LAB MANUAL

EC1002 -ELECTRONICS ENGINEERING PRACTICES

Prepared By

Dr. P. Eswaran

Department of Electronics and Communication Engineering

Faculty of Engineering and Technology,

SRM University,

Kattankulathur – 603203.

2014-2015

Department of Electronics and Communication Engineering

Program: B. Tech. Electronics and Communication Engineering

Expected learning outcome of

EC1002 Electronics Engineering Practices

Program Educational Objective → / **Student Outcomes** ↓	**PEO1:** Graduates will perform as a successful Professional engineer in related fields of Electronics and Communication Engineering.	**PEO2:** Graduates will pursue higher education and/or engage themselves in continuous professional development to meet global standards.	**PEO3:** Graduates will work as a team in diverse fields and gradually move into leadership positions.	**PEO4:** Graduates will understand current professional issues, apply latest technologies and come out with innovative solutions for the betterment of the nation and society.
a	√	√		
b	√	√		
c	√			√
k	√	√		√

Mapping of instructional objectives with student Outcomes

INSTRUCTIONAL OBJECTIVE	STUDENT OUTCOME	JUSTIFICATION
1. To familiarize the electronic components and basic electronic instruments.	(a) an ability to apply knowledge of mathematics, science, and engineering	The students will apply knowledge of mathematics, science, and engineering to conduct experiments using standard instrument and equipments.
	k) An ability to use the techniques, skills and modern engineering tools necessary for engineering practice.	The students are taught to design and simulate various electronic circuits using electronic components.
2. To make familiar with PCB design and various processes involved.	(a) an ability to apply knowledge of mathematics, science, and engineering	The students will apply knowledge of engineering to design and conduct experiments using PCB design software..
	c) an ability to design a system, component, or process to meet desired needs within realistic constraints such as economic, environmental, social, political, ethical, health and safety, manufacturability, and sustainability	The students will have the ability to identify, formulate, and solve engineering problems related to PCB design and generate manufacturing files.
	k) An ability to use the techniques, skills and modern engineering tools necessary for engineering practice.	The students are taught to design and simulate various PCB circuits using industry standard PCB design software tools like Orcad, Power PCB and TINA packages.

INSTRUCTIONAL OBJECTIVE	STUDENT OUTCOME	JUSTIFICATION
3. To provide in-depth core knowledge in the fabrication of Printed	(a) an ability to apply knowledge of mathematics, science, and engineering	The students will apply knowledge of engineering to design and conduct experiments using designed circuits.
	b) An ability to design and conduct experiments as well as to analyze and interpret data.	The students will have the ability to conduct experiments using designed and assembled circuits
	c) an ability to design a system, component, or process to meet desired needs within realistic constraints such as economic, environmental, social, political, ethical, health and safety, manufacturability, and sustainability	The students will have the ability to identify, formulate, and solve engineering problems associated with assembly and testing of electronic circuits
	k) An ability to use the techniques, skills and modern engineering tools necessary for engineering practice.	The students are taught to design and simulate various electronic PCB required for prototyping and testing using software tools and testing equipments.
4. To provide the knowledge in assembling and testing of the PCB based electronic circuits.	b) An ability to design and conduct experiments as well as to analyze and interpret data.	The students will apply knowledge of engineering to design and conduct experiments using designed circuits.
	c) an ability to design a system, component, or process to meet desired needs within realistic constraints such as economic, environmental, social, political, ethical, health and safety, manufacturability, and sustainability	The students will have the ability to identify, formulate, and solve engineering problems associated with assembly and testing of electronic circuits
	k) An ability to use the techniques, skills and modern engineering tools necessary for engineering practice.	The students are taught to design and simulate various electronic PCB required for prototyping and testing using software tools and testing equipments.

Syllabus

EC1002 Electronic Engineering Practices

UNIT I-INTRODUCTION TO BASICS OF ELECTRONIC COMPONENTS AND INSTRUMENTS (4 hours)

Study of electronic components- active & passive, Electronic Instruments: CRO, Function generator, Power Supply, Multi-meter, IC tester. Solder practice.

UNIT II -SCHEMATIC CAPTURE (6 hours)

Introduction to OrCAD schematic capture tool, Simulation of simple electronic circuit, Schematic to layout transfer, Layout Printing.

UNIT III-PCB DESIGN PROCESS (6 hours)

Conception Level Introduction: Specifying Parts, Packages and Pin Names, Libraries and Checking foot prints of the components, Partlist, Netlist, Making Netlist Files, Placing Parts, Routing Traces, Modifying Traces, Mounting Holes, Adding Text, PCB Layout, DRC, Pattern Transfer.

UNIT IV-PCB FABRICATION PROCESS (6 hours)

Etching, cleaning, drying and drilling.

UNIT V-ASSEMBLING AND TESTING (8 hours)

Identifying the components and its location on the PCB, soldering of active and passive components, Testing the assembled circuit for correct functionality.

REFERENCES

1. Department Laboratory Manual.
2. Orcad User manual.
3. Printed Circuit Boards: Design, Fabrication, and Assembly (McGraw-Hill Electronic Engineering-2006) by Raghbir Singh Khandpur.

Academic Course Description

SRM University
Faculty of Engineering and Technology
Department of Electronics and Communication Engineering

EC1002 - Electronics Engineering Practices

Second Semester, 2014-15

Course Description

To provide hand-on experience in PCB Circuit design using software and to familiarize with PCB Fabrication process. To provide hands on experience in assembly and Testing of electronics circuit.

Compulsory/Elective course: Compulsory for ECE students

Credit hours: 1 credits

Venue: Basic Engineering Laboratory 2nd Floor BEL307 & BEL308, Electronics Engineering Practices Laboratory

Course coordinator: Mr. P.Eswaran

Instructor(s)

Name of the instructor	Class	Venue	Day order Class hours	Email (domain: @ktr.srmuniv.ac.in)	Consultations
Mrs. G.Suganthi Brinda Mrs. S. Hannah Pauline	A	Workshop-Electronics Engineering practices Laboratory 2nd Floor Basic Engineering Workshop Building	Day-2 1st and 2nd	suganthibrindha.g hannahpauline.s	1.00 pm-1.30 pm
Mrs. Kavitha Narayanan Ms. S. Suhashini	B		Day-1 5th and 6th	kavitha.n suhasini.s	1.00 pm-1.30 pm
Ms. S. Sudarvizhi Mr. B. Ananda venkatesan	C		Day-2 3rd and 4th	sudarvizhi.s anandavenkatesan.b	1.00 pm-1.30 pm
Mrs.V.PadmaJothi Ms.A Ramya	D		Day-2 6th and 7th	padmajothi.v ramya.a	1.00 pm-1.30 pm
Mrs. Rashmitha Routary Mr. P.Eswaran	E		Day-3 3rd and 4th	rashmita.r eswaran.p	1.00 pm-1.30 pm
Mrs. C. Vimala Ms. A. Bhavani	F		Day-3 5th and 6th	vimala.c bhavani.a	1.00 pm-1.30 pm

Name of the instructor	Class	Venue	Day order Class hours	Email (domain: @ktr.srmuniv.ac.in)	Consultations
Mrs. S. Murugaveni Mrs.S. Diana Emarald Aasha	G		Day-3 3^{rd} and 4^{th}	murugaveni.s dianaemeralaasha.s	1.00 pm-1.30 pm
Mr.M. Ramchandran Mrs. S. Latha	H		Day-3 1^{st} and 2^{nd}	ramachandran.md latha.k	1.00 pm-1.30 pm
Mr.M. Ramchandran Mr.T. Saminathan	I		Day-5 3^{rd} and 4^{th}	ramachandran.md saminathan.t	1.00 pm-1.30 pm
Mr. G. Elavel Viswanathan Mr.S. Vijayananth(ITCE)	J		Day-3 1^{st} and 2^{nd}	elavelvisuvanathan.g vijayananth.s	1.00 pm-1.30 pm
Mr. S. Vijayananth(ITCE) Mr. B. Ananda Venkatesan,	Bio Medical		Day-1 1^{st} and 2^{nd}	vijayananth.s anandavenkatesan.b	1.00 pm-1.30 pm

Relationship to other courses

Pre-requisites: Nil

Assumed knowledge:

Basic Electronic Components
Tools used for assembly
Procedure to handle equipments and instruments
Basic knowledge to use computer

Following courses:

Text book(s) and/or required materials:

Lab manual; additional materials posted on SRM web.

References

1. Department user Lab manual.
2. ORCAD.PADS software User manual.
3. R S Khandpur, Printed Circuit Boards- Design Fabrication, Assembly and Testing, Tata Mc Graw Hill Publishing Company Limited, Ist edition 2008.

Computer usage

OrCAD Pspice and Power logic and Power PCB for schematic and CAM File generation for PCB.

Hardware Laboratory Usage

Each laboratory station is equipped with a Power supply, CRO, Function generator, Digital Multimeter, Hand tool kit for circuit assembly, soldering station, components and PCBs. Students work in groups of two, but maintain individual laboratory notebooks and submit individual reports.

Detailed Session Plan

Ex. No.	Lab Exercises	Lab Sessions	Correlation of lab exercises with Objectives	
			IOs	SOs
1	Study of Electronic Components	1	1	a,k
2	Study of Instruments and Equipments (DMM, Power supply, CRO, FG)	2		
	Introduction to PCB Design software	3	2,4	a,b,c,k
3	Generation of CAM Files for single side PCB (Measuring voltage Drop)	4		
	Generation of CAM Files for single side PCB (Measuring voltage Drop)	5		
4	Generation of CAM Files for single side PCB (Full wave Rectifier)	6		
	Generation of CAM Files for single side PCB (Full wave Rectifier)	7		
5	Introduction to soldering practice	8	1,3,4	a,c,k
	PCB Assembly and Testing (Measuring voltage Drop)			
	PCB Assembly and Testing (Measuring voltage Drop)	9		
6	PCB Assembly and Testing (Full wave Rectifier)	10		
	PCB Assembly and Testing (Full wave Rectifier)	11		
7	Study of single side PCB Fabrication process	12	3,4	a,b,c,k
	Study of single side PCB Fabrication process	13		
	Study of single side PCB Fabrication process	14		
	Repeat class	15		
	University Practical Exam	16		

Evaluation methods

Attendance	-	5 Marks
In-lab experiment	-	15 Marks
Lab Report	-	15 Marks
Viva voce	-	10 Marks
Model exam	-	15 Marks
Final exam	-	40 Marks

Laboratory Policies and Report Format

Reports are due at the beginning of the lab period. The reports are intended to be a complete documentation of the work done in preparation for and during the lab. The report should be complete so that someone else familiar with digital design could use it to verify your work. The prelab and postlab report format is as follows:

1. A neat thorough prelab must be presented to your faculty incharge at the beginning of your scheduled lab period. **Lab reports should be submitted on A4 paper.** Your report is a professional presentation of your work in the lab. Neatness, organization, and completeness will be rewarded. Points will be deducted for any part that is not clear.

2. In this laboratory students will work in teams of three. However, the lab reports will be written individually. Please use the following format for your lab reports.
    a. **Cover Page:** Include your name, Subject Code, Section No., Experiment No. and Date.
    b. **Objectives:** Enumerate 3 or 4 of the topics that you think the lab will teach you. DO NOT REPEAT the wording in the lab manual procedures. There should be one or two sentences per objective. Remember, you should write about what you will learn, not what you will do.
    c. **Design:** This part contains all the steps required to arrive at your final circuit. This should include diagrams, tables, equations, K-maps, explanations, etc. Be sure to reproduce any tables you completed for the lab. **This section should also include a clear written description of your design process.** Simply including a circuit schematic is not sufficient.
    d. **Questions:** Specific questions (Prelab and Postlab) asked in the lab should be answered here. **Retype the questions presented in the lab and then formally answer them.**

3. Your work must be original and prepared independently. However, if you need any guidance or have any questions or problems, please do not hesitate to approach your faculty incharge during office hours. Copying any prelab/postlab will result in a grade of 0. The incident will be formally reported to the University and the students should follow the dress code in the Lab session.

4. Each laboratory exercise (circuit) must be completed and demonstrated to your faculty incharge in order to receive working circuit credit. This is the procedure to follow:
    a. Circuit works: If the circuit works during the lab period (3 hours), call your faculty incharge, and he/she will sign and date it.. This is the end of this lab, and you will get a complete grade for this portion of the lab.
    b. Circuit does not work: If the circuit does not work, you must make use of the open times for the lab room to complete your circuit. When your circuit is ready, contact your faculty incharge to set up a time when the two of you can meet to check your circuit.

5. Attendance at your regularly scheduled lab period is required. An unexpected absence will result in loss of credit for your lab. If for valid reason a student misses a lab, or makes a reasonable request in advance of the class meeting, it is permissible for the student to do

the lab in a different section later in the week if approved by the faculty incharge of both the sections. Habitually late students (i.e., students late more than 15 minutes more than once) will receive 10 point reductions in their grades for each occurrence following the first. **Student attendance less than 75% is detention**.

6. Final grade in this course will be based on laboratory assignments. All labs have an equal weight in the final grade. Grading will be based on pre-lab work, laboratory reports, post-lab and in-lab performance (i.e., completing lab, answering laboratory related questions, etc.,).The faculty Incharge will ask pertinent questions to individual members of a team at random. Labs will be graded as per the following grading policy:

Attendance	-	5 Marks
In-lab experiment	-	15 Marks
Lab Report	-	15 Marks
Viva voce	-	10 Marks
Model exam	-	15 Marks
Final exam	-	40 Marks

7. **Reports Due Dates**: Reports are due one week after completion of the corresponding lab.

8. **Systems of Tests:** Regular laboratory class work over the full semester will carry a weightage of 60%. The remaining 40% weightage will be given by conducting an end semester practical examination for every individual student if possible or by conducting a 1 to 1 ½ hours duration common written test for all students, based on all the experiment carried out in the semester.

9. General Procedures

a. **Handout:** Class representatives are advised to collect the relevant handout from their faculty.
b. **Tools:** Students has to borrow the required tools and components from store.
c. **Attendance:** Minimum of 75% to be maintained
d. **Punctuality:** Late entry to the lab Not permitted.
e. **Dress Code:** Should wear lab coat and shoe, **No slippers, sandals or footwear that exposes the foot is allowed**. Students with long hair should have it tied up.
f. **Conduct:** No eating or drinking is allowed. Unnecessary roaming around the lab to be avoided. Noise level is to be kept to the absolute minimum.
g. **Safety:** Students are to observe safety regulations at all times.
h. **Equipment Usage:** All mains and electrical equipment are to be **switched off when not in use** or when the lab session ends.
i. **General House Keeping:** Students should keep their work station neat and clean.

Prepared by : Dr. P. Eswaran

Dated: 05/01/2015

Revision No.: 00 **Date of revision:** NA **Revised by:**

Program Educational Objectives

(i) To prepare students to compete for a successful career in Electronics and Communication Engineering profession through global education standards.

(ii) To enable the students to aptly apply their acquired knowledge in basic sciences and mathematics in solving Electronics and Communication Engineering problems.

(iii) To produce skillful graduates to analyze, design and develop a system/component/ process for the required needs under the realistic constraints.

(iv) To train the students to approach ethically any multidisciplinary engineering challenges with economic, environmental and social contexts

(v) To create awareness among the students about the need for life long learning to succeed in their professional career as Electronics and Communication Engineers.

Program Outcomes

a. an ability to apply knowledge of mathematics, science, and engineering
b. an ability to design and conduct experiments, as well as to analyze and interpret data
c. an ability to design a system, component, or process to meet desired needs within realistic constraints such as economic, environmental, social, political, ethical, health and safety, manufacturability, and sustainability
d. an ability to function on multidisciplinary teams
e. an ability to identify, formulate, and solve engineering problems
f. an understanding of professional and ethical responsibility
g. an ability to communicate effectively
h. the broad education necessary to understand the impact of engineering solutions in a global, economic, environmental, and societal context
i. a recognition of the need for, and an ability to engage in life-long learning
j. a knowledge of contemporary issues
k. an ability to use the techniques, skills, and modern engineering tools necessary for engineering practice.

Name of the instructor	Class/ Section	Email (domain: @ktr.srmuniv.ac.in)	Signature
Mrs. G.Suganthi Brinda	ECE -A	suganthibrindha.g	
Ms. S. Suhashini	ECE-B	suhasini.s	
Ms. S. Sudarvizhi	ECE-C	sudarvizhi.s	
Mrs.V.PadmaJothi	ECE-D	padmajothi.v	
Mr. P.Eswaran	ECE-E	eswaran.p	
Ms. A. Bhavani	ECE-F	bhavani.a	
Mrs. S. Murugaveni	ECE-G	murugaveni.s	
Mr.M. Ramchandran	ECE-H	ramachandran.md	
Mr.M. Ramchandran	ECE-I	ramachandran.md	
Mr. G. Elavel Viswanathan	ECE-J	elavelvisuvanathan.g	
Mr. B. Ananda Venkatesan,	Bio Medical	anandavenkatesan.b	

Course Coordinator	**Academic Coordinator**	**Professor In-Charge**	**HOD/ECE**
(Dr. P. Eswaran)	(Mrs. R. Monahari)	(Dr. Shanthi Prince)	(Dr. S. Malarvizhi)

List of Experiments

1. Study of Electronic Components
2. Study of Instruments and Equipments (DMM, Power supply, CRO, FG)
3. Generation of CAM Files for single side PCB (Measuring voltage Drop)
4. Generation of CAM Files for single side PCB (Full wave Rectifier)
5. PCB Assembly and Testing (Measuring voltage Drop)
6. PCB Assembly and Testing (Full wave Rectifier)
7. Study of single side PCB Fabrication process by photo resist Method

Laboratory Report Cover Sheet

SRM University
Faculty of Engineering and Technology
Department of Electronics and Communication Engineering

EC1002 Workshop- Electronics Engineering Practices

First Semester, 2014-15 (Even semester)

Name :

Register No. :

Day / Session :

Venue : 2th Floor, BEL307 & BEL308, Engineering Basic Lab Building.

Title of Experiment :

Date of Conduction :

Date of Submission :

Particulars	Max. Marks	Marks Obtained
Design / Assembly / Testing/ Fabrication	15	
Lab Performance and Lab report	15	
Viva voce	10	
Total	**40**	

REPORT VERIFICATION

Date :

Staff Name :

Signature :

EXERCISE NO 1

STUDY OF ELECTRONIC COMPONENTS

OBJECTIVES

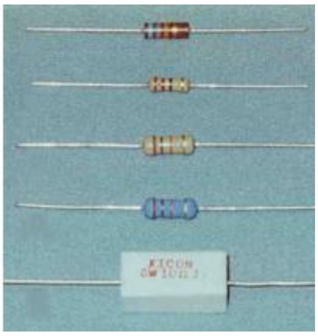
a. To get familiar with basic electronic components such as Resistor, capacitors, Inductor, diodes, transistors, integrated circuits (IC), light emitter diode (LED), switches, fuses, batteries, power plugs, connectors, wires and cables.
b. To test and understand the function of various electronic components.

RESISTORS

A resistor is a passive two-terminal electrical component that implements electrical resistance as a circuit element. The current through a resistor is in direct proportion to the voltage across the resistor's terminals. This relationship is represented by Ohm's law. A device used in electrical circuits to maintain a constant relation between current flow and voltage. Resistors are used to step up or lower the voltage at different points in a circuit and to transform a current signal into a voltage signal or vice versa, among other uses. The electrical behaviour of a resistor obeys Ohm's law for a constant resistance; however, some resistors are sensitive to heat, light, or other variables.

Resistors are one of the most used components in a circuit. Most are color coded, but some have their value in Ohms and their tolerance printed on them. A multimeter that can check resistance can also be helpful, providing the resistor is already removed from the board (measuring it while still soldered in can give inaccurate results, due to connections with the rest of the circuit). They are typically marked with an “R” on a circuit board.





