Introduction

This manual describes the use of PCB design software, its basic principles, and few application circuits.

The order of chapters follows the typical process from drawing a schematic circuit to a ready-to-use layout.

Electronics have an exponentially growing influencing our lives. Most of the things used by an individual in day to day life include the contribution of electronics to some extend. PCB's have definitely contributed in a significant manner as a means of interconnection of various electronic components. The field of PCB designing is a mutual contribution of wide range of various other fields. For example, the knowledge of mechanics, chemistry and electronics design, production and process management is an essential need for the construction of any particular PCB. The PCB is widely used in fabrication of electronics as well as electrical circuits since they provide a bae for the connections amongst various components used in that particular circuit.

The PCB based circuits are highly reliable as they consist of fewer inter-connection wires. The testing and repairing of components are a lot easier in these (PCB based) circuits.

What is Printed circuit board (PCB)?

Printed circuit boards are dielectric substrates with metallic circuitry photo chemically formed upon that substrate. There are thee major classifications:

1- Single sided boards

2- Double sided boards

3- Multi layer boards

Steps to PCB design using Express PCB/Orcad.

1. Design circuit using schematic entry package (Capture). 2. Generate netlist for PCB package.
2. Import netlist into PCB package (LayoutPlus). 4. Place components, route signals.
3. Generate machining (Gerber) files for PCB plant.

This document is a 'quick start', describing some of the most commonly used operations for PCB design using Orcad. For more details see on-line help and also the pdf manuals which are usually in Program Files\Orcad\Document. These pdf files seem generally much more comprehensive than the on-line help.

Schematic Design

- Use Capture to enter your design. Multiple schematic pages for same design can be used.
- Tip: label nets you may want to locate at the pcb stage – net names are carried through to the pcb design process.
- Select project in project window (as opposed to schematic window), select Design Rule Check for Tools menu. Correct any errors in design.
- Select project in project window, select Create Netlist from Tools menu. Choose Layout tab (to generate Layout compatible netlist), generate netlist.
  Choose units (English or metric) compatible with what you will use in your pcb design.

PCB Layout

- Run LayoutPlus. Choose File/New.
- Select a ‘‘technology file’’ appropriate for your design. These are in Program Files\Orcad\Layout_Plus\data and set defaults for things like track spacing, hole sizes etc. Some examples:

  1BET_ANY.TCH – allows single track between pins on standard DIP; **2BET_SMT.TCH** – for surface mount and mixed smt/through hole designs, 2 tracks between pins of standard DIP;
  3BET_THR.TCH – through hole boards, up to 3 tracks between pins.

  You're best using 1bet_any.tch if at all possible, since this is the least demanding pcb technology.

- Choose your netlist file (.mnl extension). If the units (English/metric) are not the same you won't be able to load it. Just go back to Capture and generate the netlist again with the right units.
- If some of your components chosen from the Orcad Capture libraries did not have PCB footprints associated with them you will get ‘‘Cannot find footprint for...’’ messages. If this happens, choose ‘‘link existing footprint to component’’. Browse footprint libraries to find the required footprint (preview of footprint shown on screen). You can often guess footprints from names.

  Examples: TM = through hole mounted (as opposed to surface mount) BCON100T = block connector, 0.1’’ pitch, through hole BLKCON.100/VH/TM1SQ/W.100/3 = block connector, 0.1’’ pitch, vertical (as opposed to right angle), through hole, pin 1 square pad, width 0.1’’, 3 pins. Library DIP100T = dual in line packages, through hole, 0.1’’ between pins.

  If you can't find the right footprint then you'll need to make your own. See ‘‘Creating a new Footprint’’ at the end of this document.

Draw Board Outline

- Click obstacle toolbar button.
- Somewhere in design, right click, select new.
- Right click again, select properties.
- Select: board outline, width = 50 (or as required), layer = global layer, OK.
- Left click to place one corner of board, then right click on successive corners. Draw a board the required size. Right click, select finish when done (only need to do 3 corners, finish will complete the outline). Notice that the dimensions are shown on status bar at bottom of screen as you draw the board – can be

helpful for creating particular board size. (You can do all this later, after you've placed and routed everything if you prefer.)

Choose Layers

- Use spreadsheet toolbar button to see the Layers spreadsheet.
- Enable only the layers you want for routing, set other layers to unused (double click on the spreadsheet entry, select unused routing). For a single sided board you probably want only the ‘‘bottom’’ layer, for double sided you probably want ‘‘top’’ and ‘‘bottom’’, for a 4 layer board you probably want these plus power and ground plane layers. Tip: you can select multiple layers using click with ctrl key, then right click, select properties, then set/clear unused routing to simultaneously enable or disable several layers.

Place Components

- Select ‘‘component’’ tool from toolbar, click on required component and drag it where you want. Right click to see some options, including rotate.
- Auto/place board will attempt to place components automatically for you within board outline. You may want to move components manually as well.

Track Thickness

- To change, select ‘‘nets’’ spreadsheet, double click on required net to set its properties. Net names are inherited from your schematic diagram – explicitly naming nets helps you identify them in the PCB design. For a simple through hole board you probably want about 20mil tracks. For surface mount boards you probably want 10 or 12 mil tracks. Our pcb plant will make 5 mil tracks if you really need them, but there's an increased risk of part of the track being lost in the etching process. 10 or 12 mil can be reliably made. You may want to make power and ground tracks thicker.

Routing

- Automatic routing is ok, but you can manually route as well (use toolbar buttons).
- Make sure you've set the track thicknesses as you want before routing (see above).
- You may want to route power and ground first, especially if it's a 1 or 2 layer board. Use the nets spreadsheet to enable/disable those nets you want to route at any one time. Tip: select Routing Enabled column, right click, disable to disable all nets, then enable the ones you want.
- You may want to give priority to critical nets (those that need shortest paths), to optimally route those. Priority can be selected from the ‘‘nets’’ spreadsheet for each net.
- To automatically route, select Tools, auto, route board. To put everything back to the rat's nest net, tools/auto/unroute board.
- You can auto route just one component by selectingautoroute/component then click on a pin on that component.
- After an autoroute/board is completed, orcad thinks it's finished, and if you run it again (eg to route some more signals that were disabled the first time) it says all sweeps done or disabled and won't run again. To run autoroute again you have to remove "done" from all autoroute passes. Click the spreadsheet toolbar button and select strategy/route pass. Select the whole "enable" colum, right click, select properties. Remove the "done" tick and click OK. Close the spreadsheet and you can now run the autorouter again.

Copper Pour

- Copper pour fills selected unused board area with copper. This allows creation of large ground (and/or power) areas which improves noise properties. Also reduces amount of copper that needs to be etched off the board by manufacturing process.

- Tip: don't do this until you've finished placement and routing.
- Select required layer (eg TOP or BOTTOM).
- Select obstacle tool (toolbar button), right click in design, select new.
- Right click again, select properties
- Select copper pour, net = GND (or as required), OK. (This example would connect copper pour to the GND net.)
- Draw (by left click and drag) the outline for the copper fill.
- Repeat as required for other copper pour areas and/or layers.
- If you want to delete it, select it by using obstacle tool then ctrl left click on the pour. Then press the delete key.

Set Datum (origin)

- From Tool menu select dimension/move datum.
- Left click at the bottom LH corner of your board to set the origin. (Exact placing doesn't matter.)

Generate Machining Files (Gerber files)

- The machining files required to manufacture the PCB are generated by the ‘‘post processor’’. From options menu choose ‘‘Post Processing’’. In the spreadsheet, select the layers you need to manufacture. You need at least the routing layers you have used (eg top, bottom) and the drill information.
- Choose Auto/Post Process to generate the files. The files generated by the post processor are the only ones needed for the PCB plant to make the board.

Tips

- Use the colour toolbar button, click on a colour box and press the –

key to toggle its visibility. With copper pour in place it can be hard to see what you've got.

- Silk Screen - to see it, you need to add it to the colours table. Use Colours tool, right click, new. Select layer SST (silk screen top), rule = default, OK.
- Use manual place (auto place doesn't optimise placement for noise considerations).
- When placing, set critical nets (eg op amp inputs) to a distinctive colour (via nets spreadsheet) so you can easily see them to optimise placement.
- Place connectors first – they need to be in a convenient place (eg near the edge).
- Some components have multiple parts within one package. Place an additional part in the schematic, choosing, for example, the B part. The ‘‘annotate design’’ tool will combine them into one package (same component identifier). Don't forget to connect unused inputs to appropriate places (eg power, ground) in the schematic, particularly for digital circuits.
- You can go back and change your schematic. Then, when you generate the netlist again, be sure to select the box ‘‘Run ECO to Layout’’ (Engineering

  Change Order). The PCB will be appropriately changed (you may need to then tidy things up).

