Module 10: Global Editing

Module 10: Global Editing

Module Seq = 10

10.1 Editing Multiple Text Objects

One of the powerful features of Altium Designer is the data editing system. The Schematic Editor includes a feature that provides a very efficient mechanism to edit text strings in your schematics. The reason this is powerful is that the same process using find similar objects would take three separate operations on netlabels, ports and sheet entries.

10.1.1 Find and Replace Text

You can perform complete or partial substitutions on text using the following methods.

- To target a section of a string, include the * or ? wildcards as appropriate. In Figure 1, the combination of the Text to Find, the Sheet Scope and the Restrict to net Identifiers option will result in any net label, port, sheet entry or power port whose net attribute starts with the letters RB being found.

- If you wish to replace the entire contents of a text field with a new value, simply enter the new value in the Replace With field.

- Partial string substitutions can be performed using the syntax {oldtext=newtext}. This means you can change a portion of the current string (oldtext) to a new string (newtext). In Figure 1, the letters RB will be replaced with LCD. Any other characters in each found net identifier will remain the same. For example, the following changes would occur:
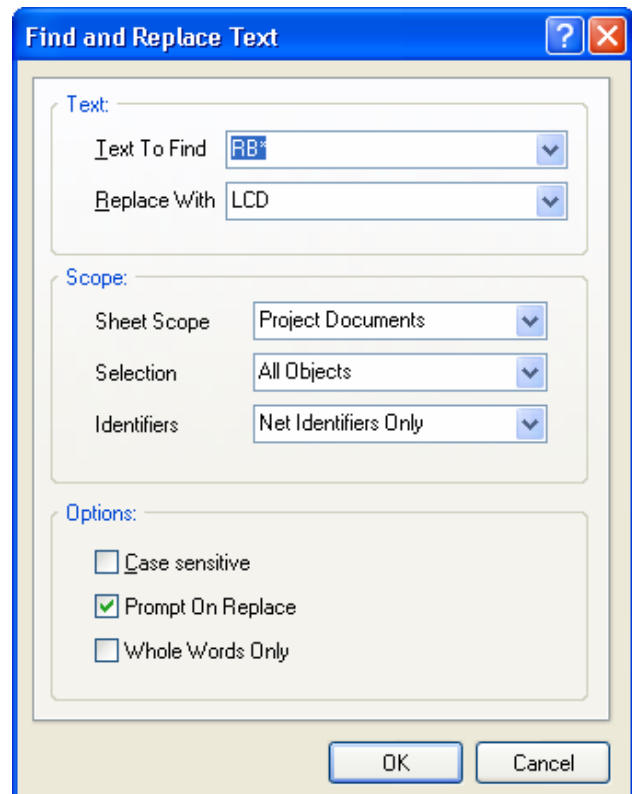
Figure 1. Performing a partial string substitution

Before Find and Replace	After Find and Replace
RB1	LCD1
RB200	LCD200
RBout	LCDout
RB_CLK	LCD_CLK
RB[0..7]	LCD[0..7]

10.1.2 The Parameter Manager

- The Parameter Manager allows you to control all your parameters in one single editor. Open the Parameter Manager by selecting **Tools » Parameter Manager** from the menus. User-defined parameters can be added, removed or renamed in the Parameter Manager. You can modify the values of system-level parameters but these cannot be added, removed or renamed.

- You can select which parameters will be included in the *Parameter Table Editor* by limiting the types of parameters you wish to use in the *Parameter Editor Options* dialog. For example, you can exclude all system parameters, or only use document-level or part parameters.
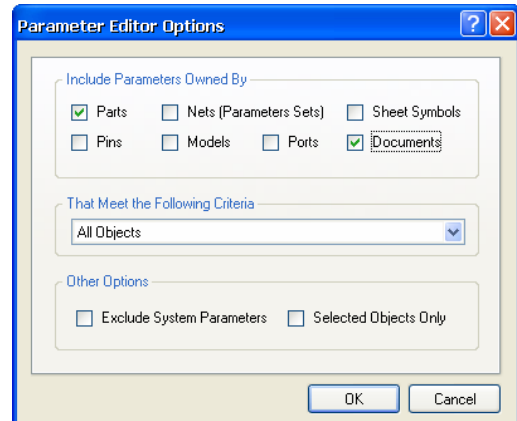
- Changes to the values or names of parameters are made in the *Parameter Table Editor* and then an ECO is generated to execute the changes in the design or schematic library.
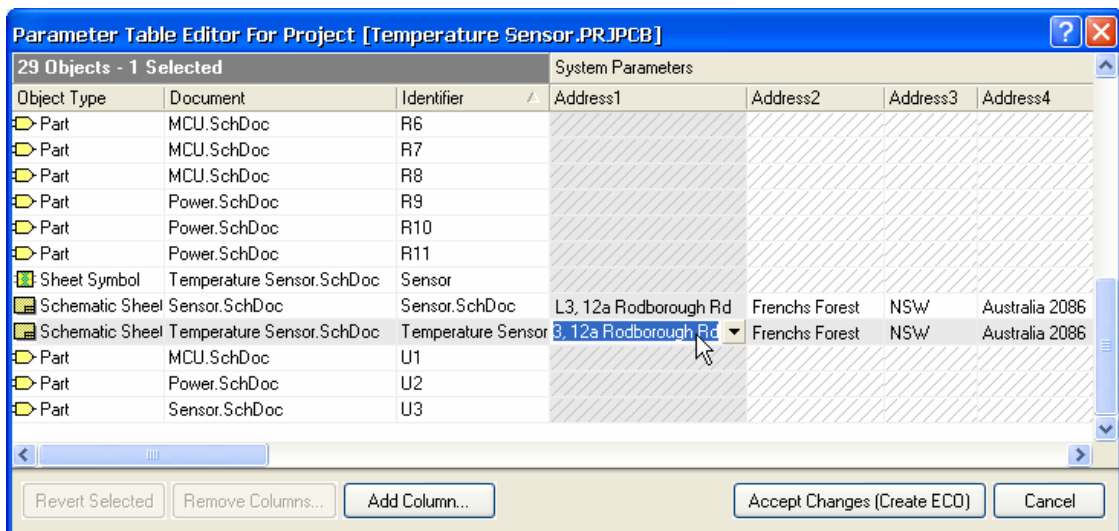
Figure 2. Parameters can be modified using the Parameter Table Editor.

Tips for using the Parameter Manager

- Editing in the *Parameter Table Editor* is similar to editing in an Excel spreadsheet. For example, press **F2** or SPACEBAR to edit, type in the value or select it from a drop-down list, if available, and then press **Enter**. Use the arrow keys to move through the spreadsheet.

- You can edit multiple instances of the same parameter value by selecting the cells, right-clicking and selecting **Edit** for the drop-down menu. Type in the new value and press **Enter**. Right-click and choose **Revert** to undo changes to selected cells.

