Using Altium Documentation
Modified by Jun Chu on Aug 17, 2018

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 can also contain a large number of design objects that make up the board. During the course of the design process, the placement and properties of these objects need to change as the designer works to balance out the various design requirements.

Fundamentals of Object Placement

Grids and Cursors

Before placing objects in the Schematic Editor, set the grids to enable easier placement. Altium offers three grid types: visible grids for navigation; snap grids for placement and electrical grids for aiding the creation of connections. Grids are document options, meaning that they are saved with the individual design, so grid settings may differ between one design document and the next. Set your grids initially in the *[Document Options](#)* dialog (**Design » Document Options**).
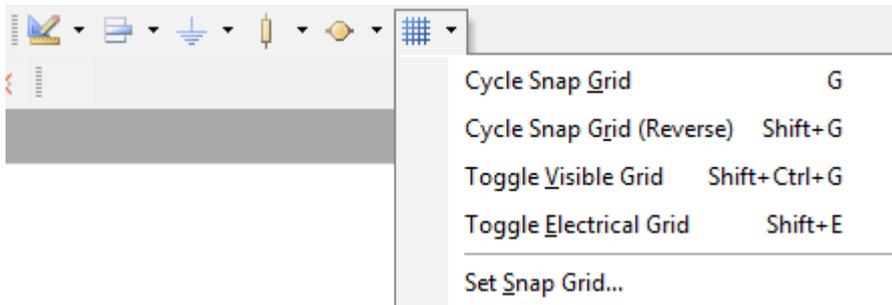
Visible grids appear whenever the zoom level will allow them to be sufficiently spaced, displayed as either lines or dots. The Snap grid is the grid that the cursor is locked to when placing or moving schematic design objects. Overriding Snap grids are Electrical grids, which allow connections to be made to off-grid parts. When moving an electrical object in the workspace and it falls within the electrical grid range of another electrical object that you could connect to, it will snap to the fixed object and a hotspot (red cross) will appear. The Electrical grid should be set slightly lower than the current Snap grid or else it becomes difficult to position electrical objects one snap grid apart.

Grids can be quickly modified or toggled between enabled and disabled, through keyboard or mouse shortcuts, for example, press **G** to cycle through the Snap grid settings of 1, 5 and 10. You can also use the **View » Grids** sub menu or the Grids right-click menu. Use the Grids tab under the Schematic folder in the Preferences dialog (**Tools » Schematic Preferences** or shortcut **T, P**) to set Imperial and Metric grid presets.

You can also change the Cursor type to suit your needs in the **Cursor** section of the [Schematic – Graphical Editing](#) page of the *Preferences* dialog. For example, a large 90 degree

cross that extends to the edges of the design window (Large Cursor 90 option) can be handy when placing and aligning design objects.

Placing Design Objects

The basic steps for placing schematic design objects are outlined below.

1. Select the object type that you want to place. You can do this by selecting an object type from the Place menu (e.g. **Place » Wire**) or by clicking on one of icons on the placement toolbars. Shortcut keys for placement are also available (e.g. **P**, **W** to place a wire). To place components (parts), you can also click the **Place** button in the *Library* panel, or select the component name from an available library in the *Library* panel and drag it into the document.
2. When an object is selected for placement the cursor will change to a crosshair, indicating that you are in editing mode, and, if relevant, the object will appear "floating" under the cursor.
3. Press the **Tab** key to edit the properties of the object before placing it. This will open the Properties dialog for the particular object, allowing you to change various options. Once you have finished setting the properties, click **OK** to return to placement mode. The advantage of editing during placement is that objects that have a numeric identifier, such as a designator, will auto-increment.
4. Position the cursor and left-click, or press **Enter**, to place the object. For complex objects, such as wires or polygons, you must continue the position-and-click procedure to place all vertices of the object.

> Note: The **AutoFocus** options on the **Schematic - AutoFocus** page of the *Preferences* dialog control the state of the schematic display. For example, the schematic can be configured to automatically zoom in when placing or editing connected objects or dim all wiring not related to the wire currently being placed. Other zooming and panning options are available using the shortcut keys or mouse wheel. Use the **Ctrl** key and scroll the wheel mouse to zoom in and out, push the wheel button down and move mouse up to zoom in or move mouse down to zoom out when placing. You can setup the behavior of your mouse on the **Schematic - Mouse Configuration** page of the *Preferences* dialog.

5. After placing an object you will remain in placement mode (indicated by the crosshair cursor), allowing you to place another object of the same type immediately. To end placement mode, right-click or press the **Esc** key. In some cases such as placing a polygon, you may need to do this twice; once to finish placing the object and once to exit placement mode. When you exit placement mode, the cursor will return to its default shape.

Re-Entrant Editing

The Schematic Editor includes a powerful feature called re-entrant editing which allows you to perform a second operation using the keyboard shortcuts without having to quit from the operation you are currently carrying out. For example, pressing the **Spacebar** when placing a part will rotate the object but will not disrupt the placement process. Once you place the part, another part will appear ready upon your cursor, already rotated.

Another example of when re-entrant editing is very useful is if you start placing a wire that it needs to be connected to a port which you have not placed yet. There is no need to exit **Place Wire** mode; just press the Place Port shortcut keys (**P**, **R**), place the port, press **Esc** to exit Place Port mode and then connect the wire to the port.

Measuring the Distance on a Schematic Document

The Schematic Editor has a distance tool located in the **Reports** menu (**Reports » Measure Distance** as well as the **Ctrl+M** shortcut keys). You can use this tool to measure the distance between two points on a schematic document.

After launching the command, you are prompted to click on two points on the schematic document. Once you have chosen two points, an *Information* dialog appears with an overall Distance value, with the X Distance and Y Distance values displayed accurate to two decimal places.
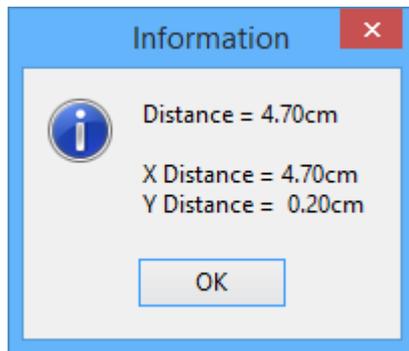
The measurement units are determined by the **System Units** chosen for the schematic document (**Design » Document Options**). If the dialog does not include measurement units, it means that the document is currently set to use DXP default units, where 1 unit is 10 mils. You can switch to Imperial or Metric units by toggling the System units (**View » Toggle Units**).

Graphically Editing Placed Objects

It is generally easier to edit the look of an object in the workspace graphically. To do this, you must first select the object(s).
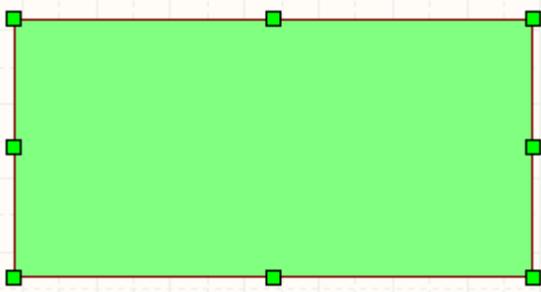
When an object is selected, you can move the object or edit its graphical characteristics. Click on an object to select it and its 'handles' or vertices are displayed. To graphically change a selected object, click and hold on an editing handle. That point of the object will then become attached to the cursor, so simply move the mouse to a new location and release to resize. Click anywhere on a selected object to move it, or press the **Delete** key to delete it.

Using Copy and Paste

In the Schematic Editor, you can copy and paste objects within or between Schematic Documents, e.g. component(s) from a schematic can be copied into another Schematic Document. You can copy objects to the Windows clipboard and paste them into other documents. Text can be copied from the Windows clipboard into a schematic text frame. You can also directly copy/paste a table-type selection from another application such as Microsoft Excel, or from

any grid style control within Altium Designer.

More advanced copy/paste actions can be performed using the Smart Paste feature.

Select the object(s) you want to copy, select **Edit » Copy (Ctrl+C)** and click to set a copy reference point on the object which will be used to accurately position the object during pasting. You will only be prompted to click to set a reference point if the **Clipboard Reference** option is enabled in the **Schematic - Graphical Editing** page of the *Preferences* dialog.

