

Summary

Underlying Altium Designer's schematic and PCB editors is a powerful query engine. By entering queries into this engine you can filter down to find and edit precisely those objects you require.

Perhaps the greatest challenge when working on a complex electronics design is managing the large number of design objects. From the components, net labels and text strings on the schematic sheets, through to the hundreds of components and tens of thousands of track objects that make up the routing on the PCB, as the designer you need to be able to access, review and edit these objects.

Like most Windows application you can of course double-click on an object and edit its properties. This is fine for a single object, but not something you would like to do if you need to edit the stroke width for the 300+ component designator strings, or change the annular ring for all the vias on the PCB. For this you need to access multiple objects simultaneously – to inspect them, to edit them, or apply a PCB design rule to them. Ideally you should be able to access different types of objects simultaneously too – the query-based filtering system lets you do precisely this.

Figure 1 shows a diagram of the Altium Designer filtering system, you can see how the Query Engine is used control the Filter Engine, allowing you to exactly control what objects are returned.

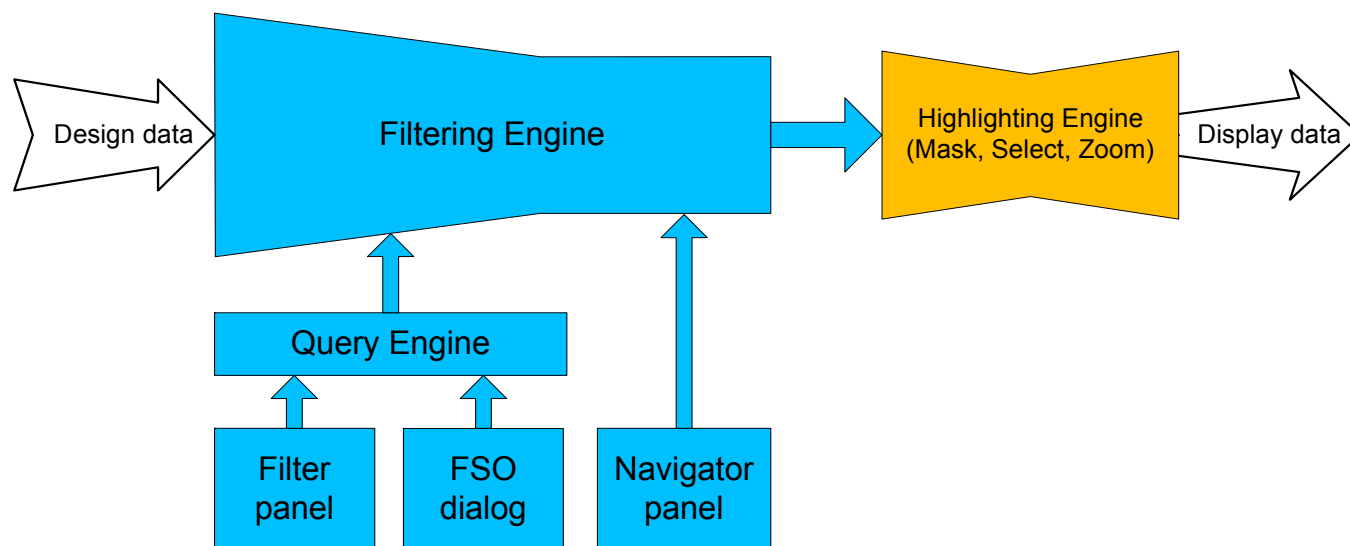


Figure 1. You write a query to filter your design data, which can then be viewed and modified.

What is a Query?

The powerful data filtering and editing system in Altium Designer lets you instruct the software to return you a specified set of objects. This instruction is entered in the form of a *Query*. A query is a string you enter using specific keywords and syntax, which will return the targeted objects. What you do with those objects is up to you. Perhaps you want to highlight them, dimming out all other objects. Or perhaps you want to browse or sort their properties, and modify specific attributes that they all share. There are a number of places where you can apply a query but command central is the **Filter** panel, as shown in Figure 2. Press **F12** to display/hide the **Filter** panel.

