

A project report entitled

**‘Preparing PCB using dry photoresist film photolithography and circuit’**

Subtitle

**‘Pattern transforming using dry photoresist film photolithography’**

Submitted to:-

**Savitribai Phule Pune University, Pune**



Submitted by:-

**Miss.Pratiksha Sadashiv More**

Guided By:-

**Prof. Aishwarya Pawar**

Department of Physics

K.S.K.W College,Nashik

**Year:- 2020-2021**

**M.V.P Samaj's K.S.K.W College, Nashik.**



**CERTIFICATE**

This is to certify that **Miss. Pratiksha Sadashiv More**, student of M.Sc. (Physics) has satisfactory completed her project work on **Preparing PCB using dry photoresist film photolithography and circuit**, towards the partial fulfillment of his M.Sc. (Physics) Semester-IV during academic year 2020-2021.

Place: Nashik

Date:

Prof. Aishwarya Pawar  
Project guide

Dr. P.G. Loke  
Head Of Department

Internal Examiner

External Examiner

## Acknowledgement

The completion of this project could not have been accomplished without the advice, support and direction from those who all present around me.

I would like to express my sincere gratitude to all those who helped me in the successful completion of my venture. First of all, I would like to thank our **Principal Dr. J.D. Sonkhaskar** who provided me with all facilities and amenities for the development of my project.

I would like to thank **Prof. Aishwarya Pawar** for his help throughout the duration of the project. He gave his valuable advice and ideas while letting me able to work autonomously.

I express our special thanks to **Head of department Dr. P.G. Loke** for her encouragement and intense motivation.

Last but not least I would also like to thanks this prestige institution and its faculty member and especially to the staff member of our department to provide us materials and with standing all the pain through the journey of our report, without them it would have been a distant reality.

## Index

Sr.no	Title	Page.no
1.	Abstract	5
2.	Introduction	6
3.	Types of PCB	7
4.	Photolithography	14
5.	Using this PCB to make LPG leakage detector circuit	27
6.	Components use in the circuit	31
7.	Advantages & disadvantages of PCB	39
8.	Reference	40

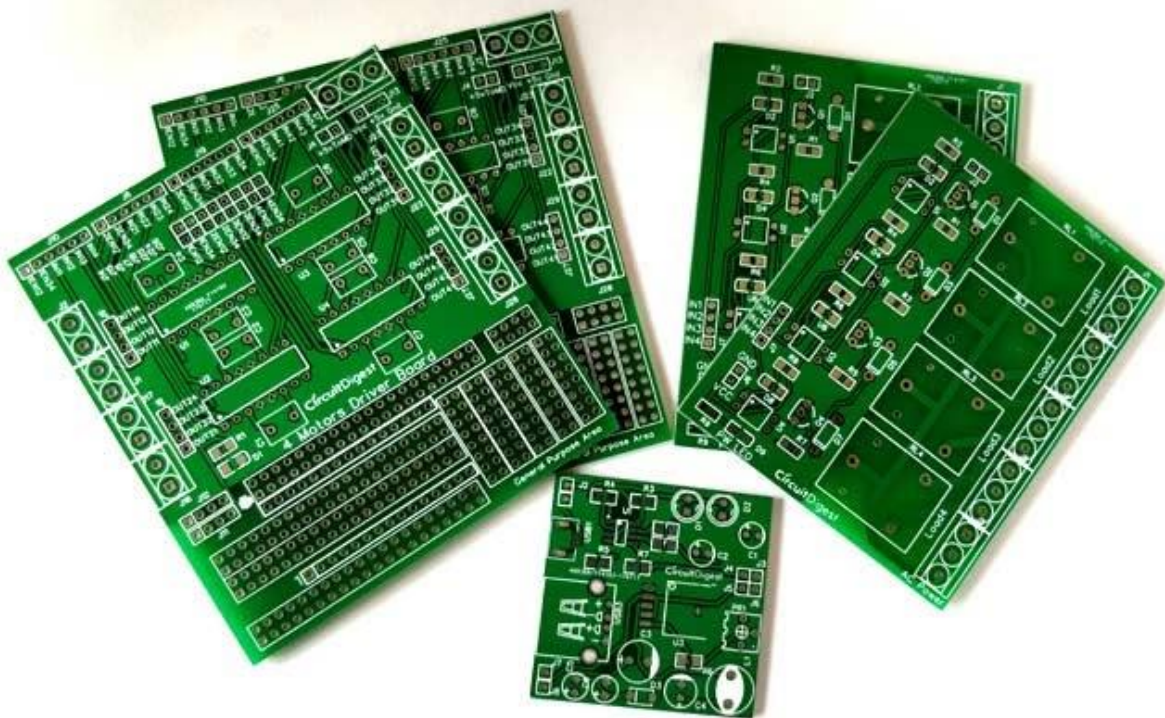
## ABSTRACT

The object of project preparing pcb using dry photoresist film photolithography and built circuit. Low-cost method of fabricating a printed circuit board (pcb) is presented. The process is favorable over conventional pcb fabrication due to fewer processing steps and lower capital investment. A laser printer is used to print a circuit layout onto ink-jet paper or specialized transfer paper (i.e., pnp-blue from technics, inc.). The image is then heat transferred to a copper clad board using a standard household clothes iron. The transferred toner acts as an etch resist in a ferric chloride ( $\text{FeCl}_3$ ) or other etchant bath, such as ammonium persulfate. Using this method, it is possible to fabricate pcbs with state-of-the art feature sizes as small as 4 mils using a 300 dpi laser printer. A higher resolution printer is expected to give finer resolutions. Tradeoffs between the ink-jet paper and the pnp-blue paper is analyzed and discussed. This approach has been used for graduate-level courses including a vhdl class project to implement an alu using a 44-pin cpld and a microwave class project to implement a microstrip matching network for a high-gain, 875 mhz transistor. Over 200 students have successfully processed pcbs using this technique

# INTRODUCTION

## Printed circuit board(PCB)

PCB is a copper laminated and non-conductive Printed Circuit Board, in which all electrical and electronic components are connected together in one common board with physical support for all components with base of board. When PCB is not developed, at that time all components are connected with a wire which increases complexity and decreases reliability of the circuit, by this way we cannot make a very large circuit like motherboard. In PCB, all components are connected without wires, all components are connected internally, so it will reduce the complexity of the overall circuit design. PCB is used to provide electricity and connectivity between the components, by which it functions the way it was designed. PCBs can be customised for any specifications to user requirements. It can be found in many electronics devices like; TV, Mobile, Digital camera, Computers parts like; Graphic cards, Motherboard, etc. It also used in many fields like; medical devices, industrial machinery, automotive industries, lighting, etc.





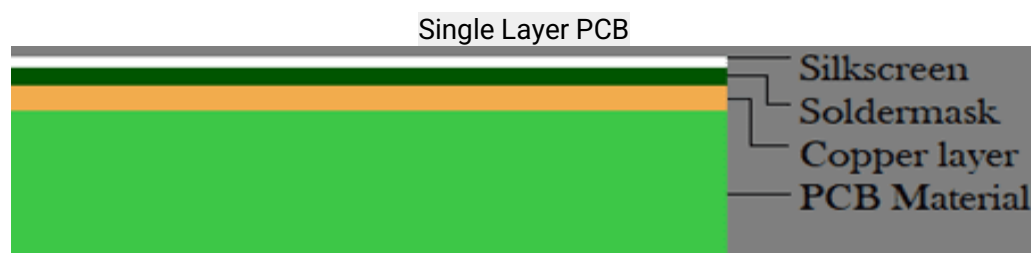
## Types of PCB:

There are several types of PCB available for the circuit. Out of these types of PCB, we have to choose the appropriate type of PCB according to our application.

