



GTU Innovation & Startup Center



A REPORT ON PCB DESIGN & HAND ON SESSION

Speaker: Mr. Jaydeep Bhadani

Date: 14th-15th July, 2018

Time: 10:00am to 5:00pm

Venue:

GTU Innovation and Start-up Center – Rajkot Hub AVPTI Campus,

Opp. Hemugadhavi Hall, Tagore Road, Rajkot – 360001

Introduction about PCB

A printed circuit board (PCB) mechanically supports and electrically connects electronic components or electrical components using conductive tracks, pads and other features etched from one or more sheet layers of copper laminated onto and/or between sheet layers of a non-conductive substrate. Components are generally soldered onto the PCB to both electrically connect and mechanically fasten them to it.

Printed circuit boards are used in all but the simplest electronic products. They are also used in some electrical products, such as passive switch boxes.

Alternatives to PCBs include wire wrap and point-to-point construction, both once popular but now rarely used. PCBs require additional design effort to lay out the circuit, but manufacturing and assembly can be automated. Specialized CAD software is available to do much of the work of layout. Mass-producing circuits with PCBs is cheaper and faster than with other wiring methods, as components are mounted and wired in one operation. Large numbers of PCBs can be fabricated at the same time, and the layout only has to be done once. PCBs can also be made manually in small quantities, with reduced benefits.

PCBs can be single-sided (one copper layer), double-sided (two copper layers on both sides of one substrate layer), or multi-layer (outer and inner layers of copper, alternating with layers of substrate). Multi-layer PCBs allow for much higher component density, because circuit traces on the inner layers would otherwise take up surface space between components. The rise in popularity of multilayer PCBs with more than two, and especially with more than four, copper planes was concurrent with the adoption of surface mount technology. However, multilayer PCBs make repair, analysis, and field modification of circuits much more difficult and usually impractical

History

Before the development of printed circuit boards electrical and electronic circuits were wired point-to-point on a chassis. Typically, the chassis was a sheet metal frame or pan, sometimes with a wooden bottom. Components were attached to the chassis, usually by insulators when the connecting point on the chassis was metal, and then their leads were connected directly or with jumper wires by soldering, or sometimes using crimp connectors, wire connector lugs on screw terminals, or other methods. Circuits were large, bulky, heavy, and relatively fragile (even discounting the breakable glass envelopes of the vacuum tubes that were often included in the circuits), and production was labor-intensive, so the products were expensive.

Development of the methods used in modern printed circuit boards started early in the 20th century. In 1903, a German inventor, Albert Hanson, described flat foil conductors laminated to an insulating board, in multiple layers. Thomas Edison experimented with chemical methods of plating conductors onto linen paper in 1904. Arthur Berry in 1913 patented a print-and-etch method in the UK, and in the United States Max Schoop obtained a patent to flame-spray metal onto a board through a patterned mask. Charles Ducas in 1927 patented a method of electroplating circuit patterns.

About Speaker

Mr. Jaydeep Bhadani

(Founder of ACDC)



Mechanical Courses

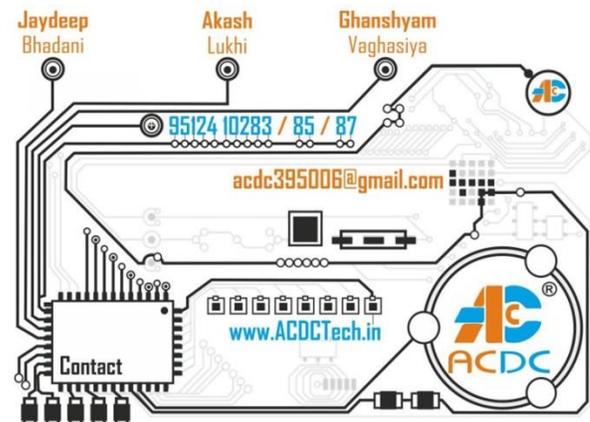


ATL (Atal Tinkering Labs)