To select an obstacle (eg board outline or copper pour) select the obstacle tool (toolbar button). Hold ctrl key down and left click on obstacle to select it (becomes highlighted – usually white). Or, click on corner of obstacle and drag as required. Or, select obstacle by drawing a box (with obstacle tool selected) which includes some part of the object.

Make sure you do a save fairly often. You can save to a different file name if you want, then you have a partial pcb design you can go back to if you change your mind how to do things. Do a save before trying anything daring. Then if it doesn't work out you can just exit without saving and start again with your previously saved design. Beware: the "undo" option is only occasionally helpful.

The on-line manuals are ok, but more detailed information is in the pdf files contained in the Orcad Family folder (eg information on technology files, strategy files and many more complex operations than those mentioned here).

Creating a New Footprint

An easy way to create a new footprint is to find an existing one that's similar, edit it and save it with a new name.

- Start Layout+ and choose tools/library manager.
- The list of libraries is displayed in the top part of the window. Click on one you think may be useful.
- The list of footprints in that library is now shown in the bottom window. If you click on a footprint it is displayed in the window on the right.
- Browse to a footprint that's close to what you want. (eg Right number of pins but wrong width, or vice versa.)
- To move a pin, choose pin tool, click on pin, move cursor to where you want it. (You can also use the arrow keys.) The coordinates and distance moved are shown in the status bar at the bottom.
- Another, possibly easier way to move a pin to the right place is to edit its properties in the footprints spreadsheet. You can just type in the required x,y coordinate of the pin here. You can also take a copy of a pin (ctrl C) if you need to add pins.
- To move text use the text tool, click on the text and drag it to where you want.

- To change the place outline and detail, select them using the obstacle tool and either delete (delete key) or drag to where you want. If you delete and redraw, make sure it's the right obstacle type (right click, properties). The place outline shows the board space taken – other footprints can't be placed within this outline.
- When you want to save, do a “save as” (don't overwrite the original library

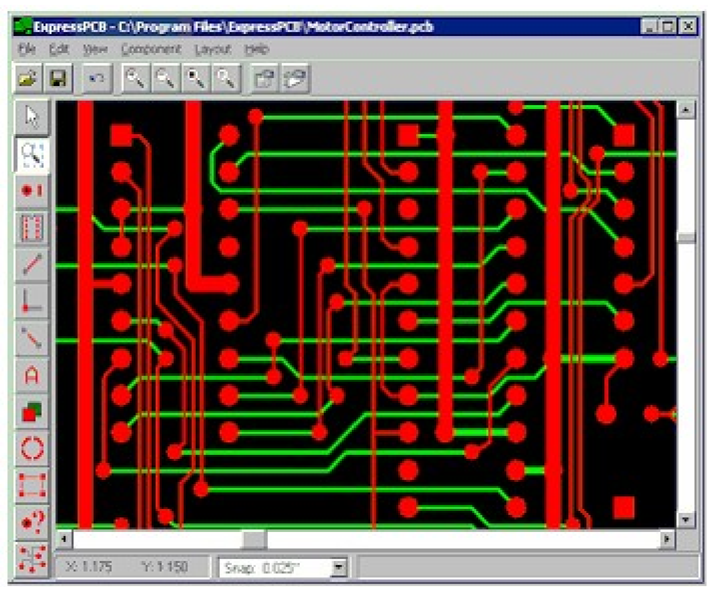
Quick Start Guide To ExpressPCB

There are two parts to ExpressPCB, our CAD software and our board manufacturing service. Our CAD software includes ExpressSCH for drawing schematics and ExpressPCB for designing circuit boards. We also provide a low cost, high quality and fast source for having your boards made. Here is how it works:

1. We recommend that you begin by drawing a schematic using ExpressSCH.
2. Next, use the ExpressPCB program to lay out your PC board. If you link your schematic to ExpressPCB, it will guide you through the wiring process.
3. When your layout is complete, determine the exact cost to have boards made with the Compute Board Cost command.
4. To order the boards, enter your name, address and billing information into ExpressPCB and press the Send button within the Order Boards Via The Internet dialog box.
5. In a few business days (typically 2 or 3) an overnight courier will deliver your PC boards.

Designing a PCB

ExpressPCB is a very easy to use Windows application for laying out printed circuit boards. While not required, we suggest that you draw a schematic for your circuit using the ExpressSCH program. By linking your schematic, ExpressPCB will guide you by highlighting the pins that should be connected together with traces.



Note: ExpressPCB defaults to coordinates in inches. The exact position of the mouse is displayed on the status line.

The Side Toolbar

The toolbar along the left side of the ExpressPCB main window is used to select the editing modes, such as place component and place trace. These are the side toolbar buttons:

Select, drag or set properties of an item in the layout.	Place text in the layout.
Zoom to view a selected area.	Place a rectangle.
Place a pad (either a through hole or surface mount pad).	Place a circle or an arc.
Place a component.	Place a filled power or ground plane.
Place a trace.	Insert an edge segment in the board perimeter.

At the bottom of the side toolbar are 4 additional buttons that toggle on and off features of the display:

Toggle display of silkscreen layer.	Toggle display of bottom copper layer.
Toggle display of top copper layer.	Toggle Snap-to-grid.

The Top Toolbar

When you click on a side toolbar button, additional buttons are added to the top toolbar, specific to the function selected. These buttons are always displayed on top:

Open a schematic file.	Zoom in.	Zoom to previous size.
Save schematic file.	Zoom out.	Display *Options* dialog box.
Undo last operation.	Zoom to see all.	Display the *Component Manager*.

These buttons are added on top as needed:

Move selected items to top layer.	Set rotation up.	Highlight first net.
Move selected items to bottom layer.	Set rotation left.	Highlight previous net.
Select top silkscreen layer.	Set rotation down.	Highlight next net.
Select top copper layer.	Set rotation right.	Highlight last net.
Select bottom copper layer.	Set text weight to normal.	Set text weight to bold.

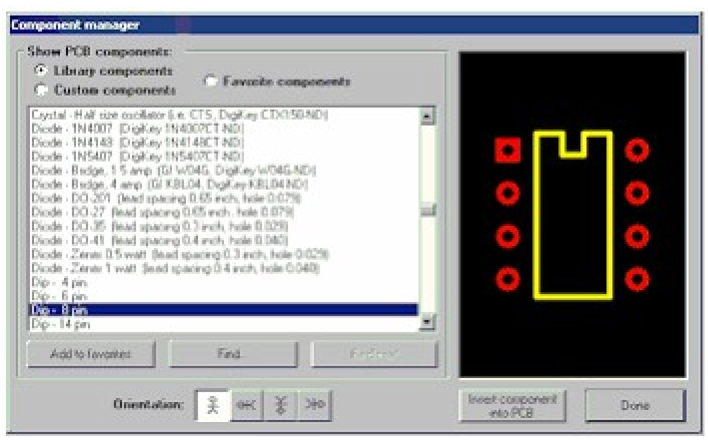
Beginning a New Layout

1. Begin a new layout by running ExpressPCB. If you are designing a four-layer board, select Board properties from the Layout menu and check the 4-Layer option.
2. In the main window, the yellow rectangle defines the perimeter of the PC board. Set the size of your board by moving three of its four corners (the upper left corner is fixed at 0,0). Move the corners by dragging them with the mouse, or by double-clicking them and entering coordinates. Additional corners can be added to the perimeter to change its shape (see ExpressPCB: Changing the Board Perimeter).
3. Select the size of the Default via. In some cases as you place traces, ExpressPCB inserts via (plated-through holes) when a trace changes between the upper and lower layers. When inserted, these via pads are always visible. Set the Default via in the Board properties dialog box.
4. If you have drawn a schematic of your circuit using ExpressSCH, link the schematic file to your circuit board layout using the Link schematic to PCB command found under the File menu.
5. Finally, give your board a name by selecting Save As from the File menu.

Zooming and Panning

The easiest way to move around your layout is with the scroll wheel on the mouse. Turning the wheel zooms in and out. Pressing the wheel and dragging the mouse pans.

The Grid



The Snap-to-grid is an invisible grid that helps you align traces and components. With the Snap-to-grid on, you will notice that as you place objects, they gravitate to a grid boundary. The Snap-to-grid spacing is set from the Options dialog box or from the Snap listbox located on the lower statusbar. To toggle the Snap-to-grid on or off, press the G key or click .

Placing Components

The easiest way to place components in your board layout is to use the Component Manager.