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 can 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.
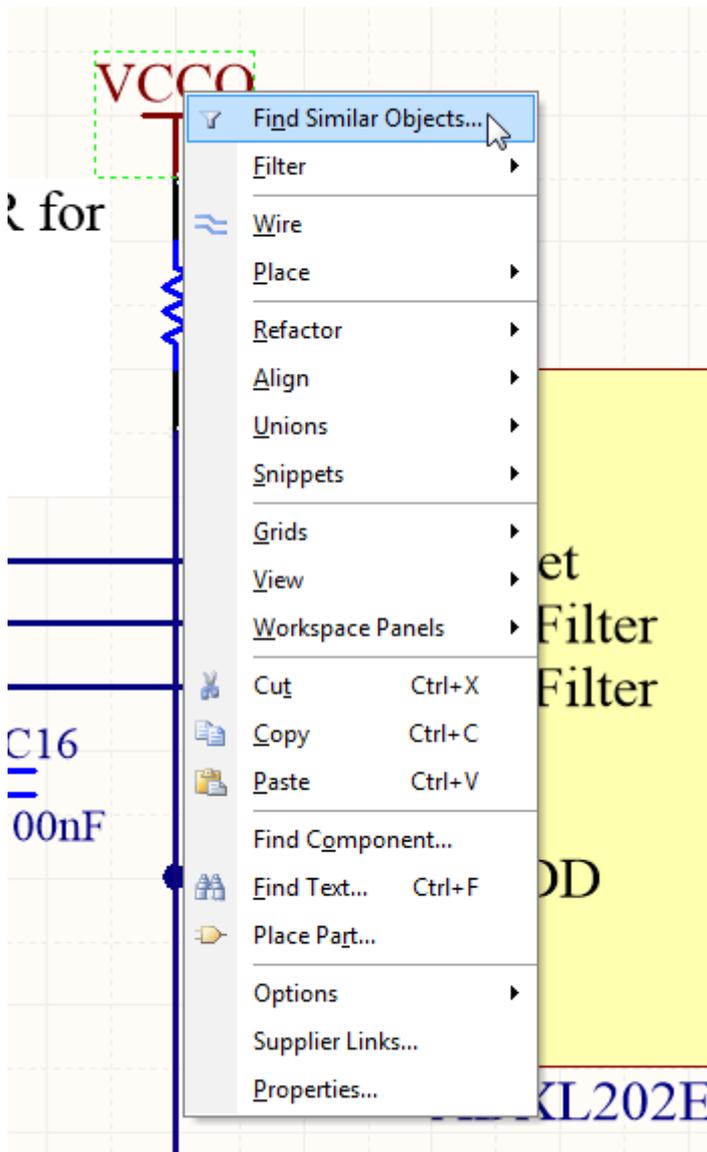
Let's walk through the process using an example. Let's say we need to change the name of a

power net in the schematic from VCC to 3V3. The name of the power net is defined by the **Net** attribute of the Power Port object. To change the name, all of the VCC power ports on all of the schematic sheets need 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** from the context menu.
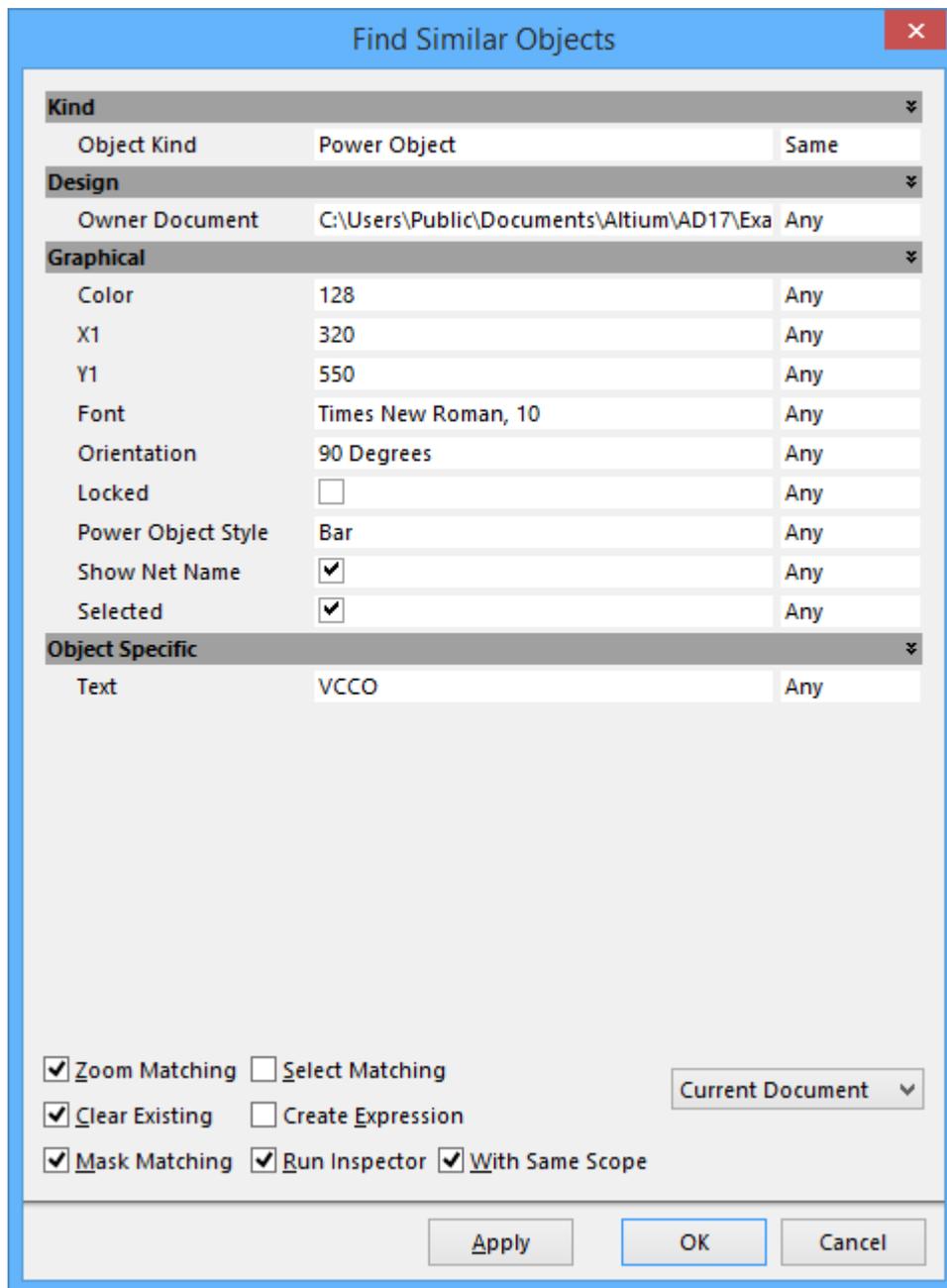

The image above shows the *Find Similar Objects* dialog after right-clicking on a schematic Power Port. It is important to note that the dialog lists the properties of the object you clicked on, so the contents of this dialog will be different if you clicked on something else. You can see that the dialog has two columns: the first column on the left shows the current properties of the object you clicked on – down the bottom you can see that the net name **Text** is currently **VCCO**.

The second column on the right is where you configure how to match other objects. For each property of the object, you 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 in the image below, 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 other object is a **Power Object** with a net name of **VCCO**.

