Editing Multiple Objects

Summary

Tutorial

TU0115 (v2.2) November 29, 2005

This document describes various techniques for applying edits globally to multiple objects in your design. It covers using the Find Similar Objects dialog and Inspector panel combination, as well as the Parameter Manager and the Model Manager. Finally, it introduces queries and the List panel, a powerful technique for finding and editing design objects.

Electronic design is the process of capturing a logical design in the schematic, then representing that design as a set of objects in the PCB workspace. Even for a small circuit, the schematic can include many components, each with numerous models and parameters, and the PCB workspace will end up containing a large number of design objects that make up the board. During the course of the design process, the properties of these objects will need to be changed as the designer works to balance out the various design requirements.

To support the task of editing many objects, previous versions of Altium design tools included a feature called Global Editing. The basic approach of this feature was to edit one object, and then push those changes onto other objects.

With the introduction of the DXP platform, the technique for applying an edit globally changed. The basic approach now is to *select* the objects to be edited, *inspect* their properties, and then *edit* them.

Keeping this *select – inspect – edit* sequence in mind, let's look at each step in detail.

Selecting multiple objects

There are actually a number of ways of selecting objects, for example the Windows standard mouse click shortcuts can be used. This approach is ideal when the number of objects to be selected is small, or perhaps when there are different kinds of objects to be edited simultaneously.

To select many objects, including over a number of schematic sheets, you use the *Find Similar Objects* dialog. To open this dialog, right-click on one of the objects to be edited, and select **Find Similar Objects** from the context menu.

Let's walk through the process using an example. Let's say we need to change the name of a power net in the schematic from VCC to 3V3. This requires all the VCC power ports on all of the sheets to have their Net attribute changed. The first step is to find a VCC power port on the schematic, right click on it, and select **Find Similar Objects.**

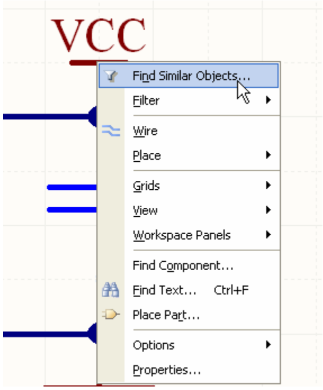


Figure 1. Right click and choose Find Similar Objects

Figure 1 shows the *Find Similar Objects* dialog after right-clicking on a schematic Power Port. It is important to note that the dialog lists the properties of the object you clicked on, so the contents of this dialog will be different if you clicked on something else.

You can see that the dialog has two columns, the column highlighted in Figure 2 shows the current properties of the object you clicked on – down the bottom you can see that the net name Text is currently VCC.

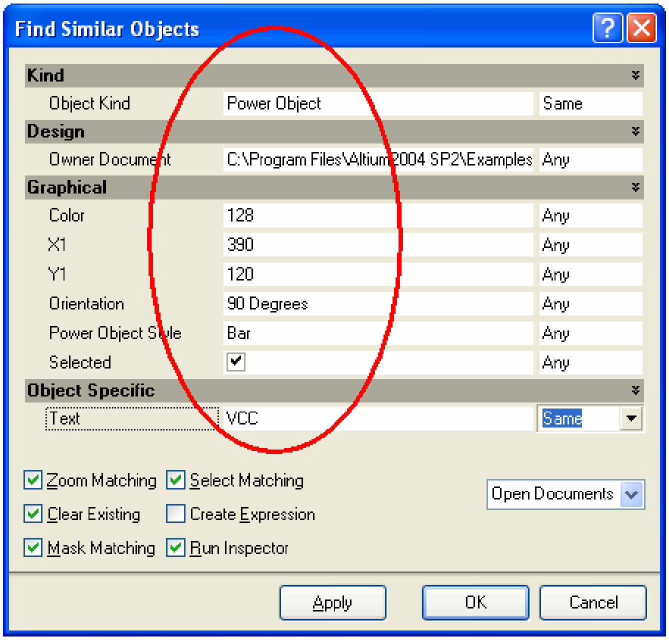


Figure 2. Properties of the current object loaded into the Find Similar Objects *dialog*

The second column in the *Find Similar Objects* dialog is where you instruct it how to match other objects. For each property of the object, you can instruct it to match target objects when this property value is the *Same*, match when the target has a *Different* value, or set it to *Any* when you are not interested in matching by this property.

Note that in Figure 3, the matching will occur when the **Object Kind** is the *Same*, and when the net name **Text** is the *Same.* Or to say that another way, match when the object is a Power Object with a net name of VCC.

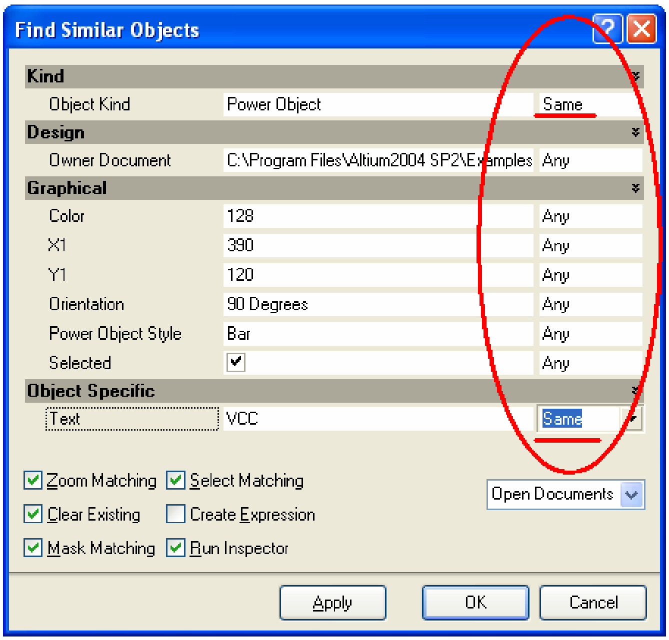


Figure 3. Which properties should be used to match by?

The next step is to set the scope of the Find action, should it be on the Current Document only, or all Open Documents. In Figure 4, you can see that this has been set to **Open Documents**. For this editing action to apply to all the sheets in the project, they must be opened first.

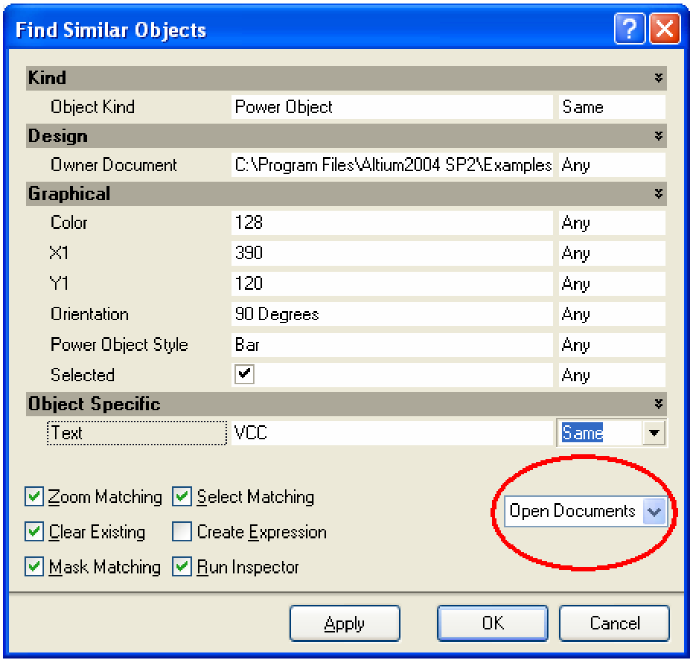


Figure 4. What documents should this edit apply to?

The final step is to define what should happen after it has found all the Power Objects that have net name Text of VCC, in all Open Documents.

Figure 5 shows the important settings for this edit operation. The highlighted options are **Select Matching** (to *select* all the VCC power ports), and **Run Inspector**, which will open the Inspector panel with the selected objects loaded into it.

Click the **OK** button to select the matching Power Ports.

The Apply button will also select the matching power ports and open the Inspector, but the *Find Similar Objects* dialog will remain open too – use this if you are not sure you have your matching criteria correct.

