Hardware Implementation Entitled

LED Array for Light Therapy

Project by:

Name of student	G.R. No.
1. Krutika Jagtap	17U294
2. Prachiti Gaikwad	U1410128
3. Pranali Sonawane	21820006

For the Course: Skill Development (Electronic Workshop Practice-II)

Under Guidance of: Prof. R. S. Pol

Department of Electronics & Telecommunication

Bansilal Ramnath Agarwal Charitable Trust's

Vishwakarma Institute of Information Technology

Pune-48

Academic Year 2018-19

INDEX

Abstract

The use of LEDs in applications like displays, information and advertising panels, traffic signals, biomedical applications, automotive and agricultural lighting is becoming more and more common. A simple LED driver can be termed as an electrical circuit that is used to power a light emitting diode (LED). LED array drivers with proper configuration and constant current regulation offer a solution that is suitable for most of the applications mentioned above. Hence, we have found a biomedical application of LED array driver.

Light therapy or Phototherapy with LED array using blue LED panel which is used for biomedical application. This panel helps to cure disease like jaundice in infants. A simple LED array driver circuit is used for this project.

Literature Review / Motivation

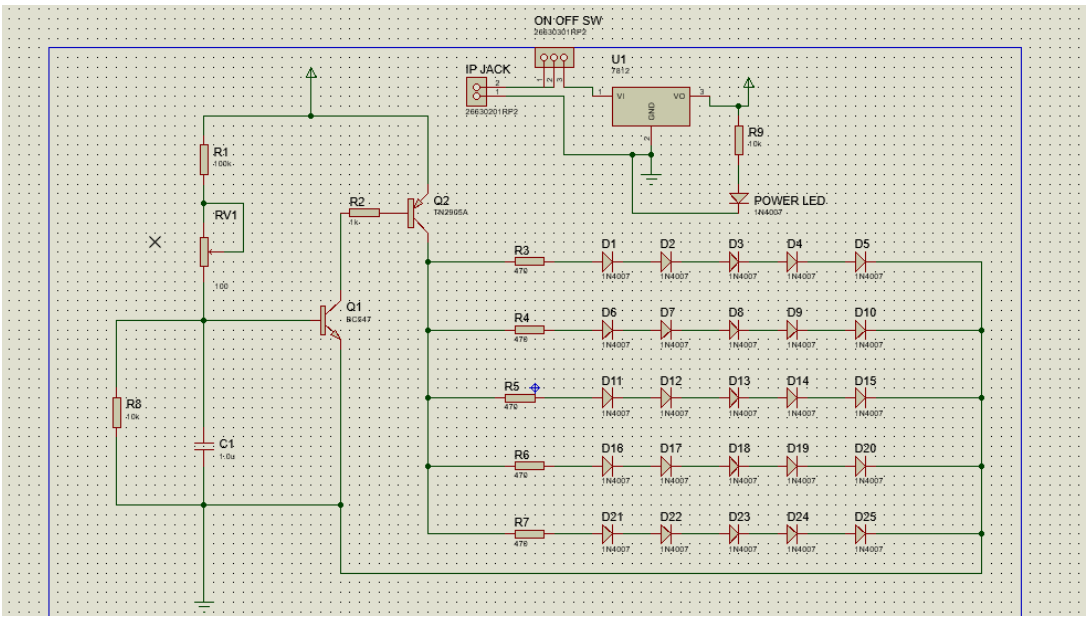
Basically, a LED array is a parallel or series combination of several LEDs. The simple LED array driver circuit is an electrical circuit that powers the LEDs in the circuit. The circuit diagram uses 25 blue LEDs that can be called as LED panel and an associated driving circuit. This circuit can be implemented using knowledge of LED drivers and LED array. Many of the educational sources provide these kinds of circuits. Electronics related journals, papers and books can be used too, which is one of the sources through which we gained the knowledge about LED array driver circuit.

The biomedical application of this project implies to jaundice treatment in infants. The LED panel having blue LEDs cure the jaundice by altering the level of bilirubin in infant's body. The different levels of bilirubin and according variation in treatment information was collected from a renowned hospital, Noble Hospital, Pune.

