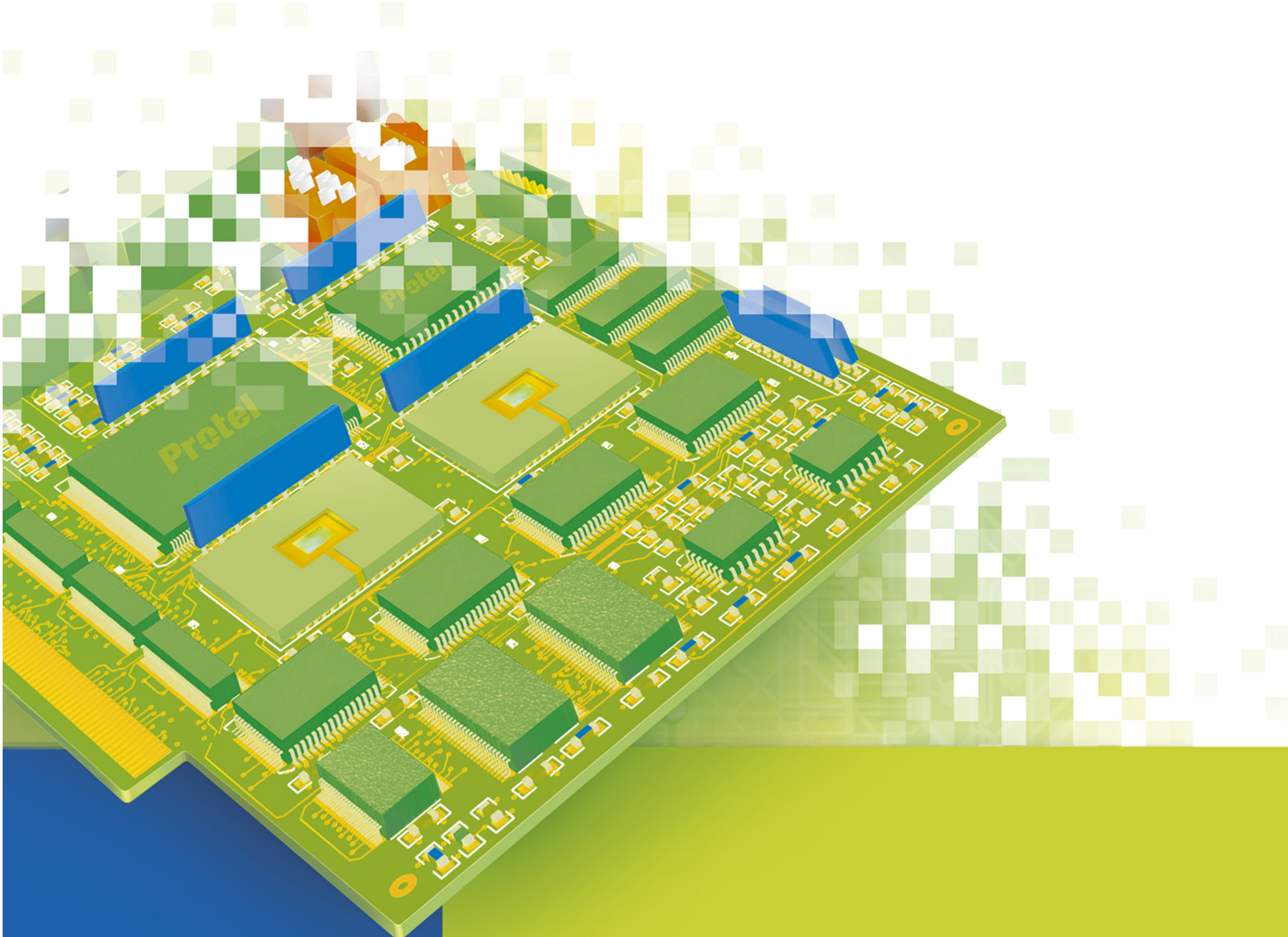
DESIGN EXPLORER DXP TECHNOLOGY BY ALTIUM

Introducing Protel DXP™

Tutorial



Protel®
Board-level design system from Altium

A step-by-step introduction to
Altium's complete board-level design system

Table of Contents

Welcome to Protel DXP

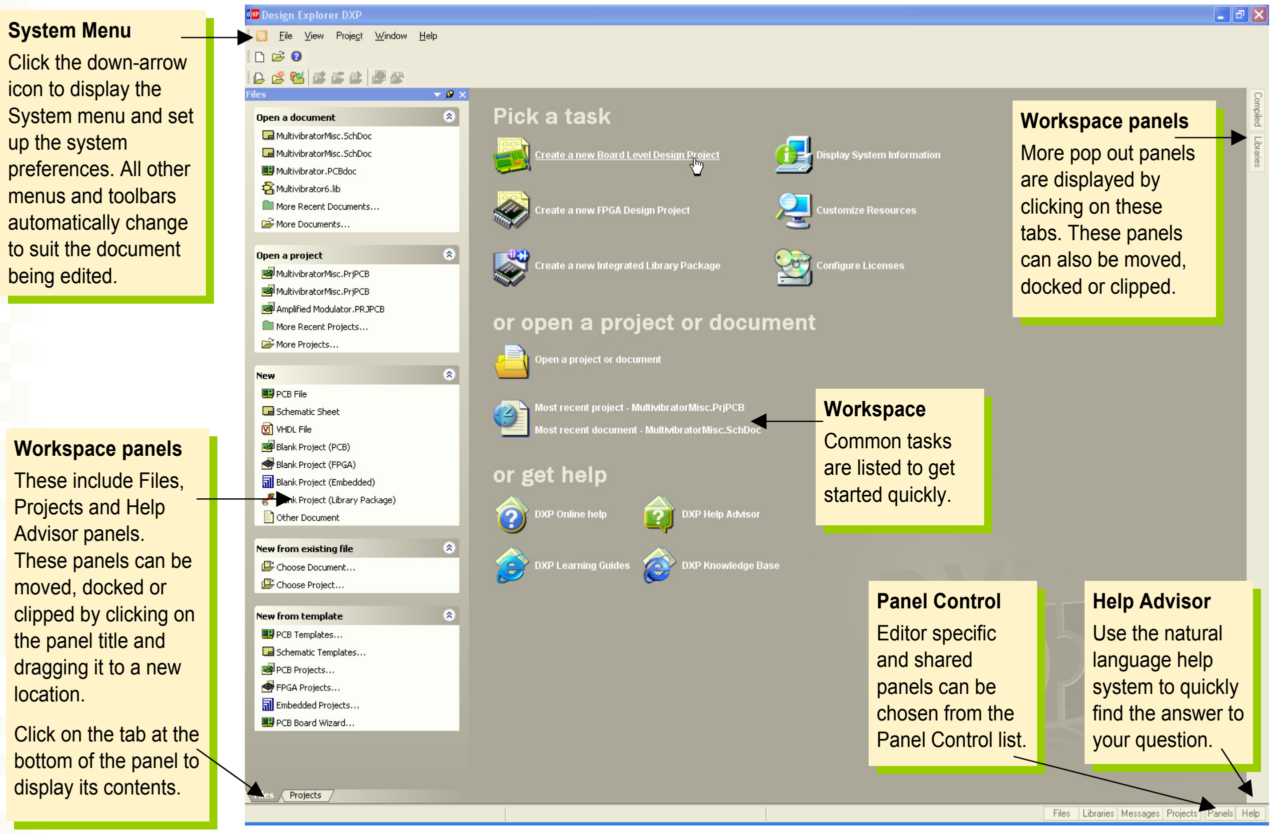
Welcome to the world of Protel DXP – a complete 32- bit electronic design system for Windows NT/XP. Protel DXP provides a completely integrated suite of design tools that lets you easily take your designs from concept through to final board layout.

All Protel DXP tools run within a single application environment – the *Design Explorer*. Start Protel DXP and the Design Explorer opens, putting all your design tools at your fingertips. You benefit from a single, consistent, customizable user environment.

This tutorial is designed to give you an overview of how to create a schematic, update the design information to a PCB and generate manufacturing output files. It also investigates the concept of projects, integrated libraries and circuit simulation.

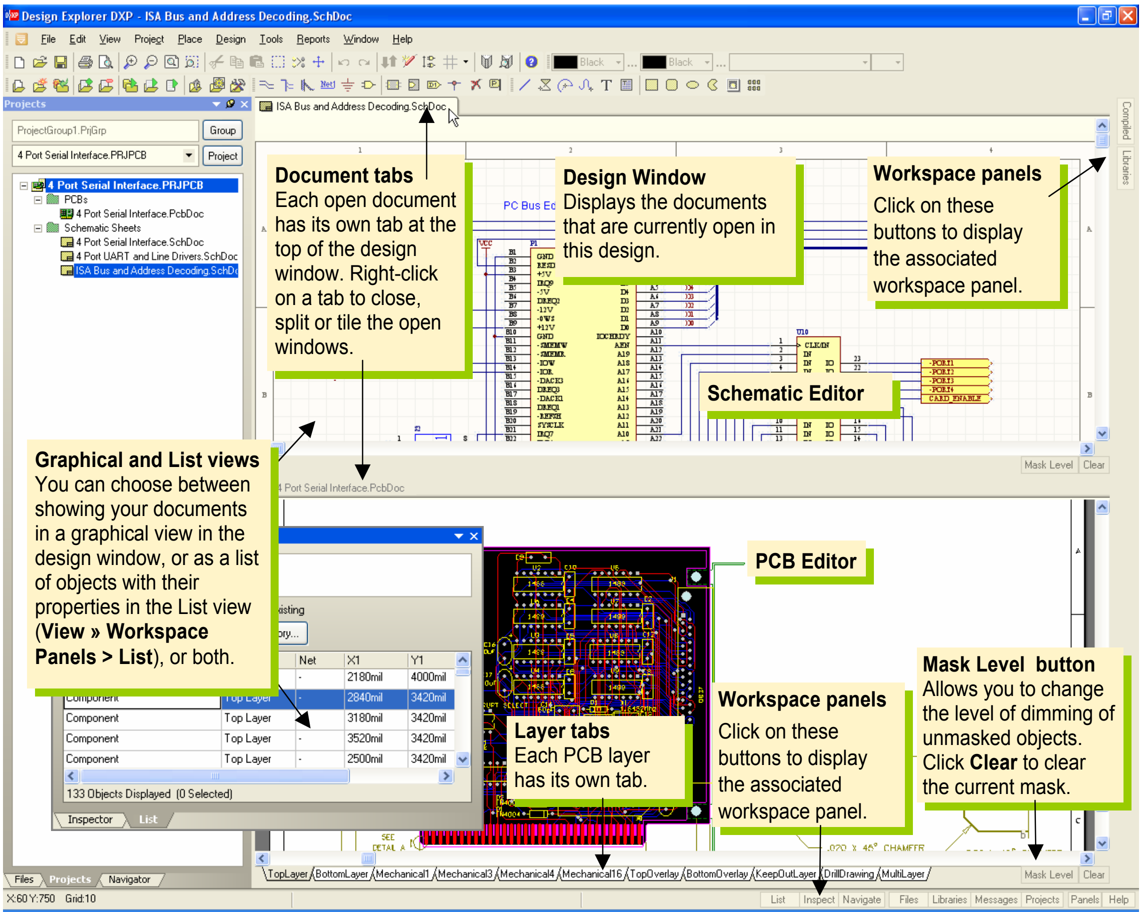
The Protel DXP Design Explorer

The Design Explorer is your interface to your designs and the design tools. To start Protel DXP and open the Design Explorer, select **Programs** » **Altium** » **Protel DXP** from the Windows Start menu. When you open Protel DXP, the most common initial tasks are displayed for easy selection.



As you create your design documents, you can easily switch between editors, for example, the Schematic Editor and the PCB Editor. The Design Explorer will change toolbars and menus according to the editor you are currently working in. The name of some workspace panels will initially be displayed down the right side of the workspace. Click on these names to pop out the panels, which then can be moved, docked or clipped to suit your work environment.

The following diagram shows the Design Explorer when several documents and editors are open at the same time and the windows have been tiled.



How the design documents are stored

Protel DXP stores all the design documents and output files on your hard disk as individual files. You can use the Windows Explorer to search for them. Project files can be created that contain links to the design documents and are necessary for design verification and synchronization.

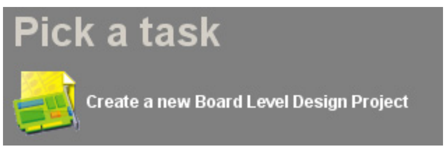
Creating a new project

A project in Protel DXP consists of links to all documents and setups related to a design. A project file, e.g. `xxx.PrjPCB`, is an ASCII text file that lists which documents are in the project and related output setups, e.g. for printing and CAM. Documents that are not associated with a project are called ‘free documents’. Links to schematic sheets and a target output, e.g. PCB, FPGA, embedded (VHDL) or library package, are added to a project. Once the project is compiled, design verification, synchronization and comparison can take place. Any changes to the original schematics or PCB, for example, are updated in the project when compiled.

The process of creating a new project is the same for all project types. We will use the PCB project as an example. We will create the project file first and then create the blank schematic sheet to add the new empty project. Later in this tutorial we will create a blank PCB and add it to the project as well.

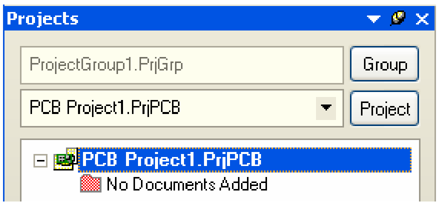
To start the tutorial, create a new PCB project:

1. Click on **Create a new Board Level Design Project** in the **Pick a Task** section of the design window.



Alternatively, you could click on **Blank Project (PCB)** in the **New** section of the **Files** panel. If this panel is not displayed, select **File » New**, or click on the **Files** tab at the bottom of the Design Manager panel.

2. The **Projects** panel displays. The new project file, `PCB Project1.PrjPCB`, is listed here with no documents added.



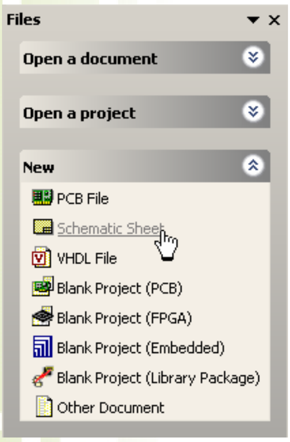
3. Rename the new project file (with a `.PrjPCB` extension) by selecting **File » Save Project As**. Navigate to a location where you would like to store the project on your hard disk, type the name `Multivibrator.PrjPCB` in the File Name field and click on **Save**.

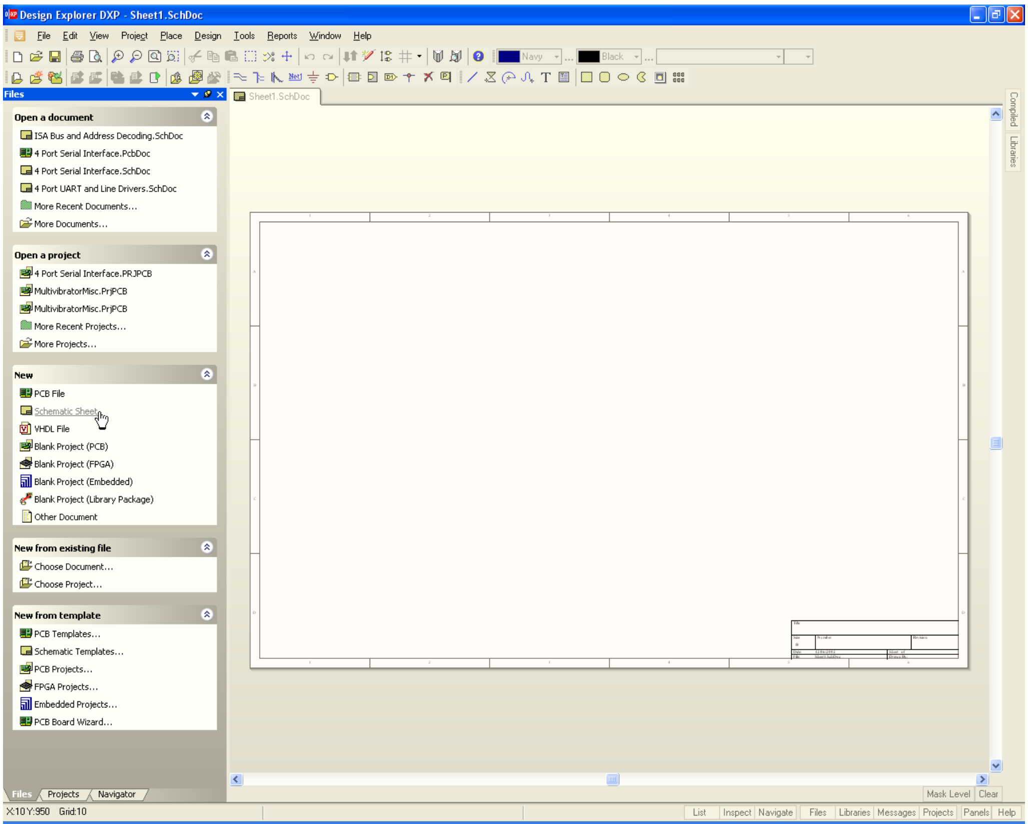
Next we will create a schematic to add to the empty project file. The schematic will be for an astable multivibrator circuit.

