PCB Manufacture Steps:

Process 1# : PCB Design and GERBER file

Process 2 # : GERBER to Photo Film

Process 3 # : Inner layer printing: Photo resists and Copper

Process 4 # : Removing the Unwanted Copper

Process 5 #: Layer Alignment and Optical Inspection

Process 6: Layer-up and Bonding

Process 7: Drilling

Process 8: Plating and Copper Deposition

Process 9 #: Outer Layer Imaging

Process 10: Plating of Outer Layer:

Process 11: Etching

Process 12: Solder Mask Application

Process 13: Surface Finish

Process 14 # : Silkscreen

Process 15 #: Electrical Test

Creating the printed circuit pattern on the substrate

Additive and Subtractive process:

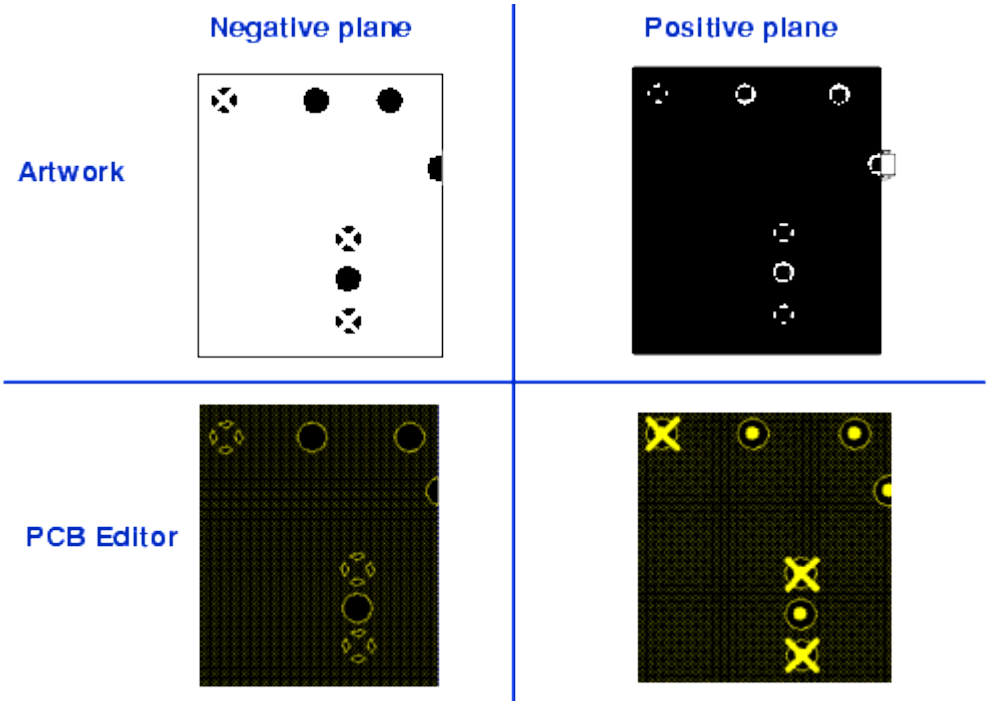
Subtractive: manufacturing is the earliest PCB production process, and it is also a relatively mature manufacturing technology. Photosensitive resist materials such as dry films are generally used to complete the pattern transfer, and the material is used to protect the traces and via holes.

The copper layer of the unprotected area is removed by an acidic or alkaline etching solution, forming the circuit traces by subtracting the copper layer.

Subtractive is also known as Tenting/Negative film PCB production process.

Additive: printing the circuit pattern on the substrate without copper foil, then plating to forming the copper traces. Since the traces need to be added to the printed circuit board, it is called additive PCB production process.

