SRM VALLIAMMAI ENGINEERING COLLEGE

(An Autonomous Institution)

SRM NAGAR, KATTANKULATHUR – 603 203.

DEPARTMENT OF MEDICAL ELECTRONICS

LABORATORY MANUAL

1910604 – MEDICAL ELECTRONIC SYSTEM DESIGN LABORATORY

Regulation - 2019

Semester/Branch	**:VI Semester / MEDICAL ELECTRONICS**
Academic Year	**:2024 -25(EVEN)**
Prepared By	**:Mr.Satheeswaran.C AP(OG)**

SRM VALLIAMMAI ENGINEERING COLLEGE

(An Autonomous Institution)

SRM Nagar, Kattankulathur – 603 203

DEPARTMENT OF MEDICAL ELECTRONICS

VISION OF THE INSTITUTE

Educate to excel in social transformation

MISSION OF THE INSTITUTE

- To contribute to the development of human resources in the form of professional engineers and managers of international excellence and competence with high motivation and dynamism, who besides serving as ideal citizen of our country will contribute substantially tothe economic development and advancement in their chosen areas of specialization.
- To build the institution with international repute in education in several areas at several levels with specific emphasis to promote higher education and research through strong institute-industry interaction and consultancy.

VISION OF THE DEPARTMENT

To excel in the field of electronics and communication engineering and to develop highly competent technocrats with global intellectual qualities.

MISSION OF THE DEPARTMENT

M1: To educate the students with the state of art technologies to compete internationally, able to produce creative solutions to the society`s needs, conscious to the universal moralvalues, adherent to the professional ethical code

M2: To encourage the students for professional and software development career

M3: To equip the students with strong foundations to enable them for continuing education and research.

SRM VALLIAMMAI ENGINEERING COLLEGE

(An Autonomous Institution)
SRM Nagar, Kattankulathur – 603 203

PROGRAM OUTCOMES

1. **Engineering knowledge:** Apply the knowledge of mathematics, science, engineering fundamentals, and an engineering specialization to the solution of complex engineering problems.
2. **Problem analysis:** Identify, formulate, review research literature, and analyze complex engineering problems reaching substantiated conclusions using first principles of mathematics, natural sciences, and engineering sciences.
3. **Design/development of solutions:** Design solutions for complex engineering problems and design system components or processes that meet the specified needs with appropriate consideration for the public health and safety, and the cultural, societal, and environmental considerations.
4. **Conduct investigations of complex problems:** Use research-based knowledge and research methods including design of experiments, analysis and interpretation of data, and synthesis of the information to provide valid conclusions.
5. **Modern tool usage:** Create, select, and apply appropriate techniques, resources, and modern engineering and IT tools including prediction and modeling to complex engineering activities with an understanding of the limitations.
6. **The engineer and society:** Apply reasoning informed by the contextual knowledge to assesssocietal, health, safety, legal and cultural issues and the consequent responsibilities relevant to the professional engineering practice.
7. **Environment and sustainability:** Understand the impact of the professional engineering solutions in societal and environmental contexts, and demonstrate the knowledge of, and need for sustainable development.
8. **Ethics:** Apply ethical principles and commit to professional ethics and responsibilities and norms of the engineering practice.
9. **Individual and team work:** Function effectively as an individual, and as a member or leader in diverse teams, and in multidisciplinary settings.
10. **Communication:** Communicate effectively on complex engineering activities with the engineering community and with society at large, such as, being able to comprehend and write effective reports and design documentation, make effective presentations, and give and receive clear instructions.
11. **Project management and finance:** Demonstrate knowledge and understanding of the engineering and management principles and apply these to one's own work, as a member and leader in a team, to manage projects and in multidisciplinary environments.
12. **Life-long learning:** Recognize the need for, and have the preparation and ability to engage in independent and life-long learning in the broadest context of technological change.

PROGRAM SPECIFIC OUTCOME(PSOs)

PSO1: Ability to apply the acquired knowledge of basic skills, mathematical foundations, principles of electronics, modeling and design of electronics based systems in solving engineering Problems.
PSO2: Ability to understand and analyze the interdisciplinary problems for developing innovative sustained solutions with environmental concerns.
PSO3: Ability to update knowledge continuously in the tools like MATLAB, NS2, XILINIX and technologies like VLSI, Embedded, Wireless Communications to meet the industry requirements.
PSO4: Ability to manage effectively as part of a team with professional behavior and ethics.

SYLLABUS

1910604 – MEDICAL ELECTRONICS SYSTEM DESIGN LABORATORY

OBJECTIVES:

The student should be made:

- To familiarize with the electronic components used in medical electronics system.
- To enhance the use of sensors used for medical applications.
- To design electronic circuits for various signal processing applications.
- To perform PCB design using software and to explain the various processes involved.
- To provide the knowledge in assembling and testing of the PCB based Medical electronic circuits.

LIST OF EXPERIMENTS

1. Design a Power Supply for Low Power Wearable Devices.
2. Design of a Bio Amplifier.
3. Design and setup a notch filter circuit and Active Band Pass Filter.
4. Design and setup a threshold detector, sample and hold circuit using op-amp.
5. Design and setup a Patient Isolation Circuit.
6. Design of body temperature measuring circuit using thermistors.
7. Design of Plethysmography circuit.
8. Design of Pace Maker circuit.
9. Design and setup a circuit for skin contact impedance.
10. Design of Bio-Telemetry using IC4046.
11. Design of Bio-Electrode Equivalent Circuit.
12. Study of PCB design software (open source) like KiCad, Eagle, etc.

TOTAL: 60 PERIODS

OUTCOMES:

At the end of the course, the student should be able to:

- Understand the usage of electronic components and medical sensors used in medical electronics system.
- Create electronic circuits for various signal processing applications in medical electronic system.
- Conduct experiments using designed and assembled circuits for medical applications.
- Design and simulate various electronic PCB required for prototyping and testing using software tools and testing equipments.
- Identify, formulate, and solve engineering problems associated with assembly and testing of Medical electronic circuits.

LIST OF EXPERIMENTS

CYCLE-I

1. Design a Power Supply for Low Power Wearable Devices.
2. Design of a Bio Amplifier.
3. Design and setup a notch filter circuit and Active Band Pass Filter.
4. Design and setup a threshold detector, sample and hold circuit using op-amp.
5. Design and setup a Patient Isolation Circuit.
6. Design of body temperature measuring circuit using thermistors.

CYCLE-II

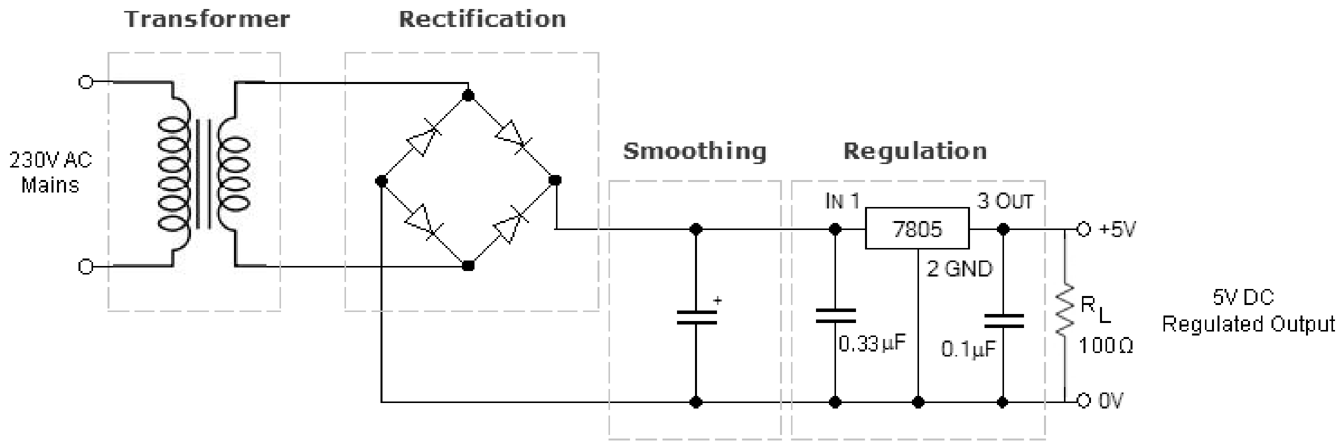
1. Design of Plethysmography circuit.
2. Design of Pace Maker circuit.
3. Design and setup a circuit for skin contact impedance.
4. Design of Bio-Telemetry using IC4046.
5. Design of Bio-Electrode Equivalent Circuit.
6. Study of PCB design software (open source) like KiCad, Eagle, etc.

ADDITIONAL EXPERIMENTS

1. Characteristics of PLL.

INDEX

CIRCUIT DIAGRAM



EXPT. NO: 1 DESIGN OF POWER SUPPLY FOR LOW POWER WEARABLE DEVICES

AIM:

To design a power supply for low power wearable devices.

APPARATUS REQUIRED

S.NO	ITEM	RANGE	QUANTITY
1.	Diode	1N4007	04
2.	IC	IC7805	01
3.	Step down transformer	230 V/ 9 V, 1A	01
4.	Resistor	100Ω	01
5.	Capacitor	0.33μF, 0.1μF	Each 1
6.	Electrolytic Capacitor	1000μF/25V	01
7.	Function Generator	3 MHz	01
8.	RPS	(0-30)V	01
9.	CRO	30 MHz	01
10.	Bread Board		01
11.	Connecting Wires	Single stand	few

PRE-LAB QUESTIONS

1. Design and create a SPICE model of a bridge-type, full-wave rectified, dc power supply using a filter capacitor.
2. Analyze the circuit for different values of filter capacitors. Observe the change in ripple content and comment on your observation.
3. Analyze the circuit for different load conditions. Observe the change in ripple content and comment on your observation.
4. From the IC 7805 datasheet, write down the minimum, typical and maximum values of the output voltage V0.
5. Determine the smallest value of the input voltage VI for which IC7805 can still work as a voltage regulator.

THEORY:

Every electronic circuit is designed to operate off of supply voltage, which is usually constant. A regulated power supply provides this constant DC output voltage and continuously holds the output voltage at the design value regardless of changes in load current or input voltage.

The power supply contains a rectifier, filter, and regulator. The rectifier changes the AC input voltage to pulsating DC voltage. The filter section removes the ripple component and provides an unregulated DC voltage to the regulator section. The regulator is designed to deliver a constant voltage to the load under varying circuit conditions. The two factors that can cause the voltage across the load to vary are fluctuations in input voltage and changes in load current requirements.

Load regulation is a measurement of power supply, showing its capacity to maintain a constant voltage across the load with changes in load current. Line regulation is a measurement of power supply, showing its capacity to maintain a constant output voltage with changes in input voltage.

DESIGN

Design a 5 V DC regulated power supply to deliver up to 1A of current to the load with 5% ripple. The input supply is 50Hz at 230 V AC.

Selection of Voltage regulator IC:

Fixed voltage linear IC regulators are available in a variation of voltages ranging from -24V to +24V. The current handling capacity of these ICs ranges from 0.1A to 3A. Positive fixed voltage regulator ICs have the part number as 78XX. The design requires 5V fixed DC voltage, so 7805 regulator IC rated for 1A of output current is selected.

Selection of Bypass Capacitors:

The data sheet on the 7805 series of regulators states that for best stability, the input bypass capacitor should be 0.33µF. The input bypass capacitor is needed even if the filter capacitor is used. The large electrolytic capacitor will have high internal inductance and will not function as a high frequency bypass; therefore, a small capacitor with good high frequency response is required. The output bypass capacitor improves the transient response of the regulator and the data sheet recommends a value of 0.1µF.

Dropout voltage:

The dropout voltage for any regulator states the minimum allowable difference between output and input voltages if the output is to be maintained at the correct level. For 7805, the dropout voltage at the input of the regulator IC is Vo +2.5 V.

$V_{dropout} = 5+2.5 = 7.5V$

Selection of Filter Capacitor:

The filter section should have a voltage of at least 7.5V as input to regulator IC.

