SOLAR CHARGE CONTROLLER: A SERVICE LEARNING PROJECT

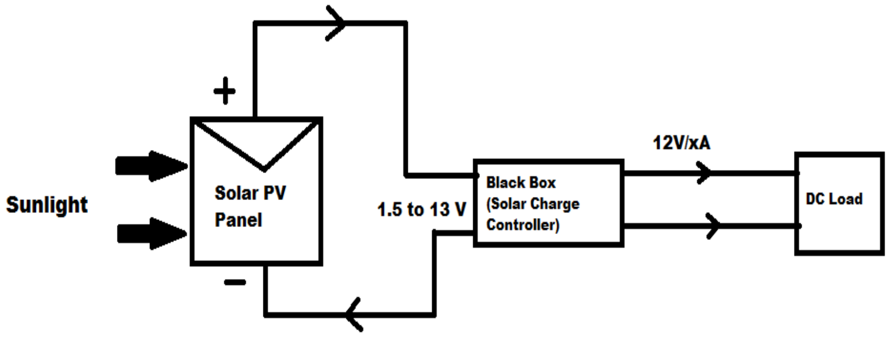
ABSTRACT:

Product design lab focuses to design a solar charge controller to convert solar energy to dc voltage which is stored in batteries. Our lab was divided into 3 parts consisting of designing the schema of the solar charge controller in KiCad software, followed by breadboard testing and finally, PCB mounting of components on glass epoxy. The main aim of this project was to participate actively in community service thereby installing the solar charge controller to produce electricity in government schools in Kolwade village. Initially we acquired hands-on experience of a real life circuit through breadboard testing and designing of circuit virtually on KiCad software. In the later stage, we mounted the entire circuit on PCB for final testing and then got the circuit fabricated on the glass epoxy PCB.

SOLAR PHOTOVOLTAIC SYSTEM:

A photovoltaic system, also PV system or solar power system, is a power system designed to supply usable solar power by the means of photovoltaics. It consists of an arrangement of several components, including solar panels to absorb and convert sunlight into electricity, a solar inverter to change the electric current from DC to AC, as well as mounting, cabling, and other electrical accessories to set up a working system. PV systems range from small, rooftop-mounted or building-integrated systems with capacities from a few to several tens of kilowatts, to large utility-scale power stations of hundreds of megawatts. Nowadays, most PV systems are grid-connected, while off-grid or stand-alone systems only account for a small portion of the market. PV systems have developed from being niche market applications into a mature technology used for mainstream electricity generation. A rooftop system recoups the invested energy for its manufacturing and installation within 0.7 to 2 years and produces about 95 percent of net clean renewable energy over a 30-year service lifetime. Due to the exponential growth of photovoltaics, prices for PV systems have rapidly declined in recent years. However, they vary by market and the size of the system. A photovoltaic system converts the sun's radiation into usable electricity. It comprises the solar array and the balance of system components. PV systems can be categorized by various aspects, such as, grid-connected vs. stand alone systems, building-integrated vs. rack-mounted systems, residential vs. utility systems, distributed vs. centralized systems, rooftop vs. ground-mounted systems, tracking vs. fixed-tilt systems, and new constructed vs. retrofitted systems. About 99 percent of all European and 90 percent of all U.S. solar power systems are connected to the electrical grid, while off-grid systems and are somewhat more common in Australia and South Korea.

Circuit Diagram of Solar Photovoltaic System



BASICS OF KiCad:

➢ What is KiCad?

KiCad is a free software suite for electronic design automation (EDA). It facilitates the design of schematics for electronic circuits and their conversion to PCB designs. KiCad was originally developed by Jean-Pierre Charras. It features an integrated environment for schematic capture and PCB layout design. Tools exist within the package to create a bill of materials, artwork, Gerber files, and 3D views of the PCB and its components.

