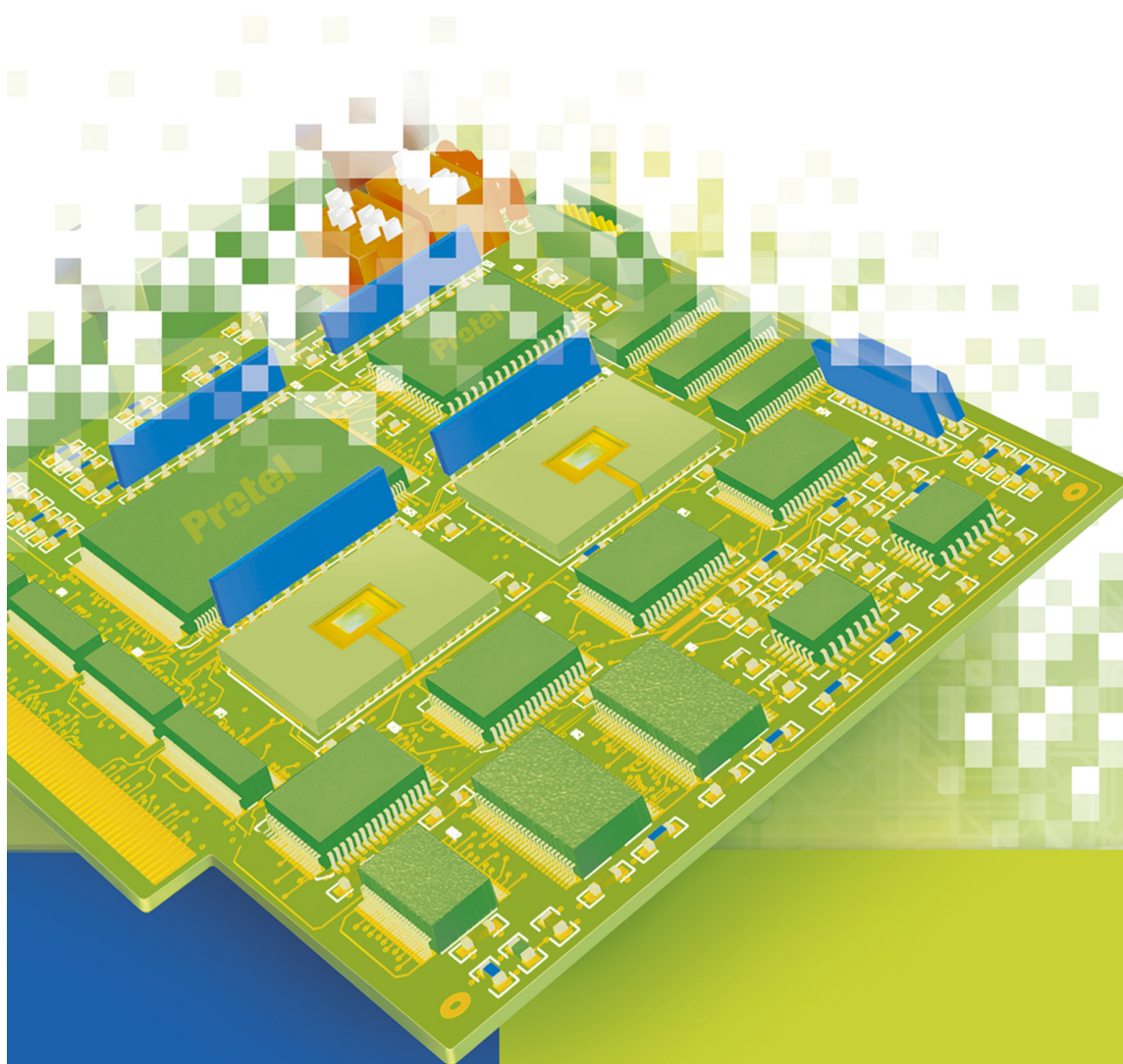


Integrated Libraries

Protel*DXP*TM

Tutorial



Protel[®]

Board-level design system from Altium.

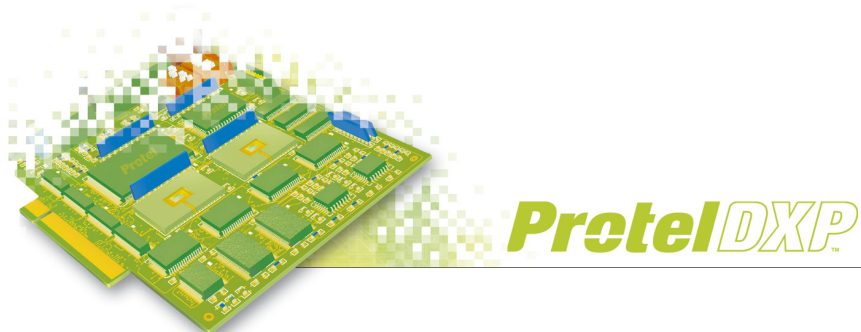


Table of Contents

DXP integrated libraries	2
Using DXP integrated libraries	2
Adding and removing libraries	3
Finding a component in integrated libraries	3
Creating an integrated library	5
Creating a schematic library	5
Creating a PCB library	6
Creating the source Library Package	6
Adding source libraries to the Library Package	7
Adding models to the Library Package	7
Compiling the integrated library	10
Modifying an integrated library	10

DXP integrated libraries

This tutorial looks at using, creating and modifying integrated libraries in Protel DXP.