- Cells are highlighted in the *Parameter Table Editor* according to whether the parameter exists or has current values.

4.7K	the object possesses the parameter and the string entry in the field is its value.
	the object possesses the parameter, but it currently has no value
	the object does not possess the parameter.

- When you modify a parameter, markers in the right-hand top corner of the cell indicates what changes will be made.

10uF ◥	the value assigned to the parameter will be changed.
＋	the parameter will be added to the object but, in this case, no value will be assigned.
／／／／／／／ ─	the parameter will be removed from the object.

- Note that any changes made within the table are virtual changes that will not be implemented until the execution of an Engineering Change Order.
- Press **F1** in the Parameter Manager dialogs for more information.

10.1.3 Exercises – Using the Parameter Manager

10.1.3.1 Adding new parameters using the Parameter Manager

1. This exercise adds a new parameter, named Part Number, to all components in the design. If it is not open, re-open the 4 Port Serial Interface example project in the Reference Designs.

2. In the Schematic Editor with the required schematic documents open, select **Tools » Parameter Manager** to display the *Parameter Editor Options* dialog.

3. In the **Include Parameters Owned By** section of the dialog clear the **Documents** checkbox, enable the **Parts** checkbox, then select **All Objects** as the criteria. Click **OK**.
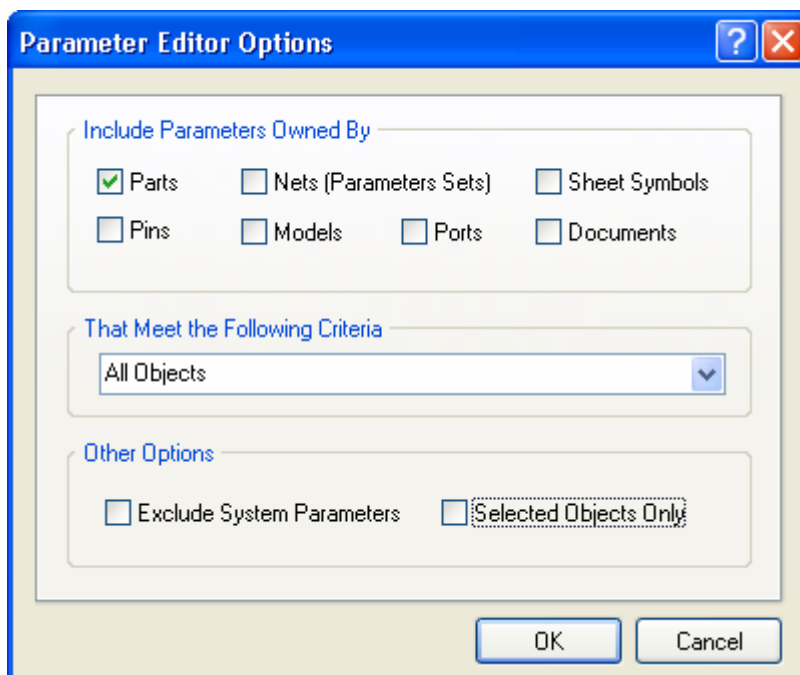
4. In the *Parameter Table Editor* dialog, click on **Add Column**. The *Add Parameter* dialog displays. Type in a new parameter name, e.g. `Part Number`, and enable **Add to all objects**. Click **OK** to create the new parameter column. Enter values in the new Part Number column as required. Click on **Accept Changes (Create ECO)**.

5. Click on **Validate Changes**. If the validation is successful, click on **Execute Changes**.

6. When the changes have been executed, click **Close**. The new parameters are added to the components in the schematic. These can be checked by double-clicking on the components in the schematic document to display the *Component Properties* dialog. The new parameter is added to the Parameters list.
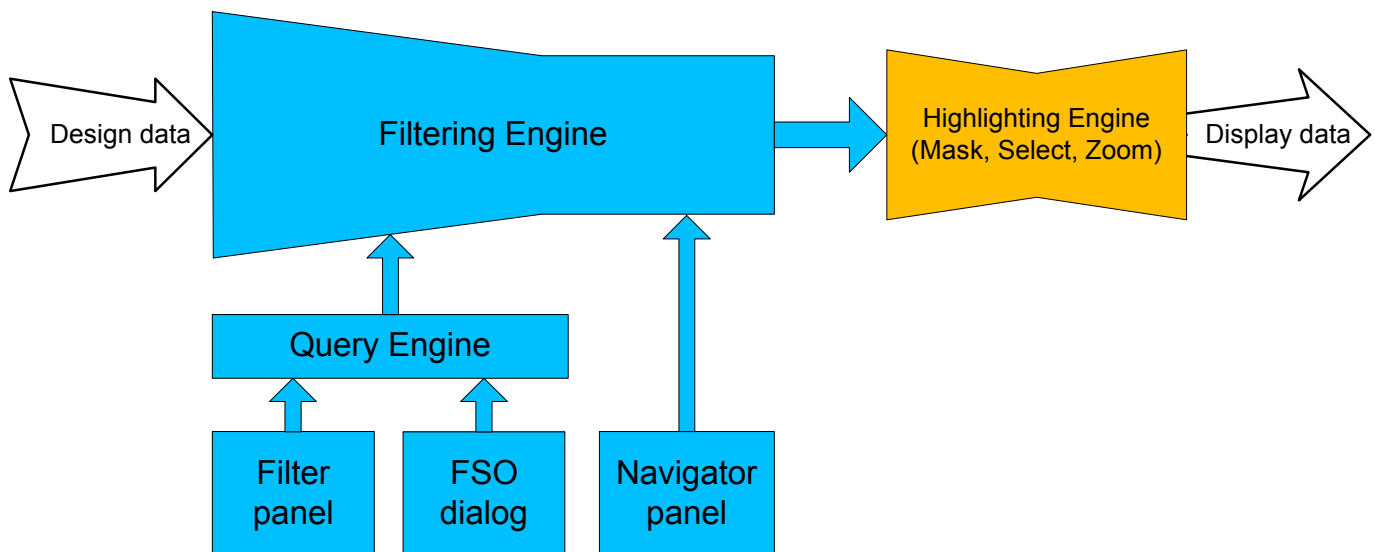
10.2 The Data Editing System

Figure 3. Diagram of the filtering/highlighting system

One of the greatest challenges you face as a designer is managing the large amounts of design data that is created during the design process. To facilitate this, Altium Designer has a powerful data editing system. This system allows you to manage, find and edit design data in a variety of ways.

To provide flexible and appropriate methods of editing data, three alternate views of the data can be used to access and edit design objects:

- The traditional graphical view

- The **Inspector** panel (press **F11** to toggle it on and off)

- The **List** panel (press **Shift+F12** to toggle it on and off)

- The **Filter** panel (press **F12** to toggle it on and off)

The **Inspector** displays the attributes of the currently selected object(s), with the total number selected being listed at the bottom. Note that the **Inspector** can be used to edit different kinds of objects simultaneously.