The next step is to set the scope of the find action, should it be on the **Current Document** only, or all **Open Documents**. For the example we will set the scope to **Open Documents**. For this editing action to apply to all the sheets in the project, all sheets in the project must be opened first.
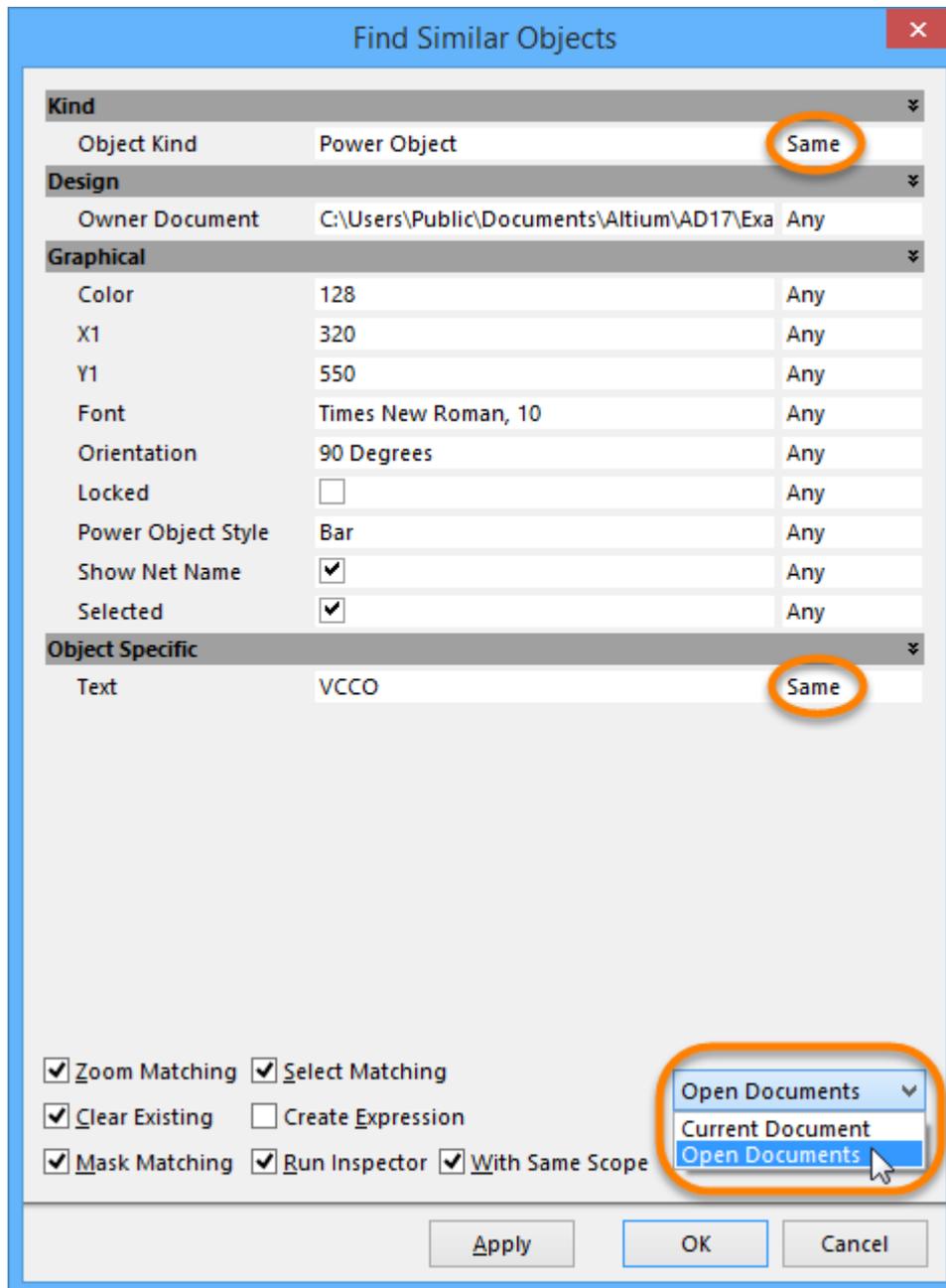
The final step is to define what should happen after all the **Power Objects** that have net name Text of **VCCO**, in all **Open Documents** have been found. The image below shows the settings that need to be set for this edit operation.

The key options to be set are:

- **Select Matching** (to select all the Power Objects with net name Text of VCC), and
- **Run Inspector**, which will open the *Inspector* panel with the selected objects loaded into it.
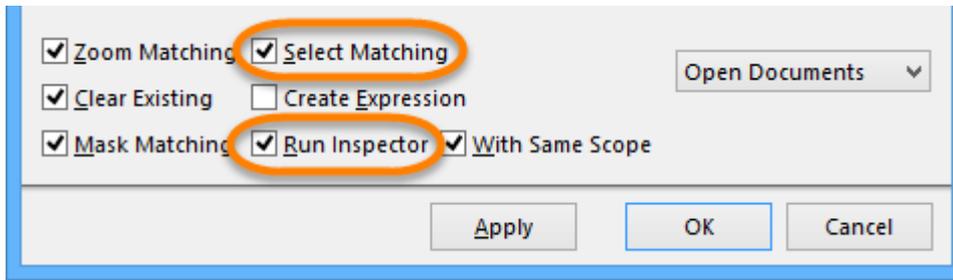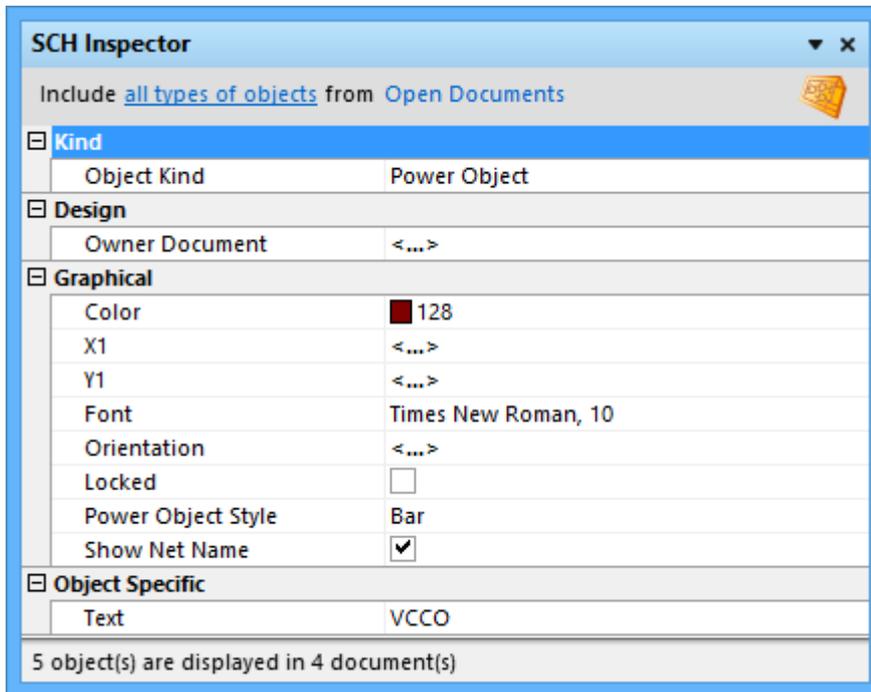
Click the **OK** button to select the matching Power Ports. Alternatively, you can use the **Apply** button, this will select the matching power ports and open the *Inspector*, but the *Find Similar Objects* dialog will remain open too – use this approach if you are not sure you have the correct matching criteria.

Moving selected objects:

- Selected schematic or PCB objects can be 'nudged' by the current snap grid value by pressing the arrow keys while holding down the **Ctrl** key.
- Selected objects can also be 'nudged' by the snap grid value x10, by pressing the arrow keys while holding down the **Ctrl+Shift** keys together.
- For the schematic objects, the current Snap Grid is displayed on the status bar and is defined in the *Document Options* dialog (**Design » Document Options** or shortcut **D**, **O**). The available Snap Grids are configured in the **Schematic – Grids** page of the *Preferences* dialog – press **G** to cycle through the available grid values as you work.
- For the PCB objects, the current Snap Grid is displayed on the Status bar and is defined in the *Grid Editor* dialog, press **Ctrl+G** to open the dialog and enter a new value.
- You can select multiple PCB components and reposition them individually (in the order that you selected them in) using the **Reposition Selected Components** command (**Tools » Component Placement » Reposition Selected Components** or shortcut **T**, **O**, **C**). The components can be selected directly on the PCB, via the *PCB* panel, or in the schematic editor if **Cross Select Mode** is enabled (**Tools** menu).

Inspecting the Objects

SCH Inspector

Include all types of objects from Open Documents

Kind	
Object Kind	Power Object
Design	
Owner Document	<...>
Graphical	
Color	■128
X1	<...>
Y1	<...>
Font	Times New Roman, 10
Orientation	<...>
Locked	☐
Power Object Style	Bar
Show Net Name	✔
Object Specific	
Text	VCCO

5 object(s) are displayed in 4 document(s)

Both the schematic and PCB editors include the *Inspector* panel. The basic behavior of the *Inspector* is that it lists the properties of all currently selected objects. The set of selected objects can be the same kind of object – for example the image above shows the properties of ten power ports – or they can be different types of objects.

Properties that are identical for all the selected objects have their value displayed, for example all ten power ports currently have the same **Color**, the same **Orientation**, the same **Style** and the same **Text** (net name).

For each power port property that has a different value, you will see **<...>** instead, for example 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.

Note in the image above that the *SCH 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 of the selected power ports loaded into the *SCH Inspector* you must set this to **open documents** or **open documents of the same project**.

What is the *Inspector*?

