

PCB Wizard 3 Tutorial 2

Drawing a 555 timer circuit



Step 1 of 10: Introduction

This tutorial shows you how to design and make an electronic circuit with PCB Wizard. You should follow this tutorial to learn the basic skills you will need to use PCB Wizard effectively.

Difficulty Level:  **Medium** (suitable for moderately experienced users)

Getting started

In this tutorial you will create a 555 astable timing circuit similar to the one shown on the right. The circuit will flash an LED on and off.

Along the way, you will learn how to:

- [Add components from the Gallery](#)
- [Wire components together](#)
- [Change component values](#)
- [Convert the circuit into a PCB layout](#)
- [Add text to the PCB](#)
- [View how the finished PCB will look](#)

Components

To make this circuit you will need:

8-pin dual-in-line (DIL) socket
555 timer integrated circuit (IC)
Red LED

1K ohm resistor:

- Brown, Black, Red and Gold (4 band)*
- Brown, Black, Black, Brown and Gold (5 band)*

680 ohm resistor:

- Blue, Grey, Brown and Gold (4 band)*
- Blue, Grey, Black, Black and Gold (5 band)*

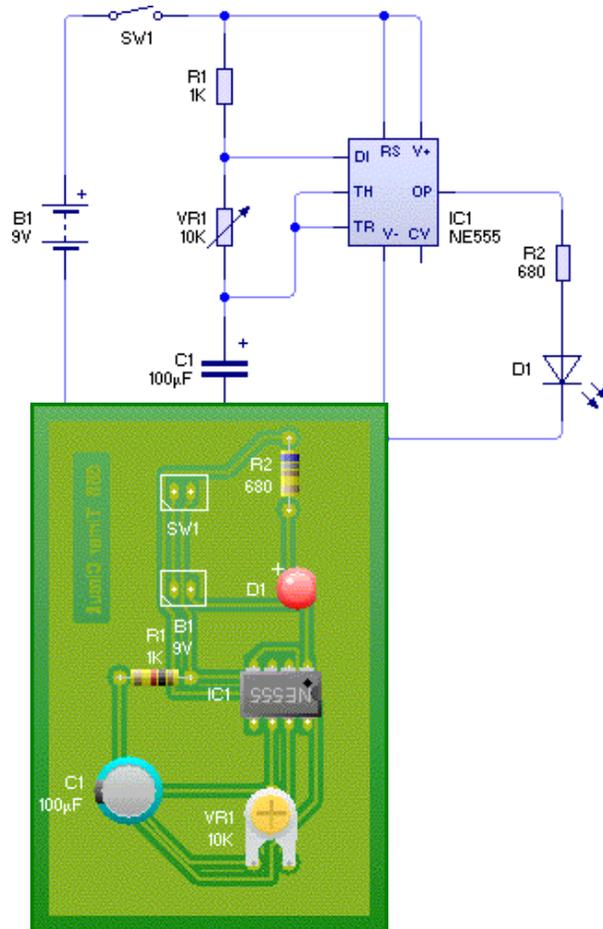
10K ohm variable resistor

100 μ F electrolytic capacitor

PP3 battery and clip

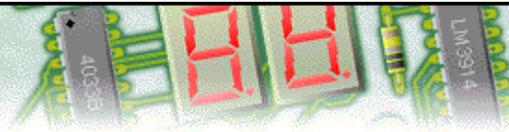
Single pole, single toggle (SPST) switch

plus suitable PCB making equipment



PCB Wizard 3 Tutorial 2

Drawing a 555 timer circuit



Step 2 of 10: Adding components

You will begin by creating a new (empty) document in which to draw your circuit. To create a new document, click on the New button or choose **New** from the **File** menu.

Next you will learn how to use the Gallery to add components to your circuit. If the Gallery is not currently open, click on the Gallery button on the top toolbar to open it. Select the **Circuit Symbols** option.



In the Circuit Symbol Gallery window, you will be able to see all the components that are available within PCB Wizard.

Components within the Gallery are grouped according to their function. At the top of the window, a drop-down list box allows you to select which group is shown.

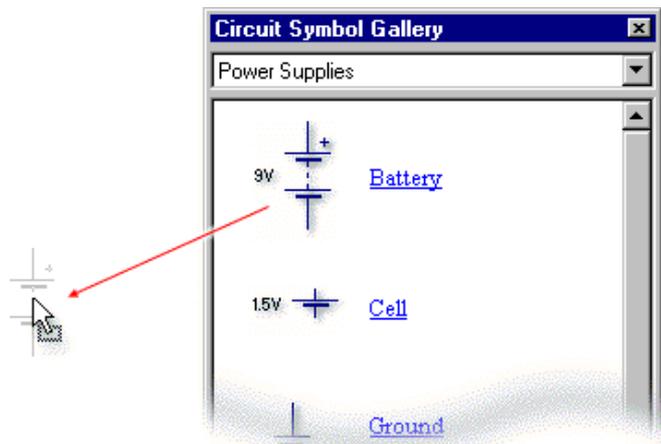
From the **Power Supplies** group, add a **Battery** component from the Gallery to your circuit.