➢ Basic Files and Folders Used in KiCad

***.pro**	Small file containing a few parameters for the current project, including the component library list.
***.sch**	Schematic files, which do not contain the components themselves.
***.lib**	Schematic component library files, containing the component descriptions: graphic shape, pins, fields.
***.kicad_pcb**	Board files containing all info but the page layout.
***.kicad_mod**	Footprint files, containing one footprint description each.
***.gbr**	Gerber files, for fabrication
***.drl**	Drill files (Excellon format), for

	fabrication.
***.pdf**	Plot files (pdf format), for documentation

Using KiCad:

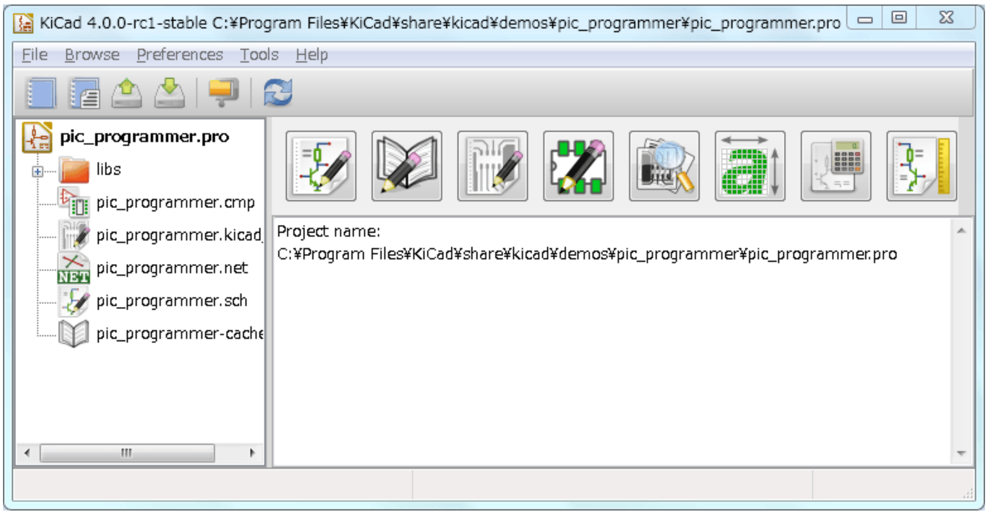
The KiCad Manager (kicad or kicad.exe file) is a tool which can easily run the other tools (editors, gerber viewer and utility tools) when creating a design.

Running the other tools from KiCad manager has some advantages:

- cross probing between schematic editor and board editor.
- cross probing between schematic editor and footprint selector (CvPcb).

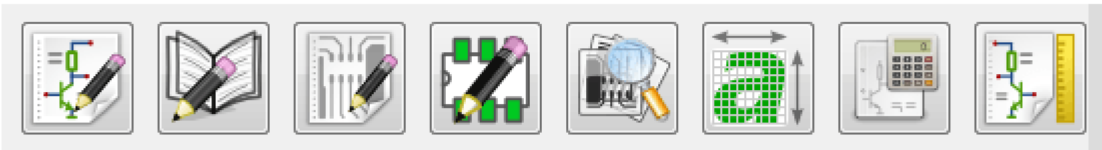
But you can only edit the current project files.

➢ Main Window:



The main KiCad window is composed of a project tree view, a launch pane containing buttons used to run the various software tools, and a message window. The menu and the toolbar can be used to create, read and save project files.

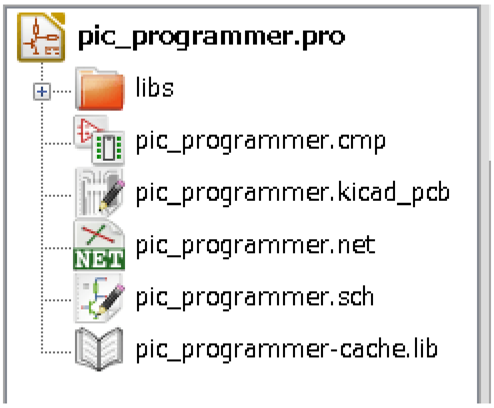
➢ Utility Launch Window:



KiCad allows you to run all stand alone software tools that come with it.