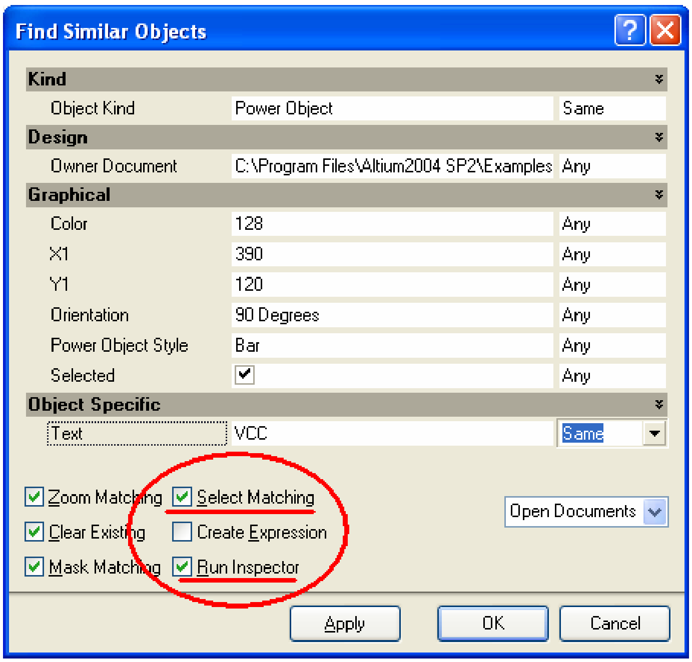


Figure 5. What should be done with the found objects?

Inspecting the objects

Both the schematic and PCB editors include a panel called the Inspector. The basic behavior of the Inspector is that it lists the properties of all objects that are currently selected. The set of selected objects could be the same kind of object, for example Figure 6 shows the properties of ten power ports.

Properties that are identical for all the selected objects have their value displayed, for example all ten power ports currently have an Orientation of 90 degrees.

For each power port property that has a different value, you will see <…> instead (e.g., the X1 location). This means that not all these ten objects have the same X1 value, which makes sense since they each have a different location.

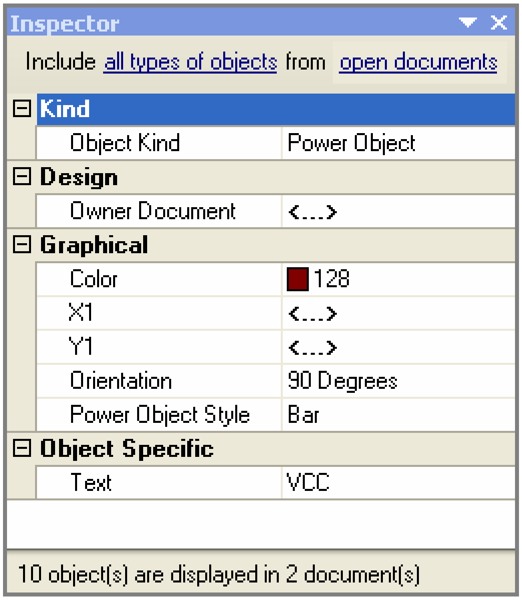


Figure 6. The Inspector shows the properties of the selected objects

Note in Figure 6 that the Inspector includes two options at the top. It is important that you set the second of these, which sets the display of where the found objects are from – from the *current document*, *open documents*, or *open documents of the same project*. To have all the selected power ports loaded into the Inspector, you will need to set this to *open documents* or *open documents of the same project*.

What is the Inspector?

The Inspector is a panel that displays the properties of whatever is currently selected. This could be one object, or many objects. If more than one object is selected, only properties that are common to all selected objects will be listed. Common properties that have the same value will show that value, otherwise the value will display <…>. When you type a value into the Inspector and press **Enter** on the keyboard, the value of that property is immediately changed for all selected objects.

The Inspector has certain characteristics that make it very handy for everyday use.

The first is that because it is a panel, it can be visible all the time so you do not need to double-click to open a dialog. This means that you can click to select any object in the workspace and its properties will be displayed immediately. This can be much more efficient if you are reviewing settings in your design. For example, you might want to check the designator text height of a few components on the PCB. If the Inspector is open, you simply click on a designator, read the value, click on the next one, read the value, and so on. This would be much faster than double-clicking on one designator, reading the height, closing the dialog, double-clicking on the next designator, and so on.

The second advantage of the Inspector is that it can display the common properties of different objects, and let you edit them. We'll see how this can be useful later in the tutorial.

Note that at the bottom of the Inspector the total number of selected objects is displayed, always check this and confirm that it is what you expect.

Editing the objects

OK, so far you have *selected* the objects you want to edit, *inspected* the properties in the Inspector, so now you are ready to *edit* them.

When you click to edit the net name Text, a pair of curly braces appear. These are needed when you want to perform a partial string substitution. For this edit, we will be replacing all the text, so we simply replace the entire contents of the cell with the new text, 3V3.

The change you make to the Text value is applied to all the selected objects as soon as you press **Enter** on the keyboard, or cl. T on another cell in the Inspector.

If you change your mind during the edit, press the **Esc** key on the keyboard to abort the edit. To Undo an edit that has been applied, select **Edit » Undo** from the menus. If the edit has been applied to multiple schematic sheets, you will need to perform an Undo action in each sheet.

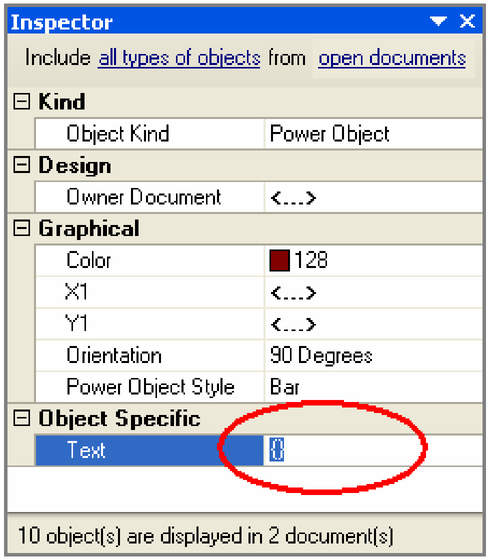


Figure 7. Editing the net name Text

Figure 8 shows the Inspector after changing the text and pressing **Enter**, next to one of the edited power ports.

You can use this approach to apply an edit globally to any type of object in the Schematic or PCB editors.

After performing the edit, you will probably find that all the other objects on the schematic are faded out, or masked. While something is masked it cannot be edited, to remove the mask click the Clear button at the bottom right of the workspace (shortcut, **Shift+C**).

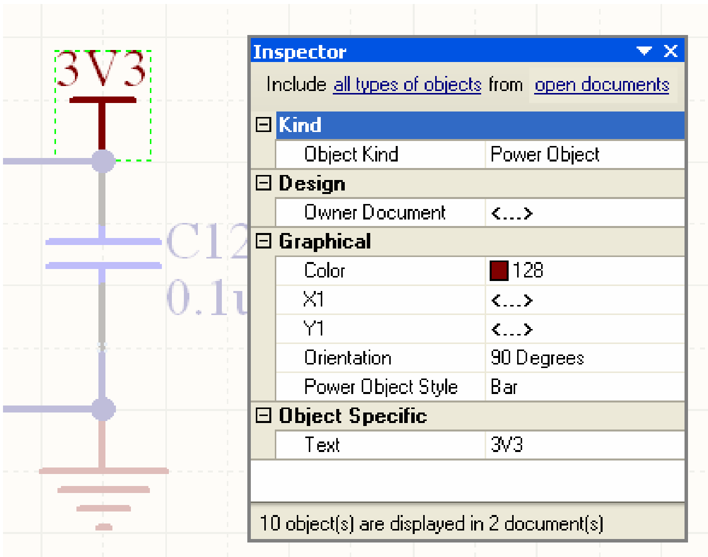


Figure 8. One of the ten updated power ports

Editing group objects

The edit that we just performed was on a primitive object, that is, one of the basic objects used in the Schematic editor. More complex objects such as components are called group objects; these are essentially a collection of primitive objects. For example, a component on a schematic is a collection of drawing objects, strings, parameters, pins, and references to models. The primitive objects that belong to a group object are sometimes referred to as the *child* objects, and the group object is their *parent* object.

Let's look at an example of a typical group object edit that you might want to perform. Your design includes a number of 100uF 16V capacitors, using the footprint CAPPR2-5x6.8. Currently the voltage is specified as part of the components' comment string. You need to change this and specify the voltage as a parameter of the component instead, and make this parameter visible on the schematic.

The steps we need to perform are:

1. Select capacitors, that have a value of `100uF 16V`, and a footprint of `CAPPR2-5x6.8`.
2. Change their comment to be `100uF` (remove the 16V text).
3. Add a new parameter to these components, with a name of Voltage, and a Value of `16V`.
4. Change the visibility of this parameter so the voltage is displayed on the schematic.

While this might seem a complex set of edits to perform in one go, it is actually quite straightforward.

Step 1. Selecting the capacitors

Firstly, to select all the capacitors, right-click on the component symbol of one of the 100uF 16V caps, and choose **Find Similar Objects** from the menu.

We use the approach covered in the previous example, except this time you want to match on components that have the same Part Comment, and the same Current Footprint, as shown in Figure 9.

Note that we can also match on components that have a designator starting with the letter C. This is done by changing the component designator value in the *Find Similar Objects* dialog from what it opened as, to `C*`, as shown in Figure 9.

Click the **OK** button to select the matching capacitors.

