



Space product assurance

Design rules for printed circuit boards

Foreword

This Standard is one of the series of ECSS Standards intended to be applied together for the management, engineering and product assurance in space projects and applications. ECSS is a cooperative effort of the European Space Agency, national space agencies and European industry associations for the purpose of developing and maintaining common standards. Requirements in this Standard are defined in terms of what shall be accomplished, rather than in terms of how to organize and perform the necessary work. This allows existing organizational structures and methods to be applied where they are effective, and for the structures and methods to evolve as necessary without rewriting the standards.

This Standard has been prepared by the ECSS-Q-ST-70-12C Working Group, reviewed by the ECSS Executive Secretariat and approved by the ECSS Technical Authority.

Disclaimer

ECSS does not provide any warranty whatsoever, whether expressed, implied, or statutory, including, but not limited to, any warranty of merchantability or fitness for a particular purpose or any warranty that the contents of the item are error-free. In no respect shall ECSS incur any liability for any damages, including, but not limited to, direct, indirect, special, or consequential damages arising out of, resulting from, or in any way connected to the use of this Standard, whether or not based upon warranty, business agreement, tort, or otherwise; whether or not injury was sustained by persons or property or otherwise; and whether or not loss was sustained from, or arose out of, the results of, the item, or any services that may be provided by ECSS.

Published by: ESA Requirements and Standards Division
 ESTEC, P.O. Box 299,
 2200 AG Noordwijk
 The Netherlands

Copyright: 2014© by the European Space Agency for the members of ECSS

Change log

ECSS-Q-ST-70-12C 14 July 2014	First issue
----------------------------------	-------------

Table of contents

Change log	3
Introduction	14
1 Scope	15
2 Normative references	16
3 Terms, definitions and abbreviated terms	17
3.1 Terms from other standards.....	17
3.2 Terms specific to the present standard.....	17
3.3 Abbreviated terms.....	25
4 Principles	27
4.1 Qualified PCBs.....	27
4.2 Manufacturing tolerances.....	27
4.3 Reliability of design.....	28
5 Design review and MRR	29
5.1 Overview.....	29
5.2 Documentation.....	29
6 General design and production requirements	31
6.1 Reliability of design.....	31
6.2 Choice of materials and build-up.....	31
6.2.1 Overview.....	31
6.2.2 Material selection.....	34
6.3 Selection of the PCB manufacturer.....	34
6.4 Traceability and marking.....	34
7 Rigid PCBs	35
7.1 PCB build-up.....	35
7.1.1 General.....	35
7.1.2 Copper styles.....	35

7.1.3	Dielectric thickness	37
7.2	PCB dimension	40
7.3	Thickness of PCB	40
7.3.1	General	40
7.3.2	Polyimide PCB	41
7.3.3	Epoxy PCB	41
7.3.4	Number of copper layers in PCB	41
7.3.5	Aspect ratio of vias	41
7.4	Track width and spacing	42
7.4.1	General	42
7.4.2	Manufacturing tolerances for width and spacing	42
7.4.3	External layers	43
7.4.4	Normal pitch tracks on internal layers	44
7.4.5	Fine pitch tracks on internal layers	45
7.4.6	Routing to AAD footprint on internal layers	46
7.5	Pad design	47
7.5.1	Non-functional pad removal	47
7.5.2	Pad dimensions	47
7.5.3	Non-circular external pads	49
7.6	Copper planes in rigid PCB	50
7.7	Design considerations for the prevention of sliver and peelable	51
7.8	PCB surface finish	51
7.8.1	Metallization	51
7.8.2	Solder mask	52
8	Flex PCBs	53
8.1	Overview	53
8.2	Dynamic applications	53
8.3	PCB build-up	53
8.3.1	General	53
8.3.2	Dielectric materials	53
8.3.3	Copper cladding	54
8.3.4	Copper planes in flex PCB	54
8.4	Track design	55
8.5	Through holes	56
8.5.1	Annular ring	56
8.5.2	Vias and pads	57
8.5.3	Tear drop pad for flex PCB	57

8.6	Bending radius.....	58
8.6.1	Overview.....	58
8.6.2	General.....	58
8.7	Sculptured flex PCB.....	59
8.7.1	Overview.....	59
8.7.2	General.....	59
8.7.3	Copper foil dimensions for build-up.....	59
8.7.4	Connection finger.....	61
8.7.5	Through-holes.....	61
8.7.6	Bending radius.....	62
9	Rigid-flex PCBs.....	63
9.1	Overview.....	63
9.2	General.....	63
9.3	Build-up.....	64
9.4	Cover layer.....	65
9.5	Interface of rigid part and flexible part.....	65
9.6	Pads.....	65
10	Thermal rules and heat sinks.....	66
10.1	Overview.....	66
10.2	General requirements.....	66
10.3	Specific requirements for external heat sink.....	66
10.3.1	Overview.....	66
10.3.2	Construction of the interface between PCB and heat sink.....	66
10.3.3	Dimensional requirements.....	68
10.4	Specific requirements for internal heat sink.....	69
10.4.1	General.....	69
10.4.2	Cu thickness and type.....	70
10.4.3	CIC and Molybdenum inserts.....	70
10.4.4	Dimensional requirements.....	71
11	HDI PCBs.....	73
11.1	Overview.....	73
11.2	Justification.....	73
11.3	Microvia technology.....	74
11.4	Microvias.....	74
11.4.1	Build-up of microvia layers.....	74
11.4.2	Design of microvias.....	76

11.4.3	Pad design for microvia.....	77
11.4.4	Annular ring for microvias.....	77
11.5	Core PCB for HDI	78
11.5.1	General build-up	78
11.5.2	Annular ring on vias for fine pitch footprint	78
11.5.3	Track width and spacing on external layers.....	79
11.5.4	Track width and spacing on internal layers for impedance control and routing to AAD.....	79
11.5.5	Track width and spacing on internal layers for differential pair routing within the footprint of 1,0 mm pitch AAD.....	80
11.5.6	Aspect ratio of vias for footprint of AAD with 1 mm pitch	81
12	PCBs for high frequency applications.....	83
12.1	Material selection.....	83
12.2	Build-up of RF PCB	83
12.3	Embedded film resistors	83
12.4	Thickness of RF PCB	85
12.5	Track width and spacing	85
12.5.1	External layers	85
12.5.2	Internal Layers	86
12.6	Pad design	86
12.6.1	Pad dimensions	86
12.6.2	Non-functional pads	86
12.7	Surface finish.....	86
12.8	Profiled layers and vias.....	86
13	Electrical requirements for PCB design	88
13.1	Overview	88
13.2	General.....	89
13.3	PCB drying	89
13.4	Electrical characteristics	90
13.5	Floating metal.....	90
13.6	Current rating.....	91
13.6.1	Overview.....	91
13.6.2	Requirements for temperature increment	91
13.6.3	Requirements for the model IPC-2152 for current rating	91
13.6.4	Amendments to the model from IPC-2152	93
13.7	Provisions to prevent open circuit failure on critical tracks	94
13.7.1	Overview.....	94

13.7.2	Routing	94
13.8	Voltage rating	94
13.8.1	Overview.....	94
13.8.2	General requirements	95
13.8.3	Spacing on flex and rigid-flex laminate	97
13.8.4	Conformal coating.....	97
13.9	Double insulation design rules for critical tracks.....	98
13.9.1	Overview.....	98
13.9.2	Critical nets	98
13.9.3	Prevention of short circuit.....	99
13.10	Insulation distance of combined requirements on rigid PCB	103
13.11	Controlled impedance tracks	108
13.11.1	Definitions specific to controlled impedance	108
13.11.2	General rules	108
13.11.3	Microstrip and stripline	108
13.11.4	Line impedance termination for end-to-end configuration.....	109
13.11.5	Line impedance termination for multidrop configuration	109
13.12	Digital PCB	110
13.12.1	Overview.....	110
13.12.2	Zone management and routing	110
13.12.3	Criticality of digital signals	111
13.13	Analog PCB.....	112
13.13.1	Overview.....	112
13.13.2	Criticality of analog signals.....	112
13.13.3	Routing and shielding.....	113
13.14	Mixed analog-digital PCB.....	114
14	Design for assembly.....	115
14.1	Overview	115
14.2	General.....	115
14.3	Placement requirements.....	116
14.3.1	Conductive patterns	116
14.3.2	Components.....	117
14.3.3	Component pads.....	119
14.3.4	Fan out of SMT pads.....	120
14.3.5	Fan out of PTH.....	122
14.4	Specific requirements for fused tin-lead finish.....	123
14.5	Dimensional requirements for SMT foot print	123

14.5.1	Overview.....	123
14.5.2	General.....	124
14.5.3	Bipolar components	125
14.5.4	SOIC components.....	126
14.5.5	J-leaded components.....	127
14.5.6	LCC components	128
14.5.7	Flat pack components.....	131
14.5.8	AAD components	132
15	Design of test coupon.....	133
15.1	Design rules for test coupon	133
15.2	Test coupon design	133
Annex A	(normative) PCB definition dossier - DRD	136
A.1	DRD identification.....	136
A.1.1	Requirement identification and source document.....	136
A.1.2	Purpose and objective.....	136
A.2	Expected response.....	136
A.2.1	Scope and content	136
A.2.2	Special remarks	142
A.2.3	Example figures	142
Annex B	(normative) PCB manufacturing dossier – DRD.....	147
B.1	DRD identification.....	147
B.1.1	Requirement identification and source document.....	147
B.1.2	Purpose and objective.....	147
B.2	Expected response.....	147
B.2.1	Scope and content	147
B.2.2	Special remarks	148
Annex C	(informative) Example of capability list of PID	149
Annex D	(informative) Track current rating computation methodology.....	151
D.1	Introduction of the three models.....	151
D.1.1	Overview.....	151
D.1.2	Formulae for the three models	152
D.1.3	Example of current rating	153
D.2	Track current rating computation based on IPC-2152.....	153
D.3	Track current rating computation based on CNES/QFT/IN.0113.....	157
D.4	Track current rating computation based on IPC-2221A.....	161

Annex E (informative) Example of calculation of PTH pad dimensions	165
Annex F (informative) Prevention of resin starvation.....	167
F.1 Prevention of resin cracks.....	167
F.2 Prevention of resin starvation	167
Annex G (informative) Example of MRR checklist.....	169
Bibliography.....	173

Figures

Figure 3-1: Simplified build-up of HDI PCB.....	21
Figure 3-2: Projected peak-to-peak insulation distance	24
Figure 7-1: Example of automated fine pitch routing and possible improvements.....	46
Figure 7-2: Comparison between circular and oblong pads showing annular ring and the centre of the hole misregistered with the centre of the pad.....	49
Figure 7-3: Grid copper plane with openings	50
Figure 7-4: Example of peelable (left) and sliver (right).....	51
Figure 8-1: Clearance of tracks on flex PCBs.	55
Figure 8-2: Tracks on flex, defining termination and bending zones.	56
Figure 8-3: Teardrop reinforcement of terminal pads in flex PCB.	57
Figure 8-4: Bending radius of assembled flex.....	58
Figure 8-5: Sculptured flex circuit	59
Figure 8-6: Build-up of sculptured flex circuit.....	60
Figure 8-7: Connection finger of sculptured flex circuit	61
Figure 8-8 Side view of a component hole for sculptured flex.....	62
Figure 9-1 Example of a build-up of a 6 layer symmetric rigid-flex.....	63
Figure 10-1: Lay out of heat sink and PCB	69
Figure 10-2: drilling holes and slots in internal heat sink.....	72
Figure 10-3: Angle of intersection of overlapping holes	72
Figure 11-1: Build-up of HDI PCBs with staggered (left) and stacked (right) microvia.....	74
Figure 11-2: Dimple on a microvia.....	76
Figure 11-3: Tear drop pad design	79
Figure 11-4: Example of routing two differential pair tracks between vias in a footprint of an AAD with 1,27 mm pitch.	80
Figure 11-5: Example of routing two differential pair tracks between vias in a footprint of an AAD with 1 mm pitch.	82
Figure 13-1: Cross section of PCB with insulation distances.	89
Figure 13-2: Example of double insulation by increasing distance in X,Y and by not superimposing copper on adjacent layers	102

Figure 13-3: Example of double insulation by increasing distances in X,Y and by using two insulators in Z direction.	102
Figure 13-4: Edge coupled differential striplines	109
Figure 14-1: Illustration of surface pattern to adjacent surroundings.....	117
Figure 14-2: Illustration of component placing on PCB w.r.t. adjacent surrounding.....	118
Figure 14-3: Illustration of pad placing on PCB with respect to adjacent surroundings	120
Figure 14-4: Fan out from SMT pad to via	121
Figure 14-5: Example of point symmetric AAD footprint.....	122
Figure 14-6: Track width (l_c) ratio to PTH pad (D) diameter.	122
Figure 14-7: Track length (L_c) between soldering pad of PTH (D_1) and via pad (D_3)	123
Figure 14-8: Illustration of bipolar component pads	125
Figure 14-9: Illustration of SOIC component pads	126
Figure 14-10: Illustration of J-leaded component pads	127
Figure 14-11: Illustration of LCC component pads.....	129
Figure 14-12: Illustration of FP and QFP component pads	131
Figure 14-13: Illustration of AAD component pads.....	132
Figure A-1 : Example of PCB mechanical layout	142
Figure A-2 : Example of a drilling drawing	143
Figure A-3 : Example of a table indicating as-designed plated hole diameters	144
Figure A-4 : Example of build-up data	145
Figure A-5 : Example of the archive file with CRC	146
Figure D-1 : Comparison between CNES and IPC-2152 current rating at 5°C increase	152
Figure D-2 : IPC-2152: Current rating [A] vs cross sectional area [mm^2] in double log scale	154
Figure D-3 : IPC-2152: Track width [mm] vs cross sectional area [mm^2]	154
Figure D-4 : IPC-2152: Current rating based on Figure D-2, range 0-25 A	155
Figure D-5 : IPC-2152: Current rating based on Figure D-2, range 0-10 A	155
Figure D-6 : IPC-2152: Current rating based on Figure D-2, range 0-5 A	156
Figure D-7 : IPC-2152: Current rating based on Figure D-2, range 0-2 A	156
Figure D-8 : CNES/QFT/IN.0113: Current rating [A] vs cross sectional area [mm^2] in double log scale	158
Figure D-9 : CNES/QFT/IN.0113: Track width [mm] vs cross sectional area [mm^2]	158
Figure D-10 : CNES/QFT/IN.0113: Current rating based on Figure D-8, range 0-25 A	159
Figure D-11 : CNES/QFT/IN.0113: Current rating based on Figure D-8, range 0-10 A	159
Figure D-12 : CNES/QFT/IN.0113: Current rating based on Figure D-8, range 0-5 A	160
Figure D-13 : CNES/QFT/IN.0113: Current rating based on Figure D-8, range 0-2 A	160
Figure D-14 : IPC-2221A: Current rating [A] vs cross sectional area [mm^2] in double log scale	162
Figure D-15 : IPC-2221A: Track width [mm] vs cross sectional area [mm^2]	162

Figure D-16 : IPC-2221A: Current rating based on Figure D-14, range 0-25 A	163
Figure D-17 : IPC-2221A: Current rating based on Figure D-14, range 0-10 A	163
Figure D-18 : IPC-2221A: Current rating based on Figure D-14, range 0-5 A	164
Figure D-19 : IPC-2221A: Current rating based on Figure D-14, range 0-2 A	164
Figure E-1 : PCB manufacturing tolerances for registration and annular ring for HDI PCB	165
Figure E-2 : PCB manufacturing tolerances for registration and annular ring for rigid- flex sequential PCB	166

Tables

Table 6-1: Characteristics of some example dielectric materials for laminates	33
Table 7-1: As-designed versus as-manufactured copper foil thickness.....	37
Table 7-2: Example of worst case as-manufactured insulation distance for laminate with double-sided copper cladding	40
Table 7-3: Tolerance on track width and spacing of internal and external layers	43
Table 7-4: Minimum as-manufactured track width and spacing for external layers as a function of copper thickness	44
Table 7-5: Minimum as-manufactured track width and spacing for internal layers as a function of copper thickness	45
Table 8-1: Minimum as-manufactured dimensions of sculptured flex conductor	60
Table 12-1: Minimum spacing and width as-designed for RF elements on external layers	86
Table 13-1: Legend of terms	88
Table 13-2: Current overload limits.....	90
Table 13-3: Minimum insulation distance as function of voltage on as-manufactured PCB for rigid laminate.	96
Table 13-4: Minimum insulation distance as function of voltage on as-manufactured PCB for flex laminate.....	97
Table 13-5: Minimum double insulation distance as function of voltage on as- manufactured PCB for rigid laminate.....	100
Table 13-6: Minimum double insulation distance as function of voltage on as- manufactured PCB for flex laminate.....	100
Table 13-7: Minimum insulation distances on rigid PCB as function of all combined requirements (<i>part 1 of 4</i>).....	104
Table 14-1: Minimum distance as-manufactured of the surface pattern to the adjacent surroundings	116
Table 14-2: Minimum distance as-manufactured of components to adjacent surroundings	118
Table 14-3: Minimum distance as-manufactured to component pads	119
Table 14-4: Minimum track length to component pads.....	122
Table 14-5 Legend for dimensions of components and footprint.....	124

Table 14-6 As-manufactured pad sizes for bipolar components	125
Table 14-7: As-manufactured pad sizes for SOIC components	126
Table 14-8: As-manufactured pad sizes for J-leaded components	128
Table 14-9: As-manufactured pad sizes for LCC components	130
Table 14-10: As-manufactured pad sizes for FP and QFP components	131
Table 14-11: As-designed pad diameter for AAD components	132
Table C-1 : Example of technical capabilities of PCB manufacturer as specified in the PID.....	149

Introduction

PCBs are used for the mounting of electronic components to produce PCB assemblies that perform electrical functions. The PCBs are subjected to thermo-mechanical stress during assembly such as soldering of components, rework and repair under normal terrestrial conditions. In addition the assembled PCBs are exposed to the launch and space environment. The reliability of the circuit depends on the robustness of the design, among other factors. Moreover, PCB design with high technological complexity enables the use of complex components with advanced functionality.

1 Scope

This standard specifies the requirements for the supplier and PCB manufacturer for PCB design.

This standard is applicable for all types of PCBs, including sequential, rigid and flexible PCBs, HDI and RF PCBs.

This standard can be made applicable for other products combining mechanical and electrical functionality using additive or reductive manufacturing processes, as used in PCB manufacturing. Examples of such products are slip rings and bus bars.

This standard may be tailored for the specific characteristics and constraints of a space project in conformance with ECSS-S-ST-00.

2

Normative references

The following normative documents contain provisions which, through reference in this text, constitute provisions of this ECSS Standard. For dated references, subsequent amendments to, or revision of any of these publications do not apply. However, parties to agreements based on this ECSS Standard are encouraged to investigate the possibility of applying the more recent editions of the normative documents indicated below. For undated references, the latest edition of the publication referred to applies.

ECSS-S-ST-00-01	ECSS system – Glossary of terms
ECSS-Q-ST-70-02	Space product assurance - Thermal vacuum outgassing test for the screening of space materials
ECSS-Q-ST-70-08	Space product assurance - Manual soldering of high-reliability electrical connections
ECSS-Q-ST-70-10	Space product assurance - Qualification of printed circuit boards
ECSS-Q-ST-70-11	Space product assurance - Procurement of printed circuit boards
ECSS-Q-ST-70-38	Space product assurance -High-reliability soldering for surface-mount and mixed technology
ECSS-E-ST-20	Space engineering – Electrical and electronic
ECSS-E-ST-20-06	Space engineering - Spacecraft charging
IPC-2152, August 2009	Standard for determining current carrying capacity in printed board design
IPC-4101D, April 2014	Specification for base materials for rigid and multilayer printed boards
IPC-4562A, April 2008	Metal foil for printed wiring applications

Terms, definitions and abbreviated terms

3.1 Terms from other standards

For the purpose of this Standard, the terms and definitions from ECSS-S-ST-00-01 apply, and in particular for the following terms:

supplier

NOTE In the context of this standard the supplier is also responsible for the design of the PCB.

3.2 Terms specific to the present standard

3.2.1 annular ring

ring of copper pad surrounding the drilled hole

NOTE The measurement of annular ring is different on internal and external layers. See clause 6.3.1 from ECSS-Q-ST-70-11.

3.2.2 area array device (AAD)

surface mount package wherein the solder terminations are formed in a grid on the bottom of the package

NOTE BGA and CGA are specific types of AAD.

3.2.3 artwork

graphical representation of individual layers

NOTE Examples of artwork are: conductive layers, solder mask, silk screen, selective finishes, heat sink.

3.2.4 as-designed

state of the PCB in the design phase

NOTE This typically refers to dimensions associated with the designed PCB, which does not take into account manufacturing tolerances.

3.2.5 as-manufactured

state of the PCB after manufacturing

NOTE This typically refers to dimensions measured on the manufactured PCB, the final product. The dimensions are measurements that include manufacturing tolerances.

3.2.6 as-manufactured hole

hole in as-manufactured PCB after all process steps

NOTE In case of a plated through-hole, an as-manufactured hole includes plating and surface finish.

3.2.7 aspect ratio

ratio of the as-designed thickness of the build-up and the diameter of the drilled hole

3.2.8 assembled PCB

PCB with all its electronic and mechanical components mounted, having undergone all the manufacturing operations

NOTE Examples of manufacturing operations are wiring, soldering, wire-bonding, gluing, screwing, potting and conformal coating.

3.2.9 assembly house

company performing assembly of PCB

3.2.10 back-drilled hole

via with part of its metallisation removed on one side by depth controlled mechanical drilling with a larger diameter drill

3.2.11 ball grid array (BGA)

surface mount package wherein the solder balls for terminations are formed in a grid on the bottom of the package

NOTE BGA is a specific type of AAD.

3.2.12 basic copper

copper foil layer that excludes etching and plating steps

3.2.13 blind via

type of via exposed only on one side of the PCB

NOTE One method to manufacture a blind via can be by depth controlled drilling. A second method can be by sequential lamination of minimum 2 half-stacks.

3.2.14 bondply

type of cover layer with adhesive on both sides for the purpose of bonding flex laminate

NOTE Bondply is a trademark manufactured by DuPont.

3.2.15 build-up

technical representation of individual conductive layers and dielectric materials of the specific PCB design

NOTE Examples of a build-up are given in Figure 3-1 and Figure A-4

3.2.16 buried via

type of via connecting internal layers without being exposed on either surface

3.2.17 column grid array (CGA)

surface mount package wherein the columns for terminations are formed in a grid on the bottom of a package

NOTE CGA is a specific type of AAD.

3.2.18 conductive layer

electrically conductive parts of a PCB on the same layer

NOTE The conductive layer can contain tracks, planes or pads.

3.2.19 conductor

conductive elements within the PCB

NOTE Elements within the conductive layer and vias are conductors.

3.2.20 critical net

conductive circuit with a specific functionality that requires redundant solutions to avoid loss of functionality in case of any credible single failure

NOTE Redundant solutions can be double insulation, increased copper cross section, multiple vias.

3.2.21 critical track

track that is part of a critical net

3.2.22 double insulation

barrier between tracks or elements of an electronic circuit that provides insulation of tracks or elements of an electronic circuit in case of any credible single failure

3.2.23 drilled hole

hole after drilling and before plating

NOTE In case of a plated through-hole, drilled hole excludes plating and surface finish

3.2.24 electrical field

voltage per insulation distance between two conductors

NOTE This is a simplified representation of electrical field applicable for PCB design.

3.2.25 fine pitch

spacing of tracks or pads that is more dense than for normal pitch

NOTE The exact perimeter of fine pitch on external and internal layers is described in clause 7.4.

3.2.26 FR4

type of laminate and prepreg with a specific epoxy resin

3.2.27 half-stack

sequence of laminated layers that are included in further lamination sequences to form the finished PCB

3.2.28 heat sink

layer of thick metal with the purpose of thermal dissipation internal or external of the PCB

NOTE Term “thermal drain” is synonymous with the word “heat sink”.

3.2.29 high density interconnect (HDI)

technology on PCB related to the pattern that allows a smaller pitch in the footprint on the surface pattern and a higher density of internal signal routing than for conventional PCBs

NOTE 1 The exact perimeter of HDI technology is described in clause 11.

NOTE 2 This technology can be required for assembly of AAD.

NOTE 3 An example of HDI build-up is shown in Figure 3-1.

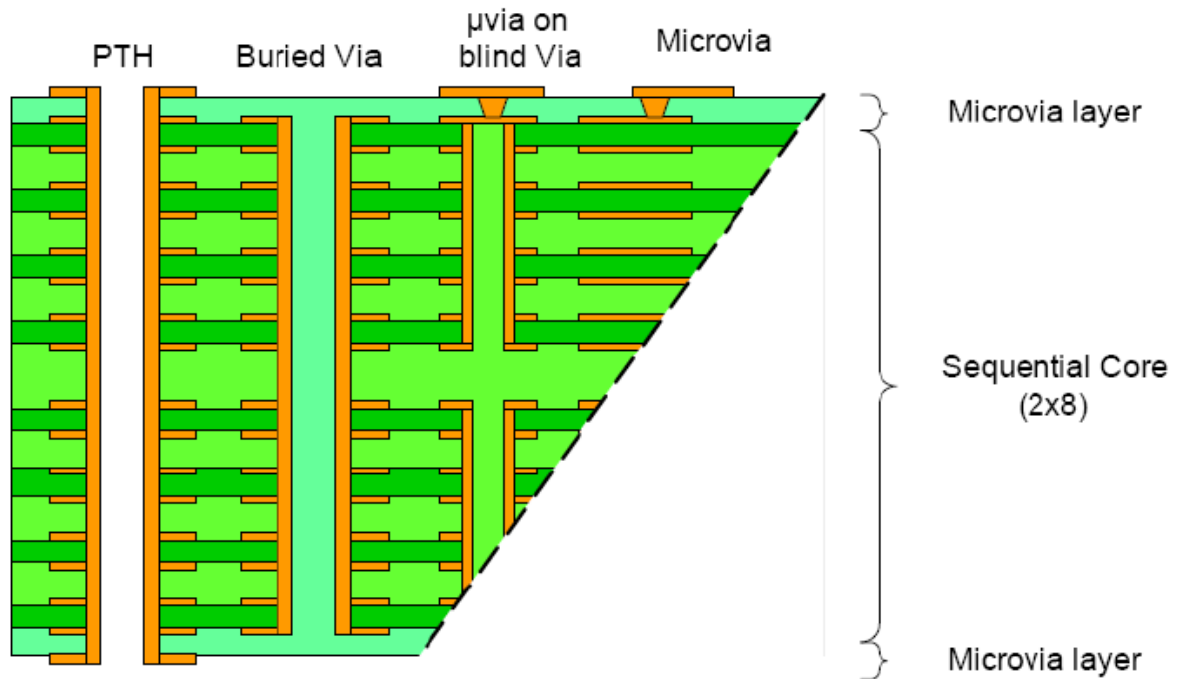


Figure 3-1: Simplified build-up of HDI PCB

3.2.30 high speed signal

electronic functionality that requires specific design precautions to maintain time dependant signal integrity

NOTE This is further specified in IPC-2251 and IPC-2141A.

3.2.31 hole wall pull away

adhesion defect between copper of the hole wall and resin

3.2.32 HTE

grade of copper foil with the grade designation "high temperature elongation electrodeposited"

NOTE This is specified in IPC-4562A.

3.2.33 intralayer

within the same layer in X,Y direction

3.2.34 interlayer

in between two superpositioned layers in Z direction

3.2.35 JTC

grade of copper foil

NOTE This grade of copper foil is produced by GOULD Electronics GmbH.

3.2.36 laminate

sheets of fully cured, C-stage, resin with copper cladding

NOTE Rigid laminate includes reinforcement, for example by woven glass fibres or non-woven aramid fibres.

3.2.37 microvia

blind via manufactured by laser ablation with a diameter smaller than conventional vias.

NOTE 1 Microvias are required for routing internal signals in HDI PCBs.

NOTE 2 Example of microvia configuration is shown in Figure 3-1.

3.2.38 microvia layers

layers of conductive pattern that contain microvias

NOTE microvia layers are layers 1 and 2 and opposite layers

3.2.39 mil

unit of length equal to 25,4 μm

NOTE The unit mil is in some cases customary in PCB design and therefore preferred above the SI unit.

3.2.40 no-flow prepreg

type of prepreg that has a reduced flow of resin during press cycle

NOTE The term “low-flow prepreg” is synonymous with the word “no-flow prepreg”. The terms do not specify the exact amount of flow.

3.2.41 non-functional pad

pad on internal copper layer without electrical connection

3.2.42 non-plated hole

hole in a PCB that does not contain plating or other type of conductive reinforcement

3.2.43 normal pitch

standard spacing of tracks and pads

NOTE See “fine pitch” for another category of pitch.

3.2.44 number of layers

number of layers in a PCB containing conductive elements

NOTE For example, the numbering convention is from top to bottom layer: L1, L2, ..., Ln-1, Ln.

3.2.45 panel

area of laminated layers that contains PCBs and coupons processed as a single unit

3.2.46 PCB manufacturer

entity that manufactures the PCB

3.2.47 peelable

areas of photo resist or copper that have poor adhesion to underlying substrate because of a tapered shape with a narrow dimension on one end

3.2.48 plated through-holes (PTH)

metal-plated holes drilled through all the layers used for assembly of components

3.2.49 printed circuit board (PCB)

product resulting from the process of selectively etching unwanted copper from surfaces of copper clad insulating substrates to form a desired circuitry pattern which is metal-plated and laminated

NOTE Examples of specific PCB technologies are rigid, flexible, rigid-flex, double sided, multilayer, sequential, RF and HDI PCBs

3.2.50 prepreg

sheets of partly cured, B-stage, resin with reinforcement

NOTE Examples of reinforcement are woven glass fibres and non-woven aramid fibres.

3.2.51 RF elements

conductive pattern on PCB with specific RF functionality

NOTE For example filters, combiners, splitters, couplers. RF elements do not have voltage ratings or solder connections.

3.2.52 resin starvation

region in a PCB that has an insufficient amount of resin to wet out completely the reinforcement

NOTE This is shown by low gloss, dry spots or exposed fibres.

3.2.53 review item

design feature to be included into the PCB definition dossier and reviewed during the PCB design review

3.2.54 projected peak-to-peak insulation distance

worst-case minimum thickness of dielectric material that includes thickness tolerance of the laminate core and the maximum roughness of copper surface treatment

NOTE Figure 3-2 specifies how this is measured.

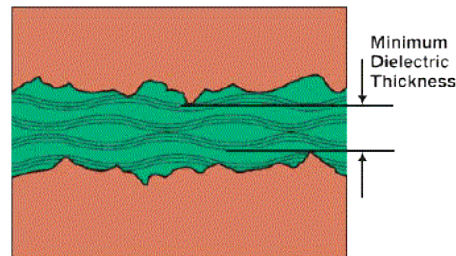


Figure 3-2: Projected peak-to-peak insulation distance

3.2.55 serialization

process of numbering of each unique PCB by the PCB manufacturer for traceability

3.2.56 sliver

pieces of photo resist or copper that have poor adhesion to underlying substrate because of the small surface area.

NOTE For example, a surface area below 0,01 mm²

3.2.57 soldering pad

conductive part intended for soldering components on the PCB

3.2.58 spacing

insulation distance

NOTE The term “gap” is synonymous with the word “spacing”, but this term is not used in the present standard.

3.2.59 stack

assembly comprising of one sequence of laminated layers

NOTE Half-stack identifies the sub-assembly.

3.2.60 test pad

pad that is dedicated for electrical testing on the PCB

3.2.61 track

conductive part routing the electrical connection between the pads

NOTE The term “line” and the term “conductive track” are synonyms with the word “track”

3.2.62 via

metal-plated hole drilled through all layers, only used for interconnection between layers

NOTE A via is not used for assembly

3.2.63 X,Y direction

orientation in the plane of the PCB, along length and width of the PCB

3.2.64 Z direction

orientation perpendicular to the plane of the PCB, along height of the PCB

3.3 Abbreviated terms

For the purpose of this Standard, the abbreviated terms from ECSS-S-ST-00-01 and the following apply:

Abbreviation	Meaning
AAD	area array device
AC	alternating current
AOI	automated optical inspection
BGA	ball grid array
CAD	computer aided design
CAE	computer aided engineering
CGA	column grid array
CIC	Copper – Invar - Copper
CoC	certificate of conformance
CRC	cyclic redundancy check
CTE	coefficient of thermal expansion
Cu	copper (element)
DC	direct current
DRD	document requirements definition
DWV	dielectric withstanding voltage
GND	electrical signal to ground
EMC	electromagnetic compatibility
FAI	first article inspection
FP	flat pack
FR4	type of epoxy resin for PCBs
HDI	high density interconnect
i.a.w.	in accordance with

Abbreviation	Meaning
I/O	input/output
LCC	leadless chip carrier
LF	low frequency
MRR	manufacturing readiness review
PCB	printed circuit board
PID	process identification document
ppm	parts per million (10^{-6})
PTH	plated through hole
PTFE	polytetrafluoroethylene
QFP	quad flat pack
RF	radio frequency (high frequency)
SOIC	small outline integrated circuits
SPF	single point failure
sq	square (in unit Ω/sq)
TDR	time domain reflectometry
Tg	temperature of glass transition
Th	thickness
Vcc	power supply voltage
Vrms	voltage root mean square

4

Principles

4.1 Qualified PCBs

Qualified PCBs for space application meet the following conditions:

- a. PCB is procured in conformance with requirements of ECSS-Q-ST-70-11,
- b. PCB is procured from a PCB manufacturer that is qualified in conformance with requirements of ECSS-Q-ST-70-10,
- c. the PCB technology is qualified in conformance with requirements of ECSS-Q-ST-70-10 for that PCB manufacturer,
- d. the PCB design is in conformance with the requirements of ECSS-Q-ST-70-12, and
- e. the PCB design, the manufacturing process and materials are in conformance with the PID of the PCB manufacturer.

PA requirements of space projects use this standard as applicable document. Based on their heritage, suppliers can propose the re-use of existing PCB designs that are “recurrent” and that are not in compliance with all design requirements of this standard. The possible acceptability of those cases is reviewed by the project on case by case basis. The PCB design is “recurrent” only when no changes are made in the artwork, routing, lay-out, build-up, material selection.

4.2 Manufacturing tolerances

Dimensional measurements of PCBs are “as-designed” except when explicitly mentioned “as-manufactured”.

Design values are affected by the manufacturing tolerances. The minimum value on the PCB as-manufactured equals the design value subtracted with the manufacturing tolerance. See definition of terms in clause 3.2.

4.3 Reliability of design

Specific design features of PCBs are recorded as Review Items in the PCB definition dossier. This is done when the design features include a higher technological complexity that can affect manufacturability or (thermal) reliability. These design features are specifically evaluated by qualification prior to inclusion in the PID.

Requirements in this standard generally state minimum dimensions permitted. It is good practice to use minimum dimensions only when necessary to achieve the desired functionality and to design with margin when possible. For instance, it is recommended to implement minimum insulation distance between tracks only when space limitations prevent larger insulation distance. It is recommended to implement fan out of tracks which allows for larger insulation distances where more space is available.

The robustness of the PCB can be affected when combining multiple design features at the limit of the requirement or recorded as Review Item. The PCB manufacturer and the supplier evaluate this risk during the design review and provide formal approval during the MRR. The reliability of the manufactured PCB is evaluated by inspection on representative coupons.

5

Design review and MRR

5.1 Overview

This clause specifies the iterative process of designing a PCB in collaboration with the PCB manufacturer and specifies associated documentation for the formal authorization for manufacture.

5.2 Documentation

- a. The supplier shall issue the PCB definition dossier in conformance with the DRD in Annex A.
- b. The supplier shall list design features as Review Items in conformance with clause A.2.1<7> from the DRD of the Annex A.
- c. The supplier may list other design features as Review Items .
- d. The supplier shall submit the PCB definition dossier to the PCB manufacturer for the design review.
- e. The supplier should submit to the PCB manufacturer the draft PCB definition dossier or a list of specific Review Items specified in the requirement 5.2b in the design phase, when those Review Items are at the limit of the PID capability.

NOTE Examples of Review Items that can be on the limit of PID capability are: dimensions, number of layers, maximum aspect ratio, number of sequences, material selection or a combination of these. Consultation with PCB manufacturer at this early design stage is recommended to avoid unnecessary risks. Multiple iterations can be performed to achieve an agreed PCB definition dossier.

- f. The PCB manufacturer shall review the PCB definition dossier including Review Items for compliance with the PID.

-
- g. As a result of the design review, the PCB manufacturer may identify to the supplier additional Review Items in the PCB definition dossier.

NOTE Review Items can exist for PCB designs even within the capability of the PID. These Review Items and methods to mitigate them are specified in this standard as best practices and recommendations.

- h. The PCB manufacturer shall provide the PCB manufacturing dossier in conformance with DRD in the Annex B.
- i. The PCB manufacturer shall issue to the supplier a MRR checklist, which is part of the PCB manufacturing dossier, in conformance with the DRD in the Annex B.

NOTE An example of a MRR checklist is given in Annex G.

- j. During the MRR, supplier and PCB manufacturer shall approve the PCB manufacturing dossier, the PCB definition dossier and all Review Items.

NOTE The approvals are recorded in the MRR checklist.

- k. The MRR shall provide authorisation for manufacture to the PCB manufacturer.

6

General design and production requirements

6.1 Reliability of design

- a. The supplier shall design the PCB with margin.

NOTE This is also described in clause 4.3.

6.2 Choice of materials and build-up

6.2.1 Overview

For the selection of materials for PCB a number of physical properties are important, such as Tg, Young's modulus, dielectric constant, CTE, hygroscopy. Table 6-1 provides an overview of some commonly used materials and their physical properties. The CTE of the as-manufactured PCB depends on the total copper thickness, which has a CTE of about 16 ppm/K.

It is important that for the selection of the technology for the PCB design the following is taken into account:

- CTE mismatch between devices and PCB,
- moisture sensitivity of PCB function,
- interconnection constraints,
- lay-out constraints.

Rigid-flex boards can be preferred to reduce the interconnection constraints.

Polyimide and epoxy are the most commonly used materials. Polyimide has an advantage over epoxy because it is more thermally stable and outperforms epoxy in thermal cycling and assembly. At the same time, polyimide is more hygroscopic than epoxy, which affects PCB ground use and assembly. Epoxy is available with different glass transition temperatures (Tg). Higher glass transition temperature is recommended for epoxy because it provides better thermal stability. Thermal expansion of resin in Z direction below and above Tg is typically higher than expansion of copper. Therefore it is beneficial for thermal endurance to use materials with low CTE in Z direction. Flame

retardants in base materials are commonly required for ground based applications but are not necessary for space based PCBs.