That is Vdc = 7.5 V

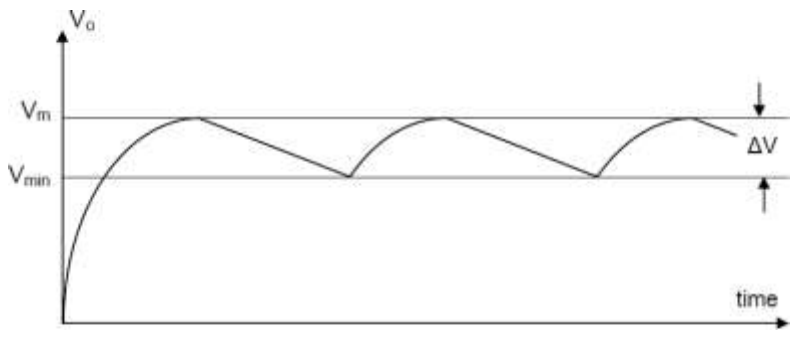


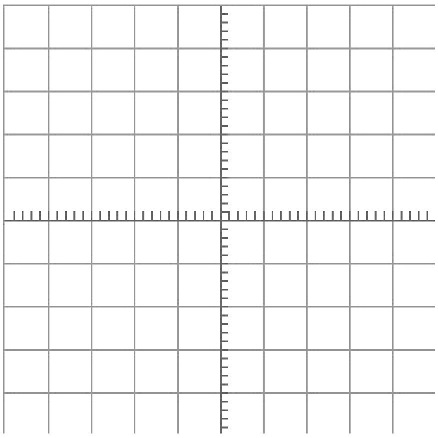
Figure 2: Output wave shape from a full-wave filtered rectifier

PROCEDURE:

1. Power Supply

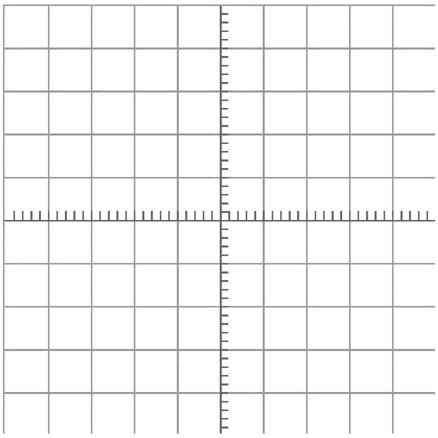
1. Connect the circuit as shown in Figure 1 .
2. Apply 230V AC from the mains supply.
3. Observe the following waveforms using oscilloscope

Volt/div = Time/div =



Graph 1: Waveform at the secondary of the transformer

Volt/div = Time/div=



Graph 2: Waveform after rectification

Volt/div = Time/div =

Volt/div = Time/div=

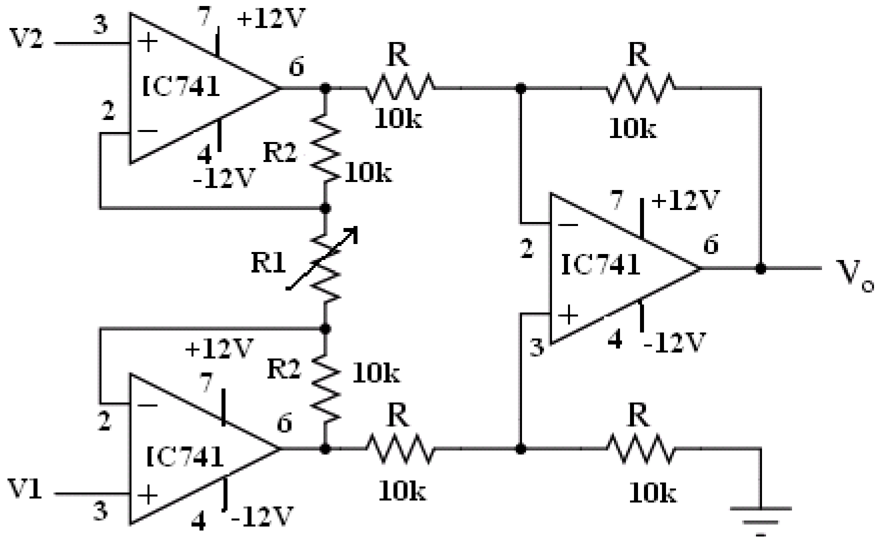
POST-LAB QUESTIONS

1. Why is the ripple voltage larger at full load?
2. Under full load conditions, what is the power dissipated by the regulator IC?
3. Comment on the efficiency of the circuit for a minimum output voltage and a maximum output voltage.
4. Identify the short-circuit current of 7805 from data sheets.
5. What modification needs to be done to obtain a variable output voltage?

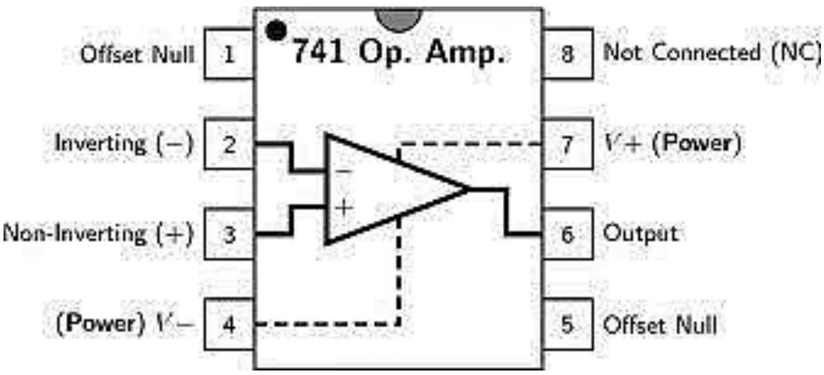
RESULT:

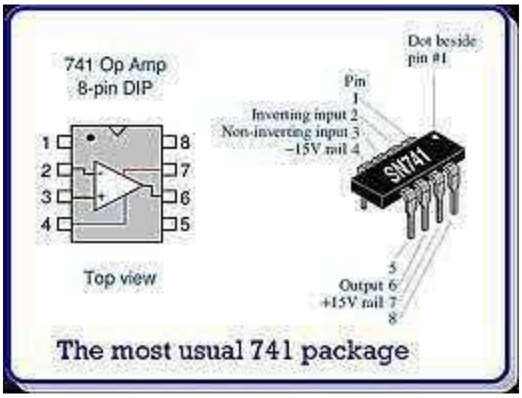
Thus the regulated power supply was designed for low power wearable devices.

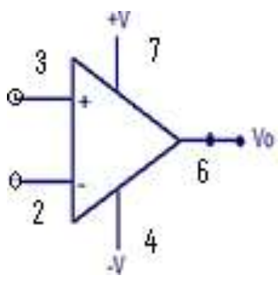
CIRCUIT DIAGRAM:



PIN DIAGRAM







EXP NO .2 DESIGN OF BIO-AMPLIFIER

AIM:

To design and setup a bio amplifier for a gain of 10 and to calculate the CMMR.

DESCRIPTION

Input : Set the dc input voltage in both the input terminals V1 and V2.
Output : Vary the resistor R1 to obtain the output voltage with varying gain.

APPARATUS REQUIRED :

S.NO	ITEM	RANGE	QUANTITY
1	Op-amp	IC741	1
2	Resistors	10kΩ	6
3	Decade Resistance box	-	1
4	Multimeter	-	1
5	RPS	DUAL(0-30) V	2

PRE-LAB QUESTIONS

1. What is an op-amp? Why it is called so?
2. Define the term input offset voltage, input bias current, gain bandwidth product.
3. Define slew rate in an op-amp.
4. List the ideal characteristics of an op-amp.
5. What are the building blocks of an op-amp?

DESIGN:

Assume the resistors in the differential amplifier stage to be of the same value R =10kΩ

Using Superposition theorem the output of the Instrumentation amplifier is

$V_o = -(R/R)V_a + (1+R/R)V_b = V_b-V_a$

Where $V_a = -R2/R1(V1-V2) +V2$

and $V_b = R2/R1(V1-V2)+V1$

Thus $V_o = V_b-V_a =(V1-V2)\ (1+2R2/R1)$

Gain of an Instrumentation amplifier is

$Av = V_o/(V1-V2) = (1+2R2/R1)$

For $Av = 3$

$3=(1+2R2/R1)$

Then $R2 = R1$

Let R2 = 10kΩ then set the variable resistor R1 = 10KΩ

THEORY:

Instrumentation Amplifiers are high gain differential amplifiers with high input impedance and a single ended output. They are mainly used to amplify very small differential signals from strain gauges, thermocouples or current sensing resistors in motor control systems. They also have very good common mode rejection in excess of 100dB at DC. The negative feedback of the top op-amp causes the voltage at Va to be equal to the input voltage V1. Likewise, the voltage at Vb is equal to the value of V2. This produces a voltage drop across R1 which is equal to the voltage difference between V1 and V2. This voltage drop causes a current to flow through R1, and as the two inputs of the buffer op-amps draw no current (virtual earth), the same amount of current flowing through R1 must also be flowing through the two resistors R2. This voltage drop between points Va and Vb is connected to the inputs of the differential amplifier which amplifies it by a gain of 1 (assuming that all the "R" resistors are of equal value). Then the overall voltage gain of the instrumentationamplifier circuit is
The differential gain of the circuit can be changed by changing the value of R1.

$$V_{out} = (V_1 - V_2)\left[1 + \frac{2R_2}{R_1}\right]$$

TABULATION :

V1 (Volts)	V1 (Volts)	R1 (Ohms)	Vo (Volts)	Gain	
				Theoretical 1+2R2/R1	**Practical V0/V1-V2**

Thus the gain of the instrumentation amplifier can be varied over a range of <u>1.5 to 1.2</u> by varying the variable resistance R1 from <u>20K to 50K</u>.

PROCEDURE :

1. Connect the components as per the circuit diagram.
2. Initially set the variable resistor R1 to the minimum value and note down the output voltage and compute the gain.
3. Repeat the procedure in step 2 for various values of R1 and note down the corresponding output voltage to find the range of gain.

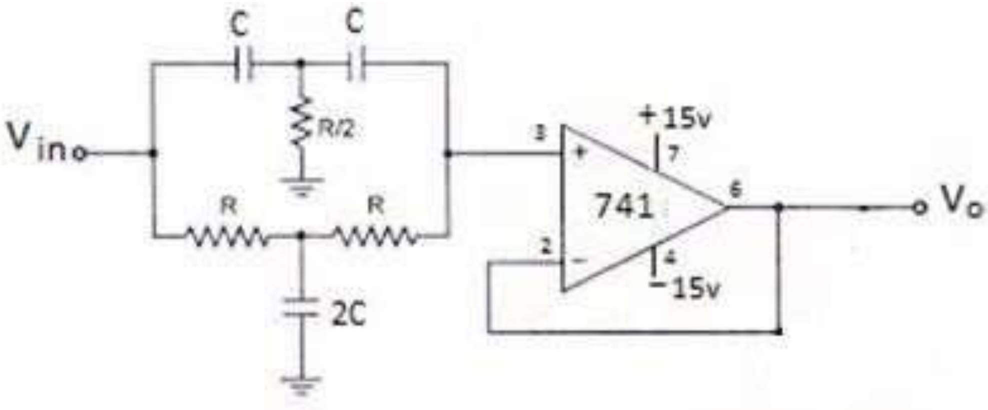
PRE-LAB QUESTIONS

1. What are the advantages of negative feedback in an op-amp?
2. List the linear and non-linear applications of an op-amp.
3. What is an instrumentation amplifier ?
4. List some applications of an instrumentation amplifier.

