

Altium ***Designer***

Module 4: Schematic Capture

Module 4: Schematic Capture

4.1	Introduction to Schematic Capture	4-1
4.2	The Schematic Editor workspace	4-2
4.2.1	Document Options.....	4-2
4.2.2	Preferences	4-8
4.3	Libraries and components	4-9
4.3.1	Locating and loading libraries	4-9
4.3.2	Locating components	4-11
4.3.3	Browsing libraries	4-13
4.3.4	Exercises – Libraries and components	4-14
4.4	Placing and wiring	4-17
4.4.1	Placing components	4-17
4.4.2	Pin-to-pin wiring.....	4-18
4.4.3	Exercise – Drawing the schematic	4-19
4.4.4	Exercise – The Sensor schematic.....	4-21

Software, documentation and related materials:

Copyright © 2009 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, Board Insight, Design Explorer, DXP, LiveDesign, NanoBoard, NanoTalk, P-CAD, SimCode, 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.

Module Seq = 4

4.1 Introduction to Schematic Capture

The Schematic Capture training session covers how to create single sheet schematics and multi-sheet hierarchical projects from initial setup through to component placement, wiring, design verification and printing. The functionality of the Schematic Editor will be explored and a series of exercises will show you how to capture a design as a schematic, ready for PCB design.

Figure 1 outlines the workflow to be followed when creating a schematic in Altium Designer.

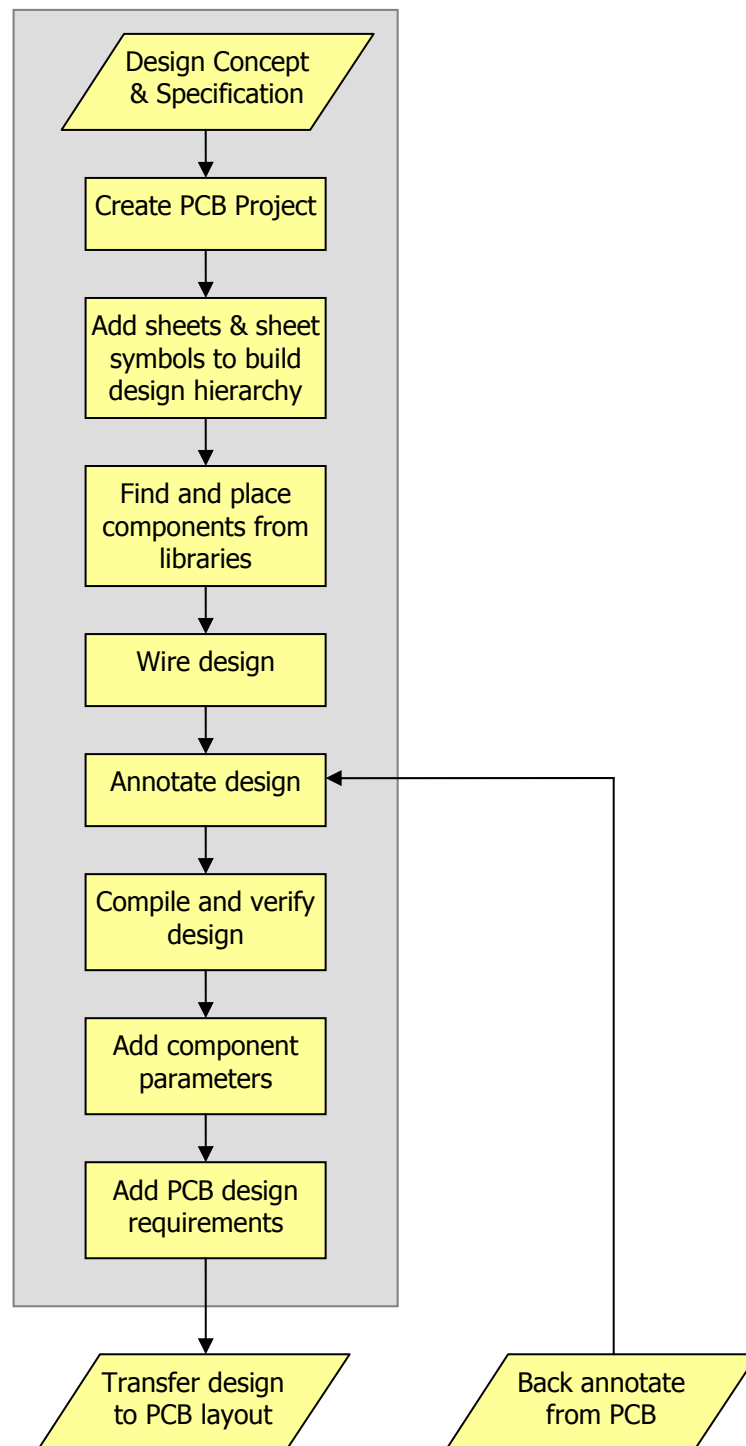


Figure 1. The Altium Designer schematic capture workflow

4.2 The Schematic Editor workspace

This section describes how to set up and browse the Schematic Editor workspace, done via the *Document Options* and *Preferences* dialogs. Sheet options, such as grids and templates, as well as preferences and defaults can be set through these dialogs.

To open the Schematic Editor, simply create a new schematic document (**File » New » Schematic**) or open an existing .SchDoc document in Altium Designer.

4.2.1 Document Options

The *Document Options* dialog allows you to:

- Set parameters relating to *individual schematic files*.
- The settings in this dialog are saved with that schematic file.
- The *Document Options* dialog is displayed by double-clicking on the sheet border, or by choosing the **Design » Document Options** menu command.

The tabs of the *Document Options* dialog are described in the following sections.

4.2.1.1 Sheet Options tab

The **Sheet Options** tab of the *Document Options* dialog is shown in Figure 2. The options in each of the sections are explained below.

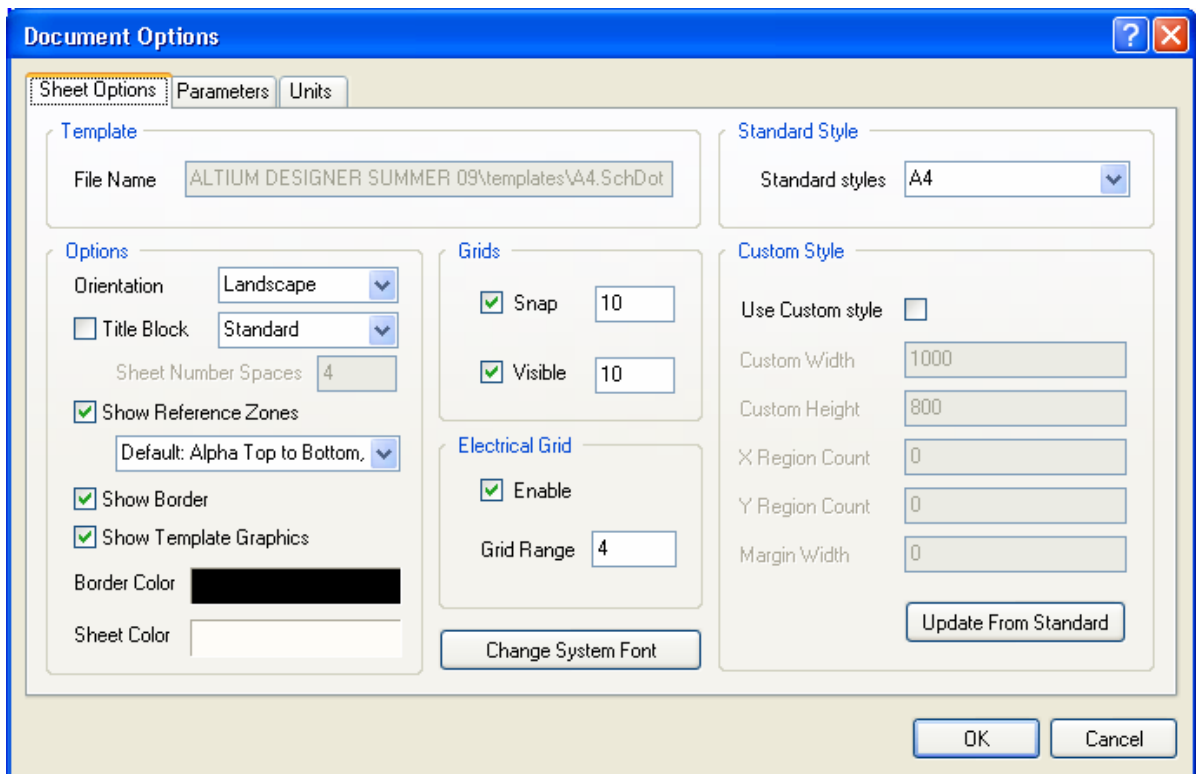


Figure 2. Sheet Options tab of the Document Options dialog

Template section

Displays the filename of the associated template, if any. Use the **Template** options in the **Design** menu to apply, update or remove the associated template. Set the default template in the **DXP » Preferences » Schematic » General**.

