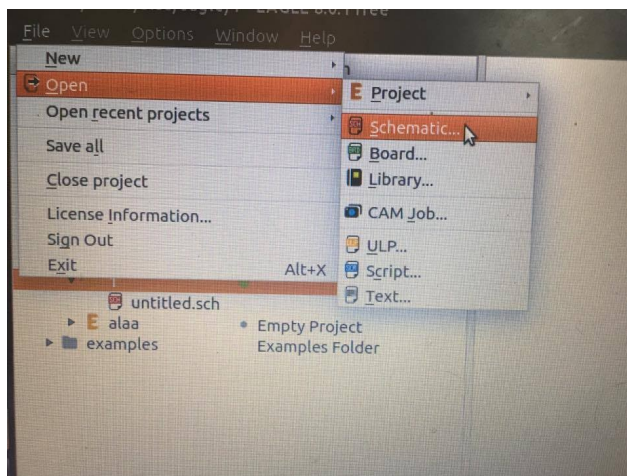EAGLE works in two stages: Schematic capture and PCB design. In the first, the circuit connectivity is designed In the second stage, the components are positioned on the printed circuit board and then traces between connected components are laid out.
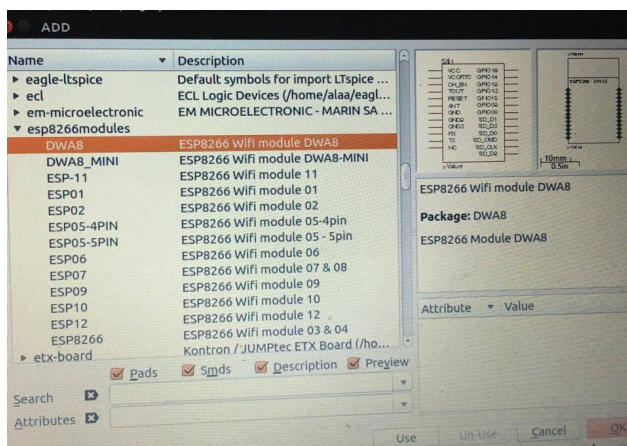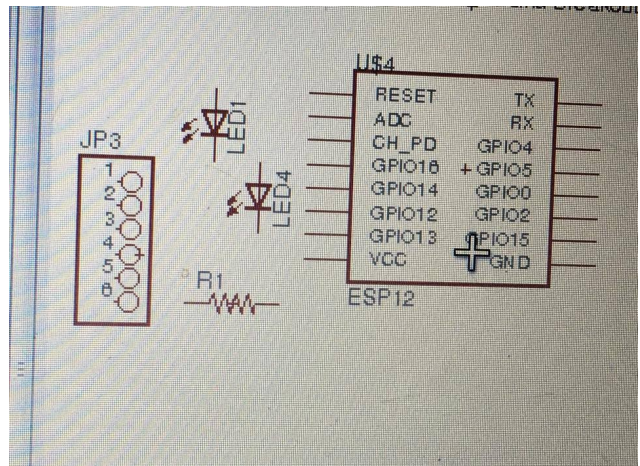
Creating the Schematic

