SCH Inspector

Modified by Rob Evans on Jun 2, 2015

Parent page: [Sch Panels](#)

SCH Inspector	▼ ✕
Include only _Parts_ from _current document_	

Kind	
Object Kind	Part
Design	
Owner Document	SL_LCD_SW_LED_2E.SchDoc
Graphical	
X1	<...>
Y1	430
Orientation	90 Degrees
Locked	☐
Mirrored	☐
Display Mode	Normal
Show Hidden Pins	☐
Show Designator	☑
Object Specific	
Description	3216x4 Resistor Array
Lock Designator	☐
Lock Part ID	☐
Pins Locked	☑
File Name	*
Configuration	*
Library	SpiritLevel_2E.SchLib
Symbol Reference	MNR34
Component Designator	<...>
Current Part	
Part Comment	4K7
Current Footprint	MNR34T
Component Type	Standard
Database Table Name	
Use Library Name	☑
Use Database Table Name	☑
Design Item ID	MNR34
Vault	
Item	
Revision	
Symbol Vault	
Symbol Item	
Symbol Revision	
Parameters	
✕ AEMS Code	1300330
Add User Parameter:	

3 object(s) are displayed in 1 document(s)

Manually select Schematic objects to populate the _SCH Inspector_ panel with design objects to be viewed or edited.

Summary

The _SCH Inspector_ panel enables you to interrogate and edit the properties of one or more design objects in the active schematic document (or all open schematic documents). The panel can be used to make changes to multiple objects of the same kind, from one convenient location.

Panel access

To display the *SCH Inspector* panel, click the **Inspector** button in the **Schematic** group under the **View** tab (**View | Schematic | ▦**), or use the **F11** keyboard shortcut

Panels can be configured to be floating in the editor space or docked to sides of the screen. If the *SCH Inspector* panel is currently in the group of docked *Workspace* panels on the left, use the **SCH Inspector** tab located at the bottom of the panels to bring it to the front.
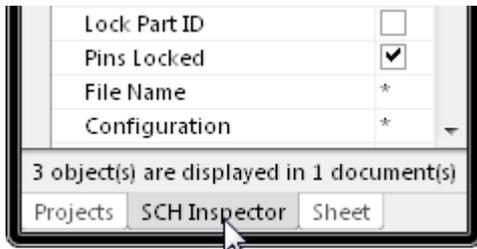

Defining panel display scope

When a large number of objects have been selected in the editor workspace, there is often a need to edit the properties of certain object types only, without losing or having to alter your selection. This can be achieved using the underlined controls at the top of the panel, to essentially define a 'display scope' for the panel:

Use the the left-hand underlined control to set the type of objects that can be displayed. Clicking on the control will reveal a selection pop-up.
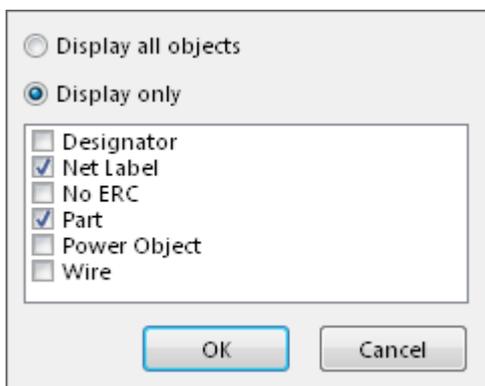
Use the pop-up to choose which object types to include in the panel for display and editing - either all objects or specific objects. To choose one or more specific object types, enable the **Display only** option and then enable the check box next to the required object(s) in the list beneath. The list will only contain those object types currently selected in the main workspace.

When enabling specific object types for display, the control will reflect the choice by listing the enabled types, separated by commas.

Note that if you have enabled display filtering by one or more object types, a control will appear at the bottom right corner of the panel, allowing you to quickly toggle between display of all objects ✖️ and only those specified by the **Display Only** option 🔽.