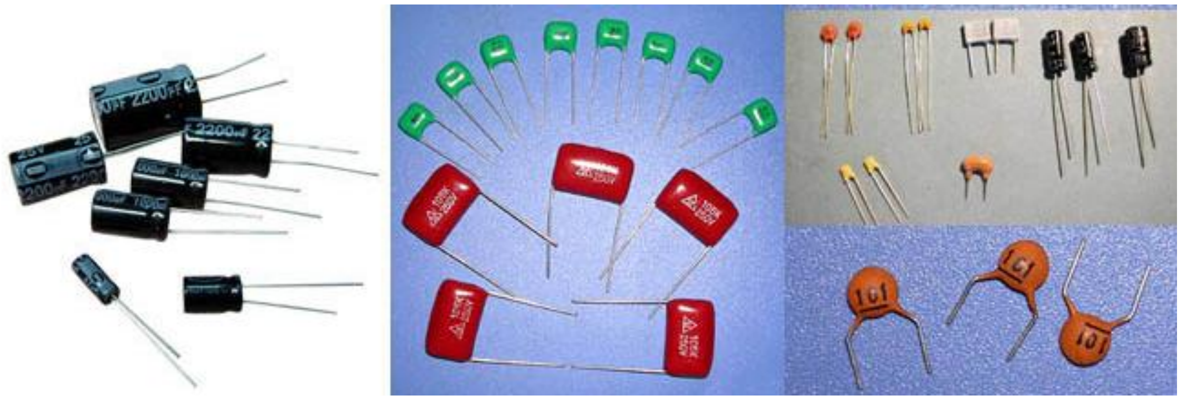
POTENTIOMETERS

Potentiometers are variable resistors. They normally have their value marked with the maximum value in Ohms. Smaller trimpots may use a 3-digit code where the first 2 digits are significant, and the 3rd is the multiplier (basically the number of 0′s after the first 2 digits). For example, code 104 = 10 followed by four 0′s = 100000 Ohms = 100K Ohms. They may also have a letter code on them indicating the taper (which is how resistance changes in relation to how far the potentiometer is turned). They are typically marked with an “VR” on a circuit board.

CAPACITORS

A capacitor (originally known as a condenser) is a passive two-terminal electrical component used to store energy electrostatically in an electric field. By contrast, batteries store energy via chemical reactions. The forms of practical capacitors vary widely, but all contain at least two electrical conductors separated by a dielectric (insulator); for example, one common construction consists of metal foils separated by a thin layer of insulating film. Capacitors are widely used as parts of electrical circuits in many common electrical devices.

Capacitors are also very commonly used. A lot have their values printed on them, some are marked with 3-digit codes, and a few are color coded. The same resources listed above for resistors can also help you identify capacitor values. They are typically marked with an “C” on a circuit board.



INDUCTORS

An inductor, also called a coil or reactor, is a passive two-terminal electrical component which resists changes in electric current passing through it. It consists of a conductor such as a wire, usually wound into a coil. When a current flows through it, energy is stored in a magnetic field in the coil. When the current flowing through an inductor changes, the time-varying magnetic field induces a voltage in the conductor, according to Faraday’s law of electromagnetic induction, which by Lenz's law opposes the change in current that created it.

Inductors, also called coils, can be a bit harder to figure out their values. If they are color coded, the resources listed for resistors can help, otherwise a good meter that can measure inductance will be needed. They are typically marked with an “L” on a circuit board.

TRANSFORMERS

A transformer is a static electrical device that transfers energy by inductive coupling between its winding circuits. A varying current in the primary winding creates a varying magnetic flux in the transformer's core and thus a varying magnetic flux through the secondary winding. This varying magnetic flux induces a varying electromotive force (emf) or voltage in the secondary winding.

Transformers are normally pretty easy to identify by sight, and many have their specs printed on them. They are typically marked with an “T” on a circuit board.

FUSES

In electronics and electrical engineering, a fuse is a type of low resistance resistor that acts as a sacrificial device to provide overcurrent protection, of either the load or source circuit. Its essential component is a metal wire or strip that melts when too much current flows, which interrupts the circuit in which it is connected. Short circuit, overloading, mismatched loads or device failure are the prime reasons for excessive current. A fuse interrupts excessive current (blows) so that further damage by overheating or fire is prevented.

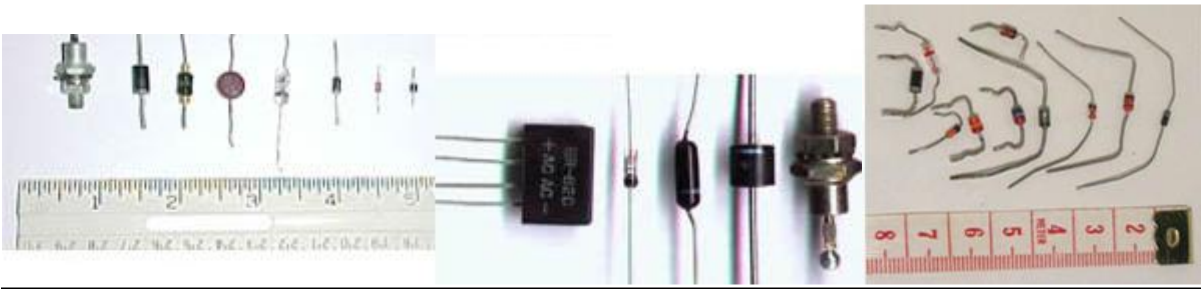
Fuses can be easy to identify, and typically have their voltage and amperage rating marked on them.

SEMICONDUCTORS

DIODES

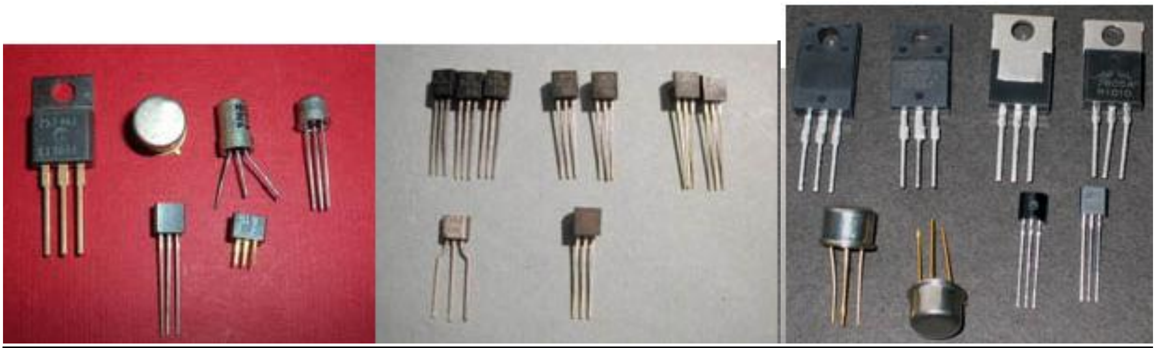
In electronics, a diode is a two-terminal electronic component with asymmetric conductance, it has low (ideally zero) resistance to current flow in one direction, and high (ideally infinite) resistance in the other.

Semiconductors, such as Diodes (typically marked with an “D” on a circuit board).



TRANSISTORS

A transistor is a semiconductor device used to amplify and switch electronic signals and electrical power. It is composed of semiconductor material with at least three terminals for connection to an external circuit. A voltage or current applied to one pair of the transistor's terminals changes the current through another pair of terminals. Because the controlled (output) power can be higher than the controlling (input) power, a transistor can amplify a signal. Today, some transistors are packaged individually, but many more are found embedded in integrated circuits.
Transistors (typically marked with an “Q” on a circuit board).



BRIDGE RECTIFIERS

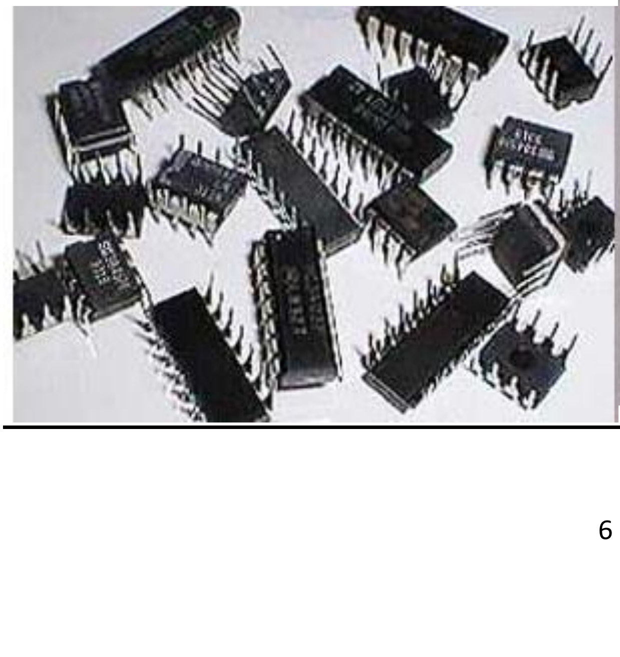
A diode bridge is an arrangement of four (or more) diodes in a bridge circuit configuration that provides the same polarity of output for either polarity of input. When used in its most common application, for conversion of an alternating current (AC) input into a direct current (DC) output, it is known as a bridge rectifier. A bridge rectifier provides full-wave rectification from a two-wire AC input, resulting in lower cost and weight as compared to a rectifier with a 3-wire input from a transformer with a center-tapped secondary winding.

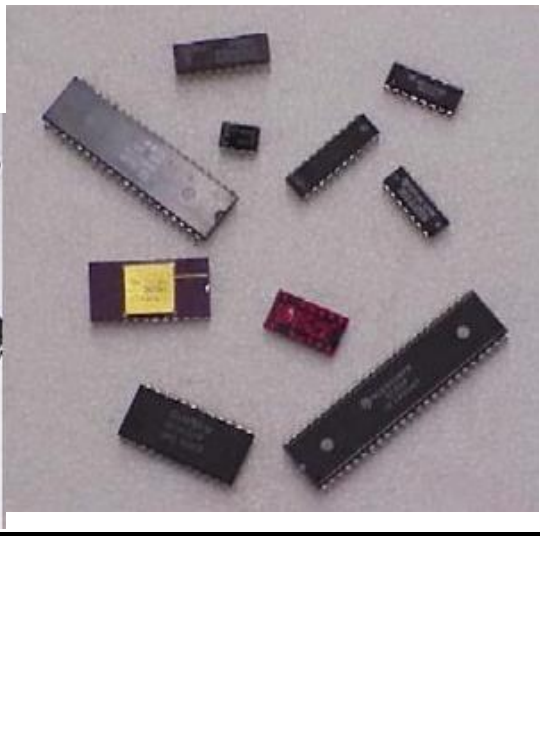
Bridge Rectifiers (typically marked with an “BR” on a circuit board)

INTEGRATED CIRCUITS

An integrated circuit or monolithic integrated circuit (also referred to as an IC, a chip, or a microchip) is a set of electronic circuits on one small plate ("chip") of semiconductor material, normally silicon. This can be made much smaller than a discrete circuit made from independent components. Integrated circuits are used in virtually all electronic equipment today and have revolutionized the world of electronics. Computers, mobile phones, and other digital home appliances are now inextricable parts of the structure of modern societies, made possible by the low cost of producing integrated circuits.

Integrated Circuits (typically marked with an “U” or “IC” on a circuit board)

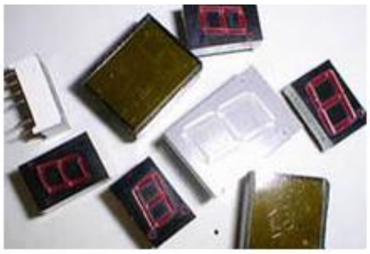




LED AND LED DISPLAY

A light-emitting diode (LED) is a semiconductor light source. LEDs are used as indicator lamps in many devices and are increasingly used for other lighting. LEDs emitted low-intensity red light, but modern versions are available across the visible, ultraviolet, and infrared wavelengths, with very high brightness.





SWITCHES

In electrical engineering, a switch is an electrical component that can break an electrical circuit, interrupting the current or diverting it from one conductor to another. The most familiar form of switch is a manually operated electromechanical device with one or more sets of electrical contacts, which are connected to external circuits. Each set of contacts can be in one of two states: either "closed" meaning the contacts are touching and electricity can flow between them, or "open", meaning the contacts are separated and the switch is nonconducting.

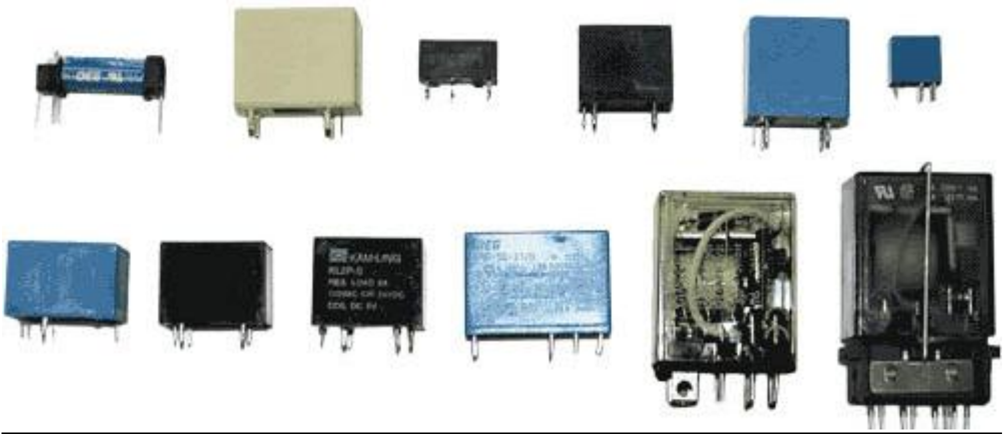
BATTERIES

In electricity, a battery is a device consisting of one or more electrochemical cells that convert stored chemical energy into electrical energy.
Batteries are also pretty easy to identify, and are well marked with their specification.

RELAYS

A relay is an electrically operated switch. Many relays use an electromagnet to operate a switching mechanism mechanically, but other operating principles are also used. Relays are used where it is necessary to control a circuit by a low-power signal (with complete electrical isolation between control and controlled circuits), or where several circuits must be controlled by one signal.

Relays are typically enclosed in plastic, and many have their specs printed on them. They are typically marked with a “K” on a circuit board.



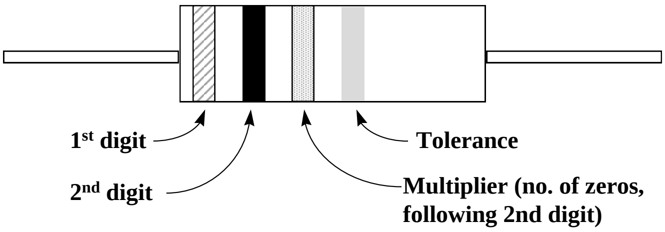
LAB ACTIVITY

EQUIPMENT REQUIRED

Electronic components (Resistor, capacitors, diode, Transistor)
Digital multimeter (DMM)

A1. Determining Resistor values

Resistor Colour Codes



Colour-code bands on a resistor.

Colour	Band 1	Band 2	Band 3	Band 4
Black	0	0	×1	–
Brown	1	1	×10	±1%
Red	2	2	×100	±2%
Orange	3	3	×1000	–
Yellow	4	4	×10000	–
Green	5	5	×100000	±0.5%
Blue	6	6	×1000000	±0.25%
Violet	7	7	×10000000	±0.1%
Grey	8	8	–	–
White	9	9	–	–
Gold	–	–	×0.1	±5%
Silver	–	–	×0.01	±10%

Resistance measurement

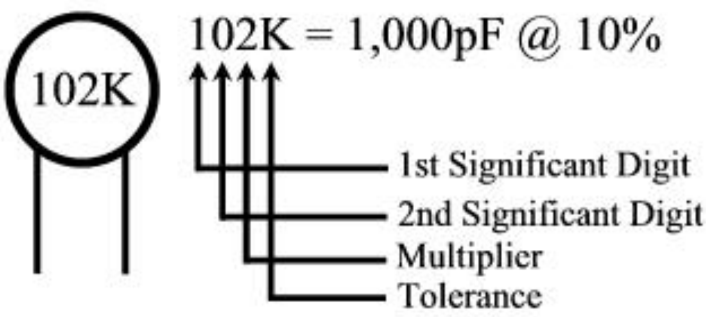
Procedure

1. Connect probes: black probe to COM terminal and red probe to terminal marked with 'Ω'
2. Set function to resistance measurement
3. Set to the appropriate range (refer to above)
4. Connect the two probes' crocodile clips to the resistor (or to the resistor circuit via jumper wires) to make measurement
5. Note the reading, adjust range if necessary
6. Take the more accurate reading.

Determine the value for the given data

No.	Colour code	Actual Value	Measured Value (DMM)
1	Red, red, black		
2	Red, black, orange		
3	Blue, gray, green		
4		10M	
5		33K	

A2. Determining capacitor values



Code	Tolerance
C	±0.25pF
J	±5%
K	±10%
M	±20%
D	±0.5pF
Z	80% / -20%

Determine the value of the ceramic capacitors

No.	Code Number	Actual Value
1	104	
2	223	
3	68	
4		0.47 μ F
5		33nF

A3. Diode Testing

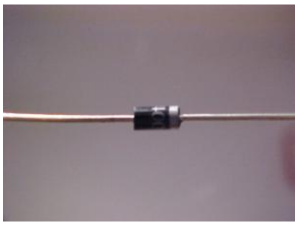
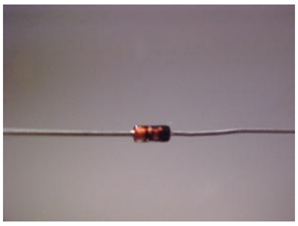
Draw the schematic symbol of a 1N4001 diode and identify the leads (Anode and Cathode) in the box below.

a. Set the Lab DMM to *Diode Testing* mode.

b. Measure the forward and reverse bias voltages of the given diodes and record them

i. Forward bias voltage:
Place the RED probe on the Anode. Touch the BLACK probe to the Cathode and record the reading

ii. Reverse bias voltage:
Place the RED probe on the Cathode. Touch the BLACK probe to the Anode and record the reading

No.	Diode Number	Forward Bias Voltage	Reverse Bias Voltage
1	1N4001 		
2	1N914 		

Result:

Laboratory Report Cover Sheet

SRM University Faculty of Engineering and Technology Department of Electronics and Communication Engineering
EC1002 Workshop- Electronics Engineering Practices **First Semester, 2014-15 (Even semester)**

Name :

Register No. :

Day / Session :

Venue : 2th Floor, BEL307 & BEL308, Engineering Basic Lab Building.

Title of Experiment :

Date of Conduction :

Date of Submission :

Particulars	**Max. Marks**	**Marks Obtained**
Design / Assembly / Testing/ Fabrication	15	
Lab Performance and Lab report	15	
Viva voce	10	
Total	**40**	

REPORT VERIFICATION

Date :

Staff Name :

Signature :

EXPERIMENT NO 2

2A DIGITAL MULTI METER

INTRODUCTION

A Multimeter is an electronic device that is used to make various electrical measurements, such as AC and DC voltage, AC and DC current, and resistance. It is called a Multimeter because it combines the functions of a voltmeter, ammeter, and ohmmeter. Multimeter may also have other functions, such as diode test, continuity test, transistor test, TTL logic test and frequency test.

PARTS OF MULTIMETER

A Multimeter has three parts:

- Display
- Selection Knob
- Ports

The **display** usually has four digits and the ability to display a negative sign. A few multimeters have illuminated displays for better viewing in low light situations.

The **selection knob** allows the user to set the multimeter to read different things such as milliamps (mA) of current, voltage (V) and resistance (Ω).

Two probes are plugged into two of the **ports** on the front of the unit. **COM** stands for common and is almost always connected to Ground or '-' of a circuit. The **COM** probe is conventionally black but there is no difference between the red probe and black probe other than color. **10A** is the special port used when measuring large currents (greater than 200mA). **mAVΩ** is the port that the red probe is conventionally plugged in to. This port allows the measurement of current (up to 200mA), voltage (V), and resistance (Ω). The probes have a *banana* type connector on the end that plugs into the multimeter. Any probe with a banana plug will work with this meter.