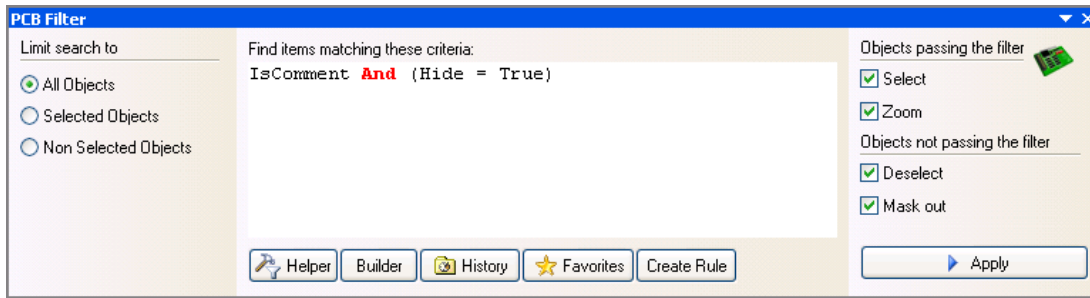


Figure 2. The PCB filter panel, the query shown will return all component comment strings that are hidden.

Note the controls on either side of the **Filter** panel, these are used to define which objects the query will be applied to, and how to handle the objects that pass the query as well as those that fail.

Where are the Query Results?

So you have written a query in the Filter panel, now how do you access the objects returned by your query? There are three ways that objects can be accessed and edited in the Altium Designer environment.

Graphically – after applying a filter the default behavior is to mask (fade and make non-editable) all objects that failed the query, leaving only those that passed. This mode is excellent for locating and reviewing objects in the workspace. Since the masked objects are not editable, commands like **Ctrl+A** (select all) only apply to the objects returned by the query.

List panel – the **List** panel (**Shift+F12**) is a spreadsheet-like view into your design data. By default it displays all design objects, once a filter has been applied it displays only the objects returned by the query. The List has standard spreadsheet-like behaviors for sorting and selecting, allowing you to edit single or multiple cells directly. Note that the top of the List panel has a number of controls, press **F1** when the cursor is over the panel for help with these controls and other general panel behavior.

Inspector panel – where the List presents the objects' properties in a table, the Inspector collates the common attributes of the selected objects into a single column. Note the total number of selected objects is displayed at the bottom of the panel. After editing the value in a cell press Enter to apply the change. Press **F1** when the cursor is over the Inspector panel for more information.

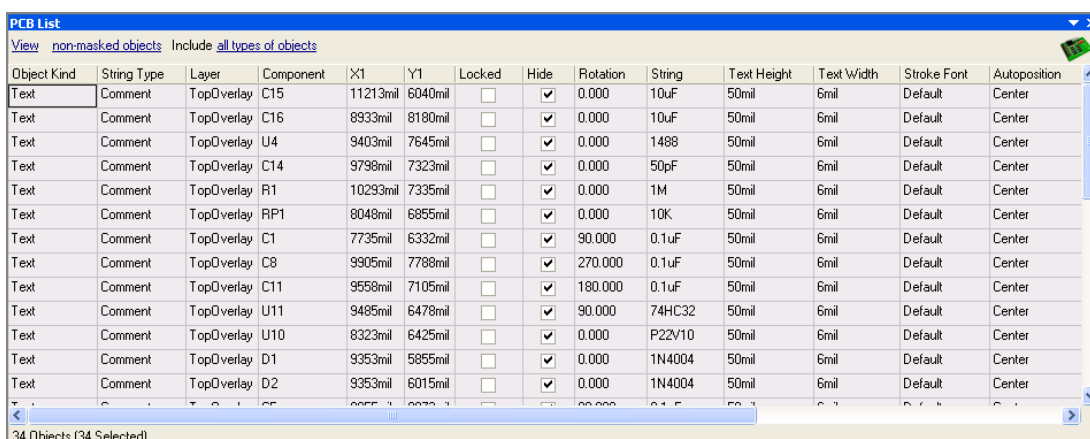
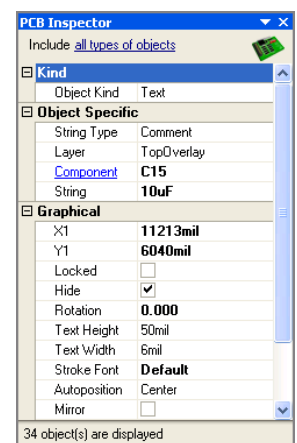


Figure 3. The List and Inspector panels, display the result set from the query applied in Figure 2.

The first thing you will want to know after applying a query is "how do I get rid of it, I want the rest of the objects back!"

