PCB Handbook

AcubeSAT-OBC-G-009

Theodoros Katzalis

August 18, 2020
Version: 0.3

Aristotle University of Thessaloniki

Aristotle Space and Aeronautics Team
CubeSat Project

2020

Contents

Changelog

Date	Version	Document Status	Comments
18/08/2020	0.3	INTERNALLY RELEASED	Updated: • Rules of thumb. • Identification of design and PCB design guidelines regarding high speed applications. The last is referred as Design for Performance. • Links are archived. Added: • References and image sources. • Footnotes • Design for Testing and Design for Manufacturing. • An overview of simulation and analysis. • Checklist Deleted: • The Disclaimer section
15/08/2019	0.2	INTERNALLY RELEASED	Remove WIP from title and fix some typos.
13/08/2019	0.1	INTERNALLY RELEASED	Initial revision.

This is the latest version of this document (`0.3`) as of August 18, 2020. Newer versions might be available at `https://helit.org/mm/docList/AcubeSAT-OBC-G-009`.

1 Preface

The purpose of this report is to make the reader feel familiar with the PCB fabrication and to give an insight about the PCB guidelines when the time for design comes. So, it is divided in two main conceptual parts: **"What is a PCB?"** and **"How to design a PCB?"**. In the first part, we will make a short introduction about different types of PCBs, their materials and the overall structure. The second part isn't oriented on how to use a CAD tool, but highlights the potential problems in signal integrity and the countermeasures.

2 What is a PCB?

PCB is a **P**rinted **C**ircuit **B**oard and is a "sandwich" of conductive and non-conductive materials that cooperate to control the electrical flow between electrical components. It's the place, where these components can be hosted and communicate with each other and can be described as their mechanical and electrical support. PCBs have developed a lot throughout the years: The irreversible progress of modern electronics changed radically the performance and the specifications along with the considerations and the new challenges that the whole industry must face. Things are getting smaller, faster, light weight and reliable. Just check the figure 2 and you will feel how time change things!

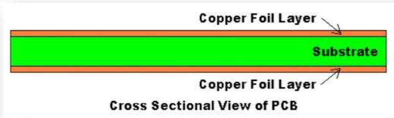
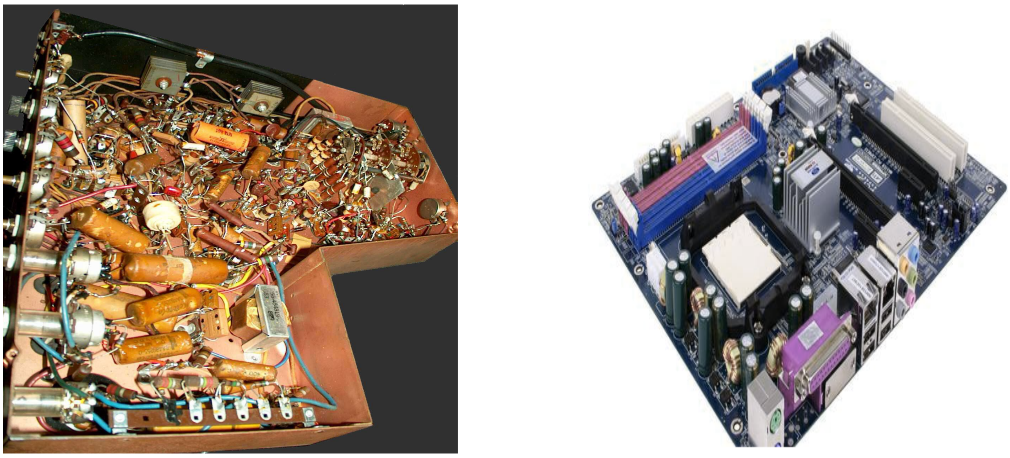


***Figure 1**: 2-layer PCB (image source)*



***(a)** image source* ***(b)** image source*

***Figure 2**: An old school TV's PCB on the left and a modern PCB on the right*

2.1 Materials

As mentioned earlier, PCB is mainly a stack of conductive and non-conductive materials:

- Conductive: **Copper foils**

- Non-conductive: Dielectric substrate (the base that holds the PCB together and separates the copper foils) like **FR-4**. The choice of the substrate is a function of thermal and electrical performance. For example, ceramic substrate is preferred in demanding thermal applications due to the higher thermal conductivity.

To be more precise, FR4 laminate coated with copper is called copper clad laminate (CCL). The copper foil is glued within high temperatures with the pre-preg (pre-impregnated) that is fiberglass with impregnated epoxy resin (uncured FR4). Thus CCL is the fundamental block of a PCB [24].

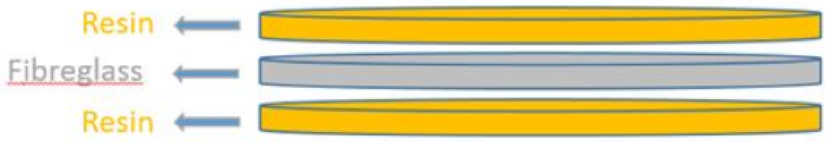


Figure 3*: Prepreg, uncured FR-4 (image source)*

Are PCBs consisted only of copper and substrate? Actually, no! In PCB fabrication, more complementary processes are required (subsection 2.4).

But, first of all let's visualize and comprehend the structure and the manufacturing process of these magic boards! PCBs can have different numbers of layers, but can also come in changing inflexibilities.

2.2 Inflexibility

In terms of inflexibility, there are three main types: 1) rigid, 2) flex and 3) rigid-flex PCBs:

2.2.1 Rigid

The first thing that is coming to your mind when thinking a PCB is probably the rigid one. A motherboard is a characteristic example. Basically, a solid substrate material is used that prevents the board from changing its shape. In this report, for the analysis, we will stick to this type.

2.2.2 Flex

As the name implies, the advantage of the flex boards is the ability to flex! When there are tight mechanical constraints and little space, this type of boards are usually preferred. They can be either used as connectors reducing significantly the bulky size of typical connectors or as a fully assembled board. Common flex-materials to utilize these features are plastic like polyimide and polyester [6]. Of course all of this comes with an extra cost than the rigid one.

2.2.3 Rigid-Flex

Rigid flex boards merge technology from both flexible and rigid circuit boards and they are consisted of flexible and rigid materials. A typical example could be two rigid boards connected with a flex one.

(a) OBDH EM *(b) image source* *(c) image source*

***Figure 4**: Rigid, flex and rigid-flex PCBs*

From now on, we will be focused on simple and common rigid PCBs.

2.3 Layers

Printed Circuit Boards can be divided into layers. The increasing technological demands result to multi-layer boards and the stack up (the arrangement of copper layers and insulating layers) is a very important factor. The number of layers range typically from 1 to 16 and usually, in the industry, even numbers are used. Using the term layers we are basically referring to the number of copper foils.

2.3.1 2-Layer

A 2-layer board can be described as a copper clad laminate as we mentioned earlier. On these copper sides, the routing takes place. These copper paths, called traces, serve as the channel for the electrical components to interact with each other. In other words, the point of the copper is to carry electrical signals to and from the PCB, much like your nervous system carries signals between your brain and your muscles.

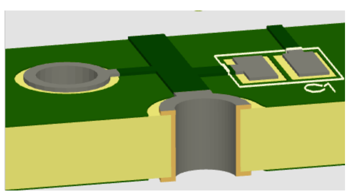


***Figure 5**: 2-layer PCB, source: Eurocircuits buildup feature*

Now, you may be wondered, how it is possible traces that belong to different copper layers to be connected with each other? The answer is via (vertical interconnection access) and it is applied also to multi-layers. In short, they are holes plated with copper.

About the core, it is a non-conductive substrate usually made of FR4. FR4 is used because provides a solid, strong base and the most cost effective solution regarding

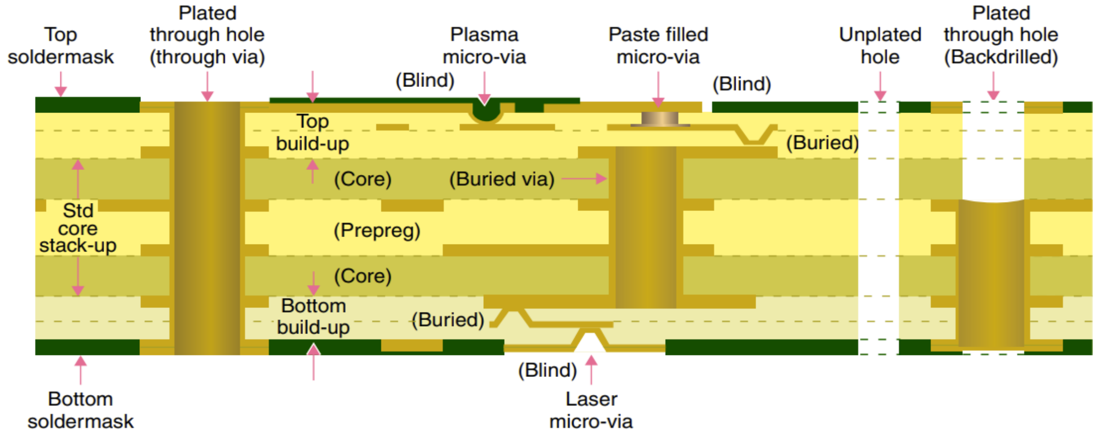
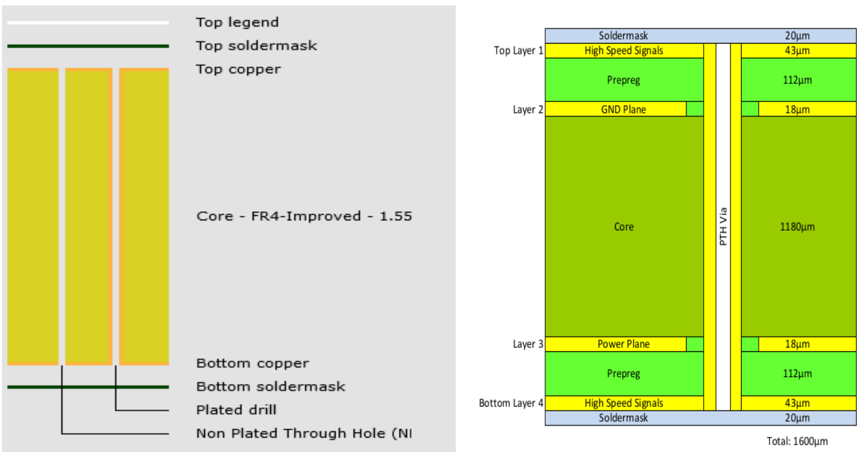


***Figure 6**: Different types of via for a multi-layer board [20, p.4]*

dielectric strength, losses and heat dissipation. There are many types of FR4 though. Eventually, it may be helpful to think of the substrate as the PCB's "skeleton".

2.3.2 4-Layer

Now, that we have a basic understanding about a 2-layer PCB, let's jump to a 4-layer one:



***(a)** source: Eurocircuits buildup feature* ***(b)** image source*

***Figure 7**: Typical stack-up dimensions fro a 2 (left) and 4 (right) layers boards*

The multi-layer boards are basically a set of 2-layer boards glued together and this is also the main reason why even numbers are used. Furthermore, lamination is the process by which the core(s) of a printed circuit board is melted together through heat and pressure with copper layers and prepreg layers. In other words, it is the process that creates a "sandwich" or multiple "sandwiches" connected together. It requires specific heating and pressure for specific periods of time based on materials used to ensure the PCB is made properly. These laminates and the copper thickness comes in different sizes, too.

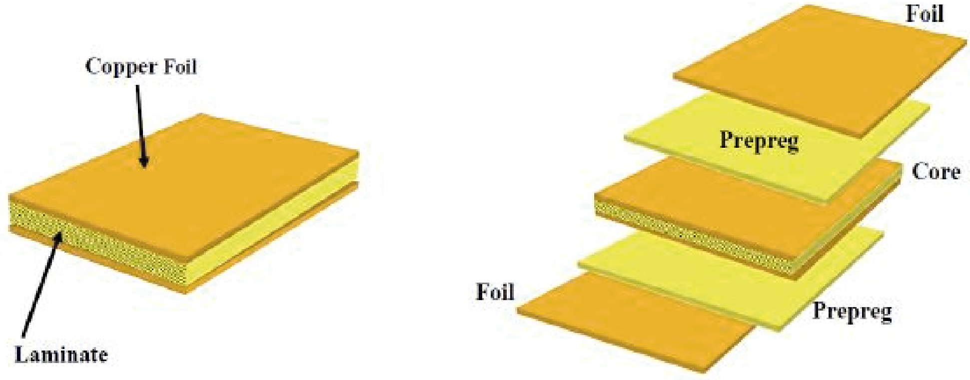


***Figure 8**: image source*

Noticeable differences between the figure 7a and figure 7b are: 1) the height is approximately the same to these boards (0.063 inches = 1.6 mm) and 2) the **prepreg**. Someone could say, how it is possible for the overall thickness to be the same in 2 and 4 layers PCBs? It's because laminates and copper foils of different thickness are used to prevent increasing the height as the layers are increasing. So a 4-layer board is actually a set of two 2-layer boards separated by the core, but the overall thickness is the same with a 2-layer design, because the pairs have smaller thickness. A rule of thumb is to have the same thickness above and below of the core (multi-layers boards can be mirrored)!.

As we have stated, prepreg is a fiberglass impregnated with resin (FR4). The resin isn' t hardened yet, so it can be used to bond the required copper foils. It is different with the core because there is still the modular feature of curing, of bonding things together. The core is already cured with copper foils in both sides (copper clad laminate). Although, there might be some differences in the materials and the electric and thermal properties between these two.

2.4 Complementary layers

Except the fundamental copper layers and the substrate, there are some complementary processes during manufacturing that can be defined as extra layers[1] and they are very useful and necessary nowadays. These are the soldermask, the surface finish/treatment, the silkscreen and the conformal coating. We will not go into great depth for the materials and their properties though.

