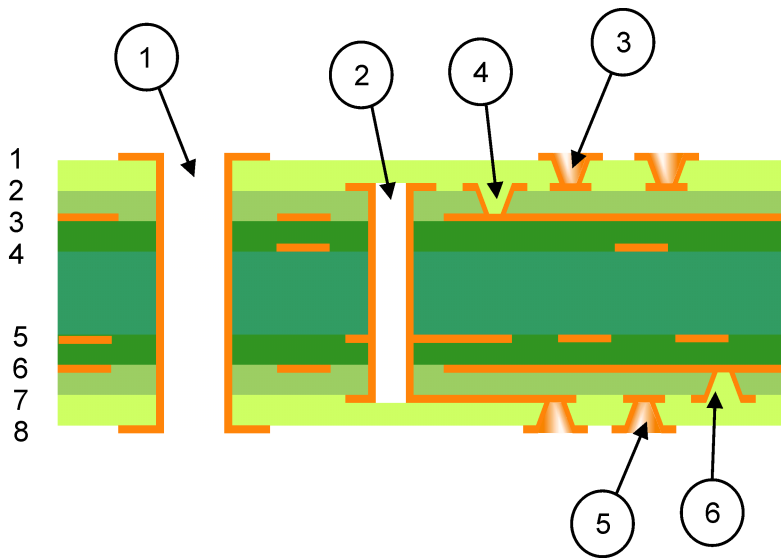


A multilayer board in PowerPCB

Let's make a sample 8 layer stackup, including vias. See the picture below with two drilled vias (1,2) and 4 laser vias (3-6).

The process is described in details. Note that some PowerPCB functions used here are described later in the book.

First, it is essential to contact the fabricator to confirm that the planned pad, annular ring and drill sizes and spacings are applicable to the process. Using laser vias will limit the number of fabricators who can be used to make boards.



8 layer board. Different layers described by colours

Steps in PowerLogic or PowerPCB:

1. "*Setup-Layer Definition*", set layer count to 8 by "*Electrical Layers-Modify*".
2. It is usual to use copper fill for all free areas, especially in the inner layers. If you want to use autorouter (*BlazeRouter*), they all cannot be defined as "*Split/Mixed*", because *BlazeRouter* avoids Split planes, with the exception of reaching a component pin which is located inside a plane area.

It is better to select the main Power and Ground layers, 3 and 6 in this example, and use Copper Pour for the rest later in the design process.