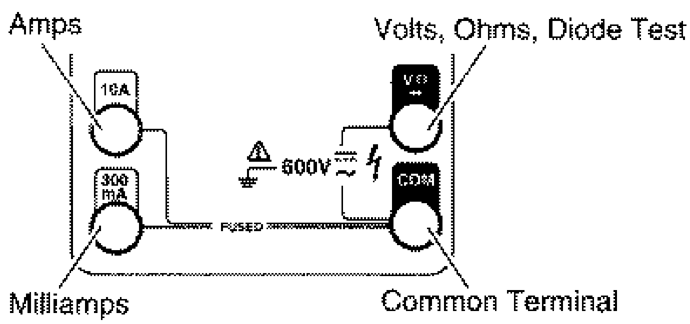
	Diode test mode has been selected.		Caution, see (5) above.
	Indicates that the meter battery voltage		Caution, risk of electric shock.
uA, mA, A	Units for current measurements	A.C.V.	A.C. Voltage ranges.
mV, V	Units for voltage measurements	D.C.V.	D.C. Voltage ranges
, K, M	Units for resistance measurements	D.C.A.	D.C. Current ranges.

SAFETY MEASURES

- Be sure the test leads and rotary switch are in the correct position for the desired measurement.
- Never use the meter if the meter or the test leads look damaged.
- Never measure resistance in a circuit when power is applied.
- Never touch the probes to a voltage source when a test lead is plugged into the 10 A or 300 mA input jack.
- To avoid damage or injury, never use the meter on circuits that exceed 4800 watts.
- Never apply more than the rated voltage between any input jack and earth ground.
- Be careful when working with voltages above 60 V DC or 30 V AC rms. Such voltages pose a shock hazard.
- Keep your fingers behind the finger guards on the test probes when making measurements.
- To avoid false readings, which could lead to possible electric shock or personal injury, replace the battery as soon as the battery indicator appears.

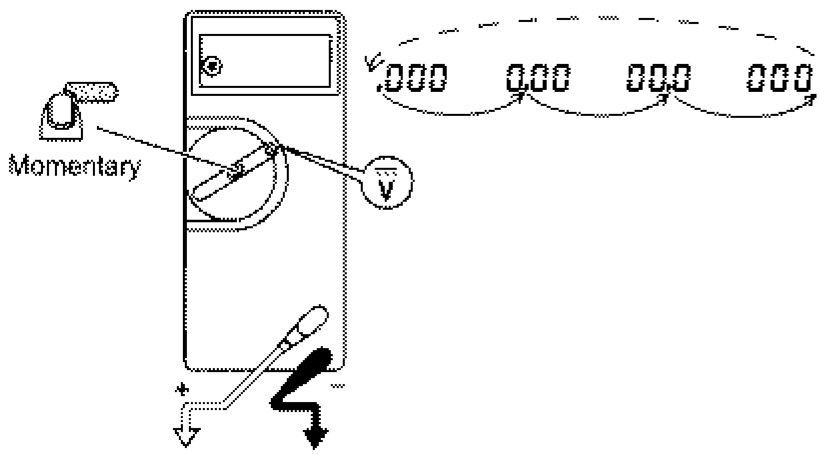
INPUT JACKS

The black lead is always plugged into the common terminal. The red lead is plugged into the 10 A jack when measuring currents greater than 300 mA, the 300 mA jack when measuring currents less than 300 mA, and the remaining jack (V-ohms-diode) for all other measurements.



RANGE FIXING

The meter defaults to autorange when first turned on. You can choose a manual range in V AC, V DC, A AC, and A DC by pressing the button in the middle of the rotary dial. To return to autorange, press the button for one second.

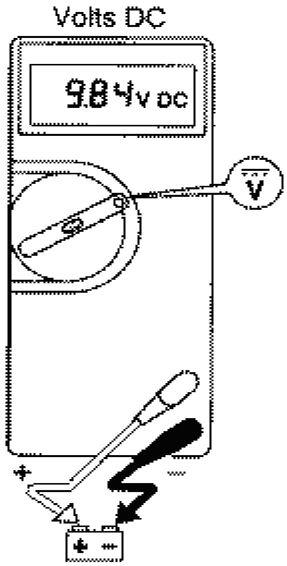


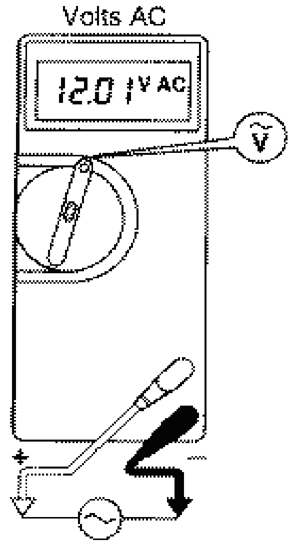
PROCEDURE FOR MEASUREMENT

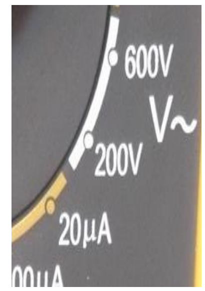
VOLTAGE MEASUREMENT

D.C. / A.C. Voltage Measurment

1. Connect the positive(red) test lead to the 'V/mA' jack socket and the negative(black) lead to the 'COM' jack socket.
2. Set the selector switch to the desired mV D.C./D.C.V/A.C.V range.
3. Connect the test leads to the circuit to be measured.
4. Turn on the power to the circuit to be measured, the voltage value should appear on the digital display along with the voltage polarity(if reversed only).

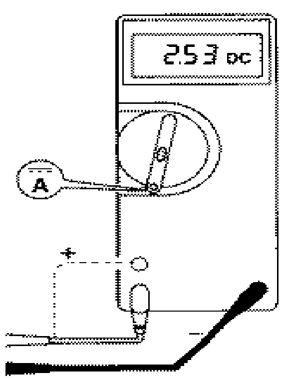






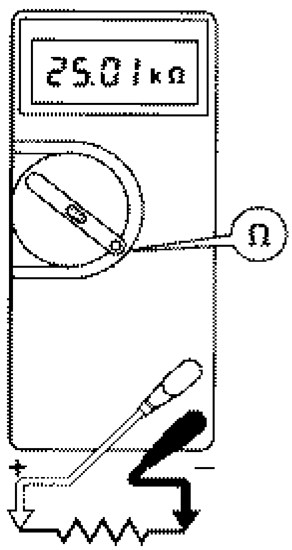
CURRENT MEASUREMENT

1. Connect the positive(red) test lead to the 'V/mA' jack socket and the negative(black) lead to the 'COM' jack socket(for measurements up to 200mA). For measurements between 200mA and 10A connect the red test lead to the '10mA' socket.
2. Set the selector switch to the desired uA/mA/A range.
3. Open the circuit to be measured and connect the test leads in **SERIES** with the load in which current is to be measured.
4. To avoid blowing an input fuse, use the 10A jack until you are sure that the current is less than 300 mA. Turn off power to the circuit. Break the circuit. (For circuits of more than 10 amps, use a current clamp.) Put the meter in series with the circuit and turn power on.



RESISTANCE MEASUREMENT

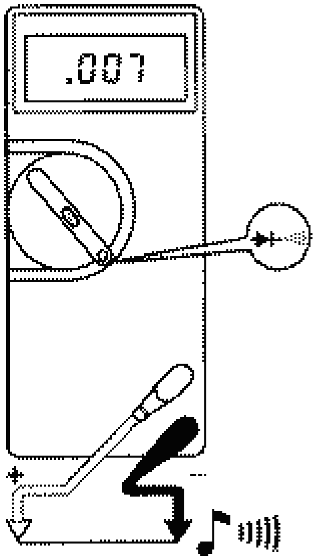
1. Connect the positive(red) test lead to the ‘V/mA’ jack socket and the negative(black) lead to the ‘COM’ jack socket.
2. Set the selector switch to the desired ‘OHM **Ω**’.
3. If the resistance to be measured ia part of a circuit, turn off the power and discharge all capacitors before measurement.
4. Connect the test leads to the circuit to be measured.
5. The resistance value should now appear on the digital display.
6. If the resistance to be measured is part of a circuit, turn off the power and discharge all capacitors before measurement.

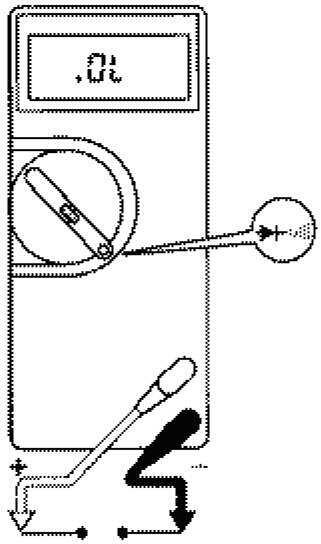


CONTINUITY TEST

This mode is used to check if two points are electrically connected. It is often used to verify connectors. If continuity exists (resistance less than 210 ohms), the beeper sounds continuously.

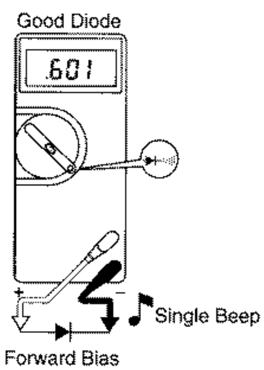
1. Connect the positive(red) test lead to the ‘V/mA’ jack socket and the negative(black) lead to the ‘COM’ jack socket.
2. Set the selector switch to the position.
3. Connect the test leads to two points of the circuit to be tested. If the resistence is Ohms the buzzer will sound.
4. If the resistance to be measured is part of a circuit, turn off the power and discharge all capacitors before measurement.

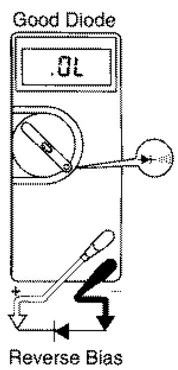


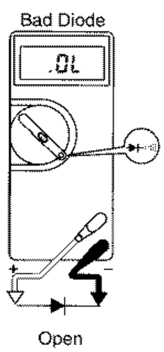


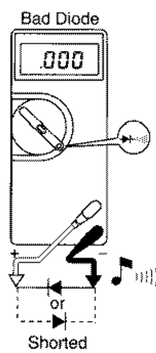
DIODE TEST

1. Connect the positive (red) test lead to the ‘V/mA’ jack socket and the negative(black) lead to the ‘COM’ jack socket.
2. Set the selector switch to the ⯈| position.
3. Connect the test leads to be measured.
4. Turn on the power to the circuit to be measured and the voltage value should appear on the digital display.
5.









General Operation

Connection of Probes:

All multimeters come with two probes. They are to be connected to the terminals on the meter itself. The Black probe is to be connected to the COM terminal. Red probe is to be connected to terminal marked with :

- 'V-Ω' for voltage measurement,
- 'mA' or '20A' for current measurement (there are two terminals, one for 2A range and the other for 20A range)
- 'V-Ω' for resistance measurement.

Setting of function:

The multimeter uses different circuits internally to measure different things. Therefore, you must select the correct function before using it.

Setting of Range:

You can change the sensitivity of the meter by selecting different range for measurement. Set the range to the first range that is higher than the maximum value you expect to measure. This will give a more accurate reading. If you do not know what to expect, use the highest range first. After a reading is obtained, set the range to the appropriate one to get a better reading.

When the value measured exceeds the existing range, the display will flash. When this happens, set the multimeter to a higher range until some values are displayed.

Precautions:

1. For current measurement, the maximum input current is 2A (if the RED mA and BLACK COM terminals are used) or 20A (if the RED 20A and BLACK COM terminals are used). Excessive current will blow the fuse on the 2A range, which must be replaced. The 20A range, however, is not protected by the fuse.

2. For voltage measurement (use the RED V-Ω and BLACK COM terminals), the maximum input voltage on :
    - all DC ranges is 1200V DC or peak AC
    - the 20V, 200V and 1000V AC ranges is 1000V rms continuous
    - the 2V and 200mV AC ranges is 1000V rms for not more than 15 seconds.

3. To avoid electrical shock and/or instrument damage, do not connect the COM input terminals to any source of more than 500 volts DC or peak AC above earth ground.

Resistance measurement

Procedure

1. Connect probes: black probe to COM terminal and red probe to terminal marked with '□ '
2. Set function to resistance measurement
3. Set to the appropriate range (refer to above)
4. Connect the two probes' crocodile clips to the resistor (or to the resistor circuit via jumper wires) to make measurement
5. Note the reading, adjust range if necessary
6. Take the more accurate reading.

Voltage measurement

Procedure

7. Connect probes: black probe to COM terminal and red probe to terminal marked with ‘V’
8. Set function to voltage measurement
9. Set to the appropriate range (refer to above)
10. Set the AC-DC selection - depends on what type of signal you want to measure
11. Touch the two points where you want to make measurement
12. Note the reading, adjust range if necessary
13. Take the more accurate reading.

Note

Reading obtained is the voltage of where the red probe touch with reference to where the black probe touches. This may not indicate the voltage level from ground. To find the voltage level of a point from ground, black probe should be touching a ground point and the red probe on the point you want to measure.

Current measurement

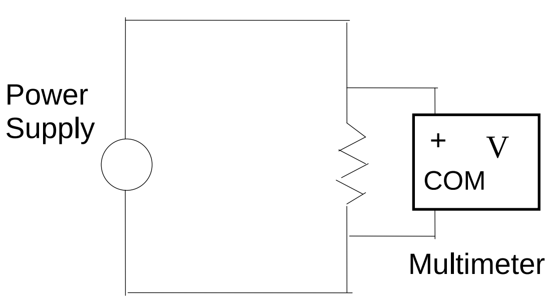
Procedure

1. Connect probes: black probe to COM terminal and red probe to terminal marked with ‘A’
2. Set function to current measurement
3. Set to the appropriate range
4. Set AC-DC selection - depends on what type of signal you want to measure
5. Off the power to the circuit
6. Break the path which we want to make measurement
7. Connect the path with the two probes so that current now flow through the multimeter
8. On the power
9. Note the reading, change range if necessary
10. Take the more accurate reading.

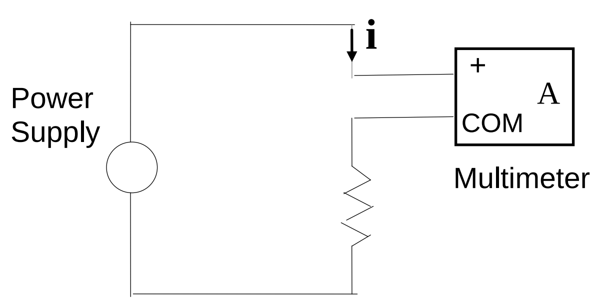
Note

Use the 20A range if you are not sure of the current to be measured.

Connection Methods to Measure Voltage and Current



Voltage Measurement



Current Measurement

2B DC VARIABLE POWER SUPPLY

Objective

- To study the function and operation of regulated power supply.

Equipment required

- Multimeter
- Dual DC variable regulated Power supply (0-30) Volts

Theory

A **power supply** is a device that supplies electric power to an electrical load. The term is most commonly applied to electric power converters that convert one form of electrical energy to another, though it may also refer to devices that convert another form of energy (mechanical, chemical, solar) to electrical energy. A regulated power supply is one that controls the output voltage or current to a specific value; the controlled value is held nearly constant despite variations in either load current or the voltage supplied by the power supply's energy source.

A power supply may be implemented as a discrete, stand-alone device or as an integral device that is hardwired to its load. Examples of the latter case include the low voltage DC power supplies that are part of desktop computers and consumer electronics devices.

Commonly specified power supply attributes include:

- The amount of voltage and current it can supply to its load.
- How stable its output voltage or current is under varying line and load conditions.

POWER SUPPLIES TYPES

- Battery
- DC power supply
- AC power supply
- Linear regulated power supply
- Switched mode power supply
- Programmable power supply
- Uninterruptible power supply
- High voltage power supply
- Voltage multipliers

DC POWER SUPPLY

SPECIFICATION

1. Adjustable 0~30V/0~2A
2. The design is limit the voltage overload
   The power supply input **220V, 230V, 240V AC**
3. Output voltage: 0-30V DC
4. Work temperature: -10°C-40°C

MAIN FUNCTION

1. Output constant current adjustable.
2. Output constant voltage adjustable.
3. LCD voltage and current display.
4. Constant voltage and current operation in individual.
5. Over current protection.

Adjustable power supply

2C CATHODE-RAY OSCILLOSCOPE (CRO)

Objective

- To introduce the basic structure of a cathode-ray Oscilloscope.
- To get familiar with the use of different control switches of the device.
- To visualize an ac signal, measure the amplitude and the frequency.

Equipment Required

- Cathode-ray Oscilloscope
- Function Generator
- BNC connector

Theory

The device consists mainly of a vacuum tube which contains a cathode, anode, grid, X&Y-plates, and a fluorescent screen (see Figure below). When the cathode is heated (by applying a small potential difference across its terminals), it emits electrons. Having a potential difference between the cathode and the anode (electrodes), accelerate the emitted electrons towards the anode, forming an electron beam, which passes to fall on the screen.

When the fast electron beam strikes the fluorescent screen, a bright visible spot is produced. The grid, which is situated between the electrodes, controls the amount of electrons passing through it thereby controlling the intensity of the electron beam. The X&Y- plates are responsible for deflecting the electron beam horizontally and vertically.

A sweep generator is connected to the X-plates, which moves the bright spot Horizontally across the screen and repeats that at a certain frequency as the source of the signal. The voltage to be studied is applied to the Y-plates. The combined sweep and Y voltages produce a graph showing the variation of voltage with time.

Experimental Figures

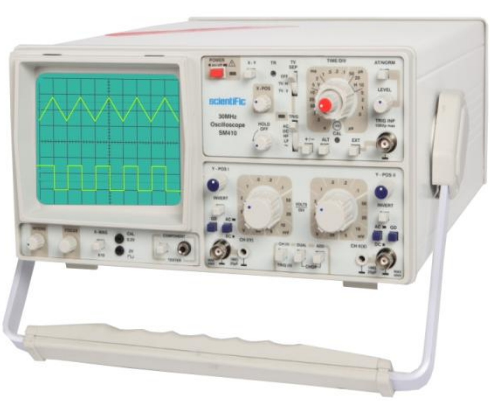


Fig 1. Cathode Ray tube Oscilloscope

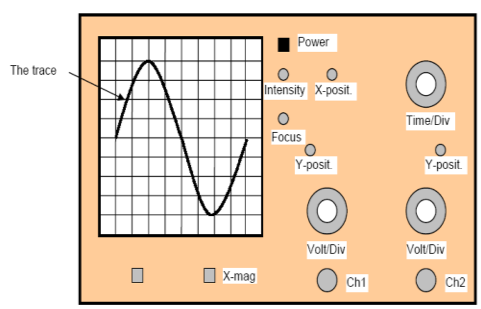
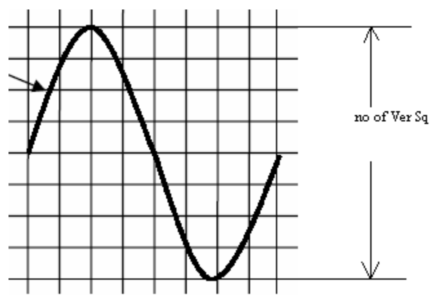


Fig 2. Out Line Diagram of CRO

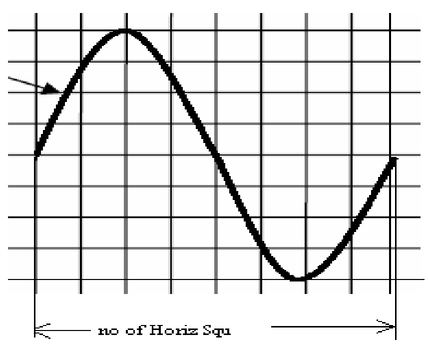
Procedure

1. Turn on the Oscilloscope
2. Adjust the intensity and the focus of the trace.
3. Use the X & Y knobs to center the trace horizontally and vertically.
4. Connect the cable from Ch1 of the CRO to Function generator.
5. A signal will appear on the screen.
6. Make sure that the inner red knobs of the Volt/Div and the Time/Div are locked clockwise.
7. Set the frequency of the generator to 100 Hz.
8. Adjust the Volt/Div and the Time/Div knobs so that you get a suitable size signal
9. Count the number of vertical squares lying within the signal, then calculate the peak to peak value as:



Vp-p = No. vertical Div *Volt/Divs

10. Count the number of horizontal squares lying within the one Duty Cycle, then calculate time value as:



Time = No. Horizontal Div *Time/Divs

11.Calculate the Frequency of signal by using the formula:

$$Freq = 1 / Time$$

Result:

2D FUNCTION GENERATOR

Objective

1. To get familiarization and study the operation of a function generator instrument
2. To identify key function generator specifications
3. To visualize the types of waveforms produced by a function generator

Equipment Required

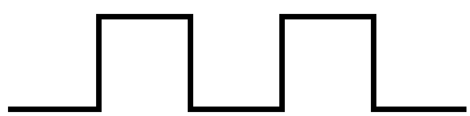
- Oscilloscope
- Function generator.
- BNC connector cable

Theory

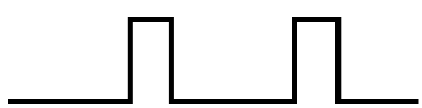
A **function generator** is electronic test equipment used to generate different types of waveforms over a wide range of frequencies. Function generators are capable of producing a variety of repetitive waveforms, generally from the list below

- ***Sine wave:*** A function generator will normally have the capability to produce a standard sine wave output. This is the standard waveform that oscillates between two levels with a standard sinusoidal shape.

