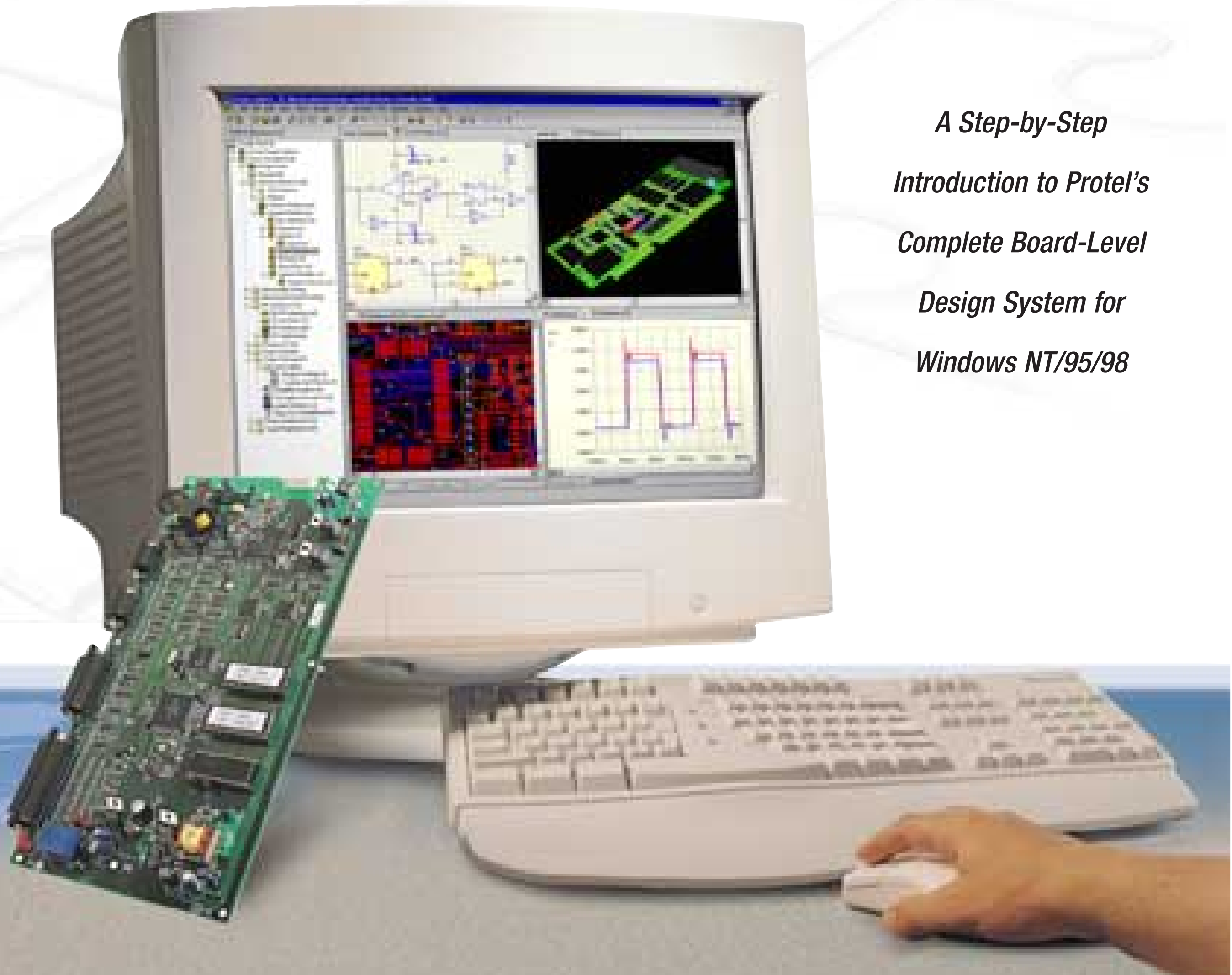
Introductory Tutorial

Exploring Protel 99 SE

A Step-by-Step Introduction to Protel's Complete Board-Level Design System for Windows NT/95/98



Protel Offices

NORTH AMERICA

Protel Technology Inc.
5252 N. Edgewood Dr.
Suite 175, Provo, UT 84604
Phone: (801) 224-0333
Fax: (801) 224-0558
Toll Free: 1 800 544 4186
E-mail sales: salesusa@protel.com
E-mail support: helpme@protel.com

EUROPE

Protel Europe AG (Switzerland)
Hinterdorfstrasse 33 CH-4334 Sisseln
Phone: +41 62 866 41 11
Fax: +41 62 866 41 10
Freecall sales:
In deutscher Sprache: 00800 776 776 77
In Dutch or English: 00800 776 776 44
En Français: 00800 776 776 55
Support phone:
In deutscher Sprache: +41 62 866 41 31
In English: +41 62 866 41 32
En français: +41 62 866 41 30
E-mail sales: eda@protel.ch
E-mail support: support@protel.ch

JAPAN

Protel Japan KK.
351-1 Sunayama-cho
Hamamatsu, SHIZ 430 0926
Sales phone: +81 53 453 6705
Support phone: +81 53 453 8058
Fax: +81 53 453 6707
E-mail sales: sales@protel.co.jp
E-mail support: support@protel.co.jp

AUSTRALASIA

Protel International Limited
PO Box 1876
Dee Why NSW 2099, Australia
Phone: +61 2 9984 0016
Fax: +61 2 9984 0017
Freecall sales: 1 800 030 949
Freecall support: 1 800 676 684
E-mail sales: sales@protel.com.au
E-mail support: support@protel.com.au

For more information on products and resources for designers, visit: www.protel.com

Protel
Making Electric Design Easy™

Table of contents

Welcome to Protel 99 SE

Welcome to the world of Protel 99 SE – a complete 32-bit electronic design system for Windows 95/98/NT. Protel 99 SE provides a completely integrated suite of design tools that lets you easily take your designs from concept through to final board layout.

Protel 99 SE brings to the EDA desktop a new level of integration. Unlike other tool suites that provide separate applications for each phase of the design process, all Protel 99 SE tools run within a single application environment – the *Design Explorer*. Start Protel 99 SE and the Design Explorer opens, putting all your design tools at your fingertips. You benefit from a single, consistent, customizable user environment – no need to grapple with different applications for different design tasks.

A single environment also means Protel 99 SE eliminates import/export hassles as you progress through the design process. For example, autorouting a board in Protel 99 SE is done directly in the PCB document window at the touch of a single button – no need to fiddle with separate router files. With Protel 99 SE you can compile a circuit for implementation in a programmable logic device directly from the schematic, producing an industry-standard JEDEC device programming file in a single action.

Simulation is also integrated into Protel 99 SE. You can set up and run mixed analog/digital simulations directly from the schematic. Protel 99 SE comes with comprehensive libraries of simulation-ready components, with full support for the industry-standard simulation language, SPICE 3f5.

Protel 99 SE comes with extensive schematic and PCB footprint libraries. The Protel Library Development Center continually updates and adds to these libraries, ensuring you will always be up-to-date with the latest devices and packaging technologies from all major manufacturers. Updated libraries are available for download from the Protel web site at www.protel.com.

Protel 99 SE also includes integrated spreadsheet, charting, text editing and macro creation tools, allowing you to manage all aspects of a design project without leaving the Design Explorer environment.

Because the Design Explorer is built on an open client/server architecture, you can extend the functionality of Protel 99 SE 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 99 SE 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.

Install Protel 99 SE and you have at your fingertips all the tools you need to produce sophisticated designs with unprecedented speed and ease.

The Protel 99 SE Design Explorer

The Design Explorer is your interface to your designs, and the design tools. To start Protel 99 SE and open the Design Explorer, select Protel 99 SE from the Protel 99 SE Program Group in the Windows Start menu – the Design Explorer will open, ready for work. Before starting your own design you might like to explore some of the example designs included with Protel 99 SE – these are located in the \Program Files\Design Explorer 99 SE\Examples\ folder.

• Document/Folder Tabs

Each open document and folder has its own Tab (the icon shows the active Tab). Right-click on a Tab to split the window, click and drag to move a Tab to a different region of the window.

• System Menu

Click the down-arrow to display the system menu – use this to set up the preferences, compact a database and customize the resources.

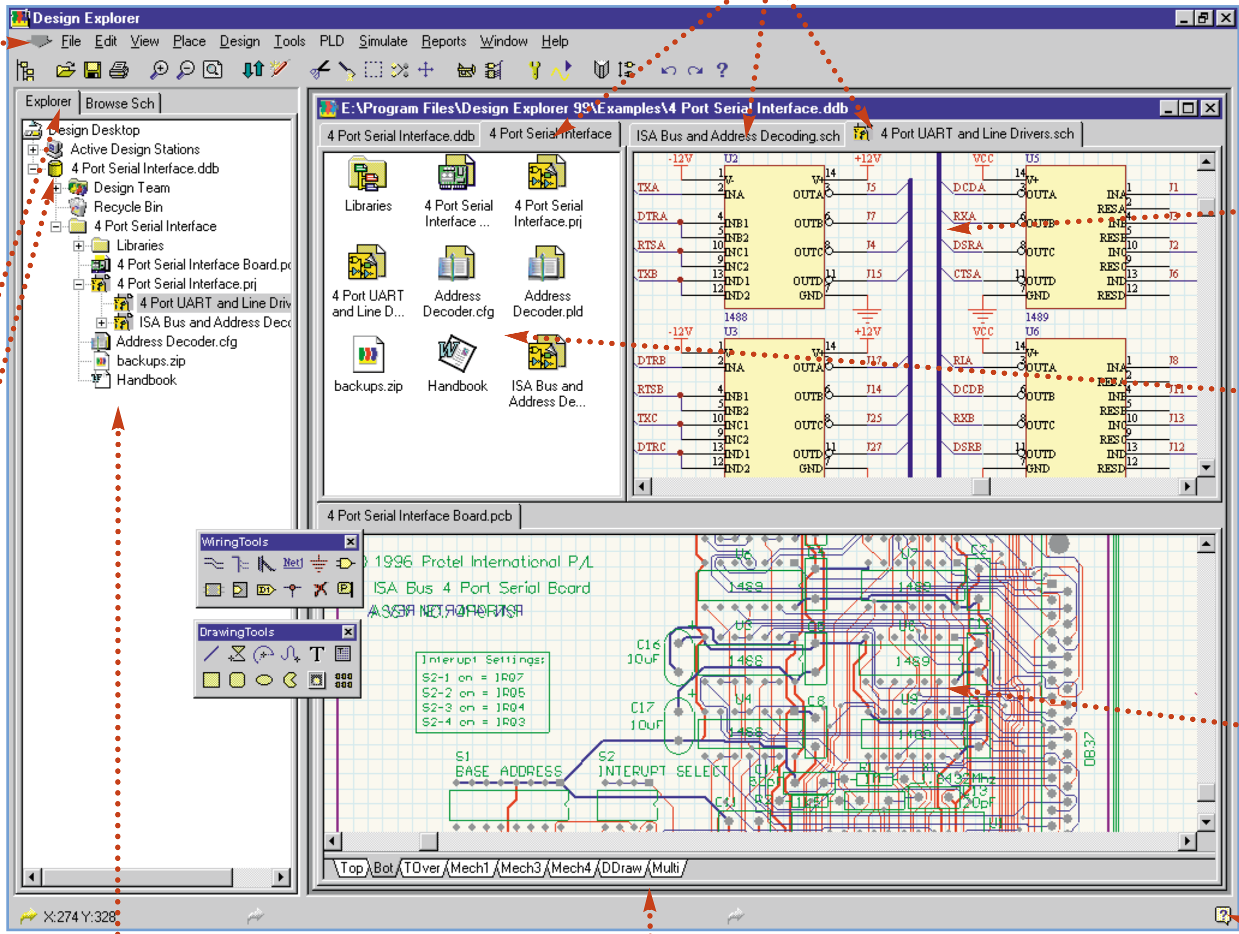
All other menus and toolbars automatically change to suit the document being edited.

• Explorer and Editor Panels

As well as the Design Explorer navigation panel, other editor panels are available for each editor (schematic, PCB, text, etc).

• Active Database

All design documents are stored in an integrated design database.



• Navigation Panel

Displays all the documents and folders in the design database. The active document is highlighted, right-click on a document or folder for a menu of options.

• Design Window

Displays the documents that are currently open in this design – there is a Tab for each open document and folder. The window has been split into 3 regions to display a folder, a schematic sheet and the PCB.

• Schematic Editor

Note the Tabs at the top – there are 2 schematics currently open.

• Folder View

Use folders to organize the design documents. Right-click to create a new document, click and drag to move a document to a different folder.

• PCB Editor

You can open and work on the PCB at the same time.

• Help Advisor

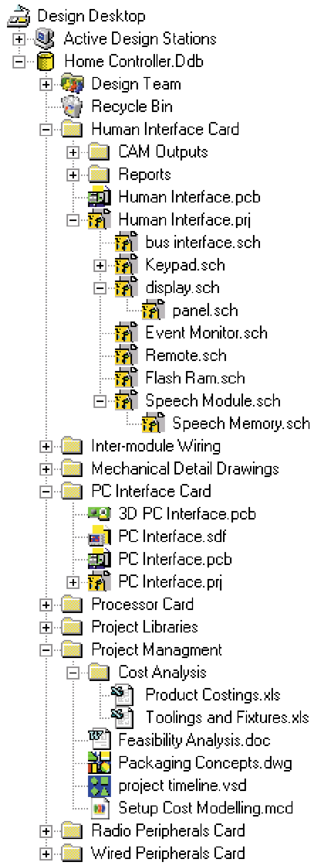
Use the natural language help system to quickly find the answer to your question.

Working in the Design Explorer

Working in the Design Explorer is just like working in the Windows File Explorer – if you are familiar with the Windows Explorer then you are ready to go!

Like the Windows Explorer, there are 2 regions to work in – the navigation panel on the left, that displays the contents of the design in a tree-like structure, and the view of what is currently selected in the tree is displayed on the right. Simply click on a document or folder in the tree to display it on the right.

All the documents in a design are displayed in the same window, each on a separate tab. Right-click on a tab to split the window and view multiple documents simultaneously.



Create folders and store all the documents for your design in the one design database

How the design documents are stored

Protel 99 SE stores all the design documents – including the schematic sheets, PCB, libraries, simulation results, manufacturing files, and so on – in an integrated design database. There is no need to find and open each document individually, you simply open the database, and from there you can access all of the design documents.

As well as storing documents within a database, you can also create folders to organize the documents.

To make it easy to manage all the documents in your project you can also store any other type of document, including Microsoft Word and Excel documents, AutoCAD drawings, and so on, in the design database. Documents created by "OLE compliant servers", such as Word and Excel, can be opened directly from the Protel 99 SE design database, simply double-click to edit them.

Tips on working in the Design Explorer

- Select **File » New** to create a new document or folder.
- Right-click in a folder and select Import to import any document into a design database.
- Right-click on a document (or folder) to export it from the design database.
- Select **Design Utilities** from the System menu (the down-arrow) to compact an MS Access database.
- Click-and-drag to re-organize documents and folders in a design.
- Hold the ALT key as you open a design to stop documents automatically re-opening.

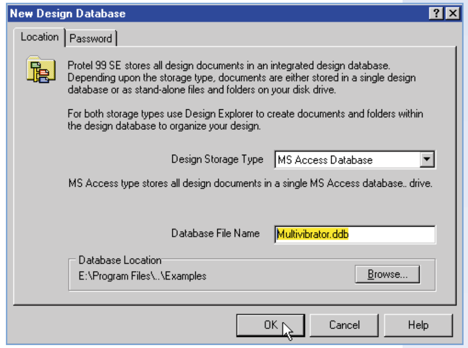
Creating a new design

To start the tutorial, create a new design database by completing the following steps:

➤ Close any design databases that are currently open.

➤ Select **File » New** from the menus.

The *New Design Database* dialog will appear. Two types of design storage options are available – a **MS Access Database**, where all the design documents are stored within the one database file on the disk drive, and **Windows File System**, where the design documents are stored separately on the disk drive. Regardless of which storage option you choose, the way you work in the design database is exactly the same.



The first step is to create a new design database

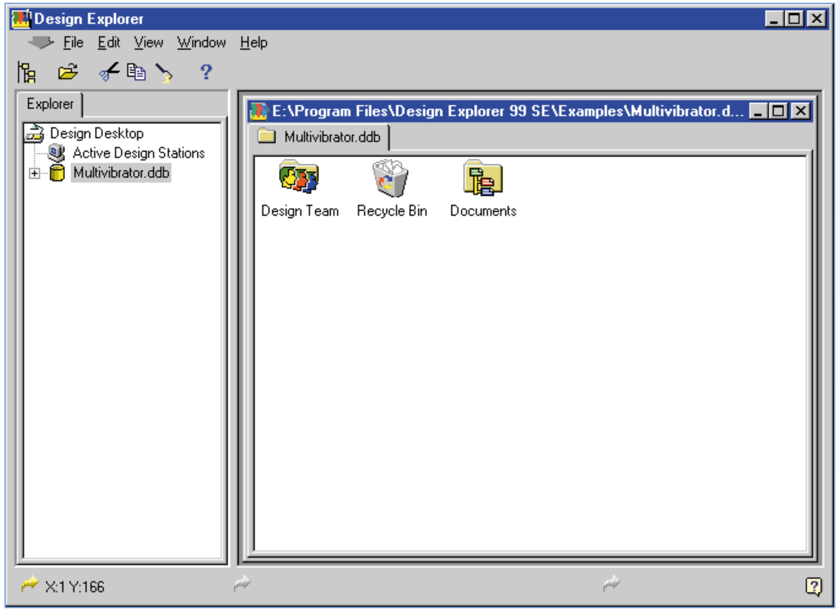
➤ For the tutorial we will use an Access database – set the **Design Storage Type** to **MS Access Database**.

➤ Type the name Multivibrator.ddb in the **Database File Name** field.

➤ Click the **Browse** button and navigate to a location where you would like to store the design.

➤ Click **OK** to close the *New Design Database* dialog – the new database will open in the Design Explorer.

You are now ready to start capturing the schematic.



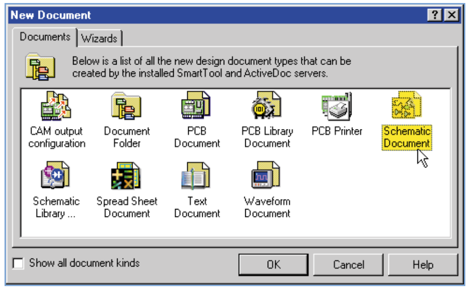
The new Multivibrator design database open in the Design Explorer

Capturing the schematic

Creating a new schematic sheet

To create a new blank schematic sheet complete the following steps:

- In the panel on the left, expand the navigation tree by clicking on the small + symbol next to the Multivibrator.ddb icon.
- Click on the **Documents** folder icon to display the contents of the folder on the right.
- From the menu bar select **File » New**.
- In the *New Document* dialog click on the **Schematic Document** icon to select a new schematic sheet.
- Click the **OK** button to close the dialog and create a new schematic sheet icon in the Documents folder.



- The icon is created with a default name. Select **Edit » Rename** and type in the new name for the sheet, Multivibrator.sch.
- Double-click on the Multivibrator.sch icon to open the schematic sheet.

When the blank schematic sheet opens you will notice that the workspace changes. The main toolbar includes a range of new buttons, two new toolbars are visible, the menu bar includes new items, and the Panel changes to include schematic component library management tools.

You can customize many aspects of the workspace. For example, you can reposition the two "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. To find out how to customize other aspects of the workspace, see the topic, *Using the Design Explorer*, in the main help file.

Setting the schematic options

Protel 99 SE 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.

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

- From the menus choose **Design » Options**, 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.
- Use the scroll bar to scroll up to the **A4** style and click to select it.
- Click the **OK** button to close the dialog and update the sheet size.

To make the document fill the viewing area again, select the **View » Fit Document** menu item. In Protel 99 SE 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.

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

Refer to the *Sheet Templates topic* in the *Setting Up the Schematic Workspace* chapter of the Protel 99 SE Designer's Handbook for complete details on creating a schematic template.

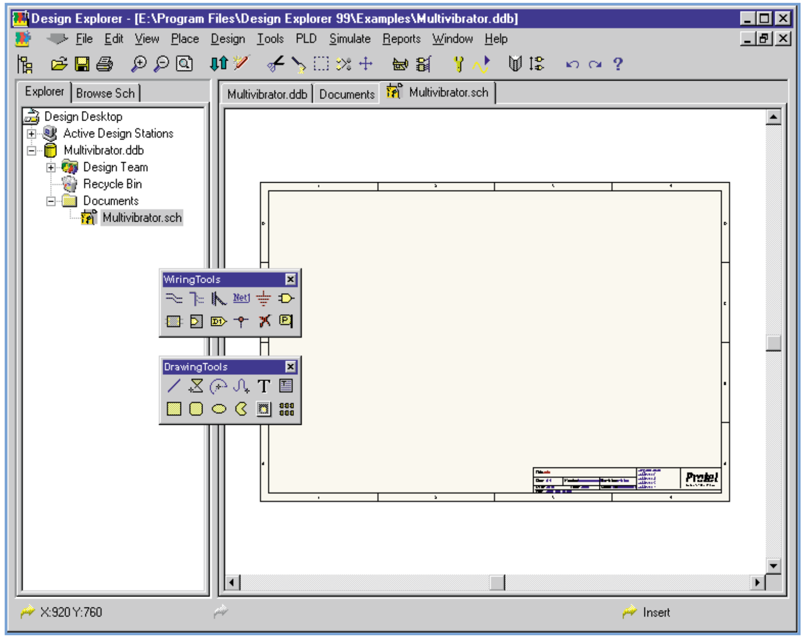
Next we will set the general schematic preferences:

- 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.
- Click on the **Default Primitives** tab to make it active and enable the **Permanent** check box. Click the **OK** button to close the dialog.

Before you start capturing your schematic, save this schematic sheet:

- From the menus select **File » Save** [shortcut `F`, `S`].

Your blank sheet will be saved in the Documents folder.



The Multivibrator design database open in the Design Explorer. The schematic sheet is open, ready to start the design.

Drawing the schematic

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

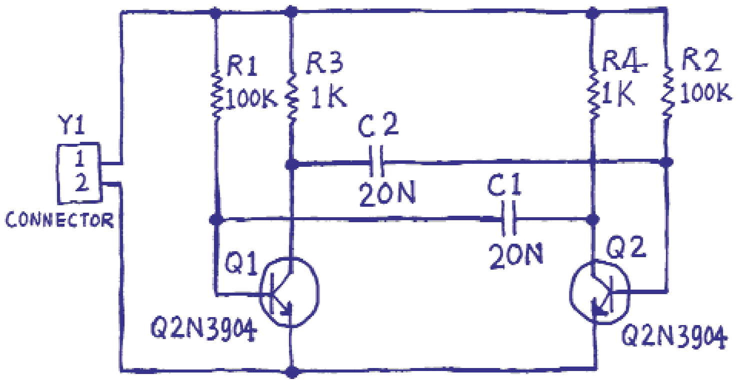


Figure 1
A stable multivibrator

Locating the component and loading the libraries

To manage the thousands of schematic symbols included with Protel 99 SE, the Schematic Editor provides powerful library search features. Work through the following steps to locate and add the libraries you will need for the tutorial circuit:

➤ From the menus select **Tools » Find Component** [shortcut T, O], or press the **Find** button in the Schematic Editor Panel. This will open the *Find Schematic Component* dialog.

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

➤ Ensure there is a tick in the box next to the **By Library Reference** field (click in the check box to enable it if necessary).

➤ We want to search for all references to 3904, so in the **By Library Reference** text field type *3904* (the * symbol is a wildcard used to take into account the different prefixes and suffixes used by different manufacturers).

➤ Ensure that the **Search Scope** is set to **Specified Path**, and that the **Path** field contains the correct path to your schematic libraries. If you accepted the default directories during installation, the path should be C:\Program Files\Design Explorer 99 SE\Library\Sch\. Ensure that the **Sub directories** and **Find All Instances** boxes are ticked.

➤ Click the **Find Now** button to begin the search. There are over 60,000 components supplied with Protel 99 SE, so the search may take a couple of minutes. If you have entered the parameters correctly, 2 libraries will be found and displayed in the **Found Libraries** list.

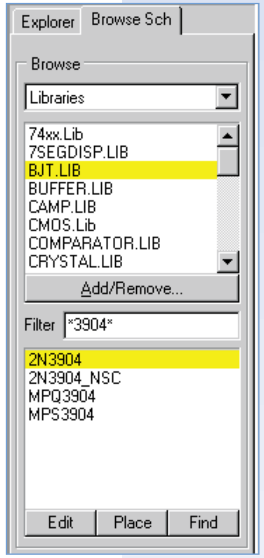
➤ Click on the **BJT.LIB** library to select it. This library has symbols for all the available simulation-ready BJT transistors. Like all the simulation-ready libraries, the BJT library is in the **SIM.DDB** library database.

➤ Click the `Add To Library List` button to make this library available to your schematic.

You will notice that a number of libraries appear in the **Browse Sch** panel, which should be visible behind the open dialog – when you add one library from a library database all libraries in that database are added to the list of available libraries. Because all simulation-ready libraries are now in the library list, we will not need to search for the other components.

➤ Close the *Find Schematic Component* dialog.

The libraries in the Sim.ddb will appear in the **Browse Sch** panel whenever the Browse mode at the top of the panel is set to **Libraries**. 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.



Use the filter to quickly locate the component you need

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.

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

➤ Set the **Browse** mode to **Libraries** in the **Browse Sch** Editor Panel.

➤ Q1 and Q2 are BJT transistors, click on the BJT.LIB library to make it the active library.

➤ In the **Filter** section of the panel, type *3904* and press the ENTER key. A list of components which have the text "3904" as part of their **Part Type** field will be displayed.

➤ Click on the **2N3904** entry in the list to select it, then click the **Place** button.

➤ 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.

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 *Part* dialog for the component.

➤ In the **Attributes** tab of the dialog set the following values: for **Footprint**, type TO92A and for **Designator**, type Q1. Leave all other fields at their default values.

➤ Click the **OK** button to return to placement mode.

You are now ready to place the part.

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

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

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

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.

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°

floating on the cursor. This feature of Protel 99 SE allows you to place multiple parts of the same type. So let us now place the second transistor.

Because this transistor is the same as the previous one, there is no need to edit its attributes before we place it. Protel 99 SE 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.

- 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.
- 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.
- 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.

Use the following shortcut keys to change your view of the sheet:

- `PAGEUP` zooms in one step
- `PAGEDOWN` zooms out one step
- Shortcuts `V,D` redraws the sheet to fit the current window
- Shortcuts `V,F` redraws the sheet so that all placed objects are visible in the current window
- The `END` key redraws the screen

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.

- 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.

Next we will place the four resistors:

- In the list of libraries in the **Browse Sch** Panel, scroll down and select the **Simulation Symbols.Lib** library.

The Simulation Symbols.Lib library includes generic components such as resistors, capacitors, voltage and current sources, and linear and non-linear dependant sources.

- In the **Browse Sch** panel set the **Filter** to `*res*` and press `ENTER`.
- Click on **RES** in the components list to select it, then click the **Place** button. You will now have a resistor symbol floating on the cursor.
- Press the `TAB` key to edit the resistor's attributes.
- In the **Attributes** tab of the *Part* dialog, set the **Footprint** to AXIAL0.4, the **Designator** to R1, and the **Part Type** to 100k.
- Click **OK** to close the dialog and return to part placement mode.
- 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.
- Next place the other 100k resistor R2 above the base of Q2 (the designator will automatically increment when you place the second resistor).
- The remaining two resistors, R3 and R4, have a value of 1k, so press the `TAB` key to call up the *Part* dialog and change the **Part Type** field to `1k`, then click **OK** to close the dialog.
- Position and place R3 and R4 as shown in the schematic diagram in Figure 1.

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

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

Now place the two capacitors:

- The capacitor part is also in the **Simulation Symbols.Lib** library, which should already be selected in the **Browse Sch** panel.
- Type `*cap*` in the **Filter** in the panel and press the `ENTER` key.
- 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.
- Press the `TAB` key to edit the capacitor's attributes. In the **Attributes** tab of the *Part* dialog, set the **Footprint** to `RAD0.1`, the **Designator** to `C1`, and the **Part Type** to `20n`.
- Click **OK** to close the dialog and return to part placement mode.
- Press the `SPACEBAR` to rotate the capacitor by 90° so it is in the correct orientation.
- Position and place the two capacitors in the same way that you placed the previous parts.
- Right-click or press `ESC` to exit placement mode.

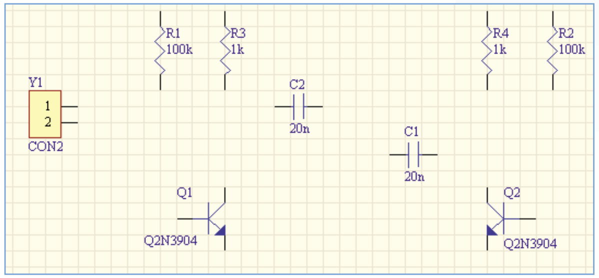
When you are in any editing or placement mode (a cross hair cursor is active), moving the cursor to the edge of the document window will automatically pan the document. If you accidentally pan too far while you are wiring up your circuit, press the `V,F` key sequence (equivalent to the **View » Fit All Objects** menu item) to redraw the schematic window, showing all placed objects. This can be done even when you are in the middle of placing an object.

The last component to be placed is the connector. Connectors are stored in the Miscellaneous Devices.ddb library database.

- Using the steps described earlier, add the Miscellaneous Devices.ddb library database to the current library list.
- Select the **Miscellaneous Devices.lib** library from the list of libraries in the **Browse Sch** panel.
- The connector we want is a two-pin socket, set the **Filter** to `*con2*` and press `ENTER`.
- Select **CON2** from the parts list and click the **Place** button. Press `TAB` to edit the attributes and set **Footprint** to FLY4 and **Designator** to Y1. Click **OK** to close the dialog.
- Before placing the connector, press `X` to flip it horizontally so that it is in the correct orientation. Place the connector on the schematic.
- Right-click or press `ESC` to exit part placement mode.
- 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, 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:

> You can start the tutorial at this point by opening the schematic *Multivibrator placed.sch* located in the download design database *Multivibrator tutorial.ddb*

- ➤ To make sure you have a good view of the schematic sheet, select **View » Fit All Objects** from the menus [shortcut V, F].
- ➤ 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.
- ➤ Position the cursor over the bottom end of R1. When you are in the right position a circle will appear at the cursor location. This indicates that the cursor is over an electrical connection point on the component.
- ➤ 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.
- ➤ 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.
- ➤ Position the cursor over the base of Q1 until you see the connection circle. Left-click or press ENTER to connect the wire to the base of Q1.
- ➤ 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).
- ➤ 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.
- ➤ Move the cursor horizontally till it is directly over the wire connecting the base of Q1 to R1. A connection circle will appear.
- ➤ 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.
- ➤ Wire up the rest of your circuit, as shown in Figure 3.
- ➤ When you have finished placing all the wires, right-click or press ESC to exit placement mode. The cursor will revert to an arrow.

> 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 left-click once.
>
> Small "editing handles" will appear at each vertex (corner). In Protel 99 SE we call this "focusing" an object.
>
> Click-and-drag an editing handle to change the shape of the object.

> Whenever a wire runs across the connection point of a component, or is terminated on another wire, Protel 99 SE will automatically add 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.

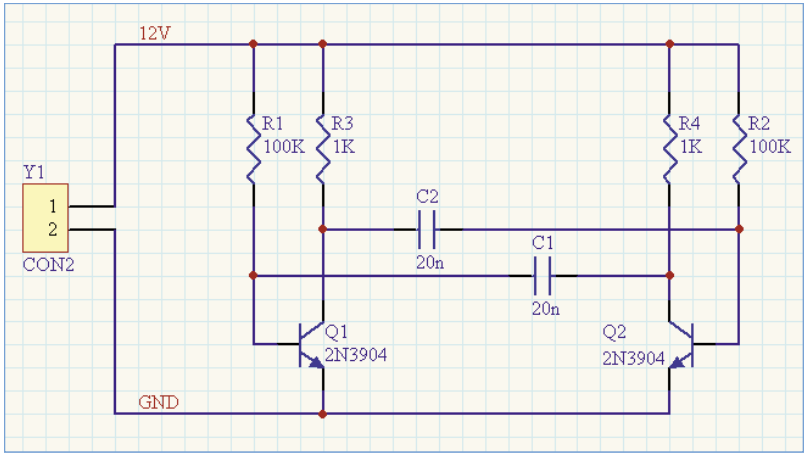


Figure 3
The fully wired schematic

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 2 power nets complete the following steps:

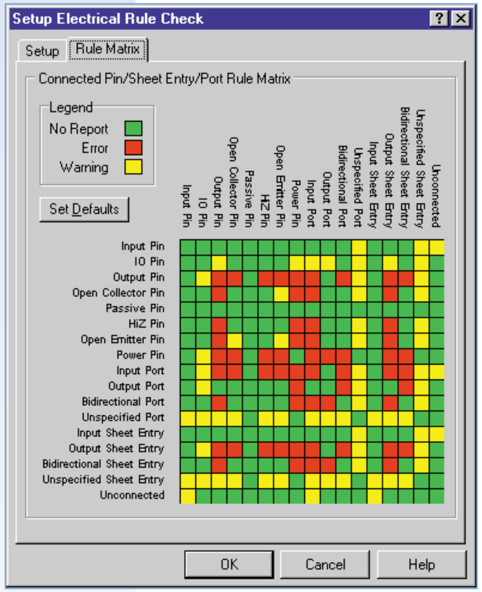
- ➤ Select **Place » Net Label** from the menus. A dotted box will appear floating on the cursor.
- ➤ To edit the net label before it is placed press the TAB key to display the *Net Label* dialog.
- ➤ Type 12V in the **Net** field, then click **OK** to close the dialog.
- ➤ Place the net label such that the bottom left of the net label touches the upper most wire on the schematic, as shown in Figure 3.
- ➤ After placing the first net label you will still be in net label placement mode, press the TAB key again to edit the second net label before placing it.
- ➤ Type GND in the **Net** field, click **OK** to close the dialog, then place the net label as shown in Figure 3.
- ➤ Select **File » Save** [shortcut F, S] to save your circuit.

> 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.

Congratulations! You have just completed your first schematic capture. Before we turn this schematic into a circuit board, let's take a look at some of the features Protel 99 SE includes to help you check your schematic design.

Checking the electrical properties of your schematic

Schematic diagrams in Protel 99 SE are more than just simple drawings – they contain electrical connectivity information about the circuit. You can use this connectivity awareness to verify your design. To do this, perform an Electrical Rules Check (ERC) by completing the following steps:



- ➤ Select **Tools » ERC** [shortcut T, E] from the menus. This will open the *Setup Electrical Rule Check* dialog.
- ➤ The **Setup** tab of this dialog includes checks for drafting-type errors. Leave the settings on this tab at their default values.
- ➤ Click on the **Rule Matrix** tab. This controls the electrical characteristics of the ERC.

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 left 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 red indicating that an Output Pin connected to an Open Collector Pin on your schematic will generate an error condition when the ERC is run. 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 ERC will detect unconnected input pins:

- ➤ Look down the row labels to find **Input Pin**. Look across the column labels to find **Unconnected**. The square where these entries intersect indicates the error condition when an Input Pin is found to be Unconnected in the schematic. The default is a yellow square, which indicates that a warning will be generated for unconnected input pins.
- ➤ Click the **OK** button to run the ERC. A text window will open with an ERC report for your circuit.

If your circuit is drawn correctly, the ERC should be blank. If the report gives errors, check your circuit and ensure all wiring and connections are correct. If you can't find the error in your schematic, try opening the **Multivibrator.Sch** file in the **Tutorial_1.ddb** download database and repeating the above steps.

We will now deliberately introduce an error into our circuit and re-run the ERC:

- ➤ Click on the **Multivibrator.sch** tab at the top of the window to make the schematic sheet the active document.
- ➤ 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 to indicate that it is focused). Press the DELETE key to delete the wire.
- ➤ Re-run the ERC by selecting the **Tools » ERC** menu item [shortcut T, E] and then clicking the **OK** button in the *Setup Electrical Rule Check* dialog.

A text window with the new ERC report 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, there is a special option to check for floating input pins in the **Setup** tab of the ERC dialog.

- ➤ To view the ERC report and the schematic sheet together, right-mouse click on the **Multivibrator.ERC** tab at the top of the ERC text window, then select **Split Horizontal** from the floating menu. The design window will split into 2 regions, the upper region displaying the schematic, the lower region the ERC report.
- ➤ Click on the schematic sheet to make it active and then select **View » Fit All Objects** from the menus [shortcut V, F] to resize your schematic to fit the new window size. Note the crossed red circle on Q1 indicating the error.

When an ERC detects a number of errors in a circuit, you can use Protel 99 SE's "Cross Probe" feature to easily find particular errors. While only one error is present in our circuit, you can see how the Cross Probe feature works by completing the following steps:

- ➤ Click on the tab of the ERC report to make it active.
- ➤ The error information includes designator and pin details (eg Q1-2), and location details (eg @430,390). Double-click on one of the location details to select it.
- ➤ Click on the **Cross Probe** tool in the main toolbar. The corresponding error will be centered in the schematic window, and the cursor will point to the location.

> Cross Probing works between many different types of documents in Protel 99 SE. For example, you can cross probe from a schematic component to the corresponding footprint in a PCB design. The Cross Probe feature is a powerful tool when working in complex designs.

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

- ➤ Click on the tab of the schematic sheet to make it active.
- ➤ Select **Edit » Undo** from the menus [shortcut E, U]. The wire you deleted previously should now be restored.
- ➤ To check that the undo was successful, re-run the ERC by selecting **Tools » ERC** from the menus [shortcut T, E] and clicking **OK** in the resulting dialog. The ERC report should show no errors.
- ➤ Select **File » Close** from the menus to close the error report window, then select **View » Fit All Objects** [shortcut V, F] from the menus to restore your schematic view.

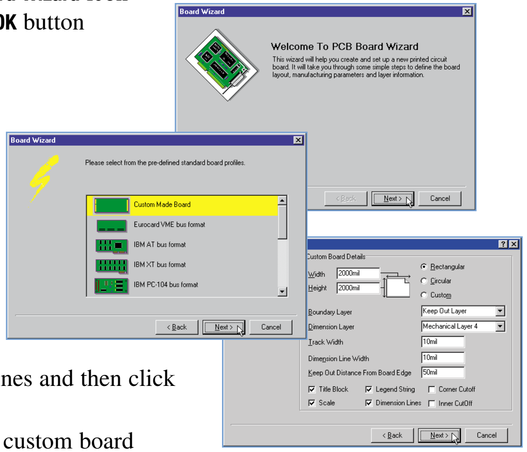
Congratulations! You are ready to transfer the design to PCB layout.

Designing the PCB

Creating a new PCB file

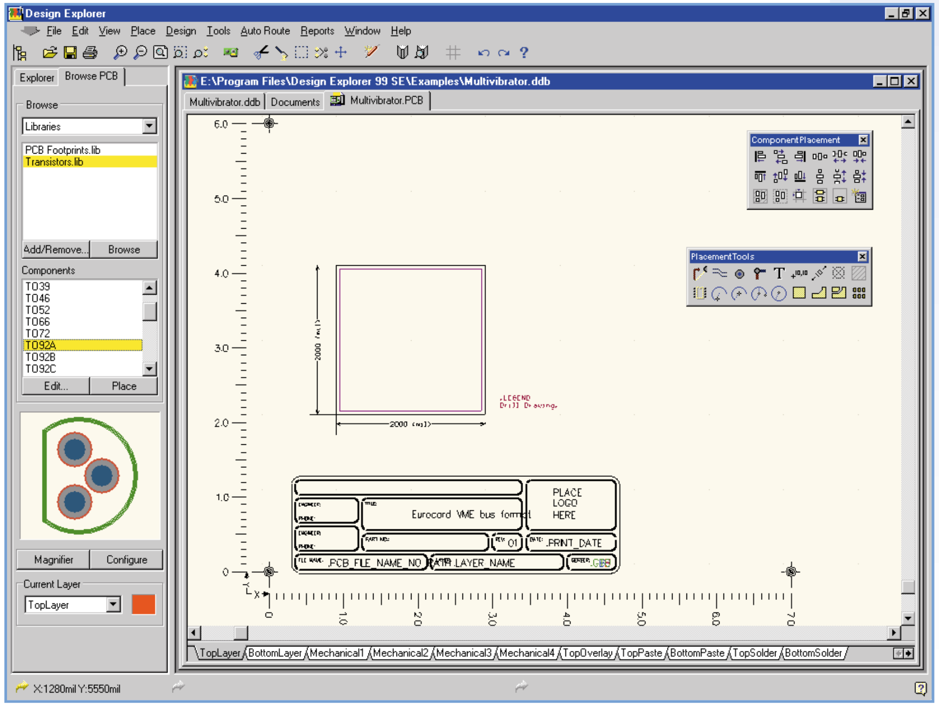
Before you transfer the design from the schematic editor to the PCB editor you need to create the "blank PCB". The easiest way to create a new PCB design in Protel 99 SE is to use the PCBMaker Wizard, which allows you to choose from over 60 industry-standard board outlines as well as create your own custom board sizes. To create a new PCB using the PCBMaker Wizard, complete the following steps:

- Open the **Multivibrator.ddb** design database, then click on the **Documents** folder in the navigation tree on the left.
- Select **File » New** from the menus [shortcut F, N], then click on the **Wizards** tab in the *New Document* dialog.
- Click the **Printed Circuit Board Wizard** icon to select it, then click the **OK** button to start the Wizard.



- The first screen you see is the introduction page. Click the **Next** button to continue.
- The second page allows you to select the board outline you wish to use. For this tutorial we will enter our own board size. Select **Custom Made Board** from the list of board outlines and then click the **Next** button.
- 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. Type 2000 in both the **Width** and **Height** fields. Also, turn off the **Corner Cutoff** and **Inner Cutoff** options. Leave the other fields at their default values. Click the **Next** button to continue. Note that the default units for this dialog are mils: 1000mil = 1inch.
- The next page displays the board outline, you can adjust the sizes if necessary. We will leave the options on this page as they are, click the **Next** button to continue.
- The next page allows you to select the physical makeup of the board. Leave the options on this screen set to their defaults. Click the **Next** button to continue.
- The next page allows you to select the types of vias used in the design. Leave the options on this screen set to their defaults. Click the **Next** button to continue.
- The next page allows you to set the routing options. Select the **Thru-hole components** option, and set the number of tracks between adjacent pads to **One Track**. Click the **Next** button to continue.
- 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.
- The next 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 press the **Next** button to continue.
- The PCB Wizard has now collected all the information it needs to create your new board. Click **Finish** to close the Wizard. The PCB document editor will open with your new board outline.
- Save the document by selecting **File » Save** from the menus, then close it by selecting **File » Close**.
- The new board has been created with the name PCB1.PCB. To rename the PCB click on the **Documents** folder in the navigation tree, then click once on the **PCB1.PCB** icon displayed on the right to select it. Select **Edit » Rename** from the menus, type in the new name, Multivibrator.pcb, and press ENTER.
- Double-click on the **Multivibrator.pcb** icon to open the PCB, ready to start the board design.

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



The PCB workspace with the custom board outline

Adding the PCB Footprint libraries

Recall that when we placed the components on our schematic, we typed entries in the **Footprint** field of each component's attributes dialog. Before Protel 99 SE knows how to handle the various schematic components you have used in your circuit, it needs a representation or "footprint" for each part. The text we typed into the footprint fields indicates which footprint to use in the PCB design. Protel 99 SE comes supplied with over 35 PCB footprint libraries, including a number of IPC standard libraries. Before we begin our PCB design we need to make sure the appropriate footprint libraries are available.

To add the necessary footprint libraries, complete the following steps:

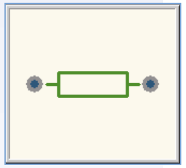
➤ Select **Design » Add/Remove Library** [shortcut D, L] from the menus to open the *PCB Libraries* dialog.

➤ The footprint library databases are located in 3 sub-folders in the \Program Files\Design Explorer 99 SE\Library\PCB\ folder. Navigate to the \Generic Footprints\ folder to display a list of available generic library databases. For our tutorial we want two library databases: **Advpcb.ddb**, which includes a number of general axial and radial component footprints, and **Transistors.ddb**, which contains transistor footprints. Find and select each of these libraries in turn and press the **Add** button in the *PCB Libraries* dialog to add them to the **Selected Files** list. When you have added both libraries, click the **OK** button to close the dialog.

➤ To check that the correct libraries are available, select **Libraries** from the drop down list in the **Browse PCB** Editor Panel. **PCB Footprints.lib** and **Transistors.lib** should both appear in the list.

➤ Remember that we set the resistors in our schematic to have a footprint entry of AXIAL0.4. To view this footprint, click on **PCB Footprints.lib** in the library list in the **Browse PCB** Panel. Now scroll down the **Components** list until you see the AXIAL0.4 entry. Click on the entry to make it active. A "thumbnail" of the footprint will appear in the MiniViewer window in the PCB Editor Panel, as shown in Figure 4.

Figure 4
Use the MiniViewer to preview the component footprints



We now have all the PCB footprint libraries loaded. Let's turn our attention to loading our circuit design.

Transferring the design

Transferring the design from the schematic editor to the PCB editor is easy in Protel 99 SE. To transfer the design:

➤ Click on the **Multivibrator.sch** tab at the top of the design window to make the schematic the active document.

➤ Select **Design » Update PCB** from the menus.

The *Update Design* dialog will appear. Uncheck the 2 **Classes** options at the bottom of the **Synchronization** tab, leave all the other settings at their defaults.

➤ Click on the **Preview Changes** button at the bottom of the dialog to examine what changes will be carried out. The **Changes** tab should list 35 macros, detailing what components, nets and net nodes will be added to the PCB.

A macro is created for each design change that needs to be performed to get the contents of the PCB file to correspond to the schematic. Any macro that has an error listed will not be executed. Click on the **Help** button for information on resolving macro errors.

➤ Click the **Execute** button to close the dialog and execute the netlist macros.

➤ Once the design transfer is complete, click on the **Multivibrator.pcb** tab at the top of the window to display the board.

The components in your design will appear to the right of the board outline. Before we start positioning the components on the board we need to set up the PCB workspace.

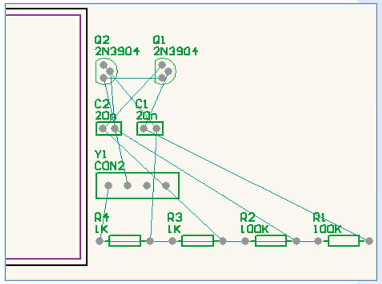


Figure 5
The components next to the board, ready for positioning

Setting up the PCB workspace

Before we start positioning the components we need to ensure that our placement grid is set correctly. 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. 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:

You can start the tutorial at this point by opening the PCB *multivibrator components.pcb* in the download design database *Multivibrator tutorial.ddb*

➤ Select **Design » Options** from the menus [shortcut D, O] to open the *Document Options* dialog, and click on the **Options** tab to make it active.

➤ 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 *Document Options* dialog.

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

➤ Select **Tools » Preferences** from the menus [shortcut T, P] to open the *PCB Preferences* dialog. In the **Editing Options** section of the **Options** tab, ensure 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.

➤ Click the **Display** tab in the *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, then close the *Preferences* dialog.

Defining the layer stack and other non-electrical layers

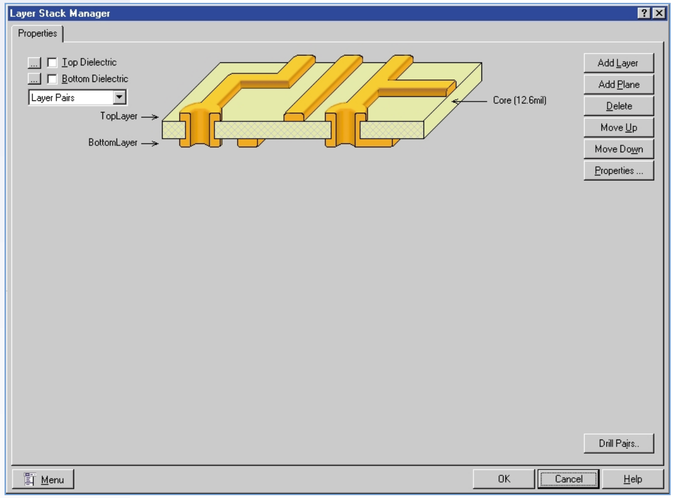
If you look at the bottom of the PCB workspace you will notice a series of layer tabs. The PCB Editor is a multi-layered environment, and most of the editing actions you perform will be on a particular layer.

There are 3 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. Select **Design » Mechanical Layers** to display the *Setup Mechanical Layers* dialog, where you can add, remove and name mechanical layers.

Special layers – these include the top and bottom silkscreen layers, the solder and paste mask layers, drill layers, the keepout 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 *Document Options* dialog (**Design » Options**).



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

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:

- Press the L shortcut key to display the **Layers** tab of the *Document Options* dialog.
- Click the **Used On** button to disable all layers except those that have something on them.
- Disable the 4 **Mask** layers and the **Drill Drawing** layer, then click **OK** to close the dialog.

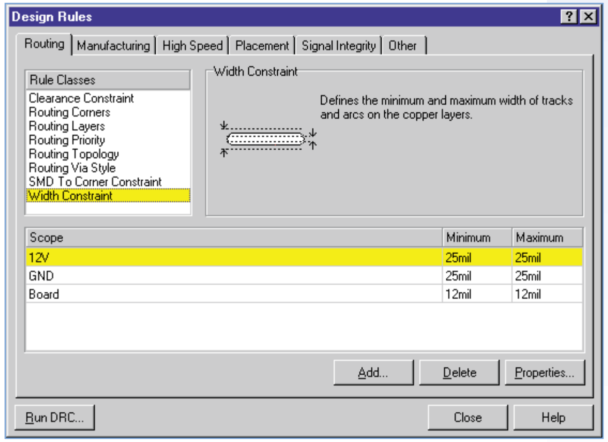
Setting up new design rules

The Protel 99 SE's PCB editor is called a "rules-driven environment", but exactly what does this mean? 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.

There are 48 design rules, divided into 6 categories. These rules cover routing, manufacturing, high speed design, 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:

- With the PCB as the active document, select **Design » Rules** from the menus.
- The *Design Rules* dialog will appear. Each category of rules is displayed on a separate tab. Click on the **Routing** tab to make it active.
- Click once on each rule in the **Rules Classes** list to select it. As you click on each rule, the lower half of the dialog displays the rules of that kind that are already set up. These rules are either defaults, or have been set up by the Board Wizard when the new PCB document was created.
- Click on the **Width Constraint** rule in the **Rule Classes** list. One rule will be displayed in the lower half of the dialog.



Rules are displayed in their order of priority

One of the powerful features of Protel 99 SE'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).

Currently there is one width constraint rule for your design, which applies to the whole board. We will now add 2 new width constraint rules, one for the 12V net and another for the GND net.

To add new width constraint rules complete the following steps:

➤ Click the **Add** button to add a new width constraint rule. Like all the rule definition dialogs, there are 2 regions to the *Max-Min Width Rule* dialog. On the left of the dialog you define the scope (what you want this rule to target), and on the right of the dialog you define the rule attributes.

➤ Let's set this rule up to target the 12V net. Select **Net** from the **Filter Kind** list. When you do a new field titled **Net** will appear, select **12V** from this list.

➤ In the **Rule Attributes** section of the dialog change the **Minimum Width, Maximum Width** and **Preferred Width** fields to 25mil, then click **OK** to close the dialog.

➤ The lower region of the *Design Rules* dialog now includes the new width constraint design rule.

➤ Now add another new width constraint design rule for the GND net.

➤ Double-click to edit the original Board scope width rule, and confirm that the **Minimum Width, Maximum Width** and **Preferred Width** fields are all set to 12mil.

➤ Close the *Design Rules* dialog.

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

Positioning the components on the PCB

Before we start positioning the components on the PCB let's change the view to zoom in to the board:

➤ Press the V, F shortcut keys to zoom in on the board and components.

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.

The thicker green/red line is a placement force vector, it indicates better/worse placement locations.

➤ 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.

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

➤ 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 6.

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

➤ Reposition the remaining components, using Figure 6 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 6. Don't forget to re-optimize the connection lines as you position each component.

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 99 SE'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.

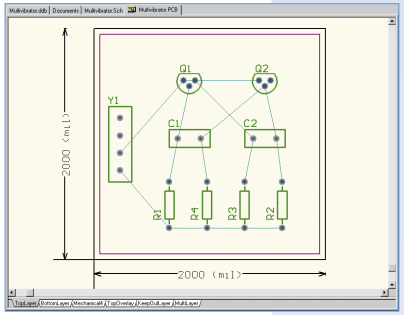
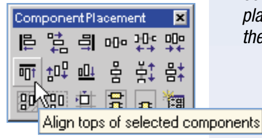


Figure 6 Components placed on the PCB

Protel 99 SE also includes powerful interactive placement tools, let's use these to ensure that the 4 resistors are correctly aligned and spaced.

➤ Holding the SHIFT key, left-click on each of the 4 resistors to select them. Each resistor's color will change to the selection color (the default is yellow).

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



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

➤ Press the X, A shortcut keys to deselect all the resistors.

The four resistors are now aligned and equally spaced.

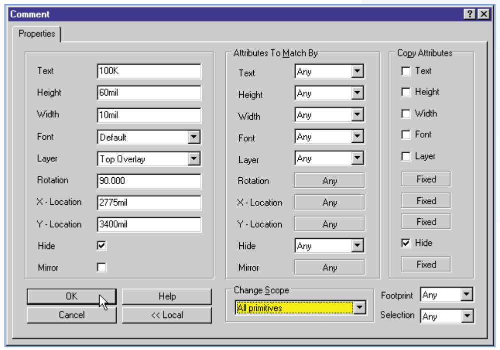
Global editing

Both the schematic and the PCB editors include a powerful editing aid – referred to as global editing. Global editing is the ability to apply the same change that you are currently making, to other objects in the workspace. Let's do an example to demonstrate it.

You have finished positioning the components and you decide that you do not want the component comments to be displayed. To hide all the component comments complete the following steps:

➤ Double-click on one of the comment strings to open its *Comment* dialog, then click to enable the **Hide** option.

➤ To apply this change globally to all components, click on the **Global** button. This displays the global editing options for the dialog.



There are now 3 sections to the dialog; the original attributes on the left, the **Attributes to Match By** in the center, and the **Copy Attributes** on the right.

The options in the **Attributes to Match By** section define which other comment strings you want this change to apply to. For example, if you only needed to change the attributes of a few of the comment strings you could select those strings first, then during the global edit you would match by **Selection**, targeting only the required comment strings.

➤ In this case we want the change to apply to all the comment strings so leave the **Attributes to Match By** options unchanged.

- The options in the **Copy Attributes** section define what changes will be copied to each of the matched objects. Ensure that the **Hide** option in the **Copy Attributes** section has a tick next to it.
- Set the **Change Scope** at the bottom of the dialog to **All primitives** (free primitives are those primitives that are not part of a component).
- Click the **OK** button to close the dialog. A confirmation message will appear asking if you want to make 8 changes. Click **Yes** to apply the changes.

The comment string for each component will disappear. You can unhide a component comment by double-clicking on the component.

- Now reposition the component designator strings.
- When you have finished repositioning the component designators save your board by selecting **File » Save** from the menus [shortcut `F`, `S`].

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

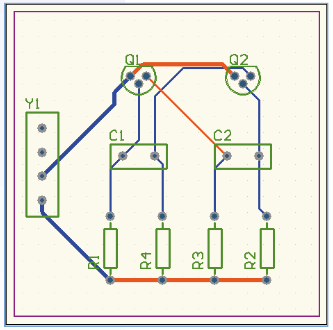
Automatically routing the board

You can start the tutorial at this point by opening the PCB *multivibrator placed.pcb* in the download design database *Multivibrator tutorial.ddb*

Routing is the process of laying tracks and vias on the board to connect the components. Protel 99 SE makes the job of routing easy by providing a number of sophisticated manual routing tools as well as a powerful and easy-to-use shape-based autorouter, which optimally routes the whole or part of a board at the touch of a button. To see how easy it is to autoroute with Protel 99 SE, complete the following steps:

- Select **Autoroute » All** from the menus [shortcut `A`, `A`]. The *Autorouter Setup* dialog will appear, click the **Route All** button to route the board.
- When the autorouter has finished, press the `END` key to redraw the screen.

Figure 7
Fully autorouted board



It's as simple as that! Protel 99 SE's autorouter provides results comparable with that of an experienced board designer. And because Protel 99 SE routes your board directly in the PCB window, there is no need to wrestle with exporting and importing route files.

- 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 board Wizard. You will also notice that the 2 power net tracks running from the connector are wider, as specified by the 2 new **Width Constraint** 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 the routing be.

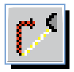
Manually routing the board

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:

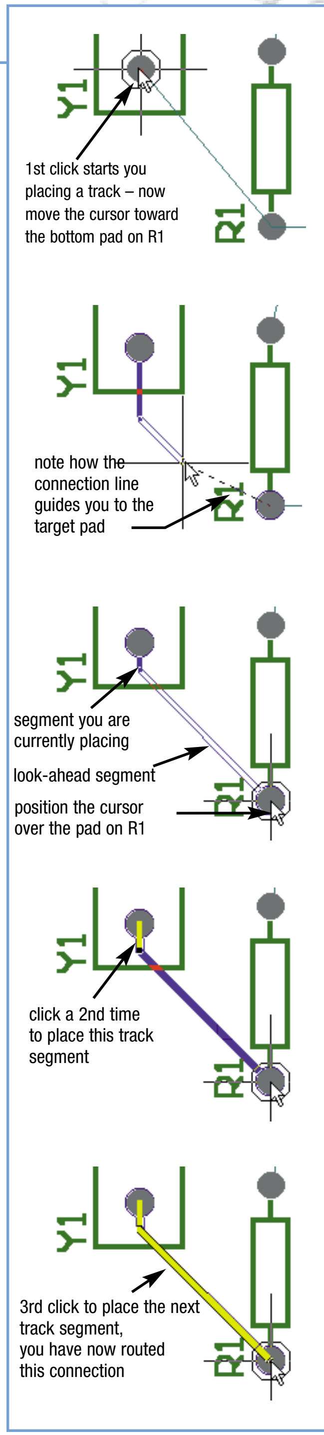
- To remove the tracks placed by the autorouter, select **Tools » Un-Route » All** from the menus [shortcut `U`, `A`]. The board will be completely un-routed.

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

In Protel 99 SE, 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 99 SE 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).



- 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.
- 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.
- 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.
- 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.

➤ 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 yellow, indicating that it has been placed. Move the cursor around a little and you will see that you still have two segments 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.

➤ 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. It will turn yellow, indicating it has been placed.

You have just routed the first connection.

➤ 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. It will now turn yellow.

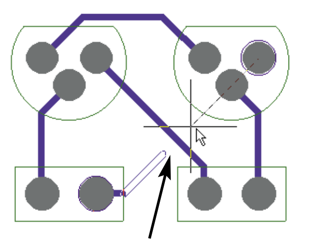
➤ Now move the cursor to the bottom pad of resistor R3. Note that this segment is not solid blue, but drawn in outline indicating it is a look-ahead segment. This is because each time you place a track segment the mode toggles between starting in a horizontal/vertical direction and starting at 45°. Currently 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.

➤ 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.

➤ 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.

➤ You can now route the rest of the board in a similar manner to that described in the previous steps. Figure 8 shows the manually routed board.

➤ Save the design.



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.

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 yellow.
- 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 "un-place" 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 99 SE 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 focus it. The segment's "editing handles" will appear (the rest of the track will be highlighted in yellow). Press the DELETE key to clear the focused track segment.
- Re-routing is easy in Protel 99 SE – 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.

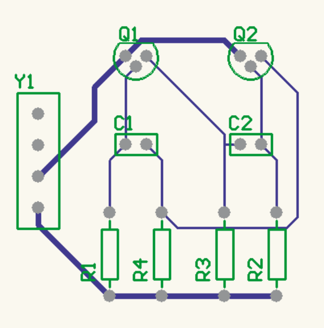


Figure 8 *Manually routed board, with tracks placed on the bottom layer*

Congratulations! You have manually routed your board design.

Because we originally defined our board as being double-sided in the PCBMaker 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 99 SE will automatically insert vias if necessary when you change layers.

Verifying your board design

Protel 99 SE 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, then verify that the design complies with the rules at the end of the design process.

You can start the tutorial at this point by opening the PCB *multivibrator routed.pcb* in the download design database *Multivibrator tutorial.ddb*

Earlier in the tutorial we examined the routing design rules and added 2 new width constraint rules. We also noted that there were already a number of rules that had been created by the board creation wizard.

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

➤ Choose **Design » Options** from the menus [shortcut D, O], and if necessary click the **Layers** tab to make it active.

You can also use the PCB editor panel to find design rule violations. Set the **Browse** mode at the top of the panel to **Violations**, then use the **Details, Highlight** and **Jump** buttons to find and analyze the design rule errors.

To make it easier to see the design rule violations you can display the primitives in draft mode. Select **Tools » Preferences**, click on the **Show/Hide** tab, then click the **All Draft** button. When you close the dialog all primitives will be shown in draft mode, except those that are marked with a DRC error marker. Click the **All Final** button in the **Show/Hide** tab to display the primitives as solids again.

➤ In the **System** section of this dialog, ensure that the **DRC Errors** option is checked.

➤ Click the **OK** button to close the dialog.

➤ 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 Check* dialog. Leave all options at their defaults and click the **Run DRC** button.

The DRC will run and the results will be displayed in a text file. Look through this DRC report file. It lists each active design rule, and 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. If you switch to your PCB document, you will see that the transistor pads are highlighted in green, indicating a design rule violation.

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.

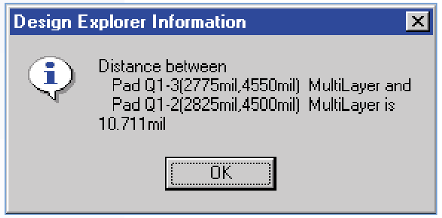
To find out the actual clearance between the transistor pads:

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

➤ Select **Reports » Measure Primitives** from the menus [shortcut R, P]. The cursor will change to a cross hair and you will be prompted on the status bar to "Choose First Primitive".

➤ 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.

➤ 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 edge of the two pads is 10.711mil.

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

Let's look at the current clearance design rules.

➤ Select **Design » Rules** from the menus [shortcut D, R] to open the *Design Rules* dialog. Click on the **Routing** tab to make it active, then in the **Rule Classes** list click once on the **Clearance Constraint** to select 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.

➤ Click the ADD button to add a new clearance constraint rule.

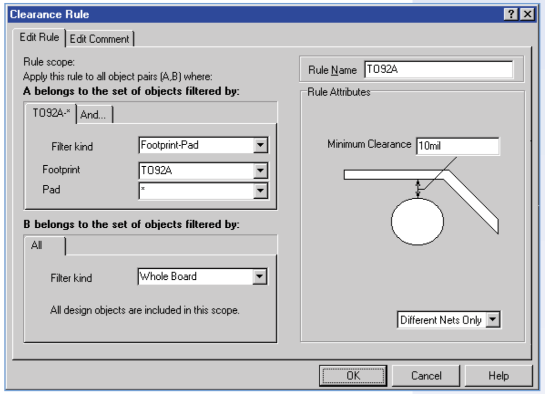
➤ In the **Rule Attributes** section of the resulting dialog, set the **Minimum Clearance** to 10mil.

➤ In the **Rule Scope** section, use the drop-down list to change the first **Scope Kind** field to Footprint-Pad.

➤ Select **TO92A** in the **Footprint** field.

➤ Type an * (asterisk) in the **Pad** field, indicating "any pad" on the TO92A.

➤ Click **OK** to close the dialog.



The new rule will appear in the lower region of the *Design Rules* dialog.

➤ You can now re-run the DRC from the *Design Rules* dialog by clicking the **Run DRC** button, then clicking the **Run DRC** button in the *Design Rules Check* dialog.

A new report will be generated – this report should show no rule violations.

You have completed the PCB layout, and are ready to produce the output documents.

```
Protel Design System Design Rule Check
PCB File : Documents\Multivibrator.PCB
Date     : 10-Jan-2000
Time     : 15:47:33

Processing Rule : Broken-Net Constraint ( (On the board ) )
Rule Violations :0

Processing Rule : Short-Circuit Constraint (Allowed=Not Allowed) (On the board ),(On the board
Rule Violations :0

Processing Rule : Broken-Net Constraint ( (On the board ) )
Rule Violations :0

Processing Rule : Width Constraint (Min=12mil) (Max=12mil) (Prefered=12mil) (On the board )
Rule Violations :0

Processing Rule : Clearance Constraint (Gap=13mil) (On the board ),(On the board )
Rule Violations :0

Processing Rule : Width Constraint (Min=25mil) (Max=25mil) (Prefered=25mil) (Is on net 12V )
Rule Violations :0

Processing Rule : Width Constraint (Min=25mil) (Max=25mil) (Prefered=25mil) (Is on net GND )
Rule Violations :0

Processing Rule : Clearance Constraint (Gap=10mil) (pad * in footprint TO92A ),(On the board )
Rule Violations :0


Violations Detected : 0
Time Elapsed        : 00:00:01
```

Protel 99 SE supports fully-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.

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 99 SE includes a sophisticated printing engine, called Power Print, that gives you complete control over the printing process. Using Power Print you first preview the drawings (called printouts), where you can define precisely what mix of PCB layers you want to print, set the scaling and orientation, and see exactly how it will look on the page before you print it.

Print/preview setups are stored as .PPC documents in your design – they can be renamed, reopened at any time, and copied from one design to another.

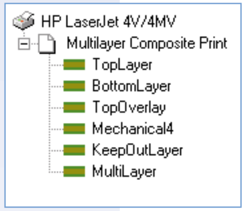
To create a print preview document:

➤ Select **File » Print/Preview** from the PCB menus.

The PCB will be analyzed, and a default printout displayed in a new PPC window. The default composite preview includes 1 printout.

➤ To examine the set of PCB layers that are included in the printout, click on the **Browse PCBPrint** tab at the top of the panel to display the printout set, then click on the small + symbol to expand the Multilayer Composite Print, as shown in Figure 9.

Figure 9 *Default composite printout, includes all the used layers on the board*



A print preview document (.PPC) can include any number of printouts, and each printout can include any combination of layers, overlaid in any order you require.

➤ To change the default composite printout to a composite drill guide, select **Tools » Create Composite Drill Guide** from the menus, clicking **Yes** in the *Confirm Create Print-Set* dialog.

The PCB is re-analyzed, and a new printout displayed.

➤ Click on the small + symbol next to the **Combination Drill Guide** printout in the **Browse PCBPrint** panel to display the layers in the printout.

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, to remove it right-click on the **DrillGuide** layer in the panel and select **Delete** from the floating menu.

Note the printer icon at the top of the **Browse PCBPrint** panel – the printouts will target your default windows printer.

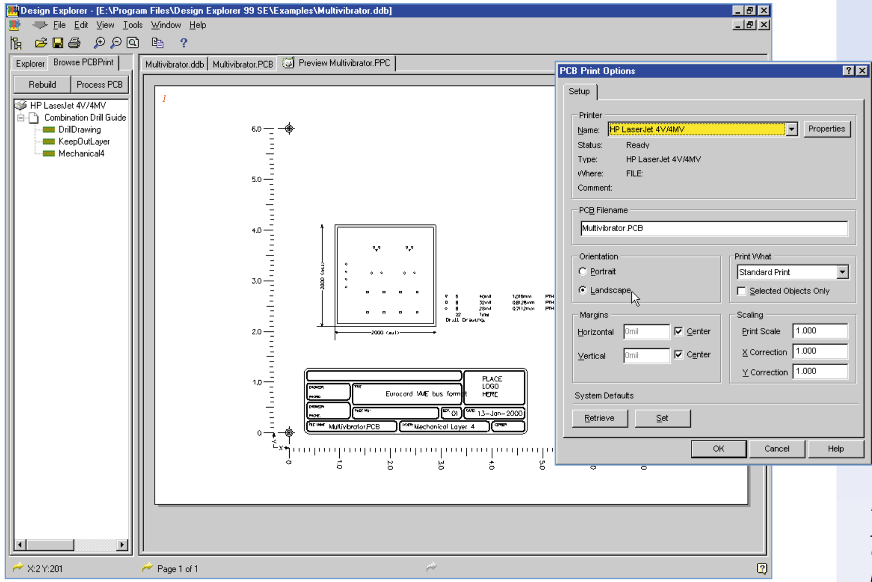
Select **Edit » Copy** from the menus to copy the current printout to the clipboard and paste it into another Windows application.

Select **File » Export** to export the printouts to the hard disk as WMF or EMF files.

➤ To change the target printer, and set the page orientation and scaling, select **File » Setup Printer** from the menus – then choose your preferred printer and set the **Orientation** to landscape in the *PCB Print Options* dialog.

The preview will be re-analyzed and displayed, ready for printing.

➤ Select **File » Print All** from the menus to print the drill drawing. For information on the various printing options refer to the **Print Options** menu item in the **Help** menu.



Power Print gives you complete control over the printing process

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.

All of these files are configured in and produced by Protel 99 SE's CAM Manager. The setups for the manufacturing documents are stored in a .CAM document in the design database.

➤ To create the manufacturing files for the tutorial PCB, make the PCB the active document, then select **File » CAM Manager** from the PCB Editor menus.

A new CAM document is created, and the Output Wizard launched.

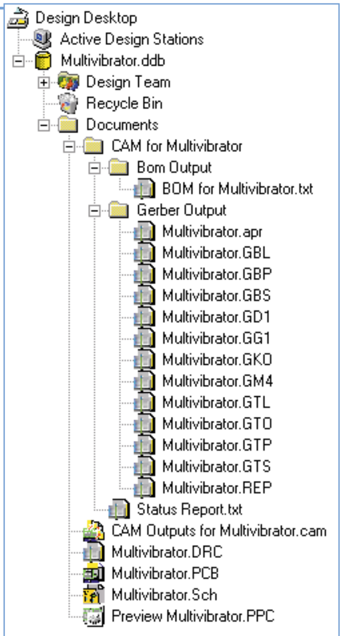
➤ The Wizard can be used to create each of the supported output file types, click **Next** to display the list of available types.

➤ Select **Gerber** from the list and click **Next** for each page of the Wizard, leaving the options on each page at the defaults, then **Finish** on the last page.

➤ Once the Wizard closes, the CAM document will display one CAM output setup, called **Gerber Output 1**. The setup of the Gerber files can be changed at any time, double-click on the name **Gerber Output 1** to display the *Gerber Setup* dialog.

Each Gerber file corresponds to 1 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.

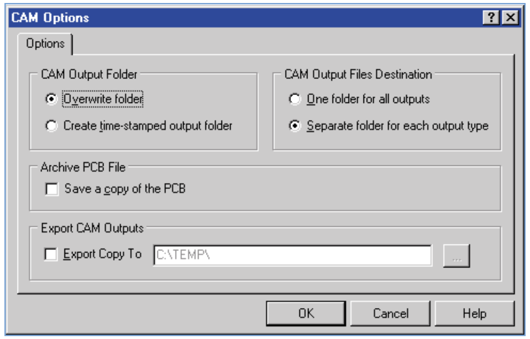
- ➤ To include a Bill of Materials in the Output setups, right-click in the CAM document and select **Insert Bill of Materials** from the floating menu.
- ➤ In the *Bill of Materials Setup* dialog enable the **Text** format, disable the other 2 formats, and click **OK** to close the dialog. Note that 3 formats are supported, **Text**, **CSV** (Comma Separated Values), which loads straight into a spreadsheet editor or can be imported into a database, and **Spreadsheet**, which automatically opens in Protel 99 SE's spreadsheet editor.



The output documents have been created and stored in folders in the design database.

You are now ready to generate the Gerber and BOM output documents. Before you do, we will configure the generation options.

- ➤ Select **Tools » Preferences** from the menus to display the *CAM Options* dialog. Note that you can automatically export the documents to a disk drive.
- ➤ Enable the **Separate Folder for Each Output Type** option and click **OK** to close the dialog.
- ➤ Select **Tools » Generate CAM Files** from the menus to create the Gerber and BOM files.
- ➤ When the generation process is complete a new folder will appear in the navigation tree in the **Explorer** panel, called **CAM for Multivibrator**.



- ➤ Click on the small + symbol to expand the view and display the **Bom Output** and **Gerber Output** sub-folders, then click on the small + symbol next to each to display the contents of each folder.
- ➤ Click on the **BOM for Multivibrator.txt** icon to open the Bill of Materials.

If required, the Gerber files could be exported from the design and shipped to the PCB manufacturer.

Congratulations! You have completed the PCB design process.

Simulating the design

Protel 99 SE 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

You can start the tutorial at this point by opening the schematic *multivibrator sim1.sch* in the download design database *Multivibrator tutorial.ddb*

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.

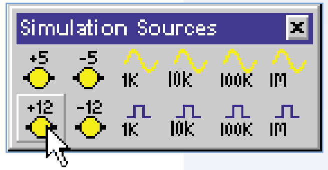
- ➤ Click on the **Multivibrator.sch** tab at the top of the window to make the schematic the active document.

The first step is to replace the connector with a voltage source.

- ➤ To delete the connector, click once on the body of the connector to focus it (a dotted focus box will appear around the connector), then press the `DELETE` key on the keyboard.

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 focus 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.
- ➤ Repeat this process for the dangling end of the GND wire, moving it toward the bottom of the sheet.
- ➤ Select **View » Toolbars » Simulation Sources** from the menus to display the **Simulation Sources** toolbar.



- ➤ 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. Leave the **Footprint** field blank. Click the **OK** button to close the dialog and then place the source between the dangling ends of the 12V and GND wires.
- ➤ 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 10.

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 – ensure that this corner touches the wire.

- ➤ Select **Place » Net Label** from the menus [shortcut `P, N`]. Press the `TAB` key to edit the net label attributes. In the resulting dialog, set the **Net** field to `Q1B` and close the dialog.
- ➤ Position the cursor over the wire coming from the base of Q1. Left-click or press `ENTER` to place the net label on the wire.

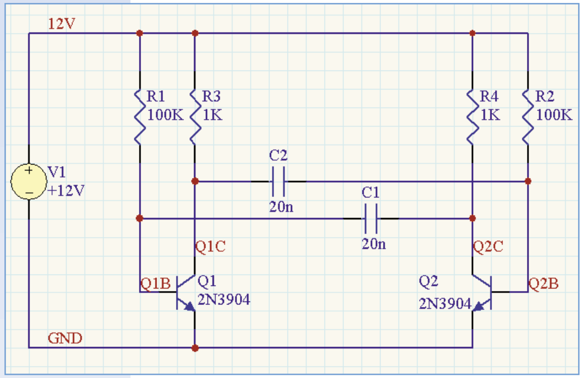


Figure 10
Simulation-ready schematic

- ➤ Press the TAB key and change the **Net** field to Q1C.
- ➤ Position the cursor over the wire coming from the collector of Q1 and left-click to place the second net label.
- ➤ Similarly place net labels with designators Q2B and Q2C on the base and collector wires of Q2 respectively.
- ➤ When you have finished placing the net labels, right-click or press the ESC key to exit placement mode.
- ➤ 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.sch in the *Save As* dialog.

Running a transient analysis

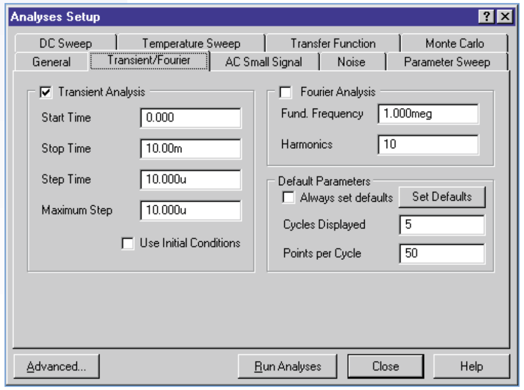
You can start the tutorial at this point by opening the schematic *multivibrator sim2.sch* in the download design database *Multivibrator tutorial.ddb*

Your schematic now has all the necessary additions, so let's set up to run a transient analysis of the circuit.

In our tutorial circuit, the RC time constant is 100k x 20n = 2 milliseconds. To view 5 cycles of the oscillation we will set up to view a 10ms portion of the waveform.

- ➤ Select **Simulate » Setup** from the menus to display the *Analyses Setup* dialog. All the simulation options are set up here.
- ➤ Click on the **Transient/Fourier** tab at the top of the dialog.
- ➤ In the **Default Parameters** region of the dialog disable the **Always Set Defaults** option. When you do, the options in the **Transient Analysis** region of the dialog will become available.

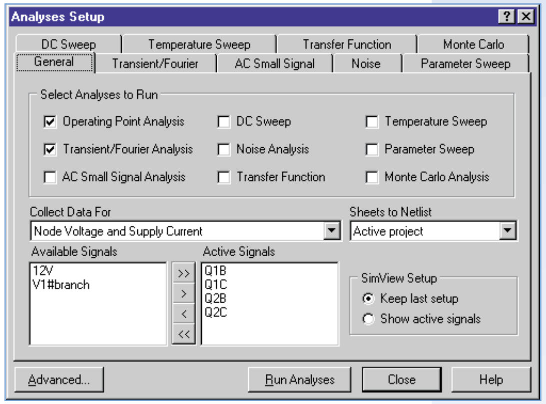
Setting up the transient analysis



- ➤ To specify a 10ms simulation window set the **Stop Time** field to 10m.
- ➤ Now set the **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).
- ➤ During simulation the actual timestep is varied automatically to achieve convergence. The **Maximum Step** field limits the variation of the timestep size, set the **Maximum Step** to 10u.

The last thing to set up is the nodes in the circuit that you wish to observe. To do this:

- ➤ Click on the **General** tab in the *Analyses Setup* dialog
- ➤ 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.
- ➤ 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.

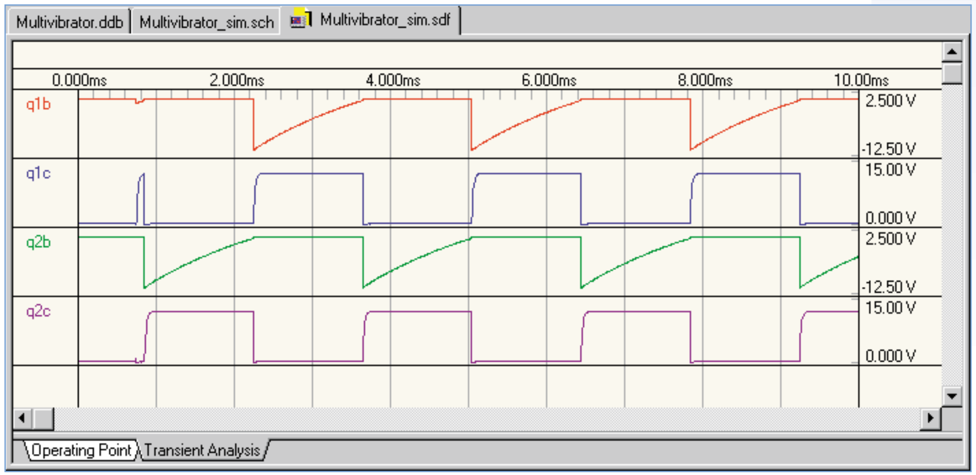


You are now ready to run a transient simulation.

- ➤ Click on the **Run Analyses** button at the bottom of the dialog to run the simulation.
- ➤ The simulation will be performed, when it is finished you should see output waveforms similar to those shown in Figure 11.

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.

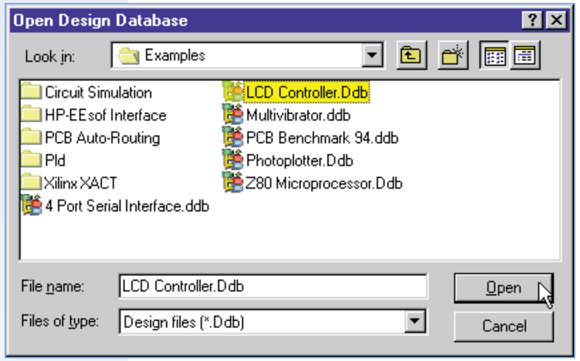
Figure 11
Output waveforms from the multivibrator



Further explorations

This tutorial has introduced you to just some of the powerful features of Protel 99 SE. 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 99 SE. But we've only just scratched the surface of the design power provided by Protel 99 SE.

Once you start exploring Protel 99 SE 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\Design Explorer 99 SE\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 99 SE.



Check out the extensive examples to explore more of Protel 99 SE's design capabilities

Check out the \Circuit Simulation\ sub-folder to explore Protel 99 SE'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 (741 Operational Amplifier.ddb), and even a vacuum tube example!

If you're interested in PLD design, the \PLD\ sub-folder has a number of schematic-based pld designs, including **Wait.ddb** which can be compiled to produce the **JEDEC** download file. If you're interested in implementing a PLD design using a **Hardware Description Language**, the \Reference\ sub-folder includes a number of designs that demonstrate the various capabilities of Protel 99 SE's **CUPL HDL**, including simulating the PLD design.

With faster logic switching and design clock speeds, the quality of the digital signals becomes more important. Protel 99 SE 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** 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.

If you are interested in exploring Protel 99 SE's signal integrity analysis features why not download the Signal Integrity Analysis Introduction PDF file from the download page of Protel's web site, www.protel.com.

Shortcut keys

Common Schematic and PCB Shortcuts

Keys	Function
Left-click	Focus object
Left double-click	Edit object
Left-click and hold	Move object/selection
Left-click, hold and drag	Select inside area
Shift+left-click	Select/deselect object
Right-click	Popup floating menu/esc from current operation
X, A	De-select all
Alt+Backspace	Undo
Ctl+Backspace	Redo
V, D	View Document
V, F	View Fit placed objects
PageUp	Zoom in (zooms around cursor, position the cursor first)
PageDown	Zoom out
Home	Redraw screen with center at the cursor point
End	Redraw the screen
Spacebar	Abort screen re-draw
Tab	While an object is floating on the cursor to edit its attributes
Shift	While autopanning to pan at higher speed
X	Flip object along the X-axis
Y	Flip object along the Y-axis
M	Pop-up Move sub-menu
S	Pop-up Select sub-menu
Esc	Escape from current process

Schematic Shortcuts

Keys	Function
Insert	While placing an object to clone a placed object (of the same kind)
Backspace	While laying a wire/bus/line/ polygon to delete the last vertex
Spacebar	While placing to rotate object by 90 degrees
Ctrl+click (then release Ctrl)	Drag object, maintaining wire connectivity
Ctrl (hold)	Temporarily disable the Snap grid
Left-click and hold, then press Insert	When a wire is focused to add a vertex
Left-click and hold, then press Delete	When a wire is focused to delete a vertex

PCB Shortcuts

Keys	Function
Q	Toggle units (metric/imperial)
Ctrl+click	Drag track/drag track end
Ctrl+shift+click	Break track
Backspace	Remove last track corner during track placement
Ctrl	Disable the Electrical grid during placement
Alt	Temporarily switch from Avoid Obstacle to Ignore Obstacle mode
Ctrl+Spacebar	Cycle through connection lines on a pad when routing
Right-click and hold	Display slider hand and slide view of PCB
G	Pop up snap grid menu
Ctl+G	Pop up snap grid dialog
Shift+E	Toggle electrical grid on/off
Shift+R	Cycle through routing modes (ignore obstacle, avoid obstacle, push obstacle)
Shift+S	Toggle single layer mode on/off
L	Layers Tab of Document Options dialog
Ctl+Delete	Clear all selected
Ctl+H	Select connected copper
Spacebar	Rotate object being moved (anti-clockwise)
Shift+Spacebar	Rotate object being moved (clockwise)
Spacebar	Toggle start/end mode during track placement
Shift+Spacebar	Cycle through all placement modes during track placement
Ctrl+Spacebar	Cycle through connection line on a pad - use this after starting to route from a pad with multiple connection lines
L	Flip component being moved to the other side of board
N	Hide net connections while moving a component
*	Cycle through signal layers (numeric keypad)
+	Cycle forwards through layers (numeric keypad)
-	Cycle backwards through layers (numeric keypad)

Design Explorer Shortcuts

Keys	Function
Ctrl (during file open)	Open design without re-opening documents
Tab	Switch from Panel to active document/folder
Shift+Tab	Switch from text document to Navigation Panel
Ctrl+Tab	Cycle through open documents/folders
Ctrl+F6	Cycle through open design databases
Ctrl+F4	Close active document/folder
Alt+F4	Close Design Explorer
F5	Refresh the display