SIERRA CIRCUITS

DESIGN FOR ASSEMBLY HANDBOOK

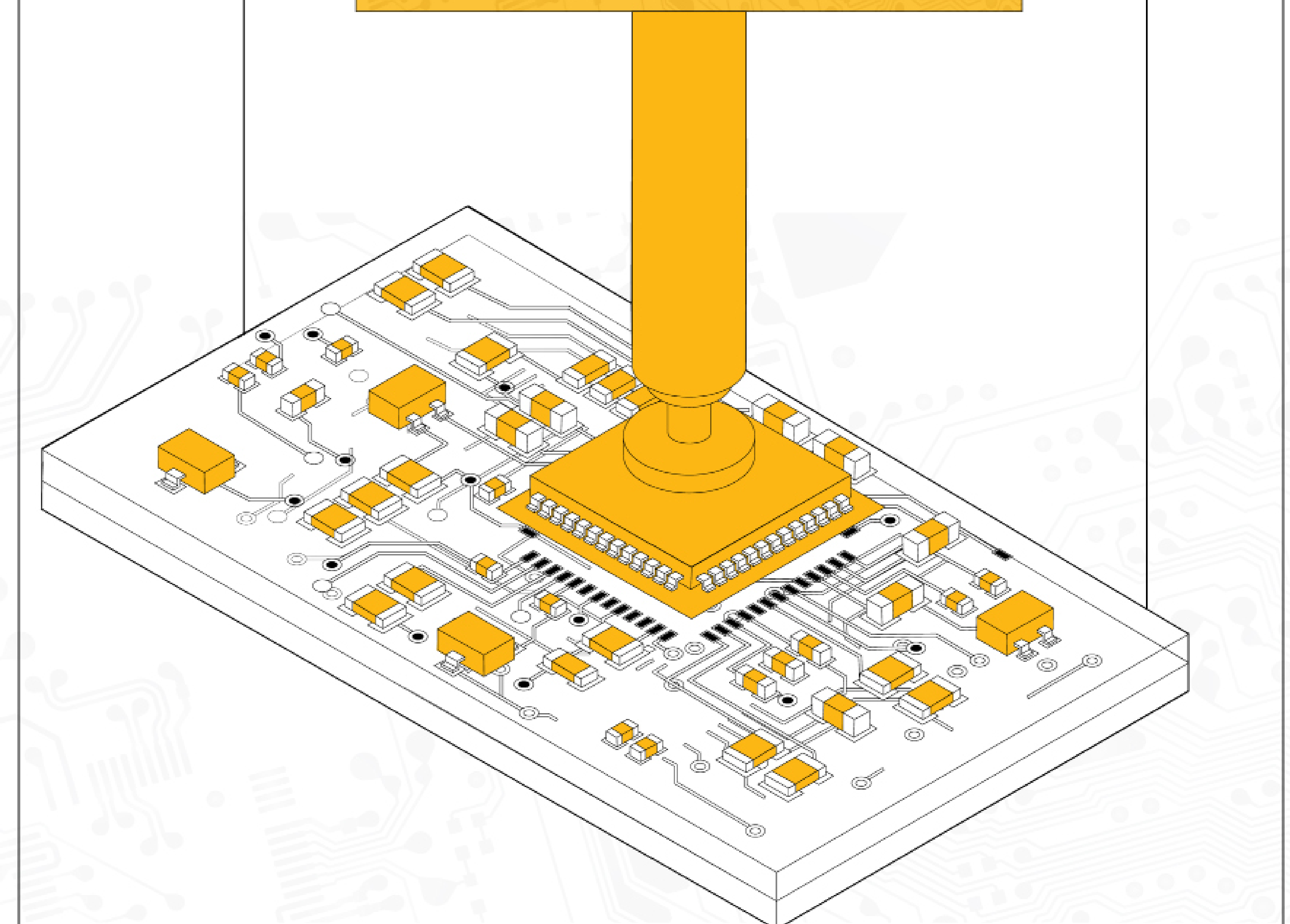


Table of Content

Table of Content

Table of Content

1. DFA definition and overview

1.1 What is design for assembly (DFA)?

DFA is a systematic process that is used to reduce the assembly costs of a product by simplifying its design. This is done by reducing the number of components or parts in the product design and ensuring the parts are easily assembled. DFA techniques are intended to arrive at a simpler product structure and assembly system.

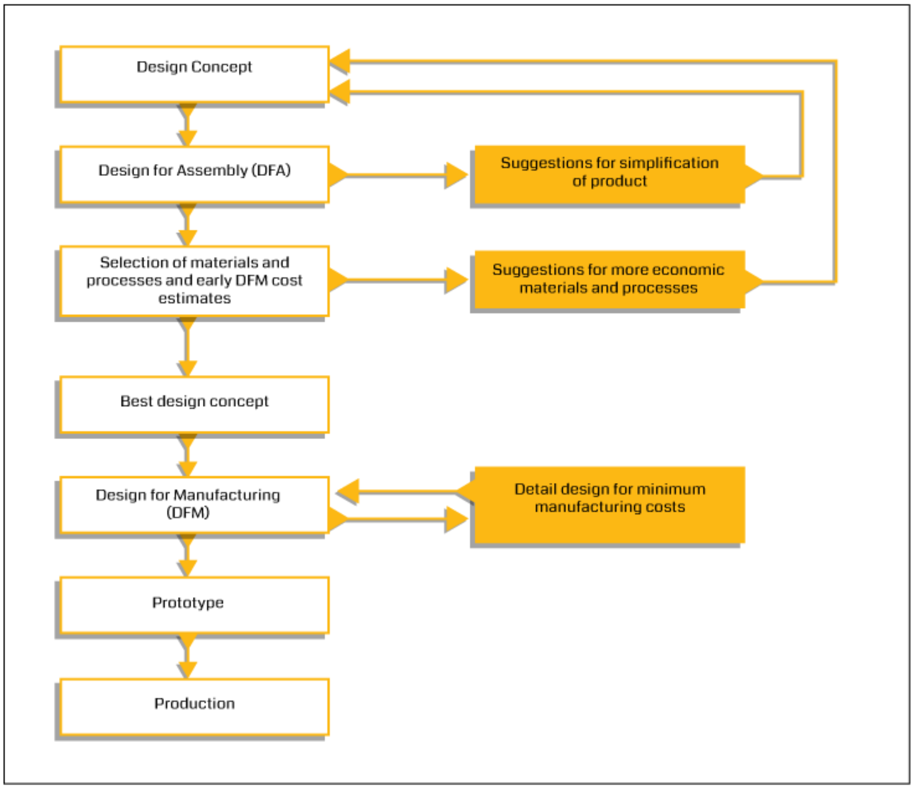


Figure 1: Role of DFA and DFM in PCB manufacturing.

Design for assembly includes a distinct design methodology that considers product functionality along with the cost and efficiency of the product assembly process. Design choices go a long way in ensuring the quality and reliability of the end product.

Pre-PCB manufacturing and assembly DFA includes:

- Firstly, checking the product BOM (bill of materials) and identifying obsolete and end-of-life products. Also, identifying non-washable products and through-hole components.
- Secondly, identifying whether the part numbers in the BOM match with the footprints on the PCB.
- Thirdly, validating whether the components are too close to each other and inspecting problems for assembly.

1.2 PCB design for assembly

Designers and production staff need a crystal clear understanding of DFA to cut down on component procurement and component placement costs. This is responsible for a major portion of the cost involved in PCB assembly. DFA methods can help mitigate project lead time and prevent assembly issues that might need rework or re-design.

PCB DFA is a critical consideration for any board manufacturing enterprise. When implemented accurately, the advantages of DFA are considerable such as minimizing development costs, shortening the product cycle. It also ensures a seamless flow from the prototype to the production stage of manufacturing. Product engineering teams forego the significance of high volume production factors such as component placement and fixtures in favor of accelerated design and development timeframes. It might lead to unforeseen costs in the assembly stage in terms of rework and increased assembly time and cost. This is why DFA is crucial to ensure the process of circuit board assembly is optimized before production begins.

It is significant to note that the DFA considerations vary between prototype board assembly and production board assembly. Prototype and production PCB assembly calls for different design skill sets and appropriate equipment for every stage. The board prototype stage requires a focus on the validation of circuits and board functionality leading to high-quality circuit board prototypes. The PCB production stage places importance on testability and manufacturing guidelines ensuring a smooth and precise production assembly.

1.3 The objectives of DFA

The objectives of DFA are as follows:

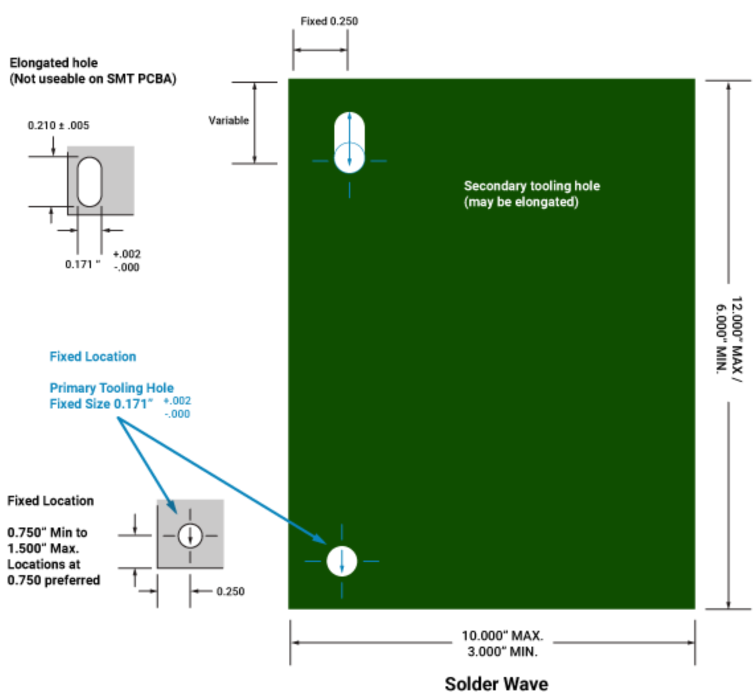
- The components selected in the BOM are available and not obsolete or end-of-life products.
- The components' manufacturing part numbers (MPN) must match with the footprint of the PCB.
- The component placement, sizes, and distances between the components are compatible with the assembly manufacturing processes.
- The solder mask and solder paste layers are correct and meet the production requirements.
- The DNI (do not include) components are correctly marked and verified.

1.4 DFA tips for PCB assembly

Board assembly can be greatly simplified and made more efficient by following these tips:

- Select easily available components and confirm their continued production. This will preclude or reduce possible future production delays.
- Implement component spacing guidelines. [Component placement] impacts how your board can be assembled, the [soldering techniques] that can be used and the type of thermal dissipation required. It may also affect signal integrity.
- Stick to component manufacturer's recommendations of footprints. This is needed to prevent pad mismatches and ensure that accurate markings and identification are present.
- Follow spacing clearances and tolerances that are within your manufacturer's capabilities. This ensures design for manufacturability (DFM).

- Stick to size, spacing, and tolerances for drill holes that fall within your fabricator's capabilities. This also ensures the manufacturability of the design.



Typical example of PCB size specification

Figure 2: PCB size Specification

- Consider the operating environment of the PCB. Boards subjected to significant vibration will need through-hole components as they can be more securely attached compared to SMDs.
- Ensure the use of adequate thermal relief. This guarantees good quality solder joints and rules out certain soldering issues.
- Adopt efficient trace routing techniques. Inadequate solder connections and rework might result from traces misaligned with pads or vias.
- Implement board edge guidelines. Board shape and component placement can largely affect the panelization method used.

2. Aspects of DFA

2.1 Reduction and simplification

One of the prime aspects of DFA is to see if any component can be combined with another component or eliminated. Whenever possible, decide the minimum quantity of components that is required for the PCB assembly. One technique for deciding what part quantities are necessary for the assembly is to go through this checklist:

- If the component can be fabricated utilizing the same material as the other components
- If it will affect the ease of disassembly when combined with another component
- If there will be greater ease of manufacture when combined with other components

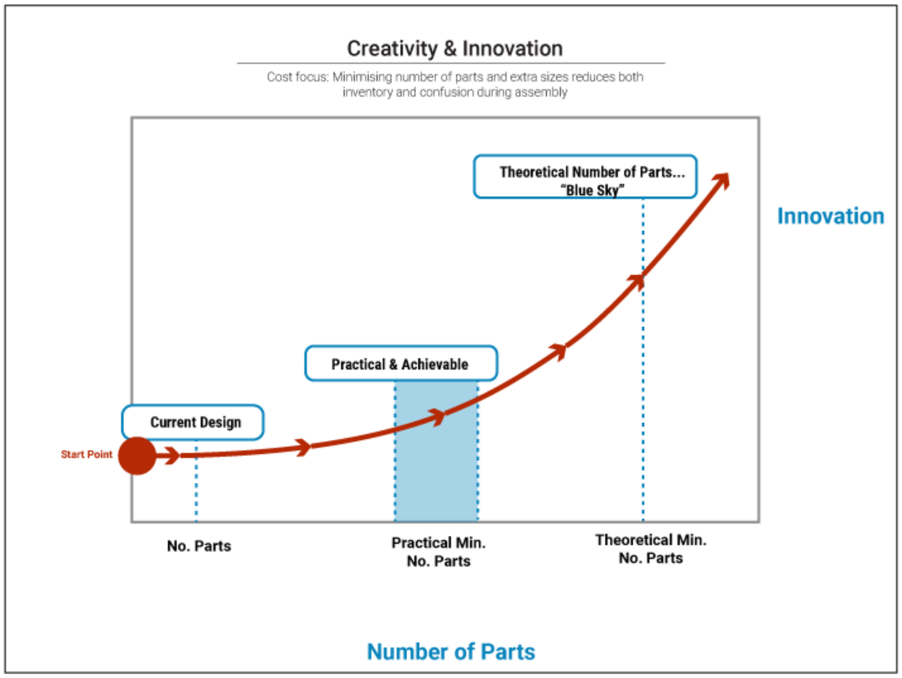


Figure 3: Relation between innovation and component count in a design

When you reduce the number of components on a board, the number of assembly steps will also decrease proportionally. This will in turn reduce the number of assembly errors that might result during the assembly process.

2.2 Standardization of components

The use of standard components in PCB design presents many benefits such as decreased development time and cost. It is a fundamental concept that the use of a custom solution that is complex will drive up the cost of a product. This might even lead to feasibility issues when it's time for high-volume production.

Using standard components helps simplify a product's supply chain and minimize component supply issues. Another clear advantage of using standard and easily available components is that they can be verified easily compared to other components in the design.

2.3 Efficient fastening methods

PCB assembly with the use of fasteners will drive up assembly costs. Thus it is best to reduce the use of fasteners in the PCB assembly. One way to minimize their use iis to utilize surface mount components for power ICs along with heat sinks in the board design.