Both the schematic editor and the PCB editor include an *Inspector* panel – which is a panel that displays the properties of whatever is currently selected. This can be one object, or many objects – the selected object count is displayed at the bottom of the panel.

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 *SCH Inspector* and press **Enter** on the keyboard, the value of that property is immediately changed for all selected objects.

The *Inspector* panels have certain characteristics that make them very handy for everyday use.

The first is that because it is a panel, it can be visible all the time. 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 *PCB Inspector* is open, you simply click on a designator, read the value, click on the next one, read the value, and so on. This is much faster than double-clicking on one designator, reading the height, closing the dialog, double-clicking on the next designator, and so on. And, if you need to change the text height for those components, simply select the required designators, then edit the height of all of them in a single action in the *Inspector*.

The second advantage of the *Inspector* panels is that they can display the common properties of different objects, and let you edit them.

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

Editing the Objects
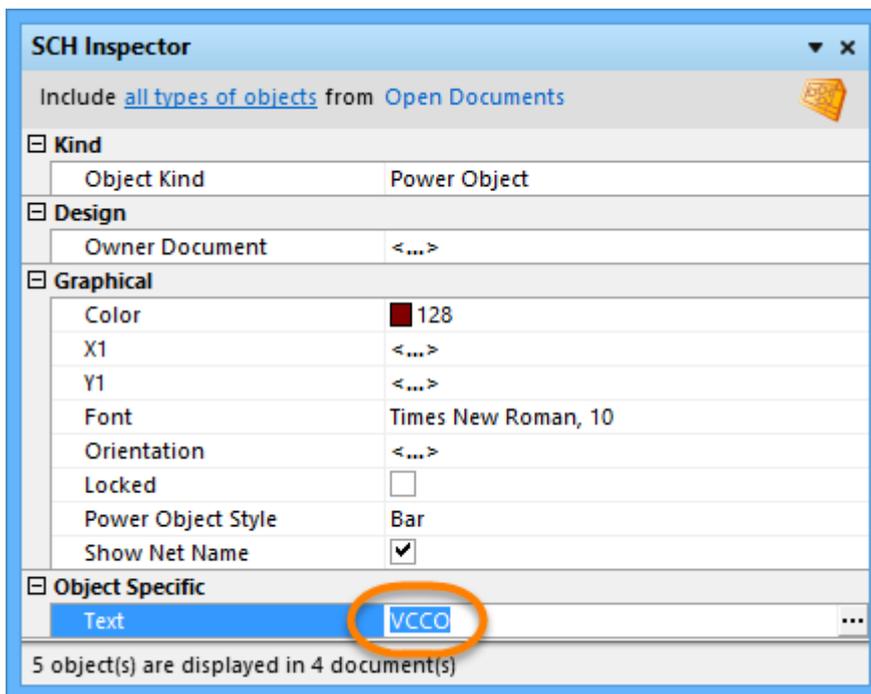
So far you have *selected* the Power Objects you want to edit and *inspected* their properties in the *Inspector*. Now you are ready to edit them.

Click to edit the net name text simply type in the new value. The ⚹ browse button appears at the end of the **Text** field, click this 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 click another cell in the *SCH 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.

The image below shows the *SCH Inspector* panel after changing the text and pressing **Enter** next to one of the edited power ports.
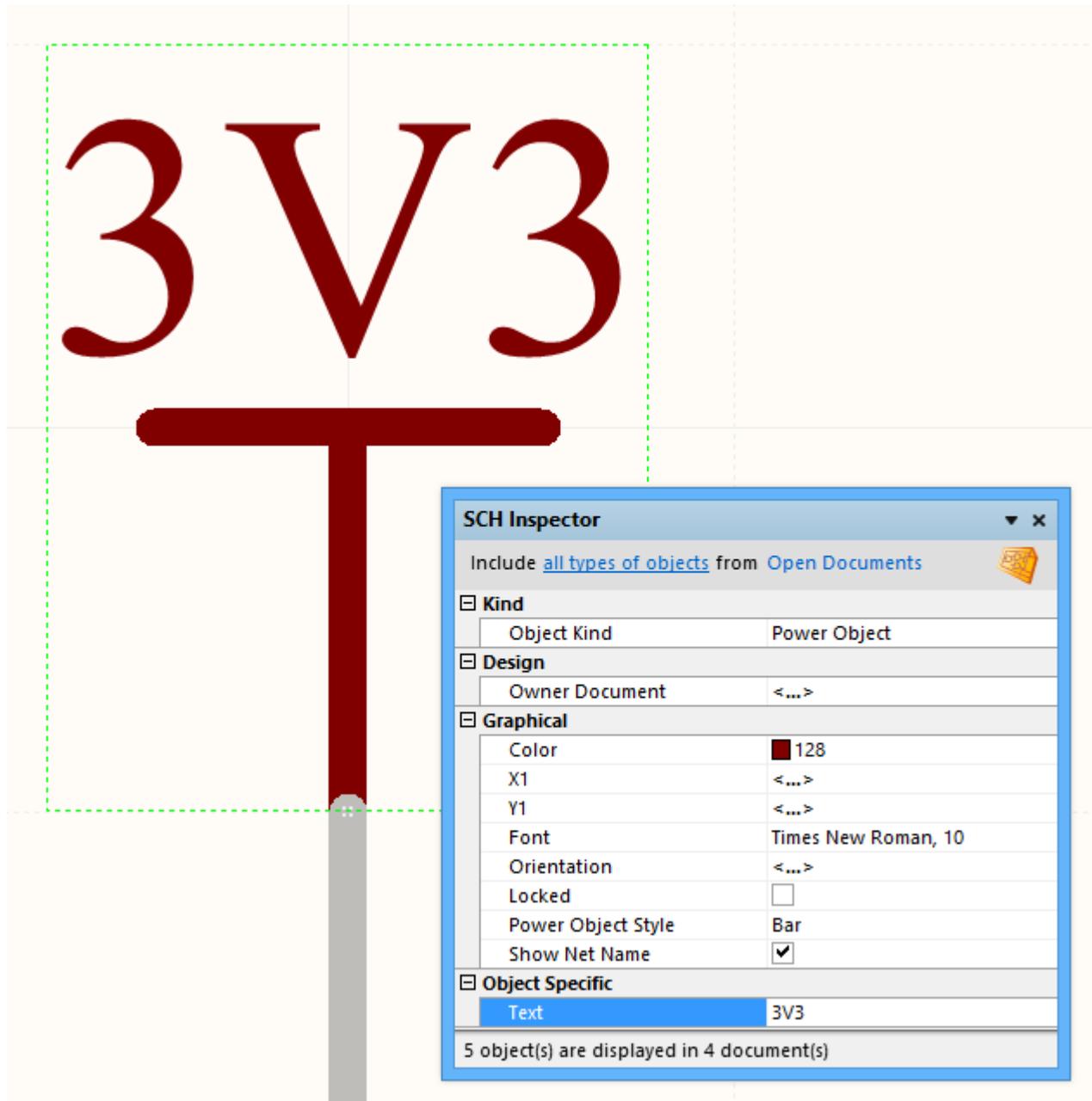
You can use this approach to apply an edit globally to any type of object in the Schematic or PCB Editors.

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**).

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 a typical group object edit that you might want to perform. Your design includes several **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 component parameter instead, and make this parameter visible on the schematic.

