

english



Modernisation of VET through  
Collaboration with the Industry

**Santiago Silvestre**  
**Jordi Salazar**  
**Jordi Marzo**

## Printed Circuit Board (PCB) Design Process and Fabrication



**Erasmus+**

This project has been funded with support from the European Commission.  
This publication reflects the views only of the author, and the Commission cannot  
be held responsible for any use which may be made of the information contained  
therein.

**Title:** Printed Circuit Board (PCB) Design Process and Fabrication  
**Author:** Santiago Silvestre,  
Jordi Salazar,  
Jordi Marzo  
**Published by:** Czech Technical University of Prague  
Faculty of electrical engineering  
**Contact address:** Technicka 2, Prague 6, Czech Republic  
**Phone Number:** +420 224352084  
**Print:** (only electronic form)  
**Number of pages:** 45  
**Edition:** 1st Edition, 2019

**MoVET**

Modernisation of VET through  
Collaboration with the Industry

<https://movet.fel.cvut.cz>



This project has been funded with support from the European Commission.  
This publication reflects the views only of the author, and the Commission cannot be held responsible for any use which may be made of the information contained therein.

## EXPLANATORY NOTES



Definition



Interesting



Note



Example



Summary



Advantage



Disadvantage

---

## ANNOTATION

This module describes main characteristics of Printed Circuit Boards (PCBs). A brief history of PCBs is introduced in the first chapter. Then, the design processes and the fabrication of PCBs are addressed and finally a study case is presented in the last chapter of the module.

## OBJECTIVES

After studying this module, students will know the meaning of PCBs and the important role they play in electronics. They will discover the applications of the several categories of PCBs with their own particular manufacturing specifications, material types and usages.

## LITERATURE

- [1] R. S. Khandpur. Printed Circuit Boards: Design, Fabrication, Assembly and Testing. Tata McGraw-Hill Education, 2005. 696. ISBN 0-07-058814-7.
- [2] William Ho and Ping Ji. Optimal production planning for PCB assembly. Springer series in advanced manufacturing, 2006. 121. ISBN 1-84628-499-6
- [3] Barco Graphics N.V. Gerber RS-274X Format - User's Guide. 55 p., 2001, [http://d1.amobbs.com/bbs\\_upload782111/files\\_11/ourdev\\_450330.pdf](http://d1.amobbs.com/bbs_upload782111/files_11/ourdev_450330.pdf)
- [4] Sinclair, Ian Robertson; Dunton, John. Practical electronics handbook. Elsevier, 2007. 543. ISBN 978-0-7506-8071-4.
- [5] Kraig Mitzner. Complete PCB Design Using OrCAD Capture and PCB Editor. Elsevier, 2009. 471. ISBN 978-0-7506-8971-7.
- [6] Lianyu Fu, Jian Wang, Qiang Guo, (2013) "Characterization of PCB routing process and optimization of tool design based on the investigation of routing temperature", Circuit World, Vol. 39 Issue: 4, pp.212-216, <https://doi.org/10.1108/CW-08-2013-0026>.
- [7] Ravi Kansagara. How to Choose The Best PCB Design Software. <https://circuitdigest.com/tutorial/best-pcb-design-software>
- [8] Liyan Gong. Top 9 Best PCB Design Software of 2018. 2017 <http://www.seeedstudio.com/blog/2017/03/13/pcb-design-software/>
- [9] Mahmoud Wahby. PCB design basics. EDN Network. 2013. <https://www.edn.com/Home/PrintView?contentItemId=4424239>

- [10] PCB design guidelines for reduced EMI. Texas Instruments. 1999.  
<http://www.ti.com/lit/an/szza009/szza009.pdf>
  
- [11] Sam Sattel. The top 10 PCB component placement tips for the PCB beginner. AUTODESK. <https://www.autodesk.com/products/eagle/blog/top-10-pcb-component-placement-tips-pcb-beginner/>
  
- [12] Edwin Robledo & Mark Toth. Ten best practices of PCB design. EDN Network. March 07, 2014. <https://www.edn.com/electronics-blogs/all-aboard-/4429390/1/Ten-best-practices-of-PCB-design>
  
- [13] Nikola Zlatanov. PCB Design Process and Fabrication Challenges. PCB West Conference, 2012.
  
- [14] PCB manufacturing process. <http://www.circuitstoday.com/pcb-manufacturing-process>
  
- [15] PCB Manufacturing Process — A Step-by-Step Guide.  
<https://www.pcbcart.com/article/content/PCB-manufacturing-process.html>

# Index

<b>1</b>	<b>Introduction .....</b>	<b>7</b>
<b>2</b>	<b>The History of Printed Circuit Boards.....</b>	<b>9</b>
<b>3</b>	<b>PCB classification.....</b>	<b>11</b>
<b>4</b>	<b>Implementation process.....</b>	<b>13</b>
<b>5</b>	<b>Design .....</b>	<b>16</b>
5.1	CAD software for PCB design .....	17
5.2	Creating library components .....	19
5.3	Schematic design.....	20
5.4	Floor plan .....	21
5.5	Placement and routing .....	22
5.6	Design rules checking .....	25
5.7	PCB documentation.....	26
<b>6</b>	<b>Fabrication.....</b>	<b>27</b>
6.1	Film Generation.....	28
6.2	Etching.....	29
6.3	Electroless Copper Deposition (Plating) .....	30
6.4	Solder Mask.....	31
6.5	Silkscreen .....	32
6.6	Surface finish.....	33
6.7	Drilling .....	34
6.8	Electrical test .....	35
6.9	Assembling.....	36
<b>7</b>	<b>Study Case.....</b>	<b>37</b>

# 1 Introduction

Printed Circuit Boards (PCBs) are used in almost all electronic applications. From mobile phones to aircrafts, going through medical devices or industrial machinery, all of the electronic products used daily are supported by PCBs.



---

A printed circuit board: PCB, is a non-conductive substrate plate that is used for the assembly and interconnection of electronic components through routes or tracks of a material conductor recorded on the substrate.

---

The PCBs are most commonly made out of fiberglass, composite epoxy, or another composite material. Different electronic products have different requirements on the PCB base material.

Most PCBs for simple electronics are simple and composed of only a single layer. However, more sophisticated electronic circuits such as computer graphics cards or motherboards use to have multiple layers, sometimes up to sixteen or more, called “Multilayer” PCBs.

Typical PCB substrates are made from a composite of:

- Some kind of Reinforcement Material or core providing the "skeletal" support for the laminate.

This part of the PCB gives strength and dimensional stability. Teflon fiber or Quartz cloths are used as reinforcement materials. The FR-4 composite material is a good example of core material for PCBs.



---

The FR-4 is a composite material composed of woven fiberglass cloth with an epoxy resin binder that is flame resistant used as common reinforcement material for PCBs fabrication.

---

- Resin Binder System:

It provides the "glue" to bind the laminate together. Some examples of this kind of binder materials are thermoplastics, thermoplastics or general purpose resins.

- Conductive or Catalytic Surface Coating :

It provides the basis for creating the circuit pattern.

Once the PCB is manufactured, the components assembled and the PCB verified two types of products are usually applied to the same PCB. First, some kind of cleaner or flux solvent is applied to the PCB. Then a protective coating is applied covering the circuit. This final protective coating avoids the copper to oxidize and gives a greater durability to the plates.

Main advantages of the use of PCB's in electronic circuits are the following [1-2]:

- The size of component assembly is reduced as well as the weight.

- Quantity production can be achieved at lower unit cost and the wiring and assembly can be automated.
- They ensure a high level of repeatability and offer uniformity of electrical characteristics from assembly to assembly.
- The location of parts is fixed, which simplifies identification and maintenance of electronic equipment and systems. Moreover, Inspection time is reduced because printed circuitry eliminates the probability of errors.
- Printed wiring personnel require minimal technical skills and training. In addition, chances of short-circuited wiring are minimized.
- Circuit characteristics can be maintained without introducing variation in inter-circuit capacitance and parasitic effects.