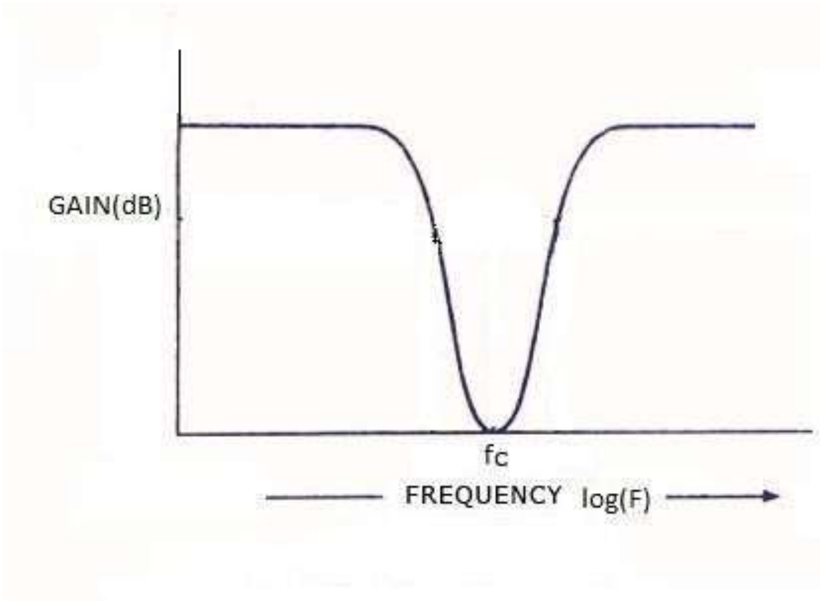
RESULT :

Thus the bio amplifier was designed using Op-Amp 741 IC and the gain was calculated.

CIRCUIT DIAGRAM-NOTCH FILTER



MODEL GRAPH



Frequency Response

EXP NO .3(A) DESIGN AND SETUP A NOTCH FILTER CIRCUIT

AIM:

(i) To design and setup a notch filter for a notch out frequency of 50Hz and plot the frequency response characteristics.

.

COMPONENTS REQUIRED:

S.NO	ITEM	RANGE	QTY
1.	IC	741	03
2.	Resistor	15KΩ, 33KΩ , 1KΩ, 1.5KΩ 5KΩ, 10 KΩ	Each 2
3.	Capacitor	0.1 µF,0.2 µF	Each 2
4.	Function Generator	2MHz	01
5.	RPS	(0-30)V	01
6.	DSO	30MHz	01
7.	Bread Board		01
8.	Connecting Wires	Single stand	few

PRE-LAB QUESTIONS

1. What is the stop band frequency of a notch filter?
2. How deep should the notch of an active notch filter be?
3. What are the disadvantages of a notch filter circuit?

THEORY:

Filter is a frequency selective circuit that passes electric signals of specified band of frequencies and attenuates the signals of frequencies outside the band. Notch filter is actually a 12 narrow band reject filter. This is because of its higher Q (> 10) it is commonly used for the rejection of a single frequency. The most commonly used notch filter is the twin –T network.

This network consists of two T networks, one consisting of two resistors and a capacitor and the other consisting of two capacitors and a resistor. At the notch frequency, the filter shows maximum attenuation and the filter will pass all other frequency signals. The notch out frequency is given by

$$f_N = 1 / 2\pi RC$$

The passive twin T-network has a relatively low figure of merit Q. To increase the value of Q, a voltage follower network is connected at the end of the twin T network. They are generally used in communication and bio medical instruments for eliminating the undesired frequencies like power line frequency.

Design:

Notch frequency $F_N = 1/2\pi RC$

$F_N = 50Hz$, Let $C = 0.1\mu F$

$R = 1/(2 \times 3.14 \times 50 \times 0.1 \times 10^{-6}) = 31847\Omega \approx 33K\Omega$

TABULATION

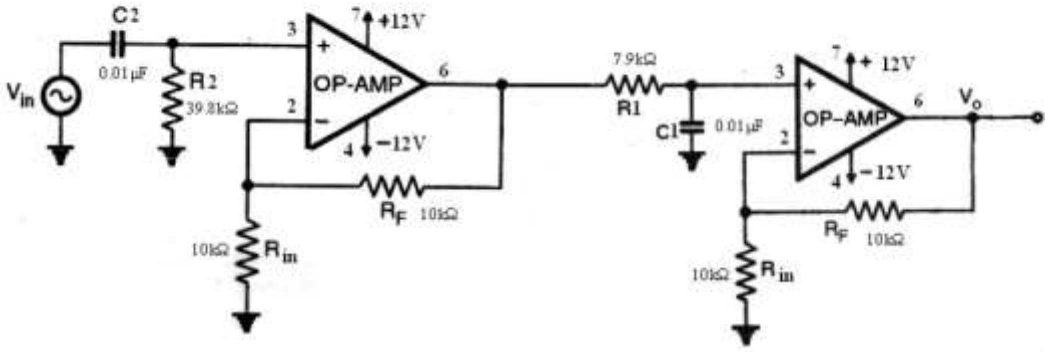
Vi = 1 Vpp

F(Hz)	Log F	V_0(V)	Gain (dB)

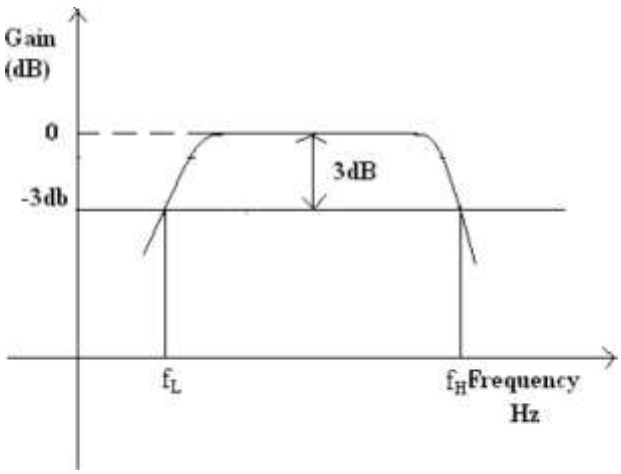
PROCEDURE:

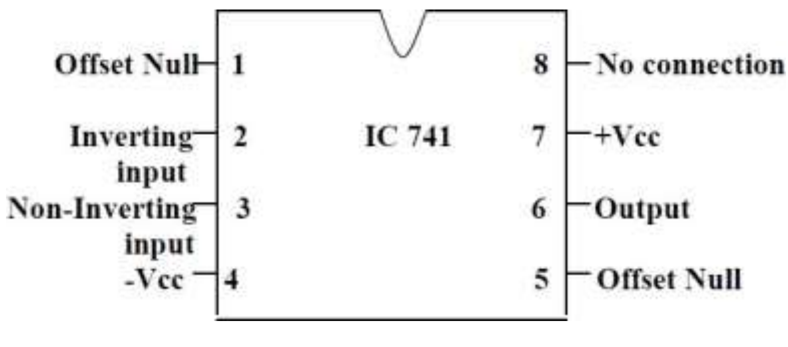
1. Setup circuit on the bread board.
2. Connect the ±15V power supply.
3. Feed a 1 Vpp sinusoidal input and vary the frequency from 10Hz to 1 KHz
4. Vary the frequency of the sine wave in steps and note down the corresponding output voltage.
5. Plot the frequency response curve on a graph sheet.
6. Find the notch frequency.

CIRCUIT DIAGRAM-BPF



MODEL GRAPH





EXP NO .3 (B) **DESIGN AND SETUP AN ACTIVE BAND PASS FILTER.**

AIM:

Design a second order active Butterworth low pass filter using IC 741 having upper cut-off frequency 1 KHz, also determine its frequency response and Design a second order active band pass pass filter using IC 741.

COMPONENTS REQUIRED:

S.NO	ITEM	RANGE	QUANTITY
1	Op-amp	IC741	1
2	Resistors	10KΩ,1.5KΩ,5.6 KΩ	1
3	Capacitor	0.1 μF	1
4	CRO	-	1
5	RPS	DUAL(0-30) V	1

PRE-LAB QUESTIONS

1. What is an op-amp? Why it is called so?
2. List the ideal characteristics of an op-amp.
3. What are the building blocks of an op-amp?
4. What is an electric filter?
5. State the advantages of an active filter

DESIGN:

Given:
Upper cut-off frequency f_H = 1 kHz
$f_H = 1/(2\pi RC)$
Let C = 0.1 μF, R = 1.6 KΩ
For n = 2, α (damping factor) = 1.414,
Passband gain = $A_o = 3 - \alpha = 3 - 1.414 = 1.586$.

Transfer function of second order Butterworth Low-pass filter is:

$$H(s) = \frac{1.586}{S^2 + 1.414\,s + 1}$$

Now $A_o = 1 + (R_f / R_1) = 1.586$

Thus, $R_f = 0.586R_i$
Let R_i = 10 kΩ, then R_f = 5.86 kΩ

THEORY:

Band Pass Filter:

The cut-off frequency or fc point in a simple RC passive filter can be accurately controlled using just a single resistor in series with a non-polarized capacitor, and depending upon which way aroundthey are connected, we have seen that either a Low Pass or a High Pass filter is obtained.By connecting or “cascading” together a single Low Pass Filter circuit with a High Pass Filter circuit, we can produce another type of passive RC filter that passes a selected range or “band” of frequencies that can be either narrow or wide while attenuating all those outside of this range. This new type of passive filter arrangement produces a 17 frequency selective filter known commonly as a Band Pass Filter or BPF for short.

PROCEDURE:

1. Connect the circuit as shown in diagram.
2. Connect the DSO to the probes and switch it on.
3. Check the graph for both positive and negative voltage and write down the output.

TABULATION

S.No.	Frequency (Hz)	Output VoltageV_O(Volts)	Voltage Gain Av=20 log Vo/Vi (dB)

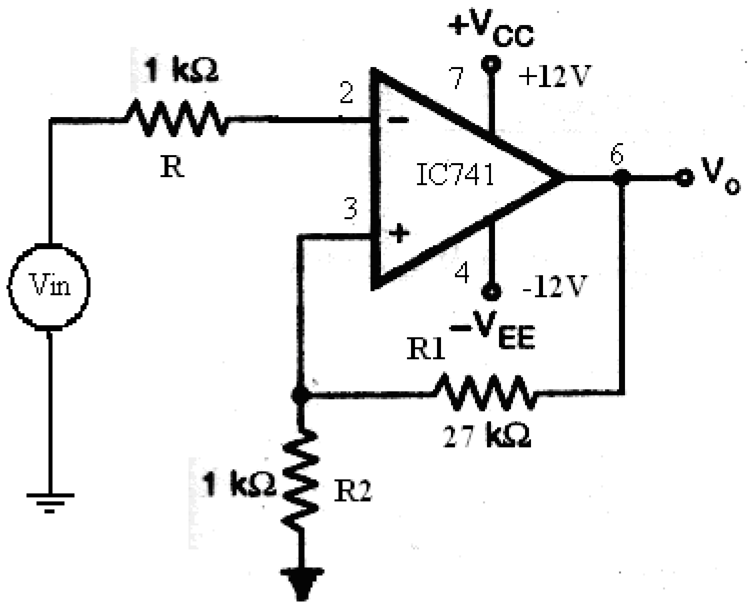
POST-LAB QUESTIONS

1. What is a band-pass filter?
2. What is the frequency response curve of a band pass filter?
3. What is the voltage gain magnitude of the bandpass filter?
4. What is a second-order band pass filter transfer function?

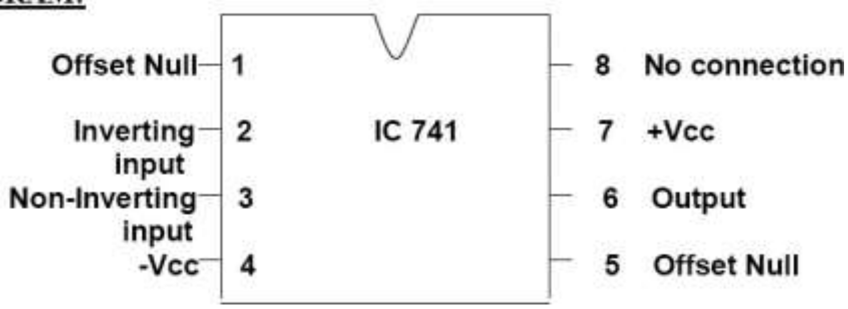
RESULT:

Thus the notch filter and active band pass filter was designed and the frequency response was analyzed.

CIRCUIT DIAGRAM



PIN DIAGRAM:



EX.NO. 4(A) : DESIGN AND SETUP A THRESHOLD DETECTOR

AIM

To design design and setup a threshold detector, sample and hold circuit using op-amp for the following specifications;V_{UT} = 0.5 v , V_{LT}= -0.5v, Vsat = +12 V and -V_{sat}=-12V and V_{ref}=0v.

DESCRIPTION

Comparator:

Input : Sine input of 4V at 10KHz

Output : Square Signal of 24 Vp-p at 10 KHz.