2.4.1 Soldermask

If you encountered a PCB, then the first thing that you would notice is the green color. This is called soldermask and is a thin polymer applied to the top and the bottom of a PCB. Overall, the soldemask assists the functionality and durability of the board by protecting it from oxidation, corrosion, moisture and dust. It is used also for preventing manufacturing defects like solder bridges. There are two basic categories of

[1]The term layers is used quite a lot and can have several meanings (stack-up, soldermask, coatings).

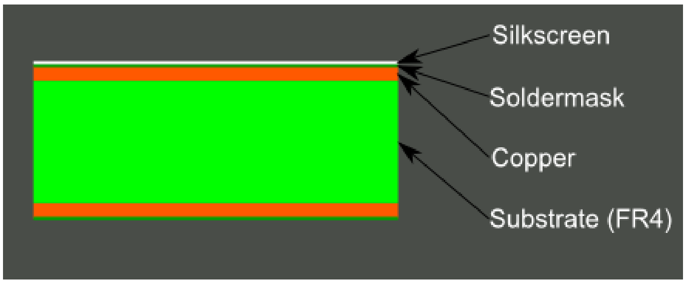


Figure 9: *Image source*

soldermask materials: liquid screen printed and photoimageable masks. Most of the times the soldermask choice in special applications like aerospace will be dictated by standards. It is worth mentioning that the green color is the most distinctive but of course soldermask can have other colors. There are a few reasons why the green is the most common one like better visual inspection [20].

2.4.2 Surface finish

A PCB surface finish is a coating applied above the pads, the exposed copper areas. It is applied for two basic reasons: to ensure solderability and to protect exposed copper circuitry. As there are many types of surface finishes, selecting the right one is no easy task, especially as SMDs have become more complex and regulations such as RoHS and WEEE have changed industry standards.

There are many types of surfaces. Namely we have: Hot air solder leveling (HASL), Immersion Tin (ImSn) or Silver (ImAg), Electroless Nickel Imerssion Gold (ENIG) and Electroless Nickel Electroless Palladium Immersion Gold (ENEPIG), Organic Solderability Preservative (OSP) [25]. Each one of those has of course advantages and disadvantages. The most popular in the old days was HASL. The most common one probably is the ENIG nowadays.

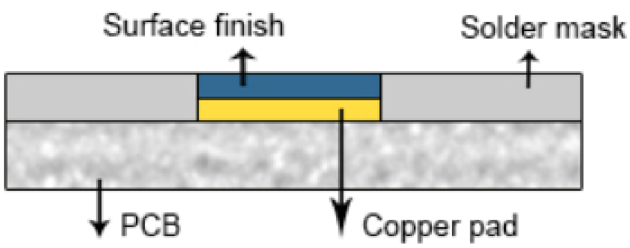


Figure 10: *image source*

2.4.3 Silkscreen

Silkscreen has no electrical purpose and is referred to the human readable letters and shapes on the surface of the PCB. It is white and normally used to identify components, test points, PCB part numbers, warning symbols, company logos, date codes and manufacturer marks. This, in turn, makes it easier for electronics assemblers to place each electrical component in the proper place and in the right direction. It gives a more artistic tone in the PCB, too!

2.5 Assembly

After the design and the bare board fabrication have finished, the time for soldering[2] and component placement has come (populated board). The process that is responsible for this is called assembly and the item with the components combined with the board is called PCBA (Printed Circuit Board Assembly).

PCBs can be assembled **manually** or by **automatic** machinery. Manually assembled is related with placing and soldering the components by hand. This is suitable for prototypes and low volume production. A mixed process with hand placement and automatic soldering can also be used as a strategy regarding the manual method. For the automated assembly, pick and place machines are substituting the hands of the assemblers and the soldering process can be divided to reflow and wave soldering.

The first one is used for SMDs and the second one mostly for through hole or for a mixed set up with both package types. As the high speed applications are increasing, SMT is replacing the through-hole due to better electrical performance (inductance and capacitance of the leads) and size costraints. Thus the wave soldering is used less except some special applications (e.g. power devices).

2.5.1 Conformal coating

Very briefly, conformal coating is an additional step of the assembly process. After the placement of the components in the PCB, a polymeric coating is applied to the top of the populated board to protect it from harsh environmental conditions.

2.6 Integrated circuit packages

Until now, we have discussed about PCB structure and fabrication. But what about the electrical components that can be placed on them. What are the prerequisites? What are the form factors? Why to choose the one or the another?

Packages are a fundamental part of the IC fabrication and PCBA. They are the house, the structure of the actual circuitry of the die and act as the interface with the rest of the distributed electrical components in a PCB via leads, solder balls and pads. They play a crucial role in electrical and thermal performance too.

Standardization makes the job of PCB industry a lot more productive and efficient. Thus, most electronic components come in standardized packages. A package type has a well defined set of physical dimensions that the component has to conform to. For each package, normally the pitch spacing, height and general shape is defined. However there are non standardized packages for special applications. In general, based on the

[2]Soldering is used both to attach components physically to the PCB and to provide electrical conductivity between the component's leads and the PCB traces. The most common one is the Sn63/Pb37.

mounting technology ICs can they can be divided in two main categories: 1) SMD (Surface Mount Device) and 2) Through-hole. More about packages can be found in the AcubeSAT-THE-BC-032 report.

2.7 External links

- PCB material guide by PCBChart
- What is the difference between prepreg and core?
- Stack-up dimensions by CBS Electronics
- JLCPCB fabrication cycle video
- Eurocircits fabrication cycle video
- Eurocircuits: Making a PCB step by step
- Technical specifications of Eurocircuits services
- A list of IC packages

3 Workflow: From design to reality

Now that we have developed an intuition about the PCB, we will try to make it more specific and analyze the design aspect. What engineers actually do before the PCB is sent for fabrication?

The development cycle of the PCB can be divided into six main parts: 1) component selection, requirements and specifications, 2) design (PCB layout), 3) simulation, 4) manufacturing, 5) assembly and 6) testing. Testing is integrated both in the assembly and manufacturing. The design can be categorized to design for manufacturing, for testing and for performance.

For the design part, specialized software tools are used in the industry. These are called CAD or EDA and provide features such as schematic capture, library management, PCB layout and generation of standardized files for fabrication along with other of course capabilities to make the life of designers easier. The engineer, very briefly, first organize the libraries for the components, then captures the schematic and finally starts the PCB layout, placing the components and routing the connections. Actually in large corporations, a lot of people are contributing to this: library manager, circuit engineer, layout engineer, test engineer.

A free and open source tool used by the AcubeSAT team for the first engineering models of OBDH and EPS is the **KiCad**. To start learning KiCad, useful resources are the well-written documentation from KiCad website and a detailed book, KiCad Like a Pro. Of course, the best way to learn a tool is start using it!

3.1 Libraries

Having reliable and well configured libraries is a very important part of the PCB making. They are the fundamental blocks of the design, the bricks of a building. Always check if the data is compliant with the needs of the fabricators and the datasheets, especially for the pinout and the dimensions.

The libraries are referred to schematic symbols used for the schematic capture, the footprints for the PCB layout and the 3D models for the visualization of the actual board.

3.1.1 Schematic symbols

3.1.2 Footprints

Footprint or land pattern is an arrangement of exposed copper areas (pads) for the physical attachment and electric connection of the components with the bare board. In other words, it is the interface between the board and the component. Manufacturers tend to work with standardized footprints to ensure consistency, quality and productivity. The most well know industrial standard for this purpose is the IPC-7351.

Should the designers build all the footprints from the scratch for each component? Usually vendors (TI, ST Microelectronics, NXP) provide the required data (schematic symbols, footrpints, 3D models) compliant with IPC for a wide range of EDA tools. Otherwise designers can make use of other third party libraries like SnapEDA, Ultra-librarian, SamacSys. In the engnineering model of the OBDH, most of the footprints mined from the SamacSys library (IPC-7351B). There is also a quiet useful plugin for combining it with KiCad.

If you can't find or build one by IPC wizards (some EDA tools like Altium feature an integrated wizard to build IPC footprints), then most of the times, especially for the custom packages, the vendors will provide in the documentation guidelines on how to configure it. For example in case of the ADCS subsystem, the RM3100 (Geomagnetic sensor) has a custom package developed by PNI and in the figure 11 is depicted the recommended data for the footprint's implementation. When a package isn't standardized, it will probably increase the cost, because the manufacturer should adapt their flow to meet the needs of this particular component.

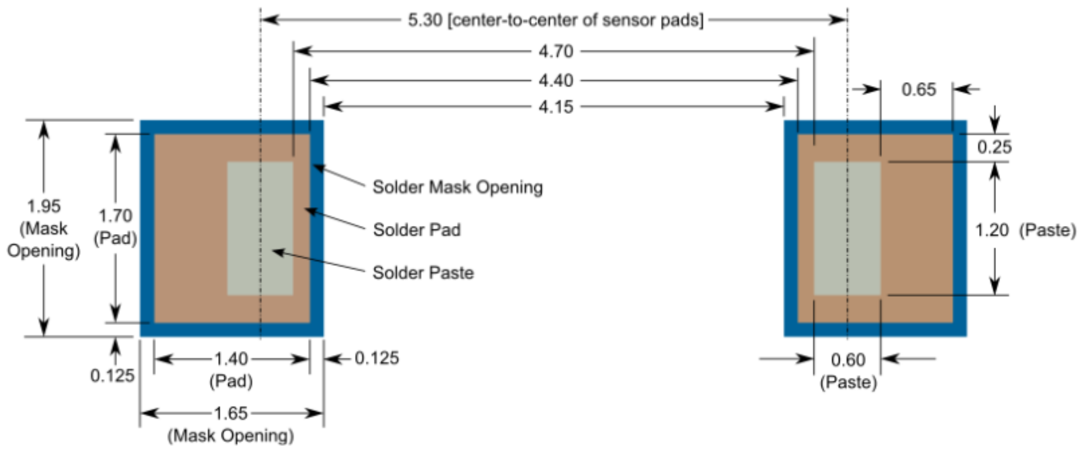


***Figure 11**: RM3100 recommended custom footprint configuration [26, p.9]*

How important are footrpints? A lot of problems originates from poor library implementation. So, it would be wise to always review your libraries. Precisely, check each pad if it is properly assigned for each type of layer (e.g. copper, solder paste, soldermask). Sometimes ready footrpints offered by third party libraries might have some issues like assigning copper areas as a drawing (no electric connection) that can cause many problems. This actually happened in the EPS's board. The drawing had the exact same color with the layer associated with the copper one, so it was very error-prone! For the mistakes made in the engineering models you can check the AcubeSAT-OBC-EC-012.

3.1.3 3D models

3D models help a lot with the visualization and can be used for debugging. For instance, by viewing your board in 3D, you can inspect if objects are overlapping and if everything fits together. This of course can be done in the PCB layout enabling the layer that is dedicated for the mechanical dimensions (if your footprint contain the required data) like the Courtyard in KiCad, as we have mentioned previously.

3.2 Fabrication data

3.2.1 Manufacturing data

Next step after the finished design is the bare board fabrication. For this purpose, the generation of files with special format called **Gerber** is the de facto standardized way to connect manufacturers and designers. Each EDA tool has the option to generate these type of files. In short they contain information about the copper of each layer of the PCB, the soldermask and the silkscreen. The **NC** (Numeric Controlled) **drill files** are also generated along with the Gerber ones to specify the data for the drilling machines to create the holes of the PCB.

3.2.2 Assembly data

Each assembly house requires certain formats about the data requested for the process, so some differences might exist among them. Assembly data is typically referred to the following:

- Bill of Materials (**BOM**). The materials used for a cooking recipe! It is usually a csv file that lists everything that the assembler needs to know about the reference designators (component identifiers) and the manufacturing part numbers of the ordered components. Be careful with the default generated format from the EDA. It should be adjusted to the assembly house requirements.
- Component **location** and **orientation**. For KiCad, this is satisfied by the "CtrYd" (Courtyard) and the "Fab" layers respectively.
- Solder **paste** location. The paste data is used to manufacture an SMD stencil required for reflow soldering. It is a dedicated layer in the EDA tools (for KiCad "Paste" layer). The solder area is almost equal to the size of the pad.
- For automatic assembly, specific position files readable by Pick and Place machines are needed, the so called **X,Y files**. These provide information about the

component position and orientation (.pos extension file for KiCad).
- Files with location data for the tests points (**test fixture**), if testing is integrated in the assembly process.

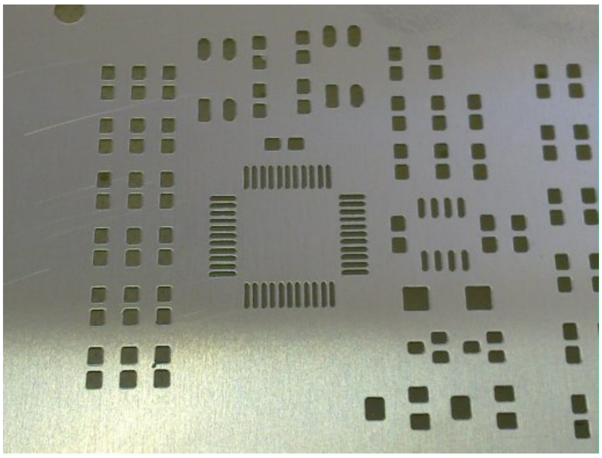


Figure 12: *Stencil (image source)*

4 Identifying the design

Let's start talking about the challenging part of PCB development cycle, the **design**. All the problems of the PCB can be categorized to **Signal Integrity** (SI): "Involving the distortion of signals", **Power Integrity** (PI): "Involving the noise on the interconnects and any associated components delivering power to the active devices" and Electromagnetic Compatibility[3](**EMC**): "The contribution to radiated emissions or susceptibility to electromagnetic interference from fields external to the product". These definitions aren't very helpful and sometimes problems of one category may overlap with the another but we are going to investigate them more in the DFP section.