2.4 Single or double-sided PCB

A single-sided board is one where all components are placed on one side of the PCB. A double-sided circuit board refers to the board which has components mounted on both sides of the PCB. The designer will have to decide which one is better to fabricate and assemble:

- A smaller board with components placed on either side of the board
- A larger PCB with components placed only on one side of the board

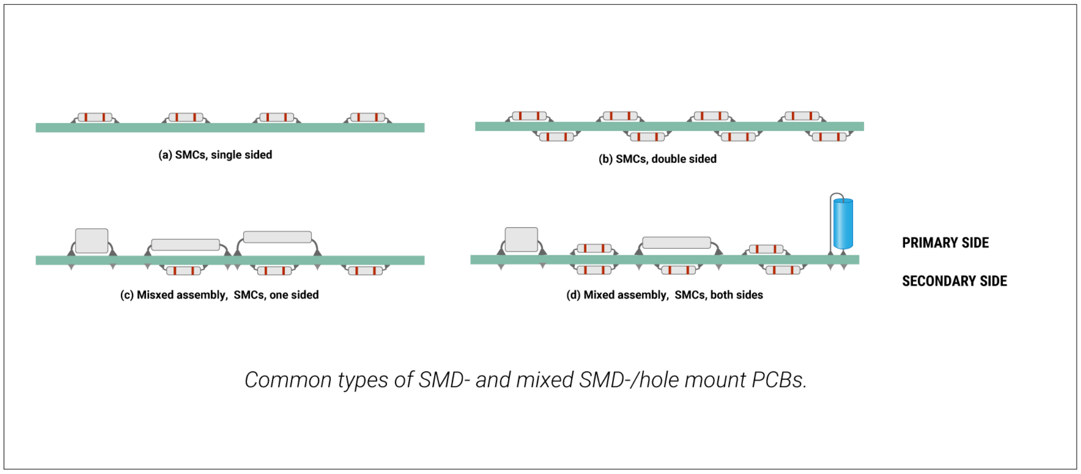


Figure 4: Single-sided and double-sided board assemblies

2.5 Board repositioning and handling

Repositioning a board during the assembly process will significantly increase the assembly time. The repositioning will be most needed in double-sided circuit boards where components need to be installed on both sides of the PCB.

It is good practice to use only surface mount components on one side of the board to facilitate a single reflow soldering step. This can be followed by the mounting of through-hole components with wave soldering or manual soldering.

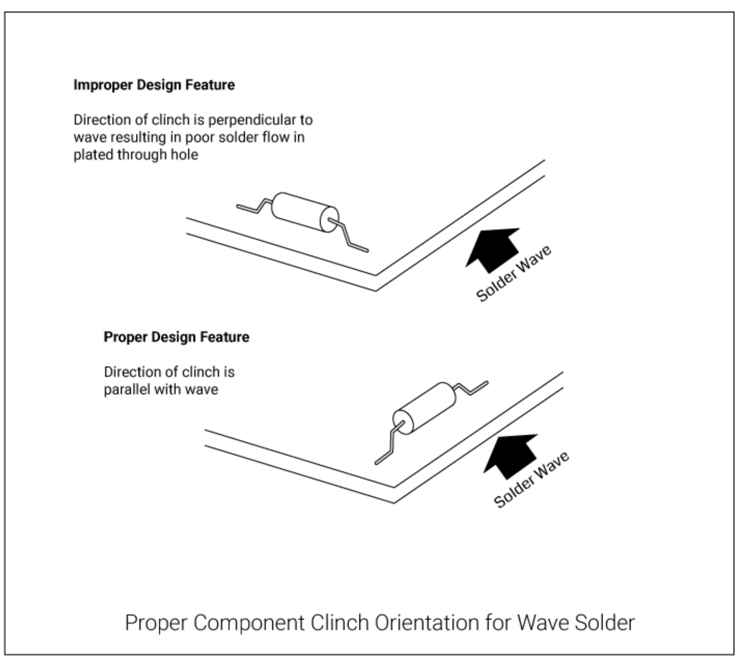


Figure 5: Component orientation for proper wave soldering

2.6 Design for automation

Automated assembly aims to minimize the cost of assembly, component inventory, and enhance the assembly quality and reliability. To ready a product for an automatic assembly, you will need to use components of consistent quality. Such components should conform to close geometric and dimensional tolerances to mitigate the chance of assembly defects due to component mismatch.

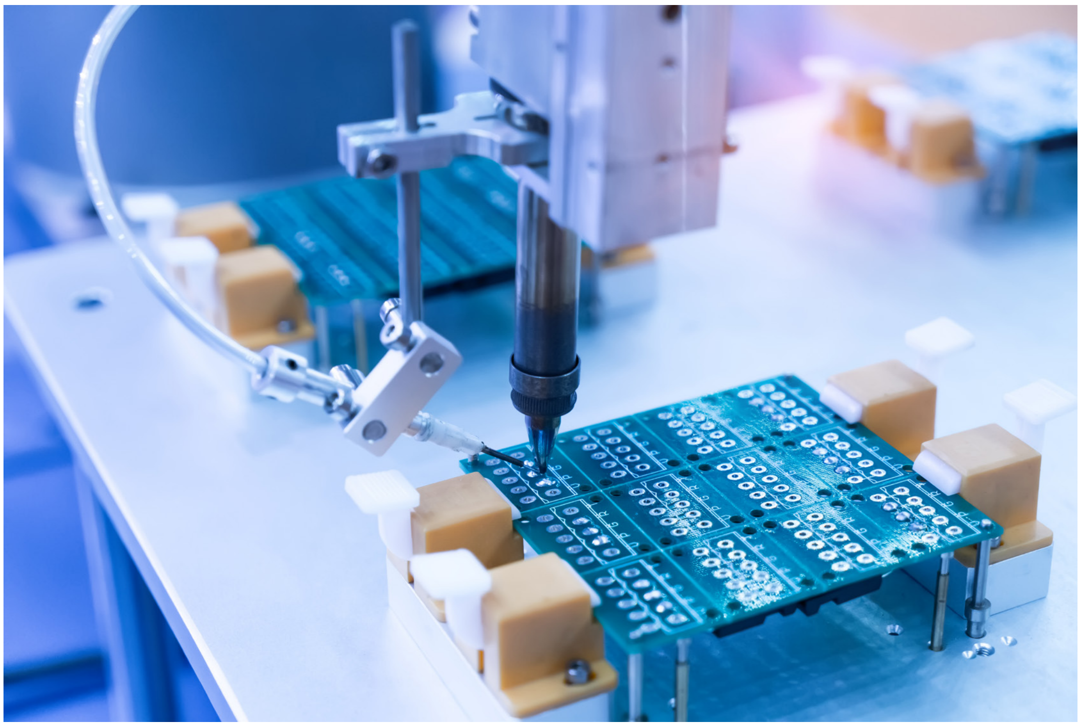


Figure 5a: Automated soldering station

Here are some steps that designers can incorporate to enable easy assembly automation:

- Wherever possible, opt for components that snap together (press fit).
- Try to minimize the number of fasteners used in the assembly. Instead, use self-aligning and self-locating features including guide pins, dimples, and chamfers. When fasteners cannot be excluded, ensure the screw heads used are consistent throughout the design.
- Every component selected during the design should be able to withstand the forces that will be exerted during the assembly process.
- Choosing components that can be oriented easily will reduce the turnaround time resulting in cost and time savings. This will eliminate or at least reduce the requirement of reorienting stations before being fed to the assembly line.

2.7 Design for test (DFT)

DFT methodologies function as a holistic set of techniques to design a PCB layout for the reliable, economical, and speedy testing of completed circuit boards.

There are 3 phases for DFT implementation during the circuit board design and layout process.

- Decide on what connections on the PCB require an electrical in-circuit test.
- Define the mechanical requirements for circuit board alignment and how to allot areas for the test probe to contact the board.
- Generate the necessary data for fixture fabrication and test programming to equip testing machinery.

PCB testing is done in two parts, before assembly (bare-board test) and after assembly (in-circuit test, or ICT)

2.7.1 Pre-assembly test - Bare board tests

Isolation testing: This includes the measurement of the resistance between electrical connections.

Continuity testing: This includes checking for open circuits, which is basically a variation on isolation testing.

Short circuit testing: This includes checking for any undesired electrical connections due to manufacturing or design errors.

Bare board tests are generated using a netlist which is in turn generated from the design data. When you send design data to the manufacturer in ODB++ data format the design files will include a netlist as well. In the case of Gerber design files, the CAM team will generate a netlist using the Gerber files and schematic. Flying probe test or bed of nails tests can be employed for bare board testing.

2.7.2 Post assembly test - ICT or in-circuit testing

In-circuit testing verifies the working of a PCB assembly that is also known as white-box testing. In this method, we employ electric probes to verify the PCB for opens, shorts, and values of capacitance, resistance, and other parameters.

Conventionally ICT used fixture-based testing also known as "bed of nails." Every PCB assembly needed a custom ICT fixture including a group of spring-loaded pogo pins that contact the board assembly at the required test points. Each pogo pin is meant to connect with a node or test point in the assembly when testing. This method is expensive and time-intensive which is why it led to the adoption of the flying probe test.

2.7.2.1 Flying probe test

As suggested by the name, [flying probe testing] makes use of probes that "fly" from one test point to another as instructed by testing software. As there is no requirement for a custom fixture, this method is also known as a fixtureless in-circuit test when used for a board assembly. This method is known to be highly cost-effective for prototypes with low to mid-sized production runs.

Figure 6: Flying probe tester at the Sierra Circuits assembly facility

Flying probe test was developed for bare board testing, and became the default standard. This testing involves checking for opens and shorts between the copper conductor features on the bare PCB. The main electrical parameter that is evaluated is the resistance between two nodes or points.

FPTs have evolved to analyze the inductance and capacitance along with the resistance, making them indispensable to board assemblies as well. The various reasons for the adoption of this mode of testing for board assemblies are as follows:

- Cost-effective when it comes to smaller production runs.
- Elimination of accessibility issues that are present in the testing method that uses pogo pins mounted on fixtures. FPT probes can access much smaller pads as compared to fixture testing. Also while test points need to be designed specifically for fixture-based testing, FPT does not require such cost-intensive measures.
- FPT probes are controlled by a software program that is easy to modify, enabling easy implementation of flexible testing strategies. Probe landing positions are a lot easier to adjust this way.
- Enhanced test coverage gained from better access to test points through automated probing and independent of specially designed test points.

2.7.2.2 Flying probe test process

Testing a board assembly in a FPT requires an FPT test program. This program is usually generated on a PC, offline, similar to an SMT program that is created for [pick-and-place machines] utilizing an SMT programming application.

Creation of the program:

Each FPT offers a test program generating application that operates on a PC. This application needs the board assembly's BOM and eCAD file. You would need the eCAD to be in ODB++ format/ IPC-2581 format /native eCAD design file format as required by the software used to design the board. The BOM on the other hand needs to be in an Excel format.

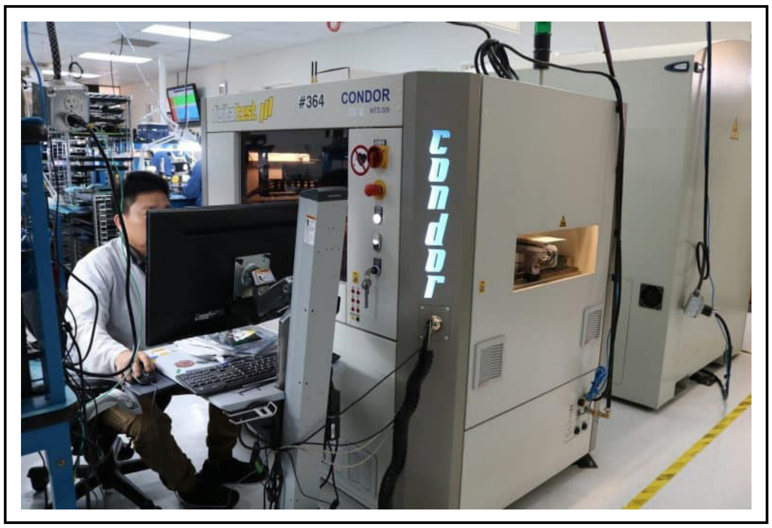


Figure 7: An expert working on the tester program

After the test program is created, it is then loaded into the FPT tester. The PCB assembly that is to be tested is placed on a conveyor to travel inside the tester area where the probes are present.

After the test program is initiated, the probes will contact the pads and unmasked vias according to the pre-loaded program, applying test signals and power to make measurements.

Such measurements are then processed within the tester to find if the circuit sections are providing the expected values within given tolerances. Through this method, FPT will detect unit defects. FPT hardware includes various types of sensors, frequency counters, DC and AC power supplies, signal generators, and more. Such instrumentation is used to provide signals to excite the board points and make measurements on component and interconnection nodes in the assembly.

FPT tries to isolate the component sections between the probes and the rest of the board. Such virtual isolation of components from the rest of the interconnections on the board enables accurate measurement of component values, while still on the PCB.

This tester is also equipped with a camera to aid in the automatic inspection of component polarity. It also executes diode impedance tests on the inputs of integrated devices whose functioning is beyond FPT testing. Most FPTs available today are able to make measurements on IC pads that are beneath the component bodies such as BGAs and QFNs using capacitive probing.

2.8 Poka yoke

Poka-Yoke is a term that refers to mistake-proofing. This is a method that uses tooling, sensors, or other devices to eliminate or design out, the errors within a process. Poka-yoke serves the purpose of cutting down the cost of poor quality which includes repairs, rework, rejects, and returns. These elements of poor quality can be rooted out by incorporating an effective poka-yoke solution.

2.8.1 Defects and process variation

Defects are mostly caused by process variation. Process variation, in turn, occurs due to:

- Poor procedures
- Human error
- Equipment
- Non-conforming material
- Tooling, jigs, and fixtures

Apart from human error, other causes for process variation can be foreseen and remedied with corrective action to remove the cause of defects.

Poka-yoke is meant to identify process variation and close the process before an error occurs. It means catching errors before a defective product is manufactured/assembled by focusing on the process and not quality control.

2.8.2 Types of Poka-yoke systems

There are two approaches to the Poka-yoke process:

1. Control system: This method eliminates human error. An example of this is the four-slot offset tooling system making it impossible for production staff to place a panel onto a machine the wrong way.

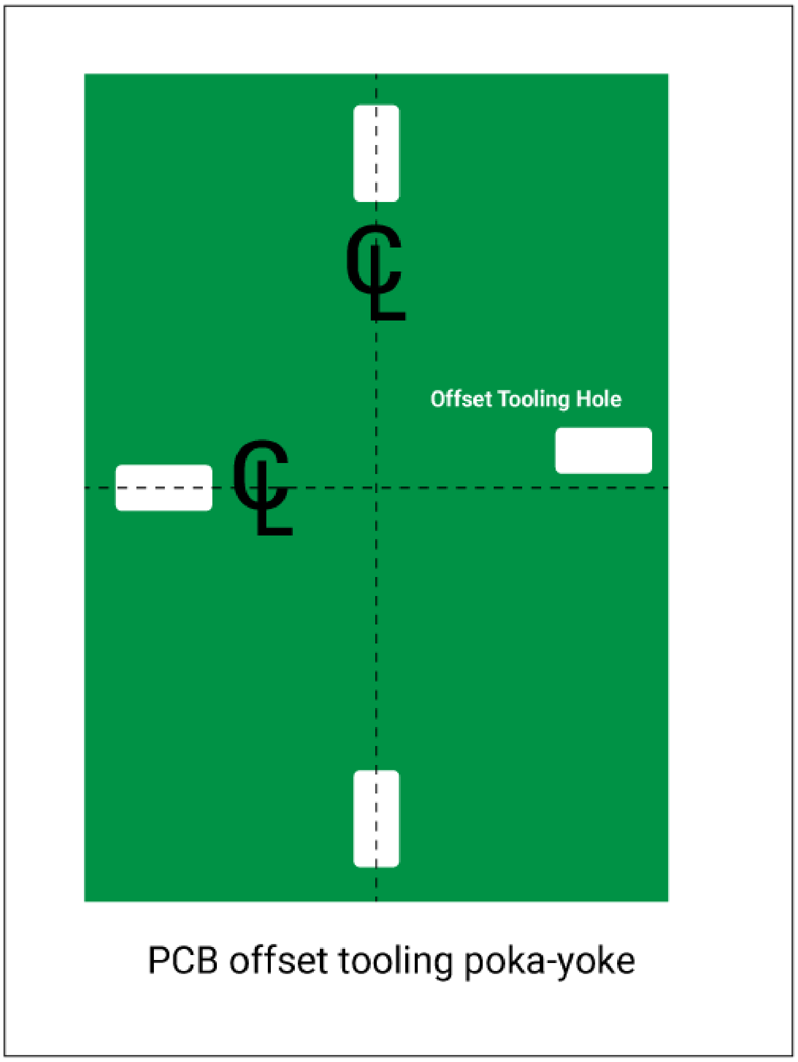


Figure 8: Poka-yoke through offset tooling

2. Warning system: When automatic shutoff for the process is not feasible, warning systems can be used. These can include audio-visual alerts such as lights and buzzers which can alert the process operator of an undesirable condition. This method is unable to root out human error and is not as reliable as the control system method, it is effective to a certain extent.