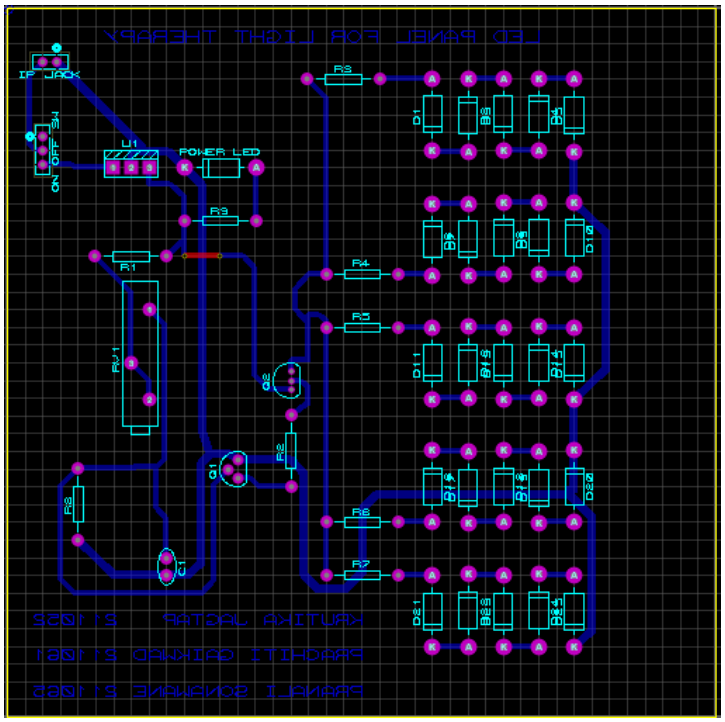
Electronic Component list and specifications

Sr. No.	Name of Component	Quantity	Value and specification
1	resistors	6	10k, 100k, 1k, 470 ohms
2	Transistor 1.BC547 2.2N2905A	2	1.Used for switching purposes And amplification 2.Small signal switching
3	IC7482	1	Voltage regulator
4	Power jack	1	To connect adaptor
5	SPDT switch	1	Switch
6	Blue LEDs	25	Blue light emission
7	Red led	1	Power LED
8	Adapter	1	12V, 1A
9	Capacitor	1	1 micro Farad
10	Variable resistor	1	100k

Circuit Diagram



PCB layout



Implementation of PCB using EDA tool

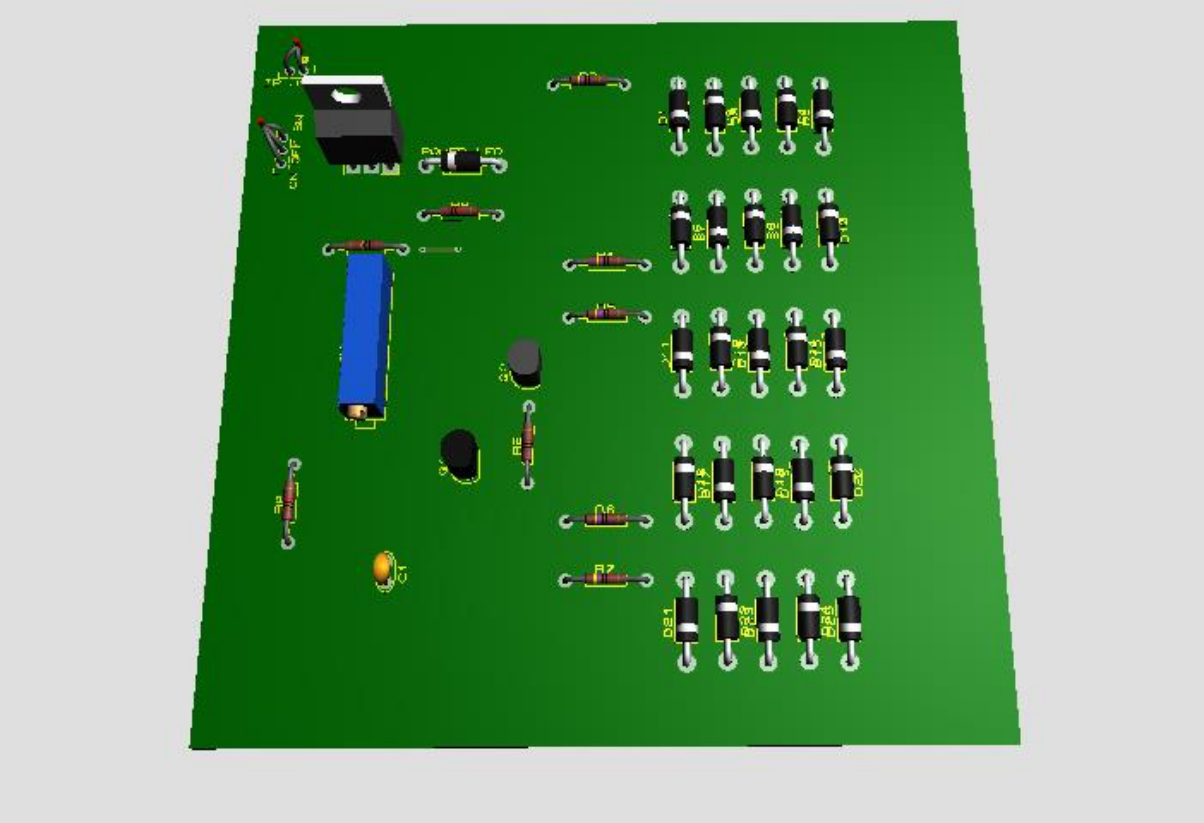
There are almost 46 types of EDA (Electronics Design Automation) tools. Out of which we have used Proteus 8. This is widely used software by students and also it has got several advantages. Some of the salient features of it are:

- Friendly and Reliable
- 3D Preview
- Placement Feature
- Manual Routing

There are some guidelines that are to be followed while designing a PCB. The standards and norms mentioned tell about width of track, distance between two tracks, thickness of tracks, pad size, etc. Some of the common rules can be specified as follows:

- Components should be arranged properly to avoid jumpers. More the spacious components placing less are the jumpers.
- Determine proper pad and hole dimensions early in the PCB designing process.
- The minimum distance between any two tracks should not be more than 0.5 mm as closer tracks can interconnection can get shorted.

3D view:



PCB Specifications:

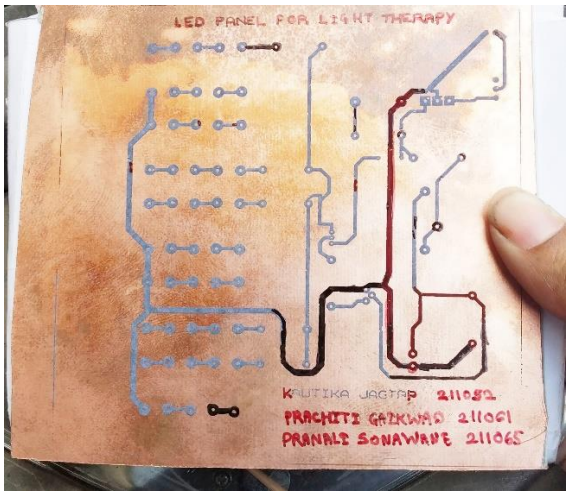
1. Size of PCB: 100X100 mm
2. Track width: 2 mm
3. Pins: 89
4. No. of components: 42
5. Holes: 91
6. No. of jumpers: 1

PCB Development Process

Once we are done with PCB designing i.e. getting an appropriate circuit diagram, make a schematic using a proper EDA tool, simulate it, make necessary changed and get a PCB layout. When the components are placed properly that includes least jumpers and also shows a distinct arrangement of components, the layout is ready for routing. The routed diagram has pads, holes and tracks.

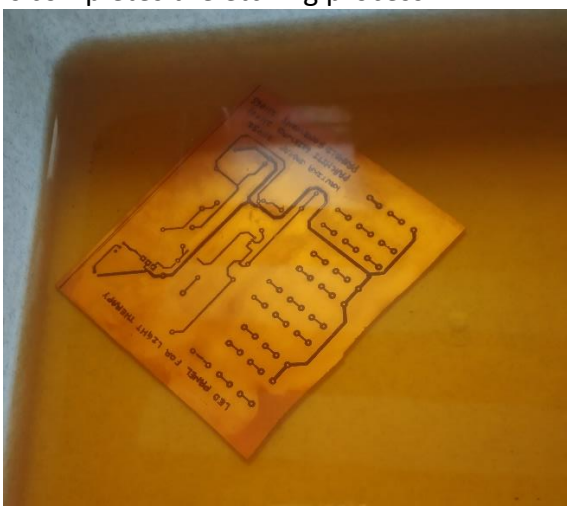
1. PCB Printing:

   The routed circuit was then taken in the form of photocopy so that it can be easily pasted on copper clad. Here, to paste the print on copper clad, we used iron. The print was placed facing towards clad and then a strongly heated iron was used to paste it. Once the whole circuit gets pasted on clad, stop ironing.



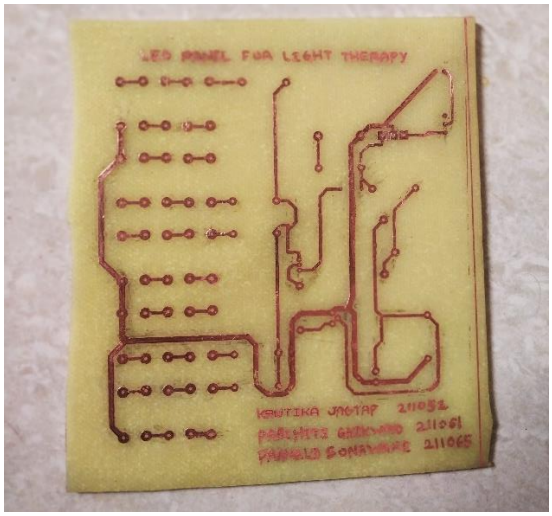
2. PCB Etching:

After pasting circuit on the copper clad, it is further processed for etching. Etching is the process that removes the excess copper. For our project we used Ferrous Chloride powder as etching solution. The powder was mixed with water in a plastic container. After constant stirring for about 45 minutes, the excess copper was removed from clad. This completes the etching process.



