Report on

1. LAPTOP AUDIO AMPLIFIER (Analog Project)
2. SIMPLE ELECTRONIC CODE LOCK (Digital Project)

Projects

In Electronics hardware workshop

T.E. Electronics & Telecommunication -Semester V

BY

MEET HARIA

MEET JAIN

MIKIN BHANUSHALI

Under the guidance of

HETAL DOSHI

Prof.--

DEPARTMENT OF ELECTRONICS & TELECOMMUNICATION

K. J. SOMAIYA COLLEGE OF ENGINEERING, MUMBAI

UNIVERSITY OF MUMBAI

2012-2013

CERTIFICATE

Project entitled

        1. Laptop Audio Amplifier. (Analog Experiment)
        2. Simple Electronic Code Lock. (Digital Experiment)

Submitted by :   **MEET HARIA**

                   MEET JAIN

                   MIKIN BHANUSHALI

For term work in Electronics Hardware work shop Semester-V   is approved.

Guide   Examiners

  Prof. HETAL DOSHI
------------------------   Internal: ----------------------------
(Name of Internal Guide)

                                    External: ---------------------------

----
Head of Department

                                    Principal

Contents

List of Tables

Table no	Name	Page no
1.1	List of Components	
2.1	Function of Various Pins	

List of Figures

CHAPTER 1: LAPTOP AUDIO AMPLIFIER

1.1   INTRODUCTION

An audio amplifier is an underline electronic amplifier that amplifies low-power audiosignals (signals composed primarily of frequencies between 20 - 20 000 Hz, the human range of hearing) to a level suitable for driving loudspeakers and is the final stage in a typical audio playback chain.

The preceding stages in such a chain are low power audio amplifiers which perform tasks like pre-amplification, equalization, tone control, mixing/effects, or audio sources like record players, CD players, and cassette players. Most audio power amplifiers require these low-level inputs to adhere to line levels.

While the input signal to an audio power amplifier may measure only a few hundredmicrowatts, its output may be tens, hundreds, or thousands of watts.

1.1.1 History

The audio amplifier was invented in 1909 by Lee De Forest when he invented the triode vacuum tube. The triode was a three terminal device with a control grid that can modulate the flow of electrons from the filament to the plate. The triode vacuum amplifier was used to make the firstAM radio.[2]

Early audio power amplifiers were based on vacuum tubes (also known as *valves*), and some of these achieved notably high quality (e.g., the Williamson amplifier of 1947-9). Most modern audio amplifiers are based on solid state devices (transistors such as BJTs, FETs andMOSFETs), but there are still some who prefer tube-based amplifiers, and the valve sound. Audio power amplifiers based on transistors became practical with the wide availability of inexpensive transistors in the late 1960s.

1.1.2 Design parameters

Key design parameters for audio amplifiers are frequency response, gain, noise, and distortion. These are interdependent; increasing gain often leads to undesirable increases in noise and distortion. While negative feedback actually reduces the gain, it also reduces distortion. Most audio amplifiers are linear amplifiers operating in class AB.

1.1.3 Filters and preamplifiers

Since modern digital devices, including CD and DVD players, radio receivers and tape decks already provide a "flat" signal at line level, the preamp is not needed other than as a volume control and source selector. One alternative to a separate

preamp is to simply use passive volume and switching controls, sometimes integrated into a power amplifier to form an *integrated amplifier*.

1.1.4 Further developments in amplifier design

For some years following the introduction of solid state amplifiers, their perceived sound did not have the excellent audio quality of the best valve amplifiers (see valve audio amplifier). This led audiophiles to believe that valve sound had an intrinsic quality due to the vacuum tube technology itself. In 1972, Matti Otala demonstrated the origin of a previously unobserved form of distortion: transient intermodulation distortion (TIM), also called slew rate distortion. TIM distortion was found to occur during very rapid increases in amplifier output voltage.[3] TIM did not appear at steady state sine tone measurements, helping to hide it from design engineers prior to 1972. Problems with TIM distortion stem from reduced open loop frequency response of solid state amplifiers. Further works of Otala and other authors found the solution for TIM distortion, including increasing slew rate, decreasing preamp frequency bandwidth, and the insertion of a lag compensation circuit in the input stage of the amplifier.[4][5][6] In high quality modern amplifiers the open loop response is at least 20 kHz, canceling TIM distortion. However, TIM distortion is still present in most low price home quality power amplifiers.[citation needed]

The next step in advanced design was the Baxandall Theorem, created by Peter Baxandall in England.[7] This theorem introduced the concept of comparing the ratio between the input distortion and the output distortion of an amplifier. This new idea helped audio design engineers to better evaluate the distortion processes within an amplifier.

Figure 1.1 Following are different watts speakers driven by audio amplifier

1.2 CIRCUIT DESCRIPTION

Usually, the audio output from a laptop's built-in speakers is low. A power amplifier is required to get a high volume. Here is a simple circuit to amplify the laptop's audio output.

The circuit is built around power amplifier IC LA 4440 (IC1) and a few other components. LA4440 is a dual channel audio power amplifier. It has low distortion over a wide range of low to high frequencies with good channel separation. Inbuilt dual channels enable it for stereo and bridge amplifier applications.

In dual mode LA4440 gives 6 watts per channel and in bridge mode 19- watt output. It has ripple rejection of 46 dB. The audio effect can be realised by using two 6-watt speakers. Connect pins 2, 6 and ground of IC1 to the stereo jack which is to be used with the laptop.

Assemble the circuit on a general-purpose PCB and enclose in a suitable cabinet. The circuit works off regulated 12V power supply. It is recommended to use audio input socket in the circuit board. Use a proper heat-sink for LA4440.

By varying the voltage applied or tone/volume control in Laptop, the volume of the amplifier can be varied.

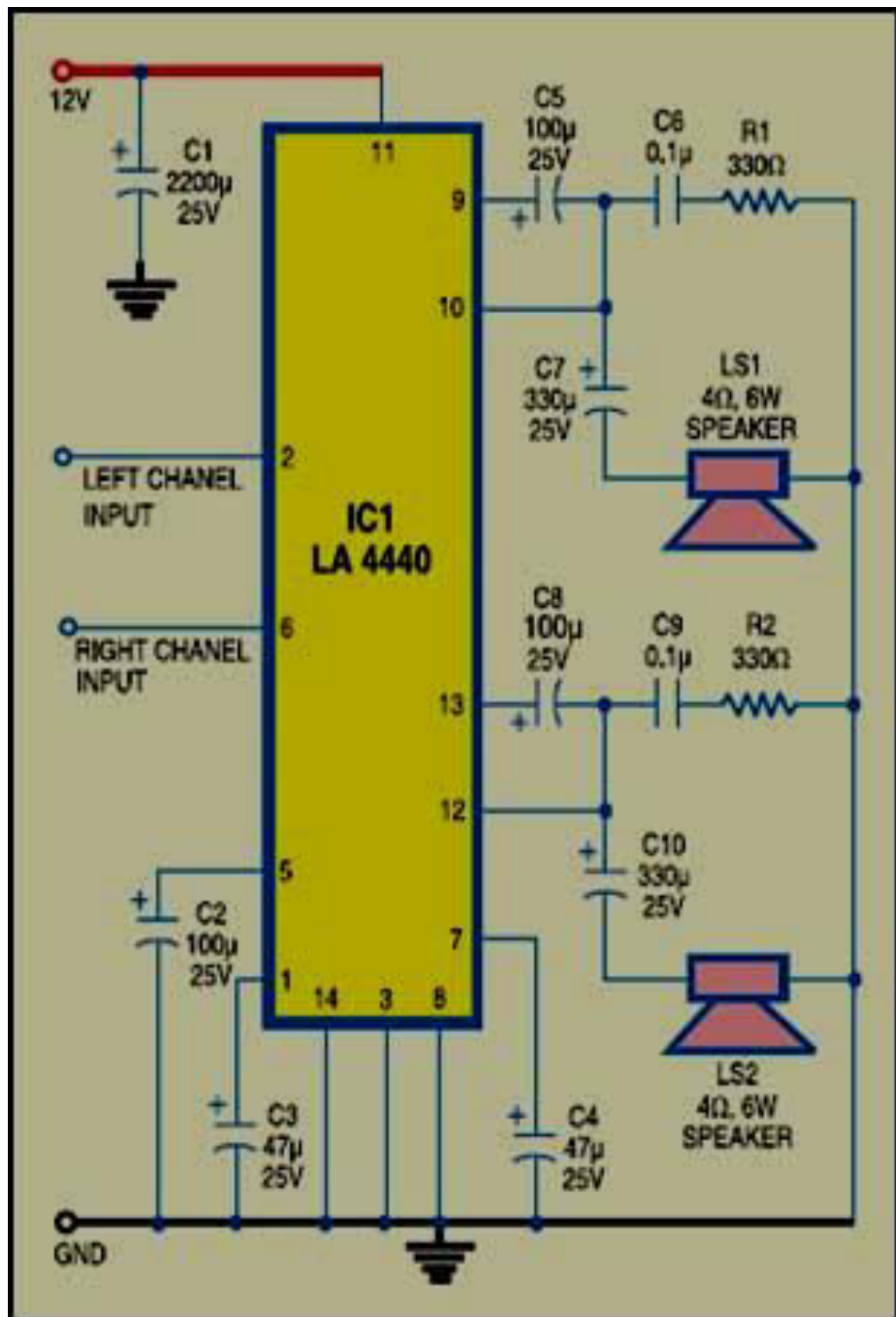| NAME OF COMPONENT	NO. OF COMPONENTS	VALUE	TOLERANCE
1) IC LA 4440	1	--------	6WATTS*2
2) CAPACITOR	3	100uf	25V
3) CAPACITOR	2	330uf	25V
4) CAPACITOR	2	0.1uf	25V
5) CAPACITOR	2	47uf	25V
6) CAPACITOR	1	2200uf	25V
7) RESISTOR	2	330ohm	0.25 WATTS
8) AUXILARY CABLE	1	-------	------
9) SPEAKERS	2	4ohm	6WATTS