For now, it should be noted that it is very important to distinguish in the design the existence of transmission lines. They are a subset of the root cause of many problems regarding signal integrity (reflection and the associated side effects) and you will understand why soon.

Introduction to distributed model and transmission line theory. Electromagnetism phenomena can be divided to 1) high power and low frequency (electric machines and plants, transformers) and 3) high frequency and low power (mobile devices, modern systems). The focus in this report will be on the second part, so let's take a look on why high frequency really matters? Is actually the frequency the root cause of circuit issues?

In the PCB industry, designers for a long time in the low-frequency era neglected the **electromagnetism** and build their designs according to its **abstract** and less complicated version, the **circuit theory**. The so called traditional **lumped** model is based on principles such as 1) everything can be modeled via resistors, capacitors, inductors 2)

[3]Usually Electromagnetic Interference (EMI) is referred to the cause of the problem and EMC is referred to the solutions

everything happens instantly (there aren't time transients) and 3) interconnections that connect components are equipotential surfaces. This abstract model is a very efficient and practical method to make your application functional without worrying with the complex nature of electromagnetic fields. But as technology is progressing and things are getting smaller and signals are getting faster, the electromagnetic phenomena can not be neglected anymore. This is where **transmission line theory** along with the wave properties are coming to the surface, that connect the electromagnetism with the circuit theory and introduces the **distributed** model. According to this model, circuit theory can also be a great tool in high speed applications but with some modifications. The space parameter now really matters and the tracks that carrying signals can't be treated like equipotential surfaces but as an RLC circuit that extends as a function of the length of the track (figure 14). This result to a much different electric behavior and should be consider.

This different electric behavior is related with the fact that signals are starting to act like waves. But why and how? Waves in nature have some fundamental characteristics. The **wave** nature of voltage signals is starting to take shape when frequency is very high and length of the interconnections is such that segments of wavelength can fit to the conductive traces. Just like ocean waves and ropes reflect when they meet an obstacle, a voltage signal can potentially reflect back to the source. A very nice video by AT&T can help a lot with the visualization to consolidate the analogy! These reflections is the cause of the important aspect of impedance matching. In the PCB case, they can be observed when there are discontinuities (impedance mismatch) regarding the geometry of the conductive path along the way that the signal propagates. Another feature that needs to be considered is the **radiation** aspect. When charges accelerate, electromagnetic fields are taking shape. The higher the frequency, the shorter the rise time, the faster the acceleration, the greater the radiation. The goal of the transmission lines is to guide the energy with maximum power transfer to an antenna and without radiating energy. So we could summarize that in high frequencies, voltage signals behave like waves and the charge acceleration radiate electromagnetic energy that it can be coupled to undesired pathways if we don't pay attention. We will discuss more in the section Design for Performance about the problems and the solution regarding high speed signals.

But why transmission lines can be bad? What are the transmission line effects? They can cause timing issues due to the delay of the signal to reach from the driver to the source (tuning traces) and reflection issues such as false triggering, ringing, more EMI and crosstalk, overshoot, distortion etc. As it seems transmission lines can affect severely the signal integrity so we need first to identify them and secondly to address the problems of reflection and delay with impedance matching, proper termination and tuning traces.

Sometimes we are very good at learning individual things but connecting the dots is equally important. A recommended book to connect electromagnetism and circuit theory about electronics is "The Fields of Electronics: Understanding Electronics Using Basic Physics" by Ralph Morrison. It is very important to start thinking in fields and space rather traces that carrying signals. Viewing traces as the boundary of the space, is more helpful as a designing mindset in the AC world.

4.1 To be or not to be a transmission line

A very important question that a designer should ask in the early phase of the decision making about the designing strategies is **when should I worry about transmission line theory?**[4] To answer this question we need to know: 1) The propagation velocity of the signals of interest, so the constant of the material that the EM fields are propagated through. In our case, we need the dielectric (typical FR-4) constant of the PCB. 2) The frequency of the signal that can determine afterwards the wavelength. Wavelength is quite interesting because is referred to an oscillation, a wave, a sinusoidal function. Circuits are often described by how they respond to sine waves of various frequencies. What happens to the non-sinusoidal inputs though (e.g. square waves)? How to analyze them [22]?

First about the sinusoidal ones, for the transmission line model (microstrip, depicted figure 13), using the equation $c = \lambda * f$ and knowing the frequency, we can calculate the wavelength in free space ($c = 3 * 10^8$, the speed of light). Including the dielectric constant of the medium, the wavelength of interest is $\lambda_d = \lambda/\sqrt{\epsilon_r}$[5]. For the typical dielectric choice FR4 in PCBs, the constant is $\epsilon_r = 4.5$ (velocity factor, $1/\sqrt{\epsilon_r} = 0.47$) [29, p.46]. It is worth mentioning that **it isn't the frequency** that really matters, it is about comparing the length of the tracks that carrying the signal (voltage) with its wavelength. When these two are "comparable", then the designers need to incorporate in their thinking the transmission line theory. To define what is comparable, a rule of thumb well know in the industry with some differences among the references is that the length of the line should not be under $\lambda/10$! For example for a 100MHz signal with FR4 as medium, the length of the tracks should kept under 0,141 m for a transmission-free design! **But this rule of thumb should not be taken too seriously**. Designer may face transmission line effects even in cases of $\lambda/40$ or can ignore them until the length is comparable with $\lambda/4$. It depends on the acceptable noise margins, the sensitivity of the circuit and the tolerances. So the best response for the question when the designer should worry about transmission line effect is: **When these effects become important/noticeable to the design**. In other words the real issue that needs to be considered is how much reflection and coupling can the circuit tolerate and still be functional.

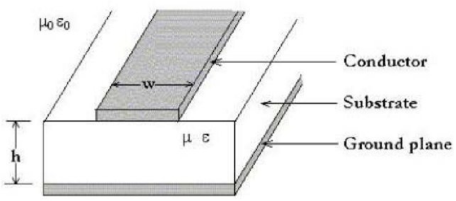


***Figure 13**: Microstrip [4]*

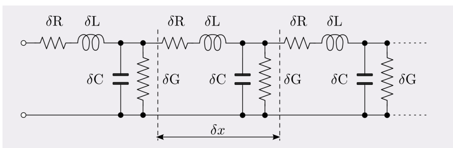


***Figure 14**: Transmission line model (image source)*

[4]Transmission line mainly can be referred to any pair of signal and return conductors. In this case we worry about the reflection aspect of transmission lines.

[5]This formula is valid in a stripline environment that the field is surrounded by the dielectric. In the microstrip, the effective dielectric should be used due to the air. Propagation velocity of stripline is a stricter rule than the microstrip though.

4.1.1 Digital signals

Non sinusoidal signals can be analyzed as the sum of sinusoidal ones (fourier transfrom). This is a very powerful concept that can connect the sine wave analysis with any signal. A very common non-sinusoidal signal with great interest in the digital world is of course the **square wave** (actually the trapezoidal) and this will be the base to understand the analysis for non-sinusoidal inputs.

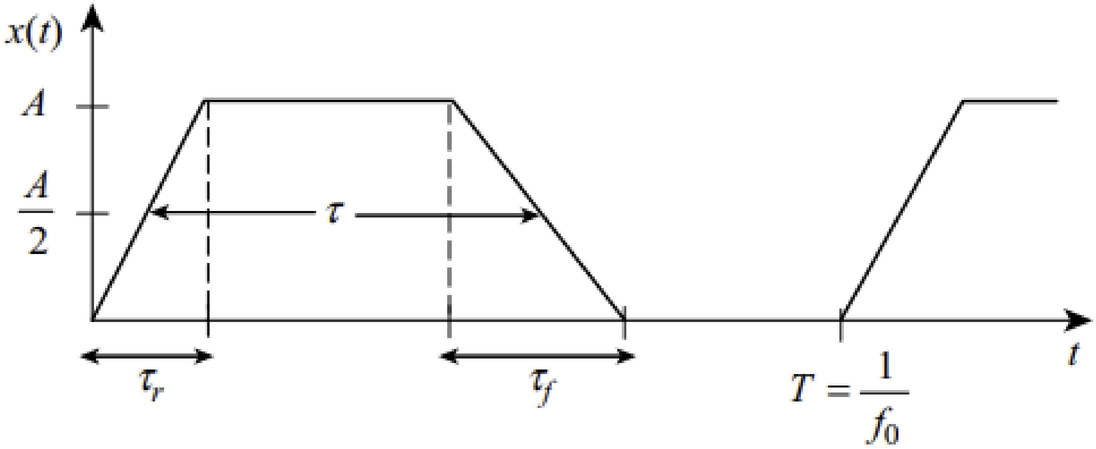


Figure 15: *Typical digital clock [23, p.2]*

The **response** of a pulse can be analyzed as the sum of the responses of the sine waves that constitute the pulse. With that in mind, a pulse response contain a large amount of information regarding a range of frequencies. So when dealing with such inputs, the analysis of the circuits is getting more complex because the designer needs to think not one sine input but a frequency spectrum. In the case that we want to determine when the length of the trace is critical for mitigation of electromagnetic phenomena, we will introduce the terms **rise time** (time domain) and **bandwidth** (frequency domain).

What is rise time? Rise time usually referred as the time needed to go from 10% of the amplitude of the signal to the 90%. In high speed design, it can be approximated by the 10% of the clock period. In some cases like FPGA and ASIC, rise time can reach 1%! However, wafers could be designed in such a way that can affect it dramatically despite operating in low frequency. It's a function of the logic family and the chip technology. Another rule of thumb claims that it can be estimated as the **7% of the period**. As we said, most of the times rise time will be 10%, but is better to underestimate than overestimate, that's why you could use 7%. Rise time is essential as an input for design strategies and the why will become clear soon [9].

From the time domain we will jump to the frequency one. As we have said, a pulse can be described as an infinite sum of sinusoidal functions, but the **infinite** term is quite troublesome for the analysis. Do we need all the harmonics for an adequate representation of a pulse? We can actually neglect some frequency components due to the very low magnitude. In other words when they are not significant. Hence we can define **bandwidth**[6] as the highest sine-wave frequency component that is significant

[6] The term bandwidth is used as the highest frequency component because in the frequency spectrum digital signals start at the DC.

in the spectrum [9]. But how to define the "significant". There are some variations regarding this:

First approach by Clayton R. Paul. If we are going to plot the frequency spectrum of a trapezoidal signal we can observe that while the frequency increases, the magnitude decreases. Precisely in the figure 16 we see that after the breakpoint $1/\pi\tau$ the levels of harmonics are rolling off at a rate of 20dB/decade, then at the second breakpoint at a rate of 40 dB/decade. If we go past the second one by a factor of about 3 to the frequency that is the inverse of the rise and fall time, $f = 1/\tau_r$ then the levels of the component we will be reduced further by 20dB. This breakpoint will be $f = 3 * 1/\pi\tau_r \approx 1/\tau_r$. Hence we can claim that above this frequency the components will be negligible and we define the bandwidth of the trapezoidal clock as [23]:

$$\text{BW} = \frac{1}{\tau_r}$$

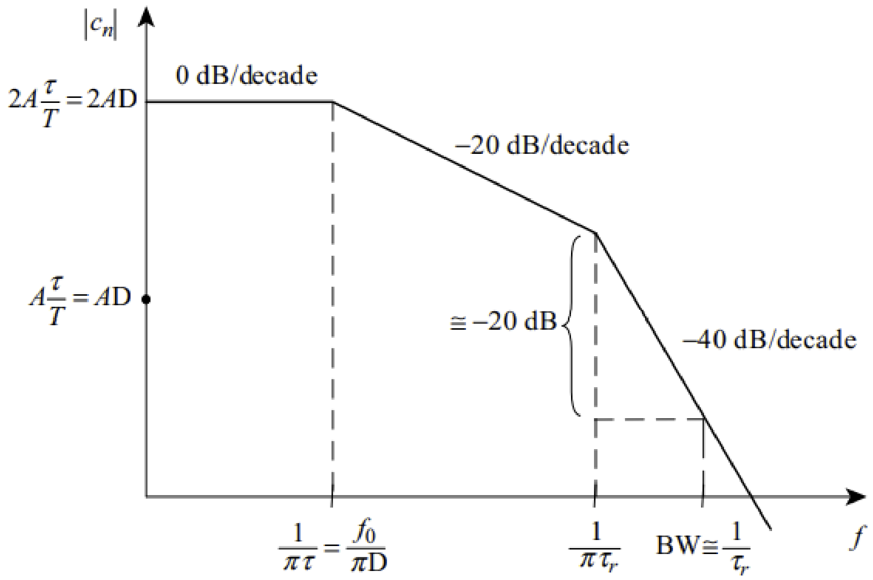


***Figure 16**: Bounds on the spectral coefficients of the trapezoidal pulse train for equal rise and fall times [23, p.16]*

Second approach by Eric Bogatin. Let's try to re-create an ideal square wave (zero rise time) by adding harmonics. We can see in figure 18 that whenever we add more harmonics, the rise time becomes shorter and a trapezoidal signal is starting to take shape. Eventually the trapezoidal could become an ideal square if infinite harmonics were added. We are not interested in an ideal square, but in the trapezoidal ones that resembles the digital signals. So, for these signals, from which frequency and after, adding harmonics isn't significant anymore? Let's take the harmonics of an ideal trapezoidal signal and a square one. We are looking for the harmonic of the trapezoidal that its power is 50% less than the power of the same harmonic of the square wave or when the amplitude is 70% less. Then this harmonic is the highest frequency component and thus the bandwidth is defined.