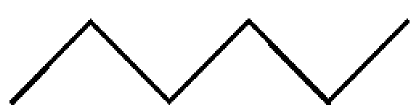
- ***Square wave:*** A square wave is normally relatively easy for a function generator to produce. It consists of a signal moving directly between high and low levels.

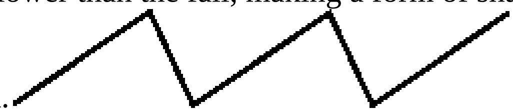


- ***Pulse:*** A pulse waveform is another type that can be produced by a function generator. It is effectively the same as a square wave, but with the mark space ratio very different to 1:1.



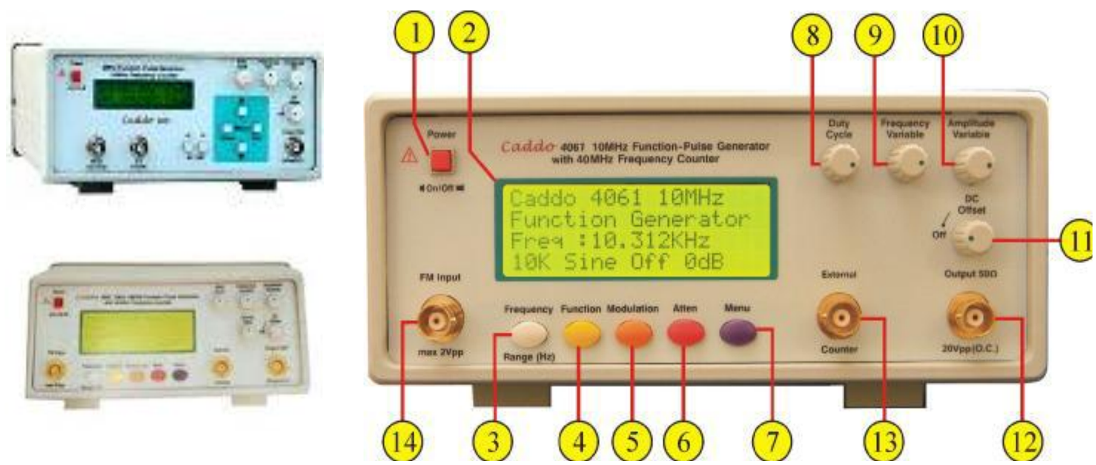
- ***Triangular wave:*** This form of signal produced by the function generator linearly moves between a high and low point.



- ***Saw tooth wave:*** Again, this is a triangular waveform, but with the rise edge of the waveform faster or slower than the fall, making a form of shape similar to a saw tooth. 

These waveforms can be either repetitive or single-shot Function generators are used in the development, test and repair of electronic equipment.

Types of Function Generator



PROCEDURE

1. Turn on the oscilloscope
2. Connect the function generator to one vertical channel of the oscilloscope using BNC Connector
3. Select the type of wave form by pressing Function control button.
4. Set the waveform at desired frequency by adjusting Frequency variable control button.
5. Now adjust the amplitude control of the function generator to establish a 4 V peak-to-peak (p-p) sinusoidal waveform on the screen.

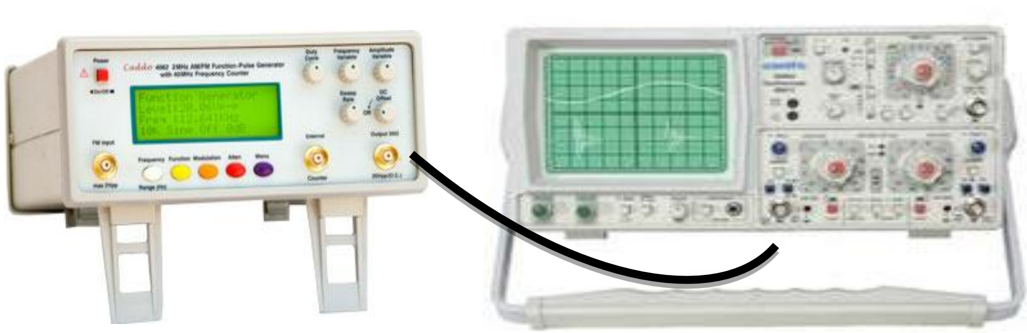
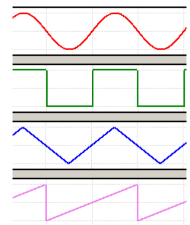


Figure: Function Generator

Function Generator Controls

Knobs Number	Control Name	Functions
1	Power	Push button switch to power ON the instrument.
2	LCD Display	20 x 4 Character bright back lit Liquid Crystal Display.
3	Frequency	Used for selection of frequency range step by step.
4	Function	Used for selection of Particular waveform. A total number of 6 different waveforms : ❖ *Sine* ❖ *Square* ❖ *Triangle* ❖ *Ramp* ❖ *Pulse* ❖ *TTL are available.* 
5	Modulation	Used for selection of Frequency Modulation.
6	Attenuation	Used for Selection of 20dB or 40dB attenuation
7	Menu	Used for selection of Function Generator/Frequency counter mode.
8	Duty Cycle	When pulse output function is selected, this controls the pulse duty cycle from 15% to 85%.
9	Frequency Variable	In conjunction with frequency range, selected by frequency key on front Panel.
10	Amplitude Variable	In conjunction with attenuators (6), this varies the level of output.
11	DC Offset	This control provides DC offset. Approximately ±5VDC is superimposed on the output. Keep the control off if DC offset is not required.
12	Output (BNC connector)	Output of 10 MHz function generator i.e. 20Vpp (Open Circuit)
13	External Counter (BNC Connector)	Input BNC connector for measuring the frequency of external signal when External Counter mode is selected by Menu key on the LCD display. 
14	Modulation Input	Maximum modulation Input i.e. 2Vpp.

LAB Activity

2A DIGITAL MULTI METER

observation

Description	Value
Measurement of line voltage (ac)	
Measurement of resistance	
Measurement of continuity	

2B DC VARIABLE POWER SUPPLY

Set 5V in both channel

2C CATHODE-RAY OSCILLOSCOPE (CRO)

Self test of CRO	
Description	**Value**
Measurement of voltage	
Measurement of Time	
Calculated frequency	

2D FUNCTION GENERATOR

- **Set sine wave _____Hz and display in CRO**
- **Set Triangle/ Square wave and display in CRO**

observation

Description	Sine wave	Square Wave
Measurement of voltage		
Measurement of Time		
Calculated frequency		

Result

General Instructions to Create Schematic Design

OBJECTIVES:

- Learn to create a schematic drawing with PADS Logic

PART 1 – PADS LOGIC

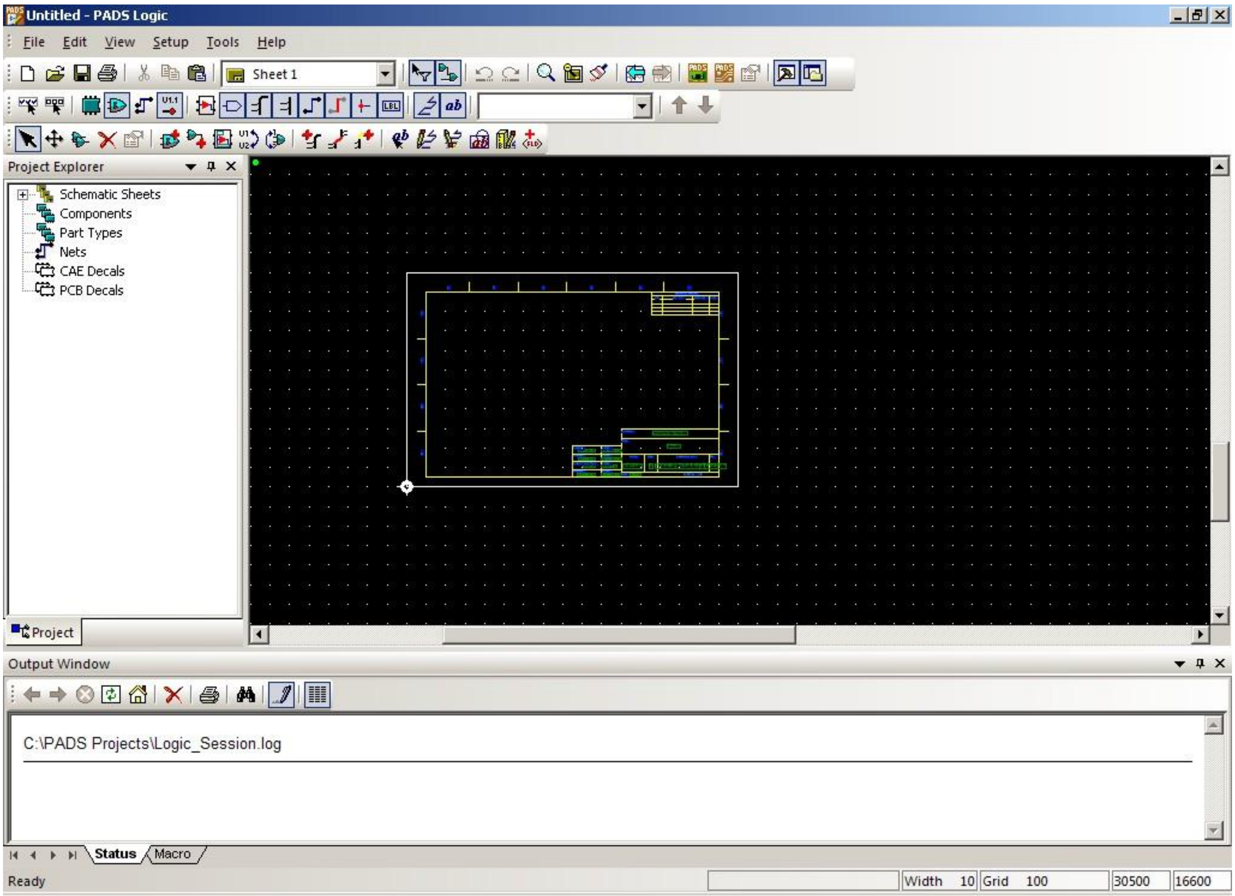
Launch PADS Logic: Move your cursor to the Toolbar and tips will appear as you move from one icon to another. Spend a few minutes to familiarize yourself with the **Menu** bar, **Toolbars** and **Icons.**

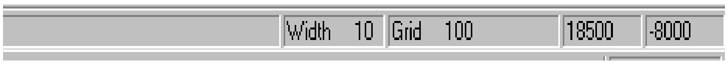
At the top of the window, other than the Menu Bar, there are 3 Toolbars:

(i) **Standard** Toolbar

(ii) **Selection Filter** Toolbar

(iii) **Schematic Editing** ToolbarThere is also a Status Bar at the bottom of the screen.



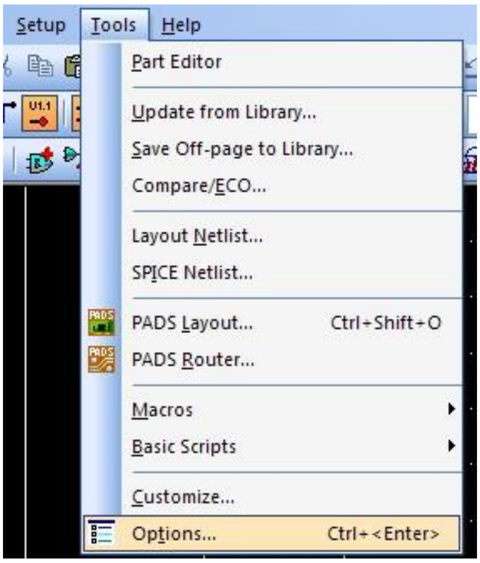


Default width
Grid Size
X, Y Cursor location

General Design Settings

Set up Drawing Environment

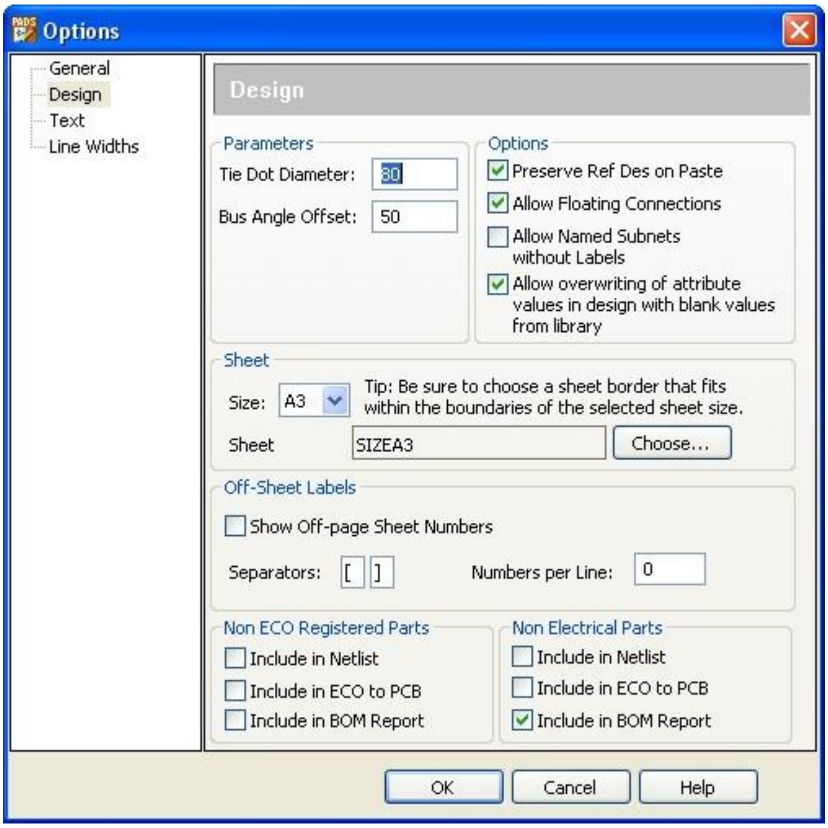
To start, Goto*Tools>>Options*



Set Sheet Size and Tie Dot Diameter

Select **Design** and change at the **Sheet** area: **Size** and **Sheet** to **A3**.

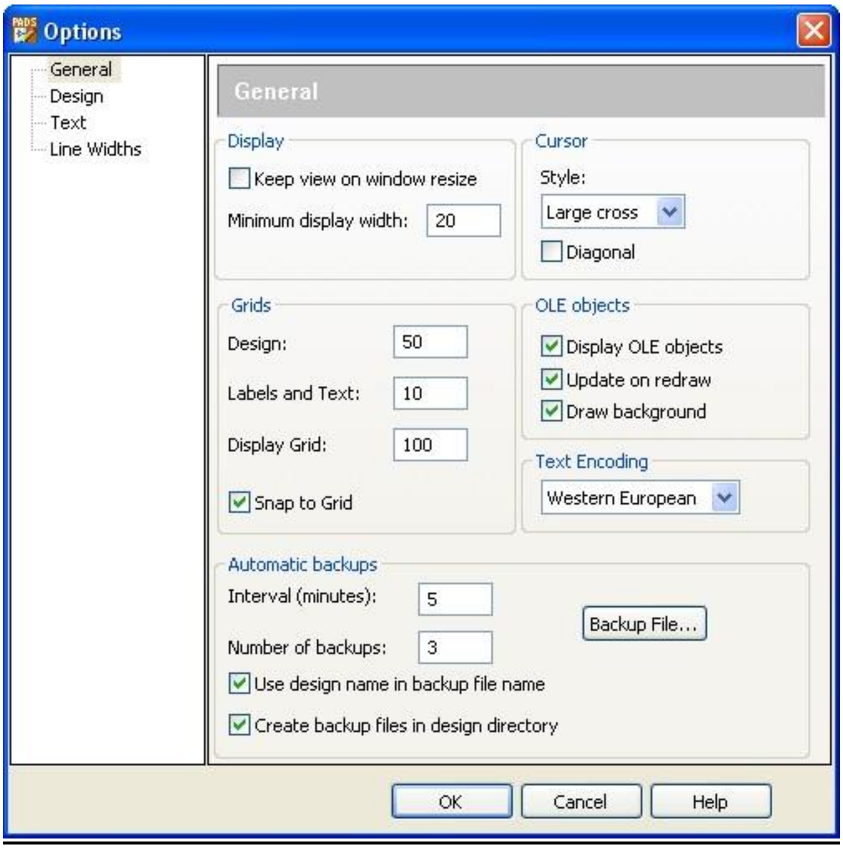
Also change the **Tie Dot Diameter** to **80.**



Set Display and Design Grid

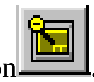
Next, select **General.** Set the **Display Grid** to **100**and **Design Grid** to **50**as shown.

Display Grids appear as dots on the screen. Design Grid are not shown but can be snapped to.



Click **OK** to make changes.

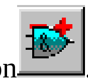
Zoom In/Out

To see the whole schematic on the screen, click on the **Sheet** icon.

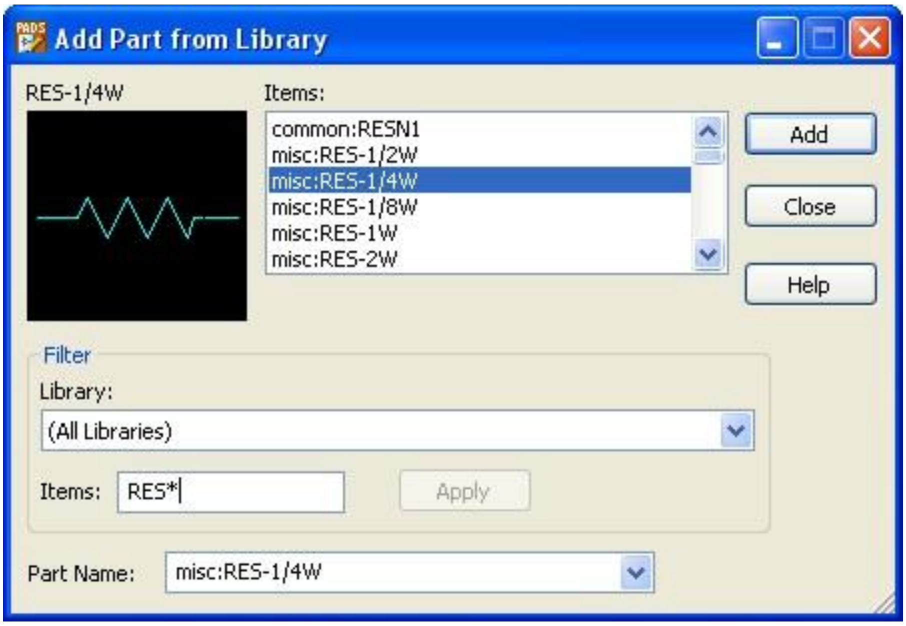
To Zoom-In or Out of the drawing, use the **PgeUp** or **PageDown** buttons on the keyboard or press the **CTRL** key and roll the **mouse-wheel**.