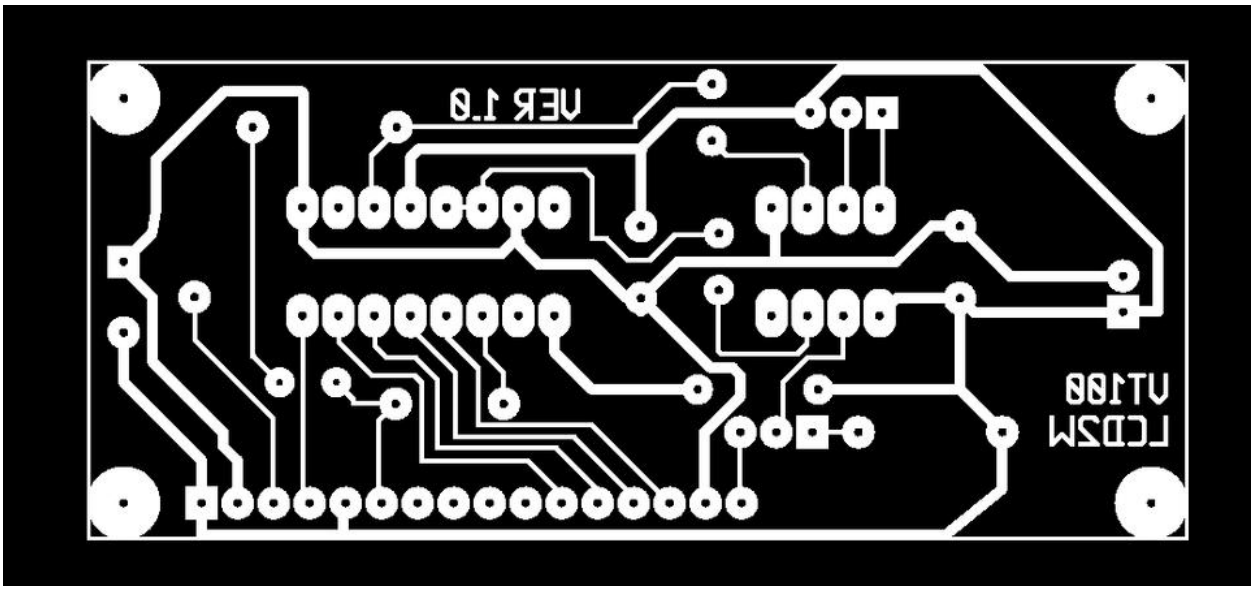
Positive vs Negative PCB or art work:



Negative Plane

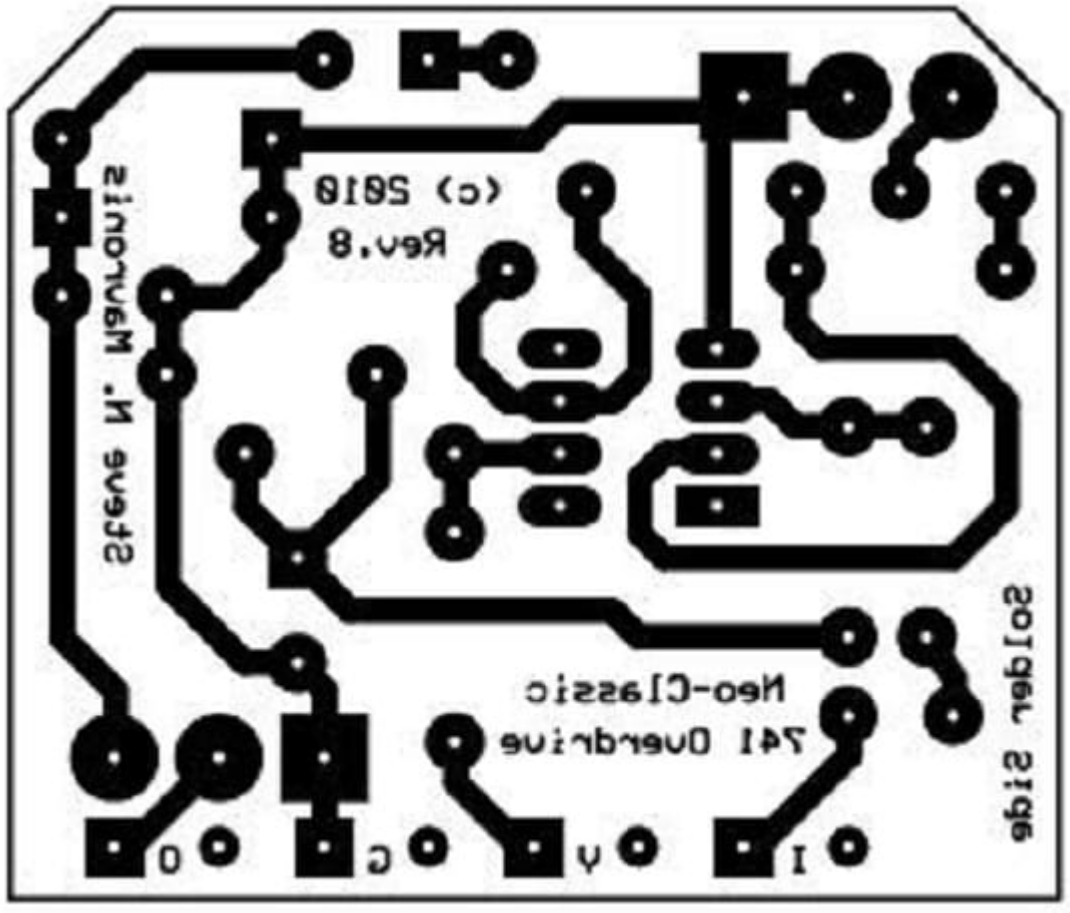
One advantage is that when you use the vector Gerber (6x or 4x) format, the artwork file size required to plot this copper area is much smaller because no data is required to fill the polygon, modern computers are fast enough to handle such task easily .

