Module 1: Getting Started With Altium Designer

Module 1: Getting Started With Altium Designer

Module Seq = 1

1.1 Introduction to Altium Designer

Underlying the Altium Designer environment is a software integration platform that brings together all the tools necessary to create a complete environment for electronic product development, in a single application.

Altium Designer includes tools for all design tasks: from schematic and HDL design capture, circuit simulation, signal integrity analysis, PCB design, and FPGA-based embedded system design and development. In addition, the Altium Designer environment can be customized to meet a wide variety of user requirements.

1.1.1 The Altium Designer Integration Platform

When you select **All Programs » Altium Designer Summer 09** from the Windows Start menu to run Altium Designer, you are actually launching DXP.EXE. The DXP platform underlies Altium Designer, supporting each of the editors that you use to create your design.

The application interface is automatically configured to suit the document you are working on. For example, if you open a schematic sheet, appropriate toolbars, menus and shortcut keys are activated. This feature means that you can switch from routing a PCB, to producing a Bill of Materials report, to running a transient circuit analysis, and so on – and the correct menus, toolbars and shortcuts will be readily available.

Also, all toolbars, menus and shortcut keys can also be configured to suit how you like to configure your design environment.

Figure 1. Altium Designer's software integration architecture

1.2 The Altium Designer environment

The Altium Designer environment consists of two main elements:

- The main document editing area of Altium Designer, shown on the right side in Figure 2.
- The *Workspace Panels*. There are a number of panels in Altium Designer, the default is that some are docked on the left side of the application, some are available in pop-out mode on the right side, some are floating, and others are hidden.

When you open Altium Designer, the most common initial tasks are displayed for easy selection in a special view, called the *Home Page*.

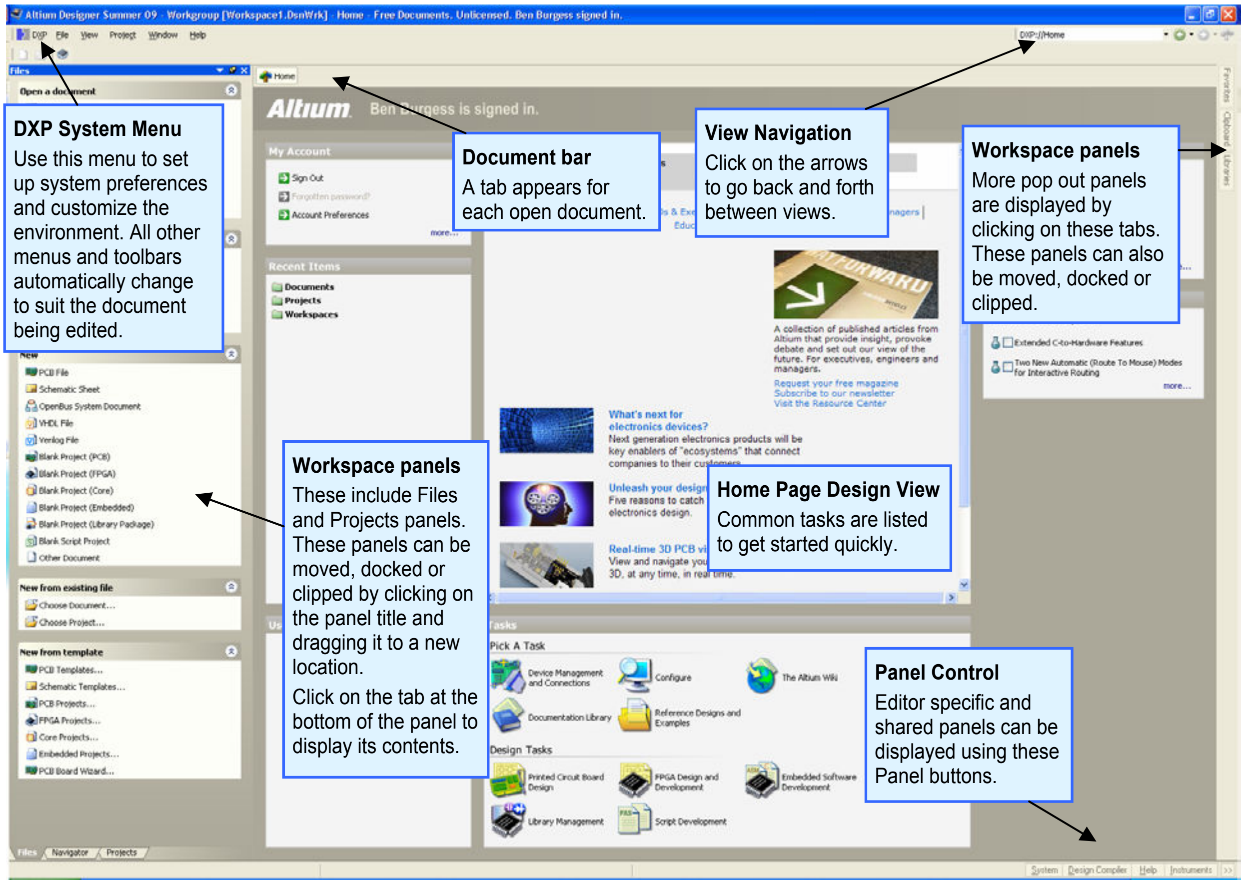

Figure 2. Altium Designer with the DXP Home Page displayed.

> **Note**: To move an individual panel, click and hold on the panel name. To move a set of panels, click and hold on the panel caption bar away from the panel name. To prevent panels stacking together, hold the CTRL key. To change a docked panel to pop-out mode click the small pin icon at the top of the panel, to change it back to docked click the pin icon again.