To do this:

Move the mouse over the Battery symbol. Press and hold down the left mouse button.

With the left mouse button still held down, move the mouse to drag the symbol onto the circuit.

Finally, release the mouse button when the circuit symbol is in the required position.



To make the 555 timer circuit you will also need several other components.

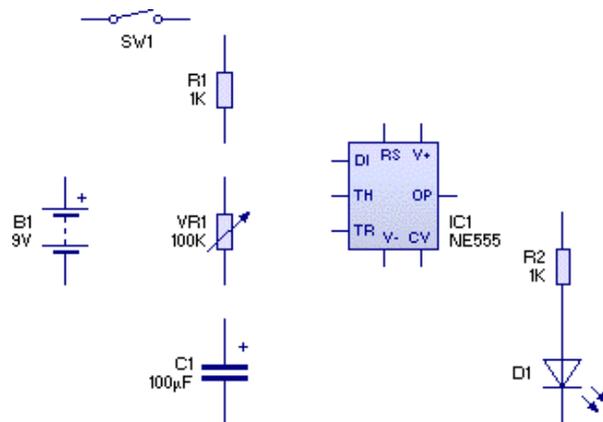
Add an **SPST Switch** and a **Variable Resistor** the **Input Components** group; two **Resistors** and an **Electrolytic Capacitor** from the **Passive Components** group; a **555 Timer** from the **ICs (Analogue/Mixed)** group and finally an **LED** from the **Output Components** group.

It will help later on if you position the components neatly before you start adding wires to the circuit.

You can move components by clicking on the Select button from the top toolbar.



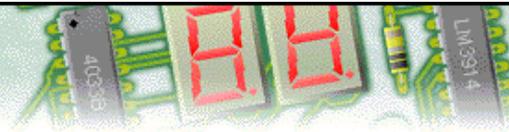
In Select mode the cursor will appear as a standard pointer:



Using the above layout as a guide, try repositioning the components. Thinking about the position of components at the start can help produce a much neater circuit diagram.

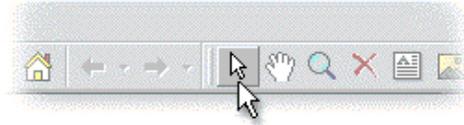
PCB Wizard 3 Tutorial 2

Drawing a 555 timer circuit



Step 3 of 10: Wiring components together

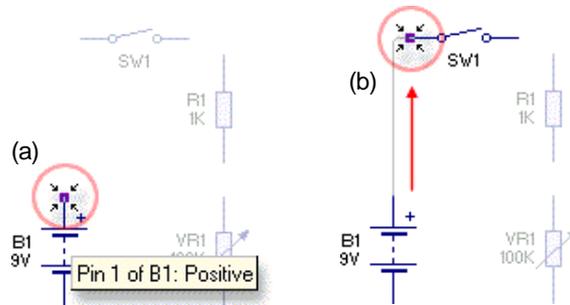
Once the components have been placed, you can start to wire the components together. To do this you must first click on the Select button from the top toolbar.



Next, move the mouse over the top pin of the battery (a). As you hold the mouse over the pin you will notice a hint appear describing that particular component pin.

Press and hold down the left mouse button. With the mouse button still held down, move the mouse to place a wire.

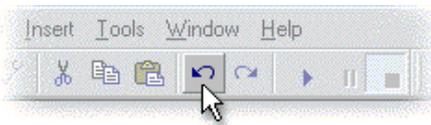
To complete the wire, release the mouse button over the left pin of switch **SW1** (b).



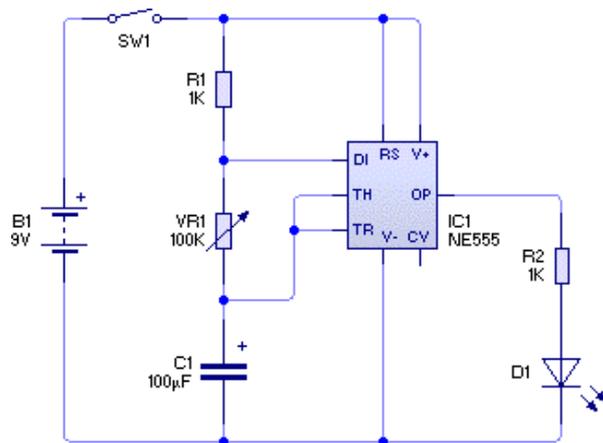
 When drawing a wire you can add a bend to a wire by releasing the mouse button over or clicking on an empty part of the circuit.

