How to stack up a 4-layer board

By: Yahia Ahmed Saber

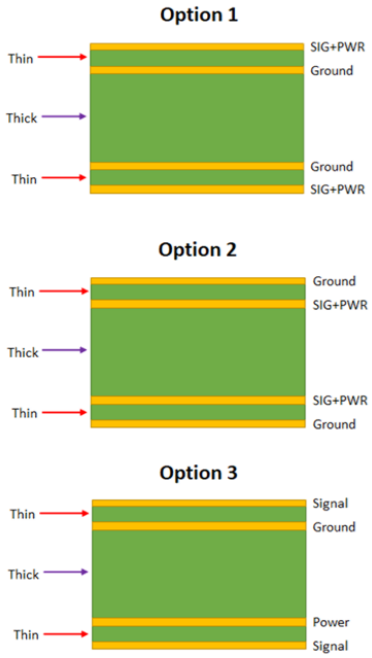
There is variety when choosing the correct way to stack up a 4-layer board and it depends on the application it is used for. The three most common configurations include:

1. Two internal GND planes

These are mostly used in high speed PCBs as they are perform well in such applications where routing on both sides is required and for high speed signals. One of its main advantages is having ground planes adjacent to signal layers meaning a return path is always available. However, the space free for routing is a bit limited.

2. Two external GND planes

Considered to be the inversion of the previous configuration, it does not excel in high speed routing as one of its significant drawbacks is the possibility of crosstalk between signals. Instead, it is used in low noise systems as a result of its exterior ground shielding from noise. Using this stack-up also gives more opportunity for reducing costs.



3. Signal-Ground-Power-Signal

This configuration is often used where different signals should be produced and each with their levels of power needed. The ground and power planes being separated helps in noise reduction. Nevertheless, the bottom signal layer may not work too well with high speed signals and this stack-up is a bit complex so could be more expensive.