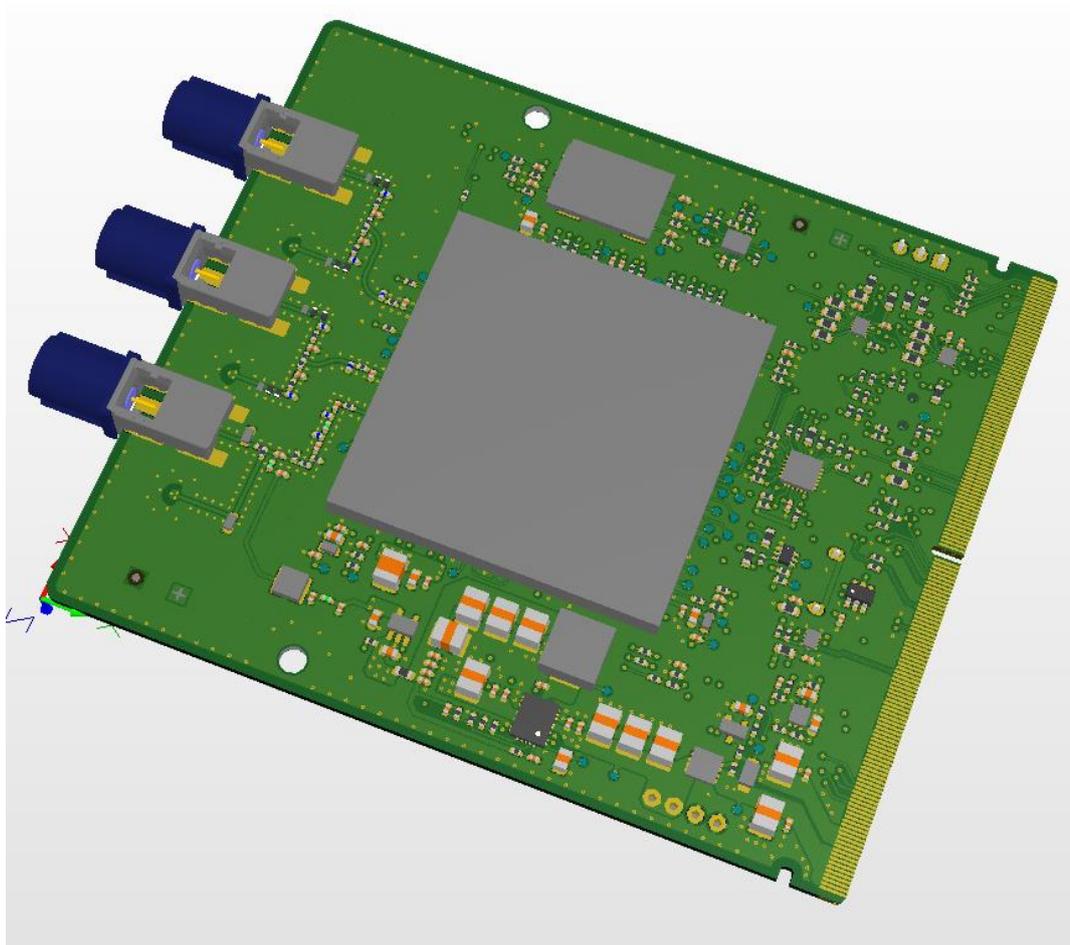


Fig. 1. Example of PCB.

## 2 The History of Printed Circuit Boards

Before PCBs became the common component used in electronic circuits, point to point construction was used. This meant some extremely bulky and unreliable designs that required large sockets and regular replacement. The first PCBs appeared in the 1920s. These PCBs were basically made from Bakelite and thin pieces of wood. The process consisted on drill holes in the material rivets of flat brass wires were made. These PCBs were mostly used in the fabrication of early tube style radios and gramophones at the time.

Dr. Paul Eisler in Austria began making the first real operational printed circuit boards in 1943 and the first double sided PCB's with plated through holes was produced in 1947. By the 50's and early 60's laminates using different types of resins mixed with all sorts of different materials were being introduced but the PCBs were still single sided. The wiring would be printed on one side and the electrical components would be on the other.

The first multilayer PCBs were produced commercially also in the 60's. In the next decade hot air soldering methods began to be used. Moreover, aqueous developed LPs (liquid photo imageable masks) were introduced for screen processes, making it the global industry standard. This new fabrication process allowed to important reductions on the size of the PCBs. Gerber Scientific introduces RS-274-D as a machine-based format for vector photoplotters. This standard open ASCII vector format was used by the PCB industry software to describe the printed circuit board images: copper layers, solder masks and legends.

This size reduction continued along the 1980s and 1990s when surface mount parts where introduced along with the increase of the quality of the product. In 1986, the RS-274X released as an enhancement to RS-274-D data format as an extended version [3]. This new version supports embedded aperture information relieving the need for external aperture definition files.

Rigid-flex and flexible PCB options became more affordable because of the use of multi-layer circuit boards to effectively meet the needs of rapidly growing technology. Moreover, in the late 1990s HDI (High Density Interconnect) PCBs were developed by using micro-via technology.



---

A via or VIA (Latin for path or way, also known as vertical interconnect access) is an electrical connection between layers in a physical electronic circuit that goes through the plane of one or more adjacent layers.

---

The HDI technology allows shorter traces on the board by reducing the amount of time it takes an electrical signal to travel between components. Moreover, smaller vias between devices help to reduce the overall capacitance in the circuit, which reduces the rise time for the signal. On the other hand, the overall inductance is also reduced with the high-density design offered by HDI, lowering the effect on neighbouring leads and pins.

A side benefit to the reduction in overall capacitance has led to a reduction in required voltage in HDI devices. In turn, this drop in voltage not only means that

smaller batteries are required, but also that heat generation is reduced; meaning heat dissipation is less problematic. The elimination of large heat sinks or cooling fans in many HDI applications has contributed to the development of mobile and wearable electronics.



---

HDI: **High Density Interconnect**, printed circuit board with a higher wiring density per unit area than conventional PCBs. They have finer lines and spaces ( $\leq 100 \mu\text{m}$ ), smaller vias ( $\leq 150 \mu\text{m}$ ), and capture pads ( $\leq 400 \mu\text{m}$ ), and a very high connection pad density.

---



---

Nowadays, the PCB industry is still evolving further. The PCBs are expected to evolve according to user behaviour following miniaturization trends.

---

### 3 PCB classification

There are several overall types of PCB boards each with their own particular manufacturing specifications, material types and usages.

Table 1 shows a classification of most common PCBs used in electronics including main characteristics and applications.