Tip: Use the Find button to search for a PCB footprint by its name.

To place a component:

1.Click the button located on the top toolbar to display the Component Manager.

2. Select one of these categories:· Library components - Components that are included with the program· Custom components - Components that you have drawn· Favorite components - Components or symbols that you have book-marked

3. From the list box, choose the item to insert then select the component's orientation (rotated up, left, down, or right) by clicking one of these buttons:

4. Press the Insert component into PCB button, then drag the component to the desired location.

5. Assign the component's Part ID (such as R1 or U2). To do this, select then double-click on the component to display its Component properties dialog box.

Note: *Setting the Part ID is required if you link your schematic and PCB. In order to mate a schematic component with one on the board layout, the Part IDs must match.*

Placing Pads

Individual pads are used both to build new components and as via. To insert a pad:

1.From the side toolbar, select or press the P shortcut key.

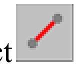
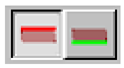
2.Select a pad size from the drop down list box on the top toolbar. There are several pad types: round, square, surface mount and via. Vias are used to pass traces between layers. Regular pads differ from via pads in that ExpressPCB may eliminate a via if the traces connecting to it are all on the same layer.

Tip: *If you cannot find a pad the size you need, a custom pad is easily created. See: ExpressPCB: Making Custom Pads*

3.To place the pad, click on your layout at the desired location.

Placing Traces

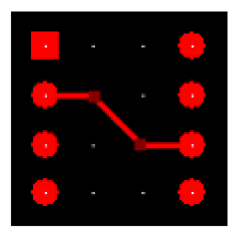
These are the steps to add traces to your layout. If you have drawn a schematic for your circuit, be sure to read the section Linking the Schematic and PCB before you begin.

1. From the side toolbar, select  or press the T shortcut key.
2. Select the trace width from the drop down list box on the top toolbar: A width of 0.010" is a good default for digital and analog signals. For power lines, use traces 0.05" or wider.
3. Select the layer by clicking  or by pressing the L shortcut key.
4. Move the mouse to the trace's first end point and click. Drag the trace to the second end point, then click again. Continue placing trace segments (use the L key to change layers) until you have reached the final end point.

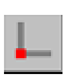
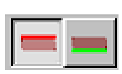
    Tip: *Keep an eye on the statusbar when connecting traces to components. It will display the pin and part number to which the trace is connected.*

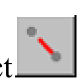
5. As you drag the trace, the L key changes layers, the Del key deletes the previous segment, the + and – keys zoom in and out, the G key toggles the snap-to-grid on and off. The Spacebar sets the trace, and the Esc key cancels it.
6. To complete the operation, press the Spacebar or click right.

Working with Traces

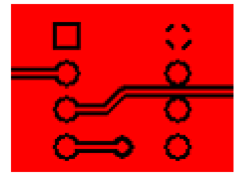


Traces themselves cannot be moved. The path of a trace is determined by the straight line between its two connections. Therefore, to move a trace, you need to connect it to something different or to move what it is connected to.Corners in traces allow them to bend. They are displayed as small square blocks at the ends of traces. A trace with two corners is shown here. In your final PC board layout, the corners will not be included.

Corners can be dragged, inserted or deleted to change the trace's path. To insert a corner, select  then click  to choose the layer on which the corner will be placed. Next click on the trace at the point where you want to insert the corner.

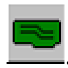
To disconnect a trace and reconnect it elsewhere, select  from the side toolbar. Next, click on the trace near the point you want to disconnect, then drag the trace to a new pin.

Placing Filled Planes



Filled Planes are used to add ground or power planes to a circuit, usually on double-sided boards. They can be placed on the top or bottom layers. The perimeter can have the shape of any polygon and the interior is automatically insulated from traces and pads.

Note: *The Filled Plane feature is not supported in Windows 95, 98 or ME.*

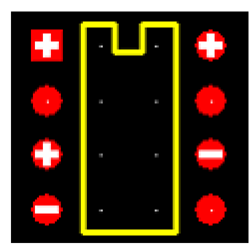
To place a Filled Plane, select , then on the top toolbar choose the Draw filled plane option, along with the layer. Next use the mouse to draw the perimeter. Do this by clicking the left mouse button at each corner, then clicking right after you have placed the last corner.Connect a pad to a plane by right clicking on the pad. In the popup menu, select one of the Top layer pad shape or Bottom layer pad shape options.

Important: After creating a plane, you will want to inspect it for possible problems. Learn how here: ExpressPCB: Placing Power and Ground Planes

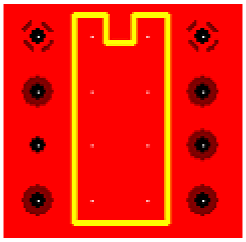
Tip: *After adding a Filled Plane, you may want to lock or hide it using the Options dialog. If planes are hidden, they still will be included when the boards are manufactured.*

Designing a Four-Layer PCB

Begin a four-layer design by selecting the 4-Layer option in the Board properties dialog box. This adds two inner layers to your layout, a Power plane and a Ground plane.



Any through-hole pad in a layout can be connected to either the Power or Ground internal planes by right clicking on the pad. In the popup menu, select one of the Power inner layer pad or Ground inner layer pad options. Connections to the Power plane are displayed with a + over the pad. Ground plane connections are marked with a - symbol.

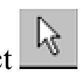


To look at the actual inner layer planes, select *View Power layer or View Ground layer* in the *Options* dialog box. This example shows pins 1, 3, and 8 connected to the Power plane. The other pads are isolated.You will notice that pins 1 and 8 are connected with *Thermal* pads, while pin 3 is connected with a Solid pad. Typically *Thermals* are used with pads connecting to a plane because they improve the solderability of the pad, reducing the chance of a "cold solder joint".

When viewing inner layers, you can draw split planes and keep out areas.

Setting Properties of an Item

All the items in your layout have properties. For example, the properties of a pad include the pad type and its XY coordinates.

To view or change the properties of an item, select  then double click on the item.

Copying, Deleting and Moving Items

ExpressPCB uses standard Edit commands for Copy, Cut and Paste. To copy an item or several items, first select them. From the Edit menu choose Copy, then from the Edit menu choose Paste.

Deleting items from your layout is as easy as selecting them and pressing the Del key.

There are three ways most items can be moved. Typically, items can be moved by selecting and dragging them. They can also be moved with the arrow keys by selecting them and then pressing Ctrl-right, Ctrl-left, Ctrl-up or Ctrl-down. Alternately, an item can be moved by changing the coordinates set in its properties dialog box.

Tip: To copy a group of items from one layout file to another, select and copy the items of interest into the clipboard. Load the second file into ExpressPCB and paste. It is not possible to copy items between two programs running at the same time.

Changing the Layer of Items

To move traces, rectangles and text between the top and bottom copper layers, select them with the mouse then click to move the selection to the top layer; click to move it to the bottom.

Linking the Schematic and PCB

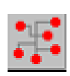
Running the traces from component to component is simplified if the schematic and PCB are linked. Embedded in Express SCH schematic files are wiring lists (called Net Lists). When linked, the Express PCB program highlights which pins need to be wired together. Before linking your schematic, check it for errors. Run the command Check schematic for net list errors from the File menu of Express SCH. If errors are reported, correct them, then save the schematic file. For more information, see: <u>Express SCH: Checking the Schematic for Errors</u>

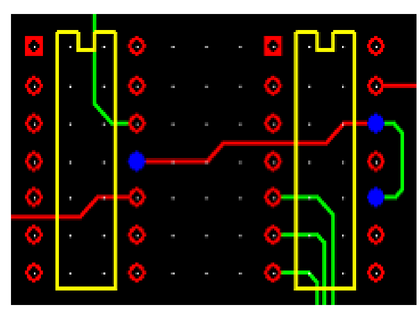
Next, for every part referenced in your schematic, you need to insert the component footprint for that part in your PCB design. Also, the Part ID values for each footprint must be set. To learn how to insert the footprints and set the IDs, see: <u>Placing Components</u>

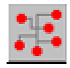
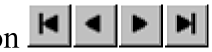
Note: The Part IDs are used to match a schematic component with its PCB footprints when highlighting a net. (i.e. a Part ID of R12 in the schematic will be matched with a PCB footprint that also has the Part ID set to R12).

Now link your schematic and PCB files. From the File menu of the ExpressPCB program, select the command Link schematic to PCB.

Once linked, ExpressPCB can show you which pins should be connected together by highlighting them in blue.

To highlight a "Net", select  from the side toolbar, then click on one of the pads of a component. This highlights the other pads that should be wired to the pad you've selected. If the schematic has nothing connected to that pin, then no pads are highlighted.