1. Single-layer PCB
2. Double-layer PCB
3. Multi-layer PCB
4. Flexible PCB
5. Aluminium backed PCB
6. Flex-rigid PCB

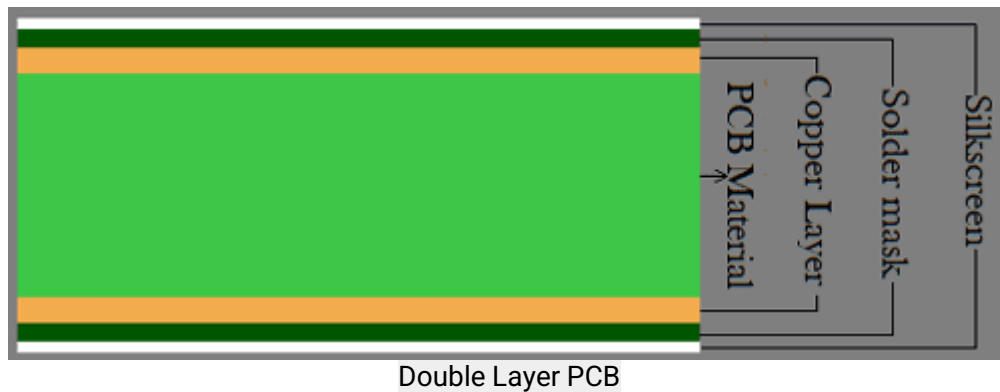
### 1) Single Layer PCB:

A single layer PCB is also known as **single sided PCB**. This type of PCB is simple and most used PCB because these PCBs are easy to design and manufacture. One side of this PCB is coated with a layer of any conducting material. Generally, copper is used as conducting material for PCB, because copper has very good conducting characteristic. A layer of solder mask is used to protect PCB against oxidation followed by silk screen to mark out all of the components on the PCB. In this type of PCB, only one side of the PCB is used to connect different types of electrical or electronics components like resistor, capacitor, inductor, etc. These components are soldered. These PCBs are used in low cost and bulk manufacturing application like calculators, radio, printers and the solid-state drive.



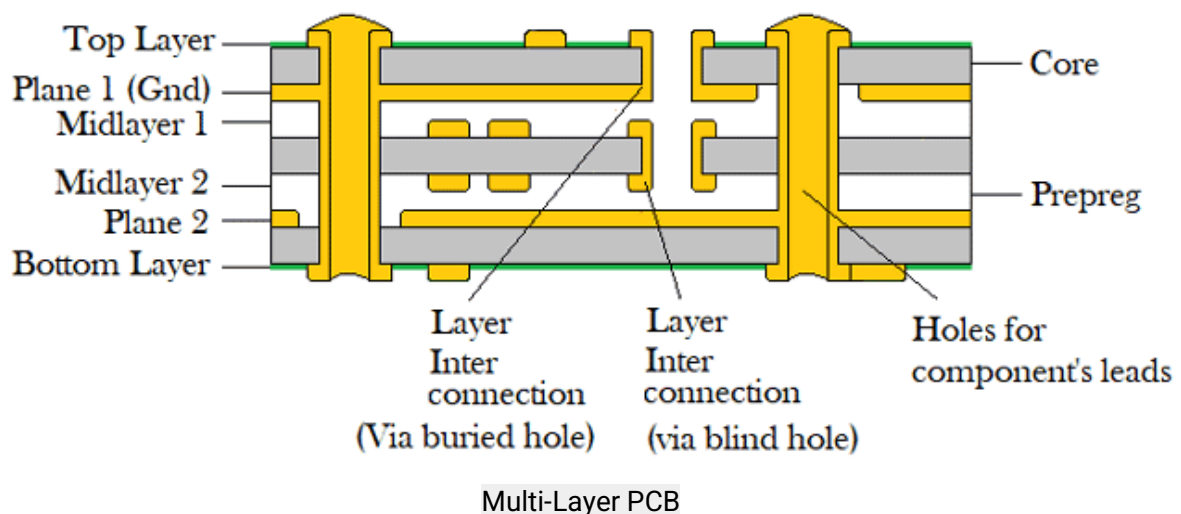
### 2) Double Layer PCB:

Double layer PCB is also known as **double sided PCB**. As name suggests, in this type of PCB, a thin layer of conducting material, like copper is applied to both top and bottom sides of the board. In PCB, on different layer of board, consist via, which has two pads in corresponding position on different layers. These are electrically connected by a hole through the board, which is shown in figure-2b. More flexible, relatively lower cost, and most important advantage of this type of PCB board is its reduced size which makes circuit compact. This type of PCB is mostly used in industrial controls, converter, UPS system, HVAC application, Phone, Amplifier and Power monitoring systems.



### 3) Multi-Layer PCB:

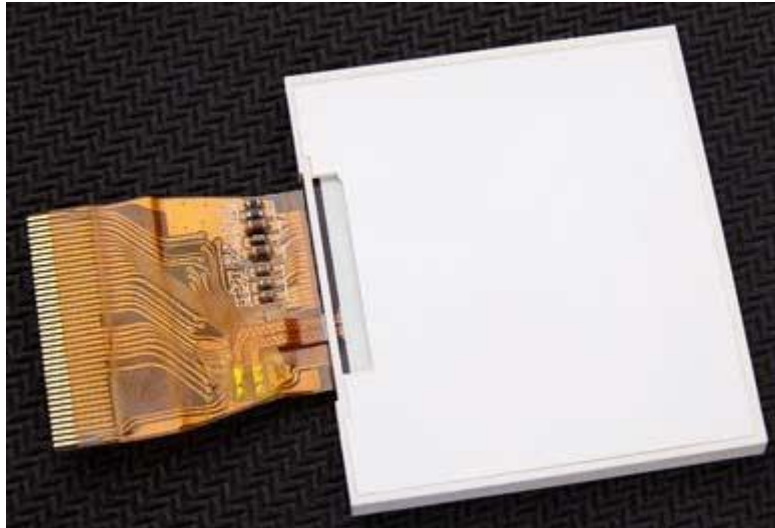
Multilayer PCB has more than two layers. It means that, this type of PCB has at least three conductive layers of copper. For securing the board glue is sandwiched between the layer of insulation which ensures that the excess heat will not damage any component of circuit. This type PCB designing is very complex and used in very complicated and large electrical task in very low space and compact circuit. This type of PCB is used in large application like GPS technology, satellite system, medical equipment, file server and data storage.



### 4) Flexible PCB:

Flexible PCB is also known as **Flex circuit**. This type of PCB used flexible plastic material like polyimide, PEEK (Polyether ether ketone) or transparent conductive polyester film. The circuit board is generally placed in folded or twisted. This is a very complex type of PCB and it also contains different ranges of layers like single-sided flex circuit, double-sided flex circuit, and multi-sided flex circuit. Flex circuit is used in organic light-emitting diode, LCD fabrication, flex solar cell, automotive industries, cellular telephones, cameras, and complex electronics devices like laptops.





Flexible PCB

#### 5) Rigid PCB:

Rigid PCBs are made out of solid material which don't allow PCB from twisting. Same as flex PCB, Rigid PCB also have different layer configuration like single layer, double layer and multi-layer Rigid PCB. Shape of this PCB is not change after installation. This PCB cannot be bent according to shape of base that's why this PCB is known as RIGID PCB. Lifespan of this type of PCB is very high, so this is used in many parts of computer like **RAM, GPU and CPU**. Simple in design and most used and most manufacture PCB is single sided rigid PCB. Multi-layer rigid PCB can more compact by containing 9-10 layers.

#### 6) Flex-rigid PCB:

Combination of Flexible circuit and rigid circuit is most important board. Flex-rigid boards consists of multiple layers of flexible PCB attached to a number of rigid PCB layer. Flex-rigid board is as shown in figure. It is used in cell phones, digital cameras and automobiles etc.

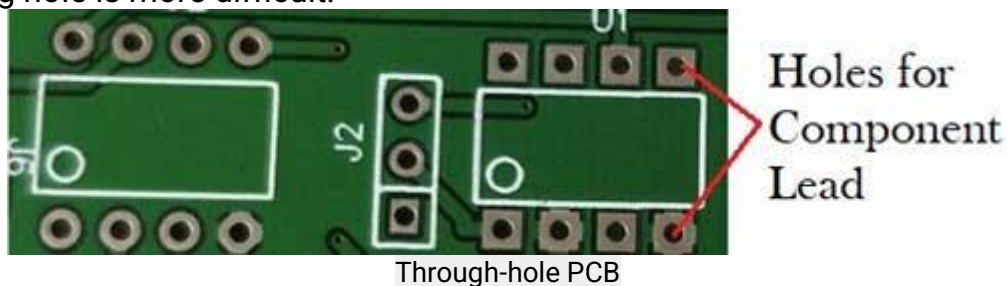


## Types of PCBs According to Mounting System

1. Through-hole PCB
2. Surface mounted PCB

### 1) Through-hole PCB:

In this type of PCB, we have to make hole using drill on PCB. In these holes, leads of components are mounted and soldered to pads situated on opposite side of PCB. This technology is most useful because it gives more mechanical support to electrical components and very reliable technology for mounting of components but drilling in PCB make it more expensive. In single layer PCB, this mounting technology is easy to implement, but in case of double layer and multi-layer PCB making hole is more difficult.