For high frequency performance typically PTFE based materials are used. These materials require special lamination cycles as adhesion is generally poorer compared to polyimide and epoxy.

IPC-4121 provides a comparison of various properties for core constructions as function of resin and glass type.

Table 6-1: Characteristics of some example dielectric materials for laminates

Type of material	Dielectric constant		Dissipation Factor		Tg (°C)	water absorption (%)	Volume density (g/cm ³)	CTE (ppm/K) below Tg			CTE (ppm/K) above Tg			Thermal conductivity (W/m.K)	Example
	at 1 MHz	at 10 GHz	at 1 MHz	at 10 GHz				X	Y	Z	X	Y	Z		
Epoxy HTg	4,24	3,92	0,015	-	180	0,15	-	13 - 14	13 - 14	45	14 - 17	14 - 17	230	0,4	370 HR (Isola)
	4,24	3,92	0,015	-	170	0,15	-	13 - 14	13 - 14	45	14 - 17	14 - 17	230	0,4	IS420 (Isola)
Polyimide / Glass	4,2	-	0,01	-	> 250	0,26	1,6	16	16	51	-	-	158	0,2	35N (Arlon)
	4,2	-	0,01	-	250	0,27	1,6	16	16	55	-	-	149	0,2	85N (Arlon)
Polyimide / Aramid	3,6	-	0,014	-	250	0,6	1,37	6 - 9	6 - 9	93	-	-	279	0,2	85NT (Arlon)
Polyimide flex laminate	3,4	3,3	0,003	-	220	0,8	-	25	25	-	-	-	-	-	AP (DuPont)
Ceramic reinforced PTFE	-	2,94	-	0,0012	-	0,02	2,1	16	16	24	-	-	-	0,6	RT/Duroid 6002 (Rogers)
Hydrocarbon and ceramic reinforced glass	-	3,38	-	0,0027	> 280	0,06	1,79	11	14	46	-	-	-	0,71	RO 4003 (Rogers)
Glass reinforced PTFE	2,20	2,20	0,0004	0,0009	-	0,02	2,2	31	48	237	-	-	-	0,2	RT/Duroid 5880 (Rogers)
Ceramic thermoset	-	9,8	-	0,002	-	0,16	2,77	19	19	20	-	-	-	0,76	TMM10i (Rogers)

6.2.2 Material selection

- a. Selection of the base laminate materials shall be performed in compliance with the requirements from the clause 5.2.1 of the ECSS-Q-ST-70-11.
- b. The material selection for all types of PCBs shall be in compliance with the requirements from the clauses 7.1 to 7.6 of the ECSS-Q-ST-70-11.

NOTE Examples of types of PCBs are: single sided, double sided, multilayer, sequential, rigid, rigid-flex, flexible, sculptured flex, HDI, RF.

- c. The as-manufactured PCB shall be in conformance with the outgassing requirements of the clause 5.5.3 from the ECSS-Q-ST-70-02 or project specific requirements on cleanliness.

NOTE The outgassing requirements are CVC_M<0,1% and RML<1,0%.

6.3 Selection of the PCB manufacturer

- a. The supplier shall procure PCBs from a PCB manufacturer with a qualification approval for an identified technology in conformance with the requirements from the clause 5 to clause 9 of the ECSS-Q-ST-70-10.
- b. The supplier shall procure the PCBs in conformance with the requirements from the clause 5 to clause 7 of the ECSS-Q-ST-70-11.

6.4 Traceability and marking

- a. The PCB shall be marked with part number and revision number from the PCB definition dossier as given by the supplier.
- b. All individual PCBs within the panel shall be identified by a serial number.
- c. The serialization of the PCB shall be established before the start of the fabrication.
- d. The serial number shall identify the position of the PCB on the panel.
- e. The serial number methodology shall be identified in the PID of the PCB manufacturer.

7

Rigid PCBs

7.1 PCB build-up

7.1.1 General

- a. The build-up of the PCB and each sub-assembly should be symmetric.

NOTE This is done to avoid warp and twist.
- b. The supplier shall record asymmetric build-up as a Review Item in the PCB definition dossier.

NOTE The build-up includes the copper thickness and its distribution.
- c. In case Molybdenum or CIC layers are used, they shall be included in the build-up and recorded as a Review Item in the PCB definition dossier.

NOTE 1 Specific requirements for the build-up are listed in the PCB definition dossier in A.2.1<5>a.3(h).

NOTE 2 On external layers, copper clad laminate typically achieves better peel strength whereas copper foil achieves better registration.

7.1.2 Copper styles

- a. The thickness of the copper cladding on both sides of the laminate should be equal except on outer layers of the PCB or half-stack.

NOTE Equal copper thickness is needed when laminate is etched in same process steps, as is the case for internal layers. The exception on outer layers is made because each side is etched in different process steps and because thin copper on external layers decreases the total thickness after plating.

- b. In case the thickness of the copper cladding on the laminate is asymmetric, this asymmetric build-up of the PCB shall be recorded as a Review Item in the PCB definition dossier.

NOTE 17/70 copper thickness is more risky than 17/35 or 35/70 copper thickness. The presence of fine pitch tracks in combination with asymmetric copper further increases the difficulty to process.

- c. Except the case specified in the requirement 11.4.1e, the external and internal layers shall use basic copper thickness 70 μm , 35 μm or 17 μm .

NOTE 1 Copper layers can be implemented as copper clad laminate or as separate copper foils.

NOTE 2 Requirement 11.4.1e specifies microvia layers on HDI PCBs.

- d. Copper distribution within a layer should be homogeneous.

NOTE This is done to ensure even pressure distribution during lamination, which can cause filling voids, cracks or glass compression.

- e. Balancing of copper should be performed by including dummy non-functional copper.

NOTE This is more important for partial planes on thick copper layers such as 70 μm .

- f. Signal tracks should not be routed on copper plane layers.

- g. In case the total as-designed thickness of copper is above 700 μm , this shall be recorded as a Review Item in the PCB definition dossier.

NOTE High layer count in combination with thick copper layers can have significant impact on manufacturability and long-term reliability.

- h. Copper quality "HTE" shall be used for copper clad laminate.

NOTE Type HTE can be procured in conformance with IPC-4562 Type E, grade 3.

- i. Copper quality "JTC" or "HTE" shall be used for copper foils.

- j. The values from the Table 7-1 should be used in the design calculations for copper foil thickness.

NOTE The values indicated in Table 7-1 are minimum as-manufactured thickness for the specified as-designed foil thickness.

Table 7-1: As-designed versus as-manufactured copper foil thickness

As-designed foil thickness (μm)	Minimum as-manufactured thickness for non-plated inner layers (μm)	Minimum as-manufactured thickness for plated layers with $\geq 25 \mu\text{m}$ plating (μm)
9	6	31
12	9	34
17	11	38
35	25	53
70	56	84

7.1.3 Dielectric thickness

- a. A minimum of two sheets of prepreg shall be used for insulation between two layers.
- b. A minimum of two sheets of glass should be used in glass-reinforced laminates.
- c. In case a single sheet of glass is used in a glass-reinforced laminate it shall be recorded as Review Item in the PCB definition dossier.

NOTE Two sheets of glass reinforcement are specified to mitigate the risk of reduced insulation caused by contamination. This is also specified for laminates for class 3/A in accordance with IPC-6012 Appendix A. In case a single reinforcement in laminate is used, it is recommended to implement other means to mitigate the risk of contamination in laminate. IPC-4121 provides various properties, such as resin-to-glass ratio, for core laminate constructions.

- d. The insulation distance as-designed between two layers in rigid laminate or prepreg in Z direction should be in conformance with the values from Table 13-7.

NOTE This table specifies a minimum distance of 100 μm .

- e. Except the case specified in the 11.4.1h, the insulation distance as-manufactured between two layers in rigid laminate or prepreg in Z direction shall be in conformance with the values from Table 13-7.

NOTE 1 This table specifies a minimum projected peak-to-peak distance of 70 μm .

NOTE 2 Requirement 11.4.1h specifies microvia layers on HDI PCBs..

- f. Double-sided copper clad laminate with a thickness as-designed of 4 mil (~100 μm) shall not be used unless:
1. the voltage is less than 30V,
 2. the copper foil thickness on both sides of the laminate is 17 μm or 35 μm ,
 3. the laminate batch is screened by the PCB manufacturer to have a nominal thickness of the etched laminate of $\geq 3,8$ mil,
 4. the insulation distance as-manufactured is verified by the PCB manufacturer to be in compliance with requirement 7.1.3e, and
 5. it is recorded as a Review Item in the PCB definition dossier.
- NOTE 1 Requirement 7.1.3f.1 is in compliance with the values from Table 13-3.
- NOTE 2 The nominal thickness of etched laminate from the requirement 7.1.3f.3 can be read from the CoC. However, this does not indicate worst-case thickness as mentioned in the requirement 7.1.3e. Therefore the use of 4 mil laminate on the lower end of the tolerance between 3,5 mil and 3,8 mil is prohibited by this requirement.
- NOTE 3 The precautions specified in this requirement are defined to prevent a worst-case insulation value of less than 70 μm due to high tolerances with rough copper surface treatment as indicated in Table 7-2.
- g. Double-sided copper clad laminate with a thickness as-designed of 5 mil (~125 μm) shall not be used above 30 V unless:
1. the thickness of laminate as-manufactured is in conformance with the Table 13-3,
 2. the laminate batch is screened by the PCB manufacturer to have a nominal thickness of the etched laminate of $\geq 4,8$ mil,
 3. the as-manufactured insulation distance is verified by the PCB manufacturer to be in compliance with requirement 7.1.3g.1,
 4. it is recorded as a Review Item in the PCB definition dossier.
- NOTE 1 The thickness value of the requirement 7.1.3g.1 is ≥ 100 μm .
- NOTE 2 This value specified in the requirement 7.1.3g.2 can be read from the CoC. It does not indicate worst-case thickness as mentioned in the requirement 7.1.3g.1. This prohibits the use of 5 mil laminate on the lower end of the tolerance between 4,5 mil and 4,8 mil.
- NOTE 3 Precautions specified in this requirement are defined to prevent a worst-case insulation value of less than 100 μm due to high tolerances with rough copper surface treatment as indicated in Table 7-2.

- h. For insulation distance in Z direction of a laminate, the minimum projected peak-to-peak insulation distance specified in the requirement 7.1.3i shall be used.
- i. The minimum projected peak-to-peak insulation distance shall include the thickness tolerance of the laminate core and the maximum roughness of copper surface treatment.

NOTE Projected peak-to-peak insulation distance is specified in Figure 3-1 of IPC-4101.

- j. If available, laminates should be class D in conformance with requirements from the IPC-4101 except the cases specified in the requirement 7.1.3k.

NOTE 1 Class D guarantees a minimum projected peak-to-peak insulation distance.

NOTE 2 Typically used materials can be unavailable in class D even though this specification exists.

- k. In case other laminate classes are used, the PCB manufacturer shall calculate minimum projected peak-to-peak insulation distance in conformance with requirement 7.1.3i.

NOTE An example of this is shown in Table 7-2 for class C laminate and commonly used copper profiles.

- l. The PCB manufacturer shall define laminate and prepreg styles and amount to achieve the insulation distances as designed by the supplier in conformance with requirements of the clause 13.8.

NOTE The supplier is responsible to design insulation distances required to meet design voltages in conformance with the requirement 13.8.2c.

Table 7-2: Example of worst case as-manufactured insulation distance for laminate with double-sided copper cladding

As-designed laminate thickness	Laminate thickness tolerance i.a.w. IPC4101 Table3.7 class C.	Copper foil 18 or 35 μm		Copper foil 70 μm	
		"Very low profile" i.a.w. IPC4562 Table3.1	Worst-case as-manufactured projected peak-to-peak insulation distance	"Low profile" i.a.w. IPC4562 Table3.1	Worst-case as-manufactured projected peak-to-peak insulation distance
101,6 μm (4 mil)	$\pm 13 \mu\text{m}$	5,1 μm	78,4 μm	10,2 μm	68,2 μm
127 μm (5 mil)	$\pm 18 \mu\text{m}$	5,1 μm	98,8 μm	10,2 μm	88,6 μm
152 μm (6 mil)	$\pm 18 \mu\text{m}$	5,1 μm	123,8 μm	10,2 μm	113,6 μm
203 μm (8 mil)	$\pm 25 \mu\text{m}$	5,1 μm	167,8 μm	10,2 μm	157,6 μm
254 μm (10 mil)	$\pm 25 \mu\text{m}$	5,1 μm	218,8 μm	10,2 μm	208,6 μm

7.2 PCB dimension

The maximum dimensions of the PCB depend on the panel dimensions used and qualified by the PCB manufacturer. A typical large panel size is 45x60 cm (18x24 inch). Not all surface area of the panel is available for the PCB design as mandatory coupons are included in the panel.

7.3 Thickness of PCB

7.3.1 General

- a. PCB thickness shall be measured from bottom insulation to top insulation.

NOTE 1 The manufacturing tolerance is provided by the PCB manufacturer.

NOTE 2 It is important to include the manufacturing tolerance of 10% in conformance with the requirement 7.6.b.2 from the ECSS-Q-ST-70-11.

NOTE 3 Tolerances on individual copper clad laminate of 100 μm is about $\pm 18\%$. Tolerance on the total PCB stack is typically lower.

NOTE 4 Specific requirements for maximum thickness over metal surface finish are specified in the PCB definition dossier.

7.3.2 Polyimide PCB

- a. The thickness as-manufactured of a rigid and rigid-flex polyimide PCB shall be $\leq 4,0$ mm.
- b. The as-designed thickness of a rigid and rigid-flex polyimide PCB should be $\leq 3,6$ mm, except the case specified in the requirement 7.3.2c.
- c. The as-designed thickness of a rigid and rigid-flex polyimide PCB may be $> 3,6$ mm in case:
 1. the thickness as-manufactured is in conformance with requirement 7.3.2a,
 2. the manufacturing tolerance is $< 10\%$, and
 3. it is recorded as a Review Item in the PCB definition dossier.

7.3.3 Epoxy PCB

- a. The thickness as-manufactured of a rigid and rigid-flex epoxy PCB shall be $\leq 2,4$ mm.
- b. The as-designed thickness of a rigid and rigid-flex epoxy PCB should be $\leq 2,2$ mm, except the case specified in the requirement 7.3.3c.
- c. The as-designed thickness of a rigid and rigid-flex epoxy PCB may be $> 2,2$ mm in case:
 1. the thickness as-manufactured is in conformance with requirement 7.3.3a,
 2. the manufacturing tolerance is $< 10\%$, and
 3. it is recorded as a Review Item in the PCB definition dossier.

7.3.4 Number of copper layers in PCB

- a. Number of copper layers for polyimide PCB shall be ≤ 26 .
- b. Number of copper layers for epoxy PCB shall be ≤ 20 .

7.3.5 Aspect ratio of vias

- a. Except the case specified in the requirement 11.5.6a, the aspect ratio of vias on a rigid and rigid-flex PCB shall be ≤ 7 .

NOTE 1 There is no additional requirement for the minimum diameter of plated via in case the diameter conforms to the aspect ratio of ≤ 7 .

NOTE 2 Requirement 11.5.6a specifies aspect ratio for AAD with 1 mm pitch on HDI PCBs.

7.4 Track width and spacing

7.4.1 General

- a. A verification should be performed to prevent non-functional changes of direction of tracks.

NOTE Some automated routing software can cause unnecessary changes of direction that have no function.

7.4.2 Manufacturing tolerances for width and spacing

- a. The relative manufacturing tolerances on the track width and spacing of internal and external layers shall be as specified in Table 7-3.
- b. The maximum absolute manufacturing tolerances on the track width and spacing of internal and external layers shall be as specified in Table 7-3.

NOTE 1 For example, an internal track width of 400 μm on 35 μm copper thickness has a relative tolerance of $\pm 20\%$, i.e. 80 μm , which is limited by the maximum absolute tolerance of 50 μm .

NOTE 2 The tolerance applies to as-designed dimensions. For example, a spacing as-designed of 200 μm with a tolerance of $\pm 20\%$ results in a spacing as-manufactured of 160 μm - 240 μm . Typically, the minimum spacing of 160 μm is important for PCB design.

- c. The use of planarization, copper reduction or surface preparation processes shall be excluded in the definition of the total copper thickness.

NOTE 1 Plating of sequential half stacks can be done with different plating methods, like panel plating, selective pattern plating or a combination and can include copper surface reduction processes to reduce copper surface thickness and give best options for fine pitch etching.

NOTE 2 The plated copper thickness as-designed is 25 μm . This can be a thicker layer as-manufactured.

Table 7-3: Tolerance on track width and spacing of internal and external layers

Thickness category [μm]	Basic Cu [μm]	Plated Cu [μm]	Total Cu thickness [μm]	Relative tolerance on track width and spacing	Maximum absolute tolerance on track width and spacing
Th≤17	≤ 17	0	≤ 17	± 20 % FOR 11.5.5a.5: -10% to +20%	± 30 μm
17<Th≤60	≤ 17	1x25	≤ 42	± 20 %	± 50 μm
	35	0	35		
60<Th≤70	35	1x25	60		± 50 μm
	≤ 17	2x25	≤ 67		
70<Th≤95	70	0	70	± 70 μm	
	35	2x25	85		
	70	1x25	95		

Note: Explanation of "FOR" is given in Table 13-1.

7.4.3 External layers

- a. The as-manufactured track width and spacing as a function of copper thickness for external layers of rigid PCB shall be in conformance with the values specified in the Table 7-4.
- b. The as-designed track width and spacing for external layers of rigid PCB should be in conformance with the values specified in the Table 13-7 except the case specified in the requirement 7.4.3c.
- c. In case the as-designed track width and spacing for external layers of rigid PCB is less than the values specified in the requirement 7.4.3b, the following conditions shall be met:
 1. the manufacturing tolerances as specified in the PID are smaller than as specified in Table 7-3,
 2. the as-manufactured dimensions specified in Table 7-4 and Table 13-7 are met, and
 3. it is recorded as a Review Item in the PCB definition dossier.
- d. Tracks should not be routed on external layers.
- e. In case tracks are routed on external layers they should not be routed under components.
- f. The basic copper foil thickness of fine pitch shall be 17 μm.

- g. Fine pitch track width and spacing may be used in case:
1. fine pitch tracks are used to route to the footprint of a fine pitch component,
 2. the length of fine pitch tracks is ≤ 20 mm from the solder pad,
 3. conformal coating is applied on fine pitch tracks or pads in conformance with the requirement 13.8.4b,
 4. the voltage is $\leq 30V$, and
 5. it is recorded as a Review Item in the PCB definition dossier.

NOTE A long fine pitch track can cause solder pearls.

- h. Spacing and width of pads shall be in conformance with the requirements for tracks specified in 7.4.3a, 7.4.3b, 7.4.3c, 7.4.3f and 7.4.3g.

Table 7-4: Minimum as-manufactured track width and spacing for external layers as a function of copper thickness

Basic Cu [μm]	Plated Cu [μm]	Thickness category [μm]	Pitch	As-manufactured	
				width [μm]	spacing [μm]
17	1x 25	17<Th \leq 60	fine/normal	120/160	120/160
17	2x 25	60<Th \leq 70	fine/normal	120/160	120/160
35	1x25	17<Th \leq 60	normal	160	160
35	2x25	70<Th \leq 95	normal	240	240
70	1x25	70<Th \leq 95	normal	240	240

7.4.4 Normal pitch tracks on internal layers

- a. The as-manufactured track width and spacing as a function of copper thickness for internal layers of rigid PCB shall be in conformance with the values specified in the Table 7-5.
- b. The as-designed track width and spacing for internal layers of rigid PCB should be in conformance with the values specified in the Table 13-7 except the case specified in the requirement 7.4.4c.
- c. In case the as-designed track width and spacing for internal layers of rigid PCB is less than the values specified in the requirement 7.4.4b, the following conditions shall be met:
 1. the manufacturing tolerances as specified in the PID are smaller than as specified in Table 7-3,
 2. the as-manufactured dimensions specified in Table 7-5 and Table 13-7 are met, and
 3. it is recorded as a Review Item in the PCB definition dossier.
- d. The thickness category of copper on internal layers shall be as specified in Table 7-3.

Table 7-5: Minimum as-manufactured track width and spacing for internal layers as a function of copper thickness

Thickness category [μm]	Pitch	As-manufactured	
		width [μm]	spacing [μm]
$\text{Th} \leq 17$	fine/normal	80/104	96/104
$17 < \text{Th} \leq 60$	normal	120	120
$60 < \text{Th} \leq 70$	normal	160	160
$70 < \text{Th} \leq 95$	normal	240	240

7.4.5 Fine pitch tracks on internal layers

- a. Except the case specified in the requirement 11.5.5a, the track width and spacing for fine pitch on internal layers shall be in conformance with the as-manufactured dimensions specified in Table 7-5.

NOTE Requirement 11.5.5a specifies track width and spacing for impedance controlled routing to AAD with 1 mm pitch on HDI PCBs .

- b. In case fine pitch tracks are used, this shall be recorded as a Review Item in the PCB definition dossier.
- c. Fine pitch shall be limited to the functionality of impedance control or signal routing to AAD.
- d. Fine pitch tracks shall be limited to ≤ 30 V.
- e. For routing to AAD, fine pitch tracks shall fan out to normal pitch as soon as they are outside the AAD footprint.
- f. In case no other circuitry is adjacent that requires the use of fine pitch, the routing of fine pitch tracks shall increase the insulation distance and track width as specified for normal pitch in the Table 7-5.

NOTE 1 CAD software can implement different minimum insulation distances and track widths in areas with different space available.

NOTE 2 The Figure 7-1 shows examples of possible improvements in routing.

- g. Fine pitch shall be manufactured by using 17 μm basic copper thickness without plating steps.

7.4.6 Routing to AAD footprint on internal layers

- a. Tracks in AAD footprint shall be placed equidistant between pads to increase insulation distance to both pads.

NOTE See example of nonconformance indicated by blue arrow in Figure 7-1.

- b. Tracks shall exit from AAD footprint before changing direction to keep the insulation distance to the pad in compliance with normal pitch as specified in Table 7-5.

NOTE See example of nonconformance indicated by green arrow in Figure 7-1.

- c. Widening of tracks shall occur close to AAD footprint to provide a wider track where no other circuitry is adjacent that requires the use of fine pitch.

NOTE See example of nonconformance indicated by yellow arrow in Figure 7-1.

- d. Changing of direction of track should be in compliance with the requirement 7.4.1a.

NOTE See example of nonconformance indicated by white arrow in Figure 7-1

- e. Pads of AAD footprint may be designed with teardrop reinforcement

NOTE Pads in Figure 7-1 are not designed with teardrop reinforcement.

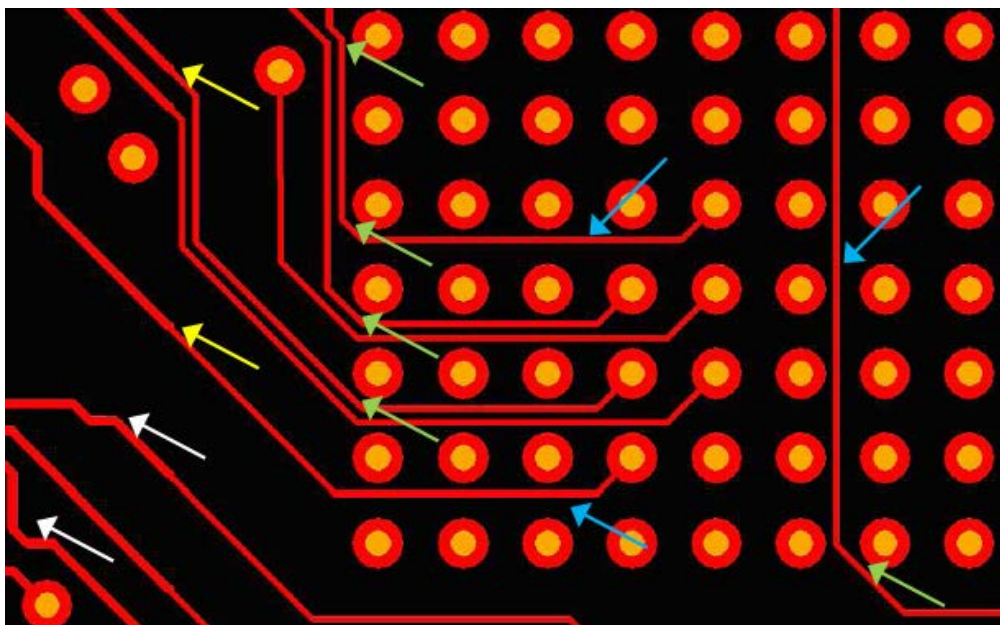


Figure 7-1: Example of automated fine pitch routing and possible improvements

7.5 Pad design

7.5.1 Non-functional pad removal

- a. Non-functional pads shall be present on all layers, except two cases specified in the requirement 7.5.1b.

NOTE The presence of non-functional pads reduce the risk of hole wall pull-away and reduce the risk of cracks in resin-rich areas.

- b. Non-functional pads may be removed:
1. when the presence of the pads degrade the electrical performance, they can be removed by the supplier, or
 2. to prevent high pressure area during lamination that cause resin starvation in the prepreg, the pads can be removed by the PCB manufacturer.

NOTE Removal of non-functional pads is approved by supplier and PCB manufacturer during MRR in conformance with the requirement 7.5.1c.6

- c. When non-functional pads are removed as specified in the requirement 7.5.1b, the following conditions shall be met:
1. non-functional pads are present on all copper plane layers,
 2. non-functional pads are present on flex laminate,
 3. maximum half of the non-functional pads in the pad stack are removed,
 4. non-functional pads are removed on maximum two consecutive layers,
 5. non-functional pads are removed on maximum one side of a laminate, and
 6. it is recorded as a Review Item in the PCB definition dossier.
- d. Non-functional pads should not be removed on layer 2 and n-1.

7.5.2 Pad dimensions

- a. Except the case specified in the requirement 11.5.2a, the annular ring as-manufactured on internal layers shall be in conformance with the requirement 9.5.b.15(d) of ECSS-Q-ST-70-10.

NOTE 1 This requirement specifies 50 μm .

NOTE 2 Requirement 11.5.2a specifies annular ring of internal pads for fine pitch footprint on HDI PCBs.

- b. The annular ring as-manufactured on component side of external layers shall be in conformance with the requirement 9.5.b.15(b) of ECSS-Q-ST-70-10.

NOTE This requirement specifies 100 μm .

- c. Except the case specified in the requirement 7.5.3b.2, the annular ring as-manufactured on solder side of external layers shall be in conformance with the requirement 9.5.b.15(a) of ECSS-Q-ST-70-10.

NOTE This requirement specifies 200 μm

- d. For component holes on solder side the minimum diameter of external pads should be the diameter of as-manufactured hole plus $\geq 0,8$ mm.
- e. For component holes on solder side the minimum diameter of external pads may be the diameter of as-manufactured hole plus $\geq 0,6$ mm in case:
1. the as-manufactured annular ring as specified in the requirement 7.5.2c,
 2. the component holes are a footprint for a nano D connector,
 3. it is recorded as Review Item in the PCB definition dossier.
- f. For via holes and component holes on component side the diameter of external pads should be the diameter of as-manufactured hole plus $\geq 0,6$ mm.
- g. For via holes and component holes on component side the diameter of external pads may be the diameter of as-manufactured hole plus $\geq 0,5$ mm in case:
1. the as-manufactured annular ring as specified in the requirement 7.5.2b, and.
 2. it is recorded as a Review Item in the PCB definition dossier.
- h. The diameter of internal pads should be the diameter of drilled hole plus $\geq 0,3$ mm.

NOTE 1 The drilled diameter is typically 0,2 mm larger than the as-manufactured minimum hole diameter.

NOTE 2 It is important to consider the number of drilling and lamination sequences and the PCB manufacturing tolerances for the design of the diameter of the pads.

- i. The diameter of pads may be the diameter of the drilled hole plus $< 0,3$ mm in case this is recorded as a Review Item in the PCB definition dossier.

NOTE Tear drop reinforcement of pads to tracks can be used for a more robust design and to mitigate the risk of a smaller designed diameter.

7.5.3 Non-circular external pads

- a. Pads for component holes on external layers shall be circular, except cases specified in the requirement 7.5.3b.
- b. Non-circular pads may be used for component holes, in case the following conditions are met:
 1. the surface area of the oblong pad is not smaller than the surface area of the circular pad as specified in the requirements from the clause 7.5.2,
 2. the oblong pad meets the requirement for annular ring of 100 μm as-manufactured, and
 3. it is recorded as a Review Item in the PCB definition dossier.

NOTE 1 For the requirement 7.5.3b.1 $A_{(\text{oblong})}$ in Figure 7-2 is the surface area of the oblong pad and $A_{(\text{circular})}$ in Figure 7-2 is the surface area of a circular pad

NOTE 2 For the requirement 7.5.3b.2 see Annular ring_(oblong) in Figure 7-2

NOTE 3 The design in the requirement 7.5.3b.2 is a deviation of the requirement 7.5.2c (and ECSS-Q-ST-70-10 requirement 9.5b.15.a) of 200 μm annular ring for pads on solder side (see Annular ring_(circular) in Figure 7-2) in one direction, compensated by a larger annular ring in the other direction.

NOTE 4 Other non-circular pad designs are possible and typically used on external layers for AAD and SMT components. Dog-bone shapes or key hole shapes can be used for the verification of solder joint quality or to prevent thermal stress on (blind or micro) vias.

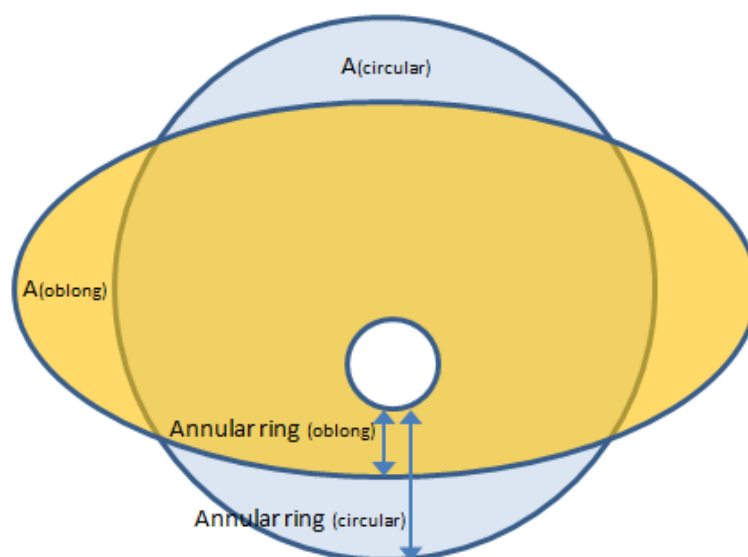


Figure 7-2: Comparison between circular and oblong pads showing annular ring and the centre of the hole misregistered with the centre of the pad

7.6 Copper planes in rigid PCB

- a. Copper planes should have additional openings in a grid format.
 - NOTE 1 For example, planes larger than 10 cm² can be designed using a grid.
 - NOTE 2 An example of a grid plane is shown in Figure 7-3.
- b. The grid should be offset between layers.
 - NOTE These additional openings act as venting holes for desorption of humidity out of the PCB. For example, a grid can be 0,4 mm lines at a pitch of 0,6 mm and at 45° angle. The thermal and electrical resistance across such hatch plane is a factor of 2,5 larger than across a solid plane, assuming pattern as specified above. This can affect both power distribution and signal integrity, in particular for RF or digital PCB.
- c. For partial copper planes, balancing of copper should be performed in conformance with requirements 7.1.2d and 7.1.2e.
- d. Copper layers should be placed symmetrically within the build-up.
 - NOTE This is done to limit warp and twist.
- e. In case copper layers are placed asymmetrically within the build-up, this shall be recorded as a Review Item in the PCB definition dossier.

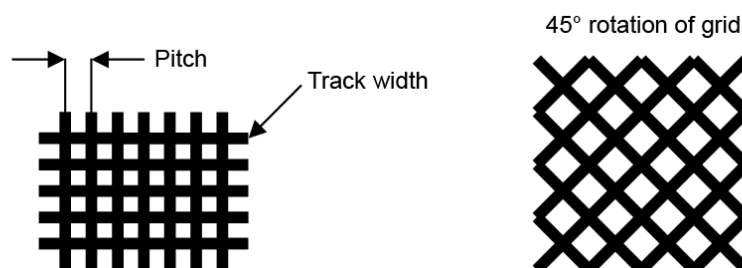


Figure 7-3: Grid copper plane with openings

7.7 Design considerations for the prevention of sliver and peelable

- a. All areas of the conductive patterns as well as areas of insulating dielectric shall have a dimension of $\geq 0,1$ mm in X and Y direction after etch compensation.

NOTE This is done to prevent sliver or peelable see Figure 7-4.

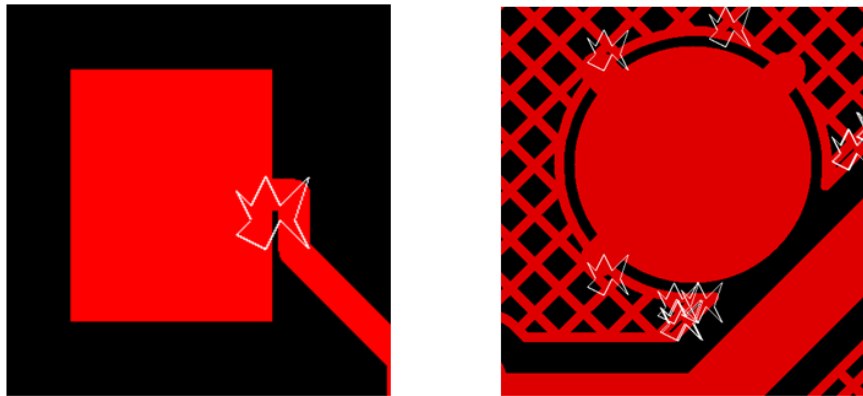


Figure 7-4: Example of peelable (left) and sliver (right)

7.8 PCB surface finish

7.8.1 Metallization

- a. Plating of metallic layers and finishes shall be done in conformance with the requirements from the clause 5.2.3 of the ECSS-Q-ST-70-11.

NOTE Qualified surface finishes are tin-lead reflowed in hot oil and electrolytic gold. Electrolytic gold can be plated over copper or nickel.

- b. Except cases specified in the requirement 7.8.1d, each finish shall be on separate outer layer nets when a single PCB uses a mixed surface finish of electrolytic gold and fused tin-lead.
- c. Separate outer layer nets with mixed surface finishes shall have insulation distance of $> 500 \mu\text{m}$.

- d. Mixed surface finish may be used on the same outer layer net in case the following conditions are met:
 - 1. a tin-lead overlap on electrolytic gold of $\geq 200 \mu\text{m}$ is used, and
 - 2. the distance of gold plating to a solder pad is $\geq 200 \mu\text{m}$.
 - NOTE 1 Condition specified in the requirement 7.8.1d.1 is to ensure coverage of the plated copper.
 - NOTE 2 Condition specified in the requirement 7.8.1d.2 is to avoid gold-tin embrittlement in the solder joints.
- e. A PCB with mixed surface finishes shall be recorded as a Review Item in the PCB definition dossier.
- f. Other surface finishes than the ones specified in the requirements from 7.8.1a to 7.8.1d may be used for non-soldering purpose.

7.8.2 Solder mask

7.8.2.1 Overview

Solder mask has been used traditionally to prevent solder flowing away from AAD in a dog-bone footprint, possibly with through-hole vias. Instead, it is recommended to design with via-in-pad (blind, micro or filled via). Problems seen with solder mask include poor adhesion to tin-lead, poor outgassing performance, high risk associated with selective stripping of tin-lead, the implementation of a width of $\geq 200 \mu\text{m}$ to ensure adhesion and a transition zone of solder mask overlapping on tin-lead to prevent exposed copper and allow for registration tolerance.

7.8.2.2 Use of solder mask

- a. Solder mask shall not be used.

8

Flex PCBs

8.1 Overview

This clause specifies requirements for flex PCBs with the purpose to maintain flexibility and achieve mechanical strength.

8.2 Dynamic applications

- a. Dynamic applications with flex PCBs shall be project qualified.

NOTE The testing of flex PCBs only covers static applications and the integration of the PCB.

8.3 PCB build-up

8.3.1 General

- a. The maximum number of copper layers on a flex PCB shall be 2.
- b. Thickness of a flexible PCB shall be $\leq 0,6$ mm over the cover layers as-designed and as-manufactured.

NOTE This assumes maximum thickness of laminate, copper, adhesive and cover layer.

8.3.2 Dielectric materials

- a. Flex PCBs shall be manufactured with copper clad flexible polyimide laminate without adhesive between copper and laminate.

NOTE 1 This laminate type is named "adhesiveless".

NOTE 2 The laminate typically used is of type Pyralux AP from DuPont.

- b. The as-designed thickness of the flex laminate shall be 25 μm , 50 μm , 75 μm , 100 μm or 150 μm .

NOTE 50 μm thickness is preferred.

- c. The tolerance for the as-manufactured thickness of the flex laminate shall be:
1. for $\leq 50 \mu\text{m}$ flex laminate: $\leq \pm 12,5 \%$, and
 2. for $\geq 75 \mu\text{m}$ flex laminate: $\leq \pm 10 \%$.

NOTE This implies that for the as-manufactured PCB, the minimum insulation between two layers of flexible laminate in Z direction is $22 \mu\text{m}$.

- d. The thickness of the polyimide cover layer shall be $25 \mu\text{m}$ or $50 \mu\text{m}$.
- e. The thickness of the acrylic adhesive on the cover layer shall be $25 \mu\text{m}$ or $50 \mu\text{m}$.

NOTE A typical cover layer is of type Pyralux LF from DuPont.

8.3.3 Copper cladding

- a. The thickness of copper cladding shall be $17 \mu\text{m}$, $35 \mu\text{m}$ or $70 \mu\text{m}$.
- b. Thickness of copper cladding $70 \mu\text{m}$ should not be used.

NOTE This is recommended to avoid risk in the bonding cycle of the cover layer.

- c. In case copper cladding of $70 \mu\text{m}$ is used, this shall be recorded as a Review Item in the PCB definition dossier.
- d. Copper cladding on flex laminate of $>17 \mu\text{m}$ shall be the type "rolled and annealed".
- e. Copper cladding on flex laminate of $17 \mu\text{m}$ shall be the type "rolled and annealed" or "electrodeposited".

