Open up just about any electronic product and inside will be a Printed Circuit Board (PCB). This board provides the mechanical mounting for the electronic components that make up the design, as well as the electrical connections between them. Having found common use in the electronics industry for over half a century, PCBs have evolved into complex items created by skilled designers and manufactured using procession processes.

While understanding the way a PCB is manufactured is not mandatory for designers, those who have a grasp of the processes involved are far better equipped to design lower cost PCBs that benefit from higher manufacturing yields.

The PCB (Printed Circuit Board) is at the heart of just about every electronics product.

**The Anatomy of a PCB**

The following sections take a look at various types of PCB - from single-sided, to rigid-flex - and the key elements that are a common factor in the fabrication of all.

**Single-Sided Board**

The simplest PCB to manufacture is called a single-sided PCB, because it only has conductors on one side; usually the bottom side.
A single-sided PCB with room for components on the top side and soldering on the bottom side.

Single-sided PCBs begin their life in much the same way as all PCBs, and that is as an insulating substrate called the Core. The Core can be made from a multitude of materials depending on the desired properties of the final circuit, but the most common material is fiberglass.

The insulating Core is typically fabricated using a material known as FR4. This is Flame Retardant, type 4 woven glass reinforced epoxy laminate - a strong, rigid insulator that retains its high mechanical and electrical insulating properties in both dry and humid conditions, and also has good fabrication properties.

The Core is coated completely on one side with a thin layer of copper. After drilling the holes that will later be used for mounting the components, unwanted copper is removed, using a chemical etching process, to leave the tracks and pads needed to electrically connect the circuit’s components together.

The top side of the board is called the Component Side because through-hole components are usually mounted on this side so that their leads protrude through the board to the bottom side, where they can be more easily soldered to the copper pads and tracks. Surface mount components are the exception to this rule since they need to be mounted directly to the copper pads and can only ever exist on the Solder Side.

**Double-Sided Board**

Only slightly more complex than the single-sided PCB is a double-sided PCB, with copper traces on both the top and bottom sides of the core. This allows for more complex routing. By convention, through-hole components remain mounted on the Top Layer and surface mount components on the Bottom Layer, as per single-sided PCBs.

**Plated Through Holes (PTH)**

Double-sided boards typically rely on the leads of through-hole components to provide the electrical connection between top and bottom layers. However this is not always possible, since traces will sometimes need to traverse between the two layers at locations that don't coincide with a component lead. Therefore, a common addition to double-sided PCBs is Plated Through Holes (PTH).
Hole plating is achieved using an electrolysis process to deposit copper inside the hole after it has been drilled. This creates a conduction path between copper on the top and bottom layers, without relying on the lead of a through-hole component.

**Top and Bottom Solder Masks**

Most PCB assemblies are soldered using either wave or reflow soldering processes. In either case, there is the potential for solder bridging to occur between adjacent traces unless a *Solder Mask* is applied. The solder mask, as its name implies, provides a repellant (or mask) that helps prevent solder from indiscriminately adhering to copper in areas of the board that would otherwise cause a malfunction. As a secondary benefit, solder masks also prevent the otherwise exposed copper on the PCB traces from corroding.

While just about any color is possible, solder masks have traditionally been colored green and are responsible for the characteristic green that most people recognize PCBs as having. The solder mask is painted onto the top and bottom layers of the PCB using a precision screen printing process.

**Silkscreen Layers**

When visible information such as company logos, part numbers, or instructions need to be applied to the board, silk screening is used to apply the text to the outer surface of the circuit board. Silk screen information is usually colored white so as to contrast with the chosen solder mask, however any color can be used. Where spacing allows, screened text can indicate component designators, switch setting requirements, and additional features to assist in the assembly process.
Multi-Layered Boards

So far, only PCBs containing one or two copper layers have been described, however it is possible to create PCBs that contain many more layers. These PCBs are called *multi-layered PCBs* and they can offer much denser routing topologies, as well as better electrical noise characteristics. Each layer within a multi-layered PCB will either be a signal, or plane layer.