CAPTURE THE SCHEMATIC

To Add a Part

To add a Resistor to the schematic, from **Schematic Editing Toolbar**, select the **Add part** icon.

A window for part selection will appear.

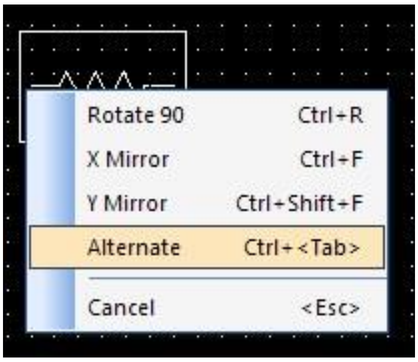


- In the **Filter** section, we want to search through **All Libraries.**
- In **Items** edit box enter **RES***(The Asterisk *is a wildcard character). Then click the **Apply**button.

Look for **misc: RES-1/4W.** Click **Add** Buttonto add the part.

Alternate Symbols

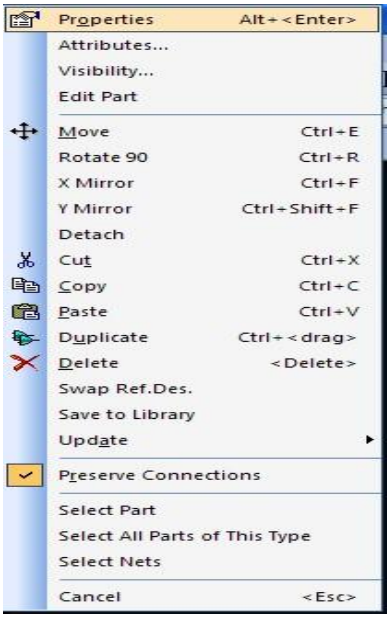
- Click the **right-mouse button** and select **Alternate**. Do this several times.



- Finally, click the **left-mouse-button** to drop the part onto your schematic. However, only a few commonly used parts have alternate symbols.
- Depending on how you would like the part to appear, you may also carry out the following operations on the part: **Rotate 90, X Mirror and Y Mirror**.
- Search and add the other parts given in the Appendix to your schematic.

Change Part Properties

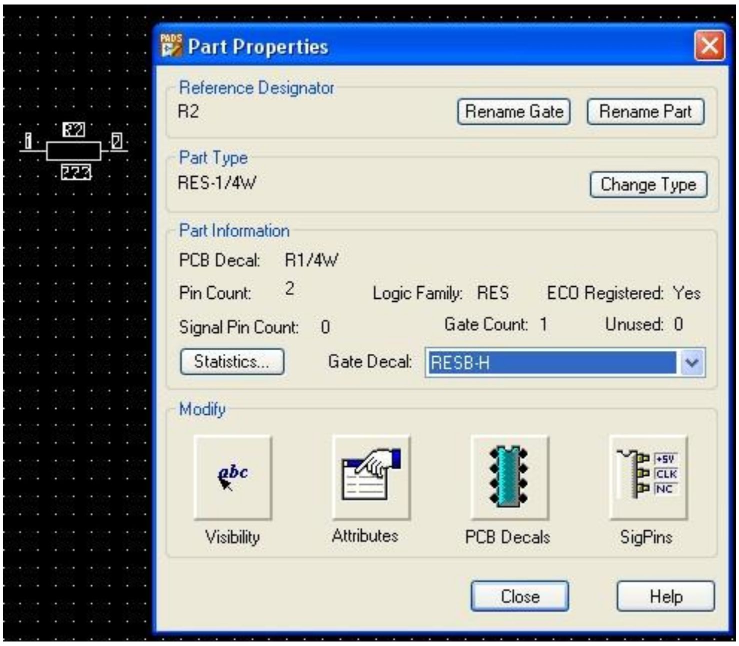
To see or change properties of the part, **Select** the part (click the Part once and it will turn White), click the **Right-Mouse Button** and then choose **Properties** from the drop down menu shown below.



The **Part Properties** window will pop-up and you can:

- change values or set colors in **Attributes**
- view the part decal using **PCB Decal**
- change power connection of ICs using **SigPins**

You can also **double-click** on the part to get to the Part Properties window:



Deleting Parts

To delete a part, select the part and press **Delete** Key or select the **Delete icon**

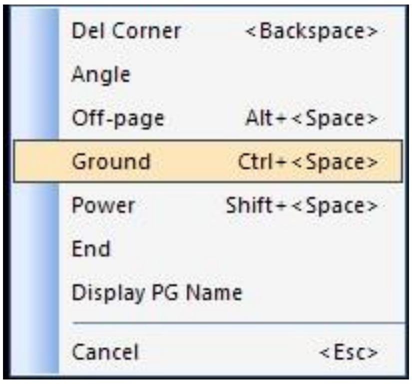
Add Connection

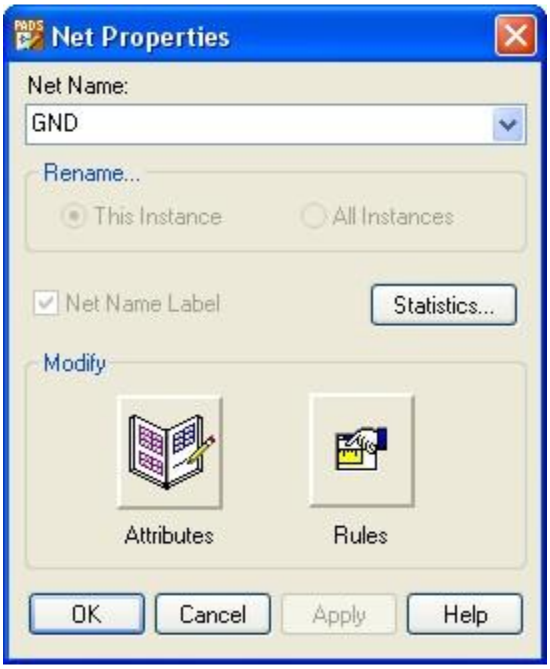
Select the **Add Connection icon** . Click near to the tip of the pins to start the connection.

Add a Power/ Ground Connection

To add a ground symbol:

- Add and extend a **connection** from a pin of a part.
- Move the cursor to the location that you want the ground symbol to be.
- **Right-mouse-click** and select "**Ground**"





To display the **GND** symbol text, Double-click on the **GND** Symbol and the Net Properties window will appear. Scroll to GND and Tick the **Net Name Label** box.

Do likewise for the other Power Nets: **VIN &VCC**. Type-in **VIN** and **VCC i**f they are not listed.

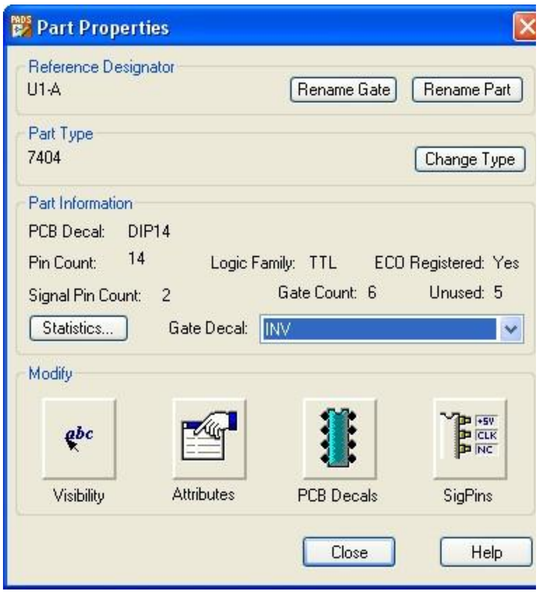
Delete Connection

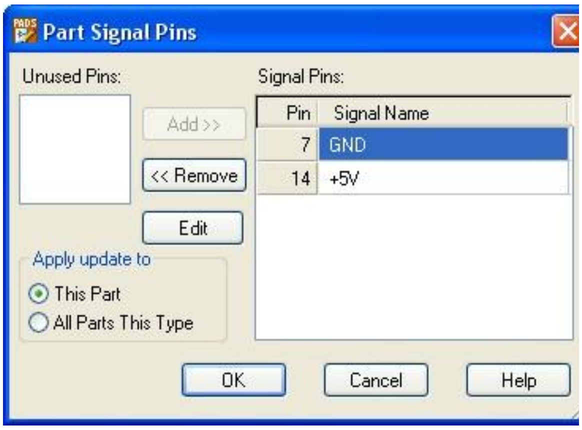
To delete a connection, choose the **Delete** icon and then select the connections to delete.

Power the IC (Take Note, Especially if your design has ICs)

Some ICs can be powered at different voltages. Select **SigPins** button and specify the Net that the Power or GND pins are connected to.

Eg.

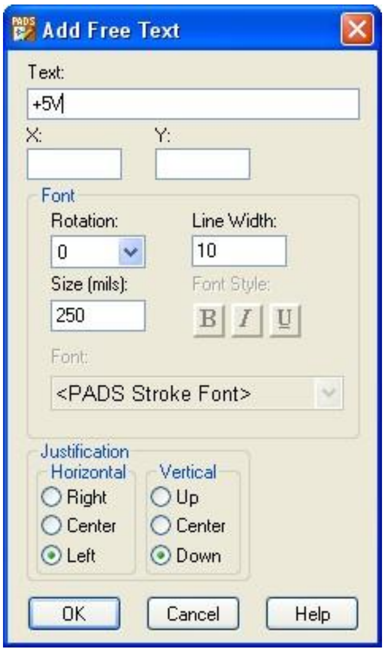




Add Info to Title Block

Double-Click on the Company Field in the Title Block. A Field Properties window will appear. Type in your "**Name / Login ID / Date**". Then Type "**Exercise A**" in "Company Field

To add other Text in the schematic, select Add Text icon .



Save your File

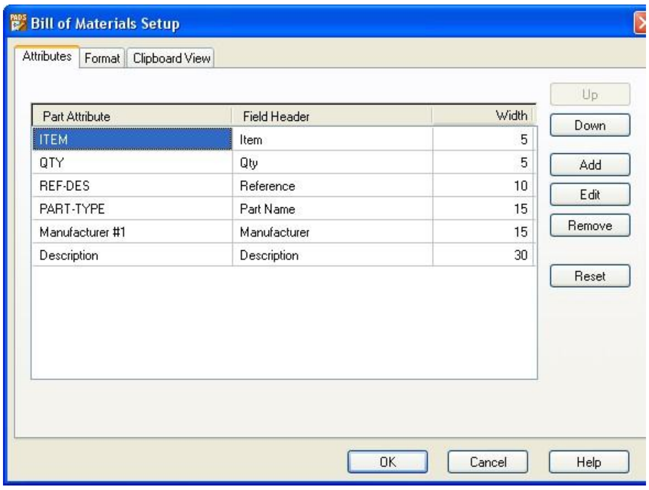
Save your work regularly.

- From Menu bar, Choose **File / Save As.** The “File Save As” dialog box will appear**.**
- Type the file name and then press **Save** or **Save As**

Note the file extension is **.sch** for schematic capture. The file will be saved in the server in your default directory.

BOM (Bill of Material)

Bill of Material (also known as part list) is a document which provides information on the quantities, values, and part type etc of the components used on the printed circuit board.



.

Designers generate the BOM to list the parts required to assemble the boards.

1. Choose **File>>Report** from the main menu.
2. Tick **Bill of Materials** and click on the **Setup** button.
3. Use **Add** button to add in the Reports menu and notepad will display the BOM as indicated below.
4. The file will be saved in your folder.

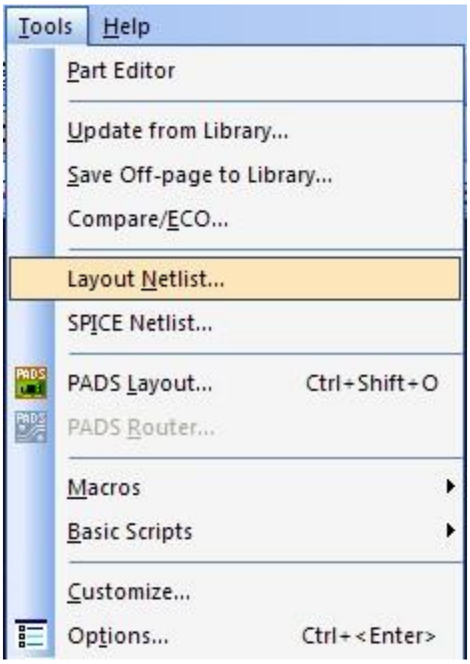
Creating Net-List

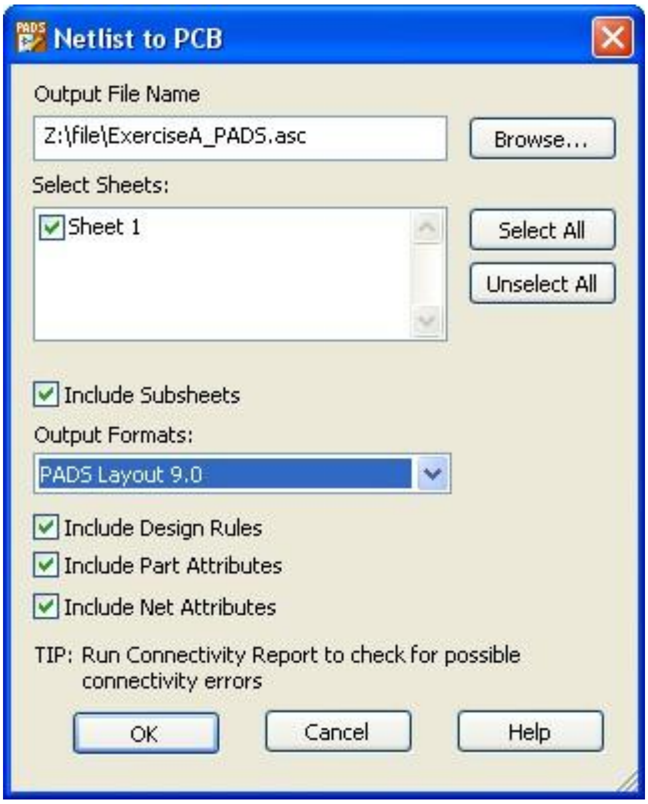
A net list contains a list of parts, their part types and all the nets (connections). The PADS Logic software passes data to the PADS Layout via the net list.

To create a net list:

1. From the menu, select **Tools>>Layout Netlist**. The Netlist dialog box appears.
2. In the output edit box, make sure the directory is current directory. The filename should be *.asc. This is an ASCII file with file extension (.asc). Click on the **OK** button to generate the Netlist file.

3. Study the contents of the netlist file. DO NOT PRINT. Save your work.





General Instructions to design PCB Layout using PADS Layout

OBJECTIVES:

- Learn to design a PCB layout using PADS Layout

Launch the PADS Layout. Other than the **Menu Bar**and the **Standard Toolbar**, there are 5 other Toolbars. Enable them by selecting one of the icons  .

The 5 toolbars, as suggested by their names, are for:

(i) Drafting

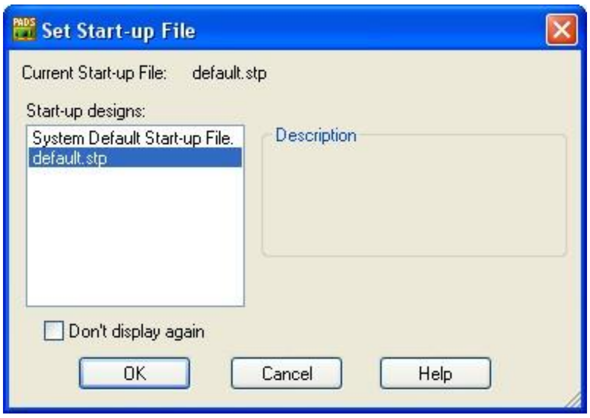
(ii) Design

(iii) Dimensioning

(iv) ECO

(v) BGA

If you are using the program for the first time, go to **File>>set Start-up File**



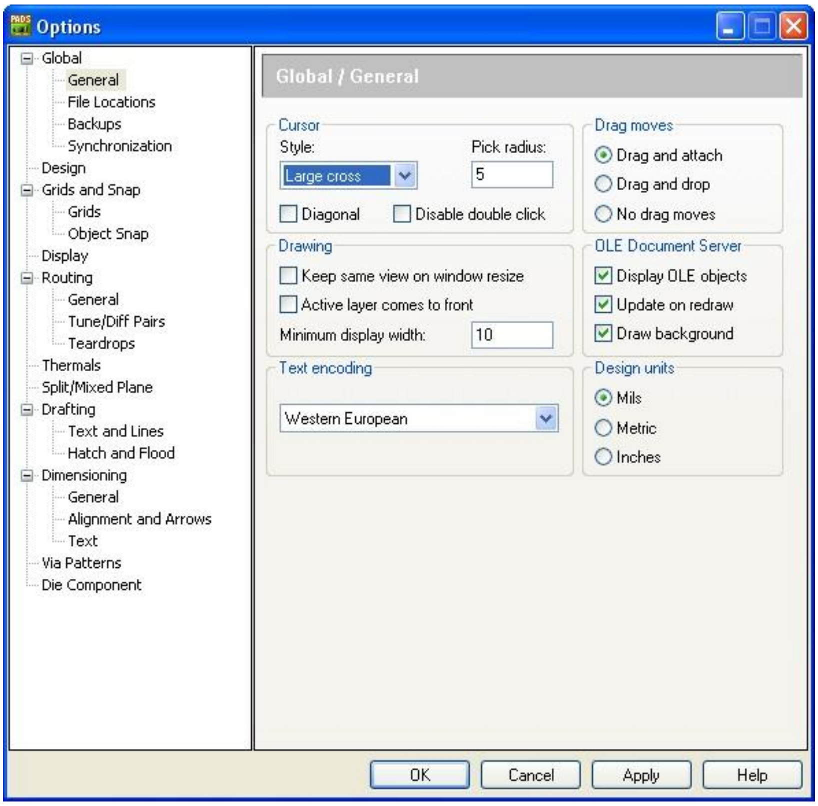
Select **default.stp**. This file will determine the colors used for the various drawing elements. You can check the colors setting by going to **Setup>> Display Colors.**

Select the other startup file and see what happens to the color set.

Set Units of Measure.

As usual, setup the design environment before working on it.

From Main Menu, select Tools>>Options **or select the** Options **icon**  **, which can be found in the Drafting, Design and Dimensioning Toolbar.**



Go to Global>>General. **Change the Design units to Mils.**

Next, go to Grids. **Change the Display grid to 100 and the Design grid to 25 (for both X and Y)**

Zoom-in or Zoom-out till you can see the display grids and till you feel that the screen can fit your PCB board outline.

Set Origin

Before drawing the PCB outline, set the Origin first.

From the menu select **Setup**>>**Set Origin.** Click (precisely) on a display grid. The origin of the workspace, or (0, 0) coordinate location, is represented by a large **white datum (cross).**

Create a Board Outline

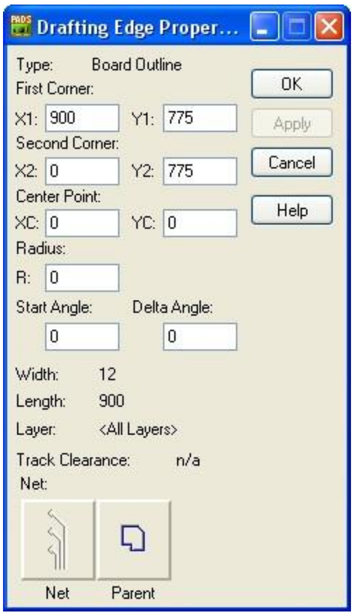
Before drawing the outline of the board, identify the co-ordinates of all the corners.

From the Drafting Toolbar, select the **Board Outline and Cutout icon** **.**

Click on the **Origin** and then extend the board outline to the next co-ordinate point.

Modifying the Board Outline

Double-click on a board outline and a **Drafting Edge Properties** window will appear. Adjustments to the board outline can be changed by entering the correct co-ordinates.



Import Netlist File - ASCII (.asc)

A **Netlist** typically contains a list of all the parts and their connections.