8.3.4 Copper planes in flex PCB

- a. Full copper planes should not be used except the case specified in the requirement 8.3.4b.
- b. In case full copper planes are used, this shall be recorded as a Review Item in the PCB definition dossier.
- c. In case a copper plane is used in a flex PCB, a grid with openings should be implemented as specified in the Figure 7-3.

NOTE The purpose of the openings is to improve adhesion of the cover layer, desorption of humidity and flexibility of the PCB. A grid can be $0,4 \text{ mm}$ lines at a pitch of $0,6 \text{ mm}$ and at 45° angle. It is preferable to have the grid offset between layers. In this case, the electrical and thermal resistance increases by a factor of approximately $\times 2,5$ compared to a solid plane.

8.4 Track design

- a. Tolerances on track width and spacing shall be as specified in the Table 7-3.
- b. Track width shall be as specified in the Table 7-5 for normal pitch.
- c. Track width shall be minimum 120 μm as-manufactured.
- d. Spacing shall be as specified in the requirements of the clause 13.8.3.
- e. Tracks should be evenly distributed over the entire width of the PCB.
- f. Non-functional tracks may be used to achieve an even distribution of tracks over the entire width of the PCB.
- g. Tracks on double-sided boards should not be superimposed and a clearance distance in X,Y direction should be implemented.

NOTE This is done to reduce the local thickness. An example of off-set tracks is shown in Figure 8-1.

- h. Edges of tracks should not be superimposed.

NOTE This is done to prevent localised high pressure during lamination.

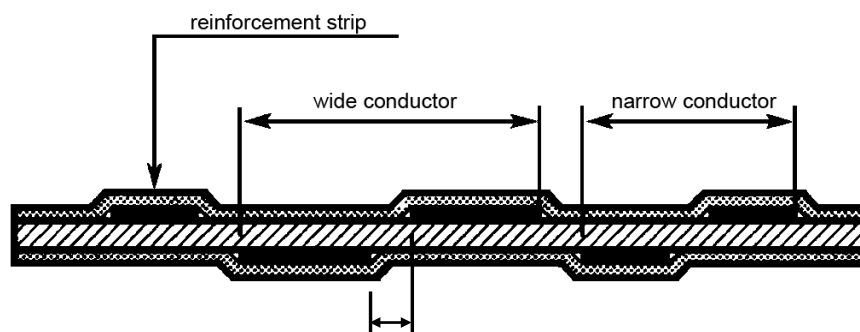


Figure 8-1: Clearance of tracks on flex PCBs.

- i. Tracks shall not change direction within the bending zone, as specified in Figure 8-2.
- j. The edge of the PCB should include on one side of the laminate a non-functional track as mechanical reinforcement with a width of $\geq 0,3$ mm.

NOTE 1 This is indicated as "Ir" in Figure 8-2.

NOTE 2 The non-functional track for mechanical reinforcement is named reinforcement strip.

- k. The reinforcement strip should be implemented on the following areas:
 1. Termination zone when no other mechanical fixing is designed apart from the termination leads,
 2. Zone where a change of direction of tracks or copper plane occurs,
 3. Bending zone.

- l. The distance of tracks or reinforcement strip to the edge of the board shall be $\geq 0,5$ mm.

NOTE This is indicated as “b4” in Figure 8-2.

- m. The distance of tracks or reinforcement strip to the edge of the board should be $\geq 1,0$ mm.
- n. Track width and spacing shall be in conformance with the values specified in the Table 13-4.
- o. When tracks change direction, the angle (p) shall be $\geq 135^\circ$ as specified in the Figure 8-2.

NOTE It is recommended to design tracks as straight as possible.

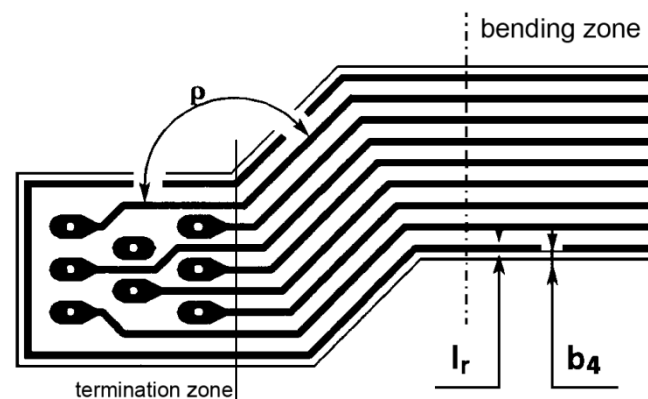


Figure 8-2: Tracks on flex, defining termination and bending zones.

8.5 Through holes

8.5.1 Annular ring

- a. Annular ring on a flex PCB shall be the distance of the surface copper from the hole wall to the clearance opening in the cover layer.
- b. Adhesive protruding from underneath the cover layer onto the surface copper shall be subtracted from the annular ring measurement.
- c. Annular ring as-manufactured on a component hole shall be $\geq 0,25$ mm.
- d. Annular ring as-manufactured on non-soldering hole shall be $\geq 0,1$ mm.

8.5.2 Vias and pads

- a. PTH shall be located only in termination zones, not in the bending zone, as specified in Figure 8-2.

- b. Terminal pads shall include PTH.

NOTE Terminal pads on single sided board have risk to be damaged during assembly. Terminal pads need to be reinforced by plating.

- c. The cover layer shall include a clearance for assembly of a PTH in conformance with the clause 8.5.1.

8.5.3 Tear drop pad for flex PCB

- a. The diameter of a circular pad with or without teardrop reinforcement shall be D_1 as specified in Figure 8-3.

- b. The diameter of an oblong pad shall be equal to D_2 as specified in Figure 8-3.

- c. The track width should be at least half the diameter of the pad except the case specified in the requirement 8.5.3d.

NOTE In the Figure 8-3 l_c is track width and D_1 , D_2 are diameters of the pad.

- d. The track width may be less than half of the diameter of the pad in case the following conditions are met:

1. a teardrop reinforcement is added to the connection of the track to the pad, and
2. the reinforced zone has a length $\geq 0,8x$ the diameter of the pad.

NOTE In Figure 8-3 Z_r is the length of the reinforced zone.

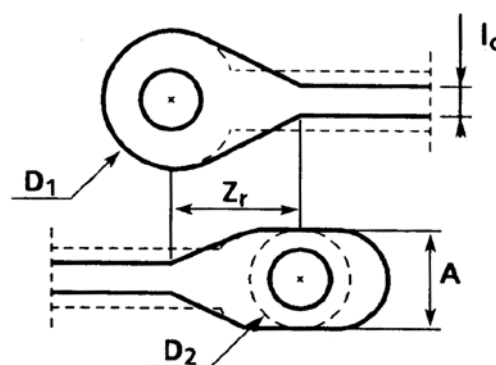


Figure 8-3: Teardrop reinforcement of terminal pads in flex PCB.

8.6 Bending radius

8.6.1 Overview

The PCB is designed to bend in L, U or S shapes with a radius of flexion that enables the PCB to be mounted and dismantled without damage. These radii depend on the type of board, the thickness of laminate and copper layers and a static or dynamic bending application, see Figure 8-4.

8.6.2 General

- a. The radius on a flex PCB having 50 μm laminate thickness and 35 μm double sided copper thickness and including cover layer shall be $\geq 12x$ total thickness of the flex for a static application.

NOTE In the Figure 8-4 R is the bending radius of a flex PCB

- b. Other PCB build-ups or dynamic applications shall be qualified by the supplier.

NOTE Other PCB build-ups include thicker laminate, thicker copper, or multilayers

- c. The termination zone shall include all termination pads.
- d. Bending of the flex PCB shall be designed not to occur on the termination zone and on an additional distance of 2 mm.

NOTE 1 In the Figure 8-4 the additional distance is indicated as X.

NOTE 2 Even though no bending is designed to occur on the termination zone, some bending can still occur on the assembled PCB due to the stress on the flex section. A stiffening of the termination zone by implementing double cover layer or adhesive can decrease the bending locally.

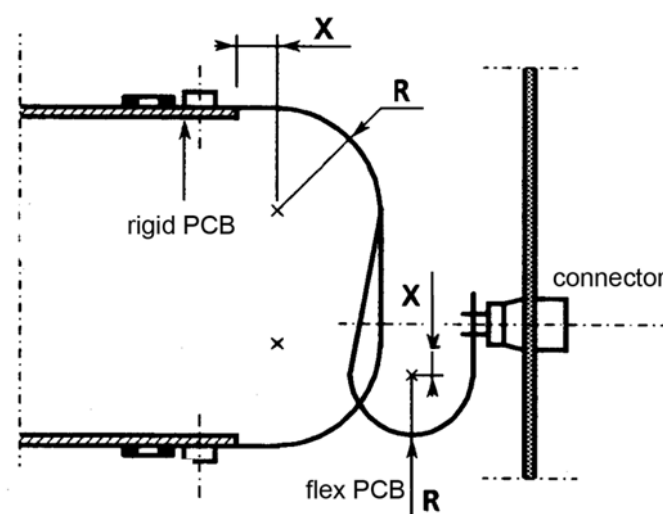


Figure 8-4: Bending radius of assembled flex

8.7 Sculptured flex PCB

8.7.1 Overview

Sculptured flex PCBs can be used as a connection between 2 PCBs or connectors. Sculptured flex circuits comprises of a copper circuit with different thickness sandwiched between two cover layers, as illustrated in Figure 8-5 and Figure 8-6.

8.7.2 General

- a. The thickness of the cover layer, excluding adhesive, shall be 25 μm .
- b. The thickness of the acrylic adhesive of the cover layer shall be 50 μm or 75 μm .

NOTE A typical cover layer is type Pyralux LF 0210 from DuPont.

- c. The surface finish shall be in conformance with the requirement 7.8.1b.

NOTE 1 This requirement specifies the use of hot oil reflowed tin-lead.

NOTE 2 The connector pins are cut after reflow and exposed copper is permitted.

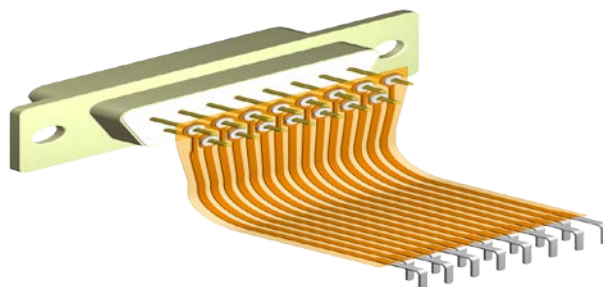


Figure 8-5: Sculptured flex circuit

8.7.3 Copper foil dimensions for build-up

- a. A single foil of copper shall be used in the build-up.

NOTE Copper clad laminate is not used.

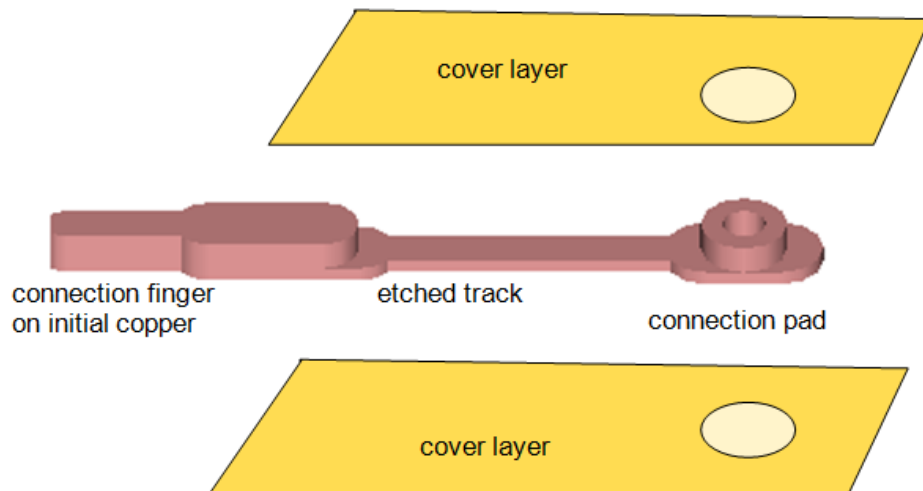
- b. The initial thickness of the copper foil shall be 250 μm or 300 μm .

NOTE Copper type CW004 grade R240 (1/2 hard)

- c. The sculptured shape of the copper shall be manufactured by selective etching of the copper foil.
- d. The build-up, initial and etched copper thickness, track width and insulation distance shall be in conformance with the minimum dimensions specified in Table 8-1.

Table 8-1: Minimum as-manufactured dimensions of sculptured flex conductor

	Thick initial copper		Thin initial copper	
	initial copper	etched copper	initial copper	etched copper
Initial copper thickness	300 μm		250 μm	
Track width for initial copper	400 μm		300 μm	
Insulation distance for initial copper	500 μm		400 μm	
Connection finger width	500 μm		350 μm	
Etched thickness		150 μm		100 μm
Track width for etched copper		300 μm		200 μm
Insulation distance for etched copper		350 μm		250 μm


Figure 8-6: Build-up of sculptured flex circuit

8.7.4 Connection finger

- a. The connection fingers may be bend in a staggered pattern to fit the footprint of the connecting PCB

NOTE An example of staggered bend connection fingers is shown in Figure 8-7.

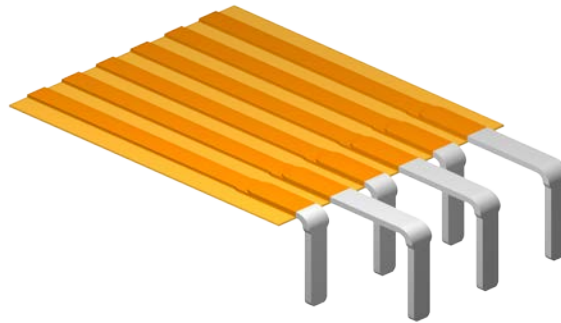


Figure 8-7: Connection finger of sculptured flex circuit

8.7.5 Through-holes

- a. Through-holes shall be manufactured by drilling into the initial copper thickness.
- b. Oblong pads should be used only in case circular pads cannot be used due to insufficient spacing in one direction.
- c. The drill diameter of the hole shall be $\geq 0,5$ mm.
- d. Component holes with circular pads should have a pad diameter as-designed of $\geq 0,4$ mm larger than the drilled diameter.

NOTE This results in a pad size of $200 \mu\text{m}$ as-designed.

- e. The minimum as-manufactured pad size on the component side may be 0.

NOTE The pad size is similar to annular ring but this term is not used for sculptured flex technology. The as-manufactured pad size with misregistration is indicated with the red arrow in Figure 8-8.

- f. There shall be no adhesive inside the hole.

NOTE Adhesive can be squeezed from underneath the cover layer and misregistration of the cover layer on component side can risk to have adhesive inside the hole.

- g. The minimum as-designed pad diameter on the solder side should be $\geq 0,7$ mm larger than the drilled diameter.

- h. The annular ring on solder side shall be in compliance with the requirement 7.5.2c.

NOTE 1 Requirement 7.5.2c specifies 200 μm .

NOTE 2 The pad diameter on solder side is indicated with the blue arrow in Figure 8-8.

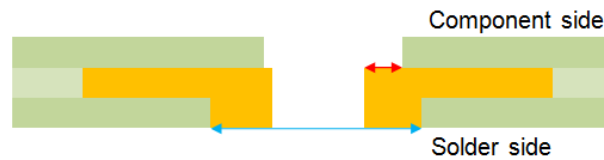


Figure 8-8 Side view of a component hole for sculptured flex

8.7.6 Bending radius

- The bending radius of sculptured flex circuit shall be $\geq 12 \times$ thickness and $\geq 3,2$ mm.
- Bending shall only occur on the etched track.
- Bending shall not occur on the full thickness copper.

NOTE The etched track is shown in Figure 8-6 and copper thickness is specified in Table 8-1.

9

Rigid-flex PCBs

9.1 Overview

Multiple flex laminates with cover layers can be incorporated within a PCB. Figure 9-1 shows an example of 1 flex laminate incorporated in the rigid-flex PCB.

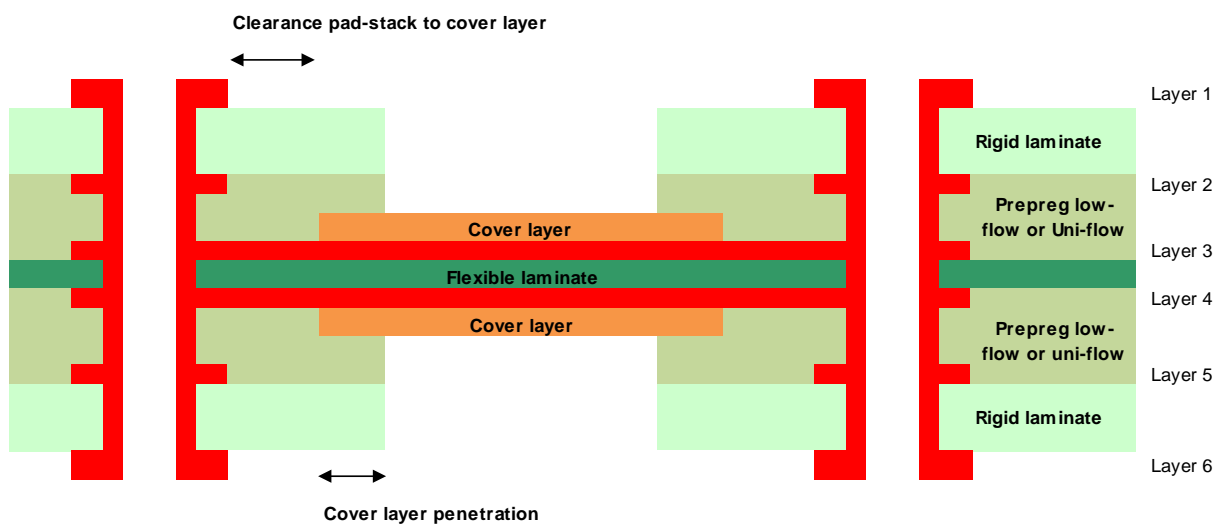


Figure 9-1 Example of a build-up of a 6 layer symmetric rigid-flex

9.2 General

- a. The design of rigid parts of a rigid-flex PCBs shall be performed in conformance with the requirements from the clause 7.
- b. The design of flex parts of a rigid-flex PCBs shall be performed in conformance with the requirements from the clause 8.

NOTE Design of rigid and flex parts include track width and spacing.

- c. No termination zone shall be defined for the flex section of a rigid-flex PCB.

9.3 Build-up

- a. The symmetry of the build-up shall be in conformance with the requirement 7.1.1a and 7.1.1b.
- b. In the rigid part of the PCB the flex layers shall be bonded by using prepreg.
- c. In the rigid part of the PCB adhesive shall not be used for bonding of the flex layers.

NOTE There are different manufacturing principles that either use standard flow prepreg as used in rigid boards, or modified no-flow type prepreps. No-flow prepreg cannot be used for resin filling of blind vias, and thereby a mix of prepreg styles can be necessary, when a sequential rigid-flex board is designed.

- d. Copper foil thickness of 70 μm shall not be used on flex or rigid layers bonded with no-flow prepreg unless it is recorded as a Review Item in the PCB definition dossier.

NOTE This can add risk for bonding in cover layer bond cycles or the bond cycle with no-flow prepreg.

- e. The number of flex laminates shall be ≤ 6 , which can result in 12 copper layers when double sided flex laminates are used.

NOTE Flex layers can be bonded together by using a bondply cover layer with adhesive on both sides. The advantage of this is improved cleanliness between layers and improved control of impedance. The disadvantage is the flexibility and bending radius of the flex stack.

- f. Three flex laminates or more in the build-up shall be recorded as Review Item in the PCB definition dossier.

NOTE Three flex laminates can include 6 copper layers.

- g. Except the cases specified in the requirement 9.3h the rigid-flex PCB shall be designed and delivered with a support frame.

- h. The rigid-flex PCB may be delivered without a support frame in case the following conditions are met:

1. the PCB manufacturer delivers the rigid-flex PCB with a support that prevents damage to the PCB during shipment, and
2. the customer implements a support structure that prevents damage to the rigid-flex PCB during assembly and handling.

9.4 Cover layer

- a. The cover layer shall be placed in the flexible section of the PCB.
- b. The cover layer shall not be placed in the rigid section of the PCB, except for the interface to the flex section as specified in the requirement 9.4c.
- c. The cover layer shall extend into the rigid section of the PCB by $\geq 1,5$ mm.
- d. In a multilayer rigid-flex PCB where flex laminates are bonded together with bondply, the ends of the cover layers and bondply should be off-set in the rigid section by 1 mm to prevent a line of weakness.
- e. The cover layer shall not overlap the internal pad of a via in the rigid section.

NOTE This is done to reduce stress on the hole wall due to high Z-expansion of cover layer adhesive and to maintain a good hole quality by not using dissimilar materials.

9.5 Interface of rigid part and flexible part

- a. The hole wall shall be ≥ 2 mm separated from the edge of the rigid part of the PCB.

NOTE A larger separation between hole wall and edge of the rigid part is recommended in case it fits within the PCB lay-out.

- b. Tracks on flex layers shall not change direction within 1 mm on both sides of the interface between flex and rigid sections.

NOTE Tracks extend 1 mm out of the rigid part of the PCB before they change direction. Tracks extend 1 mm into the rigid part of the PCB before they change direction.

- c. Bending of the flex section shall be performed in conformance with the requirements from the clause 8.6.
- d. Bending of the flex PCB shall not occur within 1,5 mm from the rigid section of the PCB.

9.6 Pads

- a. The diameter of the pads on flex layers in the rigid section of the PCB shall be $\geq 0,6$ mm larger than the diameter of the drilled hole except the case specified in the requirements 9.6b.
- b. The diameter of pads on flex layers in the rigid section of the PCB may be $< 0,6$ mm larger than the diameter of the drilled hole in case the following conditions are met:
 1. the annular ring as-manufactured is achieved in conformance with requirements from the clause 7.3.4.3.3 of ECSS-Q-ST-70-10, and
 2. it is recorded as a Review Item in the PCB definition dossier.
- c. Non-functional pads on flex laminate shall be in conformance with requirement 7.5.1c.2.

10

Thermal rules and heat sinks

10.1 Overview

Heat sinks have the purpose of improving the thermal management of the PCB especially in the vicinity of components with a high thermal dissipation or current carrying tracks. Heat sinks also have an effect on in-plane thermal expansion.

Metal inserts, such as invar or molybdenum, can also be used to restrict in plane thermal expansion under large leadless components, such as LCC and AAD. Metal inserts are considered as heat sinks and are covered by this clause.

Heat sinks can affect the solderability of the PCB as they increase the thermal mass.

10.2 General requirements

- a. Design with heat sink shall be recorded as a Review Item in the PCB definition dossier.
- b. Heat sinks shall be electrically connected in conformance with the requirement 13.5a.

10.3 Specific requirements for external heat sink

10.3.1 Overview

Heat sinks can be smaller, larger or the same size as the PCB.

10.3.2 Construction of the interface between PCB and heat sink

- a. Heat sink shall be made of copper or aluminium core bonded to the outer layer of PCB.

NOTE Bonding can be done by no-flow prepreg, cover layer or adhesive.

- b. The copper or aluminium core shall have a protective layer to prevent corrosion.

NOTE The type of protective layer is specified in the PCB definition dossier. Examples of protective layers to prevent corrosion are anodization, alodine, nickel plating, gold plating. These corrosion protections can have specific drawbacks for further PCB processing.

- c. The heat sink shall be manufactured by milling or etching.

NOTE Milling is preferred because it can provide better edge definition. Etching of thick layers result in a wider edge profile.

- d. The PCB manufacturer shall inform the supplier of the method of bonding the heat sink.

NOTE This is done to verify the thermal properties of the assembly.

- e. Minimum two non-plated through-holes shall be diagonally placed on the PCB for registration of the heat sink.

- f. No conductive surface pattern shall be present beneath the heat sink, except the case specified in the requirement 10.3.2g.

- g. A conductive surface pattern may be present beneath the heat sink in case:

1. the adhesion of the heat sink to the external layer is ensured,
2. the insulation of tracks of different potential is ensured,
3. the surface area of tin-lead under the heat sink is minimised, and
4. it is recorded as a Review Item in the PCB definition dossier.

NOTE 1 For conditions specified in the requirements 10.3.2g.1 and 10.3.2g.3 to ensure adhesion on conductive surface pattern and tin-lead, it is an option to bond a cover layer on tin-lead or to perform selective stripping of tin-lead and bond with prepreg.

NOTE 2 For condition specified in the requirement 10.3.2g.2 to ensure insulation, it is an option to bond a cover layer on conductive surface pattern.

NOTE 3 The behaviour of tin-lead on a PTH pad during assembly can affect the design of the bonding method of the heat sink.

10.3.3 Dimensional requirements

- a. The mechanical stability of individual clearances of the pattern of the heat sink shall be recorded as a Review Item in the PCB definition dossier.

- b. Distance between 2 clearances shall be $\geq 1,5$ mm along straight edges.

NOTE Distance along straight edges is indicated as "Iz1" in Figure 10-1.

- c. The length of the straight edges shall be reviewed for mechanical stability in conformance with requirement 10.2a.

- d. Distance between 2 clearances shall be $\geq 1,0$ mm between circular edges.

NOTE Distance between circular edges is indicated as "Iz2" in Figure 10-1.

- e. Distance between pads on the PCB and the edge of the clearance of the heat sink shall be $\geq 0,3$ mm.

NOTE Distance between pads on the PCB and the edge of the clearance of the heat sink is indicated as "i" in Figure 10-1.

- f. Distance between a clearance in the heat sink and the edge of the PCB laminate shall be $\geq 0,5$ mm.

NOTE Distance between a clearance in the heat sink and the edge of the PCB laminate is indicated as "b6" in Figure 10-1.

- g. Distance between the clearance in the heat sink and the edge of the heat sink shall be $\geq 1,5$ mm.

NOTE Distance between the clearance in the heat sink and the edge of the heat sink is indicated as "b7" in Figure 10-1.

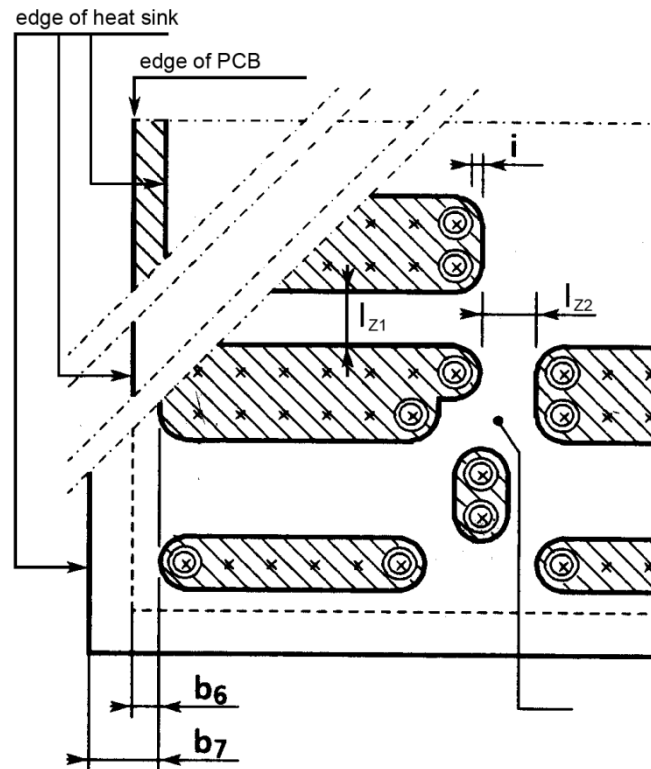


Figure 10-1: Lay out of heat sink and PCB

10.4 Specific requirements for internal heat sink

10.4.1 General

- a. Heat sink shall be made of copper, copper-molybdenum-copper or CIC.

NOTE 1 Heat sinks can be foils covering the entire surface of a layer or inserts placed locally under a specific component.

NOTE 2 Heat sinks that consist of a copper layer can be made by using copper foil or copper clad laminate. The advantage of a foil is the ability of prepreg to fill clearances from both sides. The advantages of copper clad laminate is better registration, ease in handling and processing and the possibility of more complex electrical routing in case of split ground planes on heat sinks.

- b. There should be no tracks on the same layer as a heat sink.

NOTE Local heat sink inserts can have tracks on the same layer.

10.4.2 Cu thickness and type

- a. Foils and laminates with as-designed copper thickness $\geq 105 \mu\text{m}$ shall be designed in conformance with the requirements from clause 10 for heat sinks.
- b. The number of copper heat sink layers shall be ≤ 2 .
- c. The thickness of copper layer shall be $105 \mu\text{m}$ to $210 \mu\text{m}$ in case 2 layers of heat sinks are used.
- d. The thickness of copper heat sink shall be $\leq 700 \mu\text{m}$ when 1 layer of heat sink is used.
- e. The copper foil shall be procured in conformance with the requirements from the IPC-4562.

NOTE Types of copper foil types commonly used are: HTE grade 3, and JTC grade 1.

- f. Copper foil for heat sinks shall be in conformance with the requirement 7.1.2i.

NOTE High quality copper foil type HTE or JTC is necessary to ensure reliable electrical connection with vias.

10.4.3 CIC and Molybdenum inserts

- a. CIC may be used to reduce in plane CTE.
 - NOTE 1 Material designation IPC-CF-152/2 CIC W 6D as specified in IPC-CF-152B.
 - NOTE 2 Most common composition of CIC is 12,5% copper, 75% invar, 12,5% copper.
- b. The thickness of CIC shall be $\leq 150 \mu\text{m}$.
- c. The number of heat sink layers with CIC shall be ≤ 2 .
- d. The CIC layers shall have a tolerance of $\leq \pm 10\%$.
- e. Molybdenum may be used to reduce in plane CTE and improve heat transfer.

NOTE Standard specification for molybdenum is according to ASTM-B386-03.

- f. The thickness of molybdenum inserts shall be $0,5 \text{ mm}$.
- g. The number of layers containing molybdenum inserts shall be ≤ 1 .
- h. The molybdenum inserts shall have a tolerance of $\leq \pm 0,035 \text{ mm}$.

10.4.4 Dimensional requirements

- a. The mechanical stability of individual clearances of the pattern of the heat sink shall be recorded as a Review Item in the PCB definition dossier.

NOTE Clearances that are separated by a long straight edge known as 'thin web' are a concern for mechanical stability.

- b. For heat sinks that are designed to be within the edge of the PCB, the distance between the edge of the heat sink and the edge of the PCB shall be $\geq 1,5$ mm.

NOTE Sufficient resin from the edge of the heat sink to the edge of the PCB ensures adequate insulation and mechanical stiffness needed for subsequent milling of the PCB.

- c. Heat sinks that are aligned with the edge of the PCB or exposed on one side by depth controlled milling shall have a protective coating for corrosion protection.

NOTE Example of protective coating is nickel plating.

- d. The distance between a plated hole and the heat sink shall be in conformance with the requirement 13.8.2k and values specified in the Table 13-3.

- e. The distance between two circular clearances shall be $\geq 2x$ the copper thickness of the heat sink.

- f. In case the distance between the two clearances is less than two times the copper thickness, the remaining metal shall be removed.

NOTE This remaining metal is named "thin web". This is illustrated in Figure 10-2.

- g. In case holes are overlapping, the angle of intersection, as specified in Figure 10-3, shall be minimum 110° .

NOTE This applies to etching as well as drilling and milling.

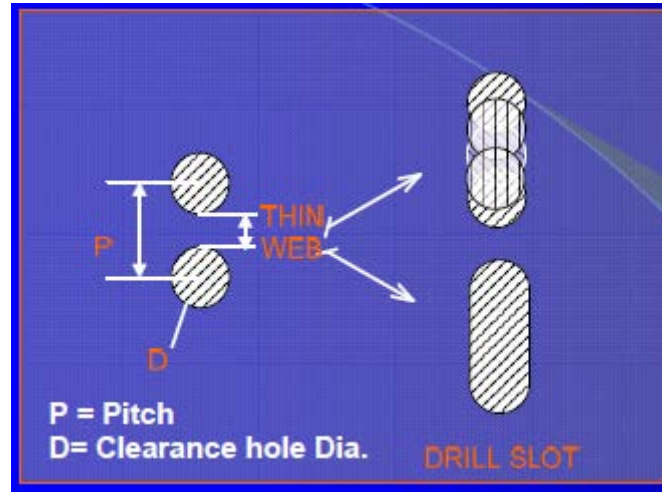


Figure 10-2: drilling holes and slots in internal heat sink

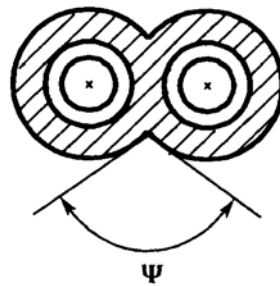


Figure 10-3: Angle of intersection of overlapping holes

11

HDI PCBs

11.1 Overview

This clause defines additional technology for HDI PCBs, such as microvias. In addition, this clause specifies deviations for the following features:

- a. For dielectric thickness of microvia layers on HDI PCBs, a deviation to requirement 7.1.3e is specified in requirement 11.4.1h.
- b. For copper foil thickness on microvia layers on HDI PCBs, a deviation to requirement 7.1.2c is specified in requirement 11.4.1e.
- c. For fine pitch track width and spacing of impedance controlled routing to 1 mm pitch AAD on HDI PCBs, a deviation to requirement 7.4.5a is specified in requirement 11.5.5a.
- d. For annular ring of internal pads for fine pitch footprint on HDI PCBs a deviation to requirement 7.5.2a is specified in requirement 11.5.2a.
- e. For aspect ratio of 1 mm pitch AAD on HDI PCBs, a deviation to requirement 7.3.5a is specified in requirement 11.5.6a.

11.2 Justification

- a. A PCB may be classified as HDI PCB during the design review in case the routing cannot be performed using conventional dimensions due to the assembly of high I/O components.
- b. An HDI PCB shall be designed in conformance with the requirements of the clause 11.
- c. All design features of the clause 11 shall be recorded as a Review Item in the PCB definition dossier.

11.3 Microvia technology

- a. For the design of an HDI PCB the following technologies shall apply:
1. without microvias,
 2. with microvia staggered with buried via without cap,
 3. with microvia stacked on cap plated filled buried via.

NOTE 1 HDI technology without microvias as specified in requirement 11.3a.1 is typically of lower complexity and of higher reliability compared to HDI technologies with microvias. HDI technology with staggered microvias as specified in requirement 11.3a.2 is of lower complexity compared to stacked microvias as specified in requirement 11.3a.3. It is recommended to design using less complex technology.

NOTE 2 Examples of stacked and staggered microvias in the build-up of an HDI PCB are shown in Figure 11-1. The offset achieved by staggering is indicated as D.

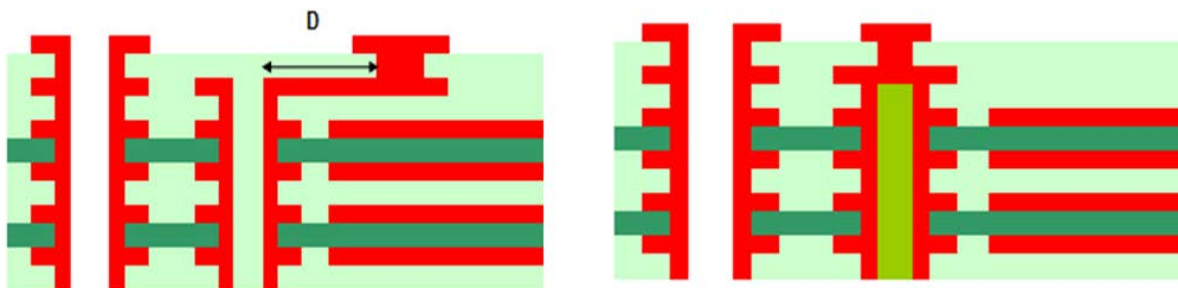


Figure 11-1: Build-up of HDI PCBs with staggered (left) and stacked (right) microvia.

11.4 Microvias

11.4.1 Build-up of microvia layers

- a. In case microvias are used, they shall connect from an external layers of the rigid section of the PCB.
- b. In case microvias are used, the number of microvia layers on either side of the core PCB shall be 1.
- c. Dummy copper plane filling should be used on internal microvia layers.

NOTE Dummy copper is used to establish a lay-out that is as homogeneous as possible. This is done to reduce embedding of copper circuit into the resin and to maximise the dielectric thickness.

- d. On microvia layers, the external and internal copper layers should be designed using basic copper thickness 17 μm .
- e. On microvia layers, the external and internal copper layers may be designed using basic copper thickness of 9 or 12 μm , in case the following conditions are met:
 - 1. the peel strength is measured on the test coupon to be ≥ 12 N/cm,
 - 2. a carrier on the foil is used during manufacturing, and
 - 3. it is recorded as a Review Item in the PCB definition dossier.

NOTE 1 The copper thickness defined in requirement 11.4.1d is in conformance with the requirement 7.1.2c. The copper thickness defined in requirement 11.4.1e is a deviation from the requirement 7.1.2c.

NOTE 2 The benefit of reduced copper thickness is to improve etching accuracy of fine features. To ensure an insulation of 60 μm between layers it can be beneficial to use thin copper on the internal microvia layer to avoid planarization processes.

NOTE 3 The potential lower peel strength of thin copper foil is a risk factor for manufacture, which is mitigated by the coupon measurement.

NOTE 4 Thin copper foil is more fragile, therefore a rigid carrier is used during manufacture. When the core surface is non-planar, it is possible the rigid carrier does not conform well to the non-planar prepreg causing lack of adhesion or deformation of the copper foil. The following features can cause non-planarity of the core surface: rigid-flex constructions, high via density, high track density, high copper thickness, use of no-flow prepreg.

- f. Microvia layers should be designed with copper foil and prepreg.
- g. Insulation of microvia layers shall be performed by using 2 sheets of prepreg in conformance with the requirement 7.1.3a.
- h. The as-manufactured insulation in Z direction in microvia layers shall be ≥ 60 μm in conformance with Table 13-3.

NOTE This is a deviation from the requirement of 70 μm as specified by the requirement 9.6.c.6 of ECSS-Q-ST-70-10.

- i. The as-designed insulation in Z direction in microvia layers shall be ≤ 100 μm .

NOTE The typical configuration to achieve ≥ 60 μm and ≤ 100 μm insulation, is by using 2 sheets of prepreg type 106.

- j. For voltages >30V microvia layers shall not have superimposed conductors.

NOTE This is done to ensure that insulation in Z direction is in conformance with the Table 13-3.

11.4.2 Design of microvias

- a. The aspect ratio of a microvia on a HDI PCB shall be ≤ 1 .
- b. The microvia shall be laser drilled.
- c. The microvia shall be filled with copper.
- d. The dimple on the surface of the filled microvia shall be $\leq 30 \mu\text{m}$.

