

PCB Wizard 3 Tutorial 1

Drawing a transistor sensing circuit



Step 1 of 7: 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:  **Beginner** (suitable for new and inexperienced users)

Getting started

In this tutorial you will create a transistor sensing circuit similar to the one shown on the right. The circuit will light an LED when the temperature gets too cold.

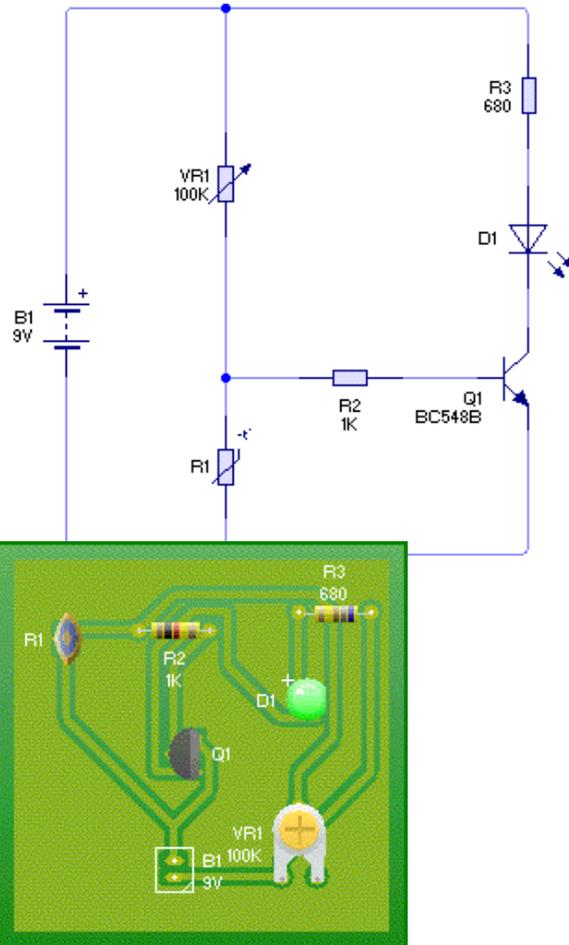
Along the way, you will learn how to:

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

Components

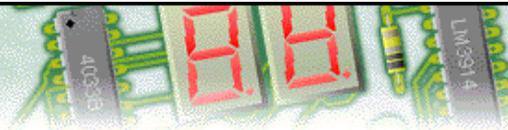
To make this circuit you will need:

- Thermistor (NTC type)
- BC548B transistor
- Green 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)
- 100K ohm variable resistor
- PP3 battery and clip
- plus suitable PCB making equipment*



PCB Wizard 3 Tutorial 1

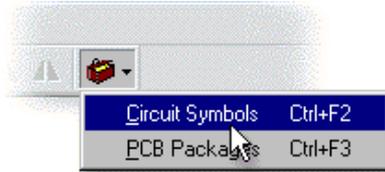
Drawing a transistor sensing circuit



Step 2 of 7: 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.

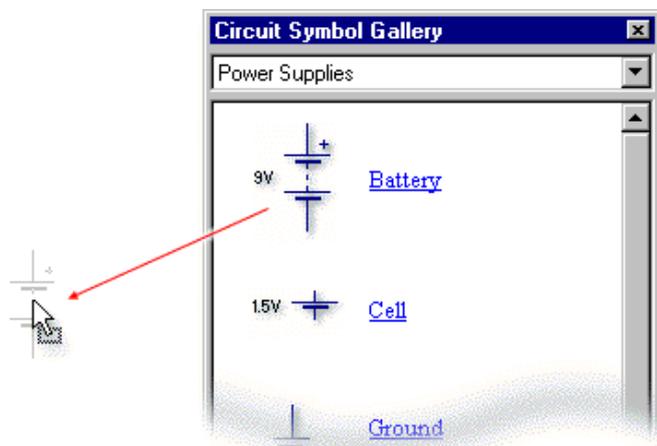
From the **Power Supplies** group, add a **Battery** component from the Gallery to your circuit.