Table 1.1 List of Components

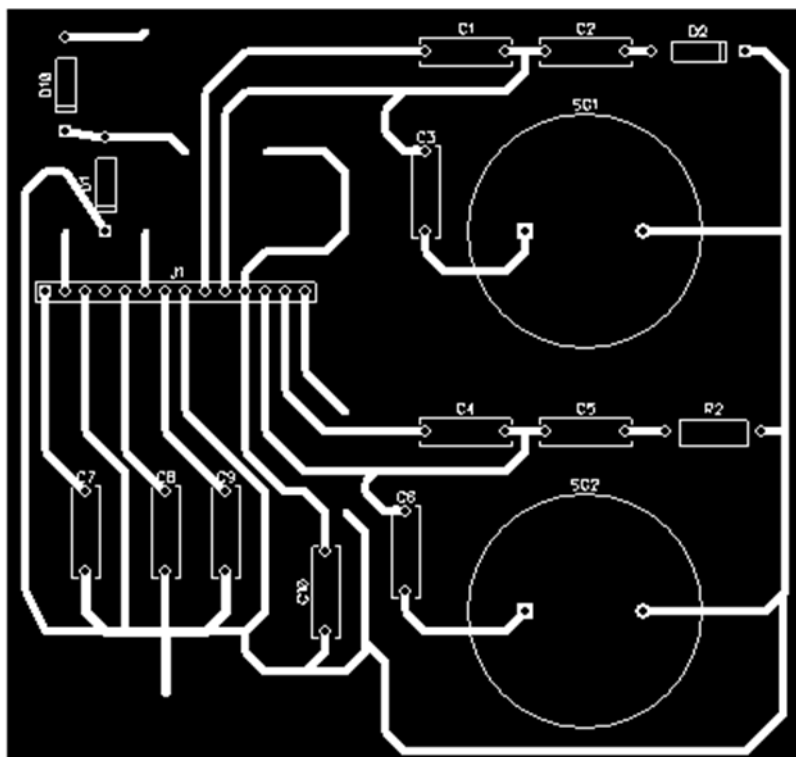**Figure 1.2 Circuit diagram of the Laptop Audio Amplifier**

Figure 1.3 PCB Layout

1.2.1 Description of IC:LA4440:

Features

• Built-in 2 channels (dual) enabling use in stereo and bridge amplifier applications.

   Dual : 6W×2 (typ.)

   Bridge : 19W (typ.)

• Minimum number of external parts required.

• Small pop noise at the time of power supply ON/OFF and good starting balance.

• Good ripple rejection : 46dB (typ.)

• Good channel separation.

• Small residual noise (Rg=0).

• Low distortion over a wide range from low frequencies to high frequencies.

• Easy to design radiator fin.

• Built-in audio muting function.

• Built-in protectors.

    a. Thermal protector

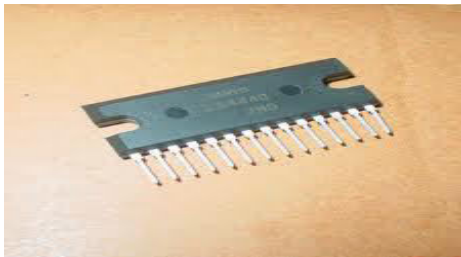   b. Overvoltage, surge voltage protector

    c. Pin-to-pin short protector

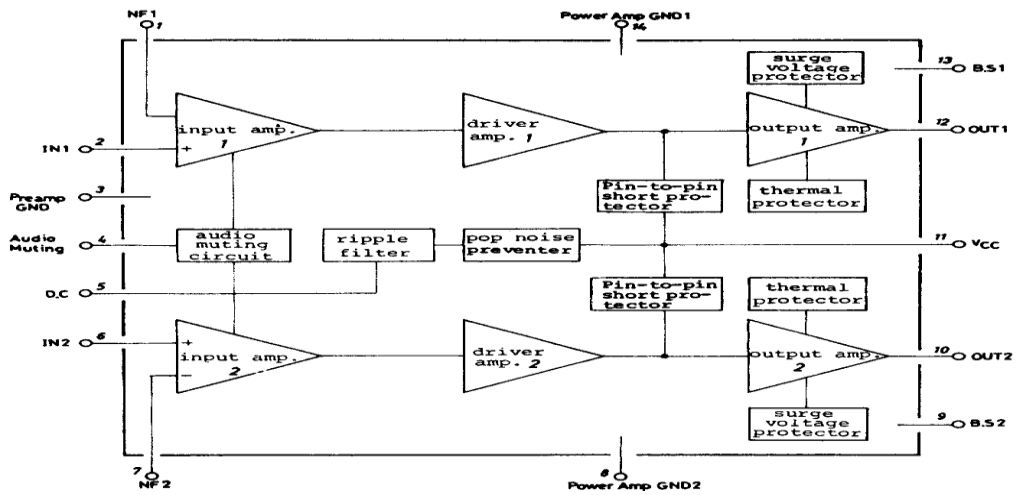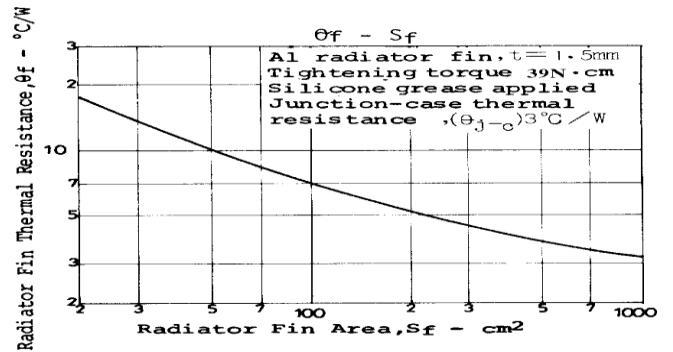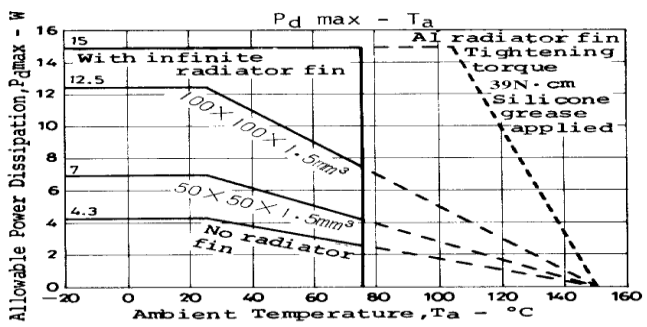**Fig 1.4 IC LA 4440**

Fig 1.5 3.5mm Auxillary Jack

Graph of power dissipation

Graph of thermal resistance v/s area

v/s ambient temperature

P_d max − T_a

Allowable Power Dissipation, P_d max − W

15
With infinite radiator fin
12.5
$100 \times 100 \times 1.5mm^3$
10
8
7
6
$50 \times 50 \times 1.5mm^3$
4.3
4
No radiator fin
2
0
−20 0 20 40 60 80 100 120 140 160

Ambient Temperature, T_a − °C

Al radiator fin
Tightening torque
39N·cm
Silicone grease applied

θ_f − S_f

Radiator Fin Thermal Resistance, θ_f − °C/W

Al radiator fin, $t = 1.5mm$
Tightening torque 39N·cm
Silicone grease applied
Junction-case thermal resistance, $(\theta_{j-c})3$°C/W

3 5 100 2 3 5 7 1000

Radiator Fin Area, S_f − cm^2

9

Proper Cares in Using IC:

Maximum ratings

If the IC is used in the vicinity of the maximum ratings, even a slight variation in conditions may cause the maximum ratings to be exceeded, thereby leading to breakdown. Allow an ample margin of variation for supply voltage, etc. and use the IC in the range where the maximum ratings are not exceeded.

Printed circuit board

When making the board, refer to the sample printed circuit pattern and be careful that no feedback loop is formed between input and output.

Oscillation preventing capacitor

Normally, a polyester film capacitor is used for 0.1μF + 4.7Ω. The capacitance value can be reduced to 0.047μF depending on the stability of the board.

Others

Connect the radiator fin of the package to GND.

1.3 APPLICATION AND FUTURE SCOPE

- ➢ The laptop audio amplifier mentioned has a major advantage of its portability. It can be connected to any laptop or mobile phones or other such devices which outputs the audio signal in milli-watts through a 3.5mm socket and with the help of 3.5mm 1:2 Aux converter, it can be applied to the designed circuit so as to produce a 6 watts output.

- ➢ The device is quite compact and is easily portable.

- ➢ Using LA4440 one can connect two speakers at a time which can cover a better audibility range with a good uniformity.