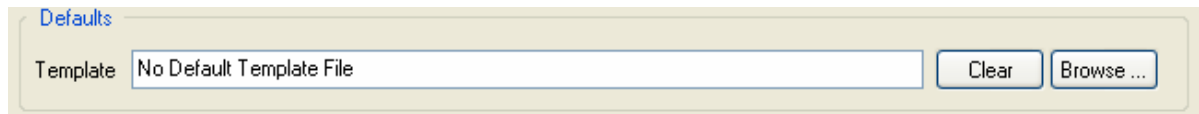


Figure 3. Setting a default template to use when creating a new schematic.

Options section

Orientation

Sets the sheet orientation to Landscape or Portrait.

Title Block

When checked, a standard title block is attached to the sheet. The format of that title block is set using the drop-down list next to this option. Note that this is typically only used when there is no associated template.

Show Reference Zones

When checked, the sheet has a reference grid defined in its border. A new option has also been added to have the zoning as per ASME Y14.1 standard.

Show Border

When checked the sheet border is displayed.

Show Template Graphics

When checked, any objects placed in the template file defined for the sheet will be displayed in the sheet. This is typically used to display a non-standard title block, in which case you would uncheck the Title Block option.

Border Color

Allows you to set the border color from the *Choose Color* dialog.

Sheet Color

Allows you to set the background color of the sheet.

Standard Style section

Allows you to select the size of the sheet from a number of standard sizes e.g. A4, A3.

Custom Style section

Allows you to define a custom sheet size and border. Use this option if you want a sheet size not covered in the Standard Style section.

Change System Font

This button allows you to change the font used to display pin numbers, pin names, port text, power port text and sheet border text.

Grids section

Grids Options allow you to set the size and turn on or off the Snap Grid and the Visible Grid.

SnapOn

The Snap Grid forces the mouse click location to the closest snap grid point. The Snap Grid is set and can be turned on or off in the *Document Options* dialog. You can also cycle through three predefined grids by pressing the **G** shortcut key at any time.

Visible

The Visible Grid displays a grid when turned on. This is independent of the Snap Grid. The Visible Grid can also be turned on or off in the **View** menu (**VV**).

Electrical Grid section

The Electrical Grid can be turned on or off and the Electrical Grid Range can be set in the *Document Options* dialog. It can also be turned on or off in the **View** menu (**VE**).

When the Electrical Grid is turned on and you are executing a command that supports the electrical grid, the cursor overrides the Snap Grid and jumps to key points on objects.

For example, if you are using the **Place » Wire** command and move the cursor to a certain distance within the Electrical Grid Range of a pin, the cursor will jump to the pin.

4.2.1.2 Parameters tab

The **Parameters** tab is used as a convenient method of editing sheet-level text. Each parameter is automatically linked to a text string on the sheet, where the text string is the same as the parameter name, except that it is preceded by an equals sign.

For example, the Parameter *Address1* is automatically linked to the text string *=Address1*. The equals sign is an instruction to the schematic editor to automatically replace the text string on the sheet with the value of a parameter with a name of *Address1*. Any number of these parameters can be added to a document, either a schematic template or a schematic sheet. Using these special strings allows template text properties, such as font, size and color, to be predefined in the template, while the actual text string value is defined when that template is applied to a schematic.

This replacement occurs automatically during printing, it can also be performed on screen by enabling the **Convert Special Strings** option in the **Graphical Editing** tab of the *Preferences* dialog (**Tools » Schematic Preferences**).

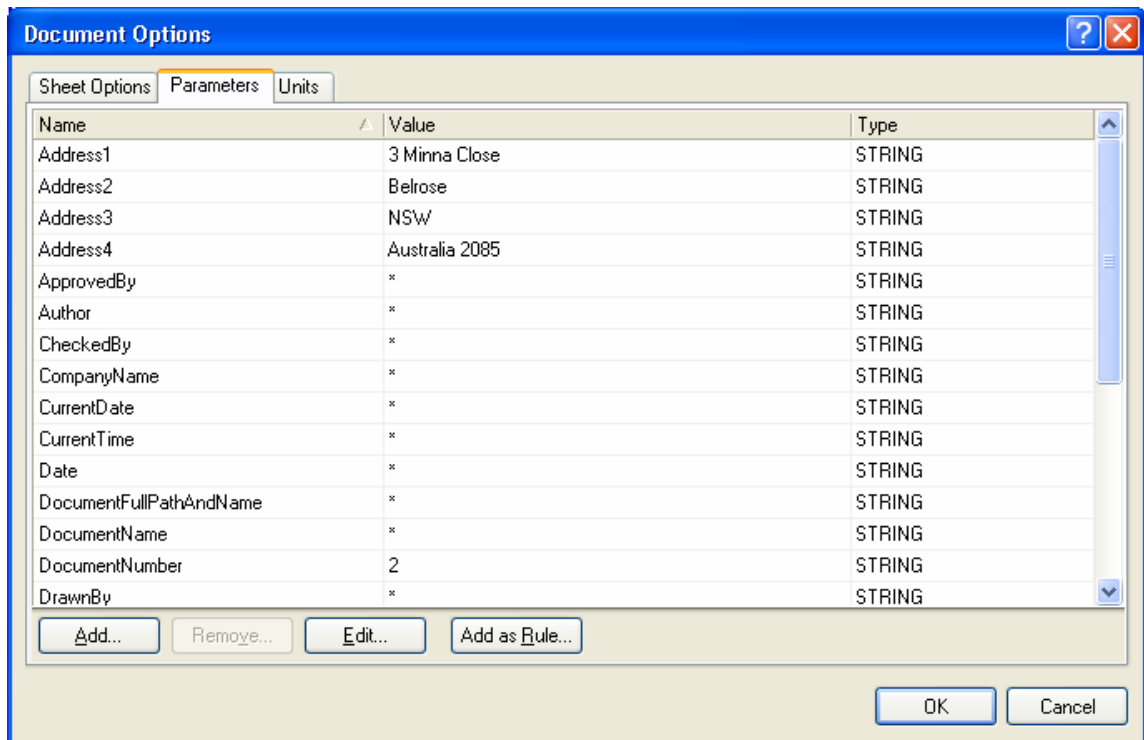


Figure 4. Parameters tab of the Document Options dialog

The default special strings are listed in the table below, but you can create custom parameters to suit your document and design requirements.

Special String	Description	Special String	Description
=Address1	Line of an address	=Engineer	Engineer's name
=Address2	Line of an address	=ImagePath	Path to image file
=Address3	Line of an address	=ModifiedDate	Computer system date of last modification to file (value entered automatically)
=Address4	Line of an address	=Organization	Organization name
=ApprovedBy	Approver's name	=Revision	Revision number
=Author	Author's name	=Rule	Rule description if added using Add as Rule option
=Checked By	Checker's name	=SheetNumber	Schematic sheet number
=CompanyName	Company name	=SheetTotal	Total number of sheets in the project
=CurrentDate	Computer system date (value entered automatically)	=Time	Time (not automatically updated)
=CurrentTime	Computer system time (value entered automatically)	=Title	Title of schematic sheet
=Date	Date (not automatically updated)	=Engineer	Engineer's name
=DocumentFullPathAndName	Filename with full path of the schematic sheet (value entered automatically)	=ImagePath	Path to image file
=DocumentName	Filename without the path (value entered automatically)	=ModifiedDate	Computer system date of last modification to file (value entered automatically)
=DocumentNumber	Document number		
=DrawnBy	Draftsperson's name		

Table 1. List of default parameters when creating a new schematic

Figure 5 shows how Special Strings are entered in a title block. Text entered as the value of a parameter in the **Parameter** tab will display where the special string is placed. The properties of the special strings (i.e. font, color) determine the properties of the text that is displayed.

You place special strings by selecting **Place » Text String** and then pressing the TAB key. The *Annotation* dialog displays. Clicking on the down arrow in the name field lists a special string for each of the parameters defined. Click on the string required and place it. Special strings display their content when the **Convert Special Strings** option is selected in the **Graphical Editing** tab of the *Preferences* dialog (**Tools » Schematic Preferences**), or when the schematic is printed or plotted.