Table 1 – PCBs classification

PCBs	Main Characteristics	Applications
Single-layer	Made out of a single layer of base material or substrate. One side of the base material is coated with a thin layer of metal. Generally their circuits and components are soldered onto one side, but also the components can be soldered on both sides of the board. Low-cost.	Calculators, cameras, radio and stereo equipment, solid state drives, power supplies, printers and other consumer electronic products.
Double-layer	A thin layer of conductive metal is applied to both sides of the board. Holes drilled through the board allow circuits on one side of the board to connect to circuits on the other. Smaller and with a lighter weight than Single-layer PCBs.	Industrial controls, Power supply, Instrumentation, LED lighting, Vending machines, Automotive dashboards or Amplifiers.
Multi-layer	Consist of a series of three or more double-layered PCBs. Complex designs.	VLSI electronics, File servers, Data storage, GPS technology and Satellite systems, Weather analysis and Medical devices.
Flexible	Made out of materials that can flex and move, such as plastic (Kapton, polyester or polyamide). Higher fabrication costs, flexibility and weight savings.	Electronic equipment subject to environmental hazards.
Rigid	Made out of a solid substrate material that prevents the board from twisting. Lower cost than flexible PCBs.	Computer motherboards, Audio keyboards, Solid-state drives, Flat-screen TVs and monitors.
Rigid-Flex	They consist of a mixture of flexible copper conductor layers on dielectric film and rigid copper material, Combine the best of both rigid boards and flexible circuits integrated together into one	Aerospace, Medical, and Military electronic applications. Portable devices industry.

	circuit. Rigid flex circuits provide higher component density and better quality control.	
High-frequency	PCBs for special signal requirements into electronic circuits that offer a frequency range of 500MHz - 2GH. Requires the use of specialized materials.	High-speed designs, Radio frequency (RF), Microwave and Mobile applications.
Aluminum-backed	Aluminum backed PCBs are composed of an Al backing, a highly thermally conductive dielectric layer and a standard circuit layer. Excellent temperature control characteristics and high levels of mechanical stability and low levels of thermal expansion.	High power and tight tolerance applications. Motor controllers and Automotive applications.

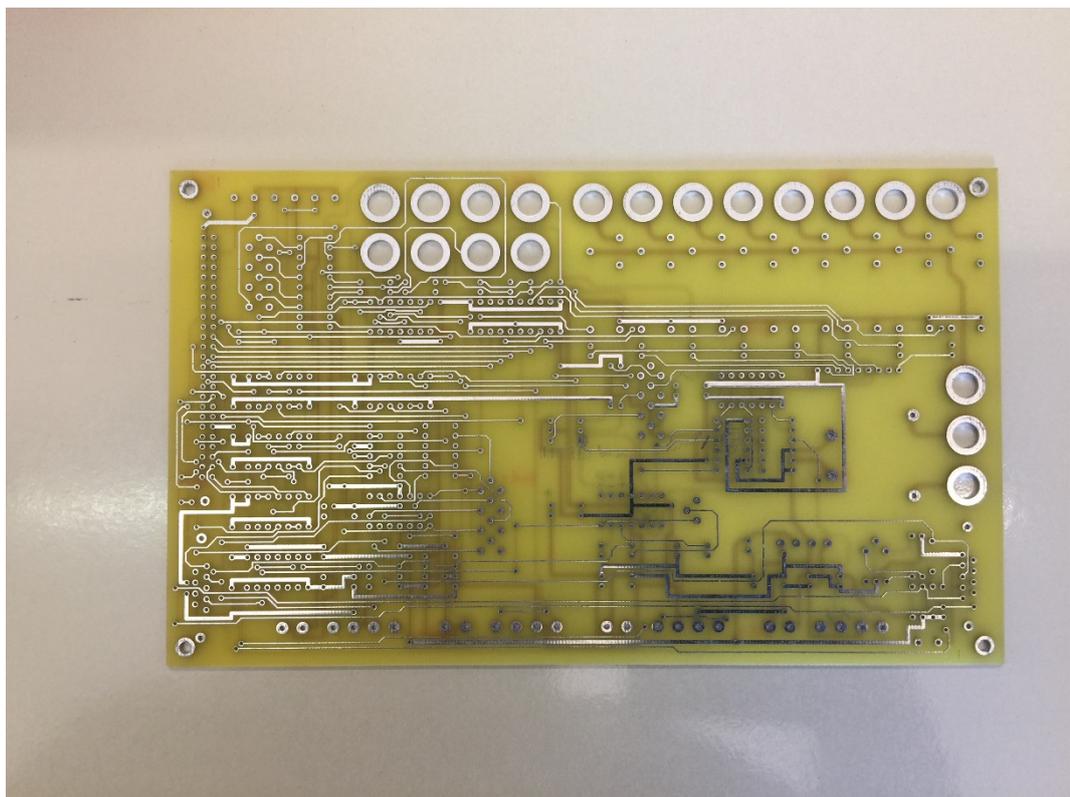


Fig. 2. Double-layer PCB.

## 4 Implementation process

The PCBs are the base for any electronic circuit construction. A PCB used to place and mount the electronic components included in the device and it provides the means of electrical connection between these components.



The necessary connections between parts with a wire in PCBs are called nets.

Software tools are used for the PCB design, including schematic capture, component footprint editor, PCB layout design and auto router. Moreover, most of these tools are including analog/digital simulation capabilities and signal integrity analysis tools. Some commercial software tools are available in the market from several companies expert in the development of software packages on this topic, as Cadence, Altium, Mentor Graphics, Zuken or Cadsoft. On the other hand, many free tools can be found for easy development of simple designs, as Kicad or Eagle.

The design of a PCB includes some steps that must be carefully full filled in order to achieve an optimized circuit. These steps are detailed in table 2.

Table 2 – PCB design. Main steps.

	<b>Main Characteristics</b>
	Schematic component symbols are created in schematic libraries. Read data sheet to identify the pins. Draw the symbol of the component. Draw the package. Adding Component Parameters.
Draw the schematic	Place the part symbols. Draw the nets and buses. Place, move and edit the electronic components.
Create the board	Transfer the schematic diagram into a drawing of the PCB. Board size and trace width. Placement of the electronic components and labeling. Definition of ground planes. Trace horizontal and vertical routing including specific angles.
Place the parts and route the signals	Components should be placed according to their connections to other components in the board. Thermal considerations, mechanical requirements, as well as signal integrity and rout ability must be taken into account. Some landpatterns (or footprints) of devices are configured on the board. All electronic components which have connections to each other should be placed in the same vicinity if possible. Each placed part consists of pins which are terminals that need to be connected in order to complete the design. Auto-Rout functions can be used in this step in most popular software design tools. These functions are useful, but it is difficult to get a good design by using just the auto-route tools.
Edit the board and check	Check the board designed to identify any errors. Several check functions are available in all software design tools.

Create PCB manufacturing data	Creation of the files needed by PCB manufacturers in order to finish the fabrication of the electronic circuit.
-------------------------------	---

Some considerations must be taken into account in the implementation process of a PCB. The following points summarize some of these relevant issues:

- **Ground planes:** The current returns must consist of large area ground planes for low impedance to high frequency currents in order to avoid parasitic resistances and inductances, especially at high frequencies. Ground planes provide a low-impedance ground connection. Eliminate the loop area return by using both separate ground plane and power plane. If you need to split the ground plane and run traces across the split, you should first connect the planes. Use solid ground planes next to power planes. In PCBs including analog and digital circuits and signals, it is always recommended to use separate supplies for digital and analog circuitry.
- **Power supply routing, clearances:** Power planes are designed using the same rules as ground planes. In mixed circuit PCBs, separate power supplies for analog and digital circuits, even if the voltages are the same. Moreover, the digital power plane does not overlap an analog power plane.
- **The power supply pins should be decoupled directly to the ground plane by using ceramic capacitors located as close as possible to the integrated circuit power pins.**
- **High frequency signals and impedance control:** High frequency circuits and signals are going to be emitting electromagnetic radiation as they travel from source to sink. In general, frequencies higher than 1GHz are considered as high frequencies.

These signals can interfere with each other or a nearby component. To avoid conflicts, try to reduce parallel signals on the board at a minimum to avoid any signal coupling or crosstalk. In addition, maintain as much distance as possible between signal traces, and even consider routing signals on a separate layer if they are particularly noisy.

The sampling clock generation and high frequency circuitries should be treated like analog circuitry and also must be grounded and heavily-decoupled to the analog ground plane. It is also important to isolate these components from noisy digital circuits.

Take into account that ground planes will minimize RF emissions.




---

The impedance is an AC characteristic, meaning that it is related to frequency. The impedance is the combination of the capacitance and inductance of a circuit when operated at high frequency. Though also measured in Ohms, it is somewhat different than resistance which is a DC characteristic.

