

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.

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.

Find Similar Ol	ojects			? 🔀
Kind	_/			¥
Object Kind	1	Power Object		Same
Design	1			¥
Owner Docum	ent	C:\Program Files\Altium	2004 SP2\Examples	Any
Graphical				¥
Color		128		Any
X1		390		Any
Y1		120		Any
Orientation	1	90 Degrees		Any
Power Object:	Siyle	Bar		Any
Selected	N			Any
Object Specific	:			¥
Text		VCC		Same 💌
Zoom Matching Select Matching				
Clear Existing Create Expression				
✓ Mask Matching ✓ Run Inspector				
			ОК	Cancel



Find Similar Objects					
Kind					
Object Kind	Power Object	Same			
Design		*			
Owner Document	C:\Program Files\Altium2004 SP2\Ex	amples Any			
Graphical		¥			
Color	128	Any			
X1	390	Any			
Y1	120	Any			
Orientation	90 Degrees	Any			
Power Object Style	Bar	Any			
Selected	v	Any			
Object Specific					
Text	VCC	Same			
✓ Zoom Matching ✓ Sele	ct Matching	Rear Desuments			
🔽 Clear Existing 📃 Crea	Clear Existing Create Expression				
✓ <u>M</u> ask Matching <u>✓ R</u> un Inspector					
		Cancel			

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.

Find Similar Objects		? 🔀		
Kind		¥		
Object Kind	Power Object	Same		
Design		¥		
Owner Document	C:\Program Files\Altium2004 SP2\Examples	Any		
Graphical		¥		
Color	128	Any		
X1	390	Any		
Y1	120	Any		
Orientation	90 Degrees	Any		
Power Object Style	Bar	Any		
Selected	✓	Any		
Object Specific		¥		
Text	VCC	Same 💌		
✓ Zoom Matching ✓ Select Matching ✓ Clear Existing Create Expression ✓ Mask Matching ✓ Run Inspector				
		Cancel		

Figure 4. What documents should this edit apply to?

Find Similar Objects		? 🗙		
Kind		¥		
Object Kind	Power Object	Same		
Design		¥		
Owner Document	C:\Program Files\Altium2004 SP2\Examples	Any		
Graphical		¥		
Color	128	Any		
X1	390	Any		
Y1	120	Any		
Orientation	90 Degrees	Any		
Power Object Style	Bar	Any		
Selected	✓	Any		
Object Specific		¥		
Text	VCC	Same 💌		
✓ Zoom Matching ✓ Select Matching ✓ Clear Existing Create Expression ✓ Mask Matching ✓ Run Inspector				
	Apply OK	Cancel		

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

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.

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.

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

In	spector		▼ ×
h	nclude <u>all types of objects</u>	from	open documents
Ξ	Kind		
	Object Kind	Pow	er Object
Ξ	Design		
	Owner Document	<>	>
Ξ	Graphical		
	Color	1 2	28
	X1	<>	•
	Y1	<>	•
	Orientation	90 D	egrees
	Power Object Style	Bar	
Ξ	Object Specific		
	Text	VCC	
10) object(s) are displayed in	2 doo	cument(s)

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

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 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**).

In	spector		▼ X
Ir	nclude <u>all types of objects</u>	from	open documents
⊡	Kind		
	Object Kind	Pow	er Object
Ξ	Design		
	Owner Document	<)	>
Ξ	Graphical		
	Color	12	28
	X1	<)	>
	Y1	<)	>
	Orientation	90 D	egrees
	Power Object Style	Bar	
Ξ	Object Specific 🦯		
	Text	8)
		-	
10) object(s) are displayed ir	n 2 do	cument(s)

Figure 7. Editing the net name Text



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.
- 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.