NOTE An illustration of a dimple on a microvia is given in Figure 11-2.

- e. The plated copper thickness on the corner of the microvia shall be $\geq 25 \mu\text{m}$.
- f. Microvia diameter should be $\geq 150 \mu\text{m}$ as-designed.

NOTE The tolerance is small therefore the diameter as-manufactured is approximately identical.

- g. The diameter of the area connecting to layer 2 shall be $\geq 100 \mu\text{m}$ as-manufactured.

NOTE This is less than $150 \mu\text{m}$ as-designed to allow for conical shape of microvia.

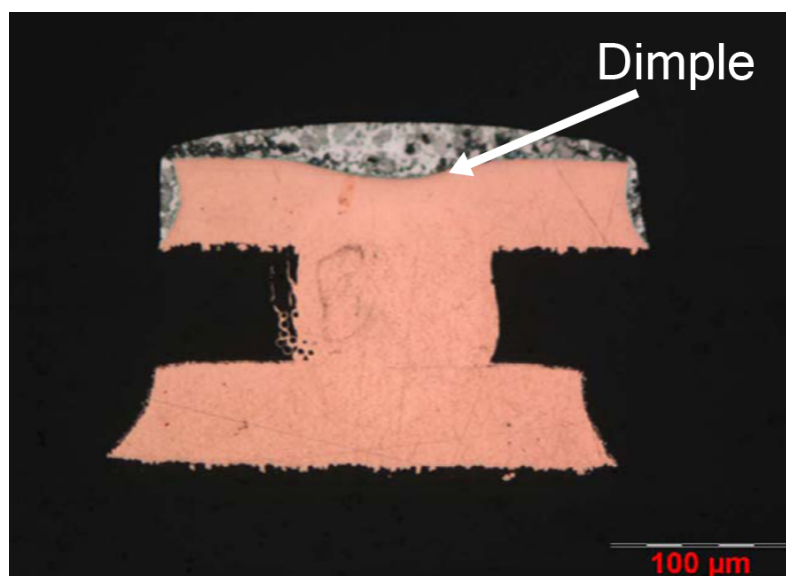


Figure 11-2: Dimple on a microvia

11.4.3 Pad design for microvia

- a. The diameter of external pads shall be the diameter of the ablated hole plus $\geq 0,15$ mm.
- b. The diameter of external pad may be the diameter of the ablated hole plus $\geq 0,1$ mm in case the following conditions are met:
 1. the annular ring is in conformance with the requirement 11.4.4b, and
 2. the diameter of external pad is recorded as a Review Item in the PCB definition dossier.
- c. The diameter of internal pads shall be the diameter of the ablated hole plus $\geq 0,15$ mm.

NOTE It is important to consider the complexity of the PCB and the PCB manufacturing tolerances for the design of the diameter of the pads.

- d. The internal pad of a staggered microvia and the pad of the buried via may be placed tangent to each other.

NOTE Assuming $100\ \mu\text{m}$ and $10\ \mu\text{m}$ for the annular ring as-manufactured of the pads of blind via and microvia, this results in a distance of about $110\ \mu\text{m}$.

11.4.4 Annular ring for microvias

- a. The annular ring as-manufactured for microvias for internal pads shall be $\geq 10\ \mu\text{m}$.
- b. The annular ring as-manufactured for microvias on external layer shall be $\geq 10\ \mu\text{m}$

NOTE The visual inspection on external layer possible on all microvias can use as criteria no breakout, i.e. tangency. This is done because it is not possible to visually inspect for this annular ring requirement.

- c. Evaluation of annular ring of microvias by microsectioning shall be in conformance with the requirement 11.4.4b.

11.5 Core PCB for HDI

11.5.1 General build-up

- a. For HDI PCB, the core PCB shall be type rigid or rigid-flex.
- b. The rigid part of an HDI PCB shall be designed in conformance with the requirements from the clause 7.
- c. The total number of copper layers in the rigid part shall be in conformance with the requirements from clause 7.3.4.

NOTE The total number of copper layers include the microvia layers.

- d. PCB thickness shall be in conformance with the requirements from the clause 7.3.
- e. Copper thickness of internal layers that do not include microvias shall be in conformance with the requirement 7.1.2a.

11.5.2 Annular ring on vias for fine pitch footprint

- a. The diameter of internal pads may be reduced in case the following conditions are met:
 1. a teardrop reinforcement is implemented,
 2. the annular ring as-manufactured in the direction of the track is in conformance with the requirement 7.5.2a,
 3. the annular ring as-manufactured in the directions that are opposite and perpendicular to the track are $> 25 \mu\text{m}$,
 4. length of wicking and drilling cracks is $\leq 25 \mu\text{m}$, and
 5. it is recorded as a Review Item in the PCB definition dossier.

NOTE 1 Requirement 7.5.2a specifies annular ring $>50 \mu\text{m}$.

NOTE 2 Requirement 11.5.2a.3 is a deviation of the requirement for annular ring 7.5.2a.

NOTE 3 In the requirement 11.5.2a.4 length of wicking and drilling cracks is the same dimension as the annular ring. This permits less wicking and drilling cracks than the requirement 7.3.4.3.3.e.1 of ECSS-Q-ST-70-10.

NOTE 4 An example of teardrop pad design is shown in Figure 11-3. In this figure "A" is the minimum annular ring between hole and circular pad design; "B" is the effective annular ring with teardrop reinforcement applied.

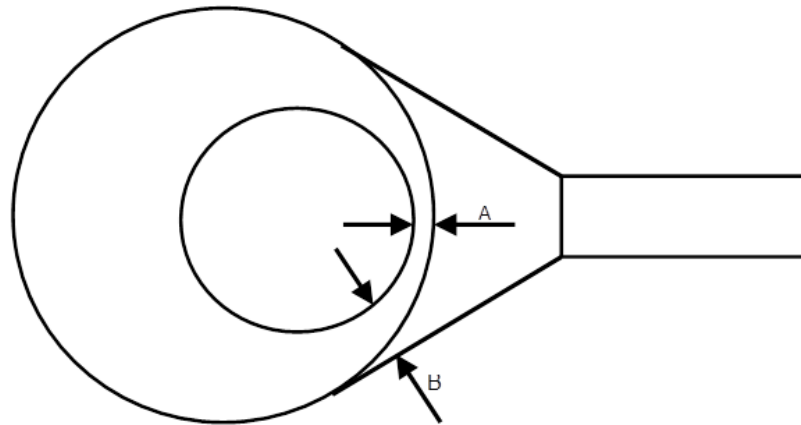


Figure 11-3: Tear drop pad design

11.5.3 Track width and spacing on external layers

- a. External layers should not include tracks.

NOTE This is in particular important for critical tracks.

- b. The spacing and width of pads and tracks for external layers may be reduced to requirements for fine pitch in conformance with the values specified in Table 7-4.
- c. The length of fine pitch on external layers shall be in conformance with the requirement 7.4.3g.

11.5.4 Track width and spacing on internal layers for impedance control and routing to AAD

- a. The spacing and width of pads and tracks for impedance control and routing to AAD on internal layers may be reduced to requirements for fine pitch in conformance with the values from Table 7-5.
- b. The requirements for fine pitch of clause 7.4.5 shall apply.

NOTE An example of routing a differential pair in a 1,27 mm pitch footprint is given in Figure 11-4. Ground planes on other layers are omitted for clarity. The following dimensions are considered:

- The available pad diameter is 710 μm .
- Allowance for design router is 10 μm .
- The maximum pad diameter is 700 μm .
- The minimum annular ring as-designed is 150 μm .
- Pad design includes teardrop and minimum annular ring as-manufactured as defined in requirement 11.5.2a.

- The maximum available drill diameter is 400 μm .
- The maximum aspect ratio is 7.
- The maximum board thickness for the drill sequence is 2,8 mm.

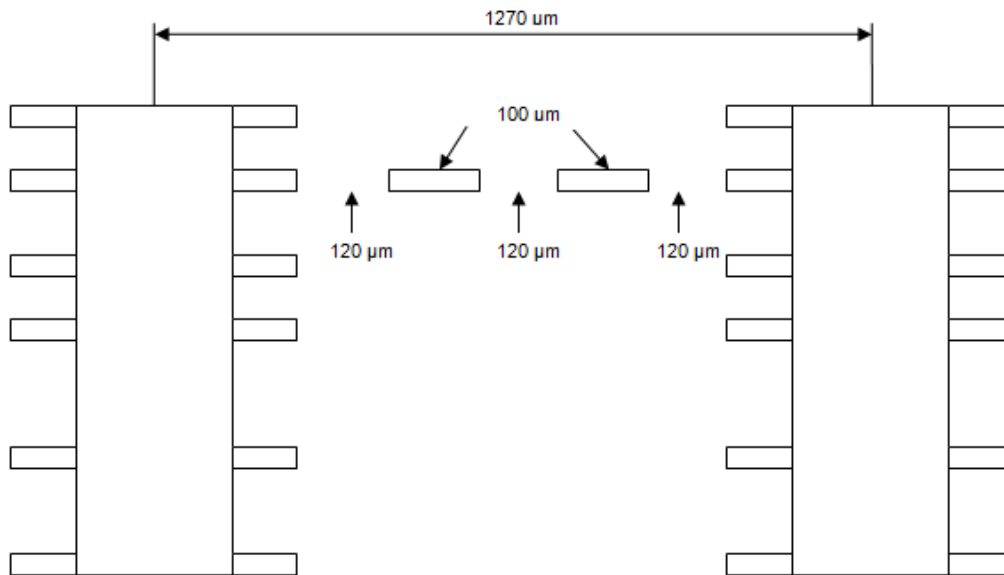


Figure 11-4: Example of routing two differential pair tracks between vias in a footprint of an AAD with 1,27 mm pitch.

11.5.5 Track width and spacing on internal layers for differential pair routing within the footprint of 1,0 mm pitch AAD

- On internal layers the width of tracks may be reduced to 80 μm and the spacing reduced to 90 μm , in case:
 - the tracks route an edge-coupled differential pair between via pads for a 1 mm pitch AAD,
 - the current is ≤ 50 mA and voltage is ≤ 10 V,
 - the spacing and track width are increased outside the footprint of the AAD to normal values in conformance with the requirements from clause 7.4.6,
 - the copper thickness is in conformance with the requirement 7.4.5g,
 - the manufacturing tolerance for 90 μm spacing is within -10% / +20%,
 - the manufacture of the tracks is inspected prior to lamination for conformance with designed track width and spacing within the required tolerance, and

7. it is recorded as a Review Item in the PCB definition dossier.

NOTE 1 NOTE to 11.5.5a.5: This reduces the tolerances from the standard $\pm 20\%$ as specified in the Table 7-3. This provides a minimum as-manufactured insulation distance of $81\ \mu\text{m}$ as specified in the Table 13-3.

NOTE 2 NOTE to 11.5.5a.6: The inspection can be performed by AOI and by dimensional measurement under microscope on the foot of the track for spacing and track width.

NOTE 3 The reduced feature sizes increase the risk of errors during etching resulting in open or short circuits.

11.5.6 Aspect ratio of vias for footprint of AAD with 1 mm pitch

- a. The aspect ratio of vias may be higher than 7 as specified in the requirement 7.3.5a in case:

1. the aspect ratio is ≤ 10 ,
2. the vias are used within the footprint of an AAD with a pitch of 1mm,
3. the plating inside vias with high aspect ratio is verified on the coupon as specified in the requirement 15.2d.6, and
4. it is recorded as a Review Item in the PCB definition dossier.

NOTE 1 It is more difficult to achieve homogeneous plating and the required thickness inside vias with a high aspect ratio.

NOTE 2 An example of routing a differential pair in a 1 mm pitch footprint is given in Figure 11-5. Ground planes on other layers are omitted for clarity. The following dimensions are considered:

- The available pad diameter is $570\ \mu\text{m}$.
- Allowance for design router is $10\ \mu\text{m}$.
- The maximum pad diameter is $560\ \mu\text{m}$.
- The minimum annular ring as-designed is $150\ \mu\text{m}$.
- Pad design includes teardrop and minimum annular ring as-manufactured is as defined in requirement 11.5.2a.
- The maximum available drill diameter is $250\ \mu\text{m}$.
- The maximum aspect ratio is 10.
- The maximum board thickness for the drill sequence is 2,5 mm.

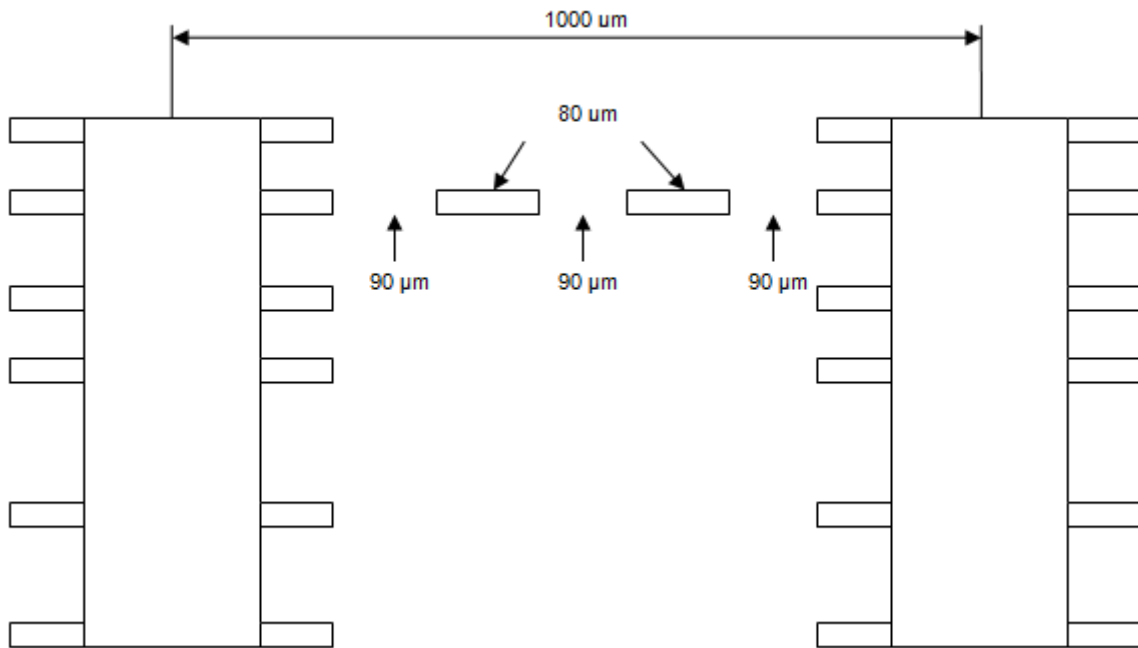


Figure 11-5: Example of routing two differential pair tracks between vias in a footprint of an AAD with 1 mm pitch.

12

PCBs for high frequency applications

12.1 Material selection

Table 6-1 presents examples of some materials used for manufacture of RF PCBs. For the selection of materials it is important to consider that materials with good RF performance typically have poorer processing characteristics (e.g. adhesion and resin flow during lamination). High performance RF materials can exhibit lower peel strength of copper. This can affect the assembly. Some RF materials based on PTFE or mixed build-ups can exhibit smear in the drilling process. For material selection it is important to consider the RF properties and the processability of the material. The possibility to mix for instance RF laminate with standard prepreg can provide good processability while maintaining acceptable RF performance.

12.2 Build-up of RF PCB

- a. The build-up of RF PCBs shall be in conformance with requirements from clause 7.1, except the cases specified in the requirements from 12.2b to 12.2e.
- b. For copper cladding and copper foils the requirements 7.1.2h, 7.1.2i shall not apply.
- c. Copper type “electrodeposited - ED” or “rolled and annealed - RA” may be used for copper cladding and copper foils.
- d. For Molybdenum and CIC layers the requirement 7.1.1c shall not apply.
- e. Molybdenum and CIC layers shall not be used in RF PCBs.

12.3 Embedded film resistors

- a. The use of embedded film resistors in internal layers shall be recorded as a Review Item in the PCB definition dossier.
- b. The tolerance on the as-manufactured resistance value shall be $\leq \pm 20\%$.

NOTE Due to the relatively high tolerance, the application of embedded film resistors is limited to line termination.

- c. Embedded film resistors shall be of type NiP or NiCr.
- NOTE Type NiP is available from OhmegaPly. Type NiCr is available from Ticer.
- d. A surface resistance of 25 Ω /sq should be used.
- e. A surface resistance of 50 Ω /sq may be used.
- NOTE A surface resistance of 25 Ω /sq is achieved on a thicker resistive layer which is beneficial for thermal reliability.
- f. Layers that include embedded film resistors should not be plated.
- NOTE This is done to achieve the best accuracy of etching and resistor value as-manufactured.
- g. No more than one plating sequence shall be performed on layers that include embedded film resistors.
- h. All embedded film resistors on a layer should be oriented in a single direction.
- i. The orientation of embedded film resistors shall be recorded as a Review Item in the PCB definition dossier.
- NOTE The best etching accuracy is obtained when embedded film resistors are oriented in the direction of the conveyor of the etching process.
- j. The width of an embedded film resistor shall be $\geq 250 \mu\text{m}$.
- k. The aspect ratio of length and width of an embedded film resistor shall be ≥ 1 .
- NOTE A higher aspect ratio achieves better accuracy of the resistive value.
- l. The ratio of length to width of an embedded film resistor shall be ≤ 5 .
- NOTE 1 A lower ratio achieves better thermal endurance.
- NOTE 2 The configurations specified in requirements 12.3d, 12.3e, 12.3k and 12.3l provide embedded film resistor values between 25 Ω and 250 Ω .
- m. Dedicated representative coupons shall be designed to allow verification of the embedded film resistor values on the etched layer before further lamination.
- n. Dedicated representative coupons shall be designed to allow verification of the embedded film resistor values on the as-manufactured PCB as specified in the requirement 15.2d.13.

12.4 Thickness of RF PCB

- a. The thickness of RF PCB as-designed should be ≤ 3 mm.
- b. The thickness of RF PCB as-designed may be > 3 mm in case the following conditions are met:
 1. the thickness as-designed is $\leq 5,5$ mm,
 2. the materials used are not based on PTFE, and
 3. it is recorded as a Review Item in the PCB definition dossier.

NOTE High PCB thickness involve a more difficult drilling process, lamination and handling in automated equipment. It is important that the PCB manufacturer confirms the good processability of the materials.

- c. The number of layers for RF PCBs should be ≤ 8 .
- d. The number of layers for RF PCBs may be > 8 in case the following conditions are met:
 1. the number of layers is ≤ 14 , and
 2. it is recorded as a Review Item in the PCB definition dossier.

12.5 Track width and spacing

12.5.1 External layers

- a. The spacing and width for external layers of RF PCBs shall be in conformance with the values from Table 7-4.
- b. For RF elements the spacing may be reduced to the values in conformance with Table 12-1 in case:
 1. the voltage between nets is ≤ 1 V,
 2. no assembly is performed on the RF elements,
 3. no critical tracks are included in the RF element,
 4. a specific visual inspection and functional measurement of the RF element is performed, and.
 5. it is recorded as a Review Item in the PCB definition dossier.

Table 12-1: Minimum spacing and width as-designed for RF elements on external layers

Total copper thickness (base + plated) [μm]	$17,5 \leq \text{Th} \leq 50$	$\text{Th} > 50$
width [μm]	125	200
spacing [μm]	125	200

12.5.2 Internal Layers

- a. The spacing and width for internal layers of RF PCBs shall be in conformance with the values from Table 7-5.

12.6 Pad design

12.6.1 Pad dimensions

- a. Dimensions of the via hole pads shall be in conformance with the requirements from clause 7.5.2.

12.6.2 Non-functional pads

- a. Removal of non-functional pads shall be performed in conformance with the requirements from clause 7.5.1.

12.7 Surface finish

- a. The surface finish for RF PCBs shall be in conformance with the requirements from clause 7.8.1.

12.8 Profiled layers and vias

- a. Z-controlled backdrilling of vias may be used in case:
1. the Z-controlled backdrilling on vias is included on the coupon, and
 2. it is recorded as Review Item in the PCB definition dossier.

NOTE Backdrilling is done to achieve better insulation or to avoid antenna effect of the via wall.

- b. The use of profiled top layers shall be recorded as a Review Item in the PCB definition dossier.

NOTE Profiled layers can be manufactured by Z-controlled milling or by laminating layers with openings. Profiled top layers can be used to expose a signal circuit on layer 2 in case the top layer is used as ground plane.

- c. The use of profiled aluminium backing used for cooling shall be recorded as a Review Item in the PCB definition dossier.

13

Electrical requirements for PCB design

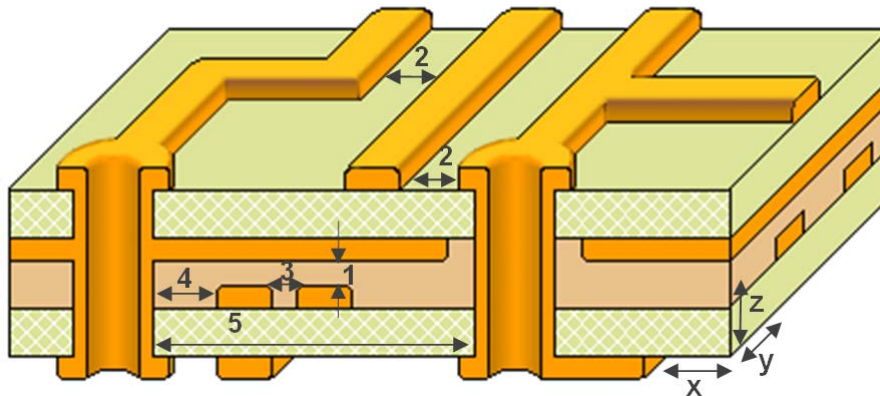
13.1 Overview

Table 13-1 explains the terms “AND”, “OR” and “FOR” as well as the used cross referencing to requirements or notes in Table 13-3, Table 13-4, Table 13-5, Table 13-6 and Table 13-7.

Table 13-1: Legend of terms

Terms	Meaning
AND	<p>The term “AND” indicates that the largest value of the two are used for the relevant voltage.</p> <p>For example: At 50 V in Table 13-3 the X,Y internal spacing is ($50 \times 1 \mu\text{m}/\text{V} = 50 \mu\text{m}$ AND $150 \mu\text{m}$). In this case $150 \mu\text{m}$ is the largest value to be used. At 200V the X,Y internal spacing is ($200 \times 1 \mu\text{m}/\text{V} = 200 \mu\text{m}$ AND $150 \mu\text{m}$). In this case $200 \mu\text{m}$ is the largest value to be used.</p>
OR	<p>The term “OR” indicates that various options are listed and any of the options can be selected.</p>
FOR	<p>The term “FOR” indicates the perimeter of a technology (for example HDI, fine pitch, uncoated connectors) for which an exemption has been specified to a requirement for standard insulation distance.</p>
Cross-reference to requirement	<p>Cross-referencing to requirements is used to trace back the short descriptions given in the tables to the normative references.</p>
Cross-reference to Note	<p>Cross-referencing to notes is used to trace back the short descriptions given in the tables to informative references.</p>

Figure 13-1 illustrates the insulation distances, numbered 1 to 5, specified in the Table 13-3, Table 13-4, Table 13-5, Table 13-6 and Table 13-7.



- 1: spacing in Z direction
- 2: spacing in X,Y direction on external layer
- 3: spacing in X,Y direction on internal layer between conductors
- 4: spacing in X,Y direction on internal layer between a conductor and a hole wall
- 5: spacing in X,Y direction on internal layer between hole walls

Figure 13-1: Cross section of PCB with insulation distances.

13.2 General

- a. The term “AND” indicates that the largest value of the two shall be used.

NOTE This is explained in Table 13-1.

13.3 PCB drying

- a. Precautions shall be taken to reduce the moisture level inside the PCB for electrical testing and for measuring electrical characteristics.

NOTE Recommended bake for bare PCB is 6 hours at 120°C. Polyimide is more hygroscopic than epoxy.

13.4 Electrical characteristics

- a. Insulation resistance between tracks shall be:
 1. intralayer: $> 10^{10} \Omega$,
 2. interlayer: $> 10^{11} \Omega$.
- b. DWV between tracks shall be 1000 Vrms/mm for intralayer and interlayer.
- c. Short-time current overload shall be in conformance with the requirements from the clause 7.3.5.1.2 of ECSS-Q-ST-70-10.
- d. Long-time current overload shall be in conformance with the requirements from the clause 7.3.5.1.3 of ECSS-Q-ST-70-10.

NOTE The values for current overload from 7.3.5.1.2 and 7.3.5.1.3 of ECSS-Q-ST-70-10 are summarised in the Table 13-2.

Table 13-2: Current overload limits

Basic copper thickness	Short-time current overload	Long-time current overload
9 μm	n/a	n/a
17 μm	n/a	n/a
35 μm	7 A for 4 s	≥ 8 A
70 μm	14 A for 4 s	≥ 16 A

- e. For power distribution a copper thickness of 35 μm or more should be used.

NOTE Thin copper tracks of below 35 μm are not suitable for power distribution. Therefore no requirement is specified for thin copper.

- f. In case sequential lamination increases the copper thickness, the current limits may be interpolated or extrapolated.

13.5 Floating metal

- a. All conductive parts and patterns shall not be left isolated, in conformance with clauses 6.3.1, 9.2.1 and 9.2.2 of ECSS-E-ST-20-06.

NOTE ECSS-E-ST-20-06 defines:

- the area of a floating surface is $\leq 1 \text{ cm}^2$
- the capacitance of the floating surface is $\leq 10 \text{ pF}$.

13.6 Current rating

13.6.1 Overview

Current rating in PCB tracks is based on the temperature increment of tracks compared to the initial PCB temperature when not powered. The method to compute the maximum allowed current is based on families of isothermal curves. These curves cover a range from 1°C up to 100°C and allow to compute the current starting from the copper cross section of the track or vice versa, for the required temperature increment.

The current rating model used in this standard is specified in the document IPC-2152 and the model is presented in graphical form. This standard is an evolution of and supersedes the previous model from IPC-2221A. Another model is presented in CNES/QFT/IN.0113, which is a more conservative model compared to IPC-2152. The mathematical fitting of the three models is presented as a guideline in Annex D. A comparison of the models is shown in Figure D-1.

13.6.2 Requirements for temperature increment

- a. The temperature increment of tracks with respect to the dielectric substrate shall be ≤ 10 °C.
- b. The temperature increment of tracks with respect to the dielectric substrate should be ≤ 5 °C.
- c. The temperature of tracks shall be ≤ 95 °C.
- d. The temperature of tracks should be ≤ 85 °C.

NOTE The increment of 10 °C is based on a temperature of the PCB substrate of 85 °C, in which case the tracks can heat to 95 °C due to self-heating. The self-heating of tracks up to 10°C compared to the substrate, regardless of overall substrate temperature, is based on heritage.

13.6.3 Requirements for the model IPC-2152 for current rating

- a. Except for the cases specified in the clause 13.6.4, the model used to calculate temperature increment shall be in conformance with the model for conductor sizing in space environment in conformance with requirements specified in chapters 1 to 5 and appendix A of IPC-2152.

NOTE The mathematical fitting of Figure 5-14 of chapter 5.2.2. of IPC-2152 and the other models is presented as a guideline in Annex D.

- b. Current rating shall apply to the conductive circuit of the as-manufactured PCB including manufacturing tolerances.

NOTE 1 Minimum as-manufactured track width and minimum as-manufactured copper thickness include tolerances specified in Table 7-1 and Table 7-3.

NOTE 2 The graphical representation of the model in IPC-2152 uses nominal dimensions for track width and copper thickness. Chapter A.4.3. of IPC-2152 specifies that the final cross sectional area can be significantly different from the nominal value and needs to be considered.

- c. The PCB configuration for which IPC-2152 is used to calculate current rating, shall be as follows:

1. PCB thickness is more than 1,5 mm.
2. PCB dielectric material has a thermal conductivity as for polyimide or epoxy.
3. PCB total surface area is more than 75x75 mm.

- d. In case the PCB configuration is not in conformance with requirement 13.6.3c, the supplier shall demonstrate the validity of the model used for the current rating.

NOTE In this case the model of IPC-2152 is not considered to be valid.

- e. The temperature rise of a track may be reduced when it is superimposed by a copper plane, in conformance with requirements specified in chapter A.4.6.2. of IPC-2152.

NOTE This chapter specifies that the presence of copper planes is recommended to be used to calculate temperature and to be considered as design margin.

- f. Current rating of a via shall use the same temperature rise as a track with equivalent cross section, in conformance with requirements specified in chapter A.3.4 of IPC-2152.

NOTE This chapter specifies that multiple vias can be used to reach the same cross sectional area of the connecting track.

- g. Multiple tracks in parallel with a spacing of less than 25 mm shall be assessed as parallel conductors, in conformance with requirements specified in chapter A.3.3 of IPC-2152.

NOTE Temperature rise of parallel conductors is determined by calculating the equivalent cross-sectional area and the equivalent current. A summary is also given in paragraph "e" of chapter 4 of IPC-2152.

- h. The current rating of polyimide and epoxy PCBs shall be in conformance with requirements specified in chapter A.4.4 of IPC-2152.

NOTE The chapter A.4.4 in IPC-2152 specifies that there is less than 2% difference.

13.6.4 Amendments to the model from IPC-2152

13.6.4.1 Overview

The requirements of the clause 13.6.4.2 specify amendments with respect to the model for conductor sizing in space environment defined in the clause 13.6.3. These amendments are specified for the current rating and for the temperature rise in the specified configurations.

13.6.4.2 List of amendments

- a. The current rating of external conductors shall be reduced by 10%.

NOTE This is an amendment to chapter A.6 of IPC-2152, which specifies that the current rating of external layers is the same as for internal layers. The justification for the 10% margin is specified in chapter 4.5 of CNES/QFT/IN.0113.

- b. For PCBs with a thickness between 1,5 mm and 1,8 mm, the temperature rise of tracks shall be increased by 20%.

NOTE The mathematical fitting of Figure 5-14 of chapter 5.2.2. of IPC-2152 presented as a guideline in Annex D is valid for PCBs of 1,8 mm. The justification for 20% margin for thinner PCBs is given in chapter A.4.2 of IPC-2152.

- c. Current rating of a via connecting to a copper plane may be 5 times higher.

NOTE This amendment to IPC-2152 is based on chapter 5.3 of CNES/QFT/IN.0113.

- d. Current rating of a track may be 5 times higher in case it is less than 2mm long and connecting a via to a copper plane.

NOTE This amendment to IPC-2152 is based on chapter 5.3 of CNES/QFT/IN.0113.

- e. In case a track leads to a PTH of a connector footprint and the track is reduced in width to pass between pads within the footprint, the current rating may be 2 times higher.

NOTE This is specified to avoid reducing annular ring to allow a wider track to pass in between pads. The increased current rating is justified by the presence of PTHs with copper pads used for assembly of connectors.

13.7 Provisions to prevent open circuit failure on critical tracks

13.7.1 Overview

The open circuit failure of a track in a PCB can lead to loss of function, a random behaviour, or worse, to the propagation of failure. As example, the open circuit failure of a connection in the DC/DC converter feedback loop can cause the loss of the DC/DC function and the failure propagation to the supplied downstream circuits.

13.7.2 Routing

- a. Criticality of tracks shall be as specified in requirement 13.9.2a.
- b. Critical tracks should be routed on a single layer.
- c. A verification shall be performed to reduce the use of vias on critical tracks.

NOTE Some automated routing software can cause unnecessary use of vias.

- d. For a critical track a second redundant via shall be used for the interconnection.

NOTE This is done to prevent open circuit failure of a single interconnection between track, via pad and via barrel.

13.8 Voltage rating

13.8.1 Overview

The insulation distance is determined by the following parameters:

- Voltage rating, as specified in requirements of clause 13.8 for single insulation and in clause 13.9 for double insulation.
- Manufacturing tolerances associated with the copper thickness and track width, as specified in requirements of clause 7.4.1.
- Manufacturing capability as function of copper thickness, as specified in requirements of clause 7.4.3 for external layers and requirements of clause 7.4.4 for internal layers.
- Exemptions made to accommodate HDI designs, as specified in requirements 11.4.1h, 11.4.1i, 11.4.1j, 11.5.2a, 11.5.3b, 11.5.4a, 11.5.5a and 11.5.6a.
- Exemptions made to accommodate the use of uncoated connectors, as specified in requirements of clause 13.8.4.

- Tolerances on laminate or prepreg, as specified in requirements of clause 7.1.3.
- Tolerances on hole wall to hole wall registration depending on number of bond sequences, as specified in requirement 13.8.2i and its Note.

All parameters listed are included in Table 13-7.

13.8.2 General requirements

- a. Voltage rating shall apply to the as-manufactured PCB.
- b. Voltage rating shall apply to the worst-case peak transient voltage.
- c. The supplier shall design the PCB with the insulation distances as specified in the requirements from the clause 13.8.
- d. The insulation distance as function of voltage of the as-manufactured PCB shall be in compliance with values specified in Table 13-3 and Table 13-4.

NOTE 1 The insulation distances also depends on manufacturing capability and tolerances as specified in clause 13.8.1 and specified in Table 7-4 and Table 7-5.

NOTE 2 The Table 13-3 and Table 13-4 cover any elevation. Withstanding voltage between two tracks in air at 87% relative humidity is 1600 V/mm. It is considered that breakdown or arcing cannot occur below this field strength as indicated in the paper by D. P Cullen and G O'Brien (see bibliography).

- e. Insulation distance in Z direction for superimposed conductors shall be in conformance with values specified in Table 13-3 and Table 13-4.

NOTE The requirements 7.1.3e and 10.3.1 also specify insulation distance of superimposed conductors, as explained in the clause 13.8.1.

- f. Conductors that are not superimposed in Z direction shall be offset in X,Y direction by the distance for X,Y internal conductor-to-conductor spacing as specified in Table 13-3 and Table 13-4.

- g. Insulation distance from conductors to plated or non-plated hole wall shall be in conformance with the values specified in the Table 13-3.

NOTE These insulation distances include an additional spacing of 50 μm to allow for wicking. The requirement does not need to be considered in addition to the requirement for X,Y internal conductor-to-conductor spacing in case the minimum annular ring as-manufactured is 50 μm and the wicking is $\leq 50 \mu\text{m}$.

- h. Insulation distance of conductors to PCB edge shall be as for X,Y internal conductor to hole wall, as specified in Table 13-3.

- i. The insulation distance between hole wall of vias on different nets shall be in conformance with the values specified in the Table 13-3.

NOTE The as-manufactured distance is not affected by the number of bond sequences. However, the tolerance on the as-designed distance increases with the number of bond sequences, as specified in the Table 13-7.

- j. The insulation distance between hole wall of vias on the same net may be less than the distances specified in the Table 13-3.

NOTE Vias on the same net can be used for redundancy. This can apply in particular for microvias.

- k. The insulation distance between the heat sink and a hole wall shall be in conformance with values from Table 13-3.

- l. Voltages above 500 V shall be subject to specific qualifications of the design.

Table 13-3: Minimum insulation distance as function of voltage on as-manufactured PCB for rigid laminate.

V	Z	X,Y external with conformal coating	X,Y external without conformal coating	X,Y internal conductor to conductor	X,Y internal conductor to hole wall	X,Y internal Hole wall to hole wall, Hole wall to heat sink
0-10V	70 μm FOR μvia layer 11.4.1h: 60 μm	120 μm	200 μm	104 μm FOR fine pitch 7.4.5a: 96 μm FOR 1 mm pitch HDI 11.5.5a: 81 μm	154 μm FOR HDI 11.5.4a: 121 μm	350 μm
11-30V			300 μm	104 μm FOR fine pitch 7.4.5a: 96 μm	FOR 1 mm pitch HDI 11.5.5a: 106 μm	
31-500V	1 $\mu\text{m}/\text{V}$ AND 100 μm	2 $\mu\text{m}/\text{V}$ AND 160 μm	5 $\mu\text{m}/\text{V}$ AND 500 μm	1 $\mu\text{m}/\text{V}$ AND 150 μm	(1 $\mu\text{m}/\text{V}$ +50 μm) AND 200 μm	2 $\mu\text{m}/\text{V}$ AND 350 μm

Note: The legend to this table is provided in Table 13-1.

13.8.3 Spacing on flex and rigid-flex laminate

- a. The spacing of conductors on flex laminate in the rigid section shall be in conformance with values for X,Y internal specified in the Table 13-3.

NOTE Conductors on flex laminate in rigid section are covered by prepreg.

- b. The spacing of conductors on flex laminate in the flex section shall be in conformance with values for X,Y internal specified in the Table 13-4.

NOTE Conductors on flex laminate in flex section are covered by cover layer.

- c. The spacing of conductors on flex laminate in the flex section for soldering a flex termination shall be in conformance with values for X,Y external specified in the Table 13-3.

NOTE 1 Conductors in flex section for soldering a flex termination are not covered by cover layer.

NOTE 2 Insulation distance in Z direction does not apply for sculptured flex, since it contains only a single layer of copper.

Table 13-4: Minimum insulation distance as function of voltage on as-manufactured PCB for flex laminate.

V	Z	X,Y internal	X,Y external
0-150 V	1 $\mu\text{m}/\text{V}$ AND 25 μm	1,5 $\mu\text{m}/\text{V}$ AND 150 μm	as specified in Table 13-3
Note: The legend to this table is provided in Table 13-1.			

13.8.4 Conformal coating

- a. Conductors on external layers should be covered with conformal coating.
- b. Fine pitch conductors on external layers shall be covered with conformal coating.
- c. Uncoated conductors on a PCB shall not be used for voltages > 30 V, except the case specified in the requirement 13.8.4d.

NOTE This is to avoid Paschen discharge when switched on during launch.

- d. Uncoated conductors may be used above 30 V for connector pads where coating or potting cannot penetrate locally between the connector pins.
- e. For the footprint of components and connectors that cannot be conformally coated, the insulation distance shall be in compliance with the requirements specified in Table 13-3 for X,Y external without conformal coating.

- f. The requirement 13.8.4e shall apply in case the conformal coating is used but cannot penetrate the footprint and cannot completely encapsulate the pads.
- g. Coating types that provide encapsulation of the PCB shall be included as conformal coating.

NOTE Higher voltage (> 30 V) PCBs designed without conformal coating are commonly encapsulated by other means, such as potting. Connectors can be potted or coated in similar way.

13.9 Double insulation design rules for critical tracks

13.9.1 Overview

The design of PCBs can have an impact on the reliability of a system. Any credible single failure into a PCB can lead to loss of a critical function of a system.