- ➢ LA 4440 IC can also be used along with different circuits involving different amplifier IC's which can increase the gain and hence the power of the amplified audio signal

- ➢ The Laptop Audio Amplifier discussed is cheaper and simple in construction on single layered pcb's and quite easy to mount.

- ➢ Important applications include :

1. Public address systems

2. Theatrical and concert sound reinforcement

3. Domestic sound systems such as a stereo or home-theatre system.

4. Instrument amplifiers including guitar amplifiers.

1.4 RESULT AND CONCLUSION:

The output of this amplifier is 6watts with dual channel which is loud enough to be heard clearly than directly from the Laptops. Thus LA4440 provides a good amplification of audio signal from Laptops which is in milliwatts and outputs 6 watts power of signal.

Thus Laptop audio amplifier designed is cheaper, easier to construct, mount and use, the volume can also be varied directly by varying volume control in laptop, is easily portable and compatible with all Laptops.

CHAPTER 2: SIMPLE ELECTRONIC CODE LOCK

2.1 INTRODUCTION

An electronic lock (more precisely an electric lock) is a locking device which operates by means   of electric current. Electric locks are sometimes stand-alone with an electronic control assembly mounted directly to the lock. More often electric locks are connected to an access control system. The advantages of an electric lock connected to an access control system include: key control, where keys can be added and removed without re-keying the lock cylinder; fine access control, where time and place are factors; and transaction logging, where activity is recorded.

2.1.1 Operation

Electric locks use magnets, solenoids, or motors to actuate the lock by either supplying or removing power. Operating the lock can be as simple as using a switch, for example an apartment intercom door release, or as complex as a biometric based access control system.

2.1.2 Types of electronic locks

The most basic type of electronic lock is a magnetic lock (commonly called a mag lock). A large electro-magnet is mounted on the door frame and a corresponding armature is mounted on the door. When the magnet is powered and the door is closed, the armature is held fast to the magnet. Mag locks are simple to install and are very attack resistant. One drawback is that improperly installed or maintained mag locks can fall on people. and also that one must unlock the mag lock to both enter and leave. This has caused fire marshals to impose strict codes on the use of mag locks and the access control practice in general. Additionally, NFPA 101, Standard for Life Safety and Security, as well as the ADA (Americans with Disability Act) require "no prior knowledge" and "one simple movement" to allow "free egress". This means that a person must be able to walk up to a door and with

one motion (no push buttons, or having another person unlock the door, or read a sign "special knowledge") and exit. Other problems include a lag time (delay) in releasing as the collapsing magnetic field is not instantaneous. This lag time can cause a user to walk into the door. Finally, mag locks by design fail unlocked, that is if power is removed they unlock. This could be a problem where security is a prime concern. Additionally, power outages could affect mag locks installed on fire listed doors, which are required to remain latched at all times. Thus, a mag lock would not meet current fire codes as the primary means of securing a fire listed door to a frame.[1] Because of this, many commercial doors (this typically does not apply to private residences) are moving over to stand alone locks, or electric locks installed under a Certified Personnel Program. The first mechanical recodable card lock was invented in 1976 by Tor Sørnes, who had worked for VingCard since the 1950s. The first card lock order was shipped in 1979 to Westin Peachtree Plaza Hotel, Atlanta, USA. This product triggered the evolution of electronic locks for the hospitality industry.

Electric strikes (also called electric latch release) replace a standard strike mounted on the door frame and receive the latch and latch bolt. Electric strikes can be simple to install when they are designed for drop-in replacement of a standard strike, although some electric strikes require that the door frame be heavily modified. Installation of a strike into a Fire Listed Door (for open backed strikes on pairs of doors) or Frame must be done under listing agency authority if any modifications to the frame are required (mostly for commercial doors and frames). Since there is no current Certified Personnel Program to allow field installation of electric strikes into fire listed door openings, listing agency field evaluations would most likely require the door and frame to be de-listed and replaced. Electric strikes allow mechanical free egress: As a user leaves, he operates the lockset in the door, not the electric strike in the door frame. Electric strikes can also be either fail unlocked (except in Fire Listed Doors, as they must remain latched when power is not present), as a mag lock, or the more secure fail locked. Electric strikes are easier to attack than a mag lock. It is simple to lever the door open at the strike, as often there is an increased gap between the strike and the door latch. Latch guards are often used to cover this gap.

Electric mortise and cylindrical locks are drop in replacements for the door mounted mechanical locks. A hole must be drilled in the door for electric power wires. Also a power transfer hinge[2] is used to get the power from the door frame to the door. Electric mortise and cylindrical locks allow mechanical free egress. Electric mortise and cylindrical locks can be either fail unlocked or fail locked. In the US, UL rated doors must retain their rating. In new construction doors are cored and then rated. In retrofits, the doors must be re-rated.

Electrified exit hardware, sometimes called panic hardware or crash bars, are used in fire exit applications. One pushes against the bar to open it, making it the easiest of mechanically free exit methods. Electrified exit hardware can be either fail unlocked or fail locked. A drawback of electrified exit hardware is their complexity which requires skill to install and maintenance to assure proper

function. Only hardware labeled "Fire Exit Hardware" can be installed on fire listed doors and frames and must meet both Panic Exit listing Standards and Fire listing Standards.

Motor operated locks are used throughout Europe. A European motor operated lock has two modes, day mode where only the latch is electrically operated, and night mode where the more secure deadbolt is electrically operated.

Numerical codes, passwords and passphrases

Perhaps the most prevalent form of electronic lock is that using a numerical code for authentication; the correct code must be entered in order for the lock to deactivate. Such locks typically provide a keypad, and some feature an audible response to each press. Combination lengths are usually between 4 and 6 digits long.A variation on this design involves the user entering the correct password or passphrase.

Security tokens

Another means of authenticating users is to require them to scan or "swipe" a security tokensuch as a smart card or similar, or to interact a token with the lock. For example, some locks can access stored credentials on a personal digital assistant (PDA) using infrared data transfer methods.

Biometrics

As biometrics become more and more prominent as a recognized means of positive identification, their use in security systems increases. Some new electronic locks take advantage of technologies such as fingerprint scanning, retinal scanning and iris scanning, and voiceprint identification to authenticate users.

RFID

Radio-frequency identification (RFID) is the use of an object (typically referred to as an RFID tag) applied to or incorporated into a product, animal, or person for the purpose of identification and tracking using radio waves. Some tags can be read from several meters away and beyond the line of sight of the reader. This technology is also used in modern electronic locks.

FIGURE 2.1 VARIOUS TYPES OF LOCK

2.2 CIRCUIT DESCRIPTION

The circuit diagram of a simple electronic code lock is shown in figure. A 9-digit code number is used to operate the code lock. When power supply to the circuit is turned on, a positive pulse is applied to the RESET pin (pin 15) through capacitor C1. Thus, the first output terminal Q1 (pin 3) of the decade counter IC (CD 4017) will be high and all other outputs (Q2 to Q10) will be low. To shift the high state from Q1 to Q2, a positive pulse must be applied at the clock input terminal (pin14) of IC1. This is possible only by pressing the push-to-on switch S1 momentarily. On pressing switch S1, the high state shifts from Q1 to Q2. Now, to change the high state from Q2 to Q3, apply another positive pulse at pin 14, which is possible only by pressing switch S2. Similarly, the high state can be shifted up to the tenth output (Q10) by pressing the switches S1 through S9 sequentially in that order. When Q10 (pin 11) is high, transistor T1 conducts and energizes relay RL1. The relay can be used to switch 'on' power to any electrical appliance.

Diodes D1 through D9 are provided to prevent damage/malfunctioning of the IC when two switches corresponding to 'high' and 'low' output terminals are pressed simultaneously. Capacitor C2 and resistor R3 are provided to prevent noise during switching action. Switch S10 is used to reset the circuit manually. Switches S1 to S10 can be mounted on a keyboard panel, and any number or letter can be used to mark them. Switch S10 is also placed together with other switches so that any stranger trying to operate the lock frequently presses the switch S10, thereby resetting the circuit many times. Thus, he is never able to turn the relay 'on'. If necessary, two or three switches can be connected in parallel with S10 and placed on the keyboard panel for more safety.

A 12V power supply is used for the circuit. The circuit is very simple and can be easily assembled on a general-purpose PCB. The code number can be easily changed by changing the connections to switches (S1 to S9).