Creating a new schematic sheet

Create a new schematic sheet by completing the following steps:

1. Select **File » New** and click on **Schematic Sheet** in the **New** section of the **Files** panel. A blank schematic sheet named `Sheet1.SchDoc` displays in the design window and the schematic document is automatically added (linked) to the project. The schematic sheet is now listed under **Schematic Sheets** beneath the project name in the **Projects** tab.





2. Rename the new schematic file (with a `.SchDoc` extension) by selecting **File » Save As**. Navigate to a location where you would like to store the schematic on your hard disk, type the name `Multivibrator.SchDoc` in the File Name field and click on **Save**.

When the blank schematic sheet opens you will notice that the workspace changes. The main toolbar includes a range of new buttons, new toolbars are visible and the menu bar includes new items. You are now in the Schematic Editor.

You can customize many aspects of the workspace. For example, you can reposition the floating toolbars. Simply click-and-hold the title area of the toolbar and move the mouse to relocate the toolbar. To dock the toolbar, move it to left, right, top or bottom edge of the main window area.

Now we can add our blank schematic to the project before proceeding with the design capture.

Adding schematic sheets to a project

If the schematic sheets you want to add to a project file have been opened as Free Documents, right-click on the schematic document in the Free Documents section of the **Projects** panel and select **Add to Project**. The schematic sheet is now listed under **Schematic Sheets** beneath the project name in the **Projects** tab and is linked to the project file.

Setting the schematic options

The first thing to do before you start drawing your circuit is to set up the appropriate document options. Complete the following steps:

1. From the menus, choose **Design » Options** and the *Document Options* dialog will open. For this tutorial, the only change we need to make here is to set the sheet size to standard A4 format. In the **Sheet Options** tab, find the **Standard Styles** field. Click the arrow next to the entry to see a list of sheet styles.
2. Use the scroll bar to scroll up to the **A4** style and click to select it.
3. Click the **OK** button to close the dialog and update the sheet size.
4. To make the document fill the viewing area again, select **View » Fit Document**.

Protel DXP has a multilevel Undo, allowing you to undo any number of previous actions. The maximum number of Undo steps is user-configurable and limited only by the available memory on your computer.

In Protel DXP, you can activate any menu by simply pressing the menu hotkey (the underlined letter in the menu name). Any subsequent menu items will also have hot keys that you can use to activate the item. For example, the shortcut for selecting the **View » Fit Document** menu item is to press the **V** key followed by the **D** key. Many submenus, such as the **Edit » DeSelect** menu, can be called directly. To activate the **Edit » DeSelect » All** menu item, you need only press the **X** key (to call up the DeSelect menu directly) followed by the **A** key.

Next we will set the general schematic preferences.

1. Select **Tools » Preferences** [shortcut **T**, **P**] from the menus to open the schematic *Preferences* dialog. This dialog allows you to set global preferences that will apply to all schematic sheets you work on.
2. Click on the **Default Primitives** tab to make it active and enable the **Permanent** check box. Click the **OK** button to close the dialog.
3. Before you start capturing your schematic, save this schematic sheet, so select **File » Save** [shortcut **F**, **S**].

You can save any schematic sheet as a document template (.dot) allowing you to include special information such as a custom company title block and logo.

Drawing the schematic

You are now ready to begin capturing (drawing) the schematic. For this tutorial, we will use the circuit shown below (Figure 1). This circuit uses two 2N3904 transistors configured as a self-running astable multivibrator.

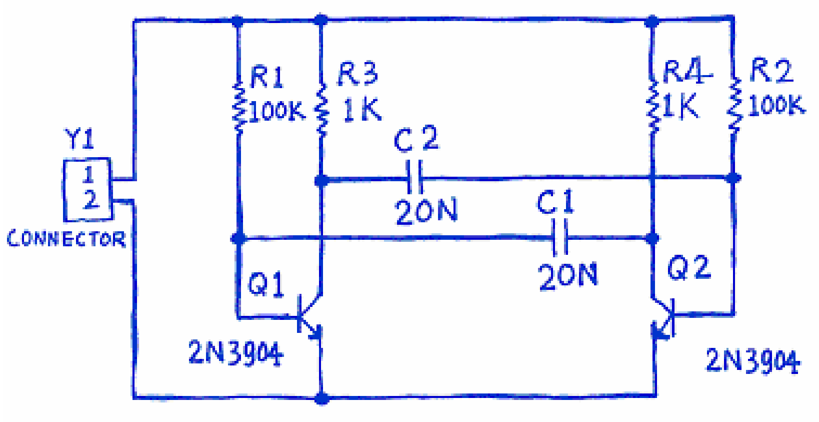
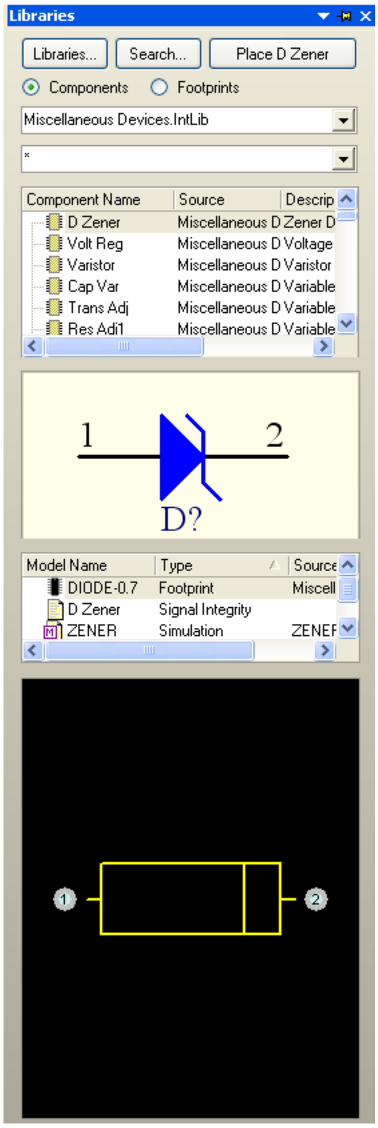


Figure 1. An astable multivibrator

Locating the component and loading the libraries



To manage the thousands of schematic symbols included with Protel DXP, the Schematic Editor provides powerful library search features. Although the components we require are in the default installed libraries, it is useful to know how to search through the libraries to find components. Work through the following steps to locate and add the libraries you will need for the tutorial circuit.

First we will search for the transistors, both of which are type 2N3904.

1. Click on the **Libraries** tab to display the **Libraries** workspace panel.
2. Press the **Search** button in the **Libraries** panel, or select **Tools » Find Component**. This will open the *Search Libraries* dialog.
3. Ensure that the Scope is set to **Libraries on Path** and that the **Path** field contains the correct path to your libraries. If you accepted the default directories during installation, the path should be `C:\Program Files\Altium\Library\`. Ensure that the **Include Subdirectories** box is not selected (not ticked).
4. We want to search for all references to 3904, so in the **Name** text field in the Search Criteria section, type `*3904*`. The * symbol is a wildcard used to take into account the different prefixes and suffixes used by different manufacturers.
5. Click the **Search** button to begin the search. The **Results** tab displays as the search takes place. If you have entered the parameters correctly, a library will be found and displayed in the *Search Libraries* dialog.
6. Click on the `Miscellaneous Devices.IntLib` library to select it. This library has symbols for all the available simulation-ready BJT transistors.
7. Click the **Install Library** button to make this library available to your schematic.
8. Close the *Search Libraries* dialog.

The added libraries will appear at the top of the Libraries panel. As you click on a library name in the upper list, the components in that library are listed below. The component filter in the panel can then be used to quickly locate a component within a library.

Placing the components on your schematic

The first components we will place on the schematic are the two transistors, Q1 and Q2. For the general layout of the circuit, refer to the schematic drawing shown in Figure 1.

1. Select **View » Fit Document** from the menus [shortcut **V**, **D**] to ensure your schematic sheet takes up the full window.

2. Make sure the **Libraries** panel is displayed by clicking on the **Libraries** tab.

3. Q1 and Q2 are BJT transistors, click on the `Miscellaneous Devices.IntLib` library to make it the active library.

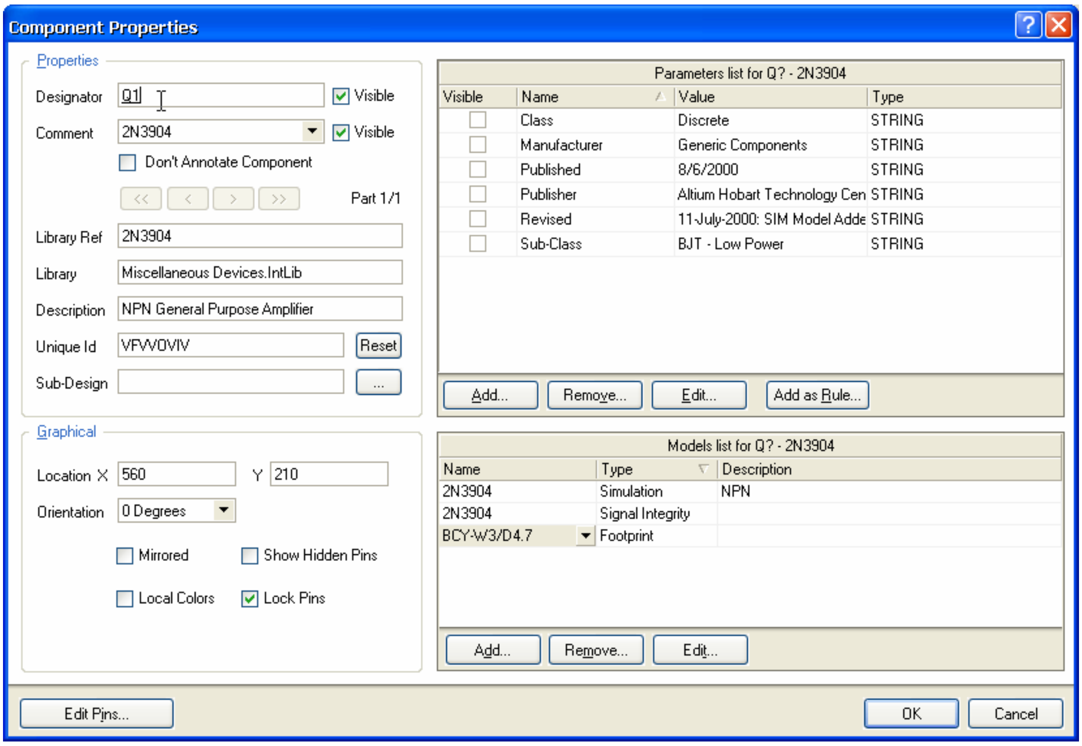
The link between the schematic component and the PCB component is the footprint. The footprint specified in the schematic is loaded from the PCB library when you load the netlist. Double-click on a schematic component to specify the footprint.

4. Use the filter to quickly locate the component you need. The default wildcard (*) will list all components found in the library. Set the filter by typing `*3904*` in the filter field below the Library name. A list of components which have the text "3904" as part of their Component Name field will be displayed.

5. Click on the **2N3904** entry in the list to select it, then click the **Place** button. Alternatively, just double-click on the component name.

   The cursor will change to a cross hair and you will have an outlined version of the transistor "floating" on your cursor. You are now in part placement mode. If you move the cursor around, the transistor outline will move with it.

6. Before placing the part on the schematic, first edit its properties. While the transistor is floating on the cursor, press the **TAB** key. This opens the *Component Properties* dialog for the component. We will now set up the dialog options to appear as below.



7. In the **Properties** section of the dialog, set the value for the first component designator by typing `Q1` in the **Designator** field.

8. Next we will check the footprint that will be used to represent the component in the PCB. For this tutorial, we have used integrated libraries which mean that the recommended models for

footprints and circuit simulation are already included. Make sure that model name `BCY-W3/D4.7` is included in the Models list. Leave all other fields at their default values.

You are now ready to place the part.

1. Move the cursor (with the transistor symbol attached) to position the transistor a little left of the middle of the sheet.

2. Once you are happy with the transistor's position, left-click or press **ENTER** to place the transistor onto the schematic.

3. Move the cursor and you will find that a copy of the transistor has been placed on the schematic sheet, but you are still in part placement mode with the part outline floating on the cursor. This feature of Protel DXP allows you to place multiple parts of the same type. So let's now place the second transistor. This transistor is the same as the previous one, so there is no need to edit its attributes before we place it. Protel DXP will automatically increment a component's designator when you place a series of parts. In this case, the next transistor we place will automatically be designated Q2.

4. If you refer to the schematic diagram (Figure 1) you will notice that Q2 is drawn as a mirror of Q1. To flip the orientation of the transistor that is floating on the cursor, press the **X** key. This flips the component horizontally.

5. Move the cursor to position the part to the right of Q1. To position the component more accurately, press the **PAGEUP** key twice to zoom in two steps. You should now be able to see the grid lines.

6. Once you have positioned the part, left-click or press **ENTER** to place Q2. Once again a copy of the transistor you are "holding" will be placed on the schematic, and the next transistor will be floating on the cursor ready to be placed.

7. Since we have now placed all the transistors, we will exit part placement mode by clicking the right mouse button or pressing the **ESC** key. The cursor will revert back to a standard arrow.

Use the following keys to manipulate the part floating on the cursor:

Y flips the part vertically

X flips the part horizontally

SPACEBAR rotates the part by 90°.

When you are in any editing or placement mode (a cross hair cursor is active), moving the cursor to the edge of the document window will automatically pan the document.

If you accidentally pan too far while you are wiring up your circuit, press the **V, F** key sequence (**View » Fit All Objects**) to redraw the schematic window, showing all placed objects. This can be done even when you are in the middle of placing an object.

Next we will place the four resistors.

1. In the **Libraries** panel, make sure the `Miscellaneous Devices.IntLib` library is active.

2. Set the filter by typing `res1` in the filter field below the Library name.

3. Click on **RES1** in the components list to select it, then click the **Place** button. You will now have a resistor symbol floating on the cursor.

4. Press the **TAB** key to edit the resistor's attributes. In the **Properties** section of the dialog, set the value for the first component designator by typing R1 in the **Designator** field.

5. Make sure that model name `AXIAL-0.3` is included in the Models list.

6. Set up a parameter field for the resistor that will display on the schematic and be used by DXP when running a circuit simulation later in this tutorial. The =Value parameter can be used for any general information about the component but discrete components use it when simulating. We can also set the Comment to read this value and this maps the Comment information to the PCB layout tool. Rather than enter the value twice (in the parameter =Value and then in the Comment field), DXP supports 'indirection' which will replace the contents of the Comments field with the parameter's string.

   Click **Add** in the Parameters list section to display the *Parameter Properties* dialog. Enter the name `Value` and a value of `100k`. Make sure **String** is selected as the parameter type and the value's **Visible** box is ticked. Click **OK**.

7. In the **Properties** section of the dialog, click on the Comment field and select the **=Value** string from the drop down list and turn **Visible** off. Click the **OK** button to return to placement mode.

8. Press the **SPACEBAR** to rotate the capacitor by 90° so it is in the correct orientation.

To edit the attributes of an object placed on the schematic, double-click the object to open its *Component Properties* dialog.

9. Position the resistor above the base of Q1 (refer to the schematic diagram in Figure 1) and left-click or press **ENTER** to place the part.

   Don't worry about making the resistor connect to the transistor just yet. We will wire up all the parts later.

10. Next place the other 100k resistor R2 above the base of Q2. The designator will automatically increment when you place the second resistor.

11. The remaining two resistors, R3 and R4, have a value of 1k, so press the **TAB** key to call up the *Component Properties* dialog and change the **Value** field to `1k` (press **Edit** when the Value name is selected in the **Parameters** list). Click **OK** to close the dialogs.