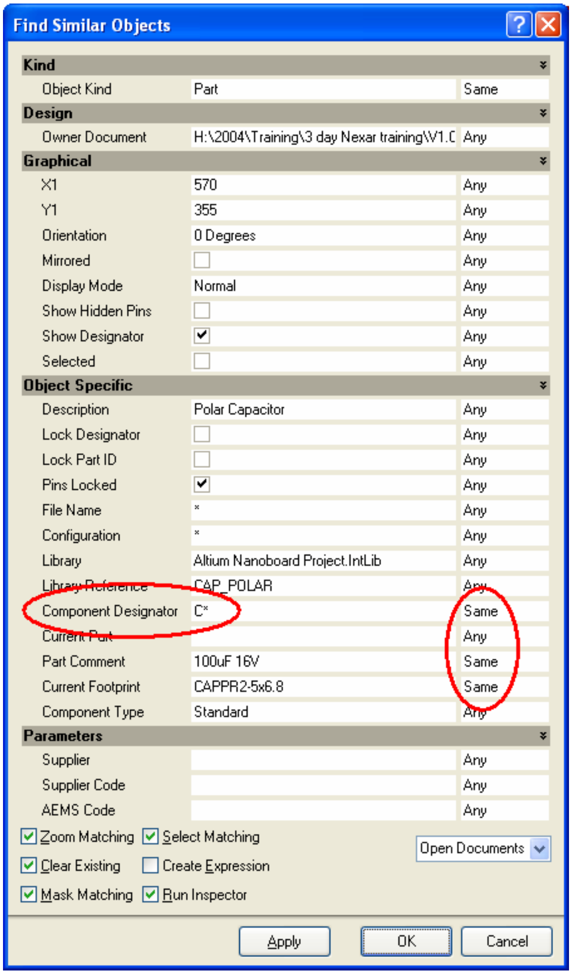


Figure 9. Finding the 100uF 16V capacitors

Step 2. Changing the Comment string

After running the *Find Similar Objects* dialog, the Inspector will open. Behind it will be the schematic sheet displaying the matching objects selected on that sheet. If the **Zoom Matching** and **Mask Matching** options were enabled in the *Find Similar Objects* dialog, then the view would be zoomed and all the objects that did not match would be faded, or masked out.

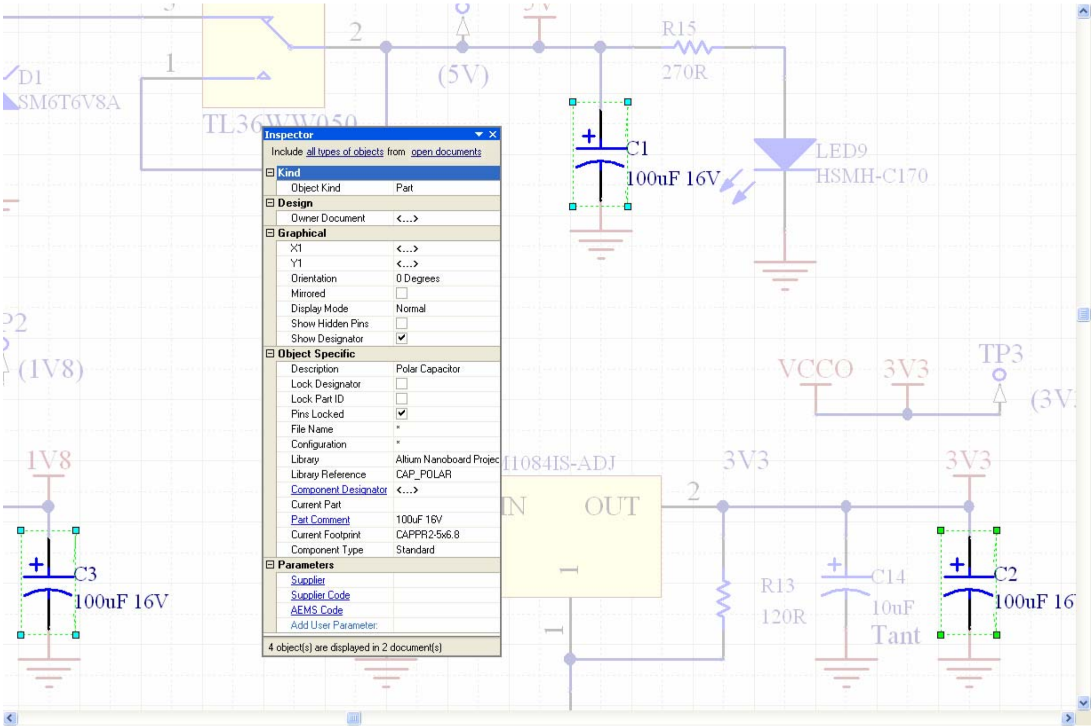


Figure 10. The view after performing a Find Similar Objects, showing the matching three capacitors found on this sheet.

Figure 10 shows the results. There are three capacitors found on this page, and from the status line of the Inspector, we can see that there is a fourth capacitor found on another document.

To change the comment string, simply delete the 16V from the string, and press **Enter** to apply the change, as shown in Figure 11.

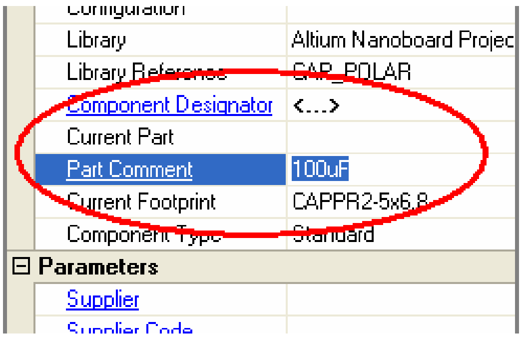
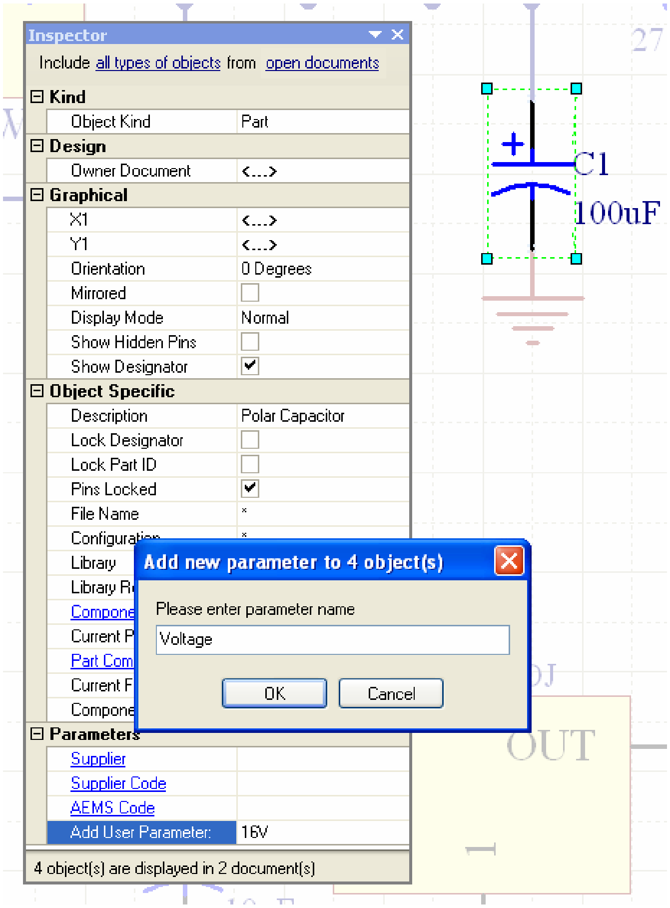


Figure 11. The capacitor value has been changed

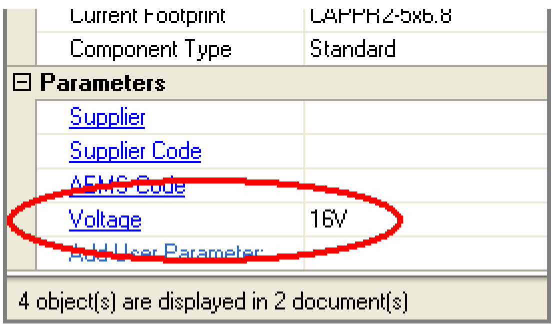
Step 3. Adding a new Parameter to the component

The next change that we need to make is to add a new parameter to these four components, called Voltage, and set the value to 16V. To do this, we use the Add User Parameter feature at the bottom of the Inspector. Note that we will enter the value first, then the parameter name.

1. Firstly, type in the value of the new parameter, 16V, into the **Add User Parameter** field in the Inspector.
2. Press **Enter** to apply the change. When you do, the *Add new parameter to XX objects* dialog will appear.
3. Type in the new parameter name and click the **OK** button.