Find Similar Objects		? 🛛		
Kind		¥		
Object Kind	Part	Same		
Design		¥		
Owner Document	H:\2004\Training\3 day Nexar training\V1.0	Any		
Graphical		¥		
X1	570	Any		
Y1	355	Any		
Orientation	0 Degrees	Any		
Mirrored		Any		
Display Mode	Normal	Any		
Show Hidden Pins		Any		
Show Designator	✓	Any		
Selected		Any		
Object Specific		¥		
Description	Polar Capacitor	Any		
Lock Designator		Any		
Lock Part ID		Any		
Pins Locked	v	Any		
File Name	×	Any		
Configuration	×	Any		
Library	Altium Nanoboard Project.IntLib	Any		
Library Reference	CAP_POLAR	App		
Component Designator	C*)	Same		
Current Part		Any		
Part Comment	100uF 16V	Same		
Current Footprint	CAPPR2-5x6.8	Same		
Component Type	Standard	Any		
Parameters		¥		
Supplier		Any		
Supplier Code		Any		
AEMS Code		Any		
Zoom Matching 🔽 Sele	ct Matching	Jacumente 🗤		
🗹 <u>C</u> lear Existing 📃 Crea	te Expression			
Mask Matching Mun Inspector				
	Apply OK	Cancel		

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.

- 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.

	Lurrent Footprint	LAPPRZ-5x6.8	
	Component Type	Standard	
Ξ	Parameters		
	<u>Supplier</u>		
	Supplier Code		
	AEMS Code		
C	<u>Voltage</u>	16V 💙	
	Add User Parameter:		
4 object(s) are displayed in 2 document(s)			

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, <u>Voltage</u>, 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.

If you wanted to return to the parent components, perhaps to edit some other property, you would do this by clicking on the <u>Owner</u> 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.

Inspector	▼ ×			
Include all types of objects from	open documents			
🗆 Kind				
Object Kind	Parameter			
🗆 Design				
Owner Document	SL_Power.SchDoc			
🗆 Graphical				
Color	8388608			
×1	<>			
Y1	4. 2			
Hide				
FontId	14			
Orientation	0 Degrees			
Horizontal Justification	Left			
Vertical Justification	Bottom			
Text Horizontal Anchor	None			
Text Vertical Anchor	None			
Show Name				
Autoposition	✓			
🗆 Object Specific				
<u>Owner</u>	<>			
Value	16V			
Parameter Name	Voltage			
Туре	STRING			
Allow Library Synchronize	✓			
Allow Database Synchroniz	✓			
4 object(s) are displayed in 2 document(s)				

Figure 12. Change the visibility of the new parameter

	Autoposition	v
P	Object Specific	
(<u>Owner</u>	<>
	Value's	16V
	Parameter Name	Voltage
	Туре	STRING
	Allow Library Synchronize	✓
	Allow Database Synchroniz	✓
4	object(s) are displayed in 2 docu	ument(s)

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.

 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**).

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

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.

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:

- Click the Layer tab for the current mechanical layer at the bottom of the PCB editor window to make it the active layer.
- 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.

Inspector	▼ ×
Include all types of objects	
🗆 Kind	
Object Kind	<>
Object Specific	
Layer	Mechanical3 🔹 💌
Net	DrillGuide 🔨 🔨
Keepout	KeepOutLayer
🗆 Graphical	Mechanical Mechanical
X1	Mechanical4 💦 📃
Y1	Mechanical16 🗟 🚞
Locked	DrillDrawing 📉
36 object(s) are displayed	

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*.

Parameter Editor Options	×
Include Parameters Owned By ✓ Parts Nets (Parameters Sets) Pins Models Ports Documents	
That Meet the Following Criteria All Objects	
Other Options Exclude System Parameters Selected Objects Only	
OK Cancel]

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.