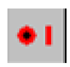


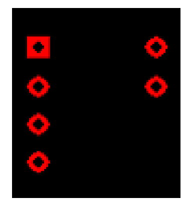
You can step through all of the "Nets", from first to last, to visually inspect your PCB for missing connections. This is done by selecting  and then clicking on  to navigate through all of the nets:

Tip: The ExpressPCB program reloads the linked schematic file every time the .PCB file is opened. If you run ExpressSCH and ExpressPCB at the same time, you can update the link by first saving the schematic within ExpressSCH, then selecting Refresh link to schematic from the File menu in ExpressPCB.

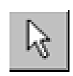
Making Custom PCB Components

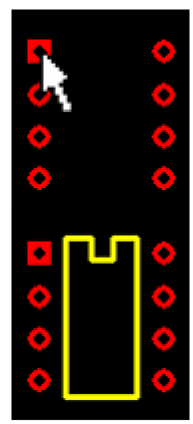
Express PCB includes many components that you can use to create your PCB. However if you need a footprint not found in our library, you can easily build your own. Here is how:

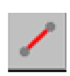
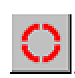
1. First add the pads by selecting  and choosing a pad type on the top toolbar. Carefully position the pads with the correct spacing. It may be helpful to change the Snap-to-grid spacing in the Options dialog.

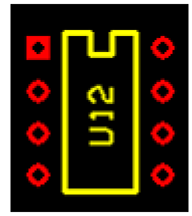


   Tip: If you cannot find a pad the size you need, create a custom pad. See: ExpressPCB: Making Custom Pads

2. Assign each pad a Pin number. This must be done if you link your PCB to its schematic. Assign pin numbers by selecting  and then double clicking on each pad to display its Pad Properties dialog box. In the Pin number field, enter the pin number.



3. Draw the component outline for the new part on the silkscreen layer. Draw straight lines by selecting  or circles and arcs using . Be sure to select the silkscreen layer before drawing the outline by clicking . The recommended line width for drawing component outlines is 0.012".
4. Group everything together to combine it into a single component as follows. Select all of the objects in your new part by clicking  and then dragging the mouse to enclose them. From the Component menu, choose the command Group



to make PCB component. Next, double click the new component to assign its Part ID (i.e. U12).

5. Save the component to add it to the Custom components list. Do this by selecting the part with the mouse and then choosing Save custom component from the Component menu.

FYI: ExpressPCB stores every component in its own file. These files have the .P extension and are saved in the directories:

- Program Files\ExpressPCB\PCBComponents_Library·
- Program Files\ExpressPCB\PCBComponents_Custom

Modifying Library Components

Sometimes it is easiest to build a new component by starting with one from our library, for example to change the pad sizes. This is done by first inserting the library component into your layout. Next select it with the mouse then choose the Ungroup PCB component command from the Component menu. Now select only the pads and double click on one. In the Pads Properties dialog, change the pad size. Finally, group the component back together as described above to create the new part. Your changes will NOT affect the original part in the ExpressPCB library.

Having PC Boards Made

When you have completed your layout, the next step is to have the boards made. Before ordering, you can determine the cost using the Compute Board Cost command from the Layout menu.

To order your boards, connect your computer to the Internet then select the Order Boards Via The Internet command from the Layout menu. Fill in the forms that follow and press the SEND button. Your order is sent to ExpressPCB for manufacturing.For more information, see the online help section Having PC Boards Made.

Circuit Board Design Tips

The engineers at ExpressPCB have assembled a few general rules of thumb that can help the beginner design their first printed circuit board. These tips are not specific to using our

CAD software, but instead provide an overview to help explain how to position the components on the board and how to wire them together with traces.

Placing Components:

Generally, it is best to place parts only on the top side of the board.When placing components, make sure that the snap-to grid is turned on. Usually, a value of 0.025" for the snap grid is best for this job.First place all the components that need to be in specific locations. This includes connectors, switches, LEDs, mounting holes, heat sinks or any other item that mounts to an external location.Give careful thought when placing component to minimize trace lengths. Put parts next to each other that connect to each other. Doing a good job here will make laying the traces much easier.Arrange ICs in only one or two orientations: up or down, and, right or left. Align each IC so that pin one is in the same place for each orientation, usually on the top or left sides.Position polarized parts (i.e. diodes, and electrolytic caps) with the positive leads all having the same orientation. Also use a square pad to mark the positive leads of these components.You will save a lot of time by leaving generous space between ICs for traces. Frequently the beginner runs out of room when routing traces. Leave 0.350" - 0.500" between ICs, for large ICs allow even more.Parts not found in the component library can be made by placing a series of individual pads and then grouping them together. Place one pad for each lead of the component. It is very important to measure the pin spacing and pin diameters as accurately as possible. Typically, dial or digital calipers are used for this job.After placing all the components, print out a copy of the layout. Place each component on top of the layout. Check to insure that you have allowed enough space for every part to rest without touching each other.

Placing Power and Ground Traces:

After the components are placed, the next step is to lay the power and ground traces. It is essential when working with ICs to have solid power and ground lines, using wide traces that connect to common rails for each supply. It is very important to avoid snaking or daisy chaining the power lines from part-to-part.One common configuration is shown below. The bottom layer of the PC board includes a "filled" ground plane. Large traces feeding from a single rail are used for the positive supply.

Placing Signal Traces:

When placing traces, it is always a good practice to make them as short and direct as possible.Use vias (also called feed-through holes) to move signals from one layer to the other. A via is a pad with a plated-through hole.Generally, the best strategy is to layout a board with vertical traces on one side and horizontal traces on the other. Add via where needed to connect a horizontal trace to a vertical trace on the opposite side.A good trace width for low current digital and analog signals is 0.010".Traces that carry significant current should be wider than signal traces. The table below gives rough guidelines of how wide to make a trace for a given amount of current.

0.010"	0.3 Amps
0.015"	0.4 Amps
0.020"	0.7 Amps
0.025"	1.0 Amps
0.050"	2.0 Amps
0.100"	4.0 Amps
0.150"	6.0 Amps

When placing a trace, it is very important to think about the space between the trace and any adjacent traces or pads. You want to make sure that there is a minimum gap of 0.007" between items, 0.010" is better. Leaving less blank space runs the risk of a short developing in the board manufacturing process. It is also necessary to leave larger gaps when working with high voltage.When routing traces, it is best to have the snap-to-grid turned on. Setting the snap grid spacing to 0.050" often works well. Changing to a value of 0.025" can be helpful when trying to work as densely as possible. Turning off the snap feature may be necessary when connecting to parts that have unusual pin spacing.It is a common practice to restrict the direction that traces run to horizontal, vertical, or at 45 degree angles.When placing narrow traces, 0.015" or less, avoid sharp right angle turns. The problem here is that in the board manufacturing process, the outside corner can be etched a little more narrow. The solution is to use two 45 degree bends with a short leg in between.It is a good idea to place text on the top layer of your board, such as a product or company name. Text on the top layer can be helpful to insure that there is no confusion as to which layer is which when the board is manufactured.

Checking Your Work:

After all the traces are placed, it is best to double check the routing of every signal to verify that nothing is missing or incorrectly wired. Do this by running through your schematic, one wire at a time. Carefully follow the path of each trace on your PC layout to verify that it is the same as on your schematic. After each trace is confirmed, mark that signal on the schematic with a yellow highlighter.Inspect your layout, both top and bottom, to insure that the gap between every item (pad to pad, pad to trace, trace to trace) is 0.007" or greater. Use the Pad Information tool to determine the diameters of pads that make up a component. Check for missing vias. Express PCB will automatically insert a via when changing layers as a series of traces are placed. Users often forget that via are not automatically inserted otherwise. For example, when beginning a new trace, a via is never inserted. An easy way to check for missing via is to first print the top layer, then print the bottom. Visually inspect each side for traces that don't connect to anything. When a missing via is found, insert one. Do this by clicking on the Pad in the side toolbar, select a via (0.056" round via is often a good choice) from the drop down listbox, and click on the layout where the via is missing. Check for traces that cross each other. This is easily done by inspecting a printout of each layer. Metal components such as heat sinks, crystals, switches, batteries and connectors can cause shorts if they are placed over traces on the top layer. Inspect for these shorts by placing all the metal components on a printout of the top layer. Then look for traces that run below the metal components.