---

The PCB traces do not behave like simple connections at high frequencies, controlled impedance helps us ensure that signals are not degraded as they route around a PCB. In fact, controlled impedance is the matching of substrate material

properties with trace dimensions and locations to ensure the impedance of a trace's signal is within a certain percentage of a specific value. Controlled impedance boards provide repeatable high frequency performance. Impedance control requires of a planning of the required stack-up and the calculation of electro-magnetic effects surrounding the traces.

- EMC recommendations: Some precautions such as minimizing crosstalk, proper grounding and namely proper layer stack-up will significantly reduce Electro Magnetic Interference (EMI) problems. An ideal stack-up will be a ground plane under each other plane.



---

In electronics, crosstalk is any phenomenon by which a signal transmitted on one circuit or channel of a transmission system creates an undesired effect in another circuit or channel.

---

- Routing techniques : When routing signals on different layers, be sure to route them orthogonally to each other. That is, on one signal layer your traces will be routed horizontally, and on the other vertically, 45 degrees, etc.

The two basic routing choices are maze routing and X-Y routing. The X-Y routing involves at least two routing layers with wires travelling in only one direction on each layer. On the other hand, the Maze routing allows the wiring of complete nets on a single layer, eliminating the need for layer changing vias.



---

The Manhattan length is the shortest path that a wire can have when it must be connected using only segments that are confined to either the X axis or Y axis.

---

Nowadays, the PCB designers have at their disposal a good number of well developed routing tools to effectively route the board. These tools use to include autorouter options. However, these options should not be considered as the only routing option.

It is important to route the nets for critical components, as connectors or FPGA components, by using either manual or auto routing techniques, because these components require routing beneath/between multiple surface mount pins.

In specific applications, it is important to conduct online measurement and the characterization of temperature during printed circuit board (PCB) routing process as well as the optimization of router design based on the investigation of routing temperature [6].

## 5 Design

This section is intended to be a basic guide to PCB design. The different steps required for the design of a printed circuit board are summarized in this section, starting from the choice of the CAD software for the PCB design, through adding components to the schematic, the design rules and checking, and ending with the generation of all PCB documentation, standards and recommendations.



---

PCB design is mainly a two-step process. First, you design the schematic of the electronic circuit and then you lay out a PCB based on that schematic.

---

## 5.1 CAD software for PCB design

There are many different ways to create a PCB design but the most common is by using a CAD software package specifically designed for PCB circuits. These software packages include board and schematic editors that work together to provide you the best design process. The board editor can import all of the components, footprints, and wires into the PCB file, which will make the PCB design process easier.

Nowadays, we can find a multitude of CAD tools in the market that allow us to design a PCB. They range from simple and intuitive to highly sophisticated, from free or inexpensive to high-end or premium, feature-less or feature-heavy [8-9]. However, the one that best suits you depends on your own needs (functionality required, typical complexity of designs, frequency requirements, etc.) and budget.

In terms of the most popular PCB design software packages, here is a list of the nine most widely used in the industry.

**Altium Designer.** It is a very popular PCB design software. Altium Designer is a commercial tool with a high cost license, mainly used in professional environments. Also, most of the universities use it in their laboratories. Altium software is generally used to design a complicated and very complex circuit with very fast and realistic results. It is efficient, easy to use, and meets the modern needs of professional electronics engineers.

**OrCad.** It is the PCB design software tool for a big group of engineers because it provides a complete environment, from the initial schematic to the final artwork. It allows you to simulate and design PCB circuits.

**Mentor PADS.** It is a PCB design package developed by Mentor Graphics. It provides quality PCB schematic and layout tools at an affordable cost for small to mid-sized companies. It has a number of high end features, including signal integrity analysis functions, an advance auto-router, thermal design consideration analysis and support for various project management functions. It is a professional level package with a competitive price point but still priced well above the hobbyist market.

**Eagle.** Eagle stands for Easily Applicable Graphical Layout Editor. This software is available in free and paid versions. In the paid version, the user is provided with technical support that includes call, mail and online chat support and provides access to latest release software. Eagle has a very simple, effective and easy interface and provide a library which contains a large number of electrical and electronics components. That is the main reason for being so popular PCB designing software among educationist, hobbyist, and professionals. In addition, you can find very good resources and tutorials from internet which is very helpful if you are beginner.

**Zuken CADSTAR.** This complete design environment takes engineers from their initial ideas through product creation, thanks to its unified suite of applications. It is an established and wide extended easy-to-use single board PCB design software with many thousands of satisfied users globally that includes schematics, PCB

layout, routing, signal integrity, manufacturing outputs. It is a professional, powerful, accessible and affordable software that meet your design requirements and budget and is supported by experienced, local resellers who provide training, support, integration and consultancy to help you maximize your return on investment.

**Zuken CR-8000** [<http://www.zuken.com/en/products/pcb-design/cr-8000>] . Electronics engineers choose Zuken CR-8000 when they require advanced PCB design software with a product-centric 3D PCB design platform. An advanced single and multi-board design solution that provides the tools to optimize a design at both the product and PCB design level. It includes 2D/3D multi-board design and implementation, FPGA I/O optimization, chip/package/board co-design, and 3D MCAD integration all in one design process.

**Mentor Xpedition.** A high-end product that combines ease-of-use with advanced functionality. It provides designers with the technology to create the most complex designs and satisfy the needs of mid-sized to large electronic companies. It includes interactive and customizable multi-pass autorouting controls for design challenges, such as differential pair routing, net tuning, manufacturing optimization and HDI/microvia and buildup technology. It also offers a 3D mechanical kernel in the PCB layout. Thus, designers can operate in a single environment and switch between 3D and 2D views.

**DesignSpark.** It is an easy-to-use free PCB design tool provided by RS Components. DesignSpark is modern and intuitive, with no practical design limitations in the area of the board, layers and number of contacts. DesignSpark generates Gerber files according to industry standards. It is offered free of charge for companies, hobbyists, educators and more. In addition, you can quickly upload your *bill of materials* (BOM) to the RS Components website using BOM-Quote service.

**KiCAD.** It is a free software suite that handles Schematic Capture, and PCB Layout with Gerber output. It provides a very easy way of routing the components and it makes easy PCB troubleshooting as well. This software is good for beginners and the biggest advantage is that this software does not require purchasing any license, it is free.



---

The PCB design software package that best suits you depends on your own needs and budget.

---

For professional PCB designers and large and complex applications, probably Mentor Xpedition, Zuken CR-8000 and Altium would be the best PCB design CAD softwares. For beginners or hobbyists, then Eagle, KiCAD or DesignSpark are good options.

## 5.2 Creating library components

Normally, all PCB design software packages are supplied with a set of libraries. These libraries can be used, modified and added to as required. You can mix and match new and existing library items together to create your own set of libraries.

Before you can make a schematic and subsequent layout of the PCB it is necessary to make sure that you have all the components in a library with their symbol (schematic symbol) and package (footprint for layout).

If not all the components are available, it will be necessary to create your own library and design the symbol and footprint of each missing component. This usually occurs when a new component that has recently been launched on the market is used in the design of the PCB.

For high volume production designs, library quality has a relevant importance. If any created footprint is not 100% well designed and verified with final production site, a big productivity failure can happen, and can have a serious impact directly to the company results. This is a big reason why large companies have their own libraries with an own Librarian exclusive team dedicated only to footprints creation.

Also is very important to have a good 3D design linked with footprint. It allows to make a better product integration (mounted PCB together with mechanical housing).