Hysterisis loop : Set the CRO in XY mode and view the hysteresis loop.

APPARATUS REQUIRED :

S.No.	ITEM	RANGE	QTY
1	OP-AMP	IC741	1
2	RESISTOR	1kΩ,27kΩ	2,1
3	CRO	-	1
4	RPS	DUAL(0-30) V	1

PRE LAB QUESTIONS

1. What is an op-amp? Why it is called so?
2. List the ideal characteristics of an op-amp.
3. What are the building blocks of an op-amp?
4. List the applications of a comparator.

DESIGN:

Given : V_{UT} = -V_{LT} =0.5V

V_{UT} = + Vsat R2/(R1+R2)

0.5 = 12 (R2/(R1+R2))

R1 = 23 R2

Assume R2 = 1kΩ

Then R1= 23 1kΩ

THEORY:

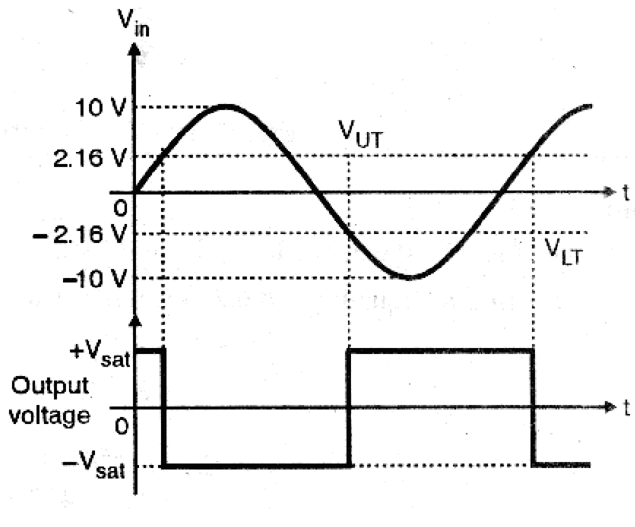
Comparator

Schmitt trigger is useful in converting a slowly varying input signal into a square waveforms. It is a regenerative comparator . The input voltage is applied to the negative terminal and the feedback is applied to the positive input terminal of the op-amp. Tthe output voltage V_o swings between the +Vsat and the –Vsat when each time the input signal V_i exceeds the threshold levels V_{UT} and V_{LT}. The hysteresis or backlash for a comparator occurs when a positive feedback isemployed. It is defined as the difference between the V_{UT} and V_{LT}.

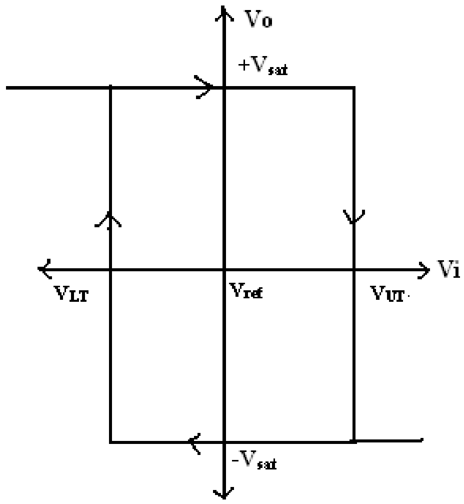
PROCEDURE :

1. The connections are made as shown in the circuit diagram.
2. Apply the input signal of desired amplitude and frequency to the negative input terminal of an op-amp.
3. Compare the input sine signal with the square wave output signal to determine the upper threshold voltage (V_{UT}) and the lower threshold voltage (V_{LT}).
4. Turn the timeperiod knob in CRO to the XY mode and determine the V_{UT} and the V_{LT} values and compare it with the previous observation in step 3.
5. Plot the input signal , output signal and the hysteresis loop in a linear graph.

MODEL GRAPH



HYSTERESIS LOOP



POST LAB QUESTIONS

1. Why Schmitt trigger is called as a regenerative comparator?
2. What is Hysteresis?
3. Mention the applications of Schmitt trigger .

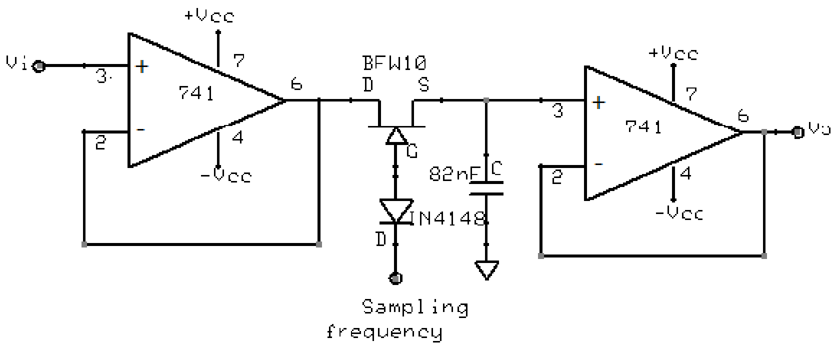
RESULT:

The Schmitt trigger circuit is connected and the waveforms are observed and theoretical and practical values of the threshold values are verified.

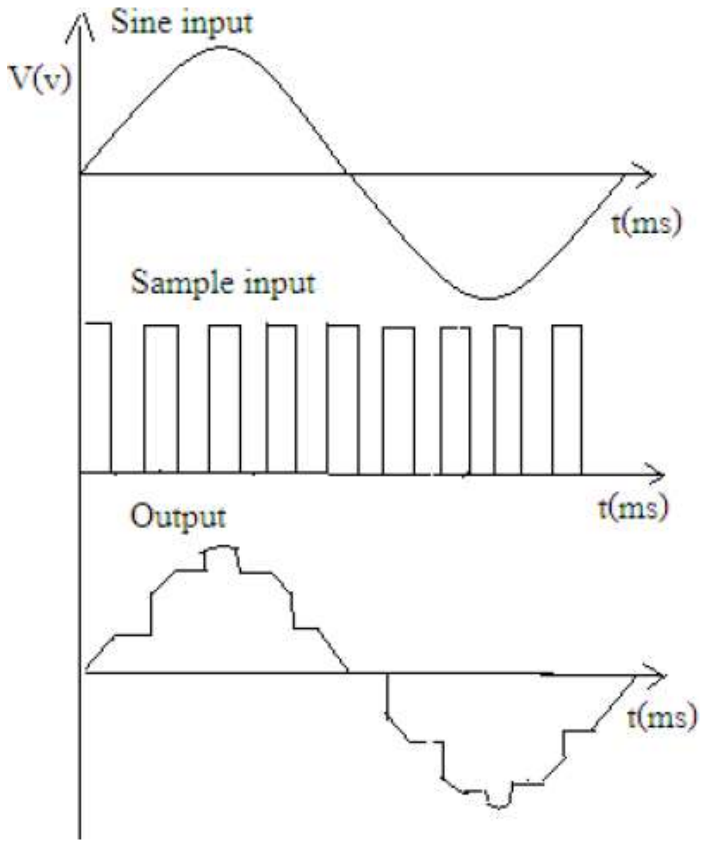
	V_{UT}**(volts)**	V_{LT}**(volts)**
Theoretical values =		
Practical values =		

(From the graph) Hysteresis width = V_{UT} - V_{LT} =

CIRCUIT DIAGRAM



MODEL GRAPH



EXPT.NO: 4B DESIGN AND SETUP A SAMPLE AND HOLD CIRCUIT

AIM:

To design and setup a discrete sample and hold circuit using op-amp.

COMPONENTS REQUIRED:

S.No.	ITEM	RANGE	QTY
1.	Bread Board		1
2.	D.S.O	(0-3)MHz	1
3.	Op-amp	µA 741C	2
4.	BFW10		1
5.	Diode	1N4148	1
6.	Capacitor	82 nF	1
7.	Regulated power supply	(0-30)V	1

PRE-LAB QUESTIONS

1. What is the process of down sampling called?
2. What is the main function of (A/D) or ADC converter?
3. The differential predictive signal quantizer system is known as?

THEORY:

The sample and hold circuit samples an input signal and holds onto the last sampled value, until the input is sampled again. The circuit mainly consists of a sampler or a switching element and a holding capacitor. In the circuit JFET used acts as the switching component. The input signal is given to the drain of the JFET through a voltage follower and sampling high frequency is given to the gate of JFET, through a diode. When the voltage at the gate is zero, the JFET turns on and the capacitor charges to the input voltage. When the voltage at the gate is negative, the JFET turns off.As the capacitor has no discharge path due to the high impedance at the output of the voltage follower circuit, it holds the last sampled voltage. When the JFET is again turns on the capacitor charges to the new input voltage. Thus sampling and holding of the input signal occurs.

PROCEDURE:

1. Check all the components.
2. Set up the discrete sample and hold circuit from a 15V supply.
3. Use dual output function generator to apply input and sampling frequency simultaneously.
4. Set the input at 1 KHz, 4V peak to peak sine wave and sampling square wave at 10 KHz, 8V peak to peak. Ensure that the negative gate voltage of BFW10 is always less than 4V.
5. Then observe the input and outputs waveforms in a DSO.

POST LAB QUESTIONS

1. Which amplifier is used in sample and hold circuit?
2. Why sample and hold circuit is used in integrated circuits?
3. Why MOSFET is used in sample and hold circuit?
4. Which MOSFET IC is used to perform sample and hold circuit operation?

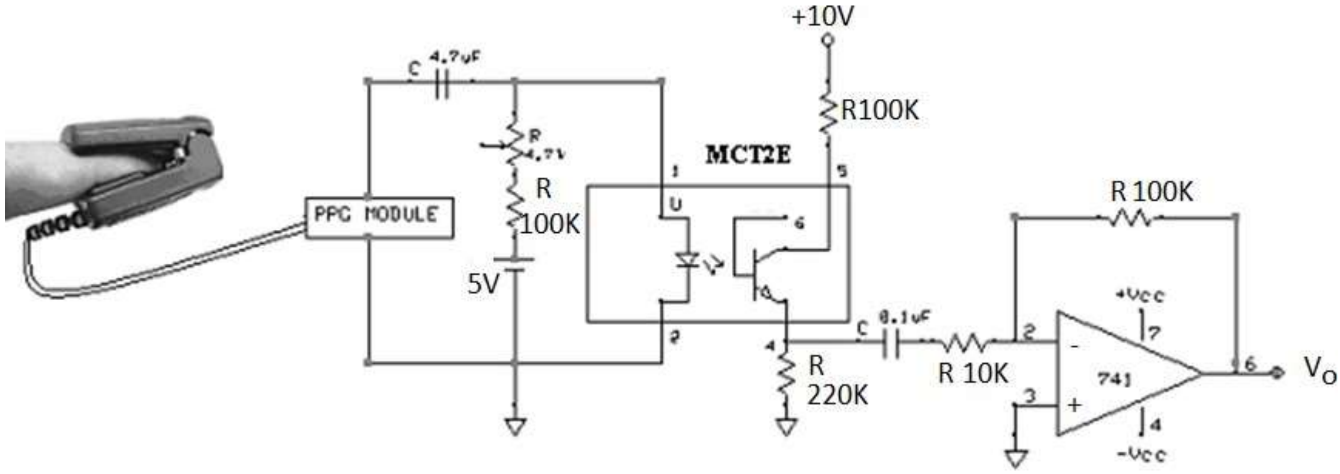
RESULT:

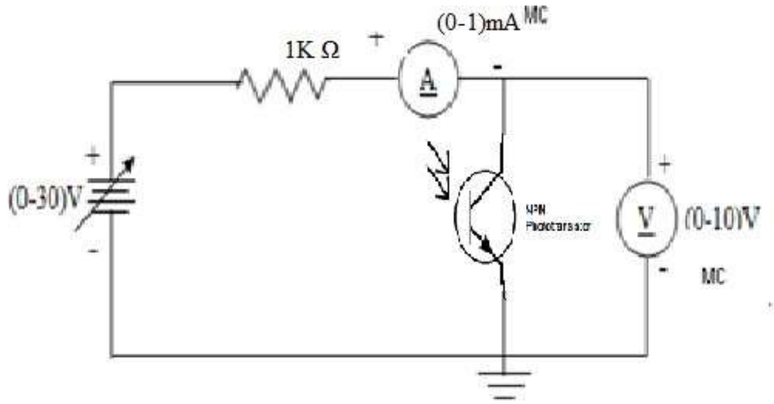
Thus the sample and hold circuit was designed and plotted the input output waveforms.