Parameter Table Editor For Project [4 Port Serial Interface.PRJPCB]												
39 Objects -	1 Selected		User Paramete	ris					^			
Object Type	Document	Identifier 🛛 🛆	Capacitance	Manufactuer	Manufactuer P/N	Part Number	Text Field1	Toerance	Voltage-Rate			
- Part	4 Port UART and Line Drivers.SchDoc	C1	0.1µF	BC Components	2222 370 22104	C001037 🔪	CAPACITOR	2.05	100V			
🕩 Part	4 Port UART and Line Drivers.SchDoc	C2	0.1µF	BC Components	2222 370 22104	C001037	CAPACITOR	0.05	100V			
i⊇- Part	4 Port UART and Line Drivers.SchDoc	C3	0.1µF	BC Components	2222 370 22104	C001037	CAPACITOR	0.05	100			
Part	4 Port UART and Line Drivers.SchDoc	C4	0.1µF	BC Components	2222 370 22104	C001037	CAPACITOR	0.05	100V			
i⊇- Part	4 Port UART and Line Drivers.SchDoc	C5	0.1µF	BC Components	2222 370 22104	C001037	CAPACITOR	0.05	100V			
Part	4 Port UART and Line Drivers.SchDoc	C8	0.1µF	BC Components	2222 370 22104	C001037	CAPACITOR		1000			
🕞 Part	4 Port UART and Line Drivers.SchDoc	C9	0.1µF	BC Components	2222 370 22104	C001037	CAPACITOR	Revert				
Part	4 Port UART and Line Drivers.SchDoc	C10	0.1µF	BC Components	2222 370 22104	C001037	CAPACITOR	Edit				
Part	ISA Bus and Address Decoding.SchDoc	C11	0.1µF	BC Components	2222 370 22104	C001037	CAPACITOR	Add				
Part	ISA Bus and Address Decoding.SchDoc	C12	0.1µF	BC Components	2222 370 22104	C001037	CAPACITOR	Remove				
- Part	4 Port UART and Line Drivers.SchDoc	C13	20pF	Kemet	C1206C200J5GACT	C001861	CAPACITOR	Rename N				
Part	4 Port UART and Line Drivers.SchDoc	C14	50pF	Kemet	C1206C510J5GACT	C001862	CAPACITOR	Undo Rename 😽				
Part	ISA Bus and Address Decoding.SchDoc	C15	10µF	Panasonic	ECA-1HHG100	C001618	CAPACITOR (Add Column	IC D			
Part	ISA Bus and Address Decoding.SchDoc	C16	10µF	Panasonic	ECA-1HHG100	C001618	1///////	Remove Columns	C			
🕩 Part	ISA Bus and Address Decoding.SchDoc	C17	10µF	Panasonic	ECA-1HHG100	C001618	1///////		- IC			
Part	ISA Bus and Address Decoding.SchDoc	D1		General Semicondu	1N4004	X001005	////////	Cut	////			
Part	ISA Bus and Address Decoding.SchDoc	D2		General Semicondu	1N4004	X001005	1///////	Сору	////			
Part	4 Port UART and Line Drivers.SchDoc	J1		Norcomp Inc.	179-037-412-571	J001009	////////	Paste				
Part	ISA Bus and Address Decoding.SchDoc	P1		EDAC Inc.	395-062-520-350	J001027	///////////////////////////////////////	Clear				
Part	4 Port UART and Line Drivers.SchDoc	R1		BC	2322 245 221.00M	R002014	////////		— ////_			
Part	4 Port UART and Line Drivers.SchDoc	R2		BC	2322 245 221.50K	R001737	///////////////////////////////////////	Report	////			
Part	ISA Bus and Address Decoding.SchDoc	RP1		BC	2322 245 2210.0K	R001817	///////////////////////////////////////	Save All	////			
Part	ISA Bus and Address Decoding.SchDoc	S1		AMP/Tyco Electron	435640-5	S001169	1////////	Save Selected				
Part	ISA Bus and Address Decoding.SchDoc	S2		AMP/Tyco Electron	435640-9	S001175	1111111	Select All	1111			
Part	4 Port UART and Line Drivers.SchDoc	U1		Maxim	MX7533JCWE	U001598	////////	Select Column	1///			
Part	4 Port UART and Line Drivers.SchDoc	U2		Texas Instruments	MC1488N	U001659	11/1/1/1		_			
Part	4 Port UART and Line Drivers.SchDoc	U3		Texas Instruments	MC1488N	U001659	///////	Cross Probe				
Part	4 Port UART and Line Drivers.SchDoc	U4		Texas Instruments	MC1488N	U001659			9V			
Part	4 Port UART and Line Drivers.SchDoc	U5		Texas Instruments	MC1489N	U001701			5V 💌			
<		ш							>			
Revert Selec	Revert Selected Remove Column Add Column Accept Changes (Create ECO) Cancel											

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

	-							
mber	Component Type	oleranc						
17	CAPACITOR	0.05						
17	CAPACITOR	0.05						
17	CAPACITOR	0.05						
17	CAPACITOR	0.05						
!7	CAPACITOR	0.05						
Figure 10. The renewed personator								

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.