So we are actually using the harmonics of a square wave to build a trapezoid signal

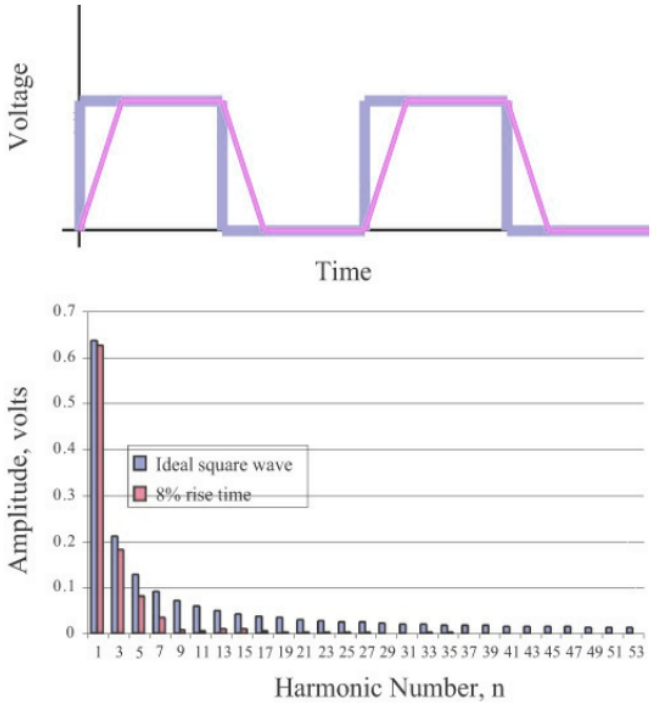


Figure 17: *Comparing the amplitudes of the harmonics between square and trapezoidal signals [9]*

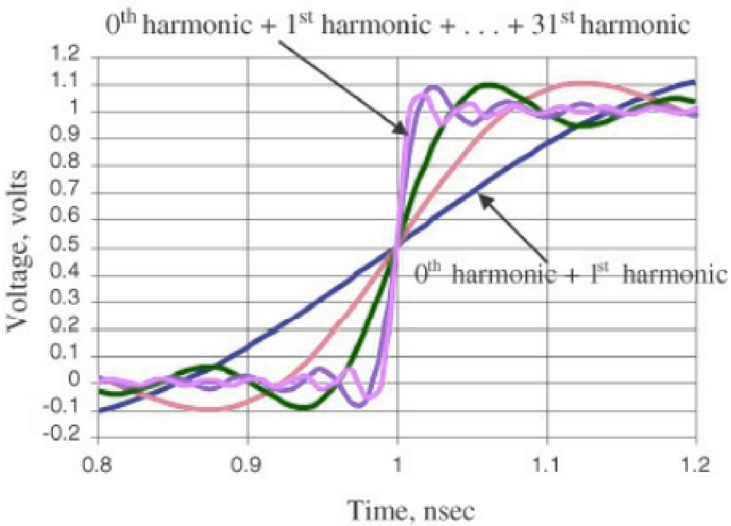


Figure 18: *[9]*

but the the highest frequency component is found by comparing the amplitude or the power of the harmonics of a trapezoidal and an equivalent square wave. According to this there is a linear relationship between bandwidth and rise time depicted in figure 19. Mathematically can be described as [9]:

$$\text{BW} = \frac{0.35}{\text{Rise time (10\%-90\%)}}$$

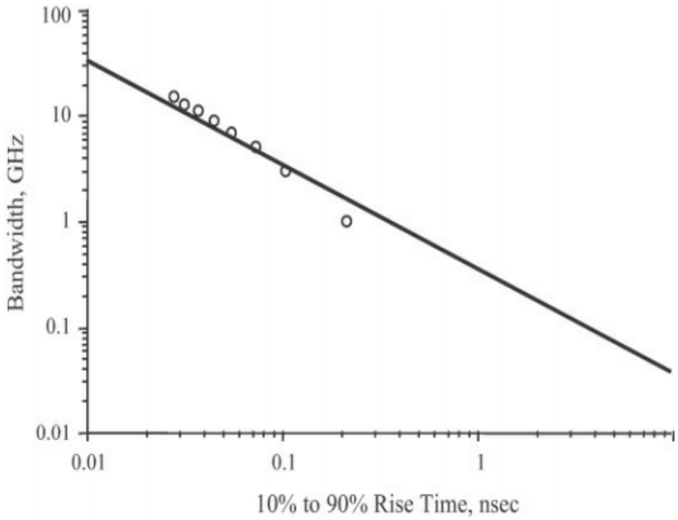


***Figure 19**: Linear relationship between bandwidth and rise time of trapezoidal signal [9]*

Third approach by Howard Johnson. Let's take the power spectral density of the signal depicted in figure 20. We can see that from the clock frequency until the so called **knee** frequency we have a 20dB/decade slop. Beyond knee frequency the amplitude rolls of faster. At this particular breakpoint the spectral amplitude is down by half (-6.8 dB) below the 20dB/decade slop. Thus we can claim that most of the energy in digital pulses is concentrated to the frequencies from the DC to the knee frequency. How to find the knee frequency based on the rise time, though? There is this formula [17]:

$$F_{\text{knee}} = \frac{0.5}{\text{Rise time (10\%-90\%)}}$$

With these three methods we are able to measure the highest significant frequency component of a digital signal. All the sine waves from DC to that frequency is important for our design, so the transmission line should be able to handle this range of spectrum. If a portion of this frequency range falls of to the category of transmission line as it was defined with the rule of thumb for the wavelength per sine wave, then high speed design guidelines should be definitely considered.

4.2 A transmission line problem

Let's see an example to comprehend the transmission line problem in digital signals. We will be based to the first approach to estimate the bandwidth. So, let's assume that the voltage source depicted in figure 21 is a clock waveform of 5V, 50 MHz, 50% duty cycle, rise time 0.5ns and the length of the interconnection is 2 in (0.0508m). The bandwidth of this waveform is

$$\text{BW} = \frac{1}{0.5 * 10^{-9}} = 2\text{GHz}$$

Let's calculate what should be the frequency of a sine wave in order for the intercon-

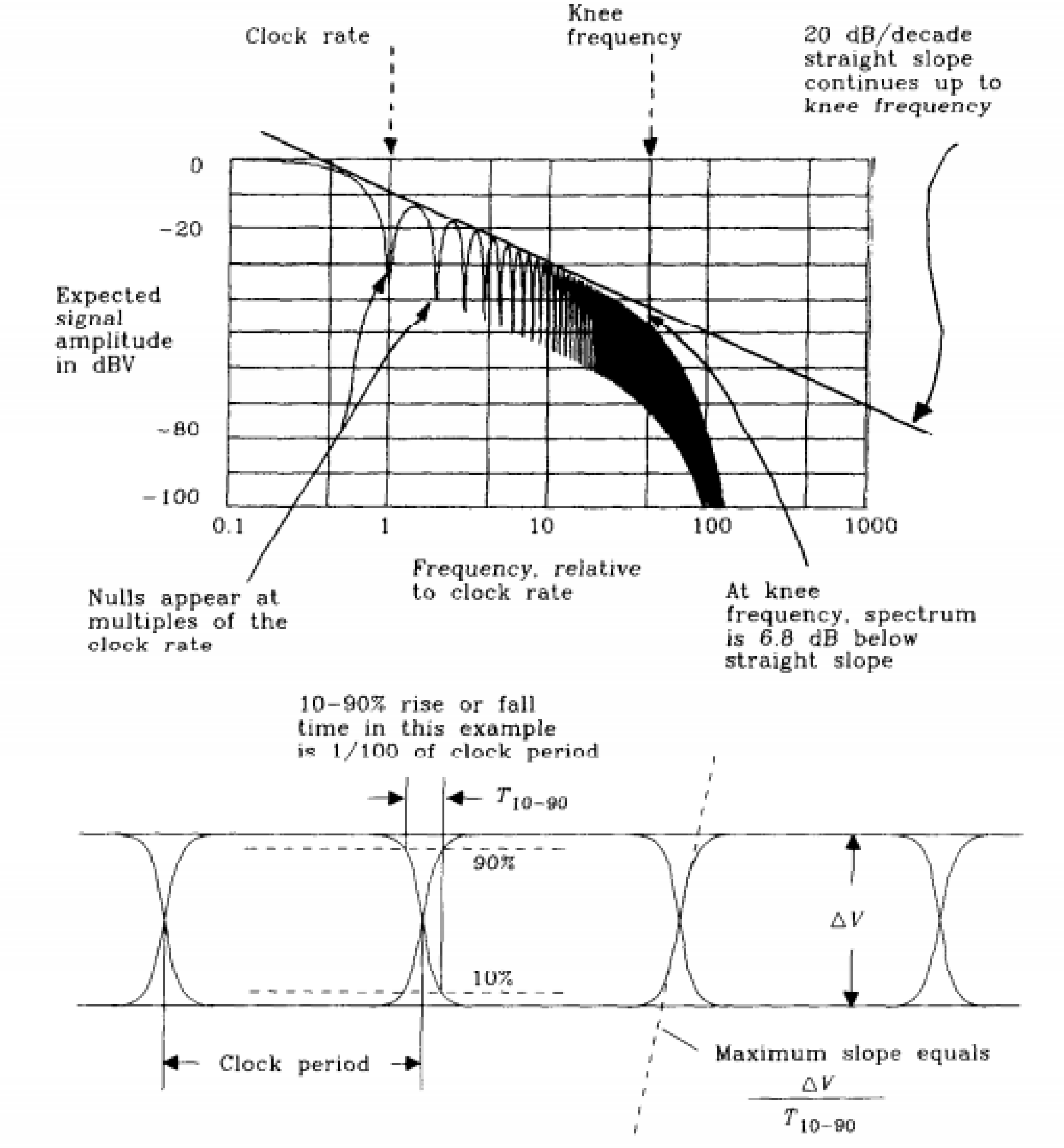


***Figure 20**: Eye diagram of a digital signal and its power spectral density [17, p.3]*

nection to act like an electrical short according to the rule of thumb of $\lambda/10$. So we set $\lambda/10 = 0.0508$m. We assume FR4 as dielectric ($1/\sqrt{\epsilon_r} = 0.47$), so

$$\lambda/10 = \frac{c}{f * \sqrt{\epsilon_r} * 10} = \frac{3 * 10^8}{f * 10} * 0.47 \rightarrow f = \frac{3 * 10^8}{0.0508 * 10} * 0.47 = 277MHz$$

We have a signal that its frequency clock is 50MHz and the bandwidth 2 GHz! For the range 277MHz - 2GHz the interconnection isn't electrical short and the distributed model should be introduced. So definitely for this, transmission line effects should be taken into account. As you can see the clock frequency of 50 MHz doesn't tell much for the criteria to be or not be transmission line...

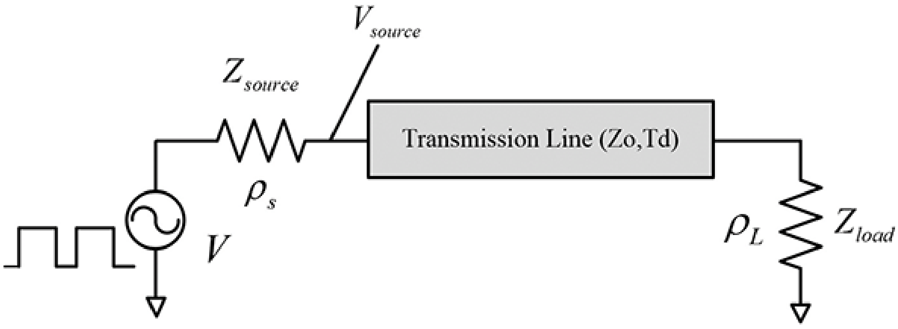


***Figure 21**: image source*

In a different way we could calculate at which length of the interconnection you should start worrying and similarly we have (f = highest frequent component, BW = 2 GHZ):

$$\lambda/10 = \frac{c}{f * \sqrt{\epsilon_r} * 10} = \frac{3 * 10^8}{2 * 10^9 * 10} * 0.47 = 0.007 \text{ m} = 7 \text{ mm}$$

Thus for length below 7 mm, we can estimate that the interconnection won't behave as a transmission line [23].

4.3 Lumped vs Distributed

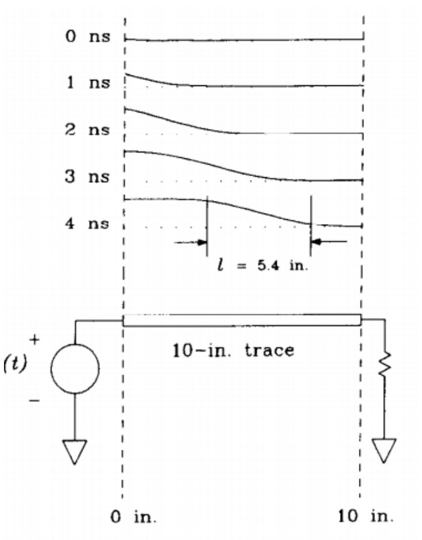


***Figure 22**: Length of the interconnection and the length of the rising edge [17, p.8]*

Another method proposed by Howard Johnson for identifying **lumped vs distributed systems** is the following:

Everything takes times and nothing can travel faster than the speed of light. So the signal from the driver to reach the load will take some time, there will be a delay. This delay is proportional to the length of the trace, but during the delay there is a possibility depending the rise time to fit a portion of the signal to this segment of trace and wave nature is coming to the surface. The rising edge is the factor that determines if the signal can fit to the segment of the trace. We can define the length of the rising edge (in m) as:

$$l = \frac{T_r}{D}$$

where T_r is the rise time (10%-90%) in ps and D is the delay, how much time took for the signal to move per unit length (ps/m). **For circuits smaller than $l/6$ are lumped** [17].