Click on the right-hand underlined control (shown above as 'current document') to choose from the following options:

- **current document** - only display target objects from the active schematic document in the design editor window
- **all open documents** - display target objects from all open schematic documents, irrespective of the parent project they belong to
- **open documents of the same project** - display target objects from all open schematic documents that are source documents of the same parent project

Note: With respect to the last two options for this control, schematic documents that are open, but hidden, will not be considered when displaying objects.

Inspecting object attributes

Clicking on a single object in the design editor window will select the object and display attributes associated to it in the *SCH Inspector* panel. Information is displayed under the following common collapsible sections:

Kind

This section of the panel contains one entry only, relating to the kind of design object that is being 'inspected'. For example, clicking on a component will display the entry 'Part', while clicking on a component designator will display the entry 'Designator', and so on.

Design

This section of the panel also contains one entry only, relating to the owner document upon which the object has been placed.

Graphical

This section of the panel contains graphical attributes of the selected object. Attributes here may include the location of the object, its orientation and colors used in its display.

Object Specific

This section of the panel contains attributes specific to the object being inspected and which are not graphical attributes. For example, selecting a component will display a range of attributes that are specific to a component:

- Description
- Lock Designator
- Lock Part ID
- Pins Locked
- File Name
- Configuration

- Component Designator
- Current Part
- Part Comment
- Current Footprint
- Component Type
- ...etc

Other selected objects will have different attributes displayed. For example, selecting a Power Object (such as a GND object) will only offer a Text attribute as specific to that object.
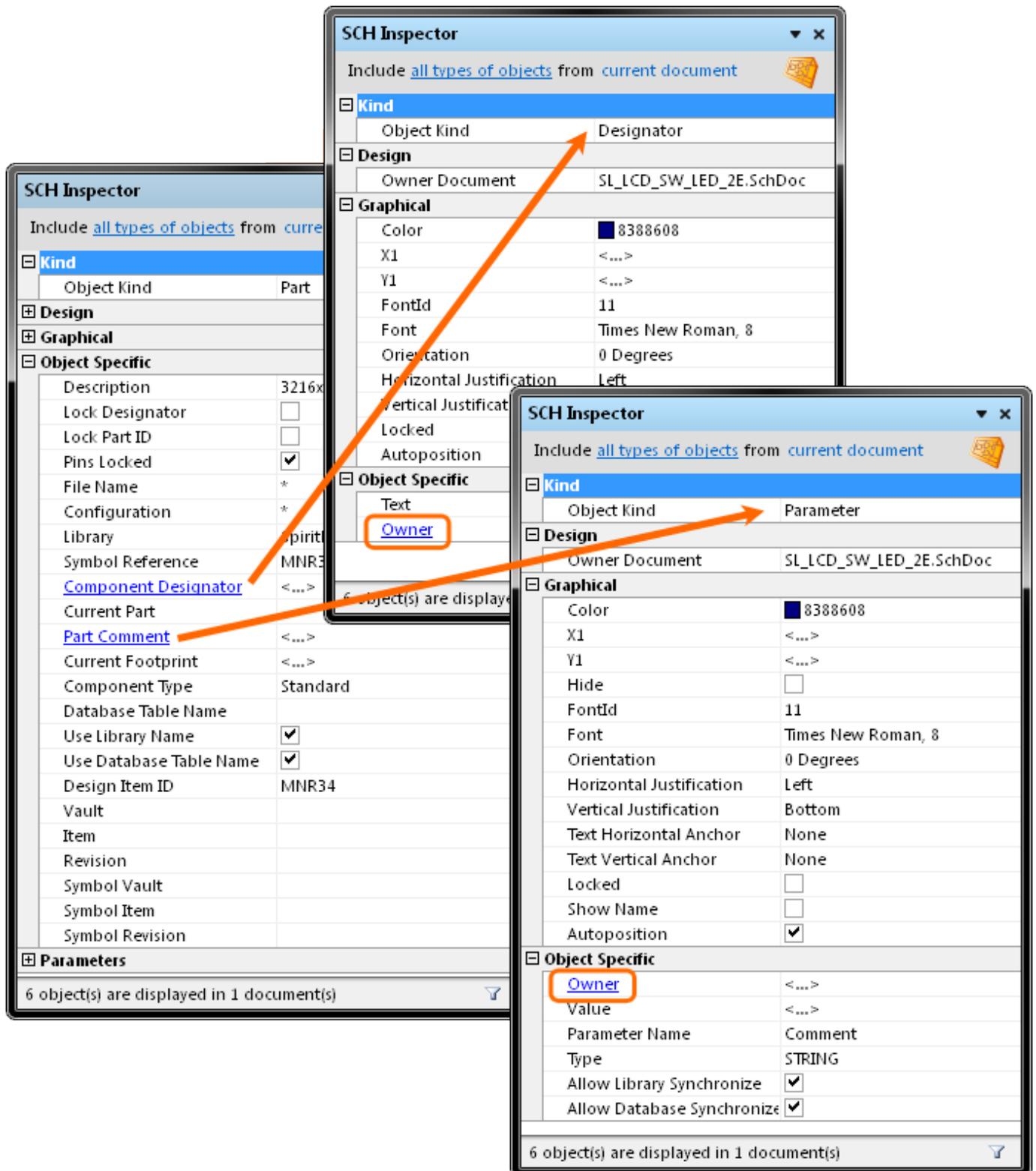
Parameters