				? 🗙					?
				>					
anufactuer P/N	Part Number	Compone	ent Type Tolerance	Voltage-Rate	ufactuer P/N	Part Number	Component Type	Tolerance	Voltage-Ra
1206C510J5GACT	C001862	CAPACI1	OR 0.05	50V	06C510J5GACT	C001862	CAPACITOR	0.05	50V
CA-1HHG100	C001618	CAPACIT	FOR (ELE 0.2	50VDC	-1HHG100	C001618	CAPACITOR (ELE	0.2	50VDC
CA-1HHG100	C001618		0.2	50VDC	-1HHG100	C001618	*	0.2	50VDC
CA-1HHG100	C001618		0.2	50VDC	-1HHG100	C001618	*	0.2	50VDC
14004	X001005		Devert	- []///////	004	X001005	*		
14004	X001005		Edit	2//////h	004	X001005	· · · · · · · · · · · · · · · · · · ·		
79-037-412-571	J001009		Add	2///////	037-412-571	J001009	√ *		
35-062-520-350	J001027		Remove	////////	062-520-350	J001027	*		
322 245 221.00M	R002014		Kelliove	_ ///////	2 245 221.00M	R002014	*	0.1	
322 245 221.50K	R001737	////	Rename	///////	2 245 221.50K	R001737	*	0.1	
322 245 2210.0K	R001817	////	Undo Rename	///////	2 245 2210.0K	R001817	*	0.1	
35640-5	S001169	[]]]]	Add Column	///////	540-5	S001169	*		
35640-9	S001175	////	Remove Columns	///////	540-9	S001175	*		
X7533JCWE	U001598		C.4		'533JCWE	U001598	*		
C1488N	U001659	[]]]]	Cut	9V	488N	U001659	*		9V
C1488N	U001659	////	Сору	9V	488N	U001659	*		9V
C1488N	U001659	/////	Paste	9V	488N	U001659	*		9V
C1489N	U001701	/////	Clear	5V 📃	489N	U001701	*		5V
C1489N	U001701	////		5V	489N	U001701	*		5V
C1489N	U001701	[[]]]	Report	5V	489N	U001701	*		5V
C1489N	U001701	////	Save All	5V	489N	U001701	*		5V
C1489N	U001701	/////	Save Selected	5V	489N	U001701	*		5V
BPAL22V10-10CN	U001669		Select All	5V	PAL22V10-10CN	U001669	*		5V
N74HC32N	U001680	////	Select Column	5V	4HC32N	U001680	*		5V
N74HC32N	U001680	1////		5V	4HC32N	U001680	*		5V
N74HC32N	U001680		Cross Probe	5V	4HC32N	U001680	*		5V
N74HC32N	U001680	//////		/ 5V	4HC32N	U001680	*		5V
CS-18-13-1	X001003				-18-13-1	X001003	+		
				>					2
		Accep	ot Changes (Create ECO)	Cancel			Accept Change	s (Create ECO)	Cancel

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.

R002014	· 0.1	R002014			0.1	R002014			0.1
R001737	· 0.1	R001737			0.1	R001737			0.1
R001817	· 0.1	R001817			0.1	R001817			0.1
S001169	Devert	S001169		. *		S001169	Switch	*	
S001175	Edit	S001175	Switch	-		S001175	Switch	•	
U001598	odd W	U001598				U001598			
U001659	Persove	U001659				U001659			
U001659	Kelliove	U001659				U001659			
U001659	Rename	U001659				U001659			
11001701	Lindo Denama	11001701				11001701			

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.

