

PBA Design-for-Manufacturing Guideline

EDM-D-005 Rigid PCB Build-Up and Density Classification

V1.5
July 2017

Contact

Geert Willems

Phone: +32 16 288962

Mobile: +32 498 919464

Geert.Willems@imec.be

IMEC

Kapeldreef 75

B3001 Heverlee

Verantwoordelijke uitgevers

Luc Van den Hove - IMEC

Copyright © imec 2017 All rights reserved.

Only an authorized person is hereby permitted to view and use this document subject to the following conditions:

1. This document may be used for informational purposes only.
2. Any copy of this document or portion thereof must include the copyright notice.
3. This information is provided "AS IS" and without warranty of any kind, express, implied, statutory, or otherwise.
4. Imec shall not be liable for any actual, direct, indirect, incidental or consequential damages arising out of the use, performance or application of this document.

Permission is not granted for resale or commercial distribution or use of the document, in whole or in part, or by itself or incorporated in another work.

The Design-for-eXcellence Guidelines principles

The PBA Design-for-eXcellence (DfX) Guidelines are designed to provide all electronic supply chain actors involved in the design, qualification, industrialization and production of Printed Board Assemblies practical guidelines to master the multi-disciplinary hardware aspects of electronic module realization and operation in a cost-effective way. The PBA DfX Guidelines are not electrical design guidelines. The PBA DfX guidelines provide the electrical designer the boundary conditions of industrial electronic manufacturing technology and operational reliability. It is intended to support the development of cost-effective, reliable PBA with a short time-to-market requiring a minimum number of design iterations.

Some of the characteristics of the PBA DfX Guidelines are:

- The PBA DfX Guidelines are oriented towards the overall optimization of the physical design of the final PBA based product.
- The guidelines refer to the relevant industry standards that are predominantly used in the international electronics industry such as those published by organizations as IPC and JEDEC. The guidelines do not replace industrial standards but define or recommend what options in the standards to use and will fill-in gaps if necessary. They provide the basis on which a company/product/product-line or application specific approach for design, industrialization and/or realization can be defined.
- Scientific argumentation and physical models form the basis of a large part of the guidelines and of the associated tools. This allows the use of the guidelines beyond the boundary of the users' experience domain. Therefore, it provides a powerful product and process innovation aid.
- The PBA DfX Guidelines will not specify, recommend or exclude specific brands of materials, components, suppliers or products. They will put forward minimal requirements on quality, physical and chemical properties and testing. They define and provide the DfManufacturing window for PBA realization.
- The PBA DfX Guidelines are based on verifiable physical models, standards and empirical data.

PBA DfX Guidelines Scope

- The PBA DfX guidelines cover lead-free SnAgCu and SnPb solder based assembly.
- The PBA DfX guidelines include: Design-for-Manufacturing, Design-for-Assembly, Design-for-Test, Design-for-Reliability, Design-for-RoHS, etc.

Acknowledgement

This document was realised in collaboration with the industrial and academic partners of imec's Center of Electronic Design & Manufacturing and the supply chain spanning Electronic Design & Manufacturing Program, an imec and Sirris initiative.

Funding organisations

IWT is acknowledged for funding the VIS projects - especially the Collective Research project CO-PBA-DfX and the VIS trajectory PROSPERITA - that have provided the scientific background for the PBA DfX Guidelines and gained the necessary industry support.

imec contributors

Geert Willems, Ph.D.

Contributing cEDM partners

ACB, Dendermonde, Belgium
ASML, Veldhoven, The Netherlands
Barco, Kortrijk, Belgium
Connectronics CTBP, Poperinge, Belgium
ED&A, Kapellen, Belgium
Electronic Apparatus, Tessenderlo, Belgium
EuroCircuits, Mechelen, Belgium
Page Electronica, Poperinge, Belgium
PsiControl, Ieper, Belgium