2.8.3 When to use

Poka-yoke finds application in any process which is open to human error and can be applied to any process in the manufacturing industry. Errors that can benefit from Poka-yoke include:

- **Processing error:** missed operations or unexecuted tasks
- **Setup error:** use of incorrect tooling or improper machine settings
- **Missing part(s):** when parts or components are missed in the lamination or plating process etc.
- **Improper component:** use of incorrect component/part in the process
- **Operations error:** incorrect implementation of a process
- **Measurement error:** inspection errors or testing errors

2.9 Factors affecting assembly speed

Board assembly forms a significant portion of the PCB design and manufacturing process. There are various aspects that come into play that decide the speed of the assembly process. Considering that the speed of the assembly process also affects the manufacturing cost, it is essential to understand the factors that impact this process. Given below are a few such aspects:

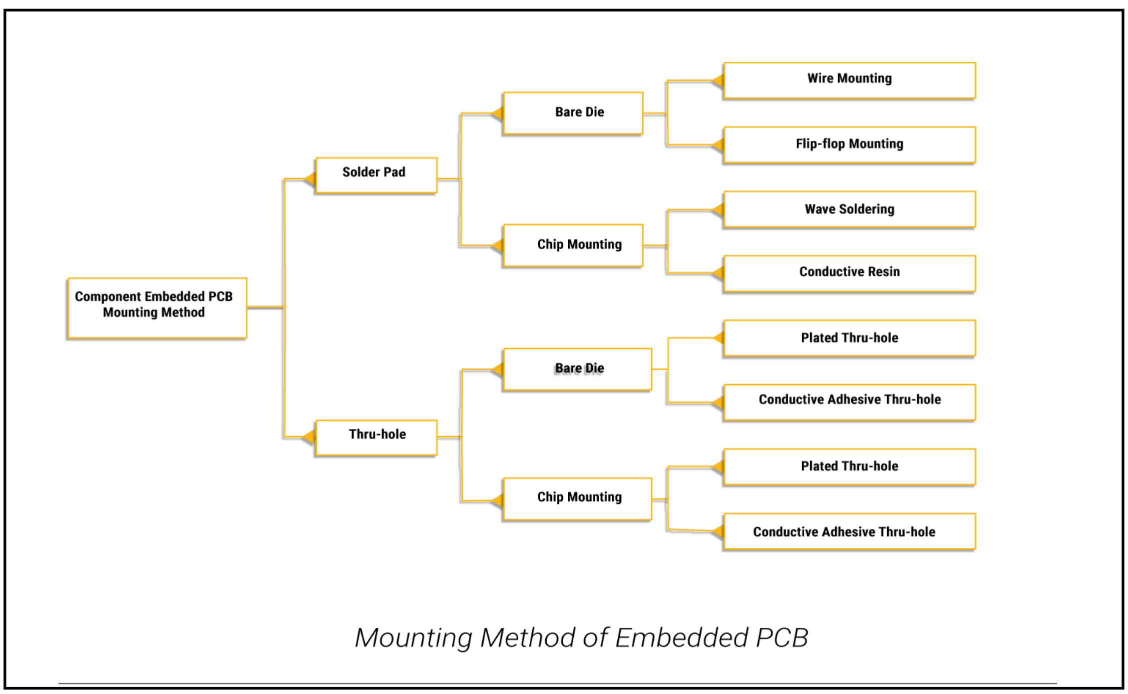


Figure 8a: Mounting methods for PCB assembly

2.9.1 PCB assembly of through-hole parts

The assembly of PCBs is a process where significant costs can be saved or spent depending on the components and assembly process used. SMT components can be automatically placed due to their uniform size, ease of automation, and use of solder reflow. This process is great for the efficient mass production of assemblies. Through-hole parts on the other hand need a more intricate assembly process. Listed below are the aspects of through-hole assembly:

Figure 8b: Reflow soldering machine

2.9.1.1 Component placement:

Even with the availability of many automated component insertion systems, most component manufacturers will prefer the manual installation of parts due to component volumes. Manual insertion of through-hole technology (THT) components before soldering is expensive compared to using pick and place machines that are used for placing surface mount components.

2.9.1.2 Automated soldering:

THT components are conventionally soldered through a wave soldering process. Wave soldering is the conventional form of soldering that is tried and tested for many years and is known to be reliable. SMT components can't be soldered using a wave soldering process. This needs the use of a specialized pallet to mask the SMT components that need not be exposed to the solder wave. Only the THT components that need to be soldered will be exposed to the wave.

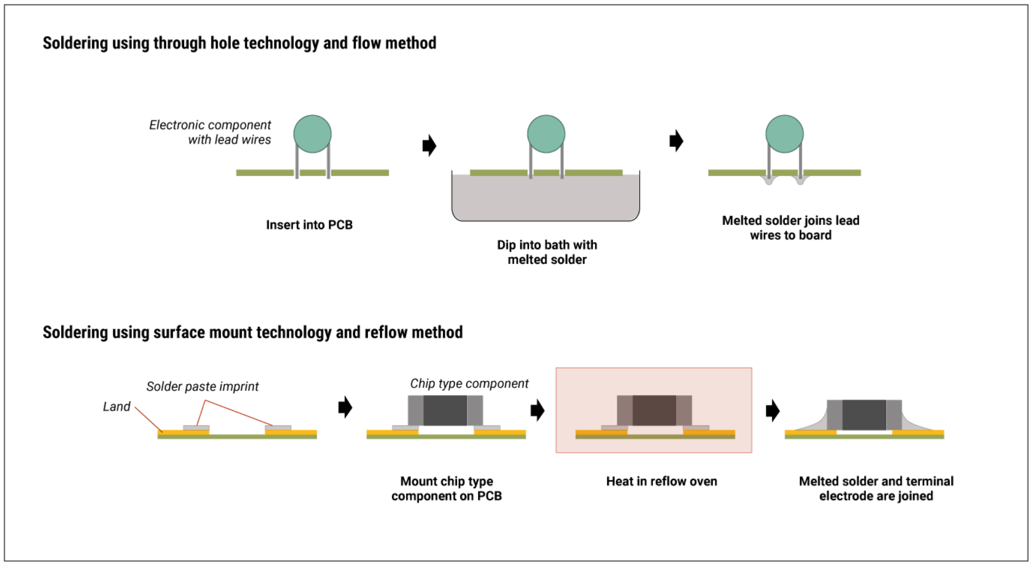


Figure 9: Through-hole flow soldering and surface mount reflow soldering

2.9.1.3 Manual soldering:

High-density through-hole components or areas that cannot be subjected to masking will be soldered manually. This is more expensive when compared to automated soldering.

2.9.2 Do not include (DNI) components

DNI components are parts that will not be procured by the manufacturer and will not be placed on the board during the assembly process. All components that have been designated as DNI on the BoM should be kept in-house. These parts can then be mounted on the board at a later date.

DNI components help prevent lead times. DNI is marked for components that involve larger or heavier objects to minimize the cost of shipping. Some clients choose DNI status for components on their prototype PCBs to experiment with possible components that could function in the board location.

2.9.3 PCB footprint creation errors

PCB footprint creation is a critical part of the PCB design process. If the footprints are not created correctly you might see these problems:

2.9.3.1 Incorrect pad sizes

Pads that are smaller than the requirement can lead to breakout problems for through-hole components and bad solder joints for SMT components. Pads that are larger than the requirement can take up room that is needed for trace routing or even cause SMT components to float out of position during the soldering process.

2.9.3.2 Incorrect pad spacing

Through-hole pads that are too close together or too far apart can lead to issues in component insertion during the assembly process. SMT pads that are too close together or too far apart can cause inadequate formation of solder joints. This will lead to an inadequate portion of the component lead being available for the soldering process.

2.9.3.3 Small component outlines

If the footprints have body outlines that are smaller than what is required, it might lead to assembly issues in automated production lines. These errors can be countered with manual assembly, which will require additional time and cost. Larger errors might not even be suitable for rework and will be rejected by the manufacturer.

2.10 Board assembly notes

PCB fab notes typically consist of design-related information that is intended to help the manufacturer carry out the assembly processes without errors. The details mentioned are as follows:

- Class of the PCB (class 1, class 2, and class 3)
- Number of layers
- Overall board thickness
- IPC standards to be followed
- Color of solder mask
- Color of silkscreen
- Layer-wise impedance details
- Cut-out details
- Stack-up details
- Drill-hole details (drill chart)
- Version number and date

All the information given in the fab notes is critical to help the fabricator assemble the board. It is also essential for future reference in terms of PCB design.

If the board is being designed for a customer it is recommended to get approval from the customer after the aforementioned step. An example of fab notes is shown below.

NOTES: UNLESS OTHERWISE SPECIFIED

1. Fabricate per IPC-600, Class 2.

2. Material: FR4 .
Board shall be 6 layer construction in accordance with Detail A.

3. Finish: Plated through hole diameter as specified are after plating plus or minus .003 inches. The thickness of the copper plating on the wall of the hole shall be no less than .0008 inch.

4. External Layers Finish to be ENGI.

5. Remove all burrs and sharp edges.

6. Soldermask color: GREEN.

7. Fabricator to clip silkscreens using solder mask layers.

8. Silkscreen: WHITE

9. Fabricator to teardrop internal and external pads on all signal nets no matter what the anular ring size. All pads need to be tear dropped.

10. Unless otherwise specified, fabricator may add thieving pattern to external layers. A minimum clearance of .100 inches shall be maintained from thieving pattern to any designed-in copper feature.

11. Cutout in Design: NO

12. No Impedance required on this design.

Figure 9a: Fab Notes

3. DFA requirements

There are various requirements to ensure that the PCB product being manufactured is designed for assembly. This can range from the right format for the design files to checking for component availability before assembly. It can also include the component spacing followed for assembly and several other design practices.

3.1 Preferred file format for assembly

Design files are the prime communication channels between designers and fabricators. Initially, Gerber files were popular in the industry since their introduction in 1980. The mid-'90s saw the introduction of the intelligent ODB++ format which was quickly followed by IPC-2581 standards.

Here is a list of the various files you can send your manufacturer for fabrication:

- Gerber files
- IPC-2581
- IPC Netlist
- Drill files
- BOM (Bill of material)
- Pick and Place files
- Fab drawing
- ODB++ files (Optional)

3.1.1 What is Gerber?

After the layout design of a board is finished and ready to be fabricated, the design is converted to a standard format called Gerber files. This format can be used by the fabricator to manufacture PCBs.

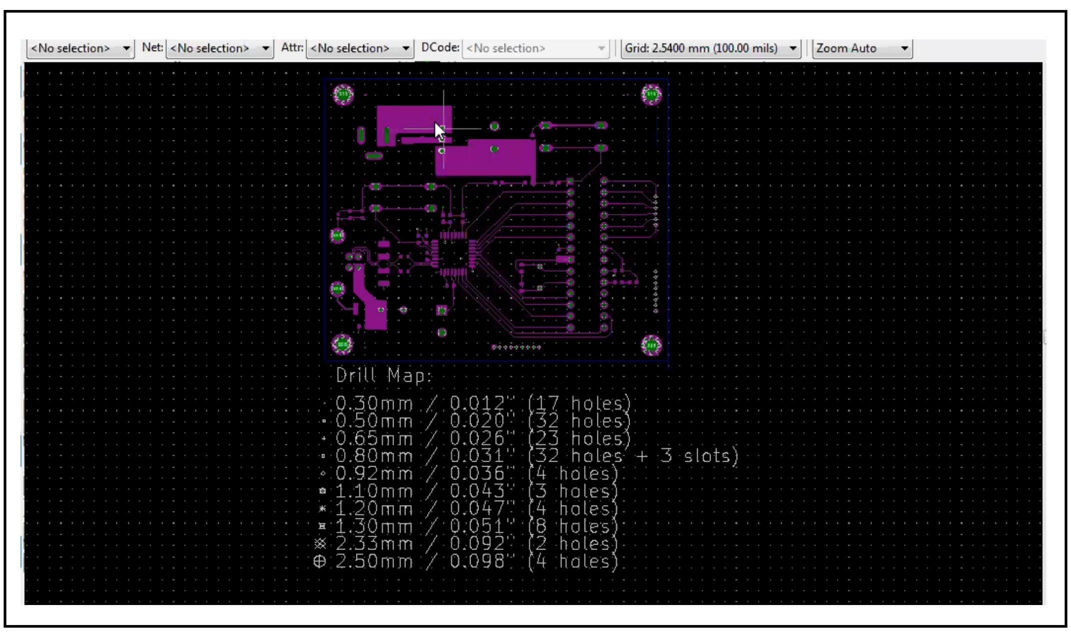


Figure 9b: Gerber file with drill map

Gerber files are created for each layer of the board. PCBs can have as many as 30 layers and manufacturers will have to generate gerbers for each layer.

3.1.2 ODB++

The ODB++ format is an intelligent one, with an individual ODB++ file or even directory containing all the data required to define a board layer.

This particular file format provides a stable framework for the necessary design data. An ODB++ file does not ensure the provided data is adequate to fabricate the design. But, it permits the designer to combine all data and implement the needed checks for reliability and manufacturability.

The ODB++ translator will input a compressed file (.tgz, .tar, .gz, .zip, or .tar), an ODB++ archive or a directory of files from the circuit layout program. Typically, the ODB++ file is sent to the manufacturer as a single file with gzip extension with all the required files being stored in this main file.

3.1.3 Why ODB++?

The ODB++ hierarchy framework enables programmers and enterprises to transfer more than just drill data and standard artwork. It can make room for more data in a single file.
For instance, the file can include data about the component placement, bill of materials, material stack-up, and also the dimension and fabrication data.

Since ODB++ can be accessed using most PCB design software such as Expedition, PADS, and Allegro, it is considered a universal format by most designers. This enables the simple and efficient production of boards without any added complexity. This highly versatile format is supported by every major vendor of CAM, CAD, and DFM tools.

3.1.4 What's the difference between ODB++ and gerber files?

Gerber RS274X is the most popular format for PCB design. It is an individual board file that comprises all the layer data, drawings, and pad shapes providing an accurate design.

Unlike an ODB++ file, a Gerber cannot define the layer stack-up and does not include the drill files. Also, ODB++ can include a large amount of data. Most manufacturers now prefer ODB++ as it aids in minimizing human errors. Using ODB++, a PCB fabricator could do away with the necessity of working with a lot of low-level files.

3.2 Good silkscreen practices

To start the creation of a good PCB silkscreen the first step is the use of optimum sizes and line widths. When font sizes are too small or drawn too wide as a line, it might look more like an inkblot than readable text. Line widths that end up being too narrow may not be drawn correctly on the board.