5 Design for Performance

Design for Performance is referred to electrical performance and it becomes crucial in high speed applications when a lot of guidelines should be implemented to meet the requirements. High speed design is something that you can't explicitly define. We mentioned some rules of thumb for the transmission line and the distributed model, but in general in the MHz region and above you should treating lines with a careful mindset[7]. It should be noted that every pair of signal and return path can be defined as a transmission line. We may keep the interconnections small and not having reflections and this is good but just a subset of the signal integrity.

Question: Even if I have very short rise times, but the pin changes its state not very often (it isn't periodic like the clock that change its time very frequently), can I assume that it isn't a critical signal eventually [15]?

First we categorized the problems in SI, PI and EMC. A more practical way to categorize the issues in the design is the following [10]:

- **E**lectro**M**agnetic **I**nterference (radiations beyond the board or susceptibility to radiations from outside the board)
- **Reflections** on a single net
- **Crosstalk** between two or more nets, in many ways a special case of EMI
- **Power system stability**

5.1 Forget the Ground think Return

In high frequency signals where voltages and currents changing/oscillating, there aren't any shorts or opens like we are used to. Even two conductors with voltage difference separated by a dielectric can act as a very low impedance path and definitely not something that current can't flow (displacement current). Another thing that we need to understand is that **current flows in loops** and always trying to find the path with the **lowest impedance** (yes impedance and not resistance. In higher frequency reactance dominates resistance). This path will be the adjacent copper area because the loop is

[7]It is should be noted that the voltage of the signal is quite important for the radiation. A fast rising time of a 5V signal would have more energy than a 3.3V. This is why the **slew rate** (how fast the voltage change) can be actually a better indicator of signal criticality.

smaller, the capacitance is higher thus the impedance is lower. So it is considered best practice to place the ground plane very close to the signal.

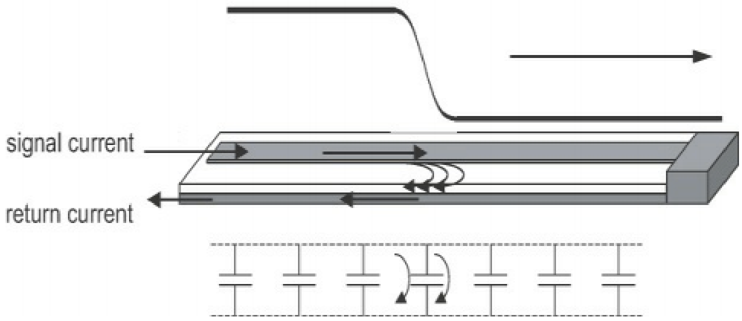
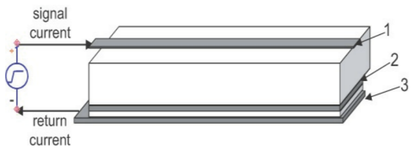
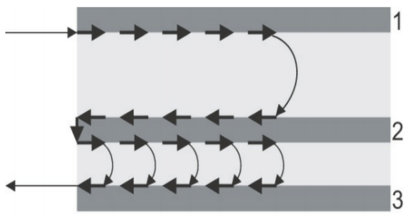


***Figure 23**: Capacitive coupling of transmission lines [9]*

But if we think that everything is fine by connecting the return path to the ground plane, this is where a lot of problems arise. As we have said the current will find the path of the lowest impedance. If for some reason there is copper area between the signal and the ground then the return path will not be in the ground but in the copper between them. In other words the current will follow a radically different route from what we estimated with side-effects such as overlapping currents, crosstalk, distortion! In the end, the current should return to ground. So a very important rule is to have an **an unbroken dielectric** between the signal and the return.



(a) Copper layer 2 disrupt the signal [9]



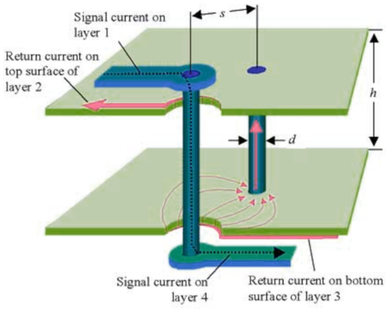
(b) Current return [9]

Figure 24

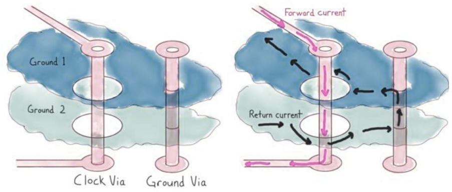
5.1.1 Changing trace layers

And if I have to change layers how to follow the rule of the unbroken dielectric? What will be the return path? This is quite interesting actually and in order to visualize it let's inspect the figure 25. In this figure we assume that layer 2 and 3 are reference planes with the same potential, thus we can connect them using a via. But before talking about the via, we can observe that in the layer 1 and layer 3 the return current follow the principle of the adjacent layer as we expected. The big challenge is what is happening between the 2 and 3, how the current can return? Thus we put the via to provide a path for the return current to transition from the 3 layer to the 2.

Let's now see the figure 15, without using a via and having as planes the ground and the power for the 2 and 3 correspondingly. How the current will return? It will spread out around the clearance created from the hole and will try to make use of the capacitive coupling between the two planes. Why it spreads? For lower inductance and for higher capacitance. This discontinuity though it will increase significant the impedance causing a voltage drop in the ground plane the so called **ground bounce**. For this case we could place a bypass capacitor near the via that will act as a low impedance path for the current (be careful with the loop inductance, the bypass capacitor eventually may not act as one). We could also minimize the clearance of the hole, making the via smaller to reduce the voltage drop. The distance between the two plates in order to increase the capacitance and lower the impedance should be kept as small as possible.



(a) *image source*



(b) *image source*

***Figure 25**: Return current when changing layers using ground transition via*

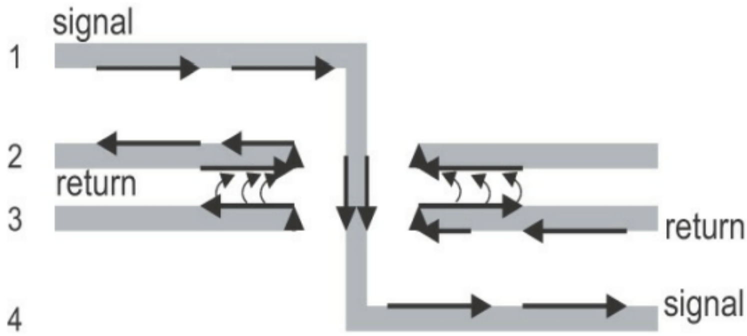
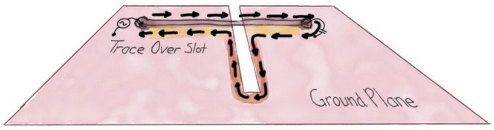


***Figure 26**: Return current without via or bypass capacitor [9]*

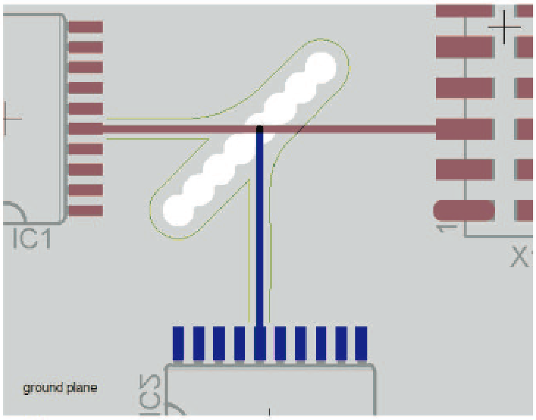
5.1.2 Slots

Another thing encountered in PCB designs is a disrupted return path with slots in the ground plane. If the return path is disrupted, the current will flow around the loop, thus the inductance increase resulting to more emissions and voltage drop (ground bounce). The field of the signal path during the slot isn't canceled by the return path. To calculate the inductance of these slots for accuracy you could use 3D field solvers. In general the bigger the area of the loop, the higher the inductance and the more the problems. It would be best to avoid slots or don't route signal over them or if you can't do anything you could place a bypass capacitor as a path of low impedance [3, 10].

Also these type of slots can be created by holes too like vias. If vias are placed very close to each other the clearance will create a gap without any copper between them. This will result to higher impedance that is of course undesired. So be careful with the clearance of the holes. Make them smaller or further the distance [16].



(a) *image source*



(b) *Vias clearance cause slots in the ground plane [16]*

Now that we analyzed the slots we will introduce another very important rule like the unbroken dielectric, **the unbroken return path**.

5.1.3 Summary

What should I keep? Two important rules for high speed design: 1) Unbroken dielectric and 2) unbroken return path. To put it in another way, you should remember: **IT IS ALL ABOUT THE FIELDS**. Start thinking in pairs of conductor and dielectric, not per signal trace [22].

Identifying the return current can solve a lot problems. Where is the source, the load and the return.

The aforementioned guidelines fall of to the category of solutions to mitigate EMI and rail collapsing noise (subset of power integrity).

5.2 Reflection

As we have said in the section about the design identification, the signals can have wave nature. Precisely, we analyzed when these kinds of reflections should be significant by comparing the wavelength of the signal with the length of the interconnection. In the case we have transmission lines and reflections, what should we do to avoid problems such as ringing, false triggering, crosstalk and distortion? The answer is **impedance matching**.

Reflections happens each time the signal face an impedance discontinuity along its way of propagation. These discontinuities can be via, junctions, traces with different geometric shape, connectors, pins of ICs etc. Our goal it to shape a path for the signal with a constant impedance along the way. But we need first to define what is **characteristic impedance** for a transmission line. It is the constant, instantaneous impedance that the signal looks. It can be defined as the input impedance of an infinite transmission line. It should be noted that there are many formulas that can calculate the characteristic impedance of any type of transmission line. In our case we are mostly interested in microstrips. Striplines, on the other hand, when EMI and crosstalk are serious issues, is a better choice.

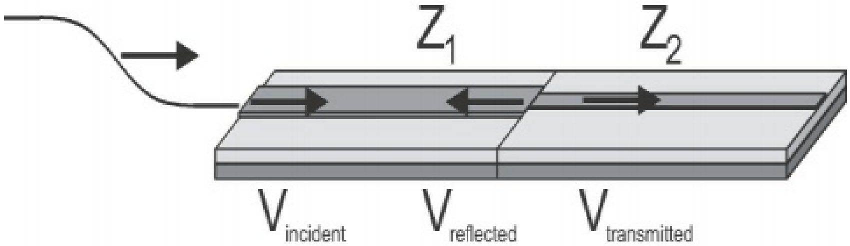


***Figure 28**: [9]*

Our goal it to match the impedance of the transmission line with the source impedance of the driver and the load (but actually only for the impedance of the interconnection we have control). Let's suppose that we have a driver, a load and an interconnection (a uniform transmission line without stubs) and we want to design the transmission line so there aren't reflections. First we need to know the source impedance and the load impedance. For a typical CMOS device that drives the signal we suppose that we have impedance of 5-20 Ohms. Similarly for the load we suppose that we have a capacitor. In most CMOS reveivers the capacitance value is very low so it can be approximated as an open circuit. The source impedance can be extracted by IBIS models or by datasheets too. So how we do the impedance match?

Usually the microstrip traces are routed in such a way adjusting the width to have characteristic impedance **50 Ohms**. In RF application, with coaxial cables and such, the traces on the PCB are designed with a 50 Ohm characteristic impedance. This is a well know standardized value for cables and RF chips. It is a convention in order to help different vendors to design their products having this value in their minds for impedance matching between different products. But in the case of PCB tracks, the density is very high and designing 50 ohms trace isn't often very convenient. If the design isn't dependent of a coaxial cable or something that forces the trace to be 50 Ohms, can I still use a different characteristic impedance?

The goal it to match the impedance. So if we had a 100 ohm characteristic impedance and the source impedance is 10 ohms, by connecting a 90 ohms resistor in series with the driver then we have impedance matching. However with this configuration, we created a voltage divider and the voltage waveform in the transmission line will be actually half of the intended. But due to the 100% percent reflection[8] at the end of the transmission line, the total waveform by adding the reflected will be V volts and when the reflected wave reach the source it will not be reflected back because of the 90 + 10 = 100 impedance.

It should be noted that terminations and impedance matching is a quite huge topic (we didn't even scratch the surface!). It is recommended to read also the documents provided by the vendors that will suggest the best way for the termination of the critical signals. For example, regarding the high-speed USB, the impedance for the differential pair should be 90 ohms.

[8]When a transmission line is opened from the load end then all the forward signal is reflected back to the source

If you want to connect multiple receivers then daisy chain routing techniques is recommended in order to keep the characteristic impedance controlled without making branches and stubs [9].

Question: If I have a via along the interconnection path? What should I do?

5.3 Crosstalk

When there is current there is magnetic field, when there are charges there is electric field. When current change, magnetic field change and this can induce current to nearby conductors if the magnetic rings spread by the source are shared with them. Because of this mutual inductance and the **inductive coupling**, noise is produced. This is called switching noise (because happens during the switching part of the current, during the rise time). Opposite current can be induced to the same conductor that created the current in the first place too. The amount of the voltage noised is determined by the total inductance (the total amount of the rings that surrounds the conductor) and of course the rate of change. This is the so called inertia, it takes time to build up current.

On the other hand when charges are oscillating, they create changing electric fields that can kick charges of the adjacent conductors and current shows up (displacement). This is called **capacitive coupling** and it is based on the mutual capacitance between two conductors. All of the aforementioned regarding the induced voltages and current when EM fields change, can be described by the notorious 4 Maxwell equations.

The change of these EM fields is actually the root cause of everything. Thus coupling happens during the edges, the rise time. When the coupling is undesired we call it crosstalk. Our goal is to have high capacitive coupling and low total inductance for the signal and the return and low capacitive coupling and low total inductance to unrelated signals.