> **Note**: If you manage to completely ruin your panel layout and wish to revert back to the factory settings, this can be done by going to the **View » Desktop Layouts » Default**. It's best to restart Altium Designer when you run this. To save a custom layout go to **View » Desktop Layouts » Save Layout**. To reload existing layouts go to **View » Desktop Layouts » Load Layout**.

1.2.1 The Altium Designer Project

- The basis of every electronic product design is the project.
- The project links the elements of your design together, including the source schematics, the PCB, the netlist, and any libraries or models you want to keep in the project.
- The project also stores the project-level options, such as the error checking settings, the multi-sheet connectivity mode, and the multi-channel annotation scheme.
- There are six project types – PCB projects, FPGA projects, Core Projects, Embedded Projects, Script Projects and Library Packages (the source for an integrated library).
- Altium Designer allows you to access all documents related to a project via the **Projects** panel.
- Related projects can also be linked under a common Workspace, giving easy access to all files related to a particular product your company is developing.
- When you add documents to a project, such as a schematic sheet, a link to each document is entered into the project file. The documents can be stored anywhere on your network; they do not need to be in the same folder as the project file. If they do exist in a directory outside where the project exists or its sub-directories, then a small arrow symbol appears on the document's icon in the **Projects** panel.

1.2.2 Demo — Opening an existing Project

1. Select the **File » Open Project** menu to display the *Choose Project to Open* dialog.
2. Navigate to the project folder, `4 Port Serial Interface`, located in the `\Altium Designer Summer 09\Examples\Reference Designs` directory. Locate `4 Port Serial Interface.PRJPCB` (the project file) and double-click on it to open it.
3. The design will now be listed in the navigation tree of the Projects panel.
4. Click on the **–** signs to contract the folders.
5. Click on + (plus) signs to expand folders.
6. Right-click on the project name (4 Port Serial Interface.PrjPcb) to display the context sensitive Projects menu.

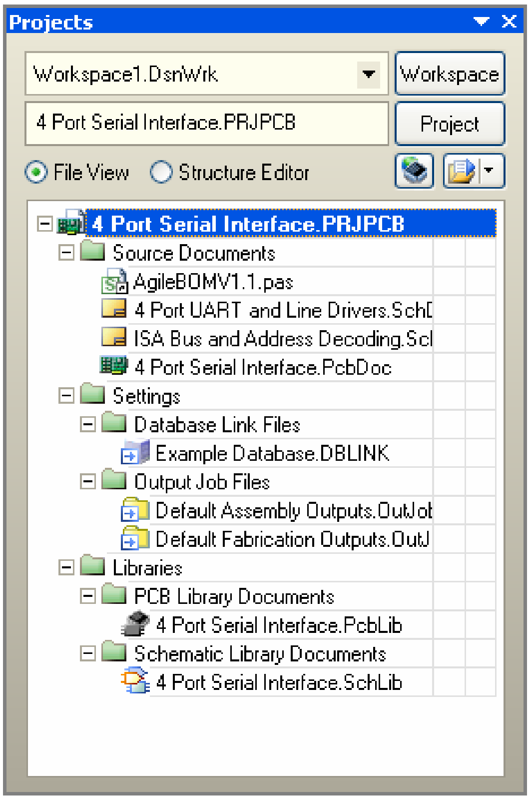

Figure 3. The open project is displayed in the Projects panel.

1.2.3 Editor View

Each different document kind is edited in an appropriate *Document Editor*, for example the PCB Editor for a PCB document, Schematic Editor for a schematic document, or VHDL Editor for a VHDL document. Figure 4 shows a schematic open for editing in the Schematic Editor.