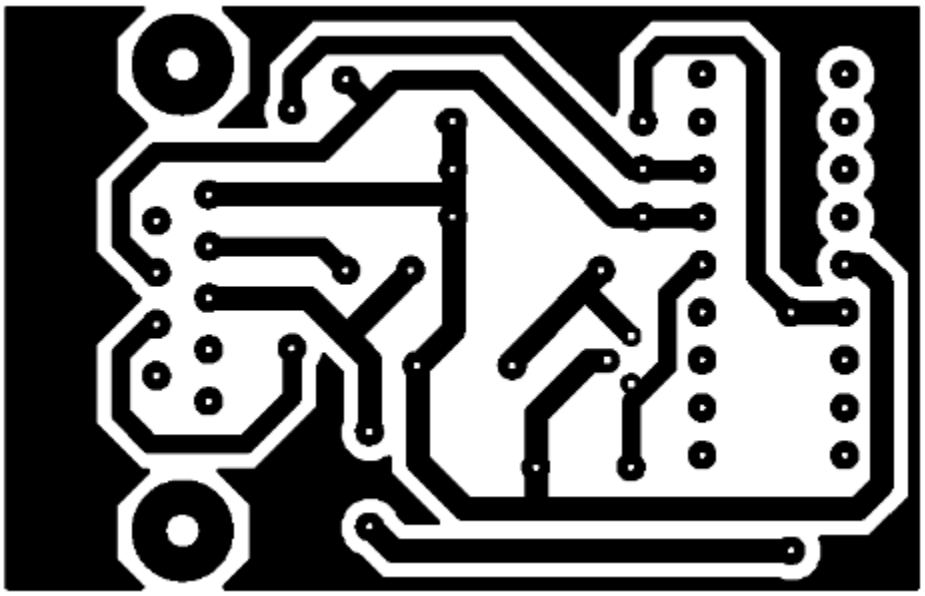
A disadvantage is that it's messy to understand and it's not used any more



Positive Image

One disadvantage is that if you are *not* generating rasterized output (RS274x), the artwork file size required to plot this copper area is larger because of the vector data required to fill the polygon.





Methods to Print PCB:

1- ***Photoresist Dry film method***

2- ***Tone Transfer***

1- *Photoresist Dry film method*

A-UV light process:

In lithography, decreasing the wavelength of light source is the most efficient way to achieve higher resolution.[9] Photoresists are most commonly used at wavelengths in the ultraviolet spectrum or shorter (<400 nm).

So we should use a UV light bulb led or incedecant or what ever lamp to cure the photoresist on the copper

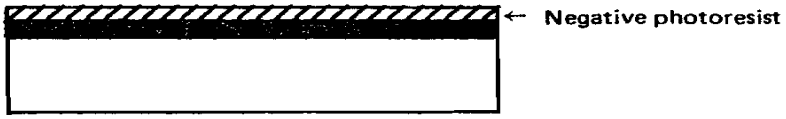
- *Positive photoresist and Negative photoresist process explained:*

  Definition:

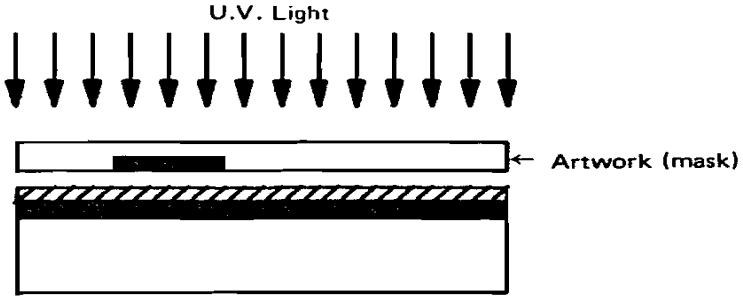
  is a process that uses a light-sensitive photoresist applied to the surface to be engraved to create a mask that shields some areas during a subsequent operation which etches, dissolves, or otherwise removes some or all of the material from the unshielded areas. Normally applied to metal, it can also be used on glass, plastic and other materials.