Integrated libraries combine schematic libraries with their related PCB footprints and/or SPICE and signal integrity models, all together in a non-editable form. All model information is copied into the integrated library from the model libraries or files and so all the component information is stored together, regardless of the location of the original source libraries. This makes integrated libraries truly portable.

Source libraries, including any number of schematic libraries and the related model libraries and files (PCB footprints, SPICE or signal integrity models) are added to a Library Package project file which is then compiled to generate an integrated library. To modify an integrated library, you must change the source library first and then recompile the integrated library.

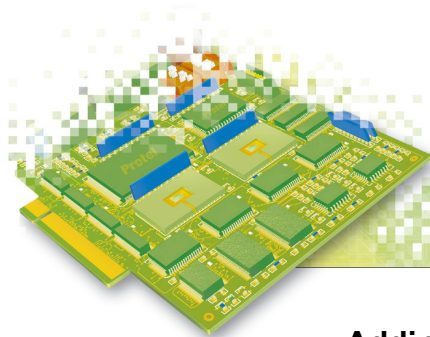
Protel DXP comes with a set of source libraries and integrated libraries (.INTLIB files) stored according to the manufacturer's name in C:\Program Files\Altium\Library. The schematic source libraries for the integrated libraries (.schlib files) are also included in folders by manufacturer's name. PCB footprint models are located in the C:\Program Files\Altium\Library\PCB folder in the form of PCB libraries (.pcblib files). SPICE models used for circuit simulation (.ckt files) are located in the SIM folder and signal integrity models are located in the SignalIntegrity folder within the C:\Program Files\Altium\Library folder.

Using DXP integrated libraries

Using an integrated library is very similar to using schematic libraries to place components and add model names. The only difference is that all the information about the component and its related models has already been added to the schematic symbol for you. You can check the **Models** list of the *Component Properties* dialog of a component to see what model names have been included with the schematic symbol. Model names can be changed or added from PCB or other model libraries once you have placed a component in a schematic sheet.

When the schematic is transferred from the Schematic Editor to a blank PCB using the **Design » Update PCB** command, the Source Reference Links fields of the *Component* dialog for each PCB footprint are populated with source library pathnames so you can easily trace where the components and models originated from if you need to change them.

Note that you can still always use any other Protel 99 SE or Protel DXP schematic or PCB libraries independently (without being in an integrated library) by adding them to the Library list as usual.

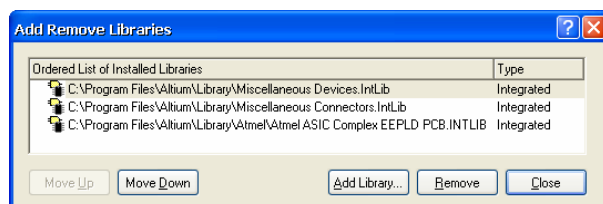


Adding and removing libraries

All libraries must be added to the Library list in the **Libraries** panel for the component symbols to become available for placement in a schematic and footprints for the components to be available when creating the PCB.

To add integrated libraries to the Libraries list:

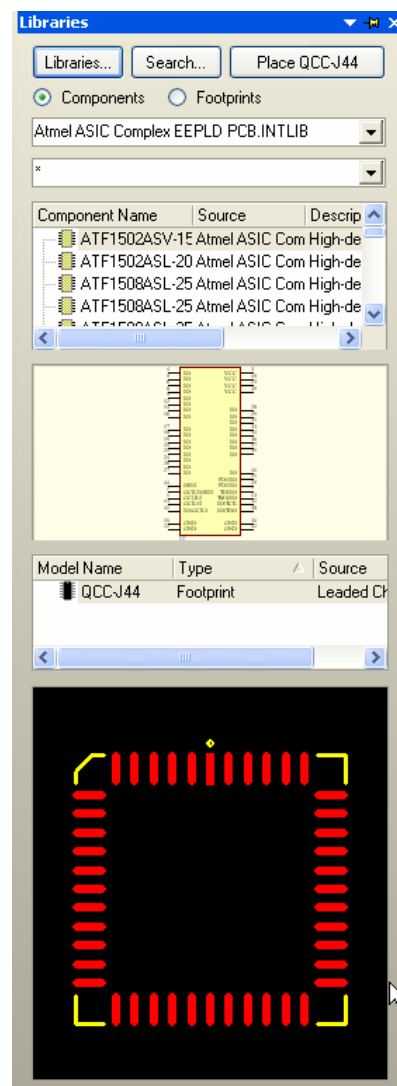
1. Click on the **Libraries** tab or select **View » Workspace Panels » Libraries**. The **Libraries** panel displays.
2. Click on the **Libraries** button at the top of the panel to open the *Add Remove Libraries* dialog.



3. Click **Add Library**.
4. Browse to the library you require in the *Open* dialog and click **Open**.
5. Click **Close** and the integrated library is added to the Libraries list in the **Libraries** panel. Select the library from the Libraries drop-down list to make it the active library.
6. Now you can select the component you wish to place from the Components list of the **Libraries** panel. Click **Place** to place it.

