PCB design is no longer a matter of simply placing tracks to create connections. In fact, many complex designs can be a minefield of conflicting requirements. High speed logic combined with smaller and more complex packaging technologies place extreme demands on the PCB Designer. It is not possible to satisfy all the requirements of the design by only considering the clearance between tracks, pads and vias. Designs today can also require that you apply specific requirements to individual nets, components or regions of the board as well as considering such issues as net impedance, crosstalk, signal reflections and path lengths. Altium Designer's rules-driven PCB Editor allows you to define design rules that monitor and test for these and other requirements.

Design rules collectively form an instruction set for the PCB Editor to follow. Each rule represents a requirement of your design and many of the rules, for example clearance and width constraints, can be monitored as you work by the Online Design Rule Checker (DRC). Certain rules are monitored when using additional features of the software, for example routing-based rules when routing a design – either interactively or using the Situs Autorouter – or signal integrity-based rules used by the Signal Integrity Analyzer when performing a detailed signal integrity analysis of a design.

Armed with a well-defined set of design rules and the PCB Editor's variety of features for checking and interrogating violations of these rules, you can successfully complete board designs with varying and often stringent design requirements.

The PCB Rules System – Fundamentals

The rules system built-in to Altium Designer’s PCB Editor has several fundamental features that set it apart from most other design rule systems employed by other PCB editing environments:

- **Rules are separate from the objects** – the defined design rules are separate from the design objects to which they are applied. A rule is not added as an attribute of a design object, but is rather added to the overall rule set and then scoped to apply to that object. In this way, rules can be applied to multiple design objects, modified, or applied to different objects. This would otherwise prove very cumbersome to do if having to change rule attributes of design objects on an individual basis.

- **Rules are targeted (scoped) by writing a query** – instead of using a set of fixed, pre-defined rule scopes, a flexible query system is used to define the objects that a rule is applied to. This gives precise control over the target of each and every design rule.

- **Rules for any design situation** – multiple rules of the same type can be defined and targeted to different sets of objects, allowing complete control over the definition of your board constraints. For example, you can set up rules to route a net at one width on the top layer, and a different width on the bottom layer. Or you can define larger solder mask openings for pads on the bottom layer than the top layer.

- **Each rule has a priority** – any design object can be targeted by multiple rules of the same type, catering for general and more specific situations. To resolve any rule contention, the rule priority is used. The system simply goes through the rules from highest to lowest priority and picks the first one whose scope expression(s) match the object(s) being checked.

- **There are two types of rules** – unary rules, or rules that define the required behavior of an object; and binary rules, rules that define the interaction between two objects.

Defining and Managing the Design Rules

Design rules are defined and managed in the PCB Rules and Constraints Editor dialog. Note that rule requirements can also be defined on the schematic and transferred to the PCB Editor during synchronization. See the section Defining Rules in the Schematic for more details.

The PCB Rules and Constraints Editor dialog has two sections. The tree on the left lists the ten rule categories. Below each category heading are the individual rule types available in that category. Click on a rule type to list all rules of that type that have been defined. Right-click on a rule type to add a new rule of that kind.
Specifying the PCB Design Rules and Resolving Violations

The right-hand side of the dialog will either list a summary of defined rules of the current type, category or entire system, or display the settings of the rule if an actual rule is selected in the tree on the left. To configure a rule’s settings either click on that rule in the tree, or double-click on it in a summary list.

Prioritizing the Rules

To simplify the process of defining and managing rules, the idea is to define general rules that cover broad requirements and then override these with specific rules in specific situations. For this to be possible you need to be able to prioritize the rules, to indicate which one to use when an object is targeted by multiple rules of the same type.

For example, to specify the most commonly used routing width on the board, you define a single rule that applies to every net on the board. You can then override this rule for a specific net (or a class of nets for that matter) by adding another rule of the same type, but with a higher priority.

Another example could be the solder mask requirements – here you would define one mask rule that targeted every pad and via on the board, which could then be overridden for the pads in a specific footprint-kind. This footprint-specific rule could further be overridden for a specific pad in that footprint, if required.

An important aspect of managing the rules is ensuring that all the priorities are set appropriately. When you add a new rule it defaults to the highest priority. Use the Priorities button at the bottom of the PCB Rules and Constraints Editor dialog to configure the priorities.

