

Stack-up and the Basics of Multi-Layer Construction

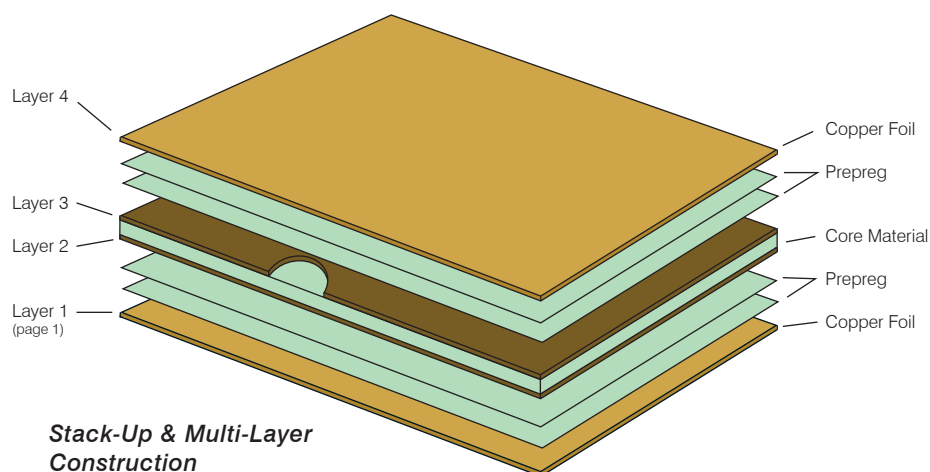
The Stack-Up is the specific call out of material thickness and copper weights required to produce a particular multi-layered printed circuit board. The description of the stack-up can be found on the PCB fabrication drawing—a drawn cross-section depicting the material and copper thickness of each layer involved. The stack-up is how the designer tells the manufacturer the way to properly build the PCB to work as designed.

Circuit board material available for multi-layer construction are primarily of two types: **Core material** that has copper laminated to both sides (cured laminate) and is usually 0.031 or less thick and **pre-preg** material (uncured resin treated glass) is used to fill the spacing between cores and out-layer foils and used to bond all layers together. **Copper foil** of various weights (half ounce or greater) is available as well for outer layer construction.

The manufacture of a multilayered board requires the combination of core and pre-preg materials. The core material is the basis for the construction of the multi-layered board. Core material is etched with the required circuitry and oxide treated to promote proper adherence to the pre-preg layers that will be applied and the stack-up is followed. During the manufacturing process this “laying-up” of materials involved is called creating a multi-layer **book**.

Using a 4 layer PCB as an example, the book is created this way: Copper foil is laid down as Layer 1 or the first page of the PCB followed by a layer or layers as required of pre-preg. The core material that contains Layers 2 and 3, or in this case pages 2 and 3, is applied on top followed by an additional layer or layers pre-preg and then covered with final page, Layer 4, and the book is complete. Depending on the manufacturing equipment, several books may be stacked and sent the press, either vacuum or hydraulic, to be baked and cured at the correct temperature cycles to become one solid board. It is during the curing cycle—a combination of a specified temperature, pressure and time for total curing—that the cross linking of molecules is complete.

After the curing is complete, the books or rather the fully cured panels, are removed from the press and trimmed of excess flash, cleaned and sent on to the drilling department to continue with the balance of the PCB manufacturing process.



Research for the above information may be from, but is not limited to IPC reference manuals, the PCB Handbook, The Bare Board PCB design manual and consultation with industry professionals. Please consult a process engineer familiar with your companies PCB assembly process before making any procedure changes.