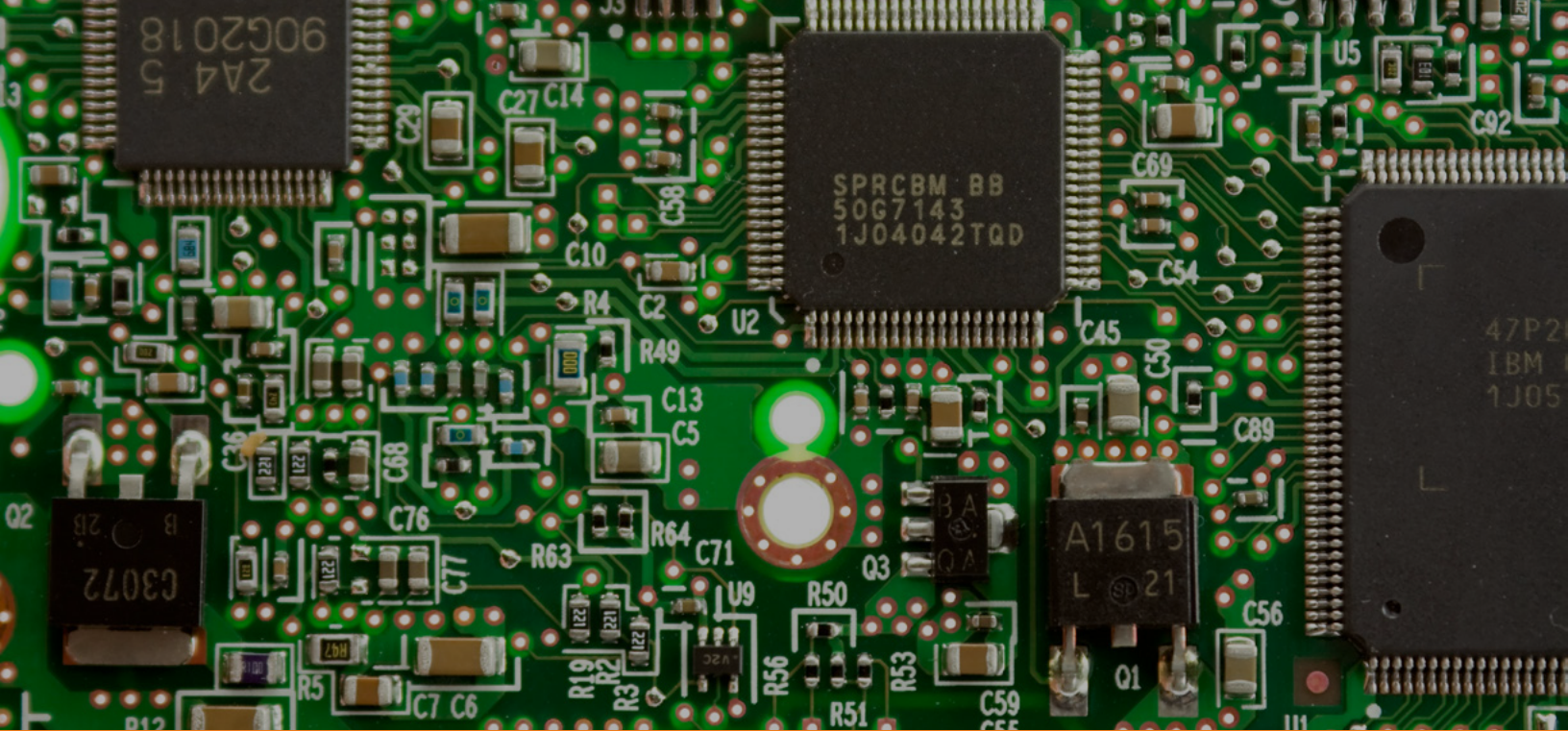




PCB Materials Basics:

What Are PCBs Made of, How to Select the Right Materials



PCB Materials Basics: What Are PCBs Made of, How to Select the Right Materials

PCBs, or printed circuit boards, are essential to most modern electronic devices. While PCBs are commonly associated with computers, they are also used in medical, aerospace, automotive, electrical, and industrial applications. Understanding the construction of printed circuit boards is necessary to select the right board materials and components to meet your unique specifications.

The quality of PCB materials greatly affects the board's performance and the device's lifespan. In this eBook, we break down how printed circuit boards are constructed, what PCBs are made of, and considerations for choosing the best PCB materials based on application.

How Are Printed Circuit Boards Constructed?

The electrical and environmental properties of the PCB dictate the board's materials and schematic, which determines the form factor and number of layers of the circuit board.