- **Signal Layers** - these layers are reserved entirely for carrying electrical signals from one component to another.
- **Plane Layers** - these layers are made up of large blocks of copper and are generally used for power supply sources such as VCC and GND. By utilizing a large surface area, plane layers are excellent at preventing and suppressing electrical noise.

Multi-layered PCBs can be manufactured in a couple of different ways but the simplest involves
laminating multiple thin, double-sided PCBs together, using a prepreg layer between each.

Prepreg - short for preimpregated - is a flexible material, typically also containing woven glass, which is supplied to the PCB fabricator partially cured (not completely cooked). It is included between the rigid layers in the layer stack during fabrication, and then heated to perform final curing, after which it becomes rigid, helping to join the layers and form the overall structure of the finished board.

The ratio of double-sided PCBs to prepreg layers can be defined according to cost, weight and electro-mechanical considerations. The following scenarios illustrate variations of layer stack for an example 8-layered board.

1. **Scenario 1** - an 8-layered board with a bias towards outer layer pairs.

   ![An 8-Layered PCB with a bias towards outer layer pairs.](image)

   In this layer stack, the copper on all four cores can be etched simultaneously and then sandwiched together (laminated) around layers of prepreg. This PCB would require the least complex manufacturing process.

2. **Scenario 2** - an 8-Layered PCB with a bias towards inner layer pairs.

   ![An 8-Layered PCB with a bias towards inner layer pairs.](image)
In this layer stack, the three cores can be etched simultaneously but then the outer prepreg and copper layers must be added separate, as part of the laminating process. The PCB as a whole must then pass through the etching process one more time to remove unwanted copper from the recently added outer layers.

3. **Scenario 3** - an 8-Layered PCB created from a single core, built up with several prepreg layers.

![An 8-Layered PCB created from a single core, built up with several prepreg layers.](image)

In this layer stack, a single PCB core is progressively built up using multiple layers of prepreg and copper. Each time a new prepreg and copper layer is added, the PCB must pass through the etching process again to remove unwanted copper from the recently added outer layer. This will occur sequentially for each of the 6 different prepreg layers. Because of the number of times the board has to pass through the copper etching process, this PCB would require the most complex manufacturing process.

**Blind and Buried Vias**

Because the cores used in creating multi-layered PCBs can be etched, drilled and plated individually, before being laminated together into a complete stack, it is possible to create vias that are only connected to internal layers and which do not surface on one or even both sides of the final board. This means that the land area that otherwise would have been occupied by the via on the outer layers of the PCB can now be used for routing. These types of via are:

- **Blind Vias** - these are vias that only surface on one side of the PCB.
- **Buried Vias** - these are vias that don’t surface on any side of the PCB.

Although the use of blind and buried vias is becoming increasingly common in advanced PCB designs, careful consideration needs to be given to the layer stackup of the PCB to ensure that the board is, in fact, able to be manufactured. Consider the layer stackup in the following image, that consists of 3 double-sided cores sandwiched around 2 prepreg layers. Consider also the via arrangement called for by an unwitting designer.
A typical 6 layered PCB stackup (top) and impossible via arrangement.

The via arrangement is impossible because it is not possible to drill (and plate) a hole that only passes through a prepreg layer. So in the image above, the 3rd and 5th vias (counting from the left) cannot be drilled. To overcome this, you would need to drill through one of the adjacent core layers and apply precise depth control to create a more plausible design, as illustrated below.

While the design above is more plausible, it limits the way the layers can be laminated together, and some PCB manufacturers may not allow this. In any case, what is proposed is far from efficient. The workflow would follow something like this:

1. Etch, drill and plate each of the double-sided core boards.
2. Laminate boards covering inner layers 2 to the bottom layer together.
3. Using controlled depth drilling, drill and plate the fifth via from left.
4. Add the final prepeg layer followed by the outer core covering the top and inner layer 1.
5. Using controlled depth drilling, drill and plate the third via from the left.

This process is quite involved and requires multiple passes through the drill, plate, and laminate processes. A better option would be to reduce steps 2-4 above into a single step, by laminating all cores together in one process and using controlled depth drilling to create the two blind vias, as illustrated below.
Finally, if controlled drilling depth is not possible with a given manufacturer, there may be no option but to drill the board all the way through at the points where a via connection is required across the prepreg layer, as illustrated below.