The next major concern is silkscreen spacing to pads or other PCB features. Silkscreen becomes unreadable when it ends up on a pad, and more importantly, it will impact the solderability of the board. Most board fabricators use these values for the best silkscreen printing:

Font size: Best results call for 0.050-inch font size. Also, the font size should not be smaller than 0.025 inches. Typically, three sizes of fonts are used - large 50/50/6, medium 35/25/5, and small 25/22/5. (text height/text width/line width)
Line width: Fonts should not have a line width smaller than 0.005-inch. Although wider lines can be used with larger font sizes for company names, part numbers and other user information should use smaller fonts. These can include reference designators, pin numbers, and polarity markings.
Clearance: Silkscreen should be placed at a minimum distance of 0.005 inches from PCB pads and other features.

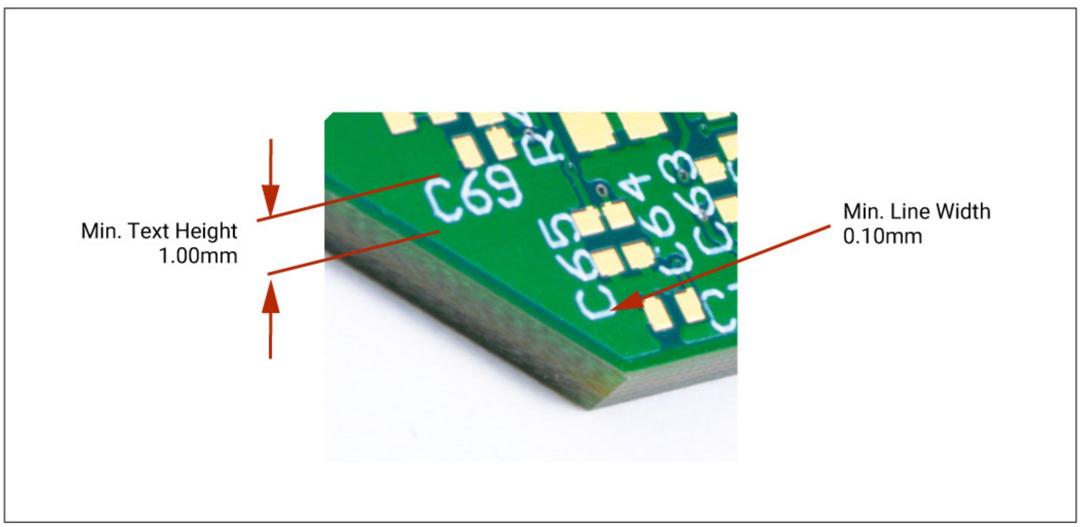


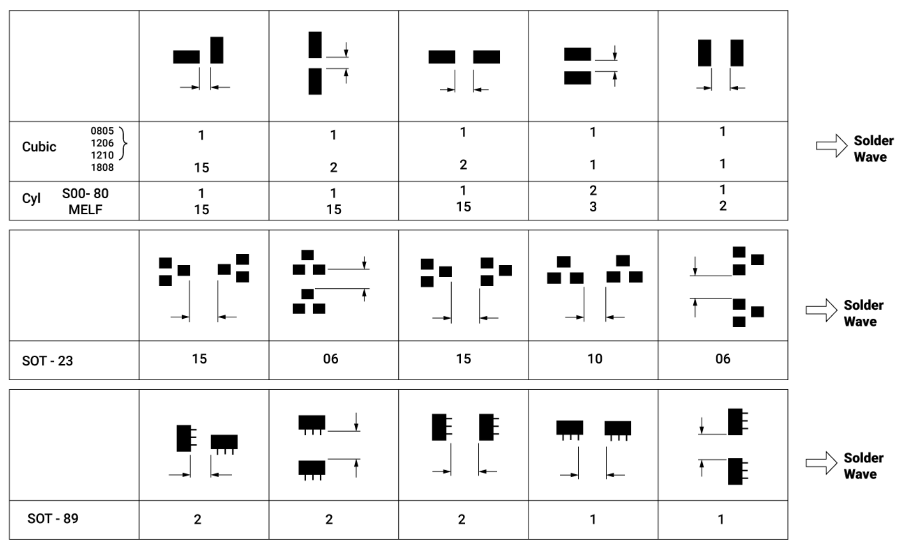
Figure 10: Silkscreen on a PCB

The consecutive step involves the arrangement of different items on the silkscreen. Reference designators should be close to their part and rotated to be easily readable. Unless otherwise required, only 90-degree rotations should be adopted for designators.

The silkscreen elements should be readable at one go on the board without rotating the board too many times. It is also essential to ensure that critical silkscreen information is not covered by placed components. When silkscreen marking is covered by components it will become difficult for technicians and inspectors to locate pin markings or components.

3.3 Component spacing

Component placement according to spacing guidelines is critical to creating a PCB that meets performance expectations. For example, bypass capacitors should be close enough to their related devices to offer an instant power reservoir and minimize parasitic inductance between them. Signal paths must be implemented to provide the shortest possible distance between pins (especially significant in high-speed signal boards). The component placement must also be such that critical routing does not cross split power planes, preventing the loss of return paths.



Minimum separation between SMD components during wave soldering

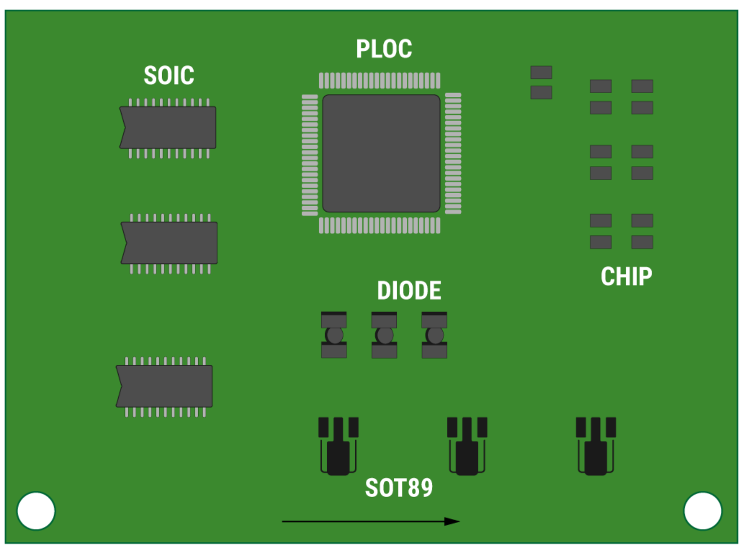
Figure 10a: SMD Component clearance in mm

If the components are placed too close to each other for performance reasons, it might affect the manufacturer's ability to fabricate the board. This might lead to additional time and cost expenses, or even redesigns. To prevent such issues, you can implement some component spacing guidelines to help ensure the success of the PCBA.

3.3.1 Part-to-part spacing

The land pattern design and spacing for each component package will affect the comprehensive reliability and timeline requirements for the PCBA along with the repairability of the board. Sufficient spacing between components on the board guards against possible faults such as solder-bridging while enabling easier manual soldering or rework. While greater spacing facilitates greater ease of assembly, some applications need close spacing to accomplish a smaller form factor.

There are a few special scenarios where you need to take extra care of sensitive component packages such as larger QFP/QFN, POP, or BGA:



Recommended Layout

Figure 11: Recommended layout for components on a PCB

It is better to place IC sockets as far away from sensitive packages as possible. This is needed as repeated plugging and unplugging of the IC into the socket might place unwarranted stress on nearby solder joints.

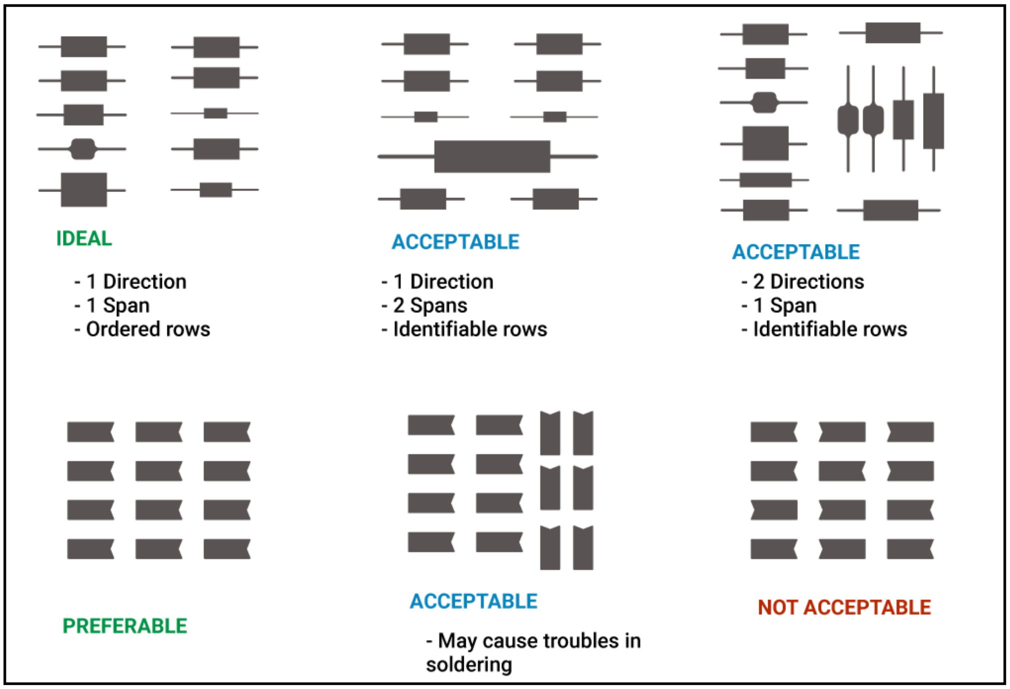


Figure 12: Acceptable and unacceptable PCB component placement

It is good practice to avoid placing sensitive component packages in the center of the PCB, as broken connections can result from bow and twist at maximum intensity in this board region.

Placing BGA and other leadless packages on only one side of the PCB is recommended. If placing such components on both sides of the board can't be avoided, then they should not be placed one beneath the other, using the same XY coordinates on both sides of the board. This might complicate X-Ray inspection and rework.

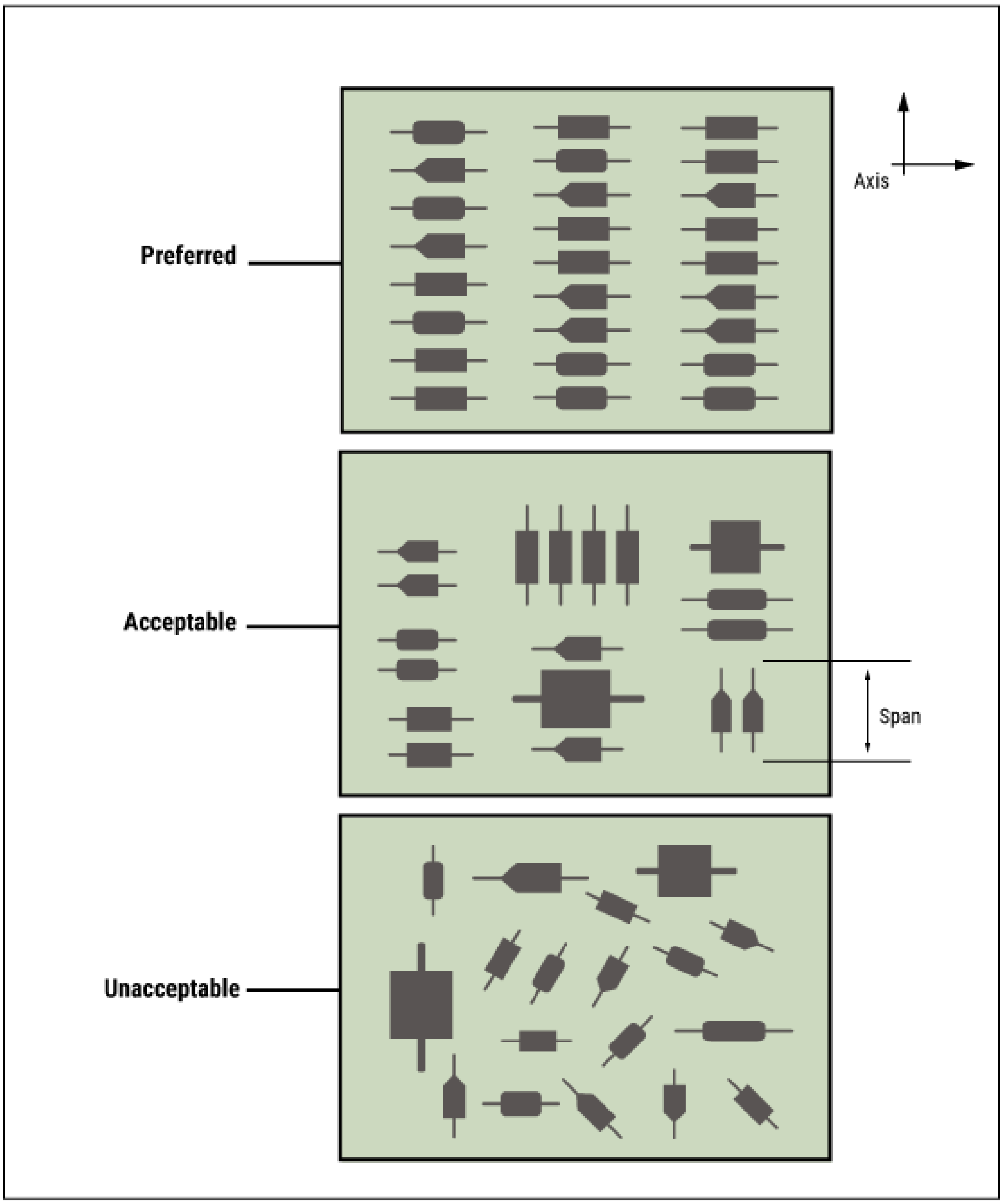


Figure 13: Acceptable and unacceptable PCB component placement

3.3.2 Part-to-edge spacing

The distance from a given component on the board to the board edge. This factor is critical for the depanelization process after the PCB assembly is complete. During the depanelization process, the components near the board edge will be subjected to stress that might compromise solder joints.

It is common practice to increase the component-to-board edge spacing on the secondary board side. This is because of the hold-down clamps used in the solder application process, to keep the PCB from moving while the solder paste is applied. Surface-mount components in this area could be blocked by the clamps and left with inadequate solder paste. Another possibility is the SMT components could be damaged.