1. Street light circuit.

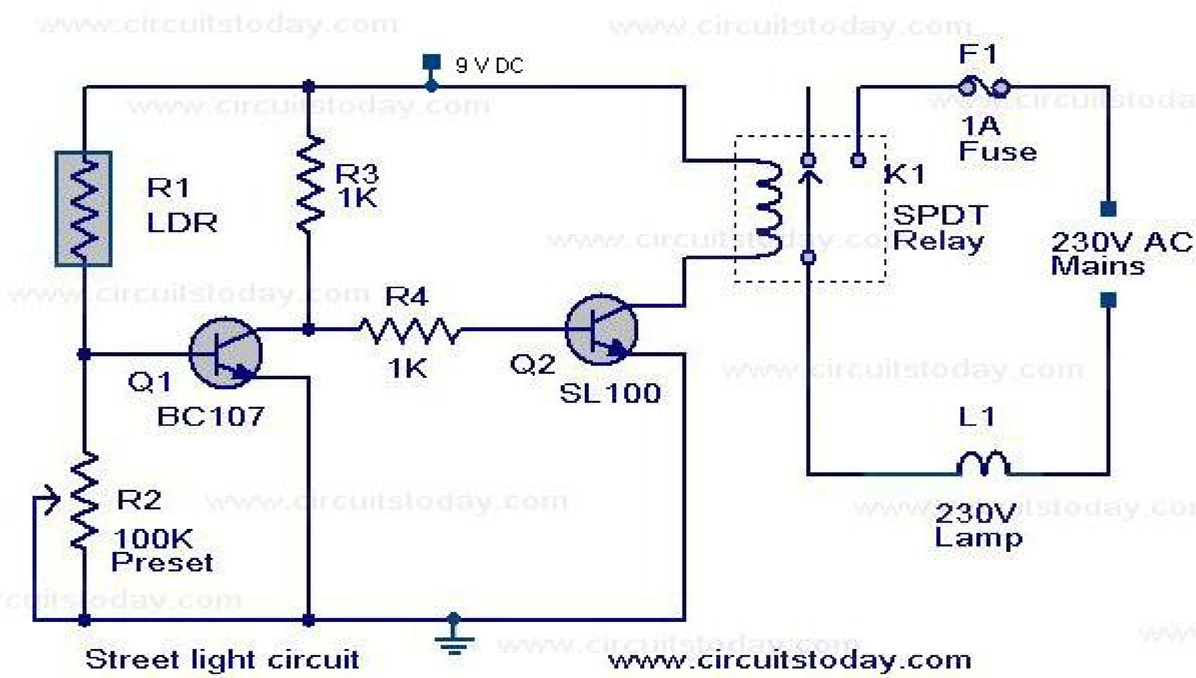
Description.

The circuit diagram present here is that of a street light that automatically switches ON when the night falls and turns OFF when the sun rises. In fact you can this circuit for implementing any type of automatic night light.

The circuit uses a **Light Dependent Resistor (LDR)** to sense the light .When there is light the resistance of LDR will be low. So the voltage drop across POT R2 will be high. This keeps the transistor Q1 ON. The collector of Q1(BC107) is coupled to base of Q2(SL100). So Q2 will be OFF and so do the relay. The bulb will remain OFF.

When night falls the resistance of LDR increases to make the voltage across the POT R2 to decrease below 0.6V. This makes transistor Q1 OFF which in turn makes Q2 ON. The relay will be energized and the bulb will glow.

Circuit diagram with Parts list.

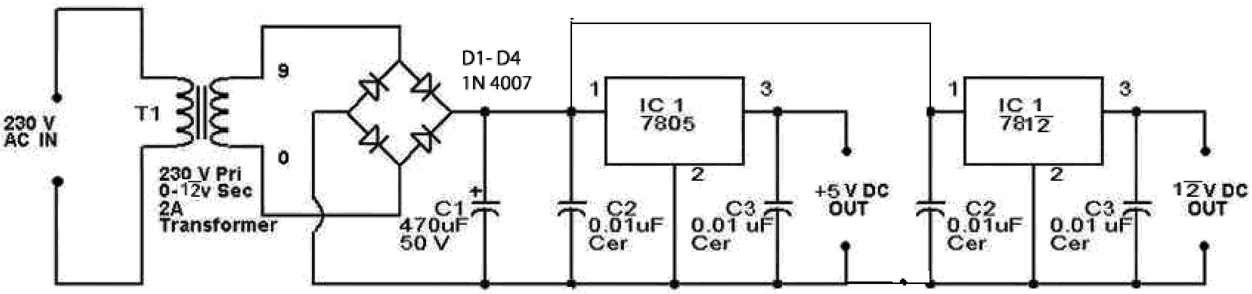


Notes.

- **P**OT R2 can be used to adjust the sensitivity of the circuit.
- **Y**ou can use bulb of any wattage, provided that relay should have the sufficient rating.
- **T**he circuit can be powered from a regulated 9V DC power supply.
- ***Click Here!*** to get the power supply circuit for this project.
- The relay K1 can be a 9V SPDT relay

2.Regulated Power Supply(5V &12V)

Circuit diagram:



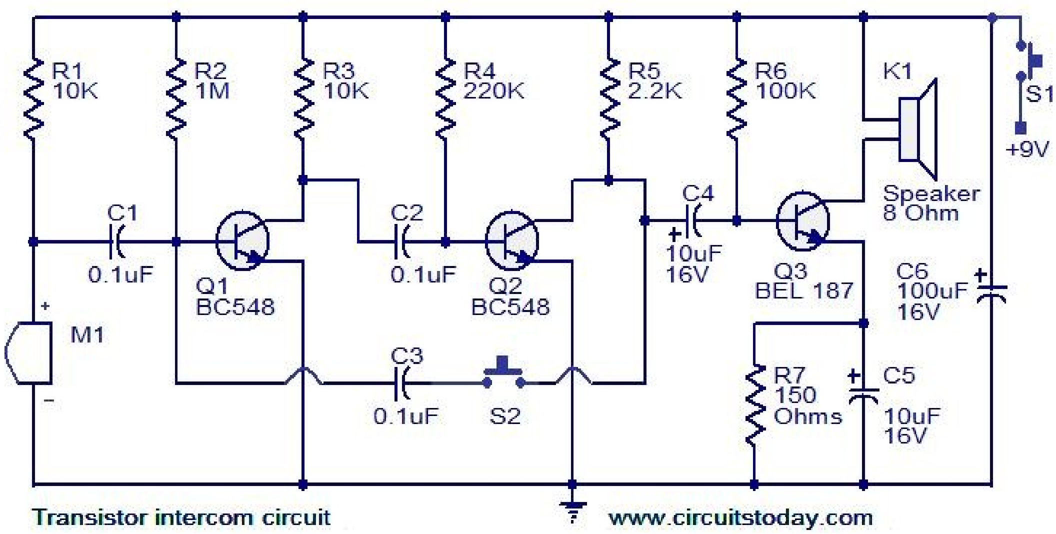
This is a small +5V regulated power supply circuit. In that case here we used 7805 and 7812 Voltage Regulator IC's. 7805 is a +5 Volt regulator IC and 7812 is a +12 Volt regulator IC from 78xx chips family. The circuit has internal current limiting and thermal protection capacity. A 12V 2A steps down transformer is used to covert 230V to 12V from mains. Here used a bridge rectifier made by four 1N 4007 diode to convert AC-DC . 470uF 50v as C1 is used for filtering. This circuit is very easy to build. For good performance recommended input voltage 8V-18V. If over 400mA current is needed at output then use a **heat sink** with the 7805 IC.

3. Transistor intercom circuit.

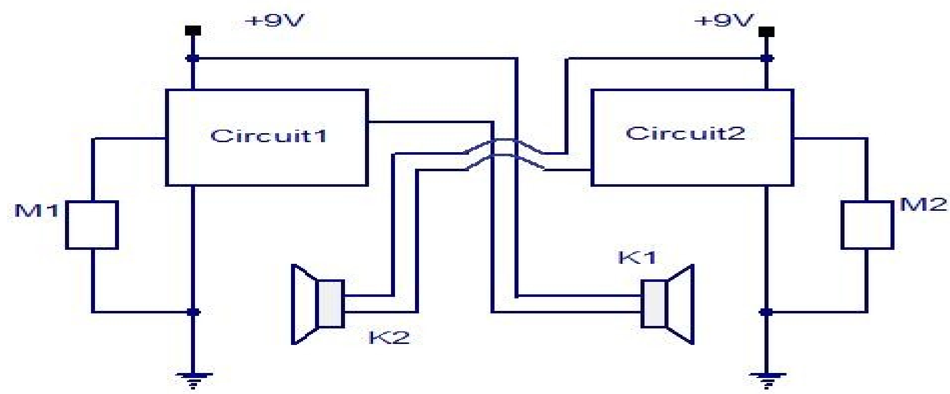
Description.

