

# PCB Design & Fabrication

# Learning Objective

Learn about the various elements of Embedded Hardware and their design principles

Refresh knowledge on the basic Analog Electronic components and circuits - Resistor, Capacitor, Diode, Inductor, Transistor etc. and their use in embedded applications

Refresh knowledge on the basic Digital Electronic components and circuits

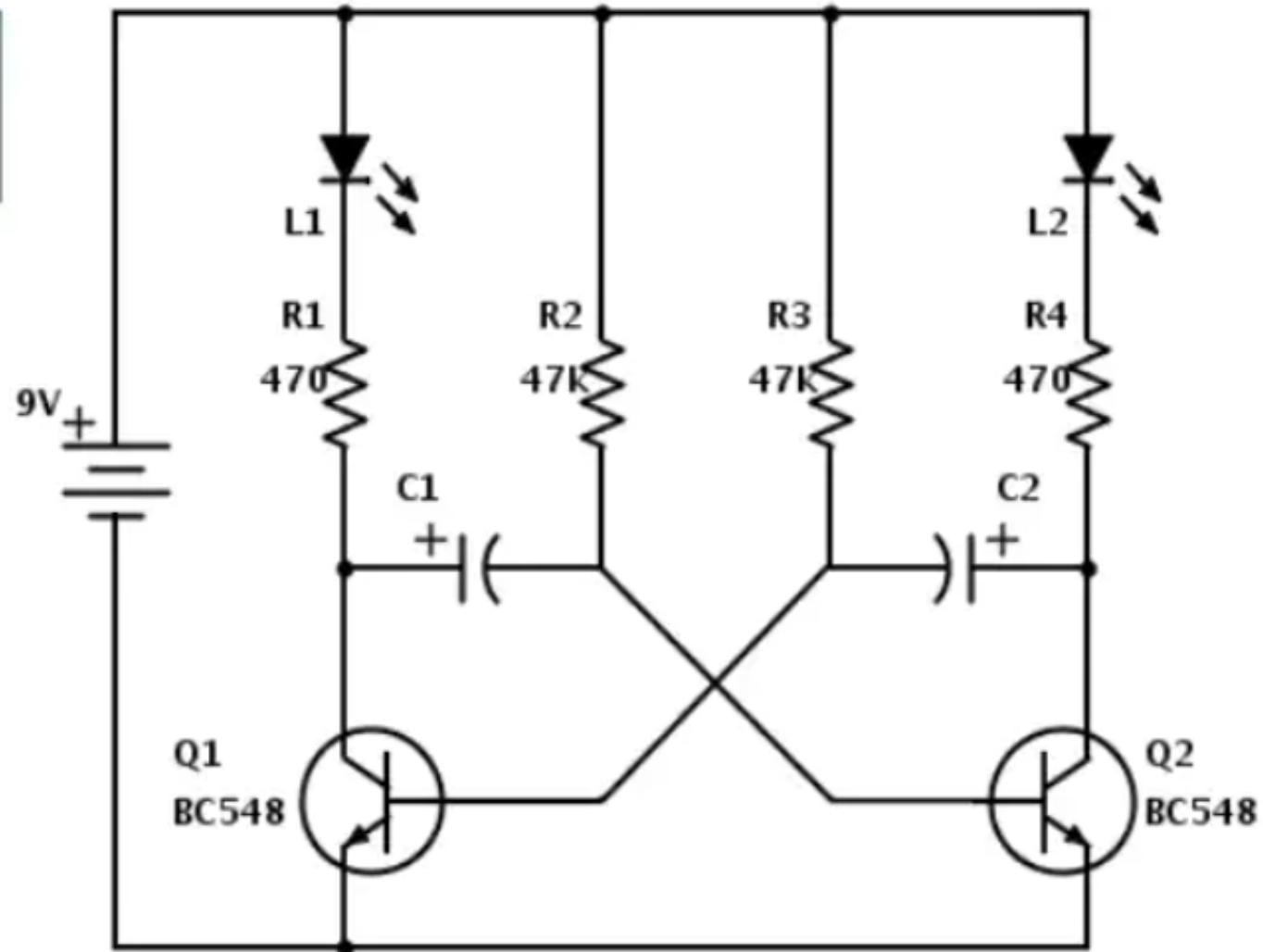
Learn about Integrated Circuits (IC's)

# Schematic Design Using

Schematic is a way of representing the different components (can be an electronic component like resistor, integrated circuit , capacitor, etc. or a mechanical/electromechanical component like push button switch, relay, etc.) involved in a hardware product and how each components are interconnected together.

Eagle tool, You can create a schematic design either by hand sketch on a paper or by electronic sketch