To remove a library from the Library list:

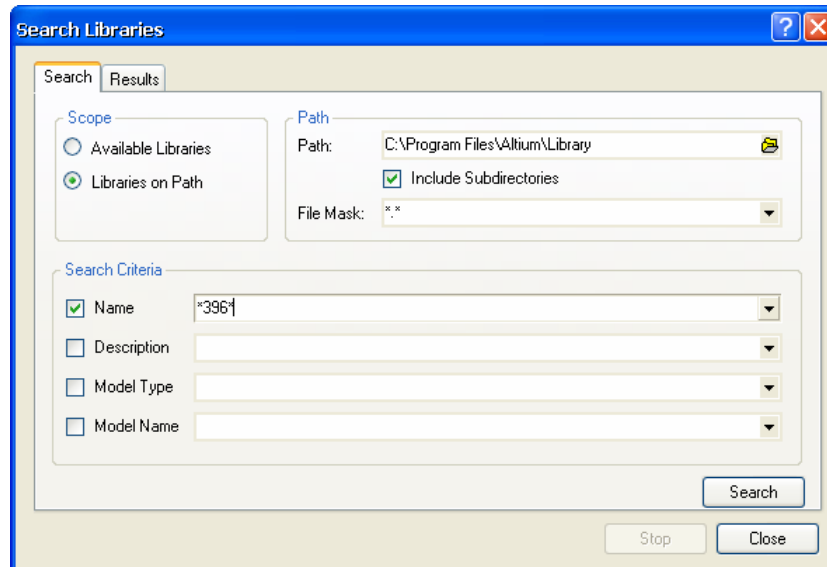
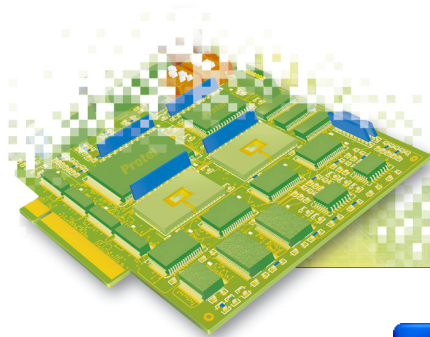
1. Click on the **Libraries** button at the top of the **Libraries** panel to open the *Add Remove Libraries* dialog.
2. Select the library you want to remove. Hold down the Shift key to multiple select libraries. Click on **Remove**.
3. You are asked to confirm your decision to remove selected libraries. Click **OK**.
4. The library pathname disappears from the Installed Libraries list. Click **Close**. The library is no longer available in the Library panel. Simply add it back in when required.



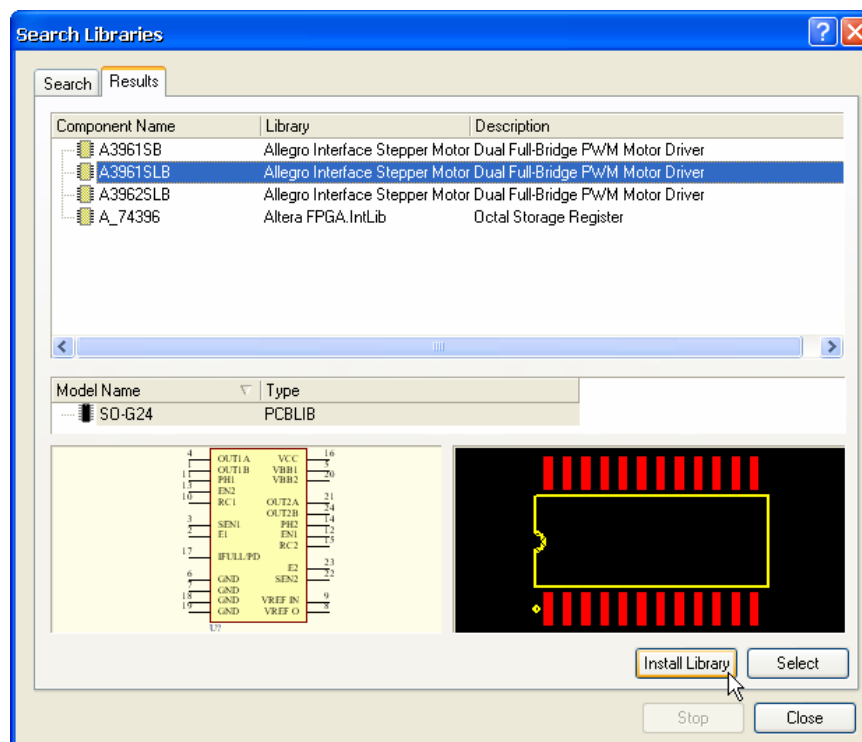
Finding a component in integrated libraries

If you do not know where the component you wish to use is located, use the Search Libraries facility.