The steps we need to perform are:

1. Select capacitors with a value of **100uF 16V** , and **CAPPR2-5X6.8** footprint.
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 it's 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

To select all the 100uF 16V capacitors, right-click on the component symbol of one of them and select **Find Similar Objects** from the context menu.
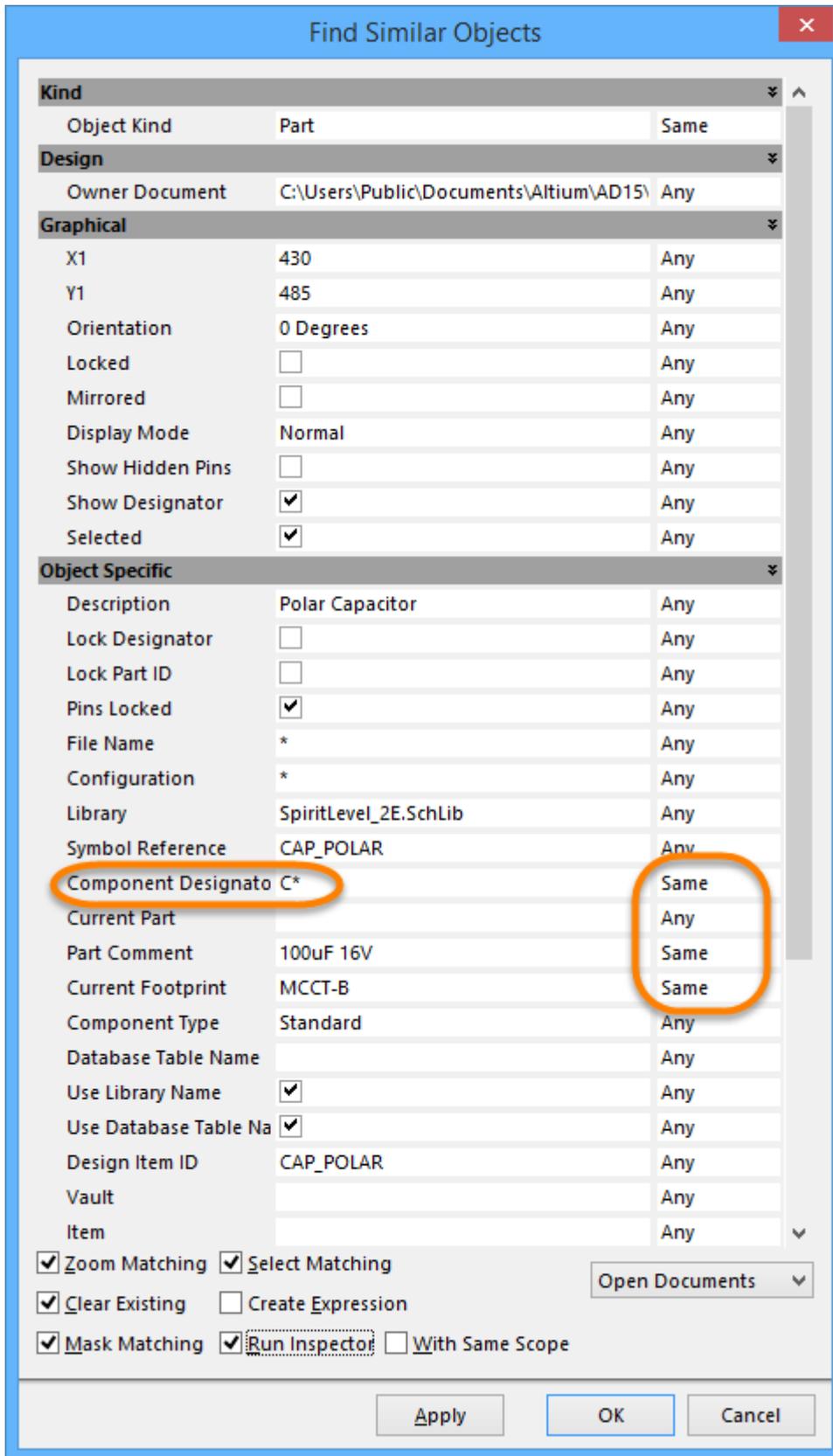
We will use the approach covered in the previous example, except this time you want to match on components that have the same **Part Comment** and the same **Current Footprint**, as shown in the image above.

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*** (above). Click **OK** to select the matching capacitors.

Step 2. Changing the Comment String
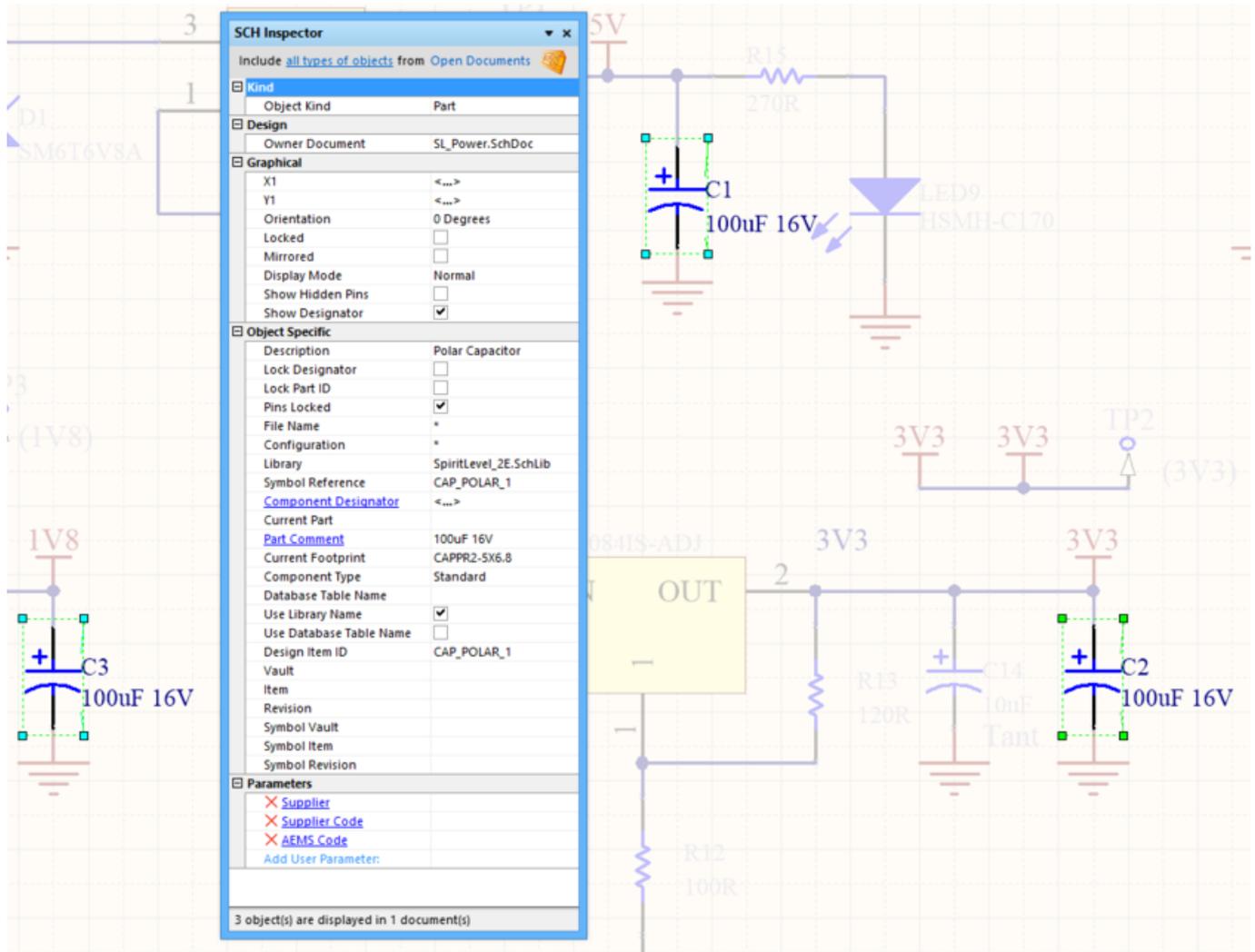
After running the *Find Similar Objects* dialog, the *SCH Inspector* panel opens (if the **Run Inspector** option in the *Find Similar Objects* dialog was enabled). Behind it will be the schematic sheet displaying the matching objects selected on that sheet. If the **Zoom Matching** and **Mask Matching** options were enabled in the *Find Similar Objects* dialog, then the view would be zoomed and all the objects that did not match are faded, or masked out.

The image above shows the results. There are three capacitors found on this current schematic sheet.

You can check the status line at the bottom of the *SCH Inspector* panel to see if the same capacitors exist on other sheets, note that it states **3 object(s) are displayed in 1 document(s)**. In this example, there are not any of the same capacitors on other sheets, since the *SCH Inspector* was configured to Include all types of objects from open documents at the top of the panel.