Inference:

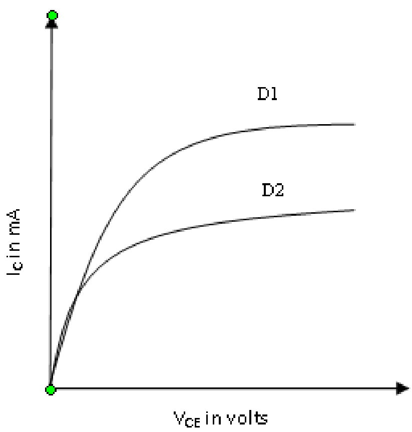
The minimum sampling frequency should be twice the input signal.

CIRCUIT DIAGRAM USING MCT2E





MODEL GRAPH



EXPT.NO: 5 DESIGN AND SETUP A PATIENT ISOLATION CIRCUIT

AIM:

To set up a patient isolation circuit using opto-coupler IC MCT2E**.**

COMPONENTS REQUIRED:

S,No	ITEM	RANGE	QTY
1.	Regulated power supply	(0 - 30V)	1
2.	Voltmeter	(0-10V)MC	1
3.	Ammeter	(0-1000µA)MC	1
4.	Photodiode	-	1
5.	Resistors	100Ω/1K Ω/10KΩ	1
6.	Bread board, connecting wires.	-	1,few

PRE-LAB QUESTIONS

1. What is an op-amp isolation?
2. Which type of amplifier provides an electrical safety barrier and isolation?
3. What type of amplifier destroys an ordinary op-amp?

THEORY:

Phototransistor is a normal transistor in which junction enclosure is transparent to allow light to fall on the base emitter junction. At any PN junction hole electron pairs are generated when light falls on the junction, so that any light falling on the base emitter junction, produces a current which is amplified by transistor action, making the device very sensitive. The dark current increase in aphoto transistor exponentially with rise in temperature and if the temperature is sufficiently high thedark and light currents may be indistinguishable. At any given temperature the dark current can be reduced to a small value by biasing the base of the transistor which is normally left open circuited. The sensitivity is decreased, but the ratio of light to dark current is increased. The advantages of the photo transistor are low power consumption, small size, immediate operation on switching ON, lowvoltage operation and long life. A photo transistor gives a high gain. This transistor is very good for digital applications because of the small rise and fall times.

MCT2E is an opto isolator chip, which is often used to provide isolation and to avoid any electrical connection between the input and output stages. The prominent components in MCT2E are a phototransistor and an LED. The phototransistor is made of silicon material and is activated by the LED made of Germanium Arsenide. The output light of the LED is infrared and the intensity is directly proportional to the potential applied between its two leads. In circuits, where optical isolation is required between two stages, the output of the first stage is given to the LED ofMCT2E and the photo transistor inside the chip transduces the emitted light in to a proportional collector current. This is given as the input to the second stage, thus providing electrical isolation. By means of providing optical isolation between input and output, interference due to leads and other such artifacts can be avoided. Opto-couplers usually finds applications in medical circuits, where patient safety is of much importance.

PROCEDURE:

1. Position the pointer at zero on the scale, when the bulb is at maximum distance away from sensors.
2. Switch ON the power supply to the instrument.
3. Measure the DC voltage output of phototransistor using a millimeter or a CRO across two terminals.
4. Gradually move the bulb towards the sensor in steps of 5mm and note the corresponding voltages.
5. Tabulate the readings and plot the graph of distance versus sensor output voltage and voltage versus current.

TABULATION:

VI CHARACTERISTICS **D=**			I-D CHARACTERISTICS **V_{AK} =**		
S.No.	Voltage V_{AK} (V)	Current (μA)	S.No.	Distance(cm)	Current (μA)

POST-LAB QUESTIONS

1. How does the mct2e work?
2. What is the difference between moc3021 and mct2e?
3. What is an mct2e opto-coupler?

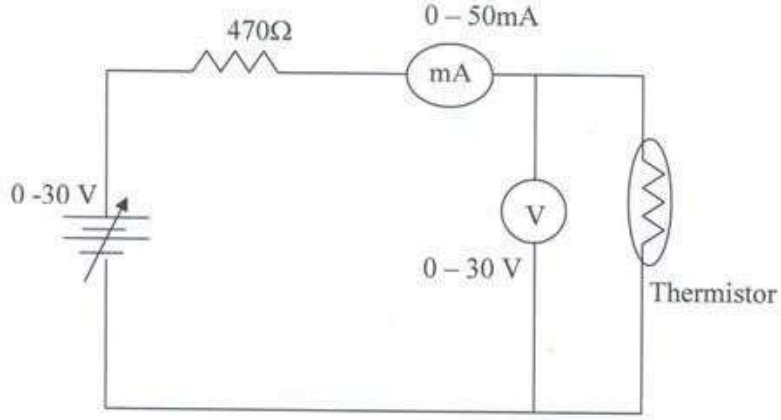
RESULT:

Thus the patient isolation circuit using opto-coupler IC MCT2E and photo transistor.

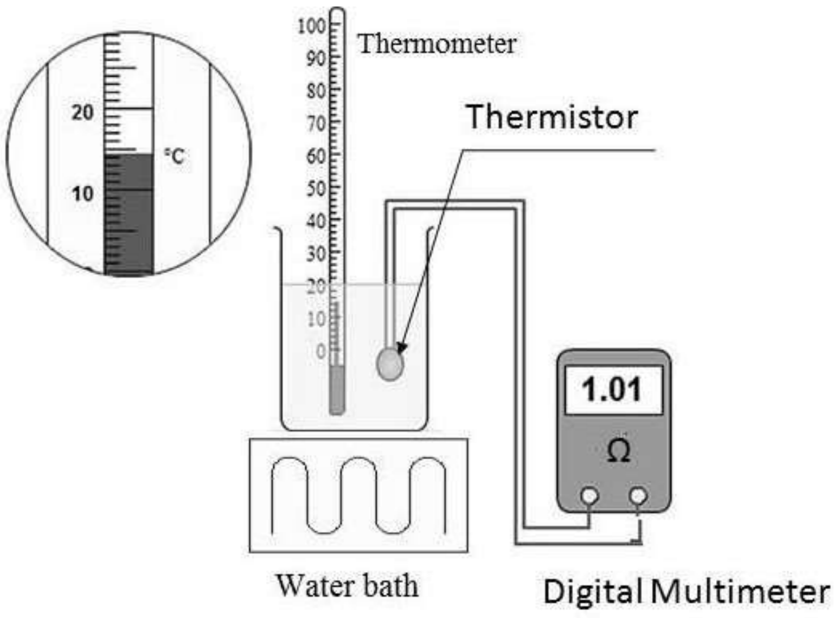
INFERENCE:

MCT2E can be used as an isolation circuit .Even without any physical contact a signal applied at the input can be reproduced at the output.

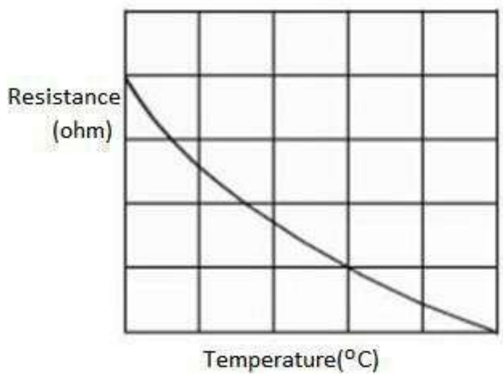
CIRCUIT DIAGRAM



EXPERIMENTAL SET UP



MODEL GRAPH



EXPT.NO: 6 DESIGN OF BODY TEMPERATURE MEASURING CIRCUIT USING THERMISTORS.

AIM:
To design the body temperature measuring circuit using thermistor and study its characteristics.

COMPONENTS REQUIRED:

S.No.	ITEM	RANGE	QTY
1.	Bread board		1
2.	Variable power supply	0-30 V	1
3.	Thermistor		1
4.	Ammeter	(0 - 50mA)	1
5.	Voltmeter	(0 – 30V)	1
6.	Thermometer		1
7.	Water bath		1
8.	Resistor	470Ω	1
9.	Bread Board		1

PRE-LAB QUESTIONS

1. What will happen to resistance, if the length of the conductor is increased?
2. Which of the following can be measured using change in resistivity? a) Temperature b) Visible radiation c) Moisture content d) All of the mentioned
3. What will happen for resistivity metal and semiconductor if the temperature is increased?
4. What is the relation of temperature coefficient of resistivity on the coefficient of thermal expansion in RTD?

THEORY:
Thermistors are the temperature sensitive resistors that exhibit a variation in resistance with temperature. In other words electrical resistance of a thermistor will vary when it is placed in an environment of changing temperature. The characteristics study of thermistor providesinformation about how its resistance changes with the changes in temperature. Based on the relationship between the resistance and temperature thermally sensitive resisters are of two types, one the posistor and its resistance increases with increasing temperature (positive temperature coefficient) and second the thermistor and its resistance decreases with increasing temperature (negative temperature coefficient). Thermistors are very essential in temperature measurement and they are made up of the oxides of certain metals like manganese, cobalt and nickel which have negative temperature coefficient of resistance. They are formed into rods, discs, washers and bead to suite various applications. The application of thermistor includes cardiac output measurements, continuous measurement of skin and core body temperature, blood flow

measurement and respiratory measurement. They are also used as resistance thermometers in low temperature measurements.

PROCEDURE:

1. Take water in a container and place over the heater.
2. Immerse thermistor and thermometer in the water bath.
3. Set up the measuring circuit on the bread board.
4. Switch on the heater.
5. Measure the temperature on the thermometer from room temperature to 98^{o}C and note down the corresponding resistance of thermistor.
6. Switch off the heater, and then take reading in decreasing order of temperature in an interval of 10^{o} C.
7. Plot a graph of temperature on X-axis and Resistance on Y-axis. This graph shows the characteristics of thermistor.

TABULATION

Temp (^{o}C)	Resistance(Ω) (Up)	Resistance(Ω) (Down)

POST LAB QUESTIONS

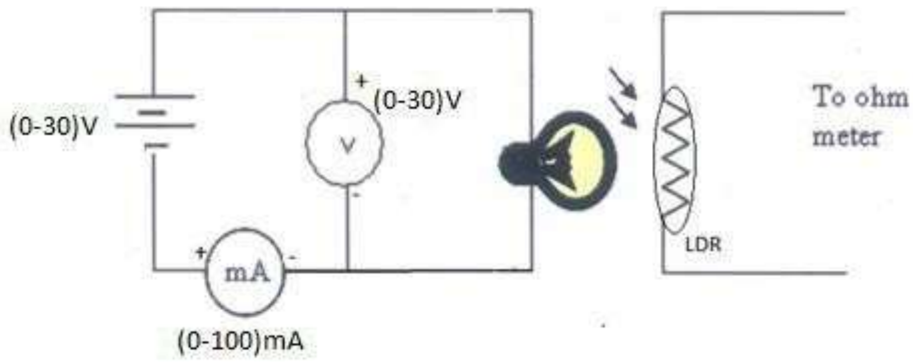
1. Name the different temperature transducers.
2. What is thermistor?
3. What is the temperature range of thermistor?
4. What is the difference between RTD and thermistor?
5. What are the materials used to manufacture thermistor?

RESULT:

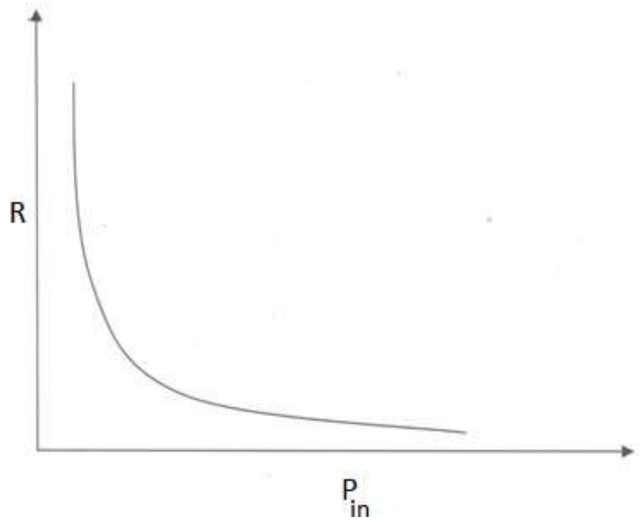
Thus the circuit for body temperature measuring circuit using thermistor was designed and and plotted the characteristics of thermistor.