For an object to which parameters can be 'attached', this section allows Parameters to be inspected, added and deleted for single or multiple objects.

Editing object attributes

The attributes of a selected object can be edited by modifying the relevant entry in the panel. The change will take effect once you click outside of the field you are editing. This is one of the advantages of using the panel to edit object properties - the panel will remain open, allowing a sequence of attributes to be changed as required, without having to close and reopen a properties dialog each time.

If the object being inspecting has child objects associated with it, they will appear in blue in the **Object Specific** section (rather like a hyperlink). Click on these entries to display the attributes pertaining specifically to these child objects. For each child object, an link will be available back to the parent object – the 'Owner' link in the Object Specific section of the panel.

Use the panel's links (in blue) to drill down to edit child object parameters, then use the **Owner** link to return to the parent object.

Another advantage of using the panel for editing is that multiple objects can be modified from the one place without having to edit, through dialogs, one object at a time. Selected objects can be of the same or differing type. Those attributes that are common to all objects in the selection will be displayed in the panel. Common attributes that have differing values between objects will be displayed as <...>. Edit the attributes as required – the changes made will be conveyed instantly to each object in the selection.

By using filtering, a query (an expression for the filter) can be applied to target a specific group of objects in the design, and the *SCH Inspector* panel then used to edit the attributes for these multiple

objects, directly.

Editing attributes with numeric values

For a numerical-based attribute of a selected object, the simplest modification to that attribute's value is made by typing a new value to replace the existing one. The plus and minus operators can be used to specify the value's sign. A value entered without a specified sign is assumed to be positive. Therefore entering 20 is the same as entering +20.

Specific units of measurement can be entered for a value. The software will convert the value into the current units defined for the document. If no units are specified, the default units set for the document will be used.

Batch replacement of string-based attributes

There are occasions when it's desirable to modify a string-based attribute that is common to multiple selected objects in the workspace. For example to rename selected data bus net label objects from D1, D2, D3, etc, to Data1, Data2, Data3, etc. To perform this type of batch replacement, the use of string substitution syntax is supported in the panel.

A string substitution entry is enclosed in braces and has the form:`{oldstring=newstring}`

An entry of this form causes all occurrences of `oldstring` found in the attribute's value to be replaced with `newstring`. In the case of the data bus net labels, you would enter `{D=Data}` in the value field for the **Text** attribute.

To replace multiple, differing string portions in the same target string, type multiple substitution entries, each enclosed in its own set of curly braces. For example, consider the following net labels, associated to the output wiring of a WB_PRTIO component, configured to have four, 8-bit I/O ports:
```
XPort8A_Out[7..0]
XPort8B_Out[7..0]
XPort8C_Out[7..0]
XPort8D_Out[7..0]
```

Now consider having placed a second WB_PRTIO component, configured to have four, 32-bit I/O ports. Instead of placing individual net labels of a similar fashion on each output wire of each port in this second device, all four of the above net labels could be selected, copied, then pasted onto the corresponding wires of the new port device and then renamed. It is in this last operation that batch string substitution comes into play.