12. Position and place R3 and R4 as shown in the schematic diagram in Figure 1.

13. Once you have placed all the resistors, right-click or press **ESC** to exit part placement mode.

Now place the two capacitors.

1. The capacitor part is also in the `Miscellaneous Devices.IntLib` library, which should already be selected in the **Libraries** panel.

To reposition any object, simply place the cursor directly over the object, click-and-hold the left mouse button, drag the object to a new position and then release the mouse button.

2. Type `cap` in the component's filter field in the **Libraries** panel.

3. Click on **CAP** in the components list to select it, then click the **Place** button. You will now have a capacitor symbol floating on the cursor.

4. Press the **TAB** key to edit the capacitor's attributes. In the **Properties** section of the *Component Properties* dialog, set the **Designator** to `C1`, check the PCB footprint model `RAD-0.3` is added in the **Models** list.

5. Set up a parameter field that will display on the schematic. Click **Add** in the Parameters list section to display the *Parameter Properties* dialog. Enter the name `Value` and a value of `20n`. Make sure **String** is selected as the parameter type and the value's **Visible** box is ticked. Click **OK**.

6. In the **Properties** section of the dialog, click on the Comment field and select the **=Value** string from the drop down list and turn **Visible** off. Click the **OK** button to return to placement mode.

7. Position and place the two capacitors in the same way that you placed the previous parts.

8. Right-click or press **ESC** to exit placement mode.

The last component to be placed is the connector, also in the `Miscellaneous Connectors.IntLib` library.

1. The connector we want is a two-pin socket, so set the filter to `*2*`.

2. Select **HEADER2** from the parts list and click the **Place** button. Press **TAB** to edit the attributes and set **Designator** to Y1 and check the PCB footprint model is `HDR1X2`. No Value parameter is required as we will replace this component with a power source when simulating the circuit. Click **OK** to close the dialog.

3. Before placing the connector, press **X** to flip it horizontally so that it is in the correct orientation. Place the connector on the schematic.

4. Right-click or press **ESC** to exit part placement mode.

5. Save your schematic by selecting **File** » **Save** from the menus [shortcut **F**, **S**].

You have now placed all the components. Note that the components in Figure 2 are spaced so that there is plenty of room to wire to each component pin. This is important because you can not place a wire across the bottom of a pin to get to a pin beyond it. If you do, both pins will connect to the wire.

If you need to move a component, click-and-hold on the body of the component, then drag the mouse to reposition it.

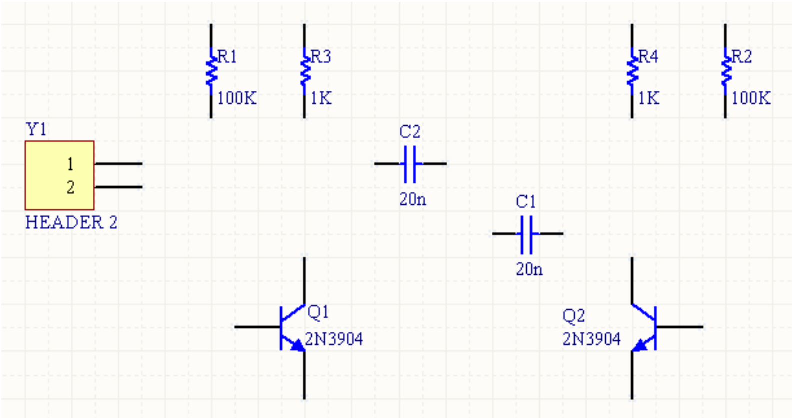


Figure 2. Schematic with all parts placed.

Wiring up the circuit

Wiring is the process of creating connectivity between the various components of your circuit. To wire up your schematic, refer to the diagram in Figure 1 and complete the following steps.

1. To make sure you have a good view of the schematic sheet, select **View » Fit All Objects** from the menus [shortcut **V, F**].
2. Firstly wire the resistor R1 to the base of transistor Q1 in the following manner. Select **Place » Wire** [shortcut **P**, **W**] from the menus or click on the **Wire** tool from the *Wiring Tools* toolbar to enter the wire placement mode. The cursor will change to a cross hair.
3. Position the cursor over the bottom end of R1. When you are in the right position, a red connection marker (large asterisk) will appear at the cursor location. This indicates that the cursor is over an electrical connection point on the component.
4. Left-click or press **ENTER** to anchor the first wire point. Move the cursor and you will see a wire extend from the cursor position back to the anchor point.
5. Position the cursor so that it is below R1 and level with the base of Q1. Left-click or press **ENTER** to anchor the wire at this point. The wire between the first and second anchor points will be placed.
6. Position the cursor over the base of Q1 until you see the cursor change to a red connection marker. Left-click or press **ENTER** to connect the wire to the base of Q1.
7. Right-click or press **ESC** to finish placing this particular wire. Note that the cursor remains a cross hair, indicating that you are ready to place another wire. To exit placement mode completely and go back to the arrow cursor, you would right-click or press **ESC** again – but don't do this just now.
8. We will now wire C1 to Q1 and R1. Position the cursor over the left connection point of C1 and left-click or press **ENTER** to start a new wire.
9. Move the cursor horizontally till it is directly over the wire connecting the base of Q1 to R1. A connection marker will appear.
10. Left-click or press **ENTER** to place the wire segment, then right-click or press **ESC** to indicate that you have finished placing the wire. Note how the two wires are automatically connected.
11. Wire up the rest of your circuit, as shown in Figure 3.

To graphically edit the shape of a wire, or any other graphical object once it has been placed, position the arrow cursor over it and click once.

Whenever a wire runs across the connection point of a component, or is terminated on another wire, Protel DXP will automatically create a junction.

When placing wires, keep in mind the following points:

- left-click or press ENTER to anchor the wire at the cursor position;

- press BACKSPACE to remove the last anchor point;

- after placing the last segment of a wire, right-click or press **ESC** to end the wire placement. The cursor will remain as a cross hair and you can begin placing another wire.

A wire that crosses the end of a pin will connect to that pin, even if you delete the junction. Check that your circuit looks like Figure 3 before proceeding.

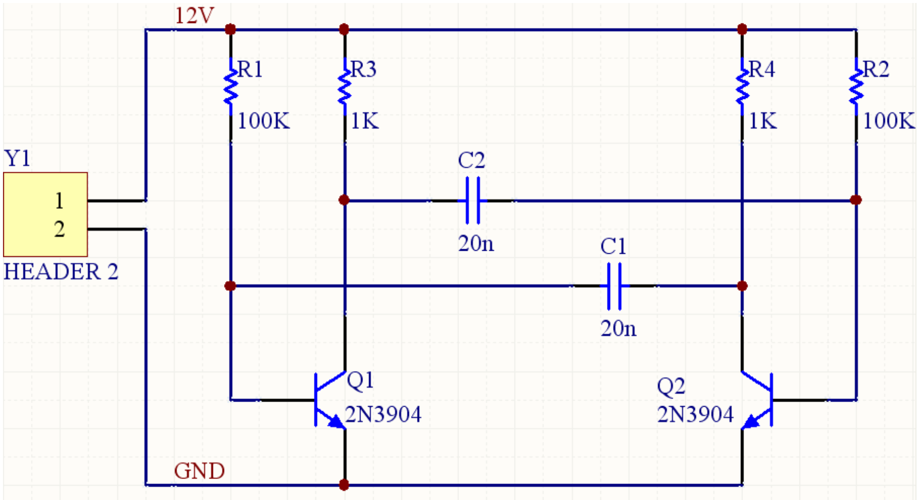


Figure 3. The fully wired schematic

12. When you have finished placing all the wires, right-click or press **ESC** to exit placement mode. The cursor will revert to an arrow.

Nets and net labels

Each set of component pins that you have connected to each other now form what is referred to as a *net*. For example, one net includes the base of Q1, one pin of R1 and one pin of C1.

To make it easy to identify important nets in the design, you can add *net labels*.

To place net labels on the two power nets:

1. Select **Place » Net Label** from the menus. A dotted box will appear floating on the cursor.
2. To edit the net label before it is placed, press the **TAB** key to display the *Net Label* dialog.
3. Type `12V` in the **Net** field, then click **OK** to close the dialog.
4. Place the net label so that the bottom left of the net label touches the upper most wire on the schematic. The cursor will change to a red cross when the net label touches the wire.
5. After placing the first net label you will still be in net label placement mode, so press the **TAB** key again to edit the second net label before placing it.
6. Type `GND` in the **Net** field, click **OK** to close the dialog and place the net label.
7. Select **File » Save** [shortcut **F**, **S**] to save your circuit.

Congratulations! You have just completed your first schematic capture using Protel DXP.

Before we turn the schematic into a circuit board, let's set up the project options.

Setting up Project Options

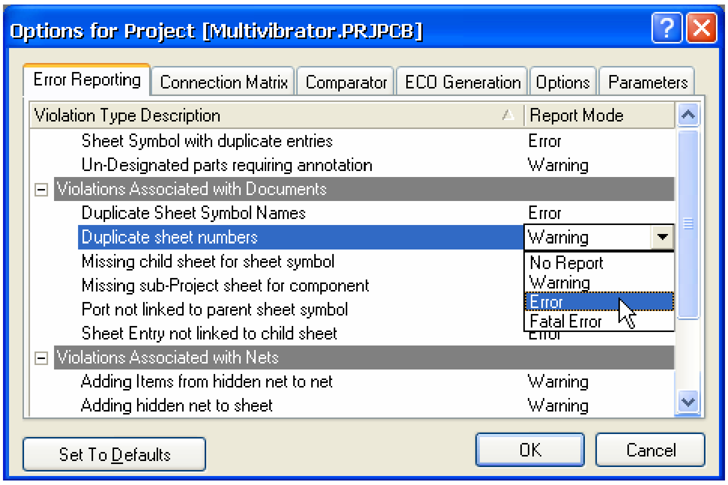
The project options include the error checking parameters, a connectivity matrix, the Comparator setup, ECO generation, output paths and netlist options and any project parameters you wish to specify. Protel DXP will use these setups when you compile the project.

When a project is compiled, comprehensive design and electrical rules are applied to verify the design. When all errors are resolved, the re-compiled schematic designs are loaded into the target document, e.g. a PCB document, by generated ECOs. The project Comparator allows you to find differences between source and target files and update (synchronize) in both directions.

All project-related operations, such as error checking, comparing documents and ECO generation, are set up in the *Options for Project* dialog (**Project » Project Options**).

All project outputs, such as netlist, simulator, documentation (printing), assembly and fabrication outputs and reports are set up in the *Outputs for Project* dialog (**Project » Output Jobs**). See *Setting up the Project Outputs* for more information.

1. Select **Project » Project Options**. The *Options for Project* dialog displays.



All project-related options are set up through this dialog.

Checking the electrical properties of your schematic

Schematic diagrams in Protel DXP are more than just simple drawings – they contain electrical connectivity information about the circuit. You can use this connectivity awareness to verify your design. When you compile a project, DXP checks for errors according to the rules set up in the Error Reporting and Connection Matrix tabs and any violations generated will display in the Messages panel.

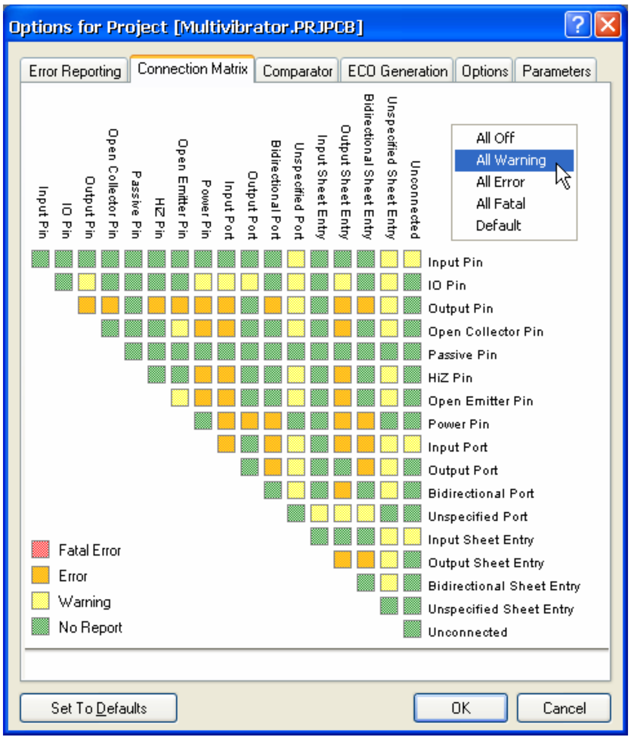
Setting up Error Reporting

The **Error Reporting** tab in the *Options for Project* dialog is used to set up design drafting checks. The Report Mode is the level of severity of a violation. If you wish to change a Report Mode, click on a Report Mode next to the violation you wish to change and choose the level of severity from the drop-down list. For this tutorial we will use the default settings.

Setting up the Connection Matrix

The Connection Matrix tab (*Options for Project* dialog) displays the severity of an error type that is produced when error reporting is run to check electrical connections within the design, i.e. connections between pins, ports and sheet entries. The matrix gives a graphical representation of different types of connection points on a schematic and whether they are allowable or not.

For example, look down the entries on the right side of the matrix diagram and find **Output Pin**. Read across this row of the matrix till you get to the **Open Collector Pin** column. The square where they intersect is orange indicating that an Output Pin connected to an Open Collector Pin on your schematic will generate an error condition when the project is compiled.



You can set each error type with a separate error level, e.g. from no report at all through to a fatal error.

To make changes to the Connection Matrix:

1. Click on the **Connection Matrix** tab in the *Options for Project* dialog.
2. Click on the box that is at the intersection of two types of connection, e.g. Output Sheet Entry and Open Collector Pin.
3. Click until the box changes to the color of the errors as listed in the legend, e.g. an orange box indicates that an error will be generated if such a connection is found.

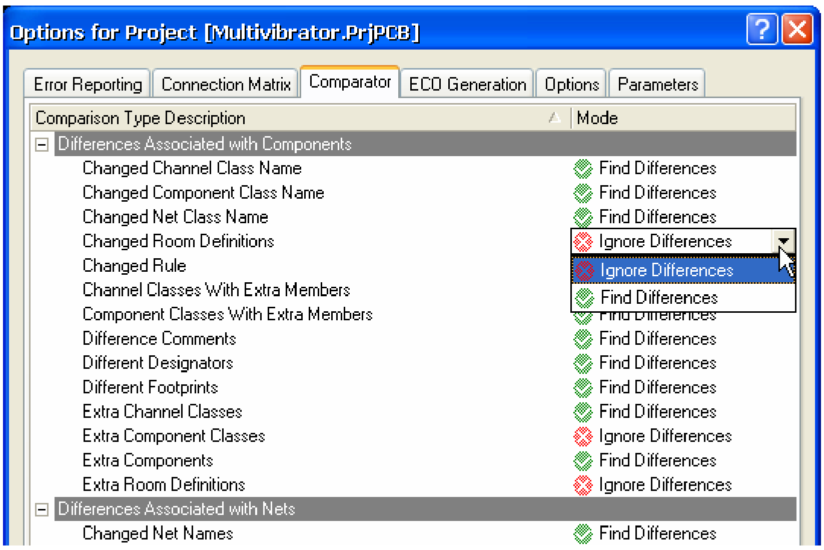
Our circuit contains only Passive Pins (on resistors, capacitors and the connector) and Input Pins (on the transistors. Let's check to see if the connection matrix will detect unconnected passive pins.

1. Look down the row labels to find **Passive Pin**. Look across the column labels to find **Unconnected**. The square where these entries intersect indicates the error condition when a Passive Pin is found to be Unconnected in the schematic. The default is a green square, which indicates that no report will be generated.
2. Click on this intersection box until it turns yellow so that a warning will be generated for unconnected passive pins when we compile the project. We will purposely create an instance of this error to check it later in this tutorial.

Setting up the Comparator

The **Comparator** tab in the *Options for Project* dialog sets which differences between files will be reported or ignored when a project is compiled. For this tutorial, we do not need to show differences between some features that refer to hierarchical schematic designs only, such as rooms. Make sure you do not accidentally ignore components when you meant to ignore component classes!

1. Click on the **Comparator** tab and find **Changed Room Definitions**, **Extra Room Definitions** and **Extra Component Classes** in the Difference Associated with Components section.
2. Select **Ignore Differences** from the drop-down list in the Mode column to the right of the these options.



Now we are ready to compile the project and check for any errors.

