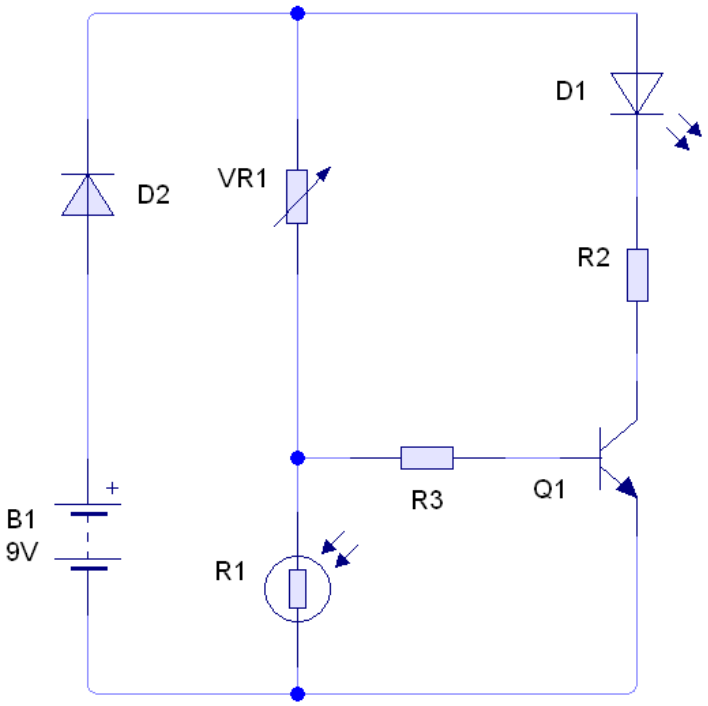
1. **Create The schematic**

   Enter the below schematic into circuit wizard.

   A description of each of the components is given underneath

   Component symbols can be associated with a variety of electrical models. Edit the property of the symbol in the schematic so that the model used is the one indicated in the table

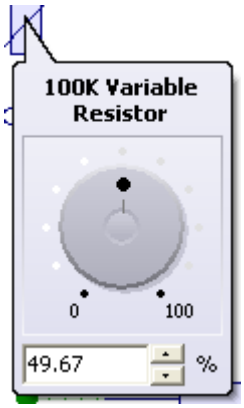


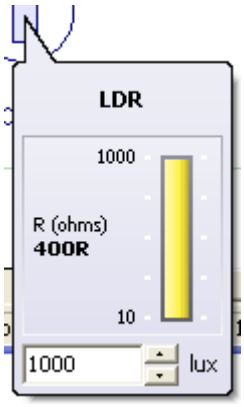
Component	Value	Model	Description
D2		1N4001	Protective diode
VR1	100K		Variable resistor for setting transistor bias point
R1			Light Dependant Resistor (LDR) This forms a potential divider with VR1
R3	2K		Resistor to limit the amount of current flowing into the transistor base
Q1		BFY51	NPN transistor
R2	270		Resistor to limit current flowing into the LED
D1			LED – switched on when light levels fall

2. **Simulate The Schematic**

Press the ► button to start simulating.

Click on VR1 and the box below (left) will appear. Hold the mouse button down to change the value of the variable resistor to be around 50%





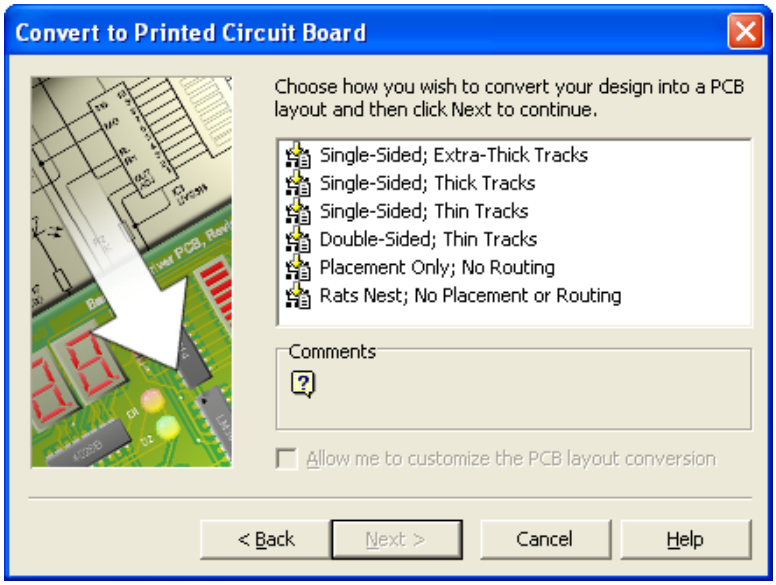
Click on the LDR (R1) and the box (above right) will appear. Click on the yellow strip and move the mouse downwards to indicate a falling light level. The LED should start to light and be fully on by the time it is fully dark.