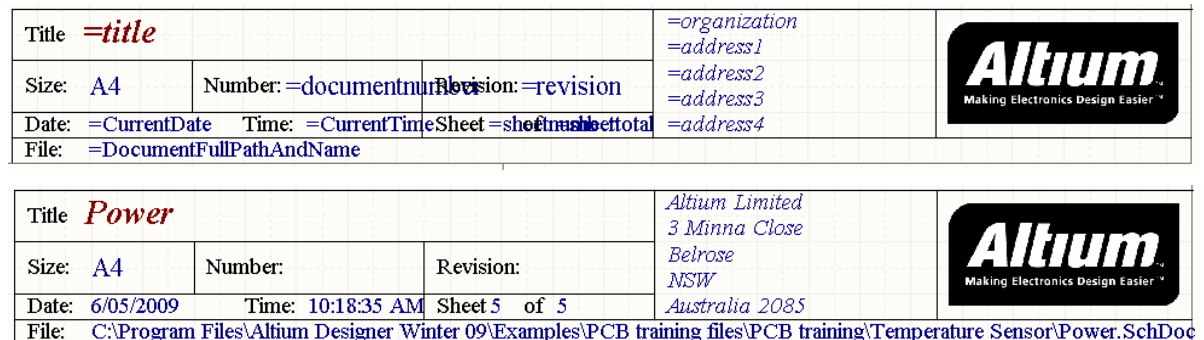


Figure 5. Special strings in a title block, with and without the Convert Special Strings option enabled

4.2.1.3 Units tab

The **Units** tab allows you to define the various units used in the Schematic Editor. Grids will be a multiples of these units so for example, using the DXP Default Units in which each *unit* is equal to 10 mils, a Snap Grid value of 5 (or 5 units) would equate to 50 mils, 10 units = 100 mils, etc. when printed actual size or 1:1.

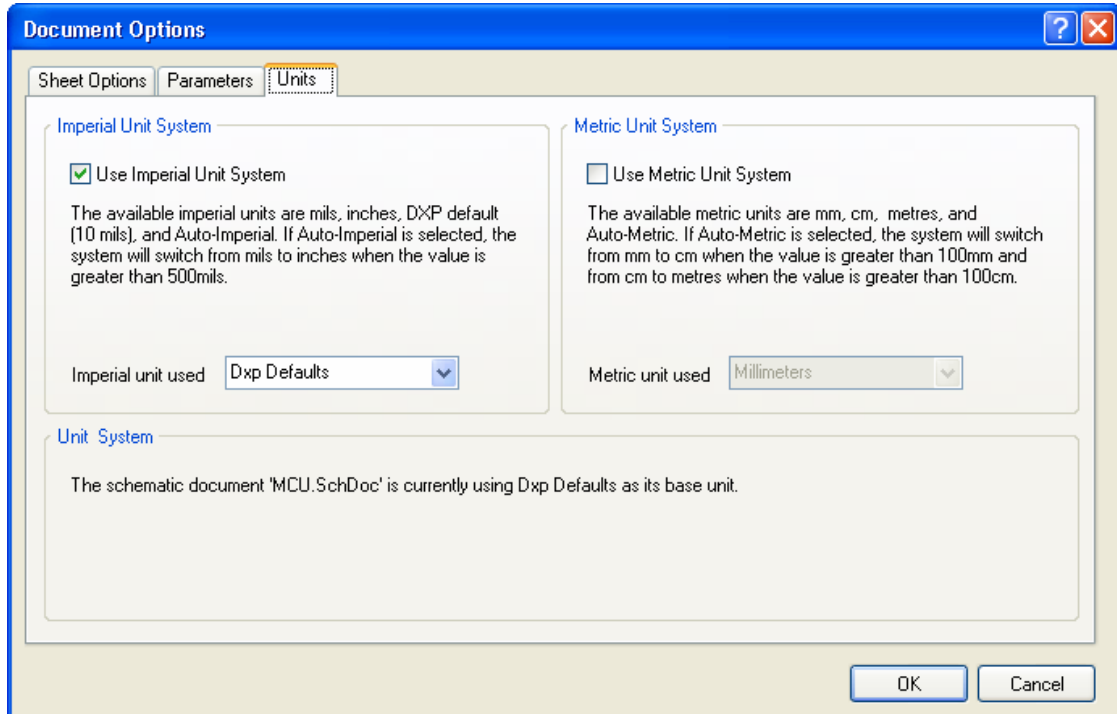


Figure 6. Units tab of the Document Options dialog

Imperial Unit System Section

Use Imperial Unit System

Check this option to use imperial units in your schematic(s). You will also want to specify which imperial units to be used (DXP defaults - 10 mil, mils, inches and auto imperial units) in the Imperial unit used drop down list.

Imperial unit used

This option is used to select from one of the available imperial units; mils, inches, DXP default units (10 mils) and auto imperial. If the Auto-Imperial unit is selected, the system will switch from mils to inches when the value is greater than 500 mils.

Metric Unit System Section

Use Metric Unit System

Check this option to use metric units in your schematic(s). You will also want to specify which metric units to be used (millimeters, centimeters, meters, and Auto-Metric units) in the Metric unit used drop down list.

Metric unit used

This option is used to select from one of the available metric units; millimeters, centimeters, meters, and Auto-Metric. If the Auto-Metric unit is selected, the system will switch from millimeters to centimeters when the value is greater than 100 centimeters.


4.2.1.4 Using templates

Standard sheet templates (*.SchDot) are supplied with Altium Designer and are accessible in the `\Altium Designer Summer 09\Templates\` folder. You can also create your own templates and store them anywhere.

- Select **Design » Template » Set Template File Name**. This option removes any existing template and uses the one you choose.
- Select **Design » Template » Update**. Use this command when a template is modified and you need to refresh the sheets which use it.
- Select **Design » Template » Remove Current Template**. This option removes the template but retains the old sheet size from the old template.

For each of these commands you will be prompted to indicate if the change is to apply to the current sheet, all open, or all in project.

4.2.2 Preferences

The **Schematic** section of the *Preferences* dialog allows you to set up parameters relating to the Schematic Editor workspace. This dialog is displayed using the **Tools » Schematic Preferences** menu command. Settings in this dialog are saved in the Altium Designer environment and therefore remain the same when you change active schematic documents. For help with the options available in the preferences, use the  button in the preferences dialog; then click on the option more information is required on.

4.3 Libraries and components

This section explores the Altium Designer libraries and how to find schematic components within them.

4.3.1 Locating and loading libraries

The supplied components are stored within a set of Integrated Libraries. An integrated library includes the schematic symbols, plus it can also include all associated models, such as footprints, spice models, signal integrity models, and so on. Most of the supplied integrated libraries are manufacturer-specific.

Integrated libraries are compiled from separate source schematic libraries, PCB footprint libraries, etc. The components in an integrated library cannot be edited, to change a component the source library is edited and recompiled to produce an updated integrated library.

There are a number of other special purpose integrated libraries, e.g. special function simulation components.

Components can also be placed directly from schematic symbol libraries if this is preferred to integrated libraries, and you can also place them from Protel 99 SE format schematic symbol libraries.

Available components are listed in the **Libraries** panel. The libraries presented in this panel include:

- Libraries in the active project. If the project that the currently active document belongs to includes any libraries, they are automatically listed.

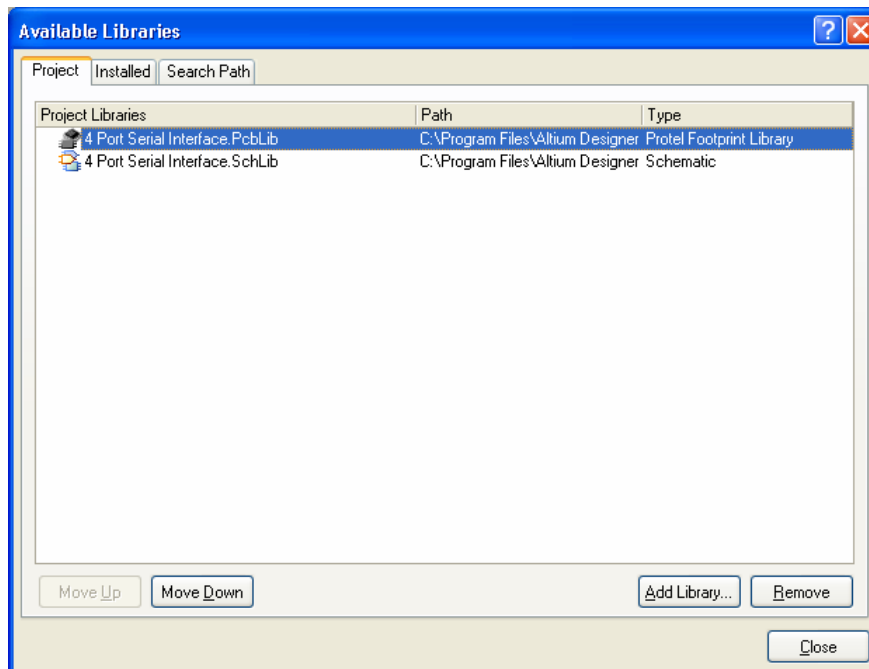


Figure 7. Available Libraries dialog, showing the project tab which is libraries in the current project

- Installed libraries. Installed libraries are those that have been made available in the environment. Use this option for company libraries that are used across different projects.