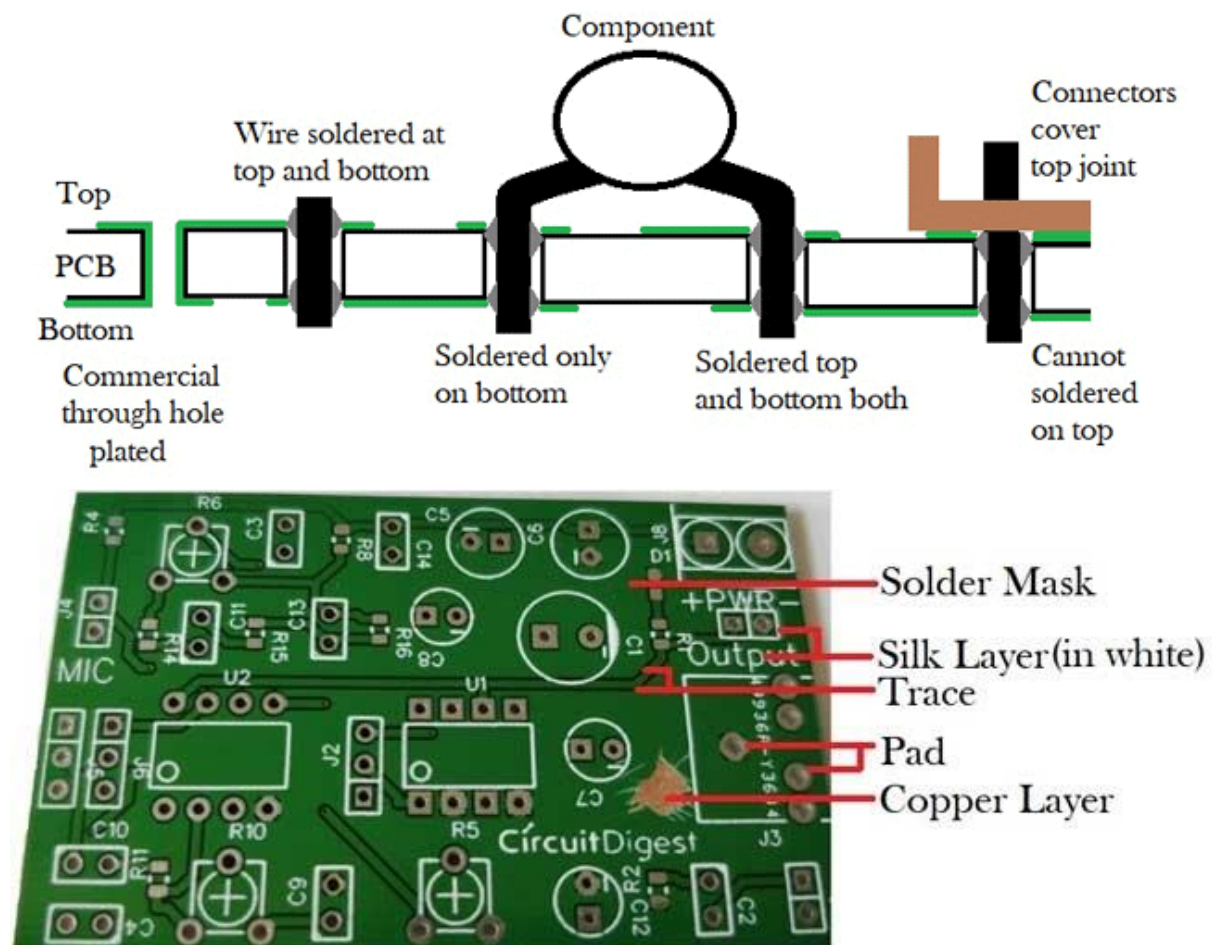


### 2) Surface mounted PCB:

In this type of PCB, components are small in size because these components have very small lead or no leads are required for mounting on the board. Here, in this technology, SMD components are directly mounted on the surface of the board and not require to make hole on board.



## Different Parts of PCB:



**Pad:** Pad is nothing but a piece of copper on which lead of components are mounted and on which soldering are done. Pad provides the mechanical support to the components.

**Trace:** In PCB, components are not connected with the help of wires. All components are connected with a conducting material like copper. This copper part of PCB which is used to connect all components that is known as trace. Trace is looks like below figure.

**Layers:** According to application, cost and available space of circuit, user can choose the layer of PCB. Most simple in construction, easy to design and most useful in routine life is single layer PCB. But for very large and complex circuit, double layer PCB or Multi-layer PCB is most preferred compared to single layer PCB. Now a day, in multi-layer PCB, 10-12 layers can be connected and most critical thing is to communicate between the components in different layer.

**Silk layer:** Silk layer is used for printing line, text or any art on the surface of PCB. Usually, for screen printing epoxy ink is used. Silk layer can be used in top and/or bottom layer of PCB according to user requirement which is known as silk screen TOP and silk screen BOTTOM.

**Top and bottom layer:** In Top layer of PCB, all components are mounted in this layer of PCB. Generally, this layer is green coloured. In bottom layer of PCB, all components are soldered through the hole and lead of components is known as bottom layer of PCB. Sometime, in top and/or bottom layer PCB is coated with green colour layer, which is known as solder mask.

**Solder Mask:** There is one additional layer on the top of copper layer called as Solder Mask. This layer generally has green color but it can be of any color. This insulating layer is used for to prevent accidental contact of pads with other conductive material on PCB.

## PCB designing software:

### **Eagle:**

EAGLE is a most popular and easiest way to design PCB. EAGLE stands for Easily Applicable Graphical Layout Editor which is previously developed by CadSoft Computer and currently Autodesk is developer of this software. For designing circuit diagram, EAGLE has a schematic editor. EAGLE file extension is .SCH and different parts and components are define in .LBR extension. Board file extension is .BRD.

### **Multisim:**

Multisim is also very powerful and easy learning software. Which is originally developed by Electronics Workbench and now it is a division of National Instruments (NI). It includes microcontroller simulation (MultiMCU) and integrated import export features to the PCB layout software. This software is widely used in academic and also in industry for circuit education.

### **EasyEDA:**

EasyEDA is a software which is used to design and simulate circuits. This software is an integrated tool for schematic capture, SPICE circuit simulation, based on Ngspice and PCB layout. Most important advantage of this software is that, it is web based software and used in browser window. So, this software is independent from OS.

### **Altium Designer:**

This software is developed by Australian software company Altium Limited. The main feature of this software is schematic capture, 3D PCB design, FPGA development and release/data management. This is first software which offer 3D visualization and clearance checking of PCB directly from PCB editor.

**KiCad:** This software is developed by jean-pierre charras. This software has tools which can create BoM (Bill of Material), artwork and 3D view of PCB as well as all components used in circuit. Many components are available in the library of this software and there is feature that user can add their custom components. This software is support many human languages.

**CircuitMaker:** This software is also developed by Altium. Schematic editor of this software includes basic component placement and this software is used to design advanced multichannel and hierarchical schematics. All schematic is uploaded to server and these files are available to view by anyone, provided that you need a CircuitMaker account.

# Photolithography

Photolithography, also called optical lithography or UV lithography, is a process used in [microfabrication](#) to pattern parts on a [thin film](#) or the bulk of a substrate (also called a [wafer](#)). It uses light to transfer a [geometric pattern](#) from a [photomask](#) (also called an *optical mask*) to a [photosensitive](#) (that is, light-sensitive) chemical [photoresist](#) on the substrate. A series of [chemical treatments](#) then either [etches](#) the exposure pattern into the material or enables deposition of a new material in the desired pattern upon the material underneath the photoresist. In complex [integrated circuits](#), a [CMOS](#) wafer may go through the photolithographic cycle as many as 50 times.

Photolithography shares some fundamental principles with [photography](#) in that the pattern in the photoresist etching is created by exposing it to [light](#), either directly (without using a mask) or with a projected image using a photomask. This procedure is comparable to a high precision version of the method used to make [printed circuit boards](#). Subsequent stages in the process have more in common with etching than with [lithographic printing](#). This method can create extremely small patterns, down to a few tens of [nanometers](#) in size. It provides precise control of the shape and size of the objects it creates and can create patterns over an entire surface cost-effectively. Its main disadvantages are that it requires a flat substrate to start with, it is not very effective at creating shapes that are not flat, and it can require extremely clean operating conditions. Photolithography is the standard method of printed circuit board (PCB) and [microprocessor fabrication](#). [Directed self-assembly](#) is being evaluated as an alternative to photolithography.