3. **Automatically Create A PCB**

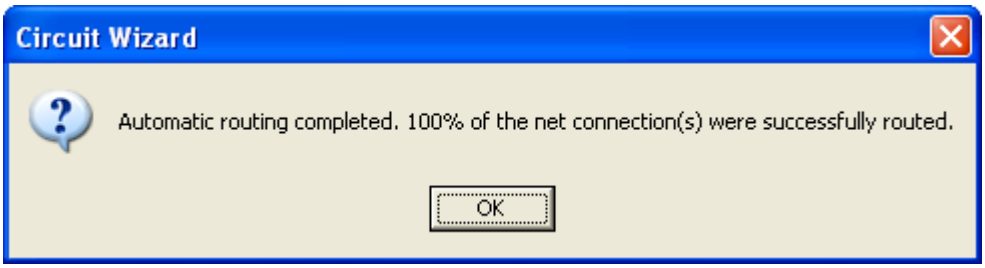
Stop simulating by pressing ■

Press the icon on the top toolbar that is called "*Convert to PCB Layout*" and press *Next*

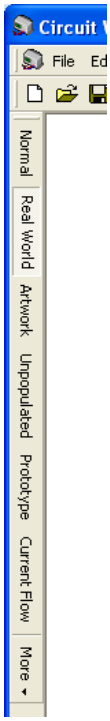
The below window will appear. Select "*Single Sided Thick tracks*" then press *Next* . In the following window, press *Next* again, then *convert*



The tool will place all of the schematic components onto a component and then try and connect them up. If successful, when finished, you should see a message as below:



4. **Examining the PCB**



Down the left hand side of the PCB window you will see various tabs. By default, the *Real World* tab is selected. This shows photographic images of how the components will look when placed, with copper interconnect on the other side of the board shown in green (the PCB is drawn as if it is transparent).

The other useful view is *Artwork*. This shows just the copper interconnect in black. It is this view that we print from and produce transparencies for PCB manufacture.

Look at the tracks – are they as wide as the could be ?

Does the PCB use more space than you think it needs to ?

Automatically creating PCB's is a fast way of designing but does not produce as effective results as designing by hand

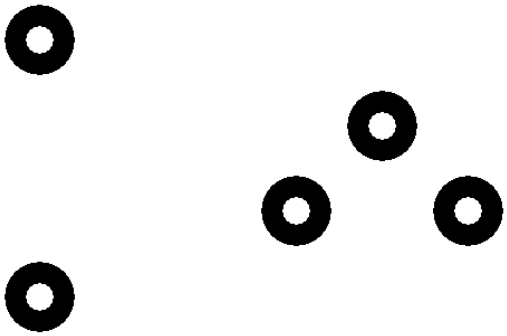
5. **Creating a PCB manually**

Delete the automatically created PCB artwork and select Real View.

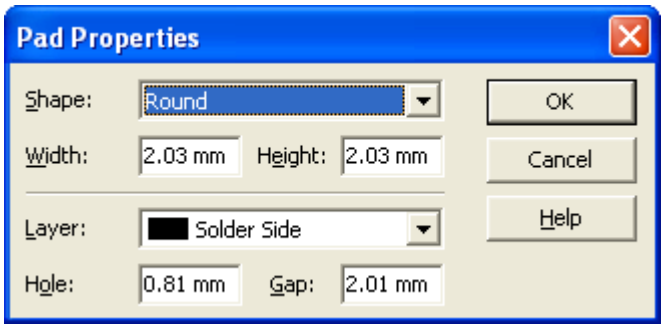
Add all of the components that appeared in your schematic and place them as close together as possible. Leave room for interconnect. Remember that you can rotate components by using the CTRL + R keys.

As you start to add components, switch to the Artwork view and notice that every component has its own footprint – a copper image of what I required to solder that particular component in place.

Below is shown the footprint for a resistor and a transistor.



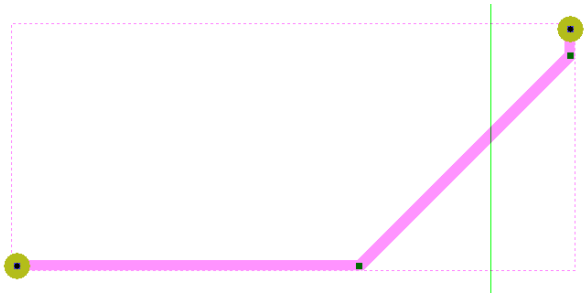
If you select a footprint in the Artwork view or a component in the Real View and then right mouse click and select Pads, the below box will appear:



This allows you to customize the size of the copper pads associated with each component lead.