Here is a simple but effective intercom circuit that is based fully on transistors. The circuit is based on a three stage RC coupled amplifier. When the pushbutton S2 is pressed, the amplifier circuit wired around T1 & T2 becomes an astable multivibrator and starts producing the ringing signals. These ringing signals will be amplified by the transistor T3 to drive the speaker. When the push button S2 is released the circuit will behave as an ordinary amplifier and you can talk to the other side through it.

To construct a two way intercom, make two identical copies of the circuit given below and connect it according to the given connection diagram. The stand by current consumption of this circuit is around 20mA.



Connection diagram.



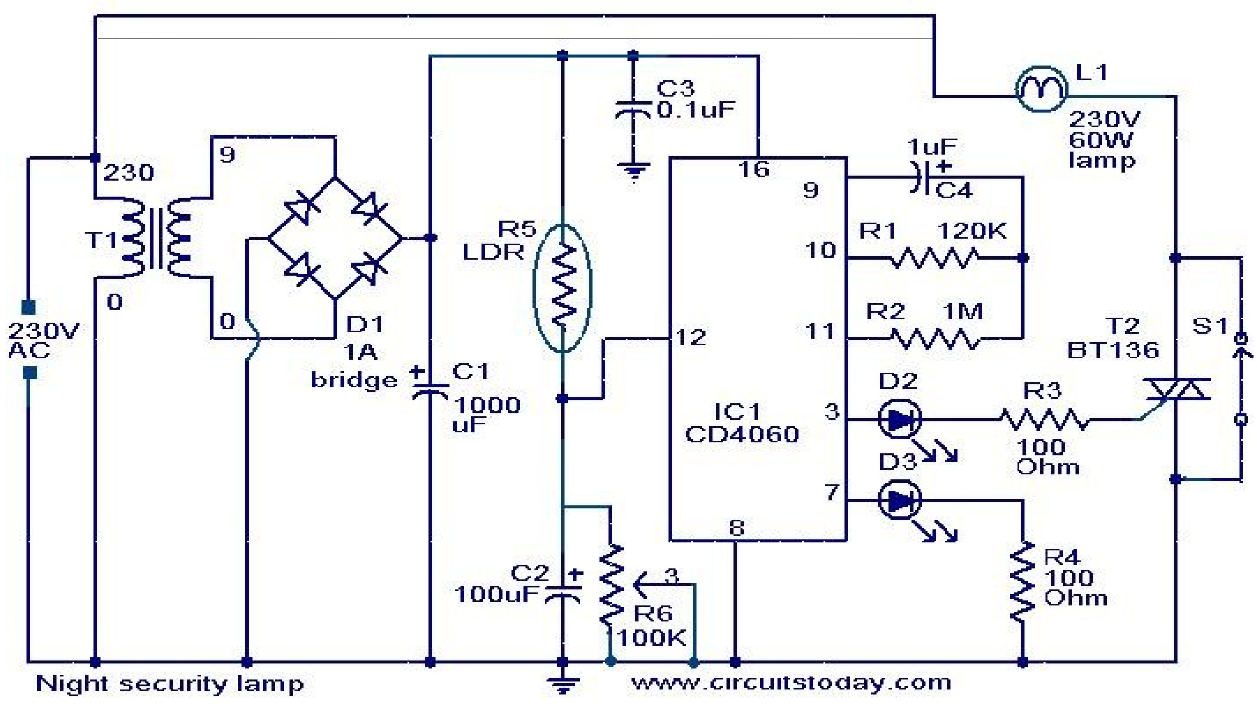
Transistor intercom connection diagram

4. Night security light

Description.

Here is a simple circuit switches on a light around 2 hours after midnight, the time at which most of the robberies taking place.

This simple circuit is build around a CMOS IC 4060 to obtain the required timing. During day time the **LDR** has low resistance and keeps the pin 12 of the IC1 high, preventing the IC1 from oscillating. When it is dark the LDR resistance becomes high and the pin 12 of IC1 becomes low and the IC starts oscillating, which indicated by the flashing of LED D3.The values of the timing components R1, R2, C4 are so selected that the out put pin3 of IC1 goes high after 8 hours. That means the high output drives the triac to switch on the lamp around 2'O clock. At morning, the LDR resistance drops and the pin 12 of IC1 goes high and stops the oscillation, making the lamp OFF. The switch S1 can be used to manually ON the lamp. The capacitor C2 prevents false triggering.



Notes.

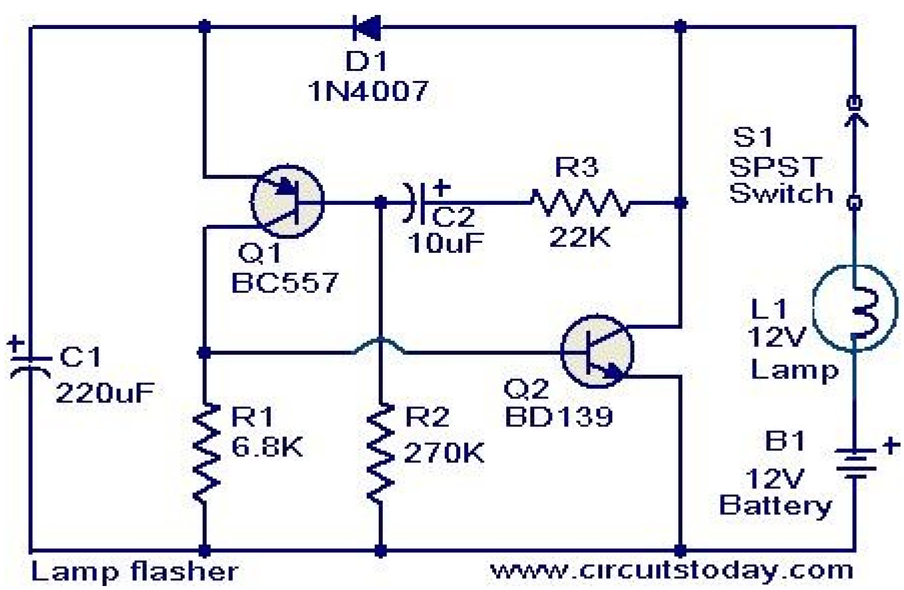
- Assemble the circuit on a good quality PCB or common board.
- The LDR can be general purpose LDR.
- The light sensitivity can be adjusted using the preset R6.
- The IC1 must be mounted on an IC holder.

5. Lamp flasher circuit

Description.

This simple circuit that can be used to flash incandescent lamps up to 10W power rating.The circuit is ideal for making flashing beacons on automobiles and other applications like that.The circuit is nothing but an astable multi vibrator based on Q1&Q2 (BC557&BD139).The capacitor C1 is the main timing element which determines the flashing rate of the circuit.The switch S1 can be used as an ON/OFF switch.

Circuit diagram with Parts list.



Notes.

- Power the circuit from a 12 V battery or 12V DC power supply.
- Assemble the circuit on a goood quality PCB or common board.
- A 12 v , 10W incandescent lamp can be used as the load.
- All capacitors must be rated 15V