If the intended format of the new net labels is to be `YPort32x_Out[31..0]`, where x represents the port (`A, B, C` or `D`), then with the four net labels selected, the *SCH Inspector* panel can be used to enter the following into the value field for the **Text** attribute:
`{X=Y}{8=32}{7=31}`

The software takes this entry and effectively performs a batch substitution - substituting for the first expression, then the second, and so on.

Smart Editing of string-based attributes

The *SCH Inspector* panel offers further support for string modification through its **Smart Edit** feature.

Simply click on a shared attribute of the selected objects, whose value is a string. A ··· button will become available at the far right of the cell. Click on this button to access the *Smart Edit* dialog.
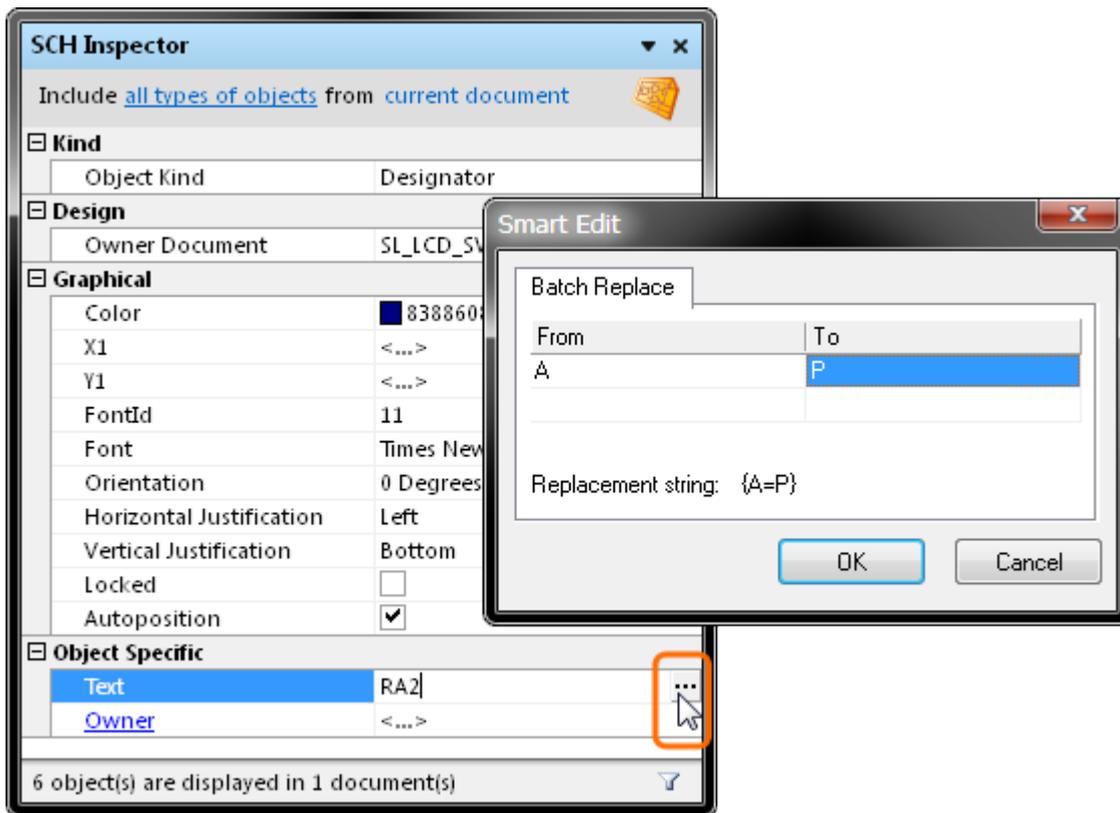

The dialog offers two methods for performing string modification, accessed from the **Batch Replace** and **Formula** tabs respectively.