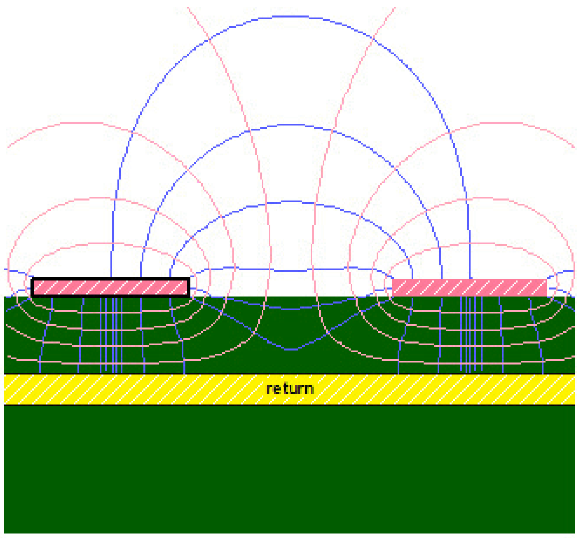


***Figure 29**: Microstrips, fields overlap (image source)*

In this kind of topic usually authors write about forward, backward and far and near crosstalk. We won't go into too much detail. The concept is that in an environment that we have two microstrips and a signal is propagating to one of them, then there are two directions that induced current can flow, the forward (the same with the signal that caused the coupling) and the backward (the opposite of the signal) and two types of noise, the near (lower in magnitude but last longer) and the far (higher in magnitude but lost shorter). The near is at the start of the trace close to where the signal started to propagate and the far is at the other end where the signal reaches the destination.

In the bottom line there are 4 main things that determine the coupling:

- Width.
  About the inductive part, the less the density of the current the less the inductance. So increasing the cross section of a conductor also leads to a lower self inductance. This is why ground plane has low inductance too. By increasing the width of a signal, the capacitive coupling between the signal and the plane is increased too.
- Space.
  Bring the return plane as close as possible to the signal to minimize the area of the loop (field cancellation, lower total inductance, less crosstalk and emissions) and further the distance of unrelated signals as much as possible, considering the density of the interconnection and mechanical constraints. We know that field intensity is proportional to the inverse square of the distance.
  But why by minimizing the space between signal and return is better? As we can see there is field cancellation and the total inductance is low.

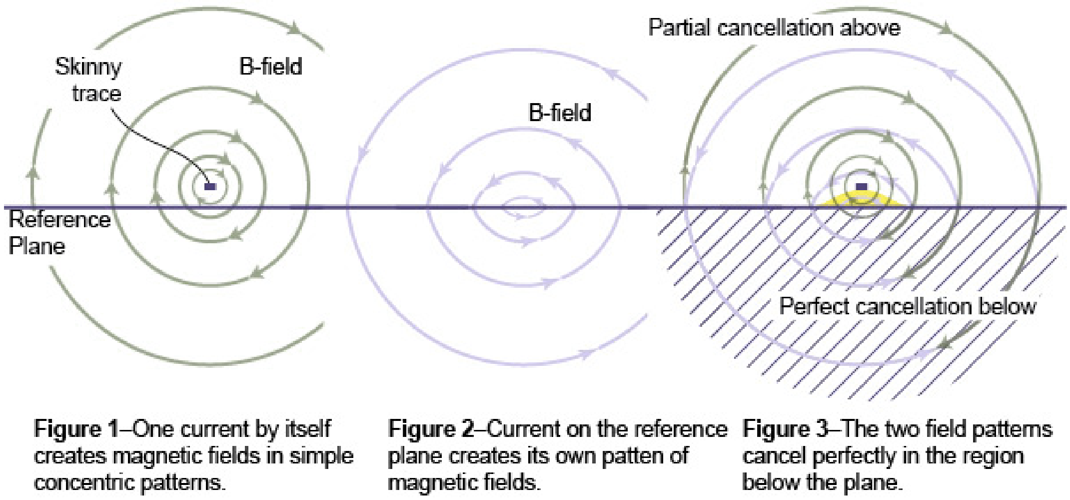


***Figure 30**: Field cancellation (source)*

- **3H rule.**
  There is also the 3H (three times the H) rule that indicates the required distance between an aggressor and a victim trace. The H is the height of the dielectric.
- Coupling length.
  Reduce the length that unrelated signals run in parallel and in general for all the traces make them as short as possible (low self inductance).
- Keep rise time high.
  As we know it is all about the rate of change! The higher the rate, more problems occur overall. But it isn't something that in most of the cases you can change.

The interesting part of crosstalk is how to simulate it. Usually 2D field or 3D tools are used to calculate the capacitive and inductive coupling and then these measurements are integrated to the modeling process.

But how much crosstalk is too much? In order to find this you need to define the noise margins and the tolerance of your design. Then with the help of superposition and with modeling you could approximate how much noise it will be coupled [21, 9].

5.4 PDN

Power Distribution Network (PDN) is the root of the power integrity. It is a fundamental part of any product and it plays a very important role in the overall performance of the board. In this network are included every part that is related with power distribution like interconnections, planes, bypass capacitors, voltage regulator modules (**VRM**). This network is responsible to feed with the necessary amount of power each component and to cycle the return currents too. It is centralized, high frequency currents are running through the power lines and noise can accumulate to these interconnections causing emissions and functional problems. So it is quite critical to have a clean power supply providing low inductance as much as possible. Planes and decoupling capacitors are essential tools for controlling the PDN.

5.4.1 Planes

Ground

Having a low impedance path (low inductance) where currents are flowing is very crucial when we want to minimize voltage drops and having a clean voltage difference. This is why the ground that acts as a reference plane should be a solid **plane**. Planes are dedicated copper layers that take all the space. The more the copper, the less the current density, the less the inductance. If indeed the return is on the ground plane, then no ground bounce problems! Additionally, the planes have the ability to contain the fields. The fields can't penetrate the copper.

Avoid ground loops, because the loop is susceptible to crosstalk, EMI due to the mutual inductance. Return to the ground with the shortest way by placing vias to the ground plane rather than routing ground traces.

Power

Power plane is recommended in order to provide a low inductance to build up the necessary charge as fast as needed when the switching transistors are calling for it (check the decoupling section to comprehend it). The plane assist the job of the decoupling capacitors.

By using a power plane, there is also a capacitance formed due to these adjacent layers (planar capacitance). Of course it isn't enough to provide the required decoupling the design needs. So don't forget the capacitors. The shorter the distance between the planes, the better though. Also, if for some reason there are unintentional return currents in the power plane, the higher the capacitance between the power and the ground, the lower the risk for integrity issues (subsection changing layers).

Planes are quite useful for controlling the impedance when reflections matters and by closing the distance between signal and ground, EMI and crosstalk are lessened [10, 9].

Question: Should I use a ferrite bead to filter the power supply?

5.4.2 Copper pours

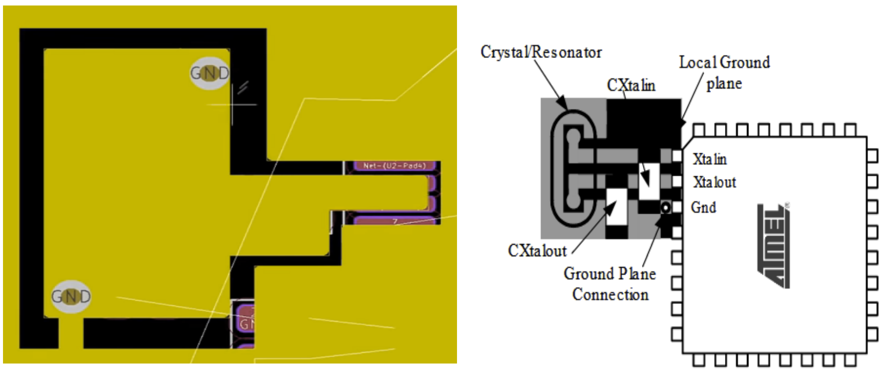
Let's suppose that we have a four layer PCB and we have already set two dedicated layers, the inner, for the GND and the power. Would it be beneficial to pour copper to the top and the bottom where traces and components exist and use via stiching[9] to connect the pour with the planes? Can nearby conductors cause undesired coupling in the pours? Am I gonna create an antenna? What about EMI?

5.4.3 Clock signals

Now that we mentioned about the importance of planes, let's check a special case of a critical high speed signal that is the heart of digital applications, the clock signal.

It is recommended not to use the same ground plane with the rest of the circuit but a local/isolated one. Clock signals are consisted of high frequency components. Running such a high frequency current in a huge copper area like a plane that can be comparable with half the wavelength, could result to a **center-fed patch antenna!** Also don't route signals near the clock traces and avoid vias [2, 1, 7].

About resources, usually application notes can be easily indexed by "Best practices for the PCB layout of Oscillators".



(a) *Isolated ground plane should be connected to one point with the main ground* *image source*

(b) *[7]*

Question: This discrete, isolated ground for the high speed signal of the clock traces can be used as a general technique for high speed signals?

[9]Via stiching is a technique of spreading across the board vias to tie electrically big copper areas.

5.4.4 Decoupling capacitors

Another huge factor that contributing to the power integrity of your design is the decoupling capacitors. It is worth mentioning that there are also the bypass capacitors and these terms are used interchangeably. So the decoupling/bypass capacitors are used to clean the power line from noise (low impedance path) and to provide a short burst of energy when the fast switching request power to drive the current, something like a temporary battery. That's why for the decoupling, in order to cover a wide range of frequencies that needs to be shunt and for energy supply reasons, we pick more than two capacitors. One for low frequency and one for high. Most of the times the vendors will provide the necessary information about the values.

As we can see in the figure 32, if we didn't place a bypass capacitor, the urgent need of transistors for the charge to build up isn't satisfied by the power supply due to the inductance, the inertia of the power line. So we need to use a temporary power supply until the time that the system can reach the desired amount of current.

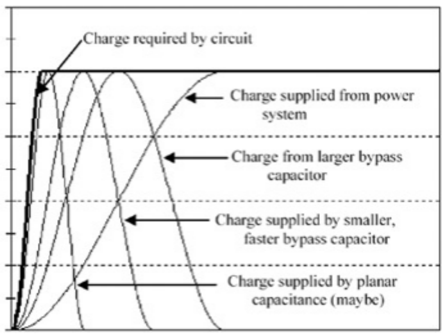


Figure 32: *Charge requirements of switching transistors [10]*

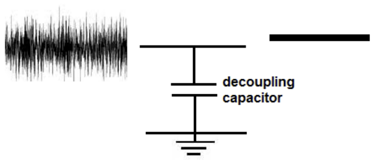


Figure 33: *Smoothing the signal (image source)*

Another important thing for the decoupling capacitors is to place them **as close as possible** to the power supply pins of the component that you are trying to decouple. The trace width should also be increased for the purpose to minimize the inductance and to provide the energy as clean and fast as possible. If you have two capacitors with different values, then place the smaller one closer [10, 16]. An interesting article for optimizing the decoupling capacitors location can be found here.

5.5 Parasitic elements

Not everything is what it seems! So far we said about the capacitance and the inductance of interconnections, but this also includes passive components, leads, connectors and so on. So it is quite important to be sure that you made the right choice of values for the passive components for the range of frequencies that you are interested along with proper PCB layout. Related to this frequency response is the so called self resonance, where for example capacitor acts like a complete pure resistor.

Most of the times we are referring to the magnitude of the impedance like high and low, but we are not mention anything about the phase. Should I care about phase? The digital circuit will work, but the analog components usually care about a lot about the

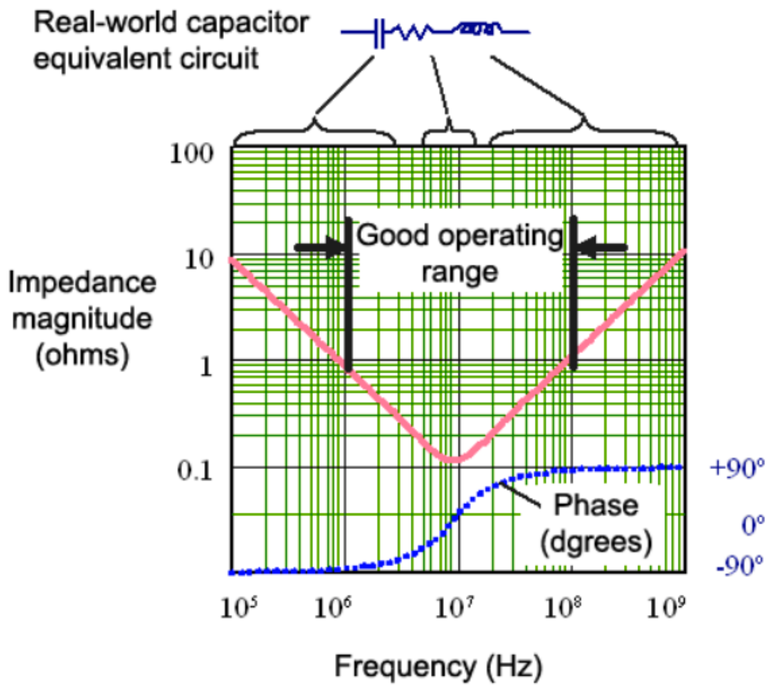


***Figure 34**: [12]*

phase so keep that in mind. Things change when frequency varies and remember in digital circuits we have a bandwidth and not a single frequency [12].

5.6 Component Placement

Before implementing any of those guidelines, the first thing that designers do is to place the component. Placement is referred as if not the most then one of the most important parts of the design. Some guidelines could be [5, 14, 8, 27]:

- Give room to breathe! Consider the density of the traces that will run across the board.
- Partition the design (RF, digital, analog). Group similar parts together.
- Connectors should be placed to the edge of the board away from circuitry. Input/Output pins is a very common way for noise to be coupled on and off the board. So for this reason should be placed away from the rest of the circuitry, in the edge of the board.
- Components that are **thermal** critical should not concentrated in a the same area in order to avoid hot spots. So distribute them and don't place them near the edge, because the heat removal won't be the best.
- Place decoupling capacitors as close as possible to power and ground pins.
- Check the **ratsnest**[10] and find the most efficient way for the traces to be short and with less vias.
- Design for assembly. It is generally preferred to have the IC the same orientation but it isn't mandatory.