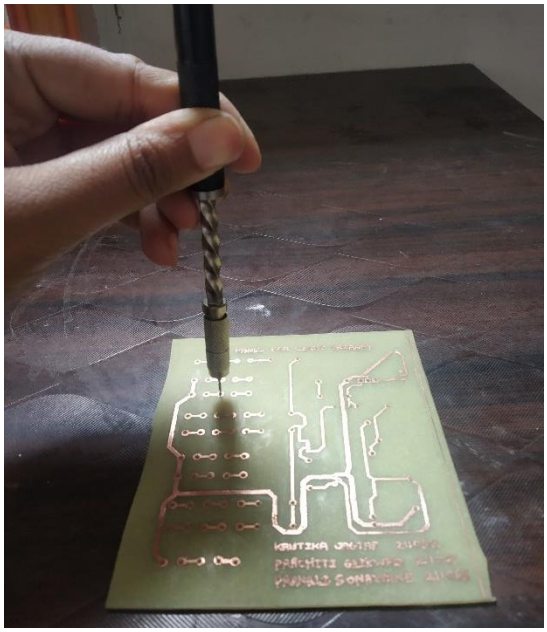
3. PCB Cleaning:

After etching the PCB, to remove the traces of print (ink) Cleaning is required. After etching, first clean the PCB in clean water and let it dry completely. For cleaning process, we used thinner to remove the black traces. And again, PCB was cleaned with cotton.



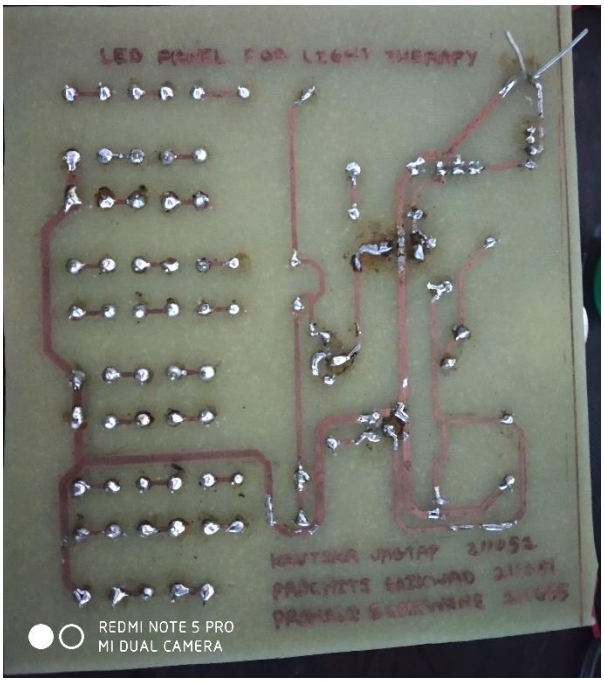
4. PCB Drilling:

The PCB is then drilled using hand-drill or an electric drill. For our project, we used and electrical drill if drill bit 1 mm. Once the PCB is drilled, it is ready for component placing.



5. PCB Soldering:

After drilling, we placed the components at their proper place by referring to the PCB layout. Then using soldering gun and wire each terminal was soldered taking care that it remains intact from any metallic contact nearby.



Troubleshooting:

Most basic PCB troubleshooting can be done with a few tools. The most versatile tool is a multi-meter. Large and complex PCBs are troubleshooted using LCR meter, oscilloscope, etc. But we have used multimeter for checking the continuity of track and voltage from one point to another.

Troubleshooting is nothing but checking the connectivity between major terminals of the PCB. It decides the reliability of PCB and tells how much precisely it can work.

Real Life Application:

This project is meant to be used for biomedical application. The main aim of the project is to treat Jaundice in new born baby. This process of treating jaundice is oftenly called as phototherapy or light therapy. The blue light emission helps to reduce the bilirubin level on the body of infant. The infant's body is exposed to blue light and thus cures jaundice.

Conclusion:

By implementing a simple project over a PCB using copper clad, we have learnt the specifications of the components. Simulation and making sure that our circuit is ready to make on PCB taught us about specifications of various components. We have recognized different techniques for making schematic, PCB layout using various EDA tools, etching of PCB, cleaning the PCB, drilling of PCB, placing components, soldering the PCB. Thus, we can conclude that we are able to implement any type of project if we have knowledge of EDA tools, components and various processes that are used to make a fine PCB.

References:

http://www.wikipedia.com
http://www.engineersgarage.com