# Circuit Making Process

## Dry Film:

Dry Film is a photo-resist, when strong UV light hits it, where the light hits, it hardens and this hardened form resists etching solutions. The film is comprised of three parts, the blue layer is in the middle, this is the polymer which hardens with UV, on each side of this polymer is a protective transparent film.

## Uses:

In very brief summary, one protective layer (the inside of curl generally) is removed, the now uncovered polymer side is attached to the PCB, the PCB is exposed to UV through a 'mask' which has transparent traces to allow UV to harden the " " traces image, the remaining protective layer is removed and the PCB is developed in a solution of washing soda (Sodium Carbonate), then the PCB is etched.

The rest of this document gives you all the steps, tips and tricks which I have learned in using Dry Film! Be sure to read it, save yourself lots of experimenting!

Practice, Practice, Practice. I advise cutting some small squares of film a couple of cm squared and using these to get the hang of affixing it to a board and developing, use some moderately dense small (section of) artwork with the finest traces you want to do, with the narrowest spaces you want to do etc. You might want to practice 5 or 10 times before doing it for real. If you mess one up, just clean it off, and start again.

Units. If you are new to designing circuits you might not know about the unit called mils. Here in New " " Zealand (and other civilised countries who use the metric system) when a somebody says oh it's about 50 " mils they usually mean 50 millimeters. But when " " " dealing with electronics mil means one one-thousandth " " " of an inch and it is used a lot even by normally " sensible metric people.

So when I write for example 10 mils, I mean 0.254 " " millimeters (one quarter of one millimeter), or 254 microns if you prefer. 8 mils is about 0.20mm, and 12 mils is about 0.30mm.

About 10 mils is achievable for a trace width (maybe even 8), and 10 mils keepout (between traces), but for practical purposes 12 mils or greater for both makes your life a lot easier :-)



### **Cleaning the PCB:**

To begin with you must clean your blank PCB surface thoroughly.

Most recently, this is my way... 16

Grab some 150 or finer (higher number) sand paper in a pack from the local cheap-chinese-shop, I would not go less than 150, but 150 is fine (it's what I'm using currently, 150 dry tan coloured sand paper).

Sand the PCB, I sand in 4 directions, vertically, then horizontally, then on an angle one way, and on an angle another way.

Wipe the PCB with a paper towel wet with acetone.

I find that this works just fine, no messing about just good old abrasion.

### **Attaching The Film:**

A good attachment without bubbles and wrinkles is the key to success.

The film needs moderate heat and pressure to properly adhere enough to resist.

Presented in this document are two methods, one using a Clothes Iron (or Hot Air, or even a Hair Dryer) and one using a Laminator.

### **Wet Lamination Method (Clothes Iron, or Hot Air):**

This method is my current favourite, it is sort of my own devising and works quite well at least for the small (7x10cm or less) boards I most commonly make.

In addition to the dry film you will need a “ ” travel spray bottle of water, you can buy such bottles at your local cheap-chinese stuff store, and some sellotape.



Start by cleaning your PCB as detailed previously.

Now with the PCB clean and dry place it on a flat surface (that's not going to care if it gets hot, your mum's dining table probably not a good place).

Set your iron to a cool setting, 60-70 degrees works for me, usually just a couple clicks off the lowest possible setting. Clothes irons are not that accurate at temperature regulation, give it a few minutes to settle before you check the temperature (if you have a temperature probe).

Take your travel bottle of (room temperature) water and give a couple pumps to put a light mist onto the PCB, you don't need to drench it, just a light spray. It doesn't matter if it's not totally even coverage.

Now take your piece of film (about 1cm larger than required for easier handling). Stick a piece of sellotape on each side of the film at one corner, pull the two pieces apart, the bottom (inside of curl) layer should pull away (this side goes against the board). Now drape the film onto the misted pcb, and take a “ ” small wad of paper towel and use that to smooth the film down from the center to the edges pushing out the water and air, check for bubbles and dust, you can usually peel the film up a bit and get rid of them at this stage.

Once you have the film smoothed down onto the pcb, take your clothes iron and literally iron the film onto the pcb. You don't need to press it through to the other side of the planet, it's really the heat you want, and mild pressure, this is what causes the adhesion.

If you don't have a clothes iron, you can also use hot air from your reflow station, and I have heard also of using a hair dryer but not tried that myself, just heat and rub the film down on to the pcb.

Pay special attention to the edges of the board, these are the problem areas, roll the iron over the edges.

If you have a little wrinkle, bubble, tear, in the polymer (photo) layer, try using the tip of the iron to “ ” buff it out, the polymer is soft at this 60-80 degree temperature and can be smooshed around with the iron “ ” tip to smooth small problems out.

After perhaps 60 seconds of ironing things should be nice and toasty, not so hot you can't hold your hand onto the PCB, but hot enough that it's not comfortable to do so :-)

A note on temperature; if you go too hot, you will get quite obvious ugly blisters under your film, this is no good those blisters will just wash off when you develop, clean it off (or cut/scrape/clean those areas and re-laminate a patch if you want to try that, sometimes you can get away with it :)), reduce your iron temperature and start again with a fresh piece.

If you go too cold, the film will also not attach well during development and etching.

Caution: clothes iron temperature regulation may be quite poor - your fancy pantaloons probably don't care if they were ironed at 80 degrees or 110 degrees so clothes irons are pretty simplistic at controlling their temperature. If you have a temperature probe for your multimeter use it to get a feel for where to set your iron's dial and note which parts of the baseplate are hotter/cooler, aim for 60 to 75 degrees at the hottest point.

For Double Sided Boards you must allow the board to cool down to room temperature again before you try and attach the second side.

You should allow the board to cool to room temperature before you expose.

If you find that the photo layer isn't stuck to the pcb totally (try peeling the top protection layer up a little on a corner) and you are sure your cleaning was well done then you know that you needed to warm it up more, or more evenly (spread the iron around), or a fraction more pressure, the most likely places to have a problem are the edges. Before exposure you can always try to iron it down a bit more to try and get that to stick.

### **Laminator Method:**

If you have a laminator, then using this can produce a good result, but you will want to do some preparation first.

Clean your blank PCB as previous.

Place PCB on a carrier and attach a piece film cut to about 5mm larger on each side than you need to the top end-stop, with tail attached to film (see next page “ ” for diagrams of my carrier setup).

It is better to cut 5mm larger on each side than you need, but if you do use right up to the cut edge of a piece of film have a careful look at the edge, sometimes there is a visible border of lighter “ ” colouring (more transparent) running down an edge (as it comes off the roll), if visibly different this border should be avoided as it's performance may be degraded a bit.

Note film has top and bottom (PCB) side, the bottom – is the inside of the curl and that goes against the PCB. If your film was sent flat, the cardboard wrapping is marked with which is top and which is bottom.

The film has a protective layer on both sides with the soft resist material in the middle. Flip film over and remove the bottom side layer easy way, stick a piece – of sellotape on each side of a corner of the film, pull the two bits of tape apart, the bottom layer will peel off (actually, it's really not a big deal if the top layer comes off instead, seems to work either way up, but inside-of-curl-down is the norm ). “ ”

Hold film vertical by the tail and feed into laminator, lowering the tail as you go, this eliminates air bubbles.  
Cut off the tail. You may then want to run it through the laminator a couple more times.

If you did get a bubble in there, prick it with the tip of a craft knife and run it through the laminator, or buff it with the tip of a cool clothes iron.

Allow to cool, then remove PCB. Laminated boards can be stored in the dark ready to use, not sure how long for, I've stored them for 2 weeks before using without any trouble, so perhaps forever is the answer. “ ”

## **Exposure:**

You need a negative artwork, transparent areas will remain on the board as resist, black areas will be removed.

I recommend a good 2mm border on your artwork - that is, no traces closer than 2mm to the PCB edge, the reason is that the most likely place for the resist to lift or get a poor exposure is at the edges.

Top tip: The resolution you can get is very high, “ ” even with our rough-and-ready setup, exposing to the sun using tracing paper. However, if you put traces very close together (about 8 mil , 0.21mm or less) any “ ” “ ” UV bleed , overexposure... can lead to traces in the resist which are touching. This is easily fixed before etching but to save you the trouble, remember that thin traces are easy, but close traces are hard.

I'd suggest sticking with a keepout of around 0.254mm “ ” (10 mil) at the minimum, and your traces could go down to 8 mil that said, for repeatability, I don't – recommend trying to get below 12 mil, sure you CAN do 8 mil, but they are so tiny that it is very very easy to damage them, and there just isn't easily fixable with a pen at that sort of size.