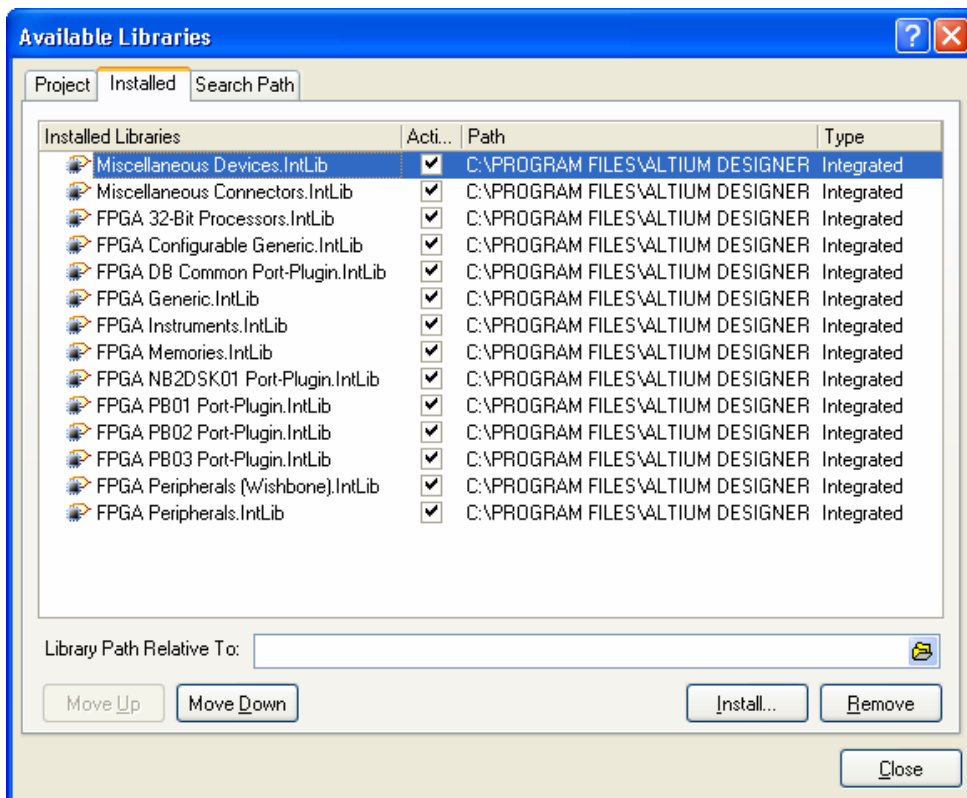


Figure 8. Available Libraries dialog, showing the installed tab which is available to any open project.

- Libraries found down the defined project search path. This option is particularly useful for accessing simulation models. Search paths are defined in the *Project Options* dialog.

4.3.1.1 Adding a library to make its components available

1. To add a library, press the **Libraries** button in the Libraries panel or select **Design » Add/Remove Library**. The *Available Libraries* dialog displays.
2. Click on the **Install** button at the bottom of the **Installed** Tab of the dialog.
3. Navigate to the required libraries directory and click on a library to select it. The library you selected will now be listed in the **Installed Libraries** list in the dialog.
4. Click **Close** when you have installed the libraries you need.

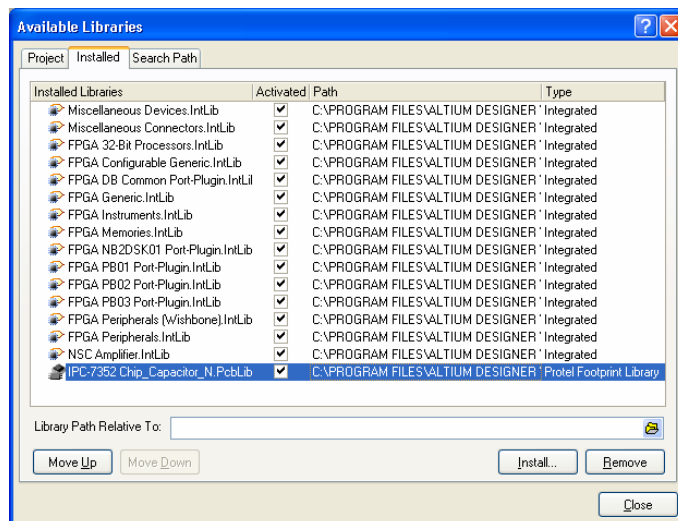


Figure 9. Available Libraries dialog, use the Installed tab to install or

Note: The supplied integrated libraries are located in the:
 \Program Files\Altium Designer Summer 09\Library\ folder.

4.3.2 Locating components

When the location of the library isn't known, use the **Search** button in the Libraries panel or the **Tools » Find Component** menu command. The *Libraries Search* dialog displays. The first time this dialog is executed, the dialog shown in Figure 10. is displayed. This initial mode is referred to as **Simple** mode. In **Simple** mode each searchable field has to be entered either manually or from a list of drop down items found within the currently installed libraries. After the field data is entered such as Name for example, an operator and value needs to be entered. The operator can be selected from a drop down list of Equals, contains, starts with and ends with. The value field can also be selected via a drop down of values found within the currently installed libraries or typed manually.

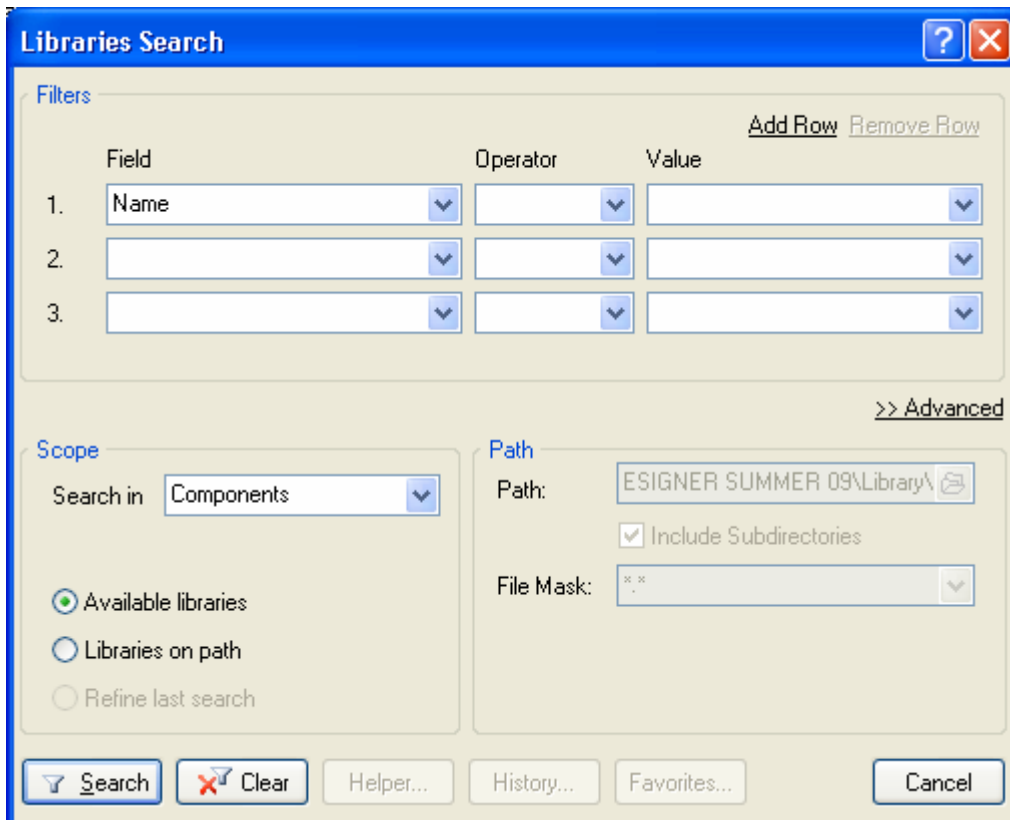


Figure 10. Simple view for library search dialog

Once data is entered, press the search button and a search starts. Going back to the search dialog and press the **Advanced** hyperlink button, the query string for the filter created in **Simple** mode is displayed.

Depending on current knowledge of the query language in Altium Designer, it is recommended, new users use the Simple mode and users familiar with the query language use the advanced mode to manually type the exact search query.