**For Example**

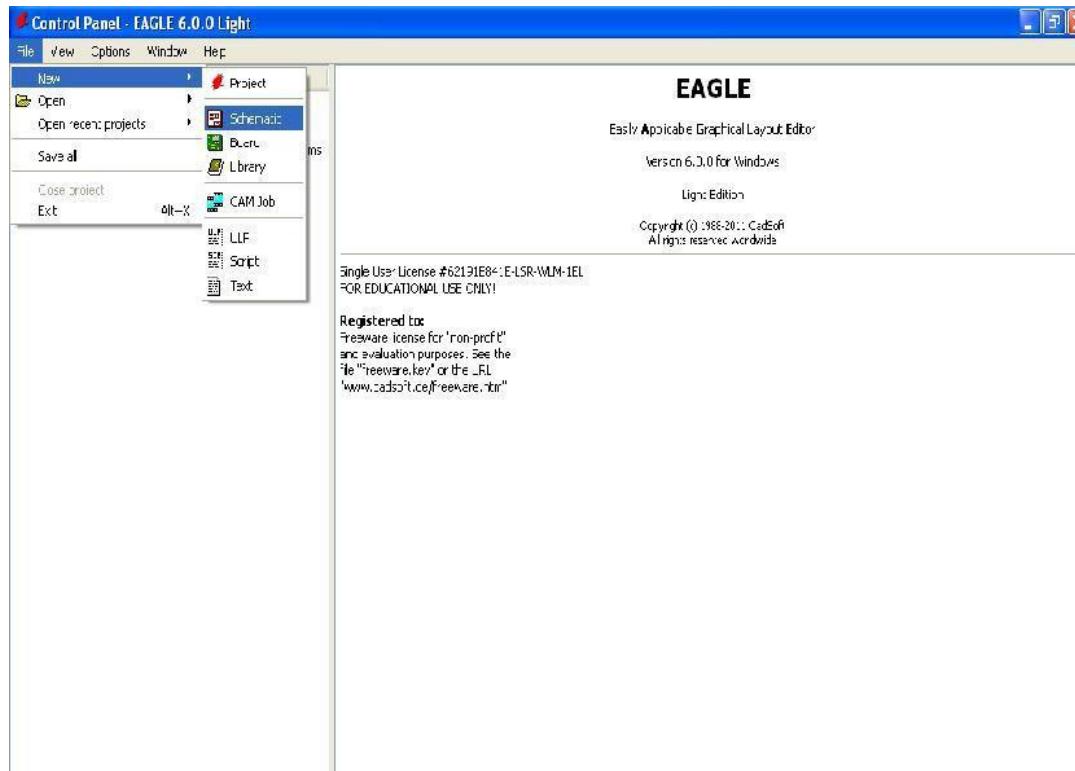


# Create New Project and S

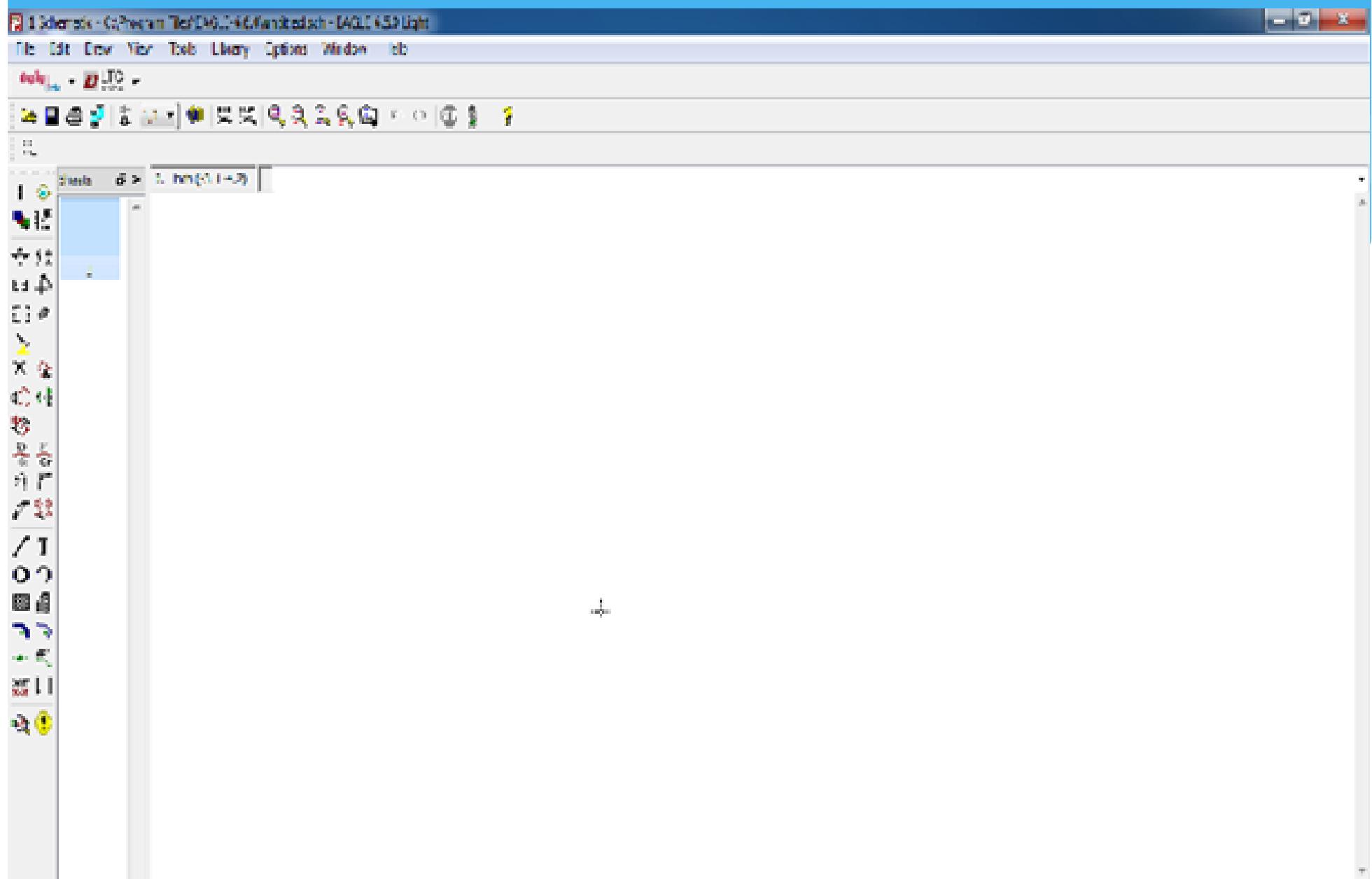
The first step is to create a new project and a new schematic in Eagle.

Choose ‘File>>New>>Project’ from the menu.

Add a new schematic to the project by selecting ‘File>>New>>Schematic’ from the menu.



# Schematic View



Name	Description
► R-TRIMM	Trimm resistor
► R-US_	RESISTOR, American symbol
▼ resistor	Resistors, Capacitors, Inductors
▼ R-EU_	RESISTOR, European symbol
R-E...	0204V
R-E...	0204/5
R-E...	0204/7
R-E...	0207/2V
R-E...	0207/5V
R-E...	0207/7
<b>R-E</b>	<b>0207/10</b>
R-E...	0207/12
R-E...	0207/15
R-E...	0309/10
R-E...	0309/12
R-E...	0309V
R-E...	0411V
R-E...	0411/12
R-E...	0411/15
R-E...	0414V
R-E...	P0613V
R-E...	P0613/15
R-E...	0617V

GS  
—>V

R-EU\_ (Version)

RESISTOR, EU

Package: 0207

RESISTOR

type 0207, grid

3D Package: 0

RESISTOR

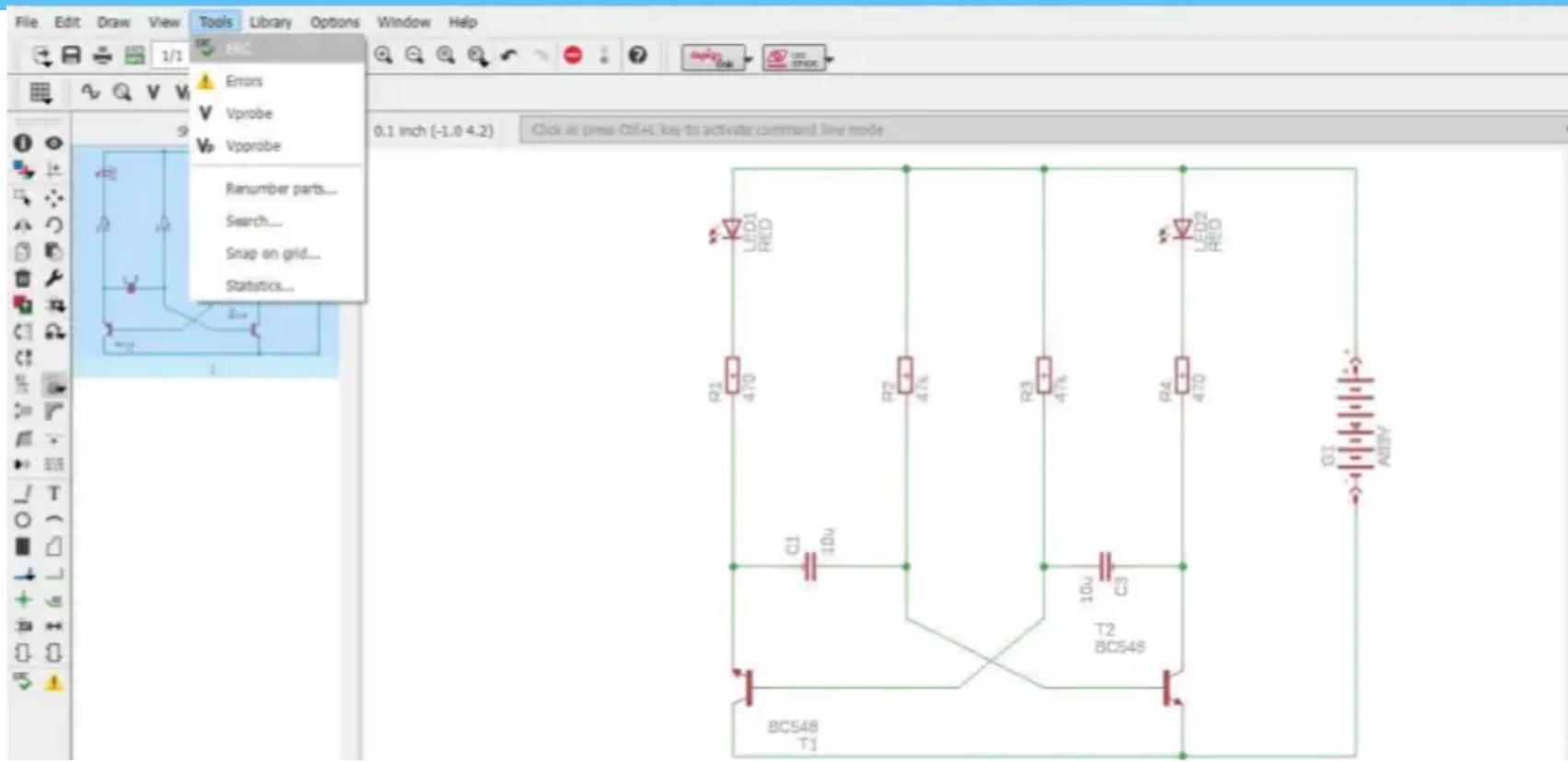
Attribute Value

- Place the resistor on your schematic by left-clicking.
- If you want to rotate the component, right click.
- After placing the component on the schematic, press Esc to