To change the comment string, simply delete the **16V** from the string, as shown in the image below, and press **Enter** on the keyboard to apply the change. Leave the objects selected and the *SCH Inspector* open, ready to perform the next edit.
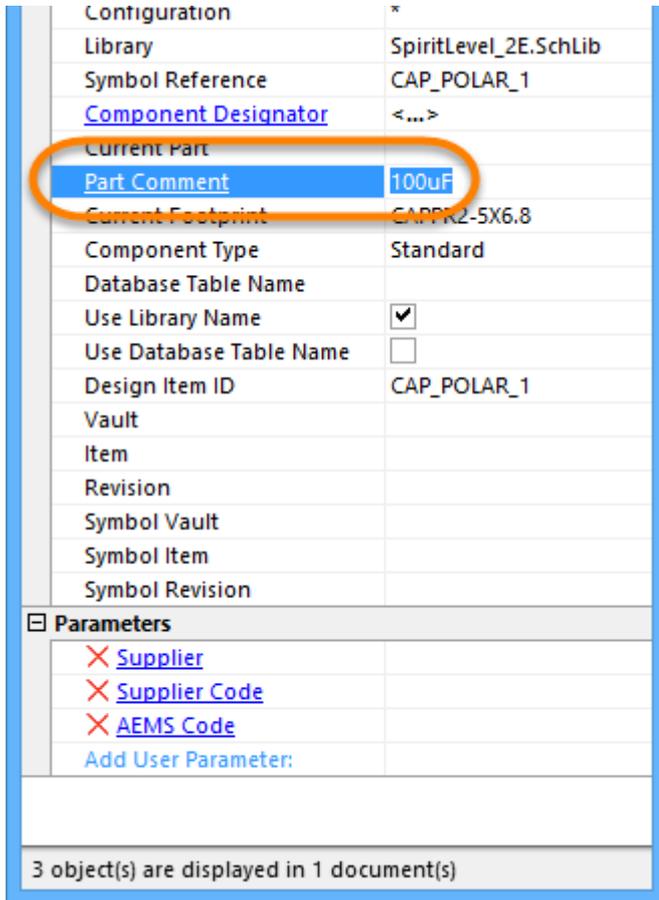
Step 3. Adding a New Parameter to the Component

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 *SCH Inspector* panel (below). Note that you enter the value first, then the parameter name.
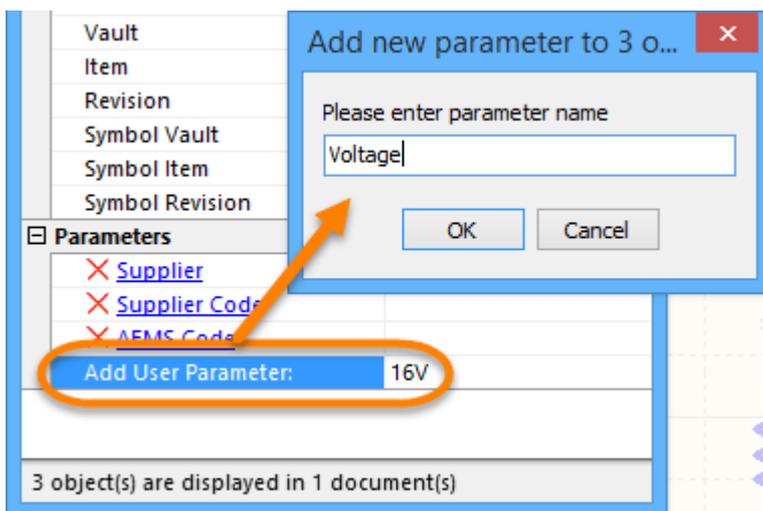
1. First, type in the value of the new parameter, **16V**, into the **Add User Parameter** field in the *Inspector*.
2. Press **Enter** to apply the change. When you do, the *Add new parameter to objects* dialog will appear.
3. Type in the new parameter name and click the **OK** button.

> Click on the red cross (X) next to each parameter to delete that parameter.

The *SCH Inspector* panel now includes the new **Voltage** parameter in the list at the bottom, with a value of **16V**, as shown in the image below. You can add as many parameters as you wish using this approach.
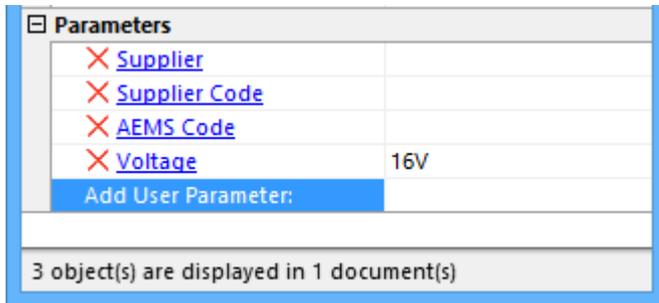
Step 4. Setting the Voltage Parameter to be Visible

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 *SCH Inspector* panel yet because it is currently 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 *SCH Inspector* panel. When you do this, the Voltage parameter properties for the selected components will be loaded into the *SCH Inspector* panel, ready to edit. You can confirm this by checking the **Object Kind** at the top of the panel – it should display the value **Parameter.**
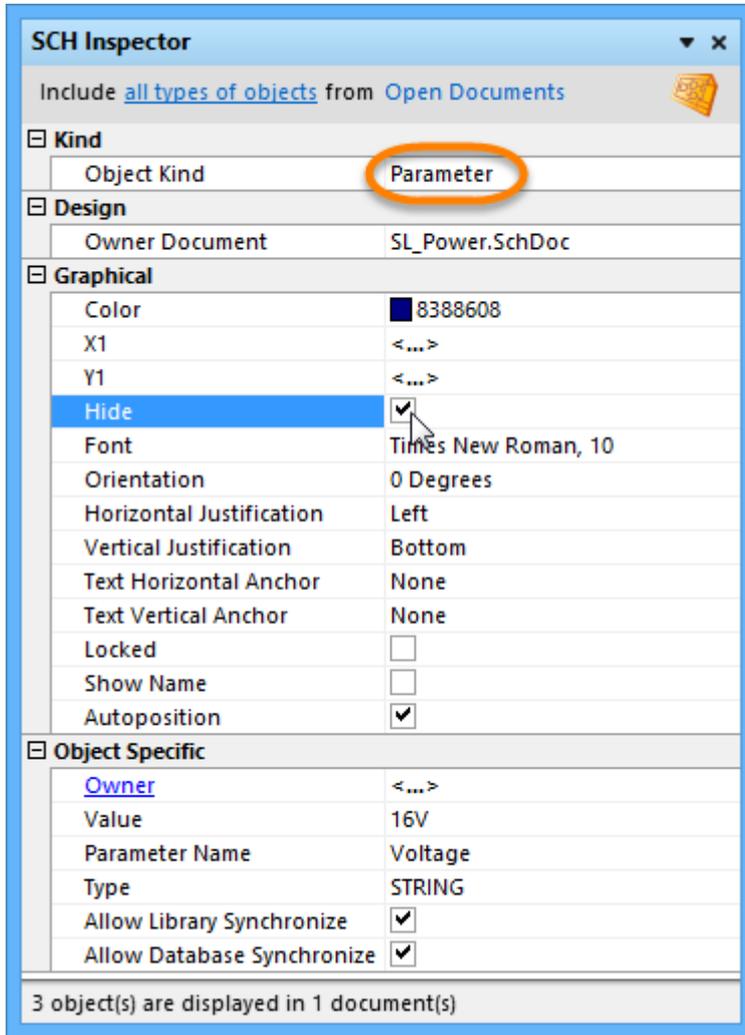
Now we can make the Voltage parameter visible on the schematic. To do this, clear the **Hide** checkbox, as shown in the image above.