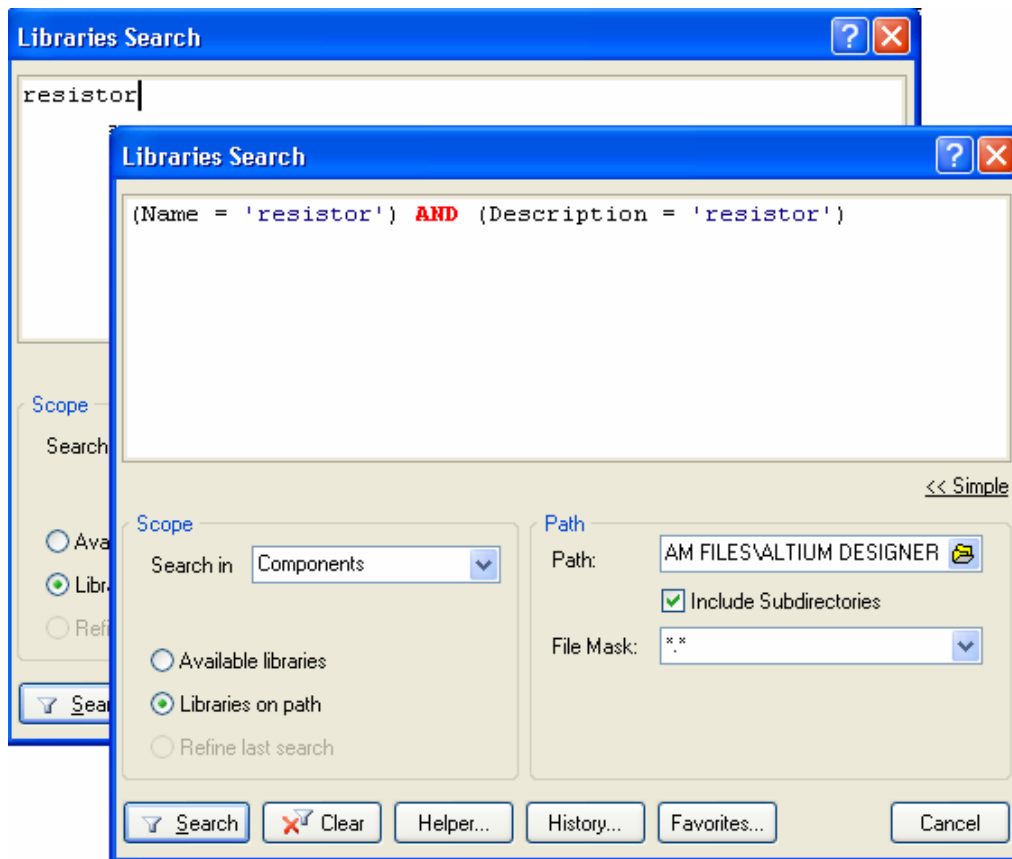


Figure 11. Advanced view for Search Libraries dialog

Tips for finding components:

- The search dialog uses a standard query to search the libraries – if the string you type does not include a query keyword it is assumed that the text is either part of the component *Name* or *Description* and a query is built automatically, as shown in Figure 11.
- The default search Scope is **Available libraries** that are those libraries currently listed in the **Libraries** panel. Change this to **Libraries on path** to search across all the supplied libraries.
- Search results are presented in the **Libraries** panel – note that the drop down where you select the current library will change to **Query Results**.
- If you attempt to place a component in the query results from a library that is not currently installed you will be asked if you wish to install that library now, you can still place without installing the library if you wish.
- The search can be terminated as soon as an instance of the part is found by clicking the button on the **Libraries** panel.
- If your search does not produce results, check that the search path is correct. Also, try searching for a component you know exists in a library to check that everything is set correctly

Note: When you are searching libraries, check that the Filter field at the top of the Libraries panel has nothing in it. If there is any thing other than a *, then the search results will be further filtered, as shown in Figure 12.

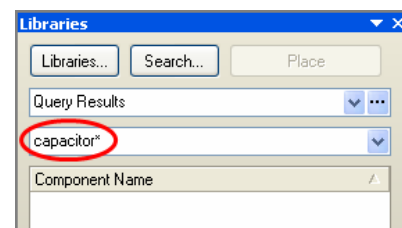


Figure 12. Ensure that the filter field is clear during searching.

4.3.3 Browsing libraries

The **Libraries** panel gives access to all components that are currently available to be placed.

- In the Panel control, bottom right, select **System » Libraries**.

- Click the **Libraries** button to display the *Available Libraries* dialog. This dialog displays all components currently available to be placed in the active project. Select the **Installed** tab, and click the **Install** button to add libraries to the library list.

- Components contained in the selected library are listed in the box below the Filter field. The Filter allows you to control what component names are listed, e.g. RES * will display only component names starting with RES.

- You can also type directly in the list of components, the type-ahead feature will automatically jump through the list as you type. Press ESC to stop performing a type-ahead action.

- Clicking on the name of a component will: display that component symbol in the viewer in the middle of the panel, list the associated models below that, and show the selected footprint model below that.

- The **Place** button places the component currently selected. Double-clicking on the name of a component also achieves this.

- The **Search** button is a powerful searching tool, allowing you to search through libraries for parts. Clicking this button pops up the *Libraries Search* dialog.

- If a component has several parts, the sub-parts will be shown in the symbol mini-viewer.

- You can control what columns are displayed in the component or model lists, right-click and choose **Select Columns** to do this. Using this with database libraries is especially useful as there is the ability to sort by a particular column like shown in Figure 14. Database libraries are covered in more detail in the Advanced PCB training course.

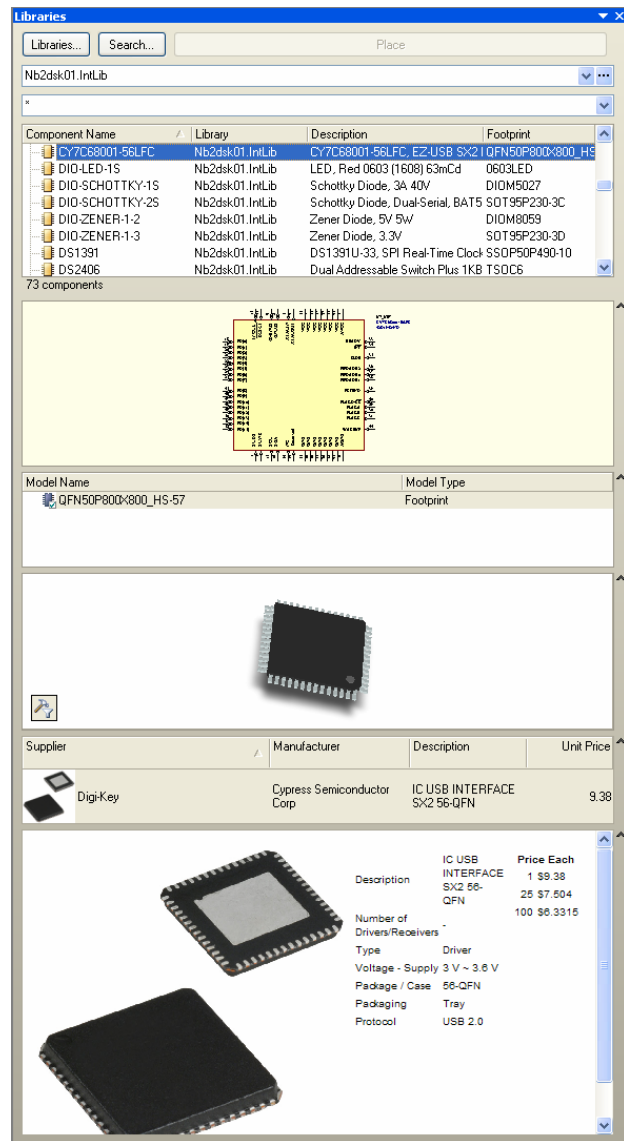


Figure 13. Browsing libraries with the Schematic Editor Panel

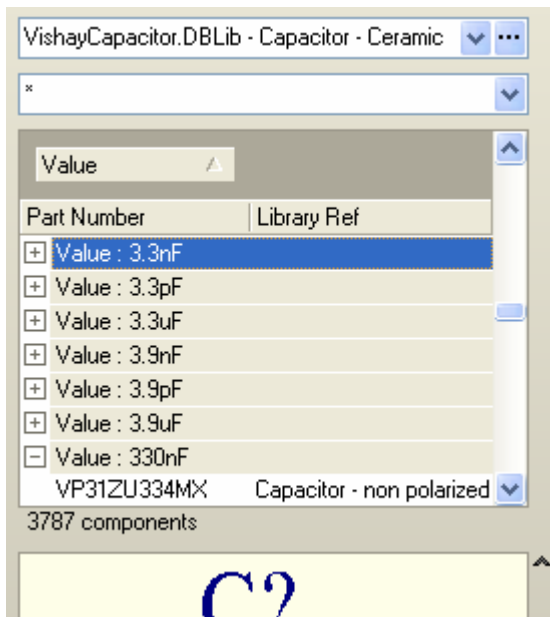


Figure 14. Using a database library and the sort by column feature to sort by Value.

4.3.4 Exercises – Libraries and components

4.3.4.1 Locating and loading libraries when the required library is known

The training design is a microcontroller driven temperature sensor. To install one of the supplied libraries and see if it includes a PIC microcontroller library, complete the following steps:

1. Open a schematic document to activate the Schematic Editor.
2. Click the **Libraries** button on the Libraries panel to display the *Available Libraries* dialog.
3. Select the **Installed** tab of dialog, then click the **Install** button and navigate to the `\Program Files\Altium Designer Summer 09\Library\` directory. This directory contains sub directories containing the integrated libraries supplied with Altium Designer's Schematic Editor.
4. Scroll down through the library directories. Open the `Microchip` folder, select and add the `Microchip Microcontroller 8-Bit PIC16.IntLib`.
5. Click the **Close** button to close the *Available Libraries* dialog.
6. Select this Microchip library in the list of libraries at the top of the Libraries panel. The library's contents will be displayed in the box below the Filter field section. Confirm that the library includes a PIC16C773/SO.