Compiling the project

Compiling a project checks for drafting and electrical rules errors in the design documents and puts you into a debugging environment. We have already set up the rules in the Error Checking and Connection Matrix tabs of the *Options for Project* dialog.

1. To compile our Multivibrator project, select **Project » Compile PCB Project**.
2. When the project is compiled, any errors generated will display in the Messages panel at the bottom of the design window. The compiled documents will be listed in the **Compiled** panel, together with a flattened hierarchy, components and nets listed and a connection model that can be browsed.

If your circuit is drawn correctly, the Messages panel should be blank. If the report gives errors, check your circuit and ensure all wiring and connections are correct.

We will now deliberately introduce an error into our circuit and recompile the project:

1. Click on the `Multivibrator.SchDoc` tab at the top of the design window to make the schematic sheet the active document.
2. Click in the middle of the wire that connects C1 to the base wire of Q1. Small, square editing handles will appear at each end of the wire and the selection color will display as a dotted line along the wire to indicate that it is selected. Press the **DELETE** key to delete the wire.
3. Recompile the project (**Project » Compile PCB Project**) to check that any errors are found.

   The Messages panel will open giving a warning message that you have an unconnected input pin in your circuit. A floating input pin error will also be generated as there is a special option to check for floating input pins in the **Error Reporting** tab of the *Project Options* dialog.
4. Click on an error in the Messages panel and the *Compile Error* window will display with details of the violation. From this window, you can click on an error and jump to the violating object in a schematic to check or correct the error.

If you wish to clear messages from the Messages panel, right-click in the window and select **Clear All**.

Before we finish this section of the tutorial, let's fix the error in our schematic.

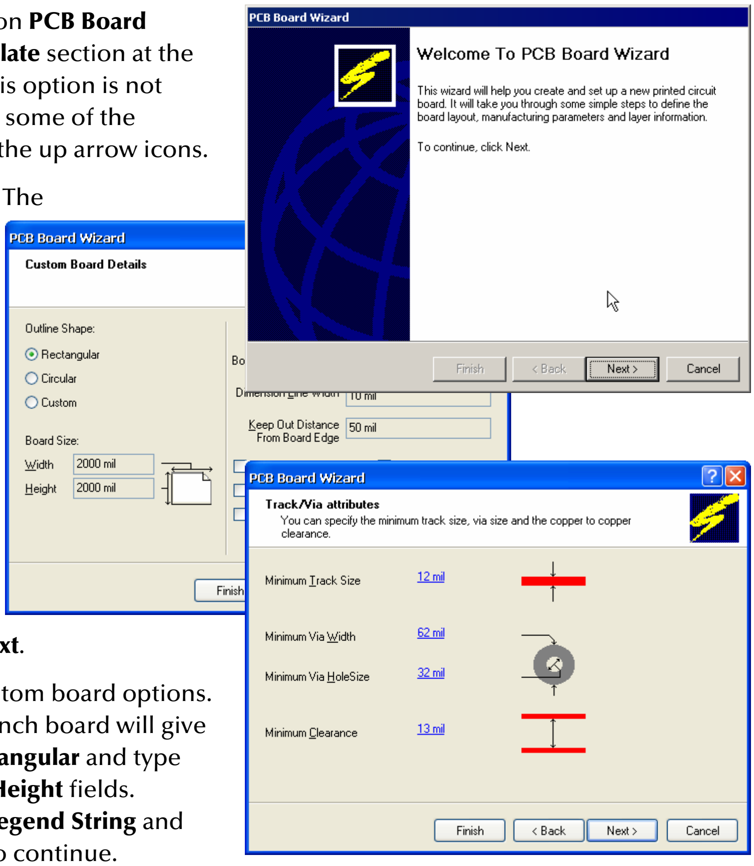
1. Click on the tab of the schematic sheet to make it active.
2. Select **Edit » Undo** from the menus [shortcut **E**, **U**]. The wire you deleted previously should now be restored.
3. To check that the undo was successful, recompile the project (**Project » Compile PCB Project**) to check that no errors are found. The Messages panel should show no errors.
4. Select **View » Fit All Objects** [shortcut **V, F**] from the menus to restore your schematic view and save your error-free schematic.

Creating a new PCB document

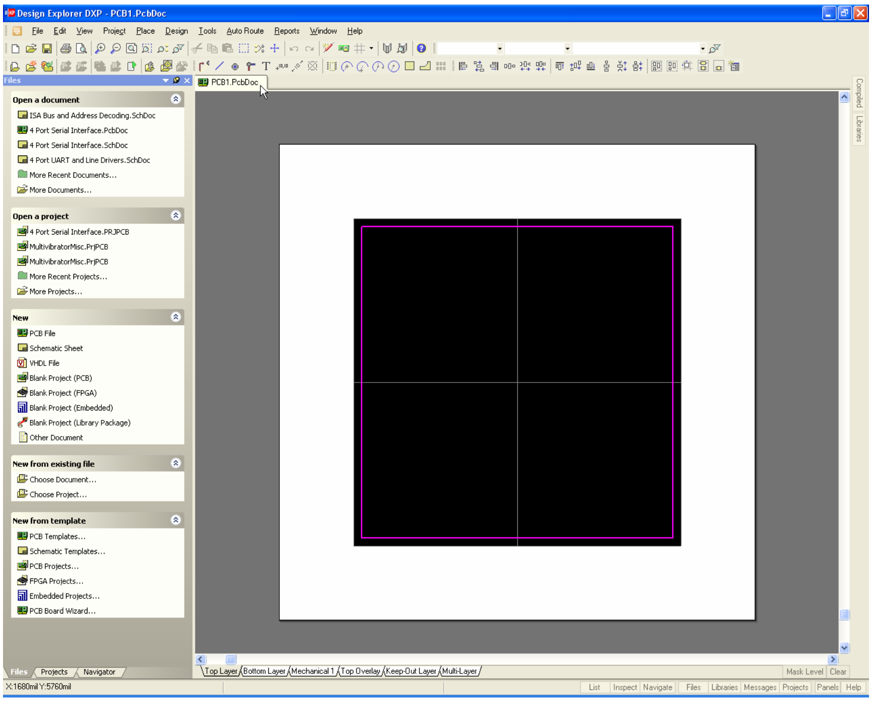
Before you transfer the design from the Schematic editor to the PCB editor, you need to create the blank PCB with at least a board outline. The easiest way to create a new PCB design in Protel DXP is to use the PCB Wizard, which allows you to choose from industry-standard board outlines as well as create your own custom board sizes. At any stage you can use the **Back** button to check or modify previous pages in the wizard.

To create a new PCB using the PCB Wizard, complete the following steps:

1. Create a new PCB by clicking on **PCB Board Wizard** in the **New from Template** section at the bottom of the **Files** panel. If this option is not displayed on the screen, close some of the sections above by clicking on the up arrow icons.
2. The PCB Board Wizard opens. The first screen you see is the introduction page. Click the **Next** button to continue.
3. Set the measure units to Imperial, i.e. 1000 mils = one inch.
4. The third page of the wizard allows you to select the board outline you wish to use. For this tutorial we will enter our own board size. Select **Custom** from the list of board outlines and click **Next**.
5. In the next page you enter custom board options. For the tutorial circuit, a 2 x 2 inch board will give us plenty of room. Select **Rectangular** and type `2000` in both the **Width** and **Height** fields. Deselect **Title Block & Scale**, **Legend String** and **Dimension Lines**. Click **Next** to continue.



6. This page allows you to select the number of layers in the board. We will need two signal layers and no power planes. Click **Next** to continue.
7. Choose the via styles used in the design by selecting **Thru-hole vias only** and click **Next**.
8. The next page allows you to set the component/track technology (routing) options. Select the **Thru-hole components** option and set the number of tracks between adjacent pads to **One Track**. Click **Next** to continue.
9. The next page allows you to set up some of the design rules that apply to your board. Leave the options on this screen set to their defaults. Click the **Next** button to continue.
10. The final page allows you to save your custom board as a template, allowing you to create new boards based on the parameters you have just entered. We do not want to save our tutorial board as a template, confirm that this option is unchecked and click **Finish** to close the Wizard.
11. The PCB Wizard has now collected all the information it needs to create your new board. The PCB Editor will now display a new PCB file named `PCB1.PcbDoc`.
12. The PCB document displays with a default sized white sheet and a blank board shape (black area with grid). To turn off the sheet, select **Design » Options** and deselect **Design Sheet** in the *Board Options* dialog.



For more tutorials, press **F1** to access the Protel DXP Help and Online Documentation system and click on the Articles & Tutorials link.

You can add your own border, grid reference and title block from other PCB templates supplied with Protel DXP. For more information about using board shapes, sheets and templates, see the *Board Shapes and Sheets* tutorial.

13. Now the sheet has been turned off, display the board shape only by selecting **View » Fit Board** [shortcut **V**, **F**].

14. The PCB document is automatically added (linked) to the project and is listed under **PCBs** beneath the project name in the **Projects** tab.

15. Rename the new PCB file (with a `.PcbDoc` extension) by selecting **File » Save As**. Navigate to a location where you would like to store the PCB on your hard disk, type the name `Multivibrator.PcbDoc` in the File Name field and click on **Save**.

Adding a new PCB to a project

If the PCB you want to add to a project file has been opened as a Free Document, right-click on the PCB document in the Free Documents section of the **Projects** panel and select **Add to Project**. The PCB is now listed under **PCBs** beneath the project in the **Projects** tab and is linked to the project file.

Transferring the design

Before transferring the schematic information to the new blank PCB, make sure all the related libraries for both schematic and PCB are available. Since only the default installed integrated libraries are used in this tutorial, the footprints will already be included. Once the project has been compiled and any errors in the schematic fixed, use the **Update PCB** command to generate ECOs that will transfer the schematic information to the target PCB.

Updating the PCB

To send the schematic information to the target PCB in your project:

You can create a report of ECOs to print out by clicking on the **Report Changes** button.

1. Select **Design » Update PCB** (`Multivibrator.PcbDoc`). The project compiles and the *Engineering Change Order* dialog displays.

2. Click on **Validate Changes**. If all changes are validated, the checks appear in the Status list. If the changes are not validated, close the dialog, check the **Messages** panel and clear any errors.

3. Click on **Execute Changes** to send the changes to the PCB. When completed, the Status changes to Done.

4. Click **Close** and the target PCB opens with components positioned ready for placing on the board. Use the shortcut keys **V, D** (View Document) if you cannot see the components in your current view.

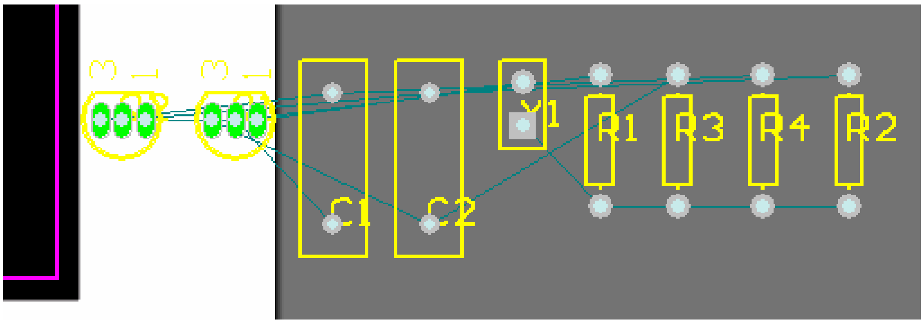


Figure 4. The components next to the board, ready for positioning.

Designing the PCB

Now we can start placing the components on the PCB and routing the board.

Setting up the PCB workspace

Before we start positioning the components on the board, we need to set up the PCB workspace, such as the grids, layers and design rules.

Grids

We need to ensure that our placement grid is set correctly before we start positioning the components. All the objects placed in the PCB workspace are aligned on a grid called the *snap grid*. This grid needs to be set to suit the routing technology that you intend to use.

The PCB Editor supports imperial and metric units. Select **View » Toggle Units** to switch.

Our tutorial circuit uses standard imperial components that have a minimum pin pitch of 100mil. We will set the snap grid to an even fraction of this, say 50 or 25mil, so that all component pins will fall on a grid point when placed. Also, the track width and clearance for our board are 12mil and 13mil respectively (the default values used by the PCB Board Wizard), allowing a minimum of 25mil between parallel track centers. The most suitable snap grid setting would, therefore, be 25mil.

To set the snap grid, complete the following steps:

1. Select **Design » Options** from the menus [shortcut **D**, **O**] to open the *Board Options* dialog.

2. In the **Grids** tab, set the value of the **Snap X, Snap Y**, **Component X** and **Component Y** fields of the dialog to `25mil`. Note that this dialog is also used to define the electrical grid. The electrical grid operates when you place an electrical object, it overrides the snap grid and snaps electrical objects together. Click **OK** to close the dialog.

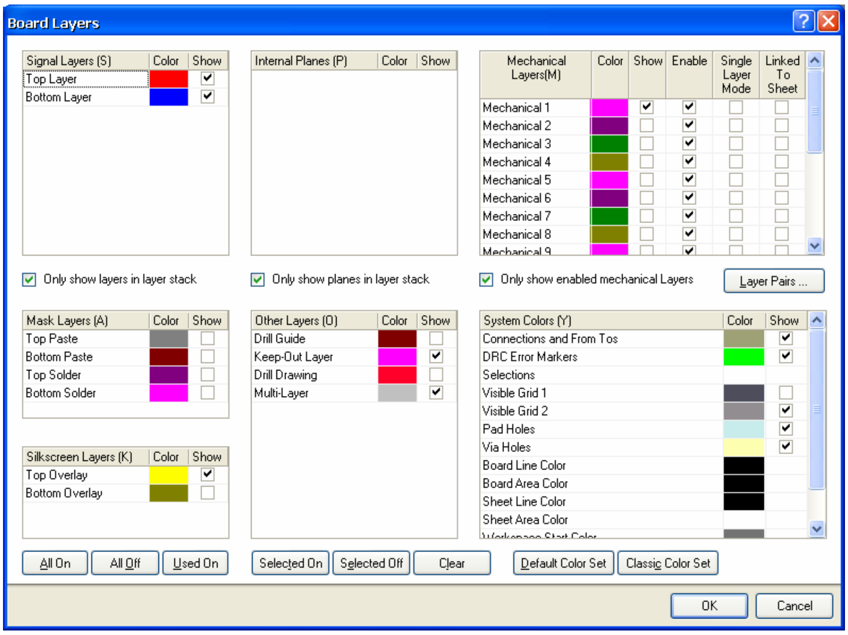
Let's set some other options that will make positioning components easier.

1. Select **Tools » Preferences** from the menus [shortcut **T**, **P**] to open the *System Preferences* dialog. In the Editing Options section of the **Options** tab, make sure the **Snap to Center** option is checked. This ensures that when you "grab" a component to position it, the cursor is set to the component's reference point.

2. Click the **Display** tab in the *System Preferences* dialog to make it active. In the **Show** section of this tab, uncheck the **Show Pad Nets, Show Pad Numbers** and **Via Nets** options. In the **Draft Thresholds** section of this dialog, set the **Strings** field to 4 pixels and then close the dialog.

Defining the layer stack and other non-electrical layers

If you look at the bottom of the PCB workspace, you will notice a series of layer tabs. The PCB Editor is a multi-layered environment and most of the editing actions you perform will be on a particular layer. Use the *Board Layers* dialog (**Design** » **Board Layers**) to display, add, remove and rename and set the colors of the layers.



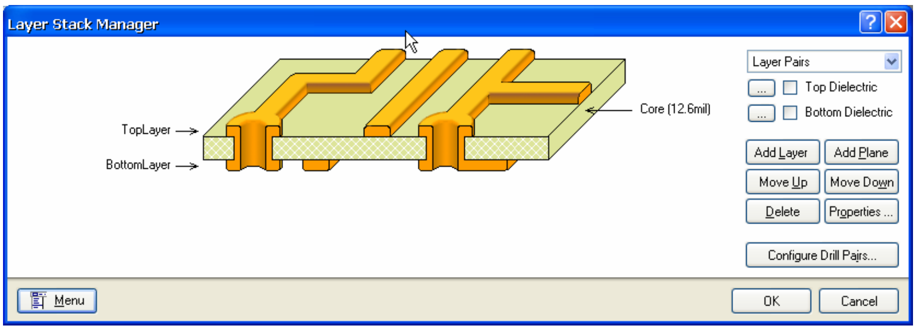
There are three types of layers in the PCB Editor:

- **Electrical layers** – these include the 32 signal layers and 16 plane layers. Electrical layers are added to and removed from the design in the Layer Stack Manager, select **Design** » **Layer Stack Manager** to display this dialog.
- **Mechanical layers** – there are 16 general purpose mechanical layers for defining the board outline, placing dimensions on, including fabrication details on, or any other mechanical details the design requires. These layers can be selectively included in print and Gerber output generation. You can add, remove and name mechanical layers in the *Board Layers* dialog.
- **Special layers** – these include the top and bottom silkscreen layers, the solder and paste mask layers, drill layers, the Keep-Out layer (used to define the electrical boundaries), the multilayer (used for multilayer pads and vias), the connection layer, DRC error layer, grid layers and hole layers. The display of these special layers is controlled in the *Board Layers* dialog.