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 transistor sensing circuit you will also need several other components.

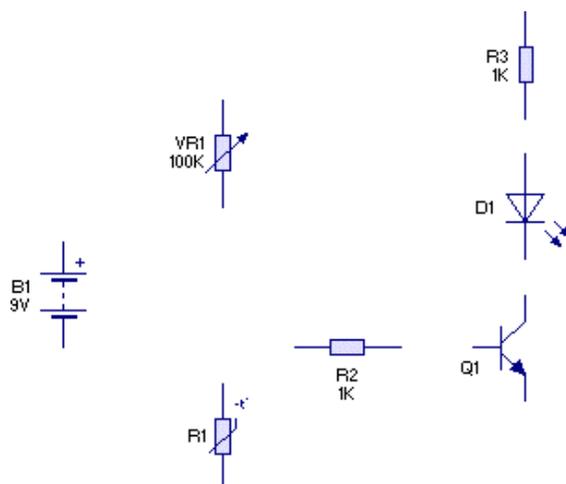
Add a **Variable Resistor** and a **Thermistor** from the **Input Components** group; two **Resistors** from the **Passive Components** group; an **NPN Transistor** from the **Discrete Semiconductors** group and finally an **LED** from the **Output Components** group.

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

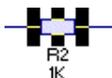
You can move components by clicking on the Select button from the top toolbar.



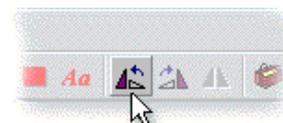
In Select mode the cursor will appear as a standard pointer:



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.

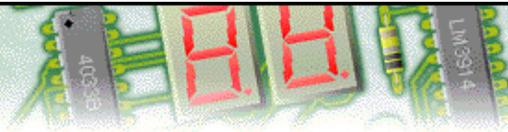


To align the pins of resistor **R2** with the base of the transistor, you will need to rotate the component. Select resistor **R2** and then click on the **Rotate Left** button.



PCB Wizard 3 Tutorial 1

Drawing a transistor sensing circuit



Step 3 of 7: 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:



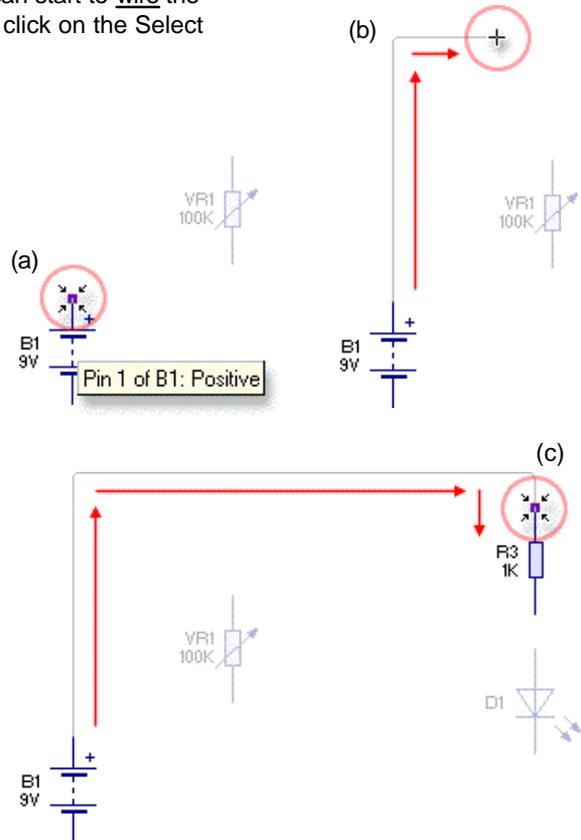
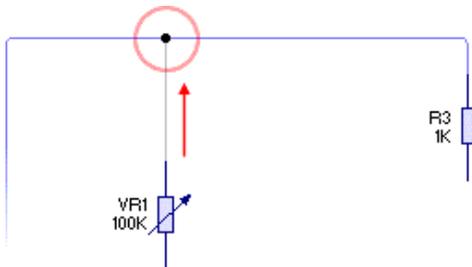
Next, move the mouse over the top pin of the battery (a). As you hold the mouse over the pin you will notice a hint appear describing that particular component pin.