Clearly this means that the land area occupied by the via on the outer layers of the PCB can no longer be used for routing, however this may be the compromise necessary to yield a PCB design that can still be manufactured for a reasonable price.

**Rigid-Flex Boards**

Rigid-flex is the name given to a printed circuit that is a combination of both flexible circuit(s) and rigid circuit(s). This combination is ideal for exploiting the benefits of both flexible and rigid circuits - the rigid circuits can carry all or the bulk of the components, with the flexible sections acting as interconnections between the rigid sections.
By joining rigid sections of PCB together via flexible sections, complex, hybrid PCBs can be designed, that can be folded to fit into their enclosures.

Flex circuits are created from a stackup of flexible substrate material and copper, laminated together with adhesive, heat and pressure. The following image illustrates a simplified view of a flex circuit, with the constituent elements summarized thereafter:

- **Substrate** - the most common substrate is polyimide, a strong, yet flexible thermosetting polymer (thermoset). Examples of polyimides often used in the manufacture of flexible circuits include: Apical, Kapton, UPLEX, VTEC Pi, Norton TH, and Kaptrex. (Note that these are registered trade names, owned by their respective trademark holders).
- **Copper** - the copper layer is typically rolled and annealed (RA) copper, or sometimes wrought copper. These forms of copper are produced as a foil and offer excellent flexibility. They have an elongated grain, it is important to orient this correctly in a dynamic flex circuit to achieve the maximum flexing lifespan. This is achieved by orienting the dynamic flex circuit along the roll (so the circuit bends in the same way the foil was coiled on the roll). The flex manufacturer normally deals with this during the preparation of fabrication panels, it only becomes an issue if the designer performs their own circuit panelization (referred to as **nesting** in flex circuit design). The copper foil is typically coated with a photo-sensitive layer, which is then exposed and etched to give the desired pattern of conductors and termination pads.
- **Adhesive** - the adhesive is typically acrylic, and as the softest material in the structure, introduces the greatest number of manufacturing challenges. These include: squeeze-out,
where the adhesive is squeezed out into openings cut into the cover layers to access copper layers; Z-axis expansion defects due to the higher CTE (coefficient of thermal expansion) of acrylic adhesive; and moisture out gassing due to the higher rate of moisture absorbance, which can result in resin recession, blow outs and delamination at plated through hole sites. Alternative adhesives and adhesive-less processes are available, these may be more appropriate in less cost-sensitive applications.

There are a number of standard stackups available for flex and rigid-flex circuits, referred to as Types. These are summarized below.

- **Type 1 (Single Layer)** - this type offers single-sided flexible wiring containing one conductive layer and one or two polyimide outer cover layers.

  ![Type 1 flex structure with 2 cover layers, access holes on both sides and no plating in the component holes.](image)

  **Functional Summary**
  
<table>
<thead>
<tr>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>One conductive layer, either laminated between two insulating layers or uncovered on one side.</td>
</tr>
<tr>
<td>Access holes to conductors can be on either one or both sides.</td>
</tr>
<tr>
<td>No plating in component holes.</td>
</tr>
<tr>
<td>Components, stiffeners, pins and connectors can be used.</td>
</tr>
<tr>
<td>Suitable for static and dynamic flex applications.</td>
</tr>
</tbody>
</table>

- **Type 2 (Double Layer)** - this type offers double-sided flexible printed wiring, containing two conductive layers with plated through holes, with or without stiffeners.

  ![A Type 2 flex structure with access holes on both sides and plated through holes.](image)

  **Functional Summary**
  
<table>
<thead>
<tr>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Two conductive layers with an insulating layer between; outer layers can have covers or exposed pads.</td>
</tr>
<tr>
<td>Plated through-holes provide connection between layers.</td>
</tr>
<tr>
<td>Access holes or exposed pads without covers can be on either or both sides; vias can be covered on both sides.</td>
</tr>
<tr>
<td>Components, stiffeners, pins and connectors can be used.</td>
</tr>
<tr>
<td>Suitable for static and dynamic flex applications.</td>
</tr>
</tbody>
</table>

- **Type 3 (Multilayer)** - this type offers multilayer flexible printed wiring, containing three or more
conductive layers with plated-through holes, with or without stiffeners.