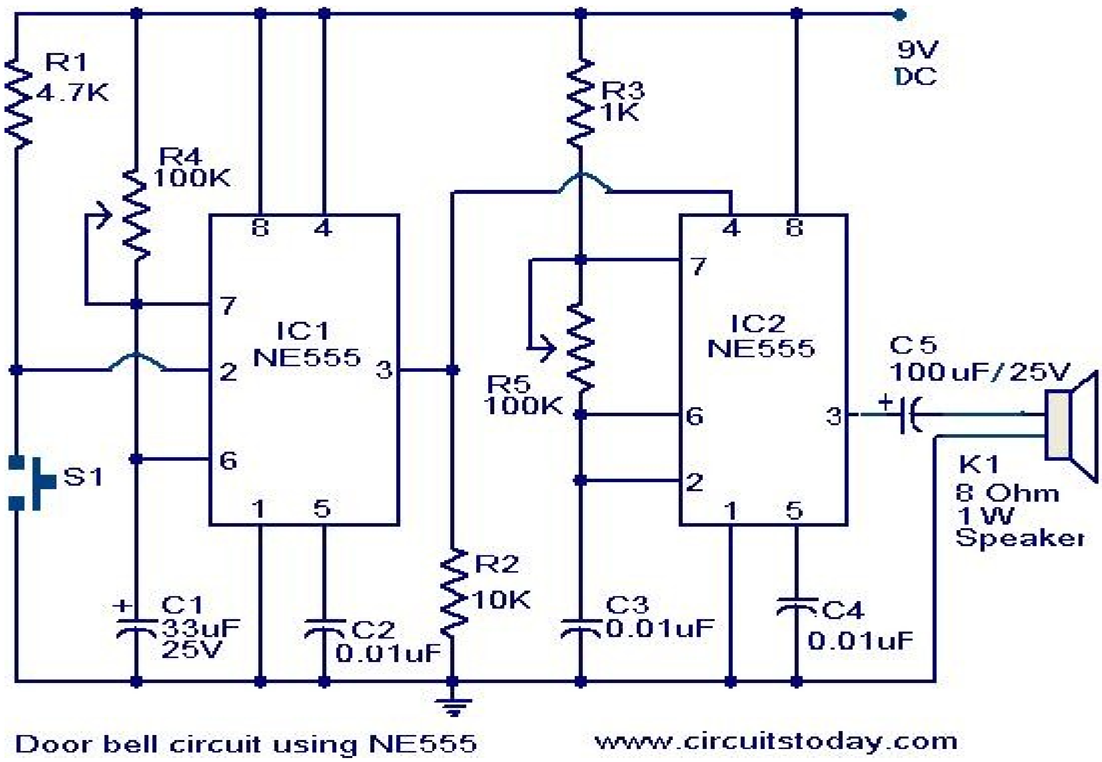
6. Door bell circuit using NE555

Description.

The main part of this doorbell circuit are two NE555 timer ICs.When some one presses switch S1 momentarily, the loud speaker sounds a bell tone as long as the time period of the monostable multivibrator built around IC1.

When the switch S1 pressed, IC1 is triggered at its pin 2 and output pin 3 goes high for a time period previously set by the values of POT R4 and POT R5.When the output ofIC1 goes high it resets IC2 and it starts to oscillate to make a bell sound through the speaker.The IC2 is configured as an astable multivibrator whose oscillation frequency can be varied with the help of POT R5.By adjusting the values of R4 & R5, modifications on the tone are possible.

Circuit diagram **with Parts list.**



Notes.

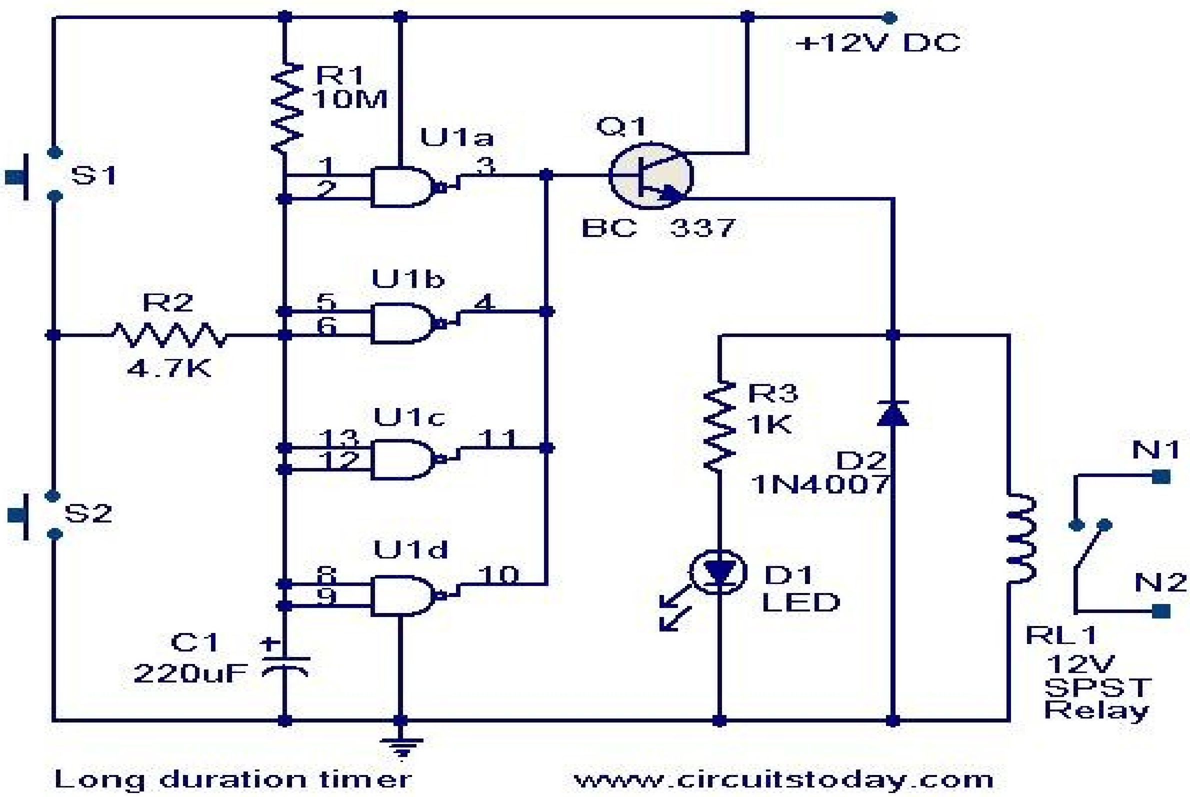
- The circuit has to assembled on a good quality PCB or common board.
- The IC1 & IC2 has to be mounted on IC holders.
- Power the circuit from a 9V battery or 9V DC power supply.
- Switch S1 is push button switch.

7. Long duration timer circuit.

Description.

This timer circuit can be used to switch OFF a particular device after around 35 minutes. The circuit can be used to switch OFF devices like radio, TV, fan, pump etc after a preset time of 35 minutes. Such a circuit can surely save a lot of power. The circuit is based on quad 2 input CMOS IC 4011 (U1).The resistor R1 and capacitor C1 produces the required long time delay. When pushbutton switch S2 is pressed, capacitor C1 discharges and input of the four NAND gates are pulled to zero. The four shorted outputs of U1 go high and activate the transistor Q1 to drive the relay. The appliance connected via the relay is switched ON. When S2 is released the C1 starts charging and when the voltage at its positive pin becomes equal to ½ the supply voltage the outputs of U1 becomes zero and the transistor is switched OFF. This makes the relay deactivated and the appliance connected via the relay is turned OFF. The timer can be made to stop when required by pressing switch S1.

- **Circuit diagram with Parts list.**



8. Water level alarm circuit

Description.

Here is a simple water level alarm circuit using 555 timer that will produce an audible alarm when the water level reaches a preset level.The circuit can be powered of a 3V battery and is very handy to use.

The circuit is based on an astablemultivibrator wired around IC1 (NE 555).The operating frequency of the astable multivibrator here will depend on capacitor C1, resistances R1,R2 and the resistance across the probes A&B. When there is no water up to the probes, they will be open and so the multivibrator will not produce oscillations and the buzzer will not beep. When there is water up to the level of probes, some current will pass through the water, the circuit will be closed to some extend, and the IC will start producing oscillations in a frequency proportional to the value of C1,R1,R2 and the resistance of water across the probes. The buzzer will beep to indicate the presence of water up to the level of the sensing prob.

Circuit

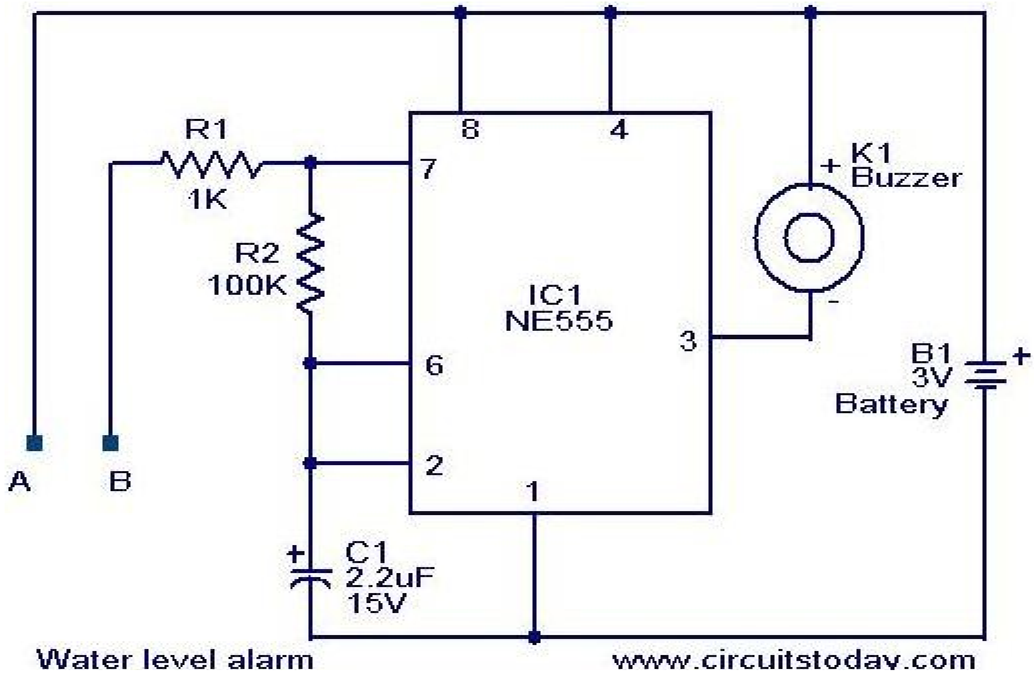


Diagram with Parts list.