The **Batch Replace** tab in the *Smart Editor* dialog provides simple straightforward substitution, along the lines of the string substitution discussed previously (but without having to enter the curly braces). Click inside the **From** field and enter the portion of the current string to be replaced. Then click inside the **To** field and enter the string to be used as the replacement. The familiar string substitution syntax is displayed at the bottom of the tab.

For example, consider several component designators that have the prefix RA that needs to be changed to RP. In this case, select the components, click on the **Component Designator** attribute in the panel and access the *Smart Edit* dialog. Then on the **Batch Replace** tab, enter RA in the **From** field and RP in the **To** field (the replacement string is therefore {RA=RP}). Note that just entering A and B in the **From/To** fields is equivalent, as the P character is unchanged.

After clicking **OK**, the component designators on the schematic will be modified accordingly.
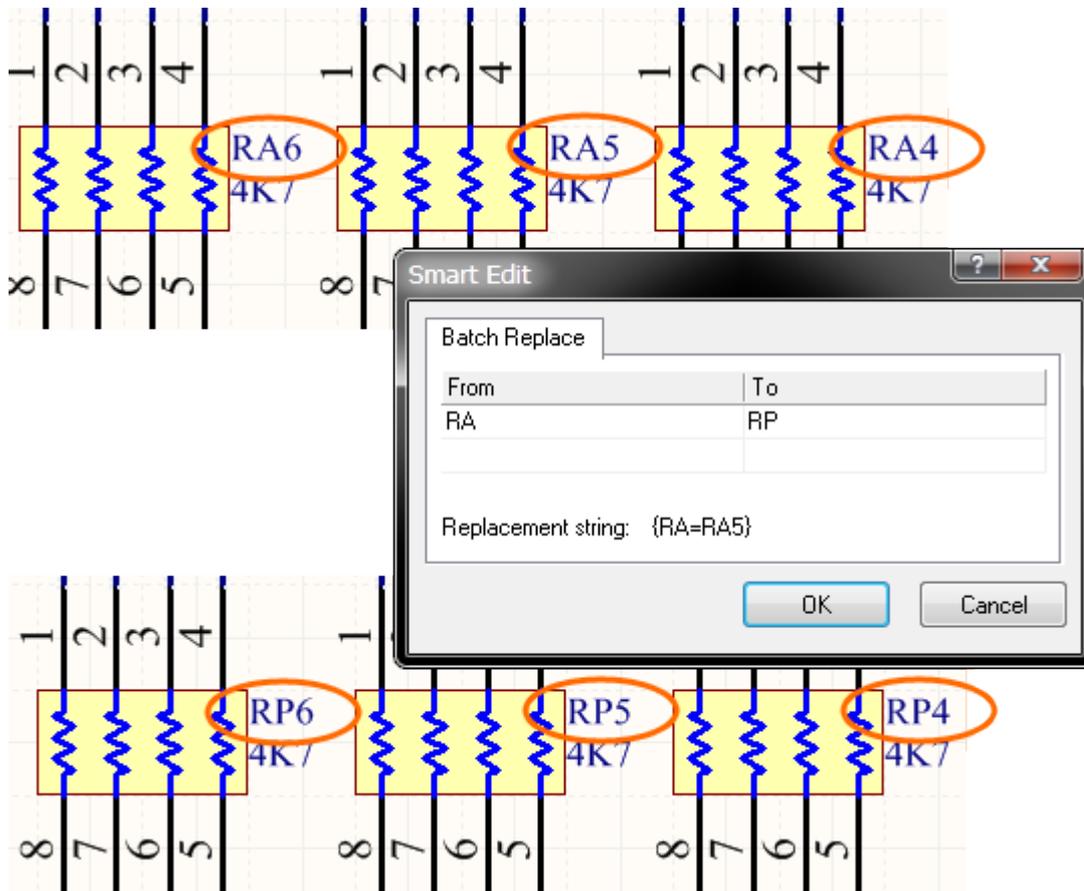
As with basic string substitution, the **Batch Replace** tab provides for replacement of multiple, differing string portions in the same target string.