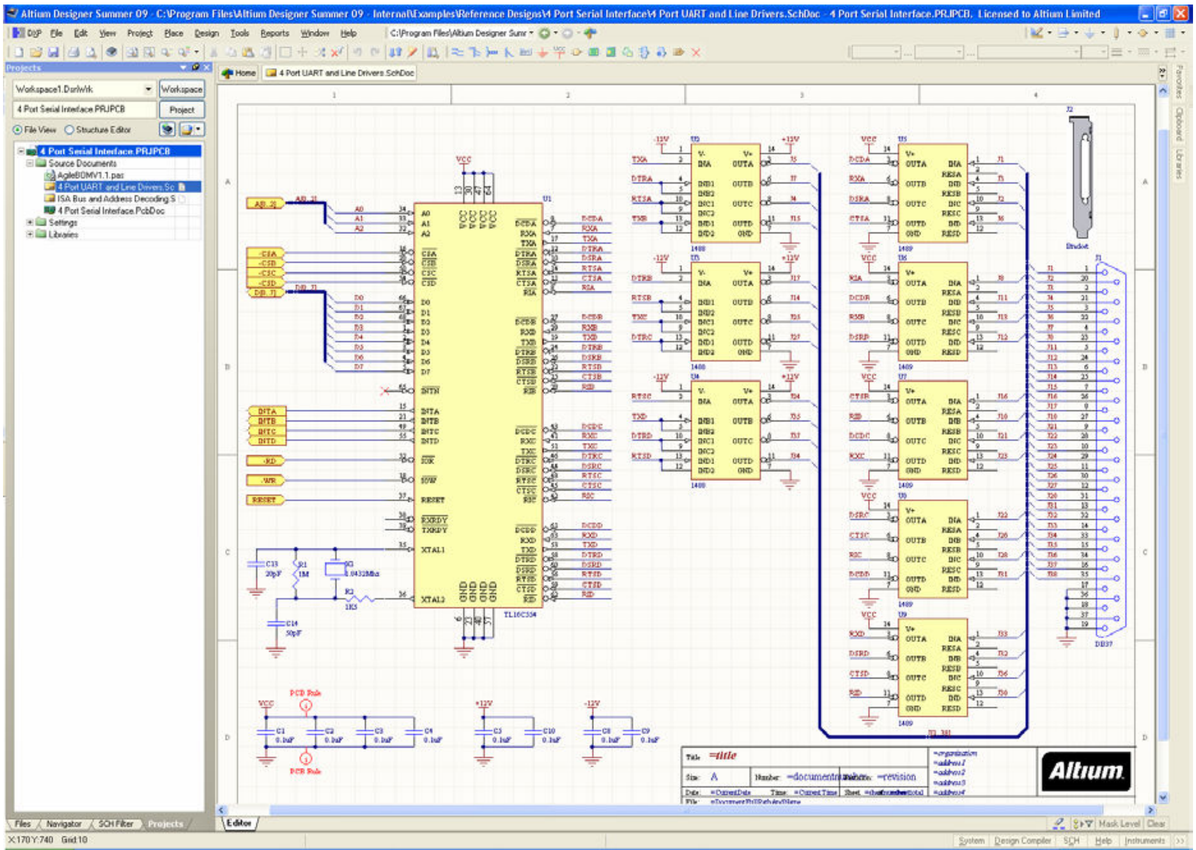

Figure 4. A schematic open for editing in the Schematic Editor View.

1.2.3.1 Document Tabs in the Documents Bar

Documents that are open are allocated a tab at the top of the application. Click on the relevant tab to display that document and make it the active document for editing. To switch between documents the *Ctrl* + *Tab* shortcut can be used. You can also tweak how *Ctrl* + *Tab* works in the preferences.

Figure 5. Tabs showing various documents open, note how the PCB tab is highlighted, indicating that it is the document currently being edited.

Right-click menu in the Documents Bar

1. Right-click on any document tab in the Documents bar.
2. Select **Tile All** from the floating menu that appears. All the opened documents are tiled in multiple screen regions.

Note: The number of opened documents determines the number of regions.

3. Right-click on a document tab.
4. Select **Close** from the menu.

5. Position the cursor at the point where two regions of a split screen meet and a double-headed arrow will display. Click and drag to resize.
6. Right-click on any one of the tabs in the tiled display and choose **Merge All**. Notice that you have converted a split screen back to a single view.

Note: Altium Designer supports multiple monitors. If your PC has multiple monitors you can use the **Open in New Window** command when you right-click on a document, or just drag and drop on to the second monitor and this will cause it to open in a separate Altium Designer application frame.

The right click menu also has options for saving and hiding individual documents as well as groups of documents like groups of schematics.

Note: There are a few options that you can tweak to gain more control of how the document bar works in Altium Designer. To do this go select **DXP » Preferences** and open the **System – View** page. At the bottom right is the *Documents Bar* section where things like auto hide, multi-line, ctrl-tab to switch can be set up.

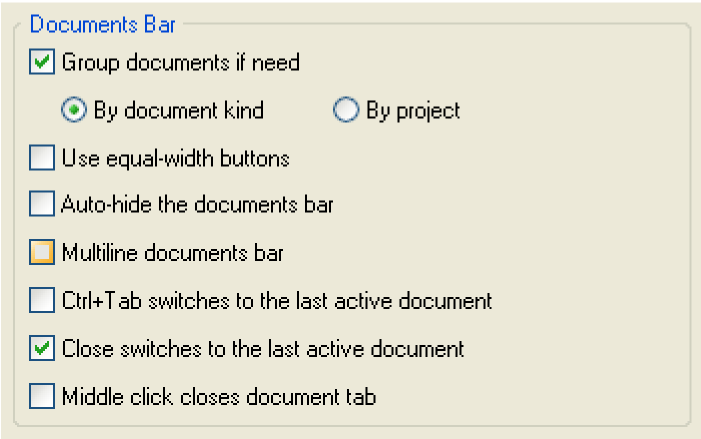

Figure 6. Document Bar options in Preferences

1.2.4 Exercises — Navigating around Altium Designer

1.2.4.1 Using the Projects panel

1. Open `4 Port Serial Interface.PRJPCB`, located in the `\Altium Designer Summer 09\Examples\Reference Designs\4 Port Serial Interface` folder.
2. Expand and then contract the contents of the navigation tree.
3. Double-click on a document in the Projects panel to open it.
4. Double-click on a few more documents in the Projects panel to open them.
5. Right click on the documents bar to see all the options.
6. Tweak some of the settings in preferences and see the results.

1.3 Document Editor Overview

To display a document in its editor, double-click on a document icon in the Projects panel. The document will be opened in the appropriate editor, e.g. Schematic Editor, PCB Editor or the Library Editors.

When you create a new document in a design you are required to select a document type, e.g. Schematic or PCB. The document type you select determines which editor is assigned to the document.

1.3.1 Working in a document editor

The sections below describe various elements in the user interface of the Altium Designer document editors.