- Robotics
- PCB Designing
- Research & Development
- Automation
- Engineering Projects
- Programming
- Embedded
- Training & Workshops
- Components Provider
- IoT
- Solar Solution
- Image Processing

“ Way Of Idea To Innovation ”

ACDC, 3rd Floor, Vallabh Nagar Chamber, B/s Maheshwar Medical,
Baroda Pristage, Varachha Main Road, Surat-395006



Contact: +91 95124 10283

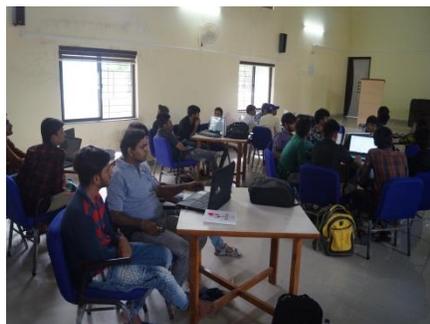
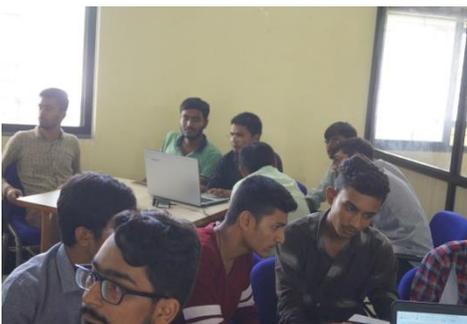
Email: bhadani_jaydeep@gtu.edu.in

LINKEDIN: <https://in.linkedin.com/in/jaydeep-bhadani-34b310b5>

Day 1

First of all sir was gave some knowledge about PCB to the attendee, there was an inter-college meet held at GIC organized “PCB DESIGN & HAND ON SESSION” for students at GTU Innovation and Startup Center-Rajkot on 14th-15th July, 2018. It was great event in which students from across various colleges and department participated and Mr. Jaydeep Bhadani conduct the Event.

As per the event timing event started at 10:00am in the morning, All Interested youngsters reached as per the time. This session is important for the Electrical & EC Department Students and they are learnt a lot from Speaker.