Inference: Temperature and resistance relationship is non-linear for a thermistor.(i.e. characteristics of thermistor are non-linear).

CIRCUIT DIAGRAM



MODEL GRAPH



TABULATION

V(volts)	I(mA)	P_{in}(mW)	R(Ω)

EXPT.NO: 7 DESIGN OF PLETHYSMOGRAPHY CIRCUIT

AIM:

To design a Plethysmography circuit using LDR.

COMPONENTS REQUIRED:

S.No.	ITEM	RANGE	QTY
1.	Multimeter		1
2.	LDR	-	1
3.	Ammeter	(0-100)mA	1
4.	Voltmeter	(0-30)V	1
5.	Regulated power supply	(0-30)V	1
6.	Bread Board	-	1
7.	Bulb	-	1

PRE-LAB QUESTIONS

1. Name some resistive transducers.
2. What is the principle of LDR?
3. What are the applications of LDR?
4. State the advantages of LDR.
5. State the disadvantages of LDR.

THEORY:

A photo resistor is an electronic component whose resistance decreases with increasing incident light intensity. It can also be referred to as a light- dependent resistor (LDR), or photoconductor. A photoresistor is made of a high resistance semiconductor. If light falling on the device is of high enough frequency, photons absorbed by the semiconductor give bound electrons enough energy to jump into conduction band. The resulting free electrons (and its hole partner) conduct electricity there by lowering resistance.Cadmium selinide or cadmium sulphide are mainly used to construct LDRs. Simple cadmium sulphide cells have a wide range of resistance from less than 100ohm in bright light to in excess of 10Mohm in darkness. The cells are also capable of reacting to a broad range of frequencies including infra-red (IR), visible light and ultra violet. They are often found on street lights as automatic on/off switches. In biomedical field it is used in photo plethismography.

PROCEDURE:

1. Setup circuit on the bread board.

2. Vary the DC input in steps from 0V to 30V

3. Note down the ammeter, voltmeter and ohmmeter readings.

4. Calculate the input power Pin = V x I.

5. Plot a graph of Power on X-axis and Resistance on Y-axis. This graph shows the characteristics of LDR.

POST LAB QUESTIONS

1. What does a plethysmograph measure?
2. How is plethysmography done?
3. What is the difference between spirometry and plethysmography?
4. What are the types of plethysmography?

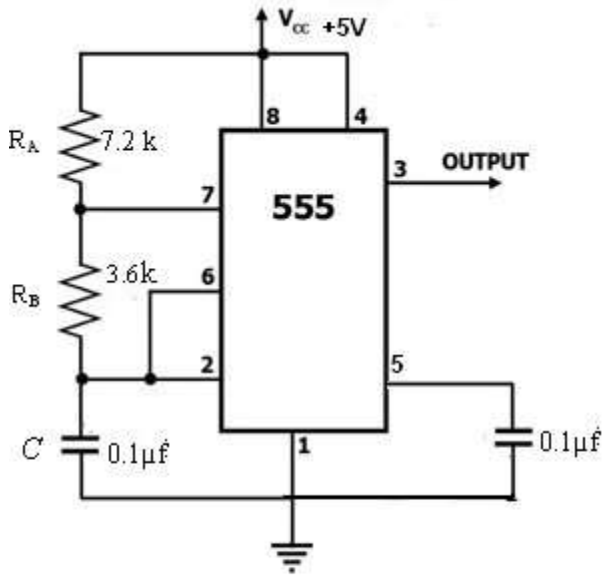
RESULT:

Thus the plethysmography circuit was designed and plotted the characteristic curve.
Dark resistance of LDR =.................. MΩ

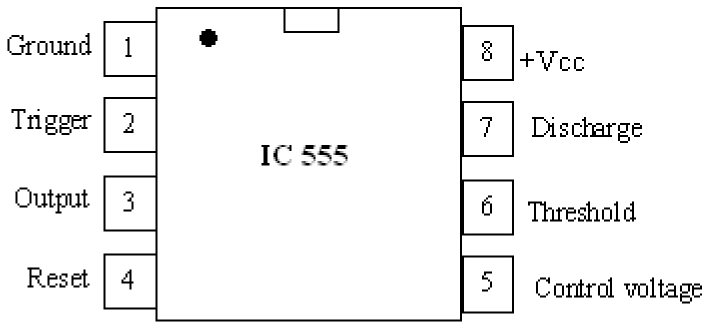
INFERENCE:

The resistance – light intensity relationship is non-linear for a LDR.(i.e. characteristics of LDR is non-linear).

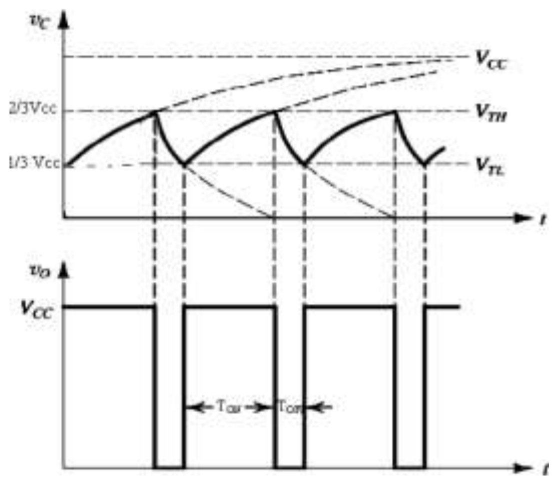
CIRCUIT DIAGRAM



PIN DIAGRAM



MODEL GRAPH



EX.NO.8 DESIGN OF PACE MAKER CIRCUIT

AIM:

To design a Pace Maker circuit using 555 TIMER for a signal of frequency of 1 KHz and duty cycle D = 75%

COMPONENTS REQUIRED:

S.NO	ITEM	RANGE	QTY
1	IC	NE555	1
2	RESISTOR	7.2kKΩ,3.6 KΩ	Each 1
3	CAPACITOR	0.01μF, 0.1μF	Each 1
4	RPS	(0-30) V	1
5	CRO	-	1

PRE-LAB QUESTIONS

1. What is a Multivibrator?

2. What is a Timer IC?

3. Differentiate amplifier and IC.

THEORY:

The IC555 timer is a 8 pin IC that can be connected to external components for astable operation. The simplified block diagram is drawn. The OP-AMP has threshold and control inputs. Whenever the threshold voltage exceeds the control voltage, the high output from the OP –AMP will set the flip-flop. The collector of discharge transistor goes to pin 7. When this pin is connected to an external trimming capacitor, a high Q output from the flip flop will saturate the transistor and discharge the capacitor. When Q is low the transistor opens and the capacitor charges.

The complementary signal out of the flip-flop goes to pin 3 and output. When external reset pin is grounded it inhibits the device. The on – off feature is useful in many application. The lower OP- AMP inverting terminal input is called the trigger because of the voltage divider. The non-inverting input has a voltage of +Vcc/3; the OP-Amp output goes high and resets the flip flop.

DESIGN :

$\mathbf{f} = 1.44/(R_A+2R_B)C$

$D = T_{ON}/T = R_A+R_B/(R_A+2R_B)$

Assume C = 0.1μf

$f = 1.44/(R_A+2R_B)C$, Given f = 1kHz

$1kHz = 1.44/(R_A+2R_B)0.1\ \mu f$

$R_A+2R_B = 14400$ -------------1

$D = T_{ON}/T = R_A+R_B/(R_A+2R_B)$

$0.75 = R_A+R_B/(R_A+2R_B)$

$R_B = 0.5\ R_A$ 2

Substitute eqn (2) in eqn (1) $R_A + 2(0.5R_A) = 14400$

Then $R_A = 7.2\ k\Omega$ $R_B = 0.5\ R_A$, $R_B = 3.6\ k\Omega$

TABULATION

Amplitude = (volts)

T_{ON} = (msec)

T_{OFF} = (msec)

Time period $= T_{ON} + T_{OFF} =$ (msec)

PROCEDURE:

1. The connections are made as per the diagram.
2. The output of the astable multivibrator I at pin 3 is measured and its frequency is verified with the theoretical frequency.
3. For monostable multivibrator the trigger input is given
4. The output is observed at pin 3 and across capacitor.
5. The theoretical frequency is verified with the practical.
6. This experiment can be repeated for various values of c for different frequency
7. Plot the output graphs.

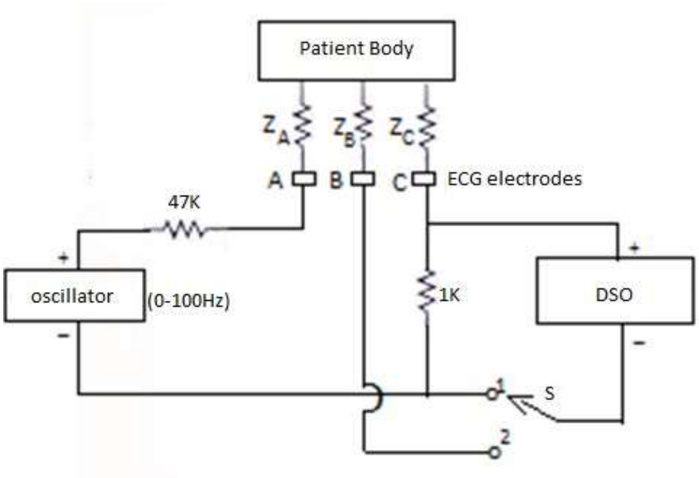
POST LAB QUESTIONS:

1. What is an astable multivibrator?
2. How astable mode of 555 Timer can be modified to get a square wave generator?
3. What is an monostable multivibrator?
4. How monostable mode of 555 Timer is designed?
5. Define duty cycle.

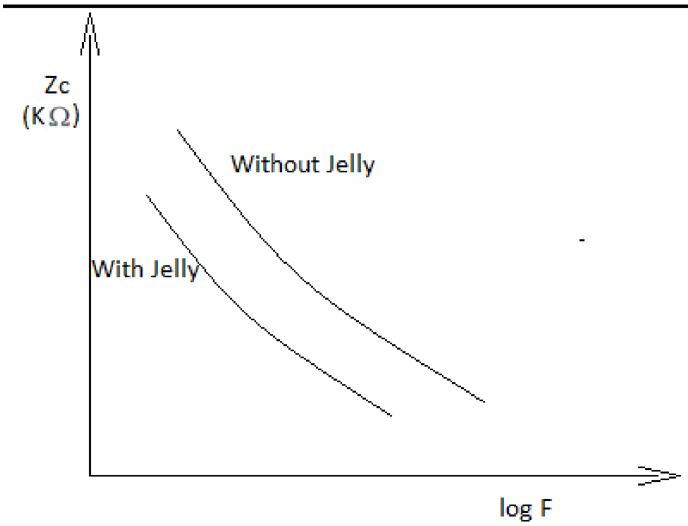
RESULT:

Thus the Pace Maker circuit. was designed using IC555 and its output waveform is traced.

CIRCUIT DIAGRAM



MODEL GRAPH



TABULATION

Without jelly					With jelly				
F(Hz)	V1(V)	I(mA)	Vzc(V))	Zc(KΩ)	F(Hz)	V1(V)	I(mA)	Vzc(V))	Zc(KΩ)

EXPT.NO: 9 DESIGN AND SETUP A CIRCUIT FOR SKIN CONTACT IMPEDANCE

AIM:

To design and setup a circuit for skin contact impedance.

COMPONENTS REQUIRED:

S.NO	ITEM	RANGE	Q.TY
1	Function Generator		
2	Resistor	1 KΩ,47 KΩ	Each 1
3	ECG Electrodes		3
4	DSO	3 MHz	1
5	Bread board	-	1

PRE-LAB QUESTIONS

1. What is impedance and what is its significance?
2. How do we reduce contact impedance?
3. How is skin electrode impedance measured?
4. What is electrode impedance?