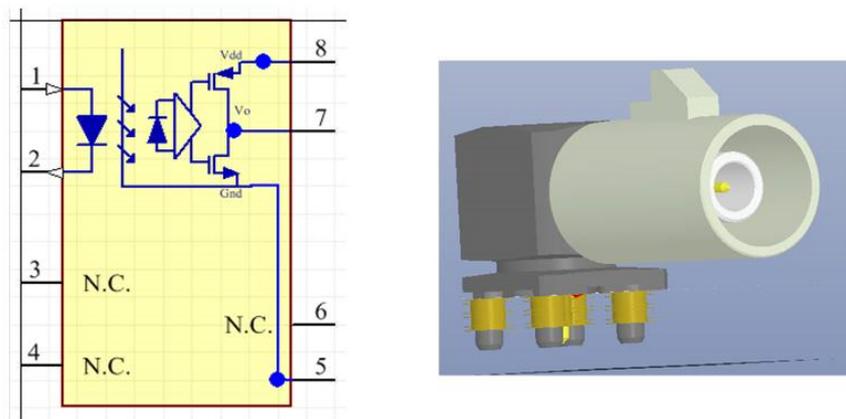


Fig. 3 Schematic symbol and 3D footprint

## 5.3 Schematic design

Before creating a schematic, the first step will be create a new project folder for our design. This folder will house both our schematic and PCB design files.

The PCB design process starts by drawing the schematic diagram of your electronic circuit into the software you have chosen. The schematic drawing controls the project, so accuracy and completeness are critical for success. A schematic includes all information that is necessary for the proper operation of the circuit. Thus, a schematic should include adequate design details, such as pin numbers, names, component values, ratings, and interconnections between electronic components.

The schematic symbol of each electronic component has a PCB footprint associated with it. The PCB footprint will define the component's physical dimensions and placement of the copper pads or through holes. Embedded within the schematic symbol of the component is the manufacturer part number used to determine price and specifications. The package specification determines the size of the footprint for each component.

Before generating a PCB, the symbols are mapped to component footprints and the symbol interconnections are converted to a netlist that specifies the connections between the component footprints in the layout process [10].

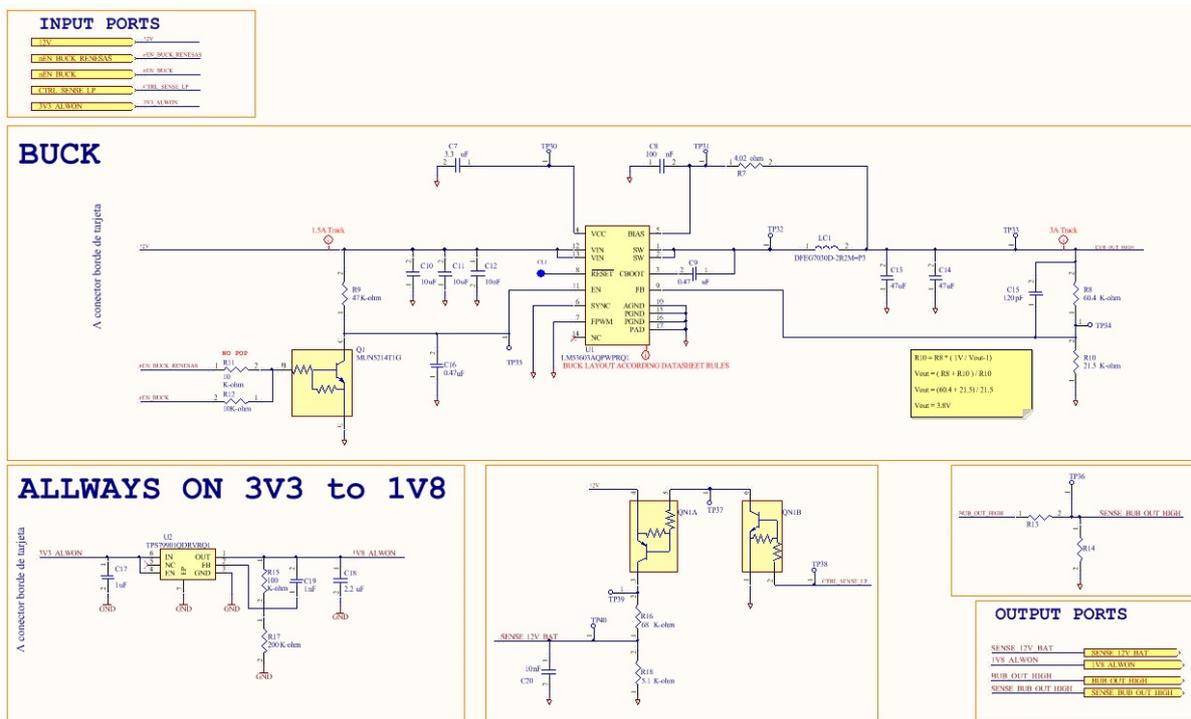


Fig. 4. PCB Schematic.

## 5.4 Floor plan

Next step is to draw the board with all the components on it. You need to transfer your schematic drawing into a drawing of your printed circuit board. That is the floor plan. A floor plan is a drawing that indicates the general location of components on the blank PCB before drawing in any traces [11].

Functional blocks of circuitry such as power conditioning, RF, digital, analog, etc., should be organized and placed as groups in order to minimize signal crossing. A pre-placement floor plan will allow you to see what the signal flow between functional blocks is and how best to plan for it. For instance, group your power conditioning parts together as much as possible in order that their signals do not have to cross through sensitive areas of RF circuitry.



---

A floor plan is a drawing that indicates general areas where blocks of circuitry are to be placed on the board. Floor plans bridge the gap between the schematic drawing and the physical layout. This gives the designer a head start on placement.

---

## 5.5 Placement and routing

Component placement is the next step in the process. This is one of the most important and critical step in the design of a PCB. It is said that the design of a PCB is 90% placement and 10% routing. What it really means is that the placement of your components will end up deciding how much time your routing will take.

It is said that PCB design is both an art and a science. There is a big set of technical know-how and measurements to consider when dealing with trace widths, layer stack-ups, schematics, PCB ground plane design, etc. The artistic side of PCB design has to do with component placement. The truth is that there is no right way to place components, and this freedom is ultimately what makes the PCB layout process so creative. If you give a schematic to 100 different engineers, you will likely get 100 different layouts back. This is why the PCB layout process is viewed as an artistic process [12].

One of the first things to do will be to adjust the board area to the maximum allowed size of the PCB. The number of layers will depend on power levels and complexity.

Here there are some general design guidelines you should know and take into account when designing a manufactural, functional and reliable PCB. Obviously, there are more, and the PCB design guidelines list below should not be thought of as a complete list [13].

When placing components make sure the snap to grid option is turned on. Start placement of components in the following order: connectors, power circuits, sensitive and precision circuits, critical circuit components (MCU, DSP, FPGA, memory, and clock devices) and then the rest. In addition, components should be placed according to their connections to other components.

Similar components must be oriented in the same direction as this will help with an efficient and error-free soldering process.

Avoid placing components on the solder side of a board that would rest behind plated through-hole components.

Place all surface mount components on the same side of the board, and all through-hole components on the top side of the board to minimize the number of assembly steps.

Place decoupling capacitors close as much as possible to  $V_{CC}$  pin on active components.

Components absorbing greater than 10 mW or conducting more than 10 mA should be considered powerful enough for additional thermal and electrical considerations. Sensitive signals should be shielded from noise sources with planes and be kept impedance-controlled.

Power management components should use ground planes or power planes for heat flow. Make high-current connections according to the acceptable voltage drop for the connection. Layer transitions for high current paths should be made with two to

four vias at each layer transition. Place multiple vias at layer transitions to increase reliability, reduce resistive and inductive losses, and improve thermal conductivity.

Always create a ground plane. This can be a large copper area on a single layer board or even an entire layer dedicated as a ground plane on multilayer boards. And once your ground plane has been added, it is simply a matter of connecting all of your components that need to go to ground with vias.