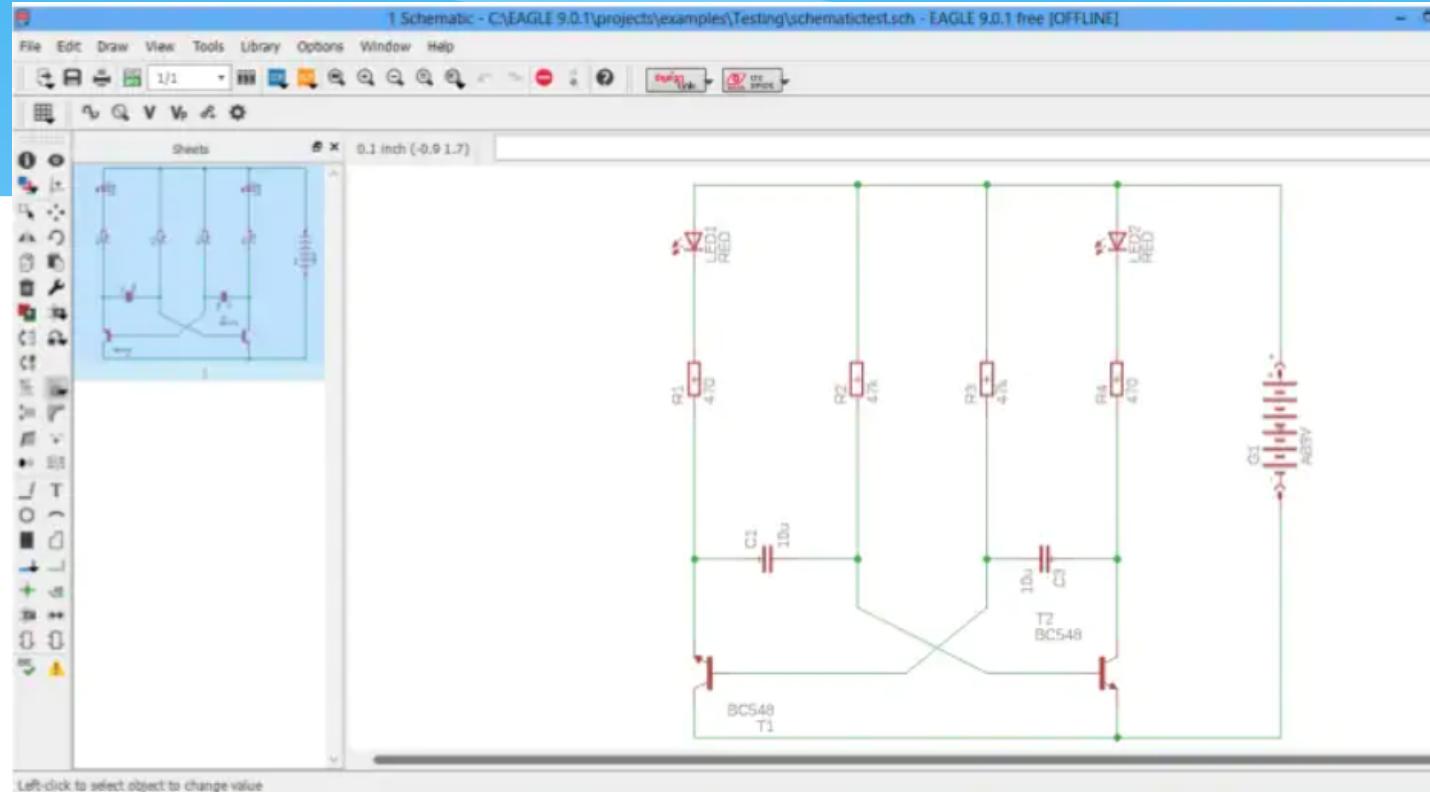


**After addition of Component Make  
Connection Between them by  
clicking toolbar icon**

# Electrical Rule Check

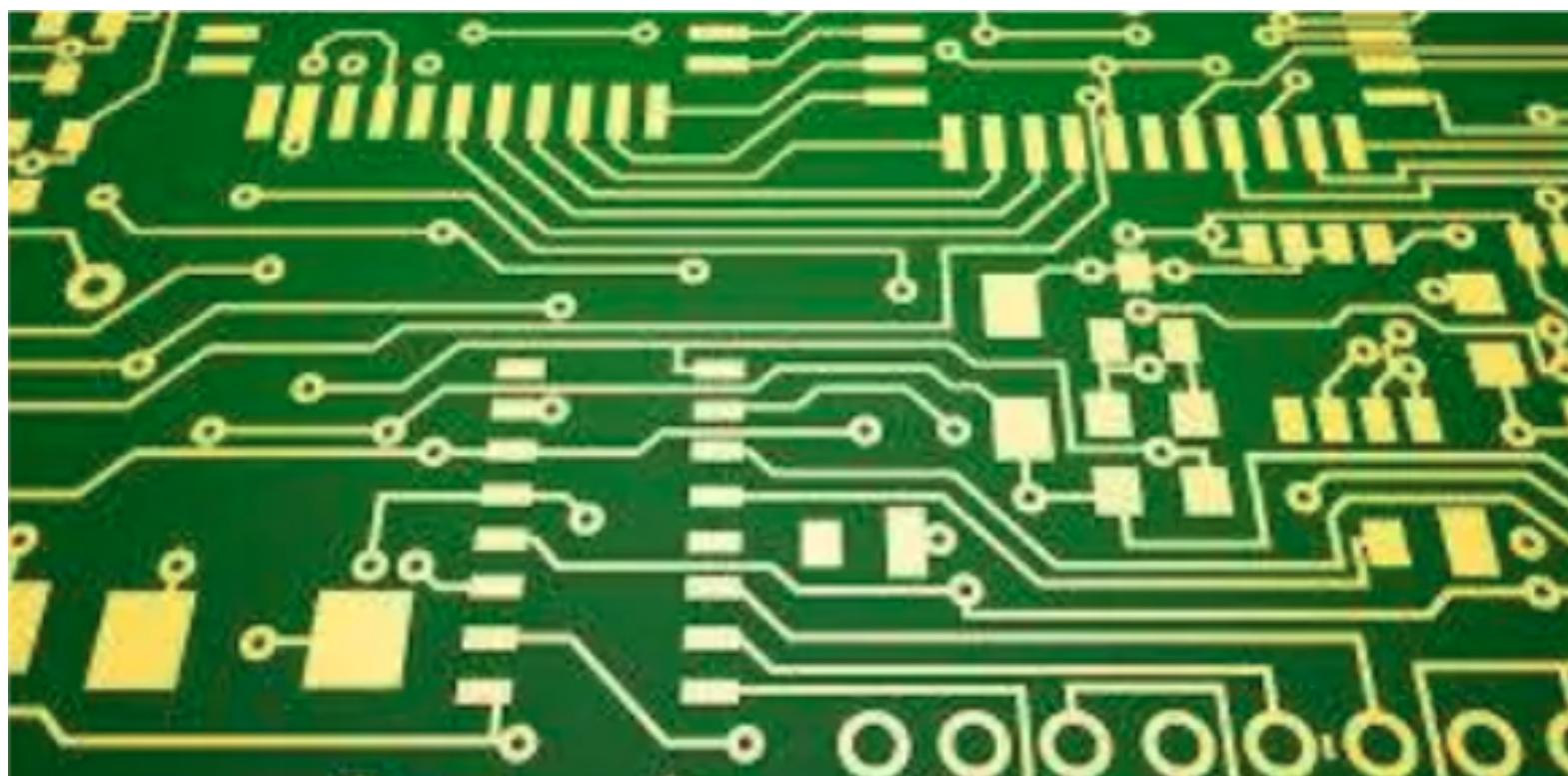


# Our Schematic is now complete.



Now, it's time to create a PCB from the schematics!