[10]Ratsnest is a bunch of air wire connections that show the distance of the connections in the PCB layout.

- Sometimes rotating the IC with an angle of 45 angle can help in the next steps of routing.
- Don't forget the mechanical dimensions. Components should not overlap.

Finally an objective/artistic comment is that if it looks good it will work!

5.7 Routing

How to connect something? Let's have an overview of the different ways of routing. Namely we have:

- Daisy chain
- Point to point
- Star connection
- Bus

5.8 Mixed design

How to approach design including analog, digital or RF components? Should I split the ground? Be aware the return paths, the slots and not to overlap currents from different functional groups.

5.9 Mounting holes

Why there is clearance on the mounting holes? Should I ground them? Should I put vias on them?

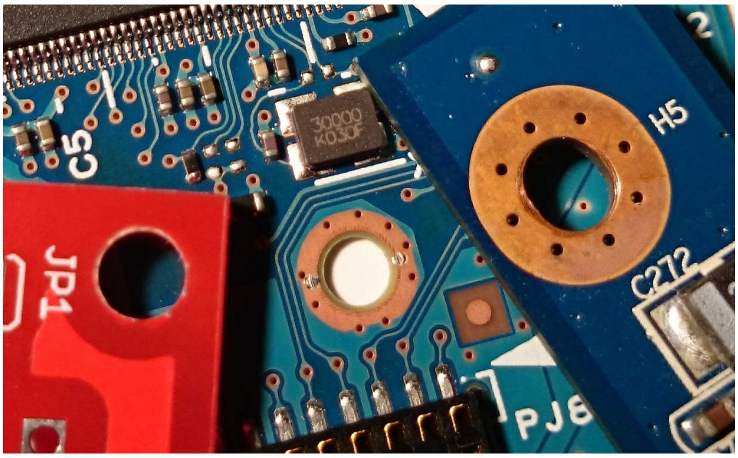


***Figure 35**: Different types of mounting holes (image source)*

5.10 Unused pins

Most of the times the datasheets will provide information what the designer should do about the unused pins, the floating pins of an MCU for example. The common solution is to tied them to ground, providing a low impedance path. But is this always the case? Is a myth to tie every floating pin to ground [18, 27]?

5.11 External links

- A very informative answer in electronics stackexchange

6 Design for Manufacturing

In order for the PCB to transition from the Gerber state to reality, it should be manufacture-able! Therefore DFM (Design for Manufacturing) guidelines should be taken into account early at the design stage[11] to avoid product failures and save time and money.

Even if something can be fabricated, small changes in the layout can reduce the total cost. This isn't always the case for small quantities but at mass production scale, even the smallest nuances will increase it significantly.

Usually EDA tools feature **DRC** (Design Rule Checking) tools, that automatically inspect the board to determine whether it meets the constraints imposed by the manufacturer. For instance, Eurocircuits offers some templates to import these scripts to the EDA tool used by the designers (for this case the supported ones are KiCad, Altium and Eagle). It should be noted that DRC is usually checking for trace width, spacing and enclosure, but this is only a subset of the DFM. So it is quite important to have an overview of the methods that are going to be used for the fabrication of the PCB. DFM is sometimes referred not only to the bare board production but also to the assembly (Design for Assembly). Actually, DFM and DFA automatic checks are the first things that the manufacturers do.

It is recommended to know in the design phase how your PCB will be assembled to integrate the corresponding considerations and rules. Some guidelines for DFM outside the scope of DRC could be [20, 8]:

- All component outlines on your silkscreen should be marked with a reference designator and polarity (pin 1 marker) indicators.
  About polarity, there is also the option to export a dedicated layer for this purpose from your EDA tool (Fab layer for KiCad) as we have already mentioned.
- Prefer to place all the components to the top side of the board. Double sided PCBs are costly. If component placement is done with automatic pick and place machines, then additional cycle is required in the assembly line to flip the board and do the placement on the bottom side.
- Orient similar components in the same direction. It is easier for inspection and testing. It looks nicer too!
- How my PCB is going to be assembled/soldered? Manually or by automated machinery? Reflow or wave soldering?
  If **wave soldering** is going to be used then for optimal soldering the designer should consider: 1) The SMDs are aligned perpendicular to the direction of the board going through the wave, 2) Large components should not "shadow" smaller

[11]The design stage is the most cost efficient stage to detect and fix potential problems

ones and 3) Dual in line packages such as SOIC have their axis aligned to the wave direction.
In general the assembly house could provide some guidelines analogous to the soldering method.

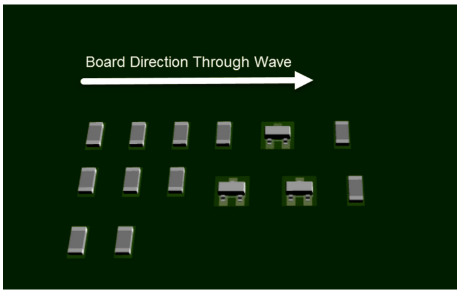


***Figure 36**: Good orientation for wave soldering (image source)*

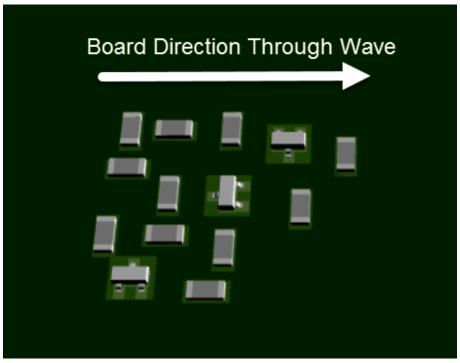


***Figure 37**: Bad orientation for wave soldering (image source)*

For manual assembly, consistency in the component placement can aid a lot the assemblers and make the process less error prone. For example it would be helpful if all ICs is oriented so the pin 1 is located in the same direction.

- Although placing via on pads is great in high speed application, assemblers tend to recommend not doing that because solder can wick into the hole thus a poor joint is created.
- Place 2 or 3 fiducial (reference points for the pick and place machine) marks, which should not be covered with the solder mask. Generally, these should be placed diagonally in the corners of the PCB. The pick and place machine finds these points using a camera and all components are placed on the coordinates relative to these points (tip: The bottom left corner of the PCB outline should be selected as the origin (0,0) point).

It is worthy mentioning that some times **conflicts** emerge when trying to design for manufacturing and performance (functionality) at the same time. Some DFM rules may violate DFP ones and the opposite, that could increase the manufacturing cost. So a **trade-off** mindset is very common! It is also important to contact the assembly house to understand and identify the requirements of the DFM.

Footprint design is also related with DFM. In order to meet manufacturing (soldering, stencil and soldermask) requirements and electric performance criteria, pad size for each layer (copper, stencil, soldermask) should be designed in a suitable manner. For this purpose footrpints are usually compliant with industry standards like IPC-7351.

7 Testing

As we have stated, testing is integrated both in the manufacturing and the assembly part of the PCB development cycle.

Each manufacturer for the bare board production is following certain guidelines to ensure and control the quality of the board. These are based on industrial standards.

For example, Eurocircuits is compliant with the IPC-A-600 Class 2, the most used one in the industry. In summary, this manufacturer combine automatic optical inspection, flying probe testing (doesn't need test fixture) and human inspection [11]. A more detailed approach for the Eurocircuit's inspection workflow can be found here.

Once the board is fabricated, the next step is the assembly process (PCBA). The assembly house is responsible to populate the bare board with components and to detect and fix the defects (e.g. solder bridges, misplaced component) along the way. Methods used to spot defects are typically:

- Automatic optical inspection (AOI).
- X-Ray inspection, also called AXI.
- Flying probe testing. Probes moving around the board trying to contact the test points and components.
- JTAG Boundary Scan. Actually JTAG isn't used only for programming and debugging MCUs, but it was created initially for assembly testing.
- Functional testing.

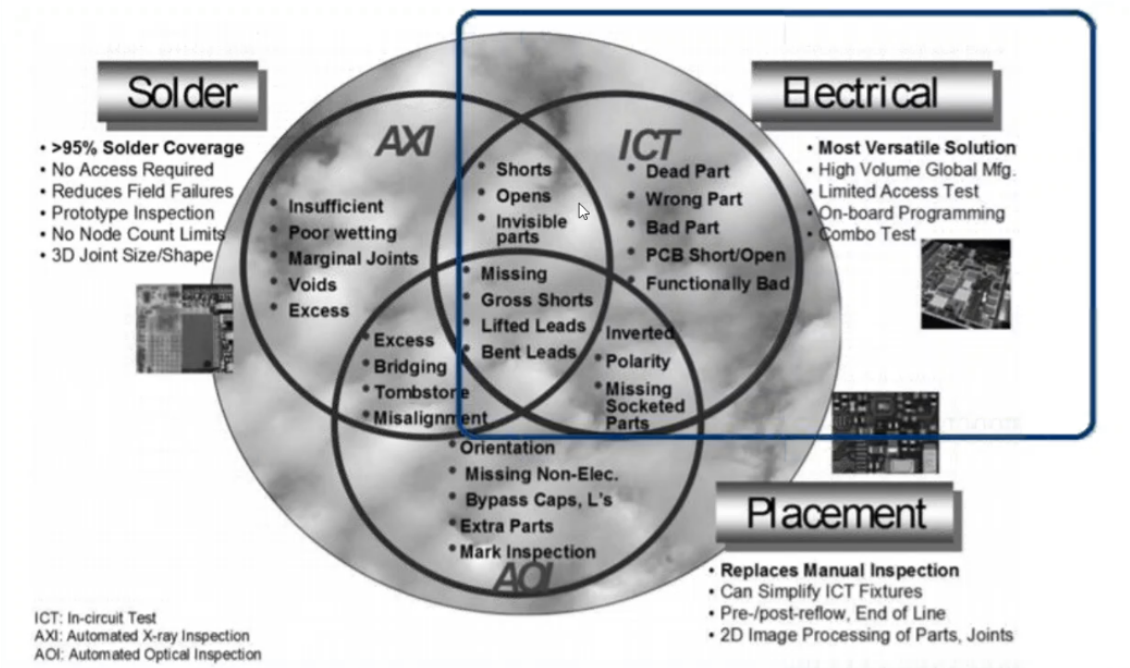


***Figure 38**: Testing overview by SMTA/TMAG TP-101E standard*

But there is also a number of defects that can't be spotted by visual inspection (accessibility limitations), so a more complex testing method along with design requirements (test fixtures) is coming to the surface.

This method is called in-circuit testing (**ICT** aka bed of nails) and can ensure that the board can move in the production line with zero defects. These defects include solder bridges, shorts, opens, resistance, capacitance, missed components etc. In short a group of probes are interfacing with usually one side of the PCB by placing test points[12] to all the nets. For this to happen, a test fixture is required that increases significantly the cost.

[12]Test point: A small exposed copper area used as a connection point to test circuitry on a PCBA

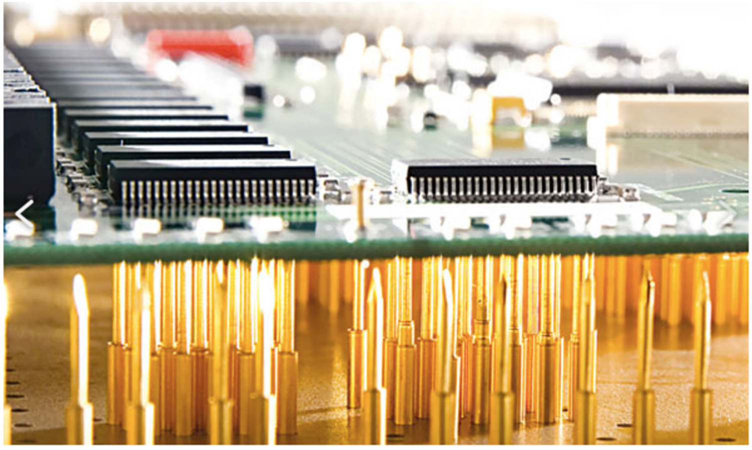


***Figure 39**: A bed of nails test fixture (image source)*

Assemblers charge the customer by the type of the testing service and by the hours needed which is a function of volume. In general testing is an expensive part of the cycle. The cost for ICT is on average 20.000 dollars [28]! Testing for electric performance (verify proper operation for analog and digital circuits) could also be integrated in the provided services as part of the ICT by performing power up tests. It can test for functionality as well as assembly defects.

On the other hand, bed of nails technique has **limitations**. For example in packages like BGAs and in high density boards, placing test points is very inconvenient. So JTAG interface comes for the rescue and aids the assemblers for the testing process. Requirement for JTAG boundary scan is the device under test to have a built in controller or to be accessed implicitly by another device capable for JTAG interface (e.g. Testing a memory module via an MCU with built in JTAG).

Finally there is the **functional testing** (FCT). Its purpose is to simulate the environment in which a product is expected to operate (Does everything work together?). Functional testers typically use a computer that is connected to test points or a test-probe point in order to perform FCT. It can check if all external analogue and digital inputs and outputs meet the requirements and the specifications. Functional testing can use the JTAG or the test fixture for the interface.

Does it **worth** to include test fixture and the additional **cost**? In circuit testing and flying probe testing is a quite common debate. In general for low volume, prototypes and low complexity boards, flying probe testing is the way to go along with visual inspection and functional testing. Flying probe testing is referred also as a type of in-circuit testing but without the need of the "bed of nails", the test fixture. The drawbacks are: more cycle time, less test coverage (how to test BGA with flying probes?), no power up test etc. But usually a combination of methods can also be used [19].