Personally, I try to stay at 13 mil or thicker traces, with 12 mil or wider keepout unless there is just no other way, it makes for much less fiddling about if you can stay above 10 mil.

Tracing paper is a good medium for printing your artwork on, you don't need to use transparencies. You can either use a single copy or print two copies, and stick them together to increase the darkness of the black areas.

Using a single copy may require a quite short exposure depending on your printer's toner density (1 minute 20- 30 seconds for me using UV leds), and any imperfections in the artwork will need to be touched up with a pen first, using two copies will mean that the exposure is longer (2 or even 3 times as long), but the imperfections in your printer's toner is less of a problem. Personally, I use a single copy, but, experiment with your own!

Can you use an inkjet? Some people do and find it works, some people tried and found it doesn't, so short answer is, maybe depends on your ink!

—

It may be useful to put a strip of packing tape (or perhaps contact book covering) over each print to protect the toner - depending on your printer, toner may rub-off tracing paper easily, tape protects it while you

are aligning your two copies, and doesn't affect the exposure much. But for very fine detail, avoid this if possible.

Apply the artwork, tonor-side-to-board, make sure it is tightly against the board (a piece of glass on both sides to make a sandwich and a couple of quick-clamps or even bulldog clips to hold it together is one way, put something on the back of the PCB in the sandwich to ensure that the board is pressed FIRMLY to the artwork) and expose to Ultraviolet light (eg, the Sun).

Time of exposure varies. Experiment to find your sweet spot. To give an idea, about 10 minutes in full Sun produces a strong exposure, possibly a bit too long. Under UV LEDs about 2 to 5 minutes for a double-copy tracing paper, about 1-2 minutes for a single-copy with an array of 5mm leds, and for my current box with 4 1Watt UV leds it's a super fast 22 seconds.

You don't need to be too precise with double-copy, there's plenty of wiggle-room if your artwork is dark enough, it's quite hard to go wrong! With single-copy it's a bit more exacting.

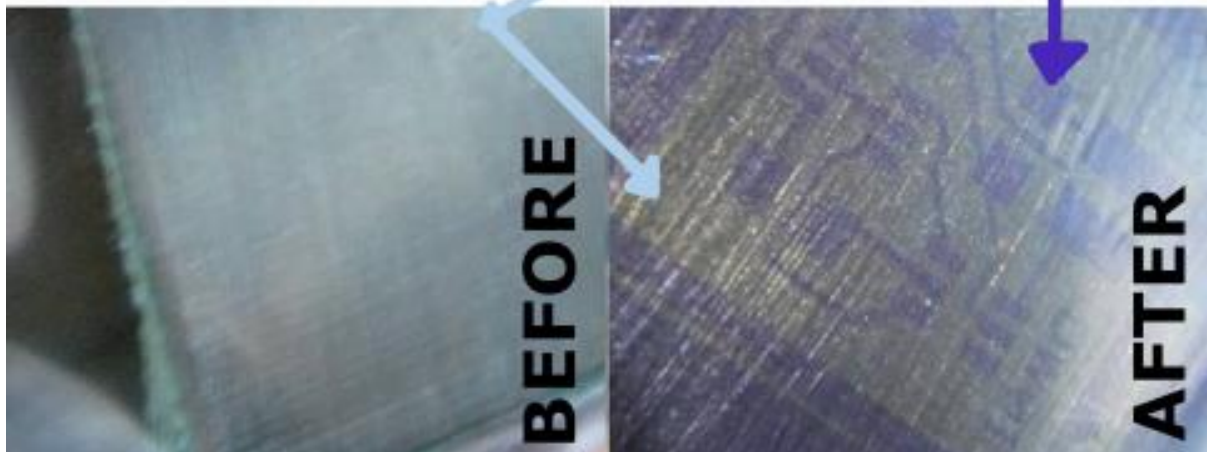
Once exposure is complete remove the artwork and you will CLEARLY see the traces burned onto the board. “ ”

Leave the board to further cure in a dark place for a few minutes.

### Example of exposure of Dry Film

Notice that the exposed areas (the bits to keep), have become a little darker.

Notice that the colour of the unexposed areas (the bits to get rid of) is about the same as it was before.



NB: Exact colour of pre-exposed film varies between manufacturers and types, but for the most part they work the same, exposure time might vary slightly.

Developing:

Carefully remove the top protective layer from the film, if your lamination technique and cleaning was good you won't pull up any of the traces, if it wasn't, you will.

Place board in a developing solution, the solution is Washing Soda (Sodium Carbonate, also often as Sodium Percarbonate, Sodium Peroxyhydrate, they will all work), which you can purchase at the supermarket.

It is used in a very wide variety of products especially in the laundry, look for powdered forms.

For example Sard Wonder Oxy Plus in “ ” any laundry aisle works well, as should any other similar powdered





laundry stuff with Sodium Carbonate/Percarbonate/Peroxyhydrate.

You can also buy pure Sodium Carbonate in the laundry aisle of most supermarkets, look for natural fabric softeners!

Mix about 30g of the sodium carbonate containing laundry powder of your choice into 1 Litre of water. Doesn't need to be precise. Mix it up real good, I put it in a pepsi bottle and give it a good old shake.

30g/L at room temperature works for me.

Let the board develop for a couple minutes (watch it though), swoosh the liquid over it. The unexposed areas will dissolve leaving the traces in place. You can lightly massage the board with your fingers to help the process along.

Using a vibrating type toothbrush very gently can really help developing. Don't scrub, just wipe really, the " " traces are strong enough to resist the brush, but not totally hard yet so you can damage them. Obviously, don't use your actual toothbrush, maybe now is a good time to buy a new one and relegate your old one to PCB duty.

It's done when it looks like all the left over gunk has gone and you have nice clean traces, look at the pads and holes, look to see if there is any gunk still sticking between close together traces.

Rinse the board under running water to get rid of all the developing solution. Then dry it by patting with a paper towel.

Eye protection is perhaps a good idea, washing soda in your eyes could be a bit unpleasant, but apart from that it's not dangerous.





After developing (2-3 minutes) and rinsing the copper to be removed is clean and all "gunk" removed.

### **Hardening Exposure:**

After developing to put the developed board in full UV (no artwork/mask necessary) for a longer exposure, this will harden up the developed traces completely. I use " " a hardening exposure twice as long as the original exposure.

After the hardening exposure, you may notice a slight blue tinge on the bare copper areas which indicates some resist residue, you can easily remove this with a few seconds in the developer and a rub with your fingers or a toothbrush, then immediately rinse in water again.



**After the hardening exposure, little change is apparent just a bit darker. Notice there is no bluing of the copper to be removed indicating the developing was**

**long enough and not overexposed. If there was bluing, it can go back in the developer briefly to clean it up then rinse.**

### **Drying:**

The developed traces will feel a bit tacky. It's a good idea now to either let the board air-dry, or use a hair dryer or hot air rework station to dry the board until it's not tacky any more.

If you try and touch-up a tacky board, you're going to have a bad time.

Reader Rodrigo from Brazil contributes...

For the "problem" of sticky/wet dry film after exposure you can quickly resolve this by applying acid to the board, it stops immediately the basic solution attached to the dry film, making it very consistent and 'hard'.

Acid can be just kitchen's vinegar! Just works great!

### **Touch Up:**

Photoetching is pretty reliable especially once you have done a few boards to get the hang of it - but you may still need to touch up now and then, a bit of dust in the wrong place, a scratch deeper than you thought, bubble, wrinkle, slightly over or under developed or exposed areas - remember you're a hobbyist probably working in your shed, not a process engineer in a clean-room, some touch up is inevitable and it's much better to do it before etching.

First have a look for any traces that just look a bit iffy to you. Any possible breaks or traces that look a bit thinner than you wanted, sharp corners that seem a bit precarious, traces that look a bit light coloured (underexposed areas).... grab your handy etch resist pen (see my Trade Me listings) and lay down some ink.

You may need to clean the pen tip by drawing a stroke on some paper now and then because of the resist residue.

Remember it's MUCH easier to prevent potential etching problems now with a dot of ink than it is to fix them after etching!

Next you want to look for any places where resist “ traces are or might be touching, or have resist gunk ” still stuck between them. Any you find, use a pin or tip of a craft knife to clean them up.

If your traces are really close together (8mil or so), it's a good idea to have a look under a magnifying glass etc to check for any gunk still lurking, it's easier to clean it up now with the tip of a pin or blade than to clean up copper bridges later.

If you are getting a lot of left over gunk, you're not developing it long enough, your developer is a bit weak/used-up, you need to agitate it a bit more, or your exposure was a bit long.