The Inspector panel will now include the new Voltage parameter in the list at the bottom, with a value of 16V. You can add as many parameters as you wish using this approach.



Step 4. Setting the Voltage parameter to be visible

The last step is to make the new Voltage parameter visible on these four capacitors. The visibility of a parameter is a property of the parameter itself, not the component, so we cannot change this in the Inspector yet because it is displaying the properties of the parent components.

To access the properties of the child parameters, click on the hyperlinked Parameter name, Voltage, in the Parameters list at the bottom of the Inspector. When you do, the Voltage parameter properties for the selected components will be loaded into the Inspector, ready to edit. You can confirm this by checking the Object Kind at the top of the Inspector – it should say 'Parameter'.

Now we can make the Voltage parameter visible on the schematic.To do this' uncheck the **Hide** checkbox, as shown in Figure 12.

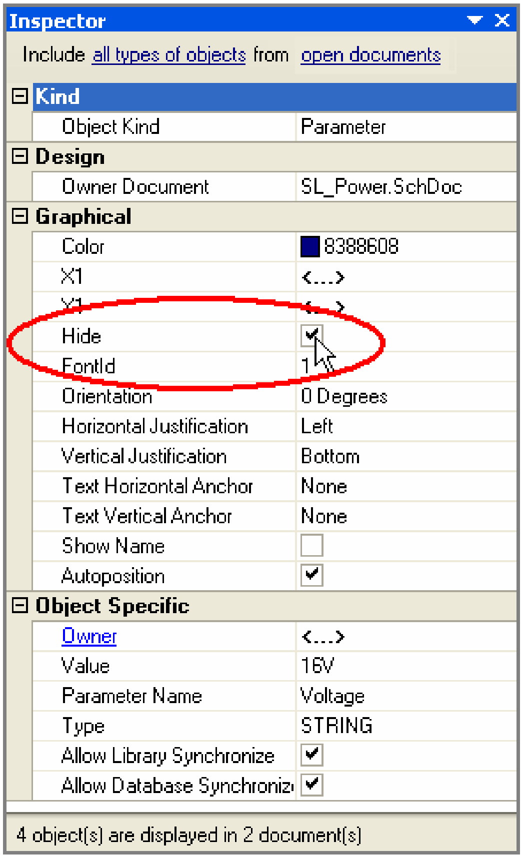


Figure 12. Change the visibility of the new parameter

If you wanted to return to the parent components, perhaps to edit some other property, you would do this by clicking on the Owner hyperlink, as shown in Figure 13.

We have now updated the comment string for all 100uF capacitors, using a CAPPR2-5x6.8 footprint. We have also added a new parameter called Voltage, set its value to 16V, and made this parameter visible.

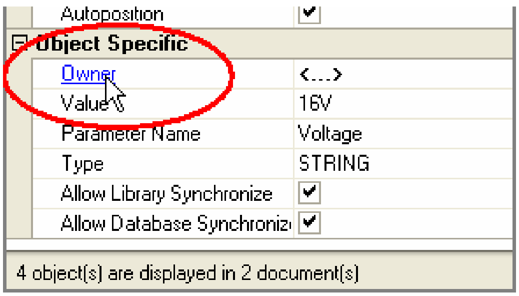


Figure 13. Returning to the parent component properties

Applying an edit to different types of objects globally

Not only can the **Inspector** panel be used to edit multiple instances of the same object, it can also be used to edit common properties of different objects.

Changing the net name for existing routing

For the first example, let's assume that you have made design changes on the schematic, removing a pin from one net and adding it to another. If the nets were already routed on the PCB, then when you update the PCB, you could end up with routing that has the wrong net name. This routing could include tracks and vias, as well as other kinds of objects.

There are a few ways this could be resolved. The easiest is to use the Inspector. Let's go through the process now.

1. In the PCB, you would select all the primitives in the routed net that needs its name changed, using the **Edit » Select » Connected Copper** command (shortcut, **Ctrl+H**).
2. If it is not already visible, you would then display the Inspector (shortcut, **F11**).
3. The Inspector will only show properties that are common to all the selected objects. If you selection was correct, one of these will be the **Net** name. To change this, simply select the new net name from the drop down list and press **Enter** to apply the change. The net property of all the different objects in the routed net will be changed.

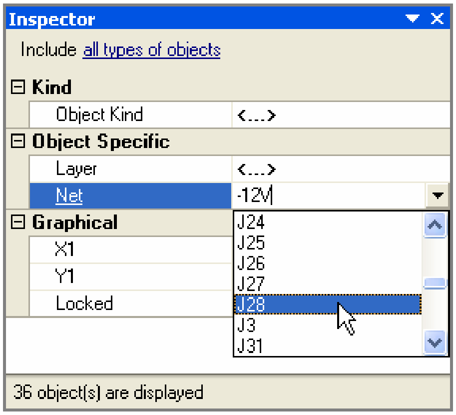


Figure 14. Changing the net name of selected tracks and vias

Changing the layer property of different objects

Another example might be that you need to move all the objects that are on one mechanical layer to another mechanical layer. To do this, you would:

1. Click the **Layer** tab for the current mechanical layer at the bottom of the PCB editor window to make it the active layer.
2. Select all the objects on that layer using the **Select » All on Layer** command (shortcut, **S, Y**).
3. If it is not already visible, display the **Inspector** (shortcut, **F11**).
4. Select the new Layer name in the drop down list, and press **Enter** to apply the change.

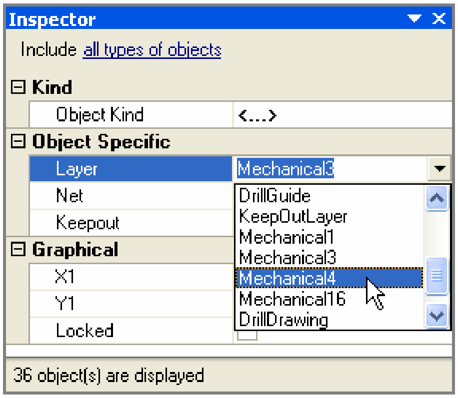


Figure 15. Changing the layer for selected objects

Editing multiple parameters using the Parameter Manager

User-defined design attributes are added to your design using parameters. Component parameters can be used to define anything from component ratings, to stock information, to PCB component class membership. You can even include links to component datasheets as a parameter. Parameters can be added to nets to specify PCB design requirements, or to include the net in a PCB net class. Document parameters can be used to define things like the title of the sheet, the designer's name, and so on.

Parameters can be added and edited individually, or you can use the *Parameter Table Editor* to add and edit them across the entire design, or across an entire library. When you open the editor, it gathers all parameter data for the entire design and presents it in a table-like editing grid. The *Parameter Table Editor* is launched by selecting **Parameter Manager** from the **Tools** menu.

After selecting Parameter Manager from the menus, the *Parameter Editor Options* dialog appears first. In this, you determine which type of parameters you want loaded into the *Parameter Table Editor*.

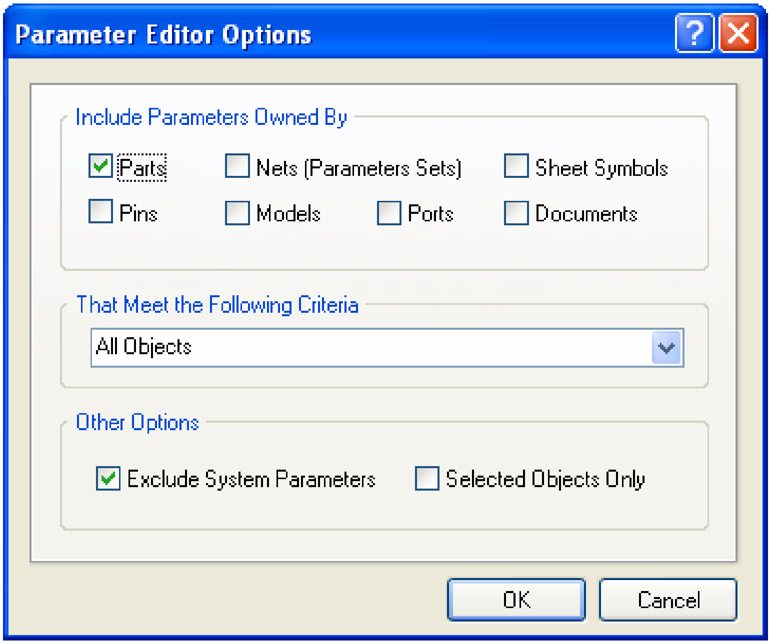


Figure 16. Choose which types of parameters that you want to edit

If you were working on component parameters you would disable all check boxes in the **Include Parameters Owned By** section, except for the **Parts** check box. If you wanted to work on document parameters, you would only enable the **Documents** check box. Note the **Exclude System Parameters** check box. System parameters include things like component model settings, document parameters that were defined in the template, and so on. Explore this option when you are more familiar with managing parameters.

