Board stack up:

It is basically refers to the arrangement of layers of copper and insulators that make up a PCB before designing the final layout of the board.

Having multiple layers increases the board's ability to distribute energy, reduces cross-interference, eliminates electromagnetic interference and supports high-speed signals.

Factors to deal with board stackup:

* Number of Layers
* the number and types of plans used (power plans and ground plans);
* sorting and sequence of levels;
* spacing between levels.

When considering the number of layers:

* the number of signals to be routed and their cost;
* operating frequency;
* whether the product will meet Class A or Class B emission requirements;
* whether the PCB will be in a shielded container or not;
* whether the design team is competent on EMC rules and regulations.

**The rules and criteria for managing a good stackup**

* ground plane boards are better because they allow signal routing in a microstrip or stripline configuration. It also significantly reduces the ground impedance and, therefore, the ground noise;
* high speed signals should be "routed" on intermediate layers located between the various levels. In this way, ground planes can act as a shield and contain the radiation coming from the tracks at high speed;
* the signal layers should be very close to each other, even in adjacent planes.
* a signal layer must always be adjacent to a plane;
* Multiple ground planes are very advantageous, since they lower the board's ground impedance and reduce radiation in a common way;
* the power and mass planes must be rigorously coupled together;

**Ounce:**

Unit of measuring for a copper thickness on a PCB

1oz is equal to 1.37mils

**Terminology:**

**Annular Ring**

The annular ring is a copper area around the drill hole that serves to ensure a good connection for vias and provide spacing for [solder mask application](https://www.tempoautomation.com/blog/understanding-soldering-part-5-solder-mask-application-process/).

There are two types of drill holes:

* non-plated through holes are used for mounting and installation
* plated through holes for current-carrying vias

**Bill of Materials**

BOM is a standard means of listing all electronics components required to build or construct a structure

**Board Thickness**

The [board thickness](https://www.tempoautomation.com/blog/what-is-the-standard-pcb-thickness/) is the total height of the board. It is well-known that the size (in the horizontal plane) of circuit boards has been decreasing to accommodate the demand for smaller electronic devices and products. The ability to route more signals, which allows for greater complexity and functionality, is the primary determinant for the number of layers in the stackup of [multilayer PCBs](https://www.tempoautomation.com/blog/5-tips-for-creating-the-best-multilayer-pcb-for-your-design/), and thus the use of increased board thickness.

**Copper Weight**

copper weight is the term used to describe the amount of copper on an external or internal layer of the board’s [stackup](https://www.tempoautomation.com/blog/top-4-tips-for-pcb-stackup-design/). This is not the total weight of copper on the actual layer surface; instead, it is defined as the amount of copper needed to cover a 1ft x 1ft area at a particular [copper thickness](https://www.tempoautomation.com/blog/using-pcb-copper-thickness-to-optimize-pcb-current-flow/), which is directly proportional to the weight (weight = thickness÷1.37) and readily measurable.

**Clearance**

is defined as the shortest distance between two points on a PCB

**Design Rule Check (DRC)**

A design rule check is a comparison of your design specifications with a set of guidelines and within specific limitations, some of which are standards and DFM parameter ranges, within which your design parameters should be selected. These restrictions that must be adhered to when creating the [PCB layout](https://www.tempoautomation.com/blog/the-top-7-pcb-layout-tips-for-manufacturing/) are known as design rules.

**Footprint**

defines the area in which the component should be placed during assembly and may include a reference indicator, polarity markings

**Ground Plane**

For circuit boards, good grounding technique is essential for effective signal routing. This is especially true for [multisignal boards](https://www.tempoautomation.com/blog/the-best-pcb-design-tips-for-grounding-multisignal-boards/) where many different signals may need to utilize a single centralized ground. This is most often accomplished by a [copper pour](https://www.tempoautomation.com/blog/5-tips-for-pcb-ground-pour-design/) that encompasses an entire layer and is known as a ground plane.

**Manufacturer Part Number**

The manufacturer part number (MPN) is a unique identifier for each component type on your board. The MPN is not to be confused with the reference identifier, which identifies each specific component in the board’s BOM and its specific location on the board

**Pad**

The location on the board that defines where a component pin and/or lead should be placed is its footprint.

**PCB design file**

The PCB design file(s) which are more accurately the [PCB manufacturing file(s)](https://www.tempoautomation.com/blog/how-are-pcb-manufacturing-files-used-during-fabrication-and-assembly/) should contain all information, specifications, and images needed by your contract manufacturer (CM) to fabricate and assemble your boards. The most common formats are the multifile Gerbers single file format, such as [IPC 2581](https://www.tempoautomation.com/blog/ipc-2581-a-better-design-file-transfer-standard/)

**Pitch**

Pitch refers to the amount of space between the center of adjacent pads (or pins) for a component. As PCBAs have become smaller and denser, so have component packages and footprints. Consequently, the amount of space between the center of adjacent pads (or pins) or the pitch is a very important design parameter. For these small components, the pitch may determine what [solder mask process](https://www.tempoautomation.com/blog/understanding-how-the-solder-mask-process-affects-your-manufactured-pcb/) can be used to avoid assembly problems, such as solder bridging, or whether [solder masking should not be used](https://www.tempoautomation.com/blog/choosing-the-right-solder-mask-clearance-for-your-pcb/) for the component pads.

**Routing**

[PCB trace routing](https://www.tempoautomation.com/blog/the-best-pcb-routing-techniques/) is the process of specifying the copper traces that connect the board elements. These connections are typically based on the netlist created during schematic generation.

**Signal Layer**

In the PCB stackup a signal layer is any layer that is designed to carry current.

**Via**

A via provides a means of passing current from one layer to another.