You can now wire up the rest of the circuit using the diagram on the right as a guide.

Remember that if you get stuck, you can always just click on the Undo button to correct any mistakes:

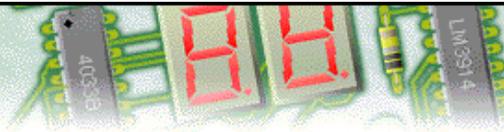


Finally, for more detailed help and information on wiring circuits, refer to the topic entitled Wiring components together in the Help.



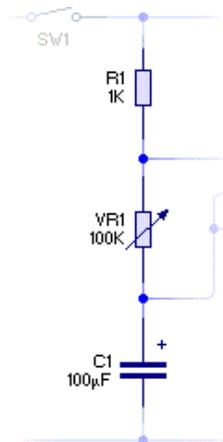
PCB Wizard 3 Tutorial 2

Drawing a 555 timer circuit



Step 4 of 10: Changing component values

Now that you have drawn the circuit diagram, you can change the component values.



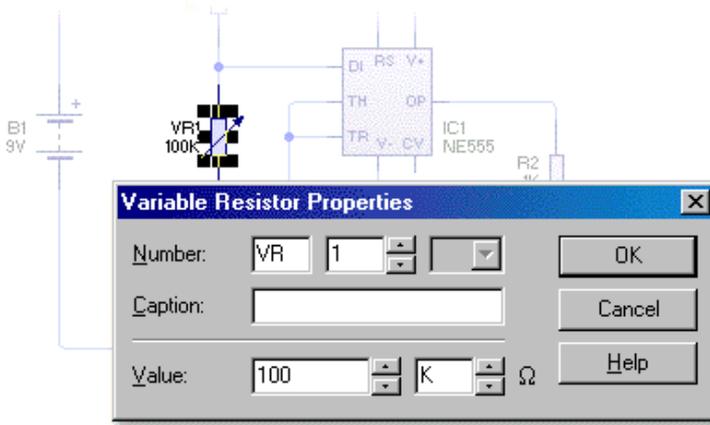
With 555 astable circuits, the timing is controlled by two resistors and a capacitor. In your circuit these are **R1**, **VR1** and **C1**.

The rate at which the LED will flash is determined by the following equation:

$$f = \frac{1.44}{(R1 + 2VR1) \times C1}$$

where **R1** is 1K (or 1,000), **VR1** is 50K (50% of 100K, or 50,000, when the slider is in the mid position) and **C1** is 100µF (or 0.0001). This gives a frequency (*f*) of 0.14 Hz (Hertz) which would result in the LED flashing about once every 7 seconds (since flash rate = 1 / frequency).

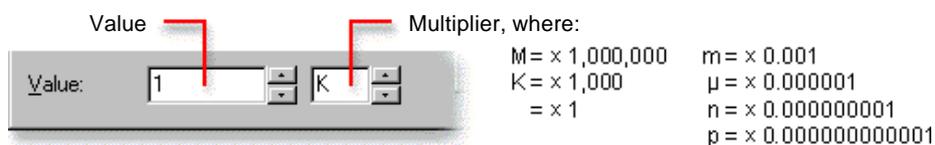
To make the LED flash at a faster rate, the 100K variable resistor will be replaced with a 10K variable resistor. Using the above formula, repeat the calculations using this new value. How often would the LED now flash?



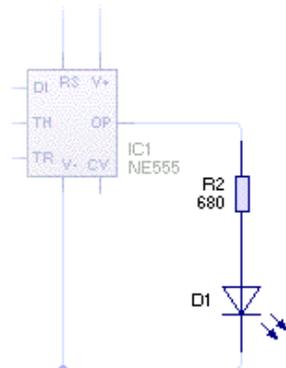
💡 It is good practice for one of the timing resistors in a 555 astable to be a variable resistor as it allows the rate to be adjusted once the circuit has been manufactured.

Double-click on variable resistor **VR1** to display the Variable Resistor Properties window (above).

The **Value** field for the variable resistor is shown at the bottom of the window and consists of both a value and a multiplier. The variable resistor's value (in ohms) is calculated by multiplying the value by the multiplier.



In your circuit, the variable resistor should have a value of 10K. Enter **10** in the first value box but leave the multiplier unchanged at K (x 1,000).



Finally, you will need to change the value of resistor **R2**. In your circuit, resistor **R2** will be used to limit the amount of current that passes through the LED. It is good practice to include current-limiting resistors when using LEDs; without them, LEDs may be damaged or even destroyed.

As a 9 volt battery has been used, the value of this current limiting resistor will need to be changed to **680 ohms** which would limit the current flowing through the LED to about **10mA** (milli-amps).

Double-click on resistor **R2** and change its value to **680**. Remember that you will also need to change the multiplier from K (x 1,000) to blank (x 1).

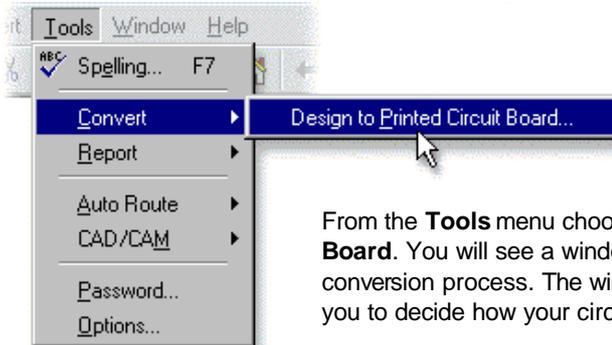
PCB Wizard 3 Tutorial 2

Drawing a 555 timer circuit



Step 5 of 10: Converting the circuit into a PCB layout (1)

Now that the 555 timer circuit is complete, you can convert it into a printed circuit board.

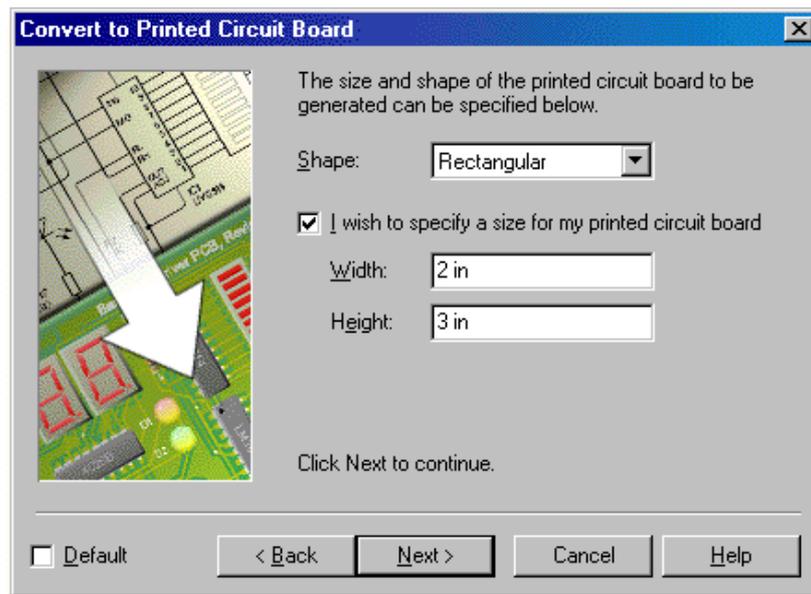


From the **Tools** menu choose **Convert | Design to Printed Circuit Board**. You will see a window appear to lead you through the conversion process. The window contains a series of pages that allow you to decide how your circuit is converted.

These pages cover areas such as the size and shape of your PCB layout, which components are used as well as more advanced features such as automatic routing and component placement.

 For more information on the options available for converting your circuit diagrams into PCB layouts, see the topic entitled [Converting to a PCB layout](#) in the Help.

Click on the **Next** button. You will then see the first page of options (see below).



Within this page you can select the size and shape of the printed circuit board that is produced.

You can either choose to enter a specific size or have PCB Wizard calculate a size based on the components within your circuit. For your 555 timer circuit a specific size will be chosen.

Click on the **I wish to specify a size for my printed circuit board** option. The **Width** and **Height** boxes will then become available. Enter **2 in** in the Width box and **3 in** in the Height box. If you wish to use metric measurements then you can enter **50 mm** and **75 mm** respectively.

 In the **Width** and **Height** fields, you can type in a different unit of measurement to the one given. For example, you could type in '65 mm', '4 in' or even '3500 mil' (where a mil is one thousandth of an inch). The available units are mm, cm, m, in, pt and mil.

To change the unit of measurement used throughout the entire application, choose **Options** from the **Tools** menu and select a different **Measurement unit** from the **General** tab.

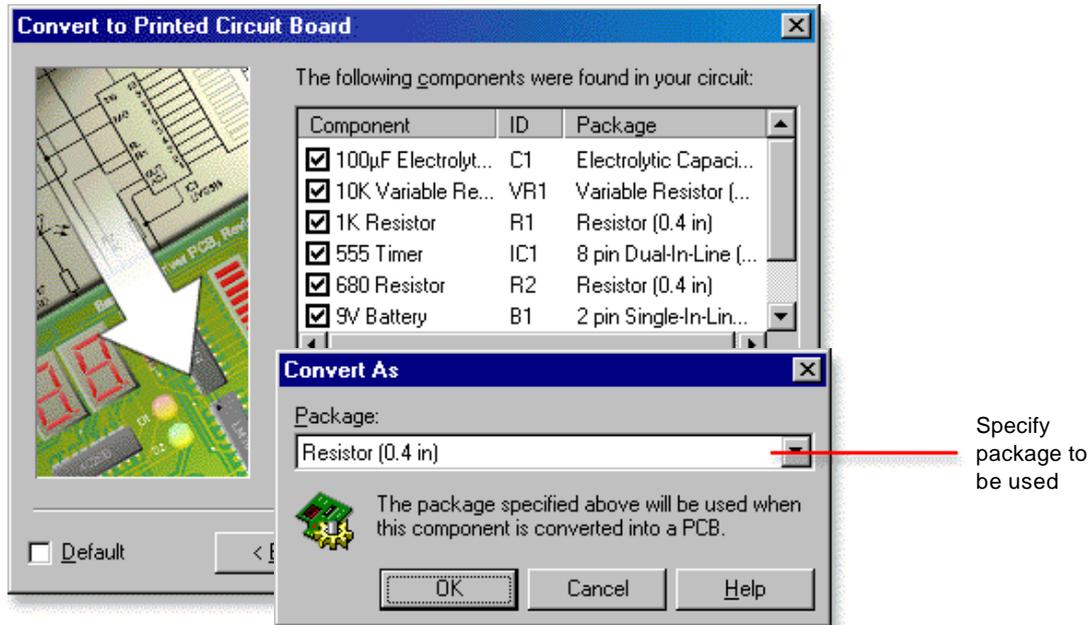
PCB Wizard 3 Tutorial 2

Drawing a 555 timer circuit



Step 6 of 10: Converting the circuit into a PCB layout (2)

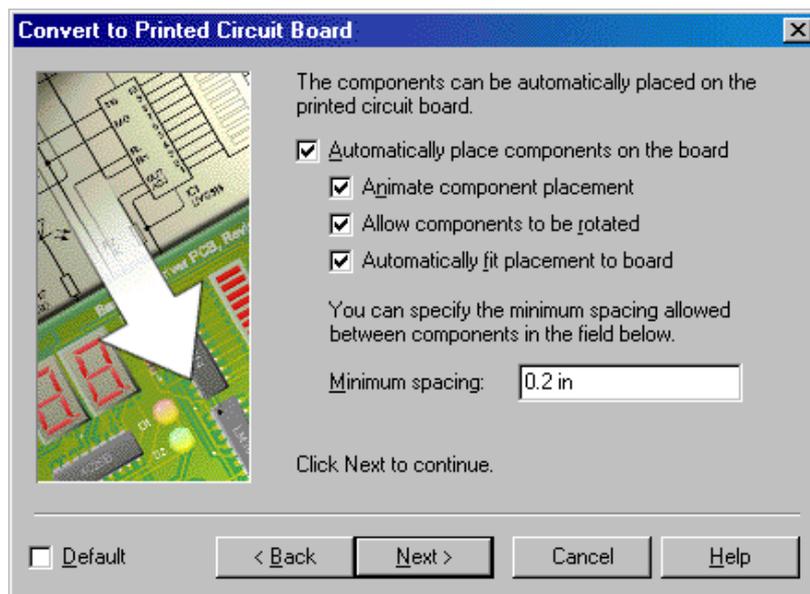
Clicking on the **Next** button will show you how your components will be converted.



The window lists each component in your circuit. If you wish to change how a particular component is converted, double-click with the left mouse button on the a component from the list and then select an appropriate PCB component from the window that appears.

Once you have specified the components, click on the **Next** button again.

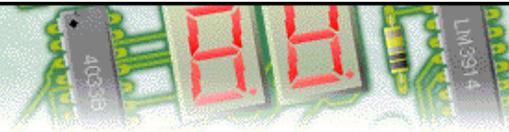
The page that appears allows you connect any digital components to a power supply such as a battery. As your 555 timer circuit does not contain any digital components, these options can be skipped so click on the **Next** button one more time. You will then see the page below.



It is within this page that you can control how the components are positioned on the board. The default settings shown above will be suitable for your 555 timer circuit.

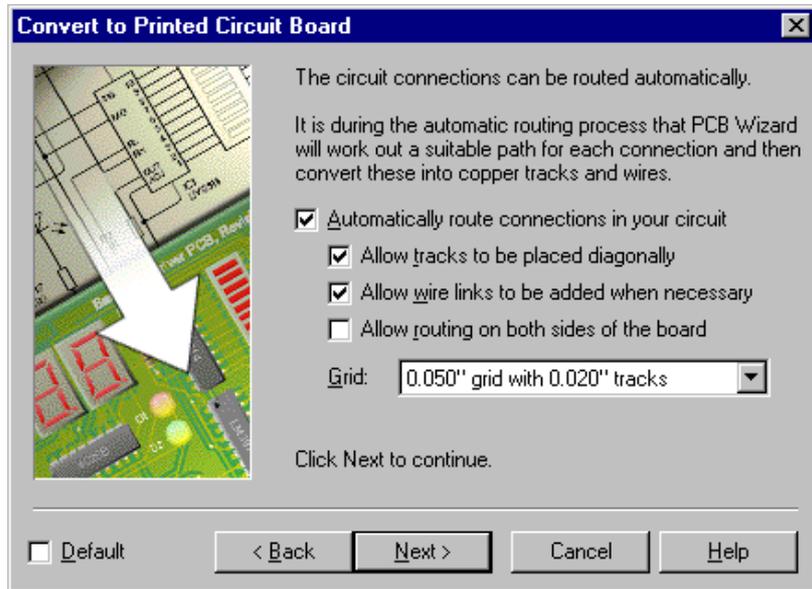
PCB Wizard 3 Tutorial 2

Drawing a 555 timer circuit



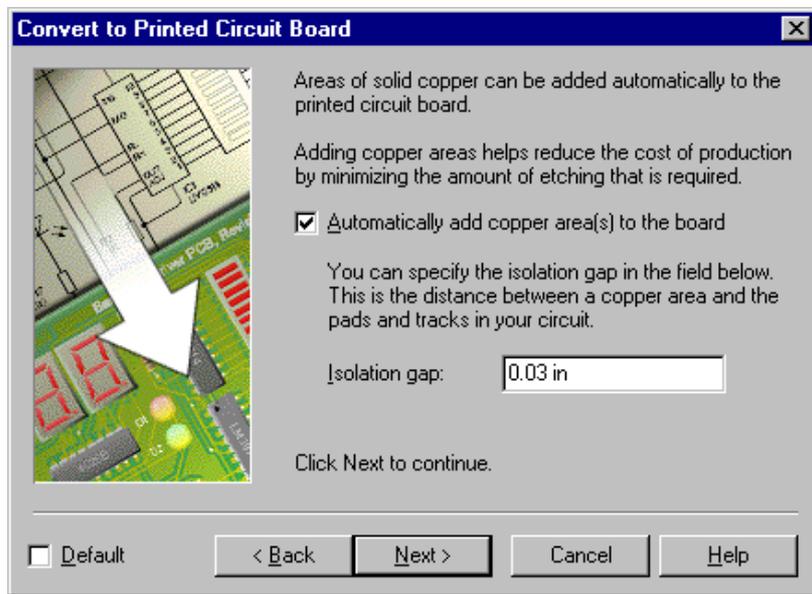
Step 7 of 10: Converting the circuit into a PCB layout (3)

After clicking on the **Next** button, the automatic routing options will be shown.



The **Grid** option should be set to **0.050" grid with 0.020" tracks**. This will make it possible for tracks to be routed through the legs of an integrated circuit (a 555 timer in your case).

With the options set as shown above, click on the **Next** button. The page below will then appear.

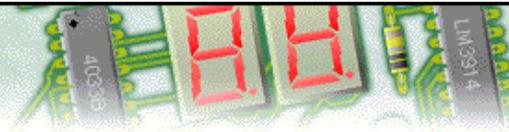


On this, the final page of options, you have the opportunity to add areas of solid copper to your printed circuit board. Copper areas help reduce costs by limiting the amount of etching solution that is required when the circuit is eventually manufactured.

By default this setting is switched on, with an isolation gap of 0.03". For your 555 timer circuit, you should keep the settings the same.

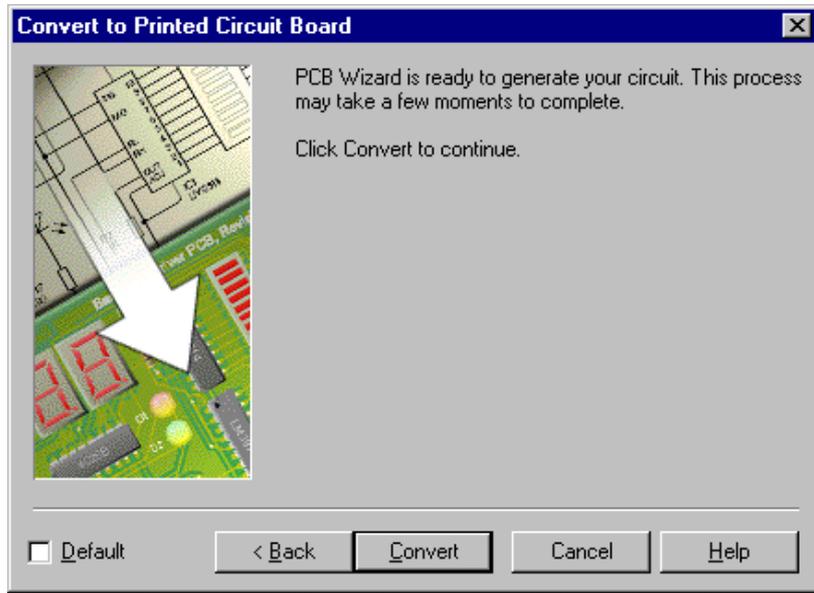
PCB Wizard 3 Tutorial 2

Drawing a 555 timer circuit

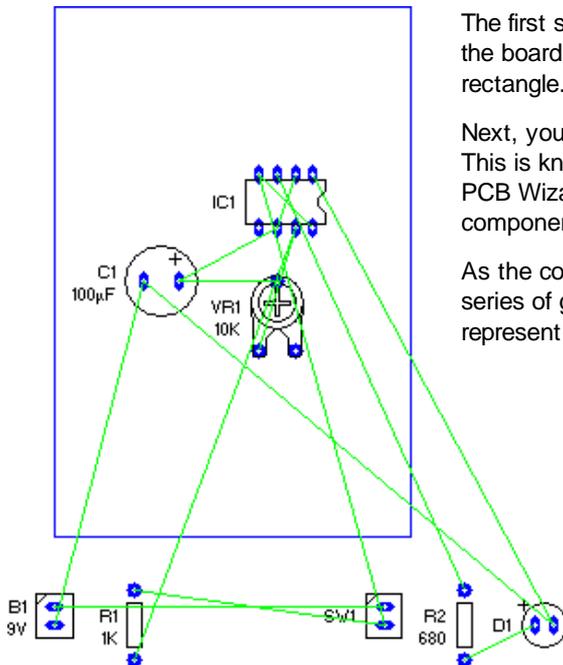


Step 8 of 10: Converting the circuit into a PCB layout (4)

With all of the options specified, you are ready to convert your circuit. Click on the **Convert** button.



PCB Wizard will now create a printed circuit board for your 555 timer circuit.



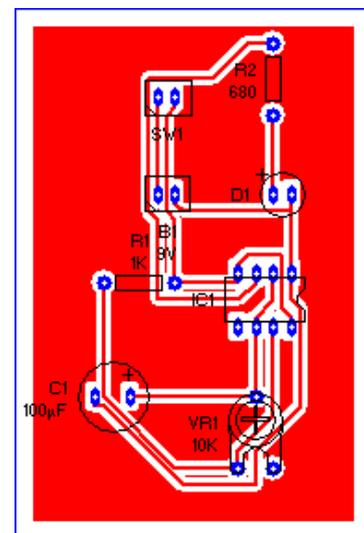
The first step in the conversion process is for an outline of the board to be created. This is shown on the left as a blue rectangle.

Next, you will see each component added to the board. This is known as **automatic component placement**. PCB Wizard calculates the optimum position for each component in your circuit.

As the components are positioned, you will also see a series of green lines. These are known as nets and represent electrical connections between the components.

With the components in position, PCB Wizard will then add the necessary copper tracks during a process known as **automatic routing**. A path, or route, is found for each connection such that it does not touch any existing tracks on your circuit. Unlike wires on a circuit diagram, copper tracks on a printed circuit board cannot overlap.

Finally, at the end of the process, a solid copper area will be added to your printed circuit board.



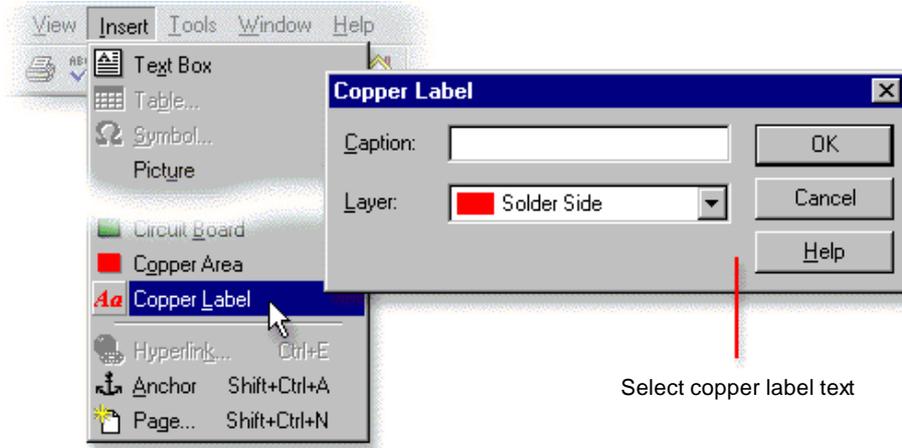
PCB Wizard 3 Tutorial 2

Drawing a 555 timer circuit



Step 9 of 10: Adding text to the PCB layout

Next you will use copper labels to add text to your printed circuit board. Copper labels will help you to identify your printed circuit board once it has been manufactured.