Clean and dry metallized substrate

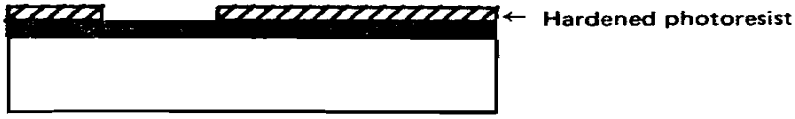
Spin on negative photoresist, dry and soft bake



Expose to U.V. light through a negative image mask



Remove mask and develop by dissolving unexposed areas in suitable solvent hard bake resist



Chemical etch metal then remove photoresist

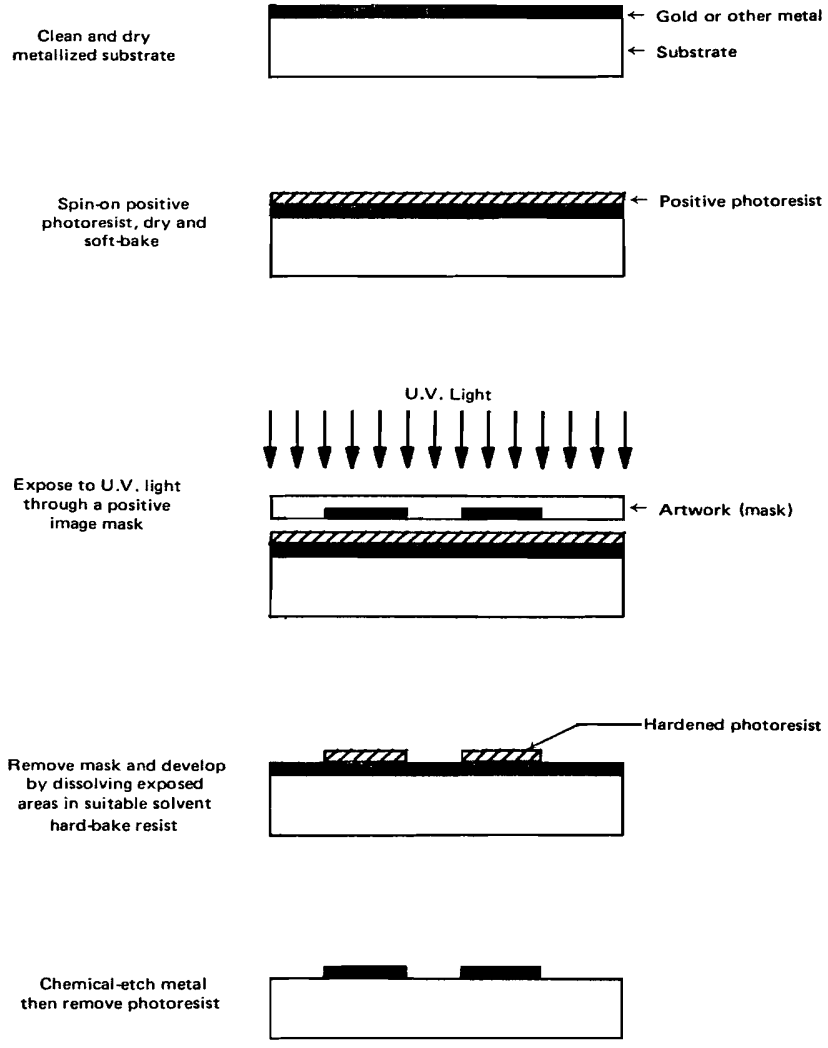


Figure 16. Steps for photolithography using positive photoresist.

