LED UV EXPOSURE UNIT FOR DIY PCBs

This manual details how to make an A4 UV LED exposure unit , using cheap and common parts.

While it shows what was used, how it was done and how well it worked , there is no guarantee you will get the same results as all batches of UV LEDs vary slightly, but with the details provided you should be able to overcome any slight problems.

Please note that no details of Mains voltages and the associated mains circuitry are given - that is entirely up to you, if you cannot design such things, then no way should you be building a unit like this.

WARNING - UV LIGHT CAN SERIOUSLY DAMAGE YOU EYES - WEAR EYE PROTECTORS AT ALL TIMES

The Box

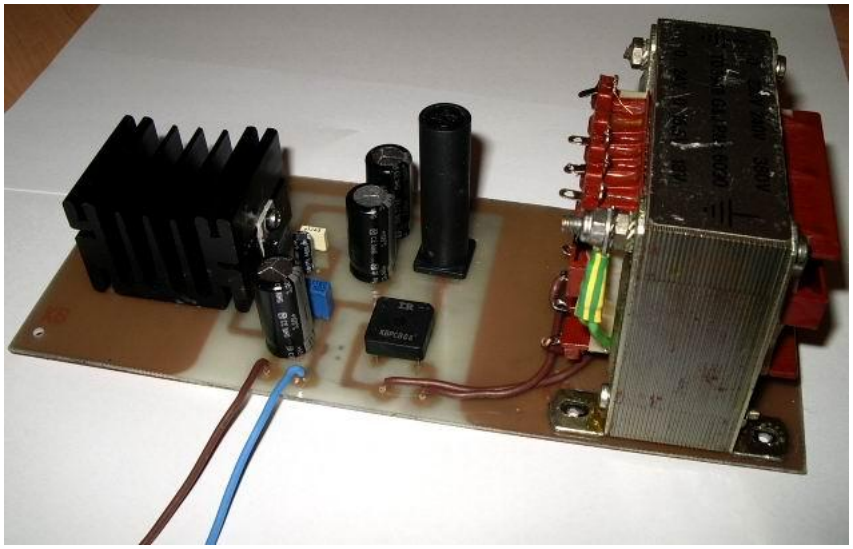
This was made from 15mm chipboard, the height needs to be about 230 – 300mm, the width and depth were made to accommodate the two A5 led boards

The Led board needs to be around 150mm or more, below the glass and you need room for the PSU as well. The main glass is in 4mm and the cover glasses are in 3mm.

The glass supports are fitted around the top edge of the box so that the top of the cover glass is set fractionally below the top of the box. When measuring the position of the glass support - the depth should be the thickness of the two glasses plus a piece of copper board.

The box lid should have a series of pieces of foam tape pads so that when the lid is closed it applies gentle pressure to the covers to ensure the mask is in good contact with the copper board.

The PSU

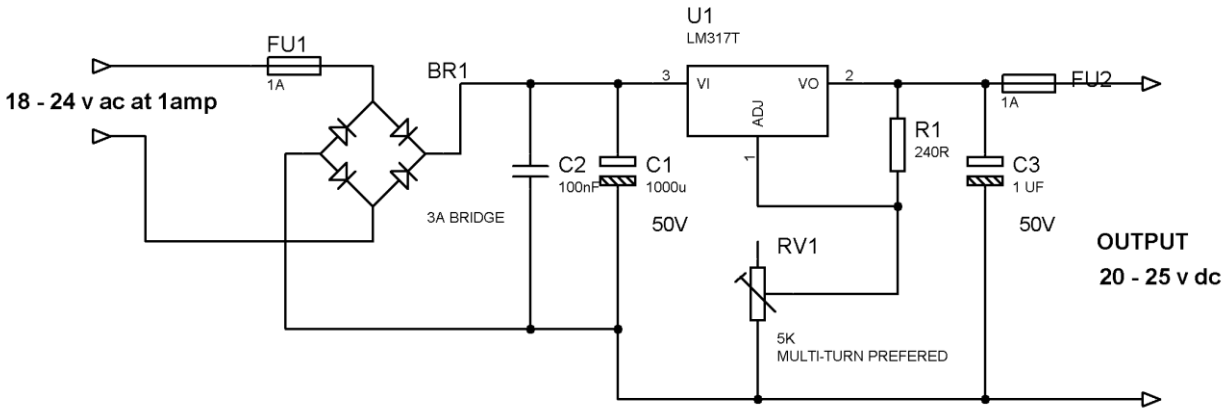


This is a very simple adjustable power supply based on an LM317 regulator.