Engineering Change Order										
Modifications					Status			^		
Enable 🗸	Action	Affected Object		Affected Document	Check	Done	Message			
 Image: A start of the start of	Add	🖻 Component Type= to U9	In							
✓	Add	🖻 Component Type= to X1	In							
✓	Add	Component Type=Connector to J1	In							
✓	Add	Component Type=Resistor Netwo	In							
✓	Add	Component Type=Switch to S1	In							
 Image: A start of the start of	Add	Component Type=Switch to S2	In							
-	Change Parameter Name(13)									
✓	Modify	C10Name: [Text Field1 -> Compor	In							
✓	Modify	C11Name: [Text Field1 -> Compor	In							
✓	Modify	C12Name: [Text Field1 -> Compor	In							
✓	Modify	C13Name: [Text Field1 -> Compor	ıln							
✓	Modify	C14Name: [Text Field1 -> Compor	In							
✓	Modify	C15Name: [Text Field1 -> Compor	ln							
✓	Modify	C1Name: [Text Field1 -> Compone	ln							
 Image: A start of the start of	Modify	C2Name: [Text Field1 -> Compone	: In							
 Image: A start of the start of	Modify	C3Name: [Text Field1 -> Component	In					≡		
✓	Modify	🔎 C4Name: [Text Field1 -> Compone	In							
 Image: A start of the start of	Modify	🔎 C5Name: [Text Field1 -> Compone	In							
✓	Modify	🔎 C8Name: [Text Field1 -> Compone	In							
✓	Modify	C9Name: [Text Field1 -> Component	In							
-	Change Parameter Value(1)									
✓	Modify	U10:Part Field 1 [CDIP24 -> blah]	In					~		
Validate Char		port Changes					Close	T		
	Iges Execute changes Ee						CIOSE			

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.

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.

Model Manager					? 🛛
	Model	∕∆ Type	Location	Description	
Mask	M LED3	Simulation		Diode	
Component 🕗 Description 🔥	SMD_LED			SMT LED; 2 Leads	
IGBT-P PNP Bipolar Junction T	C	Add			
📑 Inductor Inductor		Copy Ctrl+	-		
🕕 Inductor Adj 🛛 Adjustable Inductor		r∿Paste Ctrl+	1		
🕕 Inductor Iron Magnetic-Core Inducto		Remove Ctrl+			
🜗 Inductor Iron A Adjustable Magnetic-Co		Edit Ctrl+	-		
🕕 Inductor Iron D Magnetic-Core Inducto		100			
📑 Inductor Isolat: Isolated Inductor	Add Simula	tion ▼] [<u>B</u> e	move	Edit	
IFET-N N-Channel JFET					
JFET-P P-Channel JFET					
📑 Jumper 🛛 Jumper Wire					
📑 Lamp Incandescent Bulb					
📑 Lamp Neon 🛛 Neon Lamp 👘 👘					
📑 LEDO 🛛 Typical INFRARED Ga 🔤					
📑 LED1 Typical RED GaAs LEE					
📑 LED2 Typical RED, GREEN,					
LED3 Typical BLUE SiC LED					
MESFET-N N-Channel MESFET			2	<u></u> 1	
MESFET-P P-Channel MESFET					
📕 Meter Indicating Instrument					
📑 Mic1 Microphone					
📑 Mic2 Microphone					
MOSFET-2GN Two-Gate, N-Channel 1					
📑 MOSFET-2GP Two-Gate, P-Channel N					
MOSFET-N N-Channel MOSFET				38-	
🜗 MOSFET-N3 Two-Gate, N-Channel I 🥃					
B NOCEET NA N Channel NOCEET E					

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, rightclick 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.