4.3.4.2 Finding components when their library is unknown

Often you will want to locate a component but do not know which library it is in, or you may want to see what family types are available in the libraries. To search for components, we use the **Search** button or the **Tools » Find Component** menu command.

1. Click on the **Search** button and the *Libraries Search* dialog will appear.
2. For this exercise, **Advanced** mode is going to be used on the Library search dialog. Click on the **Advanced** Hyperlink button.
3. Set the **Scope** to **Libraries on path** and set the search **Path** to `C:\Program Files\Altium Designer Summer 09\Library` (the **Include subdirectories** option should be on).

4. The power supply in the training design uses a LM317MSTT3 adjustable regulator. To search the supplied libraries for a suitable device type the string `LM317` in the Search field at the top of the dialog and click the **Search** button.
5. Note that the library currently being searched is listed in the **Libraries** panel. Depending on the speed of the PC it will take a few minutes to search the entire 90,000+ components for the required part.
6. The result set should include components in the `ON Semi Power Mgt Voltage Regulator.IntLib`, confirm that the LM317MSTT3 part is listed.
7. To install this library so that component will be available later you can either right-click in the result list and select **Add or Remove Libraries** (this will simply open the *Available Libraries* dialog), or you can double-click on the component name in the list to place it (you can easily delete it if it is the wrong sheet), when you do the Confirm dialog will appear, giving you an opportunity to Install the library.

4.3.4.3 Locating components within an open library

1. Select the library `Miscellaneous Devices.IntLib` in the Libraries panel.
This library is one of two PCB libraries installed by default when the software is installed. It includes a variety of discrete components, including resistors, capacitors, diodes, etc.
2. Type `cap` into the Filter field. Notice that only the capacitor-type components are listed.
3. Try `diode` in the Filter field. The only components listed now are the diodes whose library reference starts with the string `diode`.
4. Now try `*diode`, this time components that include the word `diode` anywhere in their name or description will be listed.

4.3.4.4 Finding footprints when their library is unknown

1. Footprints can be searched for in the same way as component symbols; the only difference is that you need to set the **Search in**, in the *Libraries Search* dialog to **Footprints** before pressing the **Search** button.
2. Set the Search Path to `C:\Program Files\Altium Designer Summer 09\Library\Pcb\IPC-7350 Series\`.

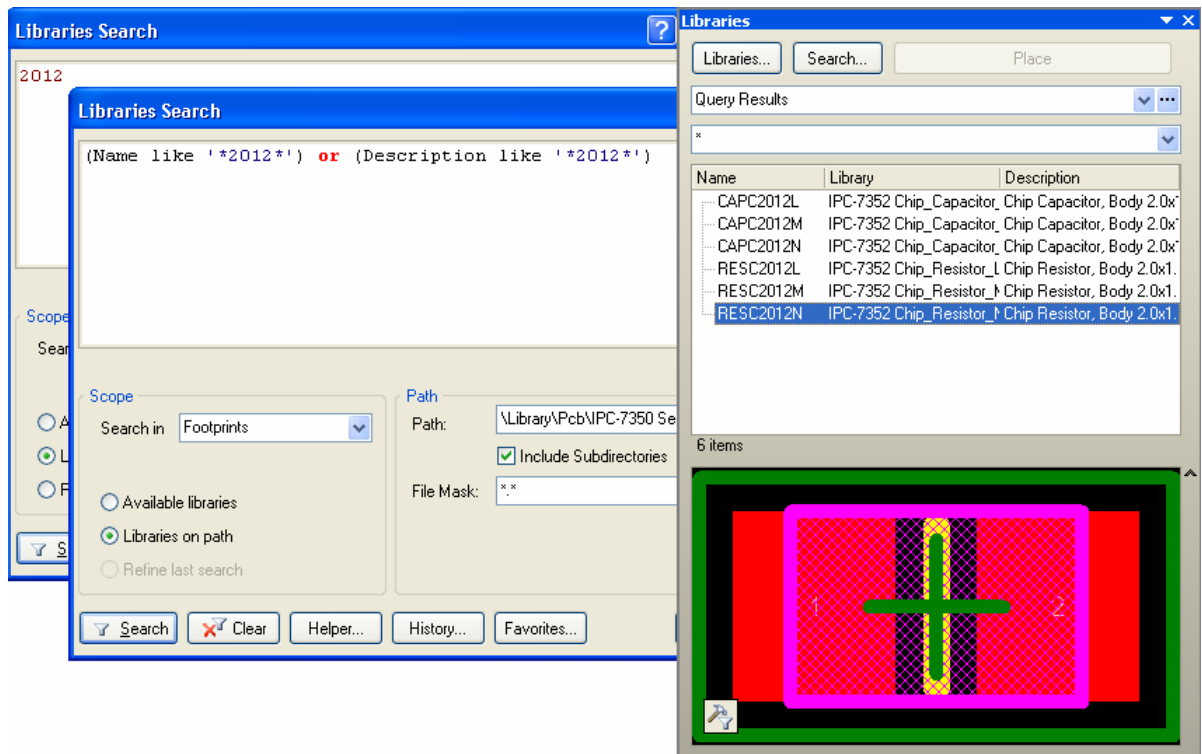


Figure 15. Searching for a footprint

3. Enter the string 2012 and click **Search**. The search results will include a number of libraries, including *IPC-7352 Chip_Resistor_N.PcbLib*.
4. Double-click on the RESC2012N footprint in the query results, a dialog will appear letting you know that the library is not currently installed, click **Yes** to install the library.

4.3.4.5 Setting the library search order

When you type in a component name, for example in the *Place Part* dialog, or when you type in a footprint name in the *Footprint Model* dialog, the available libraries are searched in a defined order. This search order is the order that the libraries are listed in the *Available Libraries* dialog. To configure the search order:

1. Click the **Libraries** button in the Libraries panel to display the *Available Libraries* dialog.
2. Click on the **Installed** tab, then in the list of Installed Libraries click to select the *IPC-7352 Chip_Resistor_N.PcbLib* to highlight it, and then click the **Move Up** button to move it to the top of the list.
3. Close the *Available Libraries* dialog.

You now have all the components and footprints required to complete the training design.

Note: Refer to the PDF, *AR0104 Component, Model and Library Concepts* article in the online documentation for further information on definitions, library search order and component to model linking.

4.4 Placing and wiring

This section looks at how to place components and then wire them together. The exercise takes you through the creation of a complete schematic sheet.

4.4.1 Placing components

- To place a component, double-click on its name in the Libraries panel.
- To edit a component's properties before you place it, press the **TAB** key. The *Component Properties* dialog displays. To step through the fields in the dialog press **TAB** (down), or **SHIFT+TAB** (up).
- New text will overwrite text that is selected.
- If you set the component designator before placing the component, then subsequent components will be automatically designated with the next designator value.
- You can also use the **Place » Part** menu command if you know the name of a component. When you select this command, you are prompted for the name of the component. Once you type the component name in, the open libraries are searched and if the component is located, it becomes attached to the cursor for placement.

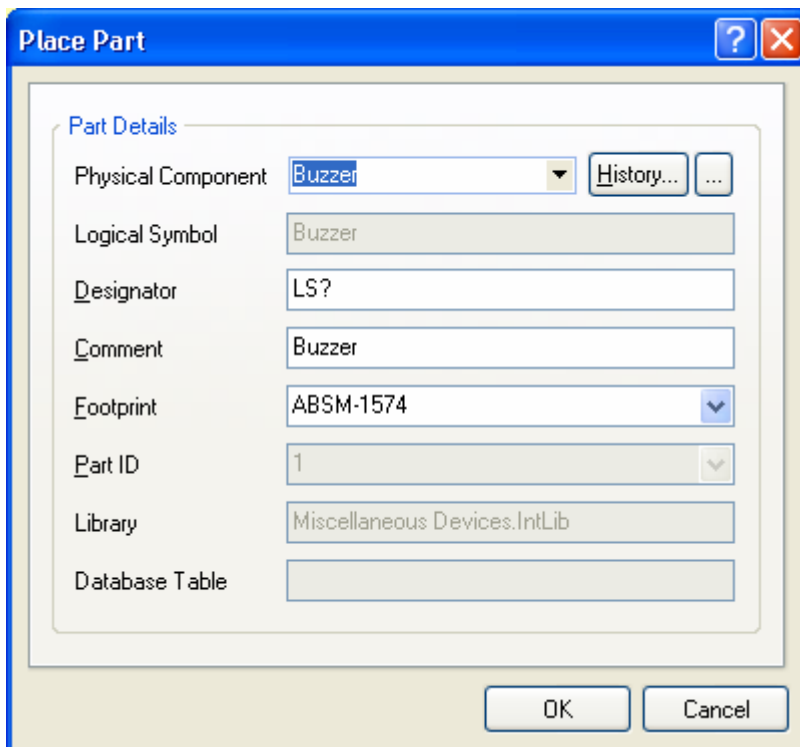


Figure 16. Place part dialog

4.4.1.1 Auto-incrementing designators

When placing a component, if the initial designator is set before placing, its designator will be assigned by incrementing the designator of the last component placed. This will only occur for subsequent parts placed after the **TAB** key was pressed to assign the initial designator. Once you stop placing this type of part, the next designator in the sequence is no longer remembered.