Scoping, or Targeting the Design Rules

As mentioned, rules are not attributes of the design objects, they are defined separately. For the system to then know what objects a given rule applies to, it needs to know the scope of the rule – the extent of its application. Scoping, or targeting the rule is done in the PCB Rules and Constraints Editor dialog.

Rather than being restricted to a predefined list of possible target options, each design rule is scoped by writing what is called a query. A query is essentially an instruction to the software that defines the set of design objects to be targeted. Queries are written using query keywords. In the same way that you can write a query in the Filter panel to find a specific set of objects, you write a query to define the objects that each rule targets. An example query might be:

`InNet (GND) And OnLayer (TopLayer)`

If this query was used as the scope for a width rule, whenever you were routing the GND net and switched to the top layer, the track width would automatically change to the specified width. Also, when you run a design rule check, any GND net routing on the top layer would have to have the specified width or it would be flagged as a violation.

For more information on queries, refer to the Introduction to the Query Language article in the Getting Started with Altium Designer document.
The Process of Defining and Scoping a Rule

As mentioned, rules are defined in the PCB Rules and Constraints Editor dialog. To add a new rule, navigate in the tree on the left to the required rule type, right-click and select Add Rule. When the rule appears you should:

- Give the rule a meaningful name to make it identifiable
- Set the constraints of the rule, such as the clearance or routing width required
- Define the scope of the rule by entering a query (or queries for a binary rule)
- Set the priority of the rule.

As well as manually typing in a query for a design rule, there are a number of other techniques that can be used to create and scope design rules.

Creating a New Rule with the Rules Wizard

A new rule can be created using the New Rule Wizard. This wizard can be accessed from the PCB Rules and Constraints Editor dialog, or directly from the main Design menu. Follow the wizard to quickly create a new design rule of any type.

Editing a Rule

To edit the scope and constraint attributes for a rule, either click on the entry for the rule in the folder-tree pane or double-click on its entry in a summary list. The main editing window of the dialog will change to give access to the controls for defining the scope and constraint attributes for that rule.

Scoping a Rule with the Built-in Options

When you edit a rule you will notice a number of options in a group titled Where the first object matches. These options provide a quick way of creating a query, you simply select one of the options and if required use the applicable drop-down list(s) to select the appropriate target, such as a Net or Layer. As you do this the query string will appear in the Full Query field. Note that this query is just an ASCII string, you can edit this as required.
Specifying the PCB Design Rules and Resolving Violations

The Advanced (Query) option enables you to write your own, maybe more complex, but also more specific query. You can type your own specific query for the rule scope directly into the Full Query region. Alternatively, two features are available to help in the creation of queries - the Query Builder and the Query Helper. These can be very useful if you are unsure of the syntax of a query, or the possible keywords that you may want to use.

Scoping a Rule with the Query Builder

Click the Query Builder button to open the Building Query from Board dialog, which enables you to create a query for targeting specific objects in the design document, by simple construction of a string of ANDed and/or ORed conditions.

The left-hand section of the dialog is where you specify the condition(s) that you require to target the set of objects needed. As you define a condition, a preview of the currently built query is shown in the right-hand type, a context-sensitive locate the keyword or object that you wish to use in the query string and then double-click on section.

Getting Help with the Query Helper

To use the Query Helper, ensure that the Advanced (Query) option is enabled and then click the Query Helper button to open the Query Helper dialog. The underlying Query Engine analyzes the PCB design and lists all available objects, along with generic keywords for use in queries.

Use the Query region of the dialog to compose a query expression. You can type directly within the region. As you type, a context-sensitive prompt list of possible keywords or objects will appear as an aid.

The Categories region of the dialog provides access to available PCB Functions, PCB Object Lists and System Functions, which can be used to create the query expression. As you click on a sub-category within each of these three areas, the corresponding list of keywords or objects will be displayed in the region to the right. Simply locate the keyword or object that you wish to use in the query string and then double-click on the entry.
Comparing to See Which Rule Applies

Depending on the board design, a fair number of design rules may need to be defined, with scopes that range from the very simple to the very complex. It is a good idea to check that the rules you have defined do indeed target their intended objects. Care at the rule definition stage can save wasted time and effort tracing violations caused through incorrect rule scoping.