Footprint Manager	- [NBP8 Xilin	x Virtex-II Pro	BGA456 Rev1.01.F	rjPcb]						? 🔀
Component List						View and Edit I	Footprints			
Drag a octume header	hore to group hu	that only man			^	4 Footprints (1 Sele	ctedl			
brag a column neader	neie to group by	that column				Footprint Name	Cur ⊽ In All	P PCB Library	Found In	
164 Components (17 S	electedJ					MCCT-B		Anu	Not Validated	
Sele 💌 Designator	Comment	 Current Foot 	 Library Reference 	💌 Part 💌 Sheet Name 🔍		MCCT-A		PassiveSMD.IntLib	Not Validated	
C31	220nF	CC1608-0603	CAP	1 NBP8 Xilinx Virtex-I		MCCT-C	ŏ	PassiveSMD.IntLib	Not Validated	
C32	220nF	CC1608-0603	CAP	1 NBP8 Xilinx Virtex-		MCCT-D	ŏ	PassiveSMD.IntLib	Not Validated	
C33	220nF	CC1608-0603	CAP	1 NBP8 Xilinx Virtex-I		-				
U34	220nF	CC1608-0603	LAP	1 NBP8 Xilinx Virtex-						
L35	220nF	LC1608-0603	LAP	1 NBP8 Xilinx Virtex-I						
L36	10uF	MCCT-B	LAP_PULAR	T NBP8 Xilinx Virtex-I						
U37 000	10uF	MULT-B	CAP_PULAR	I NBP8 Xilinx Virtex-I						
L38 C20	10uF	MULT-B	CAP_PULAR	1 NDP0 Xilinx Virtex-I						
C 40	100F	MULT-B	CAP_PULAR	1 NDP0 V/fmv V/ftex-1						
C40	10ur	MCCT-D	CAP_FULAN	1 NDF0 Allinx Villex-1						
041	10ur	MUCT D	CAP_FULAR	1 NDP0 //inx Vittex-I						
C42	10uF	MCCT P		1 NDP0 Allinx Villex-1						
043	10ur	NCCTR	CAP_FOLAR	1 NDEO Alimix Villex-1						
C44	10uF	MCCT P		1 NDP0 Allinx Villex-1						
C45 C49	10ur	MCCT-D		1 NDP0 Villey Vitex-1						
C40 C47	1005	MCCT-8		1 NBPS Viliny Virtex-						
C48	10uF	MCCT-B	CAP POLAR	1 NBPR Xiliny Virtex-I		E Marine I		Barran C.		Editor -
C/9	1005	MCCT-8		1 NBPS Viline Virtex.		<u>∎</u> <u>m</u> enu	<u>A</u> od	<u>h</u> emove	<u>_ok</u> <u>Ya</u>	lidate
C50	10uE	MCCT-B	CAP POLAR	1 NBP8 Xiliny Virtex-I						
051	220 E/4V	MCCT-D	CAP POLAB	1 NBP8 Xilinx Virtex-I						
C52	220uE/4V	MCCT-D	CAP POLAB	1 NBP8×ilins Virtes-						
C53	220uF/4V	MCCT-D	CAP POLAB	1 NBP8 Xilinx Virtex-						
C54	220uF/4V	MCCT-D	CAP POLAR	1 NBP8 Xilinx Virtex-						
C55	220uF/4V	MCCT-D	CAP POLAR	1 NBP8 Xilinx Virtex-						
C56	220uF/4V	MCCT-D	CAP_POLAR	1 NBP8 Xilins Virtex-I						
C57	220uF/4V	MCCT-D	CAP_POLAR	1 NBP8 Xilinx Virtex-I					2	
C58	220uF/4V	MCCT-D	CAP_POLAR	1 NBP8 Xilinx Virtex-I					-	
C59	10uF	MCCT-B	CAP_POLAR	1 Virtex_II_PR0_PSU.Sc						
C60	10uF	MCCT-B	CAP_POLAR	1 Virtex_II_PR0_PSU.Sc						
C61	100uF/6V/1	50 MCCT-D	CAP_POLAR	1 Virtex_II_PRO_PSU.Sc	-					
C62	22uF/10V/Y	'5\ CC3225-1210	CAP	1 Virtex_II_PRO_PSU.Sc						
C63	22uF/10V/h	′5\ CC3225-1210	CAP	1 Virtex_II_PR0_PSU.Sc	:					
C64	470pF	CC1608-0603	CAP	1 Virtex_II_PRO_PSU.Sc	~					
					-					
								Accept Chang	es (Create ECO)	Close

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.