Figure 2.2 circuit diagram of Simple Code Lock
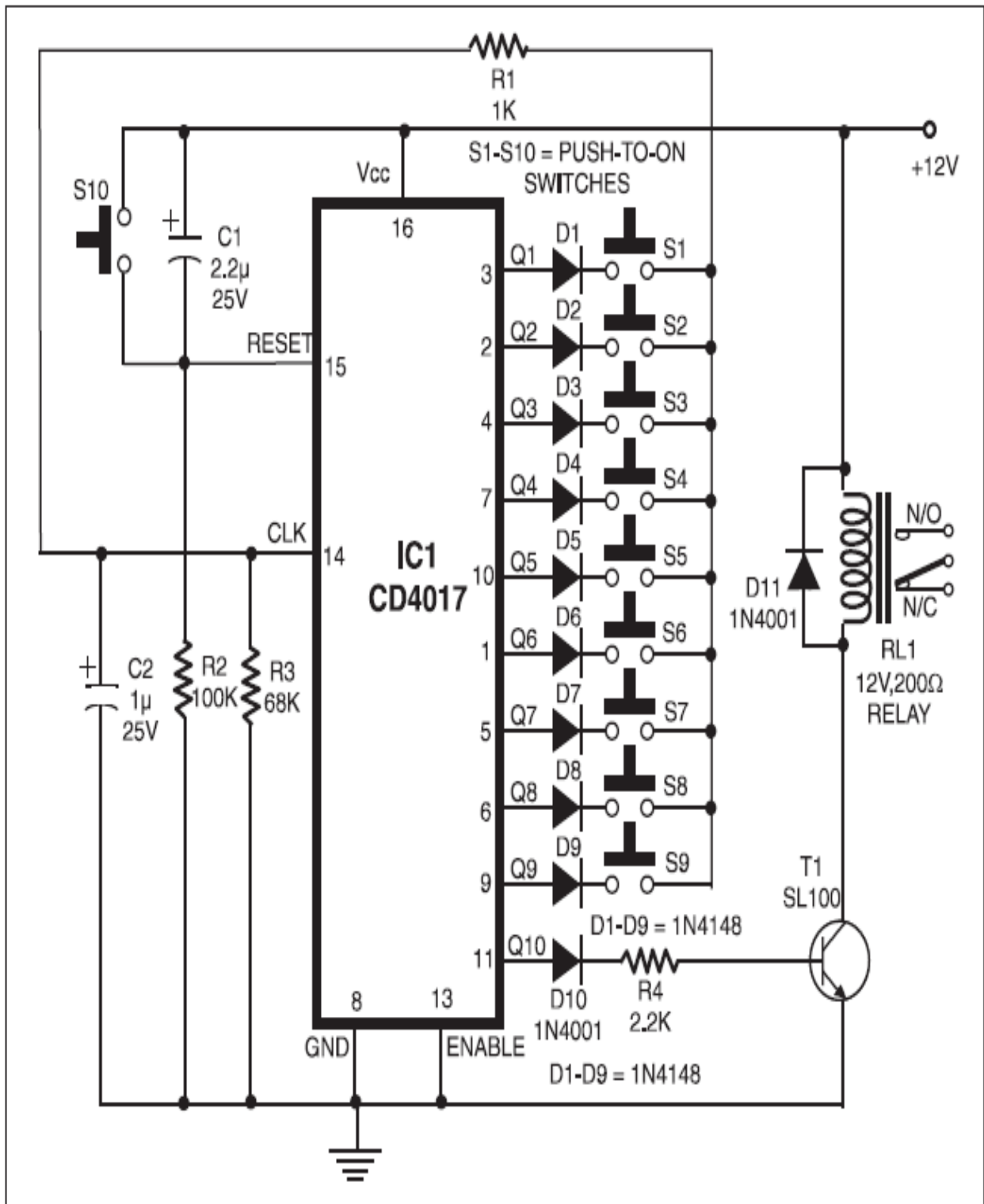
SIMPLE ELECTRONIC LOCK(DIGITAL)(needs 12v for operation)
1. ten PUSH-TO-ON switches
2. nine 1N4148 diodes
3. two 1N4001 diodes
4. IC CD4017 [DIVIDE BY 10 COUNTER (5 STAGE JOHNSON COUNTER)]
5.one 2.2uf, 25v capacitor
6. one 1uf 25v capacitor
7. one piece (1k, 100k, 68k, 2.2k) resistors(ek ek hi lena)
8. one SL100 transistor
9. one 12v,200ohm relay

Figure 2.3 PCB Layout

2.2.1 Description of IC:CD4017

4017 decade counter

The **4017 IC** is a 16-pin CMOS decade counter from the 4000 series. It takes clock pulses from the clock input, and makes one of the ten outputs come on in sequence each time a clock pulse arrives.

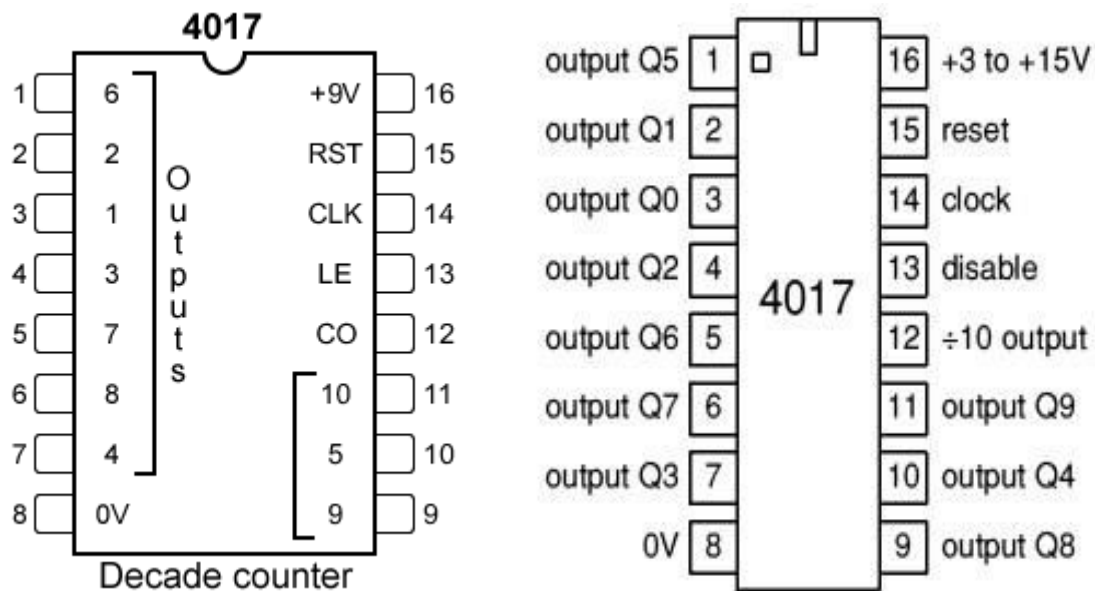**Figure 2.4 Pin Diagram of IC CD4017**

Table 2.1 Functions of various Pins

Pin number	Name	Purpose

| 1	6	The 6th sequential output
2	2	The 2nd sequential output
3	1	The 1st sequential output
4	3	The 3rd sequential output
5	7	The 7th sequential output
6	8	The 8th sequential output
7	4	The 4th sequential output
8	0 V, V_{DD}	The connection to the 0 V rail
9	9	The 9th sequential output
10	5	The 5th sequential output
11	10	The 10th sequential output
12	CO	**C**arry **o**ut output - outputs high on counts 0 to 4, outputs low on counts 5 to 9 (thus a transition from low to high occurs when counting from 9 back to 0)
13	LE	**L**atch **e**nable - latches on the current output when high (i.e. the chip counts when LE is low)
14	CLK	**Cl**oc**k** in
15	RST	**R**e**s**e**t** - sets output 1 high and outputs 2 through 10 low, when taken high
16	+9 V, V_{CC}	The connection to the +V_{CC} rail (voltage between +3 V and +15 V)

Example: Electronic Roulette
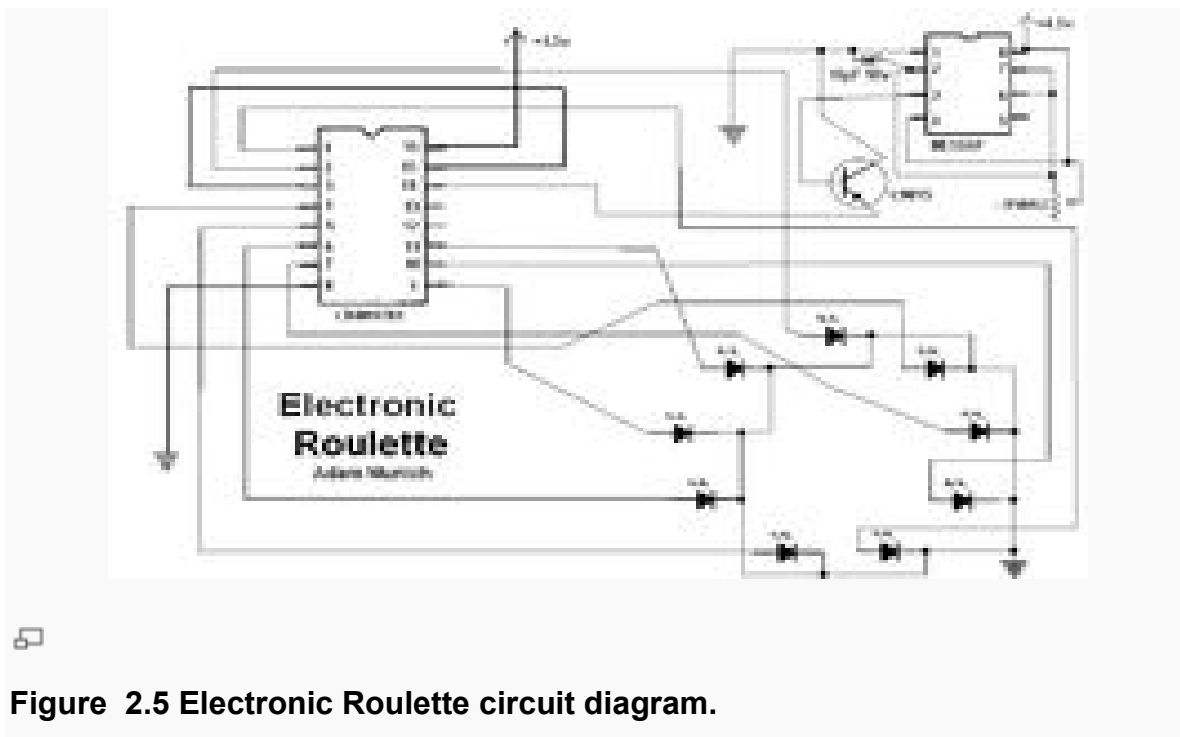
Figure 2.5 Electronic Roulette circuit diagram.