Let's do some parameter editing now. The following descriptions and images are based on the `4 Port Serial Interface` reference design example. After selecting **Tools » Parameter Manager**, configure the *Parameter Editor Options* dialog as shown in Figure 16.

Renaming a parameter

In Figure 17 below, you will notice that one of the existing parameters is called ‘Text Field1’. This needs to be renamed. ‘Component Type’ would be a more suitable name.

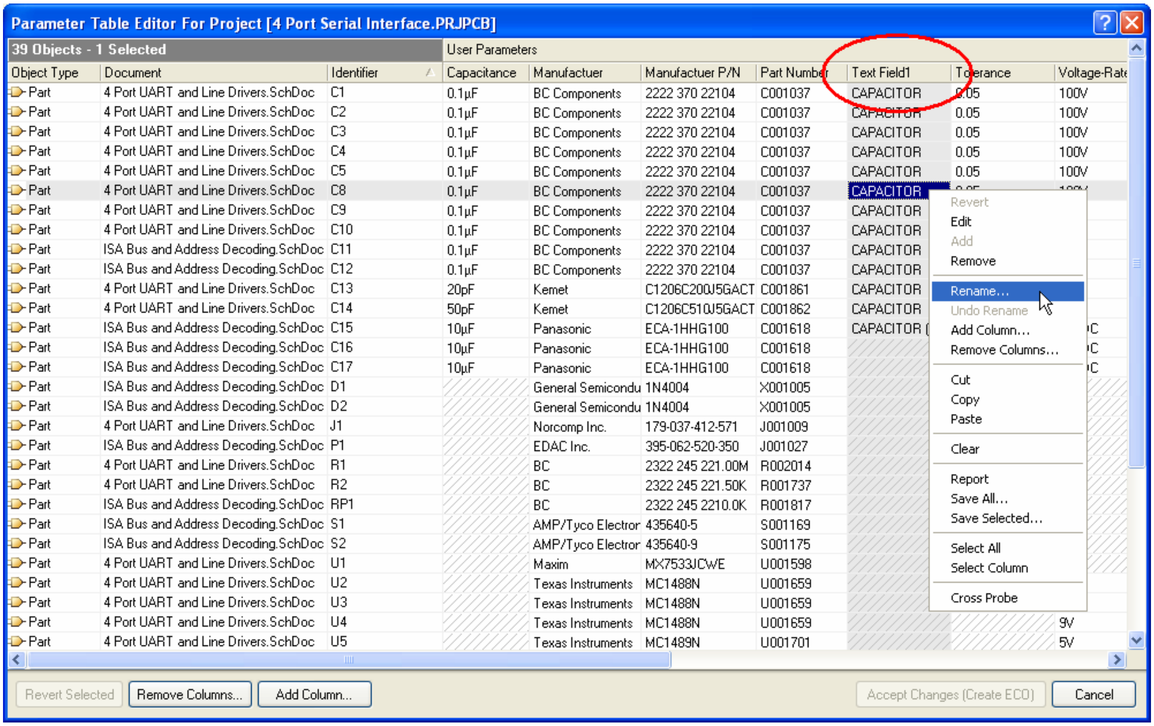


Figure 17. Using the Parameter Table Editor to rename an existing parameter

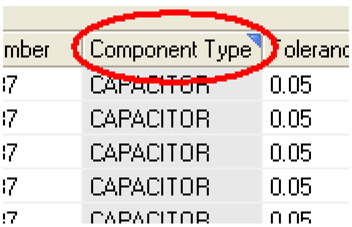


Figure 18. The renamed parameter

To rename a parameter, right-click in any cell in that column and in the Context menu that appears, select **Rename**. The *Rename* dialog will open, so type in the new name and click **OK**. Note that the column heading will have changed and now has a small blue triangle next to the name (as shown in Figure 18). This icon indicates that the value of this cell has changed. For complete details on the various icons used in this editor, press F1 when the cursor is anywhere over the dialog.

You will also notice in Figure 17 that some of the components do not have a Component Type parameter at all – this is indicated by the diagonal hatching. The next step is to add the Component Type parameter to all the other components.

Adding a parameter

To add a parameter to components that do not currently have it, select those cells in the editor using the **Shift+click** or **Ctrl+click** key combinations. Then right-click and choose **Add** from the context menu.

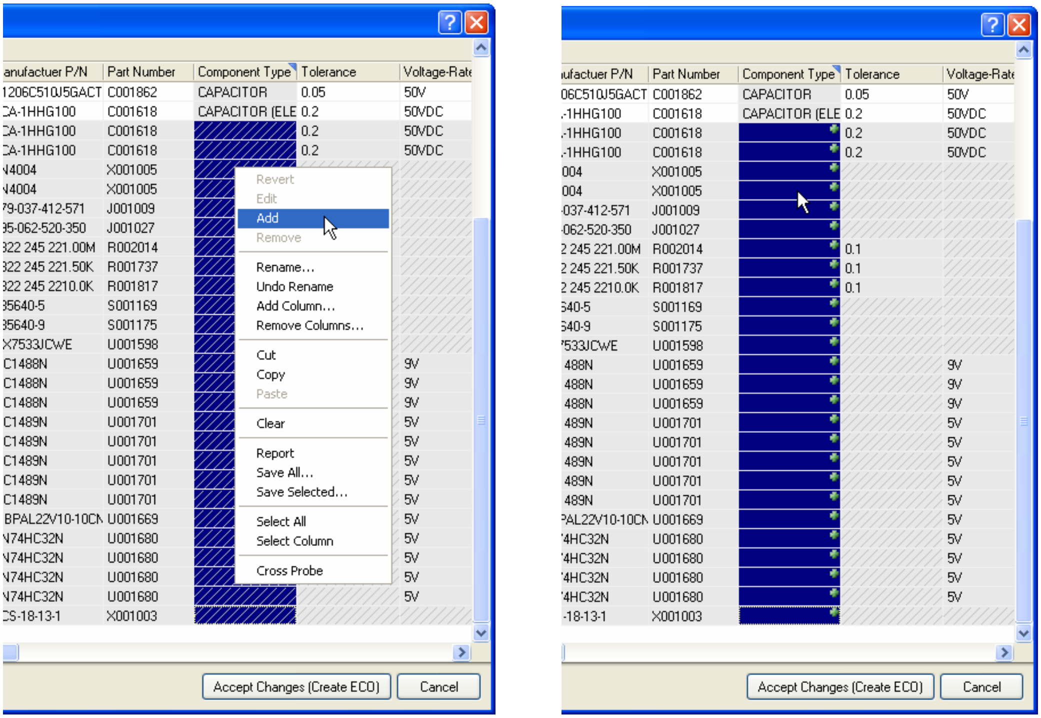


Figure 19. Adding this parameter to the selected components, before adding on the left, and after shown on the right

After selecting **Add**, you will notice that a small green plus symbol appears in each cell. This indicates that a new parameter has been added.

Now that the parameter has been added, you can define the component type for each component. The *Parameter Table Editor* supports standard table editing shortcuts. Use the cursor keys to 'walk' around the grid, press **F2** to edit a cell, and press **Enter** to apply the edit. Multiple cells can be edited in one go – select the cells, right click on the selection and choose **Edit** from the context menu, type in the new value, and press **Enter** to apply the edit to all selected cells.

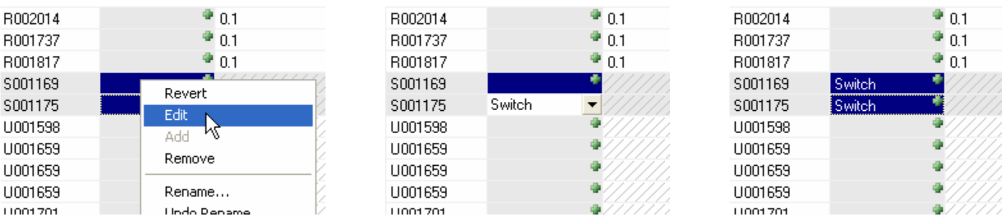


*Figure 20. Select the cells, right click and **Edit** (left), type in new value (center) and press **Enter** (right)*

Applying the parameter changes

The parameter edits that have just been carried out are currently held in the Parameter Table Editor and they have not been applied to the components on the schematic sheets yet. To apply these changes to the components, you need to generate an ECO (Engineering Change Order) and then apply the ECO to the design.

When you are satisfied with your parameter edits, click the **Accept Changes (Create ECO)** button. The Parameter Editor Table will close and the *Engineering Change Order* dialog will appear.