Layer Stack Manager

The tutorial is a simple design and can be routed as a single-sided or double-sided board. If the design was more complex, you would add more layers in the Layer Stack Manager.

1. Select **Design** » **Layer Stack Manager** to display the *Layer Stack Manager* dialog.



2. New layers and planes are added below the currently selected layer. Layer properties, such as copper thickness and dielectric properties are used for signal integrity analysis. Click **OK** to close the dialog.

The new board has opened with many more layers enabled than you will use, so let's turn off all the unnecessary layers.

To turn off layers, complete the following steps:

1. Press the **L** shortcut key to display the *Board Layers* dialog.
2. Right-click and select **Used On** to disable all layers except those that have something on them.
3. Make sure the four **Mask** layers and the **Drill Drawing** layer will not display by clicking off the tick in the **Show** button next to their layer names. Click **OK** to close the dialog.

Setting up new design rules

The Protel DXP's PCB editor is a rules-driven environment. This means that as you work in the PCB editor and perform actions that change the design, such as placing tracks, moving components, or autorouting the board, the PCB editor constantly monitors each action and checks to see if the design still complies with the design rules. If it does not, then the error is immediately highlighted as a violation.

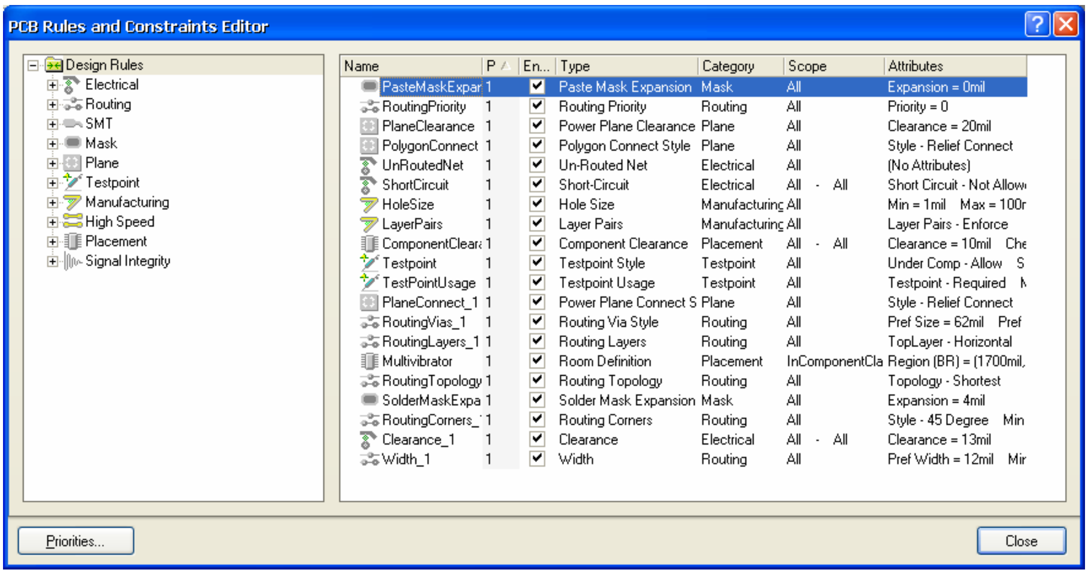
Setting up the design rules before you start working on the board allows you to remain focused on the task of designing, confident in the knowledge that any design errors will immediately be flagged for your attention.

The design rules fall into 10 categories and are further divided into design rule types. The design rules cover electrical, routing, manufacturing, placement and signal integrity requirements.

We will now set up new design rules to specify the width that the power nets must be routed.

To set up these rules, complete the following steps:

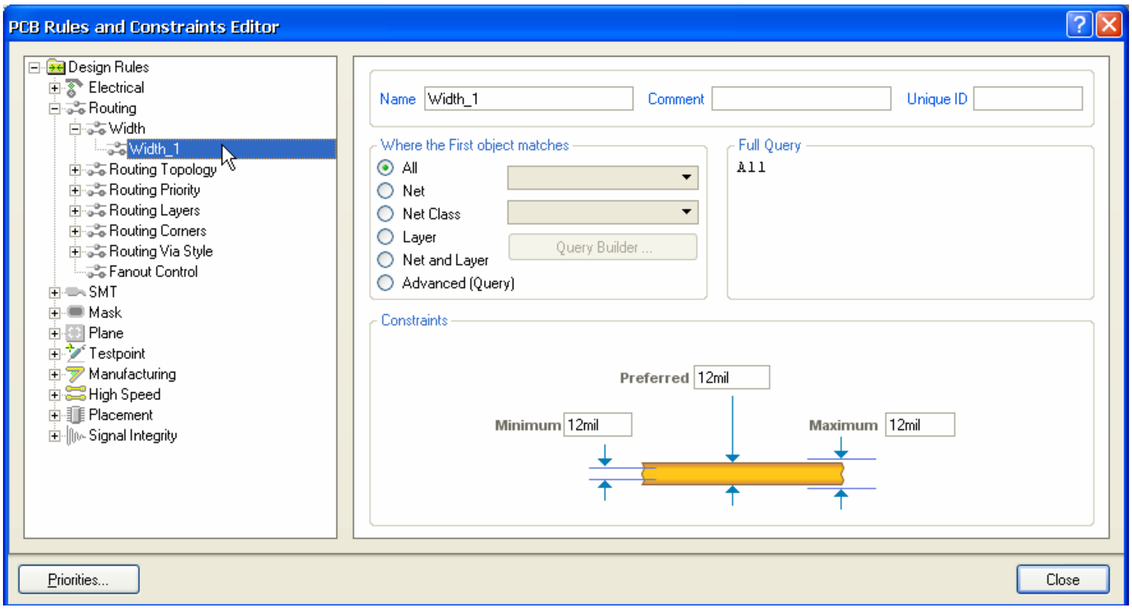
1. With the PCB as the active document, select **Design » Rules** from the menus.
2. The *PCB Rules and Constraints Editor* dialog will appear. Each category of rules is displayed in the Design Rules panel (left hand side) of the dialog. Double-click on the **Routing** category to expand the category and see the related routing rules. Then double-click on **Width** to display the width rules available.



3. Click once on each rule in the Design Rules panel to select it. As you click on each rule, the right hand side of the dialog displays the rule's scope (what you want this rule to target) in the top

section and the rule's constraints in the bottom section. These rules are either defaults, or have been set up by the Board Wizard when the new PCB document was created.

4. Click on the **Width_1** rule to display its constraints and scope. This rule applies to the entire board.



One of the powerful features of Protel DXP's design rule system is that multiple rules of the same type can be defined, each targeting different objects. The exact set of objects that each rule targets is defined by that rule's *scope*. The rule system uses a pre-defined hierarchy to work out which rule to apply to each object.

For example, you could have a width constraint rule for the whole board (meaning all tracks must be this width), a second width constraint rule for the ground net (this rule overrides the previous rule), and a third width constraint rule for a particular connection on the ground net (which overrides both of the previous rules). Rules are displayed in their order of priority.

Currently there is one width constraint rule for your design, which applies to the whole board. We will now add a new width constraint rule for the 12V and GND nets. To add new width constraint rules, complete the following steps:

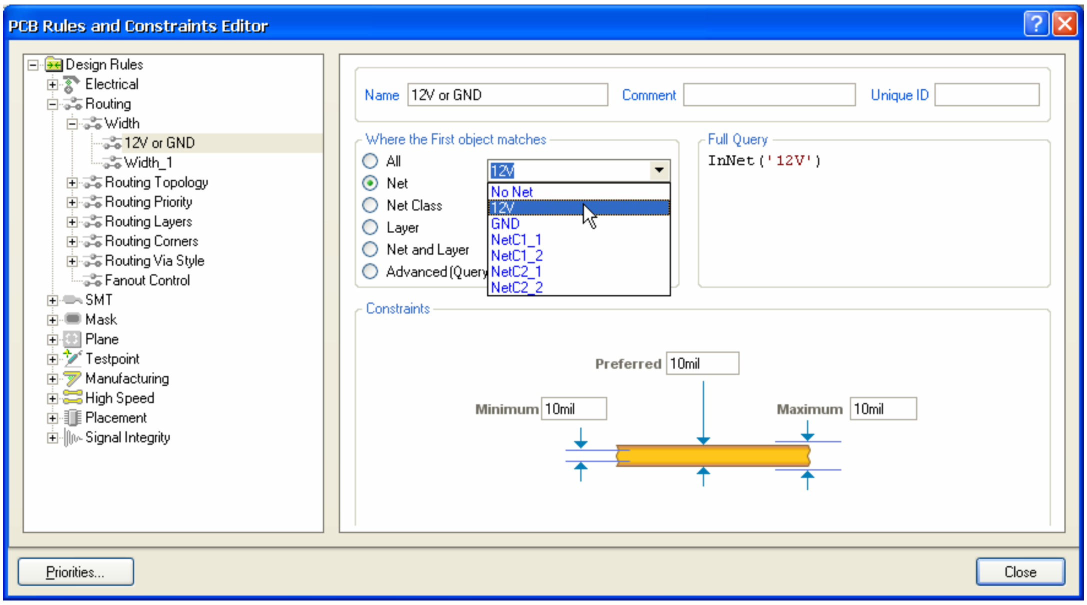
1. With the **Width** category selected in the **Design Rules** panel, right-click and select **New Rule** to add a new width constraint rule set up to target the 12V net only.

   A new rule named `Width_2` appears. Click on the new rule in the **Design Rules** panel to modify the scope and constraints.

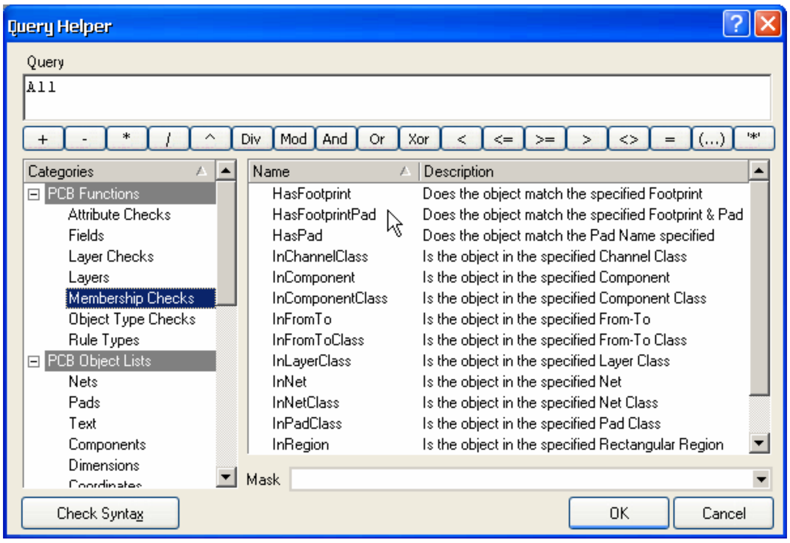
2. Type `12V or GND` in the Name field. The name will refresh in the **Design Rules** panel when you click back in the Design Rules panel when you have finished setting the rule.

3. Next we set the rule's scope using the Query Builder but you can always type in the scope directly if you know the correct syntax.

   Click on **Net** in the **Where the First object matches** section. The query `InNet( )` appears in the **Query Kind** section. Click on the dropdown list next the **All** button and select **12V** from the list of available nets. The query updates to read `InNet ('12V')`.



4. Next we will use the Query Builder to expand the scope to include the GND net as well. Click on **Advanced (Query)** and then click on **Query Builder**. The *Query Helper* dialog displays.



5. Click in the **Query** section to the right of the query `InNet('12V')` and click on the **Or** button. The query now reads `InNet('12V') or` and this sets the scope to apply the rule to both nets.

6. Click on **Membership Checks** in the PCB Functions category and double-click to select the **InNet** query in the Name section.

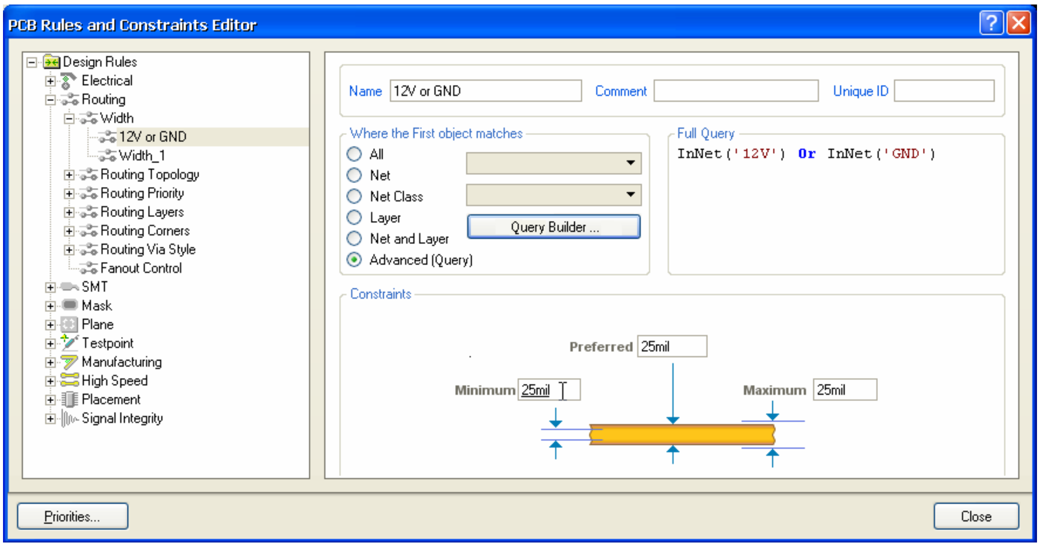
7. Click once between the parenthesis of the new `InNet( )` query in the Query section to add in the net name for the GND net. Click on **Nets** in the PCB Objects List category and double-click to select **GND** from the list of nets available. Check that the query `InNet('12V') or InNet('GND')` is created.

8. Click on **Check Syntax** to confirm the query is correct and click **OK** to close the resulting message. Fix the query syntax if an error message displays.

9. Click **OK** to close the *Query Helper* dialog. The scope in the **Full Query** section has now been updated with the new query.

10. In the bottom section of the *PCB Rules and Constraints Editor* dialog, change the **Minimum**, **Preferred** and **Maximum** width fields to `25mil` by clicking on the old constraints text (10mil) and typing in the new values. Note that you must set the Maximum width field first before you can

change the Minimum value. The new rule is now set up and will save when you choose another rule in the Design Rules panel or close the dialog.



11. Finally, double-click to edit the original Board scope width rule named `Width_1` and confirm that the Minimum, Maximum and Preferred width fields are all set to `12mil`. Click **OK** to close the *PCB Rules and Constraints Editor* dialog.

When you route the board manually or using the autorouter, all tracks will be 12mils wide, except the GND and 12V tracks which will be 25mils.

Positioning the components on the PCB

Now we can start to place the components in their right positions.

1. Press the **V, D** shortcut keys to zoom in on the board and components. The footprints have been placed in a room named 'Multivibrator'. Click on the room and press **Delete** if not required.

2. To place the connector Y1, position the cursor over the middle of the outline of the connector, and click-and-hold the left mouse button. The cursor will change to a cross hair and jump to the reference point for the part.

3. While continuing to hold down the mouse button, move the mouse to drag the component.

4. While dragging the connector, press the **SPACEBAR** to rotate it by 90°, and position it toward the left-hand side of the board (ensuring that the whole of the component stays within the board boundary), as shown in Figure 5.