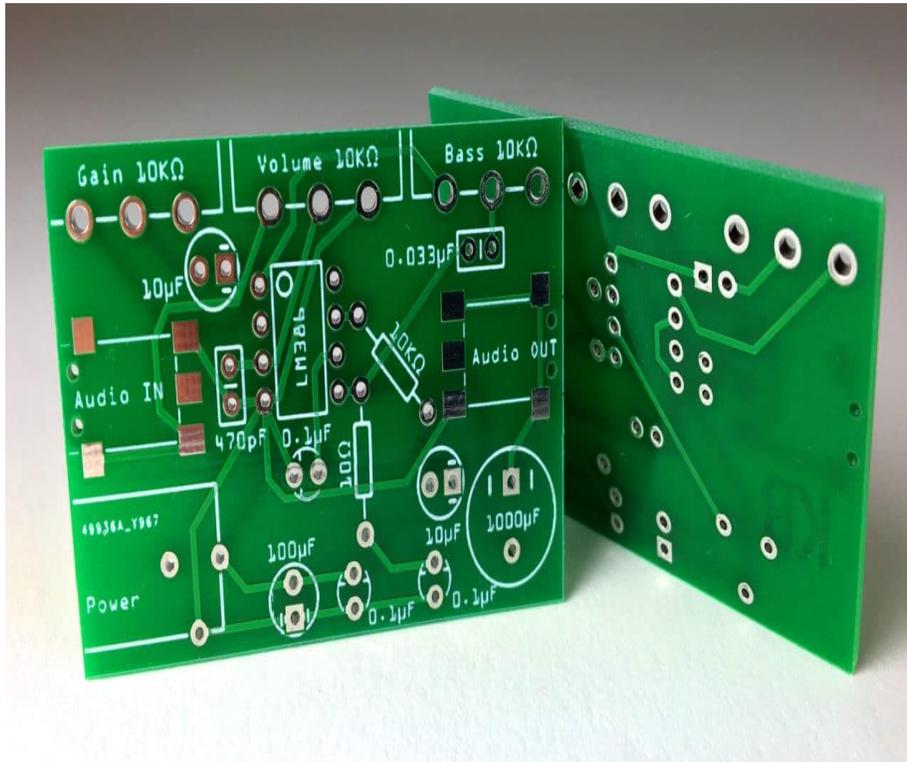


After the brief introduction about himself and also about PCB, sir gave us a basic introduction about how to Design a PCB by ourself.

A printed circuit board, or PCB, is used to mechanically support and electrically connect electronic components using conductive pathways, tracks or signal traces.

The first question is rise on our mind “How to design a PCB”?

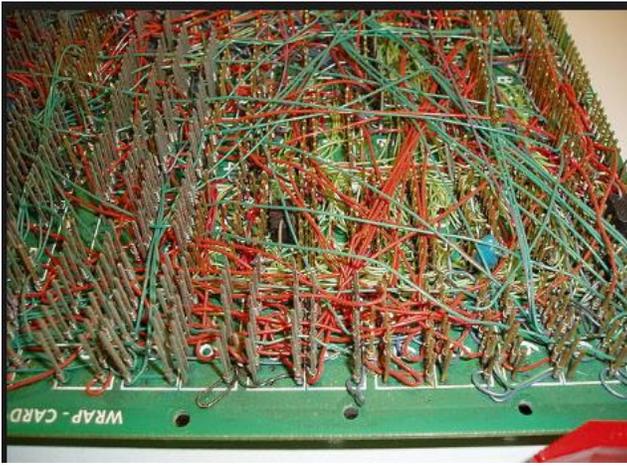
So, let's a move ahead,



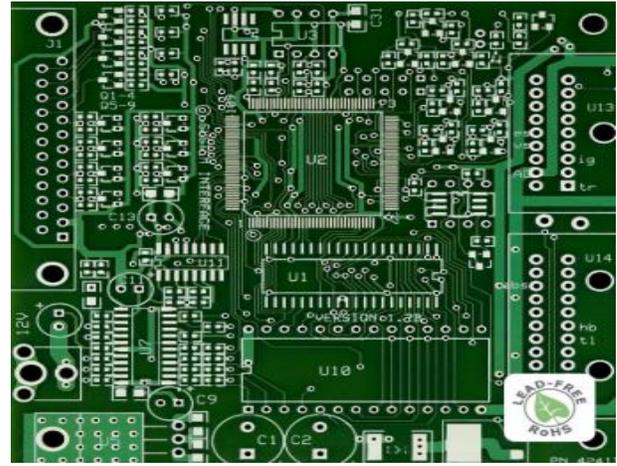
Breadboards are great for prototyping circuits, but they aren't so good for actually using the thing you're building. At some point, you'll probably want to make a project more permanent. The best way to do that is to put it on a PCB.

In this tutorial, I'll walk you through the process of designing a PCB layout and getting it printed by a custom PCB manufacturer. The performance of your circuit will depend greatly on how it's laid out on the PCB, so I'll give you lots of tips on how to optimize your design.

You can always etch PCBs at home with a process that's similar to developing prints from photographic film. But that method is messy and it uses a lot of chemicals. It's much easier (and cheaper) to get your PCB made by a professional manufacturer. To demonstrate the process, I'll use an online service called EasyEDA to design a PCB layout for an LM386 audio amplifier, then I'll have it manufactured and show you the results. Their free online design software is easy to use and the rates are very affordable.