# **How to design a PCB Layout Using Eagle**



# **Printed Circuit Board (PCB)**

A PCB or (Printed Circuit Board) is a board that contains several electronic components connected using conductive electrical tracks.

A PCB provides physical support for mounting the electronic components and facilitates the connection between them.

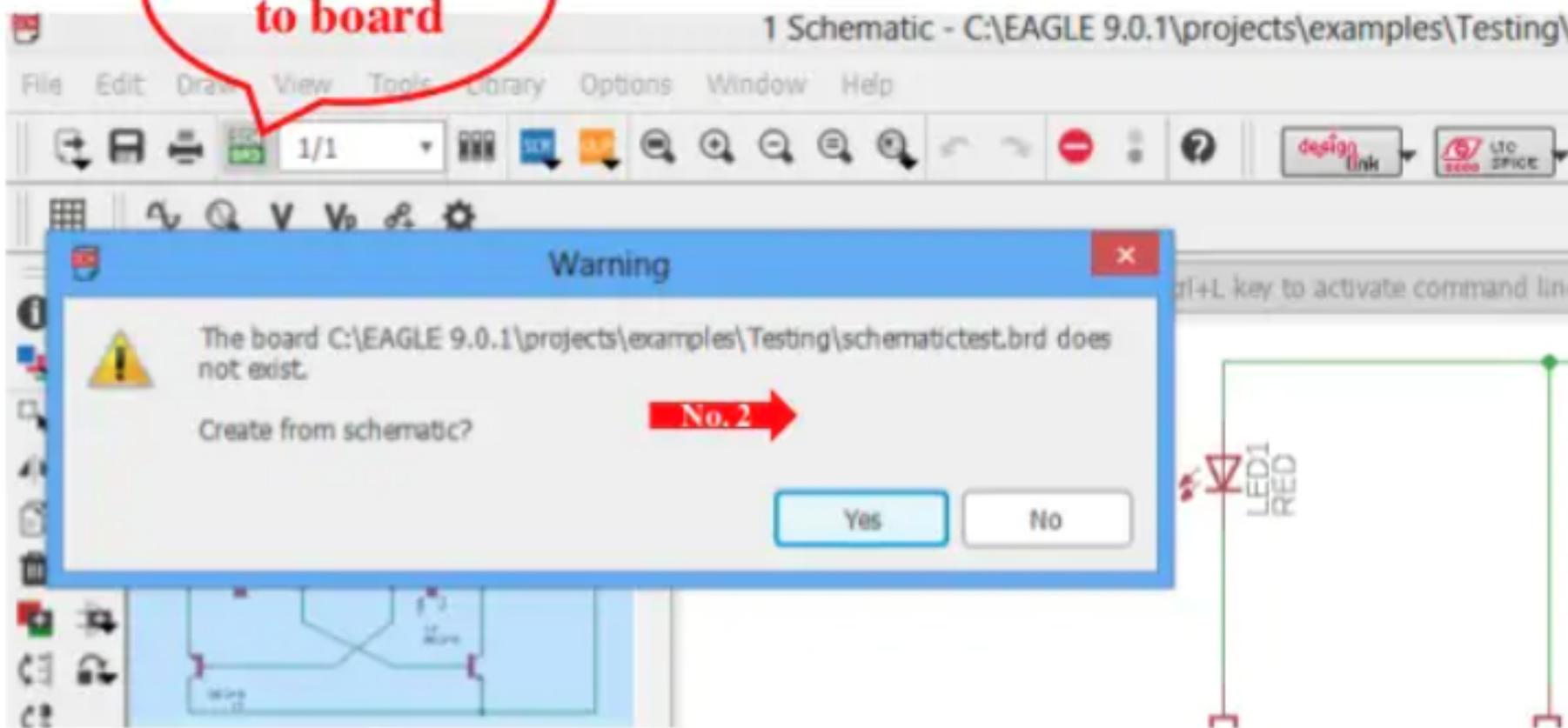
# Create PCB from Schematic

We start by opening our schematic.

Click on the ‘Board’ button (or choose ‘Switch to board’ from the top menu) to switch to the board editor for this schematic.