Choose **File>>Import**. Click on the **Yes** button when prompted to save old file before loading

The **File Import** box appears. **Open** the Netlist file "*.asc" you generated in PADS Logic.

Disperse Components

All the imported parts are now all cluttered at the origin. Use **Tools>>Disperse Components** to disperse the components.

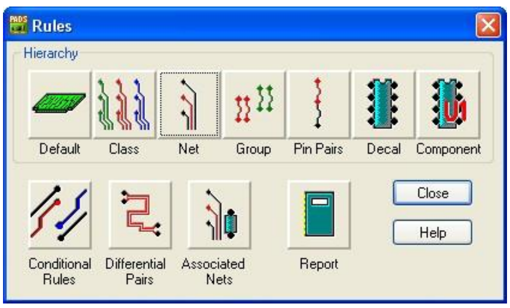
Define Design Rules

Design rules can be set for clearance, routing, high speed constraints and others. They can be assigned for all (default) or for specific nets, layers, classes, pin pairs and etc. You will set 2 sets of design rules here:

- Default Clearance Rules
- Net Clearance Rules

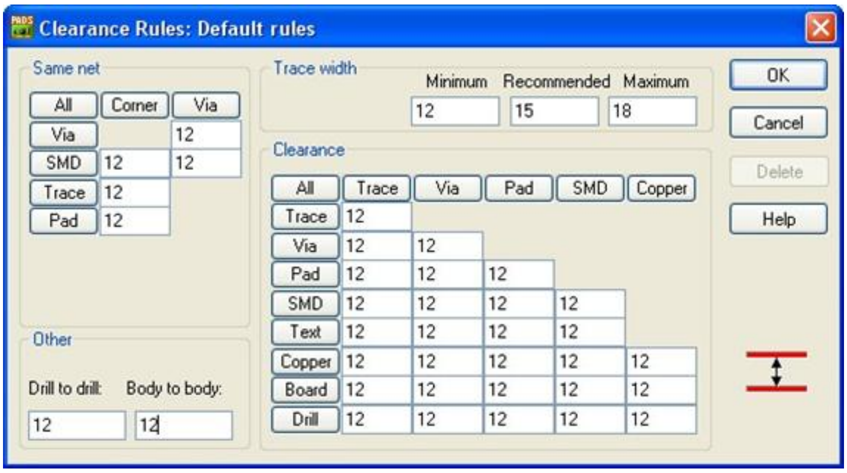
Set Default Clearance Rules

From Main Menu select **Setup>>Design Rules**. The **Rules** dialog box will appear.



Choose the **Default** icon.

The **Clearance Rules: Default Rules** dialog box will appear.

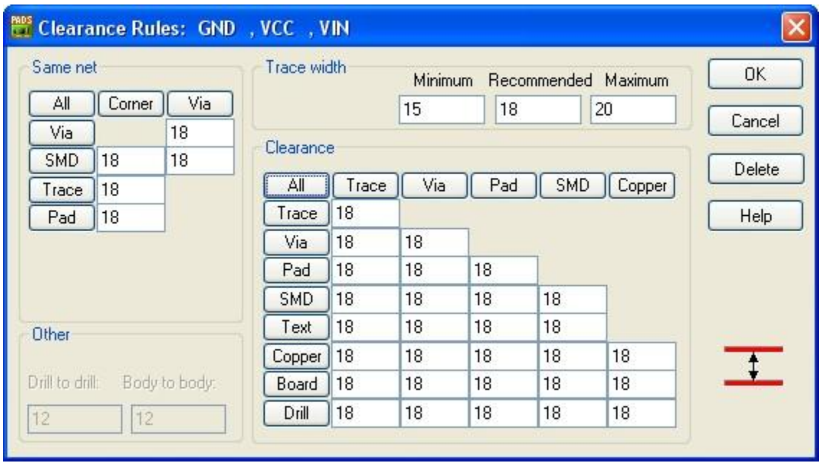


Set the default **Trace Width** and the **Clearance** as shown above (make use of the **All** button at the upper left corner of the matrixes). Click **OK** to save changes and to close the window.

<u>Set Net Clearance Rules</u>

To set rules for specific nets, click on the **NET** icon in the Rules dialog box.

Press **CTRL** key and select the nets **VIN, VCC** and **GND**. In the same way, set the trace width and the Clearance required for these nets.



Click the **OK** button to save the changes.

<u>Component Placement</u>

To move the components, select the component, then drag and drop it at the desired location.

Alternatively, select the move icon  in the Design toolbar and then choose the part to move.

You can also Rotate and Spin the part. **<u>ButDo Not "FLIP</u>"!**

Schematic-Driven Placement

PADS Logic and PADS Layout allow **Cross-Probing**.

Display the Schematic and the Layout side-by-side.

Click on a part in your schematic and the part will appear highlighted in your PCB layout.

This allows you to **place the components in the LOGICAL way that they appear in the schematic**.

If you are not able to cross-probe, in PADS Logic, go to **Tools>>PADS Layout (Link)>>Document Open (browser and select the correct document).**

ROUTING

Routing is to lay wire/traces/tracks between pins. Depending on the components and board used, the matrix below shows some of the possibilities.

Component Type	**Single-sided Board**
Through-Hole Components	Components inserted from Top. Tracks run on bottom/solder side.
Surface-Mounted Components	Components mounted on Top. Tracks on Top side

Change Grid Size

Before you route, reduce the Grids further: Eg. **Display Grid to 50**&**Design Grid to 10**

Routing Layers

Change to the correct layer before you route. Use the Drop down List Box to switch layers.

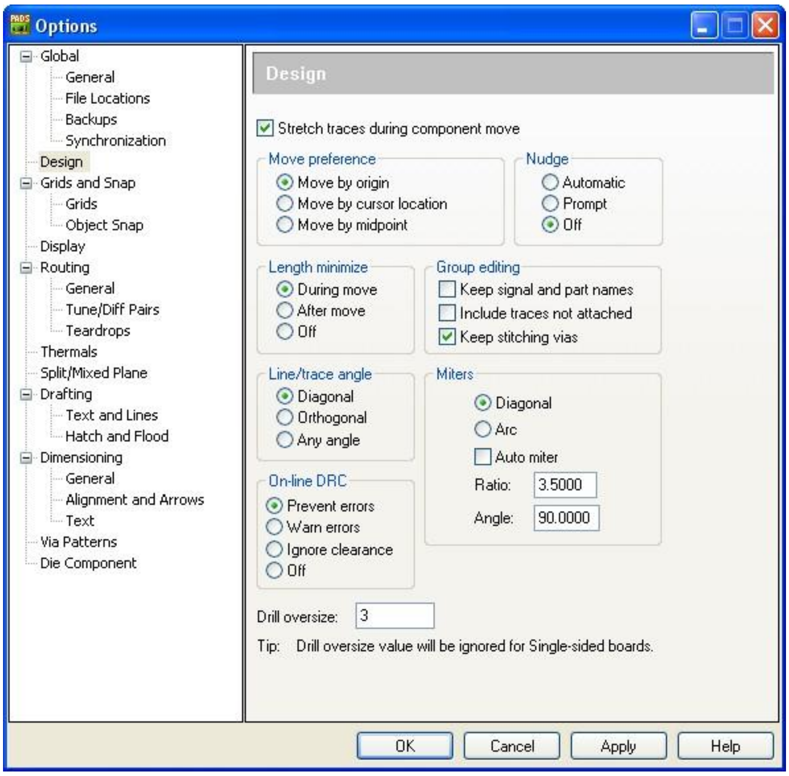
Dynamic Routing

Before you route, go to **Tools>>Options>>Design** and select:

i) **Length minimize>> During move**

ii) **Line/trace Angle>>Diagonal**

iii) **On-line DRC>>Prevent errors –** this is essential if Dynamic Routing is used. For all your exercises use Dynamic Routing.



To lay a trace, select the **Top or Bottom layer**. Click on 2 pads connected by a **Ratnest /Airwire** and the trace will be laid! Notice how dynamic route chooses a shortest path around obstacles. You can also control the route by clicking on intermediate points. Complete all the routing. The Dynamic Route Editor also offers dynamic **bus routing**. You can bus route a group of address and data busses.

Deleting Routes

Select the segment of a completed/unwanted route and press the Delete key.

Verifying your Design - Perform Clearance/Conductivity Check

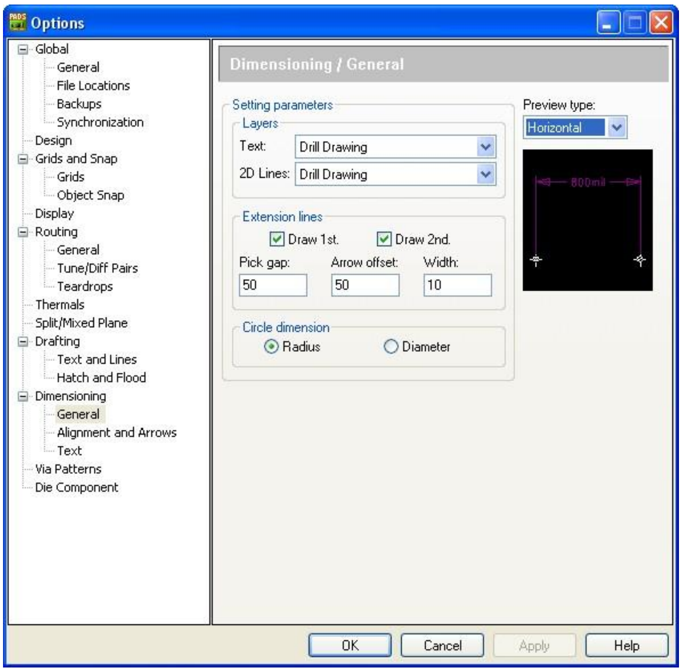
When the board is fully routed, use **Tools>>Verify Design**. Select **clearance** or **connectivity** and then press the **Start** button

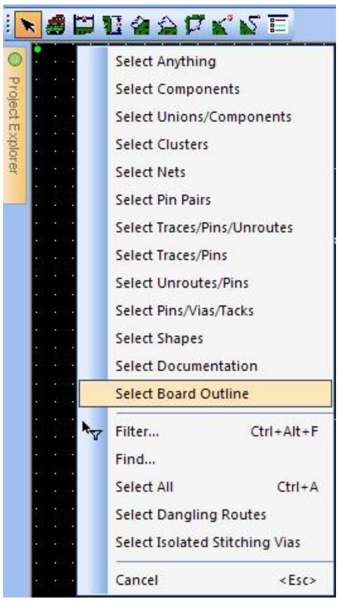
If there is a design violation, an error message will be appear. Double click on the error message (if any) and the error will be highlighted on your drawing.

Clearance checks on the in-between spacing whereas connectivity checks whether everything had been connected.

Board Dimensioning

In PowerPCB, auto-dimensioning feature allows designers to dimension the board easily. A few preferences can be set via **Tools>>Options>>Dimensioning**.





Select the **Dimensioning** Toolbar. Select the **Pointer** icon then Right-Mouse click and ensure that the **Select Board Outline** is selected.

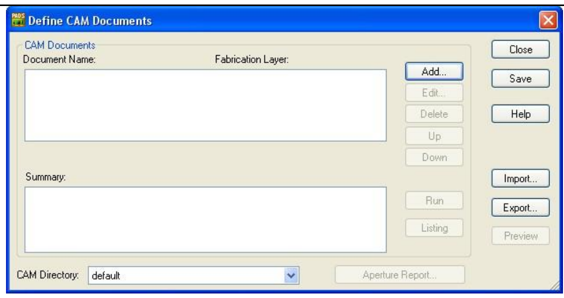
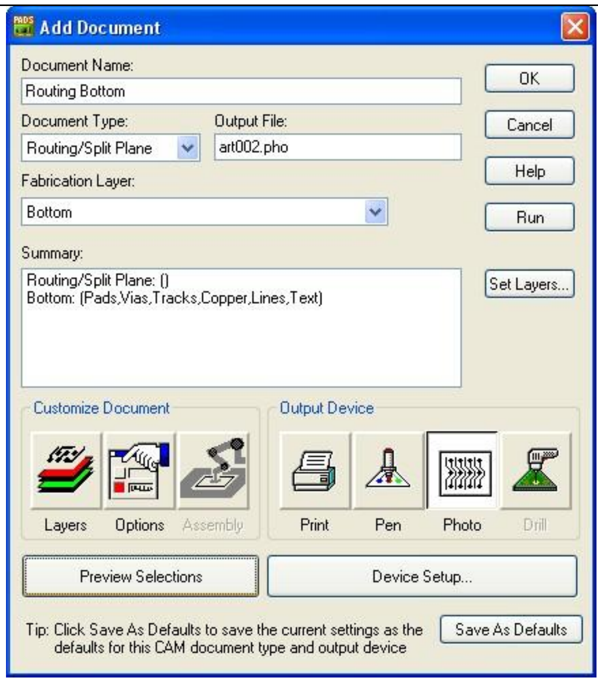
Select the **Horizontal** or **Vertical**icons, and with the mouse, pick the respective segment of the board to dimension, then Drag-and-Drop the dimensioning lines.

General Instructions to generate CAM Files

OBJECTIVES:

- Learn to generate CAM files for PCB fabrication

Creating CAM documents

<table>
<tr>
<td>From Menu Bar, choose File>>CAM. The Define CAM Document dialog box appears.
Click Add button.</td>
<td>
</td>
</tr>
<tr>
<td>The Add Document dialog will appear.
In the Document Name text box, type “Routing, Bottom”.
Select Routing from the Document Type drop down list.
The Layer Association dialog box appears.
Select Bottom and click OK.
Next, select Layers from the Customize Document area.</td>
<td>
</td>
</tr>
</table>

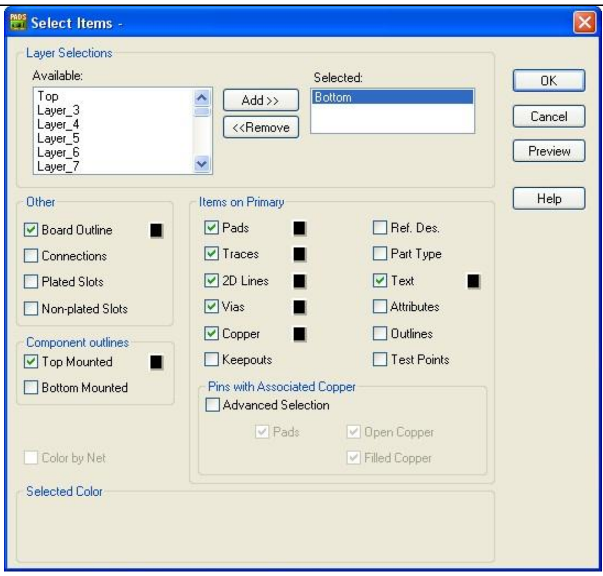
The **Select Items** dialog box will appear.

Select **Board Outline** from the **Other** area.

Check that for the **Selected** layers (Bottom in this case), the required **Items on Primary** have all been correctly selected.

The **items** are information extracted to produce the CAM file.

Choose **Preview** to and see how the artwork would look like as a photo plot.



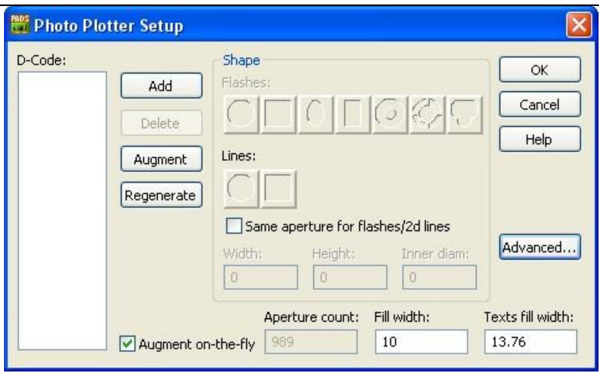
Close to exit the Preview window.

Return to the Add Document window.

Select Device Setup…

A Photo Plotter Setup dialog box would appear.

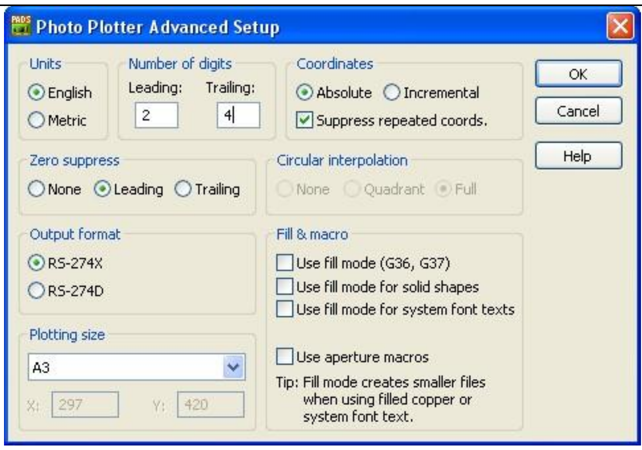
Select Advanced…



The Photo Plotter Advanced Setup is required once only.

Ensure that the Gerber Output format is RS-274X and all other settings as shown in the picture.

Click OK to go back to Define CAM Document window and ensure that you press Save.



For a **Single-Sided Board:**

Document Name	Document Type	Output file	Layer Selection	Items on Primary (for Corresponding Layer Selection)
Routing, Bottom	Routing	art002.pho	1. Bottom	1. Pads, Traces, 2D Lines, Vias, Copper, Text
Silkscreen, Top	Silkscreen	sst001026.pho	1. Top	1. Ref Des, Outlines
			2. Silkscreen Top	2. Text, Outlines
Soldermask, Bottom	Soldermask	sm002028.pho	1. Bottom	1. Pads, Test Points
			2. Soldermask Bottom	2. Lines, Copper, Text, Test Points
Drill Data	NC Drill	drl001.drl	N.A	(Note: This file is required for NC drilling)

RUN →Generate CAM output files

To generate and save all your CAM files,

1. Go to **Define CAM Document** dialog box.
   Highlight all documents in **Document Name** edit box.
2. Click **Run** button, and answer **Yes** in response to the prompt. This will output all your CAM files (artwork and its aperture table) to the D**:\CAM\default** folder in your home directory. Check your folder to verify that the files are all there.

Useful Shortcut Keys

Other Key Shortcuts

Esc	Cancel command
Spacebar	Left mouse button click
PgUp	Zoom in at cursor
PgDn	Zoom out at cursor
Insert	Centre at cursor
Home	View extents
End	Redraw
Del	Delete selected

Laboratory Report Cover Sheet

SRM University Faculty of Engineering and Technology Department of Electronics and Communication Engineering
EC1002 Workshop- Electronics Engineering Practices **First Semester, 2014-15 (Even semester)**

Name :

Register No. :

Day / Session :

Venue : 2th Floor, BEL307 & BEL308, Engineering Basic Lab Building.

Title of Experiment :

Date of Conduction :

Date of Submission :

Particulars	Max. Marks	Marks Obtained
Design / Assembly / Testing/ Fabrication	15	
Lab Performance and Lab report	15	
Viva voce	10	
Total	**40**	

REPORT VERIFICATION

Date :

Staff Name :

Signature :

EXERCISE NO 3
MEASURING VOLTAGE DROP

Objective:

1. To create a schematic with PADS Logic
2. To create a PCB Layout with PADS Layout
3. To use Copper Pour to create Power and GND planes

PART 1 – Schematic Capture

With the help of the parts list given, draw the schematic as shown in the Appendix. Save your schematic as Exercise1.sch. For your schematic, check the following:

1. Sheet Size is set to A3. Tie Dot Diameter is set to 80. Title Block is filled.
2. The component values are all correctly entered.
3. The pin numbers of the resistors are not displayed
4. Reference Designators are all upright and easy to read.
5. The netlist is named Exercise1.asc.

PART 2 – PCB Layout

1. Draw the Board Outline.
2. Set the **Origin** at the bottom-left corner of the board.
3. Import the Netlist and disperse the components.
4. Set the **design rules** as shown in the table below:

	Trace width			
Net Name	**Minimum**	**Recommended**	**Maximum**	**All clearance**
Default	10	12	15	12
+5V	15	20	30	20
GND	15	20	30	20

5. Place the components at your own discretion. Place them logically – refer to the schematic.
6 Before you route, read through the entire **Part 3** below on Copper Pouring.
7 For routing, ensure that **Prevent Error** is turned on. Route using Dynamic Route. Check that the line/trace Angle Mode is set to **Diagonal**.