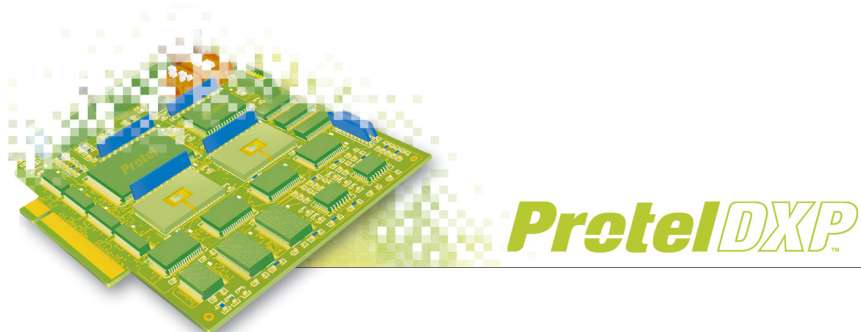
1. Click on the **Libraries** panel tab and the **Libraries** panel displays.
2. Click on the **Search** button at the top of the Libraries panel to open the *Search Libraries* dialog.



3. Select a Scope for searching in installed libraries or libraries on the search path you nominate by clicking on the folder icon in the Path field. Fill in the required Search Criteria filters making sure the filters you wish to use are ticked. Note that the Name field is the name of the schematic component. Click **OK** to start the search.
4. The integrated libraries that contain a match will display in the **Results** tab, together with component names, model names, symbols and footprints.
5. Click on a component name to display its model name and graphical representations.



6. When you have found the component you require, click on **Install Library** to add the selected library to the Libraries list in the **Libraries** panel.



7. If you have the schematic sheet open that you want to place the found component on, click on the **Select** button to switch to the Schematic Editor. The component symbol is selected in the Libraries panel, ready for placement. Click on the **Place** button to place the component.
8. Press **TAB** to display the *Components Properties* dialog while placing the symbol to set the designator.

Component Properties

Properties

Designator: ☒ Visible

Comment: ☒ Visible

☐ Don't Annotate Component

Part 1/1

Library Ref:

Library:

Description:

Unique Id:

Sub-Design:

Graphical

Location X: Y:

Orientation:

☐ Mirrored ☐ Show Hidden Pins

☐ Local Colors ☒ Lock Pins

Parameters list for U? - A3961SLB

Visible	Name	Value	Type
<input type="checkbox"/>	Class	Interface	STRING
<input type="checkbox"/>	Manufacturer	Allegro	STRING
<input type="checkbox"/>	Published	13/10/2000	STRING
<input type="checkbox"/>	Publisher	Altium Hobart Technology Cen	STRING
<input type="checkbox"/>	Sub-Class	Stepper Motor Controller/Drive	STRING

Models list for U? - A3961SLB

Name	Type	Description
S0-G24	Footprint	

9. Check the **Models** list to check that all the required model information is already added from the integrated library.
10. Click **OK** and then click to place the component symbol on the schematic sheet. Right-click or press **ESC** to end component placement mode.

Creating an integrated library

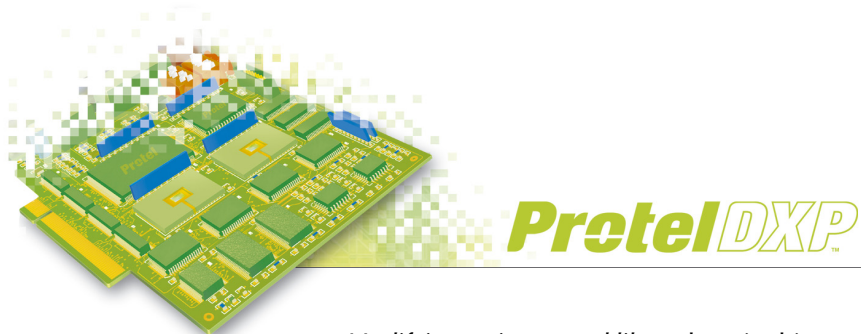
A Library Package is created first with at least all the schematic libraries added and pathnames can be set to the models libraries. Using the Project commands, the Library Package is then compiled to create the integrated library. Any errors generated during the compiling of the integrated library are displayed in the Messages panel for analysis.

Creating a schematic library

Before you can add any schematic libraries to the library package, you need to create some! You can create a schematic library out of the components that have been already placed on schematic documents in a project.

If a schematic file is not part of a project, you can still create a schematic library from it when it is open. The only difference is that it will not be added to a project and will display as a free document in the Projects panel when created.

Alternatively, you can create a schematic library from scratch using the **New » Schematic Library** command. Then create your own components using the Schematic Library Editor, or copy in components from other open schematic libraries using the **Tools » Copy Component** command. See



Modifying an integrated library later in this tutorial for more information about extracting a schematic library from an existing integrated library.

To create a schematic library from components in all schematic documents in a project:

1. Open the documents in the project by right-clicking on the project filename in the **Projects** panel and selecting **Open Project Documents**.
2. With the schematic documents that contains all the components you want to add to the new schematic library already active, select **Design » Make Project Library**.
3. The new schematic library will open in the Schematic Library Editor when it is created. All the components in the open schematic files are copied to the new schematic library, named `Projectname.SchLib`, stored in the same folder as the project file. The filename will appear in the **Projects** panel listed under **Schematic Libraries**.
4. Rename the new schematic library using **File » Save As** and close it.

Creating a PCB library

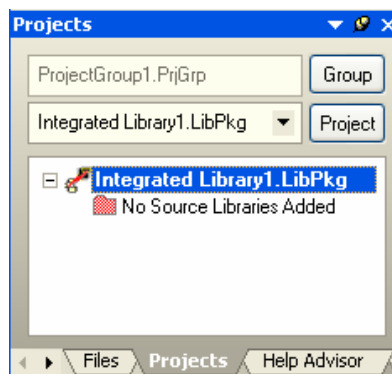
PCB libraries are supplied with Protel DXP and are stored in the default location of `\Program Files\Altium\Library\PCB`. However, you can create your own PCB library of footprints from an open PCB file, in a similar manner to creating a schematic library.

1. With the PCB document that contains all the footprints you want to add to the new PCB library already active, select **Design » Make PCB Library**.
2. The new PCB library will open in the PCB Library Editor when it is created. All the footprints in the open PCB document will be copied to the new PCB library named `PCBfilename.lib`, which is stored in the same folder as the source PCB document. The filename will appear in the **Projects** panel as a free document.
3. Rename the new PCB library using **File » Save As** and close it.

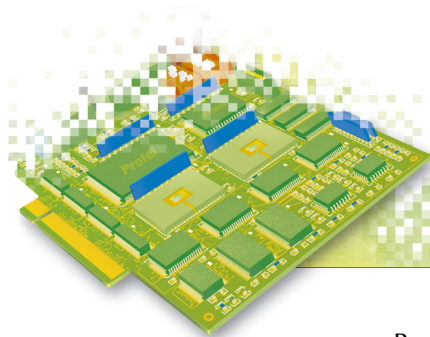
Creating the source Library Package

Now that we have some schematic libraries created, we need to add these to a library package which in turn is compiled into an integrated library.

1. Select **File » New » Integrated Library**. Alternatively, you can click on **Create a new integrated library Project** in the **Pick a task** pane which displays when no editors are open, or click on **Blank Project (Library Package)** in the **New** section of the **Files** panel.
2. The **Projects** panel displays with an empty Library Package file named `Integrated Library1.LibPkg`. There are no source libraries (schematic or PCB libraries) added to the Library Package at this stage.



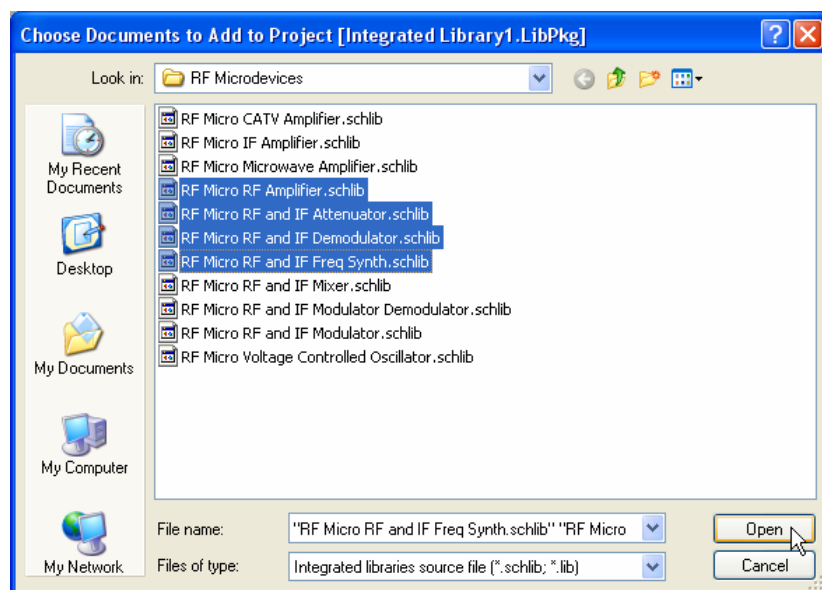
3. Rename the new Library Package using the **File » Save As** command and save it (with a `.LibPkg` extension) to your chosen location. An output folder is automatically created named `Project Outputs` for `Integrated Libraryname` in the folder you chose. The pathname to the Library



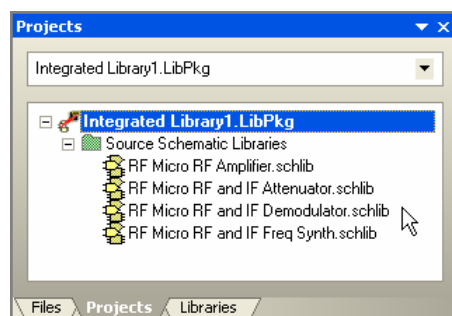
Package file is added to the Output Path field in the **Options** tab of the *Project Options* dialog (**Project » Project Options**) and the final integrated library file will be saved to this location when it is compiled.

Adding source libraries to the Library Package

1. Add in the source libraries to the Library Package by selecting **Project » Add to Project** or right-click on the selected .LibPkg file and select **Add to Project**. The *Choose Documents to Add to Project [IntegratedLibraryname.LibPkg]* dialog displays.

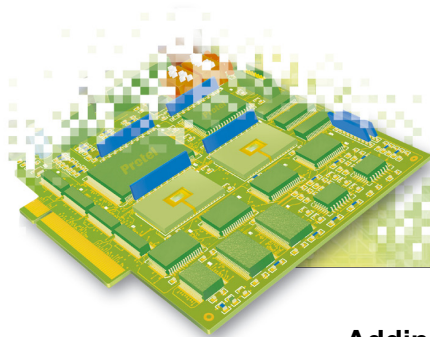


2. Browse to find the schematic libraries (.schlib) that you want to add to your Library Package. The schematic components store all the information needed to find related models in their *Component Properties* dialogs, so these are the most essential elements to be included in an integrated library.
3. Click **Open** and the added libraries are listed as Source Schematic Libraries in the **Projects** panel.



Adding models to the Library Package

You can add PCB footprint libraries (.pcb1ib), Protel 99 SE libraries (.lib), SPICE models or Signal Integrity models to a Library Package in order to keep related libraries in one integrated library (see *Adding models as source libraries* below). Note that this is optional. If you do not want to add in the model libraries and files, you can set a pathname to where they reside on your hard disk. See *Setting the pathname to model libraries and files* below.



Adding models as source libraries

Add model libraries, e.g. PCB libraries, to the Library Package in the same way as the schematic libraries are added.

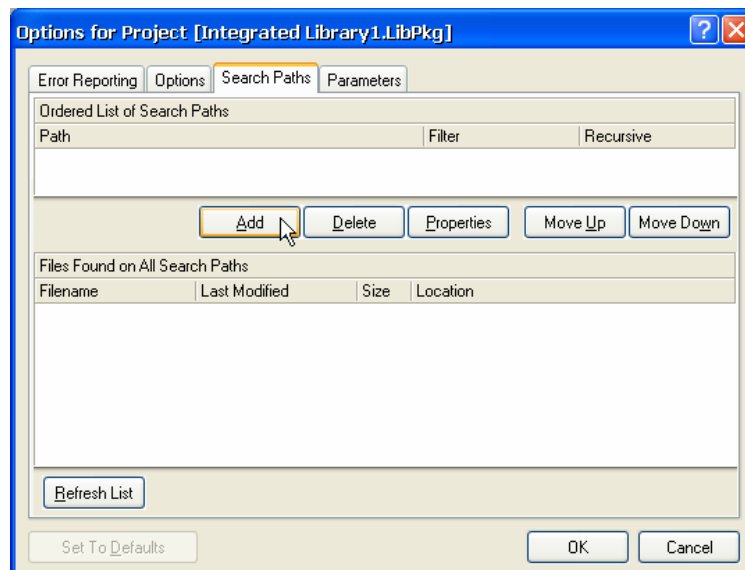
1. Select **Project » Add to Project**, or right-click on the selected .LibPkg file and select **Add to Project**.
2. Browse to find the model libraries which you want to add to your Library Package.
3. Click **Open** and the added libraries are listed as Source PCB Libraries in the **Projects** panel.


Setting the pathname to model libraries and files

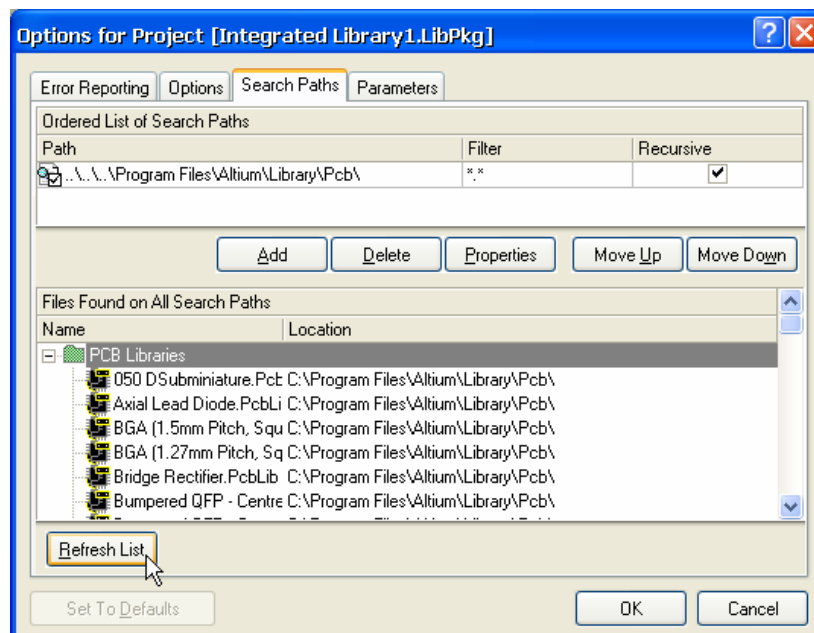
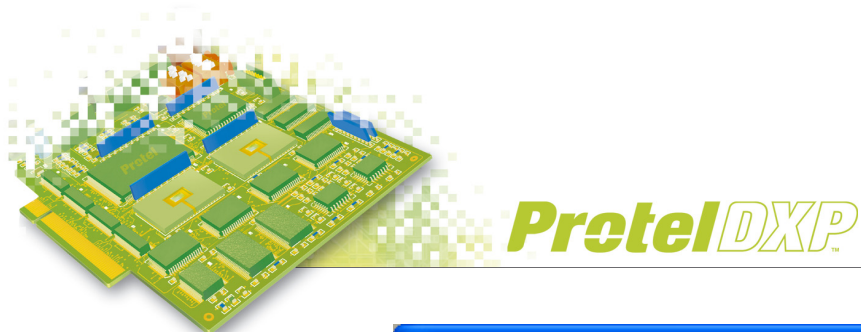
Alternatively, if the PCB footprints libraries, SPICE models or signal integrity models are not added to the Library Package, the schematic symbols in the integrated library will refer to them using the pathname set up in the *Project Options* dialog and stored in the Library Package project file.

1. Set up the pathname to the PCB libraries you want used by the schematic symbols in the integrated library by selecting **Project » Project Options**, or right-click and select **Project Options** when in the **Projects** panel. Click on the **Search Paths** tab of the *Project Options* dialog.

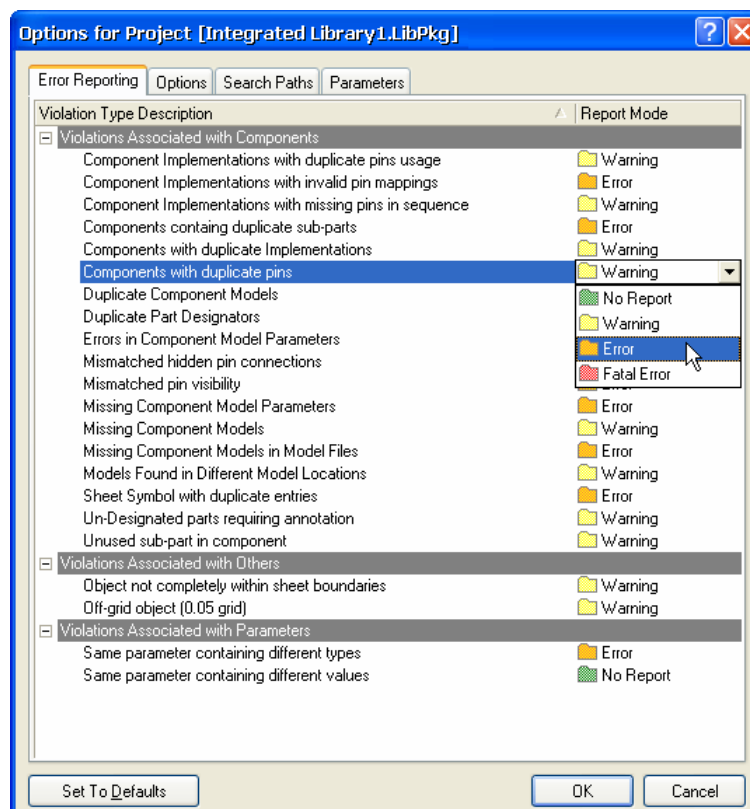
The current location of the new Library Package will be included as one search path in the Ordered List of Search Paths next time you open the Library Package.



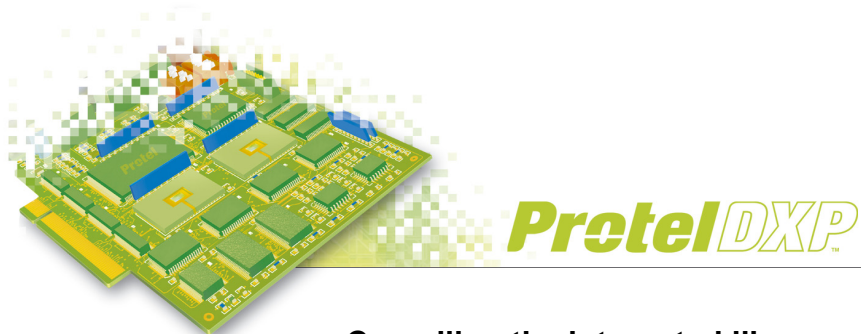
2. Add in the pathnames to the location of the footprints and models required by clicking on **Add** in the **Ordered List of Search Paths** section of the **Search Paths** tab.
3. Browse to the folders required in the *Edit Search path* dialog by clicking on the  button and locating the required model libraries and clicking **OK**. In the example below, we have added in the pathname to the folder where PCB footprint models are located, i.e. C:\Program Files\Altium\Library\PCB.
4. Click on **Refresh List** to confirm that the models are located correctly and click **OK**.



5. While you have the *Options for Project* dialog open, click on the **Error Reporting** tab to see what type of errors and warnings could be generated when the integrated library is compiled.



6. You can change the severity of the violation by clicking on the Report Mode next to the required violation type and selecting another mode from the dropdown list. Click **OK** to save the project options and close the dialog.



Compiling the integrated library

Once you have added the libraries and set any pathnames required, compile them to create the integrated library.

1. Select **Project » Compile Integrated Library** or right-click on the selected Library Package (.LibPkg) file and select **Compile Integrated Library**.
2. Protel DXP compiles the source libraries and model files into an integrated library. The compiler checks for any violations, such as missing models or duplicate pins, that have been set in the **Error Checking** tab of the *Project Options* dialog (**Project » Project Options**). Any errors or warnings found during compilation are displayed in the **Messages** panel. Fix any inconsistencies in the individual source libraries at this point and recompile the integrated library. See *Modifying an integrated library* for more information.
3. A new Integrated Libraryname.INTLIB is generated, saved in the output folder nominated in the **Options** tab of the *Project Options* dialog and displays in the **Library** panel, ready to use. The integrated library is automatically added to the current Libraries list in the **Libraries** panel.

Modifying an integrated library

The integrated libraries are used to place components and cannot be edited directly. To make changes to an integrated library, make modifications in the source libraries first and then recompile the integrated library to include the changes.

1. Open the integrated library (.IntLib) that contains the source library you need to modify. Select **File » Open** and browse to the integrated library required in the *Document to Open* dialog and click **Open**.
2. Confirm that you do wish to open the integrated library to extract the source libraries, not just add the integrated to the Libraries panel. Click **Yes**.

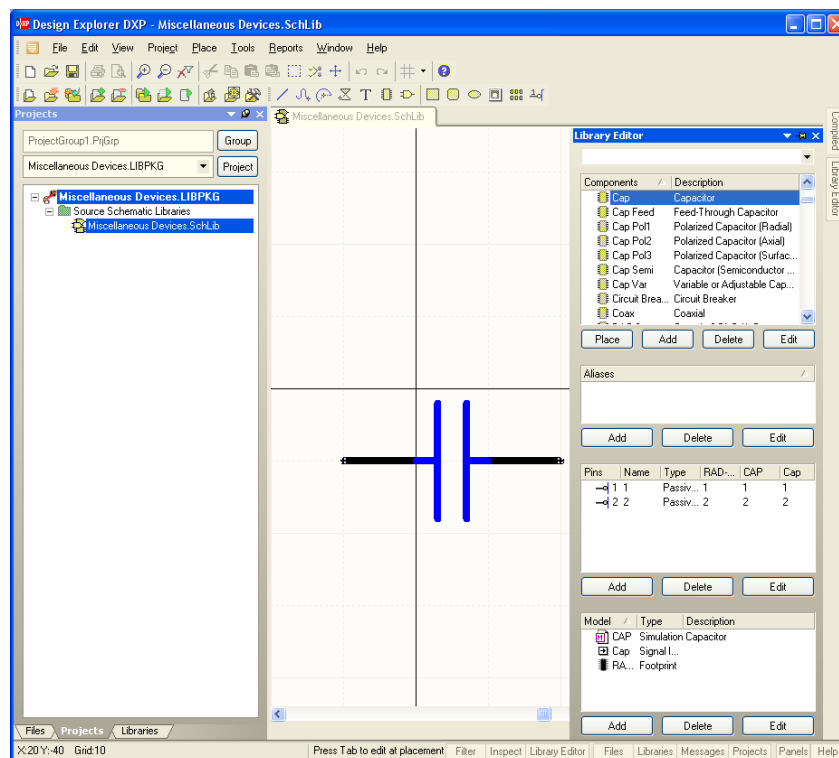
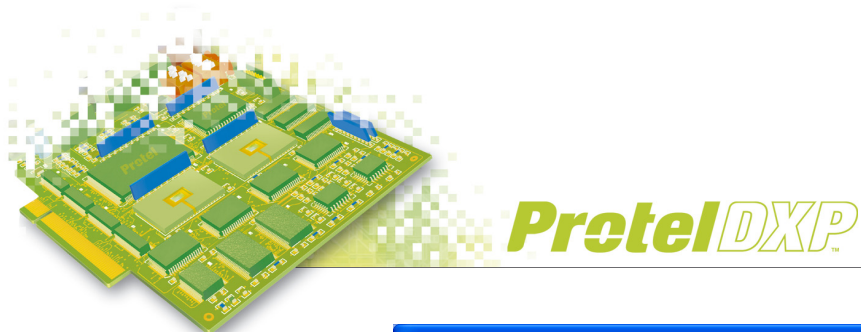
The source schematic and model libraries are generated and saved in a new folder named Integrated_libraryname, which is created in the folder storing the integrated library.

A Library Package (integrated_libraryname.LibPkg) is also created and the source schematic libraries are extracted and listed in the **Projects** panel. PCB libraries (.PcbLib) are generated as well and stored in the new library package folder but are not automatically added to the Projects panel. The pathname in the **Search Paths** tab of the *Project Options* dialog (**Project » Project Options**) indicates where the schematic components will search for when the footprints and model files are required.

3. Open the source library file you want to change. e.g. libraryname.schlib, by double-clicking on the library name in the Source Schematic Libraries list in the **Projects** panel. The library opens in the Schematic Library Editor.

If you wish to modify a footprint, you would have to add in the required PCB library before you could edit the models. Click on the **Libraries** button in the **Library** panel. Alternatively, you could use **File » Open** to open a model file.

4. Click on the **Library Editor** tab to activate the Schematic Library Editor.



5. Select the component you wish to alter from the Components list. You can make graphical changes to the component symbol by directly editing in the design window. To change component properties, such as model names, click again on the selected component in the Components list of the Library Editor panel. Alternatively, click **Edit** in the Library Editor or select **Tools » Edit Component**. The *Component Properties* dialog displays.
6. Make required alternations, such as adding a new model name to a symbol by clicking the **Add** button on the **Models** list of the *Component Properties* dialog and browsing to the location of the new model. Click **OK** to close the dialog.
7. Save the modified source library by selecting **File » Save** and close the library.
8. Select the Library Package in the **Projects** panel and select **Project » Compile Integrated Library**.
9. The integrated library is recompiled and any errors are listed in the **Messages** panel. If there are no errors or warnings, the modified integrated library is added to the Libraries panel and is ready to use.
10. Close the Library Package and save it to the same folder as the source libraries.

Thanks for participating in this tutorial.