The circuit diagram above shows how to create a game of roulette using the 4017 decade counter and various other electronic parts. The variable resistor adjusts the spin speed.

General Description:

The CD4017BC is a 5-stage divide-by-10 Johnson counter with 10 decoded outputs and a carry out bit. The CD4022BC is a 4-stage divide-by-8 Johnson counter with 8 decoded outputs and a carry-out bit. These counters are cleared to their zero count by a logical "1" on their reset line. These counters are advanced on the positive edge of the clock signal when the clock enable signal is in the logical "0" state. The configuration of the CD4017BC and CD4022BC permits medium speed operation and assures a hazard free counting sequence. The 10/8 decoded outputs are normally in the logical "0" state and go to the logical "1" state only at their respective time slot. Each decoded output remains high for 1 full clock cycle. The carry-out signal completes a full cycle for every 10/8 clock input cycles and is used as a ripple carry signal to any succeeding stages.

Features:

■ Wide supply voltage range: 3.0V to 15V

■ High noise immunity: 0.45 VDD (typ.)

■ Low power Fan out of 2 driving 74L

TTL compatibility: or 1 driving 74LS

■ Medium speed operation: 5.0 MHz (typ.)

with 10V VDD

■ Low power: 10 µW (typ.)

■ Fully static operation

<u>Applications</u>

• Automotive

• Instrumentation

• Medical electronics

• Alarm systems

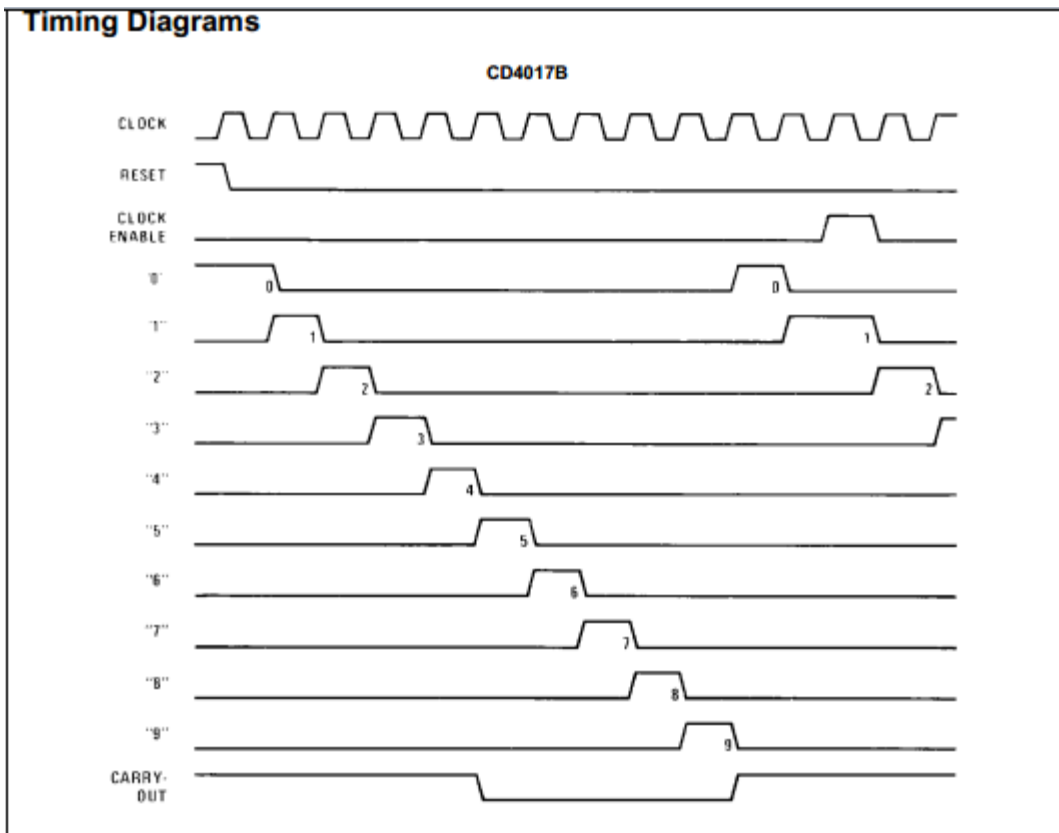• Industrial electronics
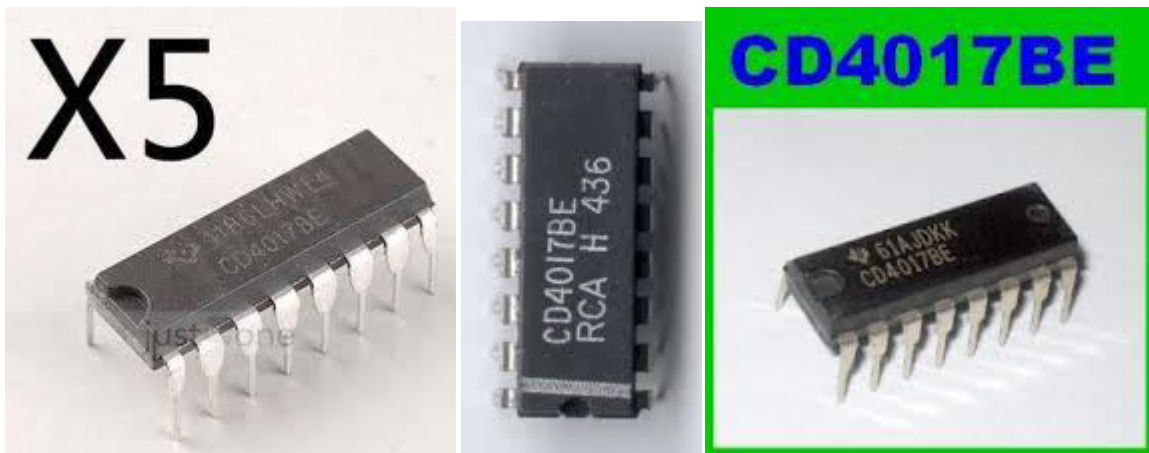
• Remote metering

**Figure 2.6 Timing Diagrams**

Figure 2.7   Different CD4017BE IC's

2.3 APPLICATION AND FUTURE SCOPE:

APPLICATION:

1.It is very simple in construction and so dependable.

2.It weight is less and that for case it can be used in any remote place and easily bearable.

3.It is perfect for locking that documents which are used all time because it is easy to operate.

4.We can easily change the combination password as we like best at any time on our demand.

5.Convenience, requiring no keys, cards or tokens which can become lost or stolen

6.Electric mortise and cylindrical locks are drop in replacements for the door mounted mechanical locks.

7.They are used for high security purposes .

8.Many industries and laboratories use digital code locking system for high security purposes.

9.Vending operator has full control of keys.

10. Any device can be connected to this lock which requires high safety while operation and can be operated at any desired voltage.

11. In future, this application will be widely used by highly developed cities for higher security purposes.
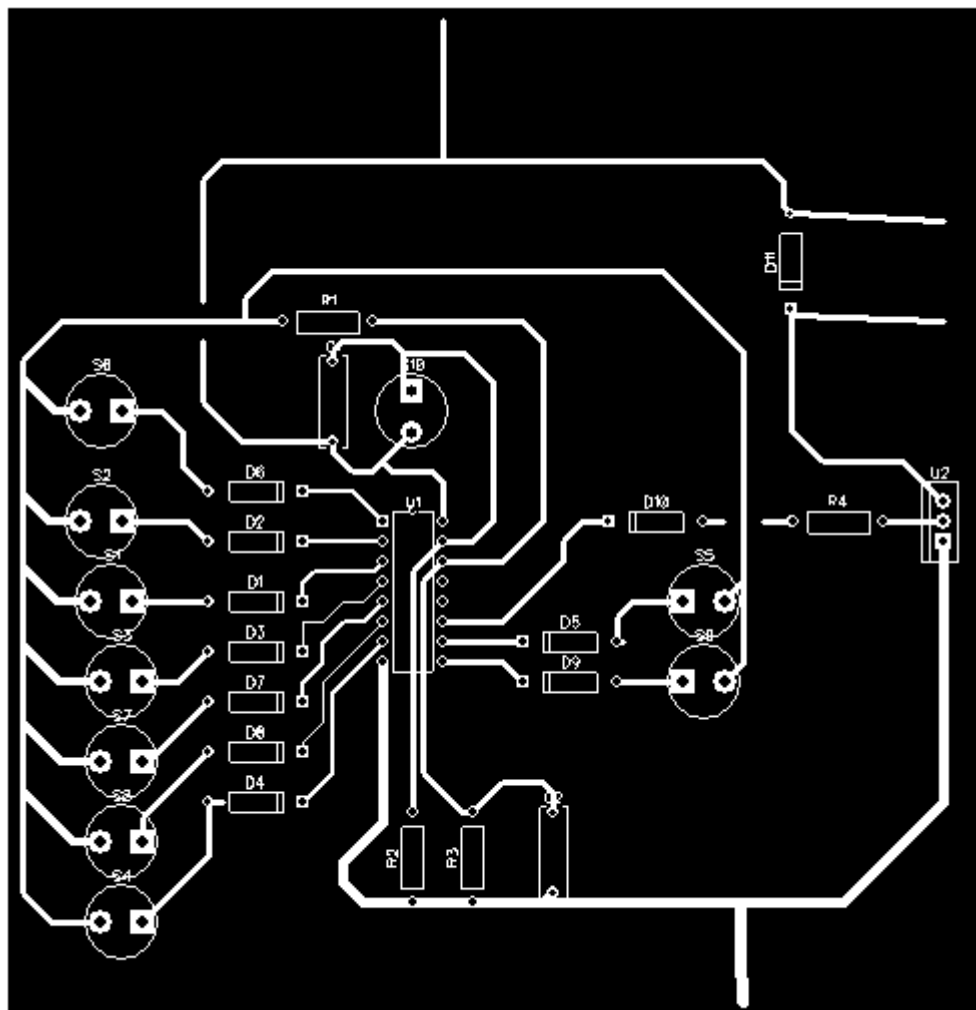
2.4 RESULT AND CONCLUSION