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

You can add a bend to the wire by releasing the mouse button over or clicking on an empty part of the circuit (b).

To complete the wire, release the mouse button over the top pin of resistor **R3** (c).

 If you make a mistake, you can move back over the wire to retrace your steps.

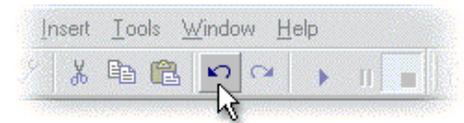


Now a second wire from the top of variable resistor **VR1** to join with the wire you have just added.

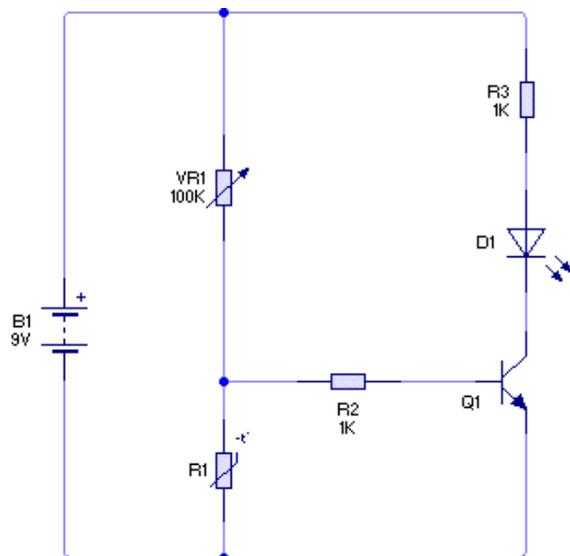
To join two wires together, simply release the mouse button over an existing wire. You will notice that when you do so, a junction is automatically added at the connecting point.

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 1

Drawing a transistor sensing circuit

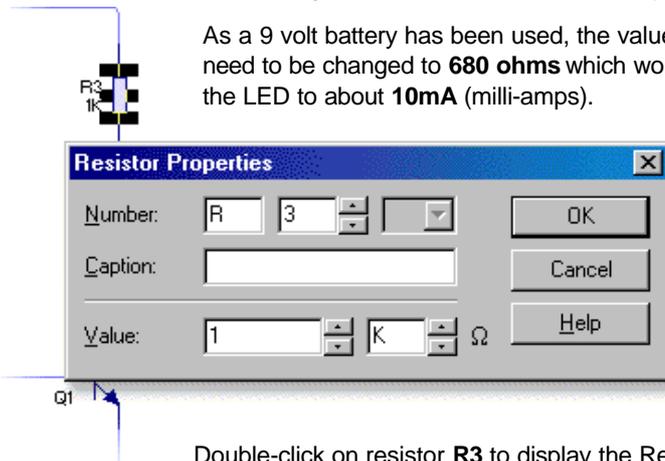


Step 4 of 7: Changing component values and models

With the circuit diagram drawn, you can now start to adjust the components.

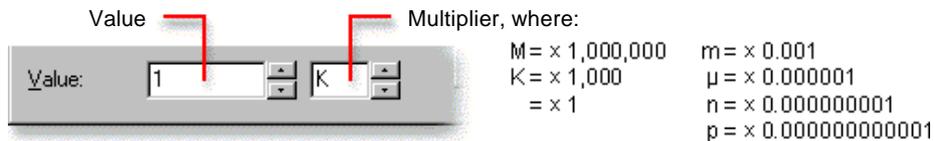
In your circuit, resistor **R3** will be used to limit the amount of current that passes through the LED. It is good practice to include current-limiting resistors when using LEDs; without them, LEDs may be damaged or even destroyed.

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 **R3** to display the Resistor Properties window (above).