Click the **Clear** button at the bottom right of the workspace, or press the **Shift+C** shortcut to clear the current filter.

Building your Query Writing Skills

How can you learn to write queries? Like anything new, it takes practice. Altium Designer includes valuable resources for learning the query language. The first is a set of example queries that are included as part of the installation. The second is a group of tools that generate queries based on your input. The third is the Query Helper, which lists all query keywords and helps you build up a query. The final source in learning the query language is in the online help.

Example and Favorite Queries

Both the schematic and PCB editors come with a set of pre-packaged example queries. To try these, press the **Y** shortcut key to display the **Filter** popup menu, where you can select the **Examples** sub-menu. When you select an entry in the menu the query behind that filter is applied (press **Shift+C** to clear it). To view the actual query behind one of the examples look in the History list after applying it, you can access this list in the **Filter** popup menu too.

You can also keep favorite queries and re-use them later. Favorites are stored in the *Expression Manager* dialog – there are a number of ways to access this dialog, via the **Favorites** button in the Filter panel, or by selecting the **Organize Favorites** menu entry after pressing the **Y** shortcut key. The great thing about favorites is that they are automatically displayed in the **Filter** menu, giving you instant access to up to ten favorite queries.

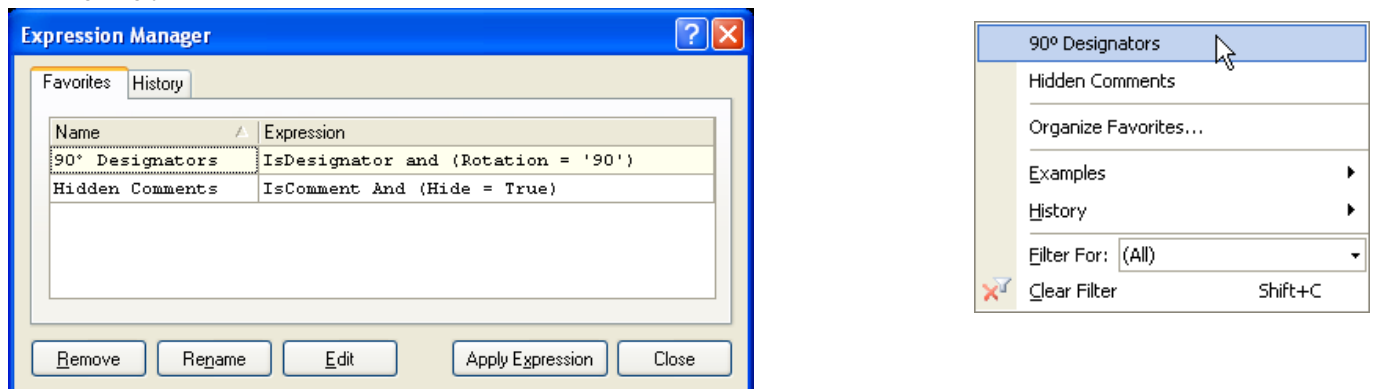


Figure 4. Store queries you use regularly as Favorites, then access them in the Filter menu.

 You can also define a shortcut key for a favorite by customizing the **Filter** menu. For more information on customizing the Altium Designer environment, refer to the [Customizing the Altium Designer Resources](#) tutorial.

Query Building Tools

Perhaps the easiest way to write a query is to get Altium Designer to write it for you! There are two ways of doing this, using the *Find Similar Objects* dialog, or the *Query Builder*.

Find Similar Objects dialog

When you use the *Find Similar Objects* dialog it generates a query to find the required objects. If the **Create Expression** option is enabled then that query will be displayed in the Filter panel. This is an excellent technique for learning different query keywords.

The Find Similar Objects dialog appears when you right-click on any unmasked object in your design document and select **Find Similar Objects** from the context menu. The idea with this dialog is that it lets you find objects similar to

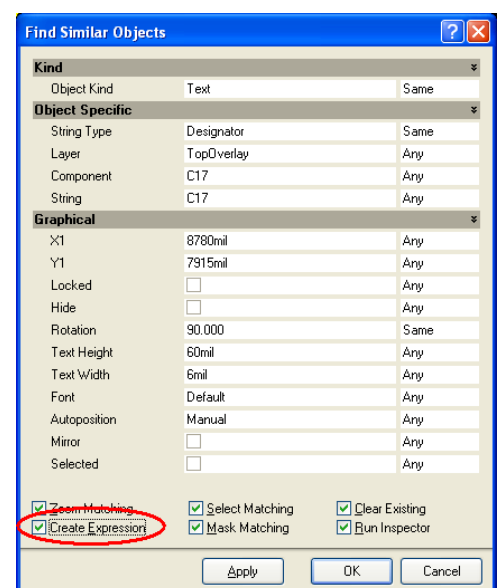


Figure 5. Using FSO dialog to write the query for you.

