A Mistake is Never Repeated

5 Common Mistakes in PCB Design

Written by: Arbel Nissan, Nistec

Both words “mistake” and “error” possess a similar meaning in dictionaries. Yet there is a slight difference between them. A “mistake” – means selecting an incorrect path, when during the time of selection, the correct path is unknown. In contrary, an “error” signifies the selection of an incorrect solution or an incorrect path, when during the time of selection all the data required for a correct decision is supposed to be known. For example: a spelling mistake or a gross error.

In this article, I have written down several common mistakes in PCB design. The consequences of these errors, which are usually detected during the PCB’s production and assembly stages, may lead to critical implications on the PCB’s quality and reliability, as well as its production cost and duration.

1. **The components selection does not suit the production technology.**

   An important stage in the PCB design process is the components’ selection. Within the components selection process, the characteristics that are being examined are certainly the component’s functionality and its suitability to the design requirements, its cost as well as its availability in the market. Nonetheless, there is yet another characteristic which is worth examining, the component’s suitability to the production technology. Selecting a component that does not suit the production technology, forces the utilization of other nonstandard means of component soldering. This degrades the component’s soldering quality, increases the work cost and prolongs the production time.

   Following are several examples depicting this issue:

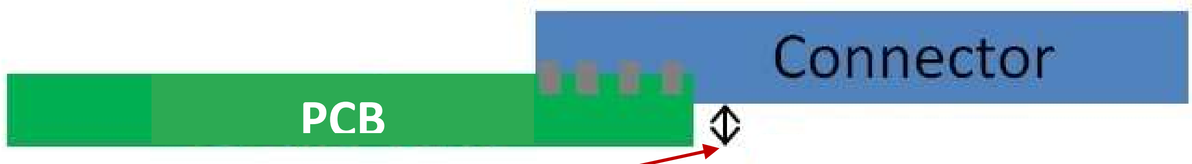
   1.1 <u>A component that does not withstand the soldering temperature</u> – when dealing with a component designated to be soldered on an SMT (Surface Mount Technology) production line, it is necessary to verify that this component does withstand the typical soldering temperature utilized in this technology. In the SMT technology production process, the heat within the reflow oven reaches a temperature of up to 230 degrees Celsius in average. There exist components in the electronics market which are defined as SMT components but their maximal soldering temperature, stated in the manufacturer’s data sheets, is 100 degrees Celsius, meaning that this type of components cannot be assembled on the automatic SMT production line and therefore must be assembled manually. In cases where the subject of discussion is a large production packet, this may increase significantly the cost of assembly since instead of performing the component's assembly automatically on the regular production line, it forces a manual assembly whose assembly cost and even time are significantly higher.

1.2 A component which is not designated for soldering on the SMT production line – there are several components that are defined as SMT components but are designated for soldering by utilizing delicate conductors' soldering in specialized machines using the Wire Bonding technology. These components cannot be assembled on the conventional SMT production line. By the way, it should be noted that for this type of components, the cost of assembly is considerably higher than the assembly cost of a typical SMT component.

1.3 Selecting cutting-edge technology components unnecessarily – the electronic components can usually be ordered in several package sizes. The smaller the component's package, the more complex is its soldering on the production line, which may instigate a non-optimal soldering quality. Sometimes, engineers unconsciously select, from among several alternatives of component package sizes offered by the manufacturer, the smaller package option. For example, selecting a QFN component with a Pitch package of 0.35mm (the distance between the component's leads) instead of a larger package containing the same components with a Pitch of 0.65mm or even a 0.55mm package. In cases where it is not necessary, selecting a larger package will be certainly more preferable and will improve the results of the component's soldering on the production line.

This section is correct except of course in cases where the components' density on the PCB is particularly higher and as a result, the utilization of smaller component packages is unavoidable in order to save precious space on the PCB.

1.4 Selecting a component that requires special assembly and installation equipment – there are electronic components whose installation onto the PCB requires the utilization of designated equipment. This increases the PCB's assembly cost and can also cause soldering results that are not sufficiently qualitative. For example, an SMT connector component which protrudes significantly from the edge of the PCB (Illustration 1). Since the component's tail is relatively long, its center of gravity is located outside the PCB and therefore it tends to bend downwards after its placement on the assembly line. This bending, on one side of the component, triggers a certain rising on the other side of the component, thereby causing a disjunction, that is deficient soldering between the lands and the component's body. The addition of rims onto the edge of the PCB will not be of any help in this case since the component's body is higher than the PCB's vertical zero axis. In order to perform an optimal soldering of this component, it is necessary to order special designated equipment which will enable the support of the component's body and prevent its downward bending. The necessity to order such equipment certainly comprises additional expenses and will also impact the schedules of assembly.



A height gap between the component's body and the PCB, which does not enable the component's stability on the PCB. In this case, the component bends downwards and thus triggers a disjunction between the component's body and the PCB and as a result, improper soldering.

In this case, we will have to prepare a special carrier apparatus which will support the component during the soldering process. The costs of this carrier apparatus are high and likewise its provision time is relatively lengthy.