The parts shown are all from my 'junk' box - nothing special apart from U1,RV1 and R1 values

With the two A5 led boards, detailed later, the PSU needs an AC input of 18 -24 v rated at least 1 amp.

The Leds demand is approximately 23.7vdc at 500 ma.

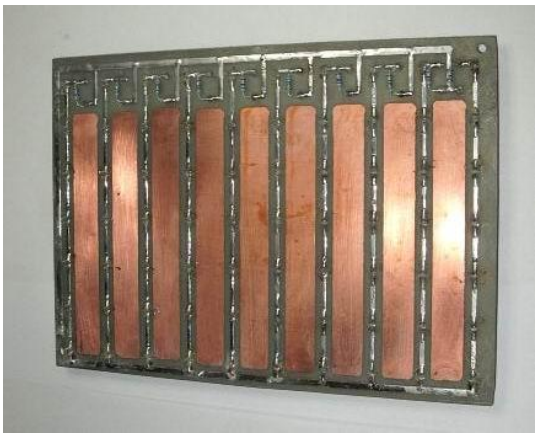


The LED PCB

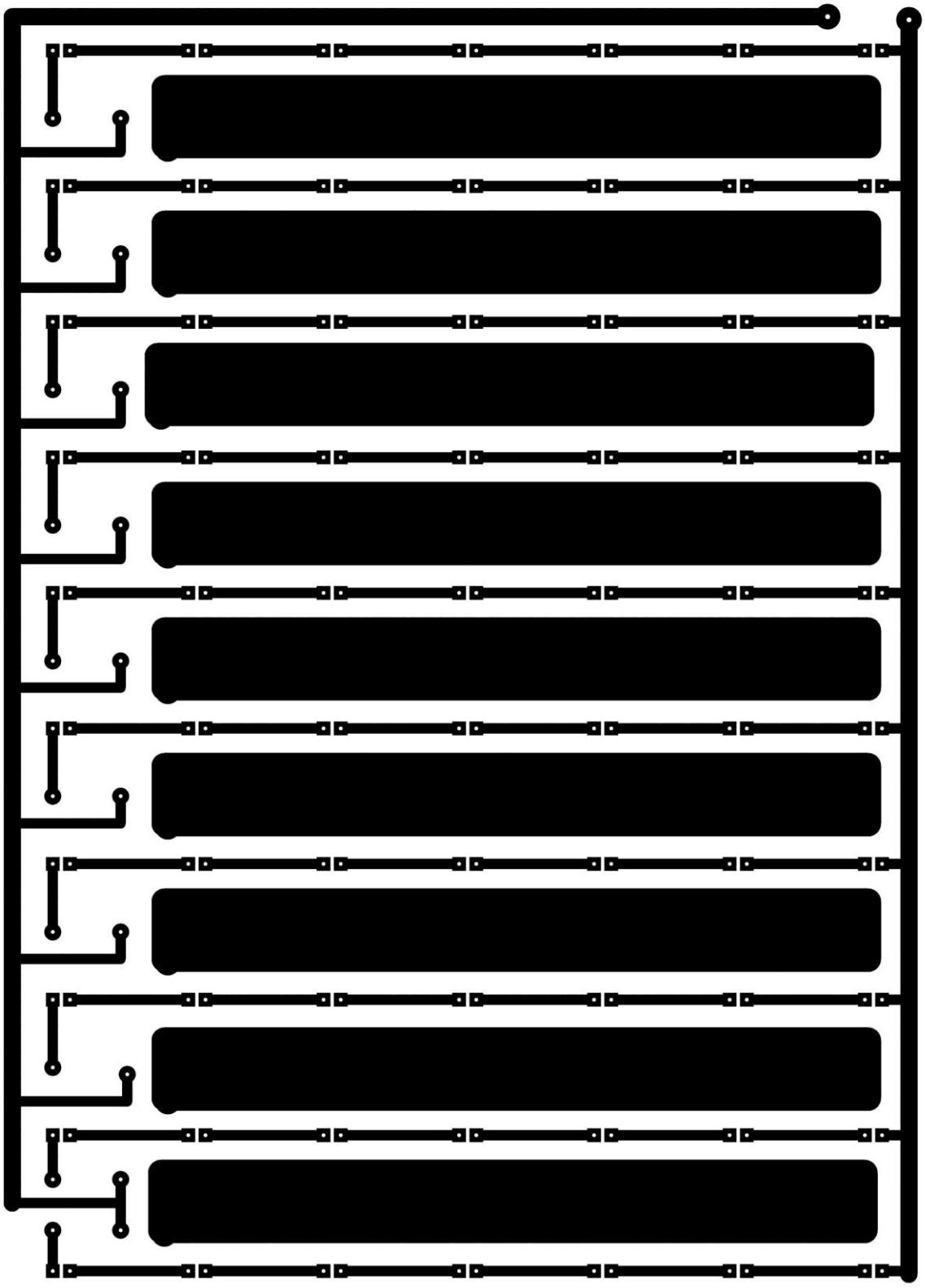
Paper transferred board prior to etching