Enter the various substitutions as distinct **From-To** entries. Consider the previous WB_PRTIO example (see [Batch replacement of string-based attributes](#), above), where strings of the form XPort8x_Out[7..0] need to be changed to YPort32x_Out[31..0]. In this case, you would enter three distinct substitution entries on the **Batch Replace** tab, as shown here.
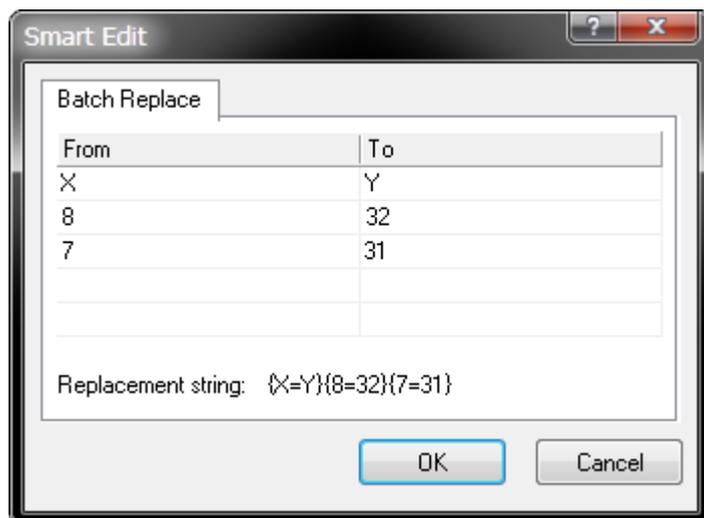
Editing/Adding Parameters

For an object to which parameters can be 'attached', an additional section is displayed in the *SCH Inspector* panel - **Parameters**.
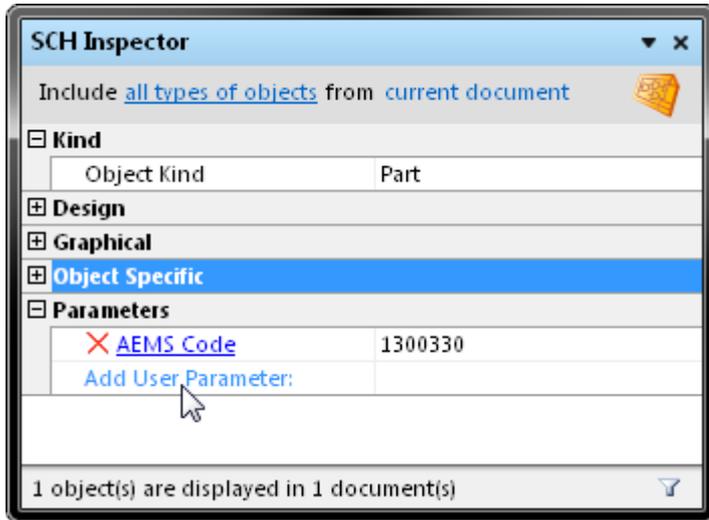
For a single selected object, this section lists all of the parameters currently defined for that object. For multiple selected objects, the section displays only those parameters that are common to all objects in the selection.

Click on the hyperlink for a parameter name to display and edit the properties for that parameter directly from within the panel. Click on the 'Owner' link to go back to editing the parent object to which the parameter belongs.

> Note that when editing multiple objects possessing the same parameter, you can switch between editing the parameter properties (which will be applied to all instances of the parameter in the selection) and the properties of the parent objects, without losing the selection.

Any number of user-defined parameters can be added from within this section of the panel. With respect to components, this allows you to define anything from component ratings, to stock information, to purchasing data, or revision history.