The **List** panel gives a spreadsheet-like, or tabular list of objects in the schematic sheet or PCB workspace. Individual or multiple cells can be edited in the **List** panel.

A powerful filtering engine is used to control the amount of data that is presented for editing in all three views. Data can be filtered using the *Find Similar Objects* dialog, the PCB editor panel, or by writing a query in the **Filter** panel. Figure 3 shows a diagram of the data editing system.

The **Filter** panel is used to type in a query that filters the entire data set, reducing both the graphical display and the **List** panel to display only those objects that satisfy the query. In the graphical display this can be shown by the fading of objects that have been filtered out (and are no longer editable).

One of the powerful features of this data editing system is the ability to edit multiple objects simultaneously. The basic approach to use the data editing system is to:

Select the required objects for editing

Inspect the objects

Edit the object attribute(s).

10.2.1 Finding and Selecting Objects

10.2.1.1 Using the PCB Panel

The **PCB** panel can be used for browsing objects in a PCB. To open the panel click **PCB » PCB** in the panel control buttons down the bottom right of the workspace.

- The options at the top of the panel control how the chosen object(s) will be highlighted:
  - **Mask:** this option fades all objects except those of interest. While masked objects are still visible, they can not be edited. Use the Mask Level control at the bottom right to control the amount of fading.
  - **Select:** select the target object(s). Selected objects are highlighted using various white shading techniques.
  - **Zoom:** zoom in to fit the highlighted objects in.
  - **Clear existing:** enable this to automatically clear any existing highlighting whenever you choose another object.
- Use the CTRL+Click combination to highlight multiple objects.
- Right-click in the panel to control which primitive kinds are displayed, this is very handy for excluding certain object kinds.
- Click the **Clear** button Clear to clear all masking and selections.
- Highlighting results are displayed in all three views – graphical, List and Inspector (if Selection is enabled).
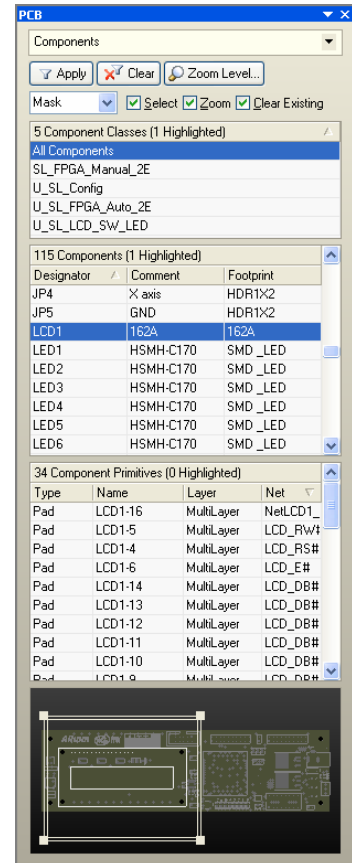

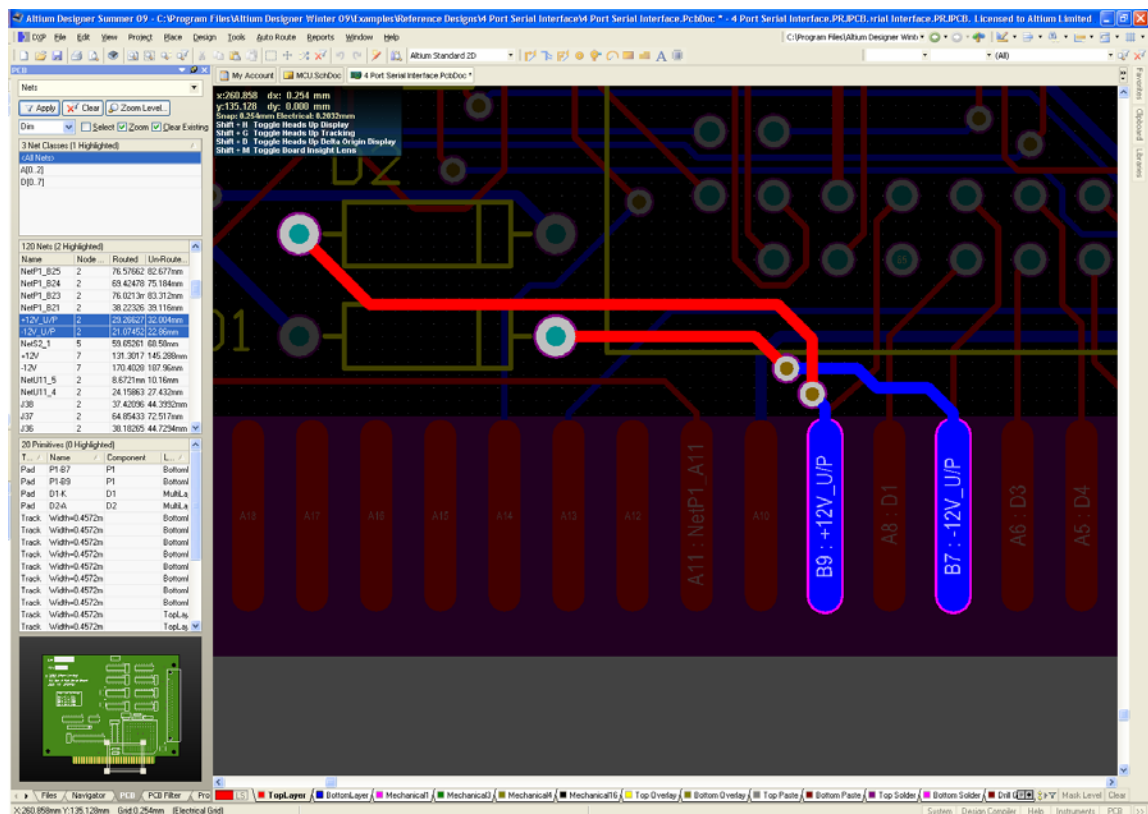*Figure 4. Use the PCB panel to find and select objects.*

Figure 5. Using the panel to highlight two nets. Note that all other objects have been faded (masked).

10.2.1.2 Using the *Find Similar Objects* dialog

The panel is ideal when working with group-type objects like components and nets. When you are working at a primitive object level, it can be more efficient to use the *Find Similar Objects* dialog. This dialog works in both PCB and Schematic.

- To launch the *Find Similar Objects* dialog, right-click on an object of interest, and select **Find Similar** from the floating menu.

- The dialog will appear, listing the attributes of that object. Next to each attribute is a drop down. Set this if you wish to use it as a matching criteria.

- Clicking **Apply** will run the search for matching objects but will leave the dialog open. Clicking **OK** will close the dialog and run the search.

- This will select all objects that match the find criteria. Figure 6 shows the *Find Similar Objects* dialog configured to find and select all PCB text strings that are component comments.