Functional Summary

<table>
<thead>
<tr>
<th>Description</th>
<th>Details</th>
</tr>
</thead>
<tbody>
<tr>
<td>Three or more flexible conductive layers with flexible insulating layers</td>
<td>between each one; outer layers can have covers or exposed pads.</td>
</tr>
<tr>
<td>Plated through-holes provide connection between layers.</td>
<td></td>
</tr>
<tr>
<td>Access holes or exposed pads without covers can be on either or both sides</td>
<td></td>
</tr>
<tr>
<td>Vias can be blind or buried.</td>
<td></td>
</tr>
<tr>
<td>Components, stiffeners, pins and connectors can be used.</td>
<td></td>
</tr>
<tr>
<td>Typically used for static flex applications.</td>
<td></td>
</tr>
</tbody>
</table>

- **Type 4 (Multilayer Rigid-Flex)** - this type offers multilayer rigid and flexible material combinations (Rigid-Flex), containing three or more conductive layers with plated-through holes. Rigid-flex has conductors on the rigid layers, which differentiates it from multilayer circuits with stiffeners.
### Functional Summary

<table>
<thead>
<tr>
<th>Description</th>
<th>Details</th>
</tr>
</thead>
<tbody>
<tr>
<td>Three or more conductive layers with either flexible or rigid insulation</td>
<td>Material as insulators between each one; outer layers can have covers or exposed pads.</td>
</tr>
<tr>
<td>Plated through-holes extend through both rigid and flexible layers (apart from blind and buried vias).</td>
<td></td>
</tr>
<tr>
<td>Access holes or exposed pads without covers can be on either or both sides.</td>
<td></td>
</tr>
<tr>
<td>Vias or interconnects can be fully covered for maximum insulation.</td>
<td></td>
</tr>
<tr>
<td>Components, stiffeners, pins, connectors, heat sinks, and mounting brackets can be used.</td>
<td></td>
</tr>
</tbody>
</table>

The Types are defined in the following standards:
- [IPC 6013B](https://www.ipc.org/standards/6013b) - *Qualification and Performance Specification for Flexible Printed Boards.*

### The PCB Manufacturing Process

The process of manufacturing a PCB is reasonably straightforward, and while it may vary slightly from manufacturer to manufacturer, understanding how this process operates will help you create PCBs that are less likely to suffer from manufacturing issues. A detailed step-by-step flow of the process used to manufacture standard multi-layer PCBs (not flex or rigid-flex) is given as a guide below.

1. **Material Selection** - the core and prepreg layers required for the final assembly are selected and cut to size.
2. **Drill & Metalize Blind/Buried Vias** - this step is only required if the board is to feature blind and/or buried vias. Drill vias and coat the via barrels with a metallization layer to ensure conduction through the core.
3. **Laminate/Expose/Develop Photoresist** - apply a photo-resistive coating to the copper clad cores. This is then exposed to UV light through a negative image of the layer’s artwork. Once developed, the exposed photoresist will harden so that it is impervious to the etchant used in the next step. Unexposed photoresist is washed away leaving exposed copper underneath.
4. **Etch** - immerse in an acid bath to remove the unprotected copper.
5. **Strip Resist** - the photoresist used to protect the copper artwork is no longer required and is removed.
6. **Laminate Cores together with Prepreg to create Panel** - stack the cores, in order, on top of one another, with layers of prepreg in between. When placed into a heated press, the prepreg will melt into an epoxy glue and will bind the core layers together to form a complete layer stack panel.
7. **Drill & Metalize Holes/Vias** - drill vias and holes through the complete panel and coat the barrels with a metallization layer to ensure conduction.
8. **Laminate/Expose/Develop Photoresist** - apply a photo-resistive coating to the outer layers. This is then exposed to UV light through a negative image of the outer layers’ artwork. Once developed, the exposed photoresist will harden so that it is impervious to the etchant used in the next step. Unexposed photoresist is washed away leaving exposed copper underneath.
9. **Etch** - immerse in an acid bath to remove the unprotected copper.
10. **Strip Resist** - the photoresist used to protect the copper artwork is no longer required and is removed.
11. **Print Solder Mask** - print the solder mask onto the board to both protect the underlying copper artwork from oxidization, and prevent solder from adhering to unintended portions of the PCB.