Real-world copper traces have resistance. This means that a trace has a voltage drop, power dissipation, and a temperature rise when current flows through it. The width of the lines should be sized according to the estimated current that flows through them. Therefore, power lines should be wider because all the current is supplied by these wires. Commonly, copper traces on PCBs have 35 micron thickness. Thus:

Table 1 – Trace width and maximum current

Trace width	Maximum current value
4 mm	10A
2 mm	5A
1.5 mm	4 A
1 mm	3 A
0.5 mm	2A
0.2 mm	0.5 A

Minimize trace lengths when placing components and avoid 90 degrees angles. Use two 45 degrees bends instead. Reasons for that are that in the board manufacturing process, the outside corner can be etched a little narrower thus changing the trace impedance. The use of 45 degrees angles shortens the electrical path between components. In addition, high speed logic signals can get reflected off the back of the angle, causing interference.

Keep digital and analog grounds separate because voltage and current spikes from digital circuits can generate interference (noise) in the analog circuits, affecting their performance.

These rules are in no particular order, can generally be applied to any PCB design project, and should prove as a useful guide for novice designers.

As a result of the placement and routing processes, we finally get the PCB layout which is the drawing where the electronic components with their footprint appear in the position they will occupy in the final PCB and the interconnection traces between pins.

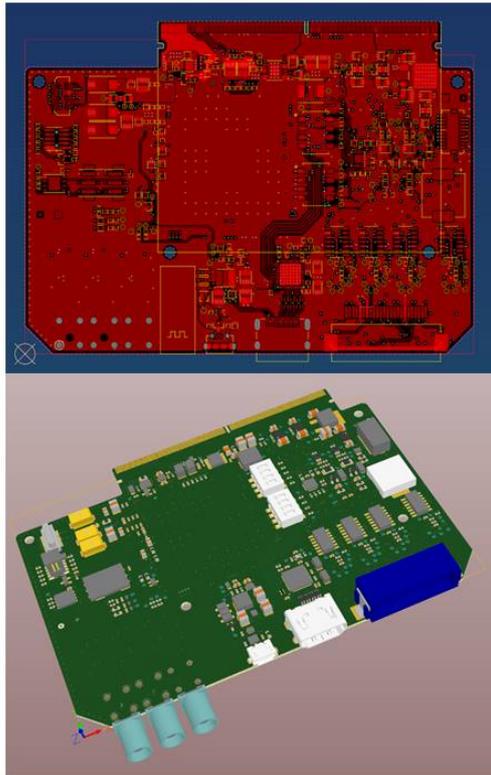


Fig. 5. PCB Layout: 2D (top) and 3D (bottom).

## 5.6 Design rules checking

Once all the traces are placed, it is time to check the routing of every signal to verify that nothing is missing or incorrectly wired (missing vias, unconnected pads, short circuits, etc.). Generally, all CAD software packages usually incorporate layout design tools such as the Design Rule Check (DRC).



---

The Design Rule Check (DRC) is a powerful automated feature that checks both the logical and physical integrity of a design.

---



---

The Design Rule Check (DRC) should be used on every routed board to confirm that it satisfies a series of recommended parameters called design rules.

---

The design rules are a series of parameters (limitations) provided by the PCB manufacturer. A Design Rule Check allows you to establish a set of boundaries for trace widths, component spacing, via diameters, etc.

## 5.7 PCB documentation

Proper documentation is necessary to reduce the chance of misunderstandings and errors during the PCB manufacturing process. PCB producers need to know the exact requirements expected of their product, and assembly teams must understand how circuit boards fit into the final build.

The PCB documentation should include at least the hardware dimensional drawings, schematic, BOM, layout file, component placement file, assembly drawings and instructions, and the Gerber files.

The Gerber files are PCB jargon for the output files of the layout that are used by PCB manufacturers to create the PCB. A complete set of Gerber files includes output files generated from the board layout file [14]:

- Silkscreen top and bottom
- Solder mask top and bottom
- All metal layers
- Paste mask top and bottom
- Component map (X-Y coordinates)
- Assembly drawing top and bottom
- Drill file
- Drill legend
- FAB outline (dimensions, special features)
- Netlist file

The special features included in the FAB outline include but are not limited to notches, cutouts, bevels, back-filled vias-in-pad (used for BGA-type IC packages that have an array of pins under the device), blind/buried vias, surface finish and leveling, hole tolerances, layer count, and more.

## **6** Fabrication

In this section, the whole PCB fabrication process in the factory will be briefly described. PCB fabrication and component assembly processes are done in an extremely clean environment where the air and components can be kept free of contamination. Most electronic manufacturers have their own proprietary processes.

The following can be considered the most common steps for the PCB fabrication [14-15].

## 6.1 Film Generation

During this initial step, the board image is created on a film. One film is generated per layer. The film is generated from the Gerber files which are send to the manufacturer or the manufacturing unit where the negative image or mask is printed out on a plastic sheet. Generally, manufacturers use a photo plotter to make these films.

For the inner layers of PCB, black ink represents the conductive copper parts of the PCB. The remaining clear portion of the image denotes the areas of non-conductive material. The outer layers follow the opposite pattern: clear for copper and black for the area that will be etched away.

## 6.2 Etching

The purpose of etching is to remove unnecessary copper traces from the substrate. There are several etching solutions available, all based on different types of chemicals like ferric chloride, ammonium per-sulphate, etc. The most commonly used are ferric chloride or hydrochloric acid.

During this process, the PCB is put on a chemical bowl and cleaned up with a brush.

## 6.3 Electroless Copper Deposition (Plating)

In order to make the through holes conductive to electrically connect to different layers of the PCB, a thin layer of copper is chemically deposited into the through holes. This copper will later be thickened through electrolytic copper plating.

## 6.4 Solder Mask

The solder mask is usually the green coating on a PCB board. Solder mask usually covers everything on the PCB board except pads and vias. The primary purpose of the mask is to restrict the areas that will be covered with solder. It also protects panels from contamination, handling damage and possible electrical shorting during assembly and installation.

## 6.5 Silkscreen

The silkscreen is normally used on the component side to identify components, test points, PCB part number, warning symbols, company logos and manufacturer marks. This is done using ink jet printing.

## 6.6 Surface finish

The Surface Finish is the most important step in the process between PCB manufacturing and PCB assembly with two main functions, one of which is to preserve the exposed copper circuitry and the other of which is to provide solderable surface when soldering components to the PCB. Surface finish is located at the outermost layer of PCB and above copper, playing a role as a "coat" for copper. There are different type of surface finish, each one has a different purpose (depending on where the PCB has to work and assembly process requirements) and also has a different application method. The most popular are: ENIG (Electroless Niquel Immersion Gold), OSP (Organic Surface Protection), HASL (Hot Air Leveling), Immersion Silver and Immersion Tin.

## 6.7 Drilling

The drilling machine is computer-controlled. The operator selects the PCB drill file from Gerber files. It tells the machine which drills to use and the X-Y coordinates of the holes. High speed drilling ensures clean hole walls to provide a secure base for good plating on the hole walls.

Drilling is a slow process as each hole must be drilled individually. Drill bit change is fully automatic. The machine selects the drill bit to use from the drill bit rack, checks that it is the correct size, and then loads it into the drill bit head. Normally, stacks of working panels (up to 5) are drilled at same time.

## 6.8 Electrical test

One of the last steps of the PCB fabrication process is to ensure the PCB is free of defects and meet the expected quality standards. Nowadays, there are many automatic testing equipments available for the high volume testing of the PCBs.

PCBs are tested for opens and shorts in the circuitry. The basic of all tests include a computer program which will instruct the electrical test unit to apply a small voltage to each contact point, and verify that a certain voltage appears at the appropriate contact points.

## 6.9 Assembling

PCB component assembly is the process of soldering or assembly of electronic components to a printed circuit board. A PCB assembly process may use through-hole assembly technology or surface mount technology (SMT) or a mix to both. In the former method, the component leads are inserted into the holes drilled in the PCB. In the latter method, a pad having the legs similar to the PCB design is inserted and the IC's are placed or fixed on top of them.

## 7 Study Case

In this section all complete PCB Design Process will be shown by following a real PCB case. The design has been done using Altium Designer Software. Main steps include the following:

- Library Creation: all the symbols for schematic design have to be done and all necessary footprints for PCB design have also to be created before to start the PCB Layout.
- Get all necessary information
  - Mechanical housing information 3D Step format, DXF (scale 1:1), PDF ..... containing inhibit areas for components in TOP and BOTTOM side of the board.
  - Final schematic: containing all connections between components, some rules related to the different nets and explanations about how to proceed with special components.
  - Key components layout recommendations from component manufacturer (this applies to microcontrollers, power components, DDR, and any component requiring special layout)
- Layer stack-up definition: define depending on PCB necessity, density, budget and time to design.

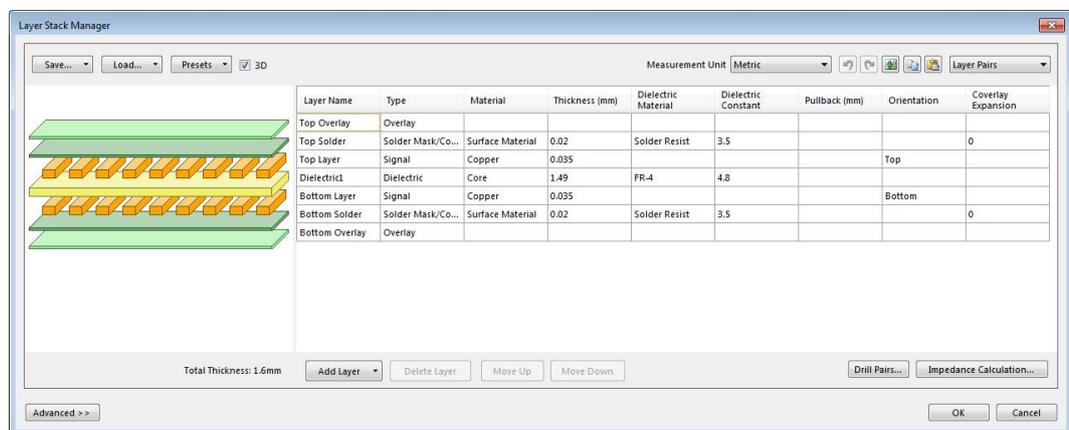


Fig. 6. PCB Layer stack-up definition.

- Import mechanical DXF to Altium special layer (mechanical layer).

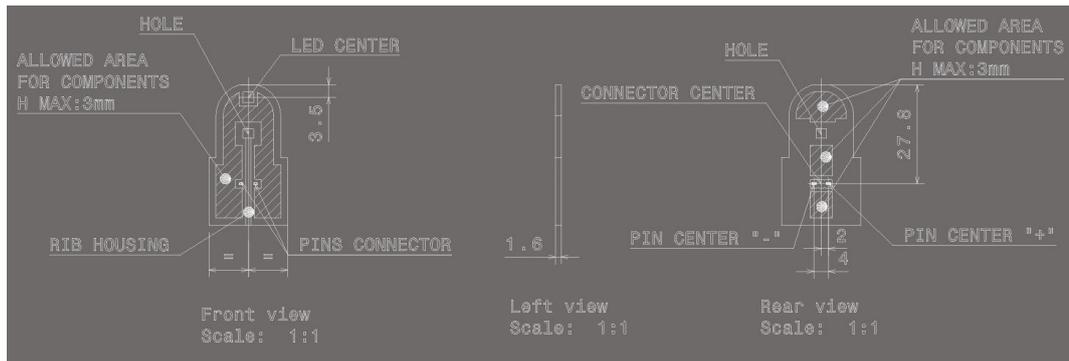


Fig. 7. Mechanical layer.

- Define the board shape from DXF in mechanical layer 2 (we will use mechanical 1 for production panel and mechanical 2 for single board design)
- Place the PCB Origin. This is very important and necessary in order to have a good reference point for design, placing the single PCB in the production panel, 3D housing placement and production information generation (gerber, drill) and mounting documentation (pick and place, ODB++ files).



Fig. 8. PCB Origin.

- Draw the inhibit areas for component placement, copper and different height restrictions in TOP and BOTTOM sides (choose any Altium mechanical layers).



Fig. 9. TOP and BOTTOM sides.

- Set all the Design Rules in Altium Designer. It's possible to set up different rules in order to help with our design. The rules are relative to Electrical and Routing restrictions (clearances), topology, layers to be used, placement, manufacturing, signal integrity and many others. We will choose the corresponding and necessary to success with our design.

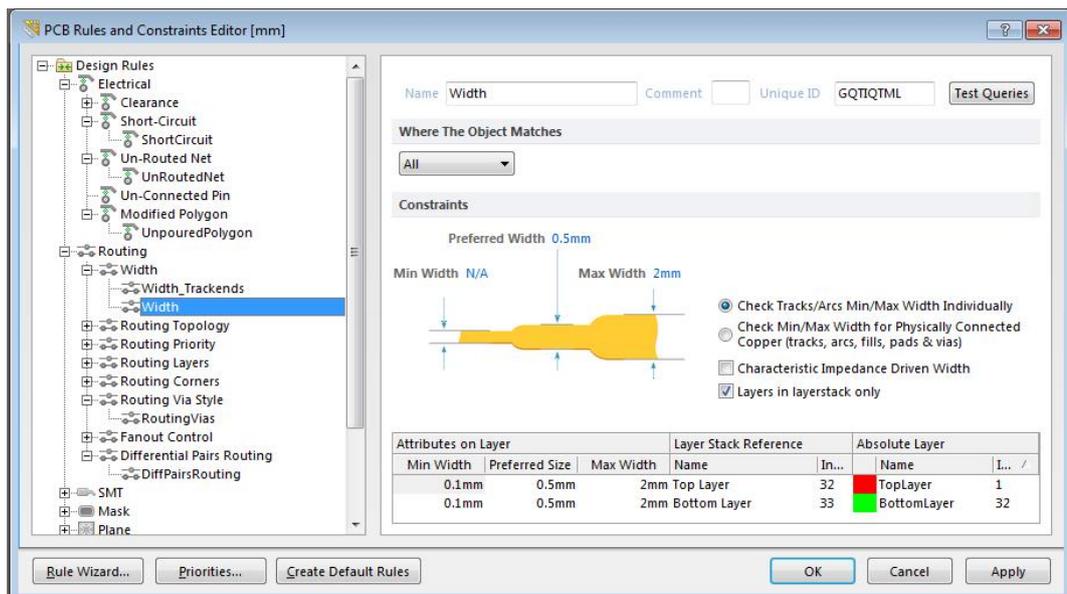


Fig. 10. PCB Rules and constraints editor.

- Perform the PCB update from schematic. All the PCB footprints corresponding with schematic symbols will appear in the PCB.

