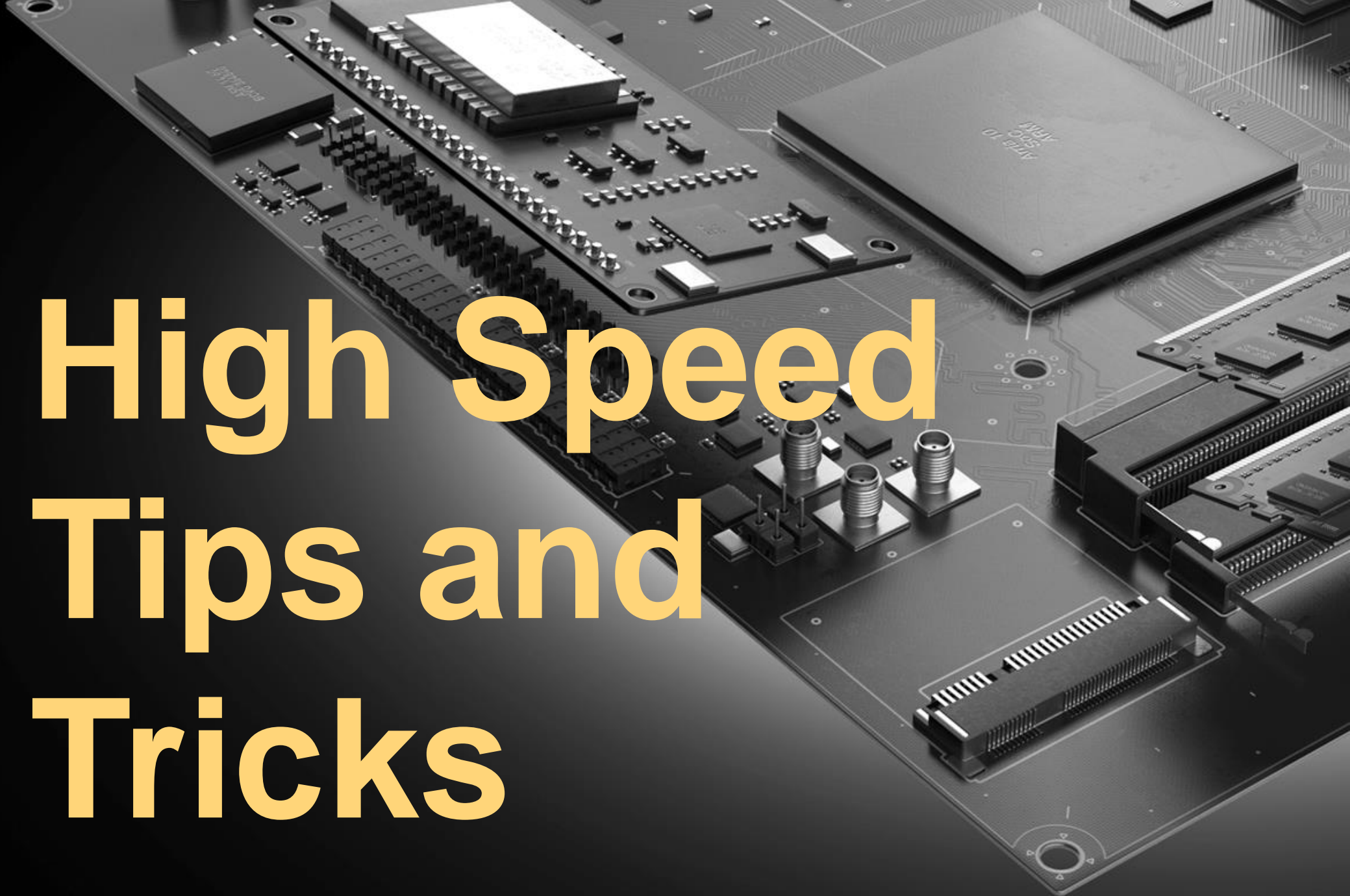


Altium

High Speed Tips and Tricks



1. FR-4 MAY BE THE BEST BOARD MATERIAL CHOICE FOR HIGH SPEED PCB DESIGN
2. BOARD LAYER STACKUP CONSIDERATIONS FOR HIGH SPEED BOARD DESIGN
3. COMPONENT SHAPE CONSIDERATIONS
4. BYPASS CAPACITOR PLACE AND ROUTE TIPS
5. TRACE TUNING FOR ROUTING LENGTH MATCHING

1

Best Board Material Choice for High Speed PCB Design

High Speed Board Materials

FR-4 has been the standard material used for the fabrication of a PCB. FR-4 is a very cost effective material and has good fabrication properties. Whereas it has a downside when it comes to excessive power, voltage or heat. There is a cost to all the advantages of high speed board materials. Some of these materials are more difficult to work with during fabrication and assembly than the others.