Generally it is easier to leave the annotation of designators until the design is complete to allow the designators to be assigned in a logical and controlled manner on each sheet. Annotation is covered in detail later in *Module 6 - Building the Project*.

4.4.2 Pin-to-pin wiring



- Wires are used to create an electrical connection between points.
- Be careful to use **Place » Wire** and not place lines by mistake.
- Press SHIFT+SPACEBAR to change the wire placement mode. Press SPACEBAR to toggle between start and end corner modes.
- Press BACKSPACE to delete the last vertex placed.
- A point on a wire must touch on the connection point of an electrical object to be connected to it, e.g. the wire must touch on the hot end of a pin to connect to it.
- Use buses to graphically represent how a group of related signals, such as a data bus, are connected on a sheet. Also, use buses to connect related signals to ports and sheet entries.
- Buses must use the bus name / bus element referencing system as shown in Figure 20, and must include the individual net labels and the bus net label.
- The bus range can increment [0..7], or decrement [7..0].
- To move a component on the schematic and maintain the wiring (referred to as dragging), hold the CTRL key as you click, hold and move the mouse (release the CTRL key once you start dragging). Press the SPACEBAR or SHIFT+SPACEBAR while dragging to change the wiring mode. Press the M shortcut to drag a selection.
- If you are having trouble seeing the grid in schematic goto **Tools » Schematic Preferences » Schematic » Grids** and change the *Grid Color* to one that is not so white.

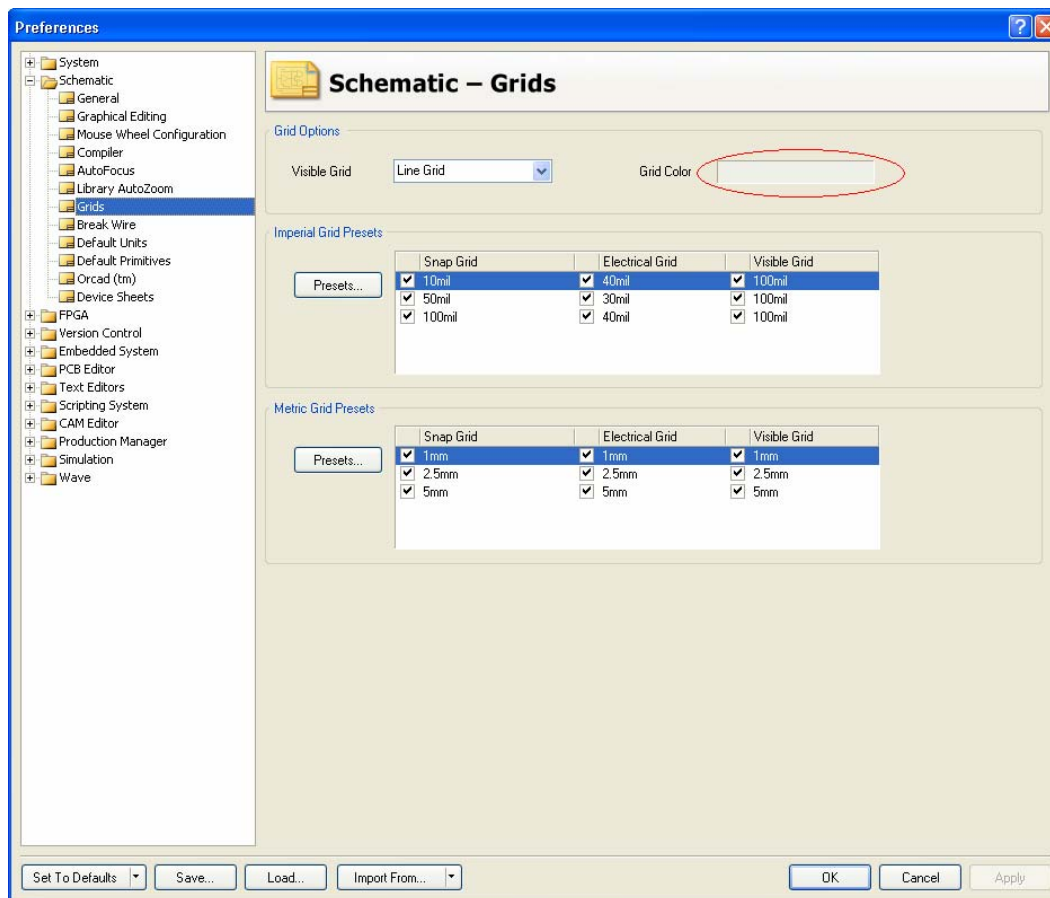


Figure 17. Grids setting in preferences

4.4.3 Exercise – Drawing the schematic

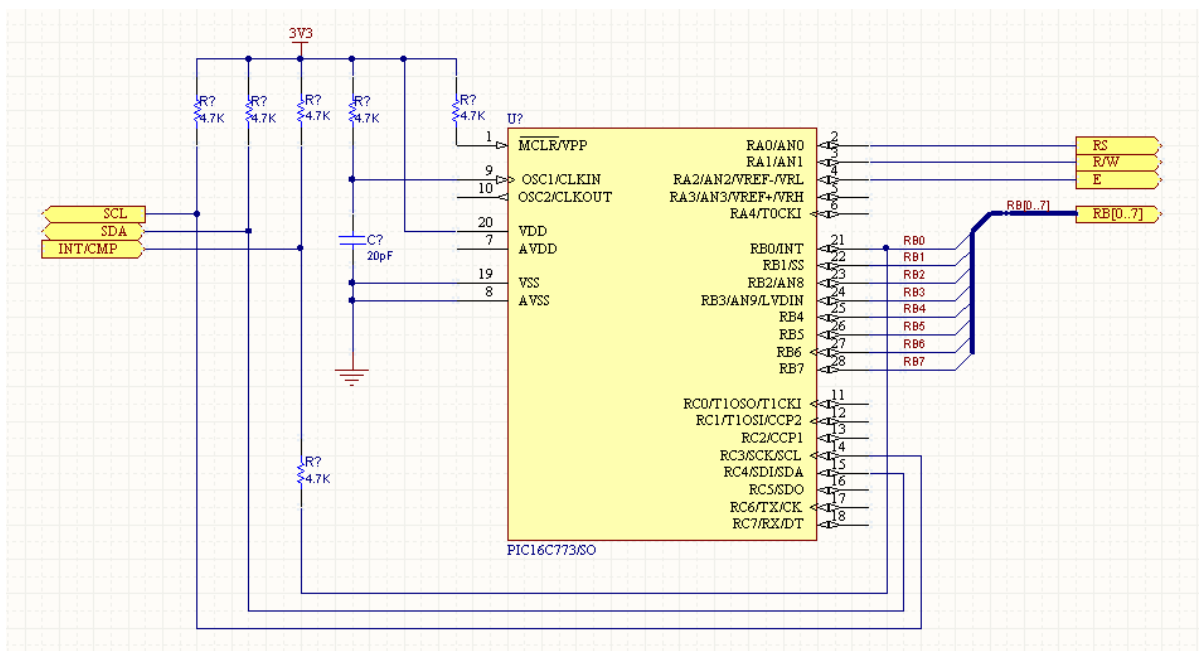


Figure 18. MCU schematic example

1. If it is not already open, re-open the project created during the *Module 1 - Getting Started With Altium Designer*, \Program Files\Altium Designer Summer 09\Examples\Training\PCB Training\Temperature Sensor\Temperature Sensor.PrjPcb.
2. Add a new schematic document to the project, to do this right-click on the project file name in the **Projects** panel and select **Add New to Project » Schematic**.
3. Right-click on the new schematic sheet in the Projects panel, and select **Save As** from the context menu. Save the schematic as MCU.SchDoc in the \Program Files\Altium Designer Summer 09\Examples\Training\PCB Training\Temperature Sensor folder.
4. Set the template for your schematic to A4.SchDot by choosing **Design » Template » Set Template File Name** and choosing the A4 size template from \Program Files\Altium Designer Summer 09\Templates folder.
5. Verify that the electrical grid is on and set to 4 and that the snap grip is on and set to 10 before placing any objects (double-click in the sheet border to open the *Document Options* dialog).
6. Draw up the schematic shown in Figure 18 above. When placing the components, press TAB to define the **Designator** and **Comment** (component value) before placing the component.