the one you right-clicked on, where you define which of the object's attributes that must be the same (or different) for a match. Suppose for example you wanted to change all GND pads in your design. You could right-clicking on one such pad, choose **Find Similar Objects**, then change the **Net** field from **Any** (the default setting) to **Same**. All of the GND pads will be selected when you Apply from this dialog, and if the **Select Matching** option is enabled you can use the **Inspector** to make the change across the entire selection. If the **Create Expression** option is enabled in the *Find Similar Objects* dialog then the following query would appear in the Filter panel:

```
(ObjectKind = 'Pad') And (Net = 'GND')
```



For more information on using the *Find Similar Objects* dialog, see the [Editing Multiple Objects](#) tutorial.

The Query Builder

The PCB editor also includes a dedicated Query Builder. This tool lets you build complex queries by choosing test conditions from drop-down lists. An advantage of the Query Builder is that it lets you create a query that targets different kinds of objects, for example Figure 6 shows the Builder being used to create a query that targets both pads and vias on the -12V net.

The Query Builder can be launched from a number of places. Click the **Builder** button in the **Filter** panel to build a query and the string loaded into the panel. Click the **Builder** button when you are editing a design rule to build a query that defines which objects this rule must apply to.

You can also call up the Builder directly in the design workspace, either by pressing the **Shift+B** shortcut keys, or right-clicking in the workspace and choosing **Build Query** from the context menu. When you run the Builder in the workspace it will look different from when it is run from the Filter panel or Rules dialog, in the workspace it applies the query immediately so it includes display-type controls, whereas from the Filter panel or Rules dialog it creates the query string instead.

Note also that the right-click option is context sensitive, the options available will change depending on what object you right-clicked on. Running the Builder from the Design Rules dialog is also context sensitive, it will only give you options that are suitable for that rule kind.

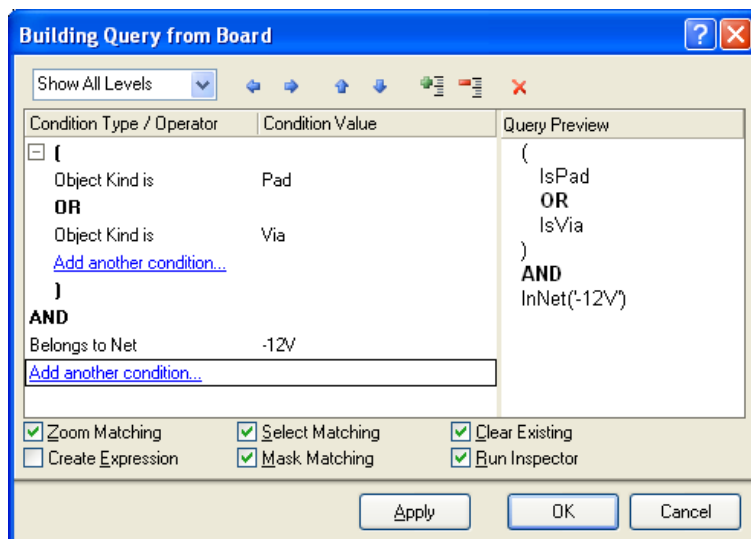


Figure 6. Use the Query Builder to generate a query for you.



For more information on using the Query Builder press F1 over the dialog, or use the What's This Help button.

Query Helper

The next step in building your query-writing skills is to use the *Query Helper*. Accessed via the **Helper** button in the Filter panel, the *Query Helper* includes a query editing box at the top, syntax buttons like **And** and **Or** in the mid-region of the dialog, as well as a complete list of all query keywords below that.

Browse through the different categories of available keywords for the one you want. Use the **Mask** field if you are not sure what the exact keyword is. For example, in the schematic editor's *Query Helper*, entering *para in the **Mask** field will show you the dozen or so keywords that apply to parameters specifically, while *par will pick up keywords for both parameters and parts. Note

that the **Mask** field works on both the keyword name field and the description field, so it can be the quickest way to find possible keywords.