Add libraries like adafruit ,sparkfun,atmel
file with extention(.lbr)
download it
but it in lbr directory in eagle folder
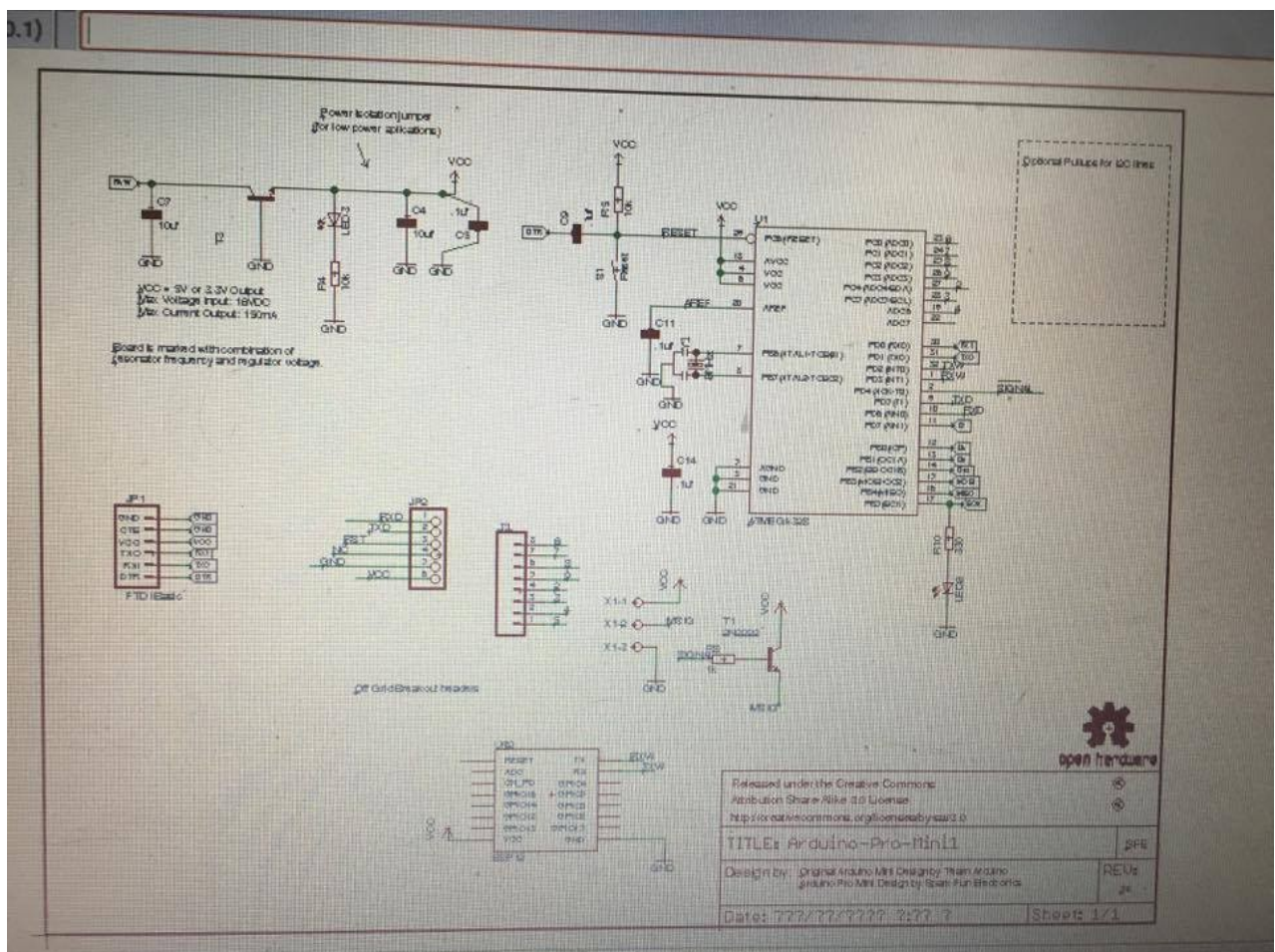from library>>use>>select file >>open
library>>update all
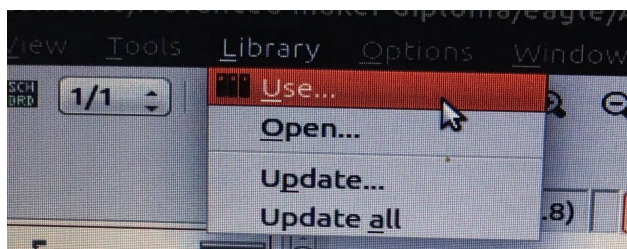
add all of the parts we will need
Click on the 'Add' icon on the left-side toolbox