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



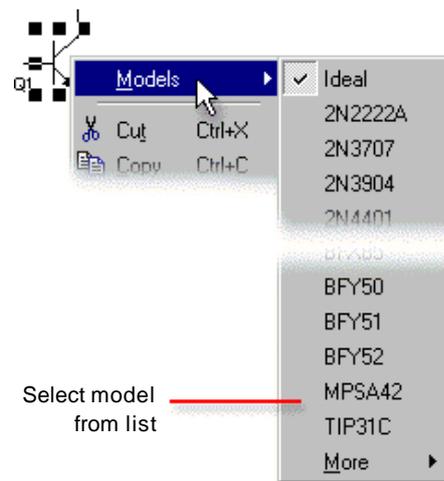
Enter **680** in the first value box and then click once on the down arrow to the right of the second box to change the multiplier from K (x 1,000) to blank (x 1).

To complete the transistor sensing circuit, you will need to specify exactly which type of transistor is to be used. These types are known as models.

Click the right mouse button over transistor **Q1** and, from the pop-up menu that appears, choose the **BC548B** transistor from the **Models** list.

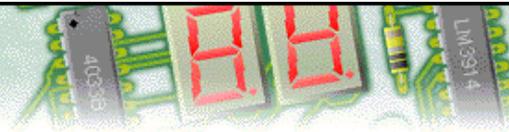
You will notice that the transistor is now labelled as a BC548B on the circuit diagram.

Most components within PCB Wizard support different models. LEDs, for example, are available in a range of different colours. Try this now. Click the right mouse button on the LED and select **Green** from the list of available models.



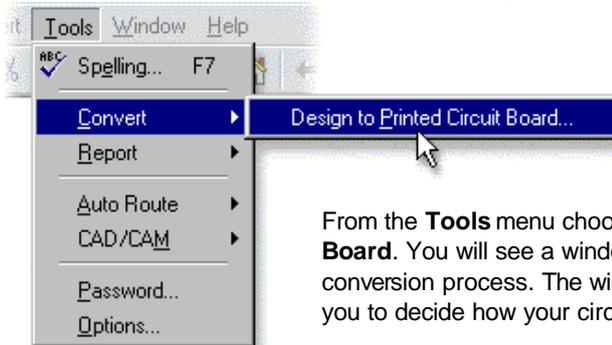
PCB Wizard 3 Tutorial 1

Drawing a transistor sensing circuit



Step 5 of 7: Converting the circuit into a PCB layout (1)

Now that the transistor sensing 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.

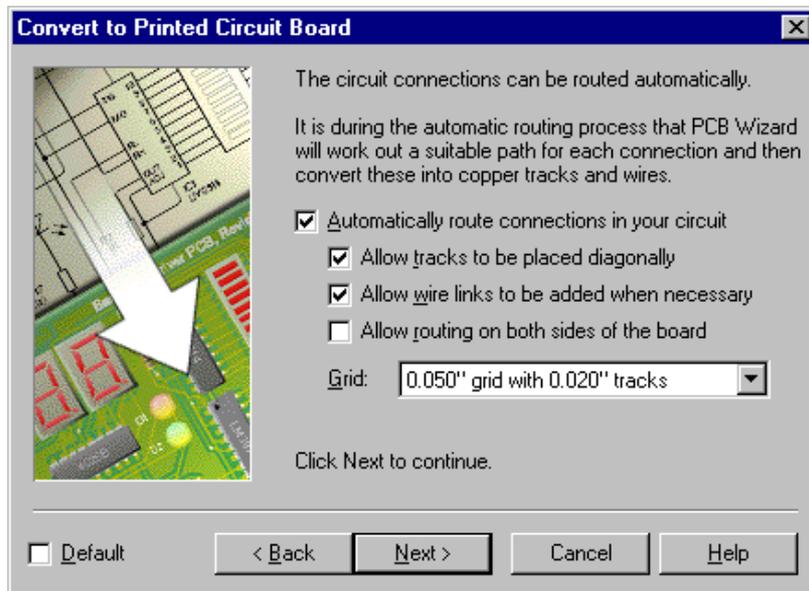
These pages cover areas such as the size and shape of your PCB layout, which components are used as well as more advanced features such as automatic routing and component placement.