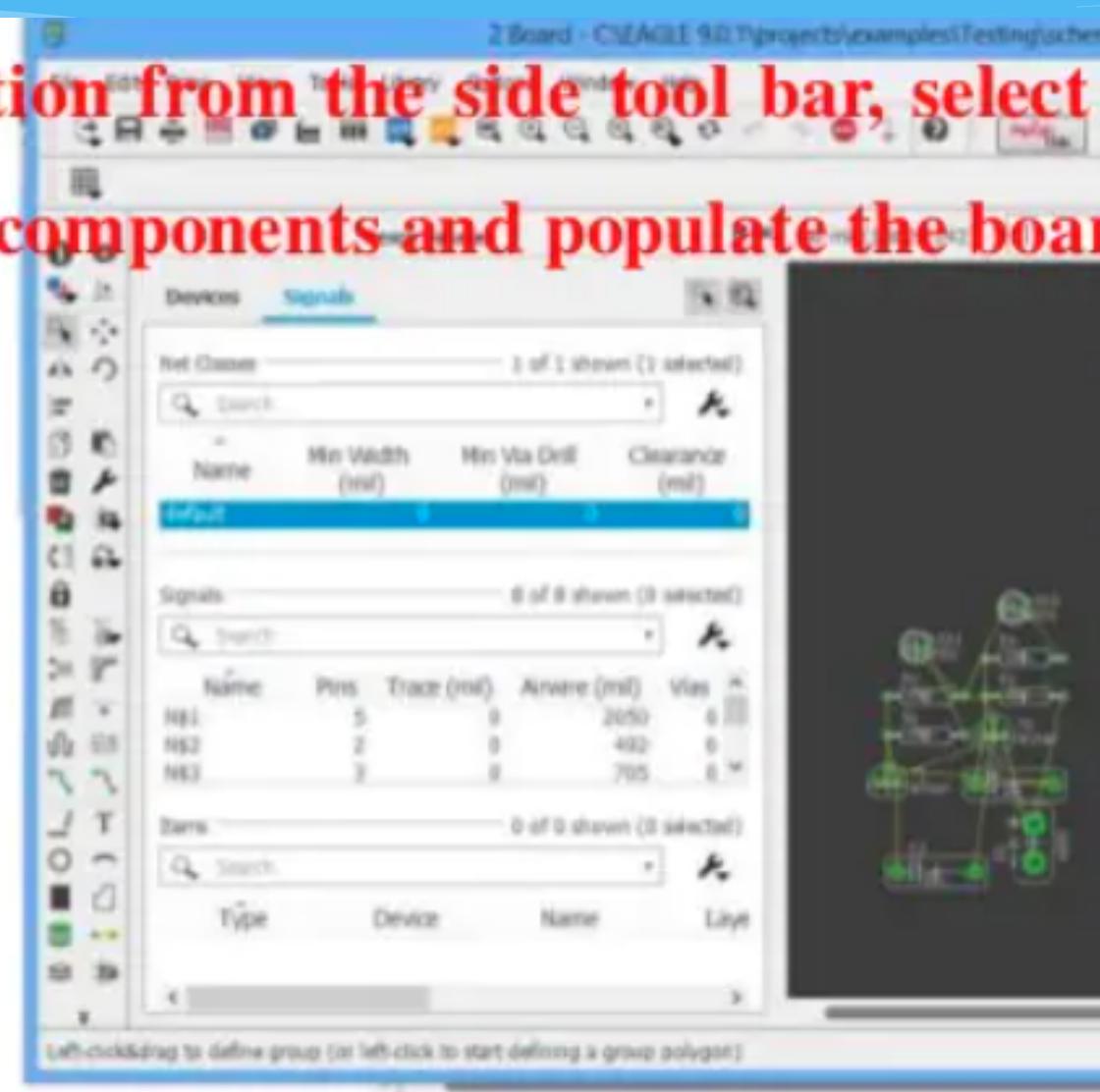
You can create the board first, or

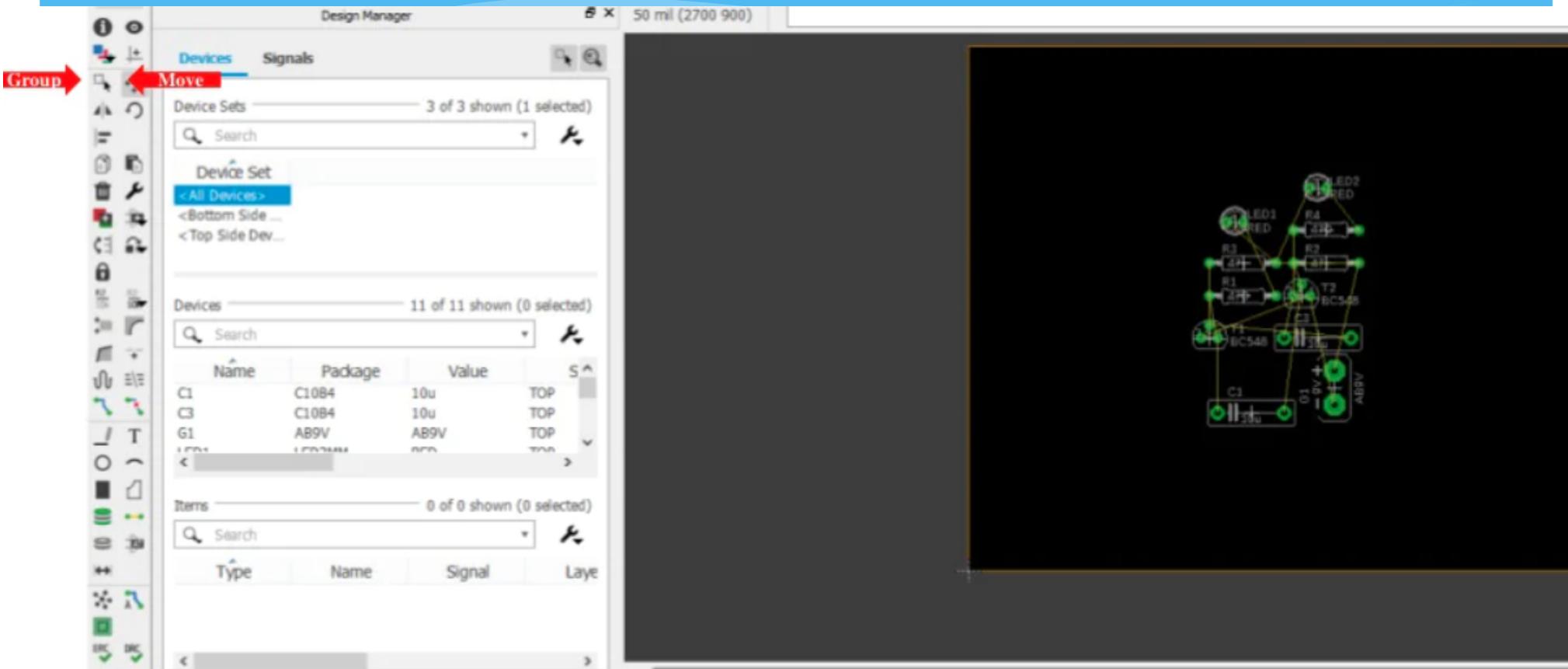
**1. Switch  
to board**



- A new window opens which is the PCB layout editor.
- The black space is the board area and all the components are at the outside bottom left of the board area.
- Now, we need to place the components in to the editor
- You can adjust the grid size of the PCB layout editor, if you want.

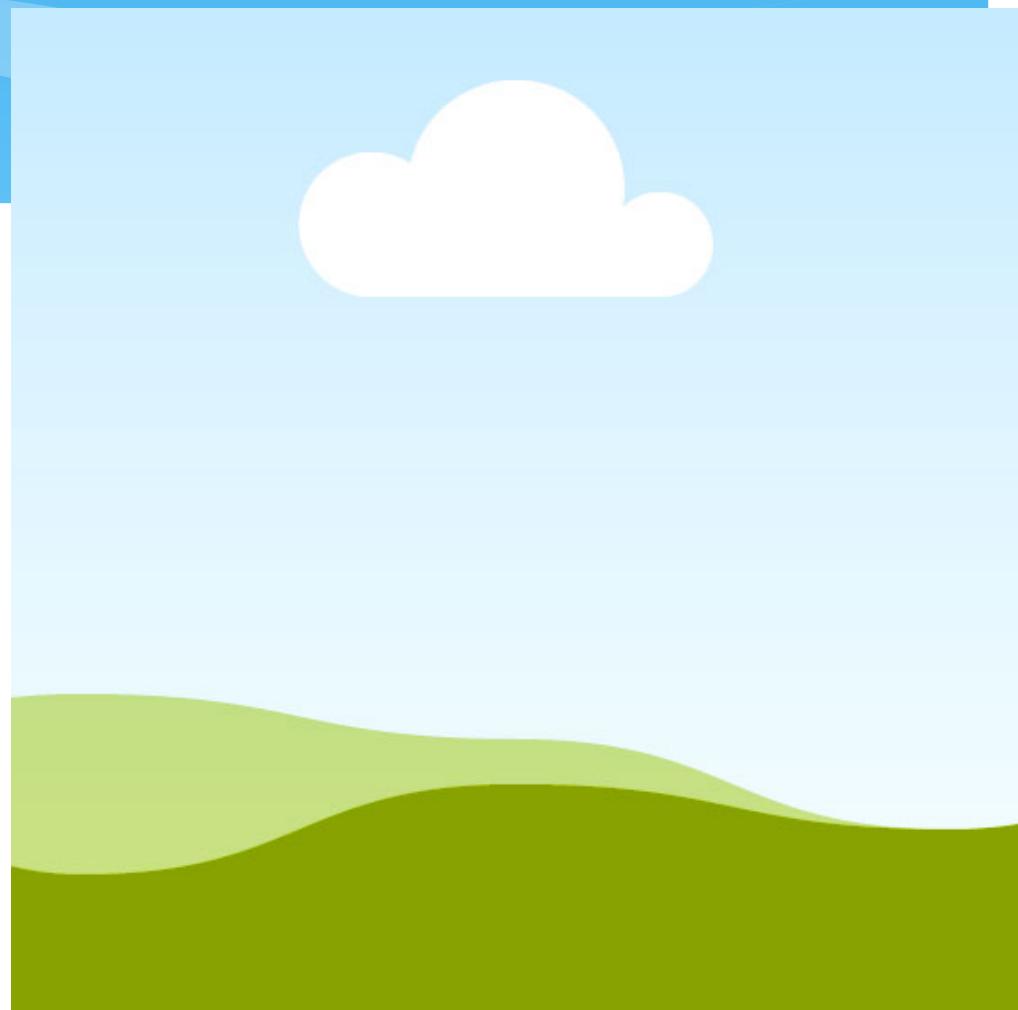
**Now using the group option from the side tool bar, select move option move all the components and populate the board**