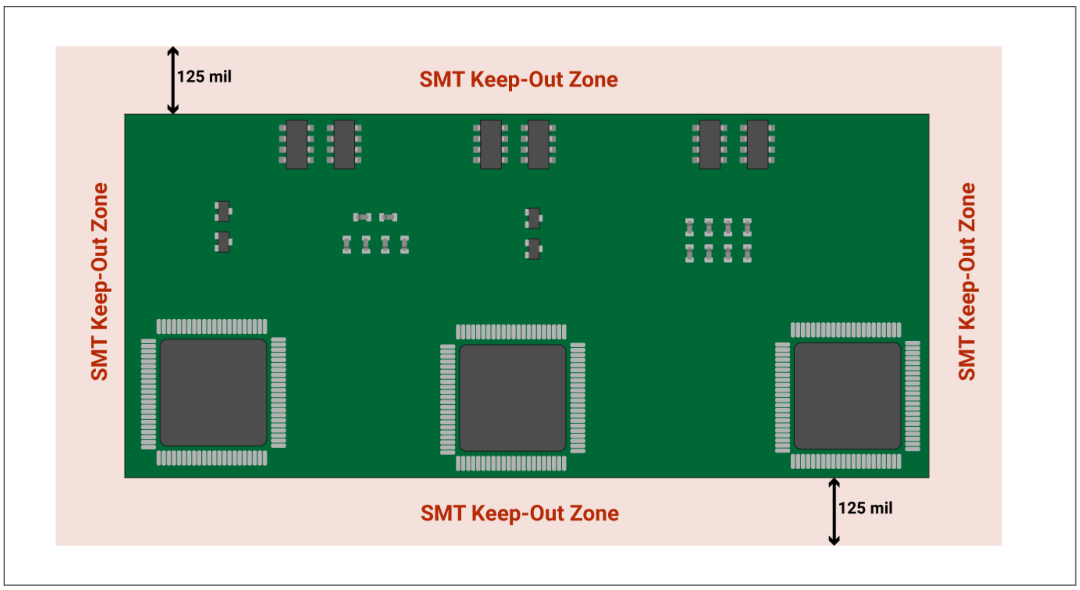


Figure 14: Part-to-edge spacing for PCBA

It is significant to note that clearances for automated assembly processes and manual assembly processes are different. This is because manual assembly allows parts to be placed closer to the edge of the board as they can be installed post reflow and depanelization.

Copper traces can also run much closer to the board edge. Both copper traces and manually installed parts need to be distanced at least 10mil from the edge of the board to permit a solder mask gap and prevent pad encroachment.

Certain designs call for copper plating at the board edge which is known as castellated holes. Such designs will need additional cost and lead time to achieve the desired copper plating.

3.3.3 Part-to-hole spacing

Part-to-hole spacing is required for both PCB vias and through-hole components. It is a requirement that determines the minimum spacing between a component pad/body and either of the hole types. Such spacing consists of two particular parameters that have to be met to achieve a quality assembly:

- **Part-to-hole wall:** This is measured from the actual hole edge in the PCB to the pad edge.
- **Part-to-annular ring:** This is measured from the edge of the hole's annular ring to the pad edge.

3.4 Component clearance

The maximum component boundary refers to the outermost boundary of the component along with the edge of the package and lead ends. The minimum placement courtyard is defined around the component including the body and the basic land pattern.

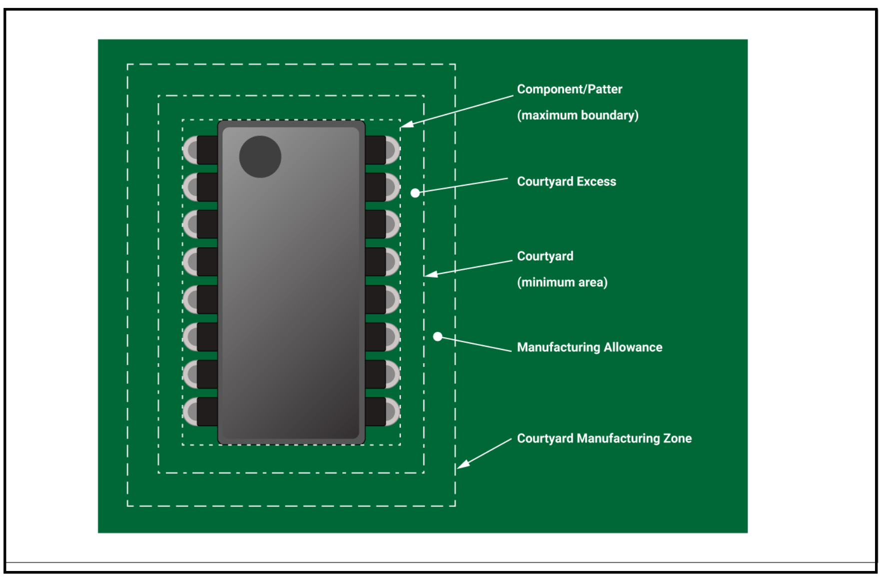


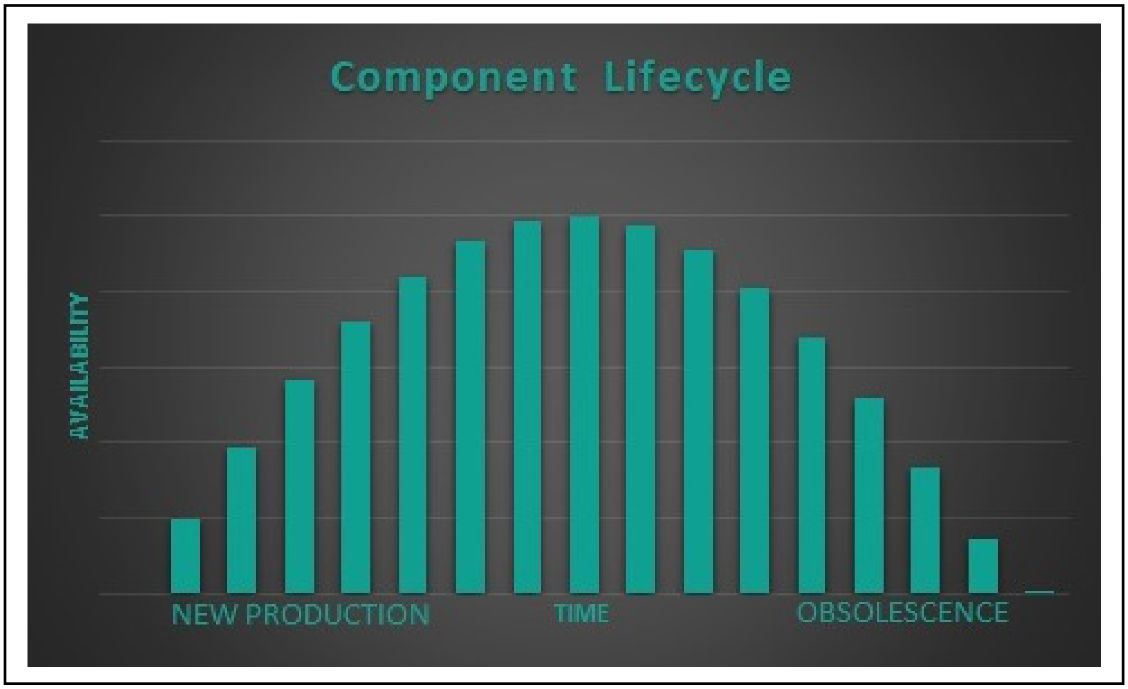
Figure 15: Component boundaries

The outermost region is the courtyard manufacturing zone that serves as clearance for other components, board edge, and hardware. This zone also provides room for rework.

- Unless otherwise specified, clearance is 0.25mm.
- BGA devices need a clearance of 1.0mm.
- Parts smaller than 0603 packages need to have a clearance of 0.15mm.
- Connectors, crystals, and canned capacitors need to include clearance of 0.5mm, along with the clearance needed for mating of the connector.

3.5 Component availability

While designing a PCB, the designers see the components existing as virtual entities, making it easy to overlook the importance of its actual physical availability. Sometimes, components might be included in the database that is no longer available resulting in major issues during PCB assembly and prototyping. There is also the possibility of a critical component reaching the end of its lifecycle before production begins. Such issues can throw a wrench into design and manufacturing plans, but these problems can be avoided by optimizing component selection based on availability during the design phase.



Component Lifecycle

3.5.1 Allocation

Allocation is when there is a shortage of components as the demand for electronic components exceeds manufacturing capacity. Allocated parts have unclear lead times and are mostly subjected to delay. Component shortages are made worse by an increase in automobile electronics and the devices which add to the Internet of things (IoT).

When parts are allocated, suppliers will not receive sufficient components from component manufacturers to fulfill the customer's orders.

What are the consequences of component allocation?

- Manufacturers and suppliers may not take on new orders including components that have critically allocated parts.
- Customer's orders might be fulfilled partially.
- Delivery dates might be delayed without information about the next available date for the delivery

- This might lead to price increases in the PCBA due to component shortage. An example of this is that the orders placed in 2017 were subjected to price increases in 2018.
- An instance of the unavailability of components in 2018 is that 100nF capacitors were essentially unavailable in large quantities. The components that were available were priced at 10-20 times the market price.

Currently, multilayer ceramic capacitors (MLCCs) are subjected to shortages and the problem of allocation.

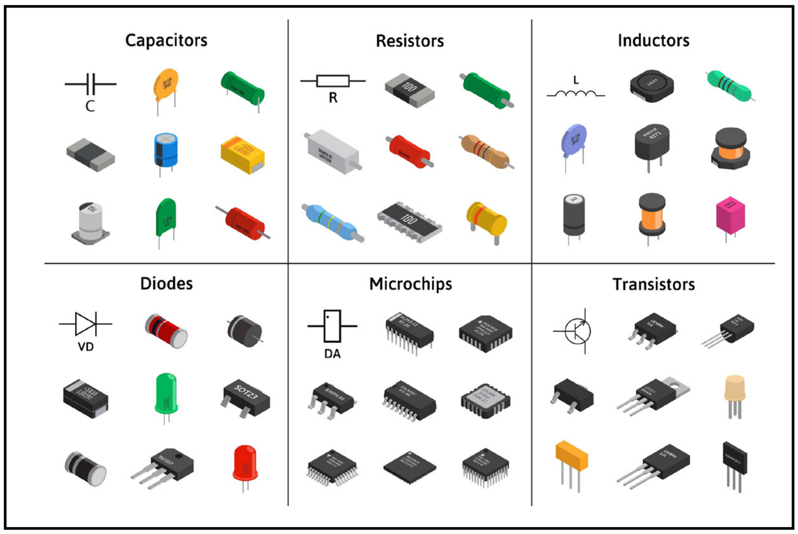


Figure 15a: Components used in PCBAs

3.5.2 Top 5 ways to mitigate PCB component availability problems

Here are five ways you can reduce disruption in the design and development of PCBs:

1. Verify the components for availability more than just once and a final time before assembling the boards.

It is standard practice in design to choose components in the initial design phase or add the said components as the design phase progresses. All these components should usually be selected right before the layout phase.

If there is a period of time between selecting the components and ordering them, then it is best to check for component availability before sending the design files to the fabrication house. This is important as some components might be out of stock at the time of ordering them, so ensure you recheck your BOM against component availability at the time of sending the design files.

This is of critical importance when it comes to the fabrication of quick-turn prototypes. Whenever there is a component out of stock the manufacturer will have to get in touch with the customer leading to delays.

2. Select alternate components before sending the files for fabrication

A way to ensure the design and development of PCBs is not disrupted is to include alternates in your bill of material. When going through the BOM during the final checks, be sure to include alternate part numbers in the BOM for parts that might go out of stock. In this manner, you can save time on substitution approvals, early on in the process.

3. Be flexible with part values when possible

When you can, be flexible with component values, and make sure your fabricator knows. This way, slight variations in the component values won't disrupt your design.

4. Consider minor redesigns for long term fabrication goals

It has been an ongoing trend that component packages with larger form factors will be phased out over time. A good place to start would be to standardize the 0402-size passive components.

Newer components such as RF and power components are only available in smaller BGA or QFN packages. While these smaller components may pose challenges, the future may see exclusive use of these components or smaller ones.

5. Respond as quickly as possible to your fabrication partner when you receive a query.

This is a fundamental practice that can prevent significant delays in manufacturing. This will prevent the fabrication house from guessing what components you need and keep pace with the manufacturing schedule.

3.6 Thermal relief

Thermal management is critical to a successful PCB design. Almost all boards, regardless of it being single-sided or multi-layered, will make use of large areas of metal for power planes or ground planes. These planes will typically connect to various component pins. In these instances, soldering would pose a challenge due to the considerable amount of metal present in the plane.

While soldering small SMT components to a large area of metal, the solder will melt faster on one side compared to the other. This will lead to the tombstone defect, where the component rises on one end.

Soldering through-hole components to a large area of metal might lead to a cold solder joint. When desoldering through-hole components that were soldered to a significant area of metal, it might lead to the application of excessive heat. This in turn might lead to PCB, and component damage.

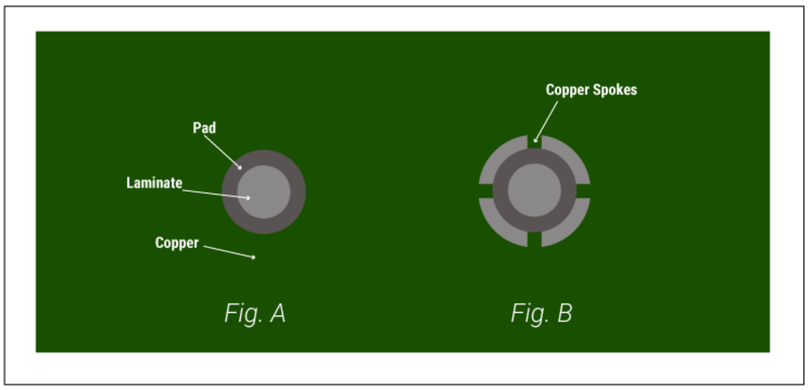


Figure 16: Pad with and without thermal relief

This is where thermal relief can prove useful in the form of a **thermal relief pad** or **thermal pad**. The thermal pad generates small gaps in the metal around the hole to make the connection to the plane through the small metal spokes. Restricting the connection to the plane through the spokes will decrease the thermal contact with the rest of the plane. This enables the power or ground lead of a through-hole component to solder at the same rate as the other leads of the component. Given below are some examples of thermal relief pads.

3.6.1 PCB thermal relief guidelines

- When connecting a through-hole pin to a power plane having a large area of metal or the metal area is wider than the component pad, utilize thermal relief pads.

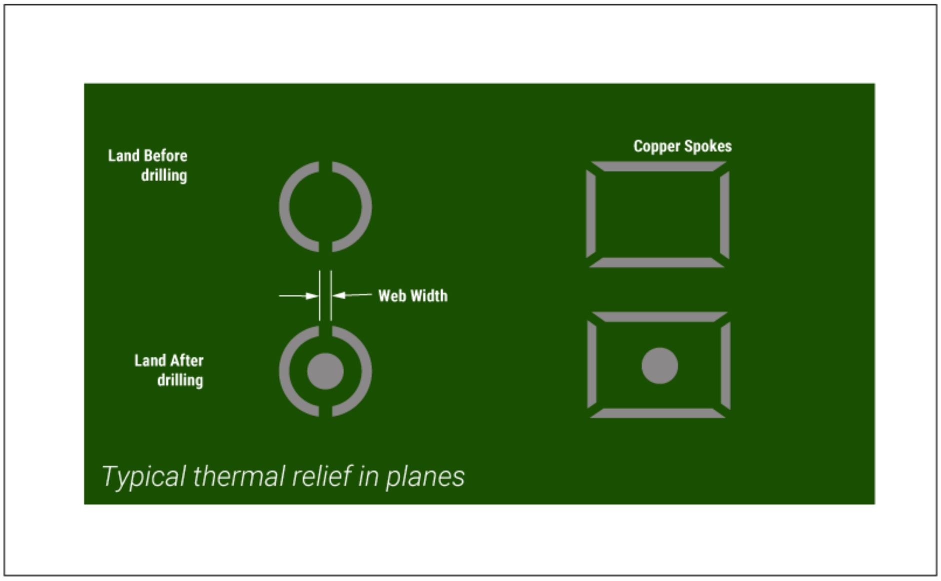


Figure 17: Thermal relief lands before and after drilling

- When the SMT component is soldered directly to large areas of metal, utilize thermal relief pads between the solder pad and the metal area.