Filter		• ×
Limit search to All objects Selected objects Non selected objects Whole Library	Find items matching these criteria: IsPin	Objects passing the filter Select Zoom Objects not passing the filter Deselect Mask out
	💦 Helper 👷 Favorites 🙆 History	Apply

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

Object Kind	×1	Y1	Orientation	Name	Show Name	Pin Designator	Show Designator	Electrical Type	Hide	Hidden Net Name
Pin	0	-60	180 Degrees	GND	~	3	~	Power		
Pin	80	-60	0 Degrees	BTN	~	5	~	Power		
Pin	0	-10	180 Degrees	EN	~	7	~	Input		
Pin	0	-20	180 Degrees	F ADJ	✓	1	~	Input		
Pin	80	-10	0 Degrees	SYNC	~	2	~	Passive		
Pin	80	-40	0 Degrees	\T\0\	~	6	~	Power		
Pin	80	-30	0 Degrees	TO	~	4	~	Power		
Pin	0	-40	180 Degrees	IN+	~	8	~	Power		
1										2

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	Н		Γ
F ADJ	TRUE	1	TRUE	0 Degrees	
SYNC	TRUE	2	TRUE	Input	
GND	TRUE	3	TRUE	OpenCollector	
ТО	TRUE	4	TRUE	Input	
RTN	TRUE	5	TRUE	OpenCollector	Γ
10/11	TRUE	6	TRUE	OpenCollector	Γ
EN	TRUE	7	TRUE	0 Degrees	Γ
IN+	TRUE	8	TRUE	OpenCollector	
					9 T

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

List										• ×
<u>Edit</u> <u>selecter</u>	d objects	from	current compo	<u>nent</u> Inclu	ide <u>all types of ob</u>	<u>ijects</u>				
Object Kind	X1	Y1	Orientation	Name	Show Name	Pin Designator	Show Designator	Electrical Type	Hide Hidden Net 1	Name
Pin	50	-50	0 Degrees	1	✓	1	×	Passive	Edit	1
Pin	50	-40	0 Degrees	2	✓	2	×	Passive	Copy	
Pin	50	-30	0 Degrees	3	Image: A start of the start	3	✓	Passive	Paste N	
Pin	50	-20	0 Degrees	4	Image: A start of the start	4	~	Passive	Show Children	
Pin	50	-10	0 Degrees	5	Image: A start of the start	5	✓	Passive		
Pin	50	0	0 Degrees	6	Image: A start of the start	6	✓	Passive	Zoom Selected	
Pin	50	10	0 Degrees	7	Image: A start of the start	7	✓	Passive	Report	
Pin	50	20	0 Degrees	8	Image: Second	8	✓	Passive	Report Selected	
									Select All Select Column	
S Objects (8 S	elected)							-	Choose Columns	>

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

List												
<u>Edit</u> <u>selected</u>	Edit selected objects from current component Include all types of objects											
Object Kind	X1	Y1	Orientation	Name	Show Name	Pin Designator	Show Designator	Electrical Type	Hide	Hidden Net Name		
Pin	50	-50	0 Degrees	F ADJ	×	1	✓	Input				
Pin	50	-40	0 Degrees	SYNC	✓	2	✓	Passive				
Pin	50	-30	0 Degrees	GND	✓	3	✓	Power				
Pin	50	-20	0 Degrees	TO	✓	4	✓	Passive 🔨				
Pin	50	-10	0 Degrees	BTN		5	✓	Power				
Pin	50	0	0 Degrees	\T\0\	✓	6	✓	Power				
Pin	50	10	0 Degrees	EN	✓	7	✓	Input				
Pin	50	20	0 Degrees	IN+		8	✓	Power				
								_	_			
<										>		
8 Objects (8 S	8 Objects (8 Selected)											

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

Software, hardware, documentation and related materials:

Copyright © 2005 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, NanoTalk, Nexar, nVisage, P-CAD, Protel, CircuitStudio, 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.