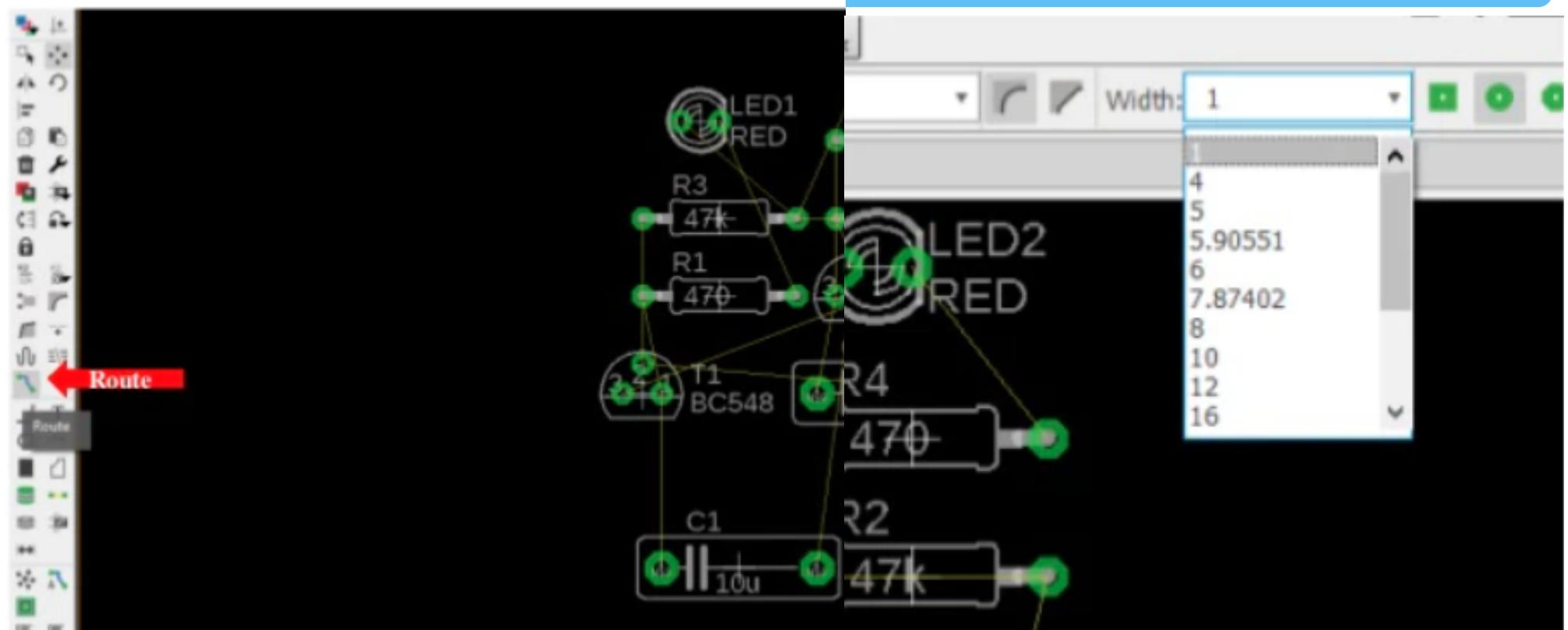


Using the move option, place the components on the board the components to be on the board.

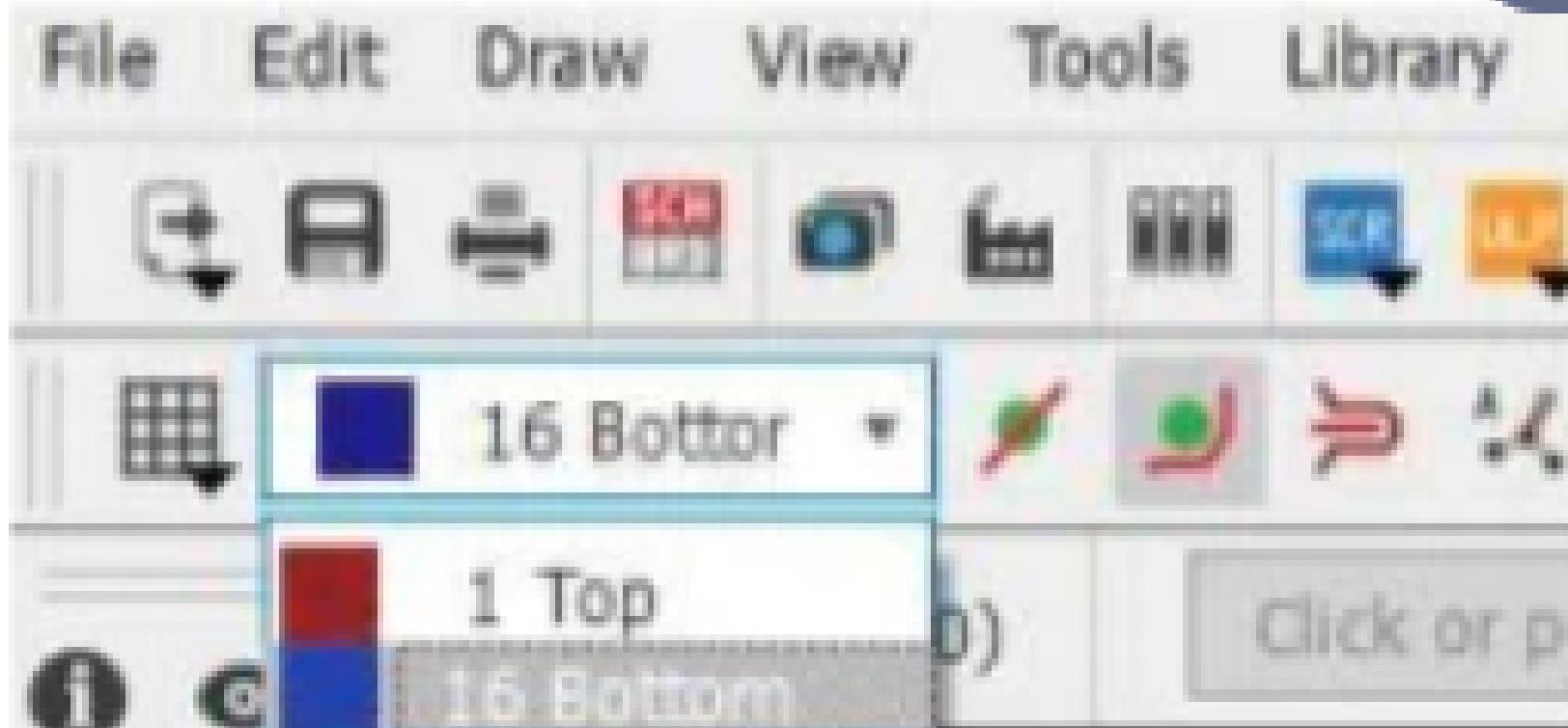
- You can see **thin yellow wires** running between the components.
- These wires are called air wires & are representation of connection between components.
- **When we route the path between components, these air wires will disappear** as an indication of successful connection.