13.9.2 Critical nets

- a. Nets shall be identified as "critical" when they include the following functionalities:
 - 1. non-protected sections of a main bus power distribution system up to and including the first protection device,
 - 2. nets that are a single point failure SPF for the system,
 - 3. nets on which a loss of insulation can result in electrical failure propagation to a critical net,
 - 4. cross-strapped functions and associated common links from source to load.
 - NOTE 1 Details for requirement 13.9.2a.1 are specified in requirement 5.8.1.c. of the ECSS-E-ST-20.
 - NOTE 2 Example for the requirement 13.9.2a.3 is primary and redundant nets on a single PCB.
 - NOTE 3 Examples of cross-strapped functions from the requirement 13.9.2a.4 are hot redundant power links, majority voters "reliable" links, specific telecommand or telemetry matrices.
- b. Critical nets shall be subject to double insulation in conformance with requirements from the Clause 13.9.
- c. PA requirements of projects may specify requirements to implement double insulation on specific nets.
- d. FAI shall be performed in case double insulation applies.

13.9.3 Prevention of short circuit

13.9.3.1 Increase insulation in the PCB

- a. The insulation in Z direction between adjacent layers as specified in the requirement 13.8.2e and Table 13-3 shall be increased by one of the three methods:
 1. Method 1: Increase the insulation distance in Z direction by implementing the distance specified in Table 13-5 and Table 13-6.
 2. Method 2: Maintain insulation distance in Z direction as specified in Table 13-3 and Table 13-4 and use a combination of two different individually cured insulators as specified in the following options:
 - (a) Prepreg with laminate and copper on one side only,
 - (b) Prepreg with flex laminate and copper on one side only
 3. Method 3: Maintain insulation distance in Z direction as specified in Table 13-3 and Table 13-4 and have conductors on adjacent layers not superimposed and off-set by an insulation distance in X,Y direction as specified in requirement 13.9.3.1c.
- b. Method 3 of requirement 13.9.3.1a shall be used on microvia layers.
- c. The insulation distance in X,Y direction shall be increased in conformance with values specified in Table 13-5 and Table 13-6.

NOTE 1 Increasing insulation distance as specified in the requirement 13.9.3.1a.1 results in a lower field strength.

NOTE 2 Figure 13-2 and Figure 13-3 show examples of double insulation methods. The letters A, B, C, D, E, F, G, H, I, L indicate insulation distance.

NOTE 3 An example of insulation by prepreg and rigid laminate as specified in the requirement 13.9.3.1a.2(a) is shown in Figure 13-2 between copper tracks 2 and 5. The copper on the bottom side of the laminate (between track 4 and 6) has been etched away such that double insulation is achieved between track 2 and 5.

NOTE 4 An example of insulation of prepreg and flex laminate as specified in the requirement 13.9.3.1a.2(b) is shown in Figure 13-3 between copper tracks 2 and 5.

NOTE 5 An example of off-set conductors as specified in requirement 13.9.3.1a.3 is shown in Figure 13-2 between copper tracks 2 and 4.

NOTE 6 An example of insulation in X,Y direction as specified in the requirement 13.9.3.1c, is shown in Figure 13-2 between copper tracks 2 and 1.

NOTE 7 For insulation in Z direction as specified in requirement 13.9.3.1a, method number 3 is

considered more reliable for electrical insulation than method 2 and method 1. Method 2 is again more reliable than method 1.

NOTE 8 For insulation in Z direction as specified in requirement 13.9.3.1a, method number 2 is subject to qualification of the PCB manufacturing processes as specified in the PID.

NOTE 9 Fine pitch tracks are not compliant with double insulation because of the increased distance specified in the Table 13-5.

Table 13-5: Minimum double insulation distance as function of voltage on as-manufactured PCB for rigid laminate.

V	Z	X,Y external with conformal coating	X,Y external without conformal coating	X,Y internal conductor to conductor	X,Y internal conductor to hole wall	X,Y internal Hole wall to hole wall Hole wall to heat sink
0-10V	100 µm	240 µm	Not permitted	240 µm	290 µm	700 µm
11-30V						
31-500	1,5 µm/V	4 µm/V	Not permitted	2 µm/V	(2 µm/V+50)	4 µm/V
	AND	AND		AND	AND	AND
	130 µm	320 µm		300 µm	350 µm	700 µm

Note: The legend to this table is provided in Table 13-1.

Table 13-6: Minimum double insulation distance as function of voltage on as-manufactured PCB for flex laminate.

V	Z	X,Y internal	X,Y external
0-150 V	2 µm/V	3 µm/V	as specified in Table 13-5
	AND	AND	
	50 µm	300 µm	

Note: The legend to this table is provided in Table 13-1.

13.9.3.2 Increase insulation of PCB assembly

- a. Conformal coating shall be applied on external conductors in case of double insulation.
- b. The insulation of conductive elements on external layers for a PCB assembly for which double insulation applies, shall be increased as follows:
 1. The insulation distance of the surface pattern to adjacent surroundings is $\geq 2x$ the distance as specified in Table 14-1 up to a distance of 1 mm.
 2. The insulation distance of components to adjacent surroundings equals the recommended data as specified in Table 14-2.
 3. The insulation distance to component pads is $\geq 2x$ the distance specified in Table 14-3.

NOTE This is done to mitigate the risk of loss of insulation due to foreign particles, voids, or cracks in laminate or prepreg.

- c. The insulation distance from a conductor to the PCB edge in conformance with the requirement 13.8.2h shall be doubled in case the PCB edge faces an external conductive element within a distance of less than 1 mm.
- d. Flexible PCB sections that include critical tracks and that face other conductive elements with an insulation distance of < 1 mm shall include an additional insulation layer of ≥ 25 μm on the conductive element.

NOTE Additional insulation can be provided by 25 μm Kapton tape.

- e. Overhang of chip components shall not reduce the insulation distance below the values specified in the requirement 13.9.3.2b.3.

13.9.3.3 Routing

- a. Critical tracks as specified in the requirement 13.9.2a should not be routed on external layers.

NOTE This is also recommended for non-critical track in conformance with requirements from the clause 7.4.3a.

- b. In case critical tracks are routed on external layers, they shall not be routed under components.

NOTE This is done to allow inspection of critical tracks.

- c. A critical net shall be covered with conformal coating including the track, pads and component leads.

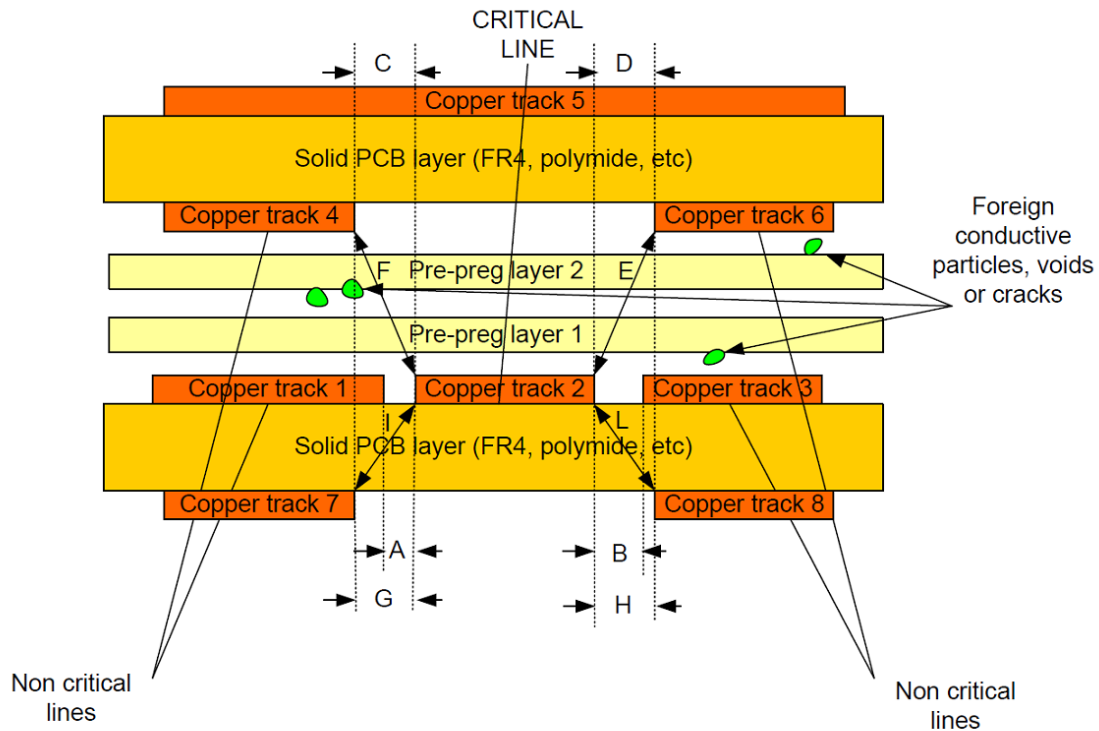


Figure 13-2: Example of double insulation by increasing distance in X,Y and by not superimposing copper on adjacent layers

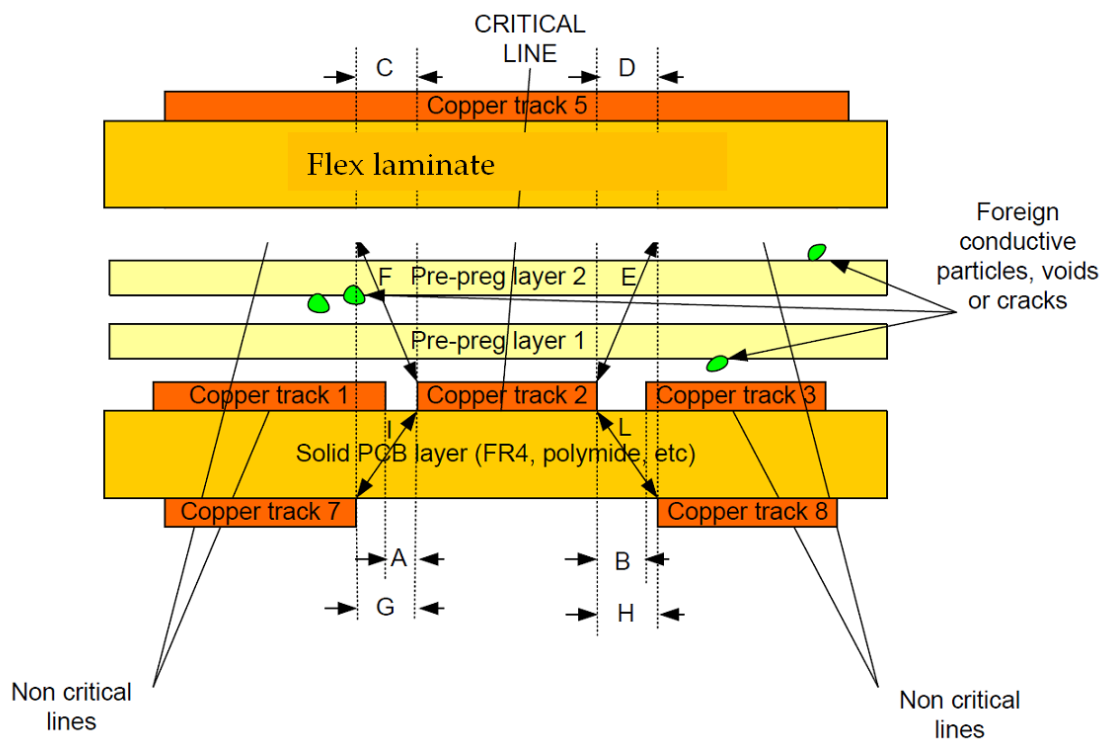


Figure 13-3: Example of double insulation by increasing distances in X,Y and by using two insulators in Z direction.

13.10 Insulation distance of combined requirements on rigid PCB

- a. As-manufactured insulation distance shall be in conformance with the values specified in Table 13-7.
- b. As-designed insulation distance should be in conformance with the values specified in Table 13-7.
- c. As-designed insulation distance may be less than the values specified in the Table 13-7 in case the following conditions are met:
 1. a dedicated inspection is performed to verify that the as-manufactured insulation distance is in compliance with the values from the Table 13-7, and
 2. the insulation distance is recorded as a Review Item in the PCB definition dossier.

NOTE 1 See the overview specified in clause 13.8.1 for a list of parameters that affect the insulation distance and that are included in the Table 13-7.

NOTE 2 For legend to Table 13-7 see the overview in clause 13.1.

Table 13-7: Minimum insulation distances on rigid PCB as function of all combined requirements (part 1 of 4)

Insulation →		Z laminate				Z prepreg	
		standard		Double 13.9		standard	Double 13.9
V (peak 13.8.2b)	Cu Th →	≤ 35 μm	70 μm	≤ 35 μm	70 μm	-	-
0<V≤10	As manufactured	70 μm		100 μm		70 μm FOR μvia layer 11.4.1h: 60 μm	100 μm FOR μvia layer offset 13.9.3.1b: 60 μm
	As-designed	≥ 4mil 7.1.3f	≥ 5mil 7.1.3g	5mil screened 7.1.3g OR 5mil class D 7.1.3j OR 6mil		-	-
10<V≤30	As-manufactured	70 μm		100 μm		70 μm FOR μvia layer 11.4.1h: 60 μm	100 μm FOR μvia layer offset 13.9.3.1b: 60 μm
	As-designed	≥ 4mil 7.1.3f	≥ 5mil 7.1.3g	5mil screened 7.1.3g OR 5mil class D 7.1.3j OR 6mil		-	-
30<V≤50	As-manufactured	100 μm		130 μm OR 100 μm with two insulators 13.9.3.1a.2.		100 μm FOR μvia layer offset 11.4.1j: 60 μm	130 μm OR 100 μm with two insulators 13.9.3.1a.2 FOR μvia layer offset 13.9.3.1b: 60 μm
	As-designed	5mil screened 7.1.3g OR 5mil class D 7.1.3j OR 6mil		See Table 7-2		-	-
50<V≤100	As-manufactured	100 μm		(1,5 μm/V AND 130 μm) OR 100 μm with two insulators 13.9.3.1a.2.		100 μm FOR μvia layer offset 11.4.1j: 60 μm	(1,5 μm/V AND 130 μm) OR 100 μm with two insulators 13.9.3.1a.2 FOR μvia layer offset 13.9.3.1b: 60 μm
	As-designed	5mil screened 7.1.3g OR 5mil class D 7.1.3j OR 6mil		See Table 7-2		-	-
100<V≤500	As-manufactured	1 μm/V		1,5 μm/V OR 1 μm/V with two insulators 13.9.3.1a.2.		1 μm/V FOR μvia layer offset 11.4.1j: 60 μm	1,5 μm/V OR 1 μm/V with two insulators 13.9.3.1a.2 FOR μvia layer offset 13.9.3.1b: 60 μm
	As-designed	See Table 7-2		See Table 7-2		-	-

Table 13-7: Minimum insulation distances on rigid PCB as function of all combined requirements (continued 2 of 4)

Insulation →		X,Y external without conformal coating				X,Y external with conformal coating					
		standard			Double 13.9	standard			Double 13.9		
max tolerance in X,Y 7.4.2b →		50 μm	50 μm	70 μm		50 μm	50 μm	70 μm	50 μm	50 μm	70 μm
V (peak 13.8.2b)	Cu Th →	fine pitch Th ≤ 70 μm	normal pitch Th ≤ 70 μm	normal pitch Th > 70 μm		fine pitch Th ≤ 70 μm	normal pitch Th ≤ 70 μm	normal pitch Th > 70 μm	fine pitch Th ≤ 70 μm	normal pitch Th ≤ 70 μm	normal pitch Th > 70 μm
0<V≤10	As-manufactured	not permitted 7.4.3g.3	200 μm	280 μm	not permitted 13.9.3.2a	120 μm	160 μm	240 μm	not permitted 13.9.3.1c and its NOTE 9	320 μm	480 μm
	As-designed		250 μm	350 μm		150 μm	200 μm	300 μm		370 μm	550 μm
10<V≤30	As-manufactured		300 μm	380 μm		120 μm	160 μm	240 μm		320 μm	480 μm
	As-designed		350 μm	450 μm		150 μm	200 μm	300 μm		370 μm	550 μm
30<V≤50	As-manufactured		not permitted, except FOR connector pads 13.8.4d: 500 μm			not permitted	160 μm	240 μm		320 μm	480 μm
	As-designed		FOR 13.8.4d: 550 μm	FOR 13.8.4d: 570 μm			200 μm	300 μm		370 μm	550 μm
50<V≤100	As-manufactured		not permitted, except FOR connector pads 13.8.4d: 500 μm			not permitted	2 μm/V AND 160 μm	240 μm		4 μm/V AND 320 μm	480 μm
	As-designed		FOR 13.8.4d: 550 μm	FOR 13.8.4d: 570 μm			2,5 μm/V AND 200 μm	300 μm		(4 μm/V + 50 μm) AND 370 μm	550 μm
100<V≤500	As-manufactured		not permitted, except FOR connector pads 13.8.4d: 5 μm/V			not permitted	2 μm/V	2 μm/V AND 240 μm		4 μm/V	4 μm/V AND 480 μm
	As-designed		FOR 13.8.4d: (5 μm/V + 50 μm)	FOR 13.8.4d: (5 μm/V + 70 μm)			(2 μm/V + 50 μm)	(2 μm/V + 70 μm) AND 300 μm		(4 μm/V + 50 μm)	(4 μm/V + 70 μm) AND 550 μm

Table 13-7: Minimum insulation distances on rigid PCB as function of all combined requirements (continued 3 of 4)

Insulation →		X,Y internal conductor to conductor									
		standard					Double 13.9				
max tolerance in X,Y 7.4.2b →		30 μm	30 μm	50 μm	50 μm	70 μm	30 μm	30 μm	50 μm	50 μm	70 μm
V (peak 13.8.2b)	Cu Th →	fine pitch Th ≤ 17 μm	normal pitch Th ≤ 17 μm	17<Th≤60 μm	60<Th≤70 μm	70<Th≤95 μm	fine pitch Th ≤ 17 μm	normal pitch Th ≤ 17 μm	17<Th≤60 μm	60<Th≤70 μm	70<Th≤95 μm
0<V≤10	As-manufactured	FOR fine pitch 7.4.5a: 96 μm FOR 1mm pitch HDI 11.5.5a: 81 μm	104 μm	120 μm	160 μm	240 μm	not permitted 13.9.3.1c and its NOTE 9	240 μm	240 μm	280 μm	360 μm
	As-designed	FOR fine pitch 7.4.5a: 120 μm FOR 1mm pitch HDI 11.5.5a: 90 μm	130 μm	150 μm	200 μm	300 μm		270 μm	290 μm	330 μm	430 μm
10<V≤30	As-manufactured	FOR fine pitch 7.4.5a: 96 μm	104 μm	120 μm	160 μm	240 μm		240 μm	240 μm	280 μm	360 μm
	As-designed	FOR fine pitch 7.4.5a: 120 μm	130 μm	150 μm	200 μm	300 μm		270 μm	290 μm	330 μm	430 μm
30<V≤50	As-manufactured	not permitted 7.4.5d	150 μm	150 μm	160 μm	240 μm		300 μm	300 μm	310 μm	390 μm
	As-designed		180 μm	188 μm	200 μm	300 μm		330 μm	350 μm	360 μm	460 μm
50<V≤100	As-manufactured		150 μm	150 μm	160 μm	240 μm		300 μm	300 μm	310 μm	390 μm
	As-designed		180 μm	188 μm	200 μm	300 μm		330 μm	350 μm	360 μm	460 μm
100<V≤500	As-manufactured		1 μm/V AND 150 μm	1 μm/V AND 150 μm	1 μm/V AND 160 μm	1 μm/V AND 240 μm		2 μm/V AND 300 μm	2 μm/V AND 300 μm	2 μm/V AND 310 μm	2 μm/V AND 390 μm
	As-designed		(1 μm/V +30μm) AND 180 μm	(1 μm/V +50μm) AND 188 μm	(1 μm/V +50μm) AND 200 μm	(1 μm/V +70μm) AND 290 μm		(2 μm/V +30μm) AND 330 μm	(2 μm/V +50μm) AND 350 μm	(2 μm/V +50μm) AND 360 μm	(2 μm/V +70μm) AND 460 μm

Table 13-7: Minimum insulation distances on rigid PCB as function of all combined requirements (continued 4 of 4)

Insulation →		X,Y internal conductor to hole wall		X,Y internal hole wall to hole wall			
		standard	Double 13.9	standard		Double 13.9	
max tolerance in X,Y 13.8.2iNOTE →		-	-	100 μm	300 μm	100 μm	300 μm
V (peak 13.8.2b)	bond sequence →	-	-	single bond sequence	sequential bonding and rigid flex	single bond sequence	sequential bonding and rigid flex
0<V≤10	As-manufactured	X,Y internal + annular ring of 50 μm 13.8.2g and its NOTE	FOR HDI 11.5.2a: X,Y internal + annular ring of 25 μm	350 μm	350 μm	700 μm	700 μm
	As-designed			450 μm	650 μm	800 μm	1000 μm
10<V≤30	As-manufactured			350 μm	350 μm	700 μm	700 μm
	As-designed			450 μm	650 μm	800 μm	1000 μm
30<V≤50	As-manufactured			350 μm	350 μm	700 μm	700 μm
	As-designed			450 μm	650 μm	800 μm	1000 μm
50<V≤100	As-manufactured			350 μm	350 μm	700 μm	700 μm
	As-designed			450 μm	650 μm	800 μm	1000 μm
100<V≤500	As-manufactured			2 μm/V AND 350 μm	2 μm/V AND 350 μm	4 μm/V AND 700 μm	4 μm/V AND 700 μm
	As-designed			(2 μm/V + 100 μm) AND 450 μm	(2 μm/V + 300 μm) AND 650 μm	(4 μm/V + 100 μm) AND 800 μm	(4 μm/V + 300 μm) AND 1000 μm

Note: The legend to this table is provided in Table 13-1.

13.11 Controlled impedance tracks

13.11.1 Definitions specific to controlled impedance

Microstrip is a track coupled to one copper plane. Stripline is a track between two copper planes. Stripline provides better shielding of the signals compared to microstrip, but it requires more plane layers.

Differential microstrip are two parallel tracks on the same layer and coupled to one copper plane. Differential striplines are two parallel tracks either edge coupled or broadside coupled and coupled to two copper planes.

For end to end configuration only a single driver and receiver is present at the extremes of a track. For multidrop configuration multiple drivers and receivers are possible along a track.

13.11.2 General rules

- a. Controlled impedance tracks shall be recorded as a Review Item in the PCB definition dossier.
- b. Controlled impedance tracks should be routed on a single layer.
- c. Vias should not be used in controlled impedance tracks except for the connection of component pads.
- d. The controlled impedance tracks should be designed with end-to-end configuration.
- e. The controlled impedance tracks may be designed with multidrop configuration.
- f. When using multidrop configuration the impedance calculation shall include the additional discrete loads.

13.11.3 Microstrip and stripline

- a. Microstrip or stripline shall be used for single ended controlled impedance tracks.
- b. Differential microstrip and differential stripline shall be used for differential controlled impedance tracks.

NOTE An example of edge coupled differential striplines is given in Figure 13-4. This figure shows the following characteristic dimensions used to calculate impedance:

- H: distance between planes as-manufactured
- H1: laminate thickness between tracks and plane.
- W: width of the top of the tracks
- W1: width of the foot of the tracks
- S: spacing between foot of tracks.

- c. Differential tracks shall be routed in parallel without change of the distance between the tracks except the case specified in the requirement 13.11.3d.
- d. Differential tracks may be routed non-parallel at the end of the track.

NOTE This is permitted to route to pads.

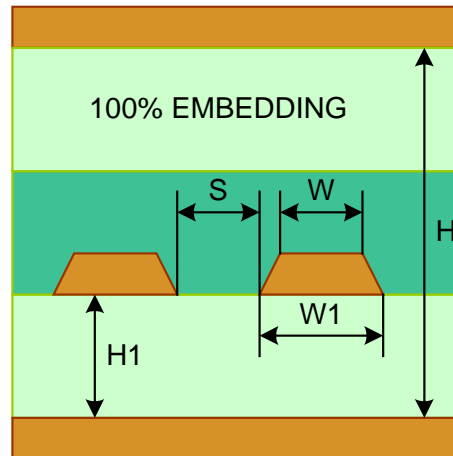


Figure 13-4: Edge coupled differential striplines

13.11.4 Line impedance termination for end-to-end configuration

- a. Series termination shall be used for end-to-end configuration.
- b. The resistor for series termination shall be placed at the driver output.
- c. AC parallel termination may be used in the end-to-end configuration at the input of the receiver.

13.11.5 Line impedance termination for multidrop configuration

- a. Parallel termination shall be used in multidrop configuration.
- b. DC or AC termination may be used in a multidrop configuration.
 - NOTE AC termination can be preferred to save power.
- c. In multidrop configuration with one driver and several receivers the driver shall be at one end of the line and the termination at the other end.
- d. In multidrop configuration with several drivers and several receivers the line termination shall be at both ends of the line.

13.12 Digital PCB

13.12.1 Overview

Digital circuitries are those in which the information on a line can assume only two values “0” or “1”. They are, in general, more robust to noise with respect to the analog ones. They can be critical during the transition of the threshold region if the input signal is not monotonic.

The criticality of digital signals is indicated as non-critical, critical and highly critical. The criteria for criticality are not the same as for critical nets subject to double insulation as specified in the clause 13.9.2.

13.12.2 Zone management and routing

- a. The layout of a PCB should be partitioned by using zone management where different logical blocks are physically separated from each other.

NOTE Different blocks to be separated in a board layout can be low speed (or static) links, analog links, power lines, high speed serial digital links, RF signals.

- b. The low speed circuit should be separated from the high speed digital circuit.
- c. High speed signals should not be routed through low speed zones.

NOTE This is done to prevent cross talk.

- d. Ground planes shall be used in high speed digital applications.
- e. Ground planes should be used in all digital applications.
- f. High speed signals shall be routed over uninterrupted planes

NOTE Examples of uninterrupted planes are Vcc or GND.

- g. The loop area of high speed signals should be minimised.

NOTE Loop area is minimised to reduce current noise. This can be achieved by routing the forwarded signal directly below or above a ground plane as return path. Further precautions can be found in the following references: chapter B.3.3 and B.3.4 of IPC-2251; chapter 6.4.6 of IPC-2221B.

- h. Circuitry and components for high speed signals should be kept close together.
- i. The length of routing in digital applications for high speed signals should be minimised, except when matched timing is used.

NOTE Reducing the length of signal paths is done, especially for high speed parallel links between components, to reduce crosstalk, voltage noise

and adaptation of line impedance. This also achieves a more compact PCB design. The technology determines how to implement minimisation. For example, rise and fall time are more critical than frequency. Tracks length lower than “critical length” defined in Table 5-4 of IPC-2251 is an example of minimisation. When “critical length” is exceeded, the tracks are considered as transmission lines. This is described in chapter 4.4.2.d of IPC-2251 and in IPC-2141A.

13.12.3 Criticality of digital signals

13.12.3.1 Non-critical signals

- a. Signals that are not sensitive to the coupling between them shall be identified as “non-critical”.

NOTE Examples of non-critical signals are: data bus, address bus.

- b. Non-critical signals specified in the requirement 13.12.3.1a may be routed together in bundles disregarding transients between them.

NOTE 1 Transients are temporary anomalies in the electrical signal, such as overshoot, undershoot, ringing, lines reflection, crosstalk.

NOTE 2 Transients are typically expired or reduced to an acceptable level before the sampling time.

- c. Direct coupling between bundles of non-critical signals specified in the requirement 13.12.3.1a should be avoided.

13.12.3.2 Critical signals

- a. Signals that require a monotonic waveform through the voltage threshold of the receivers shall be identified as “critical”.

NOTE Examples of critical signals are: clocks signals, write signals.

- b. The routing of critical signals specified in the requirement 13.12.3.2a should maintain signal integrity at the receiver end.

NOTE Signal integrity tools can be used to simulate the behaviour of the PCB after routing.

- c. The integrity of critical signals specified in the requirement 13.12.3.2a shall be verified by electrical testing of the PCB.

13.12.3.3 Highly critical signals

- a. Signals that can affect the performance of the circuit as a result of noise, crosstalk, jitter, ringing or propagation delay, shall be identified as “highly critical”.

NOTE Examples of highly critical signals are: PLL signals, low voltage and high frequency LVDS signals.

- b. Highly critical signals specified in the requirement 13.12.3.3a should be routed without adjacent signals or protected by shielding.

NOTE Separation of other highly critical signals is done to prevent crosstalk and noise pickup. Separation can be achieved by planes and rows of shielding vias.

- c. The integrity of highly critical signals specified in the requirement 13.12.3.3a shall be verified by electrical testing of the PCB.

13.13 Analog PCB

13.13.1 Overview

PCB for analog circuitries are used to interconnect heterogeneous discrete components such as resistors, capacitors, diodes, transistors, amplifiers to implement an analog function. Analog circuitries are more susceptible to noise compared to digital circuitries. The PCB design can affect the overall performance. Analog signals include all values within a range. The criticality depends on the resolution or on the bandwidth of the signal.

The criticality of analog signals is indicated as non-critical, critical and highly critical. The criteria for criticality are not the same as for critical nets subject to double insulation as specified in the clause 13.9.2.

13.13.2 Criticality of analog signals

13.13.2.1 Non-critical signals

- a. Signals that have low speed, low bandwidth or low resolution shall be identified as “non-critical”.

NOTE The signal-to-noise ratio, pickup and crosstalk noise (or any combination) are non-critical for such signals. These signals do not affect the performance or the functionality of the equipment.

13.13.2.2 Critical signals

- a. Signals for which signal-to-noise ratio, pickup or crosstalk noise can affect performance of a circuit shall be identified as “critical”.

NOTE Examples of critical signals are: feedback signals of control loop of series and switching regulators.

13.13.2.3 Highly critical signals

- a. Signals with high speed or high bandwidth or low amplitude or high resolution shall be identified as “highly critical”.

NOTE The performance degradation of such signals can affect the functionality of the equipment.

NOTE The parasitic inductance of a track and the parasitic capacitance of a pad can affect high speed analog circuit performance.

- b. The length of a track connecting an inverted input of a high speed amplifier with a passive component should be minimised.

NOTE This reduces the noise coupling to high impedance input of the amplifier. The objective is to guarantee the noise performance of the amplifier.

- c. Highly critical circuitry specified in the requirement 13.13.2.3a shall have priority in the design, placement and routing.

13.13.3 Routing and shielding

- a. Ground planes shall be used for critical and highly critical analog circuitries specified in the requirements 13.13.2.2a and 13.13.2.3a.

- b. Ground planes should be used for non-critical analog circuitries specified in the requirement 13.13.2.1a.

NOTE Ground planes shield signals from pickup noise.

- c. The track length of high impedance signals should be minimized.

NOTE This is done to reduce crosstalk and pickup noise.

- d. High impedance signals should be shielded or have guard rings implemented.

- e. “Set on test” components should be placed near to the adjusted component.

- f. Test point and jumpers for analog circuitry should be placed near to the circuit that requires adjustment.

NOTE Examples of circuit that can require adjustment are: series regulator, voltage reference, DC/DC converters.

13.14 Mixed analog-digital PCB

- a. Analog and digital circuitry on the same PCB shall be separated from each other by zone management and shielding.
- b. Planes of analog and digital circuitries should be separated.

NOTE Superimposed analog and digital planes can result in AC coupling and digital noise on the analog signal.

- c. Planes of analog and digital circuitries may be combined in case separation of the planes has an adverse effect on performance or functionality or EMC.
- d. In case planes of analog and digital circuitries are combined, power supply decoupling shall be implemented.
- e. Digital signals should not be routed through analog zones.

NOTE The accurate routing of the analog and digital signals is important for DAC, ADC and MUX applications.

14 Design for assembly

14.1 Overview

The requirements in this clause apply to the design of the PCB for PCB assembly by internal or external assembly house. The surface pattern of the PCB depends on the PCB geometry, the assembly method and the location of mechanical and external elements.

14.2 General

- a. When PCB design and PCB assembly are performed in different companies, the supplier shall confirm with the assembly house which assembly processes are used.
- b. Pad geometry and outer layers shall be reviewed by the assembly house for potential assembly problems in compliance with requirements from clause 5 to clause 15 of ECSS-Q-ST-70-08 and in compliance with requirements from clause 5 to clause 16 of ECSS-Q-ST-70-38.
- c. Pad geometry and outer layers shall be approved by the assembly house in compliance with requirements from clause 5 to clause 15 of ECSS-Q-ST-70-08 and in compliance with requirements from clause 5 to clause 16 of ECSS-Q-ST-70-38.
- d. Assemblies using SnPb solder alloys shall not be used on gold plated pads in conformance with the requirements 6.8.2a of the ECSS-Q-ST-70-08.

NOTE Today Electroless Nickel–Palladium Immersion gold surface finish is under evaluation.

14.3 Placement requirements

14.3.1 Conductive patterns

- a. The distance of the surface pattern to the adjacent surroundings shall be in conformance with values specified in Table 14-1.

NOTE An illustration is given in Figure 14-1.

Table 14-1: Minimum distance as-manufactured of the surface pattern to the adjacent surroundings

Dimension	Description	Permitted	Recommended
A 1	Dimension between track and PCB edge	0,7 mm	
A 2	Dimension between track and mechanical part	1,0 mm	2,0 mm
A 3	Dimension between track and non-plated through hole	0,4 mm	
A 4	Dimension between via pad and hole for mechanical function	1,0 mm	
A 5	Dimension between track and screw	0,5 mm	
A 6	Dimension between conductive surface pattern and PCB edge	0,25 mm	
A 7	Dimension between edge of screw hole and PCB edge	1,0 mm	1,5 mm

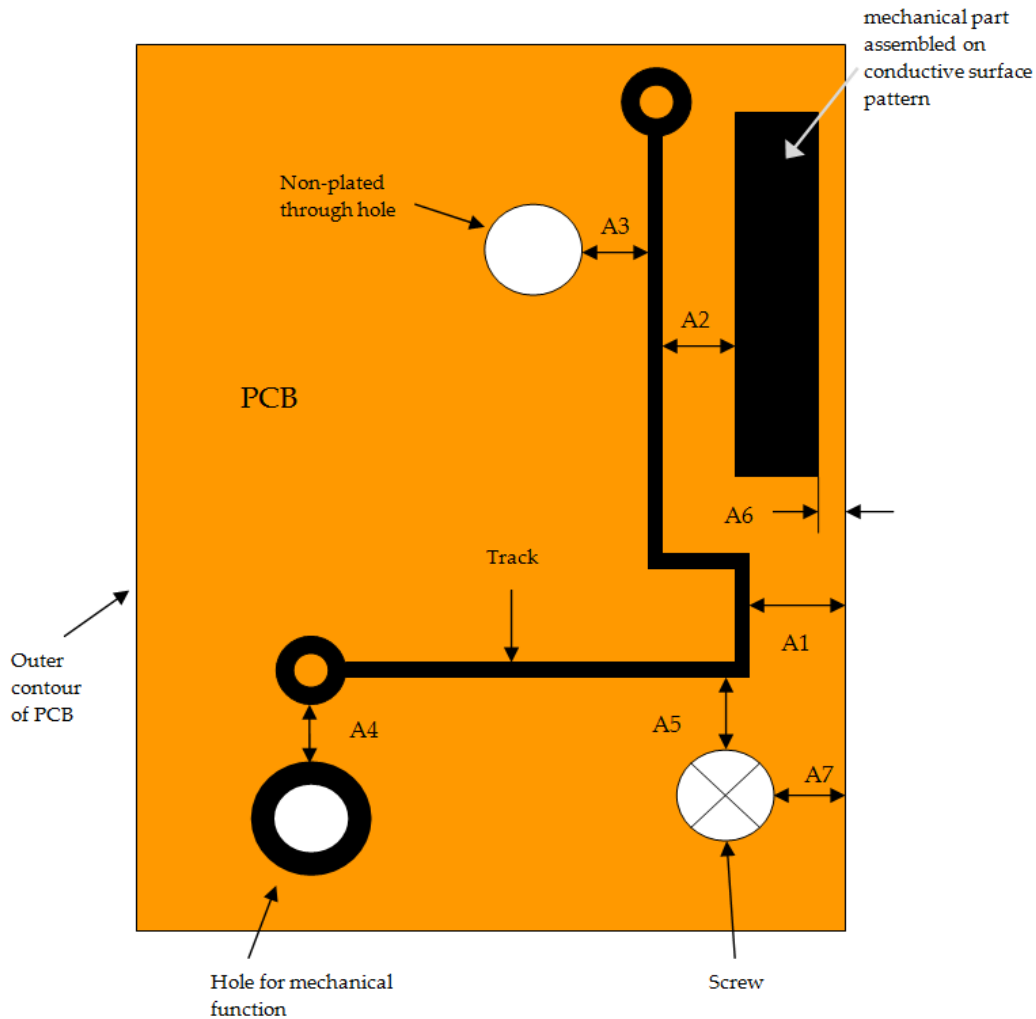


Figure 14-1: Illustration of surface pattern to adjacent surroundings

14.3.2 Components

- a. The conductive parts of components shall not be in contact with each other or with any metal in their adjacent surroundings.

NOTE For example screws, rack, and structure.

- b. The distance of components to adjacent surroundings shall be in conformance with values specified in Table 14-2.

NOTE An illustration is given in Figure 14-2.

- c. The width of the zone without components along the PCB edges should be ≥ 5 mm.

NOTE This is done to provide a clearance for the conveyor in the assembly equipment.

- d. In case the lay-out of the PCB requires the restricted zone to be less than 5 mm from the edge of the PCB, the PCB assembly house shall be consulted.