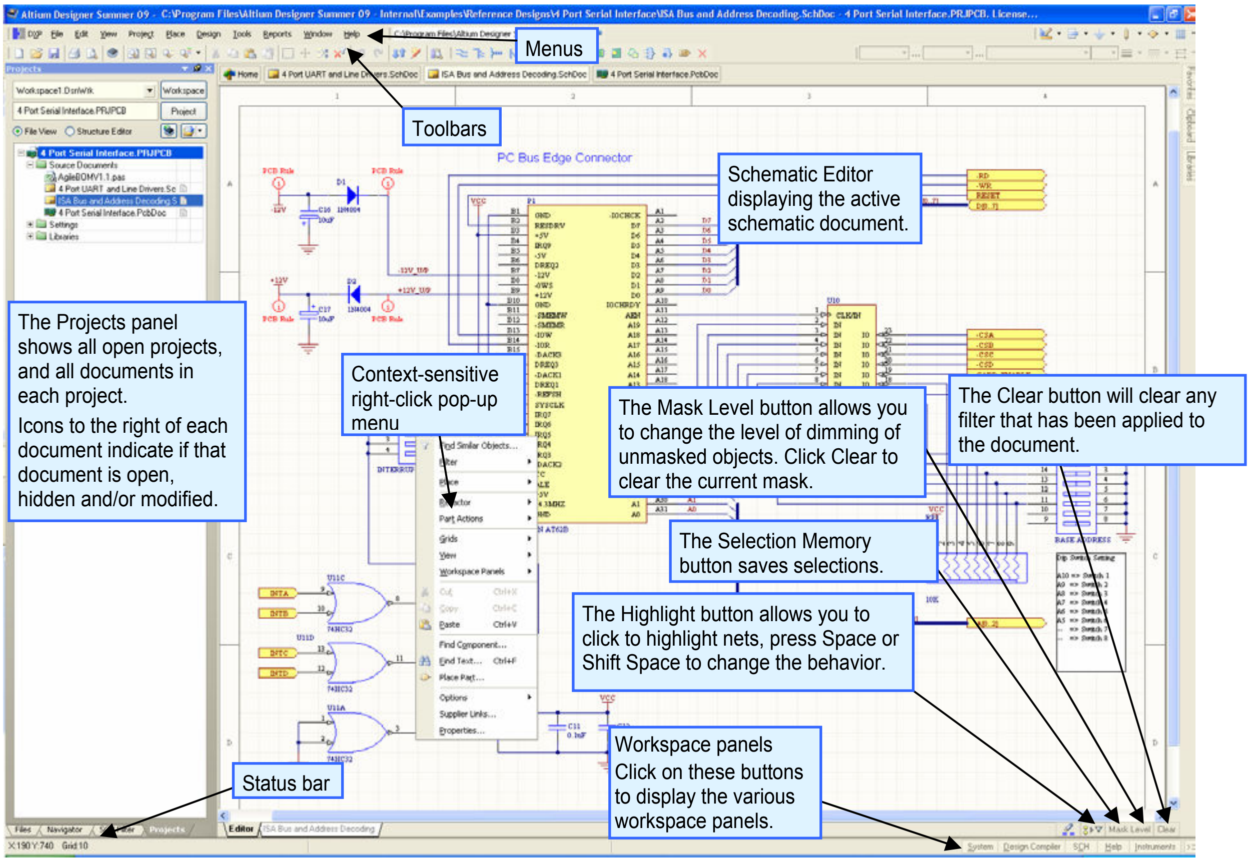

Figure 7. Schematic Editor Workspace

1.3.1.1 Menus

- Altium Designer menus are similar to standard Windows menus.
- Standard operations, e.g. opening, saving, cut, paste, etc. are consistent across editors.
- Right-click on an empty space on the menu bar or a toolbar caption to open the **Customization Editor** and customize any of the resources for that editor.

1.3.1.2 Shortcut keys and pop-up menus

- Menu commands can also be accessed using shortcut keys. The underlined letter indicates the shortcut key for a menu command, e.g. press **F** for the File menu.

- Special shortcut keys give direct access to both menus and sub-menus in the graphical editors, e.g. pressing **F** in the Schematic Editor will pop up the File menu and pressing **S** will pop up the Select sub-menu.

> **Note :** You can gain a list of every shortcut that is available in Altium Designer by looking for a document named **GU0104 Shortcut Keys.PDF** located in the Help directory of the Altium Designer Installation.

1.3.1.3 Toolbars

- Toolbars can be fixed to any side of the workspace or they can be floated.
- Click and drag to move a toolbar. The cursor must be within the toolbar but not actually on a button.
- Toolbars can be reshaped, hold the cursor over the edge of the toolbar and when the resizing cursor appears click and hold to reshape.
- New toolbars can be created and existing toolbars edited.
- Multiple toolbars can be active, right-click on a toolbar to pop up the toolbar display control menu.

1.3.1.4 System and Editor Panels

- Altium Designer uses two types of panels – system-type panels, such as the Files, Messages or Projects panels that are always available, and editor panels, such as the PCB, schematic library or PCB library panels that are only available when a document of that type is active.
- Panels can float, or be docked, on any edge of the Altium Designer workspace. Docked panels can be pinned open, or set to unpinned, where they pop out when their name button is clicked.
- Panels can be clipped together in a set by dragging and dropping one on another, and then dragged around as a set by clicking and dragging on the area of panel title bar that contains no text or icons.
- A panel can be unclipped from a set by clicking and dragging on the panel name.
- Panels can be prevented from docking on particular edges. Right-click on a panel title bar to configure this.

> **Note** : The hide and display speed of unpinned panels is configured in the **System – View** page of the *Preferences* dialog (**DXP » Preferences**). It can be useful to turn off the animation of panels on slower machines.

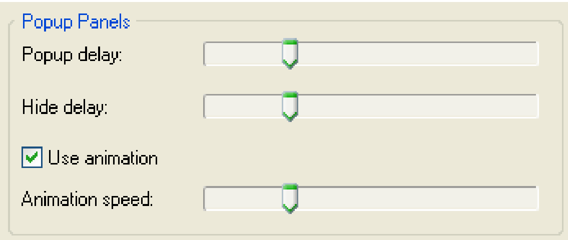

Figure 8. Configuration for panel control

1.3.1.5 Status Bar

- The Status Bar is used to display information to the user.
- The Status Bar consists of three display fields divided by separators and a set of panel display buttons. These three display fields are:
  - Cursor position
  - Prompt
  - Options.
- The fields can be re-sized by clicking and dragging on the separators.