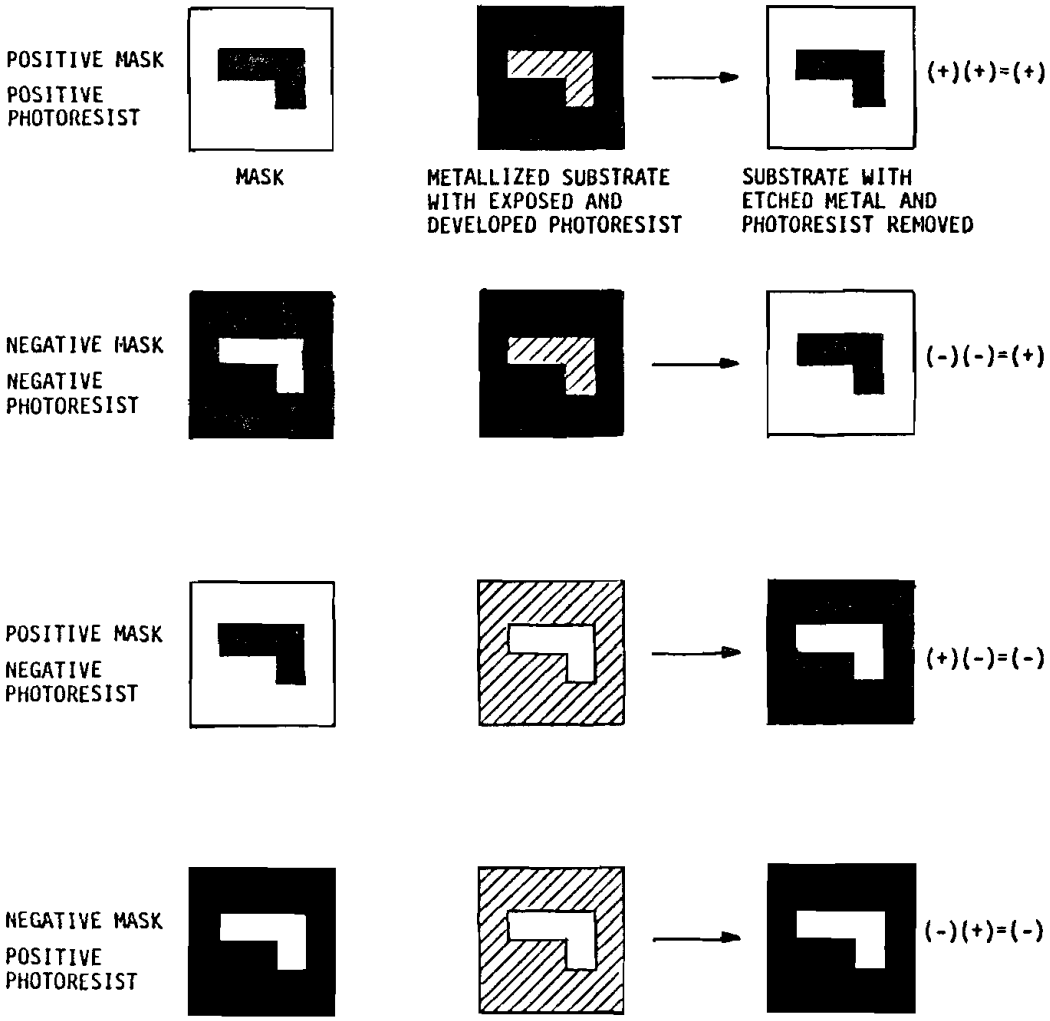


Figure 17. Mask/photoresist combinations.

Positive photoresist:

A positive photoresist is a type of photoresist in which the portion of the photoresist that is exposed to light becomes soluble to the photoresist developer. The unexposed portion of the photoresist remains insoluble to the photoresist developer.

Negative Photoresist:

A negative photoresist is a type of photoresist in which the portion of the photoresist that is exposed to light becomes insoluble to the photoresist developer. The unexposed portion of the photoresist is dissolved by the photoresist developer.

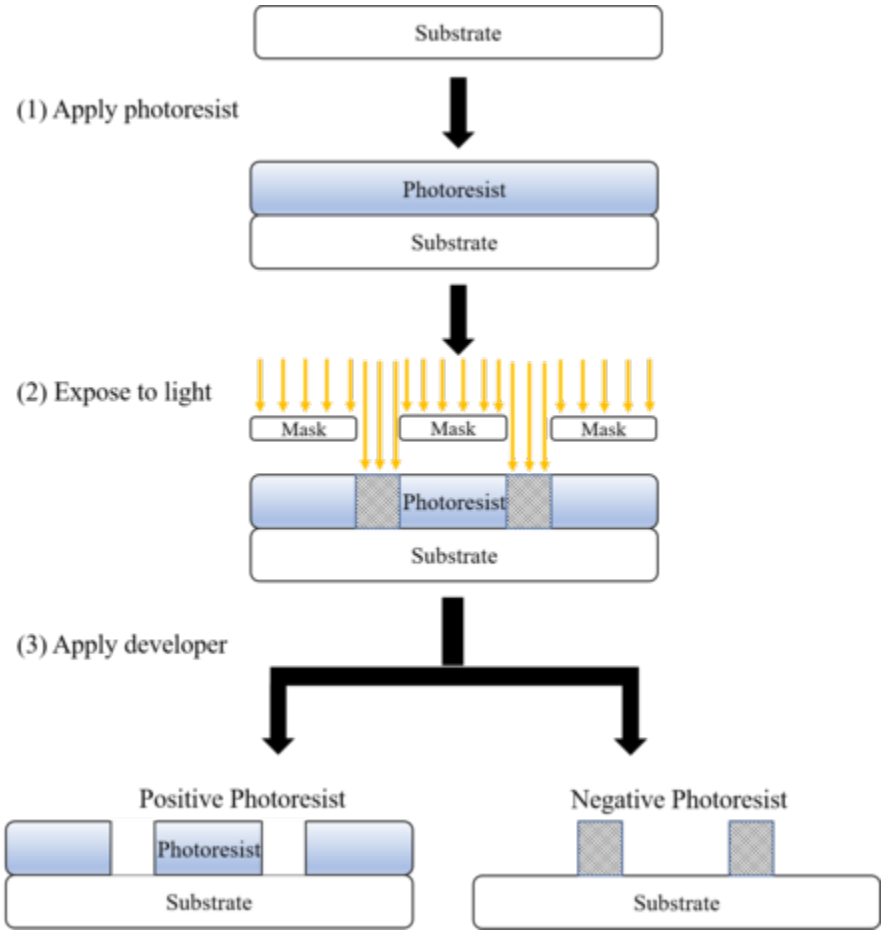
Characteristic	Positive	Negative
Adhesion to silicon	Fair	Excellent
Relative cost	More expensive	Less expensive
Developer base	Aqueous	Organic
Solubility in the developer	Exposed region is soluble	Exposed region is insoluble
Minimum feature	0.5 μm	2 μm
Step coverage	Better	Lower
Wet chemical resistance	Fair	Excellent