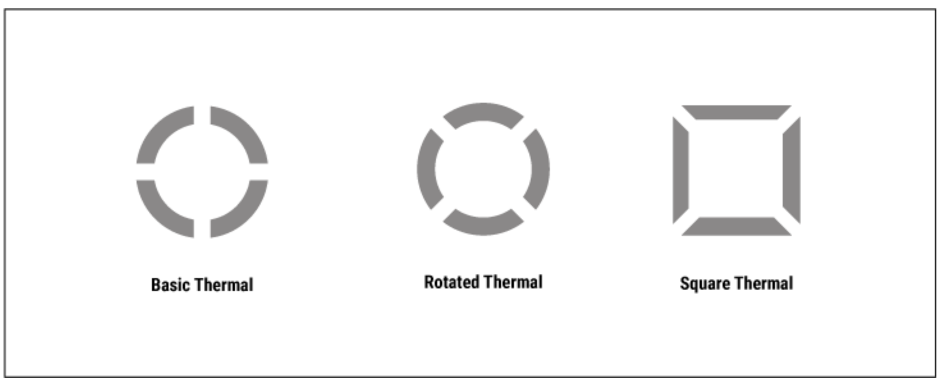


Figure 18: Different types of thermal relief shapes

- Match the width and spoke count of the thermal relief pads with the pin's power conduction level. For instance, if the power requirement calls for a minimum trace width of 40 mils, then the spokes should add up to 40mil, 4 spokes of 10mil each.

- To avoid spoke connectivity issues, set up your design software to recognize minimum thermal relief connections. The spokes in a thermal relief pad might not connect properly when two or more pads are placed too close to each other. Also connectivity issues might result when being used in a split plane or if the area of metal that they are in is too small.

3.7 PCB quality control methods

Quality during PCB manufacturing is a progressive process and the quality control process is an integral part of this. Here are a few points about the QC process in circuit board manufacturing

3.7.1 IPC certification

IPC training and certification ensure the highest levels of manufacturing quality. The IPC standards ensure a consistent and uniform approach to various assembly processes. Annual and bi-annual IPC certifications with the help of an on-site IPC trainer can help keep staff be updated with the current industry practices.

3.7.2 Component expertise

Component failures are a major issue in PCBAs. A well-documented process is needed for inspection and quality assurance to ensure the components used are of the highest quality. Component engineering specialists can work towards procurement needs to obtain the required components within time. This will help in cutting down on downtime.

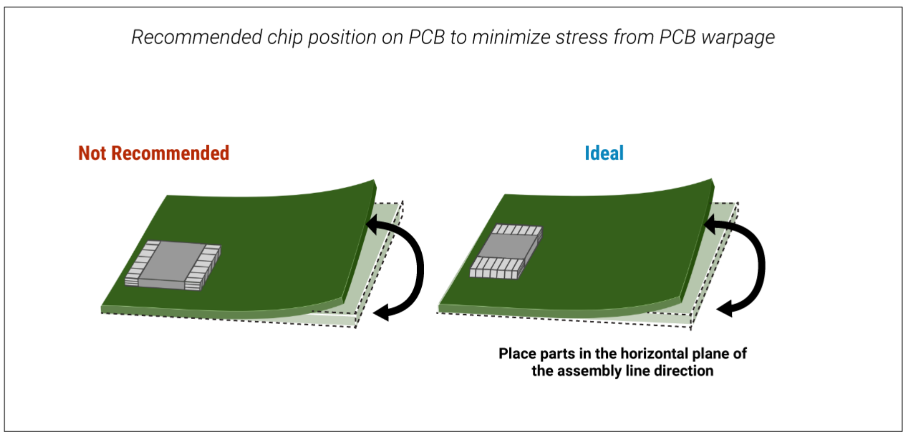


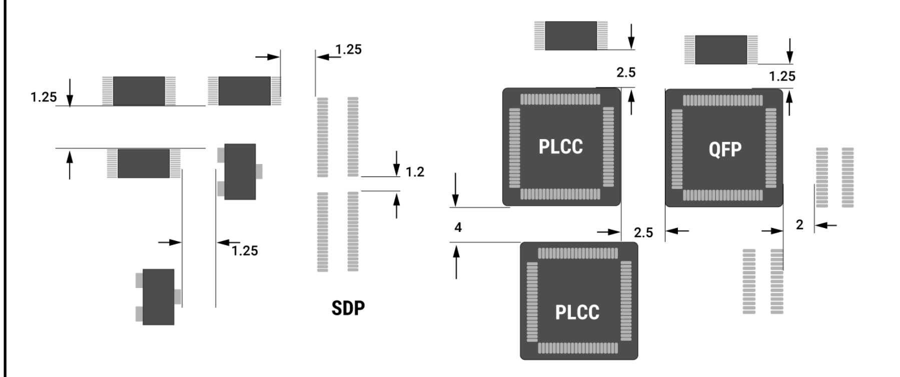
Figure 18a: Component placement to avoid board warpage stress

3.7.3 Process controls

The step-wise process of the PCBA needs to be well-documented in accordance with ISO requirements for QC processes, traceability, and risk management. The quality of the manufacturing operation needs to be governed by process controls by monitoring shelf life, cleanliness standards, soldering temperatures.

3.7.4 Assembly checks

Multiple inspection points during PCBA allow for a better chance of catching an error during the process itself, rather than when it was over. These checks can be a mix of manual and automated inspections to detect potential issues with solder joints and incorrect component placement.



Minimum spacing between Adjacent Pads among Components (unit:mm)

Figure 18b: Component spacing to facilitate an efficient PCBA process

3.7.5 Inspection and test

After the manufacture and assembly of the PCB, there should be a detailed inspection process before sending the product to the customer/client. The next step is to test the board, and based on the design, the manufacturer will have a complete test plan which needs to be implemented in its entirety.

3.7.6 Functional workspace

Fabricating boards for precision electronics require a well-sorted and organized workplace with updated equipment and processes. Attention should be paid to how well assembly lines are laid out and how accessible the equipment is to the workforce.

3.8 IPC standards for acceptability

PCB assemblies need to meet IPC standards for assembly and manufacturing to qualify as fit for use in devices. Here are two IPC standards that are relevant to DFA practices for board manufacturing:

3.8.1 IPC A 600 – Acceptability of printed circuit boards

The IPC A 610 standard in PCB manufacturing covers how boards should be handled and what the acceptable techniques are for hardware/component installation. Also specified is what constitutes acceptable soldering and several other manufacturing aspects for boards with through-hole and surface mount components.

3.8.2 IPC A 610: Acceptability of electronic assemblies

The IPC A 610 is the standard for electronic PCB assemblies that detail what is acceptable in a PCB assembly. This standard divides electronic assemblies into three classes - 1, 2, and 3.
Class 1 refers to boards required for products such as low-end toys that don't need high-quality manufacturing.
Class 2 refers to boards that are to be designed for higher reliability for non-critical electronic assemblies.
Class 3 refers to conformance with IPC standards for every aspect ranging from laminate selection and plating thickness to manufacturing processes and inspection. This class is reserved for critical electronic assemblies used in military and space applications.

Here are some of the DFA issues that we at Sierra Circuits usually face. The possible resolutions are suggested for some of them.

DFA issue	Description	Possible resolutions
Footprint mismatch	Component from BOM and CAD data footprints mismatch and not possible to assemble PCBA	–
Mounting orientation confirmation needed	• Component orientation mark is missing in silkscreen • Mismatch in silkscreen marking and actual component • Pin 1 mark in silkscreen is not matching with the actual component • When component mounting orientation is not clear etc.	–
Cathode mark confirmation needed	Cathode mark is missing/ not clear in silkscreen/ assembly drawing	
Pin1 confirmation needed	Pin 1 mark is missing in silkscreen/ assembly drawing	–
Positive terminal confirmation needed	Positive terminal marking is missing in the silkscreen/ assembly drawing. Example - For electrolytic cap, tantalum cap, all polarised cap, components like a buzzer	
Mechanical/package dimension details needed	The datasheet is not available for the component to verify in DFA	–
IBOM issue resolution not complete	IBOM resolutions/issues raised during component validation is not resolved by the customer	–
XY data missing	XY data not received from the customer	-

BOM components missing in Gerber and placement files	MPN in BOM is not available in CAD and XY data	
No polarity marking	Polarity mark is missing in silkscreen/ assembly drawing	
Component overlap	The component body is touching the adjacent component or overlapping, restricting assembly	–
No stock	Required item quantity as per BOM is not available with the vendor	• Mention alternate / correct part number from a different manufacturer who has stock available with exact specifications (using description and truncated MPN) • Mention alternate part number from a different manufacturer who has stock available with reduced specification (using description and truncated MPN) • Supply the part yourself • DNI
Layer missing	Paste/mask layer is not observed in CAD files	• Provide solder paste/mask layer • Provide the latest and correct revised files and confirm the revision

Reference designator missing in silkscreen and/or assembly drawing	Reference designators are missing in BOM / XY data / CAD data files	• Provide updated silkscreen • Provide updated XY data • Provide the latest and correct revised files and confirm the revision • Make DNI
Requote required in case of major change		–
Less stock	Required items are not available with the vendor in the needed quantity as per BOM	• Mention alternate / correct part number from a different manufacturer who has stock available with exact specifications (using description and truncated MPN) • Supply the part yourself • DNI
Wrong polarity marking	Wrong polarity mark is observed in silkscreen/ assembly drawing	–
Alternate part number	When passives are not available in the market, they are to be replaced with a direct alternate that matches the form, fit, and function	–
Paste/mask layer missing	Paste/mask layer is not observed in CAD files	• Provide solder paste/mask layer • Provide the latest and correct revised files and confirm the revision
Pad missing	Pad opening is missing in the solder mask layer and solder paste layer in CAD	–

Ref des missing	Reference designators are missing in BOM / XY data / CAD data files	• Provide updated silkscreen • Provide updated XY data • Provide the latest and correct revised files and confirm the revision • Make DNI
IBOM resolution pending for alternate part	IBOM resolutions/issues raised during component validation is not resolved by the customer	–
Incomplete SI data	NA	–
Incomplete part number	NA	–
Mismatch in quantity	There is a mismatch between the quantity and number of reference designators for items provided in the BOM	• Correct the quantity • Update the ref des • DNI
Ref des mismatch with Qty	There is a mismatch between the quantity and number of reference designators for items provided in the BOM	Provide updated XY data Provide the latest and correct revised files and confirm the revision Make DNI
Part number and description mismatch	NA	–
VPN and MPN mismatch	Mismatched description of component between VPN and MPN	• Mention alternate / correct part number with exact specifications (using description and truncated MPN)" • Mention alternate part number with reduced specification (using description and truncated MPN)

5. Common PCB assembly defects

Understanding assembly defects and their root causes can help a fabricator in enhancing the quality of the PCBA and boosting manufacturing yield. The most common board assembly defects are as follows:

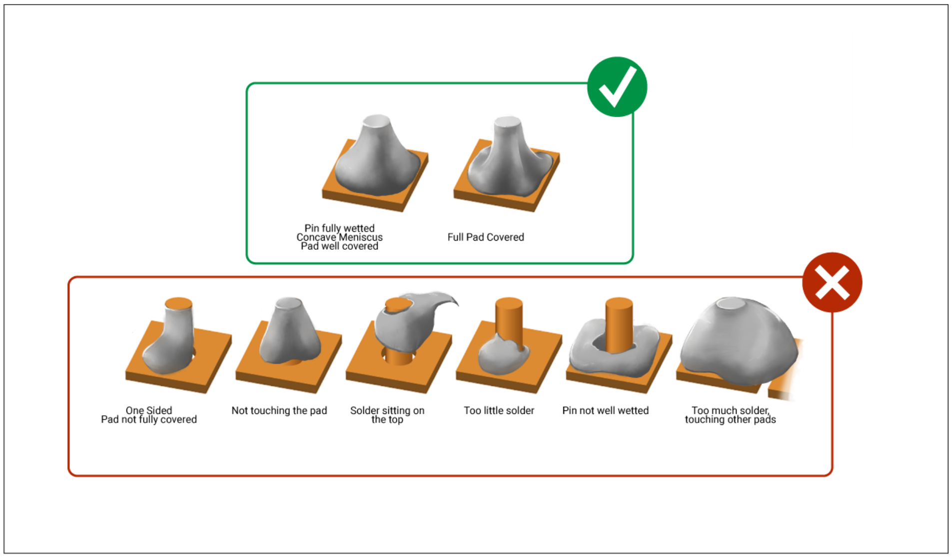


Figure 19: Soldering defects and acceptable solder joints

5.1 Open solder joints (opens)

Open solder joints happen due to a lack of bonding between the component lead and pad, resulting in an open joint/connection. It can also result from solder appearing on the pad and not at the component lead.

Root causes	**Prevention of defect (by designers and production engineers)**
Lack of solder paste or inconsistent deposit of solder paste	Ensure the correct aspect ratio is defined as the ratio of aperture width to stencil thickness
Gap or separation between the PCB pad and component lead	Proper PCB design and accurate component placement
Contamination or corrosion at the component lead or the PCB pad	Resolve lead coplanarity issues and implement correct material handling procedures for operators
Solder paste not active enough	Avoid solder paste contamination by preventing extreme environmental conditions during the manufacturing process
Poor reflow profile, not allowing all surfaces to reach target reflow temperature	Resolve PCB fabrication issues

5.2 Solder bridges (shorts)

Shorts are sometimes referred to as solder bridging. They occur when solder connects two leads that are supposed to be separate. Such shorts might even be microscopic and hard to detect as well. When a short goes unidentified, it could result in serious damage to the PCBA such as a trace burn-out and component or board damage.

Root causes	**Prevention of defect (by designers and production engineers)**
An unsuitable or erroneous reflow profile (initial ramp rate too steep)	Ensure correct reflow profile
Solder pads too large relative to the gap between the pads	Ensure the use of appropriate solder paste metal to flux weight ratio for the required application.
Excess solder on pads due to incorrect stencil specification	Ensure paste deposition is in proper resolution and quality without slump or smear before reflow.
Inactive solder paste, or not active enough	Decrease the stencil aperture dimensions by 10% or thickness of stencil to reduce the amount of solder paste that is deposited
An inadequate seal between stencil and board during printing	Check the alignment of stencil apertures with pads
Misalignment between stencil and PCB	Check the alignment of stencil apertures with pads
Improper component placement, or poor correlation between component lead and PCB pad size	Ensure required pressure and precision for component placement.

5.3 Component shift

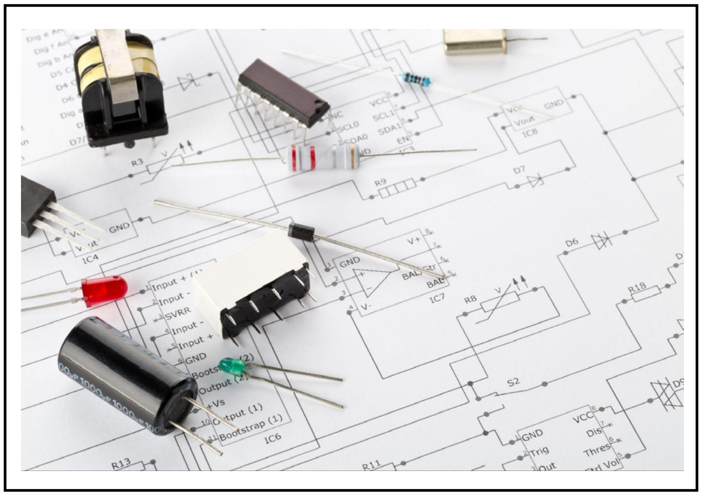
Component shift can be defined as the misalignment of the component with its intended location on the board. This may occur during the reflow process as the component floats on the solder and drifts. PCBA components having many pads such as BGA components may drift from their intended mounting location due to the surface tension of the molten solder. This makes component placement a critical requirement in board assembly.