Table 14-2: Minimum distance as-manufactured of components to adjacent surroundings

Minimum dimension	Description	Permitted data	Recommended data
B 1	Distance between component body to component body, one of which is non-conductive	> 0 mm, in case that precautions are taken to prevent damage during vibration.	0,6 mm
B 2	Distance between component lead and cover (of electronic box)	1,0 mm	1,0 mm
B 3	Component lead to component lead (B3-a) Component lead to component body (B3-b) Component lead to adjacent surroundings (B3-c) Component body to adjacent surroundings (B3-d) Distance between component body to component body, both of which are conductive (B3-e)	0,5 mm	1,0 mm

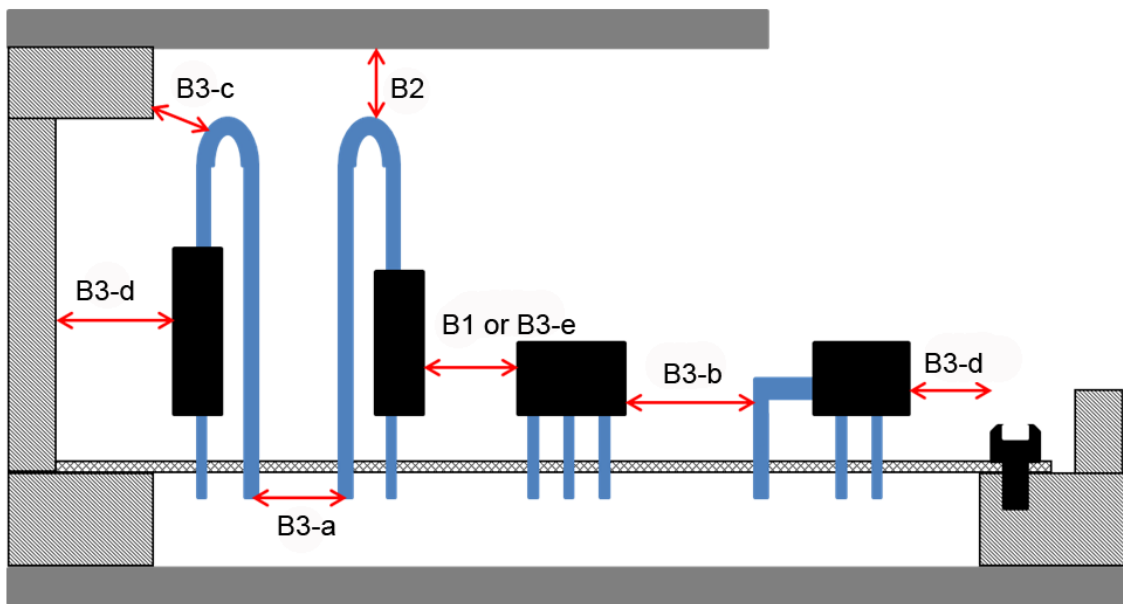


Figure 14-2: Illustration of component placing on PCB w.r.t. adjacent surrounding

14.3.3 Component pads

a. Design for assembly on component pads shall be in conformance with requirements from the clause 8 of ECSS-Q-ST-70-08 and clause 11 of ECSS-Q-ST-70-38.

b. The distance of component pads to adjacent surroundings shall be in conformance with values specified in Table 14-3.

NOTE An illustration is given in Figure 14-3

c. The location of the devices shall be such that each solder connection can be visually inspected.

d. High components should not be placed close together.

NOTE This is done to enable repair by manual soldering.

e. Around an AAD a clearance should be implemented for repair.

NOTE Requirements for PCB repair are specified in ECSS-Q-ST-70-28.

f. For PCBs submitted to conveyor wave soldering, components and tracks should be oriented in the solder direction.

g. Designed test points shall be separated from the component solder pads.

h. Test points may be incorporated in the solder pad by extending the pad dimensions so that test probing takes place outside the critical solder joint area in conformance with requirements 12.3.a.22 of the ECSS-Q-ST-70-08 and 13.3.a.21 of the ECSS-Q-ST-70-38.

i. The extended pad shall be included in the assembly qualification as the extra size affects the volume of solder in the joint.

j. Thermal relief shall be designed between component PTH and copper planes.

Table 14-3: Minimum distance as-manufactured to component pads

Minimum dimension	Description	Permitted data
A, B, C, G, J	- Dimension between pads of PTH on solder side - Pads of non-leaded SMT devices	0,3 mm
D, E, H, I	- Dimension between pads of PTH on component side - Pads of leaded SMT devices	0,5 mm
F	Dimension between component body to pad	0,5 mm

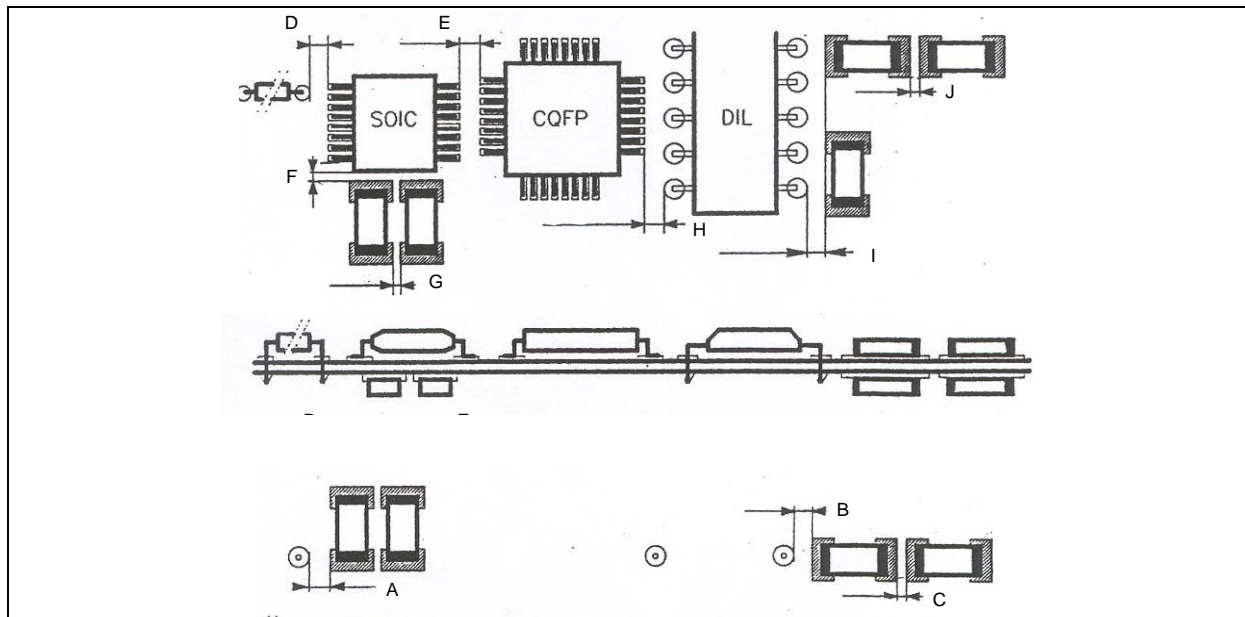


Figure 14-3: Illustration of pad placing on PCB with respect to adjacent surroundings

14.3.4 Fan out of SMT pads

- a. The track length between SMT pad and via shall be in conformance with values specified in Table 14-4

NOTE 1 In Figure 14-4: L_T is the track length between SMT and via.

NOTE 2 The through going via may absorb all tin-lead away from the pad during assembly.

- b. For other track dimensions, the supplier shall extrapolate or interpolate from the values specified in requirement 14.3.4a.

NOTE Larger distances are preferred, even though tracks are avoided on external layers.

- c. The fan out should be symmetric.

NOTE This is done to avoid unwanted movement of components during reflow soldering. An example of a symmetric fan out is given in Figure 14-4.

- d. The SMT pad of an AAD may be asymmetric.

NOTE Examples of asymmetric SMT pad are key hole, solder verification snip. The purpose of this is to verify quality of the solder joint.

- e. The four quadrants of the total footprint of the AAD shall be designed point symmetric such that each quadrant has the asymmetry in four directions.

NOTE This is done to prevent the component drifting away from the footprint during assembly. An example of a point symmetric AAD footprint is given in Figure 14-5.

- f. The width of the track connecting to an SMT pad of a chip device shall be $\leq 1/3$ of the width of the pad.

NOTE 1 An illustration of fan out from SMT pad is given in Figure 14-4.

NOTE 2 In Figure 14-4: W_T is the width of the track connecting to an SMT pad of a chip device, and W_P is width of the pad

NOTE 3 This is done for thermal relief to prevent the track acting as a heat sink and removing heat from the pad during assembly.

NOTE 4 This requirement is defined for chip devices not for flat packs.

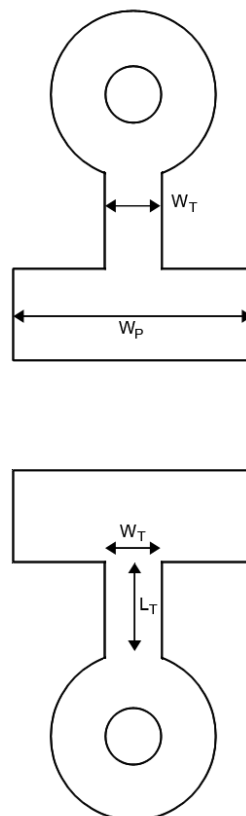
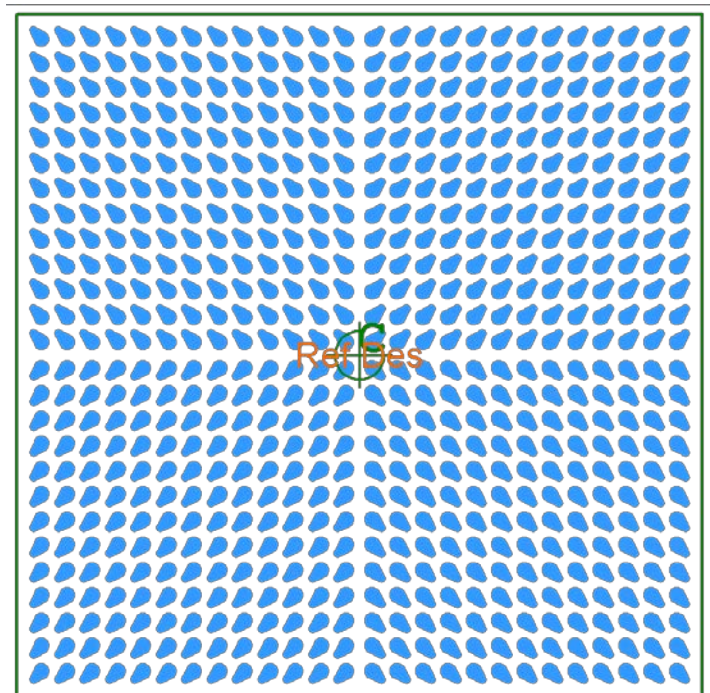


Figure 14-4: Fan out from SMT pad to via

Table 14-4: Minimum track length to component pads

W_T [mm]	L_T [mm]
0,2	$\geq 0,25$
0,3	$\geq 0,7$

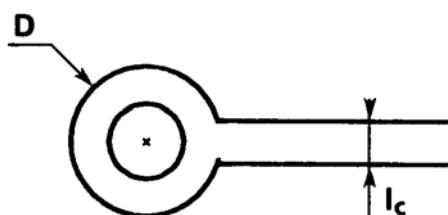

Figure 14-5: Example of point symmetric AAD footprint

14.3.5 Fan out of PTH

- a. The track width connecting to soldering pad of PTH shall be maximum half of the diameter of the pad.

NOTE 1 An illustration of track connecting to soldering pad is shown in Figure 14-6

NOTE 2 In Figure 14-6: l_c is the track width connecting to soldering pad of PTH with diameter D .


Figure 14-6: Track width (l_c) ratio to PTH pad (D) diameter.

- b. The track length between soldering pad of PTH and via shall be at least half the diameter of the largest pad

NOTE 1 An illustration of track between soldering pad and via is shown in Figure 14-7.

NOTE 2 In Figure 14-7: L_c is the track length between soldering pad of PTH with diameter D_1 and via pad with diameter D_3 .

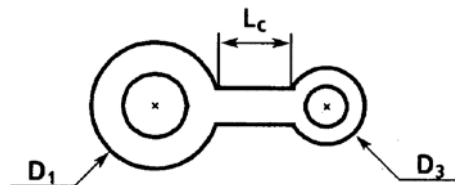


Figure 14-7: Track length (L_c) between soldering pad of PTH (D_1) and via pad (D_3)

14.4 Specific requirements for fused tin-lead finish

- a. Adhesive bonding should not be performed on fused tin-lead.
- b. Bonded devices shall have clearance for spot bonding.

14.5 Dimensional requirements for SMT foot print

14.5.1 Overview

The design rules of this clause apply for new pad designs. Successful assembly verification justifies the use of existing pad designs provided that minimum insulation distance is achieved, as specified in requirement 14.5.2c. This requirement defines permission for continued use of pad designs that have been successfully verified prior to the first issue date of this standard.

The use of blind via in component pad can act as a heat sink.

This clause specifies as-manufactured pad dimensions for various component footprints. The as-designed pad dimensions depend on following:

- a. the etching tolerance for the surface copper thickness,
- b. etching undercut
- c. tolerance of component dimensions
- d. minimum width of solder fillet
- e. misalignment of component on footprint

Table 14-5 Legend for dimensions of components and footprint

Minimum dimension	Description
i	insulation distance
M1	Solder fillet
M2	
M3	
X	Width of pad
AA	Distance between pads
BB	Total length of pad footprint
Y	Length of pad
L, E	Length of component
T	Length of termination
W, C	Width of termination
P	Pitch of AAD
D	Diameter of pad for AAD

14.5.2 General

- a. Insulation distance shall be in conformance with requirements of clauses 13.8, 13.9 and 13.10.
- b. The largest distance shall be used in case the distance between pads specified as AA in Table 14-6, Table 14-7, Table 14-8, Table 14-9 and Table 14-10 is smaller than the insulation distance specified in the requirement 14.5.2a.

NOTE 1 Insulation distance is indicated with the letter "i" in Figure 14-8, Figure 14-9, Figure 14-10, Figure 14-11, Figure 14-12 and Figure 14-13.

NOTE 2 The minimum insulation distance applies to all pad designs, including existing pad designs, new pad designs and pad designs not specified in this standard.

- c. Existing pad designs may deviate from the requirements of clauses 14.5.3 to 14.5.7, in case the following conditions are met:
 1. the existing pad designs have been already verified in accordance with clause 14 of ECSS-Q-ST-70-38 prior to the first issue date of ECSS-Q-ST-70-12, and
 2. insulation distance is in conformance with requirement 14.5.2a.

NOTE For background on requirement 14.5.2c see clause 14.5.1.

- d. Pad designs of components that are not specified in requirements from clauses 14.5.3 to 14.5.8 shall be designed in compliance with the manufacturer's datasheet and requirements from clause 14.5.2a.

14.5.3 Bipolar components

- a. The pad design for bipolar components shall be in conformance with the as-manufactured values specified in the Table 14-6, except for the case specified in 14.5.2c

NOTE 1 Examples of bipolar components are: cylindrical, rectangular and square end-capped devices

NOTE 2 An illustration is given in Figure 14-8.

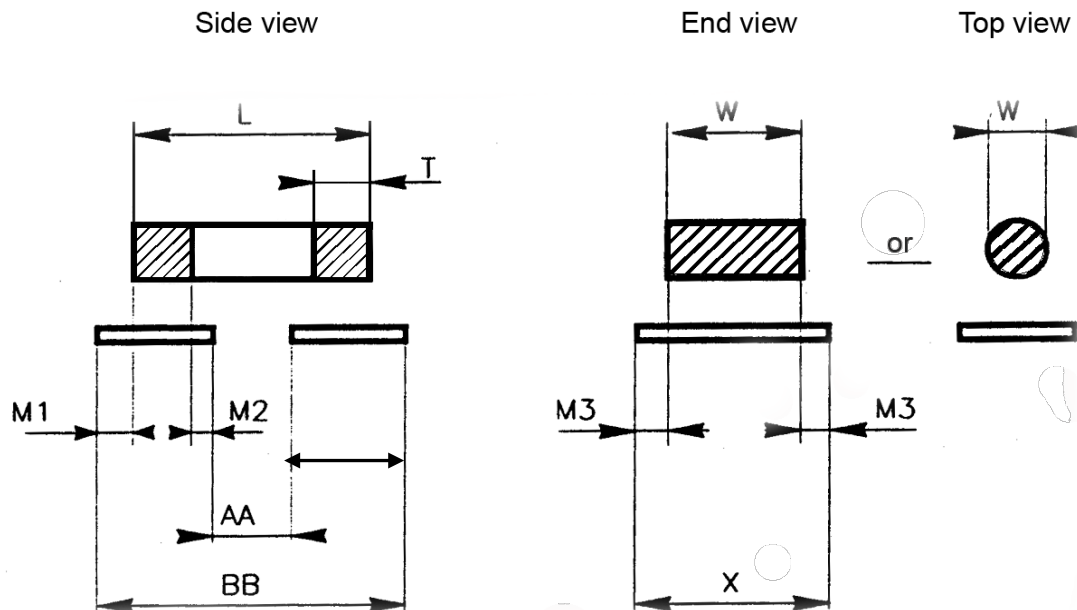


Figure 14-8: Illustration of bipolar component pads

Table 14-6 As-manufactured pad sizes for bipolar components

Minimum dimension	Permitted data
M1	0,2 mm
M2	0,1 mm
M3	> 0 mm
X	$W_{\max} + 2 * M3$
AA	$L_{\min} - 2 * (T_{\max} + M2)$
BB	$L_{\max} + 2 * M1$
Y	$\frac{BB - AA}{2}$

14.5.4 SOIC components

- a. The pad design for components of type SOIC with gullwing-leads shall be in conformance with the as-manufactured values specified in Table 14-7, except for the case specified in 14.5.2c.

NOTE An illustration is given in Figure 14-9.

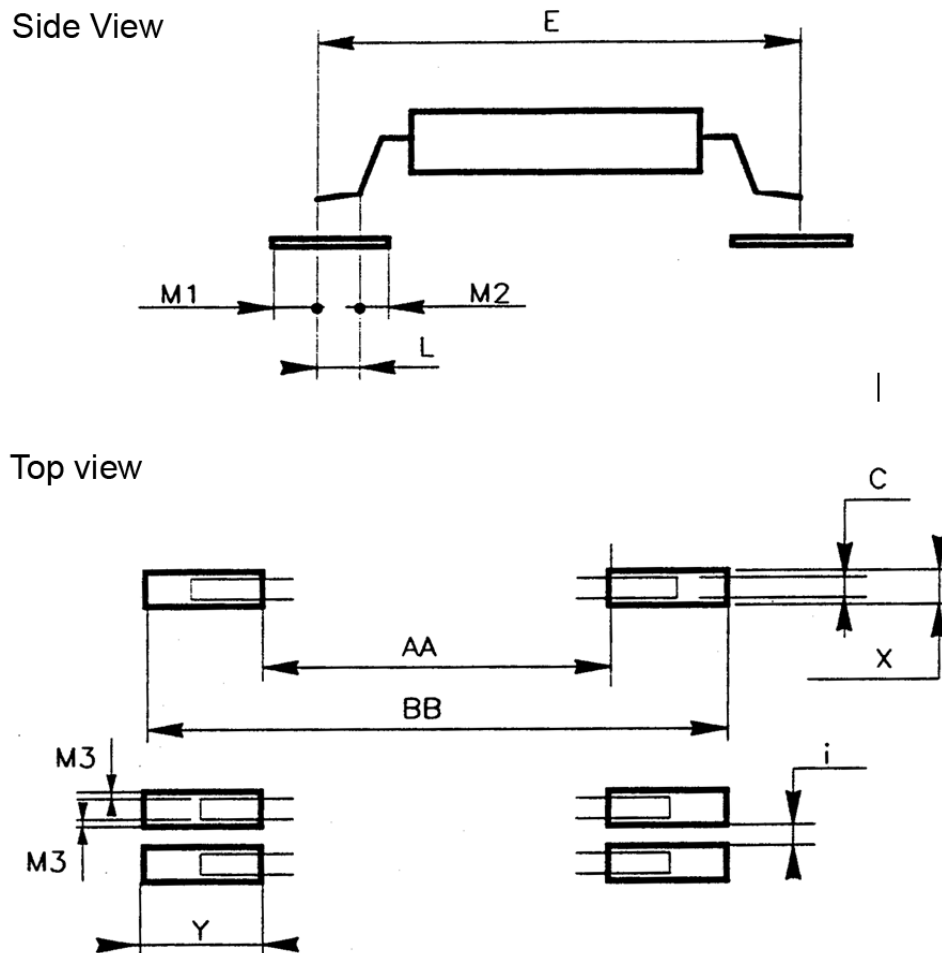


Figure 14-9: Illustration of SOIC component pads

Table 14-7: As-manufactured pad sizes for SOIC components

Minimum dimension	Permitted data
M1	0,2 mm
M2	0,2 mm
M3	> 0 mm
X	$C_{\max} + 2 * M3$
AA	$E_{\min} - 2 * (L_{\max} + M2)$
BB	$E_{\max} + 2 * M1$

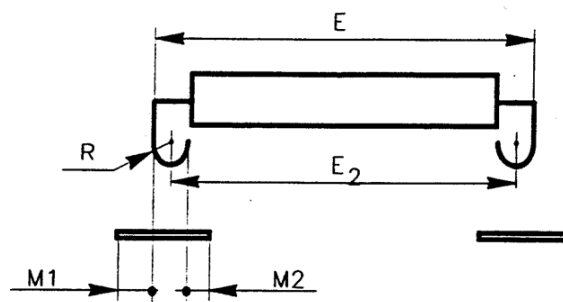
Y	$\frac{BB - AA}{2}$
---	---------------------

14.5.5 J-led components

- a. The pad design for components with J-leads shall be in conformance with the as-manufactured values specified in Table 14-8, except for the case specified in 14.5.2c.

NOTE An illustration is given in Figure 14-10.

Side view



Top view

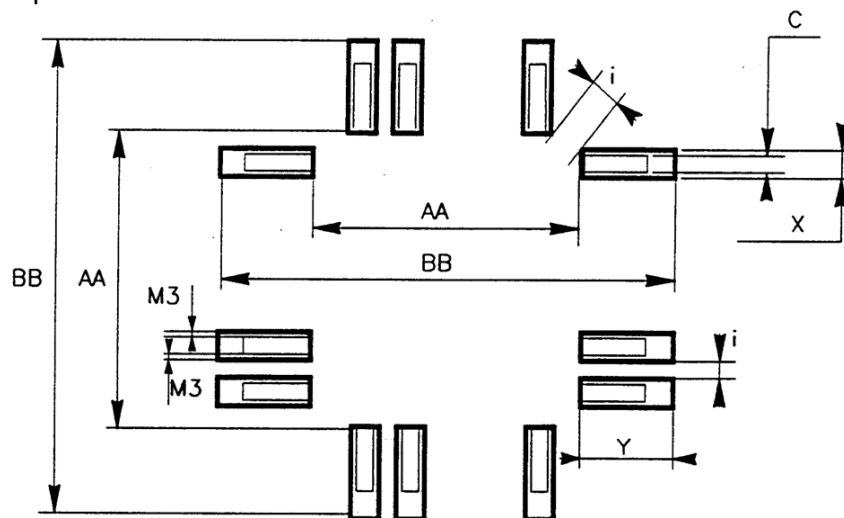


Figure 14-10: Illustration of J-led component pads

Table 14-8: As-manufactured pad sizes for J-leaded components

Minimum dimension	Permitted data
M1	0,2 mm
M2	0,2 mm
M3	> 0 mm
X	$C_{\max} + 2 * M3$
AA	$E_{2 \min} - 2 * (R_{\max} + M2)$
BB	$E_{\max} + 2 * M1$
Y	$\frac{BB - AA}{2}$

14.5.6 LCC components

- a. The pad design for components of type LCC shall be in conformance with the as-manufactured values specified in the Table 14-9, except for the case specified in 14.5.2c.

NOTE 1 An illustration is given in Figure 14-11.

NOTE 2 LCC means leadless chip carrier.

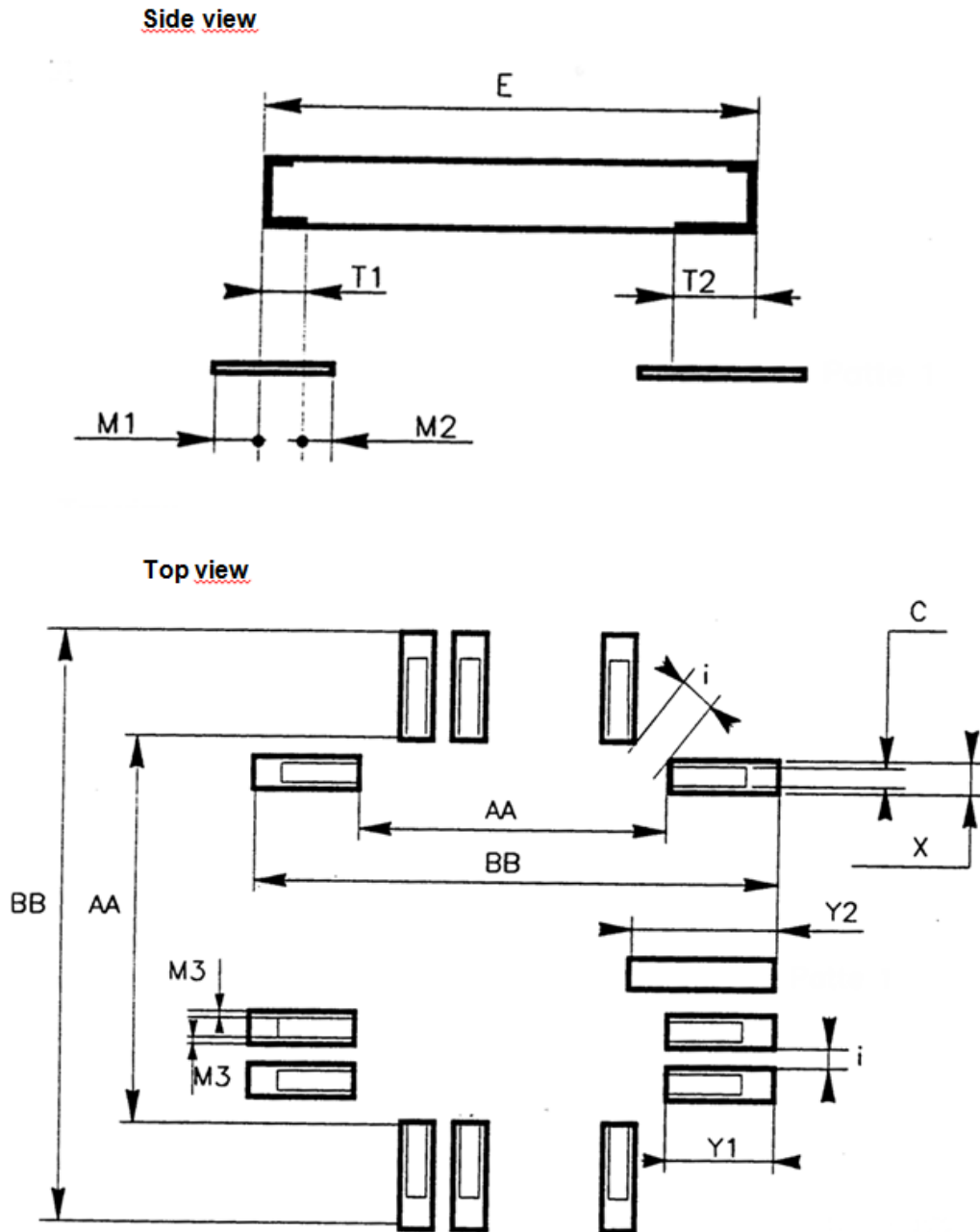


Figure 14-11: Illustration of LCC component pads

Table 14-9: As-manufactured pad sizes for LCC components

Minimum dimension	Permitted data
M1	0,2 mm
M2	0,2 mm
M3	> 0 mm
X	$C_{\max} + 2 \cdot M3$
AA	$E_{\min} - 2 \cdot (T1 + M2)$
BB	$E_{\max} + 2 \cdot M1$
Y1	$\frac{BB - AA}{2}$
Y2	$Y1 + (T2 - T1)$

14.5.7 Flat pack components

- a. The pad design for components of type FP and QFP shall be in conformance with the as-manufactured values specified in the Table 14-10, except for the case specified in 14.5.2c.

NOTE 1 An illustration is given in Figure 14-12.

NOTE 2 FP means flat pack. QFP means quad flat pack.

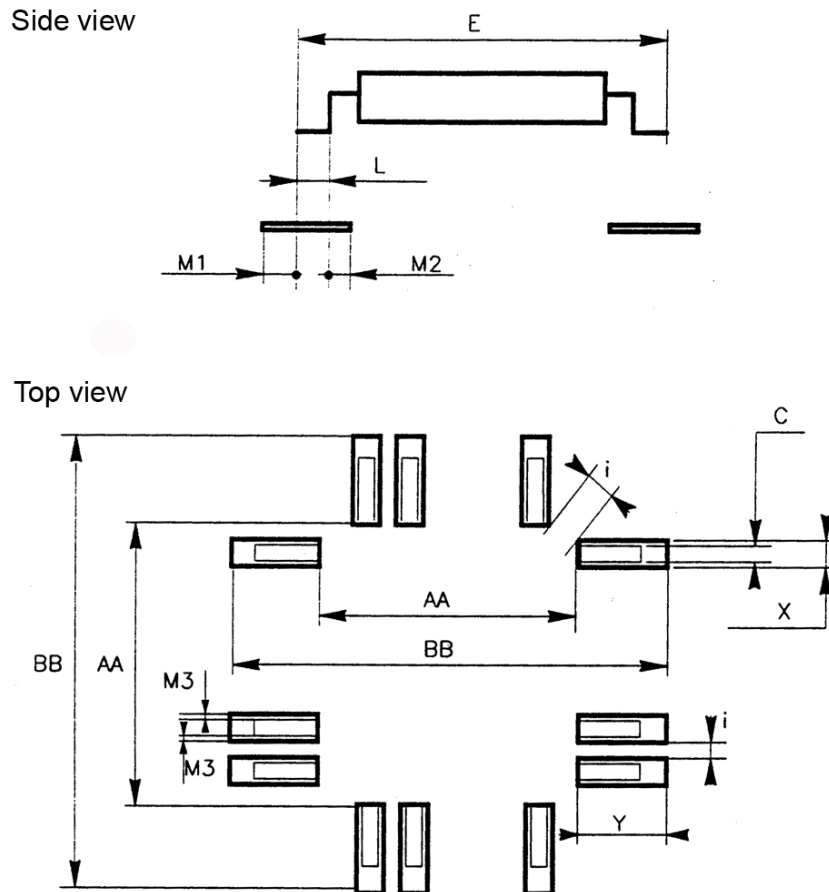


Figure 14-12: Illustration of FP and QFP component pads

Table 14-10: As-manufactured pad sizes for FP and QFP components

Minimum dimension	Permitted data
M1	0,2 mm
M2	0,2 mm
M3	> 0 mm
X	$C_{\max} + 2 * M3$
AA	$E_{\min} - 2 * (L_{\max} + M2)$
BB	$E_{\max} + 2 * M1$
Y	$\frac{BB - AA}{2}$

14.5.8 AAD components

- a. The pad design for components of type AAD should be in conformance with the as-designed values specified in Table 14-11.

NOTE An illustration of circular pad design for AAD footprint is given in Figure 14-13. An illustration of teardrop pad design of AAD footprint is given in Figure 14-5.

- b. The pad design for components of type AAD should be circular or with teardrop.
- c. The pad design for components of type AAD should be with via-in-pad technology.

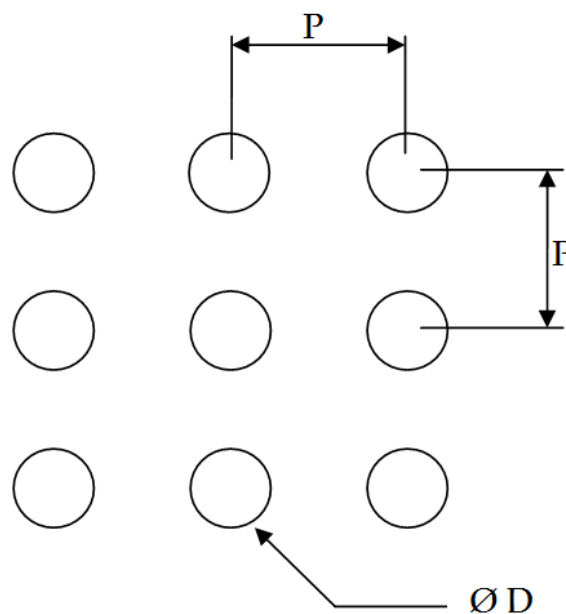


Figure 14-13: Illustration of AAD component pads

Table 14-11: As-designed pad diameter for AAD components

Component pitch (P)	Minimum pad diameter (D)
1,27 mm	0,7 mm
1,0 mm	0,6 mm

15

Design of test coupon

15.1 Design rules for test coupon

- a. The supplier shall be accountable for the design of the test coupon in conformance with requirements from the Clause 15.
- b. The design of the test coupon shall be indicated as a Review Item.

NOTE It is common practice that the PCB manufacturer designs the coupon on behalf of the supplier. However, these requirements ensure that the supplier verifies the design of the coupon and has the end responsibility.

15.2 Test coupon design

- a. The test coupon design should be representative of the PCB.
- b. In case the PCB design is complex and the coupon cannot be designed with full representativity, the inspection on the coupon should be supplemented by a FAI.

NOTE The different density of a coupon compared to the rest of the PCB can limit the representativity. A higher copper density of the footprint of an AAD compared to a typical coupon design can cause a lower plating thickness. The higher via density of the footprint of an AAD can cause problems with resin flow and filling, which can remain undetected in a coupon of different density. High pressure areas local within the PCB can cause lower dielectric spacing compared to the coupon.

- c. Test coupon location in the manufacturing panel should be representative of the worst case.

NOTE Worst-case location for registration is typically on the corners of the panel. However, worst-case for plating is typically in the centre of the

panel. In this case, it is important the PCB manufacturer compares the plating thickness in the coupon with the rest of the panel. This is typically verified during qualification.

- d. The following specific features shall be represented on the coupon in the same configuration as on the PCB:
1. PCB build-up,
 2. Copper planes,
 3. Non-functional pads,
 4. Internal heat sinks,
 5. Hole types as follows:
 - (a) PTH for components
 - (b) Through-hole vias
 - (c) Blind vias
 - (d) Buried vias
 - (e) Microvias in the same configuration as on the PCB
 6. The minimum hole size of each type.
 7. The pad stack with the minimum pad diameter
 8. Either the maximum component hole size or the most frequently used component hole size.
 9. The minimum track width and spacing on each layer.
 10. All used surface finishes and overlap zones in conformance with requirement 7.8.1d.1.
 11. All plating, drilling and lamination sequences.
 12. Backdrilling of vias.
 13. Embedded film resistors of highest and lowest resistance
 14. Flex layers including cut-out and cover layer placement
 - NOTE 1 Minimum hole size is included into requirement 15.2d.6 to verify plating in maximum aspect ratio holes.
 - NOTE 2 The pad stack with minimum pad diameter is included into requirement 15.2d.7 to verify worst-case annular ring as-manufactured and registration.
 - NOTE 3 In the requirement 15.2d.8 the supplier can decide which hole sizes to be incorporated on the coupon.
 - NOTE 4 Minimum track width and spacing is included into requirement 15.2d.9 to be verified as function of copper thickness on a dedicated pattern.

- e. In case teardrop reinforcement is used on the PCB, the following conditions shall be designed on the coupon:
1. the teardrop reinforcement is not included on the coupon, and
 2. the minimum pad diameter is included on the coupon.

NOTE This is done to ensure the annular ring requirement specified in the 11.5.2a.3 (i.e. 25 μm) is achieved without teardrop reinforcement.

- f. The coupons shall be capable of completing the following tests:

1. Thermal tests:
 - (a) Solderability,
 - (b) Rework Simulation,
 - (c) Thermal Stress.
2. Mechanical tests:
 - (a) Peel strength,
 - (b) Pull-off test,
 - (c) Flex bend cycles,
 - (d) Tape bond test.
3. Electrical tests:
 - (a) Insulation resistance,
 - (b) Interconnection resistance and continuity test,
 - (c) Inter-layer dielectric withstanding voltage,
 - (d) Controlled impedance,
 - (e) Embedded film resistance.
4. Dimensional inspection:
 - (a) Annular ring,
 - (b) Track width and spacing.
5. Inspection of general aspect in microsections.

- g. A dedicated coupon pattern shall be implemented to verify controlled impedances, when used on the PCB.

NOTE This dedicated coupon can be designed by the PCB manufacturer.

- h. With multiple small PCBs on a panel, a spare PCB may be used instead of a coupon.
- i. Each coupon shall have a serial number that ensures traceability to the panel.

Annex A (normative)

PCB definition dossier - DRD

A.1 DRD identification

A.1.1 Requirement identification and source document

This DRD is called from ECSS-Q-ST-70-12 requirement 5.2a.

A.1.2 Purpose and objective

The PCB definition dossier specifies the design and contains all information required for the tooling. The PCB definition dossier is part of the PCB manufacturing dossier under approval in the MRR.

A.2 Expected response

A.2.1 Scope and content

<1> General

- a. The PCB definition dossier shall include the following data:
 1. Description, as specified in A.2.1<2>,
 2. Mechanical layout, as specified in A.2.1<3>,
 3. Artwork data , as specified in A.2.1<4>,
 4. Drawing, as specified in A.2.1<5>,
 5. Specific electrical test, as specified in A.2.1<6>,
 6. Review Items , as specified in A.2.1<7>,
 7. Check list, as specified in A.2.1<8>,

<2> Description

- a. The PCB definition dossier shall include a description containing at least the following data:
1. Base material,
 2. Finishes,
 3. Number of layers,
 4. Size,
 5. Thickness and tolerance,
 6. Part number of PCB design: specified by supplier,
 7. Revision number of PCB design,
 8. Heat sink including:
 - (a) Heat sink thickness,
 - (b) Tolerances permitted,
 - (c) Material used,
 - (d) Type of protective surface coating.

NOTE Synonyms of revision number in requirement 7 are issue number and version number.

<3> Mechanical layout

- a. The PCB definition dossier shall include the mechanical layout containing at least the following data:
1. One or more mechanical drawings including a dimensioning system,
 2. Location and form for the PCB manufacturer to place date code and serialization.

NOTE 1 It is recommended in the requirement 1 to use a reference hole as datum point. The reference hole is recommended to be drilled in the first drill cycle.

NOTE 2 The form of numbering as specified in requirement 2 can be in ink or conductive pattern.

NOTE 3 An example of mechanical layout is shown in Figure A-1.

<4> Artwork data

a. The PCB definition dossier shall include the artwork data containing at least the following information:

1. Artwork data of each conductive PCB layer,
2. Artwork data for non-conductive layers,
3. Drilling and milling files.

NOTE 1 Examples of conductive layers for the requirement 1 are circuitry and selective finishes.

NOTE 2 Examples of non-conductive layers for the requirement 2 are solder mask and silk screen

NOTE 3 It is preferred in the requirement 3 to distinguish by different tool codes the plated and non-plated holes in the drill file even if they have identical diameter.

NOTE 4 For example, ODB++ is a format that describes all artwork of all layers in one file.