The project of 'SIMPLE ELECTRONIC CODE LOCK' is found to be successfully working. The complexity and maintainence of the circuit is small and is easily portable. For demonstration purpose one code is created.

We feel that this system is very marketable, comparatively inexpensive due to low power consumption and highly reliable.

REFERENCES

- "ELECTRONICS FOR YOU", MAGAZINE.
- WIKIPEDIA
- ALLABOUTCIRCUITS.COM
- EHOW.COM
- NEAMEN, ELECTRONIC DEVICES AND CIRCUITS
- www.extremecircuits.net
- ALLDATASHEETS.COM
- DATASHEETARCHIVE.COM

Design and Manufacturing of Printed circuit Board (PCB).

Aim: 1.Design the PCB artwork for the circuit with the help of software.

2.   Prepare the PCB by Photo Chemical Process.
3.   **Theory:**
       What is PCB?
       A printed circuit board, or PCB, is used to mechanically support and electrically connect electronic components using conductive pathways, or traces, etched from copper sheets laminated onto a non-conductive substrate. It is also referred to as printed wiring board (PWB) or etched wiring board. A PCB populated with electronic components is a printed circuit assembly (PCA), also known as a printed circuit board assembly (PCBA).PCBs are rugged, inexpensive, and can be highly reliable. They require much more layout effort and higher initial cost than either wire-wrapped or point-to-point constructed circuits, but are much cheaper and faster for high-volume production.

       Prior to manufacuring PCB, one needs tp prepare layout for the circuit. In the layout preparation we need to consider the placement of varous devices and components used in the circuit, the size of the component. The layout prepartion is done using varoius EDA (Electronics Design Automation) softwares such as ORCAD, EGLE, EXPRESS PCB, DIPTRACE etc. following tips and guidelines should be taken into account in layout preparations.

       Tips on PCB Designing

       At the start of design process, designer must follow these details:

       1.   Accurate actual sizes of the components being used.
       2.   Electrical connections between components.

3. Component mounting data.
4. PCB area.
5. Conductor width and spacing.

On the bas*s of above data, a designer must follow the following points:*

1. All components must be represented.
2. There can be no crossovers in tracks. A crossover will result in shorting of tracks .
3. Always keep point –to –point conductor runs as straight and as short as possible. When unable to use straight lines, use diagonal lines. Combination of straight lines, diagonal lines and 90 –degree bends should be used for track layout.
4. Avoid looped runs.
5. Polarity marks should be drawn for power terminals, diodes, electrolytic capacitors and any such polarized components.
6. Place all resistors, capacitors, diodes etc. parallel to each other and parallel to PCB.
7. Try to group components together on the basis of functional block, component type, and similarity in dissipation, size or polarity.
8. Put high wattage resistors and power semiconductors away from the centre of the PCB. Keeping such components near edges facilitates putting up of heat sinks.
9. Keep a margin of at least ¼" on all sides to allow a space between board edges and actual components. This space can be used only for mounting of connectors, or PCB mounting screws.
10. Keep number of jumpers minimum.
11. Mount the components as close to each other as possible, leaving only minimum space required between to service the components. Typical distance between transistors and ICs will be around 1/8" or ¼".
12. For currents up to 1A, use track size of 1/16" or 1.5mm approx. For higher currents increase the track width approximately.
13. Spacing between tracks is a function of the voltage. Typical values are –
    0 – 150 V   1/32"

    150 – 300V   1/16"

    301 – 500V   1/8"

14. Make use of power and ground bus concepts to simplify the track layout.
15. Think of symmetry while arranging components. Try to spread the component weight evenly across PCB.
16. Try to place large and small components alternately.

GUDELINES FOR PREPARING COMPONENT LAYOUT.

1. Ensure that you have all the required components with you.
2. Note down accurate measurements of each component.
3. Show PCB mounting holes at the corners at 70mmX70mm.
4. Distribute the components evenly over the area. You need not mark this area. Only make sure that this entire area is used.
5. Ensure that heavier components are towards the centre.
6. Keep components with heat sinks, power resistors, ICs near centre.
7. All axial lead components should be placed either horizontally or vertically.(i.e. parallel to PCB edges)
8. Pads for external wire connection, or the components where external wire is terminated directly, should be towards the edges of PCB.
9. For axial lead component like resistor, distance between the mounting holes will be (body length in mm +6 mm) minimum.
10. While deciding the component position, think of connected components and place the components in such a way that path length between two connected components is minimum.
11. You need not accommodate transformer, power supply& signal source on the PCB. Provide pads to terminate external wire or you can solder the external wire directly onto the appropriate component lead. In such case take care to locate that component towards the edges of PCB.
12. Show at least the mounting holes for every component. Showing component out line (Shape & size) is preferred but not must.
    You will proceed to make track layout only after your component layout is corrected & approved.

## I. Preparation of layout using DIPTRACE

1. Open the DIPTRACE Software.

   START→ Programs→DipTrace→PCB lyout.

   Following window will be seen on the monitor.
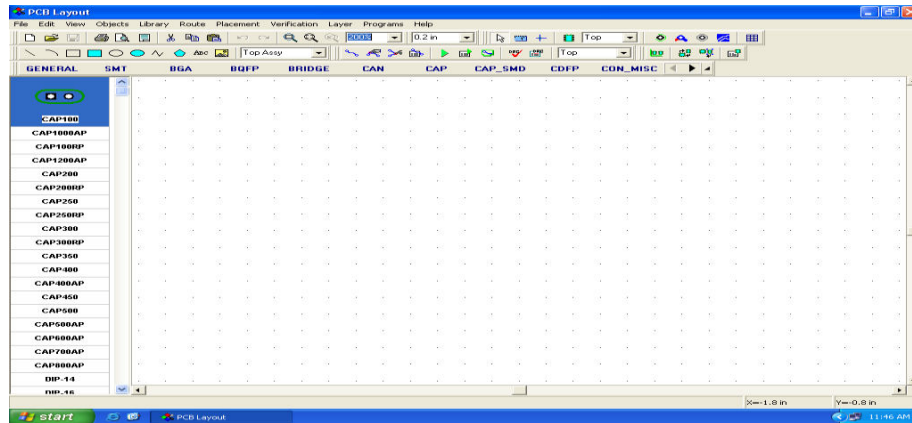
2. Standard components sizes such as resistors, capacitors, transistors and ICs with various package types, are available in the library of the DipTrace. The list of the components is displyed alphabetically on the left of the Diptrace layout window. Select, Drag and drop the required component as shown below. As the components are placed on the Top layer of the PCB, while placing the components, select the layer settings shown with arrows. The properties of the components such as rotation can be changed by selecting the component and right clicking.
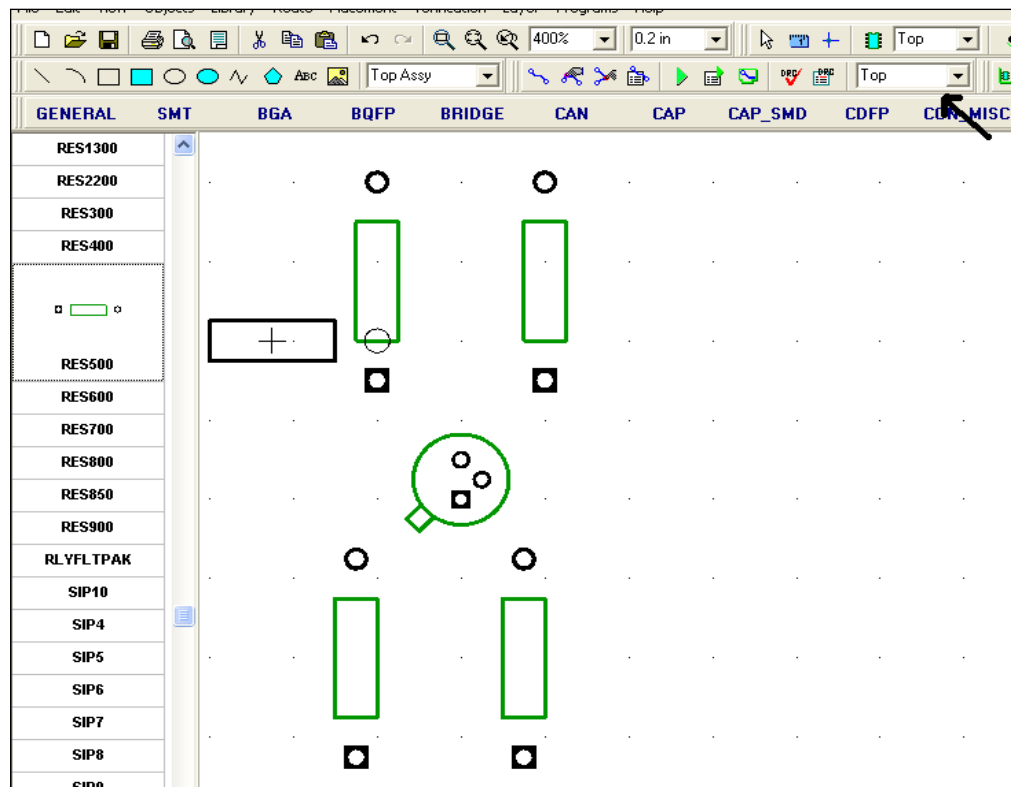