Finished Led Board

Surface painted silver or white to reflect light before fitting the leds



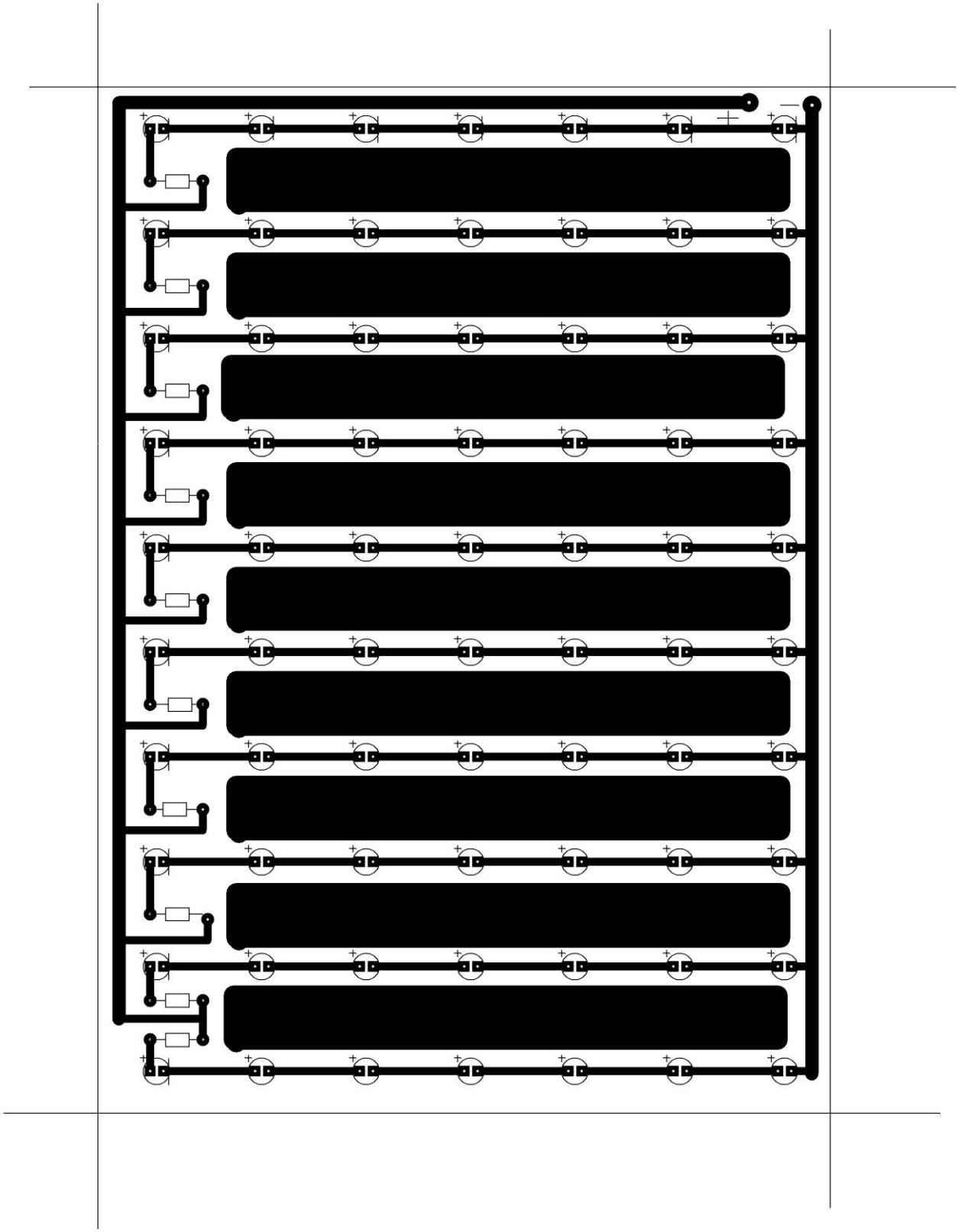
Copper Side – showing 1R resistors or wire links