<5> Drawing

a. The PCB definition dossier shall include the drawings containing at least the following:

1. Drilling drawing,
2. Drilling tables, including:
 - (a) Symbol,
 - (b) Diameter of as-manufactured holes,
 - (c) Tolerance,
 - (d) Plated or not plated,
 - (e) Quantity.
3. Build-up data, including:
 - (a) Numbered conductive layers from top to bottom,
 - (b) Total copper thickness, basic + plated, for each layers with tolerances,
 - (c) Presence of planes: ground and supply, mesh plane or full copper,
 - (d) Presence of layers with controlled impedance,
 - (e) Thickness tolerance of laminate and prepreg, including minimum as-manufactured insulation distances,
 - (f) Total PCB thickness and tolerance,

- (g) Specification if thickness is measured over bare laminate or over conductive pattern,
- (h) Specific requirements for build-up.

NOTE 1 Examples of specific requirements for build-up are:

- style of prepreg and laminate,
- use of separate copper foil or copper cladding on laminate for external layers,
- use of single or double sided laminate,
- use of two individually cured insulators to achieve double insulation.

NOTE 2 Examples of Drilling drawing, Drilling table and Build-up data are given in Figure A-2, Figure A-3, Figure A-4.

<6> Electrical test

a. The PCB definition dossier shall include the electrical test description containing at least the following data:

1. Netlist,
2. Definition of any electrical tests that are specific to the PCB design and in addition to the standard set of electrical tests,
3. Identification of conductive lines that are designed to be of higher DC resistance that are at risk not to pass standard continuity testing,
4. The definition of controlled impedance tracks including:
 - (a) Reference planes,
 - (b) Type of coupling,
 - (c) Track dimensions, including height, width, cross section,
 - (d) Spacing dimensions in X,Y and in Z direction,
 - (e) Impedance value, tolerance and method of measurement.
5. Identification of any deliberate errors.

NOTE 1 Example for requirement 4(b): broadside coupled, edge coupled.

NOTE 2 In the requirement 4(e) the standard method for measuring impedance and tolerance is TDR in case the method is not specified

NOTE 3 Example for the requirement 5 are differences between artwork and net list such as star points.

<7> Review Items

- a. The PCB definition dossier shall include all Review Items.

- NOTE The following design features are required to be recorded as Review Items by the present standard:
1. Asymmetric build-up as specified in 7.1.1b,
 2. Molybdenum or CIC layers as specified in 7.1.1c
 3. Asymmetric copper cladding as specified in 7.1.2b,
 4. Total copper thickness >700 μm as specified in 7.1.2g,
 5. Single sheet of glass reinforcement in laminate as specified in 7.1.3c,
 6. 4 mil laminate as specified in 7.1.3f,
 7. 5 mil laminate at >30V as specified in 7.1.3g,
 8. Total PCB thickness as specified 7.3.2c and 7.3.3c,
 9. Reduced as-designed track width and spacing as specified in 7.4.3c and 7.4.4c,
 10. Fine pitch tracks as specified in 7.4.3g, and 7.4.5b,
 11. Removal of non-functional pads as specified in 7.5.1c,
 12. Reduced as-designed pad dimensions on solder side and component side as specified 7.5.2e and 7.5.2g,
 13. Pad diameter on rigid laminate <0,3mm larger than drilled hole as specified in 7.5.2i,
 14. Non-circular pads as specified in 7.5.3b
 15. Asymmetric copper planes as specified in 7.6e,
 16. Mixed surface finish as specified in 7.8.1e,
 17. 70 μm copper cladding in flex laminate as specified in 8.3.3c,
 18. Full copper planes in flex laminate as specified in 8.3.4b,
 19. 70 μm copper foil in combination with no-flow prepreg as specified in 9.3d,
 20. More than 2 flex laminates as specified in 9.3f,
 21. Pad diameter on flex laminates <0,6 mm larger than diameter of drilled hole as specified in 9.6b,
 22. Heat sinks and their mechanical stability as specified in 10.2a, 10.3.3a and 10.4.4a.
 23. Conductive surface pattern below external heat sink as specified in 10.3.2g,
 24. HDI as specified in 11.2c,
 25. Basic copper layer thickness <17 μm as specified in 11.4.1e,
 26. Diameter of external pad of microvia <0,15 mm larger than the ablated hole as specified in 11.4.3b,

27. Reduced annular ring with teardrop reinforcement as specified in 11.5.2a,
 28. Differential pair routed in 1 mm pitch AAD footprint as specified in 11.5.5a,
 29. Aspect ratio of vias >7 for 1 mm pitch AAD footprint as specified in 11.5.6a,
 30. Embedded film resistors as specified in the requirements 12.3a and 12.3i,
 31. Thickness of RF PCB >3mm as specified in 12.4b,
 32. Number of layers in RF PCB >8 as specified in 12.4d,
 33. Reduced insulation distance for RF elements as specified in 12.5.1b,
 34. Z-controlled backdrilling as specified in 12.8a,
 35. Profiled top layers as specified in 12.8b,
 36. Profiled aluminium backing as specified in 12.8c,
 37. Reduced as-designed insulation distance as specified in 13.10c,
 38. Controlled impedance tracks as specified in 13.11.2,
 39. Design of test coupon as specified in 15.1b.
- b. The PCB definition dossier shall include all specific inspections required for the Review Items.

<8> Check list

- a. The PCB definition dossier shall include a check list containing at least the following data:
1. all file names included in the archive,
 2. last modified data of each file,
 3. size of the file,
 4. CRC.

NOTE The purpose of the check list is to define the content of the archive file to prevent transmission of wrong data. In Figure A-5 an example of a check list is shown. The check list indicates the content of the archive file by the four items mentioned in A.2.1<8>.

A.2.2 Special remarks

- The format of the data of the PCB definition dossier shall be agreed between supplier and PCB manufacturer.
- One single file should be used for transferring the data between supplier and PCB manufacturer.
- The transmitted file should be an archive with lossless compression.

NOTE Examples of file extensions of archives with lossless compression are: .zip, .rar, .7z, .tar.
- The supplier shall ensure the file format enables verification of file integrity to prevent corruption.

NOTE For example verification by CRC.

A.2.3 Example figures

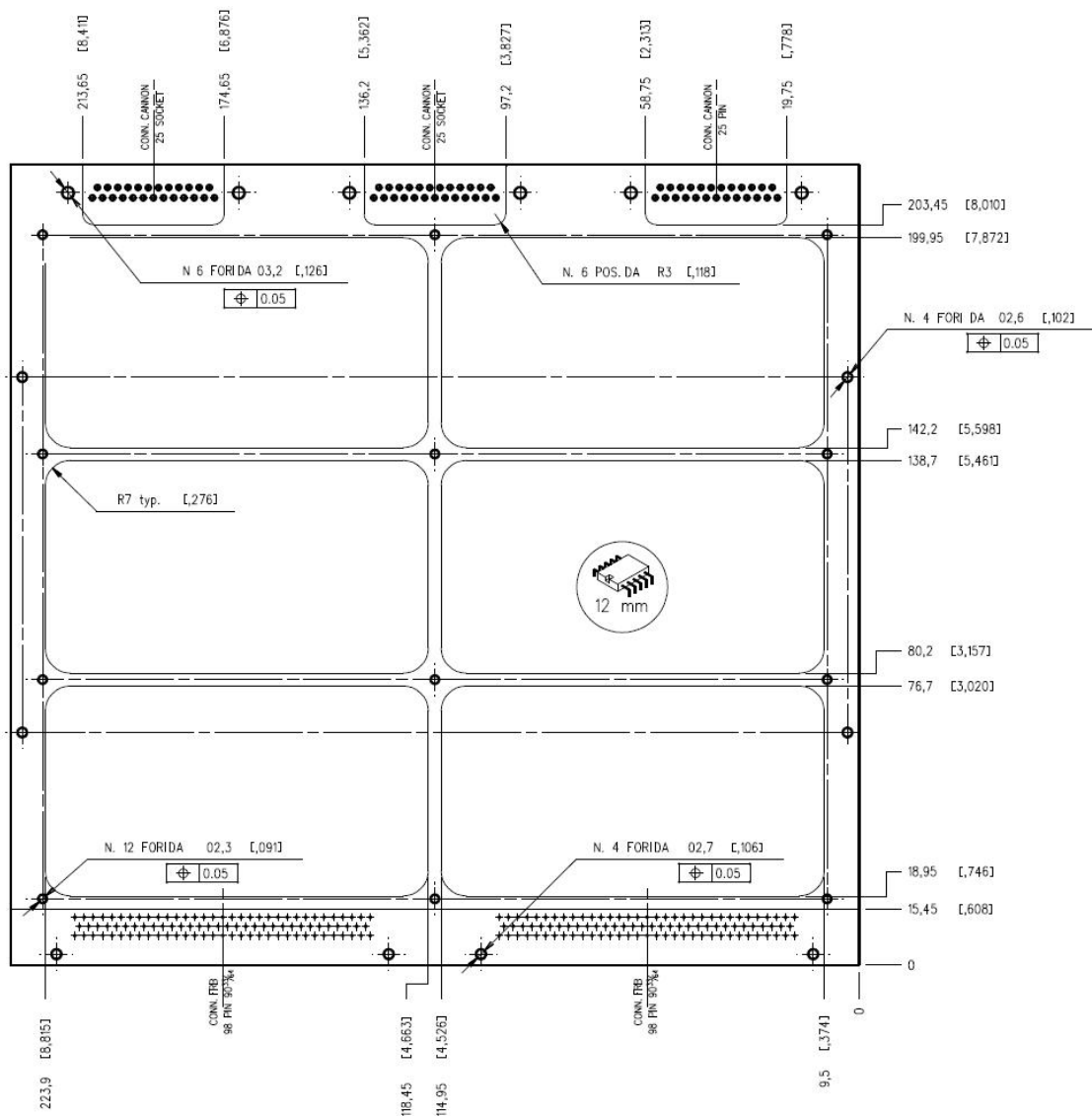


Figure A-1: Example of PCB mechanical layout

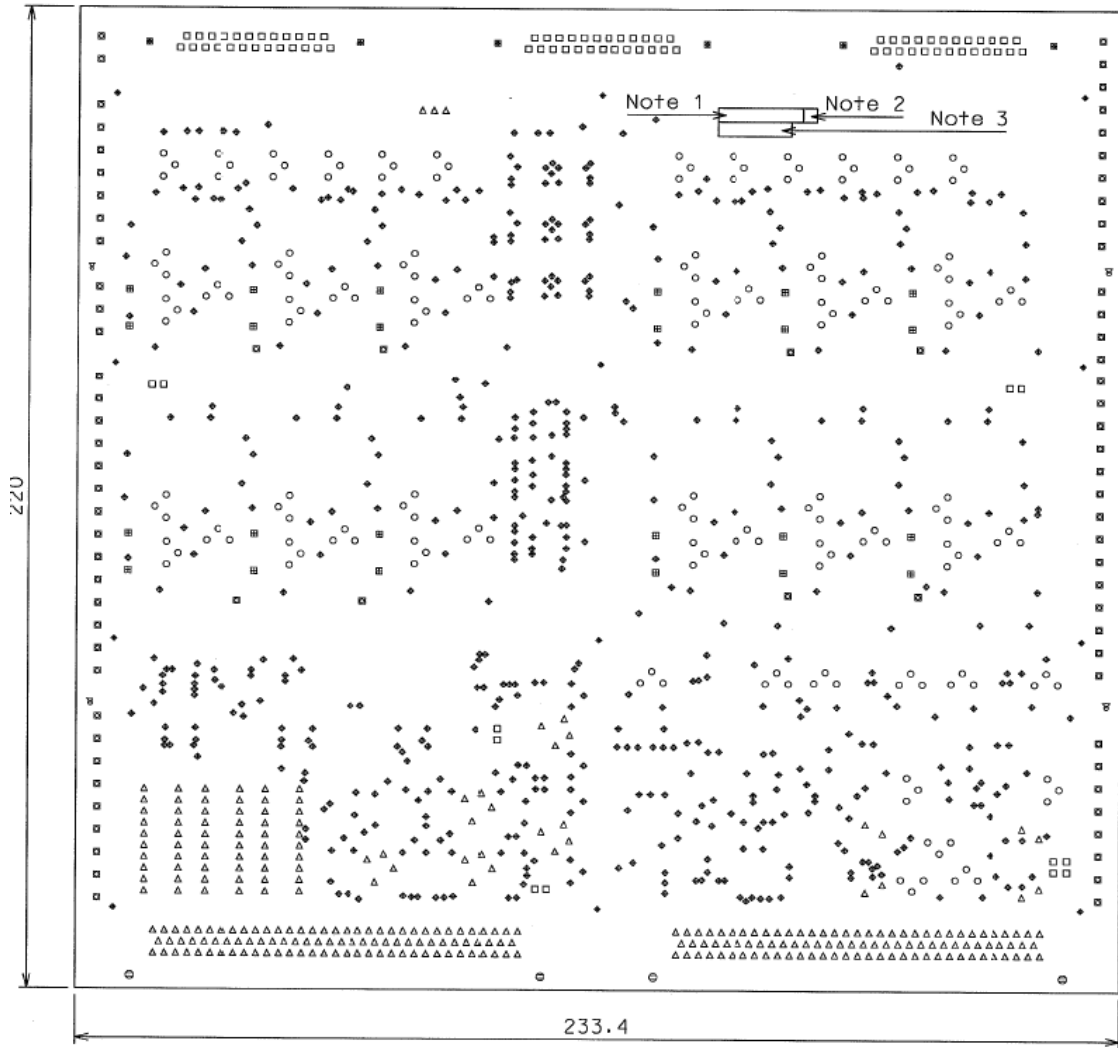


Figure A-2: Example of a drilling drawing

Through Holes			
Symbol	Diameter (mm)	Plated	Quantity
◆	0.3000	Yes	569
◻	0.5000	Yes	79
△	0.8000	Yes	287
□	0.9000	Yes	87
◆	2.3000	Yes	12
▽	2.6000	Yes	4
⊖	2.7000	Yes	4
⊠	3.2000	Yes	6
○	0.8000	No	177
⊞	1.0000	No	24

Figure A-3: Example of a table indicating as-designed plated hole diameters

Stack	Insulation thickness (as-manufactured board) (□m)	Copper plane	Base copper thickness (□m)	Drilling sequence						Total copper thickness (as-manufactured board) (□m)		Number of layers
				1	2	3	As-designed (□m)	Tolerances				
1			17						77,5	±30%	1	
	100µm min											
2			35						35	±20%	2	
	100µm min											
3			35						35	±20%	3	
	100µm min											
4			35						35	±20%	4	
	100µm min											
5			35						35	±20%	5	
	100µm min											
6			35						35	±20%	6	
	100µm min											
7		Yes	35						35	±20%	7	
	100µm min											
8		Yes	35						60	±20%	8	
	100µm min											
9		Yes	35						60	±20%	9	
	100µm min											
10		Yes	35						35	±20%	10	
	100µm min											
11			35						35	±20%	11	
	100µm min											
12			35						35	±20%	12	
	100µm min											
13			35						35	±20%	13	
	100µm min											
14			35						35	±20%	14	
	100µm min											
15			35						35	±20%	15	
	100µm min											
16			17						77,5	±30%	16	

Figure A-4: Example of build-up data















Gib700200846-01revA\							
File Name	File Type	Modified	Size	Ratio	Packed	CRC	
 DrillDrawing.gdo	File GDO	12-11-2009 10.49.38	240,904	76%	58,980	995DD56E	
 driver_mpe.dsn	Nome origine dati	12-11-2009 10.49.38	1,910	75%	482	30229A08	
 Layer10Bottom.gbr	File GBR	12-11-2009 10.49.38	82,433	73%	21,955	E78C9C4F	
 Layer1 Top.gbr	File GBR	12-11-2009 10.49.38	118,527	75%	30,156	AA09CA45	
 Layer2.gbr	File GBR	12-11-2009 10.49.38	159,644	72%	44,086	F615F858	
 Layer3.gbr	File GBR	12-11-2009 10.49.38	22,200	68%	7,091	6DD85226	
 Layer4.gbr	File GBR	12-11-2009 10.49.38	41,301	70%	12,394	45F936B2	
 Layer5.gbr	File GBR	12-11-2009 10.49.38	157,225	74%	41,549	08D530E4	
 Layer6.gbr	File GBR	12-11-2009 10.49.38	155,765	73%	41,589	0E14CD46	
 Layer7.gbr	File GBR	12-11-2009 10.49.38	27,191	68%	8,801	081B59D0	
 Layer8.gbr	File GBR	12-11-2009 10.49.38	83,037	73%	22,178	58AF5FED	
 Layer9.gbr	File GBR	12-11-2009 10.49.38	159,429	72%	43,981	496A1F1F	
 pwB700200846-01revA.dxf	File DXF	01-03-2010 12.07.14	1,899,315	91%	172,721	14331470	
 PwB700200846-01_REV_A.pdf	Adobe Acrobat Document	01-03-2010 11.55.42	117,405	10%	105,790	BC2E384F	

Figure A-5: Example of the archive file with CRC

Annex B (normative)

PCB manufacturing dossier – DRD

B.1 DRD identification

B.1.1 Requirement identification and source document

This DRD is called from the ECSS-Q-ST-70-12 requirement 5.2h

B.1.2 Purpose and objective

The PCB manufacturing dossier specifies the tooling and contains all information required for the manufacture. The PCB manufacturing dossier is under approval in the MRR.

B.2 Expected response

B.2.1 Scope and content

- a. The PCB manufacturing dossier shall contain following information:
 1. Reference to the purchase order including the following items:
 - (a) ID number
 - (b) Line item numbers for each required PCB type
 - (c) Line item numbers for each required PCB tooling
 - (d) Line item numbers for other required services like FAI
 - (e) Required quantities for each line item
 - (f) Required delivery dates for each line item
 2. Reference to the PCB definition dossier for each PCB type.
 3. Tooling files
 - (a) Imaging
 - (b) Drilling

- (c) Milling or routing
- (d) Plating
- (e) Lamination
- (f) AOI
- (g) Electrical testing
- (h) Process route card or traveller information
- (i) Inspection drawings

4. MRR checklist

NOTE An example of an MRR checklist is provided in Annex G.

- b. The supplier and PCB manufacturer shall document in the MRR checklist their approval of the PCB manufacturing dossier, the PCB definition dossier and all Review Items.

B.2.2 Special remarks

None

Annex C (informative)

Example of capability list of PID

Example of technical capabilities of PCB manufacturer as specified in the PID is shown in the Table C-1.

Table C-1: Example of technical capabilities of PCB manufacturer as specified in the PID

PCB		
PID		
Technology	DS	
	ML	
	Rigid-flex	
	HDI	
Number layers /sequence	Rigid	
	Flex	
	Drilling sequence	
	Lamination sequence	
Thickness/size	PCB Size (mm)	
	Maximum PCB Thickness (mm)	
	External minimum Cu thickness (μm)	
	External maximum Cu thickness (μm)	
	Internal minimum Cu thickness (μm)	
	Internal maximum Cu thickness (μm)	
Material	Epoxy	
	HTg epoxy	
	Polyimide	
	Flexible	

	PTFE	
	Mixed materials	
	Metal core	
Surface finish	Sn/Pb reflow	
	Ni/Au electroplated (Hard/soft)	
	ENIG	
	Tin diffusion layer	
	Soldermask	
Design features	External trace width (μm)	
	External space width (μm)	
	Internal trace width (μm)	
	Internal space width (μm)	
	Min Z insulation thickness (μm)	
Min hole size	PTH	
	Blind	
	Buried	
	Microvia	
Aspect ratio	PTH	
	Blind	
	Buried	
	Microvia	

Annex D (informative)

Track current rating computation methodology

D.1 Introduction of the three models

D.1.1 Overview

Track current rating can be computed with the following three commonly used models:

- IPC-2221A. The original chart dated from the 1950's from National Bureau of Standards as shown in figure 6.4 IPC-2221A (also referenced in Appendix A7 of IPC-2152) This model has been superseded by IPC-2152.
- IPC-2152 using the conductor sizing chart for vacuum and space environment in chapter 5.2.2 (dated August 2009).
- CNES/QFT/IN.0113.

The three models are similar for current rating computation. The CNES model is slightly more conservative than IPC-2152. An example is shown in Figure D-1.

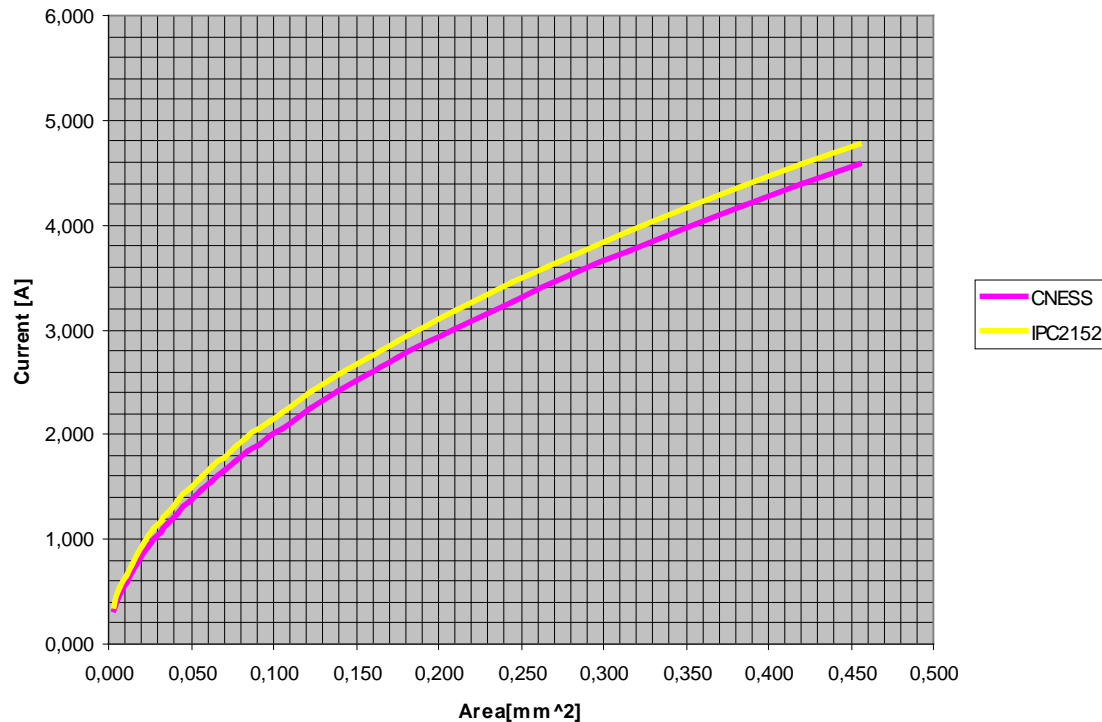


Figure D-1: Comparison between CNES and IPC-2152 current rating at 5°C increase

D.1.2 Formulae for the three models

- a. In the three models the temperature increment of tracks is calculated with respect to the initial temperature of the PCB when it is not powered.
- b. Current rating and cross sectional area are calculated in the three models by using the following formulae:

1. Current rating [I in Amp]
2. Cross sectional area of tracks [A in mm²]
3. Temperature increment of tracks [ΔT in °C].
4. Constants k_0 , k_1 , m_0 , m_1 . The values of the constants are specified in the paragraphs describing each model.
5. Conversion factor mm² to mils² $c_1=1550$.
6. Formula for current rating:

$$I = k_0 \cdot \Delta T^{k_1} \cdot (c_1 \cdot A)^{m_0 \cdot \Delta T^{m_1}}$$

7. Formula for cross sectional area:

$$A = \frac{1}{c_1} \cdot \left(\frac{I}{k_0 \cdot \Delta T^{k_1}} \right)^{\frac{1}{m_0 \cdot \Delta T^{m_1}}}$$

D.1.3 Example of current rating

Using the mathematical model of IPC-2152 specified in clause D.2, to carry a current of 1 A and to maintain a temperature increment of 5 °C, a track of as-manufactured thickness of 25 µm (as-designed thickness of 35 µm, as per Table 7-1) should have an as-manufactured track width of 925 µm (as-designed width of 975 µm, as per Table 7-3).

D.2 Track current rating computation based on IPC-2152

The IPC-2152 is the most comprehensive standard for the computation of the track current rating. It includes track sizing charts for vacuum or space environment, for several PCB thicknesses and for several copper plane thicknesses in SI (metric) and Imperial (inch) units. This standard allows to accurately calculate the current rating of a track from cross sectional area and the temperature rising of the track. However no derating are applied to take into account the complexity of the PCB.

The model presented in this clause is based on a curve fitting performed on Figure 5-14 of chapter 5.2.2. of IPC-2152.

- c. The model is valid for a range up to 30 A and up to 100°C temperature increment with respect to an initial PCB temperature of 25°C when not powered.
- d. Current rating and cross sectional area are calculated as per the formula in clause D.1.2 with the following constants:
 1. $k_0 = 0,0756$
 2. $k_1 = 0,4375$
 3. $m_0 = 0,5000$
 4. $m_1 = 0,0301$
- e. Example charts based on this model are shown in Figure D-2 to Figure D-7.

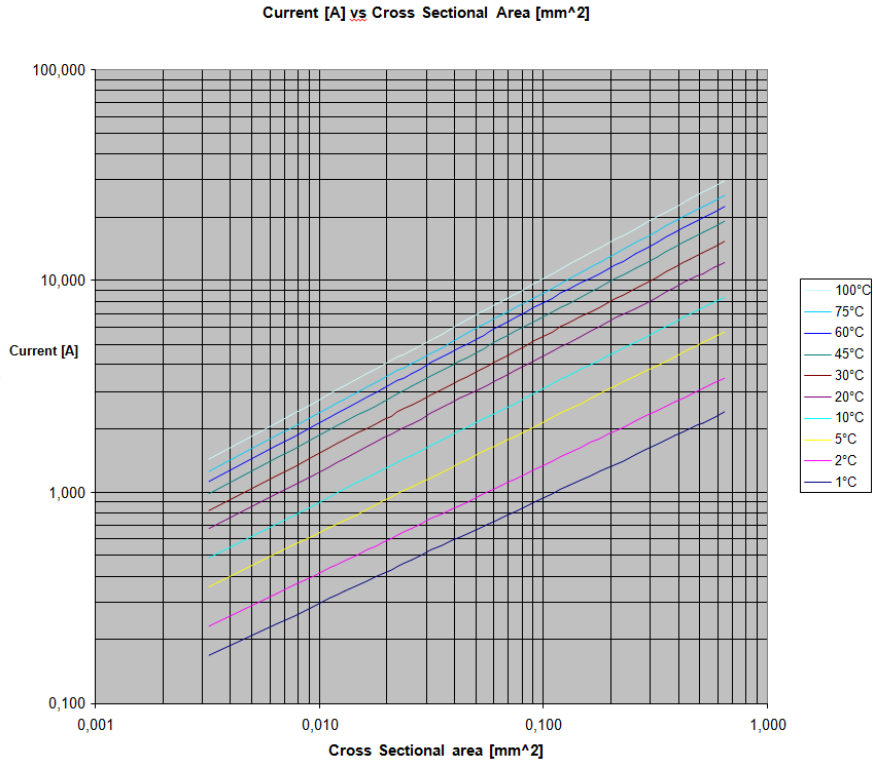


Figure D-2: IPC-2152: Current rating [A] vs cross sectional area [mm²] in double log scale

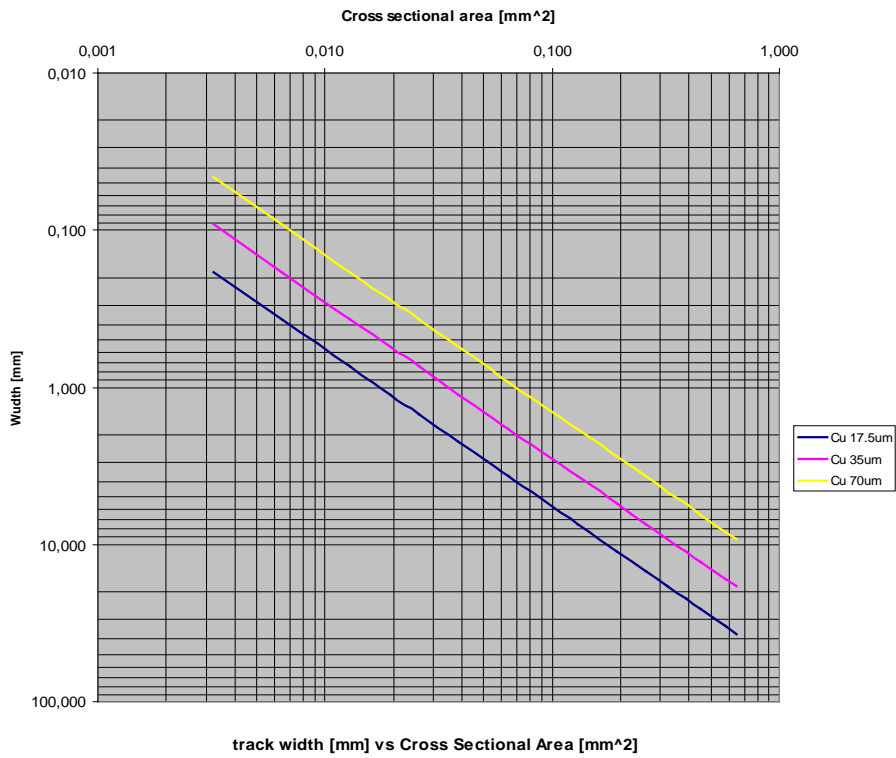


Figure D-3: IPC-2152: Track width [mm] vs cross sectional area [mm²]

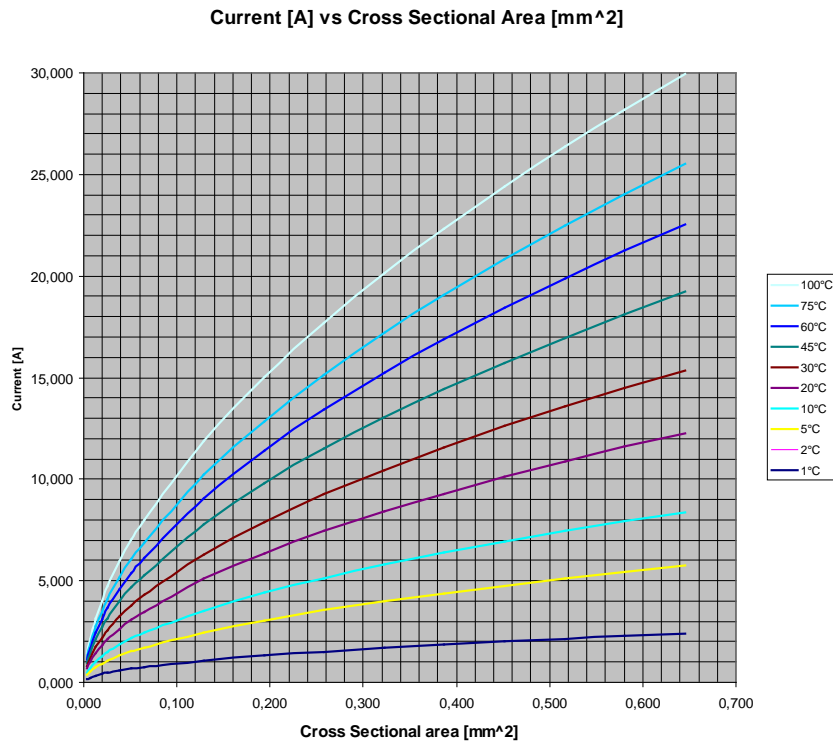


Figure D-4: IPC-2152: Current rating based on Figure D-2, range 0-25 A

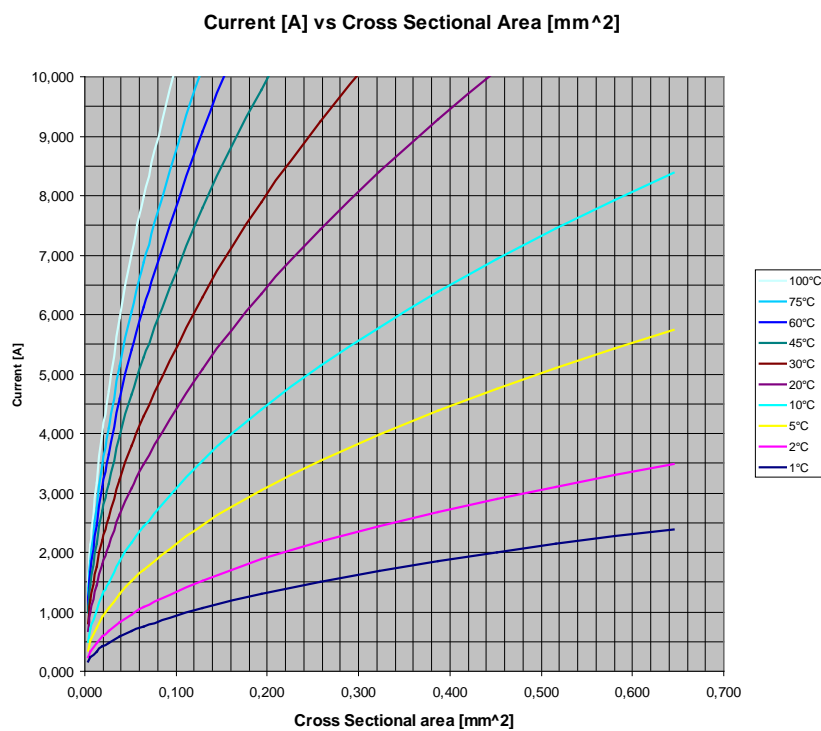


Figure D-5: IPC-2152: Current rating based on Figure D-2, range 0-10 A

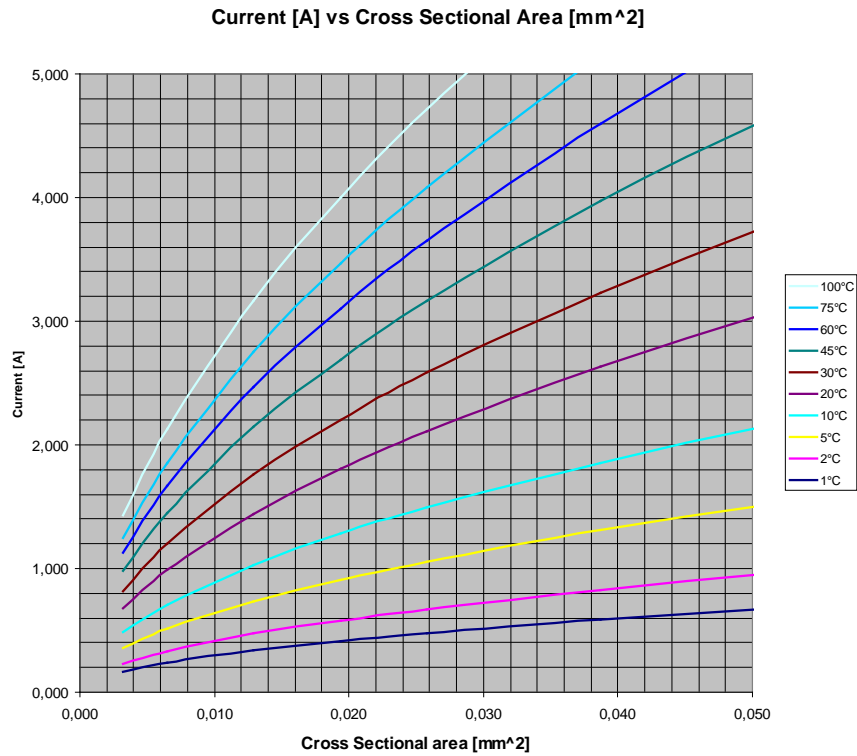


Figure D-6: IPC-2152: Current rating based on Figure D-2, range 0-5 A

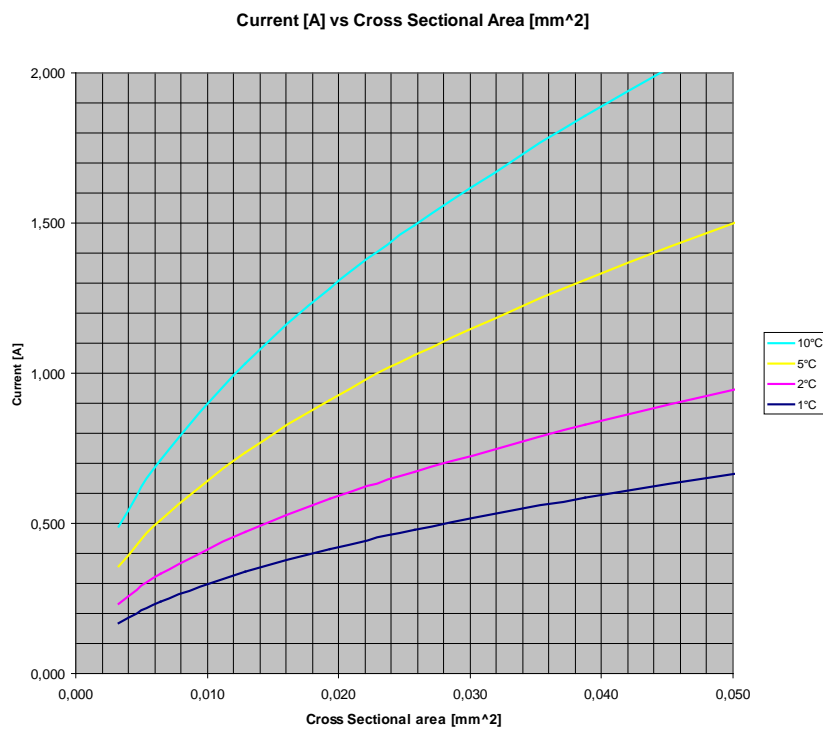


Figure D-7: IPC-2152: Current rating based on Figure D-2, range 0-2 A

D.3 Track current rating computation based on CNES/QFT/IN.0113

The CNES current rating computation is based on the thermal model correlated to experimental data. The values obtained for current carrying capacity are about 10% more conservative with reference to those of IPC-2152.

The model does not apply derating to take into account the complexity of the PCB.

The model presented in this clause is based on a curve fitting performed on the data given in the table "Tableau de quelques valeurs particulieres, Intensite=f(largeur, ΔT)" on page 10 of CNES/QFT/IN.0113".

- a. The model is derived from experimental data ranging up to 8 A and up to 100°C temperature increment with respect to an initial PCB temperature of 25°C when not powered.
- b. Current rating and cross sectional area are calculated as per the formula in clause D.1.2 with the following constants:
 1. $k_0 = 0,0594$
 2. $k_1 = 0,4800$
 3. $m_0 = 0,5420$
 4. $m_1 = 0,0034$
- c. Example charts based on this model are shown in Figure D-8 to Figure D-13.