- Enable the **Run Inspector** check box to automatically launch the Inspector, where you can edit any attribute(s) of the found and selected objects.
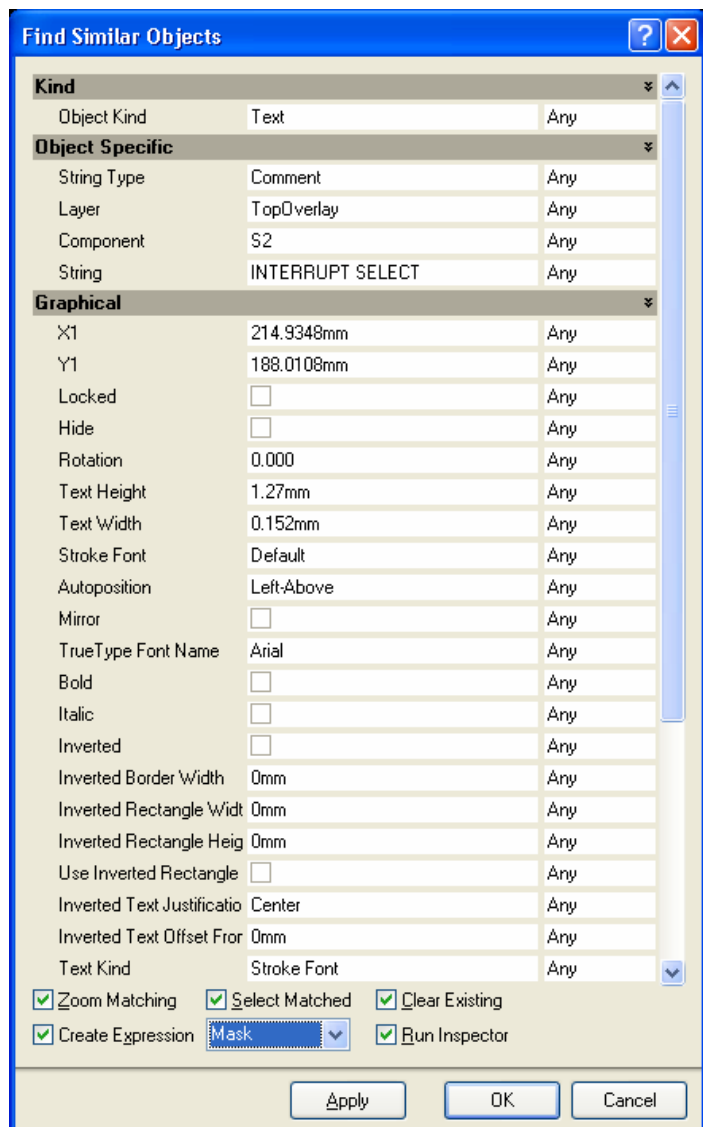

Figure 6. Using the Find Similar Objects dialog to highlight all component comments.

10.2.1.3 Using the Filter Panel

Underlying the techniques for finding objects described so far is a powerful data filtering engine. You can also access this engine directly by writing a query to describe the objects that you wish to target.

- Press **F12** to toggle the **Filter** panel on, where you write the query.

- A Query is an instruction, written using query language keywords. For example, entering the query `IsComment or IsDesignator` in the PCB editor List panel will reduce the contents of both the graphical display and the List panel to only display the component designator and comment strings on the PCB.

- For a complete list of query keywords, click the Helper button or refer to the document *TR0110 Query Language Reference.PDF* in the Altium Designer help directory. When the cursor is on a keyword, press the **F1** key in the Query Helper dialog for a complete description of that keyword. Press **F1** when the cursor is within an arithmetic operator for information on the operators.

- Refer to the document, *AR0129 An Insiders Guide to the Query Language.PDF* for detailed information on writing queries.
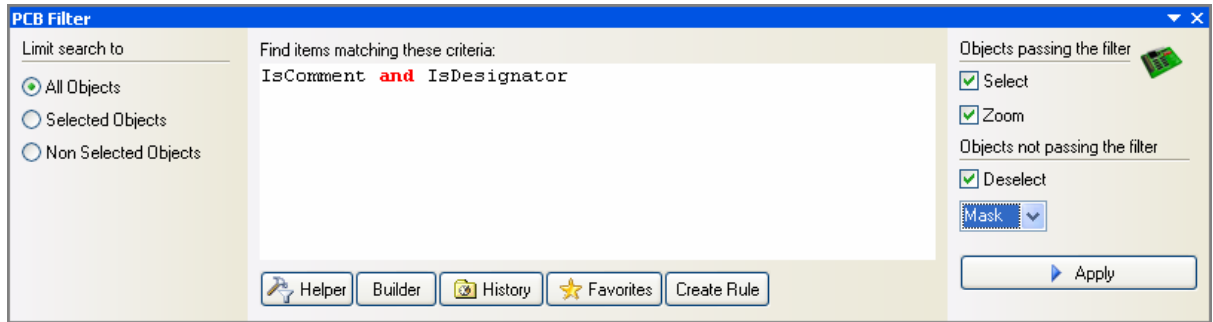
Figure 7. Use the Filter panel to query the design data and access specific objects.

10.2.1.4 Using the List Panel

The List panel can also be used to examine and edit the properties of objects. Use the List panel when you want to examine/compare attributes, edit only some of the objects or even paste in text from external editors like Excel.

- Press **Shift+F12** to toggle the **List** panel on/off.

- An individual cell in the List can be edited, press the SPACEBAR or right-click and select **Edit**.

- Multiple cells can be edited simultaneously, select them, press the SPACEBAR, type in the new value and press ENTER on the keyboard.

- Blocks of cell data can be copied and pasted to/from a spreadsheet.

- For group-type components, such as components or nets, you can include their primitive parts (child objects) by right-clicking and choosing the appropriate **Show Children** option.

- When there are multiple object types displayed, only attributes that are common to all are displayed. You can remove objects from display in the List panel, select those you wish to keep, right-click and choose **Remove Non-Selected** from the menu.

- Column display is managed by right-clicking on the column headers and selecting **Choose Columns**.

Figure 8. Using the List to examine / edit all designator and comment strings.

Note: Make sure you set the list panel to edit mode. By default it's in view mode as shown in Figure 8. This is located in the top left corner of the list panel

10.2.1.5 Inspecting and editing the selected objects

The Inspector panel is used to examine the properties of and edit the currently selected objects. Use the Inspector when you want to apply an edit to all the selected objects.

- Press **F11** to toggle the Inspector panel on/off.

- The Inspector can be used to examine 1 or many objects.

- The set of selected objects can be built up in many ways, including; manually, by writing a query, or using the Find Similar dialog.

- Dissimilar objects can be selected and edited, only their common attributes will be available for editing.

- After changing a value in the Inspector, press ENTER to apply it.