There are essentially two methods for verifying rule scopes – either by selecting design objects and interrogating the rules that currently apply to them, or by taking a rule and observing which objects fall under its scope. The method you choose, or a combination of the two, will depend largely on personal preference.

From an Object’s Perspective

For any placed object in the current design, you can quickly access information about which unary design rules apply to that object. Simply position the cursor over the object, right-click and select Applicable Unary Rules.

All defined design rules that could be applied to the selected object are analyzed and listed in the Applicable Rules dialog.

Each rule that is listed in the dialog will have either a tick (✓) or a cross (✗) next to it. A tick indicates that this is the rule with the highest priority out of all applicable rules of the same type and is the rule currently applied. Lower priority rules of the same type are listed with a cross next to them, indicating that they are applicable but, as they are not the highest priority rule, they are not currently applied.

Any rules that would apply to the object but are currently disabled also have a cross next to them and are shown using strike through highlighting.

In a similar fashion, you can also access information about the binary design rules that apply between two placed objects in a design. Simply position the cursor over any object, right-click and select Applicable Binary Rules. You will be prompted to select two objects in the design. The Applicable Rules dialog will then appear, displaying all binary design rules that apply to those objects.

From a Rule’s Perspective

If, rather than seeing which rules apply to an object (or between two objects) you would prefer to pick a rule and see which objects that rule applies to, this can be achieved from the PCB panel, when configured in Rules mode. As you click on a specific rule in the Rules region of the panel, filtering will be applied, using the rule as the scope of the filter. Only those design objects that fall under the scope of the rule will be filtered. By employing the mask highlighting feature, you can quickly see the resulting objects targeted by the rule.

This method is particularly useful when creating your query for a rule’s scope(s), since you can edit a rule directly from the panel and therefore tweak your query until the desired objects are captured by the scope(s).
Specifying the PCB Design Rules and Resolving Violations

Editing from the PCB Panel

A list of currently defined rules can be displayed in the PCB panel by enabling the Rules mode for the panel.

All rules can be viewed or only those rules associated with a particular rule type. The Rule Classes region will only include a rule type if a specific rule of that type has been defined for the active design.

Double-clicking on a rule entry will open the relevant Edit PCB Rule dialog, from where you can edit the scope of the rule and also the specific rule constraints.

Tips: For further information on any panel, press F1 when the cursor is over the panel.

Exporting and Importing Rules

Design rules can be exported from, and imported to, the PCB Rules and Constraints Editor dialog. This allows you to save and load favorite rule definitions between different designs. To export, right-click anywhere within the tree on the left of the dialog and select Export Rules. Exported rules are stored in a PCB Rule file (*.Rul).

To import, select Import Rules from the same right-click pop-up menu, then browse to and open, the required PCB Rule file.

Design Rule Reports

A report of currently defined design rules can be generated from within the PCB Rules and Constraints Editor dialog. The report can cater for all rule categories, a specific rule category or a specific rule type. Simply right-click in the respective summary list, or over the respective entry in the folder-tree and select Report. The Report Preview dialog will appear, with the appropriate report already loaded.

Use this dialog to inspect the report using various page/zoom controls, before ultimately exporting it to file or printing it.
Defining Rules in the Schematic

Design constraints (rules) can be defined prior to PCB layout, by adding parameters that are configured as design rule directives to the schematic source document(s). The scope of the corresponding PCB design rule, created when the design is transferred to the PCB document, is determined by the nature of the object to which the parameter (added as a rule) is assigned. The following table summarizes the schematic parameter-to-PCB rule scope options that are supported:

<table>
<thead>
<tr>
<th>Add a Parameter (as a rule) to a...</th>
<th>From...</th>
<th>For a PCB rule scope of...</th>
</tr>
</thead>
<tbody>
<tr>
<td>Pin</td>
<td>the Parameters tab of the Pin Properties dialog</td>
<td>Pad</td>
</tr>
<tr>
<td>Port</td>
<td>the Parameters tab of the Port Properties dialog</td>
<td>Net</td>
</tr>
<tr>
<td>Wire</td>
<td>the Parameters tab, after placing a PCB Layout Directive (Parameter Set object) on the wire using the <strong>Place » Directives » PCB Layout</strong> command</td>
<td>Net</td>
</tr>
<tr>
<td>Bus</td>
<td>the Parameters tab, after placing a PCB Layout Directive (Parameter Set object) on the bus using the <strong>Place » Directives » PCB Layout</strong> command</td>
<td>Net Class</td>
</tr>
<tr>
<td>Component</td>
<td>the Parameters region of the Component Properties dialog</td>
<td>Component</td>
</tr>
<tr>
<td>Sheet Symbol</td>
<td>the Parameters tab of the Sheet Symbol dialog</td>
<td>Component Class</td>
</tr>
<tr>
<td>Sheet</td>
<td>the Parameters tab of the Document Options dialog (<strong>Design » Document Options</strong>)</td>
<td>All Objects</td>
</tr>
</tbody>
</table>

In each case, the method of adding a rule-based parameter is the same. From the respective tab or dialog, simply perform the following:

- Add a parameter as a rule
- Select which rule type to use
- Configure the constraints for the chosen rule type.

Synchronicity through Unique IDs

When adding design rule parameters to objects on a schematic, a unique ID is given to each rule parameter. The same IDs are given to the corresponding design rules that are created in the PCB. With this Unique ID, the constraints of a rule can be edited on either the schematic or PCB side and the changes pushed through upon synchronization.

Figure 1 shows an example of adding a PCB Layout directive to a wire in a schematic. In the example, a Width rule has been defined for the wire which, when passed to the PCB design upon synchronization, appears as an additional Width rule. The important thing to notice is that the Unique ID entries for the parameter and the PCB design rule are the same.

Figure 1. Schematic-PCB rule synchronization using Unique IDs
Specifying the PCB Design Rules and Resolving Violations

Design Rule Checking (DRC)

Design Rule Checking (DRC) is a powerful automated feature that checks both the logical and physical integrity of your design. Checks are made against any or all enabled design rules and can be made online, as you work, and/or as a batch check, with results listed in the Messages panel and a generated report.

Configuring the DRC

Configuration for the check is carried out in the Design Rule Checker dialog, accessed by selecting Design Rule Check from the PCB Editor’s Tools menu.

On the left side of the dialog, each of the design rule categories whose rule types can be checked are listed under the Rules To Check folder. Click on a category to list all associated (and checkable) design rule types for that category.

The right-hand side of the dialog provides options for enabling/disabling Online (where available) and/or Batch checking for each rule type you wish to check for the selected category.

Using Online DRC

Online Design Rule Checking runs in the background, as you work, flagging and/or automatically preventing design rule violations. For a rule to be subject to the Online DRC, the following three requirements must be met:

- The rule must be enabled in the PCB Rules and Constraints Editor dialog
- The rule type must be enabled for online checking in the Design Rule Checker dialog
- The Online DRC facility must be turned on.

The last requirement is fulfilled by enabling the Online DRC option on the PCB Editor - General page of the Preferences dialog (Tools » Preferences).

If any objects are found to be in violation of an applicable design rule that is enabled for Online checking, they will be highlighted in the main workspace. By default, the outlining color used is a conspicuous bright green.

Using Batch DRC

It should be noted that Online DRC only detects new errors - ones that are created after you turn it on. Batch DRC allows you to manually run a check at any time during the board design process. So while good designers know the value of the Online DRC, they also know that board design should begin and end with a Batch DRC.

When setting up a Batch DRC, various additional options can be defined by clicking on the Report Options folder, on the left side of the Design Rule Checker dialog. These options include generation of a report.

A batch DRC is initiated by clicking the Run Design Rule Check button, at the bottom left of the dialog. After the check has completed, all violations will appear listed as messages in the Messages panel.

If you have opted to create a report, it will automatically open as the active document in the main design window, upon completion of the check. The report lists each rule that was tested and each violation that was located is listed with full details of any reference information, such as the layer, net name, component designator and pad number, as well as the location of the object.
If the Create Violations report option is enabled, clearance, length and width errors will be highlighted on the PCB document.

Resolving Design Violations

Your board is routed, those last few modifications have been brought in from the schematic and the time has come to generate the final artwork and documentation required to turn the logical image into a physical entity. With care and attention and staged design rule checking along the way, running that final Batch DRC may yield only a few design violations, if any. However, there may be a sizeable quantity of violations flagged and now the task becomes one of resolving those violations.