Component	Library Reference
Microcontroller	PIC16C773/SO
Resistors	Res1
Capacitor	Cap

7. To rotate a component press the SPACEBAR, press the **Y** key to flip it vertically, and the **X** key to flip it horizontally.
8. Set the **Port I/O Type** to match their display **Style**. Set the Ground **Style** power port net attribute to GND.
9. Set the bus name and port name to RB[0..7] so as to connect nets RB0 through to RB7 into a bus.

- To build up the nets in the bus, first create the port on the right. Copy the port and run **Edit » Smart Paste**. Select **Ports** from the left side and select **net labels and wires** on the right side. On the bottom section of the dialog select **expand buses** for signal names. You can also set the wire length to suit. Note that the spacing between the wires will be as per the current grid setting which should be on 10 so the wires and netlabels line up to the pins of the component.

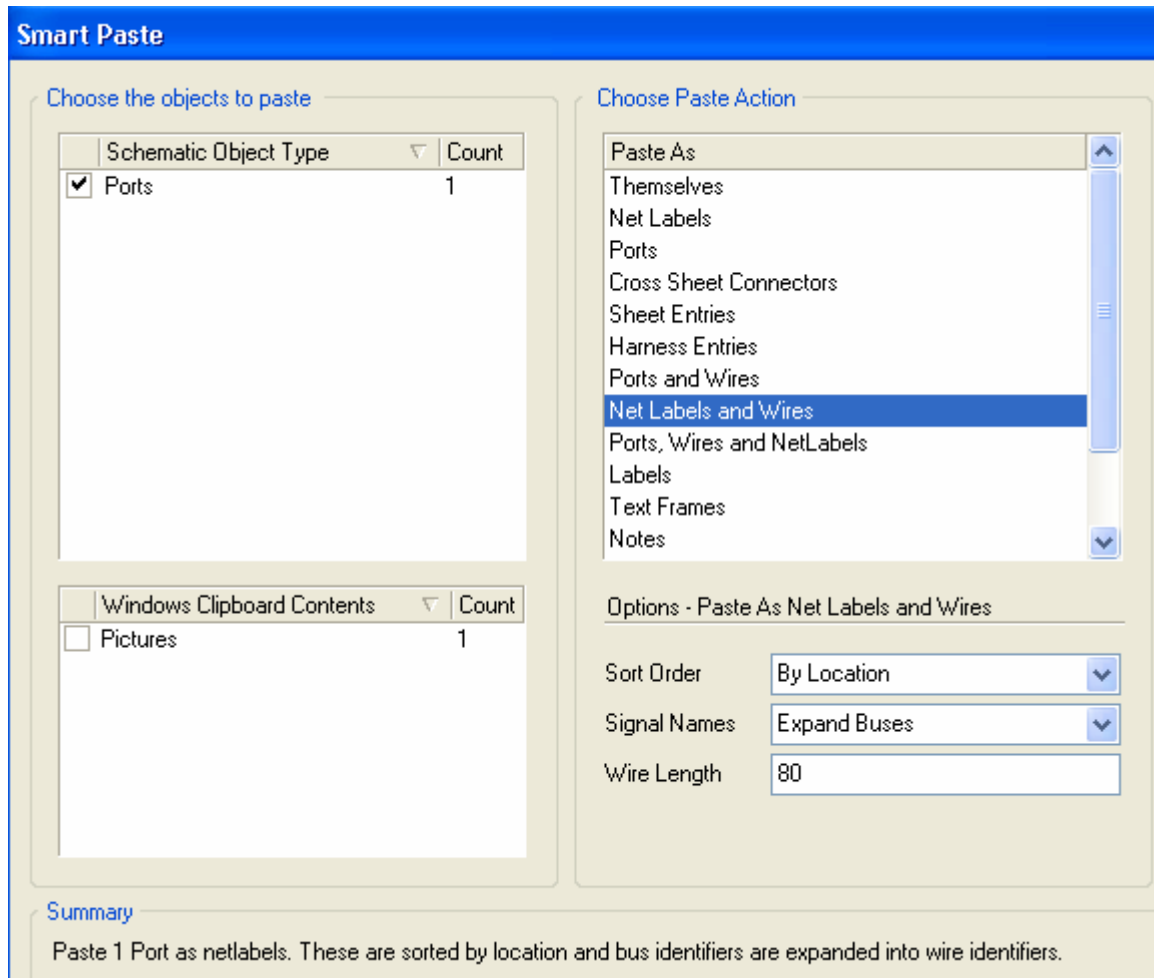


Figure 19. Smart paste dialog to create the netlabels and wires that make up the signals on a bus.

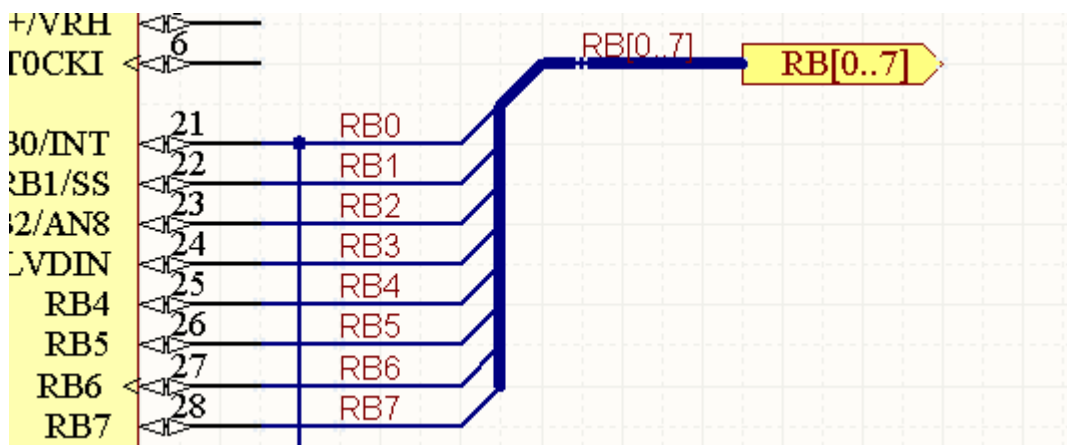


Figure 20. Buses are defined using the referencing system shown.

11. While placing the resistors and capacitors you can use the component cut wires feature to quickly cut into the wires rather than wiring around them.
12. While placing some of the ports and power ports hold down the ctrl key and drag to extend the wires with the object. This can save a lot of time when getting to the wiring stage.
13. Enter the necessary document information in the **Parameters** tab of the *Document Options* dialog. Enter the title as *PIC Microcontroller*. Don't worry about setting the sheet/Document No. at this time as there is a feature that can do this automatically over your whole project.

4.4.4 Exercise – The Sensor schematic

At this stage in the training, your Temperature Sensor project should look like Figure 21. However, the *Sensor.SchDoc* and the project libraries are not there as yet. To complete it:

1. Create a new schematic and call it **Sensor.schdoc** if you haven't done so already.
2. Save and close the *Sensor.SchDoc* sheet. Note that this sheet is just a blank schematic for now. It'll be completed later in the training.
3. Also you'll need to add the library files as shown in Figure 21 to the project and save the project PCB.

At this point we are only going to create a blank sheet. In *Module 15 - Schematic Library Editor*, we will create the sensor sheet after creating a new component.

The last step to complete the sensor design is to add the top schematic sheet which is done in the next module *Module 5 - Multi-Sheet Design*.

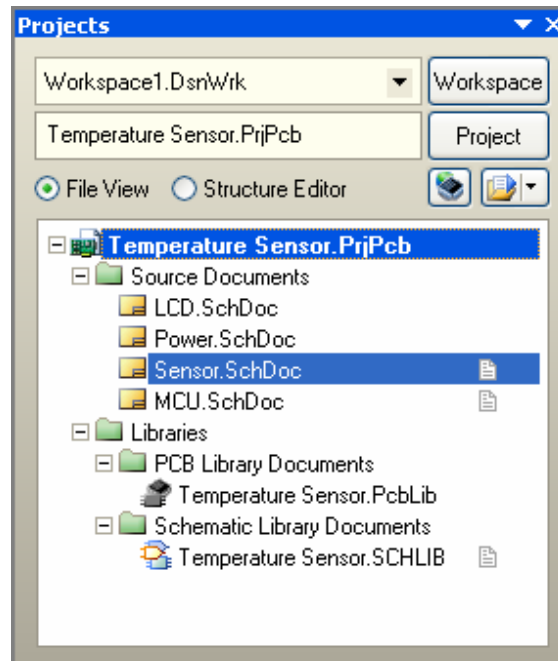


Figure 21. Project structure after completing the MCU schematic.