- The Status Bar is turned on and off using the menu command **View » Status Bar.**
- The panel display buttons can be added or removed from the Status bar by clicking on the arrow button in the far bottom left.

1.3.1.6 Tool Tips

- Tool Tips provide a brief description of how to use a particular function.
- Position the cursor over a toolbar button and leave it stationary for about a second and the Tool Tip will appear.

1.3.1.7 Right mouse click context sensitive pop-up menus

- Altium Designer makes extensive use of context sensitive right mouse menus, including in panels and dialogs.
- Right-click anywhere in the environment to pop up a context sensitive menu of commands at the current cursor position. Supported right-click locations include:
  - in a document editor, on an object
  - in a document editor, in free space
  - in the different sections of a panel
  - on the Status bar
  - on a toolbar or menu bar
  - In dialogs, especially those with a grid of information.

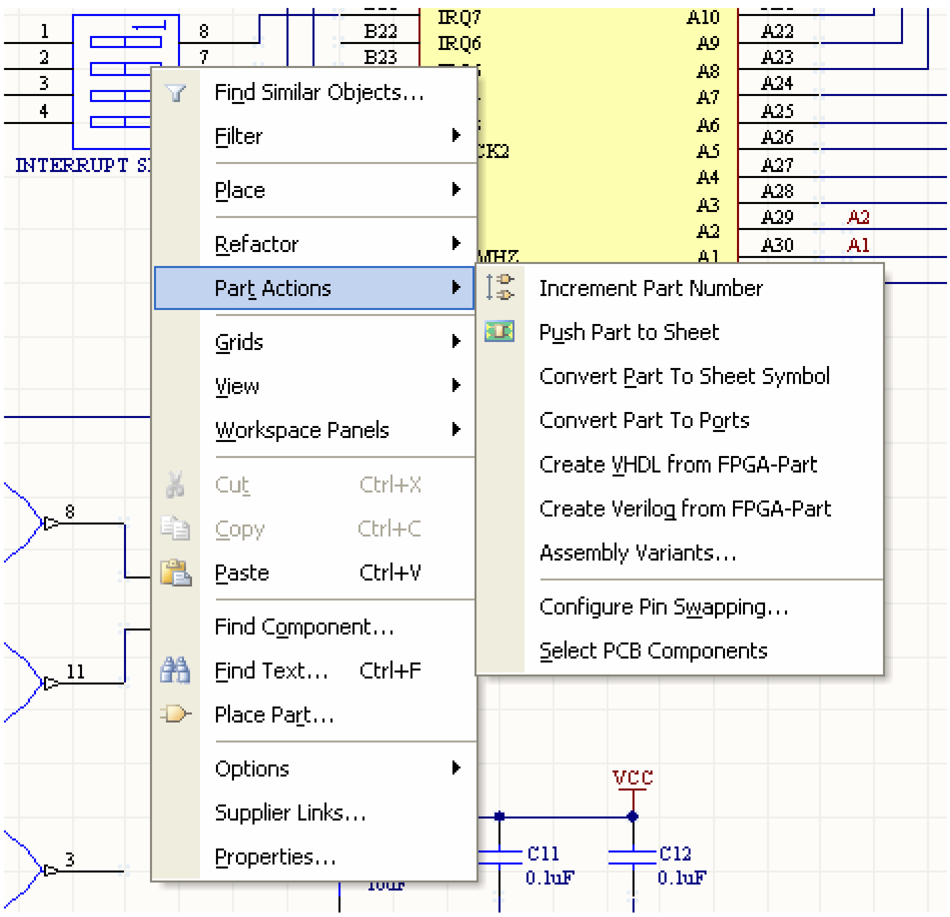

Figure 9. Context sensitive right mouse menus are available throughout Altium Designer

1.3.1.8 Dialogs

- Dialogs are used to set the parameters for various commands and objects.
- To move from one field to another in a dialog, press the *Tab* key or use the mouse. SHIFT+TAB takes you in the reverse direction.
- Most fields will have an underlined character associated with them that can be pressed (in combination with the ALT key) as an alternative to a mouse click.
- When a field is highlighted, typing can overwrite it.
- You'll find nearly all dialogs will have a question mark icon in the top right hand corner. Clicking on this icon activates the **What's This Help** (WTH) feature and will display a brief pop-up help message from the next control that you click on. For example, Figure 10 shows the WTH for the **Type** control in the component properties dialog.

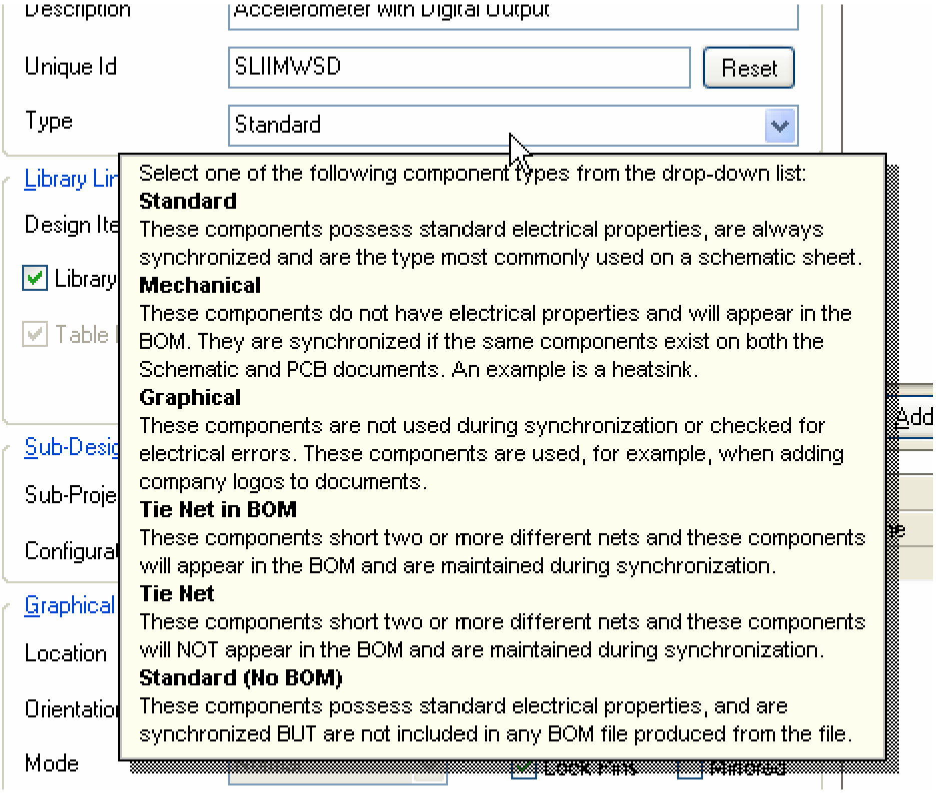

Figure 10. Using the What's This Help icon to gain help in a dialog

1.3.1.9 Undo/Redo

- Most commands can be undone or then redone using the Undo and Redo toolbar buttons. The number of schematic editor and PCB editor undos is set in the *Preferences* dialog (**DXP » Preferences**).
- The shortcut keys for Undo are CTRL+Z or ALT+BACKSPACE, and CTRL+Y or CTRL+BACKSPACE for Redo.

1.4 Working with projects and documents

A project is a set of documents that together define all aspects of your design: including schematic sheets, PCB documents, database link definition files, output job definition documents, netlists, and so on. Each project results in a single implementation, for example a PCB project results in one PCB design, and a library package project results in a single integrated library.

Each document in the project is stored as a separate file on the hard drive. The project file itself is also an ASCII document, which includes links to the documents in the project, as well as storing project-level settings.

1.4.1 Creating a new project

To create a new PCB project:

1. From the Main Menu, select **File » New » Project » PCB Project**.

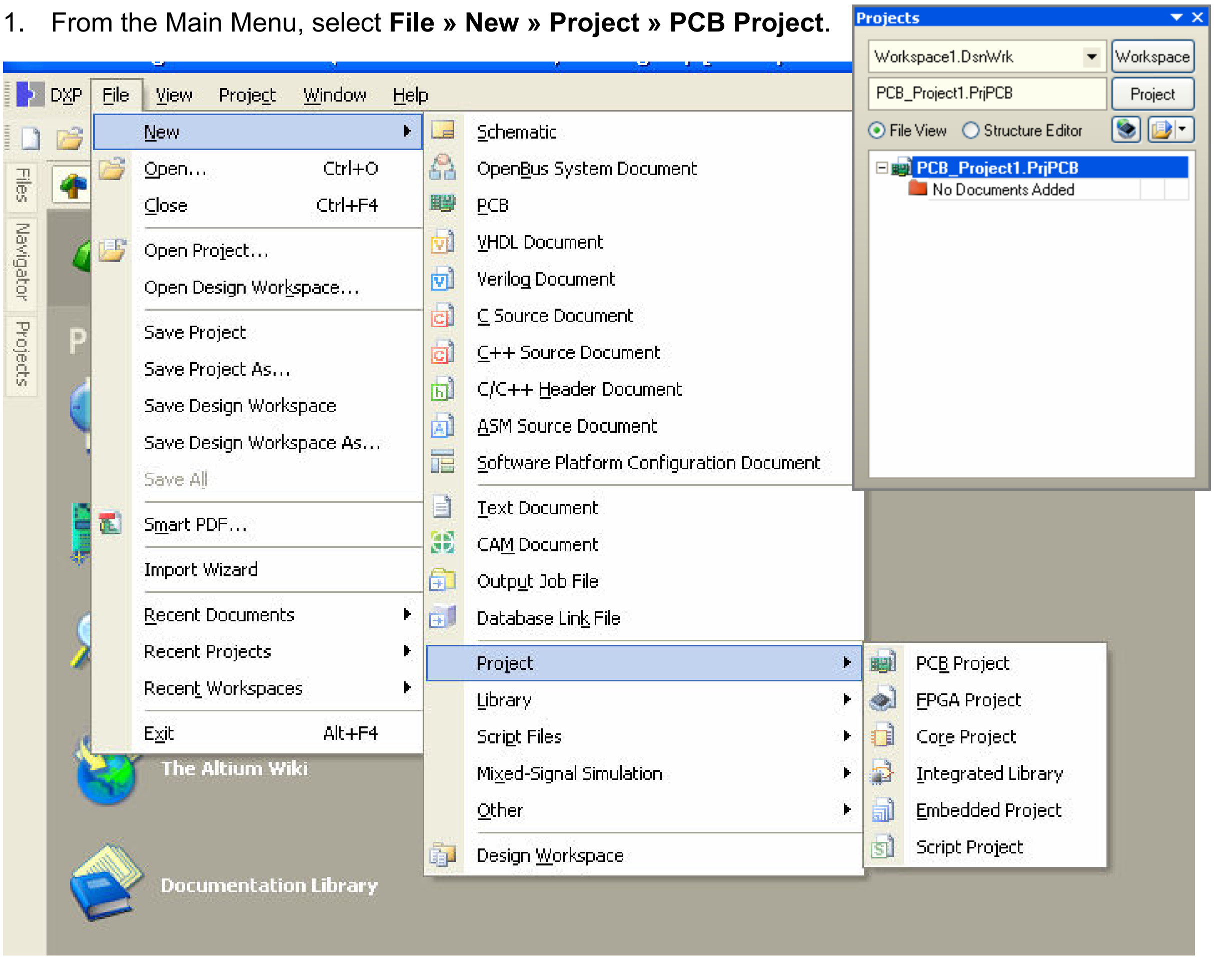

Figure 11. The new project is displayed in the Projects panel

2. Select **Save Project As** from the **File** menu to name and save the project document or you can right click on the project in the *Projects Panel* and select **Save Project As**.
3. The new project is ready to add new or existing documents to.

1.4.2 Adding a new document to the project

To add a new document to the project:

1. Right-click on the Project name in the Projects panel, and from the **Add New to Project** sub-menu, select the document kind, for example, **Schematic**.
2. Right-click on the new schematic document in the Projects panel and select **Save As** to name and save the schematic.

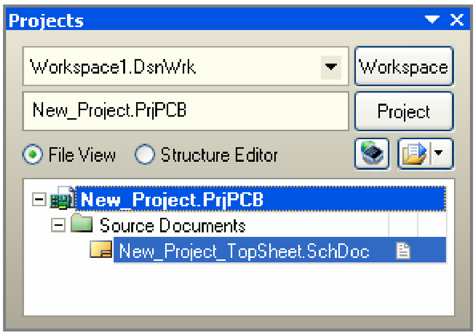

Figure 12. New schematic added to the

1.4.3 Adding an existing document to a project

To add an existing document to a project:

1. Right-click on the Project name in the Projects panel.
2. Select **Add Existing to Project** in the menu to display the *Choose Document to Add to Project* dialog.
3. Navigate to locate required file and select it.
4. Click on **Open** to add it. The document is added into the currently active project. Note that when you add a document to a project a link is added in the project file to that document. The document can be located anywhere on the hard disk (or network).

The document icon graphic indicates which Editor will be used to edit the document, e.g. a PCB document will have a PCB icon, indicating that it will be opened by the PCB Editor.

Note: You can add a document to a project using a two step process. First drag the document from the Windows File Explorer into the Altium Designer Projects panel and then when it appears as a Free Document, click and drag it into the project.

1.4.4 Moving or copying a document between projects

1. Since documents are only *linked* into the project, you can easily move a document from one project to another simply by clicking and dragging it.
2. To copy a document to another project, hold the CTRL key as you click and drag.

1.4.5 Removing a document from the project

To remove a document from a project, right-click on the document icon in the Project panel and select **Remove from Project**.

Note: The document is not deleted from the hard disk, but it is no longer linked into the project.

1.4.6 File management with the Storage Manager

The Storage Manager is a system panel that allows you to perform a variety of file management tasks. When you open the **Storage Manager** (**View » Workspace Panels » System » Storage Manager**) it presents a folder/file view of the active project's documents.

The **Storage Manager** can be used for:

- General everyday file management functions such as renaming and deleting files in the project or within the active project's folder structure.
- Management of Altium Designer backups, using the **Local History** feature.
- As a Subversion compliant interface for your Altium Designer projects.

Note: Right-click in the different regions of the panel for options.

- As a CVS compliant (Concurrent Versions System) interface for your Altium Designer projects.
- As an SCC (Source Code Control) compliant version control interface for your Altium Designer projects.
- Performing a physical and electrical comparison of any 2 versions in the **Local History**, or the CVS **Revision** list.

The **Folders** view on the left gives access to documents stored in the project folder hierarchy. Next to this the **File** list shows all documents in the selected folder. A number of highlighting modes are used to indicate the state of each document, press F1 when the cursor is over the panel for information on highlighting.

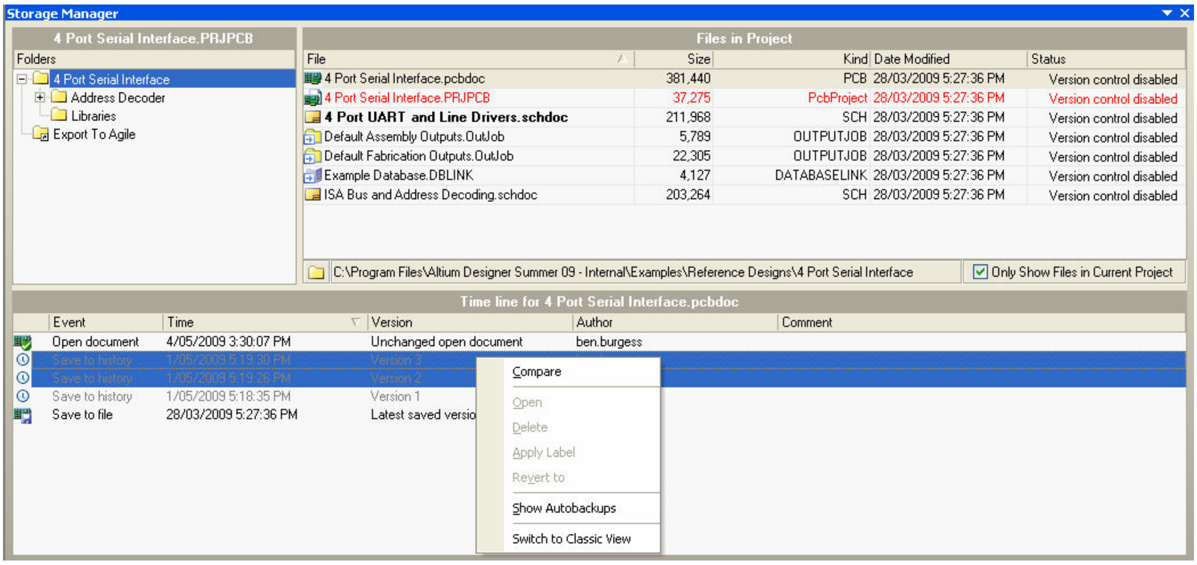

Figure 13. Use the Storage Manager to manage project files on the hard disk, and to interface to your Version control system.

Note: Press F1 over the panel for access to detailed help.

1.4.7 Including other files in the Altium Designer project

- You can include any file in your Altium Designer project, as long as the Microsoft Windows® operating system is aware of the file's associated editor.
- Add it to the project as described in section 1.4.3 (you will need to change the file filter to see non-Altium file types). The file will appear in the Project structure in the **Projects** panel, under a folder icon titled **Documentation**.

1.4.8 Libraries

- Libraries can exist as individual documents, for example, schematic libraries containing schematic symbols, PCB libraries containing PCB footprint models, discrete SPICE models (MDL and CKT), and so on.
- Altium Designer also supports the creation of integrated libraries. An integrated library is the compiled output from a library package. It includes all the schematic libraries in the original library package, plus any referenced models, including footprint, simulation and signal integrity models.
- Most of the supplied libraries are provided as integrated libraries and are stored within the `\Program Files\Altium Designer Summer 09\Library` folder. Integrated libraries can be converted back to their constituent libraries; simply open them in Altium Designer to do this. PCB libraries are also provided in the `\Program Files\Altium Designer Summer 09\Library\Pcb` folder.
- The Schematic Library Editor and PCB Library Editor are covered during the *Schematic Capture* and *PCB Design* training sessions. The basics of creating an integrated library are also covered.

> **Note**: You can use Protel 99 SE libraries directly in Altium Designer. Add them to the Libraries panel to use them without converting them to the Altium Designer format. Note that you will not get all the benefits of the enhanced parameter and model support.

1.4.9 Project Packager

An Altium Designer project can include many and varied files - source files, libraries, reports, data sheets, manufacturing files, etc. The Project Packager Wizard simplifies the task of managing and transferring the complete fileset. Guided by the settings you define the Project Packager Wizard gathers and packages the project into a portable time and date stamped ZIP file. The Project Packager supports:

- Any situation where your project must be moved, for example is moving it from one site to another, or backing up your project for secure storage.
- Packaging a complete Altium Designer project tree - ideal for linked PCB + FPGA + Embedded projects.
- Packaging a complete Altium Designer Workspace - ideal for designers that include all the board designs destined for a company product, in a single Workspace.
- Managing how directory paths are handled during packaging.
- Managing how files outside the project folder are handled during packaging.
- Including/excluding Generated files, such as reports, in the project package.
- Including/excluding History files (created by Altium Designer's built-in file history/restore system).

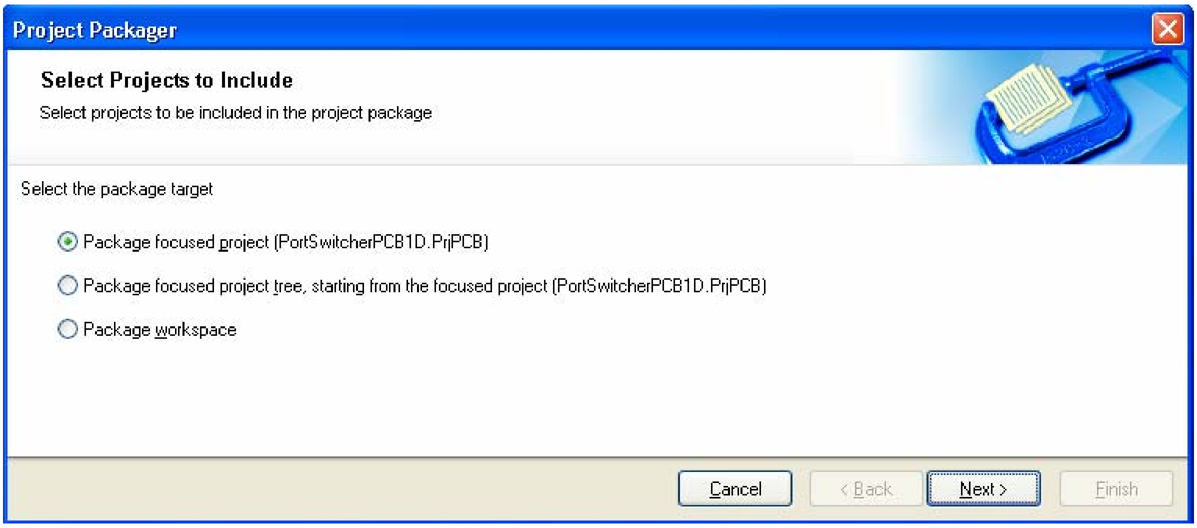

Figure 14. Project packager project choice page

1.4.10 Exercise – Working with projects and documents

This exercise looks at creating a new project and adding documents to it.

1. Create a new PCB project in the `\Altium Designer Summer 09\Examples\Training\PCB Training\Temperature Sensor` folder and name it `Temperature Sensor.PrjPCB`. We will use this project later during the Schematic Capture training session.
2. Add the following two schematic documents to the project from the `\Altium Designer Summer 09\Examples\Training\PCB Training\Temperature Sensor` folder: `LCD.SchDoc` and `Power.SchDoc`. Use **Add Existing to Project…** command from the right click menu in the **Projects** panel.
3. Save and close the new project `Temperature Sensor.PrjPCB`.
4. Check that the documents exist on the hard drive using the Windows Explorer