Before PCB



After PCB

Uses of PCB

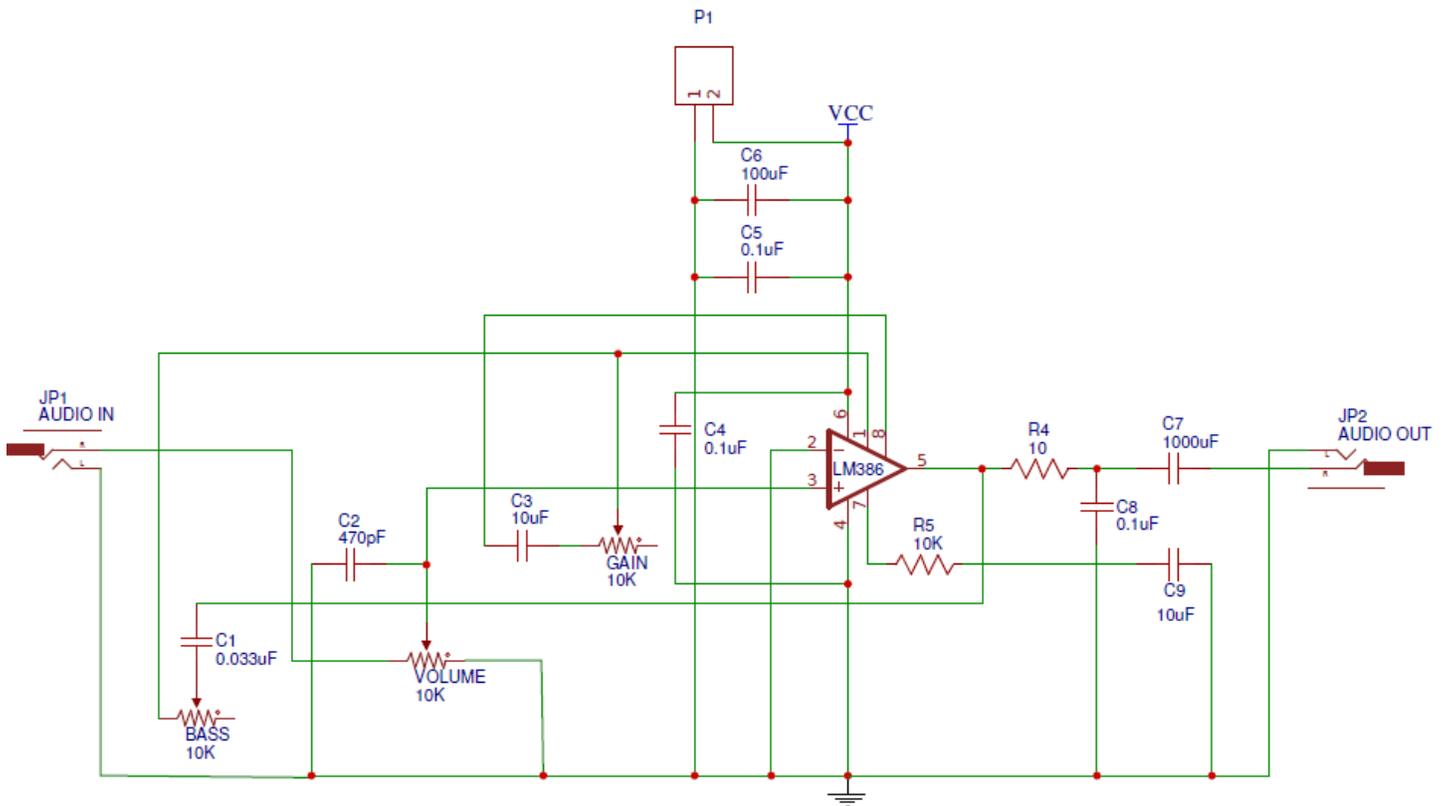
- ✓ Compact Size and Saving of Wire
- ✓ Ease of Repair and Diagnostic
- ✓ Saving of Time
- ✓ Immune to Movement
- ✓ Tight connections and Short Circuits Avoided
- ✓ Low Electronic Noise
- ✓ Low Cost
- ✓ Reliability

Types of PCB

- ✓ Single Sided PCBs
- ✓ Double Sided PCBs
- ✓ Multilayer PCBs
- ✓ Rigid PCBs
- ✓ Flex PCBs
- ✓ Rigid-Flex PCBs

It all starts with a schematic

Schematic Diagram of PCB



Before you start designing your PCB, it's a good idea to make a schematic of your circuit. The schematic will serve as a blueprint for laying out the traces and placing the components on the PCB. Plus, the PCB editing software can import all of the components, footprints, and wires into the PCB file, which will make the design process easier (more on this later).