If you press **F1** when a keyword is highlighted or the cursor is within a keyword you have typed in, an online help description for that keyword will open. This is the most valuable resource for learning the basic behavior of each query keyword. Here you will learn the nuances of that keyword, such as keywords that return children objects, parent objects or both.

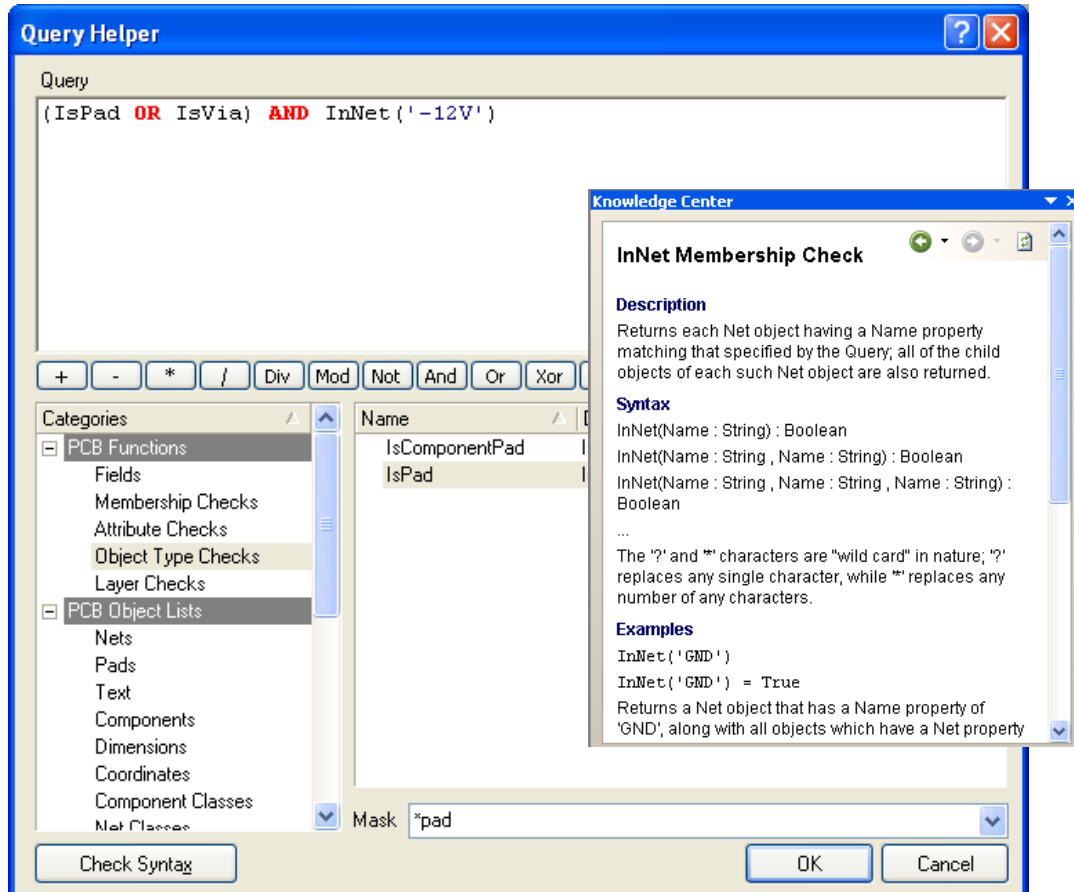



Figure 7. Use the Query Helper and the associated keyword online help to learn more about queries.

Where to Now?

Queries may seem intimidating at first, but your efforts to learn them will pay off with the improved productivity that they bring. They provide a powerful method of accessing objects in your design.

 For a more detailed description of the Query language as well as numerous example queries read the article [An Insiders Guide to the Query Language](#).

Revision History

Date	Version No.	Revision
9-Dec-2003	1.0	New product release
13-Apr-2005	1.1	Review and updated for Altium Designer
28-Nov-2005	1.2	Review and updated for Altium Designer 6.0
14-Mar-2008	1.3	Page Size updated to A4.
09-Aug-2011	-	Updated template.

Software, hardware, documentation and related materials:

Copyright © 2011 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, Board Insight, DXP, Innovation Station, LiveDesign, NanoBoard, NanoTalk, OpenBus, P-CAD, SimCode, 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.