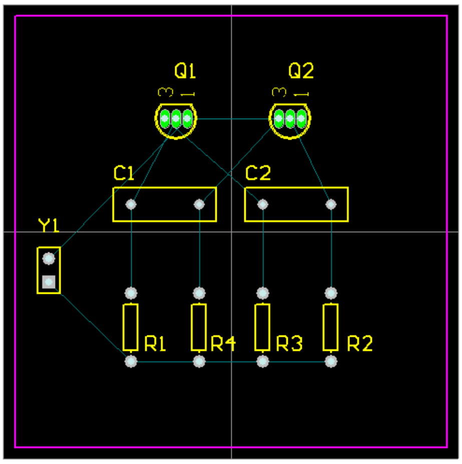


Figure 5. Components placed on the PCB

5. When the component is in position, release the mouse button to drop it into place. Note how the connection lines drag with the component.

6. Reposition the remaining components, using Figure 5 as a guide. Use the **SPACEBAR** key as necessary to rotate components as you drag them, so that the connection lines are as shown in Figure 5. Don't forget to re-optimize the connection lines as you position each component.

The connection lines are automatically re-optimized as you move a component. In this way you can use the connection lines as a guide to the optimum position and orientation of the component as you place it.

Component text can be repositioned in a similar fashion – click-and-hold to drag the text, press the **SPACEBAR** to rotate it. Before repositioning the text we will use Protel DXP's powerful global editing in the next part of the tutorial to hide the component comments (values), as these will not be required on the final board.

Protel DXP also includes powerful interactive placement tools. Let's use these to ensure that the four resistors are correctly aligned and spaced.

1. Holding the **SHIFT** key, left-click on each of the four resistors to select them. A selection box will display around each of the selected components in the color set for the system color called Selections. To change this selection color, select **Design » Board Layers**.

2. Click on the **Align Tops of Selected Components** button on the Component Placement toolbar. The four resistors will align along their top edge.

3. Now click on the **Make Horizontal Spacing of Selected Components Equal** button on the Component Placement toolbar.

4. Click elsewhere in the design window to deselect all the resistors. The four resistors are now aligned and equally spaced.

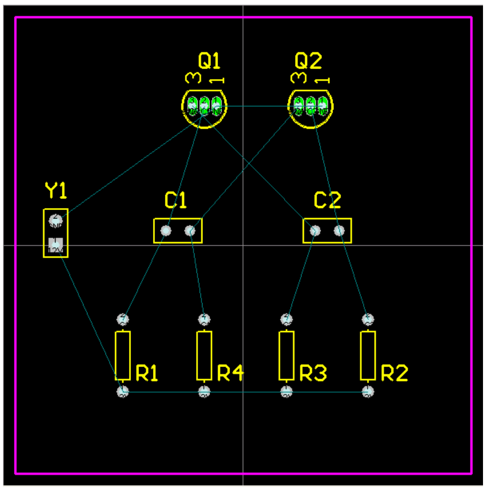
Changing a footprint

Now that we have positioned the footprints, the capacitor footprint appears too big for our requirements! Let's change the capacitor footprint to a smaller one.

1. First we will browse for a new footprint. Click on the **Libraries** panel and select `Miscellaneous Deivices.IntLib` from the Libraries list. Click on **Footprints** to display footprints available in the active library. We want a smaller radial type footprint, so type `rad` in the Filter field. Click on the Footprint names to see the footprints associated with them. The footprint RAD-0.1 will do the job.

2. Double-click on the capacitors and change the Footprint field to `RAD-0.1` in the *Component* dialog. Click **OK** and the new footprints are displayed on the board. Reposition the designators as required.

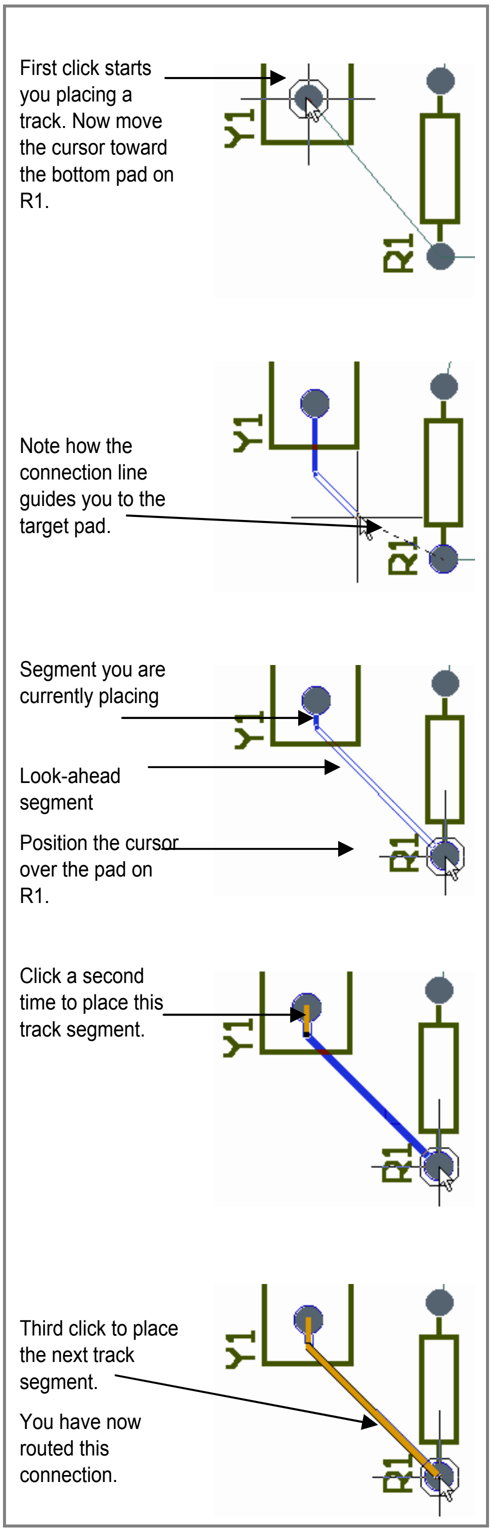
3. Your board should now look like the PCB design below.



With everything positioned, it's time to lay some tracks!

Manually routing the board

Routing is the process of laying tracks and vias on the board to connect the components. Protel DXP makes the job of routing easy by providing a number of sophisticated manual routing tools as well as the new powerful Situs topological autorouter, which optimally routes the whole or part of a board at the touch of a button.



While autorouting provides an easy and powerful way to route a board, there will be situations where you will need exact control over the placement of tracks – or you may want to route the board manually just for the fun of it! In these situations you can manually route part or all of your board. In this section of the tutorial, we will manually route the entire board "single-sided", with all tracks on the bottom layer.

We will now place tracks on the bottom layer of the board, using the "ratsnest" connection lines to guide us.

In Protel DXP, tracks on a PCB are made from a series of straight segments. Each time there is a change of direction, a new track segment begins. Also, by default Protel DXP constrains tracks to a vertical, horizontal or 45° orientation, allowing you to easily produce professional results. This behavior can be customized to suit your needs, but for this tutorial we will stay with the default.

1. Select **Place » Interactive Routing** from the menus [shortcut **P**, **T**] or click the **Interactive Routing** button on the **Placement** toolbar. The cursor will change to a cross hair indicating you are in track placement mode.
2. Examine the layer tabs that run along the bottom of the document workspace. The **TopLayer** tab should currently be active. To switch to the bottom layer without dropping out of track placement mode, press the * key on the numeric keypad. This key toggles between the available signal layers. The **BottomLayer** tab should now be active.
3. Position the cursor over the bottom-most pad on the connector Y1. Left-click or press **ENTER** to anchor the first point of the track.
4. Move the cursor towards the bottom pad of the resistor R1. Note how the track is laid. By default, tracks are constrained to vertical, horizontal or 45° directions. Also note that the track has two segments. The first (coming from the starting pad) is solid blue. This is the track segment you are actually placing. The second segment (attached to the cursor) is called the "look-ahead" segment and is drawn in outline. This segment allows you to look ahead at where the next track segment you lay could be positioned so that you can easily work your way around obstacles, maintaining a 45°/90° track orientation.
5. Position the cursor over the middle of the bottom pad of resistor R1 and left-click or press the **ENTER** key. Note that the first track segment turns blue, indicating that it has been placed on the Bottom Layer. Move the cursor around a little and you will see that you still have two segments

You may find using a larger crosshair cursor useful when placing tracks. To change the cursor size, select **Tools » Preferences**. Select **Large 90** from the Cursor Type list in the **Options** tab of the *Preferences* dialog.

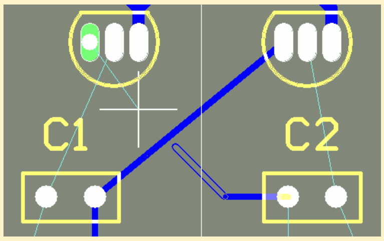
attached to the cursor: a solid blue segment that will be placed with the next mouse click and an outlined "look-ahead" segment to help you position the track.

6. Re-position the cursor over the bottom pad of R1. You will have a solid blue segment extending from the previous segment to the pad. Left-click to place the solid blue segment.

   You have just routed the first connection.

7. Move the cursor to position it over the bottom pad of resistor R4. Note a solid blue segment extends to R4. Left-click to place this segment.

8. Now move the cursor to the bottom pad of resistor R3. Note that this segment is not solid blue, but drawn in outline indicating it is a look-ahead segment. This is because each time you place a track segment the mode toggles between starting in a horizontal/vertical direction and starting at 45°. Currently it is in the 45° mode. Press the **SPACEBAR** key to toggle the segment start mode to horizontal/vertical. The segment will now be drawn in solid blue. Left-click or press the **ENTER** key to place the segment.



Note that the look-ahead segment is clipping (no longer attached to the cursor). The PCB Editor will prevent you from accidentally placing a track across another object that would cause a violation. This connection must be routed around the capacitor.

9. Move the cursor to the bottom of resistor R2. Once again you will need to press the **SPACEBAR** key to toggle the segment start mode. Left-click or press the **ENTER** key to place the segment.

10. You have now finished routing the first net. Right-click or press the **ESC** key to indicate that you have finished placing this track. The cursor will remain a cross hair, indicating that you are still in track placement mode, ready to place the next track. Press the **END** key to redraw the screen so that you can clearly see the routed net.

11. You can now route the rest of the board in a similar manner to that described in the previous steps. Figure 6 shows the manually routed board.

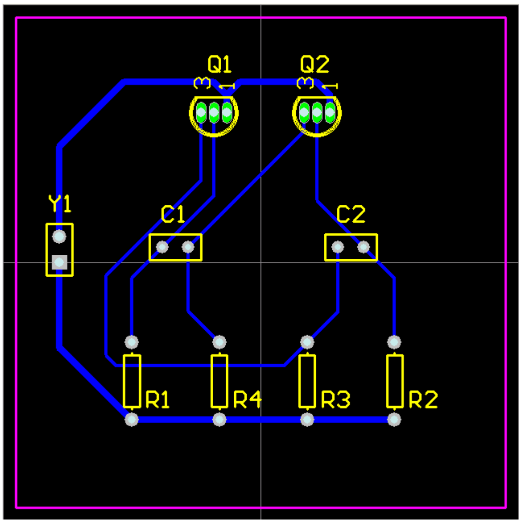


Figure 6. Manually routed board, with tracks placed on the bottom layer.

12. Save the design.

Keep in mind the following points as you are placing the tracks:

- Left-clicking the mouse (or pressing the **ENTER** key) places the track segment drawn in solid color. The outlined segment represents the look-ahead portion of the track. Placed track segments are shown in the layer color.

- Press the **SPACEBAR** key to toggle between the start horizontal/vertical and start 45° modes for the track segment you are placing.
- Press the **END** key at any time to redraw the screen.
- Press the **V, F** shortcut keys at any time to redraw the screen to fit all objects.
- Press the **PAGEUP** and **PAGEDOWN** keys at any time to zoom in or out, centered on the cursor position.
- Press the **BACKSPACE** key to "unplace" the last track segment.
- Right-click or press the **ESC** key when you have finished placing a track and want to start a new one.
- You cannot accidentally connect pads that should not be wired together. Protel DXP continually analyzes the board connectivity and prevents you from making connection mistakes or crossing tracks.
- To delete a track segment, left-click on it to select it. The segment's editing handles will appear (the rest of the track will be highlighted). Press the **DELETE** key to clear the selected track segment.
- Re-routing is easy in Protel DXP – simply route the new track segments, when you right-click to finish the old redundant track segments will automatically be removed.
- When you have finished placing all the tracks on your PCB, right-click or press the **ESC** key to exit placement mode. The cursor will change back to an arrow.

Congratulations! You have manually routed your board design.

Automatically routing the board

To see how easy it is to autoroute with Protel DXP, complete the following steps:

1. First, un-route the board by selecting **Tools » Un-Route » All** from the menus [shortcut **U**, **A**].
2. Select **Autoroute » All** from the menus [shortcut **A**, **A**].
3. When the autorouter has finished, press the **END** key to redraw the screen.

   It's as simple as that! Protel DXP's autorouter provides results comparable with that of an experienced board designer and because Protel DXP routes your board directly in the PCB window, there is no need to wrestle with exporting and importing route files.
4. Select **File » Save** [shortcut **F**, **S**] to save your board.

Note that the tracks placed by the autorouter appear in two colors: red indicates that the track is on the top signal layer of the board and blue indicates the bottom signal layer. The layers that are used by the autorouter are specified in the **Routing Layers** design rule, which was set up by the PCB Board Wizard. You will also notice that the two power net tracks running from the connector are wider, as specified by the two new Width design rules you set up.

Don't worry if the routing in your design is not exactly the same as Figure 7; the component placement will not be exactly the same, so neither will be the routing.

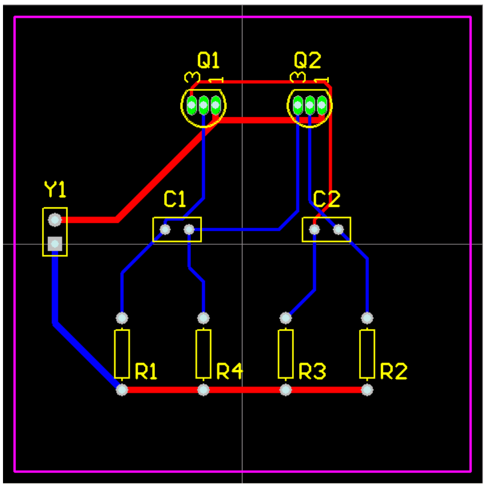


Figure 7. Fully autorouted board

Because we originally defined our board as being double-sided in the PCB Board Wizard, you could manually route your board "double-sided" using both the top and bottom layers. To do this, un-route the board by selecting **Tools » Un-Route » All** from the menus [shortcut **U**, **A**]. Start routing as before, but use the * key to toggle between the layers while placing tracks. Protel DXP will automatically insert vias if necessary when you change layers.

Verifying your board design

Protel DXP provides a rules-driven environment in which to design PCBs and allows you to define many types of design rules to ensure the integrity of your board. Typically, you set up the design rules at the start of the design process and then verify that the design complies with the rules at the end of the design process.

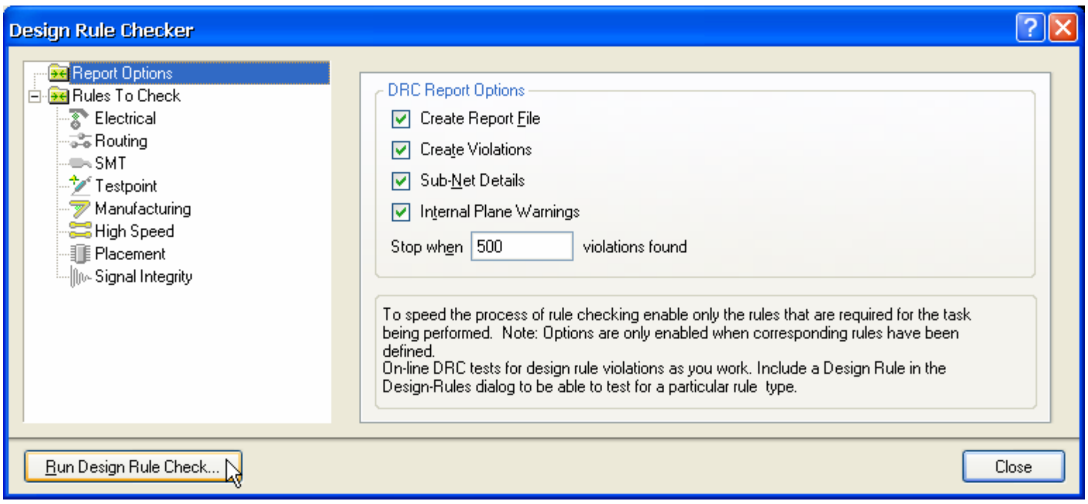
Earlier in the tutorial we examined the routing design rules and added a new width constraint rule. We also noted that there were already a number of rules that had been created by the PCB Board Wizard.