Etching:

Left as an exercise for the reader :) Use your favourite etching method.

### **Stripping:**

Once etched, to remove the resist, make a much stronger solution of Washing Soda, warm it up (or make fresh with hot water), and drop the board in it.

Leave it for a while (might take 15 minutes to an hour, hotter = faster). The resist will float off the board. A bit of a rub with fine sandpaper, or a scrubbing pad will take care of any stubborn bits and leave you with a beautiful shiny perfectly etched PCB!

Eye protection is a very good idea, strong hot washing soda in your eyes could be REALLY unpleasant.

Alternatively, you can put the PCB in bath of “ ” acetone, it will take maybe a minute to strip off in acetone, on the downside you have a bath of acetone evaporating in your office, which may not be ideal.

## Using this PCB to make LPG leakage detector circuit

The circuit for an LPG leakage detector is readily available in the market, but it is extremely expensive and usually based on a microcontroller (MCU). Presented here is a low-cost circuit for an LPG detector that you can build easily.

The main objective of the circuit is to detect LPG leakage anywhere. Fig. 1 shows the author's prototype.

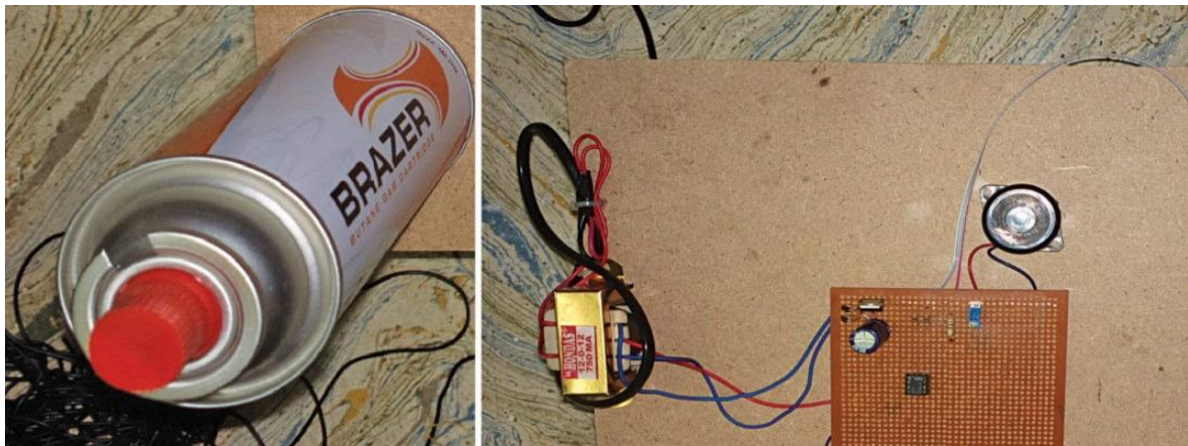
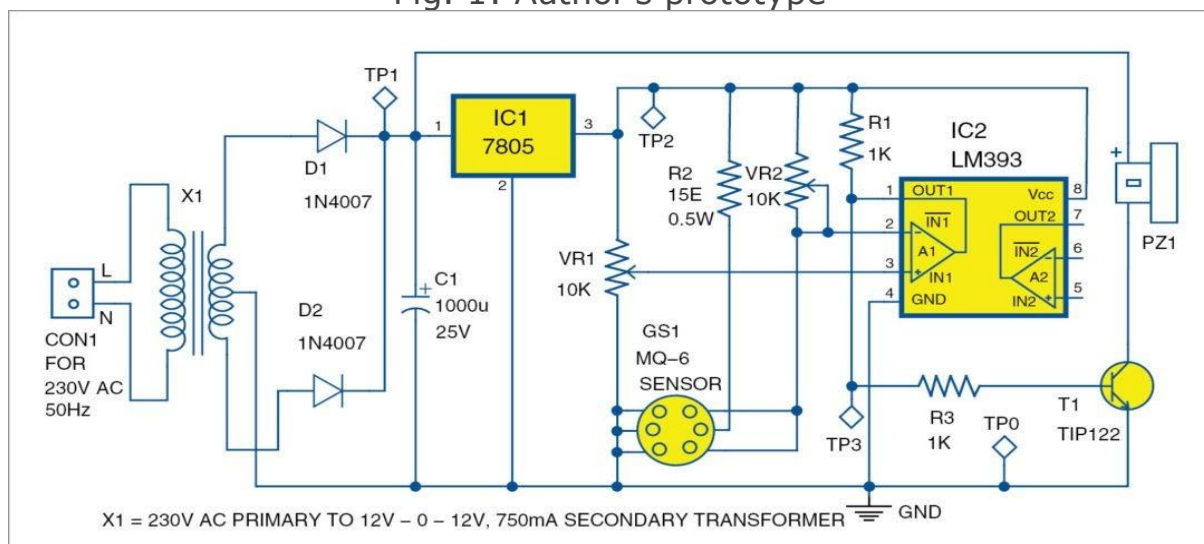


Fig. 1: Author's prototype





## Circuit and Working of the LPG leakage detector

Circuit diagram of the low-cost LPG detector is shown in Fig. 2. It is built around step-down transformer X1, two rectifier diodes [1N4007](#) (D1 and D2), a 1000 $\mu$ F capacitor (C1), [7805](#) voltage regulator (IC1), MQ-6 LPG gas sensor (GS1), dual comparator LM393 (IC2), darlington transistor TIP122 (T2), 12V high-gain siren/buzzer (PZ1) and a few other components.

components.

PARTS LIST	
<i>Semiconductors:</i>	
IC1	- 7805, 5V voltage regulator
IC2	- LM393 dual comparator
D1, D2	- 1N4007 rectifier diode
T1	- TIP122 npn darlington transistor
<i>Resistors (all 1/4-watt, <math>\pm 5\%</math> carbon, unless stated otherwise):</i>	
R1, R3	- 1-kilo-ohm
R2	- 15-ohm, 0.5W
VR1, VR2	- 10-kilo-ohm potmeter
<i>Capacitor:</i>	
C1	- 1000 $\mu$ F, 25V electrolytic
<i>Miscellaneous:</i>	
CON1	- 2-pin connector terminal
X1	- 230V AC primary to 12V-0-12V, 750mA secondary transformer
PZ1	- 12V high-gain siren/buzzer
GS1	- MQ-6 LPG sensor

The mains supply is stepped down by transformer X1, rectified by a [full-wave rectifier](#) comprising diodes D1 and D2, filtered by capacitor C1 and fed to regulator 7805 (IC1) to maintain constant 5V DC output, which is fed to the circuit.

At the heart of the circuit is dual comparator IC LM393 (IC2). It is used to compare two different voltages, namely, reference voltage and MQ-6 gas sensor output voltage.

Reference voltage at non-inverting pin 3 of IC2 is set using potmeter VR1 to adjust voltage levels based on sensitivity requirements. LPG sensor (MQ-6) output voltage is fed to inverting pin 2 of IC2.

If reference voltage (pin 3 of IC2) is less than sensor voltage (pin 2 of IC2), output goes low, which means there is no LPG leakage. With low output, T1 remains cut-off and there is no current flow through the buzzer; it does not sound and remains in silence mode.

If reference voltage is greater than sensor voltage, output goes high, which means there is LPG leakage. The high output switches on transistor T1 and the buzzer rings loudly to alert the people around.

It is very easy to find gas leakages with this circuit, which uses low-cost components and an interactive way to adjust different sensitivity levels, based on customer needs, with the help of potmeter VR1.

#### Construction and testing

An PCB pattern of the LPG leakage detector is shown in Fig. 3 and its component layout in Fig. 4.