Page 5 above - the LED PCB - Image scaled to be correct size 140mm x196mm when printed.

6

PCB SILK



LED BOARD SILK

The UV LEDS

The UV LEDS can be sourced from many places , a local web search should find suitable ones in your area.

Below is the specification of 2 types I have purchased and found to work well.

The most important factor is the 20 degree Viewing Angle and 5mm size.

When ordering allow 10% on top of what you need in case a few are below specification or you damage any when fitting.

Order your leds in one batch and mention to the supplier that you need them to be from the same batch to ensure they are all the same intensity etc.

SUPPLIER 1

WATERCLEAR ULTRA VIOLET 130mcd 5mm LED
FORWARD VOLTAGE 3.0--3.6 Vf
DOM WAVELENGTH 400--410nm
POWER DISSAPATION 170mW
LUMINOUS INTENSITY 90--130mcd
FORWARD CURRENT 20m/a
VIEWING ANGLE 20degrees

SUPPLIER 2

Source Material: InGaN
Emitting Colour: UV/Purple
Emitting Wavelength (nm): 395-400
Size: 5mm
Intensity (mcd): 100-150
Viewing Angle (deg): 20
Forward Voltage (V): 3.2-3.4
Forward Current (mA): 20-30

Testing the LEDS

Once you get the leds you should test then to check they work and have similar voltages / currents before fitting them to the PCB.

This is done by using a dc voltage typically with 4.5 or 5v with a 100 ohm resistor in series.

Most of the leds will show a very similar voltage , any that are more that 0.2 volt different keep to one side .

Building and testing the LEDS boards

My board consists of 10 rows of 7 leds each row is connected to the positive rail by a 1 ohm 0.6W resistor or wire link which is fitted after a current test.

Fit all the leds to the board, ensuring the base fits squarely to the board, the rows should look straight and even to ensure an even light pattern.

Set up the variable power supply to give an output of 20v dc.

Connect the PSU 0v to the PCB 0v and connect the PSU positive via your MA Meter to the top most led of row 1.

Hopefully the row will illuminate, so now adjust the PSU to give a reading of 20ma.

Repeat this exercise on each row in turn, but do not adjust the PSU any more - each row should give a very similar reading of around 20ma - if its more than 2ma out either way – look for the reason / fault.

When all 10 rows test ok, then fit the 1ohm resistors or wire links to connect each row to the positive rail.

Now repeat the current test on the whole board - this should show around 200ma and you can trim the psu to that.

Note that the current rises slightly as the leds warm up - so let them run for 1 minute before your final adjustment

If you make two A5 boards the total load is 400ma.

If you wish to make up a Led array of your own design this link is very helpful **http://led.linear1.org/led.wiz**

Fitting the LED Boards to the Case

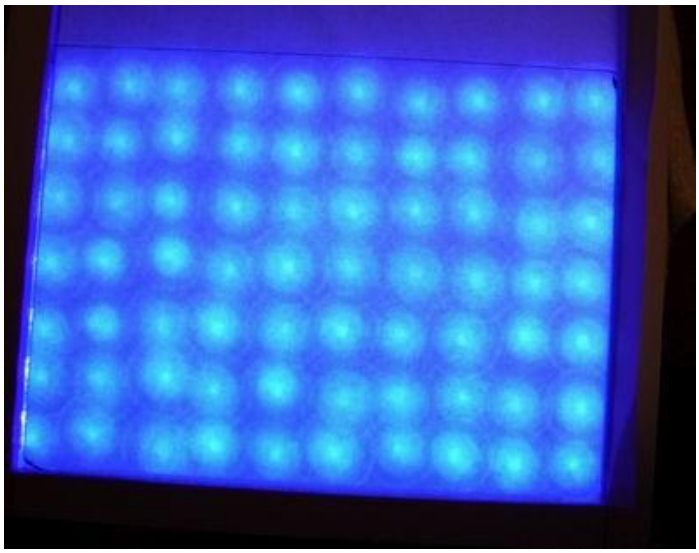
The distance between the copy glass and the led boards is important, to close and the beam from the leds creates hot spots - too far and the intensity becomes too low and exposure too long.