PCB Manufacturing Process Steps

- ✓ Step 1: Design and Output.
- ✓ Step 2: From File to Film.
- ✓ Step 3: Printing the Inner layers: Where Will the Copper Go?
- ✓ Step 4: Removing the Unwanted Copper.
- ✓ Step 5: Layer Alignment and Optical Inspection.
- ✓ Step 6: Layer-up and Bond.
- ✓ Step 7: Drill.
- ✓ Step 8: Plating and Copper Deposition.

Day 2

Continuously working as yesterday's session and moving toward to make it by on hand and we are learnt how to making it real working model. This day we have to make a real model, so we are excited to making it. But the question is again rise on our mind, i.e. "How to making PCB at home"?



Make your own printed circuit board at home to reduce problems that arise from loose connections on a breadboard.

This is usually done by converting your circuit's schematic diagram into a PCB layout using PCB layout software. There are many open source software packages for PCB layout creation and design. Some are listed here to give you a head-start:

1. Cadsoft Eagle
2. PCBWizard

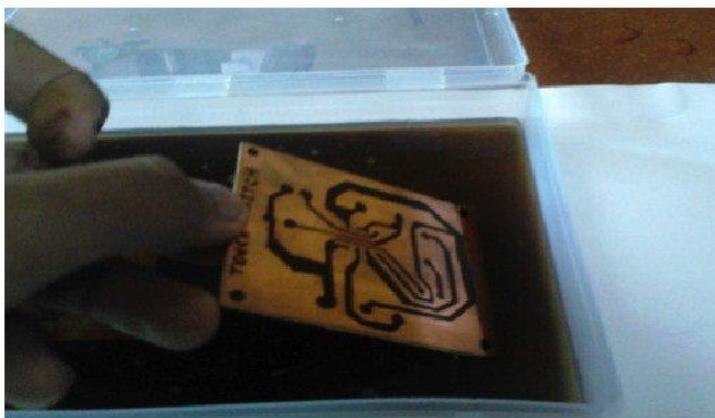
I designed my circuit schematic in Cadsoft Eagle.

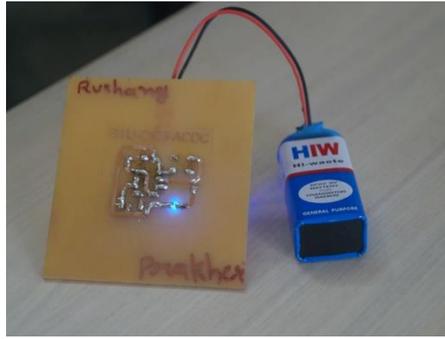
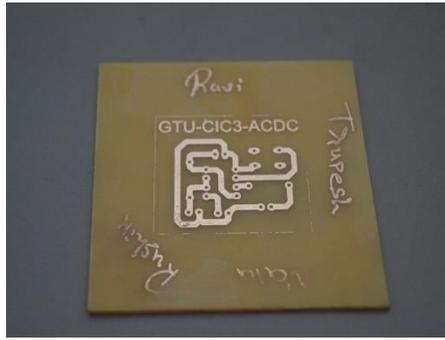
You need also: Permanent black marker, blade cutter, sandpaper, kitchen paper, cotton wool, some old clothes. To start making PCB, consider a simple project TOUCH SWITCH using IC555

Glimpses about second day event,



Making a PCB by on hand





Compiled By:
GIC Associate-Rajkot

Edited By:
GIC Team-Rajkot