Decreased Signal Loss -High speed design board materials have a much lower dissipation factor than FR-4.

More Tightly Controlled Impedance -FR-4 do not offer as precise control over dielectric constants.

Better Thermal Management-Some high speed design board materials have much better thermal conductivity than FR-4.

Increased Moisture Absorption -FR-4 can have a moisture absorption rating of nearly 50%, some of the PTFE materials are as low as 2%.

Robust Dimensional Stability - For a dense high speed design with tight tolerances, the need for dimensional stability increases.



[See original content here](#)

2

Board Layer Stackup Considerations for High Speed Board Design

Board Layer Stackup Considerations

High speed PCB designs have tighter signal integrity requirements, and constructing the board becomes part of the overall signal integrity equation.

Stackup requirements for high speed routing - High speed signals are most often routed using either microstrip or stripline methodologies. Microstrip is the simpler methodology of the two. It has a transmission line, which is a routed trace on an external layer of the board, that is separated from its referenced ground plane by a layer of dielectric material. Alternatively, stripline has a referenced ground plane both above and below the transmission line.

Stackup requirements for ground and power planes - Effective high speed routing using microstrip or stripline routing methodologies require a least one complete ground plane. Construct your board stackups so that high speed circuitry is not forced to cross plane splits or non-referenced ground planes.



[See original content here](#)

3

Component Shape Considerations

Component Shape Considerations

Schematic organization, board materials & layer configuration, critical component placement, and how high speed signals are to be routed are all aspects of high speed design that needs planning.

Pad shapes for high speed PCB design - Referred to as landing pads, these shapes are the bare metal pads that the component pins will be soldered to on the finished PCB. Usually, one or two pad shapes are duplicated to create an entire component footprint shape. To help the high speed needs of the circuit, the pad size should be reduced. Instead of oversizing the pad 30% from the actual pin size, a smaller percentage such as 5% is more beneficial.

Via shapes need consideration - The size of a via shape is driven by the diameter of the drilled hole, and therefore the design team will need to consider what via drill sizes are needed prior to layout. Smaller vias will improve high speed signal performance while at the same time increase the manufacturing costs.

Component selection and placement - It is a good practice to eliminate or minimize the use of sockets in a high speed design. Selecting the correct package, and accompanying component footprint, is also important. Component footprint shapes may need to be adjusted for thermal considerations.



[See original content here](#)

4

Bypass Capacitor Place and Route Tips

Bypass Capacitor Place & Route

Tips Some of the bypass capacitor requirements in the PDN of a high speed design, and what kind of placement and routing strategies will best help .

- Role of bypass capacitors** -A bypass capacitor is can be placed near the IC to supply the required current for these quick spikes .It is important for reducing the ground bounce that can come from digital devices that have faster switching times. Bypass capacitors also are used to filter the low-frequency noise caused by the power supply and are helpful with other signal integrity and EMI issues as well.
- Number of bypass capacitors to be used** - The amount of bypass capacitors needed for a design will depend in relation to the parts that they are assigned to, and how many of those parts are being used.
- Bypass capacitor placement and routing best practices** - Bypass capacitors should be placed as close to the device that they are assigned to. For circuits requiring multiple bypass capacitors placed near the power pin of a specific device, the capacitors should be placed next to that pin in ascending order of value. When routing a bypass capacitor, it should start from the power or ground pin of the device and go directly to the capacitor pin. From there the route can continue on to a via connecting it to the power or ground plane.



[See original content here](#)

5

Trace Tuning for Routing Length Matching

Trace Tuning for Routing Length

Matching PCB trace tuning is creating serpentine trace routing map patterns to achieve a specific overall length of the trace. This trace length must be matched with the lengths of other traces so that all of their signals arrive at their destinations at the same time.

Different Types of Trace Tuning for Trace Length Matching – High-speed design experts recommend routing in big trombone patterns to fill up the area and leave space for tuning later. When ready, fine tune the trombone routing in a tighter accordion pattern to get the exact trace length needed. In the same way that individual nets need to be tuned to other nets, the lengths of the two nets in a differential pair should also be matched.

Trace tuning functionality - Trace length rules can be set up in the high-speed routing constraints. Use processes that will automatically route the tuning patterns to the correct lengths so that there is no need to manually create the waves. Finally, use differential pair routers to match the lengths of the two traces in the pair together.



[See original content here](#)

Altium.

| HIGH SPEED TIPS AND TRICKS



ALTIUM
DESIGNER18



HIGH SPEED TIPS AND TRICKS

Altium Designer 18

New to Altium Designer?
Don't be!

Try the latest release and get time saving workflows and enhanced team
collaboration options today at:

www.Altium.com/free-trials



Altium

Thanks for your attention!