Root causes	**Prevention of defect (by designers and production engineers)**
A mismatch in part to pad geometry causing the part to pull towards the closest thermal mass	Follow the recommended PCB design guidelines that deal with component placement and clearance parameters
Bent or mis-shapen leads	Implement the recommended temperature and humidity requirements from component manufacturers
Asymmetrical component heat sink	Recalibrate and enhance the accuracy of component placement machines and processes.
Excessive vibration or rapid speed modifications in handling or conveyor system	Minimize the amount of movement resulting from the conveyor system or the other handling processes
Poor solder deposition including the wrong location, off-pad, or incorrect volume of solder paste	Enhance the solderability of components or PCBs with the required flux
Excessive convection rates	–
Undesirably high heating ramp rate leading to outgassing of flux	–
Small components are placed next to large ones where heated gas is directed from the side of the large component towards a smaller one.	–

6. Assembly Tolerance and Clearance

Assembly tolerance is defined as the allowed variation from the actual values without impacting the board's functionality. In PCB assembly, component placement, soldering, the board size, etc., all come with certain tolerances.

6.1 Component tolerance

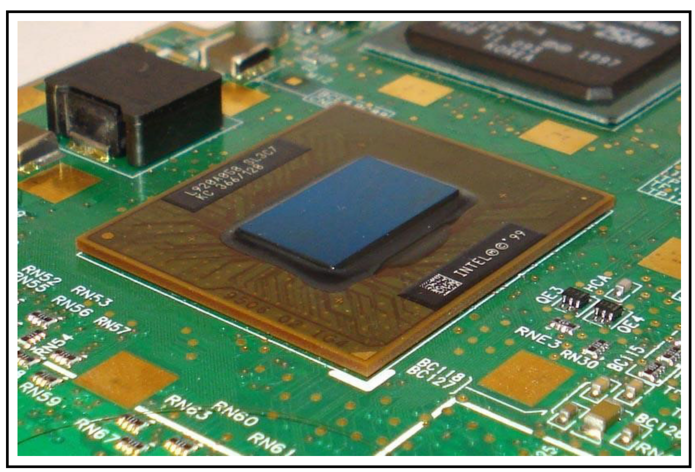


Components for assembly

As you know, components like resistors, capacitors, inductors, diodes, etc., will have tolerance. Take these tolerances into consideration while designing your circuits. Typically for resistors, the tolerances are 0.1%, 1%, 5%, and 10% of their value.

6.2 Sensitive component storage and use

Some circuits may require temperature-sensitive components to be installed in them. For instance, you should store BGA devices between temperatures of 20 to 25°C with an RH (relative humidity) of 10%. Once removed from storage, they should be assembled within 8 hours. The assembly personnel prepares them for soldering by baking them at 125°C to remove the moisture.

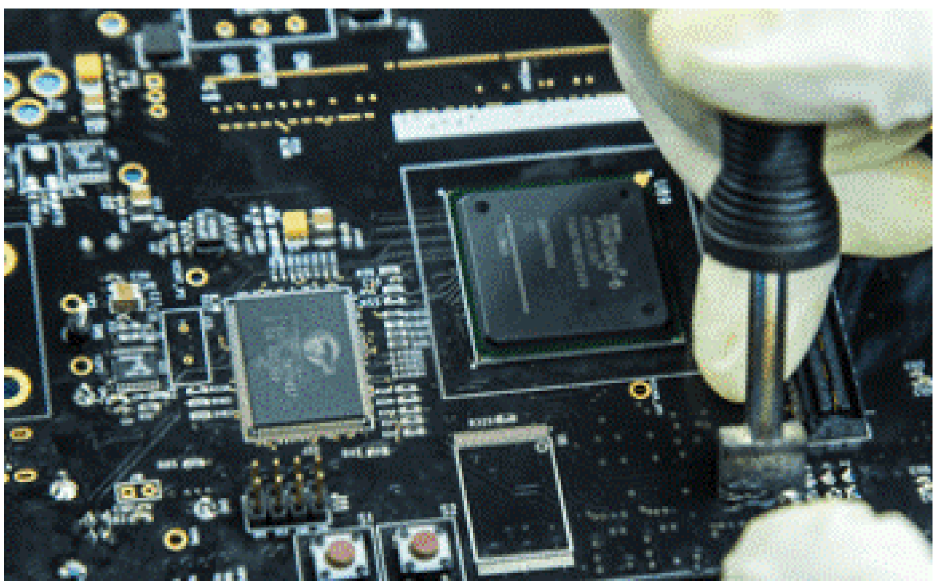


Assembly of BGA devices

6.3 Spacing between components

A lower part-to-part spacing can result in tombstoning, misalignment, solder bridging, and insufficient solder. Proper component land pattern and spacing will improve manufacturability and significantly reduce manufacturing defects.

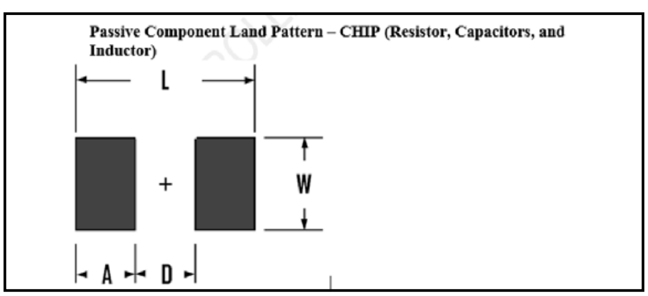
- Components smaller than 0603 packages should have **0.15 mm (6 mils)** clearance.
- Sensitive devices like BGA should have a clearance of **1 mm (39 mils)**.
- Connectors, crystals, and canned capacitors need a minimum **0.5 mm** clearance.
- To avoid solder bridges, manually soldered components should be kept away from the other parts.



Component placement

The below tables summarize the spacing of the passive components followed by Sierra Circuits.

a) Passive component land pattern- chip (resistor, capacitor, and inductor)



Component (Std)	L (in mils)	W (in mils)	D (in mils)	A (in mils)
01005	24	8	8	8
0201	34	12	10	12
0402	58	20	12	23
0603	95	37	21	37
0805	120	55	30	45
1206	200	70	70	65
1210	200	110	70	65

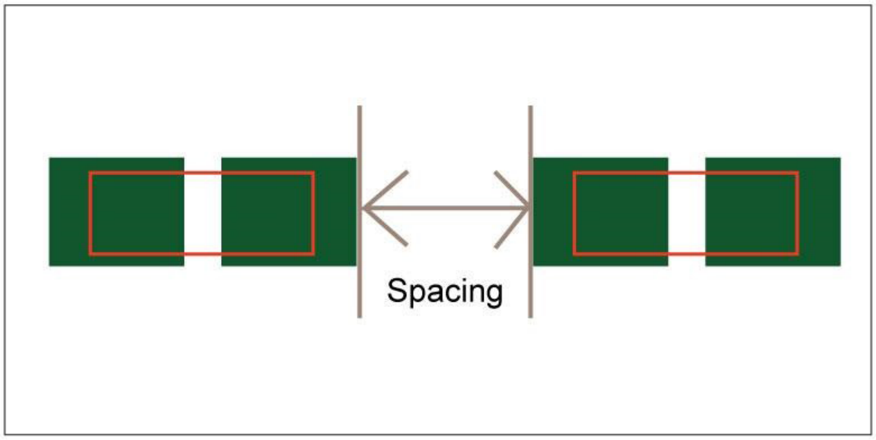
When there is a space constraint on the board, the below minimum spacing is recommended.

b) Passive component spacing- Chip (Resistor, Capacitor, and Inductor)

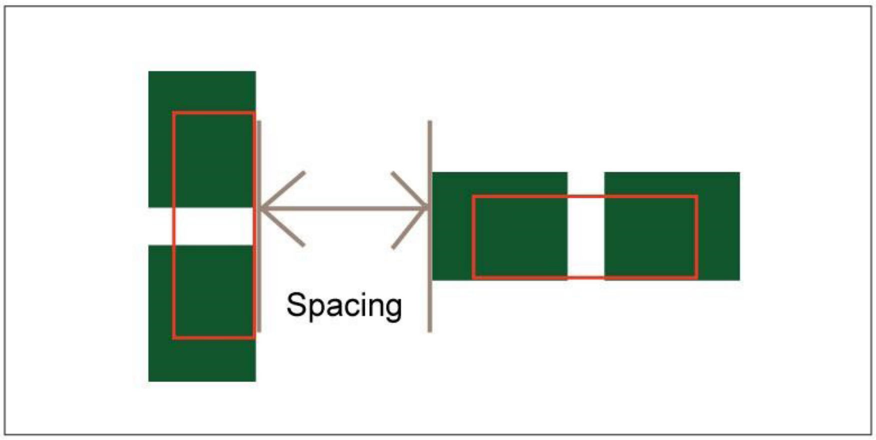
Shape Type	0201			0402			0603			0805			1206		
Rectangular	A	B	C	A	B	C	A	B	C	A	C	C	A	B	C
0201	12	15	10												
0402	12	15	10	12	18	10									
0603	12	18	10	15	20	10	15	20	12						
0805	15	20	12	18	20	15	18	25	15	20	30	18			
1206	15	20	12	18	20	15	20	25	18	25	30	20	25	30	20

Values in mils

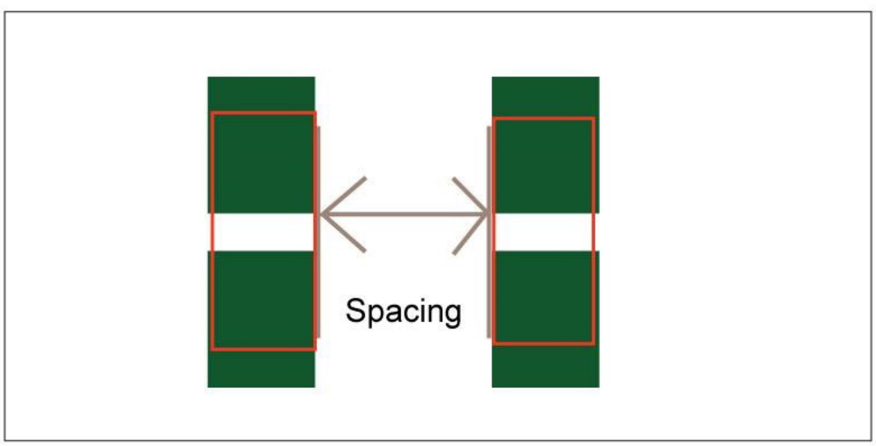
A = Component mounted side by side (End-to-end spacing between the pads)



B = Component mounted (Spacing between pad end to <edge of component body or pad end> whichever comes first)

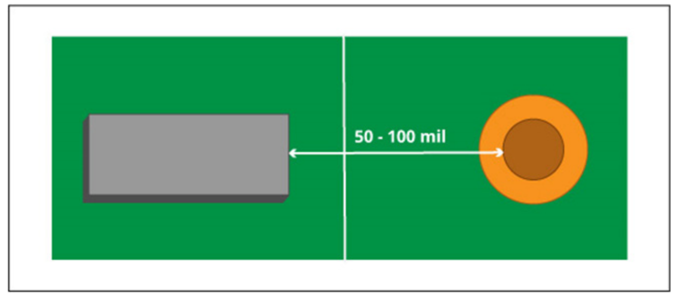


C = Component mounted side by side (Spacing between <edge of component body> to <edge of component body>)



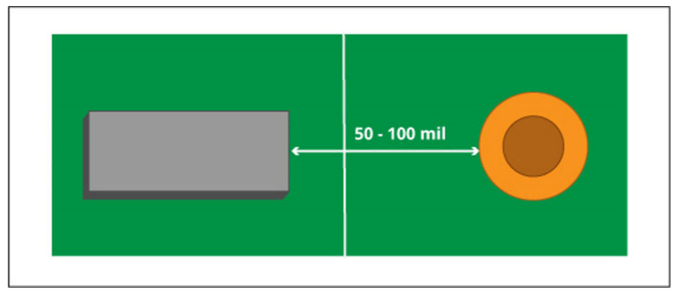
6.4 Component-to-hole spacing

Component-to-hole wall spacing: This is the spacing between the hole edge and the component pad's edge. Usually, the minimum part-to-hole wall spacing is **8 mils**. For manual soldering, this range between **50 to 100 mils**.



Component placement

Component to annular ring spacing: As suggested in the name, this is the spacing between the edge of the component pad to the edge of the annular ring. This should be a minimum of **7 mils**. However, this can be between **50** to **100 mils** for manual soldering.



Component placement

6.5 Component-to-edge spacing

This is the distance between the component to the edge of the board. This spacing becomes crucial during depanelization. The recommended clearance between the part to the board edge is shown below.

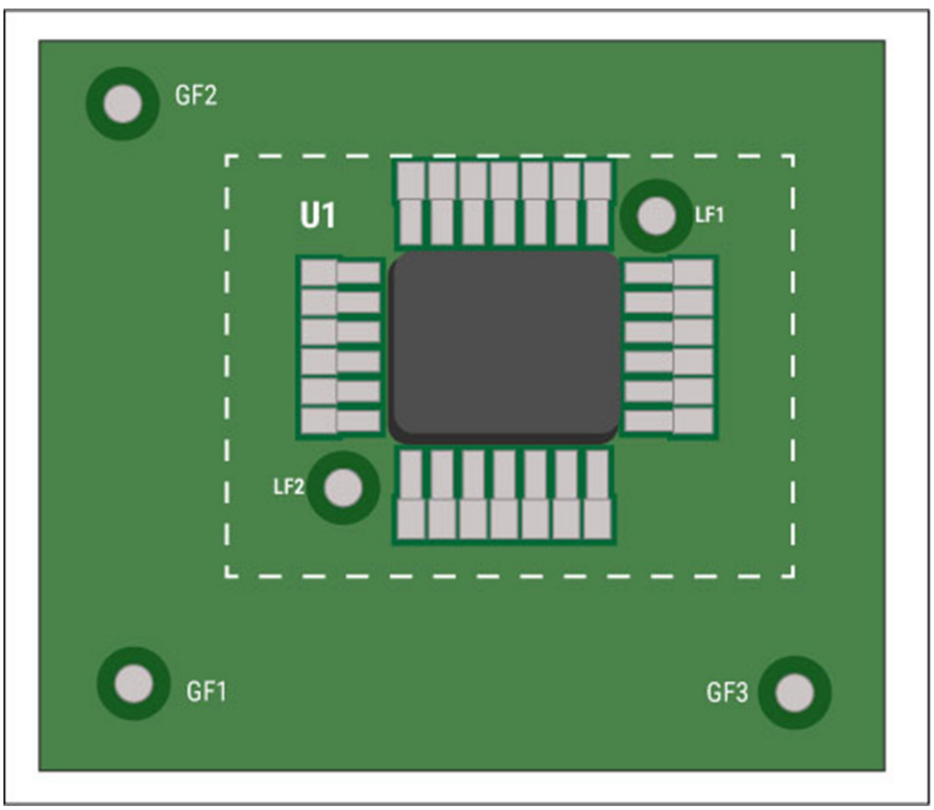
Components	**Clearance**
Larger components	125 mils
Smaller components	25 mils
Manually installed components	10 mils

6.6 Solder stencil alignment

Solder paste is applied to a board using a stencil. The alignment of the stencil to the board is vital for printing solder paste onto the pads. Hence, fiducial marks are used on both pad and stencil. Usually, you can create fiducial marks using a circular non-drilled copper layer without a solder mask.