The launch pane is made of the 8 buttons below that correspond to the following commands (1 to 8, from left to right).

➢ Project Tree View :



- Double-clicking on the Eschema icon runs the schematic editor which in this case will open the file pic_programmer.sch.
- Double-clicking on the Pcbnew icon runs the layout editor, in this case opening the file pic_programmer.kicad_pcb.
- Right clicking on any of the files in the project tree allows generic file manipulation.

➢ Top Toolbar:

KiCad top toolbar allows for some basic files operation (from left to right).

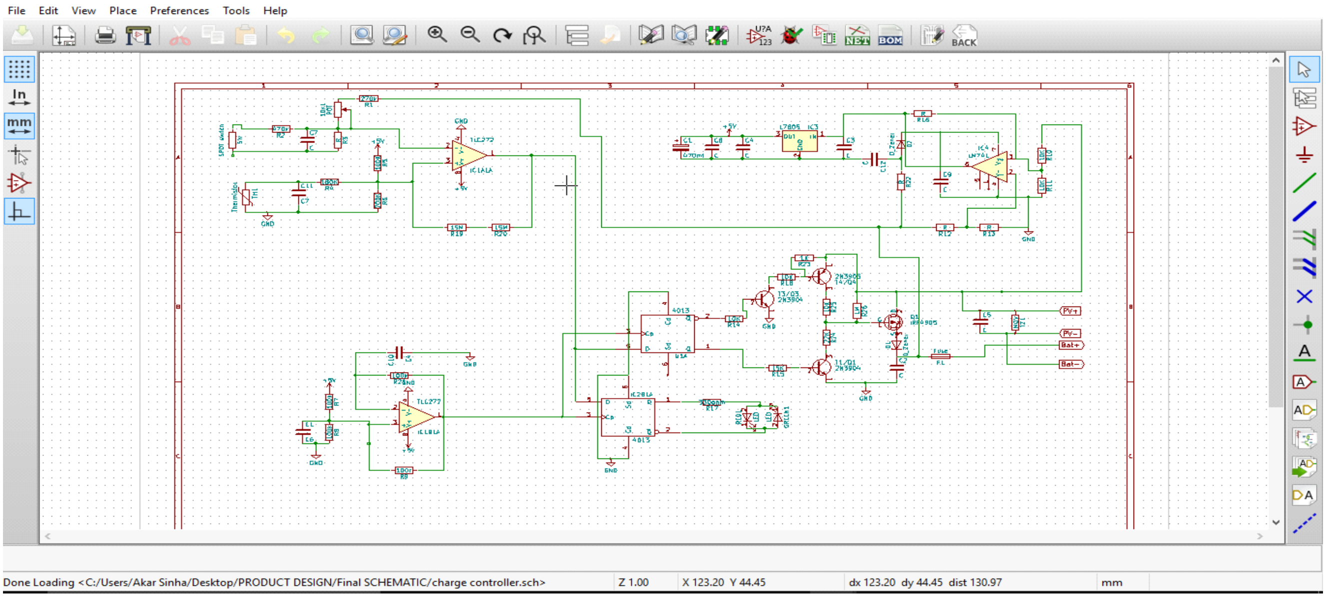
Making of Schematic Diagram Using KICAD Eschema:

1) Under Windows run kicad.exe. You are now in the main window of the KiCad project manager. From here you have access to eight stand-alone software tools: Eeschema, Schematic Library Editor, Pcbnew, PCB Footprint Editor, GerbView, Bitmap2Component, PCB Calculator and Pl Editor.

2) Create a new project: File → New Project → New Project. Name the project file. The project file will automatically take the extension “.pro”. KiCad prompts to create a dedicated directory, click ”Yes” to confirm. All your project files will be saved here. Let us begin by creating a schematic.

