

1	SCOPE	3
1.1	Scope	3
1.2	Purpose	3
1.3	Type.....	3
1.4	Notation	3
2	APPLICABLE DOCUMENTS.....	3
2.1	STANDARDS	3
2.2	Order of precedence.....	3
3	REQUIREMENTS	4
3.1	General Requirements	4
3.2	Concessions	4
3.3	Qualification of Printed Board Manufacturer	4
3.4	Performance Specification.....	4
3.4.1	Defects Not Allowed	4
3.5	Documentation Requirements	4
3.6	Material Requirements.....	4
3.6.1	Material Handling and Storage.....	4
3.6.2	Metal-clad Laminates.....	4
3.6.3	Bonding Material	4
3.6.4	Copper Foil	5
3.6.5	Tented Vias	5
3.7	Registration (Internal)	5
3.8	Laminate Integrity.....	5
3.9	Bow and Twist	5
3.10	Conductors and Conductive Surfaces	5
3.11	Solder Resist.....	5
3.12	Surface Finish	5
3.13	Hole Size Tolerances	5
3.14	Tooling Hole Positions	5
3.15	Hole Plating	5
3.16	Marking	6
3.16.1	In copper identification.....	6
3.16.2	Silk screened identification.....	6
3.17	Supplied Data.....	6
3.17.1	Data Format.....	6

3.17.2	ODB++	6
3.17.3	RS-275/Excellon Data.	6

1 SCOPE

1.1 Scope

This specification establishes the requirements for rigid printed wiring boards. The boards may be single sided, double sided with or without plated through holes, multilayer and metal cored boards. These requirements are generally in accordance with IPC-6011 and IPC-6012 with additional construction inspection and product assurance requirements specified.

1.2 Purpose

This specification provides general requirements and quality assurance provisions for rigid printed wiring boards (PWBs) for use in all high reliability products.

1.3 Type

Printed Boards shall be of the following types:

Type 1 – Single sided board

Type 2 – Double sided board

Type 3 – Multilayer board without blind and/or buried vias

Type 4 – Multilayer board with blind and/or buried vias

Type 5 – Multilayer metal core without blind and/or buried vias

Type 6 – Multilayer metal core with blind and/or buried vias

1.4 Notation

Where requirements deviate from IPC-6011 and -6012 italic font *thus* is used.

2 APPLICABLE DOCUMENTS

2.1 STANDARDS

The following specifications and standards form a part of this specification to the extent specified herein. The applicable issues of these documents shall be that or those in effect by the responsible industry association on the date of the procurement document.

IPC – Association connecting electronics industries

IPC-6011 *Generic Performance Specification for Printed Boards*

IPC-6012 *Qualification and Performance Specification for Rigid Printed Boards*

IPC-4101 *Specification for Base Material for Rigid and Multilayer Printed Boards*

IPC-4552 *Specification for Electroless Nickel/Immersion Gold (ENIG) Plating for Printed Boards*

IPC-2221A *Generic Standard on Printed Boards Design*

IPC-7351 *Generic Requirements for Surface Mount Design and Land Pattern Standard*

IPC-SM-840D *Qualification and Performance Specification of Permanent Solder Mask*

IPC-A-610 *Generic Standard on Printed Boards Design*

IPC-4562 *Metal Foil for Printed Wiring Applications*

2.2 Order of precedence

In the event of a conflict between the procurement document(s), text of this specification, or other document(s) referenced herein, the following order of precedence shall apply:

1. Contract or purchase order,
2. Printed Board Master Drawing,
3. This Specification,
4. IPC-6011 and IPC-6012,
5. Other document referenced herein.

3 REQUIREMENTS

3.1 General Requirements

Printed boards furnished to this specification shall conform to the requirements of the master drawing, this specification and IPC-6011 and IPC-6012. The detailed requirements contained within this section, although determined by examining specific quality conformance test circuitry (test coupons) and sample printed boards, shall apply to all coupons, sample boards and delivered printed boards. Defects or anomalies noted on coupons or sample boards shall be recorded and the proper corrective actions shall be initiated to eliminate the defects or anomalies.

3.2 Concessions

Concessions (deviations and waivers) to the contract or purchase order, Printed Board Master Drawing or this specification shall be submitted to the procuring activity for approval. A written concession shall be obtained prior to delivering product.

3.3 Qualification of Printed Board Manufacturer

Manufacturers' must be IPC and ISO 9000 accredited.

3.4 Performance Specification

All printed boards shall be procured in accordance with IPC-6011, IPC-6012 and this specification. Procured boards shall be Class 3 High Reliability Electronics Products, per IPC-6011 paragraph 1.2.

3.4.1 Defects Not Allowed

The following defect(s) permitted under IPC-6012 *are not allowed* for products under this specification.

3.4.1.1 Exposed Weave

Exposed weave *shall not be allowed* in as-received boards.

3.5 Documentation Requirements

Documentation supplied to the printed board manufacturer shall be in accordance with IPC-2610 Documentation Requirements for Printed Boards. The documentation shall be Class 3 Full Documentation as per paragraph 1.2 of the specification.

3.6 Material Requirements

Material requirements shall be as specified in the master drawing, this specification and IPC-6012 paragraphs 3.2.1 through 3.2.13

3.6.1 Material Handling and Storage

Material handling and storage shall be in accordance with IPC-4562 and any applicable manufacturers' data sheets. Raw material storage and handling shall be controlled to insure and prolong shelf life.