12. **Apply Surface Finish** - apply a surface finish to exposed copper areas to both protect them from the elements, and provide a suitable surface for component mounting and soldering.

13. **Print Silkscreen** - print the silkscreen text and graphics onto the completed PCB.

14. **Route** - CNC Route the completed PCB to shape.

15. **Pack and Ship Complete PCB** - encase the PCB in packaging that will protect the PCB from moisture and corrosion, before shipping the completed PCB to its customer.

This process obtains the fabricated bare-board that will then proceed - after requisite visual inspections and bare-board testing by the customer - to the Assembly house, at which time the PCB will be assembled using Pick and Place machines, and in accordance with the supplied Pick and Place and Bill of Materials manufacturing output. It is typical after assembly to also conduct in-circuit assembly testing.

**Visualizing the Manufacturing Process**

The following sections provide a more graphical look at the process involved in fabricating the bare-board, for PCBs of differing layer counts.

**Double-Sided Boards**

1. Double-sided PCBs begin their life in much the same way as single-sided PCBs, except the core is coated with copper on both top and bottom surfaces.

2. The board is pre-drilled with all holes.

3. A photoresistive mask is then applied.
A negative image of the desired tracks is carefully aligned over the top of the photoresist.

4.

The photoresistive coating is sensitive to light so when the board is illuminated under a powerful UV light, the exposed areas of photoresist harden.

5.

The negative overlay is removed and the photoresist is developed.

6.

The un-exposed portions of photoresist are washed away to reveal the copper layers below.

7.

The board is then immersed in acid. The hardened photoresist protects the copper it covers, but the exposed copper is eaten away so that only copper tracks remain.

8.

The hardened photoresist is then removed to expose the underlying copper traces.

9.
10. Next, plating is chemically applied to the via and hole walls to ensure a conduction path between the top and bottom layers.

11. The copper is then coated with a thin layer of tin to help with solder adhesion, and a solder mask is applied to repel solder away from areas of the board that don't require it. The solder mask is typically what gives the PCB its green colour however other soldermask colours are also available.

4-Layer Boards

1. For 4-layer boards, the core is etched prior to drilling and then a layer of prepreg and copper are bonded to the outer surfaces using heat and high pressure.

2. The stack is then drilled and the outer layers etched, in much the same way as previously described for double-sided boards.
6-Layers and More

1. For boards that require 6 or more layers, core and prepreg layers will be interleaved to build up the required number of layers. The cores are all etched individually and then sandwiched together with layers of prepreg on the top and bottom, as well as bonding the two cores together. As with the 4-layer board, copper sheets are also attached to the top and bottom outer surfaces.

2. The stack is then drilled and the outer layers etched in much the same way as previously described for double-sided boards. This process can be extended in the same manner to create boards with 30 or more layers.

Where to Now?

No matter what type of PCB you are wanting to manufacture - be it rigid, or rigid-flex - the first thing to do is to define the layer stackup as required. Within Altium Designer's PCB Editor, all layer stacks are defined in the Layer Stack Manager dialog (Design ▸ Layer Stack Manager). For a new board, its single default stack comprises: a dielectric core, 2 copper layers, as well as the top and bottom solder/coverlay and overlay layers, as shown in the image below.
Layer stack management is performed in the Layer Stack Manager dialog. The default single stack for a new board is shown.

The Layer Stack Manager dialog not only caters for the definition of a single layer stack, for standard single-sided, double-sided, or multi-layer boards, but also facilitates the definition of multiple stacks, in support of rigid-flex designs.

The Layer Stack Manager dialog supports the definition of any number of layer stacks.

- For more information on defining the layer stack for your board in Altium Designer, see Defining the Layer Stack.
- For more information on designing your boards with flexibility, see Rigid-Flex Design.

Source URL: https://www.altium.com/documentation/display/ADES/((The+Board))_AD