7.1 Design for Testing

Among the first things that the designer should definitely ask is how the board is going to be tested (planning ahead is crucial). There are quite a few rules that need to be addressed for testing, especially for ICT and flying probe testing. Some of them could be [8]:

- Place test points to each net being accessible from the bottom or the top of the PCBA. Preferably, to minimize the cost, use only one side (no need for an extra cycle in the assembly line)
- Distribute test points evenly over PCB. Don't have too many test points in the same area.
- Design the shape and size of test points according to the probes that are going to be used by the assembler.

A standard recommended for testability guidelines is the "SMTA/TMAG Testability Guidelines TP-101E".

Functional and electric performance testing can also be made manually. In this case, similar with the above guidelines for the automatic testing, the designer should place the test points to the nets of interest being **accessible** from the top or the bottom. The electrical instruments that are going to be used should also be considered to adjust the **shape** of the test points with the probes. Prob tips can help to clip the test points without using hands to handle them. Sometimes for **practical issues**, using vias uncovered with solder mask, can make using electrical instruments more easier than just a flat pad surface. In high speed signals, getting an accurate view of the tested signal via contacting probes to test points can be quite challenging. Thus the electrical characteristics of the probes should be considered. For example, to minimize the inductance of a long ground lead, provide a ground point near the measurement of the signal to make the loop smaller for signal integrity. Finally, place test points everywhere, the more the merrier, but be mindful regarding the following statement.

As we have already mentioned in the DFM, in a similar way the DFT can emerge conflicts with the design for performance (DFP). Creating holes to each net in a tight space isn't always an easy thing to do and in high speed applications, test points can cause performance issues. Some traces due to the above reasons might not be tested at all. Thus design for testing might have some drawbacks but it can save a lot of cost in the long run.

8 Simulation and analysis

Simulations can be thought as a method to predict the future that can save a lot of time and cost in the long run. Imagine to be able to know that your board is functional and meets the specifications only if it was manufactured. Then it would take probably ages to finalize the design. So the intermediate way to increase the confidence of the design is achieved by creating a virtual representation of the product's behavior. The virtual world is now our playground without worrying about the cost or the risk of failure. Basically, engineers are trying to catch up with potential issues that could eventually arise when the design becomes a real piece of hardware.

Creating a virtual world that is identical with the physical one is quite a challenge. Thus for practical reasons (less computational time and resources) it is recommended to simulate only critical traces like clock or other high speed signals.

The simulation can be divided into three main groups [9, 13]:

- Simulate the **circuit model**, a translation of the board layout. Don't forget the distributed model and consider the interconnections as transmission lines.
  SPICE related tools for this type of modeling that are used in the industry are LTSpice, PSpice, Multisim, Proteus and the **QUCS** (open source). Also most of the EDA tools provide built in circuit analysis based on SPICE models. KiCad can support circuit simulation and is based on the ngspice engine.
- **Electromagnetic** analysis with **S-parameters** and a corresponding simulation tool. The S parameters can be obtained either using an electromagnetic simulator having as input a compatible version of the physical layout or can be measured directly using electrical instruments such as vector-network analyzer.
  Some tools to simulate Maxwell's equations are: Ansoft's High-Frequency Structure Simulator (HFSS), Clarity 3D Solver by Cadence, CST PCB STUDIO and some open source ones like openEMS and Fast Field Solvers. Numerical solution tools like Matlab, Octave can be used for modeling with S-parameters too.
- Specialized **signal integrity simulators** like HyperLynx and Cadence.
  These commercial tools have many benefits than the traditional circuit simulators. They can import a trace layout and perform a lot of automatic checks to find and solve problems about signal integrity. They use the IBIS model (we will mention it later).

8.1 Circuit vs EM simulation tools

Neither of these tools individually is enough to have a complete representation of the circuitry and to identify the SI, PI, EMC problems. EM field simulators can handle EMC problems, resonance and non uniform wave propagation and considerations regarding the trace geometry such as how bad is this slot in a specific return path. The circuit simulator can handle switching noise (ground bounce), near field crosstalk, transmission line propagation and reflections.

PCB design is a very complex structure. Usually tools that solve 3D Maxwell equations can handle simple ones and require expertise and a very good understanding of electromagnetism. So the circuit simulation, if you need to choose between these two, in most cases is preferable. It is quicker and easier to use and can offer an adequate representation of the physical layout.

8.2 Circuit modeling

What is modeling? "Modeling refers to creating an electrical representation of a device or component that a simulator can interpret and use to predict voltage and current waveforms". The devices can be divided to the active (e.g. transistors) and the passive (e.g. interconnections) ones. For the first, there are two types of models, the SPICE and the IBIS.

The SPICE model is well known in the world of circuit simulation and is used in the analog simulation tools based on the SPICE engine. However, vendors usually struggling to offer these type of models for their products because precious information can be obtained regarding the design of the IC. Thus without revealing the intellectual property of the product, vendors usually provide the so called Input/output Buffer

Information Specification (**IBIS**) model that run in special signal integrity simulators (or behavioral simulators) of the industry, like the ones mentioned before, while containing only the necessary data for this type of analysis. Simulations tools that can handle these models are mostly commercial and probably the most complete tool among them is the HyperLynx provided by Mentor. However, IBIS models can be useful, even without simulation, by **viewing** their data that contains among others the rise time of signals in the digital ports [9].

These traditional **analog** tools, based on SPICE, can do the work for the common digital scenarios. The question is how to build or find spice circuit models for the transmitter and the receiver? How to approximate the circuit, how to test and model high speed signals? How to integrate the inductive and capacitance coupling to your circuit? Can 2D and 3D fields results interpreted as lumped elements and included in my simulation (because crosstalk can't be seen by a schematic, but affects the design)? Circuit modeling using SPICE models should be a topic for a new report!

9 Rules of thumb

- Keep traces short and wide.
- Minimize the current loop.
- Avoid ground loops.
- Spacing unrelated signals (3H rule).
- Avoid stubs formed either by vias or traces.
- Connectors to the edges.
- Avoid slots in planes or keep them short.
- Use ground and power planes. Use ground as reference.
- Use a different gorund domain for the clock signals.
- Identify the return current paths. Ground transition vias and bypass capacitors as a medium for high currents to return.
- If you can do your job with longer rise time then do it.
- Filtering the PCB supply with inductors and capacitors. Decoupling caps should be close to the pins.
- Minimize the usage of vias.
- First route the critical traces.
- 50 Ohms for impedance matching.
- Use multiple track widths. Especially for power, traces should be thicker (minimize the inductance).
- An old saying is that PCB design is 90% placement and 10% routing.

10 Checklist

- Did you read the datasheets and the docs for the hardware development of each component offered by the vendors?
- Did you know the logic family, the voltages, the current, the rise time? How critical is your design? What are the tolerances, the noise margins?
- Did you partition, zoned your design based on the functional blocks (RF, digital,

analog)?
- Do I need to take into account reflections?
- Did you place decoupling capacitors?
- Did you pay attention to the clock signals?
- Are you sure that everything fits? Mechanical constraints?
- Are you sure that ground is ground?
- Are passive components like capacitors and inductors what they are supposed to be? Parasitic elements?
- Did you run the DRC check?
- Did you design for manufacturing and testing? Who is gonna do the testing?
- Did you route first the critical signals?
- Did you make simulation (circuit and field) to increase the confidence for the functionality of your board?
- Are the assembly and manufacturing data with the proper format and compliant with the needs of the corresponding house?
- Did you document your designing decisions?
- Do you like the aesthetic aspect of your PCB (PCB is art!)? If yes then you are probably going well.
- Do you plan to improve this checklist? (It would be wise to do so!)

11 Endnote

Along the way we have mentioned a lot of rules of thumb but we didn't mention the most important one. **Don't take rule of thumb too seriously**. These are approximations and sometimes can differ a lot from reality. If you want to think something fast then it's okey to use them to guide your decisions. But simulations along with specific data for your application is a more accurate but time consuming process.

12 Further reading

12.1 Recommended Books

Warning, a biased opinion is following: "Signal and Power Integrity - Simplified" by Eric Bogatin is the bible for PCB design.

Most of the following are in the references too:

- Signal and Power Integrity - Simplified (2nd Edition) by Eric Bogatin.
- High Speed Digital Design: A Handbook of Black Magic by Howard Johnson, Martin Graham.
- Electromagnetic Compatibility Engineering by Henry W. Ott
- The Circuit Designer's Companion by Peter Wilson
- Printed Circuit Board Design Techniques for EMC Compliance by Mark I. Montrose.
- Signal Integrity Issues and Printed Circuit Board Design by Douglas Brooks.

- Complete PCB Design Using OrCAD Capture and PCB Editor by Kraig Mitzner
- Transmission Lines in Digital and Analog Electronic Systems: Signal Integrity and Crosstalk by Clayton R. Paul

12.2 Informative websites

- Be the signal. Author: Eric Bogatin
- Signal Consulting. Author: Howard Johnson
- Electromagnetic Compatibility Consulting and Training. Author: Henry Ott
- Singal integrity journal.
- EDN
- All about circuits
- incompliance magazine
- LearnEMC

References

[1] Electronics stackexchange: Competing PCB Crystal layout recommendations. https://web.archive.org/web/20200814100220/https://electronics.stackexchange.com/questions/39136/competing-pcb-crystal-layout-recommendations.

[2] Electronics stackexchange: Decoupling caps, PCB layout? https://web.archive.org/web/20200814100014/https://electronics.stackexchange.com/questions/15135/decoupling-caps-pcb-layout/15143.

[3] Electronics stackexchange: What is radiating on my PCB? https://web.archive.org/web/20200814094321/https://electronics.stackexchange.com/questions/81761/whats-radiating-on-my-pcb.

[4] Mohanad Abdulhamid and Atanasio Mugambi. Design of 2.4 GHz Microwave Bandpass Filter. *Journal of Siberian Federal University. Engineering & Technologies*, pages 773--779, 11 2019.

[5] All about circuits. PCB thermal management. https://web.archive.org/web/20200814192641/https://www.allaboutcircuits.com/technical-articles/pcb-thermal-management-techniques/.

[6] Altium. Rigid-flex Materials Selection. https://web.archive.org/web/20200814082723/https://www.altium.com/rigid-flex-circuits-resources/flex-circuit-materials.

[7] Atmel. Best Practises for the PCB layout of Oscillators. https://web.archive.org/web/20200814191225/http://ww1.microchip.com/downloads/en/DeviceDoc/Atmel-8128-Best-Practices-for-the-PCB-Layout-of-Oscillators_ApplicationNote_AVR186.pdf.

[8] Ema Design Automation. *The hitchhiker's guide to PCB design*. Blurb, 2019.

[9] Eric Bogatin. *Signal and Power Integrity-Simplified (2nd edition)*. Pearson Education, 2009.

[10] Douglas Brooks. *Signal integrity issues and printed circuit board design*. Prentice Hall Professional, 2003.

[11] Eurocircuits. Making a PCB step by step. https://web.archive.org/web/20200814085226/https://www.eurocircuits.com/making-a-pcb-pcb-manufacture-step-by-step/.

[12] Howard Johnson. Operating above resonance. https://web.archive.org/web/20200814192303/http://www.sigcon.com/Pubs/straight/resonance.htm.

[13] Howard Johnson. Planning for Signal Integrity. https://web.archive.org/web/20200814200108/http://www.sigcon.com/Pubs/straight/planningsi.htm.

[14] Todd Hubing. PCB EMC design guidelines: a brief annotated list. In *2003 IEEE Symposium on Electromagnetic Compatibility. Symposium Record (Cat. No. 03CH37446)*, volume 1, pages 34--36. IEEE, 2003.

[15] Texas Instruments. Pcb design guidelines for reduced emi. *SZZA009 November*, 1999.

[16] Texas Instruments. High-Speed Layout Guidelines. *Application report, November*, 2006.

[17] Howard Johnson, Howard W Johnson, and Martin Graham. *High-speed signal propagation: advanced black magic*. Prentice Hall Professional, 2003.

[18] LearnEMC. Not-So-Good EMC Guidelines. https://web.archive.org/web/20200814195236/https://learnemc.com/not-so-good-emc-design-guidelines.

[19] Matric. Capabilities of flying Probe Testing. https://web.archive.org/web/20200815064759/https://blog.matric.com/flying-probe-test-capabilities.

[20] Kraig Mitzner. *Complete PCB design using OrCad capture and layout*. Elsevier, 2011.

[21] Mark I Montrose. *EMC and the printed circuit board: design, theory, and layout made simple*, volume 6. John Wiley & Sons, 2004.

[22] Ralph Morrison. *The fields of electronics: understanding electronics using basic physics*. Wiley Online Library, 2002.

[23] Clayton R Paul. *Transmission lines in digital and analog electronic systems: signal integrity and crosstalk*. John Wiley & Sons, 2011.

[24] PCBWay. Copper Clad Laminate. https://web.archive.org/web/20200815090819/https://www.pcbway.com/blog/PCB_Basic_Information/Copper_Clad_Laminate.html.

[25] PCBWay. PCB Surface Finishes Introduction. https://web.archive.org/web/20200815094559/https://www.pcbcart.com/article/content/surface-finish-intro-and-comparision.html.

[26] PNI. RM3100 Geomagnetic Sensor User Manual. https://web.archive.org/web/20200814090048/https://www.pnicorp.com/wp-content/uploads/RM3100-Sensor-Suite-User-Manual-R07-1-2.pdf.

[27] M Rangu. Getting EMC design right first time. *EMC FastPass*, 2014.

[28] Wikipedia contributors. Flying probe --- Wikipedia, the free encyclopedia, 2020. [Online; accessed 15-August-2020].

[29] Tim Williams. *The circuit designer's companion*. Elsevier, 2004.