3.6.2 Metal-clad Laminates

Unless otherwise specified, metal-clad laminates for Type 1 and 2 printed boards and individual layers for type 3 through Type 6 shall be in accordance with IPC-4101/24 (NEMA FR4, MIL-S-13949/04 GFR/GFG) and have a glass transition temperature (T_g) above 150°C. Other materials should not be used without prior approval.

3.6.3 Bonding Material

The inner-layer bonding material for Type 3 through Type 6 printed boards shall be pre-impregnated and of the same type as the base laminate above.

3.6.4 Copper Foil

Unless otherwise specified the copper foil shall be class 3 in accordance with IPC-4562. The thickness shall be as specified in the master drawing.

3.6.5 Tented Vias

The tenting of vias with solder mask is not allowed.

3.7 Registration (Internal)

Registration shall be as described in paragraph 3.4.2 of IPC-6012, except that the minimum external and internal annular ring shall be as described in Table 3.5, class 3 of the above specification.

3.8 Laminate Integrity

Laminate integrity shall be as described in paragraph 3.6.2.3 of IPC-6012 for class 2 and 3 products.

3.9 Bow and Twist

Unless otherwise specified on the Printed Board Master Drawing bow and twist shall be as described in paragraph 3.4.4 of IPC-6012.

3.10 Conductors and Conductive Surfaces

Unless otherwise stated on the Printed Board Master Drawing the definition, width, thickness, spacing and imperfections of conductors and conductive surfaces shall be in accordance with paragraphs 3.5 through 3.5.4.2 of IPC-6012.

3.11 Solder Resist

Solder resist (solder mask) is required on both external faces of the printed board, it shall meet the qualification/conformance IPC-SM-840 class H. Coverage, cure and adhesion shall be as defined in paragraphs 3.8.1 to 3.8.3 of IPC-6012, *except* that no encroachment of solder resist is allowed on any surface mount or ball grid lands, **and that ALL pad patterns have solder resist slivers between individual pads. The height of the solder resist should not cause any mounting problems for surface mount components.**

Solder resist data is provided as per IPC-7351 standard, 1:1 with the land size, the manufacturer is to oversize these solder resist openings commensurate with their manufacturing procedures ensuring that *ALL* the above requirements are met, the amount of oversize to take into account the minimum track and gap dimensions as shown on the Printed Boards Master Drawing. Solder resist not related to a component pad is not to be enlarged.

3.12 Surface Finish

Surface finish shall be Electroless Nickel/Immersion Gold (ENIG) as per IPC-4552.

3.13 Hole Size Tolerances

Hole size tolerances for all plated and non-plated shall be as shown below.

< 1.00mm +/-0.05mm

>1.00mm <2.50mm +/-0.10mm

>2.50mm +/-0.15

3.14 Tooling Hole Positions

Hole centres shall be located within 0.05mm of the location shown on the master pattern.

3.15 Hole Plating

All plated holes shall be plated through to 26µm (.001") minimum thickness.

3.16 Thieving

Supplier may add thieving to the panel external to any PCB pattern, allowing at least 3mm clearance from any board outlines.

3.17 Marking

3.17.1 In copper identification.

Boards shall carry the nomenclature and the issue status of the artmasters in each layer of copper.

The panel border shall also carry the panel reference number and issue, the board reference number and any other text or detail as contained in the electronic data for the relevant layer.

3.17.2 Silk screened identification.

Boards shall be silk screen marked using white marking ink to specification A-A-56032.

Boards shall be marked, next to the nomenclature on the top layer, with the issue status of the Printed Wiring Board drawing.

Boards shall be marked, in a specified location, with the manufacturer's cage code and the week and year of manufacture, as per MIL-STD-130. The general location and layer will be shown on the documentation layer "Board Drawing".

3.18 Supplied Data.

3.18.1 Data Format.

The preferred format for all manufacturing data shall be ODB++ Version 7, supplied in the compressed format, with EDA data removed and Nets neutralised.

Where it is not possible to supply ODB++ format data, a data pack will be provided consisting of

RS-275X artmaster data

Excellon II drill data

IPC-356 Netlist data

In both cases a Printed Wiring Board drawing shall be supplied in PDF format, this will have basic notes and any exceptions from this document.

3.18.2 ODB++ .

As well as the standard artwork layer data the following layers and information shall be observed when engineering the PCB data:-

'Board Drawing' – basic board dimensions and stack up, and locations for additional manufacturing data.

'Panel Drawing' – the overall panel dimensions. Step and repeat dimension for the board pattern within the panel.

'Routing PCB Manufacturer' – any pre routing required during the manufacturing stage. This will be shown as centre justified tool paths for each size of tool required. All tool compensation has been performed, the route data to be created from the relevant tool path centre lines.

Any other documentation layers are an artefact of the ODB++ translation from the CAD data and should be ignored.

NB A layer "rout" is created automatically during the CAD > ODB++ translation, this layer is to be IGNORED, it is automatically created and its creation cannot be suppressed.

3.18.3 RS-275/Excellon Data.

The file names for each artmaster layer, subsidiary layer, drill files and ancillary data will be listed on the Printed Wiring Board Drawing supplied with the data pack.

