IPC-DR-DES-2022

PCB Design Desk Reference

2022 Edition

Developed by the 1-15 PCB Design Desk Reference Subcommittee of the 1-10 Board Design Committee IPC

Users of this publication are encouraged to participate in the development of future revisions.

Contact:

IPC

Tel 847 615.7100
Fax 847 615.7105

Table of Contents

FORWARD 1

1 INTRODUCTION 1

2 APPLICABLE DOCUMENTS 1

2.1 IPC 1

2.2 Other Standards Development Organizations 2

3 DESIGN OPTIMIZATION 2

3.1 IPC Producibility Levels. 2

3.1.1 Use a Higher IPC Producibility Level. 2

3.1.2 Use a More Advanced Technology 3

3.2 Production Costs Index 3

3.3 Parameters Hierarchy 3

3.4 Documentation 3

3.4.1 PCB Digital Files (GERBER and Drilling) 3

3.4.2 Master Drawing 3

3.4.3 PCBA Digital Files 4

3.4.4 Assembly Drawing 4

3.4.5 Other Documents. 4

4 MATERIALS SELECTION 4

4.1 Base Materials. 4

4.1.1 Base Materials Assembly 4

4.1.2 Defining Peripheral Materials 4

5 RIGID MULTILAYER STACKUP 7

5.1 Stackup Recommendations for Manufacturability of Rigid Boards 7

5.2 Sequential Stackups 8

5.3 Blind Structures 9

5.4 Combinations of Traditional Stackups, Sequential Stackups, and Blind Structures (HDI) 9

6 PAD-STACK DESIGN 10

6.1 Printed Circuit Board Overall Thickness (a). 11

6.2 Drill Hole Diameter (b) 11

6.3 Plating Thickness (c). 11

6.4 Finished Hole Size (d) 12

6.5 Etchback/Desmear (e) 12

6.6 Internal Pad Diameter (f) and External Pad Diameter (g) 12

6.7 Pad to Plane Clearance (h) . . . 12

6.8 Clearance Area in Planes (Antipad) Diameter (i) 12

6.9 Thermal Relief Web Width (j) 13

7 LASER-DRILLED MICROVIA DESIGN 13

7.1 Dielectric Thickness (J) 14

7.2 Copper Thickness Present at Drilling (***K***). 14

7.3 Microvia Diameter at Capture Pad (***b***) 14

7.4 Target Pad Diameter (C) 14

7.5 Capture Pad Diameter (d) 14

7.6 Microvia Diameter at Target Pad (a) 14

7.7 Plating Thickness (m) 15

8 COMPONENT LAND PATTERN DESIGN 16

8.1 Use of IPC-7351 16

8.2 Footprint Fanout. 17

9 SOLDER MASK AND VIA PROTECTION 18
9.1 Solder Mask (S/M) 19
9.2 Via Protection 19
9.3 S/M Thicknesses 20

10 DIMENSIONING AND TOLERANCING 20
10.1 Recommendations 22

Figures

Figure 5-1 Examples of 6-Layer Stackups. 7
Figure 5-2 Examples of Common Sequential Stackups 8
Figure 5-3 Example of Common Blind Structures 9
Figure 5-4 Examples of Common HDI Stackups 10
Figure 6-1 Pad-stack Parameters for Section 6 10
Figure 7-1 Laser-Drilled Microvia Parameters for Section 7. 13
Figure 7-2 Staggered Microvia Design 15
Figure 7-3 Stacked Microvia Design . 15
Figure 8-2 Center and right figures show the effect of asymmetrical routing on component placement. . . 17
Figure 9-1 Solder Mask and Via Protection Parameters . . . 18
Table 9-1 CAD Parameters for Liquid Photoimageable Ink (LPI) 18
Figure 9-2 Solder Mask (S/M) 19
Figure 9-3 Solder Mask (S/M) 20
Figure 10-1 Key Parameters for Dimensioning and Tolerancing 20

Tables

Table 2-1 Hierarchy of Parameters Effecting Routing Rules and Library Creation 3
Table 4-1 Material Selection for Rigid Boards. 5
Table 4-2 Material Selection for Flex and Flex-Rigid Boards. 5
Table 4-3 Material Selection for Boards. 5
Table 4-4 Materials Selection Due to Assembly and Application Requirements 6
Table 4-5 Materials Selection Due to Other Application Requirements 6
Table 6-1 Pad-stack Parameters Key for Figure 6-1 11
Table 7-1 Laser-Drilled Microvia Parameters Key for Figure 7-1 13
Table 10-1 Key Parameters for Dimensioning and Tolerancing 21