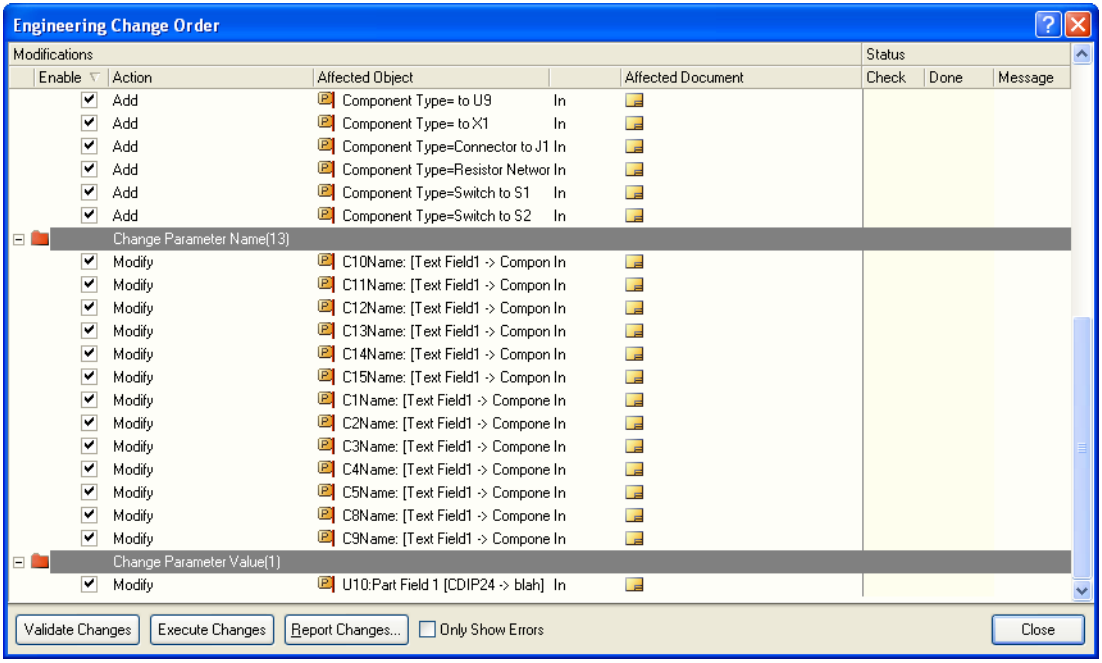


Figure 21. System applied changes are always done through the Engineering Change Order dialog

Click the **Validate Changes** button to check that the changes can be applied, then click **Execute Changes** to apply the parameter changes to the components. Once the changes have been applied, close the *Engineering Change Order* dialog.

Managing multiple component models

The schematic symbol represents the component on the schematic. The wiring then connects the component pins to create the connectivity. While this creates the schema, or the inter-connective structure of the design, other information is required to translate that into the final physical PCB.

The ability to translate the original schema into other forms, such as a PCB layout, or perhaps a circuit simulation description, is provided by the models that you attach to each component.

Different model kinds are supported, including PCB footprints, spice simulation models, signal integrity analysis models, and 3D models. While these can be defined on the schematic sheet, they are typically defined in the component library. For an individual component, it is straightforward to add a model to a component. You can add them in the model editing region at the bottom of the main schematic library editing window, as shown in Figure 22.

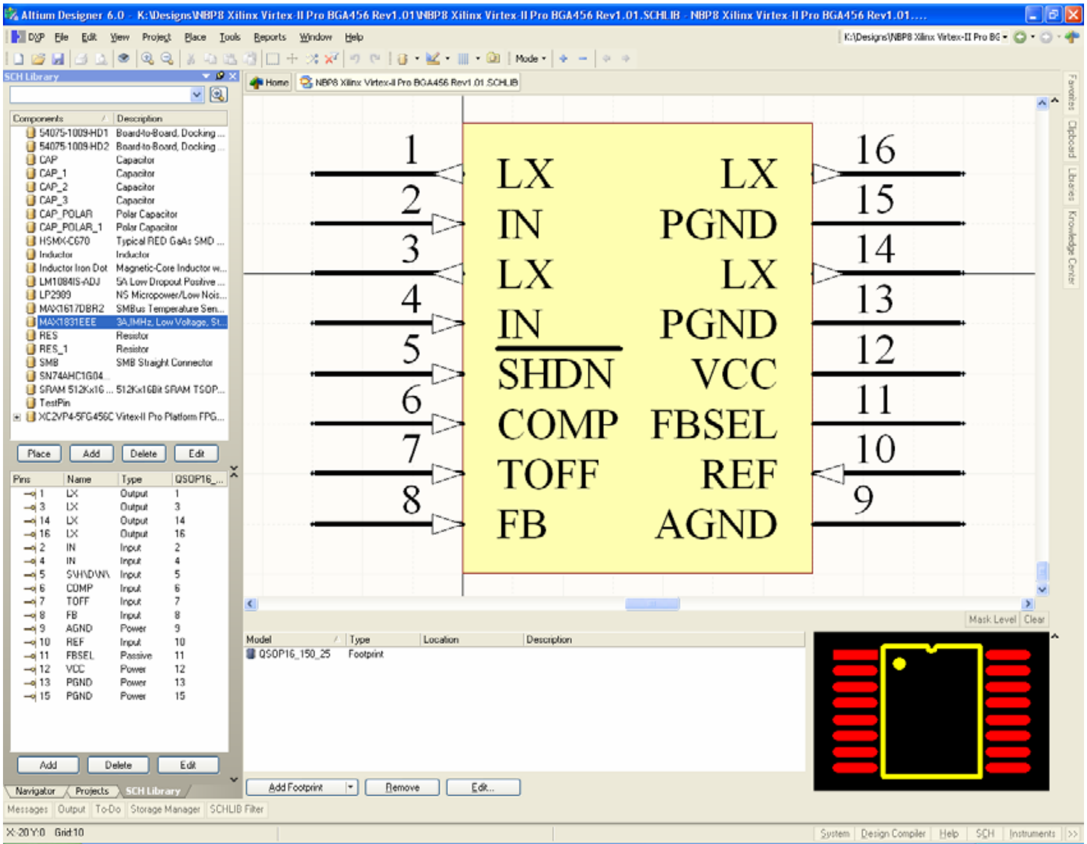


Figure 22. The Schematic Library editor, with the model editing region displayed at the bottom of the main window

For more information on creating library components and attaching models, refer to the *Creating Library Components* tutorial.

For a better understanding of component models, refer to the *Component, Model and Library Concepts* article.

Editing Multiple Objects

To add or edit model settings across multiple components, the Library editor includes a Model Manager. To open the Model Manager for the current library, select **Tools » Model Manager** from the menus. The *Model Manager* dialog will open, displaying the components in the current library down the left, click to select a component and display a list of the models currently associated with that component.

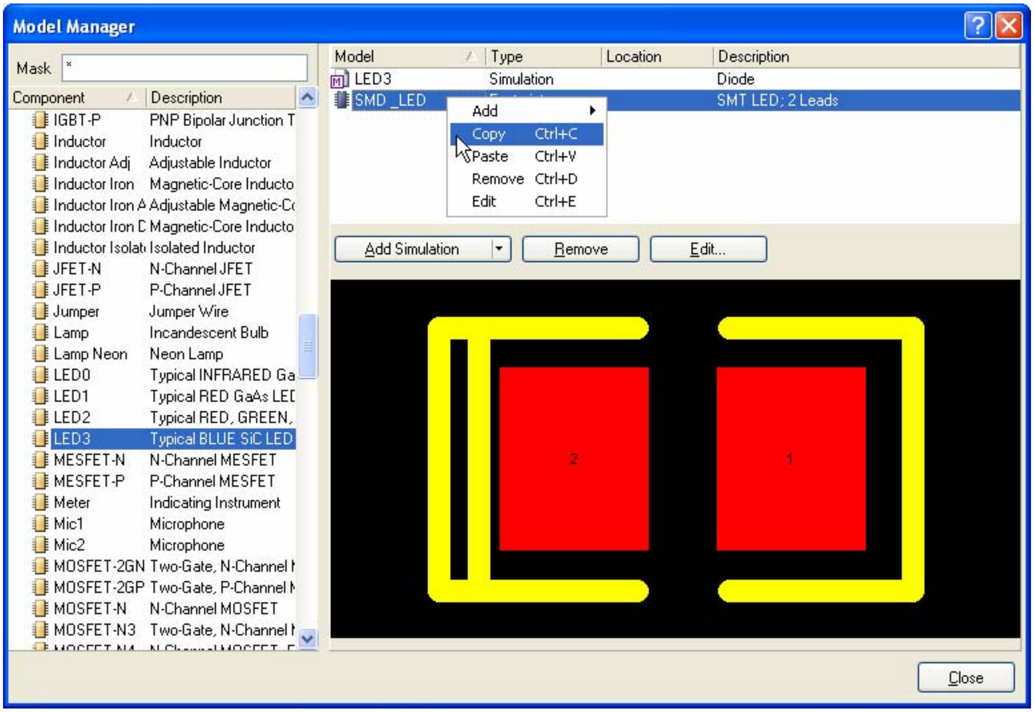


Figure 23. Use the Model Manager to manage the models across multiple components

The tasks that you can perform in the Model Manager include:

- Add a new model to one or more components
- Copy a model from one component, and paste it to one or more components
- Remove a model from one or more components
- Edit the model assigned to one or more components.