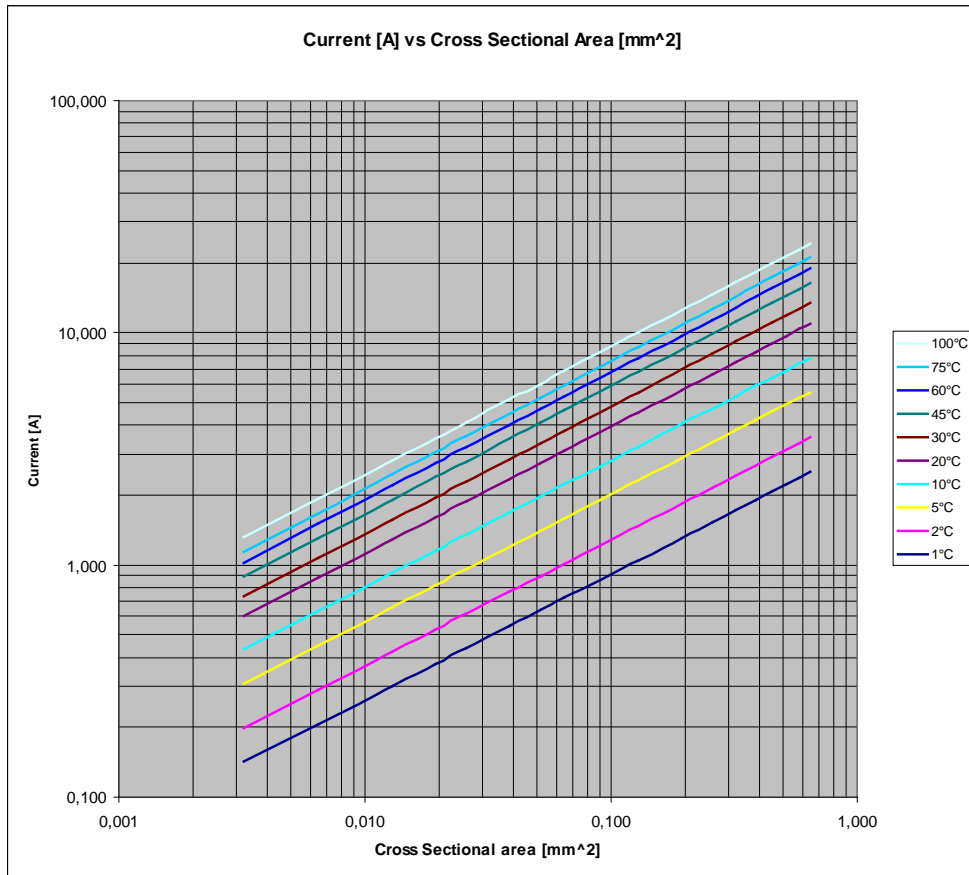


Figure D-8: CNES/QFT/IN.0113: Current rating [A] vs cross sectional area [mm²] in double log scale



Figure D-9: CNES/QFT/IN.0113: Track width [mm] vs cross sectional area [mm²]

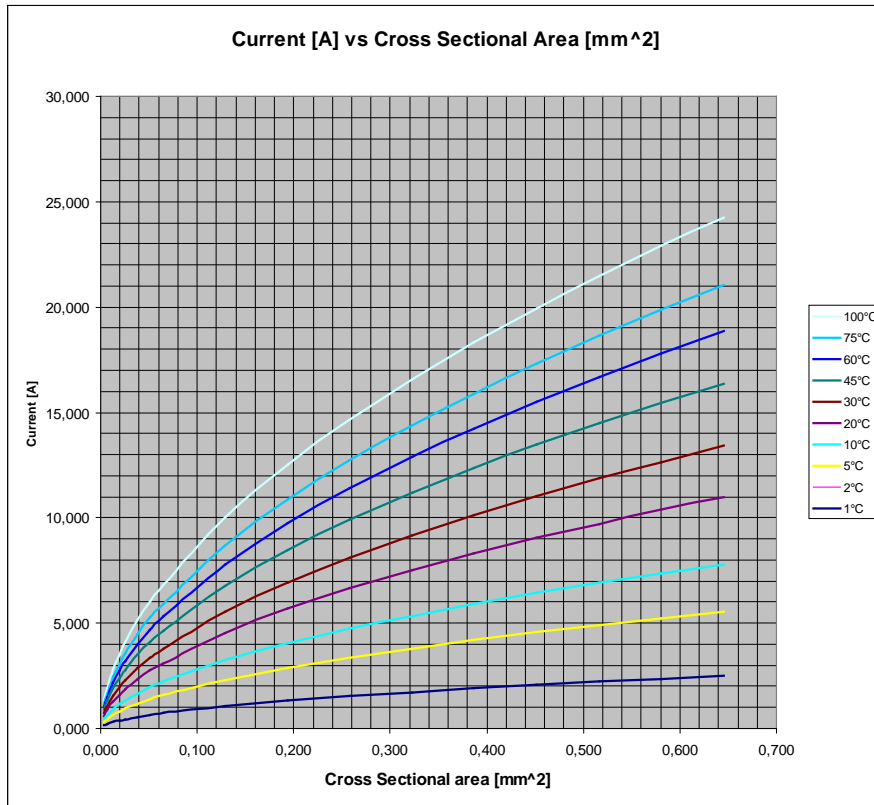


Figure D-10: CNES/QFT/IN.0113: Current rating based on Figure D-8, range 0-25 A

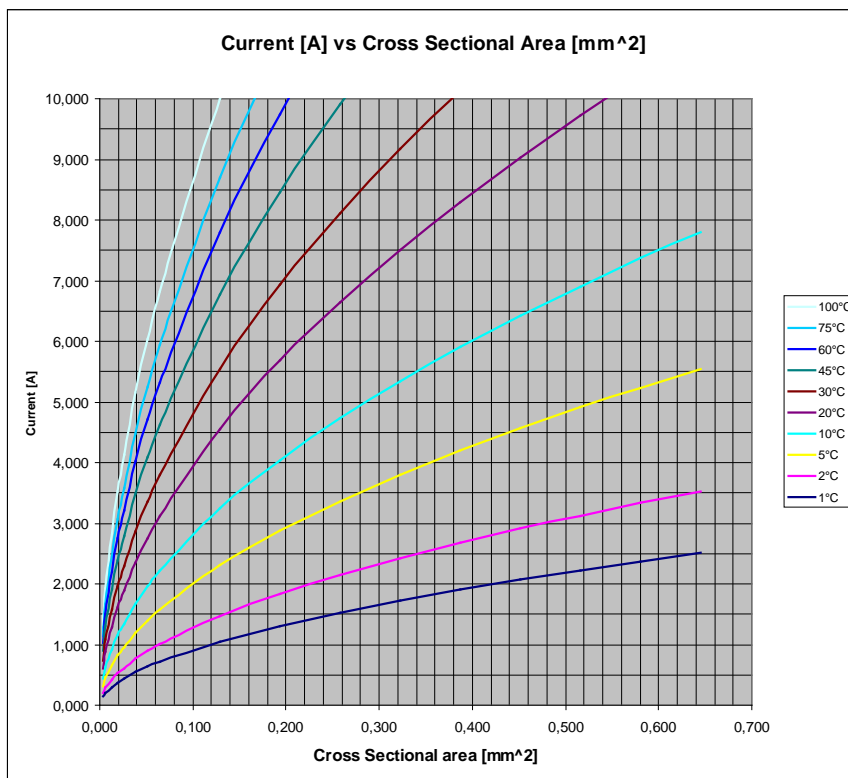


Figure D-11: CNES/QFT/IN.0113: Current rating based on Figure D-8, range 0-10 A

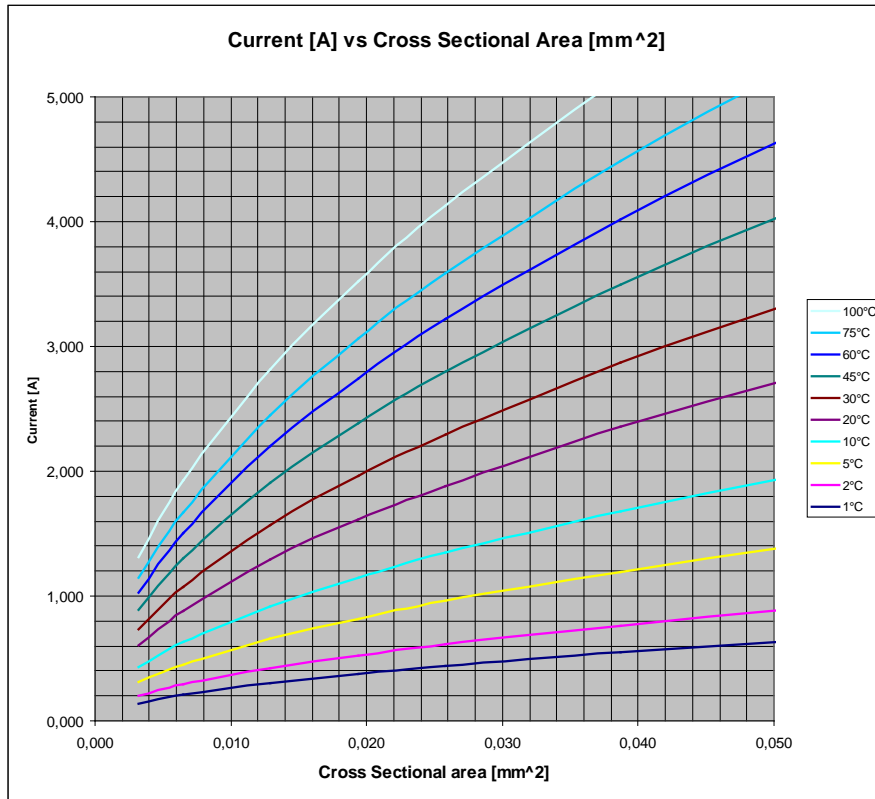


Figure D-12: CNES/QFT/IN.0113: Current rating based on Figure D-8, range 0-5 A

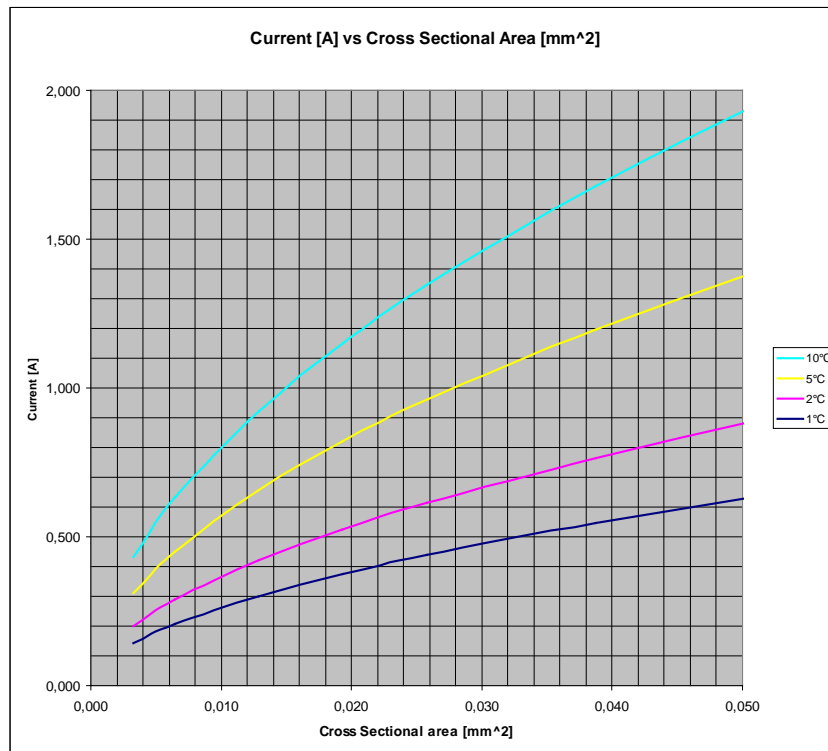


Figure D-13: CNES/QFT/IN.0113: Current rating based on Figure D-8, range 0-2 A

D.4 Track current rating computation based on IPC-2221A

The model presented in this clause is based on a curve fitting performed on figure C for internal conductors of Figure 6-4 of IPC-2221A.

IPC-2221A is the most conservative method of the three methods defined Annex D. Derating is not used in this model.

a. The current rating is calculated with the following conditions:

1. $I < 0,7A$ for Trise 1 °C
2. $I < 0,9A$ for Trise 2 °C
3. $I < 1,0A$ for Trise 5 °C
4. $I < 1,2A$ for Trise 10 °C

Within these conditions the current rating is about 70% of the current rating calculated with IPC-2152.

b. Current rating and cross sectional area are calculated as per the formula in the clause D.1.2 with the following constants:

1. $k_0 = 0,0240$
2. $k_1 = 0,4393$
3. $m_0 = 0,7252$
4. $m_1 = 0,0002$

c. Example charts based on this model are shown in Figure D-14 to Figure D-19.

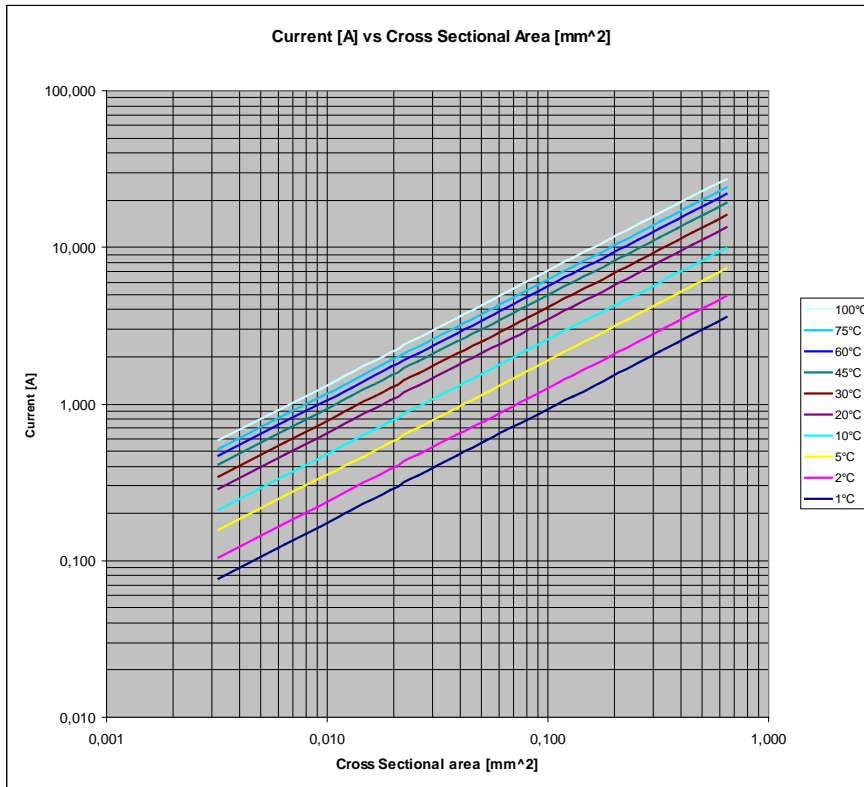


Figure D-14: IPC-2221A: Current rating [A] vs cross sectional area [mm²] in double log scale

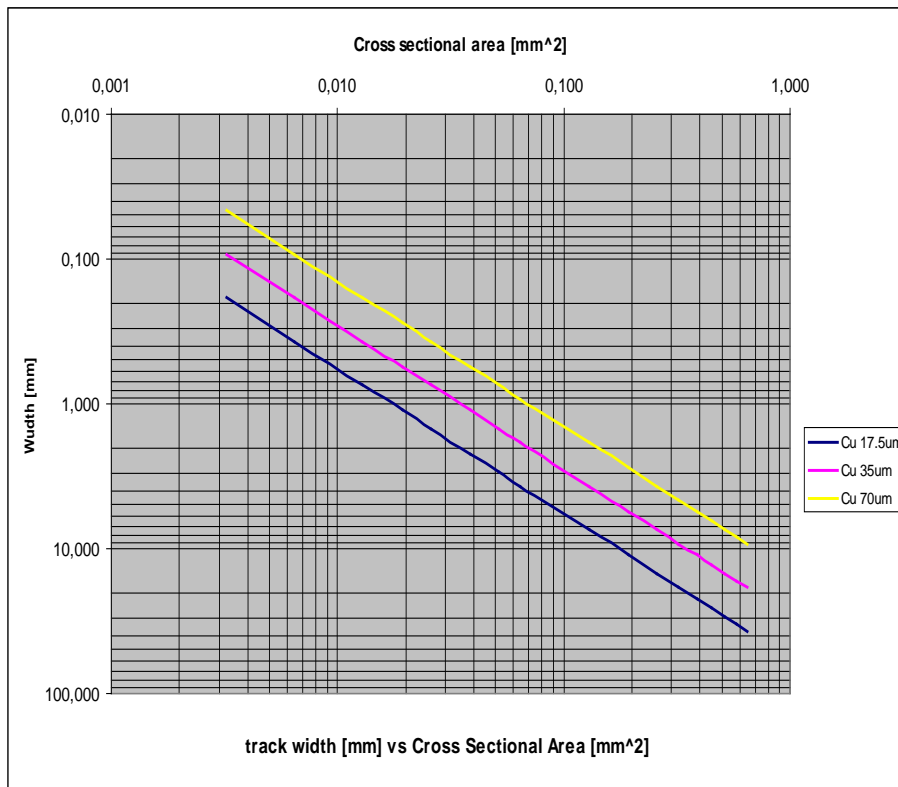


Figure D-15: IPC-2221A: Track width [mm] vs cross sectional area [mm²]

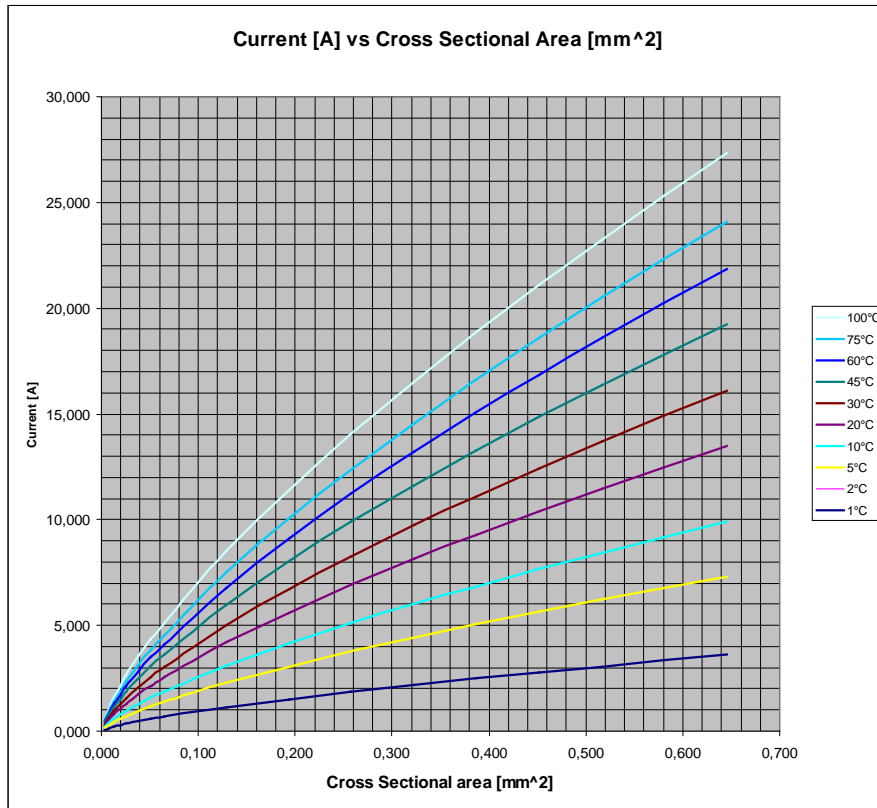


Figure D-16: IPC-2221A: Current rating based on Figure D-14, range 0-25 A

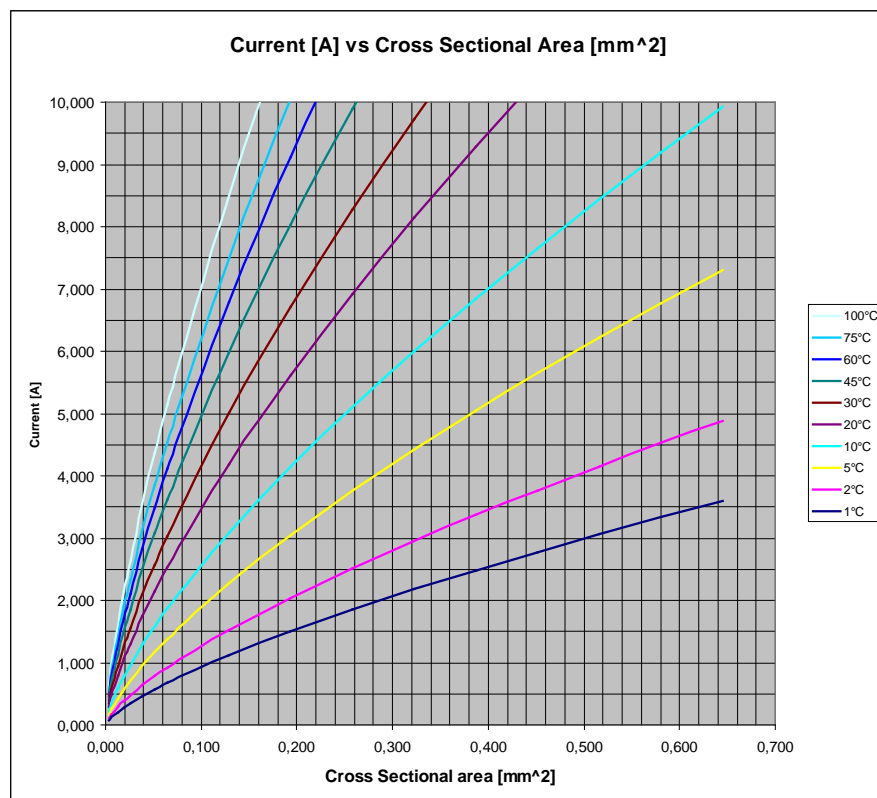


Figure D-17: IPC-2221A: Current rating based on Figure D-14, range 0-10 A

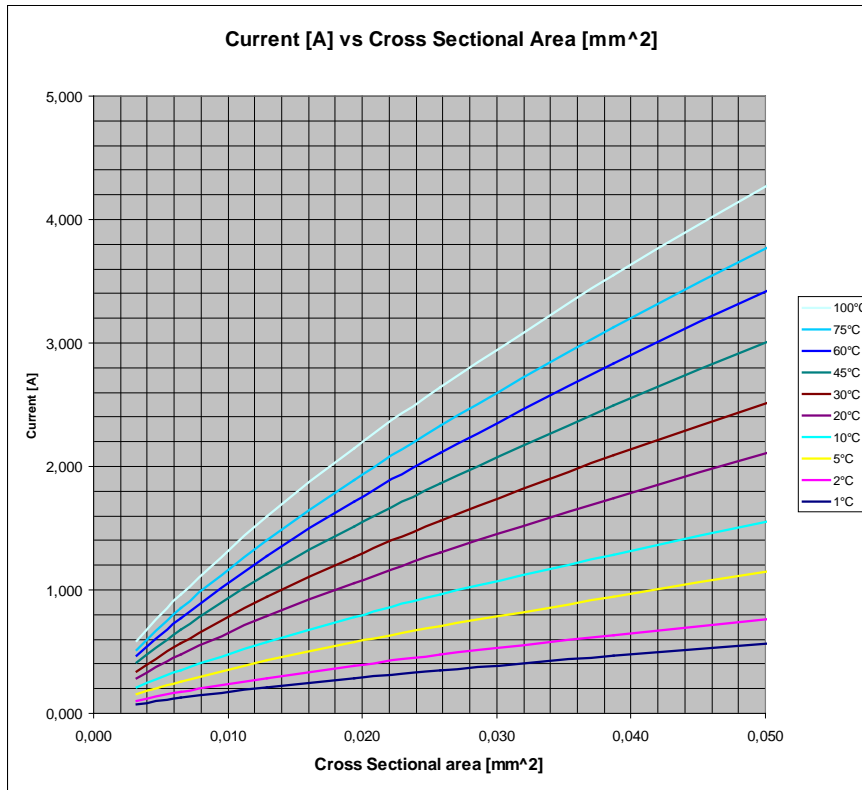


Figure D-18: IPC-2221A: Current rating based on Figure D-14, range 0-5 A

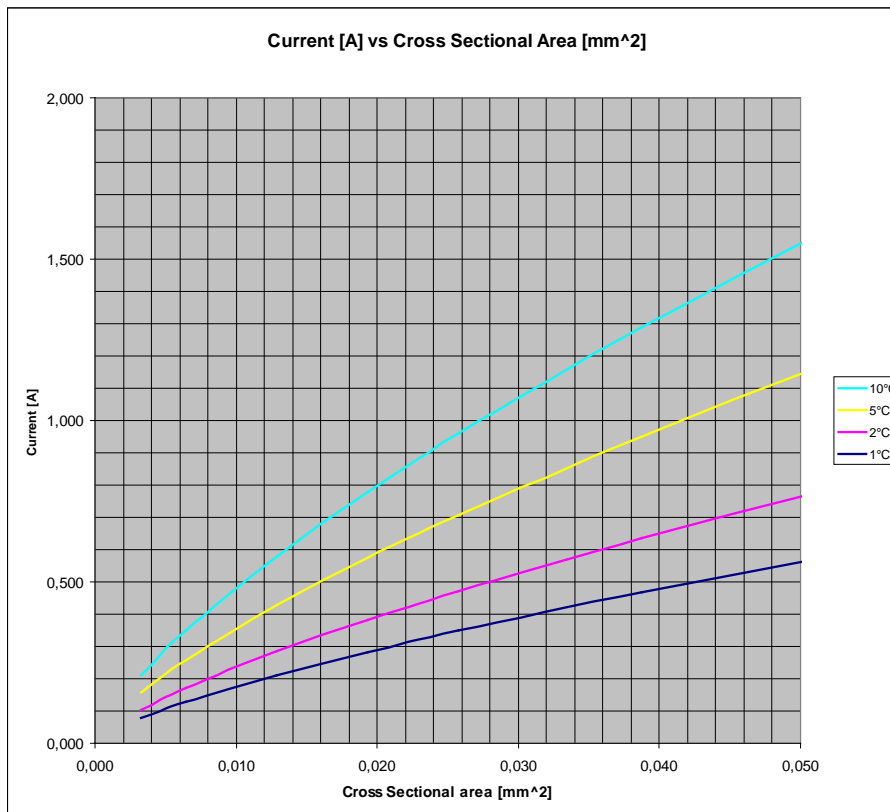


Figure D-19: IPC-2221A: Current rating based on Figure D-14, range 0-2 A

Annex E (informative)

Example of calculation of PTH pad dimensions

Two examples of calculations of PTH pad dimensions are shown below. These calculations shown in Figure E-1 and Figure E-2 are specific for the PCB manufacturer as specified in the PID and depend on the complexity of the PCB designs.

PCB manufacturing tolerances for registration and annular ring



Panel dimension (mm) : 600 x 750 ▼

Base Copper : 18µm

Min AR (calculated values) to be compliant with:		ECSS Cs	ECSS Ss
Inner layer buried via	A	150	
End layer buried via	B	110	
Inner layer buried via	C	205	
End layer buried via	D	205	
End layer buried via	E	110	
Inner layer PTH	F	205	
µvia bot	H	90	
External layer PTH	Y	150	250
External µvia	Z	100	
Ø drilled µvia		150	

Cs = Component Side

Ss = Solder Side

Ø pad mini = 2 x AR + Ø drilled hole

Ø drilled hole = Ø finished hole + 150µm (PTH: -0,05/+0,1)

Ø drilled hole = Ø finished hole + 100µm (via)

with respect to aspect ratio (e / Ø < 6 as standard or e / Ø < 8 for prototype)

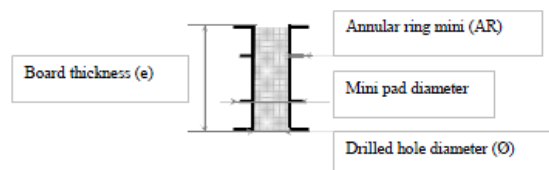
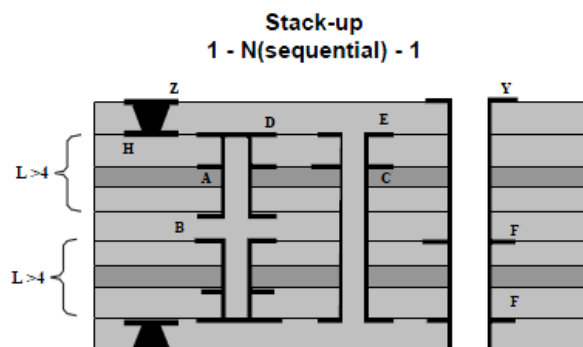


Figure E-1: PCB manufacturing tolerances for registration and annular ring for HDI PCB

PCB manufacturing tolerances for registration and annular ring

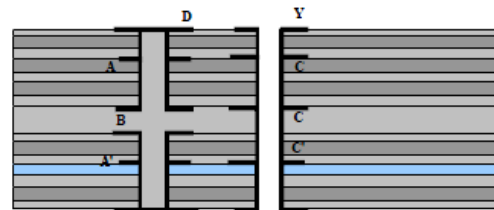


Panel dimension (mm) : 304 x 458 ▼

Base Copper : 35µm

Min AR (calculated values) to be compliant with:		ECSS Cs	ECSS Ss
Inner layer blind via	A	150	
Inner flex layer blind via	A'	210	
End layer blind via	B	110	
Inner layer PTH	C	205	
Inner flex layer PTH	C'	265	
External layer PTH	Y	150	250

Stack-up
Sequential Flex-Rigid



Cs = Component Side
Ss = Solder Side

$\text{Ø pad mini} = 2 \times \text{AR} + \text{Ø drilled hole}$
 $\text{Ø drilled hole} = \text{Ø finished hole} + 150\mu\text{m (PTH: -0,05/+0,1)}$
 $\text{Ø drilled hole} = \text{Ø finished hole} + 100\mu\text{m (via)}$
 with respect to aspect ratio ($e / \text{Ø} < 6$ as standard or $e / \text{Ø} < 8$ for prototype)

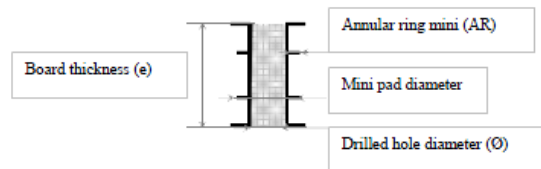


Figure E-2: PCB manufacturing tolerances for registration and annular ring for rigid-flex sequential PCB

Annex F (informative)

Prevention of resin starvation and cracks

F.1 Prevention of resin cracks.

Under some conditions, cracks can develop in the laminate resin. They are more likely to appear:

- a. In brittle resin types;
- b. In locations with only resin and no reinforcement, such as openings in thick ground planes;
- c. If the resin is mechanically stressed by drilling or routing.

The cause for the cracking can be mechanical impact or volume shrinkage during curing of the prepreg. Once a crack is formed, it can propagate into the laminate. The board manufacturing process can influence the formation of cracks. Design precautions to reduce the risk of crack formation include the following measures:

- a. Avoid large volumes of resin without prepreg reinforcement.

NOTE Heat sinks often include large volumes of resin that can increase the risk of cracks. To avoid this, prepreg inserts in openings can be used.

- b. Avoid low pressure areas due to opposite openings in superimposed copper layers.
- c. Keep non-functional pads in planes.
- d. Review the selection of material with PCB manufacturer.

NOTE Prepreg with filled resin can provide improved mechanical properties.

F.2 Prevention of resin starvation

Resin starvation is defined as a condition of insufficient resin to completely wet the glass reinforcement and fill the space between fibres. To reduce the risk for resin starvation the following can be done:

- a. Allow sufficient prepreg thickness to fill the space between the copper pattern. High profile copper features throughout the PCB layers, such as

a pad stack or superimposed tracks, can cause high pressure during lamination and can squeeze resin locally out of the prepreg if insufficient resin is available. The PCB manufacturer can implement additional prepreg between 70 μm copper layers compared to 35 μm copper layers.

Removal of non-functional pads can reduce the risk of resin starvation (see requirement 7.5.1b.2).

- b. Fill open areas in thick copper layers with a non-functional-copper pattern to reduce the amount of resin required for filling. This can also help improve the thickness uniformity of the board.

See IPC-2222A clause 9.1.4 which mentions resin starvation and cracks in a discussion of removal of non-functional pads.

Annex G (informative) Example of MRR checklist

The following is an example of “MRR checklist” document.

Supplier Specification:			
<i>(To be completed by supplier procurement with additional information where applicable from the responsible design authority.)</i>			
PCB Manufacturer		Project	
Supplier Article or Reference No:		Issue/revision	
Article title:		Issue/revision	
Supplier purchase order No:		purchase order Date	
Feature Set			
Is this a new or a recurring design?			
If recurring please define all changes made to data set since the last purchase or state none.			
If recurring please reference ALL occurrence reports raised against previous procurements of this or state none.			
Is the design fully compliant with ECSS-Q-ST-70-12?			
If the design is not compliant to ECSS-Q-ST-70-12 please list all deviations.			
Has method and position of identification marking been defined?			

Printed Circuit Board Manufacturer Declaration:			
<i>(To be completed by the PCB manufacturer)</i>			
Supplier work order No:		work order date	
If this is a recurring design please define all changes made to tooling or processing since the last purchase or state none.			

Manufacturer's PID (Process Identification Document):	
Approved PID Reference including issue and revision	
Deviations to the Approved PID List all modifications changes and nonconformances relative to the approved PID or state none	

Contract Review:	
Please confirm contract review has taken place and reference controlling procedure(s) used.	
Please list any anomalies or deviations arising from the contract review or state none.	

Design:			
Have any changes to the provided data been requested following design rule checking (DRC)?			
Please list all changes and provide references below			
Reference of requesting e-mail (date, from, to)	Reference of supplier's agreement (date, from, to)	Comments	Actions arising
Have all non-functional pads been kept?			
Has coupon sample data been included in the data pack?			
Has IST coupon sample data been included in the data pack?			
Has the position and quantity of coupon sample been agreed with supplier?			

Technical Review:	
<i>(Where there is the risk of deviation to the requirements of ECSS-Q-ST-70-11 or a risk of an identified undesirable artefact occurring this is recorded at the end of this section)</i>	
Please advise the base laminate and prepreg references to be used in fulfilment of this order.	
Is there an extended lead time in procuring these materials?	
Has this combination of materials been utilised on supplier's flight product previously?	
Is the stack-up of materials symmetrical for each pressing cycle?	
How many press cycles lamination – drilling cycles will be required in the manufacture of this design?	
Is the copper weight even on both sides of the laminate?	

Are the copper weights with the range previously supplied for supplier's flight products?		
Are there two plies of glass present between every conductive layer – laminate and prepreg?		
Where prepreg is intended to be used to backfill vias, is the resin content and board geometry suitable to support this?		
For flex rigid only, is the process open or closed window?		
How many flexible laminate layers are included in the design?		
Risks identified from the technical review	Risk Rating (1 Low, 5 High)	Risk Parameters requiring control

Additional risks:		
Have any additional risks to the manufacture of this product been identified		
Additional risk identified	Risk Rating (1 Low, 5 High)	Risk Parameters requiring control

Manufacturer Completing Signature:	
<i>Please ensure that copies of the PCB construction and a screen dump of the approved panel layout are attached to this MRR before signature.</i>	
Position	Print Name; Sign and Date

Supplier Technical Authority review:	
Is the supplier technical authority in agreement with the risks identified and the rating applied by the manufacturer?	
If no provide details of dis-agreement.	
Have any additional risks been identified by the TA as a result of this review?	
If yes give details.	

Supplier Technical Authority Recommendations:	
Recommendation	Print Name, Sign and Date
Low to intermediate levels of risk 1 - 2, manufacture may continue without project counter-signature.	
Medium levels of risk 3 - 4, risks to be entered in project risk register. Project manager responsible countersignature required to confirm acceptance and recording of risk before manufacture may commence.	
High risk 5, continuation against recommendation of supplier technical authority. To be subject IRB with inclusion in project risk register. Project manager responsible countersignature required to confirm acceptance and recording of risk before manufacture may commence.	

Authorising Signatures:	
Entity	Print Name, Sign and Date
Supplier Technical Authority	
Supplier project manager (medium and high risk only)	

Bibliography

ECSS-S-ST-00	ECSS system – Description, implementation and general requirements
ECSS-Q-ST-70-28	Space product assurance - Repair and modification of printed circuits board assemblies for space use
ASTM-B386-03, 2011	Standard Specification for Molybdenum and Molybdenum Alloy Plate, Sheet, Strip, and Foil
IPC-2141, Mar 2004	Design Guide for High Speed Controlled Impedance Circuit Boards
IPC-2221A, May 2003	Generic standard on printed board design
IPC-2221B, Nov 2012	Generic standard on printed board design
IPC-2222A, Dec 2010	Sectional design standard for rigid organic printed boards
IPC-2223C, Nov 2011	Sectional design standard for flexible printed boards
IPC-2226, Apr 2003	Sectional design standard for high density interconnect boards
IPC-2251, Nov 2003	Design guide for the packaging of high speed electronic
IPC-2315, Jun 2000	Design guide for high density interconnects (HDI) and microvias
IPC-4104, May 1999	Specification for high density interconnect (HDI) and microvias materials
IPC-4121, Jan 2000	Guidelines for selecting core constructions for multilayer printed wiring board applications
IPC-4202A, Apr 2010	Flexible base dielectrics for use in flexible printed circuitry
IPC-4204A, Oct 2013	Flexible metal clad dielectrics for use in fabrication of flexible printed circuitry
IPC-7095C, Jan 2013	Design and assembly process implementation for BGAs
IPC-7351B, Jun 2010	Land pattern and calculator
IPC-CF-152B, Dec 1997	Composite Metallic Material Specification for Printed Wiring Boards
IPC-T-50K, 2013	Terms and definitions for Interconnecting and packaging electronic circuits
RNC-CNES-Q-ST-70-101, version 8, Apr 2009	Spécifications de conception des cartes imprimées
CNES/QFT/IN.0113 rev 1, issue June 2000	Intensités admissibles en environnement spatial
D Cullen, G O'Brien, Proc. APEX 2004	“Implementation of Immersion Silver PCB Surface Finish In Compliance With Underwriters Laboratories”

NOTE See also Clause 2 for referenced normative IPC standards.