PBA Design-for-eXcellence Guideline

EDM-D-009
Signal Integrity

V1.1
October 2019

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

The Electronics Design and Manufacturing 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 basic 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 hardware realization aspects 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 Signal Integrity Design Guideline Scope

- This guideline focusses on the Signal Integrity (SI) of a PBA. This guideline is complementary to the EMC guideline EDM-D-011 and the Power Integrity guideline EDM-D-010.
- Signal Integrity (SI) refers to all problems arising in high-speed digital designs due to the interaction between the PBA interconnects and the digital signal's voltage and current waveforms that are propagating along these interconnects. Examples of SI problems are: reflections, crosstalk, attenuation, frequency dispersion, ringing, Inter-Symbol Interference (ISI), eye closure, non-monotonicity, …
- In general, sources of SI problems can be subdivided into five categories. The categories and related root causes are :
    1. Reflections: PCB layout, stack-up and material related impedance discontinuities, electrical design related impedance discontinuities and mismatches.
    2. Crosstalk between interconnects or nets: PCB layout and stack-up related.
    3. Signal losses: PCB conductor and dielectric material.
    4. Mode conversion related to any asymmetry which converts differential signals into common signals: PCB layout, stack-up, quality and material related
    5. ElectroMagnetic Immunity related: inter- or intra-system electromagnetic radiation coupling into the PBA interconnects.

  Each of these problems gets worse and harder to solve as rise times decrease or clock frequencies increase.
- This Signal Integrity guideline focusses on assessing and improving the quality of the digital signals along all interconnects. Measures to lower reflections, avoiding impedance

discontinuities, lower coupling between signal nets and reduce frequency dependent losses are described.
- ElectroMagnetic Immunity is covered by the EMC guideline EDM-I-003. However, each of the guidelines given below for good SI also contribute and may even be required to obtain good EMC of the PBA.
- This guideline applies to all types (rigid, flex, flex-rigid) and classes (IPC 1, 2 and 3) of PCB and PBA, both SnPb and lead-free soldered.

Acknowledgement

This document was realized in collaboration with the industrial and academic partners of imec's Center of Electronics Design & Manufacturing and Sirris.

Funding organizations

IWT is acknowledged for funding the VIS-traject PROSPERITA and the TETRA project DfX-Bridge that have provided the scientific background for the PBA DfM Guidelines and gained the necessary industry support.

imec contributors

Geert Willems, Ph.D.
Alain Carton

KU Leuven contributors

Davy Pissoort, Ph.D.
Bart Boesman

Contributing cEDM partners

Barco, Kortrijk

Table of Contents

1. Reference and 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-010 Power Integrity
EDM-D-011 Electro-Magnetic Compatibility (EMC)
IPC-2141A Design Guide for High-Speed Controlled Impedance Circuit Boards

[1] Eric Bogatin, “Signal and Power Integrity“, simplified 2nd edition, Pearson Education,, 2010.
[2] Keith Amstrong, “EMC Design Techniques for Electronic Engineers”, Armstrong/Nutwood UK 2010.
[3] Keith Armstrong, “EMC for Printed Circuit Boards”, Armstrong/Nutwood UK,,2010
[4] Bruce Archambeault and James Drewniak, “PCB Design for Real-World EMI”, Springer Science+Business Media, 2002.

2. Applicability of the PBA DfX Guideline EDM-D-009

- The 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.

3. PCB signal quality and Transmission Lines

3.1 This guideline focuses mainly on the **digital** part of the **Printed Board Assembly** design which often contains more than 90% digital IC components and signal nets. The remainder consists of: AC/DC, DC/DC power or high-voltage circuits; analog circuits; RF/microwave circuits; ... In a synchronous digital design two kind of signals can be distinguished: clock signals and bit-stream signals defined by their logical state. Those signals can be described or analyzed in the time domain or frequency domain. The electrical and physical design impacts the ideal digital signal’s waveform or signal quality and could result in Signal Integrity problems.

3.2 The waveform of digital signals are described by following properties: period or frequency, bit length or bit speed, delay, rise and fall time, voltage level. Typical units used are nanoseconds (ns), Mega/Giga Hertz (M/GHz) and Mega/Giga Bits per second (M/Gbps), Volts/millivolts.

3.3 The 10-90 **rise time RT** and **fall time FT** refer to the time the signal needs to go respectively from 10% of the final state to 90% (RT) of the final state or vice versa (FT). This is used in this guideline. Occasionally, the 5-95 rise and fall times are used.

3.4 Signal Integrity problems arise due to the interaction between an interconnect and a digital high-speed signal that is launched onto the interconnect. As a first step, determine which interconnect/signal combinations on the PBA are critical for SI based on the rules of thumb described in 3.4.1 - 3.4.3.