

PBA Design-for-Manufacturing Guideline

EDM-D-006
Layout Solutions

V1.2
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 realized in collaboration with the industrial and academic partners of imec's Center of Electronics Design & Manufacturing.

Funding organizations

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

imec contributors

Alain Carton
Boris Leekens
Geert Willems, Ph.D.

Contributing EDM partners

ACB, Dendermonde
Barco, Kortrijk
Connect Group, Poperinge
Eurocircuits, Mechelen
Page Electronica, Poperinge
PsiControl, Ieper

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-006	5
3. Board layout rules	6
3.1. Terminology	6
3.2. Board transport and positioning	6
3.3. PCB build up and density class selection	7
3.4. Track layout	8
3.5. Solder mask layout	9
3.6. Test point layout	10
3.7. Selective soldering layout rules	11
3.8. Repair layout rules	13
4. Foot print design	15
4.1. General	15
4.2. Reflow soldering layout	15
4.3. Wave soldering layout rules	16
5. Ball Grid Array components (BGA)	18
5.1. General	18
5.2. BGA footprint	18
5.3. BGA fan-out pattern	20
5.4. BGA break-out	21
6. Bottom Termination Components (BTC)	23
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-004	Design-for-Assembly
EDM-D-005	Rigid PCB Build-Up and Density Classification
IPC-2141	Design Guide for High-Speed Controlled Impedance Circuit Boards
IPC-2221	Generic Standard on Printed Board Design
IPC-2222	Sectional Design Standard for Rigid Printed Boards
IPC-7093	Design and Assembly Process Implementation for Bottom Termination Components
IPC-7095	Design and Assembly Process Implementation for BGAs
IPC-7351	Generic Requirements for Surface Mount Design and Land Pattern Standard
IPC-A-610	Acceptability of Electronic Assemblies
J-STD-001	Requirements for Soldered Electrical and Electronic Assemblies

2. Applicability of the PBA DfX Guideline EDM-D-006

- Specification 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 layout of printed boards and component footprint design. It treats component footprint design in a general way, specific component footprints and land pattern dimensions are out of the scope of this guideline.