Illustration 1 – Selecting a component that requires special assembly and installation equipment

2. Thermal imbalance of the PCB

The assembly process of the PCB on the production line using SMT technology includes heating of the PCB in a reflow oven. The temperature sensed within this oven is high, ranging to about 260 degrees Celsius. This heat spreads over the surface of the PCB. In the event that thermal imbalance exists, such that during the soldering process, substantial temperature differences are generated between two points on the PCB's surface (ΔT)

(more than 7 degrees Celsius), the PCB will suffer a thermal shock during the heating stage in the reflow oven. The result of this thermal shock may be PCB curvature, deficient soldering in certain components and more.

This phenomenon is greatly affected by the location of the electronic components on the PCB. The electronic components situated on the PCB are heterogeneous and they may contain small passive components (such as: resistors, capacitors etc.) whose heat signature is low, together with large active components (such as: BGA and LGA components) whose heat signature is high. The concentration, on the one hand, of electronic components whose heat signature is high in a certain area of the PCB, and on the other hand, the concentration of electronic components whose electronic signature is low in another area of the PCB, may cause significant temperature differences between the two abovementioned areas and thus instigate thermal shock.

Therefore, during the PCB's design process, it is very important to ensure that thermal balance is maintained on the PCB. One of the methods recommend to prevent thermal shock is to enable a uniform dispersion of the components whose soldering temperature is high all over the PCB's surface and disallow their concentration in a certain single area.

Another recommendation to maintain the thermal balance is to certify that the stack-up of the PCB layers is symmetrical. That is, ensuring that the copper and ground layers are evenly divided from the center of the PCB towards both sides of the PCB (Illustration 2).

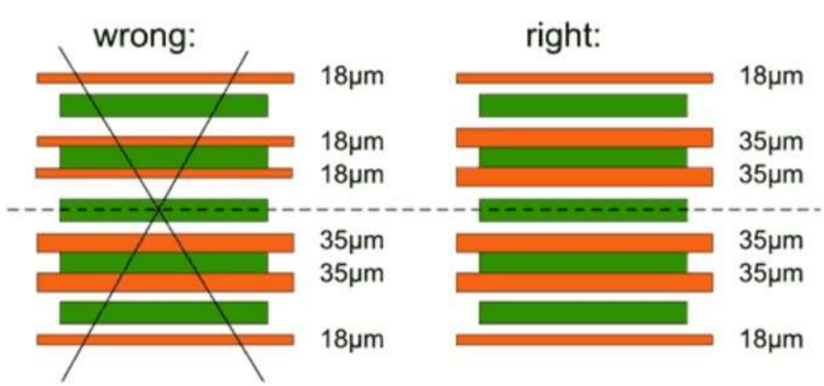


Illustration 2 – PCB layers' stack-up design

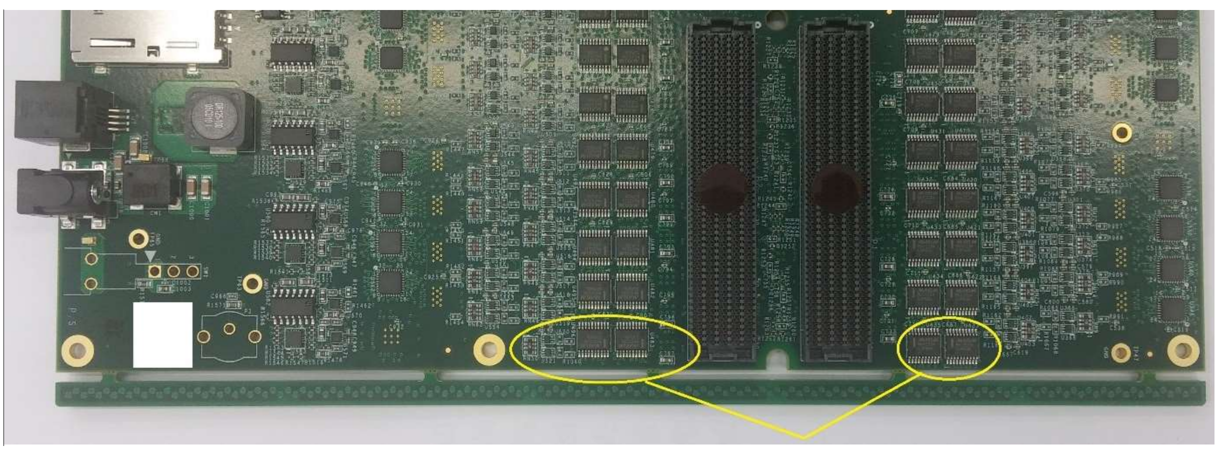
3. **Incompatibility between the manufacturing and assembly technologies**