for the connectivity! Select the 'NET' tool from the toolbox

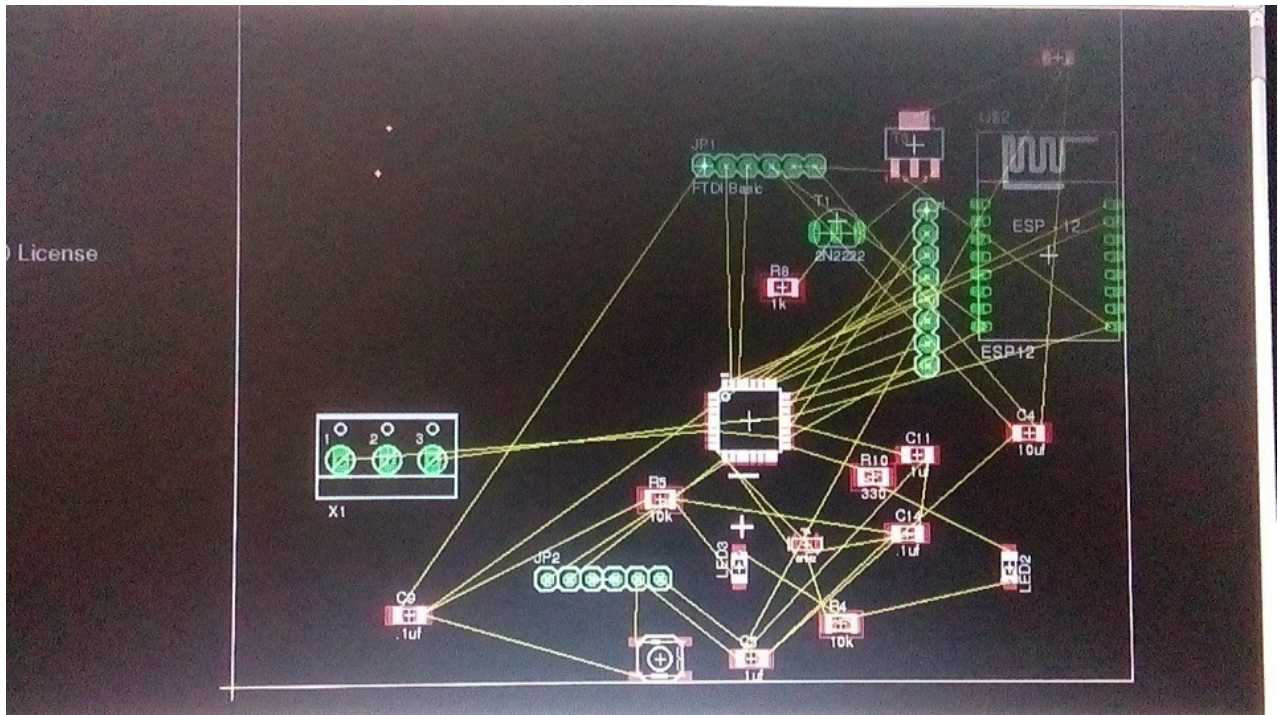Draw wires between all of the relevant terminals as per the original schematic
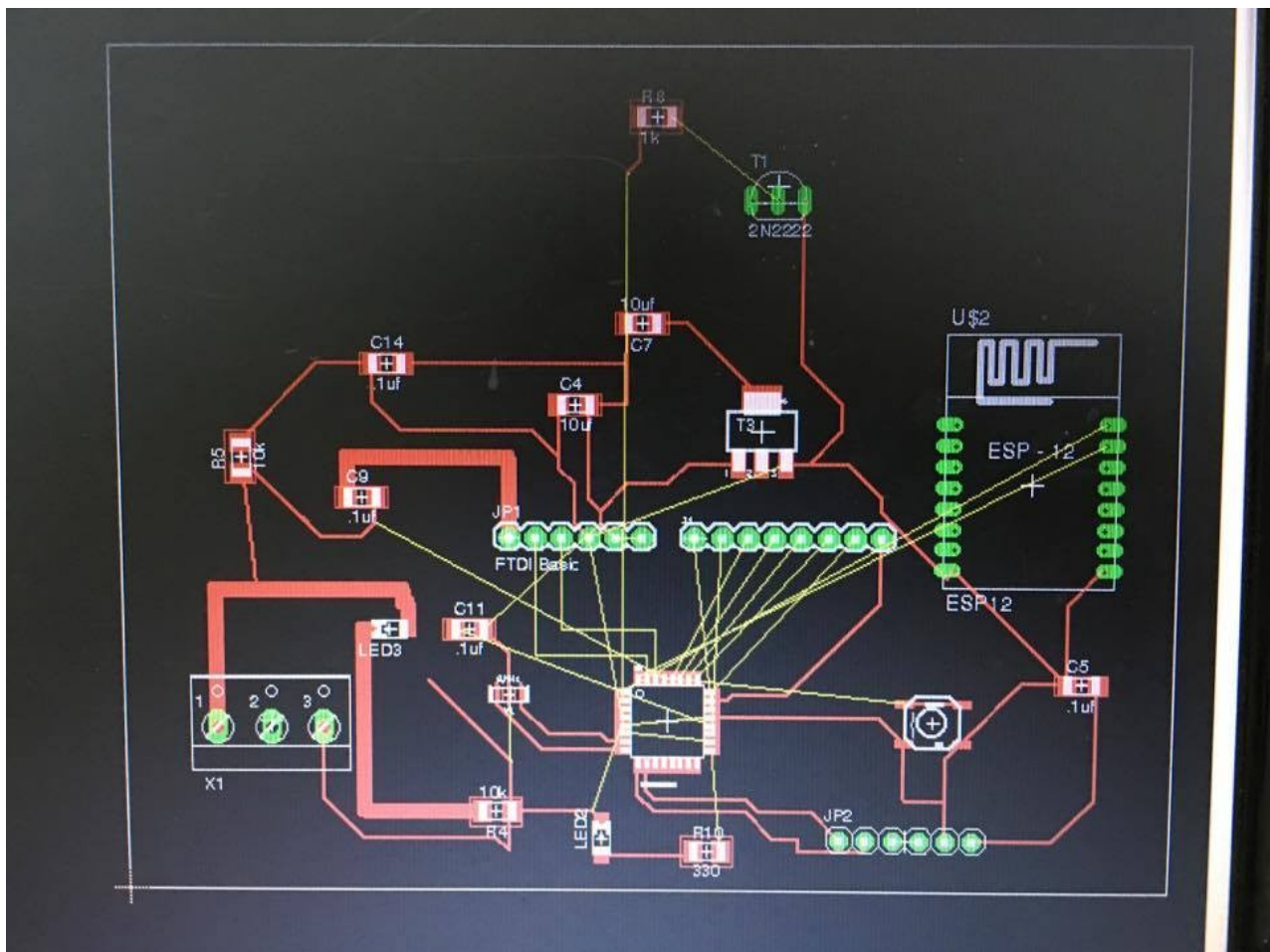

Creating the PCB

After finishing schematic
file >>>Switch to Board

We will begin by positioning the footprints on the board through a combination of rotation and movement,,Component footprints can be positioned using '**Move**' and '**Rotate**'

add the traces. Select the 'Route' tool

Make sure you are working on the side of the board you want. Layers (top or bottom normally) can be selected in the layer dropdown

To add traces to the board, first click on a connected pad,
then click to lay the trace and finally click on the ending pad indicated by the dark yellow line

Keep the traces as short as possible! Longer traces add resistance, thermal noise

Avoid 90 degree corners, 45 degrees is preferred

Make sure traces are wide enough, especially power traces.