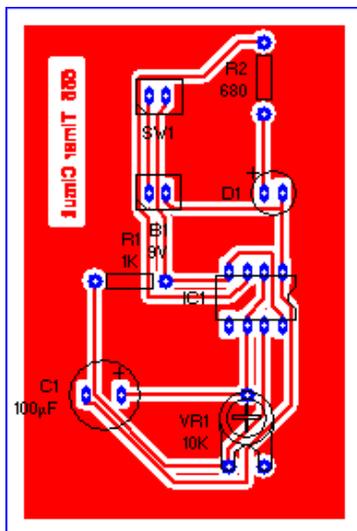


To add a copper label choose **Copper Label** from the **Insert** menu.

Next, click with the left mouse button somewhere on your circuit (you do not need to be very precise as you will be able to reposition the label later). A window will appear allowing you to type text for the label and specify the layer on which it will reside (see above).

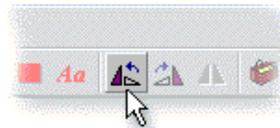


Enter a caption of **'555 Timer Circuit'** and then click on the **OK** button. Your label will then appear on the circuit.



Notice how the text in the label has been reversed. This is so that the label will appear the correct way around when the PCB is eventually made. Any copper labels placed on the solder side layer (the underside of the PCB) are automatically reversed.

To fit the label more neatly on your board, you will need to rotate the label by 90 degrees. Select the label and then click on the **Rotate Left** button on the top toolbar.



Finally, move the rotated label into the top-left hand corner of the board (as shown on the left). Remember that as the label will be made of copper (since it is on the solder side layer), it must not overlap any existing pad or track in your circuit.

PCB Wizard 3 Tutorial 2

Drawing a 555 timer circuit



Step 10 of 10: Viewing how the finished PCB will look

With your printed circuit board now created, you can see how it will look when made.



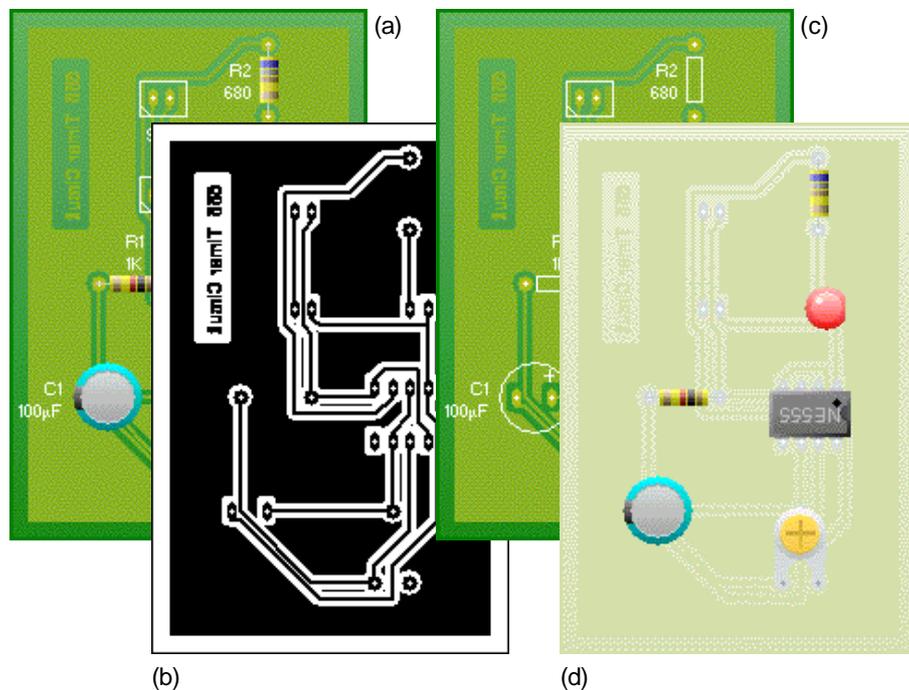
On the left-hand side of the main PCB Wizard window, you will see the Style toolbar. This toolbar shows the different ways in which your circuit can be viewed.

Click on the **Real World** button. The display of your circuit will change to show you how your circuit would look if it were professionally manufactured (a).

Next, click on the **Artwork** button. You now see the artwork (or mask) for your circuit (b). It is this artwork that you would use to make the printed circuit board.

To see how a professionally manufactured circuit would look prior to the components being soldered in place (c), click on the **Unpopulated** button.

Finally, try clicking on the **Prototype** button. This is how your circuit would look if made as a one-off prototype (d).



💡 You can use the above styles to help when manufacturing the finished printed circuit board. In particular, the Real World and Unpopulated views of your board will show where each component needs to go.