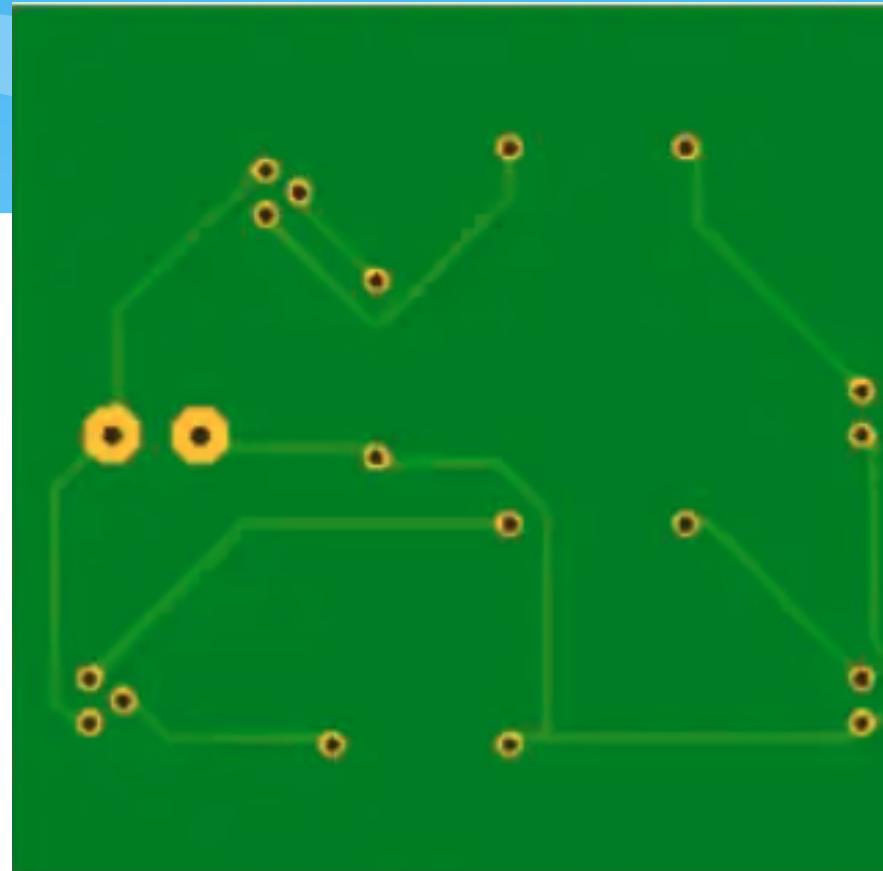
- Now, it is time to make the connections or trace the printed circuit board (PCB).
- For this, we are going to use **the Route tool** from the side tool bar.
- Also **set the width of the trace** to be routed as your requirement.
- Here, I've set the **trace width to approximately 1mm**.
- Then, select the signal layer i.e **bottom layer**, as our PCB is a single sided board.



# Select Layer



- Start routing the traces from pin to pin.
- As you progress with routing, the air wires will start disappearing.