All of these commands can be executed from the right click context menu in the model list region of the dialog and some can also be performed using the buttons below the model list region.

Figure 23 shows the Model Manager with a PCB footprint model selected and about to be copied. Once it has been copied, it can be pasted to multiple components. To do this, use **Shift+Click** or **Ctrl+Click** to select multiple components in the list. Once the required components are selected, right-click in the Model region and choose **Paste** from the context menu.

An important point to remember when you select multiple components is that only the models that are common to all the selected component will be shown. So when you go to paste a footprint model to multiple components, don't be surprised if the model list region is blank. As soon as you change to only have one component selected, the current models will appear in the list.

Managing footprints across the entire design

Altium Designer's schematic editor now includes a powerful *Footprint Manager*. Launched from the PCB Editor's **Tools** menu, the Footprint Manager lets you review all the footprints associated with every component in the entire project. Multi-select support makes it easy to edit the footprint assignment for multiple components, change how the footprint is linked, or change the Current footprint assignment for components that have multiple footprints assigned. Design changes are applied through Altium Designer's standard ECO system, updating both the schematic and the PCB if required.

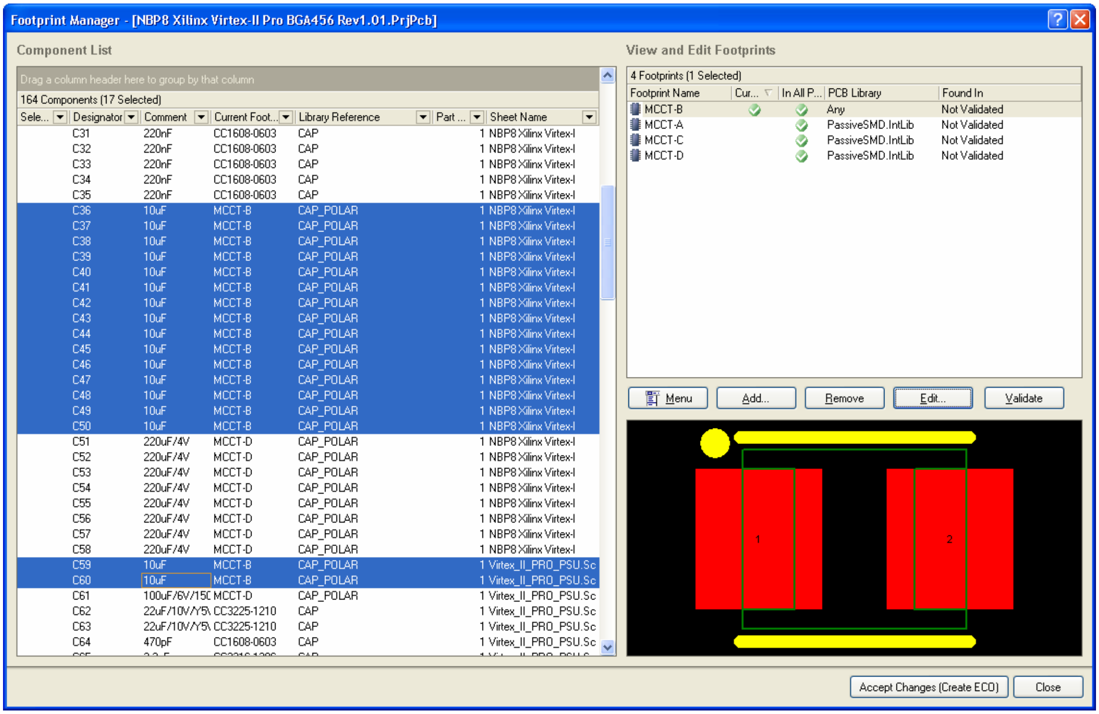


Figure 24. Use the footprint manager to review and manage the footprints across the entire PCB.

Using a query to find and edit multiple objects

Altium Designer has a powerful query engine built into it, which is used to precisely target design objects. A query is essentially a description of something that you would like to find in the design data.

Filtering to find the objects

You can query the design data in a number of different ways. One of these is to type the query in to the Filter panel. When you apply the query you are filtering the design database. Each object is tested to see if complies with the query, and if it does, it is added to the result set.

Figure 25 shows the Schematic Library editor Filter panel, with the query `IsPin` typed in. When this query is applied, every object in the Whole Library is checked (since the Whole Library option is enabled), any object that is a pin will comply and be added to the result set. All other objects are filtered out.

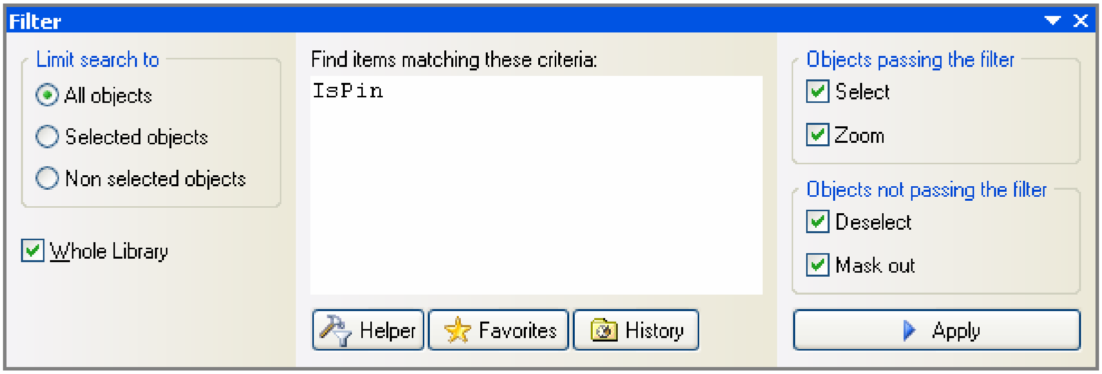


Figure 25. Using the Filter panel to query for Pins in the entire library

How the results are presented depends on the options on the right of the Filter panel. In Figure 25, you can see that objects that pass the filter (pins in this case) will be selected and zoomed. All other objects that do not pass the filter will be de-selected and masked out (faded and made non-editable).

Since the **Select** option is enabled, the pins will loaded into the Inspector. The Inspector essentially ‘stacks’ the selected objects to give one view into their common properties, which is not that useful for editing component pins (unless perhaps you wanted to change their length).

The pins will also be displayed in the List panel, which presents design data in a tabular grid, where it is easy to compare and edit one or more objects at once.

When you apply a Filter with the Mask out option enabled, the objects that are filtered out will become faded and non-editable. To remove this filter, click the Clear button at the bottom right of the workspace (shortcut, **Shift+C**).

Editing design objects in the List panel

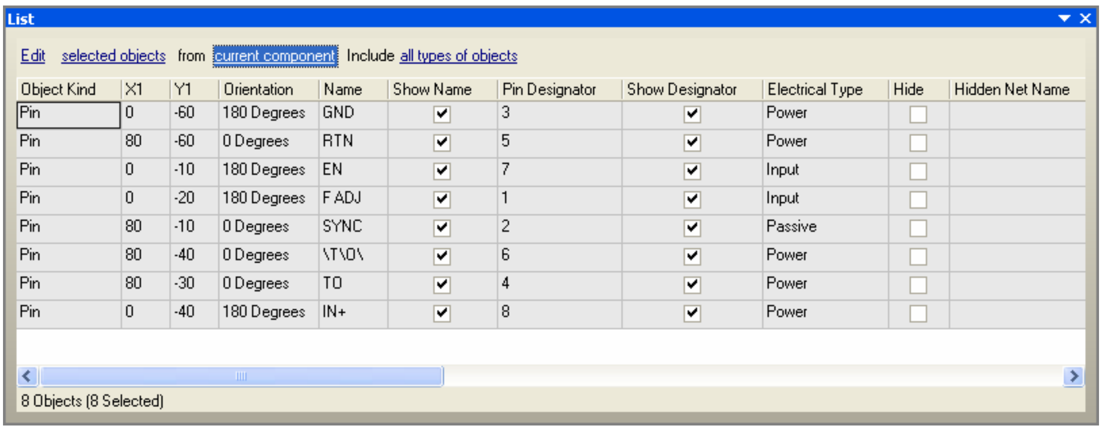


Figure 26. Pins of the current component, presented in the List panel

Figure 25 shows the Schematic Library editor List panel loaded with pins. Note that the *from* option at the top of the panel is currently set to *current component*, even though the filter was configured to select them for the whole library. There are scope controls in both the Filter panel and the List panel; this is because you control filtering separately from result display. You can use this to do things like find all pins in the current library, then switch between looking at all the pins, or just those in the current component.