To verify that the routed circuit board conforms to the design rules, we will now run a Design Rule Check (DRC):

1. Choose **Design » Board Layers** [shortcut **L**] and ensure that **Show** button next to the **DRC Error Markers** option in the System Colors section is checked (ticked) so that the DRC error markers will be displayed, if any.

Protel DXP supports hierarchical design rules. You can set any number of rules of the same class, each with a defined scope. The rule scope determines the rule's precedence.

2. Choose **Tools » Design Rule Check** from the menus [shortcut **T**, **D**]. Both the on-line and batch DRC options are configured in the *Design Rule Checker* dialog. Click on a category to see all the rules.



3. Leave all options at their defaults and click the **Run Design Rule Check** button. The DRC will run and the results will be displayed in the Messages panel. Also, you will see that the transistor pads are highlighted in green, indicating a design rule violation.

4. Look through the errors list. It lists any violations that occur in the PCB design. Notice that there are four violations listed under the **Clearance Constraint** rule. The details show that the pads of transistors Q1 and Q2 violate the 13mil clearance rule.

5. Double-click on an error in the Messages panel to jump to its location on the PCB.

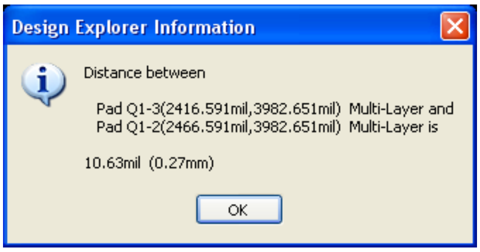
Normally you would set up the clearance constraint rules before laying out your board, taking account of routing technologies and the physical properties of the devices. Let's analyze the error then review the current clearance design rules and decide how to resolve this situation.

To find out the actual clearance between the transistor pads:

1. With the PCB document active, position the cursor over the middle of one of the transistors and press the **PAGEUP** key to zoom in.

2. Select **Reports » Measure Primitives** [shortcut **R, P**]. The cursor will change to a cross hair.

3. Position the cursor over the middle of the lower pad on the transistor and left-click or press **ENTER**. Because the cursor is over both the pad and the track connected to it, a menu will pop up to allow you to select the desired object. Select the transistor pad from the popup menu.



4. Position the cursor over the middle of one of the other transistor pads and left-click or press **ENTER**. Once again select the pad from the popup menu. An information box will open showing the minimum distance between the edges of the two pads is 10.63mil.

5. Close the information box, then right-click or press **ESC** to exit the measurement mode and then use the **V, F** shortcut to re-zoom the document.

Let's look at the current clearance design rules.

1. Select **Design » Rules** from the menus [shortcut **D**, **R**] to open the *PCB Rules and Constraints Editor* dialog. Double-click on the **Electrical** category to display all electrical rules in the right side of the dialog. Double-click on the **Clearance** type (listed in the right side) and then click on `Clearance_1` to open it. The region at the bottom of the dialog will contain a single rule, specifying that the minimum clearances for the whole board are 13mil. The clearance between the transistor pads is less than this, which is why they generate a violation when we run a DRC.

   We now know the minimum distance between transistor pads is a little over 10mil, so let's set up a design rule that allows the clearance constraint to be 10mil for the transistors only.

2. Select the **Clearance** type in the Design Rules panel, right-click and select **New Rule** to add a new clearance constraint rule.

3. Double-click on the new Clearance rule and in the **Constraints** section of the resulting dialog, set the **Minimum Clearance** to 10mil.

4. Click on **Advanced (Query)** and then click on **Query Builder** to construct the query from the Memberships Checks, or type in the query in the Query field:
   `HasFootprintPad('BCY-W3/D4.7','*')`
   The * (asterisk) indicates 'any pad' on the footprint named BCY-W3/D4.7.

5. Click **OK** to close the dialog.

6. You can now re-run the DRC from the *Design Rules Checker* dialog (**Tools » Design Rule Check**) by clicking the **Run Design Rule Check** button. There should be no rule violations.

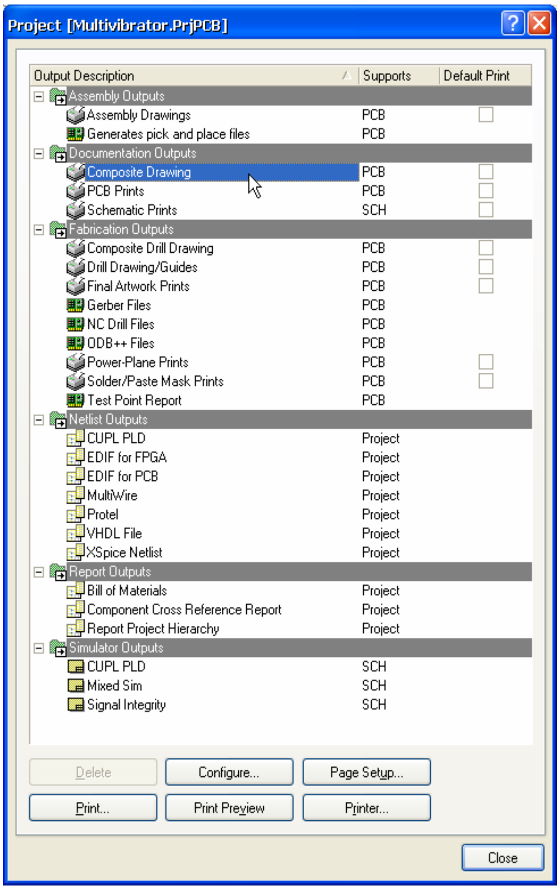
Well done! You have completed the PCB layout and are ready to produce the output documents.

Setting up the Project Outputs

Project outputs, such as printing and output files, are set up in the *Outputs for Project* dialog.

1. Select **Project » Output Jobs**. The *Project [project_name]* dialog displays.
2. Click on the output you wish to set up. If the **Configure** button is active (not grayed out) you will be able to modify the setups of that output.
3. Click **Close** when you have finished your setups.
4. If you want the outputs to be sent to individual folders according to the output type, select **Project » Project Options**, click on the **Options** tab and click on **Use separate folder for each output type** and click **OK**.

If you do not want output files to automatically open when they are created, select **Project » Project Options**, click on the **Paths** tab and deselect **Open Outputs after compile**.



Printing to a Windows printing device

Once the layout and routing of the PCB is complete, you are ready to produce the output documentation. This documentation might include a manufacturing drawing detailing the fabrication information and assembly drawings detailing component location information and loading order.

To produce these drawings, Protel DXP includes a sophisticated printing engine that gives you complete control over the printing process. You can define precisely what mix of PCB layers you want to print, preview the drawings (called printouts) and set the scaling and orientation to see exactly how it will look on the page before you print it.

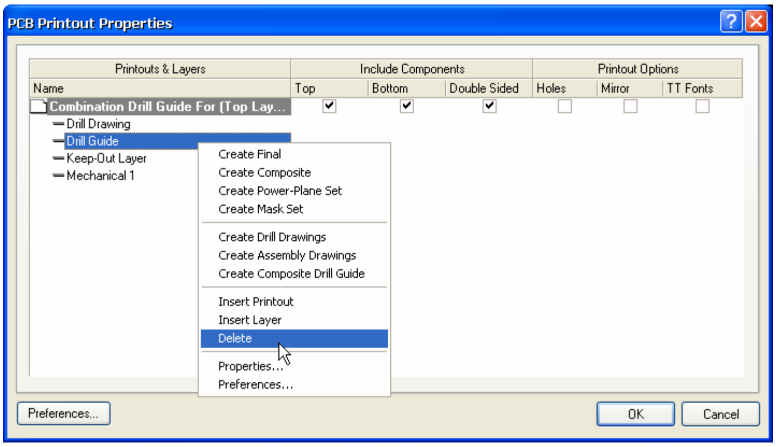
Now we will create a print preview using default output settings and then change the setups.

1. Select **File » Print Preview** from the PCB menus. The PCB will be analyzed and a default printout displayed in the print preview window. Click on **Close**.
2. To examine the set of PCB layers that are included in the printout, select **Project » Output Jobs**. The *Project* dialog displays. Select **Composite Drawing** from the **Documentation Outputs** section and click on the **Configure** button. The *PCB Printout Properties* dialog displays. You can add or delete layers using the right-click menu options. Click **OK** to close the dialog.
3. While we are in still the *Project* dialog, we will change the layer configurations for the Combination Drill Guide. Select **Composite Drill Drawing** from the **Fabrication Outputs** section and click on the **Configure** button. By default, this printout includes both the drill guide, a system layer which includes a small cross at each drill site, and the drill drawing layer, which includes a special shape at each drill site, unique for each drill size.

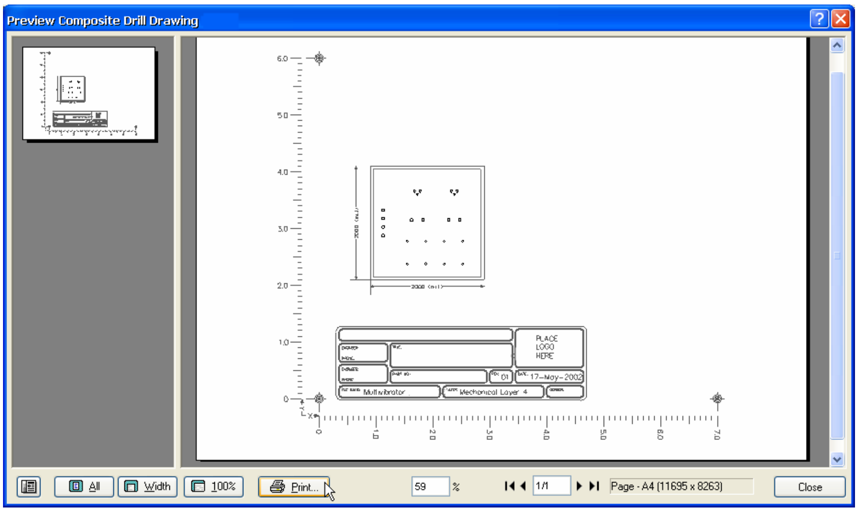
   The drill guide layer is not required in a typical drill drawing, so to remove it, right-click on the DrillGuide layer in the Printouts & Layers column and select **Delete** from the menu. Click **OK** to close the dialog.

When in the Print Preview, select **Edit » Copy** from the right-click menu to copy the current printout to the clipboard and paste it into another Windows application.

Select **File » ExportMetafile** to export the printouts to the hard disk as EMF files.



4. Now click on **Print Preview** to view your drill drawing. You can then click on **Print** to display your printer setups and then click **OK** to send your drawing to the specified printer.



5. Click **Close** to close the print preview window.

6. To change the target printer and set the page orientation and scaling, you can select the **Page Setup** button in the *Project* dialog (or **File » Page Setup** from the menus). Choose your preferred printer and set the printer page to Landscape.

7. When you have completed your setups, close all open dialogs.

Manufacturing output files

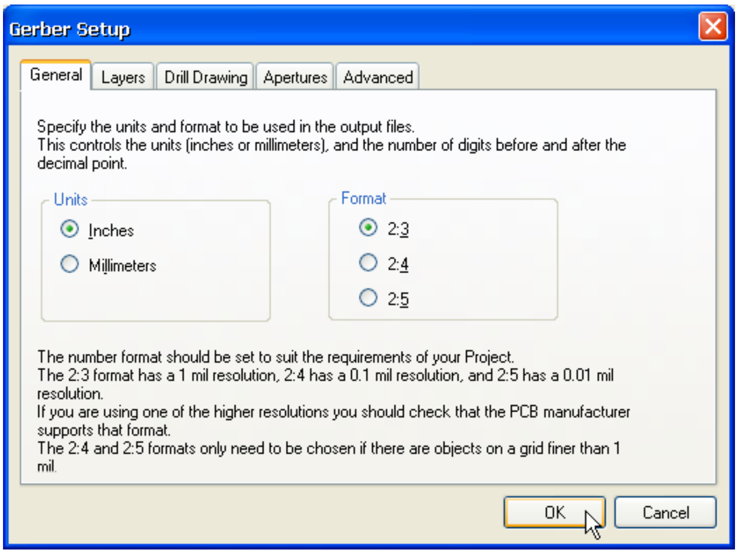
The final phase of the PCB design process is to generate the manufacturing files. The set of files that are used to manufacture and fabricate the PCB include Gerber files, NC drill files, pick and place files, a bill of materials and testpoint files. Output files can be set in the *Project* dialog (**Project » Output Jobs**) or through individual commands on the **File » Fabrication Outputs** menu. The setups for the manufacturing documents are stored as part of the project file.

Generating Gerber files

Each Gerber file corresponds to one layer in the physical board – the component overlay, top signal layer, bottom signal layer, the solder masking layers and so on. It is advisable to consult with your PCB manufacturer to confirm their requirements before generating the Gerber and NC drill files required to fabricate your design.

To create the manufacturing files for the tutorial PCB:

1. Make the PCB the active document, then select **File » Fabrication Outputs »Gerber files**. The *Gerber Setup* dialog displays.

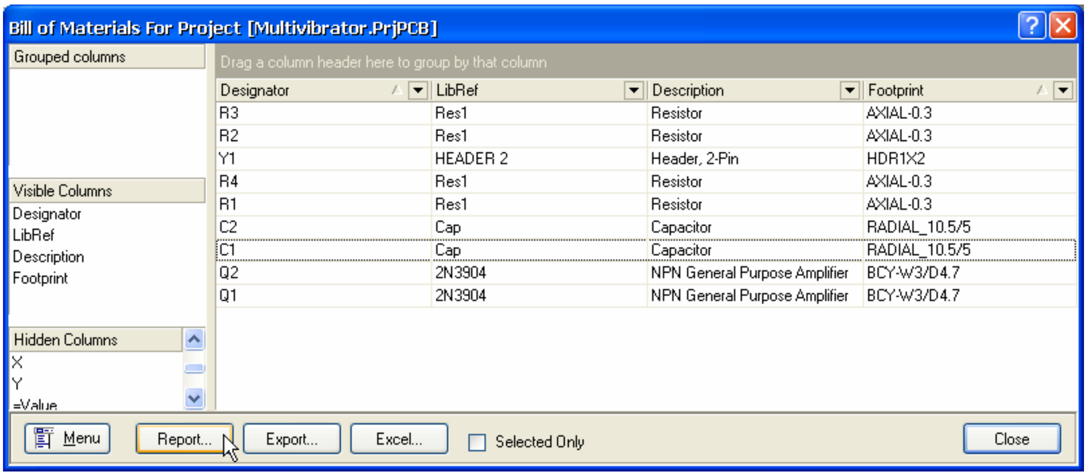


2. Click **OK** to accept the default settings. The Gerber files are produced and CAMtastic! opens to display the files. The Gerber files are stored in the `Project Outputs` folder which is automatically created in the folder where your project files reside. Each file has the file extension added that corresponds to the layer name, e.g. `Multivibrator.GTO` for Gerber Top Overlay.

Bill of Materials

1. To create a Bill of Materials, first configure your report. Choose **Project » Output Jobs** and select **Bill of Materials** in the **Report Outputs** section of the *Project* dialog.

2. Click on **Create Report**. In this dialog, you can set up which information you require for your BOM report by clicking and dragging the column headings between the Visible and Hidden Column sections.



3. Click on **Report**… to display a print preview of your BOM. This preview can then be printed using the **Print** button or exported to a file format, such as `.xls` for Microsoft Excel, using the **Export** button.

4. Close the dialogs.

Congratulations! You have completed the PCB design process.

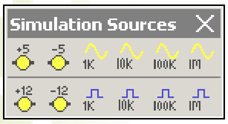
Simulating the design

Protel DXP allows you to run a vast array of circuit simulations directly from a schematic. In the following sections of the tutorial we will simulate the output waveforms produced by our multivibrator circuit.

Setting up for simulation

Before we can run a simulation we need to add a few things to our circuit: a voltage source to power the multivibrator; a ground reference for the simulations and some net labels on the points of the circuit where we wish to view waveforms.

1. Click on the `Multivibrator.SchDoc` tab at the top of the window to make the schematic the active document.
2. We must replace the connector with a voltage source. To delete the connector, click once on the body of the connector to select it (a dotted selection box will appear around the connector), then press the **DELETE** key on the keyboard.
3. At the moment there is not enough room for the voltage source, so we'll move the free ends of the wires. To move the dangling end of the 12V wire, click once on the wire to select it. When the small square editing handles appear, click once on the handle on the free end of the wire, then move the handle up almost to where the wire changes direction (but not all the way up). Click again to "drop" the handle.
4. Repeat this process for the dangling end of the GND wire, moving it toward the bottom of the sheet.