The PCB's production life cycle includes two main stages: the production of the PCB itself as well as the assembly of the components onto the PCB. An incompatibility between the technologies of these two stages may lead to the disqualification of the entire production packet. For example: defining raw materials for the PCB's manufacturer which suit the tin-lead technology while simultaneously assembling the PCB using the lead-free technology. That is, in the guidelines provided to the PCB's manufacturer, it is noted that the raw materials must suit tin-lead technology assembly which is characterized by relatively low temperatures, for instance – raw material possessing TG-130 (TG = Glass Transition Temperature, a temperature where the raw material traverses from a solid-rigid state to a flexible-elastic state). At the same time, the PCB's assembly is carried out using the lead-free technology which is characterized by high soldering temperatures, about 40 degrees Celsius higher than that of the tin-lead technology assembly process. Therefore, this case comprises incompatibility. When this PCB, whose raw materials suit low assembly temperatures, will be assembled using the lead-free technology which is characterized by high temperatures, the PCB will be distorted during its assembly process in the reflow oven, causing low quality results and even the disqualification of the entire production packet. In this case, it would have been appropriate to define raw materials with a higher TG for the PCB's manufacturer, such as TG-170, in order to ensure that the PCB withstands the high temperatures required in the lead-free technology assembly process.

4. **Components' placement at the edges of PCB's**

The PCB traverses between stations on the assembly line while moving on a conveyor belt. This conveyor belt, which is located along the entire production line, is composed of two parallel tracks that constitute the basis on which the PCB is situated during the different assembly stages. These tracks comprise a small depression that enables the PCB's stability on the conveyor belt. The projection of the depression's area on the PCB is unreachable by the machine's head and therefore it is forbidden to place components within this area. Placing electronic components at the edge of a PCB (< 5mm) may make it difficult or even prevent the machine's head from placing these components accurately on the PCB. In light of this, it is necessary to verify that a "sterile" component-free area is defined at a distance of 5mm from the PCB's edge. In exceptional cases, it will be possible to define even smaller distances, but solely upon advanced coordination with the PCB's assembler.

In the event that the PCB is very dense and it is required to utilize the entire PCB area for placing the components, it will be possible to take advantage of the margins that will be used as a support basis for the PCB during its assembly on the production line. In this manner, it will be possible to place the components even nearer to the PCB's edge, up to a distance of 1-1.5mm from it. In this case, margins will be added to the PCB, wide enough, which will constitute the PCB's point of application on the tracks of the assembly machine. These margins will be removed after the assembly in order to resume the PCB's generic size (Illustration 3).



Placing electronic components at the edge of a PCB (< 5mm) may make it difficult or even prevent the machine's head from placing these components accurately on the PCB. Adding margins to the PCB enables the placement of components on the PCB's edge, up to a distance of 1mm from its edge.

Illustration 3 – Placing electronic components at the edge of a PCB

5. Placing fiducial markers on the PCB's edge

The fiducial markers (fiducials) indicated on the PCB are used as datum points that assist the machine's head to pinpoint the precise location of the components. Correct locations and definitions of these markers during the PCB's design process have significant implications for the component's assembly quality on the PCB. According to these markers, the machines carry out their precise actions, such as: solder paste application, components' placement, AOI visual inspection, X-Ray inspection and more. Since the machine's visual field at the edge of the PCB is limited, the location of these fiducial markers on the PCB's rims may prevent the placement machine's head from identifying the markers. If the machine's head is deprived of the ability to locate the fiducial points, the components' placement onto the PCB will become impossible. Therefore, it is important to meticulously define these important fiducial markers at a distance larger than 7mm from the edge of the PCB during the PCB's design stage.

Additionally, the release of solder mask must be allowed, in order to prevent the concealment of the markers and enable the machine a rapid, clear and precise identification of the fiducial markers.

In the same context, it should be noted that it is recommended to place 3 fiducial points asymmetrically on the PCB's surface in order to allow the automatic machine to uniquely identify that the PCB has been entered into the machine's conveyor belt in the correct direction. Placing 4 fiducial markers symmetrically on the PCB's surface may cause a situation where a reversed insertion will be also perceived by the machine as a normal correct insertion and will trigger an incorrect placement of the components. Likewise, for the utilization of automatic stencil (solder paste) printers, it is necessary to place fiducial markers on the SolderPaste and SolderMask files as well.

Moreover, in order to certify that the fiducial marker is clear and easily identifiable during the use of the X-Ray machine, a component-free area (larger than 1mm) must be maintained beside the fiducial markers along the entire width of the PCB, that is, from the other side of the PCB as well. This is imperative since the X-Ray beam penetrates the entire width of the PCB and therefore photographs its other side as well. Placing the components next (< 1mm) to the fiducial markers on the other side of the PCB will not allow the identification of the markers by the X-Ray machine, hence making the examination process carried out by this machine much harder.

In conclusion, someone must have said once that "a mistake is always repeated, but when the eraser ends before the pencil – it means that you have exaggerated".

In this article I have listed several common mistakes in PCB planning and design. These mistakes stem mostly from the lack of knowledge of the engineer and/or the PCB's designer. Therefore, it is recommended that the factors involved in the planning and design of the PCB will possess enough knowledge regarding the production and assembly technologies. It is also advisable and worthwhile to consult with the PCB's designer and the PCB's assembler and to be assisted by their experience in the field while still in the design stage, in order to ensure the PCB's suitability and compatibility with these processes. Synergy between the planning and design stage and the production stage will certify that the mistake will never be repeated.