If you wanted to return to the parent components, perhaps to edit some other property, you would do this by clicking the Owner hyperlink, as shown in the image below.
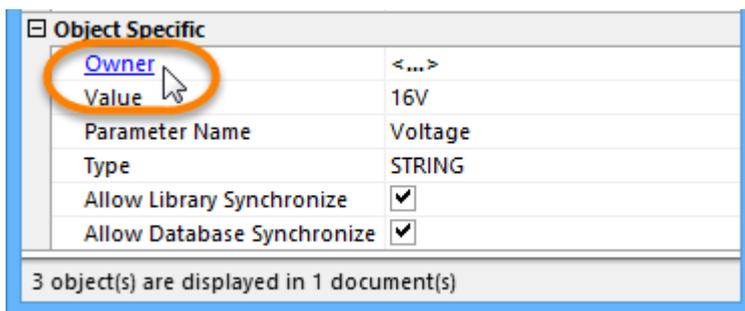
We have now updated the comment string for all **100uF** capacitors, using a **CAPPR2-5X6.8** footprint. We have also added a new parameter called **Voltage**, set its value to **16V**, and made this parameter visible.
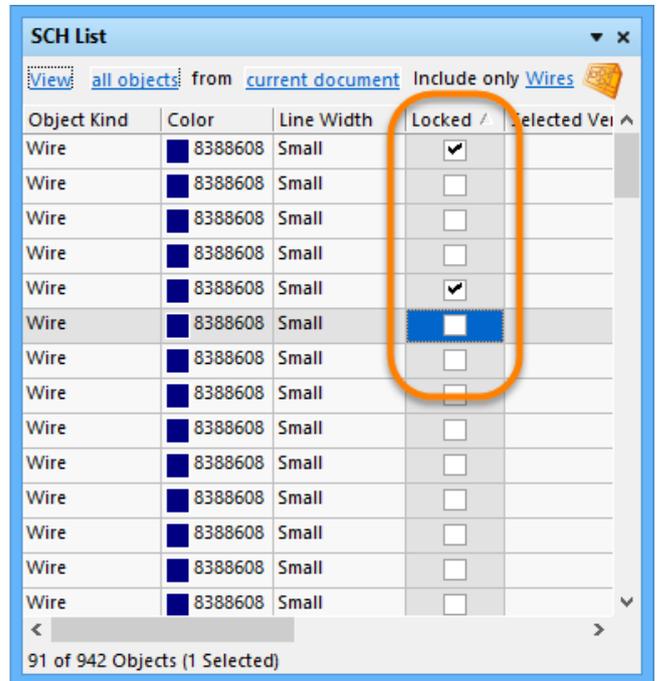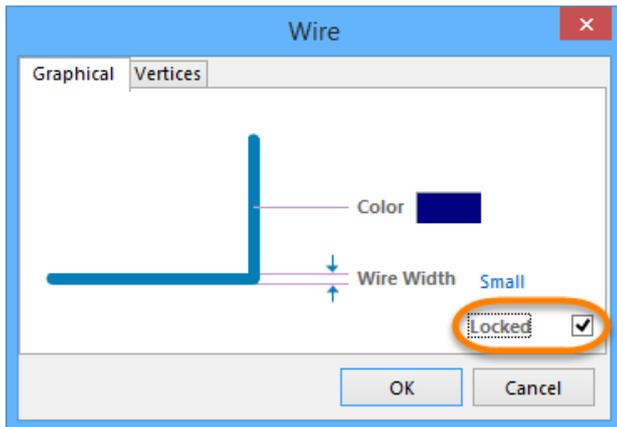
Locking Design Objects

Design objects can be locked from being moved or being edited on the schematic document by enabling their **Locked** attributes. For instance, if the position or size of specific objects are critical, lock them. This Locked attribute is available in the design objects' properties dialogs, or the Locked attributes can be toggled collectively in the SCH List

panel.

Locking Design Objects on Schematic Sheets

1. To lock a group of schematic objects, you can use the *SCH List* panel to toggle the **Locked** options as shown in the image on the right below.
2. To lock an individual object, double-click the object and when its properties dialog opens, enable the **Locked** option, as shown in the image on the left below.


If you attempt to move or rotate a design object that has its Locked property enabled, a dialog appears asking for confirmation to proceed with the edit.
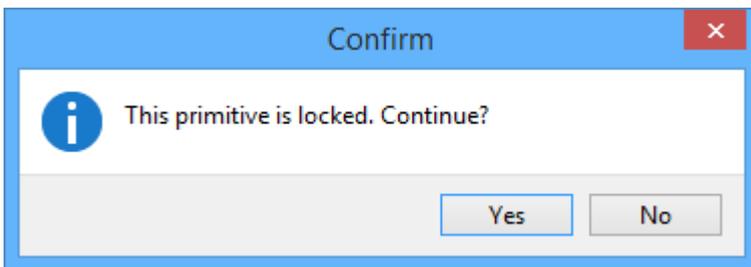
If the **Protect Locked Objects** option is enabled in the **Schematic – Graphical Editing** page of the *Preferences* dialog and the design object is *locked*, this object cannot be selected or graphically edited. Double-click on the Locked object to disable the **Locked** property, or disable the **Protect Locked Objects** option to graphically edit this object.

If you attempt to select locked objects along with other objects, only those objects that are unlocked can be selected and moved as a group when the **Protect Locked Objects** option is enabled.
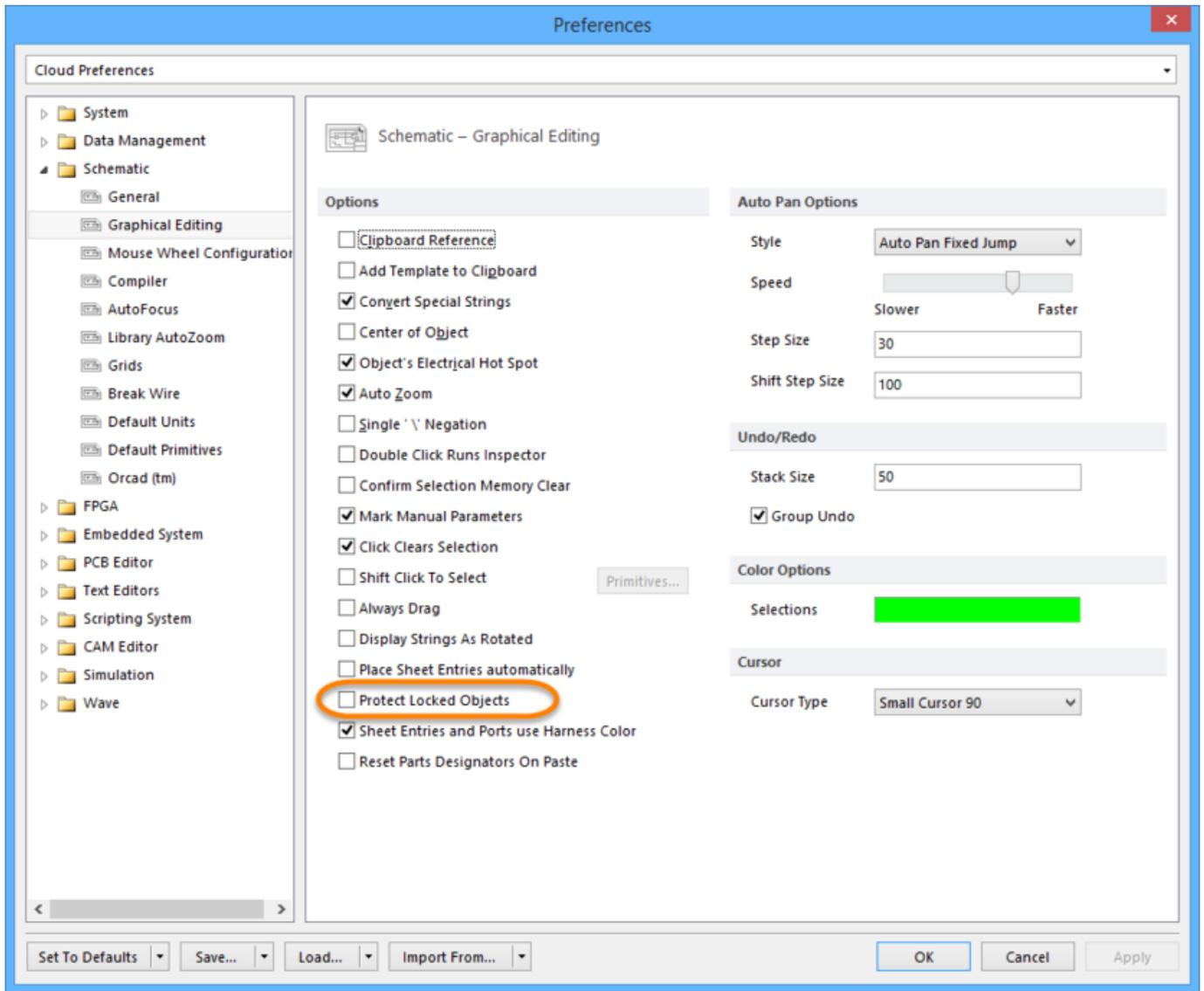
Managing Footprints Across the Entire Design
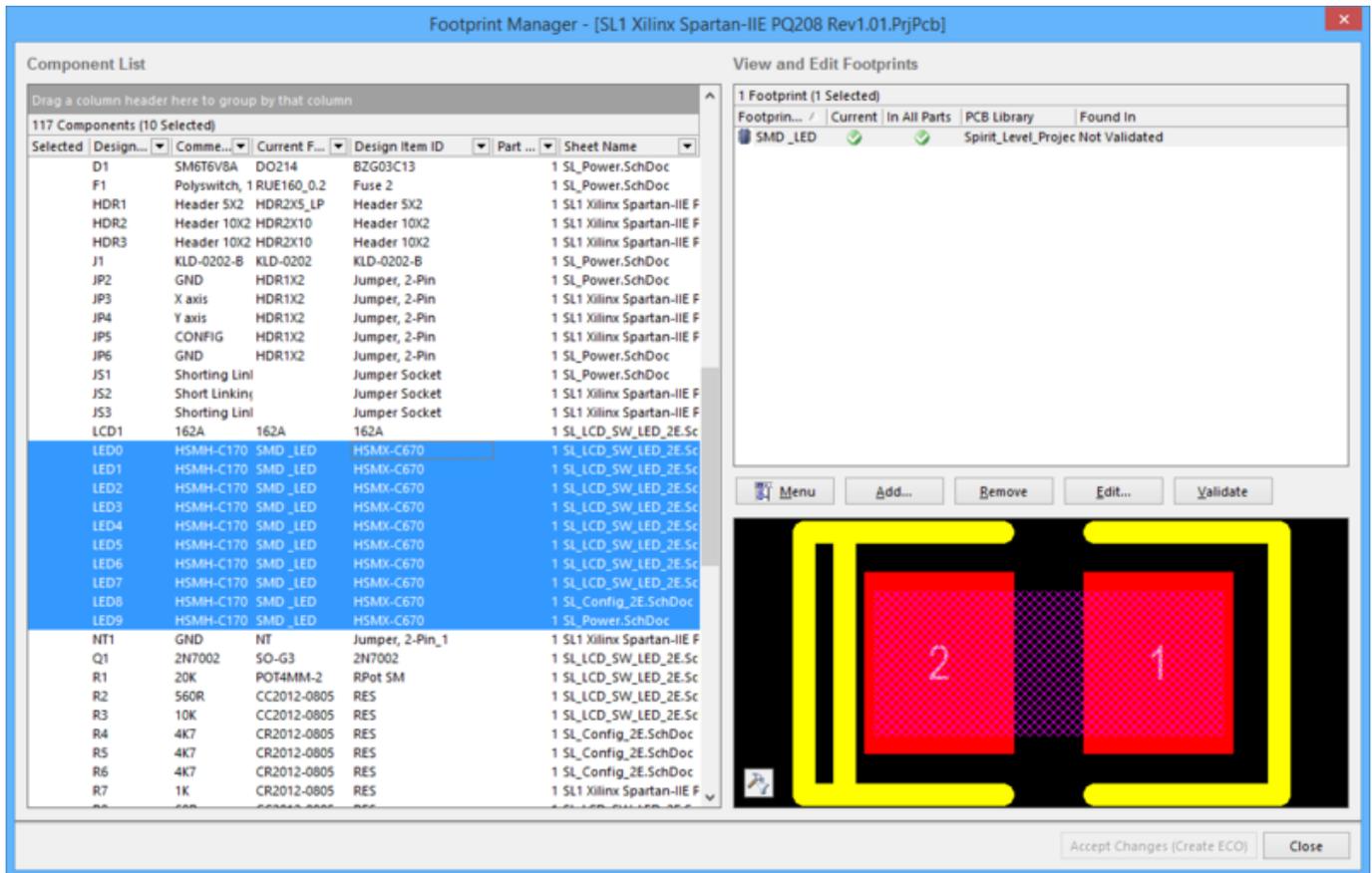
Altium Designer's schematic editor includes a powerful *Footprint Manager*. Launched from the Schematic Editor's Tools menu (**Tools » Footprint Manager**), the *Footprint Manager* lets you review all the footprints associated with every component in the entire project. Multi-select support makes it 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.