NEVAT/EMS - Technologiegroep, The Netherlands

Other contributors

Ramaer, Helmond, The Netherlands

Table of Contents

The Design-for-eXcellence Guidelines principles	2
PBA DfX Guidelines Scope	2
Acknowledgement	3
1. Applicable Documents	5
2. Applicability of the PBA DfX Guideline EDM-D-005.....	5
3. PCB Build-up	6
3.1. In-plane copper interconnection structure (X-Y).....	6
3.2. Via interconnection (Z)	8
3.3. Layer stacking	10
3.4. Solder Mask	10
3.5. Solderable finish.....	11
3.6. Screen printed ink	12
3.7. Advanced PCB solutions	12
4. PCB Density Classification.....	13
4.1. PCB Density Classification objective	13
4.2. Density classification definitions	13
4.3. Track Density Class dimensions.....	15
4.4. Via Density Class dimensions	16
4.5. Track-Via Density Class alignment for BGA break-out.....	17
4.6. Manufacturability limitations to Via Density Classes	17
4.7. MicroVia Density Class dimensions.....	18
4.8. Solder Mask Density Class dimensions	19
4.9. PCB routing	19
5. PCB Build-Up nomenclature	19
6. Guideline to PCB Density Classification application	21
6.1. PCB Density Classification design guideline.....	21
6.2. PCB supplier capability.....	22
6.3. Accuracy of PCB structure dimensions.....	22
Appendix A: PCB Density Class vs. BGA break-out.....	23
Interpad track capacity	23
Internal interland track capacity	23
Appendix B: PCB capability inquiry	25
Revisions	26

1. Applicable Documents

This PBA DfX Guideline refers as part of the guideline to the most recent versions of the following documents and standards including their amendments.

EDM-D-001	PCB Specification
EDM-D-006	PCB layout solutions
EDM-D-008	Technology and Manufacturing Capability Mapping of PBA designs
EDM-D-009	Signal Integrity
EDM-D-010	Power Integrity
EDM-D-011	Electro-Magnetic Compatibility
IPC-2152	Standard for Determining Current Carrying Capacity in Printed Board Design
IPC-2221	Generic Standard on Printed Board Design
IPC-2222	Sectional Design Standard for Rigid Printed Boards
IPC-4101	Specification for Base Materials for Rigid and Multilayer Printed Boards
IPC-4103	Specifications for Base Materials for High Speed/High Frequency Applications
IPC-4562	Metal Foil for Printed Wiring Applications
IPC-4761	Design Guide for Protection of Printed Board Via Structures
IPC-6011	Generic Performance Specification for Printed Boards
IPC-6012	Qualifications and Performance Specification for Rigid Printed Boards
IPC-6016	Qualification and Performance Specification for High Density Interconnect (HDI) Layers or Boards
IPC-6017	Qualification and Performance Specification for Printed Boards Containing Embedded Passive Devices.
IPC-6018	Qualification and Performance Specification for High Frequency (Microwave) Printed Boards
IPC-A-600	Acceptability of Printed Boards
IPC-CF-152	Composite Metallic Material Specification for Printed Wiring Boards
IPC-D-279	Design Guidelines for Reliable Surface Mount Technology Printed Board Assemblies
IPC-SM-840	Qualification and Performance of Permanent Polymer Coating for Printed Boards
J-STD-609	Marking and Labeling of Components, PCBs and PCBAs to Identify Lead (Pb), Lead-Free (Pb-Free) and Other Attributes.

2. Applicability of the PBA DfX Guideline EDM-D-005.

- Design recommendations given in the guideline are intended to help the user in making choices that improve the manufacturability, reliability, testability, etc., of the final PBA. These recommendations are of a generic nature. Therefore, in specific cases more optimal solutions may exist.
- Design specification takes precedence over this guideline.
- IPC class 2 requirements and test procedures apply unless specified otherwise in this document.
- The guideline provides general requirements and recommendations regarding the PCB build-up. Selected detailed layout solutions satisfying the general PCB build-up requirements are the subject of PBA DfX Guideline EDM-D-006.
- The PCB Density Classification system is intended to classify the manufacturing capabilities. It does not specify minimal electrical clearances between Cu features nor current carrying capacity of the Cu tracks.