Batch DRC reports can appear quite daunting to the new PCB Designer. The secret to keeping the process manageable is to develop a strategy. One strategy is to limit the number of violations that are reported. When setting up the report options in the Design Rule Checker dialog, set the Stop When Found feature to a small number. Another strategy is to run the DRC in a number of stages. If you find that the design contains a large number of violations, begin by enabling the rules one at a time. With experience you will develop a preferred approach to testing the various design rules.

Interrogating Violations

In order to effectively resolve a design violation, you must first be able to locate it. The PCB Editor provides three methods for interrogating design violations - from the Messages panel, from the PCB panel and directly within the design workspace. The first method is solely associated with having run a Batch DRC.

From the Messages Panel

After running a Batch DRC, double-clicking on a violation message in the Messages panel will cross probe to the object(s) causing that violation in the main design window.

From the PCB Panel

When running an Online or Batch DRC, any rule violations associated with a rule class or individual rule will be listed in the Violations region of the PCB panel, when the panel is configured in Rules mode. Clicking on a violation entry will apply filtering using the offending object(s) as the scope of the filter. The resulting view in the main design window will depend on the particular highlighting options enabled for the panel (Mask, Select, Zoom).

Double-clicking on a violation entry will open the Violation Details dialog, which provides information about the rule being violated and the primitive(s) responsible. From this dialog you can highlight the offending object (causing it to flash in the workspace) and jump to it, effectively providing zoom and center.

Directly in the Workspace

You can interrogate the violations associated with a particular design object directly within the PCB workspace. Simply position the cursor over the offending object you wish to interrogate, right-click and select Violations. In the example image, the offending track in the top-right corner (denoted by a yellow marker for ease of reference) is being investigated.
You can either choose to investigate individual violations associated with an object, or all violations. Choosing the former will cause the object(s) involved to be zoomed and centered in the main design window.

Irrespective of your choice, the Violation Details dialog will appear, providing further violation details and controls for highlighting and jumping to the offending object(s).

**Parting Advice**

To successfully complete a PCB design using Altium Designer’s PCB Editor, the constraints of the design should be thought out and implemented as a well-honed set of design rules. Remember that 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 success.

Verify your scopes! Too often a myriad of violations will result from a poorly scoped design rule. Checking individual rule scopes and priority levels can save much wasted effort – and more importantly time – later on.

Also, check your design at regular stages – not just at the end of the design process and five minutes before the Fab House requires the artwork. Using a combination of the Online and Batch DRC tools periodically will help to keep any violations to a more manageable number.

For further information on design rules, see the *Design Rules Reference*. This comprehensive reference includes information about each of the individual rule types and their associated constraints.
## Revision History

<table>
<thead>
<tr>
<th>Date</th>
<th>Version No.</th>
<th>Revision</th>
</tr>
</thead>
<tbody>
<tr>
<td>9-Dec-2003</td>
<td>1.0</td>
<td>New product release</td>
</tr>
<tr>
<td>26-Apr-2005</td>
<td>1.1</td>
<td>Updated for SP3</td>
</tr>
<tr>
<td>3-Mar-2008</td>
<td>1.2</td>
<td>Converted to A4</td>
</tr>
</tbody>
</table>

Software, hardware, documentation and related materials:

Copyright © 2008 Altium Limited.

All rights reserved. You are permitted to print this document provided that (1) the use of such is for personal use only and will not be copied or posted on any network computer or broadcast in any media and (2) no modifications of the document is made. Unauthorized duplication, in whole or part, of this document by any means, mechanical or electronic, including translation into another language, except for brief excerpts in published reviews, is prohibited without the express written permission of Altium Limited. Unauthorized duplication of this work may also be prohibited by local statute. Violators may be subject to both criminal and civil penalties, including fines and/or imprisonment. Altium, Altium Designer, CAMtastic, Design Explorer, DXP, LiveDesign, NanoBoard, Nexar, nVisage, P-CAD, Protel, Situs, TASKING and Topological Autorouting and their respective logos are trademarks or registered trademarks of Altium Limited or its subsidiaries. All other registered or unregistered trademarks referenced herein are the property of their respective owners and no trademark rights to the same are claimed.