PCB Design Desk Reference 2022 Edition

FORWARD

In 2018, the IPC Designers Council France (IDCF) asked French-speaking designers and CAD engineers about their design practices and especially about references they use to design their products. This survey showed that the interest designers have great interest in IPC standards but also the difficulty they face in using them as they are. Therefore, IDCF proposed to build a document that would guide them in the use of IPC design standards. This proposal was endorsed on the second Design-IPC Day and implemented by the board with the help of many designers and professionals of the trade, in working groups. This document is the collection of this work.

The IDCF board thanks all the participants and facilitators of these working groups who enabled the implementation of this document.

1 INTRODUCTION

This document was created to guide designers in their process and parameters definitions to create patterns in their CAD tools. Recommendations are shown in the order in which the designer is advised to address them.

This document is a guide. While it was created to support design in accordance with IPC standards, it cannot address every use case. Therefore, this document is far from exhaustive and there are many other constraints that may impose design choices different than those required or recommended by IPC standards. However, if these constraints are not identified, this guide will help the designer design a product that will remain Industrial; i.e., one that can be obtained from a significant number of manufacturers without severely impacting performance, reliability, and cost.

Note: Sections, tables, and figures referenced in the document are valid as of the 1st of October 2021.

2 APPLICABLE DOCUMENTS

2.1 IPC

IPC-2581 Generic Requirements for Printed Board Assembly Products Manufacturing Description Data and Transfer Methodology

IPC-T-50 Terms and Definitions for Interconnecting and Packaging Electronic Circuits

IPC-CC-830 Qualification and Performance of Electrical Insulating Compound for Printed Wiring Assemblies

IPC-SM-840 Qualification and Performance Specification of Permanent Solder Mask and Flexible Cover Materials

IPC-1902 Grid System for Printed Circuits

IPC-2221 Generic Standard on Printed Board Design

IPC-2222 Sectional Design Standard for Rigid Organic Printed Boards

IPC-2223 Sectional Design Standard for Flexible Printed Boards

IPC-2226 Sectional Design Standard for High Density Interconnect (HDI) Printed Boards

IPC-2614 Sectional Requirements for Board Fabrication Documentation

IPC-2615 Printed Board Dimensions and Tolerances

IPC-4101 Specification for Base Materials for Rigid and Multilayer Printed Boards

IPC-4103 Specification for Base Materials for High Speed/ High Frequency Applications

IPC-4204 Flexible Metal-Clad Dielectrics for Use in Fabrication of Flexible Printed Boards

IPC-4562 Metal Foil for Printed Wiring Applications

IPC-4761 Design Guide for Protection of Printed Board Via Structures

IPC-6012 Qualification and Performance Specification for Rigid Printed Boards

IPC-6013 Qualification and Performance Specification for Flexible Printed Boards

IPC-7095 Design and Assembly Process Implementation for BGAs

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

2.2 Other Standards Development Organizations

IEC 60097:4.0 Grid System for Printed Circuits

ANSI Z210.1 Metric Practices

3 DESIGN OPTIMIZATION

3.1 IPC Producibility Levels When there is enough space on a board, the designer should use level A patterns, as these protect against industrialization risks while keeping the best cost-to-reliability ratio. When there is not sufficient space to meet Level A, the designer should implement the following approach. (Wherever space remains, patterns should be evenly maximized.)

3.1.1 Use a Higher IPC Producibility Level The IPC Producibility Levels are as follows:

- **Level A:** The level of technology that the vast majority of manufacturers have the ability to manufacture. Level A usually incurs the lowest production costs.
- **Level B:** The median level of technology that roughly half of manufacturers have the ability to manufacture. Level B is a good compromise between cost and technology.
- **Level C:** The highest level of technology that roughly 20% of the most capable manufacturers can industrially achieve. Therefore it is recommended to limit Level C designs to the densest areas (e.g., fine pitch component footprints).

The higher the level, the more expensive your product should be, because it requires increasingly elaborate materials, processes and/or tools. Generally, higher design levels also require a dedicated supply chains, where the adequacy between the desired level and the manufacturer's capability should be stated through qualifications and audits.

It is feasible to use features beyond Level C, but this constitutes a significant risk that needs to be assessed and often requires the use of captive solutions.