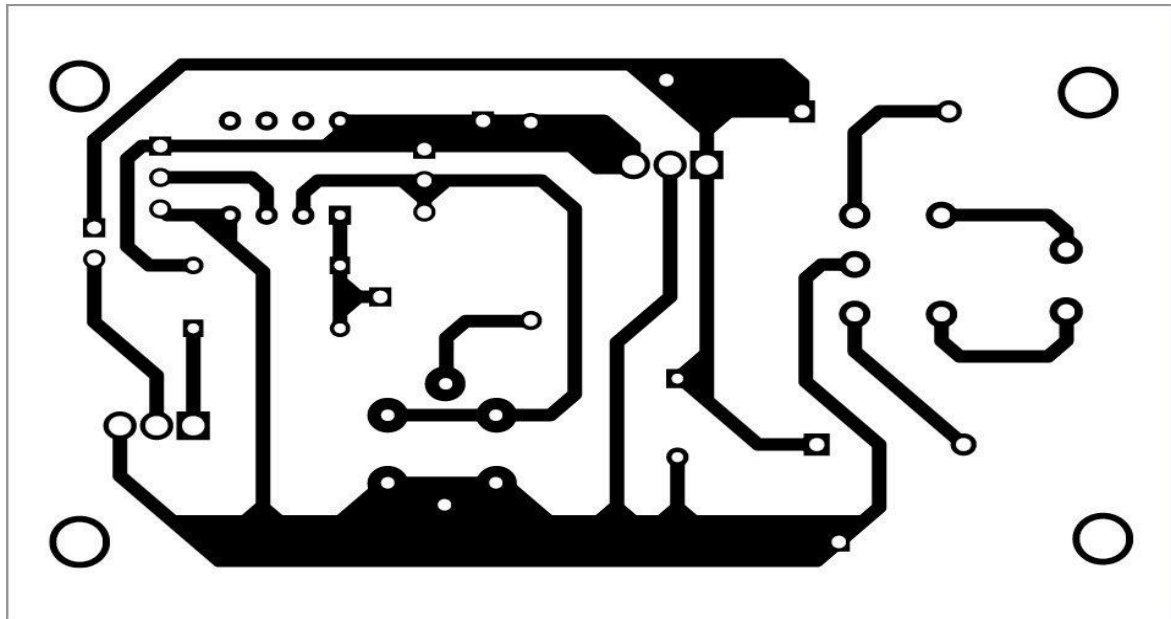


Fig. 3: PCB pattern of the LPG detector

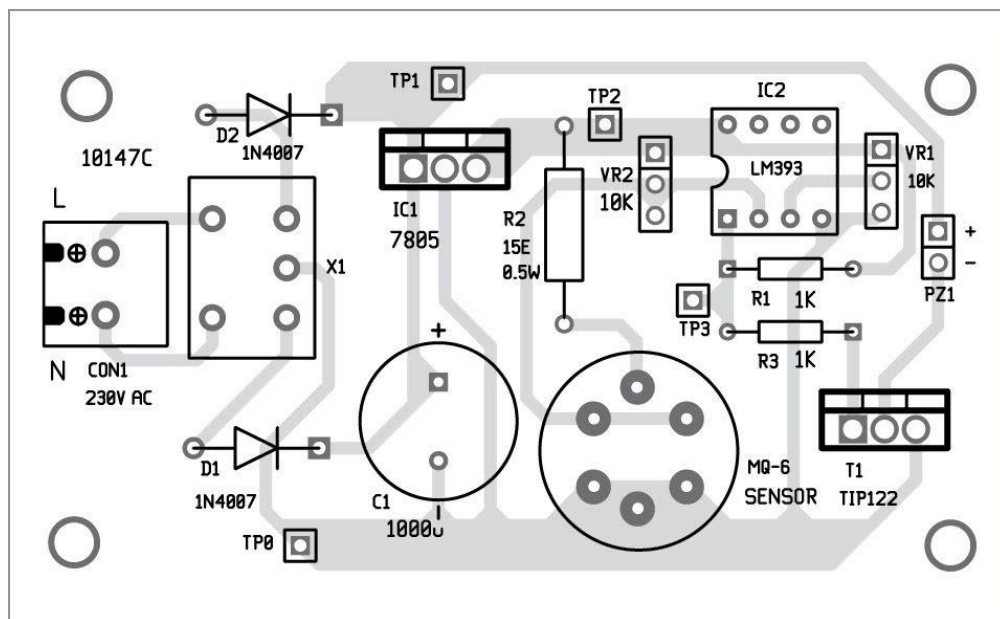
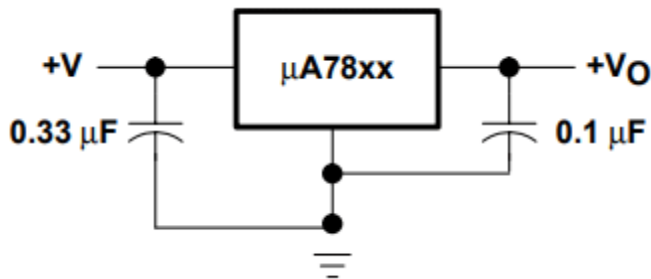


Fig. 4: Component layout of the PCB

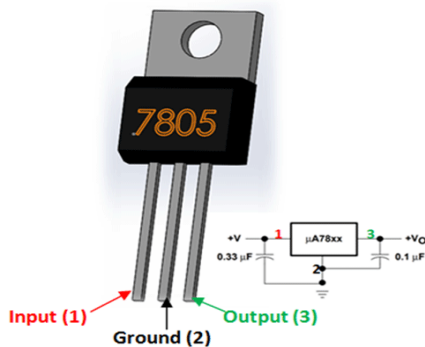
# Components use in the circuit

## 1. 7805 AS +5V VOLTAGE REGULATOR IC

This is a typical application circuit of the 7805 IC. We just need two capacitors of vale 33uf and 0.1uf to get this IC working.



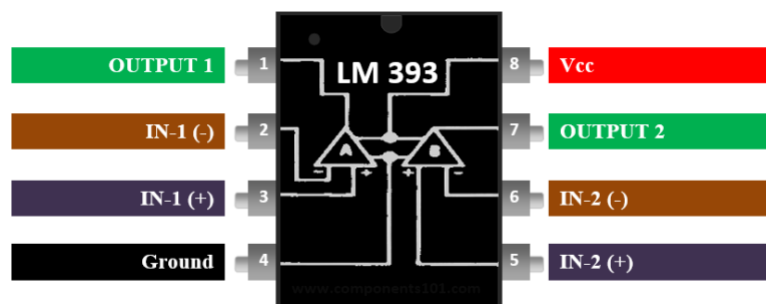
The input capacitor  $0.33\mu F$  is a ceramic capacitor that deals with input inductance problem and the output capacitor  $0.1\mu F$  is also a ceramic capacitor that adds to the stability of the circuit. These capacitors should be placed close to the terminals for them to work effectively. Also they should be of ceramic type, since ceramic capacitors are faster than electrolytic.



### Pin Configuration

Pin Number	Pin Name	Description
1	Input (V+)	Unregulated Input Voltage
2	Ground (Gnd)	Connected to Ground
3	Output (Vo)	Outputs Regulated +5V

## 2. LM393 - Dual Comparator IC



### PIN CONFIGURATION

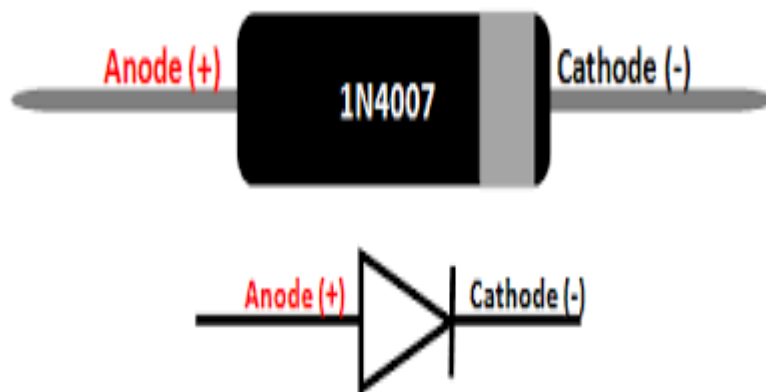
Pin Number	Pin Name	Description
1	OUTPUT1	Output of Op-Amp 1
2	INPUT1-	Inverting Input of Op-Amp 1
3	INPUT1+	Non-Inverting Input of Op-Amp 1
4	VEE,GND	Ground or Negative Supply Voltage
5	INPUT2+	Non-Inverting Input of Op-Amp 2
6	INPUT2-	Inverting Input of Op-Amp 2
7	OUTPUT2	Output of Op-Amp 2
8	VCC	Positive Supply Voltage

The **LM393 IC** can be considered as the **equivalent comparator** version of the most popular [LM358 Op-Amp](#). While any Op-Amp can be made to work as a voltage comparator, the LM393 proves itself to be advantages by providing an open collector output making it suitable to drive loads.

The output transistor can drive loads upto 50V and 50mA which is suitable for driving most of the TTL, MOS and RTL loads. The transistor can also make the Load to be isolated from the system ground. So if you are looking for a Voltage comparator to drive loads of these specifications then this IC might be the right choice for you.

### 3. 1N4007 Diode

It is a **diode** of 1N400x series in which there are also other similar **diodes** from 1N4001 to **1N4007** and the only difference between them is the max repetitive reverse voltage. ... The **1N4007 diode** is built for working with high voltages and it can easily handle voltage below 1000V.