Basic techniques engineers use when attaching components and assembling PCBs are surface mount or through-hole methods. With surface-mounted components, components are soldered on top of the board, whereas with the through-hole method, a hole is drilled in the board and long leads run through the copper-plated holes to attach the components to the board.

Before PCB assembly, following several pre-assembly recommendations can reduce the chance of defects and allow effective moisture removal. All printed circuit boards absorb moisture from the atmosphere, though some much more than others, with flexible material specifications allowing up to 6% moisture absorption by weight. Before being subjected to soldering temperatures, PCBs must have less than 0.02% moisture by weight.

To help remove moisture from your PCB's, we'd recommend the following for desiccation:

Step 1

Bake PCBs overnight for at least 10 hours at 250 °F.

Step 2

Perform soldering and assembly within two hours of this initial bake cycle. If this is not practical or possible, the PCBs must undergo dry box desiccation until board assembly.

Step 3

PCBs can also be put in a vacuum chamber of at least 50 millitorrs for two hours to achieve an acceptable level of desiccation.

Desiccation is the most crucial step before soldering, particularly with flexible and rigid-flex PCBs. Not following these moisture removal guidelines could lead to post-assembly defects.

What Are Printed Circuit Boards Made Of?

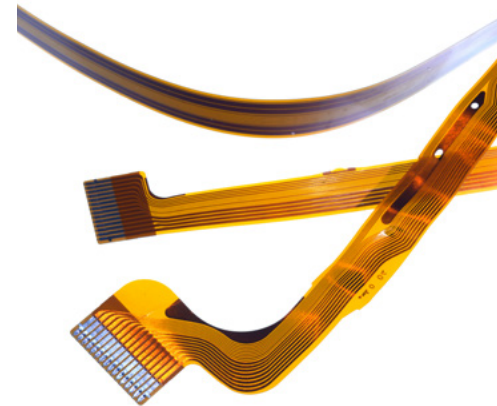
There are hundreds of materials for making printed circuit boards, but the most commonly used PCB materials are:

FR4:

A NEMA-grade flame retardant laminate constructed with a layer of woven-glass fabric and epoxy resin binder. FR4 is commonly used for rigid PCBs.

Polyimide:

Polyimide resins are used for both glass reinforced laminate and prepreg, which will be used to build rigid boards. It is also available as a plastic copper clad flexible laminate that can be used in flexible circuits and rigid flex printed circuit boards.



MEGTRON:

Low-loss, high-speed materials like MEGTRON are ideal for devices requiring higher signal speeds and signal integrity or those operating in higher-temperature environments.

PTFE/PTFE Composites:

Best for radar applications and very high-speed communications, PTFE and composites withstand environmental conditions better than resin materials like FR4.

Printed circuit boards consist of multiple material sublayers. Those layers from the bottom up (or, for double-sided circuits, from the core to the outside edges) are:

Layer 1

A thin fiberglass board with copper on both sides, called laminate or innerlayer

Layer 2

Prepreg, a resin coated glass fabric which is used to bond the layers together

Layer 3

Electronic pathways usually constructed from copper, called circuits and vias

Layer 4

Solder mask which covers and protects the outerlayer circuits from shorting and oxidation

Layer 5

Nomenclature which is the white lettering on top of the soldermask, indicating where components are to be placed

Layer 6

A final finish to the copper surfaces of the external layers, protecting the copper from oxidation and promoting solder assembly of the components



Key Considerations for Choosing PCB Materials

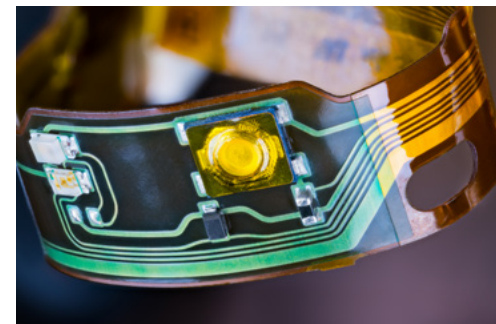
Depending upon application, PCB material requirements vary. Therefore, there are several considerations when selecting PCB materials for your products, including:

Circuit board purpose:

Understanding what the PCB will be used for, signal integrity requirements, and operating environment is essential to choosing the right PCB materials and board type for the application.

Stackup:

Understanding how the PCB will fit into the device is crucial to selecting suitable materials and proper board design. Stackup should be the first step in the PCB assembly process. Materials, circuits, and circuit geometries also determine the PCB's electrical impedance characteristics.



Thresholds for power, conductivity, and heat transfer:

Overheating is a major concern for modern electronics. Therefore, choosing PCB materials that will effectively transfer heat and withstand the necessary conductivity thresholds of a product's needs is an important consideration.

Electrical signal integrity:

Depending on whether the PCB is high-frequency, high-density, high-power, or microwave, you can decide on the best PCB materials for the board's function.