PART 3 – Copper Pouring

In order to reduce noise in a design, copper planes are often added to a PCB. Other than to reduce noise in the power and ground supply, copper pour can also be used for thermal (heat) dissipation. The steps to add copper pour planes for a typical 2 layered board are:

1. Route all the **GND signal trace** on the Bottom layer - Use **Find>>Highlight>> GND** net to find all the GND nets and **route them on Bottom Layer.**

2. Route all the **+5V signal trace** on the Top Layer - Use **Find>>Highlight>>+5V** net to find all the +5V nets and **route them on the TOP layer**.

3. Route ALL the other signal connections – The other signal lines can be routed either on the top or bottom layers.

4. Create the **outline for the GND Copper Pour** area at the Bottom Layer and **associate it to GND**.

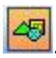
5. Similarly, create the **outline for the +5V Copper Pour area** and **associate it to the +5V**.

6. Finally Flood the areas.

Read on to learn how to create the Copper Pour Outlines and how to associate them to the correct nets.

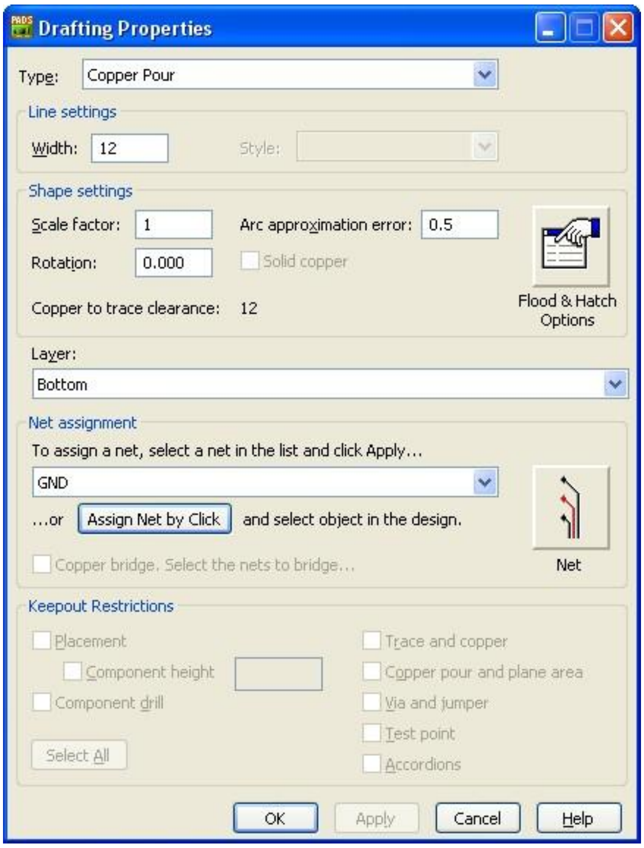
Draw the Copper Pour Outline

To create a copper pour outline on the bottom layer, 25 mils within the board outline:

1. Set the design grid to 25 (both X an Y) .
2. Set the current layer to the **Bottom** layer by selecting it from the Layer drop-down list box in the toolbar.
3. Turn on the **Drafting** Toolbar using the drafting icon .
4. Choose the **Copper Pour** icon from the Drafting
5. Click on Right-mouse-button and choose **Polygon** from the pop-up menu.
6. Position the cursor at X25,Y25 and click to start drawing a polygon which is 25 mils offset from the board outline (polygon must be **within** the board outline)
7. Select **Complete** from the pop-up menu to complete or close the polygon.

Assign the Copper Pour Outline

1. Choose the **Select** Model icon  from the Drafting toolbox to exit Copper Pour mode.
2. Press the **Shift** key and click on any section of the outline to select the whole outline.
3. Next double-click the left mouse button and a **Drafting Properties** window will appear.



4. Select the **Layer** and assign the outline to the correct **Net,** as shown above.

Flood the Copper Plane

With the **Copper Pour Outline** still selected, select the **Flood** icon .

The message "***Proceed with Flood?"*** will appear. Answer **Yes** to the prompt.

Flooding creates a hatch pattern that fills the area enclosed by the pour outline.

Copper Pour Top Layer for the +5V Plane

Repeat the above steps, but this time for the +5V plane on the top layer. Create a copper pour outline, 50 mils within the board outline. Make sure that you associate the pour outline on the TOP layer to +5V before you flood.

PART 4 – Post Processing

1. Dimension the board outline.
2. Generate the following CAM files:

   Routing Bottom

   Silkscreen Top

Solder mask Bottom

NC Drill File

3. If available, use **GC-Prevue** (Graphics code Gerber Viewer) to view your CAM files. Make sure that your artwork is neat and tidy.

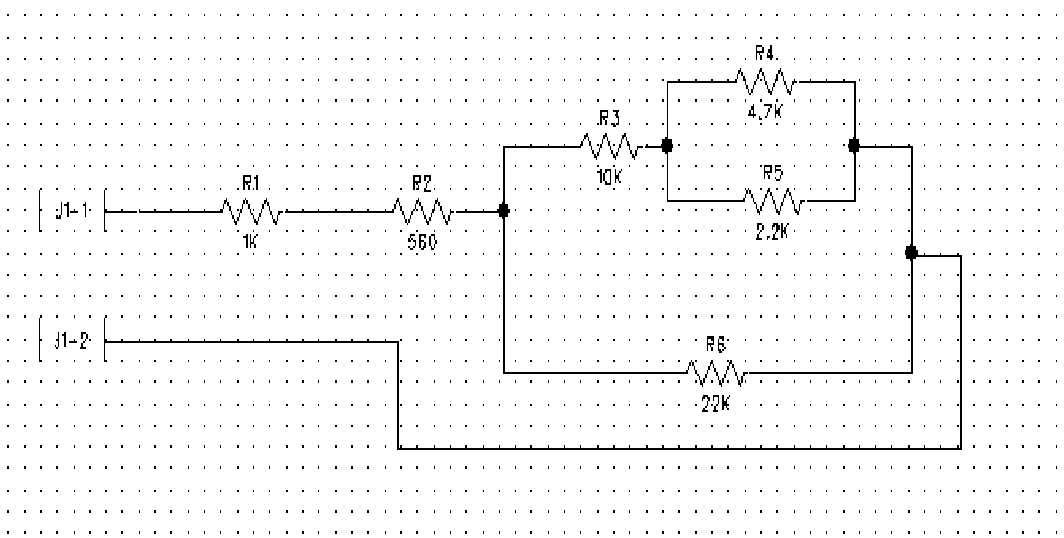
Exercise 1 – Measuring Voltage drop across resistive circuit

Part List

Library Part-type	Ref Designator	Components
RES - 1/4 W	R1 – R6	Resistor, size 1/4 Watt
CON-SIP-2P	J1	2 pins connector

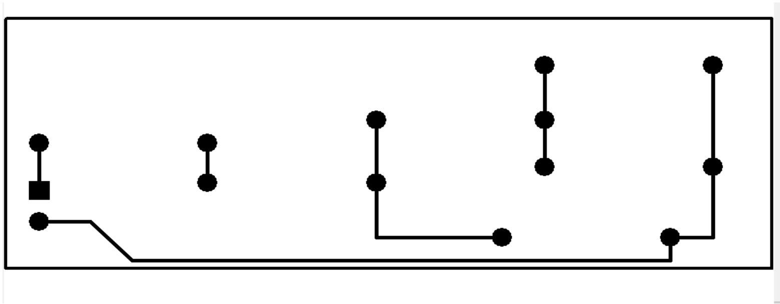
BOM

```
|-----------------------------------------------------------------------------|
|Bill Of Materials for default.sch on Thu Jul 04 15:00:37 2013                |
|-----------------------------------------------------------------------------|
|Item|Qty |Reference|Part Name      |Manufacturer  |Description               |
|----+----+---------+--------------+--------------+----------------------------|
|1   |1   |J1       |CON-SIP-2P    |              |GENERIC 2 PIN SIP HEADER .100|
|    |    |         |              |              |CENTERS                     |
|2   |1   |R3       |RES-1/4W,10K  |              |RES BODY:100  CENTERS:500   |
|3   |1   |R1       |RES-1/4W,1K   |              |RES BODY:100  CENTERS:500   |
|4   |1   |R5       |RES-1/4W,2.2K |              |RES BODY:100  CENTERS:500   |
|5   |1   |R6       |RES-1/4W,22K  |              |RES BODY:100  CENTERS:500   |
|6   |1   |R4       |RES-1/4W,4.7K |              |RES BODY:100  CENTERS:500   |
|7   |1   |R2       |RES-1/4W,560  |              |RES BODY:100  CENTERS:500   |
|-----------------------------------------------------------------------------|
```

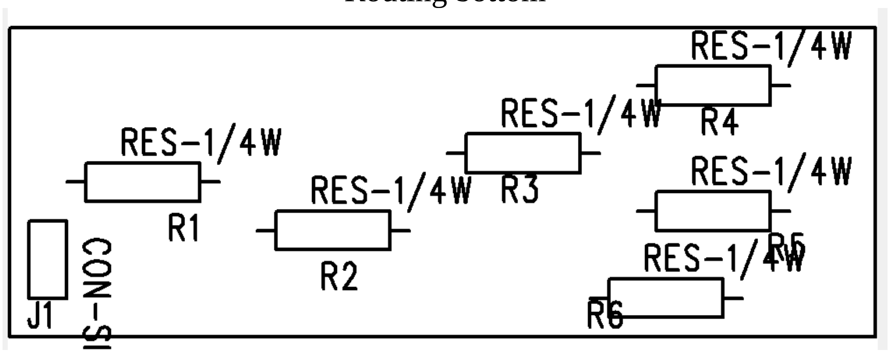


Schematic Diagram

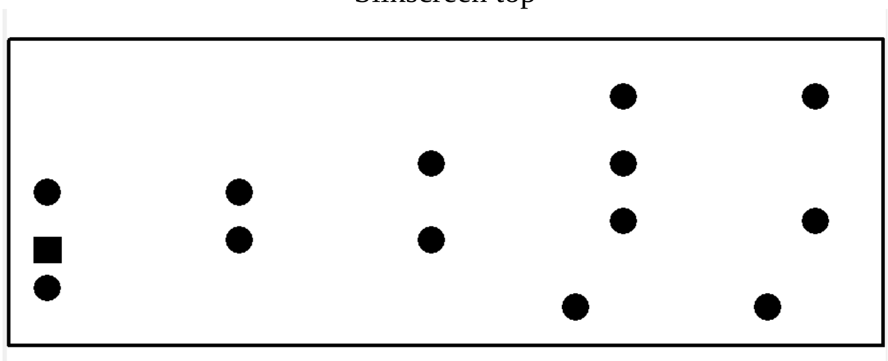
Output Files



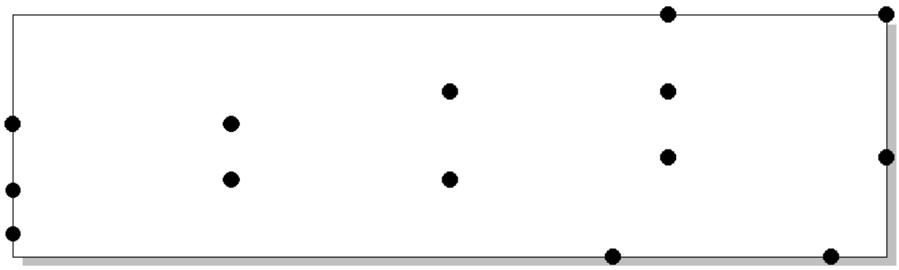
Routing bottom



Silkscreen top



Silkscreen bottom



Drill data

Laboratory Report Cover Sheet

SRM University Faculty of Engineering and Technology Department of Electronics and Communication Engineering
EC1002 Workshop- Electronics Engineering Practices **First Semester, 2014-15 (Even semester)**

Name :

Register No. :

Day / Session :

Venue : 2th Floor, BEL307 & BEL308, Engineering Basic Lab Building.

Title of Experiment :

Date of Conduction :

Date of Submission :

Particulars	Max. Marks	Marks Obtained
Design / Assembly / Testing/ Fabrication	15	
Lab Performance and Lab report	15	
Viva voce	10	
Total	**40**	

REPORT VERIFICATION

Date :

Staff Name :

Signature :

EXERCISE 4
Full Wave Rectifier Circuit

Objective:

1. To create a schematic with PADS Logic
2. To create a PCB Layout with PADS Layout
3. To use Copper Pour to create Power and GND planes

PART 1 – Schematic Capture

With the help of the parts list given, draw the schematic as shown in the General instructions. Save your schematic as Exercise2.sch. For your schematic, check the following:

1. Sheet Size is set to A3. Tie Dot Diameter is set to 80. Title Block is filled.
2. The component values are all correctly entered.
3. The pin numbers of the resistors are not displayed
4. Reference Designators are all upright and easy to read.
5. The netlist is named Exercise2.asc.

PART 2 – PCB Layout

1. Draw the Board Outline.
2. Set the **Origin** at the bottom-left corner of the board.
3. Import the Netlist and disperse the components.
4. Set the **design rules** as shown in the table below:

	Trace width			
Net Name	**Minimum**	**Recommended**	**Maximum**	**All clearance**
Default	10	12	15	12
+5V	15	20	30	20
GND	15	20	30	20

5. Place the components at your own discretion. Place them logically – refer to the schematic.
6. Before you route, read through the entire **Part 3** below on Copper Pouring.
7. For routing, ensure that **Prevent Error** is turned on. Route using Dynamic Route. Check that the line/trace Angle Mode is set to **Diagonal**.

PART 3 – Copper Pouring

In order to reduce noise in a design, copper planes are often added to a PCB. Other than to reduce noise in the power and ground supply, copper pour can also be used for thermal (heat) dissipation. The steps to add copper pour planes for a typical 2 layered board are:

1. Route all the **GND signal trace** on the Bottom layer - Use **Find>>Highlight>> GND** net to find all the GND nets and **route them on Bottom Layer.**

2. Route all the **+5V signal trace** on the Top Layer **-** Use **Find>>Highlight>>+5V** net to find all the +5V nets and **route them on the TOP layer**.

3. Route ALL the other signal connections – The other signal lines can be routed either on the top or bottom layers.

4. Create the **outline for the GND Copper Pour** area at the Bottom Layer and **associate it to GND**.

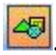
5. Similarly, create the **outline for the +5V Copper Pour area** and **associate it to the +5V**.

6. Finally Flood the areas.

Read on to learn how to create the Copper Pour Outlines and how to associate them to the correct nets.

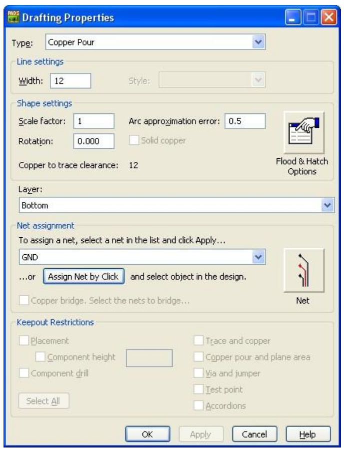
Draw the Copper Pour Outline

To create a copper pour outline on the bottom layer, 25 mils within the board outline:

1. Set the design grid to 25 (both X an Y) .
2. Set the current layer to the **Bottom** layer by selecting it from the Layer drop-down list box in the toolbar.
3. Turn on the **Drafting** Toolbar using the drafting icon .
4. Choose the **Copper Pour** icon  from the Drafting
5. Click on Right-mouse-button and choose **Polygon** from the pop-up menu.
6. Position the cursor at X25,Y25 and click to start drawing a polygon which is 25 mils offset from the board outline (polygon must be **within** the board outline)
7. Select **Complete** from the pop-up menu to complete or close the polygon.

Assign the Copper Pour Outline

1. Choose the **Select** Model icon  from the Drafting toolbox to exit Copper Pour mode.
2. Press the **Shift** key and click on any section of the outline to select the whole outline.
3. Next double-click the left mouse button and a **Drafting Properties** window will appear.



4. Select the **Layer** and assign the outline to the correct **Net,** as shown above.

Flood the Copper Plane

With the **Copper Pour Outline** still selected, select the **Flood** icon .
The message "***Proceed with Flood?"*** will appear. Answer **Yes** to the prompt.
Flooding creates a hatch pattern that fills the area enclosed by the pour outline.

Copper Pour Top Layer for the +5V Plane

Repeat the above steps, but this time for the +5V plane on the top layer. Create a copper pour outline, 50 mils within the board outline. Make sure that you associate the pour outline on the TOP layer to +5V before you flood.

PART 4 – Post Processing

1. Dimension the board outline.
2. Generate the following CAM files:

   Routing Bottom
   Silkscreen Top
   Solder mask Bottom
   NC Drill File

Measuring Voltage drop across resistive circuit

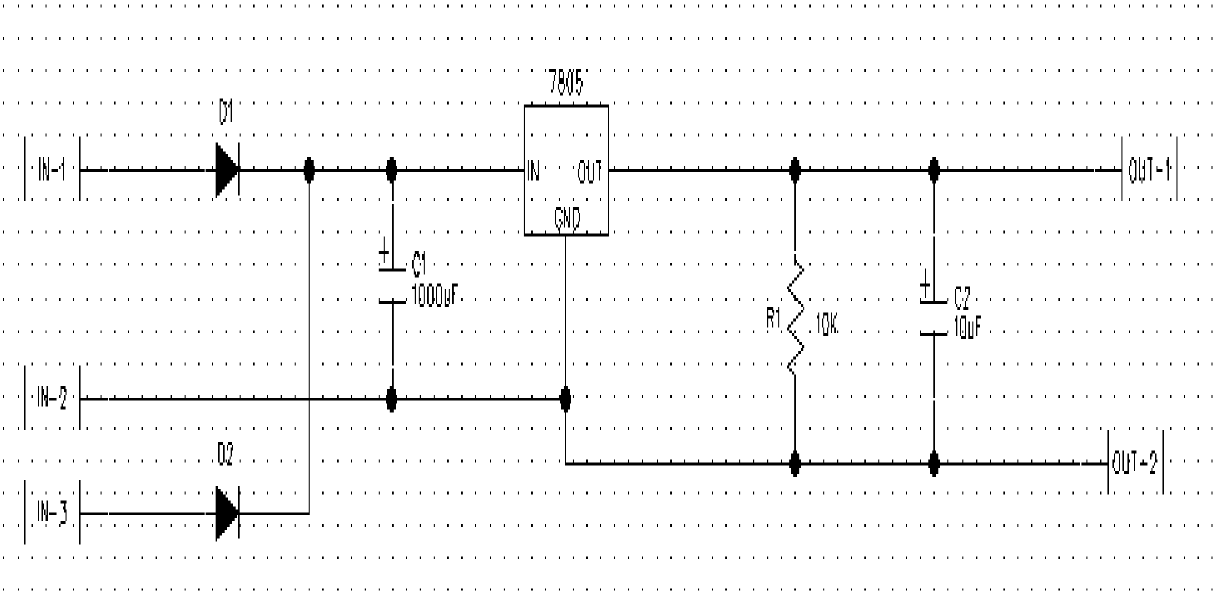
Part List

Library Part-type	Ref Designator	Components
RES - 1/4 W	R1 – R6	Resistor, size 1/4 Watt
CON-SIP-2P	J1	2 pins connector

BOM

|Bill Of Materials for schematic diagramt.sch on Thu Jul 04 14:58:36 2013 |

```
|------------------------------------------------------------------------------|
|Item|Qty |Reference|Part Name     |Manufacturer  |Description                 |
|----+----+---------+--------------+--------------+----------------------------|
|1   |1   |C2       |CAP-CX12-A,   |KEMET         |POL. DIPPED SOLID TANT. CAP.|
|    |    |         |???,10uF      |              |MIL-SPEC SIZE CX12 CASE 'A' |
|2   |1   |C1       |CAP-CX12-B,   |KEMET         |POL. DIPPED SOLID TANT. CAP.|
|    |    |         |???,1000uF    |              |MIL-SPEC SIZE CX12 CASE 'B' |
|3   |1   |OUT      |CON-SIP-2P    |              |GENERIC 2 PIN SIP HEADER .100|
|    |    |         |              |              |CENTERS                     |
|4   |1   |IN       |CON-SIP-3P    |              |GENERIC 3 PIN SIP HEADER .100|
|    |    |         |              |              |CENTERS                     |
|5   |2   |D1-2     |DIODE         |              |GENERIC DIODE W ALTERNATE   |
|6   |1   |7805     |LM7805CT      |NATIONAL      |5 VOLT, VOLTAGE REGULATOR   |
|    |    |         |              |SEMICONDUCTOR |                            |
|7   |1   |R1       |RES-1/8W,10K  |              |RES BODY:060 CENTERS:400  |
```