 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.

For many circuits you can leave the settings in the window unchanged. With your transistor sensing circuit, just one option will be changed. You will change a setting to increase the thickness of the copper tracks that will be added during the automatic routing process.

By increasing the thickness of the tracks, you will make your finished PCB layout easier to build. Note that with more complex circuits, particularly circuits involving integrated circuits (ICs), thinner tracks are normally required to allow the circuit to be routed successfully.

Click the **Next** button five times until the automatic routing page is shown (see below).



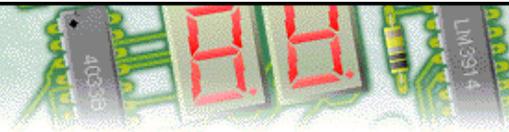
The track thickness is determined by the **Grid** option. It is so named because routed tracks are placed on a grid and it is this grid size that determines the size of the tracks.

With simple circuits, such as your transistor sensor that have few components and connections, it is often preferable to increase the track width from the default setting of 0.020" (about 0.5mm) to 0.040" (about 1mm). This will make the final printed circuit board easier to make.

Click on the **Grid** list and select the **0.100" grid with 0.040" tracks** option.

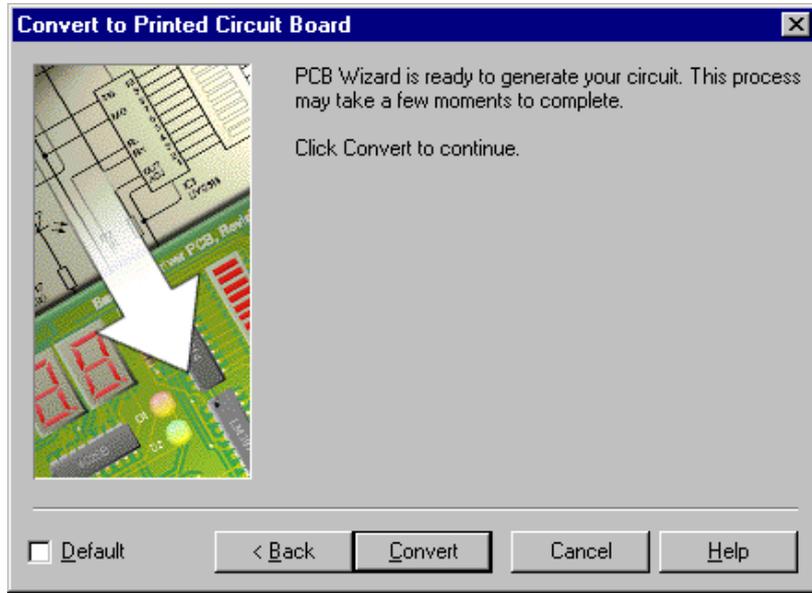
PCB Wizard 3 Tutorial 1

Drawing a transistor sensing circuit

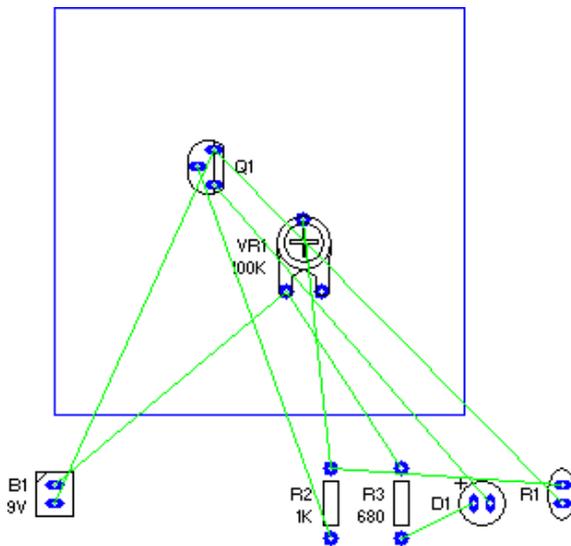


Step 6 of 7: Converting the circuit into a PCB layout (2)

With the **Grid** option changed, click on the **Next** button twice to reach the final page of the window as shown below. You are now ready to convert your circuit. Click on the **Convert** button.