5. Select **View » Toolbars » Simulation Sources** to display the Simulation Sources toolbar.
6. Click the **+12V** source button on the Simulation Sources toolbar. A source symbol will appear floating on the cursor. Press the **TAB** key on the keyboard to edit its attributes. In the resulting dialog, click the **Attributes** tab to make it active and set the Designator field to V1. Click the **OK** button to close the dialog and then place the source between the dangling ends of the 12V and GND wires.
7. Using the same technique you used to move the dangling ends of the 12V and GND wires apart, move them again to attach each wire end to either end of the voltage source, as shown in Figure 9 (simulation-ready schematic).

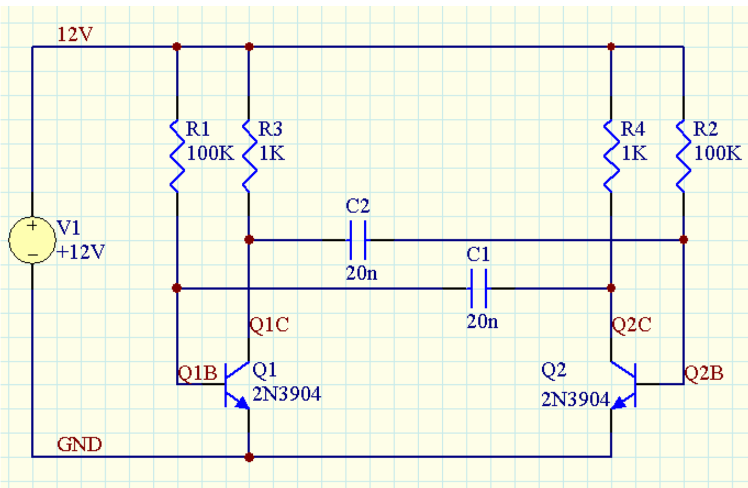


Figure 9. Simulation-ready schematic

Our last task before running a simulation is to place net labels at appropriate points on the circuit so we can easily identify the signals we wish to view. In the tutorial circuit, the points of interest are the base and collectors of the two transistors.

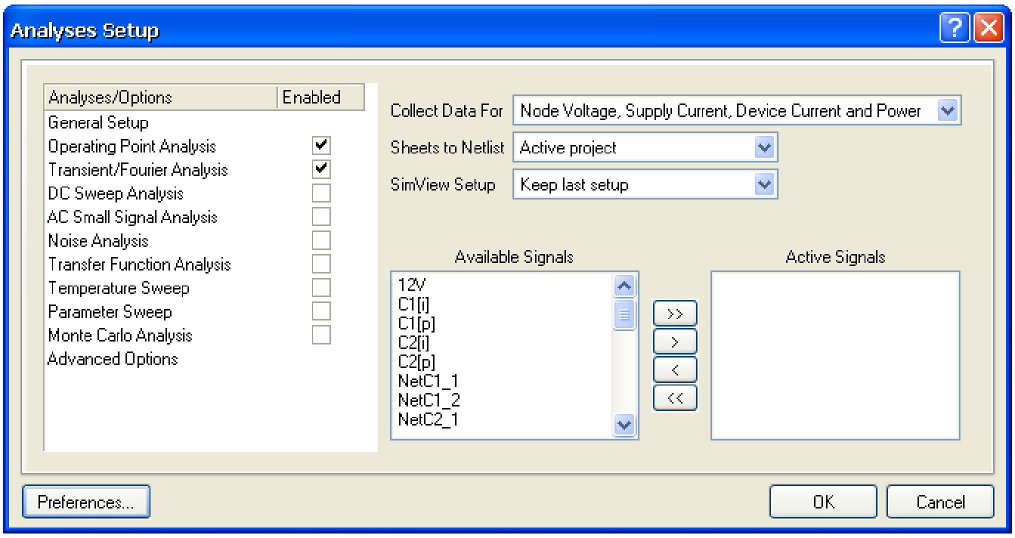
The electrical "hot spot" of a net label is the bottom left corner, so ensure that this corner touches the wire.

1. Select **Place » Net Label** from the menus [shortcut **P**, **N**]. Press the **TAB** key to edit the net label attributes. In the *Net Label* dialog, set the **Net** field to `Q1B` and close the dialog.
2. Position the cursor over the wire coming from the base of Q1. See Figure 9 for net label placement. Left-click or press **ENTER** to place the net label on the wire.
3. Press the **TAB** key and change the **Net** field to `Q1C`.
4. Position the cursor over the wire coming from the collector of Q1 and left-click to place the second net label.
5. Similarly, place net labels with designators Q2B and Q2C on the base and collector wires of Q2 respectively.
6. When you have finished placing net labels, right-click, or press **ESC**, to exit placement mode.
7. To save your simulation-ready circuit with a different name to that of your original schematic, select **File » Save As** [shortcut **F**, **A**] and type `Multivibrator simulation.SchDoc` in the *Save As* dialog.

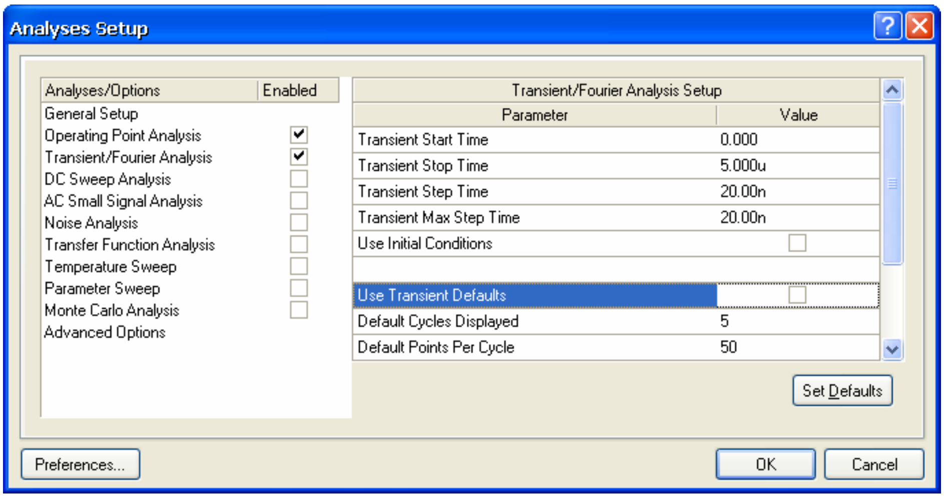
Running a transient analysis

Your schematic now has all the necessary additions, so let's set up a transient analysis of the circuit. In our tutorial circuit, the RC time constant is 100k x 20n = 2 milliseconds. To view five cycles of the oscillation, we will set up to view a 10ms portion of the waveform.

1. Select **Design » Simulate » Mix Sim** from the menus to display the *Analyses Setup* dialog. All the simulation options are set up here.



2. First we will set up the nodes in the circuit that you wish to observe. In the **Collect Data For** field, select **Node Voltage and Supply Current** from the list. This option defines what type of data you want calculated during the simulation run.
3. In the **Available Signals** field, double-click on the **Q1B**, **Q2B**, **Q1C** and **Q2C** signal names. As you double-click on each one, it will move to the **Active Signals** field.
4. Make sure the **Operating Point Analysis** and the **Transient/Fourier** checkboxes are enabled (ticked) for this analysis. If the Transient/Fourier Analysis Setup does not display automatically, click on the **Transient/Fourier** analysis name.
5. Check that the **Use Transient Defaults** option is disabled, so that the Transient Analysis parameters are available.



6. To specify a 10ms simulation window, set the **Transient Stop Time** field to `10m`.

7. Now set the **Transient Step Time** field to `10u`, indicating that the simulation should display a point every 10us (giving 1000 display points in all, enough to give an accurate picture of the results).

8. During simulation, the actual timestep is varied automatically to achieve convergence. The **Maximum Step** field limits the variation of the timestep size, set the **Transient Max Step Time** to `10u`.

You are now ready to run a transient simulation.

1. Click the **OK** button at the bottom of the *Analyses Setup* dialog to run the simulation.

2. The simulation will be performed, when it is finished you should see output waveforms similar to those shown in Figure 10.

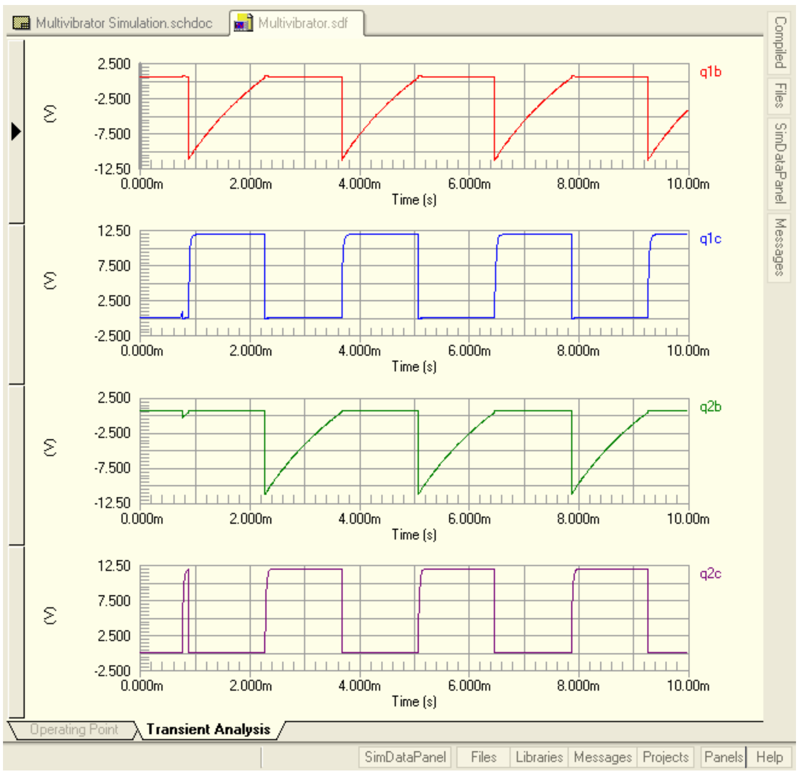


Figure 10. Output waveforms from the multivibrator.

Congratulations! You have simulated your circuit and displayed its output waveforms.

If you like, you can change the values of some of the components on your schematic and re-run the simulation to see the effects. Try changing the value of C1 to `47n` (double-click on C1 to edit its attributes) and re-running the transient analysis. The output waveforms will show an uneven mark/space ratio.

Further explorations

This tutorial has introduced you to just some of the powerful features of Protel DXP. We've captured a schematic, run a transient simulation on the design, and designed and routed a PCB, all with the integrated tools provided in Protel DXP. But we've only just scratched the surface of the design power provided by Protel DXP.

Once you start exploring Protel DXP you will find a wealth of features to make your design life easier. To demonstrate the capabilities of the software, a number of example files are included. You can open these examples in the normal way, select **File » Open** from the Design Explorer menus then navigate to the `\Program Files\Altium\Examples\` folder. As well as the board design examples in this folder, there are a number of sub-folders with examples that demonstrate specific features of Protel DXP.

Check out the `\Circuit Simulation\` sub-folder to explore Protel DXP's analog and digital simulation capabilities. As well as analog examples that demonstrate various circuit designs, such as amplifiers and power supplies, there are mixed-mode examples, a math function example, an example that includes linear and non-linear dependent sources and even a vacuum tube example!

With faster logic switching and design clock speeds, the quality of the digital signals becomes more important. Protel DXP includes a sophisticated signal integrity analysis tool that can accurately model and analyze your board layout. The signal integrity requirements such as impedance, overshoot, undershoot, and slope are defined as PCB design rules, and then tested during the standard design rule check.

If there are nets that you need to analyze in more detail, you can select **Tools » Signal Integrity » Setup and Run** to pass the design to the Signal Integrity Analyzer, where you can perform reflection and cross talk simulations. The results are displayed in an oscilloscope-like waveform analyzer, where you can examine the performance and take measurements directly from the waveforms.

Add-ons

Protel's Design Explorer is built on an open client/server architecture, so you can extend the functionality of Protel DXP with add-ons available from both Protel and third-party vendors. Add-ons integrate seamlessly into the Design Explorer environment, providing additional functions and services – extending the capabilities of your Protel DXP design tools. The Protel web site at www.protel.com provides an up-to-date list of the latest Add-ons, many of which are directly downloadable.

Thanks for participating in the Protel DXP introductory tutorial.

Shortcut keys

Design Explorer Shortcuts

Left-Click	Select document under cursor
Double Left-Click	Edit document under cursor
Right-Click	Display context sensitive pop-up menu
Ctrl + F4	Close active document
Ctrl + Tab	Cycle through open documents
Drag & Drop from	
• one project to another	Move selected document
• File Explorer to Design Explorer	Open selected document as a free document
Alt + F4	Close Design Explorer DXP

Common Schematic and PCB Shortcuts

Shift	While autopanning to pan at higher speed
Y	While placing an object to flip it along the Y-axis
X	While placing an object to flip it along the X-axis
Shift+ ↑ ↓ ← →	Move cursor ten grid increments in direction of arrow key
↑ ↓ ← →	Move cursor one grid increment in direction of arrow key
Space	Abort screen re-draw
Esc	Escape from current process
End	Redraw the screen
Home	Redraw screen with center at the cursor point
PageDown or Ctrl + mouse wheel	Zoom out
PageUp or Ctrl + mouse wheel	Zoom in (zooms around cursor, position the cursor first)
Mouse wheel	Pan Up/Down
Shift + mouse wheel	Pan Left/Right
Ctrl + Z	Undo
Ctrl + Y	Redo
Ctrl + A	Select All
Ctrl + S	Save current document
Ctrl + C	Copy
Ctrl + X	Cut
Ctrl + V	Paste
Ctrl + R	Copy and repeat paste selected objects
Delete	Delete Selection
V + D	View Document
V + F	View Fit placed objects
X + A	De-select all
Right-Click & Hold	Display slider hand & slide view
Left-Click	Focus or select object
Right-Click	Pop-up floating menu, or escape from current operation
Right-Click on object, select Find Similar	Load object under cursor into Find Similar dialog
Left-Click, Hold & Drag	Select inside area
Left-Click & Hold	Move object/selection under cursor

Left Double-Click	Edit object
Shift + Left-Click	Select/deselect object
TAB	Edit attributes while placing
Shift + C	Clear current Filter
Shift + F	Click on object to display Find Similar dialog
Y	Popup Quick Queries menu
F11	Toggle Inspector panel on/off
F12	Toggle List panel on/off

Schematic Shortcuts

Alt	Constrain object movement in line horizontally and vertically
G	Cycle through snap grid setting
Spacebar	Rotate object being moved by 90 degrees
Spacebar	Toggle start/end mode while placing a wire/bus/line
Shift + Spacebar	Step through placement modes while placing a wire/bus/line
Backspace	Remove the last vertex when placing a wire/bus/line/polygon
Left-Click, Hold +Delete	When a wire is focused to delete a vertex
Left-Click, Hold +Insert	When a wire is focused to add a vertex
Ctrl + Left-Click & Drag	Drag object

PCB Shortcuts

Shift + R	Cycle through three routing modes (ignore, avoid or push obstacle)
Shift + E	Toggle electrical grid on/off
Ctrl + G	Pop up snap grid dialog
G	Pop up snap grid menu
N	Hide the ratsnest while moving a component
L	Flip component being moved to the other side of board
Backspace	Remove last track corner during track placement
Shift + Space	Step through corner modes during track placement
Space	Toggle start/end mode during track placement
Shift + S	Toggle single layer mode on/off
O + D + D + Enter	Set all primitives to display in draft mode
O + D + F + Enter	Set all primitives to display in final mode
O + D	Show/Hide Tab of *Preferences* dialog
L	View *Board Layers* dialog
Ctrl + H	Select connected copper
Ctrl + Shift + Left-Click	Break track
+	Next layer (numeric keypad)
-	Previous layer (numeric keypad)
*	Next routing layer (numeric keypad)
M + V	Move split plane vertices
Alt	Hold to temporarily switch from avoid-obstacle to ignore-obstacle mode
Ctrl	Hold to temporarily disable electrical grid while routing
Ctrl + M	Measure distance
Shift + Space	Rotate object being moved (clockwise)
Space	Rotate object being moved (anti-clockwise)
Q	Toggle units (metric/imperial).