Schematic Diagram

OUTPUT FILES

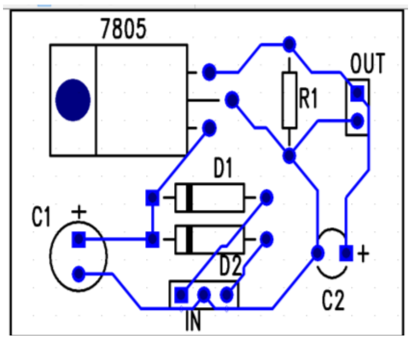


Figure 1 Bottom Routing

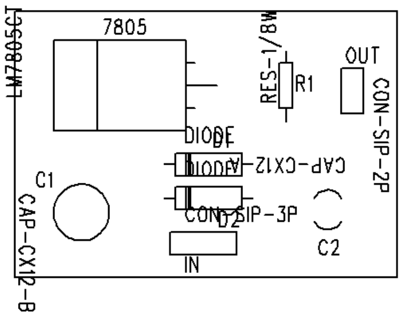


Figure 2 Silkscreen Top

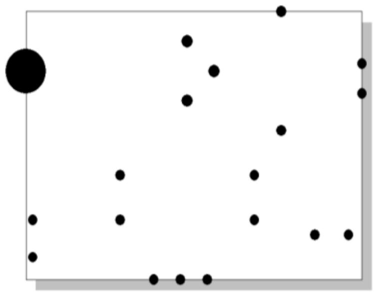


Figure 5 Drill Data

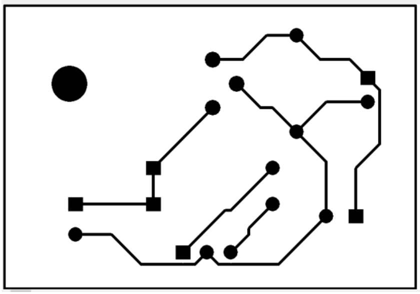


Figure 3 Bottom Routing

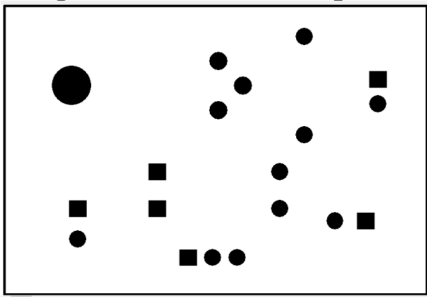


Figure 4 Solder mask bottom

Laboratory Report Cover Sheet

SRM University Faculty of Engineering and Technology Department of Electronics and Communication Engineering
EC1002 Workshop- Electronics Engineering Practices **First Semester, 2014-15 (Even semester)**

Name :

Register No. :

Day / Session :

Venue : 2th Floor, BEL307 & BEL308, Engineering Basic Lab Building.

Title of Experiment :

Date of Conduction :

Date of Submission :

Particulars	Max. Marks	Marks Obtained
Design / Assembly / Testing/ Fabrication	15	
Lab Performance and Lab report	15	
Viva voce	10	
Total	**40**	

REPORT VERIFICATION

Date :

Staff Name :

Signature :

<u>**EXERCISE 6**</u>

FULL WAVE RECTIFIER

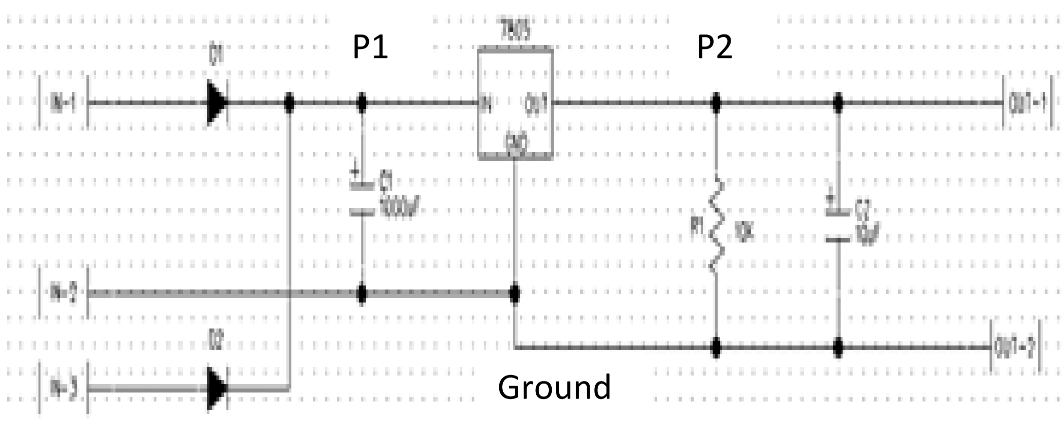
OBJECTIVE:

To assemble and testing of full wave rectifier circuit.

COMPONENTS REQUIRED:

S.NO	COMPONENTS NAME	VALUE	QUANTITY
1.	PCB for full wave rectifier circuit	-	1
2.	Digital multimeter(DMM)-	-	1
3.	Step-down transformer	(6-0-6)V	1
4.	Diode	IN4007	2 nos
5.	Capacitor	1000 µf/25V , 10 µf/12V	Each one
6.	Voltage regulator IC	IC7805	1
7.	Resistor	10kΩ	1
8.	soldering kit	-	1
9.	connecting wires	-	-

CIRCUIT DIAGRAM



PROCEDURE:

a. Components measurement

1. Connect probes: black probe to COM terminal and red probe to terminal marked with'Ω'.
2. Set the functions of resistor or capacitor or diode measurement.
3. Set the appropriate range.
4. Connect the two probes to the resistor or capacitor or diode to make measurement.
5. Note the reading and check the resistor value using colour coding technique.

6. Note the reading and check the capacitor or diode values.

b. Assembly procedure:

1. Load the components on to the dedicated component layout printed circuit board. Components must be placed well in the board.
2. Bend the resistor, capacitor and diode leads to the appropriate size before inserting them into the board. Make mechanical bend before cutting away the excess leads. Height of the each resistor or capacitor or diode must be in the same level.
3. Solder the components on the general purpose board using the point to point soldering as shown in the circuit diagram. Solder two power connection leads and connect it to the power supply whenever it's required.

TESTING OF CIRCUIT:

1. Check point to point continuity test for all the points in the PCB.
2. Measurement of voltage:
   a. Connect the power supply with the primary coils of transformer and measure the voltage across each terminals using digital multimeter.
   b. Measure the voltage across the diode P1 and the ground terminal.
   c. Measure the voltage across the diode P2 and the ground terminal.
   d. Measure the AC input voltage across the diode D1 anode and ground.
   e. Measure the AC input voltage across the diode D2 anode and ground.
   f. Measure the voltage drop across pin 1 and pin 3 of 7805 (P1 and P2)

S. No	**Measuring nodes**	**Voltage measurement**
1.	P1 and ground	
2.	P2 and ground	
3.	AC input voltage at D1 anode and ground	
4.	AC input voltage at D1 anode and ground	
5.	voltage drop across pin 1 and pin 3 of 7805	

TROUBLE SHOOTING:

1. If the capacitor is unplugged or the interchanging of positive and negative terminals or improper soldering of capacitor terminal doesn't produce any kind of output.
2. If the capacitor value differs from the mentioned values then charging and discharging period of capacitor vary.

3. When the diode is in forward bias (positive terminal of power supply connected with anode and negative terminal of power supply connected with cathode of diode) condition it allows the current flow across the diode. If the diode terminals (anode and cathode) are interchanged i.e., reverse biased then no voltage drop across the diodes.
4. If the diode is unplugged or not soldered properly then no output occurs.
5. When the diode is soldered partially then distorted output may occur.
6. When the resistor value differs from the circuit, the output value can differ.
7. If no voltage appears across the capacitor 10 µf/12V, check the connecting point of diode D1 and D2 with DMM.
   a. If voltage appear across D1 and D2 then check the voltage across the capacitor 1000 µf/25V
   b. If no voltage drops across D1 and D2 then replace the diodes or look at the continuity test at that particular point.
   c. If no dropping voltage across the capacitor 1000 µf/25V then replace the capacitor or go to continuity test at that point or check the short circuit across the track of PCB.
   d. If the voltage drops across the capacitor1000 µf/25V then check the continuity test of voltage regulator.
   e. If no voltage appears across voltage regulator terminal then replace the component or check the short circuit across the track of PCB.
   f. If any short circuit at the track of PCB then use the scratch card for open circuit.
   g. If no voltage appear across the resistor then go to the continuity test of look the condition of resistor(broken/burn).
   h. If voltage drop across the resistor then the problem with the capacitor 10 µf/12V. now look the continuity test of capacitor and replace it.

RESULT:

EXERCISE 7
STUDY OF SINGLE SIDE PCB FABRICATION

AIM

- To study the single side PCB fabrication process by photo resist method.
- To get familiarize with the materials, precautions to be observed and the fabrication procedure of NV1800 PCB fabrication machine setup.

EQUIPMENTS REQUIRED

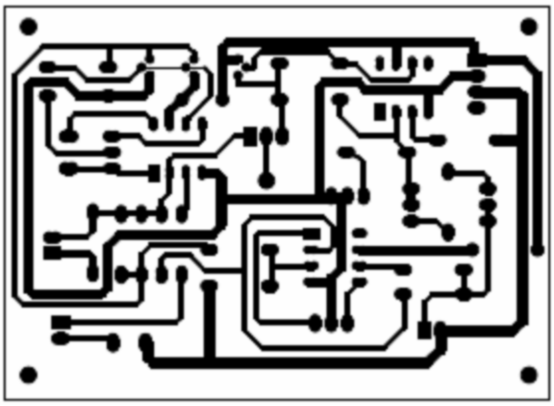
NV1800 PCB FABRICATION MACHINE

1. PCB Art Work Film Maker – NV180
2. PCB Art Work Table – NV181
3. PCB Shearing Machine – NV182
4. Photoresist Dip Coating Machine – NV183
5. PCB Curing Machine - NV191
6. UV Exposing Machine – NV184
7. Dye Tank – NV185
8. Development Tank – NV186
9. PCB Etching Machine – NV187
10. Drilling Machine – NV188
11. Solderable Lacquer Tank – NV189

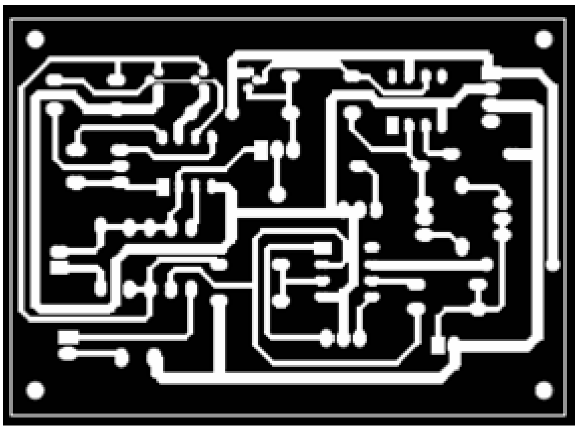
1. PCB Art Work Film Maker – NV180

PCB Layout can be system generated or hand drawn. The PCB artwork film maker is used to develop art work film. The film development is a dark room process. The lith film developed in this process is called negative. The negative and positive refers to the way the film represents the actual art work.

Positive Film



Negative Film



2. PCB Art Work Table – NV181

The artwork table is used for inspecting and correcting the film developed by Art work film maker. A new film can be directly developed with the help of accessories provided in PCB designer kit (NV190). The direct films can be developed for small circuits and it becomes complex with increase in circuit complexity.

3. PCB Shearing Machine – NV182

PCB Shearing Machine is used for cutting PCB laminate to the desired dimensions.

4. Photoresist Dip Coating Machine – NV183

PCB laminate cleaning is very important before photoresist coating. This PCB laminate coated with photoresist is used for further PCB development. The Photoresist creates a layer on the PCB laminate.

i. Take the cut PCB laminate from the shearing machine in previous procedure. **Cleaning with Cleaning Reagent- 1**
ii. Dry the PCB laminate in the air first and then in PCB Curing Machine. During drying process the care should be taken that spots should not occur on PCB laminate.
iii. **Photoresist** coating is done by mounting the PCB on the dipping assembly.
iv. Dip the PCB laminate slowly in the tank till the photoresist is coated on the PCB laminate.
v. Take out the PCB dipping assembly from the stand.

5. PCB Curing Machine (NV191).

Place the PCB laminate in the PCB Curing machine and leave the Laminate for 10 minute.

6. UV Exposing Machine – NV184

i. Place the negative on the glass of the UV exposure machine such that the solder side is in contact with the glass plate.
ii. Place the photoresist coated PCB on the negative.
iii. Close the lid of the UV exposure machine.
iv. The exposing time is about 15 minutes to 20 minutes.
v. After exposing the PCB is ready for development process.

7. Development Tank – NV186

UV exposed PCB laminate is dipped in the developer quite for few minutes and then processed for next step which is dye tank process. . Basically developer is used to remove the extra photoresist coating.

i. If no tracks are visible dip it again for 5 sec. Repeat the process until tracks are clearly visible. This process removes unused photoresist from the PCB that is not exposed to the UV light i.e. the darker side of the Negative Film.

ii. Suitable time for PCB laminate to be dipped in development tank is 1 to 5 minutes. The time must not be varied as the excess time can damage tracks and time less than the required time does not allow the developer to remove the extra photoresist.

8. Dye Tank – NV185

Dye tank is used to dip the PCB laminate after development tank process. Role of the dye tank process is to make the tracks formed in the development tank process visible.

9. PCB Etching Machine – NV187

PCB etching machine is used for the etching process. Actually the tank of the machine is filled with the ferric chloride solution in water. This solution reacts with the copper of the PCB laminate when the laminate is dipped in the solution. The uncovered copper of the laminate reacts with the ferric chloride and is removed from the PCB laminate leaving the copper tracks only which were protected actually due to the photoresist coating and dye.

i. Mount the PCB Laminate on the Stand provided for PCB etching machine. Dip it in the PCB etching machine.
ii. The required time duration to pump the PCB etching machine “ON” is entered using the push buttons and display provided on the front panel of the PCB etching machine.
iii. As soon as the spray starts functioning the ferric chloride solution goes whirling and the process of the etching starts for the time duration as entered.
iv. When the time is over and the spray ceases and the etching process is complete.
v. Open the lid of the PCB etching machine and remove out the PCB laminate from the etching machine. Check the PCB for complete etching. If the unused copper is not completely removed then start the process again from step ii.
vi. The etching time depends on the concentration of etching solution and the size of PCB.
vii. Wash the PCB laminate with cleaning reagent 2 and dry it, the process of etching is complete.

10. Drilling Machine – NV188

The PCB must be drilled as per components used in the design of the PCB

i. Place the PCB on the base of drilling machine provided for the support of drilling.
ii. Placing will be such that the drill bit will go straight to the place where hole is to be made.
iii. Drill slowly creates holes.
iv. Similarly all the holes are made and thus drilling is complete.

11. Solderable Lacquer Tank – NV189

Lacquer is chemical which creates a protecting layer on PCB for protecting it from environment reactions like oxidation.

Result

Laboratory Report Cover Sheet

SRM University
Faculty of Engineering and Technology
Department of Electronics and Communication Engineering

EC1002 Workshop- Electronics Engineering Practices

First Semester, 2014-15 (Even semester)

Name :

Register No. :

Day / Session :

Venue : 2th Floor, BEL307 & BEL308, Engineering Basic Lab Building.

Title of Experiment :

Date of Conduction :

Date of Submission :

Particulars	Max. Marks	Marks Obtained
Design / Assembly / Testing/ Fabrication	15	
Lab Performance and Lab report	15	
Viva voce	10	
Total	**40**	

REPORT VERIFICATION

Date :

Staff Name :

Signature :

EXERCISE 5
Measuring Voltage drop across resistive circuit

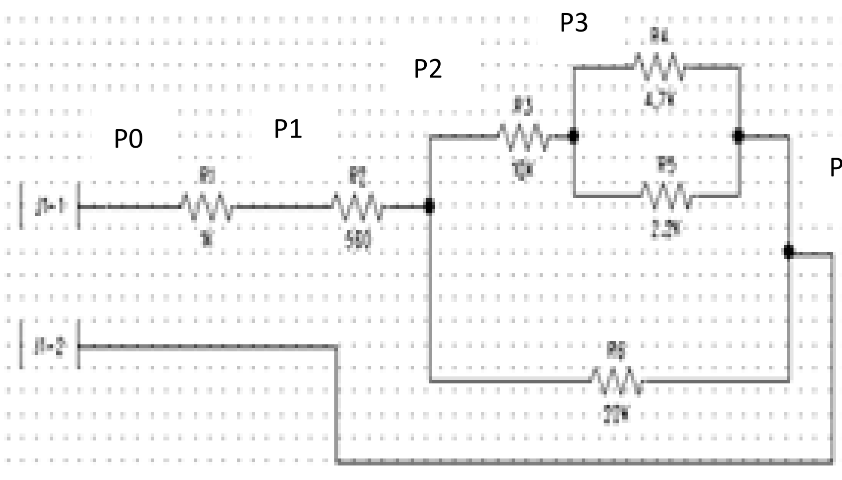
OBJECTIVE:

To assemble and testing of voltage drop across resistive circuit.

COMPONENTS REQUIRED:

S.NO	COMPONENTS NAME	VALUE	QUANTITY
1.	PCB for voltage drop resistive circuit	-	1
2	Digital multimeter(DMM)-	-	1
3	Resistors	1 kΩ ,500 Ω,10 kΩ,4.7 k,2.2 kΩ,22 kΩ	Each 1 no
4	DC power supply,connecting wires	-	1
5	soldering kit	-	1

CIRCUIT DIAGRAM:



PROCEDURE:

1) Resistance measurement using digital multimeter.

a) Connect BLACK probe to COM terminal and RED probe to terminal marked with ohm.
b) Set the functions to resistance measurement
c) Connect the two probes to the resistor to make measurements.
d) Note the reading and also check the value using color coding technique.

2) ASSEMBLY PROCEDURE

a) Load the resistor onto the dedicated component lay out Printed circuit board. Resistors should be placed well in the board. Bend the resistors lead to the appropriate size before inserting

them into the board. Make a mechanical bend before cutting away the excess lead. Height of each resistor should be in the same level.
b) Solder the resistor on general purpose PCB board using point to point soldering as shown in circuit diagram
c) Solder two power connection leads and connect a 5v power supply to the constructed circuit.

3) Testing of circuit

a) Set the DC power supply to 5v.
b) Connect the positive terminals to point P1 and negative terminal to P5.
c) Measure the voltage V1 between P0 and P1.
d) Measure the voltage V2 between P1 and P2.
e) Measure the voltage V3 between P2 and P3.
f) Measure the voltage V4 between P3 and P4.
g) Measure the voltage V5 between P2 and P4. Calculate Vb=(v4+v5)

V1	
V2	
V3	
V4	
V5	

Calculate Vin=(V1+V2+V3+V4)= + + + + =-------V
Calculate Vin=(V1+V2+V5)= + + =-------V

4) TROUBLE SHOOTING:

a) Set the digital multimeter the continuity mode. Unplug the device or switch off the circuit before attempting a continuity test. Check the continuity between all five nodes.

b) Set different voltage (eg:4v, 7v) and check the voltage across each point.

C) When the resistor value differs from the circuit, the output value can differ.

d) When the Specified value of power supply increased or decreased, then the voltage across each resistor value get increased or decreased.

RESULT: