Nistec
Excellence across the board

# The Secret Seven

## Seven Rules for PCB Design that a Hardware Engineer Must Be Acquainted With

Arbel Nissan, Nistec Design

**The design of a PCB in a manner that is suitable for manufacturing, enables to improve the quality and reliability of the PCB, shortens its introduction into the market (TTM – Time to Market) and even reduces the manufacturing costs.**
**How can this be done? Following are seven rules one should be well acquainted with during PCB development.**

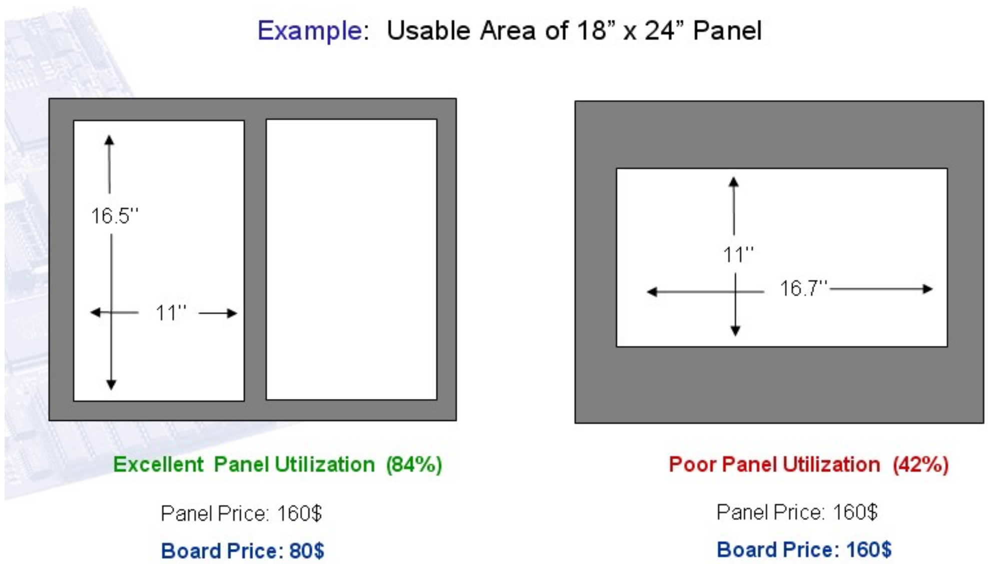
1. **PCB Size and Configuration**

   1.1 Matching the PCB size to the size of the manufacturer's raw material – the raw materials used for PCB manufacturing arrive in the form of fixed-size plates. The manufacturer cuts the raw material in accordance with the dimensions of the PCB intended for manufacturing. The cost of the raw material is approximately 30% of the cost of the PCB manufacturing process. The more efficiently the manufacturer utilizes the raw material, the lower the PCB cost will be. For instance: most manufacturers work with raw materials measuring 18 X 24 inches. Illustration No. 1 demonstrates how a 0.2 inch alteration in the PCB´s length optimizes the utilization of the raw material and subsequently leads to a substantial reduction in the cost of the PCB's manufacture. It is recommended to consult with the PCB manufacturer in order to ascertain the dimensions of the raw materials he utilizes and how this process can be maximized.

   1.2 Ensure the minimal and maximal dimensions of the PCB – the automatic machines used for the manufacture and assembly of PCB's have physical limitations related to PCB dimensions. For example, SMT PCB assembly machines can assemble a PCB that measures 60 – 508 mm in length and 60 – 450 mm in width.

1.3 PCB stability on the assembly line – the assembly lines comprise conveyors on which the PCB is conveyed during work. The PCB must be placed on the conveyors in a stable manner and it should maintain minimal rigidity. In order to allow mechanical strength to the PCB during the assembly process, a symmetrical rectangular PCB must be designed, when the PCB's length is larger than its width. When the PCB's width is larger than its length, its stability on the manufacturing line is impaired. The high pressure exerted on the center of the PCB can cause PCB bending during the assembly process. Likewise, it is advisable to maintain a component-free area measuring 5 mm along the edges of the PCB, that will be used for holding the PCB on the conveyors. In case this is not possible, rims should be added along the PCB. Asymmetrically-shaped PCB's make the PCB assembly process very difficult. In these cases, the PCB´s stability on the manufacturing line is low and hence the reliability and accuracy of the components' placement/tinning will be low. In these PCB's, we must perform an artificial supplement in order to attain a rectangular PCB size. This supplement will be removed during the manufacturing process before the PCB´s delivery to the customer.

Illustration No. 1 – Optimal utilization of the raw material which enables to reduce the manufacturing costs




2. **Panelization Design**

The panel's design is aimed at adjusting the PCB's dimensions to the manufacturing line. This activity enables on the one hand to maintain the size of the generic PCB, and concurrently, adjust it optimally and efficiently to the manufacturing line. A panel that contains several small PCB's can be designed. Through this method, the panel will be assembled on the manufacturing line and afterwards the PCB's will be dismantled from it for the purpose of further work. The time duration during which the machines will work on a single PCB, while it is placed on the panel, will be significantly shorter than when it appears as a single PCB, even if we take into consideration the time required for dismantling the PCB from the panel following its assembly, in order to restore it to its generic state. In this manner, we will be able to make better use of the manufacturing machines and hence diminish the cost of labor. The reduction is more significant when it comes to the assembly of large series. Working with a panel comprising several PCB's also streamlines the manual work carried out on the PCB, for instance, manual soldering as well as electrical tests, such that in a single panel testing cycle several PCB's are tested simultaneously. Appropriate panel design must take into consideration on the one hand the panel's strength and stability during its conveyance on the assembly lines, and on the other hand, the panel should be designed such that the PCB's separation from it after the assembly will be carried out easily and efficiently. The separation between the PCB's across the panel must be performed in the most convenient and rapid manner possible. Two main methods can be used to achieve this: a. V-CUT. b. Break-Away Tabs. It is necessary to be well acquainted with the advantages and disadvantages of each method in order to select the method that should be utilized. It is recommended to refer to the IPC-2221 Standard which details the manner of designing the parameters of both methods.

3. **Placement of the BGA Components**

The BGA (Ball Grid Array) component has a special status in the PCB assembly process. There are several reasons for this: a. Since the BGA component usually comprises a massive body, higher energy is required for its soldering compared with other components. b. In the event that its dismantling from the PCB will be required later on, this process cannot be carried out manually like the rest of the components, but by using a dedicated machine. c. The BGA component soldering test is carried out by means of an X-ray machine, unlike most of the electronic components on the PCB. d. The cost of the PCB component is usually higher than the rest of its neighbors on the PCB. The soldering process of the PCB is carried out at high temperatures of about 245 – 250 degrees Celsius. During this process, we must ensure that thermodynamic balance exists on the surface of the PCB. In light of the variety of components assembled on the PCB, high thermal variance is exhibited on its surface. Abnormal temperature difference between two points (marks) on the PCB's surface during the soldering process may lead to PCB deformation in the reflow oven, shorts and even a deformed PCB upon exiting the manufacturing line. The BGA component, as a result of its nature, is soldered at a substantially higher temperature with respect to the soldering of the other components on the PCB, and therefore close attention should be paid to the location of the BGA component on the PCB. In PCB's comprising a single BGA, it is recommended to assemble it at the center of the PCB with the aim of allowing optimal and even heat release on the PCB surface. In PCB's comprising several BGA components, it is recommended to place them at symmetrical fiducial marks across the PCB in order to ensure thermodynamic balance on the PCB's surface. Furthermore, it is recommended to refrain from placing the BGA components on the PCB's edges since these areas tend to be colder during the soldering process.

As before-mentioned, PCB dismantling, if required after the assembly, is carried out by means of a dedicated machine. Hence, in order to enable this dismantling in the future, a 3-mm component-free area should be left around the BGA component.

Moreover, in PCB's that contain a BGA component on both sides (which is not recommended unless "all hope is lost"), it is advised to make sure that the BGA components are not placed one under the other so as to allow a reliable inspection of the component using an X-ray machine. The BGA components located on both sides of the PCB, one beneath the other, are visible during the X-ray process and they can mislead the results of the component soldering test.

## 4. Spacing between Components

The components' placement on the PCB bears great importance in the success of the PCB's assembly on the manufacturing line. Minimal spacing must be maintained in between component packages. Component density that is too high may result in a low manufacturing yield. Furthermore, in certain cases, the performance of completions and amendments on PCB's with components placed in too close proximity, is almost an impossible mission. In PCB's where the spacing between the components is insufficient, the visual quality control process, both manual and automatic in the AOI machines, may not be optimal. A good rule of thumb recommended for small components (such as 0402 and below) is to maintain a minimal distance of 10 mm. For large components (such as BGA, connectors etc.), it is recommended to maintain a minimal distance of 30 mm. The IPC-2221 Standard defines the desired distance that should be maintained between the components on a PCB.