Material Types:

Photopolymeric photoresist is a type of photoresist, usually allyl monomer, which could generate free radical when exposed to light, then initiates the photopolymerization of monomer to produce a polymer. Photopolymeric photoresists are usually used for negative photoresist, e.g. methyl methacrylate.

Photodecomposing photoresist is a type of photoresist that generates hydrophilic products under light. Photodecomposing photoresists are usually used for positive photoresist. A typical example is azide quinone, e.g. diazonaphthaquinone (DQ).

Photocrosslinking photoresist is a type of photoresist, which could crosslink chain by chain when exposed to light, to generate an insoluble network. Photocrosslinking photoresist are usually used for negative photoresist.



B-Developing:

the surface of the panels is sprayed with an alkaline developer that dissolves the irradiated photoresist in the areas of the printed circuit pattern, leaving the copper foil exposed on the surface of the substrate.

Positive Developer

The 418 Positive Developer solution is used for removing exposed resist during the positive photofabrication process.

mixing one gram of sodium carbonate (Washing Soda) with one liter of water. It's ok if you don't have a scale, just use the picture I

C-Etching:

Types:

Wet Etching:

This is the simplest etching technology. All it requires is a container with a liquid solution that will dissolve the material in question. Unfortunately, there are complications since usually a mask is desired to selectively etch the material. One must find a mask that will not dissolve or at least etches much slower than the material to be patterned.