Press F2 and a set of green cross hairs will appear. You are now in routing mode. Left click and the start of a copper track will appear. Click again and your will create an intermediate point or node in the track. Double click to finish the track.

The picture below shows a piece of track that has been created between two pads. It has 2 intermediate points (nodes) in it, shown as small blue squares. nodes to can be deleted or added to any piece of track from the right hand mouse button menu. This allows you to extend or change the direction of any piece of track without starting routing again from scratch



Press F3 and you can add additional copper pads. This is useful if you wish to add solder points where pieces of wire enter the PCB for example.

To be able to select an object again at any point after pressing F2 or F3, make sure the arrow icon (select tool) is chosen from the top tool bar.

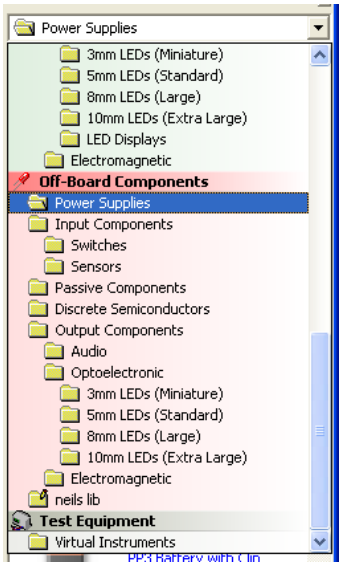
By default when using F2 or F3, the tool snaps to a predefined grid to make it easier to connect between components. Sometimes however, you may need to turn this feature off. Right click on an empty area of the screen. Select Grid/Snap & make sure that there is no √ symbol next to *Snap to Grid*. Try routing with grid snap turned off.

Use these tools to create the smallest PCB possible for your circuit

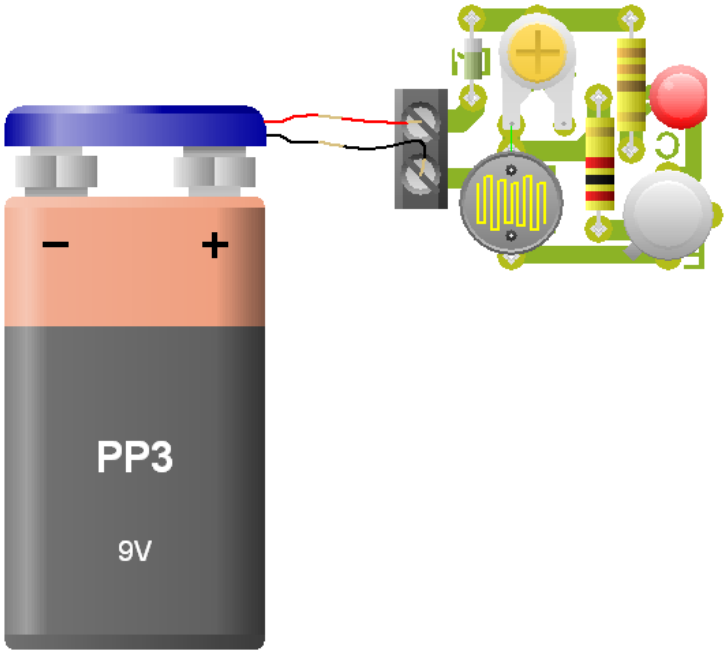
6. **PCB Simulation**

We now need to make sure that our PCB performs correctly. However, the components on the PCB don't represent the full system. Some components might exist off the PCB and be mounted in the product packaging (speakers buzzers, LEDs etc) In this case, only the battery exists off the PCB.

In the *Real World* view, in the component gallery on the right side of the screen, go down to *Off Board components* & select *Power Supplies*



Add a 9v battery to the *Real World* view and connect it up by clicking on the terminals to create flying leads



We now have a complete system and are ready to simulate.

Make sure the PCB based system performs in the same way as the schematic.

Things to Consider

1. When routing around corners try and use 45^{o} angled lines
   Diagonal lines get from A to B in the shortest distance and so you can pack more interconnect in elsewhere
2. Try and put as much copper on the board as possible
   a. Thicker lines means lower tracker resistance which means better performance
   b. Thicker lines make manufacturing defects less problematic
   c. Bigger pads make it easier to solder
   d. Less copper to etch away reduces manufacturing costs and reduces environmental impact
3. Make provision in your PCB design for the PCB to be bolted down to the casing

A prize exists for the smallest working PCB.