I found a compromise at a distance of 150mm , which although not seeming to give a totally even spread of light, when tested on actual pcbs an excellent image is produced across the whole board right up to the edges.

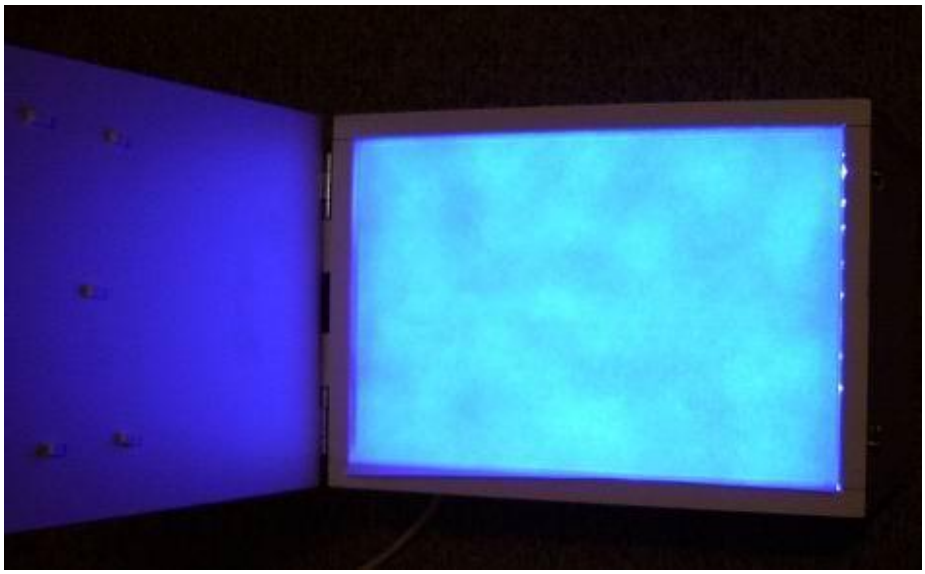
Your boards many need to be closer or further away to suit your leds

Make up a temporary adjustable shelf for the led board to rest on and place a sheet of plain white copy paper on the glass to show the led patterns and test at varying depths.

The pictures below show typical views - this is 25mm depth - too close



This is the working depth of 150mm



The completed UV box with just one board show.

Although the time is manually controlled, fitting a simpler timer does avoid over exposures.

With two A5 boards, you can make a similar small box or stand to fit on top of the glass so allowing an A5 double sided pcb to be exposed.

Exposing, Developing and Etching the Circuit Board

Preparing the Board

I have used two different manufacturers for the pre coated pcbs, and both gave equally good results and very similar exposure times.

When handling the boards, obviously minimize the handling time as much as possible to avoid any UV from sunlight etc.

For cutting the boards, you can simply use a hand saw or I find an electric tile cutter makes easy work.

The plastic cover should stay in place - once cut return to a light proof bag.

Any cut board should have a file run down its edges at an angle to remove any copper burs otherwise the board will not sit totally flush with the copy.

Exposure

I have only used a laser printer to produce the copy, but I have heard that an inkjet will also work.

The copy paper need to be transparent or semi transparent - I have used common tracing paper to good effect as well as the top quality LaserStar film.

For your initial exposure trials it is often recommended you do a step wedge test, but I found it much easier just to do simple test exposures on pieces of board about 50mm x 50mm.

Place your test copy and pcb between the glasses and expose for 4 mins - this should give some useful copy.

After developing you may find the tracks are not 100% then try again at 3 or 5mins.

This can be fine tuned but I found time changes of less than 30 seconds made little difference to the board.

Developer and Etchants

The developer and etchant should be ready together with a tray of clean water or a running tap before you expose your boards.

