Published on *Online Documentation for Altium Products* (https://www.altium.com/documentation)

Using Altium Documentation
Modified by Jason Howie on Apr 11, 2017

Altium Designer's PCB Editor uses the concept of *Design Rules* to define the requirements of a design. These rules collectively form an 'instruction set' for the PCB Editor to follow. They cover every aspect of the design - from routing widths, clearances, plane connection styles, routing via styles, and so on - and many of the rules can be monitored in real-time by the online Design Rule Checker (DRC).

Design rules target specific objects and are applied in a hierarchical fashion. Multiple rules of the same type can be set up. It may arise that a design object is covered by more than one rule with the same scope. In this instance, a contention exists. All contentions are resolved by a priority setting. The system goes through the rules from highest to lowest priority and picks the first one whose scope(s) match the object(s) being checked.

With a well-defined set of design rules, you can successfully complete board designs with varying and often stringent design requirements. And as the PCB Editor is rules-driven, taking the time to set up the rules at the outset of the design will enable you to effectively get on with the job of designing, safe in the knowledge that the rules system is working hard to ensure that success.

> For a detailed overview of the rules system in Altium Designer, see Constraining the Design - Design Rules. For an overview of the system used to verify adherence to defined rules, see Design Rule Checking.

The Design Rules Reference is comprised of the following categories:

- Electrical Rules
- Routing Rules
- SMT Rules
- Mask Rules
- Plane Rules

- Testpoint Rules
- Manufacturing Rules
- High Speed Rules
- Placement Rules
- Signal Integrity Rules