THEORY:

The bio electric events are usually measured by means of metallic electrodes placed on the surface of the body. The electrical activity generated by various muscles and nerves within the body is conducted to the electrode through the body tissues. It reaches electrodes through the skin electrode transition and is then conducted by direct wire connection to the input circuit of the recording machines. The impedance at the electrode skin interface comes in the overall circuitry of the recording machine. Hence the skin electrode impedance is significant of the final recording. Skin electrode impedance is called as the contact impedance and is of value greater than the electrical impedance of the body tissue as measured beneath the skin. The outer horny layer of the skin is responsible for the bulk of skin contact impedance. Jelly is used in order to increase the contact between the subject's skin and the skin electrodes. The skin contact impedance is reduced on the application of the jelly.

PROCEDURE:

1. Set the function generator output at 2Vpp.
2. Set up the circuit with switch in position 1 and measure the voltage across 1k by varying the input frequency from 0 to 100Hz.
3. Repeat step 2 after applying jelly.
4. Set up the circuit with switch in position2 and measure the voltage across Zcby varying input frequency from 0 to 100Hz.
5. Calculate I=V1/1K and calculate Zc=Vzc/I
6. Plot the frequency Vs impedance graph.

POST LAB QUESTIONS

1. What is skin impedance?
2. What is normal skin impedance?
3. What is skin electrode impedance?
4. How is skin electrode impedance measured?

RESULT:

Thus the circuit was setup and measured the skin contact impedance with and without jelly.

Maximum impedance without jelly = ...Ω

Minimum impedance without jelly = ...Ω

Maximum impedance with jelly =............Ω

Minimum impedance with jelly = ...Ω

INFERENCE:

The skin contact impedance is high without jelly so use jelly to reduce the contact impedance while measuring biopotentials.

EXPT.NO: 10 STUDY OF BIOTELEMETRY

AIM:

To understand the transmission and reception of biological signal using single channel telemetry system

EQUIPMENTS REQUIRED:

1. ECG Amplifier
2. Low Pass Filter – 2 Nos. FM Modulator
3. FM Transmitter
4. FM Receiver
5. FM Demodulator
6. Charger
7. Battery – 2 Nos. Electrodes

THEORY:

Telemetry is a system of sending data, usually measurements, over a distance. Telemetricdata may be physical, environmental or biological. Telemetry is typically used to gather data from distant, inaccessible locations, or when data collection would be difficult or dangerous for a variety of reasons. In telemetry, specialized instruments carry out measurements of physical quantities, and store or transmit the resulting signal, often after some initial signal processing or conversion. Biotelemetry is the electrical measuring, transmitting, and recording of qualities, properties, and actions of organisms and substances, usually by means of radio transmissions from a remote site. There are single channel and multi-channel telemetry systems. For a single channel system, a miniature battery operatedradio transmitter is connected to the electrodes ofthe subject. This transmitter broadcasts the bio potential over a limited range to a remotely located receiver, which detects the radio signals and recovers the signal for further processing. In this situation there is a negligible connection or stray capacitance between the electrode circuitand rest of the system.

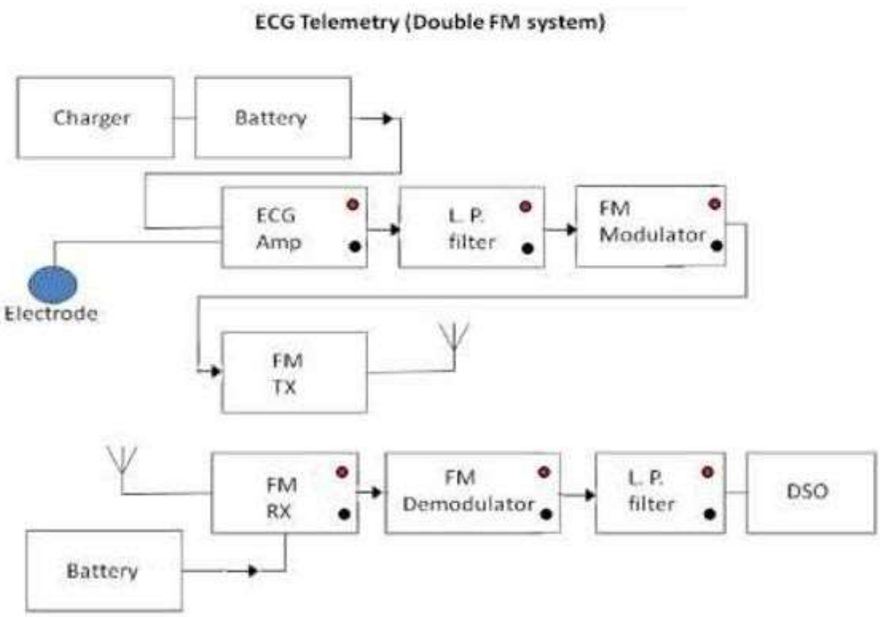
Block Diagram Description:

ECG Amplifier

ECG has amplitude of only about 1 mV, so to detect it an amplifier is required. The ECG amplifier used here has a Gain of 1000 and CMRR of more than 80dB.

Low Pass Filter

A low-pass filter allows low-frequency signals but attenuates (reduces the amplitude of) signals with frequencies higher than the cutoff frequency. When the ECG is amplified, the noise is amplified too, and often swamps the ECG signal. The noise is usually of a higher frequency than the ECG. So the noise can be reduced by low-pass filtering.



Block diagram

FM Modulator

Modulation is used to embed a message (voice, image, data, etc.) on to a carrier wave for transmission. A bandlimited range of frequencies that comprise the message (baseband) is translated to a higher range of frequencies. The bandlimited message is preserved, i.e. every frequency in that message is scaled by a constant value. Here the incoming ECG signal is modulated at around 110MHz. The modulated ECG signal is given to the FM Transmitter.

FM Transmitter

FM Transmitter sends a signal (typically 4-20mA) from a process location to a central location for control and monitoring. Here FM transmitter transmits the modulated ECG signal.

FM Receiver

A receiver receives its input through an antenna. It receives the modulated signal from the transmitter. The receiver then passes on the information to the FM Demodulator where the ECG signal is demodulated to obtain the original ECG signal. **FM Demodulator**

Demodulation, in radio is the technique of separating a transmitted audio frequency signal from its modulated radio carrier wave. Here the modulated ECG signal is demodulated ata frequency of around 100Hz and the original ECG signal is recovered.

PROCEDURE:

Connect the modules as per the block diagram. Switch ON the battery. Connect the ring electrodes to the subject. View the transmitted signal on the DSO.

The various outputs from each of the modules can be viewed on the DSO byconnecting the output pin to the desired module.

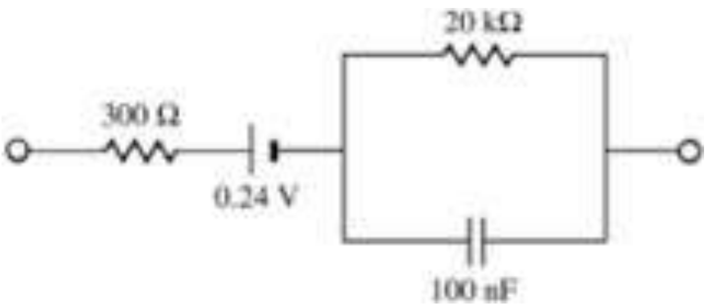
POST LAB QUESTIONS

1. Define telemetry.
2. What are the types of Telemetry?
3. What is the purpose of demodulator?
4. How does telemetry work?

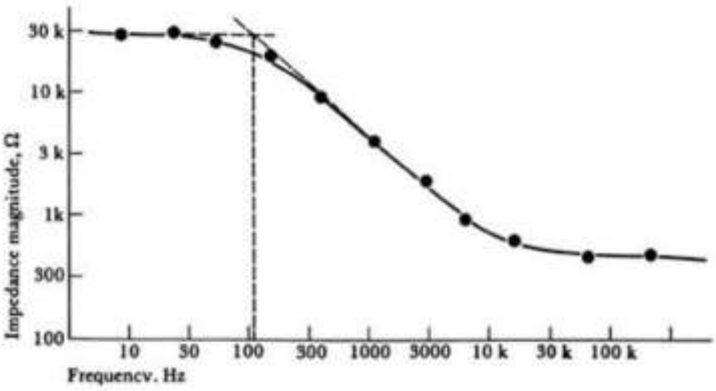
RESULT:

Thus we understand the transmission and reception of biological signal using a telemetry system.

CIRCUIT DIAGRAM



MODEL GRAPH



TABULATION

V(volts)	R(Ω)	C(F)

EXPT.NO: 11 DESIGN OF BIO-ELECTRODE EQUIVALENT CIRCUIT

AIM:

To design a bio-electrode equivalent circuit for measuring bio potential signals.

COMPONENTS REQUIRED:

S.No.	ITEM	RANGE	QTY
1.	Regulated power supply	0-30 V	1
2.	Bread Board		1
3.	Resistors	300 Ω,20 K ,DRB	1
4.	Capacitors	DCB,100 nF	1
5.	Multimeter		1

PRE-LAB QUESTIONS

1. What are bio potential?
2. How is biopotential measured?
3. Which type of electrode is preferred for bio potential measurement?
4. What is a biopotential amplifier?

THEORY:

In order to measure and record potentials (currents) in the body, it is necessary to provide some interface between the body and the electronic measuring apparatus. Current flows in the measuring circuit for at least a fraction of the period of time over which the measurement is made. Bio potential electrodes is a transducer that convert the body ionic current in the body into the traditional electronic current flowing in the electrode. Current is carried in the body by ions, whereas it is carried in the electrode and its lead wire by electrons.Half cell potential is altered when there is current flowing in the electrode. Overpotential is the difference between the observedhalf-cell potential with current flow and the equilibrium zero-current half-cell potential. Mechanism Contributed to overpotential - Ohmic overpotential: voltage drop along the path of the current, and current changes resistance of electrolyte and thus, a voltage drop does not follow ohm's law. - Concentration overpotential: Current changes the distribution of ions at the electrode- electrolyte interface - Activation overpotential: current changes the rate of oxidation and reduction. Since the activation energy barriers for oxidation and reduction are different, the net activation energy depends on the direction of current and this difference appear as voltage. Vp =VR +VC +VA.

PROCEDURE

1. The connections are made as per the diagram.
2. Input is varied from 0 to 2 V and output is measured.
3. This experiment can be repeated for various values of c
4. Plot the output graphs.

.

POST LAB QUESTIONS

1. What are the three basic types of biopotential electrodes?
2. What is bio electrode?
3. How do biopotential electrodes work?
4. What are Bioelectrodes list 3 properties and applications of Bioelectrode?

RESULT:

Thus the equivalent circuit for bio potential electrode was designed using resistors and capacitors.

EXPT.NO: 12 STUDY OF PCB DESIGN SOFTWARE (OPEN SOURCE) LIKE KICAD, EAGLE

AIM:

To Study of PCB design software (open source) like KiCad, Eagle, etc.

EQUIPMENTS REQUIRED:

KiCad software
PC

Introduction to KiCad

KiCad is an open-source software suite for creating electronic circuit schematics, printed circuit boards (PCBs), and associated part descriptions. KiCad supports an integrated design workflow in which a schematic and corresponding PCB are designed together, as well as standalone workflows for special uses. KiCad also includes several utilities to help with circuit and PCB design, including a PCB calculator for determining electrical properties of circuit structures, a Gerber viewer for inspecting manufacturing files, a 3D viewer for visualizing the finished PCB, and an integrated SPICE simulator for inspecting circuit behavior.

KiCad runs on all major operating systems and a wide range of computer hardware. It supports PCBs with up to 32 copper layers and is suitable for creating designs of all complexities. KiCad is developed by a volunteer team of software and electrical engineers around the world with a mission of creating free and open-source electronics design software suitable for professional designers.

The latest documentation for KiCad is available at https://docs.kicad.org.

Downloading and installing KiCad

KiCad runs on many operating systems, including Microsoft Windows, Apple macOS, and many major Linux distributions.

You can find the most up to date instructions and download links at https://www.kicad.org/download/.

Basic Concepts and Workflow

The typical workflow in KiCad consists of two main tasks: drawing a schematic and laying out a circuit board.