A developer frequently sold for developing pcbs is caustic soda - it does work – but has one very big problem that it develops image in about 5 seconds and if you do not stop the process by rinsing in water the actual track image will also be developed away as well. - it is really too aggressive for the beginner.

There is a product about, and you may have to search it out, call Sodium Metasilicate - sold around most of the world by RS code - 690 849 , but you may find smaller quantities locally

This is a much slower process, typically 20 seconds so you are much less likely to loose the actual image.

It should be prepared as instructed and warmed to 20c to be at its best.

Similarly, the ferric chloride should be warmed to around 35 -40c for best results.

Developing

Using gloves and tongs etc to hold the board, immerse it totally in the developer so you can see the image.

Keeping it fully submerged gently rock the tray so the developer swirls over the board.

In about 15 seconds you will see a feint image of you tracks appear - now remove the board and immediately put it in the water tray or under a running tap to stop the developing process.

Although you may see the feint image it is hard to initially know if it has completely developed - what can be done is to place the board in the ferric bath for 10 seconds then again rinse in water.

The exposed , unwanted copper areas should now all be an even pink /purple colour where the etching process has started, any area still with the coating on is very noticeable - if so, rinse the board in water again, then return to the developer bath for about another 10 seconds and repeat the process.

This may sound complicated – but it is just you getting to know the products and how they behave - it soon becomes an easy process.

Etching

Once you are happy with the development, return the board to the ferric bath and rock gently for about 10-20 mins, lifting the board out occasionally to check its progress - when you can see the drill holes in the component pads have been etched its usually ready.

Rinse the board well with water, leave the remaining coating on the board until you are ready to solder the parts.

Spray your own boards

You can buy sprays to coat your own boards, and having used one they do work.

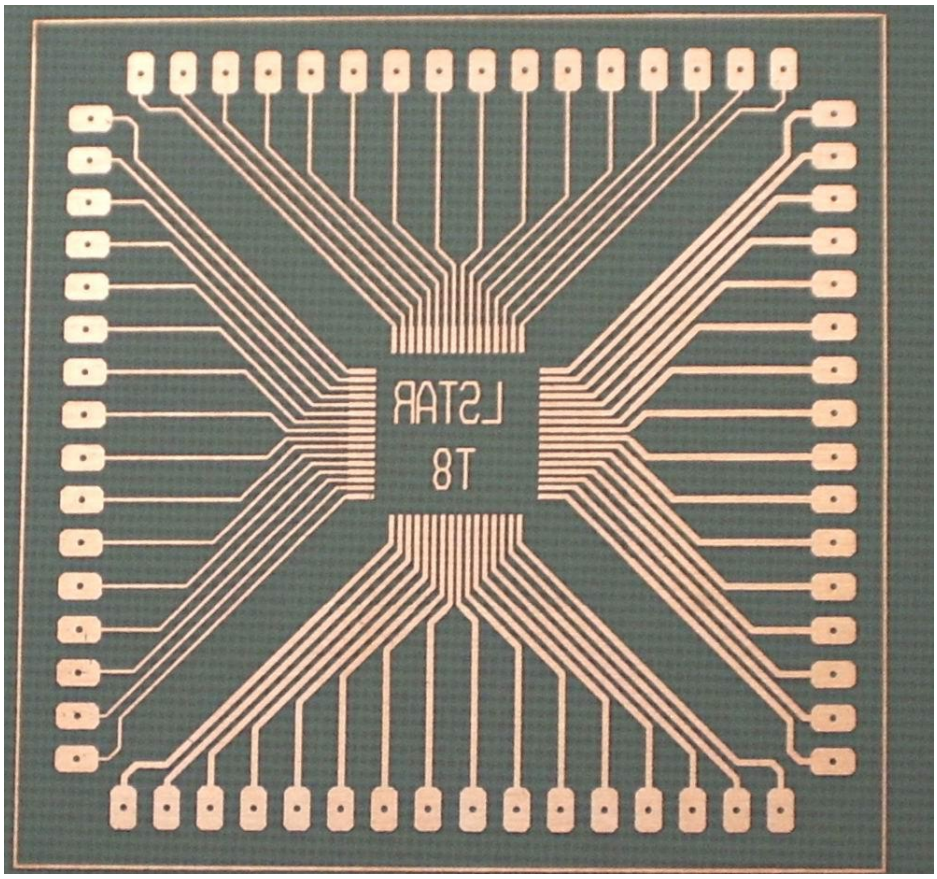
However it needs the boards to be heat treated / baked before and after spraying with the coating.