# **PCB FABRICATION PROCESS**

# Copper Laminate Cutting

PCB Shearing Machine



Copper Laminate



# Cleaning the Laminate

PCB before  
cleaning



Clean PCB surface using  
scrubber



PCB after cleaning with  
scrubber

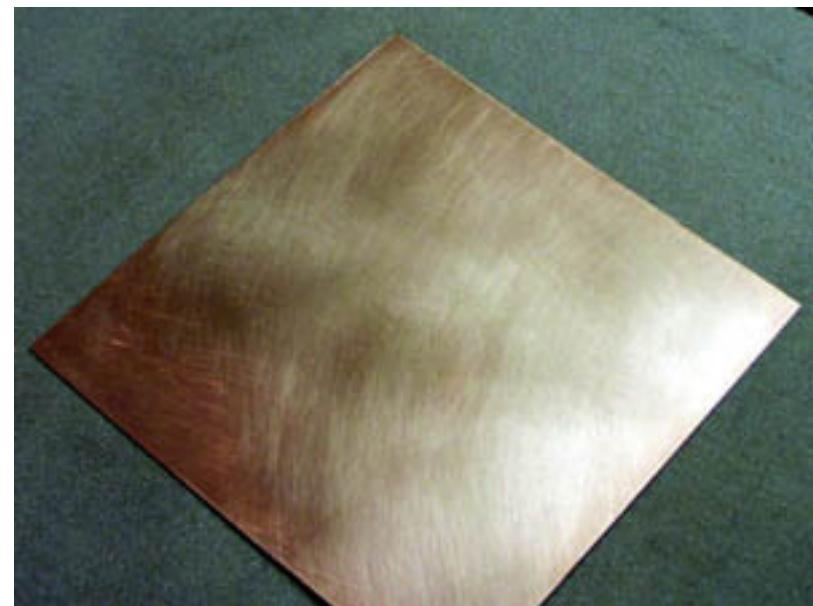


# Coating the Laminate

**Photo Resist DIP-COATING Machine**



**Coating PCB with Photoresist solution**

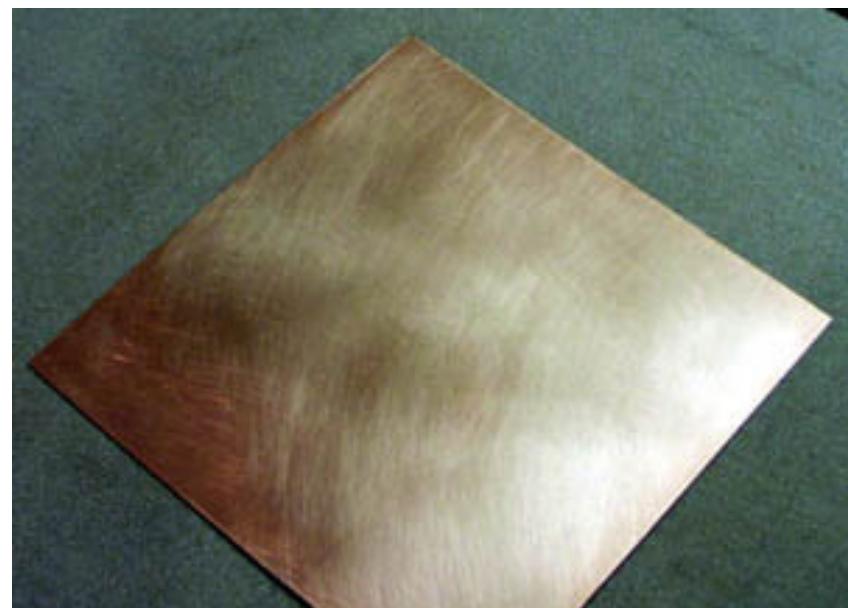


# Pre-baking

**Proto-cure baking curing  
oven**



**Coating PCB with Photoresist  
solution and Dry it in proto-cure  
Oven**



# Exposure

**Double sided UV Exposure unit**

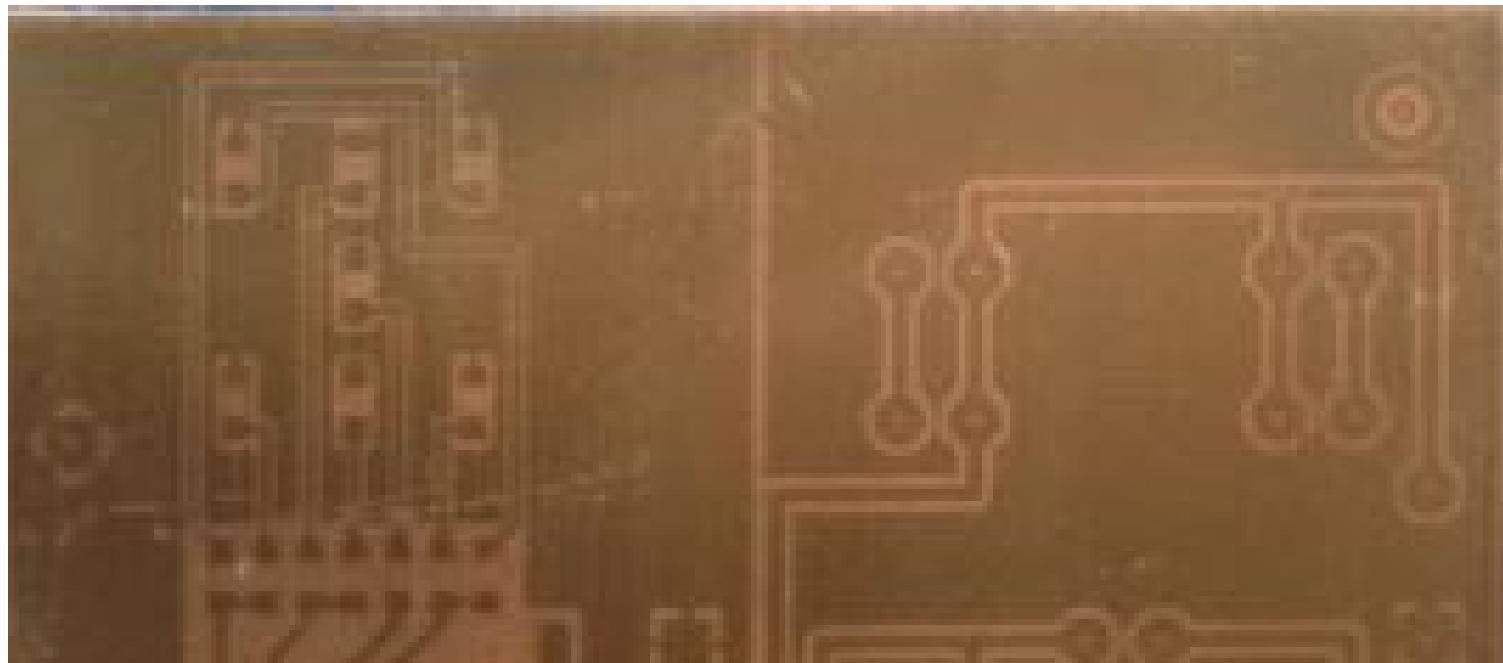


**In UV exposure keep layout on photoresist**



# Development

UV exposed PCB board after developing Stage





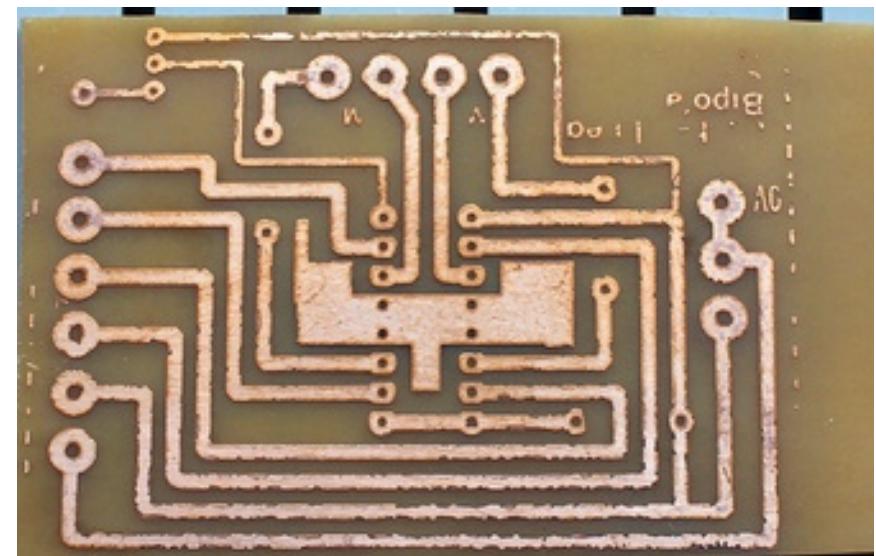
# **Wash Laminate**

# Etching

**Proto-Etch Etching  
Machine**

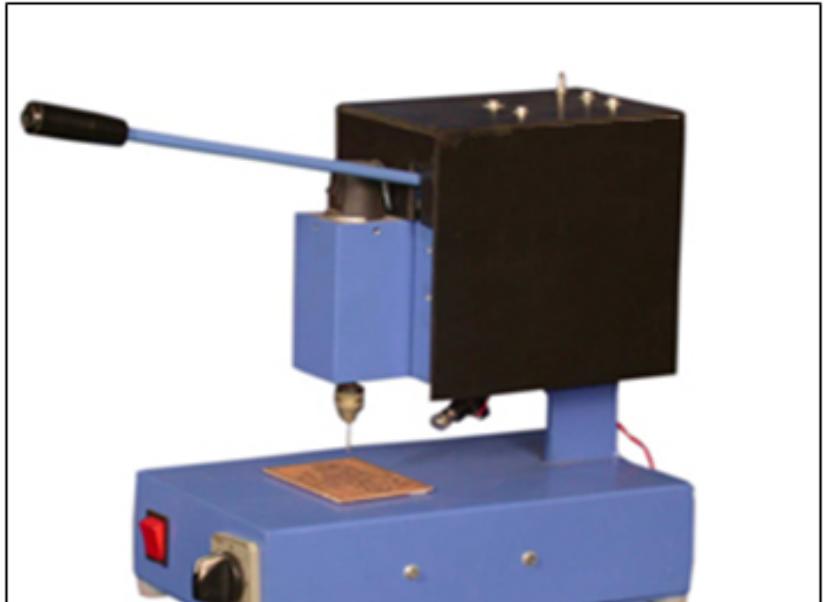


**PCB after Etching Process  
done**

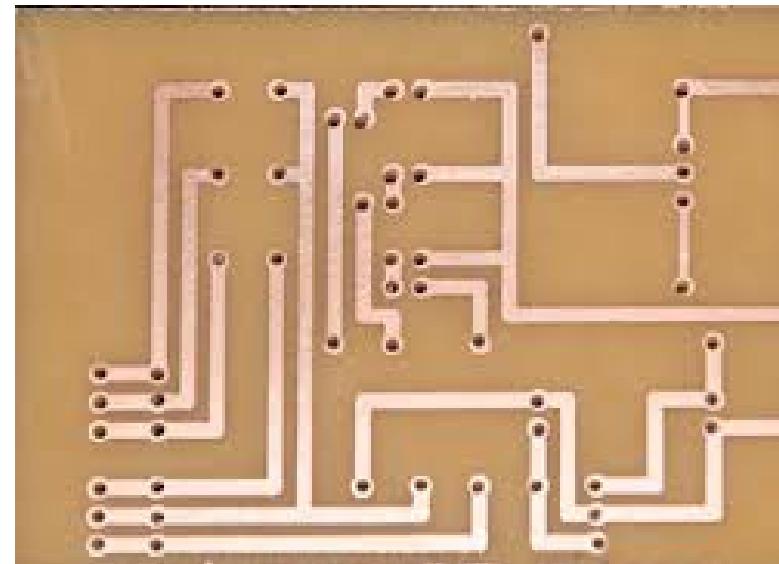


# Drilling

**PCB Drilling Machine**

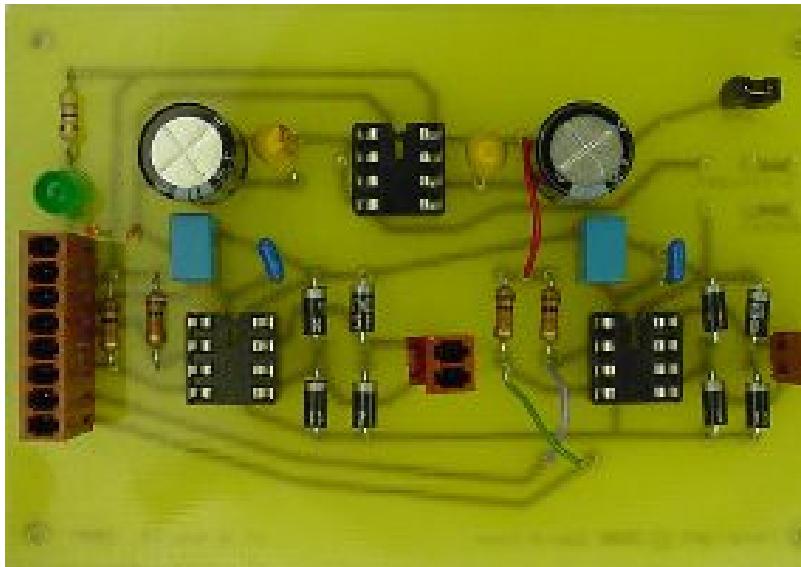


**PCB after Drilling done**



# Final PCB

Top View



Bottom View