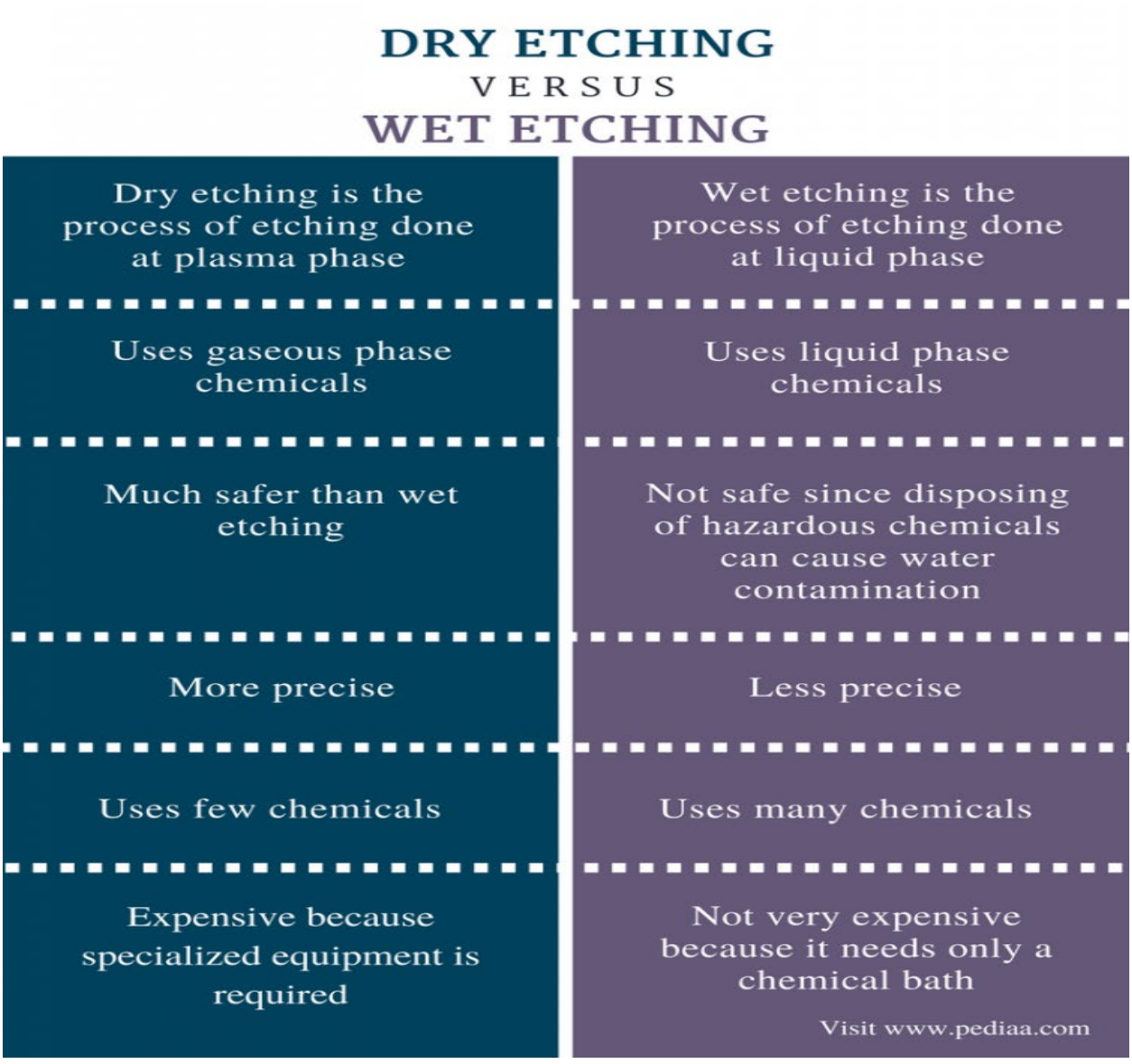
Dry Etching:

Dry etching is the process of etching done at plasma phase. Here, the material removal reactions occur in the gas phase. Dry etching is also called plasma etching. This process can be used to create designed patterns on surfaces. Dry etching can be used on printed circuit boards. There are two types of dry etching.

Non-Plasma Based Etching

Plasma Based Etching

Non-plasma-based etching uses a spontaneous reaction of an appropriate reactive gas mixture. The plasma-based etching uses radio frequency power to drive the chemical reaction.



Wet etching:

Negative Photoresist use acid Hydrochloric acid is a comparable non-critical medium for photoresist masks

Positive Photoresist use alkaline or base sodium or potassium hydroxide solution

https://hackaday.com/2012/12/10/10-ways-to-etch-pcbs-at-home/

D-How to electrically test PCB:

Opens

Shorts

Resistance

Capacitance

Inductance

Diode issues

Use UV cure ink pen or solution or a black OHP marker/pen to cure open circuit cuts in Traces

E- Solder Mask (permanent coating):

How to solder mask your PCB at home

F- PCB Simple Coating (Conformal Coating)

2- Tone Transfer:

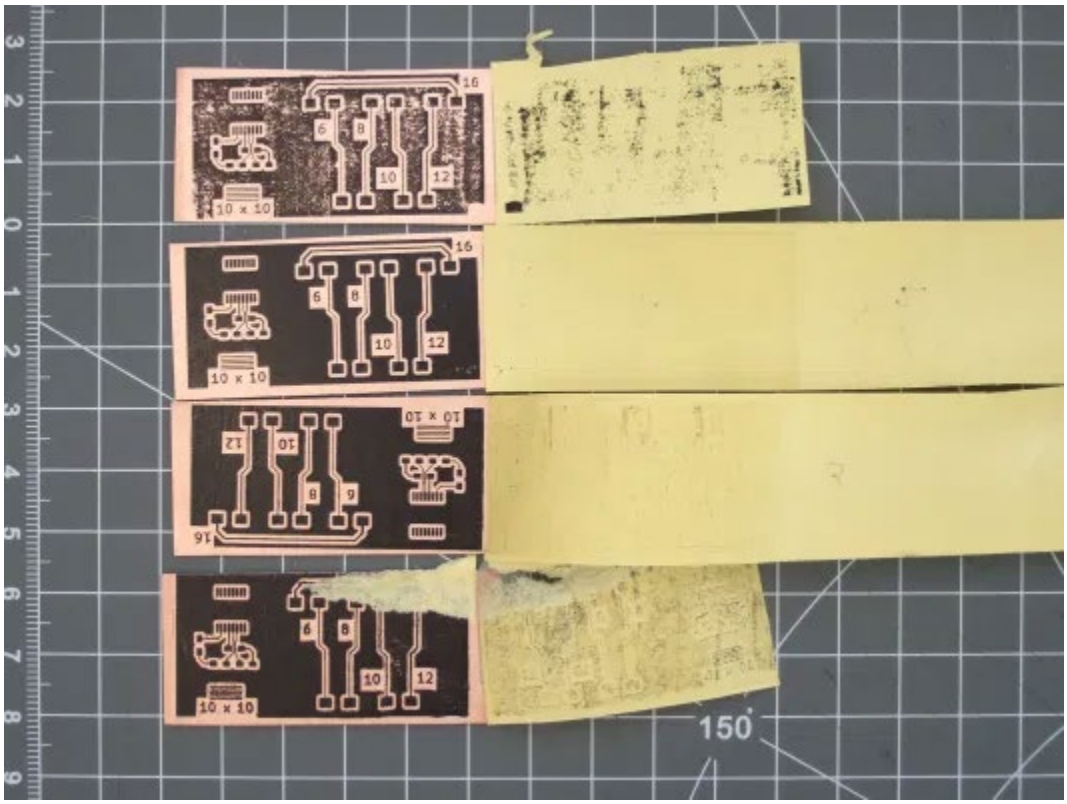
Also for this method, you need several trials, but as you get a good "preview" on the PCB before etching, you can just remove the toner with some solvents and try again. However, be prepared to produce more PCBs than you need, because pieces of the toner may separate from the PCB during etching.

Beneath the right technique of ironing (time, pressure), the paper is the most critical part. More glossy papers like those from magazines will not suck up the toner and leave more on the surface (better for the transfer), but the toner may also form a thinner film on them (not so good). And glossy papers don't soak as easily in water, so it's harder to peel them off without damaging the mask on the PCB. Some papers may contain some wax, oil or similar substances, which may also be transferred to the PCB. Clean the PCB gently with some dish liquid after transfer. (And also: clean it before the transfer with some good solvents like benzine or acetone.)

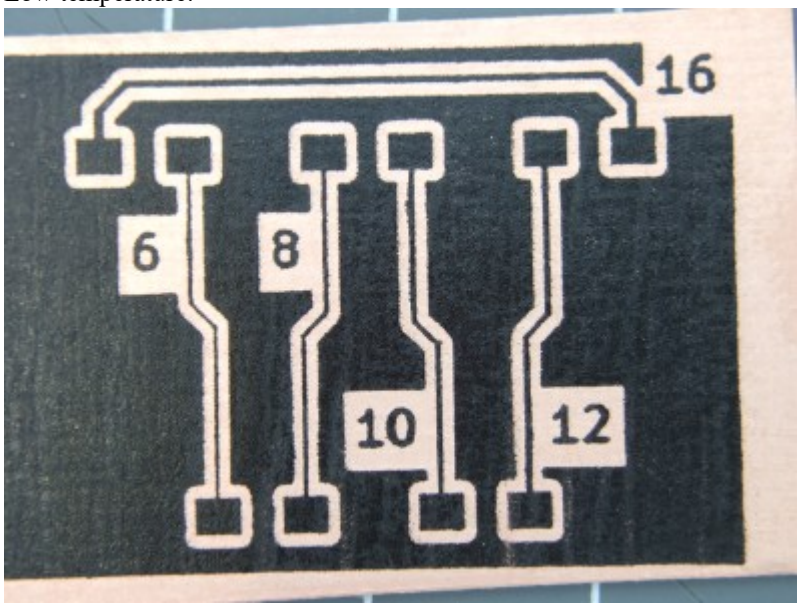
As edges are usually not that sharp and precise when using toner transfer, this method is not very suitable for PCBs with SMD components. You need a large clearance (space between tracks) and high

track widths. The bigger, the better. But if you're making PCBs very rarely and don't need too much precision, toner transfer is a good alternative.

For your board, I would increase the width by a factor of 2 or more. You can still use thinner tracks where needed, just inspect them more carefully after the transfer. Also, you are using thermals (slits around holes in the copper plane, making soldering easy). For toner transfer, I would not use them, as they are too small to come out nicely on the PCB.



Low temperature:



Hight temperature 142degree c

https://www.youtube.com/watch?v=cVhSCEPINpM