The tabular grid of the List panel is ideal for reviewing and editing objects. Once you have set the List to be in Edit mode (the option at the top left of the panel), you can use keys on the keyboard to 'walk' around and edit settings. For example, use the arrow keys to move around the grid, **F2** or **Spacebar** to edit the selected cell, **Enter** to apply a change, **Spacebar** to toggle a checkbox if that cell is active, and so on.

The List panel is completely configurable. To add or remove columns, or to change the order of columns, right-click on the column headings and select **Choose Columns** from the context menu.

Using your spreadsheet editor to edit design data

Not only can you edit data directly in the **List** panel, you can also multi-select blocks of cells and copy them from the List into your preferred spreadsheet editor, and from the spreadsheet back into the **List**. For example, you are creating a new component and you have copied all the pin data from the manufacturer's PDF datasheet into a spreadsheet. Rather than entering this data into the Schematic Library editor one pin at a time, you can:

1. Place one pin in the new schematic component, copy it, then use **Paste Array** to give you the total number of pins required.
2. Use the query `IsPin` in the Filter panel to load these pins into the List panel.
3. Set up the relevant pin data columns, so that they correspond to the arrangement of columns in the spreadsheet editor.
4. Switch to the spreadsheet editor, select the required block of pin data and copy it.
5. Switch back to the List panel, select the same block of cells, right-click and choose **Paste** from the context menu

You might want to copy a block of data from the List to the spreadsheet first, to see how the data is represented in the spreadsheet. Using this approach, you can quickly configure a large number of component pins in your new component. The following figures illustrate this sequence:

E	F	G	H	I
F ADJ	TRUE	1	TRUE	0 Degrees
SYNC	TRUE	2	TRUE	Input
GND	TRUE	3	TRUE	OpenCollector
TO	TRUE	4	TRUE	Input
RTN	TRUE	5	TRUE	OpenCollector
\T\O\	TRUE	6	TRUE	OpenCollector
EN	TRUE	7	TRUE	0 Degrees
IN+	TRUE	8	TRUE	OpenCollector

Figure 27. Pin data in the spreadsheet editor, as it is copied onto the clipboard

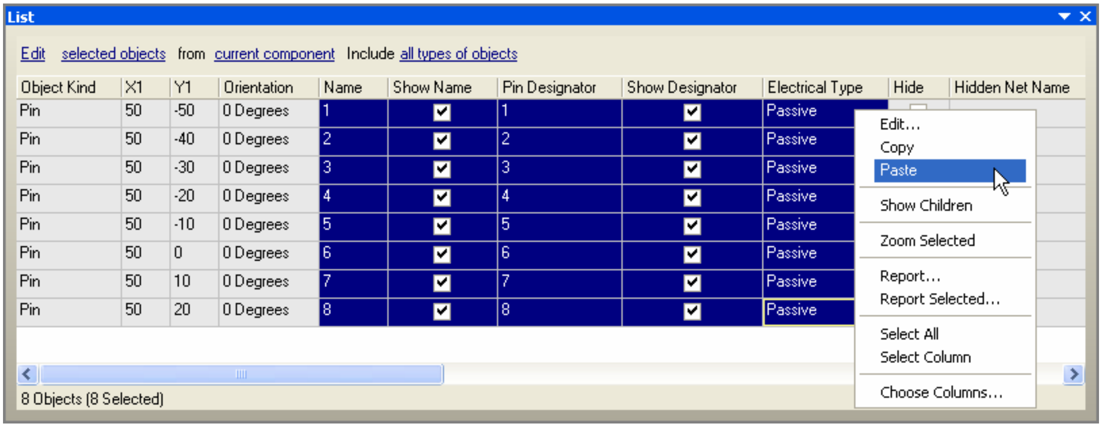


Figure 28. Select the target block of cells in the List panel, right-click and choose Paste

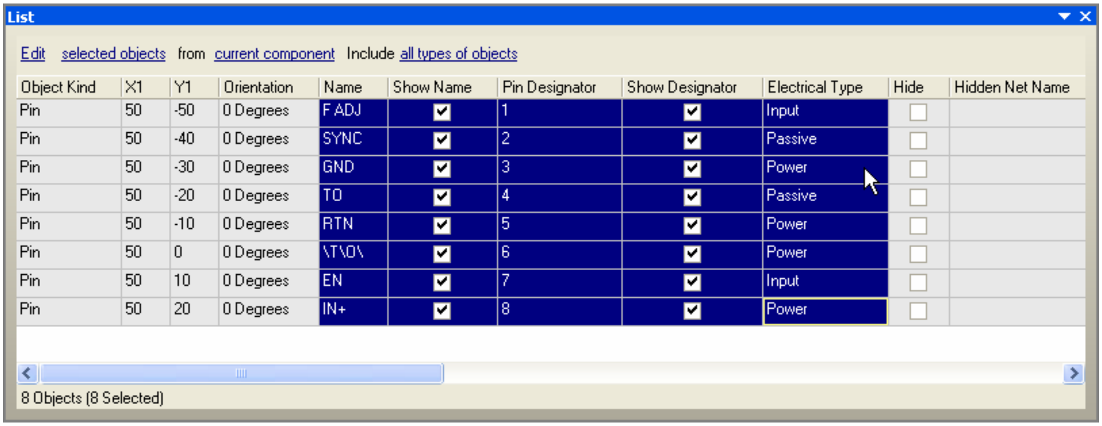


Figure 29. The List panel after the pin data has been pasted in

Filtering objects in the design workspace – how does it work?

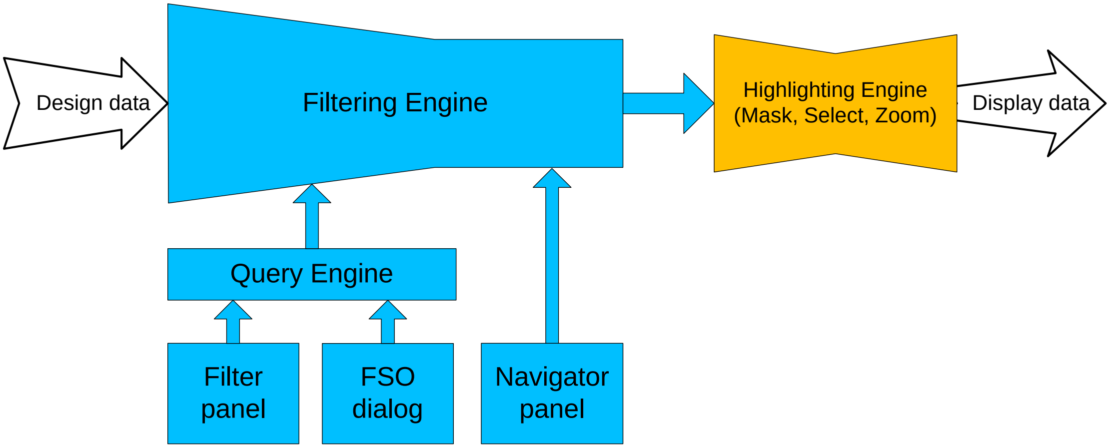


Figure 30. Diagram of the filtering/highlighting process

Figure 30 shows how design data is filtered and highlighted. Note how you can control the filtering process by writing a Query in the Filter panel, by configuring options in the *Find Similar Objects* (FSO) dialog (which actually uses a query behind the scenes), or by selecting objects in the Navigator panel. The PCB panel is not shown however, like the Navigator, it can also filter data in the PCB workspace.

The Highlighting engine determines how the filtered data will be presented.

As the user, you can access the filtered Display data in the main graphical editing window, in the Inspector (if you instructed the highlight engine to select it), or in the **List** panel.

Tips for writing queries

- Use the *Query Helper* to become familiar with the available query keywords. Press the **Helper** button in the Filter panel to display the helper.
- Press **F1** over a keyword to display on-line help for that query keyword.
- Use the **Mask** field at the bottom of the *Query Helper* dialog to search for possible keywords. If you include the * wildcard character at the start of the string you are looking for, you will find all references to that text string in the keywords and also in the descriptions.
- Click the **Check Syntax** button before you close the *Query Helper* dialog.
- Include quotation marks around a variable, for example `'DIP14'`.
- There is an order of precedence used to resolve queries, so include brackets to be sure that it is resolved in the correct sequence.

For an overview of the query system read the *Introduction to the Query Language*, for detailed information on writing queries refer to the article, *An Insiders Guide to the Query Language*.

Revision History

Date	Version No.	Revision
9-Dec-2003	1.0	New product release
1-Dec-2004	2.0	Rewritten to suit updated Inspector, List and Filter panels
13-Apr-2005	2.1	Updated for Altium Designer Service Pack 4
29-Nov-2005	2.2	Reviewed and updated for Altium Designer 6