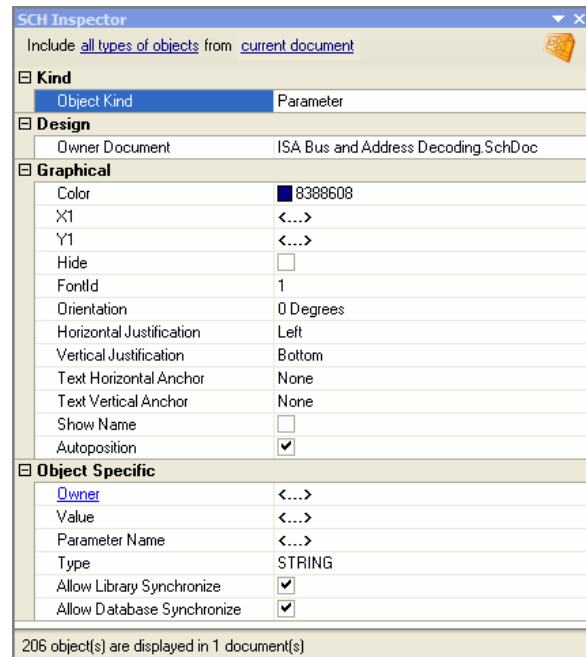- String substitutions can be performed in the Inspector panel.

Figure 9. The Inspector displaying the properties of the selected designator and comment strings.

10.2.2 Exercises – editing objects

This exercise will demonstrate different ways of changing the width of the component overlay tracks and arcs and the height of component text.

10.2.2.1 Changing the visibility of the component Comment strings

1. Locate component S1 on the 4 Port Serial Interface PCB.

2. Right-click on the comment string and select **Find Similar** from the floating menu.

3. The *Find Similar* dialog appears, presenting the attributes of the object clicked on. Note that the **String Type** attribute has a value of Comment. Set the match by setting for this attribute to **Same** (as shown in Figure 6).

4. Enable the **Run Inspector** check box, clear the **Create Expression** check box and click **OK**.

5. The Inspector will appear with 37 objects selected. Click on the **Hide** attribute, clear the checkbox and press ENTER to apply the change.

All the component comment strings will now be visible on the board.

10.2.2.2 Changing the height of designator and comment strings

1. In the editing region of the Filter panel, type the query
   IsDesignator or IsComment and then click the **Apply** button.

2. Open the Inspector panel, select the Text Height field, type in a new value of 40 and press ENTER on the keyboard to apply the change.

> **Note:** For more examples on editing multiple objects, refer to the tutorial *Editing Multiple Objects.*

10.2.2.3 Editing the width of tracks and arcs on midlayers

1. Open the **NBP8 Xilinx Virtex-II Pro BGA456 Rev1.01** example project, from *C:\Program Files\Altium Designer Summer 09\Examples\Reference Designs\Daughter Boards - 2 Connector\NBP8 Xilinx Virtex-II Pro BGA456 Rev1.01* directory then open the PCB.

2. Set the PCB panel to browse by *Nets*, enable the **Select**, **Zoom** and **Clear Existing** checkbox options at the top of the panel, and set the display mode dropdown to **Mask**.

3. Left-click on the All Nets option in the top section of the panel. This will populate the second section with all the nets. Right click in this second section and click **Select All**

4. Right-click in the Primitives region and set it so only **Tracks** and **Arcs** are displayed.

5. With only Tracks and Arcs displayed, sort the list of primitives by the **Layer** Column, by clicking on it. Once sorted find all the tracks and arcs that exist on the 4 midlayers and select them. There should be 2519 objects selected.

6. Open the PCB **List** panel. Set the options at the top of the panel to **Edit**, **Selected Objects** and **all types of objects** by clicking and selecting from the drop downs.

7. Sort by the **Width** column and select all the 0.127mm width tracks using the shift key. Once selected press F2 key, type in the new width of 0.2mm and press Enter to apply the change. All selected widths (225 in total, as shown at the bottom of the List panel) are changed to 0.2mm. View Figure 10 for details of the selection and what to set in the list panel.

Object Kind	Layer	Net	X1	Y1	X2	Y2	Width /	Keepout	Locked
Track	MidLayer3	SPI_SEL	32mm	49.3mm	32.6mm	49.9mm	0.127mm	☐	☐
Track	MidLayer3	SRAM0_A7	21.2mm	47.5mm	21.8mm	47.5mm	0.127mm	☐	☐
Track	MidLayer3	SRAM0_A7	21.8mm	47.5mm	22mm	47.3mm	0.127mm	☐	☐
Track	MidLayer3	SRAM0_A7	21mm	47.7mm	21.2mm	47.5mm	0.127mm	☐	☐
Track	MidLayer3	SRAM0_A7	21mm	47.7mm	21mm	48.5mm	0.127mm	☐	☐
Track	MidLayer3	SRAM0_A7	22mm	46.5mm	22mm	47.3mm	0.127mm	☐	☐
Track	MidLayer3	SRAM1_A13	27mm	45.5mm	27mm	46.5mm	0.127mm	☐	☐
Track	MidLayer3	SRAM1_A13	31.9mm	42.5mm	33mm	42.5mm	0.127mm	☐	☐
Track	MidLayer3	SRAM1_A13	35mm	39.5mm	36.9mm	39.5mm	0.127mm	☐	☐
Track	MidLayer3	SRAM1_A13	36.9mm	39.5mm	37mm	39.4mm	0.127mm	☐	☐
Track	MidLayer3	SRAM1_A13	37.2mm	37.5mm	39mm	37.5mm	0.127mm	☐	☐
Track	MidLayer3	SRAM1_A13	37mm	37.7mm	37.2mm	37.5mm	0.127mm	☐	☐
Track	MidLayer3	SRAM1_A13	37mm	37.7mm	37mm	39.4mm	0.127mm	☐	☐
Track	MidLayer3	SRAM1_D6	22.5mm	35mm	23mm	34.5mm	0.127mm	☐	☐
Track	MidLayer3	SRAM1_D6	23mm	31.5mm	23mm	34.5mm	0.127mm	☐	☐
Track	MidLayer3	SW4	30mm	30.5mm	30mm	32.5mm	0.127mm	☐	☐
Track	MidLayer3	VCCINT	20mm	44.5mm	23mm	44.5mm	0.127mm	☐	☐
Track	MidLayer3	VCCINT	20mm	45.5mm	23mm	45.5mm	0.127mm	☐	☐
Track	MidLayer4	IO15	29mm	30.5mm	29mm	33.5mm	0.127mm	☐	☐
Track	MidLayer4	IO18	32mm	30.5mm	32mm	33.5mm	0.127mm	☐	☐
Track	MidLayer4	IO21	33mm	30.5mm	33mm	33.5mm	0.127mm	☐	☐
Track	MidLayer4	RAM0_DATA5	27mm	30.5mm	27mm	33.5mm	0.127mm	☐	☐
Track	MidLayer4	SW2	30mm	30.5mm	30mm	33.5mm	0.127mm	☐	☐
Track	MidLayer4	SW6	31mm	30.5mm	31mm	33.5mm	0.127mm	☐	☐
Track	MidLayer4	VCCINT	19.4mm	44.5mm	23.4mm	44.5mm	0.127mm	☐	☐
Track	MidLayer4	VCCINT	19.4mm	45.5mm	22.974mm	45.5mm	0.127mm	☐	☐
Track	MidLayer4	VCCINT	22.974mm	45.5mm	23.474mm	45mm	0.127mm	☐	☐
Track	MidLayer4	VCCINT	23.474mm	45mm	24.5mm	45mm	0.127mm	☐	☐
Track	MidLayer4	VCCINT	23.4mm	44.5mm	24mm	44.5mm	0.127mm	☐	☐
Track	MidLayer4	VCCINT	24mm	44.5mm	24.5mm	45mm	0.127mm	☐	☐
Track	MidLayer1	BLUE1	10.8mm	40.1mm	15.6mm	40.1mm	0.2mm	☐	☐
Track	MidLayer1	BLUE1	12.2mm	61.4mm	12.2mm	70.2mm	0.2mm	☐	☐
Track	MidLayer1	BLUE1	15.6mm	40.1mm	15.8mm	39.9mm	0.2mm	☐	☐
Track	MidLayer1	BLUE1	15.8mm	39.9mm	16.7mm	39.9mm	0.2mm	☐	☐