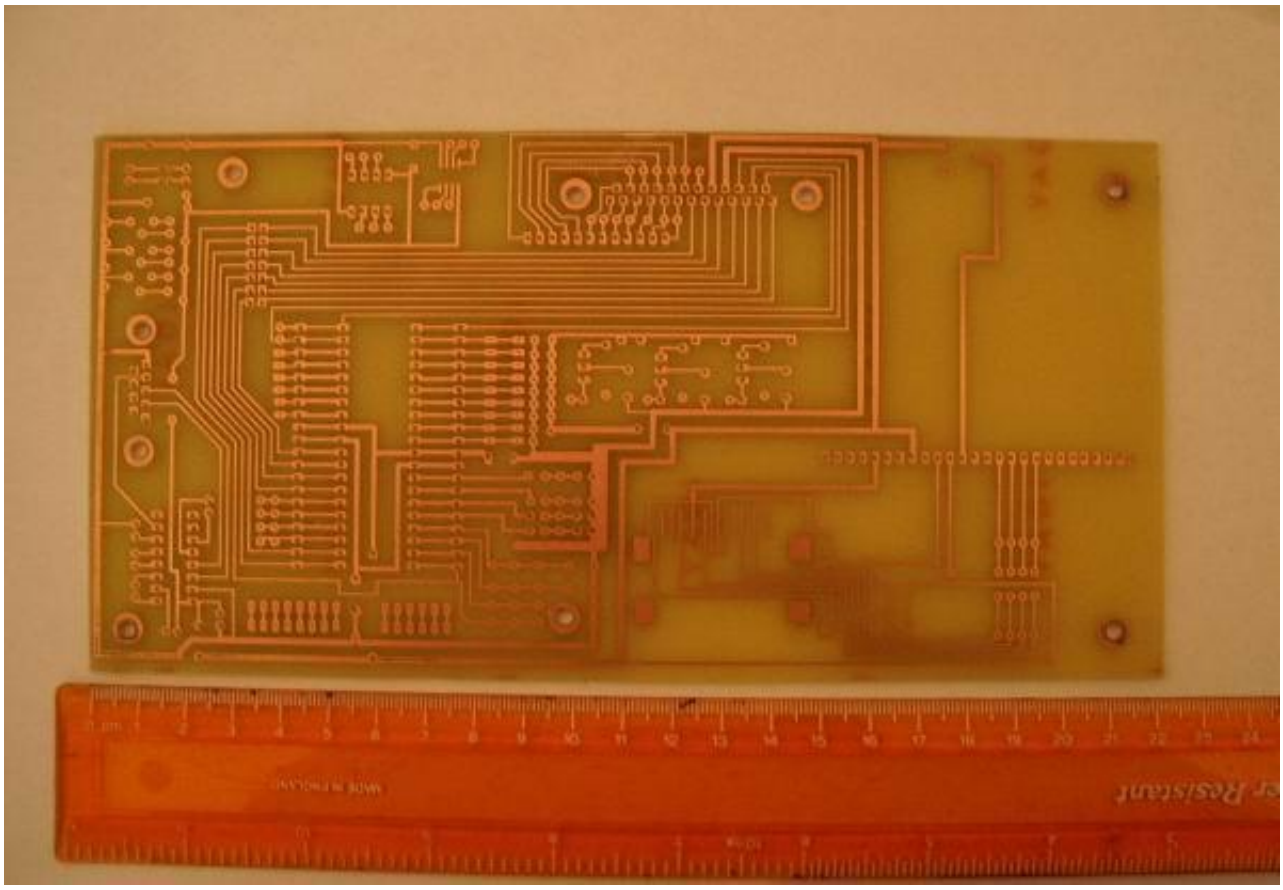
Every piece of board you spray will have different thicknesses of spray on them - is virtually impossible by hand spraying not to do this.

As a result every board needs a different exposure time because of the varying thickness of coating.

Someone very experienced at uv boards may be able to master this method, but for the beginner it is a total waste of time and money.

Some boards produced with the UV method





Finally don't forget to finish your boards with a component silk - simple done by laser printing on to glossy magazine paper and ironing on to the boards top surface - actually works easier than the copper side.

The actual results are a lot better than this picture shows.