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.

The image below shows the Schematic Library *SCHLIB Filter* panel, with the query **IsPin** typed in. When this query is applied, every object in the 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.
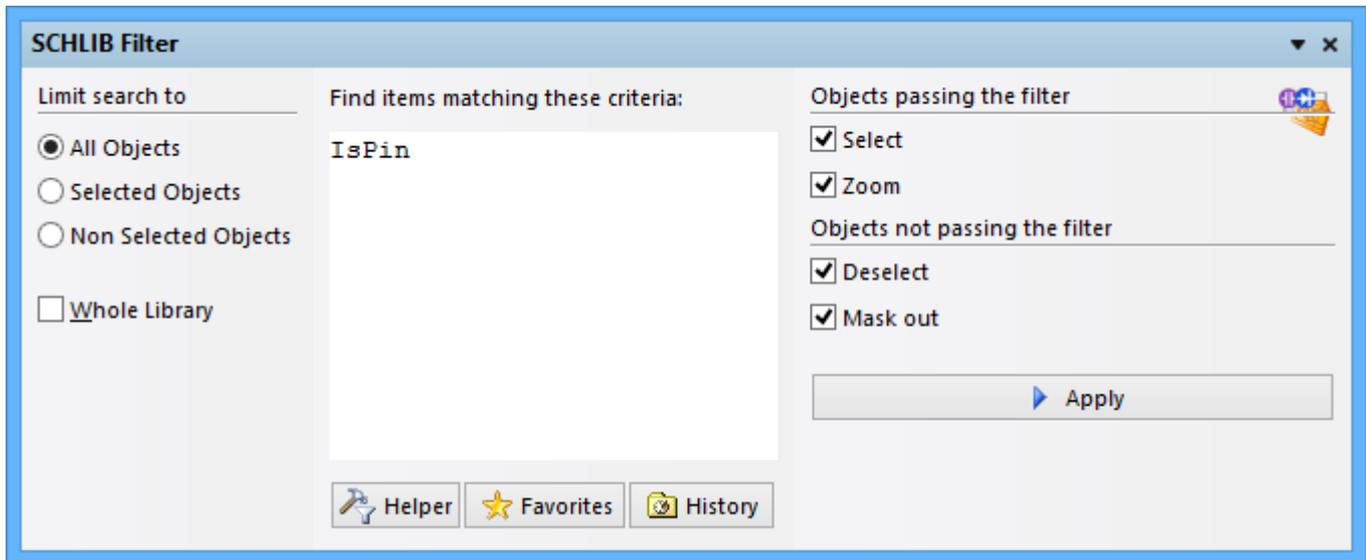
How the results are presented depends on the options on the right of the *SCHLIB Filter* panel. In the image above, you can see that objects that pass the filter (pins in this case) will be selected and zoomed. All other objects that do not pass the filter will be de-selected and masked out (faded and made non-editable).

Since the **Select** option is enabled, the pins will also be loaded into the *SCH Inspector* panel. This panel 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 *SCH 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 (**Shift+C**).

Editing Design Objects in the Schematic Library List Panel

The image above shows the Schematic Library *SCHLIB 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 *SCHLIB Filter* and the *SCHLIB List* panels; this is because you control filtering separately from the display of the results. 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 *SCHLIB List* panel is ideal for reviewing and editing objects. Once you have set the *SCHLIB List* panel 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 when that cell is active, and so on.

The *SCHLIB 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 a Spreadsheet Program to Edit Design Data

Not only can you edit data directly in the *SCHLIB List* panel, you can also multi-select blocks of cells and copy them from the *SCHLIB List* panel into your preferred spreadsheet program, and from the spreadsheet back into the *SCHLIB List* panel. For example, you are creating a new component and you have copied all the pin data from the manufacturer's datasheet into a spreadsheet.

> **Tip:** You need to set the *SCHLIB List* panel in `Edit` mode (use the control at the top of the panel) to be able to edit and paste data in this panel.

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 the `Paste Array` command 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 program.
4. Switch to the spreadsheet program, select the required block of pin data and copy it.
5. Switch back to the *SCHLIB List* panel, select the same block of cells, right-click and select `Paste` from the context menu.

You might want to copy a block of data from the *SCHLIB List* panel 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 image above and the two below illustrate this sequence:

Creating and Editing Data as you Paste from a Spreadsheet or Table

You can also use the Smart Grid Paste tools to quickly update the design objects' attributes or to create a group of primitives quickly and easily. These tools are available via the right-click menu in the Schematic or PCB Editor's List panel.

> For more information on using the Smart Grid Paste tools, see the *Sch List* page.

Filtering Objects in the Design Workspace – How does it work?

You can control the filtering process by writing a Query in the *Filter* panel, by configuring options in the *Find Similar Objects* dialog (which actually uses a query behind the scenes), or by selecting objects in the *Navigator* panel.

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 the objects), or in the *List* panel.

Tips for Writing Queries

- Use the *Query Helper* to become familiar with the available query keywords. Click the `Helper` button in the *Filter* panel to display the helper. Visit the Query Language Reference section for more information about queries.
- 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.