5. **Design in accordance with the Manufacturing and Assembly Processes**

   It is important to adjust the PCB as much as possible to its manufacturing and assembly processes. An appropriate design of the PCB within this framework, will aid in ensuring a rapid and faultless manufacture and subsequently, in reducing the product cost as well. In general, it is preferable to place the components on only one side of the PCB. A PCB that comprises SMT components on both sides, will undergo the soldering process twice. Placing the components on one side shortens by almost half the time required for working on the PCB during its assembly and saves the cost of dedicated tools (such as a stencil). Since inevitability usually does not allow the placement of components solely on one side of the PCB, it is recommendable to place irregular components on the C.S. side, for example the following components – TH, IC, FP, Oscillators, BGA, high components, heavy components and sensitive components. These components may be harmed during the manufacturing process when they are located on the P.S. side, also due to the fact that the P.S. side undergoes the soldering process twice in the reflow oven. Conversely, it is recommended to place passive components, such as resistors and capacitors, which are more durable during the soldering process, on the P.S. side.

   TH components can be currently also soldered using automatic SMT machines, and this bears many advantages. The assembly of a TH component on an automatic SMT assembly line is close to 10 times faster than its assembly using the alternative methods, which are manual or semi-automatic. In addition, in light of the fact that the SMT assembly line is automatic, the cost for assembling a TH component on this line is almost 4 times lower compared with the assembly of a TH component using the traditional methods. Therefore, it is recommended to plan the assembly of a TH component on an automatic SMT line. For this purpose, the TH component should be treated similarly to the SMT component, that is, openings must be defined in the tin stencil through which the solder paste will traverse to the holes on the PCB (Illustration No. 2). When defining the dimensions of these openings, several parameters must be taken into account, among them: the PCB's thickness, the length of the pins of the TH component as well as the diameter of the hole on the PCB. It is necessary to verify in the data sheets of the TH component that it is resistant to the soldering temperature of the SMT line (up to 260 degrees Celsius).

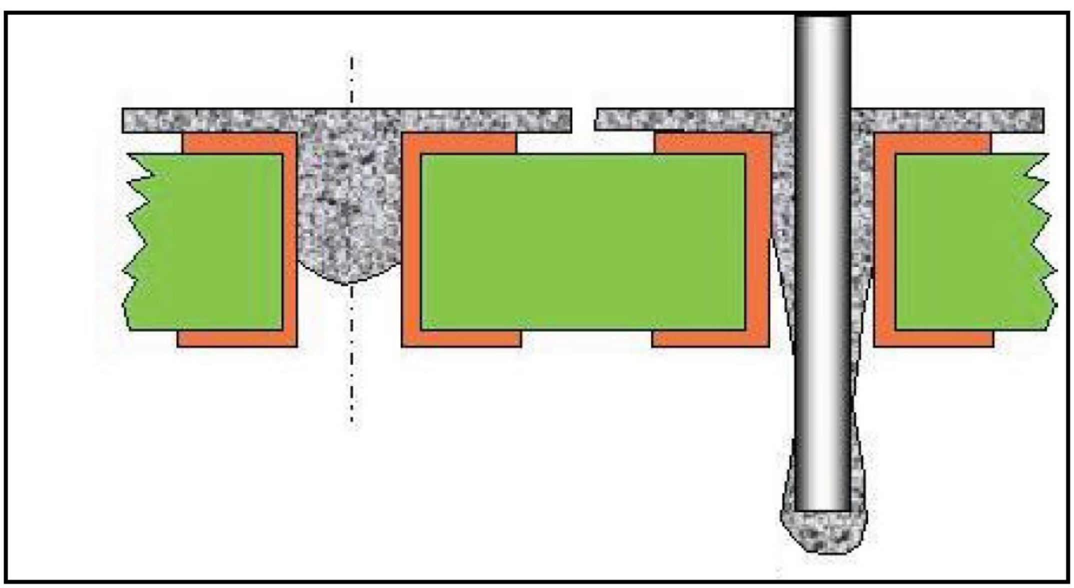


Illustration No. 2 – Soldering of a TH component on the SMT automatic assembly line
The solder paste traverses through the hole on the PCB and solders the pin of the TH component

6. **Designing Fiducial Marks**

The fiducial marks indicated on the PCB are utilized by the automatic machines on the assembly line. The correct placement and definition of these marks are of cardinal importance to the quality of PCB assembly. According to these marks, the machines carry out the precise operations, such as: component placement, tinning, visual tests (AOI). This need is highlighted when the trend for miniaturizing the electronic products becomes more and more prevalent. In dense (high-density) PCB's, every miniscule deviation in the accuracy of the PCB's placement may impose crucial consequences on the PCB's quality and reliability. Therefore, it is imperative to observe a number of rules when designing the fiducial marks on the PCB.

First, solder mask release should be enabled, thus preventing the concealment of the fiducial mark and allowing the automatic manufacturing line machine a rapid, clear and precise identification of that mark. It is necessary to place 3 fiducial marks on the PCB's surface asymmetrically in order to allow the automatic machine to recognize univalently that the PCB has been inserted onto the machine's conveyor in the right direction. The marks should not be placed on the PCB's edges (a minimal distance of 0.25 inch from the PCB's edges), and when it comes to a panel (see Rule No. 2), the fiducial mark should be placed on the PCB itself and not at the panel's edges. It is necessary to make sure that fiducial marks are placed in the Solder Paste files with the purpose of enabling the usage of automatic tinning machines (printers). Likewise, in order to ensure that the fiducial mark is clearly and easily identified during the use of an X-ray machine, a clean area must be maintained on the other side of the PCB for these fiducial points since the X-ray beam penetrates the entire thickness of the PCB.

7. **PCB Testing upon Design Completion Using Dedicated Software**

There are many guidelines related to PCB design for manufacturing, and in this article I tried to detail the main ones. Nonetheless, it should be remembered that even if we work according to all the rules, we may commit errors. Therefore, it is highly recommended to examine the design files using the appropriate tools. There are several good testing software on the market, such as GENESIS, VALOR and more. The VALOR software, by means of its new tool – PCBFLOW (Illustration No. 3), enables to carry out tests on design files in accordance with tens of thousands of design rules within just two minutes. The results of the PCB tests include several key parameters, such as: shorts, breaks, component clearance, dangling lines, minimal distances of PCB coating (solder mask bridge), the ratio between a hole and the PCB's thickness (aspect ratio), silks on the pad, and so forth. This type of test, will detect design errors in a rapid and friendly manner and it will prevent unpleasant manufacturing faults.

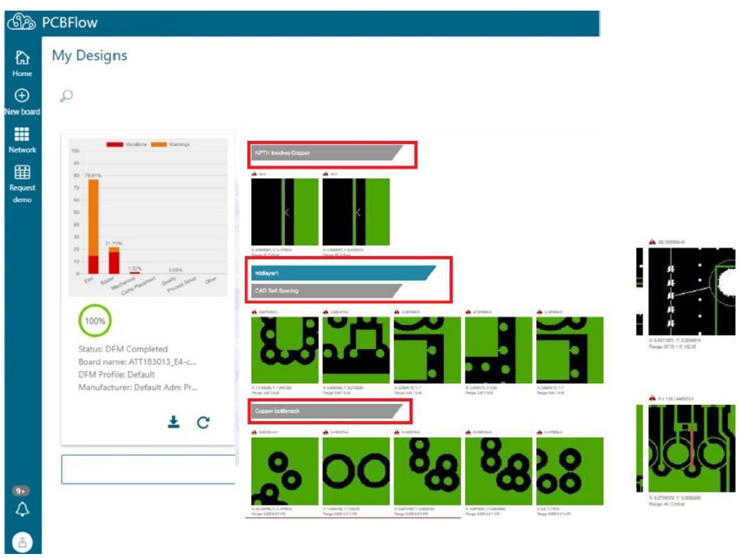



Illustration No. 3 – Running a PCB test report for manufacturing (DFM/DFA) using the VALOR tool of the Siemens Company

**In summary,** at the PCB design stage, PCB manufacturing and assembly considerations must be taken into account. An appropriate design intended for manufacturing prevents errors during the manufacturing stage, shortens the product's TTM (Time to Market), reduces the manufacturing costs and at the end of the day, yields better profits for the company. Carrying out the design in accordance with proper manufacturing rules, aids in the success of the

PCB's manufacture and assembly. The engineers and PCB designers must be aware and fully acquainted with the limitations of the machines located on the shop floor, and in consideration of these, PCB design should be carried out accordingly. The secret of success lies in the synergy between the PCB's design and its manufacture.