3) Start the schematic editor Eeschema. It is the first button from the left. Click on the Page Settings icon on the top toolbar. Set the Page Size as A4 and enter the Title. You will see that more information can be entered here if necessary. Click OK. This information will populate the schematic sheet at the bottom right corner.

4) Use the mouse wheel to zoom in. Save the whole schematic project: File → Save Schematic Project.



After creating the netlist with proper annotations and dimensions of the components of the circuit through cvpcb option, we chose a layout of 120x120 mm for our PCB. We then placed the components onto the layout properly and the wiring was done. Wiring was being taken care of by seeing that the wires do not get shorted with each other thus we used red and green wires as needed. After doing so we then created the gerber file for the layout by selecting the gerber file option and sent this file to the manufacturer for the PCB fabrication.

LIST OF COMPONENTS:

(A, b)

Material List for Solar Charge Controller

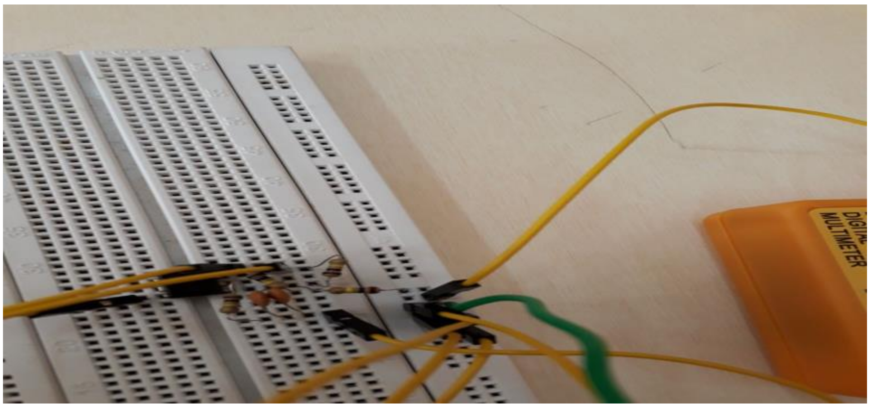
Sr. No	Name Of Material	Value Of Material	Discription Of Material
1	Tz1 V7270 or V727		Metal oxide varistor
2	IC1 TLCC2272Cp		Operational Amplifier
3	IC2 CD4013BECMOS		D Flip Flop
4	IC3 78L05		5V power supply
5	IC4 LM741N		Opamp IC
6	T1,T3 2N3904		NPN Transistors
7	T4 2N3905		PNP Transistor
8	T2 IRF 4905		P type MOSFET with heat sink.
9	Th1	2.0K,25 Degree celcius.	NTC Thermistor
10	LED		Red/Green Colour Dual.
11	POT	10K Ohm	Potentiometer
11	D1	20LIST	Diode1
12	D2	/N5818	Diode2
	SW		3 Input SPDT Switch
13	R1	270K Ohm	Resistor
14	R2	470K Ohm	Resistor
15	R3	75K Ohm	Resistor
16	R4	180K Ohm	Resistor
17	R5	100K Ohm	Resistor
18	R6	200K Ohm	Resistor
19	R7	100K Ohm	Resistor
20	R8	100K Ohm	Resistor
21	R9	100K Ohm	Resistor
22	R10	10K Ohm	Resistor
23	R11	10K Ohm	Resistor
24	R12	100K Ohm	Resistor
25	R13	100K Ohm	Resistor
26	R14	10K Ohm	Resistor
27	R15	10K Ohm	Resistor
28	R16	1M Ohm	Resistor
29	R17	330 Ohm	Resistor
30	R18	10K Ohm	Resistor
31	R19	15M Ohm	Resistor
32	R20	15M Ohm	Resistor
33	R21	100K Ohm	Resistor
34	R22	5.1 Ohm	Resistor
35	R23	1K Ohm	Resistor
36	R24	22K Ohm	Resistor
37	R25	10K Ohm	Resistor
38	R26	1M Ohm	Resistor
39	C1	470mf	Electrolytic capacitor
40	C2 to C12	100nf	Capacitor
41	8 Pin Ic Socket		
42	14 Pin IC Socket		

12-01-18

BREADBOARD MOUNTING & TESTING:

An electronics breadboard mad is actually a **solderless breadboard**. These are great units for making temporary circuits and prototyping, and they require absolutely no soldering. They can house both the simplest circuit as well as very complex circuits. Another common use of breadboards is testing out new parts, such as Integrated circuits (ICs). When you are trying to figure out how a part works and constantly rewiring things, you don’t want to have to solder your connections each time. The circuit was connected according to the schematic provided to us.We then tested the first part of circuit using components such as capacitor, ICL7805 and Zener diode and IC741 operational amplifier and other resistors.

We completed the **testing of clock oscillator** circuit on breadboard. We have given the output of 5V DC supply input to the clock oscillator input. The output frequency of that clock oscillator must be 150Hz.



Next, we did the breadboard testing of IC 741 used as **comparator** of both current input as one from Battery and another from solar PV panel. As seen in diagram both connections are given in comparator IC and then it functions or triggers voltage regulator to power other circuits such as clock oscillator and floating voltage. We also faced difficulty in arranging all the components in given space as well as understanding functioning of Zener diode and arranging it. Also for testing purpose we have given a temporary terminal on 5.1 ohms resistor for battery input which was understood. Our testing was done perfectly and then we proceeded to soldering.

Soonafter we tested the D-flip flop. For a Positive-Edge-Triggered D Flip-flop, its output Q follows input D only at every L to H transition of CLOCK, otherwise, Q keeps unchanged. The output obtained was in accordance with the above.

In a similar way, we tested all the circuits (voltage regulator) of the schema on the breadboard to check their proper functioning which will then be soldered and tested on the PCB to get more accurate results.

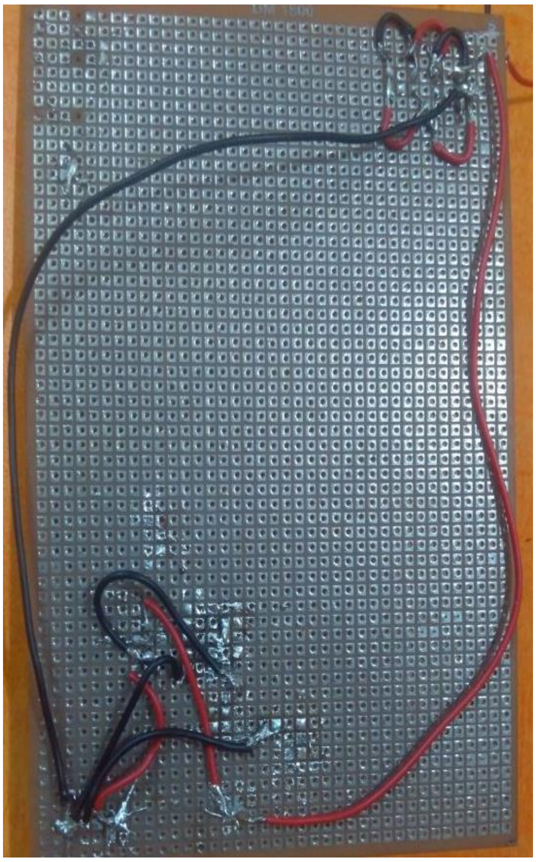
PCB SOLDERING & TESTING:

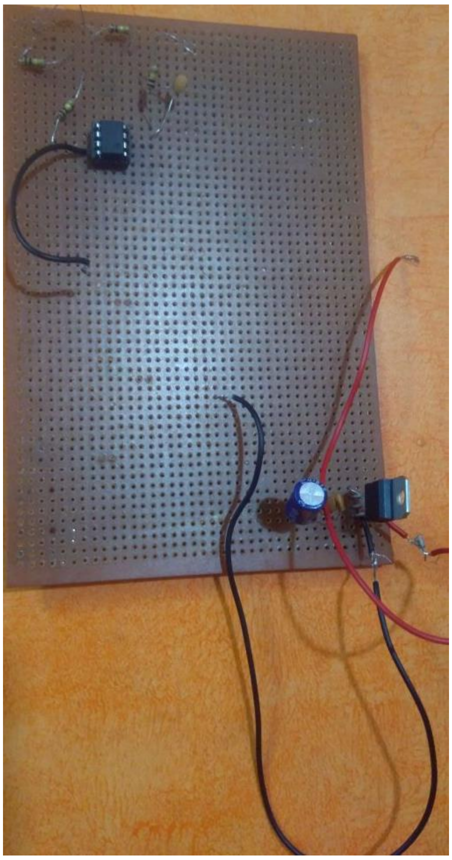
Soldering, is a process in which two or more items (usually metal) are joined together by melting and putting a filler metal (solder) into the joint, the filler metal having a lower melting point than the adjoining metal. Soldering differs from welding in a way that soldering does not involve melting the work pieces. In brazing, the filler metal melts at a higher temperature, but the work piece metal does not melt.

Soldering filler materials are available in many different alloys for differing applications. In electronics assembly, the eutectic alloy of 63% tin and 37% lead (or 60/40, which is almost identical in melting point) has been the alloy of choice. Other alloys are used for plumbing, mechanical assembly, and other applications. Some examples of soft-solder are tin-lead for general purposes, tin-zinc for joining aluminium, lead-silver for strength at higher than room temperature, cadmium-silver for strength at high temperatures, zinc-aluminium for aluminium and corrosion resistance, and tin-silver and tin-bismuth for electronics. Soldering of voltage regulator components was main aim of today’s session.

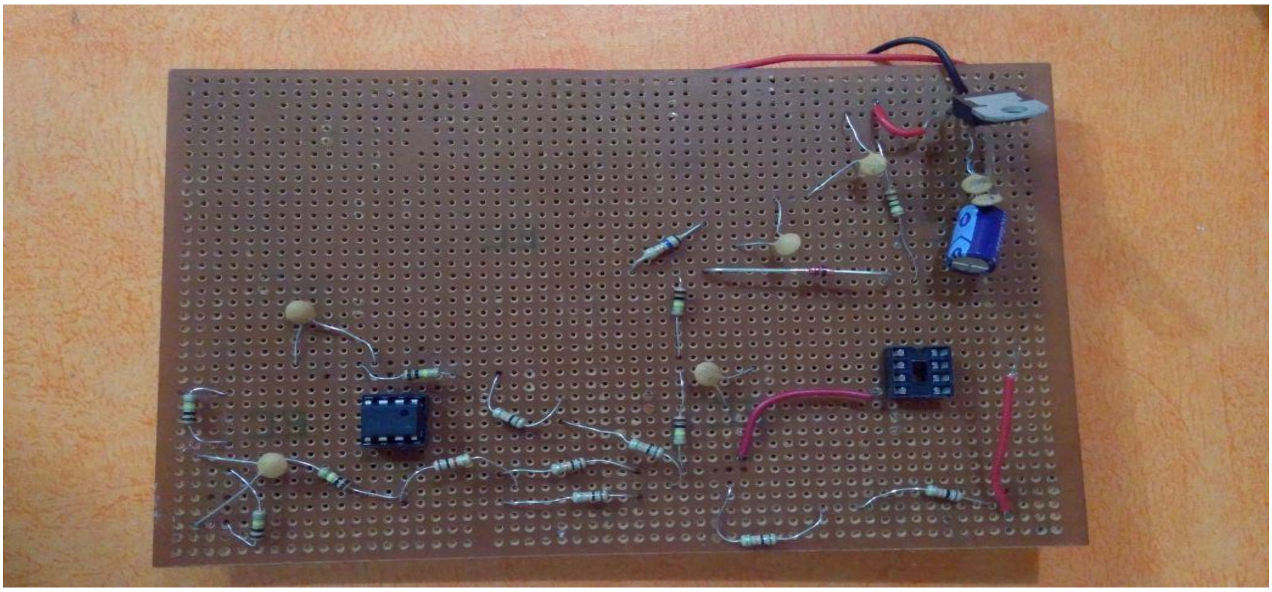
After the breadboard testing of different circuits, we started soldering them on PCB very carefully. Another important point which we noted was arrangement of connecting multi-stand wires on PCB which connects to each component. We earlier used to solder it on back side of PCB but this time we came to know a different method of arranging wires for better efficiency.

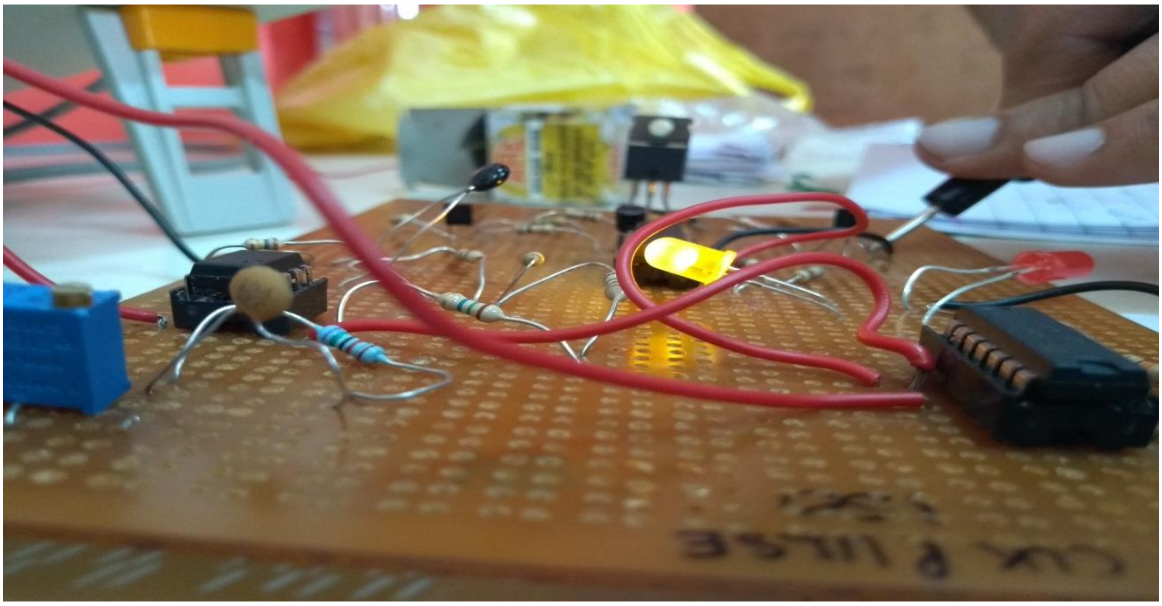
Connection of voltage regulator and clock oscillator





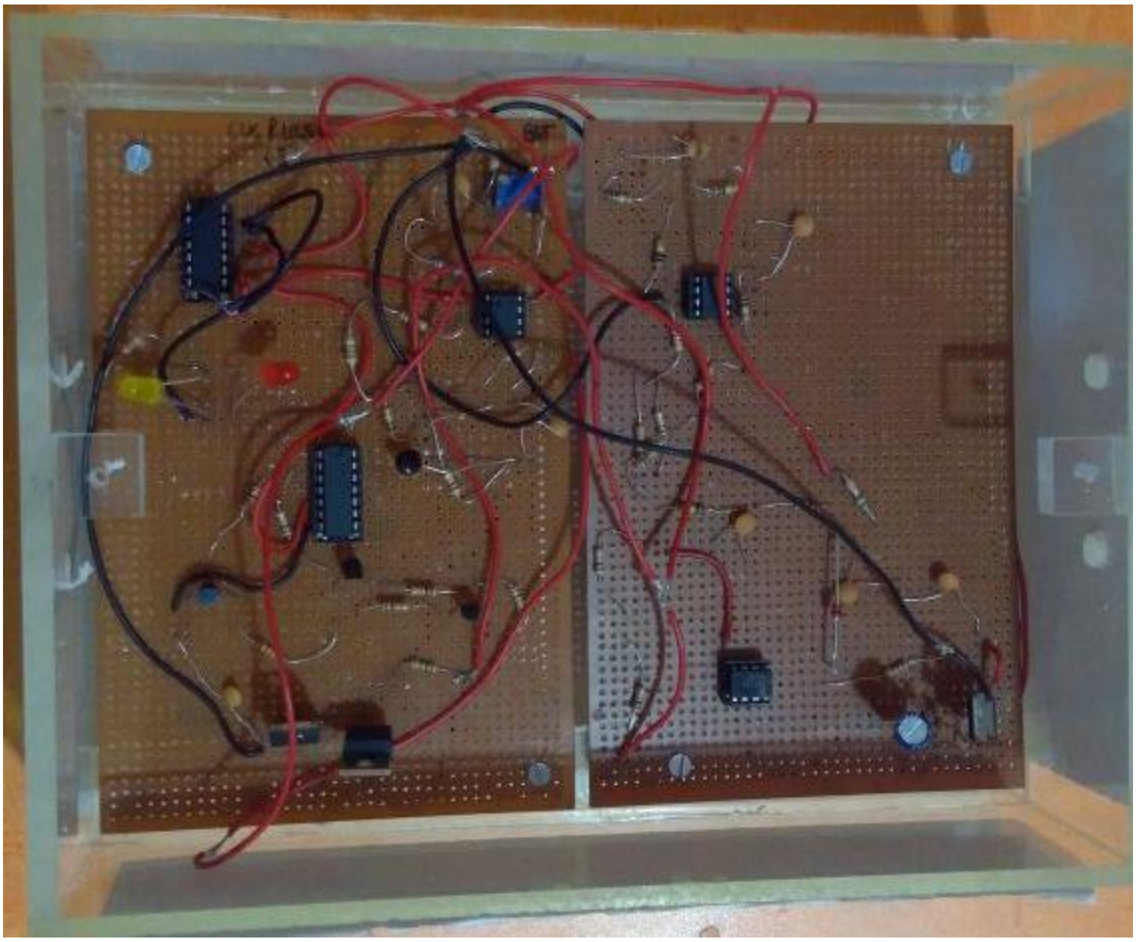
Circuit connection of above two alongwith comparator in cascaded design.





Testing the complete schema on the PCB

GETTING THE COMPLETE PCB BOXED INSIDE OF AN ACRYLIC BOX:



The arcrylic box's dimensions included 4 holes, 2 on each sides. 2 for PV i.e PV+ and PV- and 2 for battery + and battery -.We parts which were damaged during soldering/testing were replaced by new wires. Also, the parts of the circuit which were dry soldered were first desoldered and then properly soldered again.

After this, we did individual testing of different parts of the schema that included voltage comparator, voltage regulator, clock oscillator, floating potential and D flip flop. The wires that were too small to fit in the box, were increased in length and then finally we increased length of PV and battery wires so that they can be taken out from the box.

OUR FINAL PRODUCT:

INSTALLATION OF SOLAR PANEL:

The prototype of solar charge controller that we fabricated above on the PCB was made for testing purpose only. This was done to be assured of the fact that when we get our fabricated glass epoxy PCB from the KiCad gerber file, it will be performance efficient. Also, the solar charge controller that we built manually could be used as a backup in case the glass epoxy PCB does not work and can also be used for display for the juniors to test and create one for their own.

The glass epoxy PCB proved to be performance efficient upon testing and was then deployed in the government school for the installation of solar panel which will be helpful in case of power-cuts. The entire building and deployment cost of the project was borne by the students and members of the electronics and telecommunications department of Symbiosis Institute of Technology, Pune.