2519 Objects (225 Selected)

Figure 10. The list panel with the mid layer tracks we are after selected.

8. Click the **Clear** button (SHIFT+C) at the bottom right to remove all masks and selections.

Note: A number of useful queries have been stored in the query **Filter** menu. Press **Y** while your document is active to pop up this menu, then choose **Examples**. It includes an option to filter out all objects except the component tracks and arcs on the overlay. Your own Favorite queries are automatically added to the **Filter** popup menu, add these via the **Filter** panel.

10.2.2.4 Importing component pins with the List panel

1. Open the 4 port serial interface example project and open the schematic library that is contained within this project.

2. Click on **SCH Panel Button » SCH Library** in the panel control on the bottom right to open up the library panel.

3. Goto **Tools » New Component** and give the component a name of **EP2C5**.

4. Set the current grid to 10. You can see the current grid in the bottom left of the screen. Use the G shortcut to change the grid on the fly.

5. In Windows open the Excel file TestBook.xls from C:\Program Files\Altium Designer Summer 09\Examples\Training\PCB Training\Practice Documents\ directory.

6. In the spreadsheet there is a group of columns starting with **Object Kind** and ending with **Designator**. This group of columns has been created by copying and pasting sections from

---

the Altera Cyclone II datasheet and also by using Excels easy to use mathematical functions to increment. Copy these columns, including the column names, to the clipboard.

7. Back in Altium Designer, click on **SCH » SCHLIB List** in the panel control on the bottom right to open up the library list panel.

8. At the top of the List panel, select **Edit** mode, and **current component**.

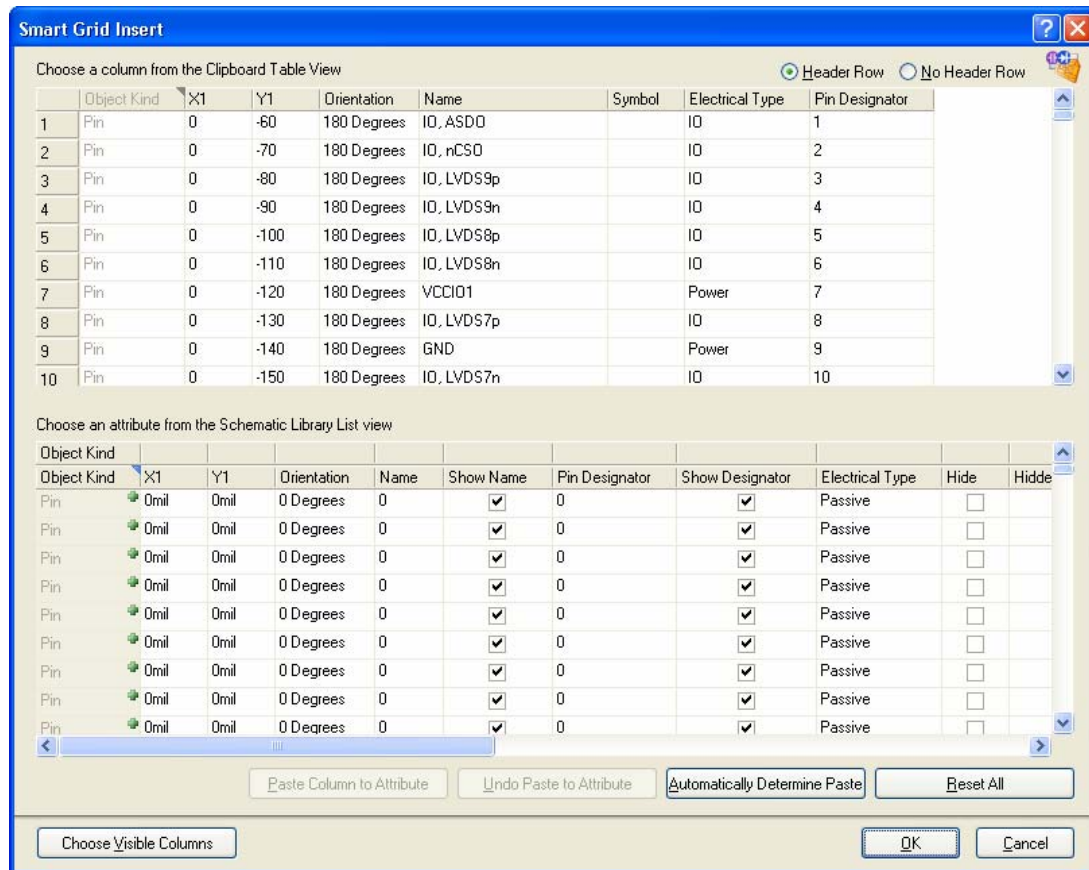9. Right click in the list panel itself, and select **Smart Grid Insert** from the floating menu.

Figure 11. Smart grid insert dialog that comes up upon pasting.

10. The Smart Grid Insert dialog will open (Figure 11). This dialog is used to map spreadsheet columns to Altium Designer pin properties. Since the column names have been pre-defined in Excel and were copied onto the clipboard, you can use the **Automatically Determine Paste** button. If the column names are not known, individual columns can be mapped one-by-one, using the **Paste Column to Attribute** button to link them.

11. Once you are satisfied with the column mapping, click **Ok**. The pins for the schematic symbol will be created based on the information in the dialog.

> **Note:** If you find that all the pins are placed on top of each other, make sure your grid has been set to 10.

10.2.3 The Footprint Manager

Component footprints across the entire project can be managed in the Footprint Manager. Use the Footprint Manager to preview, validate, add, remove or edit footprint associations. Once changes are made the ECO system updates both the schematic and the PCB if required.
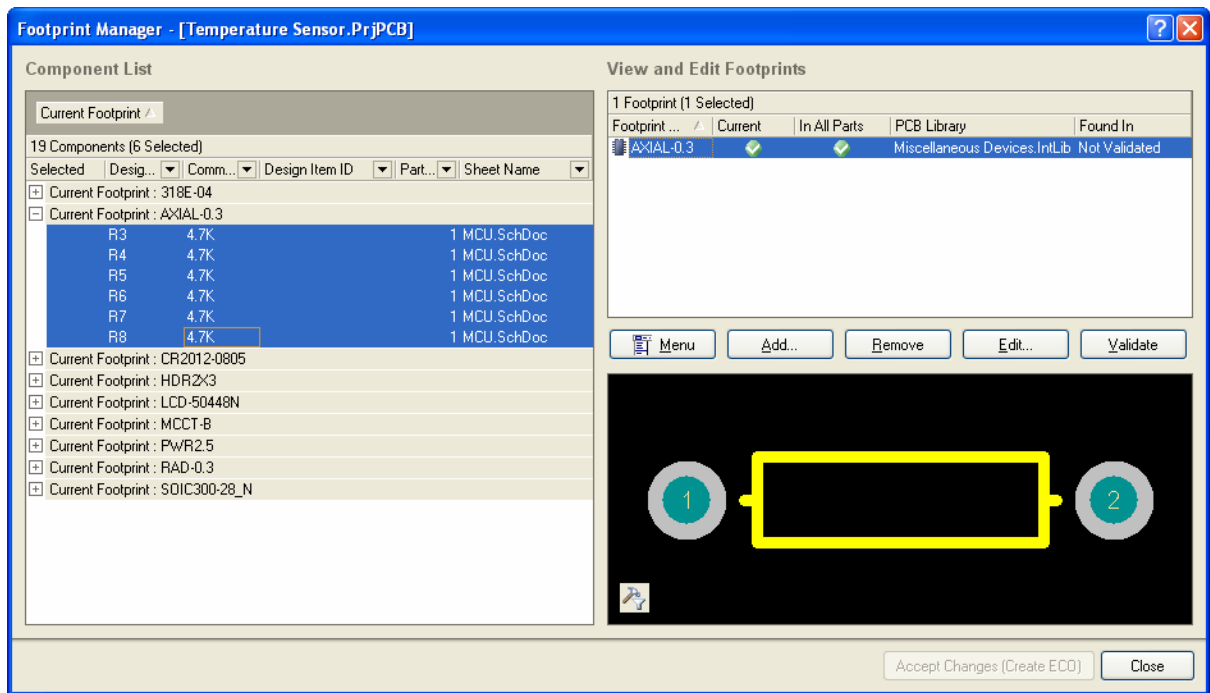
Figure 12. The Footprint Manager, with all components that use an Axial-0.3 type footprint selected.

- In the Schematic Editor, select **Tools » Footprint Manager**, to open the *Footprint Manager* dialog.
- The dialog will initially list every component individually, drag a column heading to the top left region of the dialog to group by that column. For example dragging **Current Footprint** will cluster all components that share the same footprint.

10.2.3.1 Exercise – Setting the component's footprint

The footprints currently assigned to the resistor and capacitor components are not surface mount and are not suitable for the board design. To use the Footprint Manager to change these:

1. With any of the schematics in the Temperature Sensor project active, select **Tools » Footprint Manager**.

2. Group by the **Current Footprint** column, and select the components that have the footprint **AXIAL-0.3**, as shown in Figure 12.

3. With the components selected, the top right hand side of the dialog will show all footprints that are common to all of the selected components. At this stage we can either edit the existing **AXIAL-0.3** footprint and change it to an SMD type footprint, or we can add a new footprint to the selected components. For this exercise we'll add a new footprint.

4. Click on the **Add** button, the *PCB Model* dialog will open. Before locating the new footprint, enable the **Library name** option. Doing this ensures that the selected footprint will come from the selected library.

5. Now click the **Browse** button at the top of the *PCB Model* dialog. In the *Browse Libraries* dialog that opens, click the **Find** button to search for a suitable footprint.
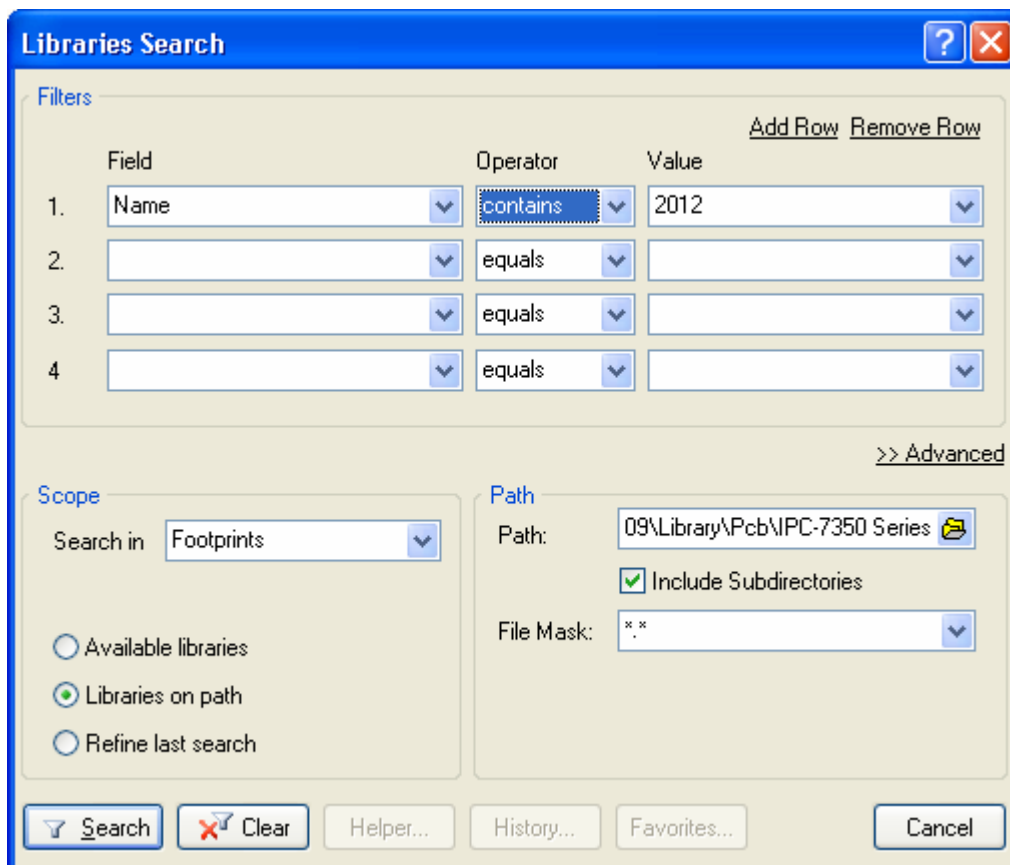

Figure 13. The Libraries Search dialog.

6. In the *Libraries Search* dialog set the Scope to **Libraries on Path**, the path to `C:\Program Files\Altium Designer Summer 09\Library\Pcb\IPC-7350 Series`, and enter the search string `2012`, as shown in Figure 13.

7. The search should return a number of matching footprints, such as **RESC2012N**. Select this footprint in the library *IPC-7352 Chip_Resistor_N.PcbLib*.

---

8. Whenever you choose a component or footprint from a search that is in a library that is not installed, you will be prompted to install the library. Click **Yes** to do this. Tick the **Don't ask again** checkbox if you always want the chosen library to always be installed.

9. The *PCB Model* dialog is now configured for the new footprint, as shown in Figure 14. Click **OK** to close the dialog and Add this footprint to the selected components.

10. At the moment, the **AXIAL-0.3** footprint is still assigned as the current footprint for the selected components, to change this right click on the newly added RESC2012N footprint in the top right hand section of the *Footprint Manager*, and select **Set as Current**, as shown in Figure 15.

11. There is also a capacitor using an inappropriate thru hole footprint, RAD-0.3. Repeat the process of locating a new footprint called **CAPC2012N**, using steps 4, 5 and 6 again. The library containing this footprint is called *IPC-7352 Chip_Capacitor_N.PcbLib*. Note you will get a confirmation prompting you to add the IPC-7352 Chip_Capacitor_N.PcbLib confirm with Yes.

12. For the capacitor, set the **CAPC2012N** footprint as the current footprint in the *Footprint Manager*.

13. Repeat for capacitors 1812.

14. The actual components on the schematic sheets have not been modified yet. To update them click on **Accept Changes (Create ECO)**, then **Execute** in the *Engineering Change Order* dialog to update the schematics.
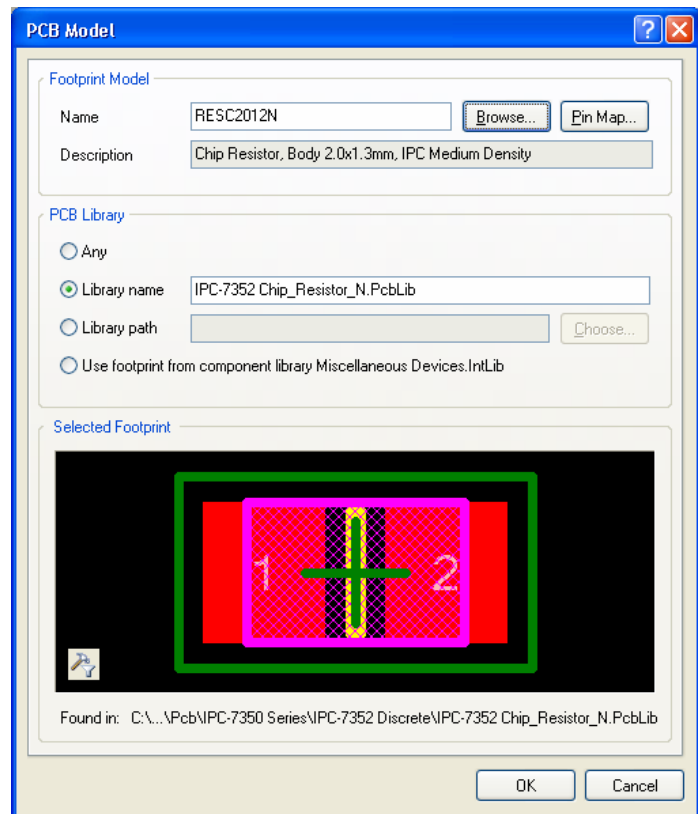

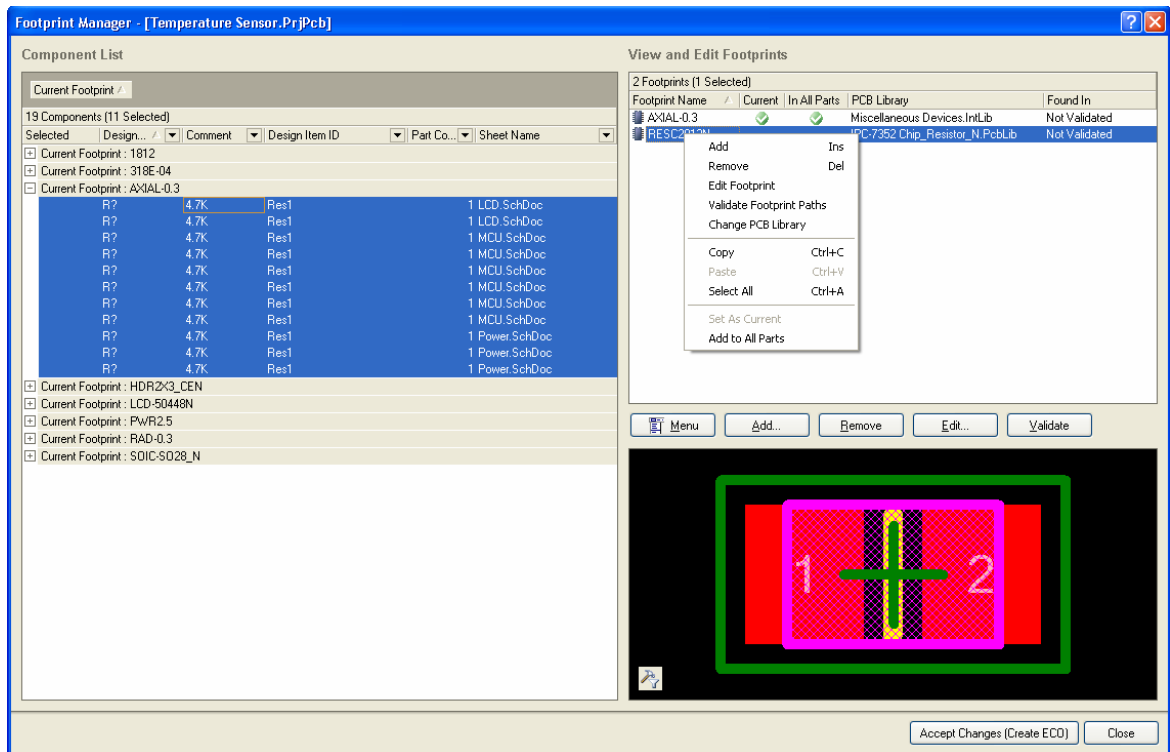*Figure 14. Configure the PCB Model dialog for the new resistor footprint.*

Figure 15. Set the Current Footprint for the selected components to the 0805 SMD footprint.

Note: As well as being able to change multiple footprints, you can also validate all the footprints, checking to see if they can be found in the available libraries.