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



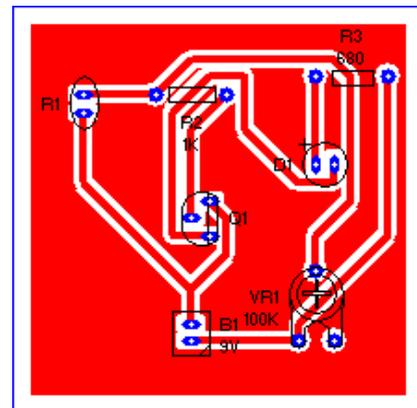
The first step in the conversion process is for an outline of the board to be created. This is shown on the left as a blue rectangle.

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.

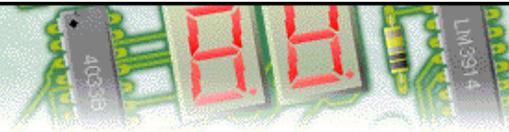
With the components in position, PCB Wizard will then add the necessary copper tracks during a process known as **automatic routing**. A path, or route, is found for each connection such that it does not touch any existing tracks on your circuit. Unlike wires on a circuit diagram, copper tracks on a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

Finally, at the end of the process, a solid **copper area** will be added to your printed circuit board. Copper areas help reduce costs by limiting the amount of etching solution that is required when the circuit is eventually manufactured.



PCB Wizard 3 Tutorial 1

Drawing a transistor sensing circuit



Step 7 of 7: Viewing how the finished PCB will look

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



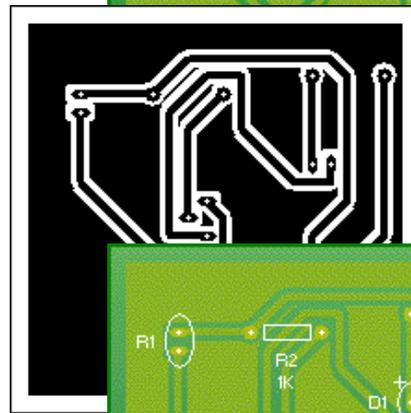
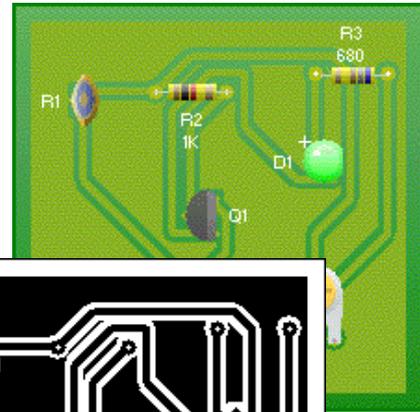
On the left-hand side of the main PCB Wizard window, you will see the Style toolbar. This toolbar shows the different ways in which your circuit can be viewed.

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).

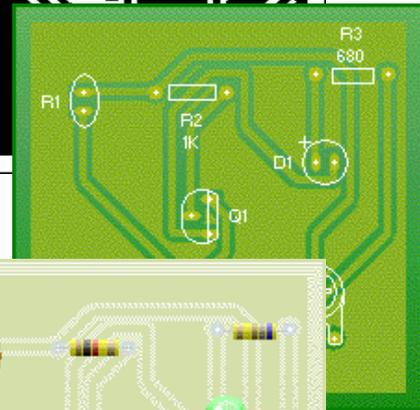
Next, click on the **Artwork** button. You now see the artwork (or mask) for your circuit (b). It is this artwork that you would use to make the printed circuit board.

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).



(b)



(c)



(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.