Simply enter the value for the new parameter in the field to the right of the **Add User Parameter** entry. When you press **Enter** or click out of the field, the *Add new parameter to n object(s)* dialog will appear, where **n** represents the number of selected design objects to which this new parameter will be simultaneously added.
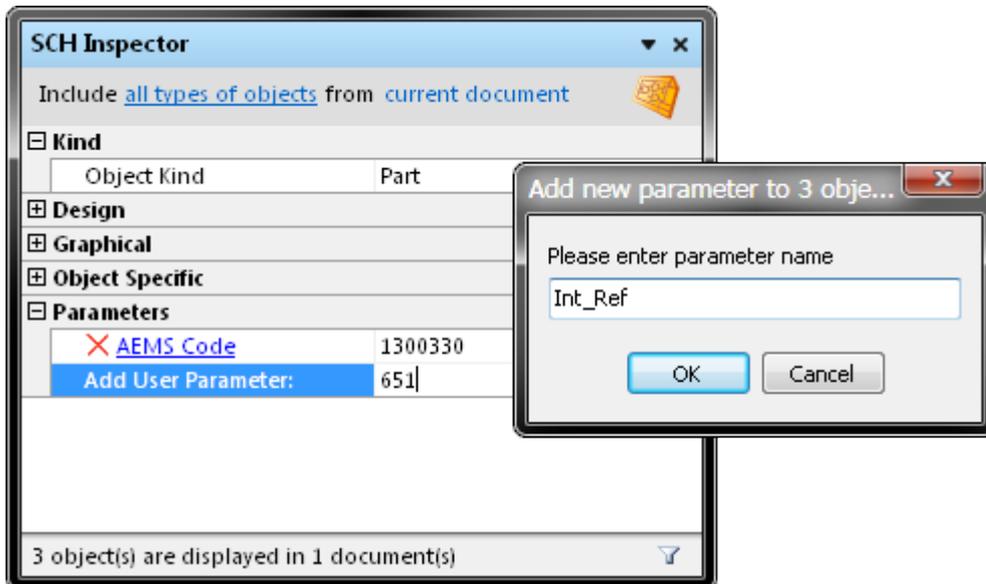
Change the name for the new parameter as required and click **OK** - the parameter will be added to the list of parameters for the selected object(s).

To remove a parameter from the selected object(s), click on the ✕ symbol to the immediate left of the parameter name. A dialog will appear prompting for confirmation to proceed with the deletion.
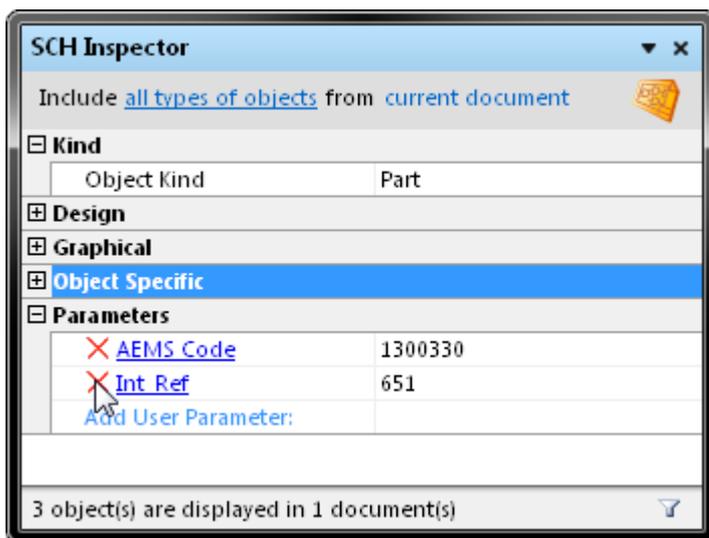
Notes

- Pressing the **F11** key will toggle the visibility of the panel in the workspace.
- Information will only appear in the panel when one or more objects have been selected in the design editor window.
- If you have defined the display scope for the panel to display specific object types, this scope will remain when the selection is cleared in the main workspace. If you subsequently select an object that is not part of this previously defined scope, the panel will display a control at its center – 'Click here to display all objects' – which, when clicked will reset the Include portion of the display scope to all types of objects.
- You can use the **Up/Down Arrow** keys to pass up and down the list of attributes in the panel respectively.