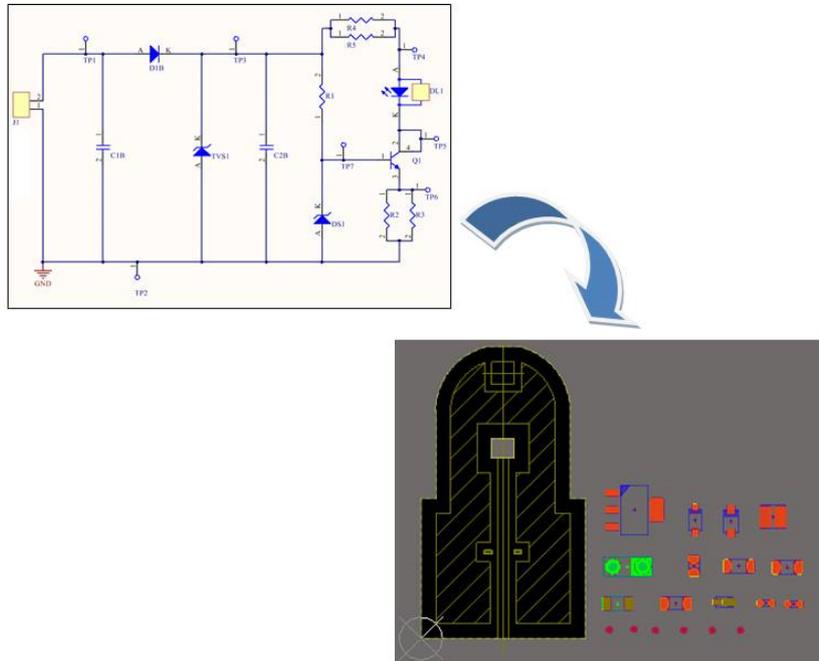


Fig. 11. PCB footprints with schematic symbols.

- Do the component placement. In that design, the target is to get minimum pattern length with only using two layers in order to minimize final product cost. Also the best power dissipation is necessary, so we need to prepare the maximum free space for making thick copper connections.

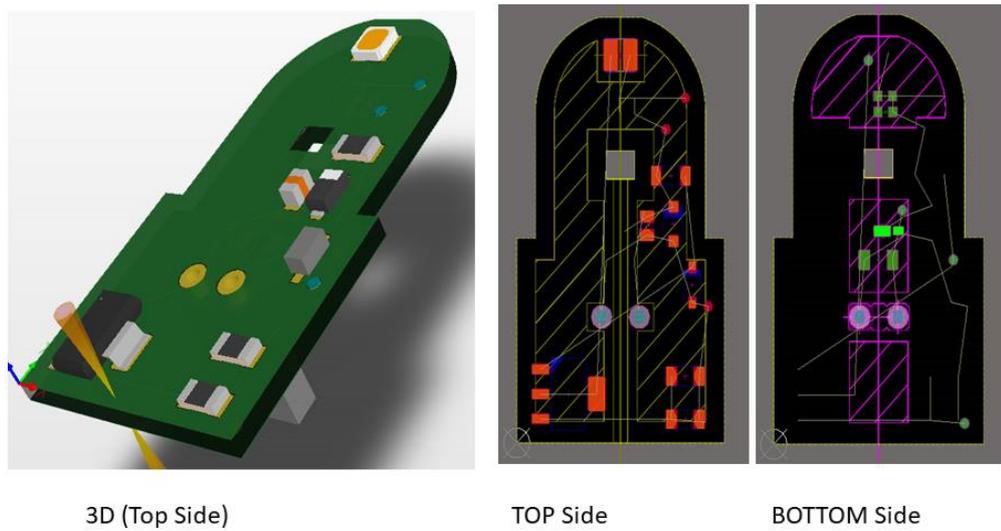


Fig. 12. 3D Top Side, TOP and BOTTOM sides.

- Place the components with fixed position (from mechanical Design) as LED and Connector.
- Take care with the defined component placement allowed areas.
- Place all components inside each circuit block as near as possible between them.
- Keep enough space for the transistor to control current through led in order to allow the maximum dissipation (high power).
- Place Voltage Dividers close together.
- Check the final placement with the real mechanical housing by embedding 3D step.

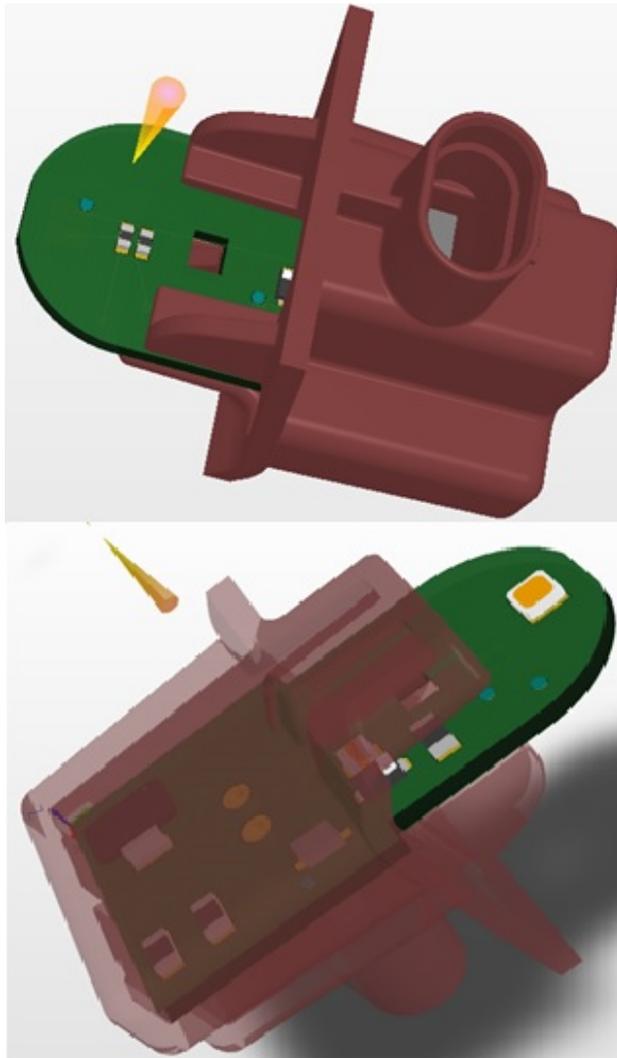


Fig. 13. 3D (Top Side), TOP and BOTTOM sides.

- Perform the PCB Routing
  - Sometimes, once start the routing some possible placement improvements are detected, so it means additional 3D checks have to be done for assure no problems with complete product.
  - Many vias are added for improving the thermal dissipation (this is special board where power dissipation is quite important).



Fig. 14. PCB routing.

- Production Panel Design. Must be done to make a quicker and cheaper assembly process. The panel contains many single PCBs with some holes and marks. Dimensions of Panel are defined according to production plant requirements and bare board manufacturer best working panel profitability. The Working panel contains many production panels.



Fig. 15. Production panel design.

- DRC (Design Rule Check) & DFM (Design For Manufacturing) check.
- DRC allows to PCB Designer to detect any possible error and assure the final PCB quality. The checks are done automatically following the stetted up Design rules. Some of the DRC checks are related to following rules:
  - Electrical: Un-Routed Net, Un-Connected Pin, Short-Circuit, Clearances, ....
  - Routing: Copper Width, Via Style, routing layers, differential pairs, ....
  - SMT: distance from any SMD to Copper Plane, Corner, ....
  - Manufacturing: Silk to Solder Mask, Silk to Silk, hole to hole, hole size, ....
  - High Speed: Vias under SMD, Parallel segment, via count, matched lengths, .....
  - Placement: Maximum height, component clearance, allowed layers, ...
  - Other specific user design rules
- DFM is done by Panel assembly production plant, and reports to PCB Designer any issue affecting the assembly process quality.
- PCB production panel documentation generation for Bare Board manufacturing and assembly:

- Necessary for PCB Bare Board manufacturing
  - Gerber Data
  - Drill Data
  - PCB Specification Sheet
- Necessary for PCB Assembly
  - Gerber & Drill Data
  - Files for SMD solder paste stencil fabrication
  - PCB Specification Sheet
  - PCB assembly information (pdf 's for any pcb layer)
  - ODB++ (universal format)
  - Test Points report: for In Circuit Test (ICT) and Functional Test
  - Pick & Place file: for components auto insertion machines