Mechanical and electrical strength:

Determining the mechanical and electrical strength of the PCB will help you select the right materials and ensure the PCB holds up in its intended application.

Component configuration:

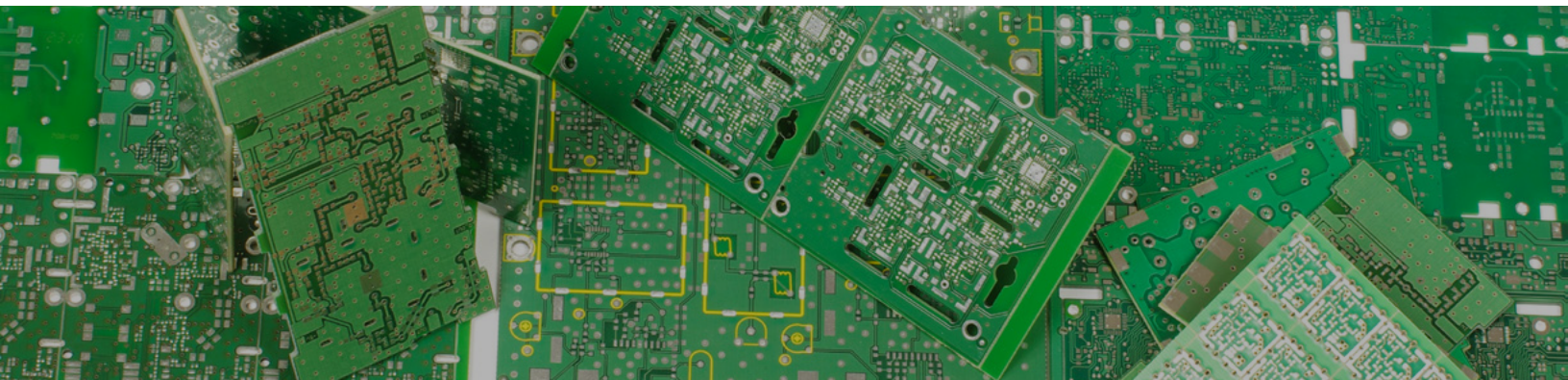
Placing components in the right place on the board is essential to prevent interference with the signal's integrity, optimize circuit routing and control heat escalation.

Flexibility:

The flexibility required for a PCB is a top consideration for selecting materials because some boards may be flexible, rigid-flex, or rigid.

Material costs:

Material costs influence PCB material selection, especially since the cost of building PCBs containing buried or blind vias, gold, or via filling is usually higher. Line and width spacing and surface finish layers used also affect the total board cost.



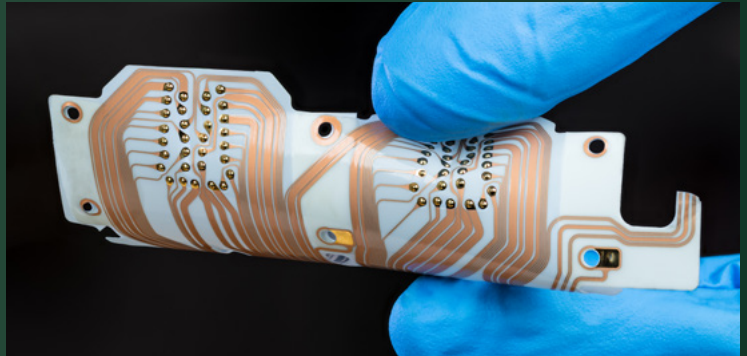
Board material and component selection also depend on the type of PCB you require. Here are the three main types.

Rigid PCBs

These PCBs feature a solid substrate, such as fiberglass, that prevents the PCB from twisting. Rigid PCBs are the most widely manufactured and can have single-layer, double-layer, or multi-layer constructions.

Flexible PCBs

Made of materials that can flex and move, like plastic, flexible PCBs can be single, double, or multi-layer to accommodate various applications.



Rigid-Flex PCBs

Combining both rigid and flexible PCB material properties, rigid-flex PCBs are made of one or more standard circuit boards and strips of flexible dielectric film that connect the boards. The standard board is made of layers of laminate, prepreg, soldermask, copper, nomenclature, and a final finish. The flexible dielectric film is usually made from polyimide.

Contact Your Fabricator to Build the Perfect PCB for Your Application

PCBs are used in many electronic devices and applications to connect and physically support electronic components. Selecting the right PCB material types and design for your application is essential to ensure optimal device and equipment performance. Whether you need a single-sided, double-sided, or multi-layered design with rigid, flex, or rigid-flex PCBs, your fabricator can help with your decision making.

Contact us today to learn more about our PCB materials and assembly capabilities.