3. After the placement of the components. Select the bottom layer and make all the

31

connections using the traces available in Menu bar shown with arrows in the following figure. Following figure is a layout for the Transitorised common emitter Amplifier. The Text is used to name the components or its value on the top layer. The width of the traces etc can be changed by selecting the trace and right click

4. After the connections are complete click "Run Autorouter" button in the toolbar to form a block of the prepared lazz
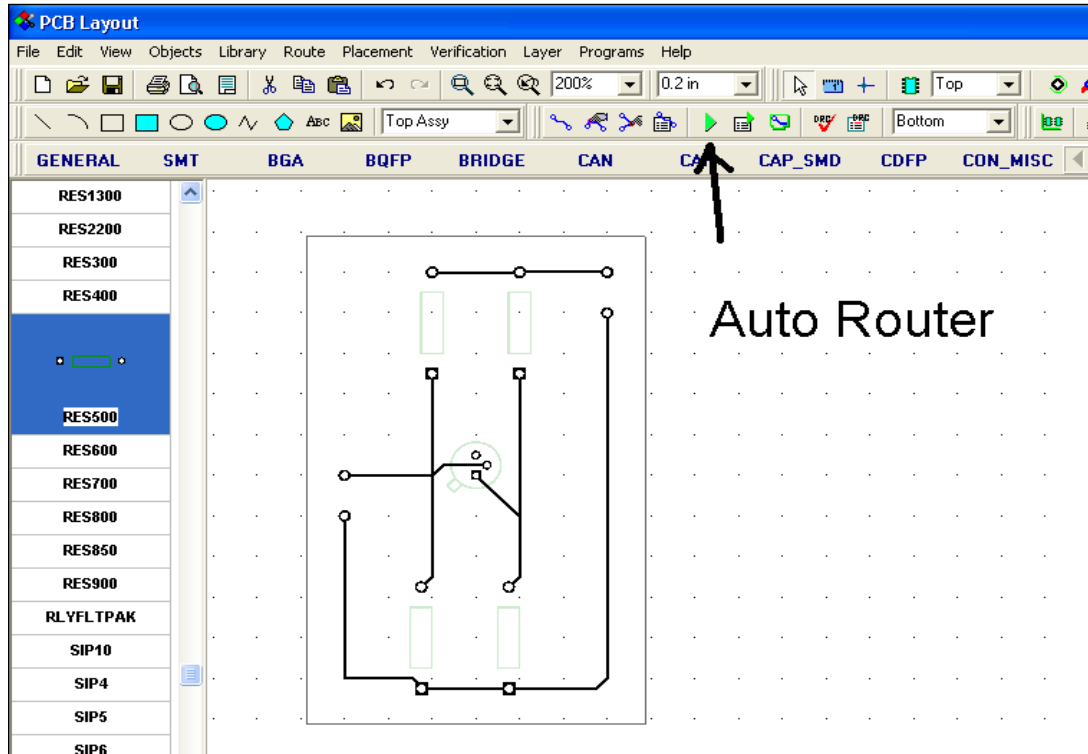
5. The photo chemical process of PCB manufacturing needs the negative of the layout. The Dip trace software can give the printout of the layout as it is (Positive) or negative of any layer. But we need the negative of the bottom layer. The following steps should be followed to get the print out.
In Dip Trace window click File→ Preview. The preview window will be opened only if the printer is connected to your PC. Use the laser printer only to take the printout of your layout negative. Do the settings in the preview window as shown below. The printout should be mirror image of the negative. Get the printout on the A$_4$ size Butter Paper. Depending on the darkness of the black Area which is actually the unwanted copper, the multiple printouts can be taken and can be overlapped.
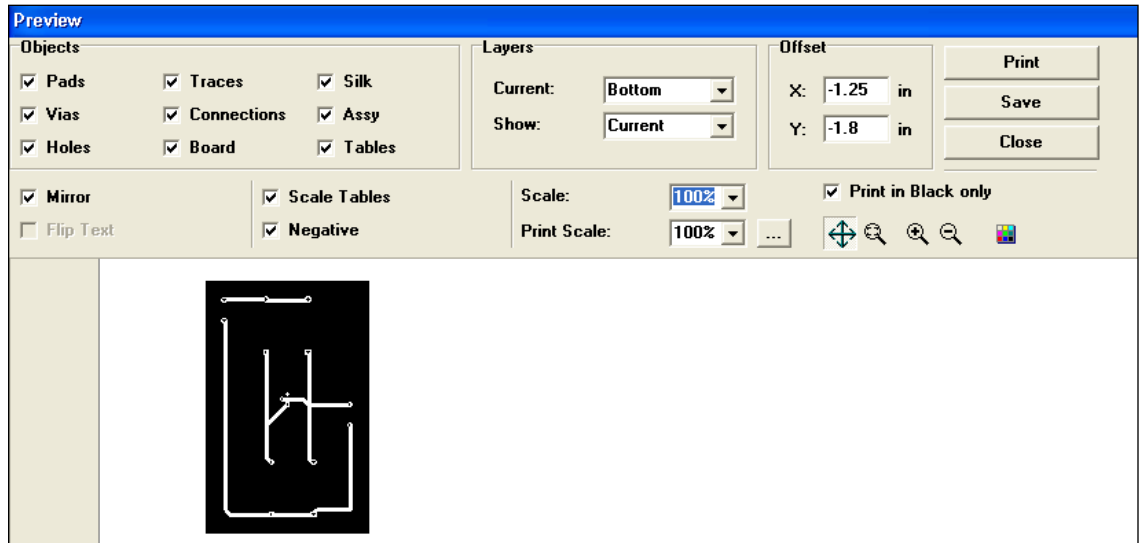
II FABRICATION OF PRINTED CIRCUIT BORD

- ➢ **Introduction –**
  The method of making your own Printed Circuit Board is described in the following guideline..

  Even if you have never fabricated a PCB before, with a little practice you will be able to produce PCB of professional grade for prototyping or even a finished product.

  The manual artwork method of PCB design is the method used by most manufacturers of PCBs. With this method, extremely accurate, high density printed circuit boards can be constructed, since the artwork can be produce 1, 2 or 4 times the size of the final PCB. This method consist of applying tapes, donuts pad, multi pad configurations (for integrated circuit) letters, numbers and words, to a sheet of Mylar film.

  If the artwork is done by computer aided design, please ensure that the output is on an accurate good quality laser, and not a dot matrix printer, as pinholes will lead to defects in the final PCB.

  There are commercial photo-sensitive resist, which are very practical for producing circuits on PCB. Placing the circuit design on a negative helps actual production of circuits. Exposing the negative while in contact with coated (photo sensitive) circuit board is done first. Then developing and other processes are required to form the circuit boards.

  There are durable photosensitive resist of organic. Solvent solutions which are Ultra Violet Light sensitive. These can be processed and used easily in laboratory.

  The resist may be applied in double coats in a dip coating machine. With a dip coater the resist is coated very accurately on the required PCB laminate and dried it is then exposed to contact with negative with the design to an actinic light source (U. V.). The non-exposed parts are dissolved by developing with the recommended developing solution.

This leaves the circuit resist on the required parts of the metal on the PCB. The non coated or unprotected metal may be etched in an etching solution such as ferric-chloride solution. After etching and washing the resist is removed with the required solvent and a very accurate circuit configuration is left on the circuit board.

While drilling the board be sure that drill bit is sharp. Best result for glass epoxy board material will be maintain with carbide tipped drill bit, but you can use an ordinary high speed steel (HSS) drill bit if that is all you have. For most component proper drill size is 1 mm. diameter and for integrated circuit .8mm is used. The specified drill sizes are not critical, but it is good if we don't use too large drill since it makes it more difficult to get a proper solder connection.

➢ **Process**
In photo transfer processes cleanliness is very important. This applies to all steps you follow to achieve the desired results. Whether you are making metal labels or Professional grade PCBs, please ensure that your work tables, apparatus, tanks, trays are completely free from dirt and dust.