Notes.

- The circuit can be powered of a 3V battery.
- Assemble the circuit on a good quality PCB or common board.
- The probes can be made of two insulated copper Aluminiun wires.
- Place the probes at the position where you have to sense the level.

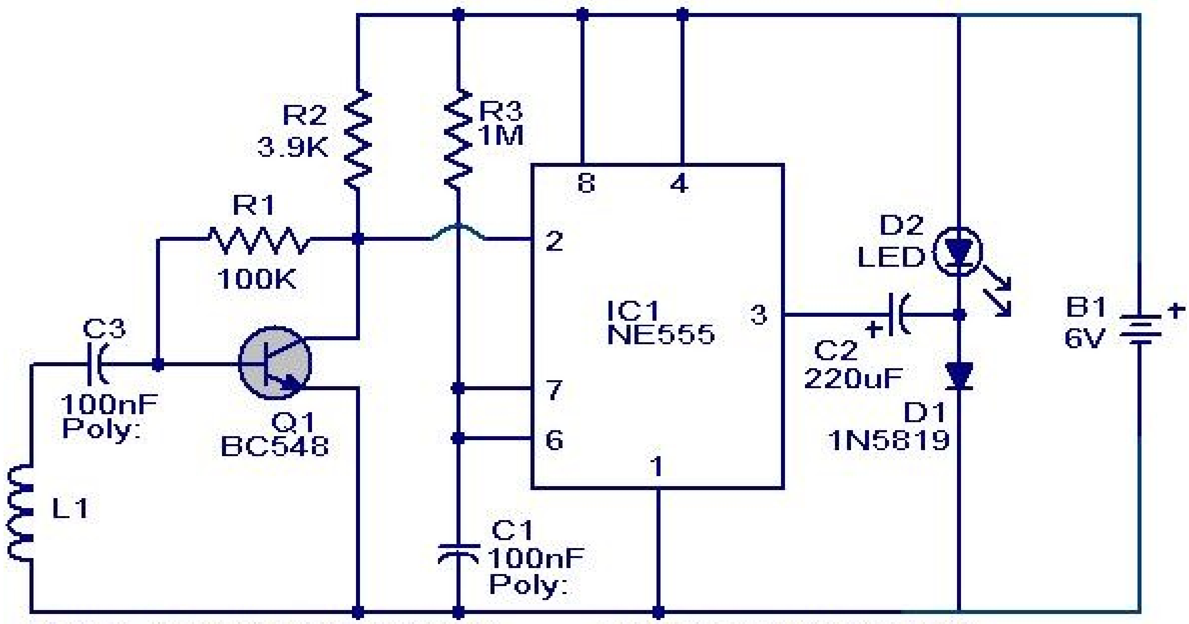
9. Mobile incoming call indicator

Description.

This circuit can be used to escape from the nuisance of mobile phone rings when you are at home. This circuit will give a visual indication if placed near a mobile phone even if the ringer is deactivated.

When a call is coming to the mobile phone, the transmitter inside it becomes activated. The frequency of the transmitter is around 900MHz.The coil L1 picks up these oscillations by induction and feds it to the base of Q1. This makes the transistor Q1 activated. Since the Collector of Q1 is connected to the pin 2 of IC1 (NE555) , the IC1 is triggered to make the LED connected at its output pin (pin 3) to blink. The blinking of the LED is the indication of incoming call.

Notes.



Mobile incoming call indicator www.circuitstoday.com

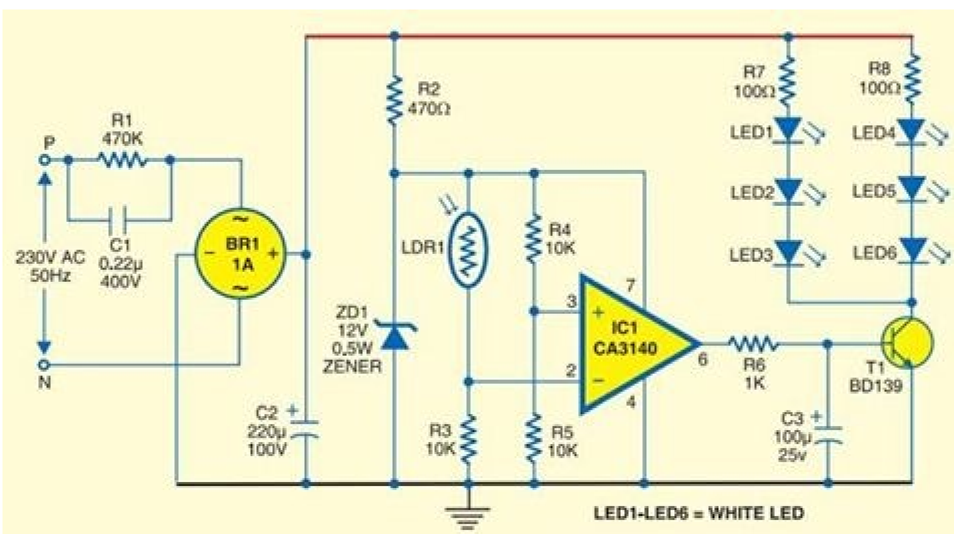
- The coil L1 can be made by making 150 turns of 36 SWG enameled copper wire on a 5mm dia plastic former. Or you can purchase a 10 uH coil from shop if available.
- The circuit can be powered from a 6V battery.
- Assemble the circuit on a good quality PCB.
- C1 & C3 are to be polyester capacitors.
- The electrolytic capacitor C2 must be rated 10V.

10. Strip LED Lamp Circuit

Strip LEDs are available in different colours powered by direct current (DC) source. These LEDs are available as surface mount devices with current limiting resistors. Usually there are 300 LEDs in a 5-metre strip. The strip can be cut into pieces so that the bits having three or four LEDs can be used with 12V DC source. The circuit given here uses the strip LEDs to make an automatic white LED lighting source. The circuit is powered by a capacitor power supply connected to AC mains. Capacitor C1 drops the 230V AC, which is further rectified by the bridge rectifier module and is made ripple-free by C2. Zener diode (ZD1) provides 12V DC to the comparator circuit.

Resistor R1 is important in the power supply as it provides discharge path to the voltage stored in capacitor C1 after the circuit is unplugged from mains. The automatic working of the circuit is based on the light-sensing property of the light-dependent resistor (LDR). Operational amplifier CA3140 (IC1) is used as a comparator with two potential dividers in its inverting and non-inverting inputs. LDR1 and resistor R3 form one potential divider that provides a variable voltage at the inverting input pin 2 of IC1. Second potential divider comprises resistors R4 and R5, which provide half of the supply voltage (6V) to the non-inverting pin 3 of IC1. The output of IC1 depends on voltage level at inverting input pin 2 of IC1 as explained below.

In daylight, LDR1 has low resistance and the voltage at inverting input (pin 2) of IC1 is more than that of non-inverting input (pin 3). This makes IC1 output low, which drives transistor T1 into cut-off condition and strip LEDs do not glow. However, at night the light incident on LDR1 is low and its resistance is high. The voltage at inverting input of the comparator decreases, making it lower than the voltage at non-inverting input. This makes IC1 output high. Transistor T1 goes into saturation, thus connecting cathodes of LEDs to ground. All the LEDs in the strip turn on and remain that way till morning.



Assemble the circuit on a general-purpose PCB and enclose it in a suitable shock-proof case. Strip LEDs are available in ribbon-shaped form. Use 5cm bits (two bits) having three LEDs each. The strip can be cut at supply-contact points. Strip LEDs are arranged on a flexible belt with double-sided adhesive on the back side, so it can be glued to any surface. Connect the LED strip in the circuit with correct polarity. EFY note. Since the circuit uses 230V AC, there is a risk of electrical shock. Do not touch or troubleshoot when the circuit is plugged in. Before connecting the circuit to the power supply section, test it using 12V DC from a battery or DC power supply.