#### 1N4007 Diode Pinout



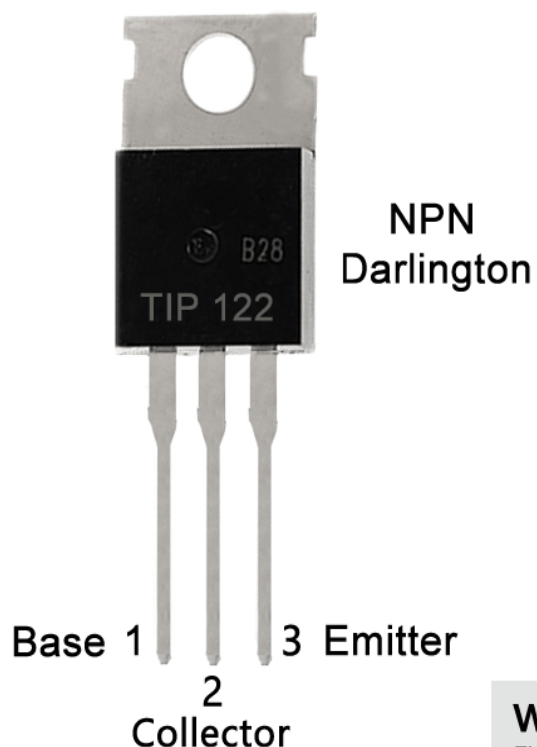
[www.componentsinfo.com](http://www.componentsinfo.com)  
Electronics Components Uses, Features, Pinouts, Equivalents,  
Applications & More.

## 4. TIP122 NPN TRANSISTOR

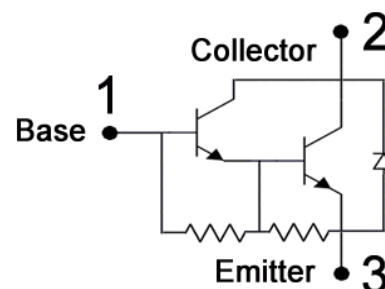
**TIP122** is a three layer NPN device within the working range, the collector current  $I_C$  is a function of the base current  $I_B$ , a change in the base current giving a corresponding amplified change in the collector current for a given collector emitter voltage  $V_{CE}$ .

# TIP122 Transistor Pinout

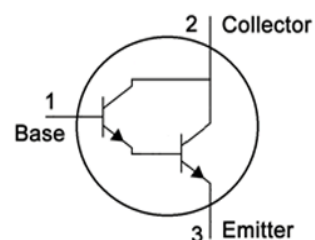
TO - 220 Package



Internal Circuit



Symbol In Diagram



[www.componentsinfo.com](http://www.componentsinfo.com)

Electronics Components Uses, Features, Pinouts, Equivalents, Applications & More...

## 5. RESISTORS

- 1-KILO-OHM



### 1k0 / 1k ohm Resistor Colour Code

Value	1 k $\Omega$ / 1000 $\Omega$
Type	4 Band Colour Code
Colour Code	Brown, Black, Red, Gold
Multiplier	Red, 100

Tolerance

**Gold Band  $\pm 5\%$**

**One kilohm** is equal to 1,000 **ohms**, which is the resistance between two points of a conductor with **one** ampere of current at **one** volt. The **kilohm** is a multiple of the **ohm**, which is the SI derived unit for electrical resistance.

- **15 OHM, 0.5W**



**Value**

**15  $\Omega$**

Type

4 Band Colour Code

Colour Code

**Brown, Green, Black, Gold**

Multiplier

**Black, 1**

Tolerance

**Gold Band  $\pm 5\%$**



- **10-KILO-OHM POTMETER**

**ROBOMART.com™**



**10 Kilo Ohm Register**  
(Pack of 10)

---

Type	: 1/4 Watt
Value	: 10 Kilo /1000 Ohm/10K

NOTE: THE PRODUCT MAY BE DIFFER FROM IMAGE GIVEN ABOVE Copyrights by Robomart.com

<b>Value</b>	<b>10 k<math>\Omega</math></b>
<b>Type</b>	<b>4 Band Colour Code</b>
<b>Colour Code</b>	<b>Brown, Black, Orange, Gold</b>

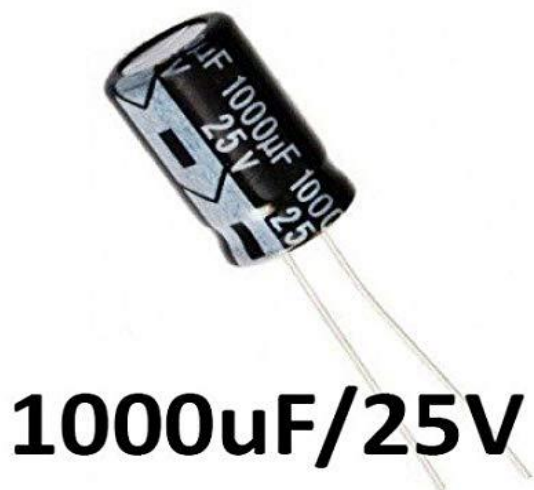
Multiplier	Orange, 1000
Tolerance	Gold Band $\pm 5\%$

As the symbol suggests a **potentiometer** is nothing but a **resistor** with one variable end. Let us assume a **10k potentiometer**, here if we measure the resistance between terminal 1 and terminal 3 we will get a value of **10k** because both the terminals are fixed ends of the **potentiometer**

## 6. CAPACITOR

- **1000  $\mu\text{F}$ , 25V electrolytic**

The microfarad (symbolized  $\mu\text{F}$ ) is a unit of capacitance, equivalent to 0.000001 (10 to the -6th power) **farad**. ... In RF scenarios, capacitances range from about 1 pF to **1,000** pF in tuned circuits, and from about 0.001  $\mu\text{F}$  to 0.1  $\mu\text{F}$  for blocking and bypassing.



# ADVANTAGES & DISADVANTAGES OF PCB

## **ADVANTAGES:**

### **Compact Size and Saving of Wire**

- A characteristics pcb includes a large no. Of electronics components on a printed circuit board. The inter connection between the components is made through. Copper track instead of using a no. Current carrying wires.
- Ease of Repair and Diagnostic. If in case of any damage, it's very easy to check and replace the particular failure components. ...
- Saving of Time. ...
- Immune to Movement. ...
- Tight connections and Short Circuits Avoided. ...
- Low Electronic Noise. ...
- Low Cost. ...
- Reliability.

## **DISADVANTAGES:**

- Easy to Cause Handling Damage.
- Process Uses a Carcinogen (Thiourea)
- Exposed Tin on Final Assembly can Corrode.
- Tin Whiskers.
- Not Good for Multiple Reflow/Assembly Processes.
- Difficult to Measure Thickness.

## REFERENCES

1. Willson, C. G., Dammel, R. R., and Reiser, A (1997). Tarascon-Auriol, Regine G (ed.). "Photoresist materials: a historical perspective". *Advances in Resist Technology and Processing XIV*. 3049: 28. Bibcode:1997SPIE.3049...28W. doi:10.1117/12.275826. S2CID 136616549.
2. ^ "Lithography".
3. ^ a b c d Lathrop, Jay W. (2013). "The Diamond Ordnance Fuze Laboratory's Photolithographic Approach to Microcircuits - IEEE Journals & Magazine". IEEE Annals of the History of Computing. 35: 48–55. doi:10.1109/MAHC.2011.83. S2CID 2562671.
4. ^ a b Weightman, Gavin (2015). *Eureka: How Invention Happens*. Yale University Press. pp. 178–179. ISBN 978-0300192087.
5. ^ "Jay W. Lathrop | Computer History Museum". [www.computerhistory.org](http://www.computerhistory.org). Retrieved 2018-06-18.
6. ^ Lécuyer, Christophe (2010). *Makers of the Microchip: A Documentary History of Fairchild Semiconductor*. The MIT Press. ISBN 978-0262014243.
7. ^ Jaeger, Richard C. (2002). "Lithography". *Introduction to Microelectronic Fabrication* (2nd ed.). Upper Saddle River: Prentice Hall. ISBN 978-0-201-44494-0.
8. ^ Zhao, X-A; Kolawa, E; Nicolet, M-A (1986). "Reactions of thin metal films with crystalline and amorphous Al<sub>2</sub>O<sub>3</sub>". California Institute of Technology.
9. ^ "Semiconductor Lithography (Photolithography) - The Basic Process".