Cleaning the Laminate-

Before the metal surface (in case of the bare PCB Copper laminate) is coated with photo resist, clean the surface thoroughly, so as to make it completely free of physical & chemical contamination .It is advisable to clean the metal by solvents like trichloroethylene to remove traces of grease or oil. Contaminants such as cupric-oxide, dirt etc can be removed by applying abrasives like pumic powder. Traces of such cleaning powder should then be removed by washing the metal plates under running water and scrubbing it with a soft brush or pad. The surface should then be quickly and thoroughly dried with forced warmed air.

Coating the Laminate-

Photo resist can be applied precisely and economically by using a dipping process, done by a Photo resist Dip Coater.

Dip coating is the best method of photo resist coating . Ti is used when the number of

Laminates to be coated is high and generally both the side of the laminates are to be coated. Stainless steel tank is used for storing the photo resist coating material. You have to dip or immerse the laminate in the tank containing photo resist and withdraw it slowly (withdrawal at rate of 30 to 40 cm/minutes) .Therefore the motorized arrangement for this operation is required.

It is difficult to predict a particular coating thickness needed for specific application because the coating methods and the equipments used change from user to user. It is therefore best to determine the optimum coating thickness experimentally.

Thinner

Use of thinner is to be done very carefully. The use of thinner will depend upon the method of coating you are following.(Photo resist's viscosity -12 secs FC4).The thinner

is used to dilute the photo resist , as it tends to get thick during non –use.

Pre-baking –

After coating the laminate, allow it to dry naturally for minimum 5 minutes. Tis helps to evaporate the solvents completely. If the coating is thick it is also desirable to bake the laminate in PCB curing machine with controlled temperature at 30º- 45º C for about 10 minutes.

All operation related to coating and pre-baking are to be carried out in safe (yellow) light

Exposure –

The coated laminate is normally exposed in contact with photographic negative or positive. Photo resist is sensitive to ultra violet radiations and therefore an Ultra-violet tubes based ,UV exposure unit can be used for exposure.

The time of exposure depends on many factors e.g. the source of light, thickness of coating, distance between the source of light and printing down frame etc.

The time of exposure depends on many factors e.g. the source of light and printing down

Frame etc.

For double side PCBs it is desirable to use double sided sandwich glass for printing frame
Development-

The exposed plate should be placed into the solvent –based developer. This will remove unexposed areas of the photo resist and will show a colorless resist image which has plastic-like appearance. The total development time is between 60-90 seconds.

The developer gets exhausted after continuous use and it is necessary to keep a periodical check on it. The exhausted developer creates scum in the non-image areas, When such

trouble occurs, developer should be replaced and the container of the old developer should be cleaned before pouring fresh developer in it.

Washing-

Immediately after development is over, wash it in running water on neutral PH. After washing and drying , dyeing is carried out to improve the visibility of the image and to find out any cracks or broken line and to facilitate the same for the purpose of retouching before etching.

Dying the image-

Immerse the dried plate in the dye developer bath. The surface of the board is covered with a dye-developer. The board is put under running water of neutral PH in order to wash off the dye stain from the unexposed area, which results into a clean dyed image of photo resist and dry.

A dyed image can show you pinholes in the image areas that may be due to unclean printed frame or dusty, negative or positive. Occasionally black spots appear in the resist areas. These may be due to foreign matters in the resist coating, which is a continuously used without filtration. It is better to repair these defects in the image area otherwise they may become the source of trouble in further process.

Etching-

This is an important and critical step in the chemical processing of the direct-etched

Boards. Although basically this operation aims at chemical, removal (etching) of unwanted copper portion (non-image area). The operation must bear in mind the important of the definition tolerance possible, under cutting and contamination of a board substrate due to enchants. Good results can be obtained by carefully studying the various aspects of the operation.

Ferric Chloride is one of the most widely used enchants for copper and copper alloys .

The main reason for its popularity is the low cost. Commercially it is available in lumps or ready to use solutions of different strengths.

Various methods are used while chemically removing the unwanted copper from the laminates.

A. In the proto-Etch Etching machine, the laminates are put in a tank filled with enchant agitated by a pump & a heater .The heater helps the enchant to become warm and helps to each PCB's faster. Fresh enchant is available to the surface

areas while the dissolved metal from the surface is rinsed away simultaneously .A sensor is provided for maintaining temperature.

B. IN the Oscillating spray etching machine, the technique of spraying the etchant with moving nozzles, has double side etching either in horizontal and vertical positioning to production is the greatest advantages of these machines but the cost of equipment is more.

Whatever the method you follow for etching the laminate, please do not forget to cleaned the etched boards after the etching is complete.

Resists Removal:

After the etching or plating job is complete it is desirable to remove the resist from the image area, photo resist stripper is used to remove the resist from PCB the cleaned surface should be washed with water and dried quickly.

Trouble Shooting:

Problem	Causes
1. Exposed coating has a spongy appearance or mat finish after development.	a) Inadequate exposure time b)Thick coating
2. Exposed coating peels off during development.	a) Inadequate exposure time b) Very thick coating c) Laminate surface not sufficient clean
3. Exposed coating does not develop time.	a) Pre-baking beyond 50°C for longer
4. Dyed- image does not look intense	a) Thin coating.

Protective Coating (Tinning):

Roller tinning unit is used for solder coating of PCB to provide an effective protection against corrosion and ensuring along storage life and improve Soldering properties. After cleaning & fluxing, the PCB to be solder coated is passed between two rotating rollers the one at the bottom being the tinned roller which partially dips into a solder bath and the top silicon rubber covered idle roller. The hot solder on the timed roller transfer on to the copper patterns of the PCB.

Drilling:

The drilling of PCB is done for making holes on the conductor pattern for inserting the component leads to the solder side of the PCB

Plated Through-Hole PCBs

Once your college already has a basic PCB making lab, without a plated- Thru-Hole PCB facility (PTH is now a commonly used PCB in industrial and consumer electronics.) the lab will be incomplete. Once your staff/ college is familiar with single sided PCB making, it may be a good time to upgrade to include a PTH Lab

The Micro – Plate Upgrade is a Prototype PTH PCB making module, ideally Suited for educational and R & D institutions. It includes all the equipment and start- up raw materials for fabrication of PTH PCB used in projects and even Commercial electronics.

III. <u>PCB Designing and Manufacturing guide lines</u>

1. **Circuit Diagram**

   Use: PCB layout, artwork, troubleshooting, user's manual and assembly.

   Information:

   a. All symbols drawn from the standard symbol drawing. which are updated as and when required.
   b. Component nos. (R1, C1 etc.) as per final component layout or component assembly.
   c. Component values with complete specifications.
   d. All pins, used or unused.
   e. Description of connectors.
   f. All test points (numbered).
   g. All components including decoupling capacitors and jumpers.
   h. Any other information. E.g. Jumper meaning. Port addresses etc.

Refer:　　　System specifications

2. Writing Diagram

Use:　　　Assembly, testing, Installation & troubleshooting.

Information:

a. Wiring related to physical parts.
b. Connector details as pin numbers and types.
c. Colour codes as of all wires, if any.
d. Wire sizes.
e. Ferrule – identification number and signal name.

Refer:　　Wiring diagram including wire length etc.

3. Component List

Use:　　　 R& D, testing, Formulation of parts list

Information:

a. Component number (one item written on one line) as on the circuit diagram as well as component layout or assembly diagram.
b. Complete description.
c. Manufacturer's item code/ Part no.
d. Remarks for alternatives or special instructions.

Refer:　　Circuit diagram, wiring Diagram, Component layout, Parts list (Bill of

Material), Structure diagram

4. Component Layout

Use:　　Assembly, Testing, Troubleshooting

Information:

a. All Components Number in Sequence either horizontal or vertical.
b. Back annotated to the circuit diagram.
c. Mechanical fixing if any.
d. PCB size and shape.
e. All Components outlines as per actual dimensions.

f.   Components numbers as per standard circuit symbol.
    g.   Silk screen (optional).

Refer: Circuit Diagram, Component layout (Bill of Material) and Drilling Details

    4. Parts List
    5. Use: Purchase, Production Planning and inventory control

Information:

    a.   Part No.
    b.   Components Number.
    c.   Full Description with make and options.
    d.   Supplier Name.
    e.   Quantity per sub-assembly or assembly, whichever preferable.
    f.   Location in stores
    g.   Remarks.

Refer: System Specifications

6. Drilling Details

Use: PCB Manufacturing

Information:

    a.   Drill size.
    b.   After or before plated through holes (PTH) normally after PTH.
    c.   Standardized color codes.
    d.   Viewing side.
    e.   Non-Plated through holes as mounting holes.
    f.   PCB number.
7. Artwork

Use: PCB Manufacturing

Information:

a. PCB number, normal or mirror as per the side of PCB.
b. Revision of artwork.
c. Component side, solder side or layer no.
d. Exact PCB dimensions in 1:1 scale.
e. The side that is matched while taking films.

Refer: Circuit Diagram, Component Layout, Drilling Details

8. Films

Use: PCB Manufacturing

Information:

a. PCB number, normal or mirror as per the side of PCB.
b. Revision of artwork.
c. Component side, solder side or layer no.
d. Exact PCB dimensions in 1:1 scale.
e. The side that is matched while taking films.

Refer: Circuit Diagram, Component Layout, Drilling Details

9. Block Diagram

Use: Explanation of complete system

Information: Block diagram of complete system, explaining all functional aspects of   the system and its blocks

Refer: System specifications.