- Mark should be 1 to 3 mm in diameter
- Clearance between the mark and edge should be 3 mm
- Place three global fiducials at the board's edge for better accuracy in identifying the PCB orientation.
- Place two local fiducials (diagonally) on the outside of the quad-packaged SMT component to recognize their footprint.

GF1, GF2, & GF3 are global fiducial markers, and LF1 & LF2 are local fiducial markers in the image below

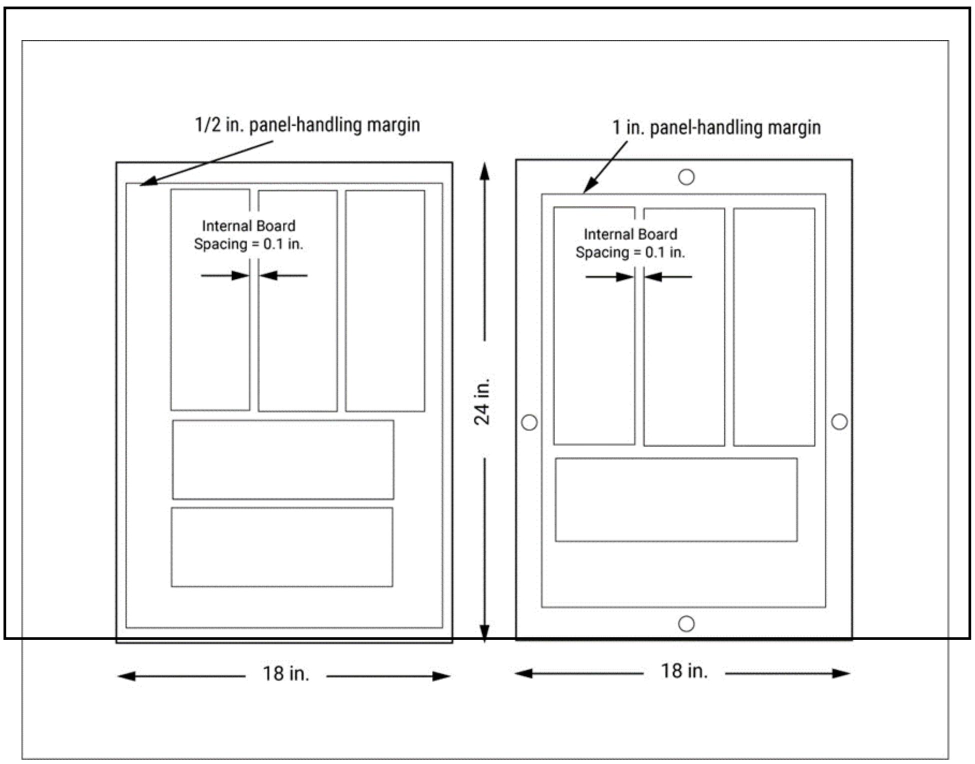


Fiducial markers on a PCB board

6.7 Minimum board size clearance

Standard sizes of PCB panels: 24 x 18 inches, 12 x 18 inches, 24 x 9 inches, and 12 x 9 inches. On a single panel, designers include multiple smaller circuit board designs. Maintain a 0.1 inch spacing between individual boards. Depending on the type of board, designers need to mention a minimum clearance.

- Single-sided boards- 1 inch
- Double-sided boards- 0.5 inches



Standard PCB panel sizes

7. Factors that impact the cost of the PCB assembly

PCB assembly costs consist of varied aspects ranging from the technology used to the number of components with a fine pitch on the board. Getting a board to be cost-effective will require considering all these factors:

7.1 Turnaround time

Turnaround time refers to the time taken to complete the PCB manufacturing and assembly process after the required inputs have been submitted. It is significant to note that assembly costs will change along with turnaround time. Cost is inversely proportional to the assembly time, with shorter assembly times driving up costs between 30 to 200% when compared to conventional assembly times. Accelerated assembly times might mean that the fabricator will have to halt other orders and focus on the expedited assembly. In certain scenarios, the manufacturing facility might have to change, further driving up assembly costs.

7.2 Technology used - through-hole or surface mount

A crucial factor impacting the assembly costs is the technology used for assembly if it is through-hole technology (THT) or surface mount technology (SMT). In some cases both THT and SMT are used in combination. SMT tends to be more cost-effective than THT due to the low setup costs, higher component densities, and higher degree of automation. In some cases, the use of THT is unavoidable and the additional drilling of boards and reduction in routing area will lead to higher assembly costs.

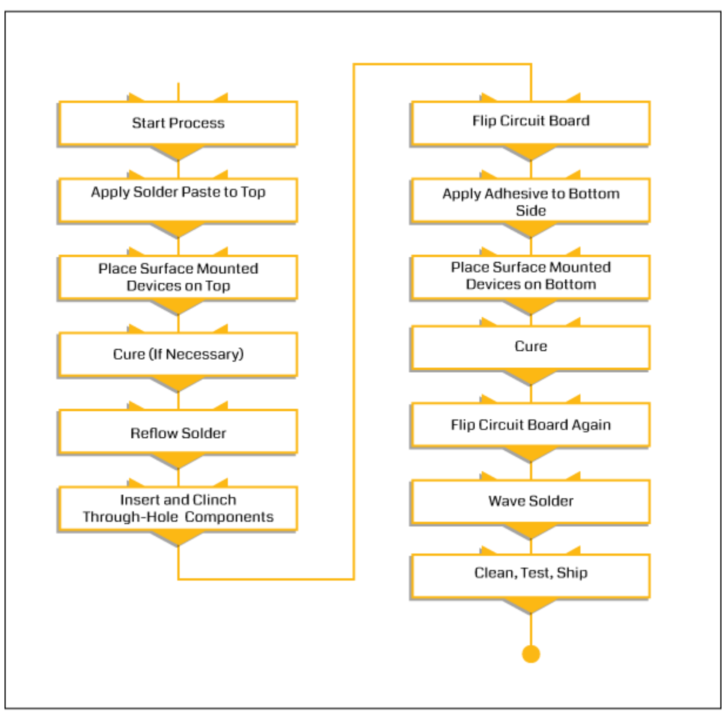
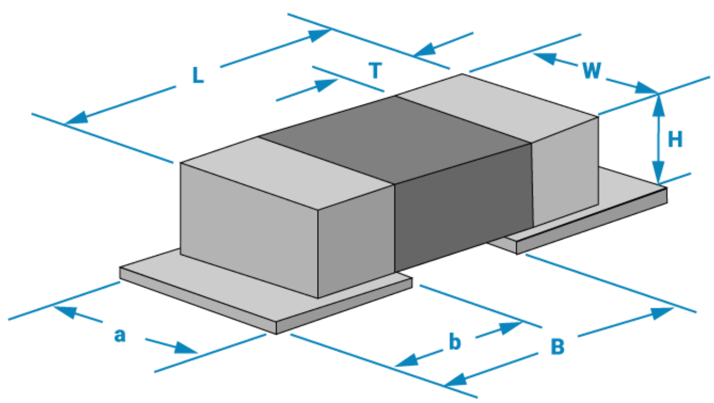


Figure 20: Process flow for through hole and surface mount soldering

7.3 Component packages

As electronic components are available in different packages, assembly costs might also depend on the component packaging. Ball Grid Array (BGA) and Quad Flat No-Leads (QFN) components come with a higher assembly cost. This is because the electrical connections beneath the components require a higher level of accuracy for component placement. These components also need X-ray inspection to look for voids and short circuits between the pins. More conventional packages such as SMD, 0603, 0805, 1206 do not have the higher costs mentioned above associated with their assembly.



Additional Dimensions of SMD component and solder lands.

Figure 21: SMD component dimensions

		Wave Soldering			Reflow Soldering		
Type	**Size**	**a**	**b**	**B**	**a**	**b**	**B**
Chip, resistors and capacitors	0603				0.9	0.8	2.3
	0805	1.45	1.2	3.65	1.45	0.8	2.65
	1206	1.7	1.4	4.85	1.7	1.0	3.65
	1210	2.75	1.4	4.85	2.75	1.0	3.6
	1808	2.25	1.5	6.45	2.25	1.1	5.2
	1812	3.25	1.5	6.45	3.25	1.1	5.2
	2220	5.3	1.6	7.6	5.3	1.2	6.2
Al electrolytic capacitors	1a	2.5	2.0	10.0	2.5	3.0	9.0
	1	2.5	2.0	14.0	2.5	3.0	12.0
Tantalum electrolytic capacitors	a	1.5	2.0	5.0	1.5	1.1	3.2
	b	1.5	2.0	6.3	1.5	1.1	4.5
	c	1.5	2.0	7.55	1.5	1.1	5.75
	d	2.75	2.0	6.3	2.75	1.1	4.5
	e	2.75	2.0	7.55	2.75	1.1	5.75
	f	3.65	2.2	8.45	3.65	1.3	6.65
	g	3.0	2.5	9.15	3.0	1.6	7.35
	h	4.0	2.5	9.65	4.0	1.6	7.85

Solder land dimensions of 0603 components

7.4 Board assembly volumes

The assembly cost for boards depends on various factors such as initial setup costs, stencil cost and programming. Economies of scale come into play here, as high-volume production runs will bring down one-time set-up costs. Prototype boards might need a different set-up for each run or even each board, which will add to the cost of the boards manufactured.

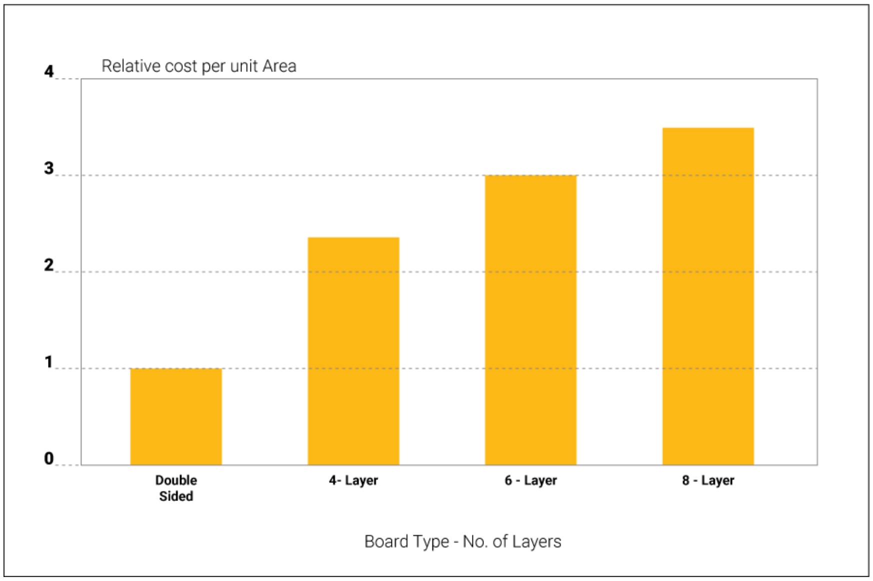


Figure 21: Cost variation according to number of board layers

7.5 Component count/density

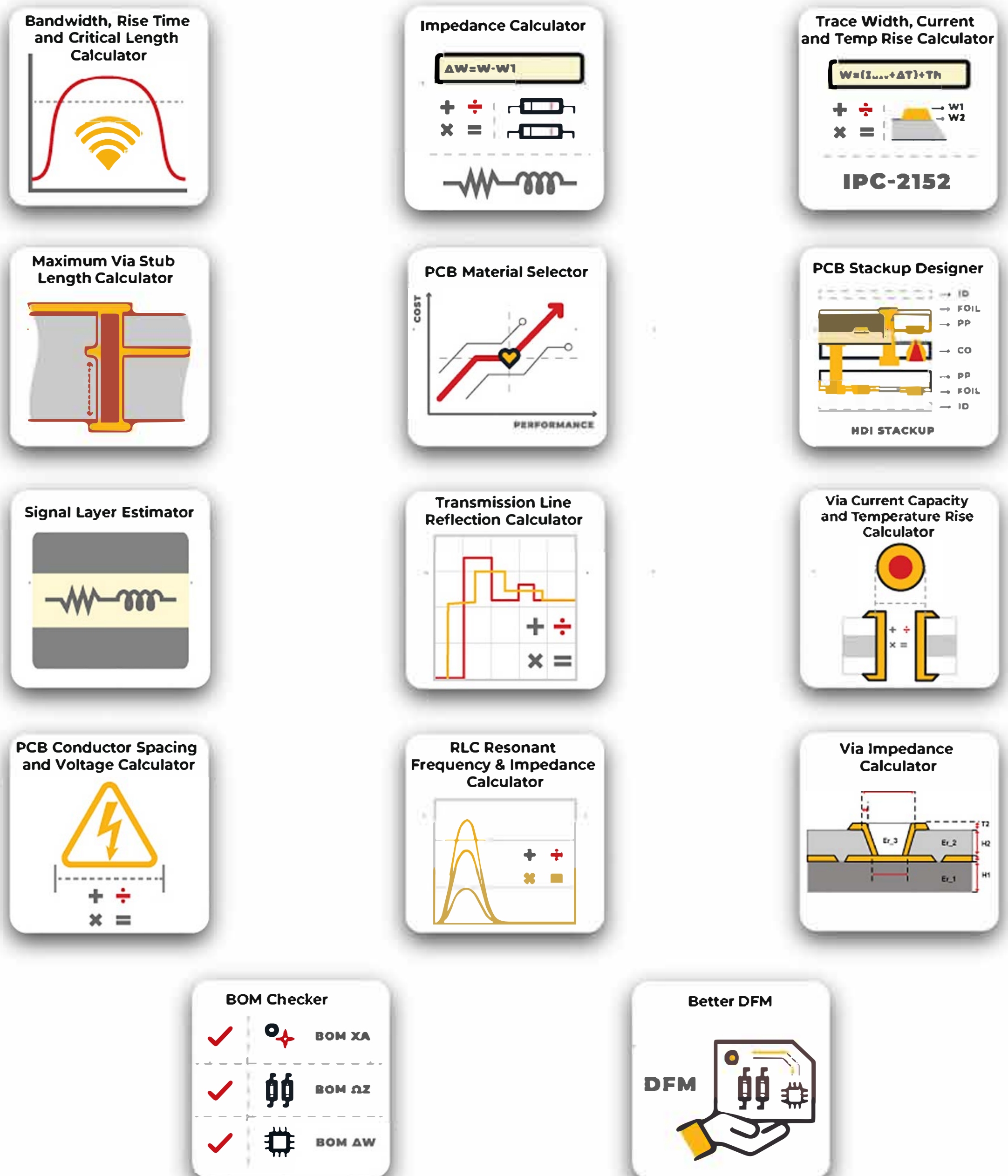
The number of components is directly proportional to the assembly costs. The more the number of components to be assembled on the board, the higher will be the cost of the assembly.

7.6 PCB Stack up cost

The PCB cost depends on a number of factors such as the number of layers, blind and buried vias, the number of laminations, and also the type of material.

Implementing DFA practices in the current market scenario is not just advantageous, but critical to ensure the product manufactured meets the requirement at an optimum manufacturing cost. With the knowledge of DFA, designers can come up with an efficient board that is easy to manufacture and assemble.

The Designer's Toolkit



TRY TOOLS

Sierra Circuits
1108 West Evelyn Avenue
Sunnyvale, CA 94086
+1 (408) 735-7137

www.protoexpress.com