The schematic is a symbolic representation of the circuit: which components are used and what connections are made between them. Schematic symbols are pictorial representations of electronic components in a schematic, such as a zigzag for a resistor or a triangle for an opamp. The schematic contains symbols for every component in the design, with wires connecting pins in the symbols. The schematic is typically drawn first, before laying out the circuit board.

The board is the physical realization of the schematic, with component footprints positioned on the board and copper tracks making the connections described in the schematic. Footprints are a set of copper pads that match the pins on a physical component. When the board is manufactured and assembled, the component will be soldered onto its corresponding footprint on the circuit board.

KiCad has separate windows for drawing the schematic ("Schematic Editor"), laying out the board ("PCB Editor"), and editing symbols and footprints ("Symbol Editor" and "Footprint Editor"). KiCad comes with a large library of high quality, user contributed symbols and footprints, but it is also simple to create new symbols and footprints or modify existing symbols and footprints.

Finally, it is important to understand that KiCad has a project-based workflow. A KiCad project is a folder with a project file, a schematic, a board layout, and optionally other associated files such as symbol and footprint libraries, simulation data, purchasing information, etc. Many project-related settings, including net classes and design rules, are stored at the project level. Opening a board outside of its associated project may result in missing design information, so be sure to keep all files associated with a project together.

PCB Design Workflow

Typically, the schematic is drawn first. This means adding symbols to the schematic and drawing the connections between them. Custom symbols may need to be created if appropriate symbols are not already available. At this stage footprints are also selected for each component, with custom footprints created as necessary. When the schematic is complete and the design has passed an electrical rules check (ERC), the design information in the schematic is transferred to the board editor and layout begins.

The schematic describes which components are in the design and how they are connected; the board editor uses this information to make layout easier and to prevent mismatches between the schematic and PCB. The layout process requires careful placement of each footprint on the circuit board. After component placement, copper tracks are drawn between components based on the connections in the schematic as well as other electrical considerations, such as trace resistance, controlled impedance requirements, crosstalk, etc.

Often the schematic will need to be updated after layout has begun; the schematic changes can be easily pulled into the board design. The reverse can often happen: any design changes made in the board layout can be pushed back to the schematic to keep the two consistent.

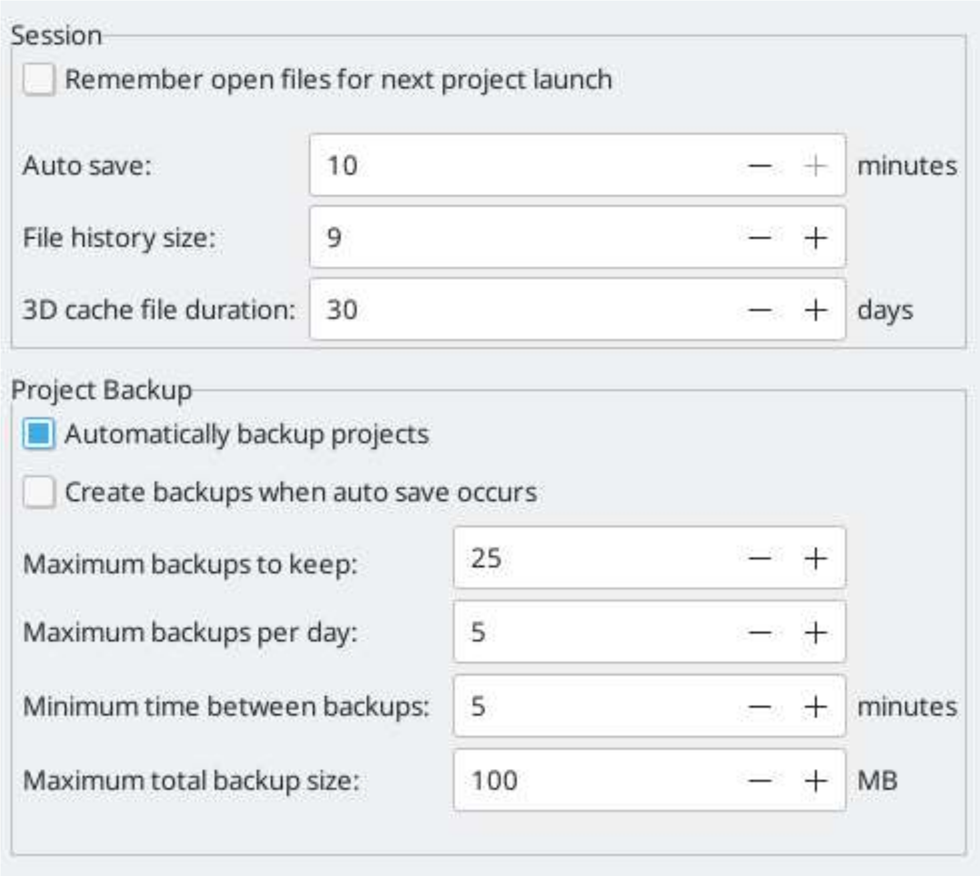
When the board layout is complete and the board has passed the Design Rules Check (DRC), fabrication outputs are generated so that the board can be manufactured by a PCB fabricator.

Tutorial Part 1: Project

The first thing to do when starting a new design is to create a new project. Opening KiCad will bring up the Project Window. Click File → New Project, browse to your desired location, and give your project a name, such as getting-started. Make sure the *Create a new folder for the project* checkbox is ticked, then click Save. This will create your project files in a new subfolder with the same name as your project.

At left, the Project Files pane lists the files in the new project. There is a project file with the extension .kicad_pro, a schematic file with the extension .kicad_sch, and a board file with the extension .kicad_pcb. These files all share a name with your project.

There may also be a -backups directory: KiCad will automatically create backups of your project when you save, and optionally at fixed time intervals. The backup settings are configurable by going to **Preferences → Preferences → Common → Project Backup**.



At right of the Project Window, there are buttons to launch the various tools that KiCad provides. Launching these tools will automatically open the associated design file (schematic or PCB) from the current project. Start by opening the Schematic Editor.

Tutorial Part 2: Schematic

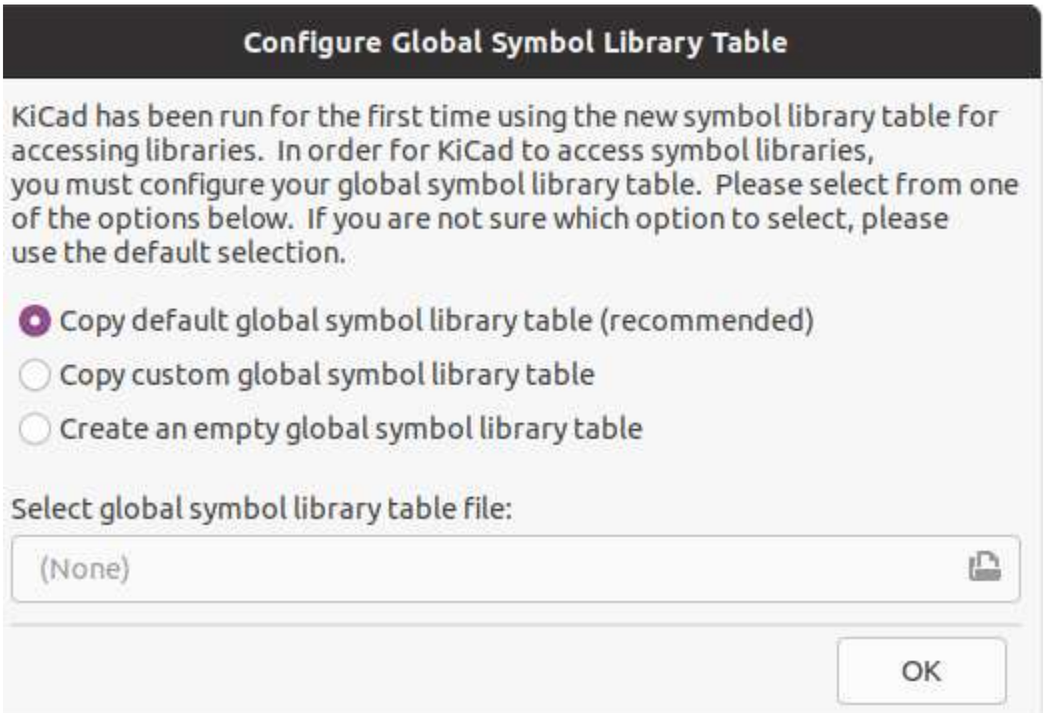
Symbol Library Table Setup

The first time the schematic editor is opened, a dialog will appear asking how to configure the global symbol library table. The symbol library table tells KiCad which symbol libraries to use and where they are located. If you have installed the default libraries with KiCad, which is recommended, select the default option: *Copy default global symbol library table (recommended)*.

If KiCad cannot find the libraries in their expected installation location, this option will be disabled. In this case, the user should choose the second option, *Copy custom global symbol library table*. Click the folder button at the bottom, and browse to the location given below. Select the sym-lib-table file.

The location of the default library table files depends on operating system and may vary based on installation location:

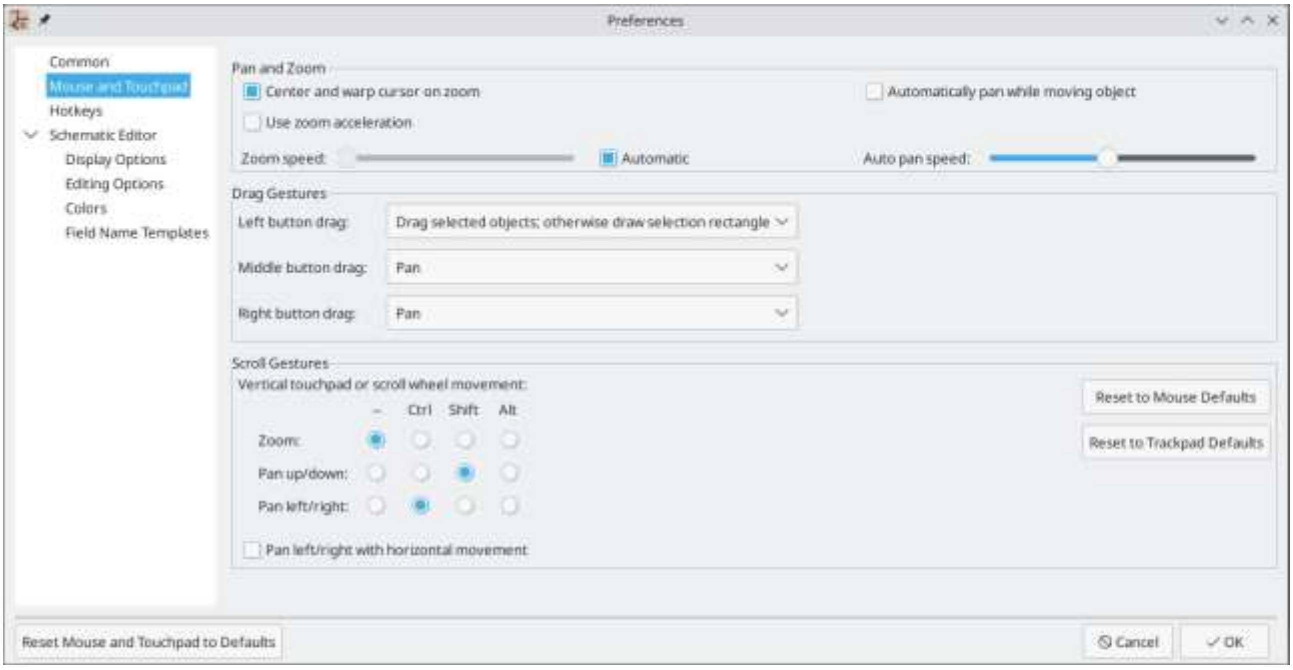
- Windows: C:\Program Files\KiCad\6.0\share\kicad\template\
- Linux: /usr/share/kicad/template/
- macOS: /Applications/KiCad/KiCad.app/Contents/SharedSupport/template/



<u>Schematic Editor Basics</u>

To pan around the schematic, click and drag with the middle mouse button or right mouse button. Zoom in and out with the mousewheel or F1 and F2. Laptop users may find it useful to change the mouse controls to be better suited to a touchpad; the mouse controls are configurablein **Preferences → Preferences… → Mouse and Touchpad**.

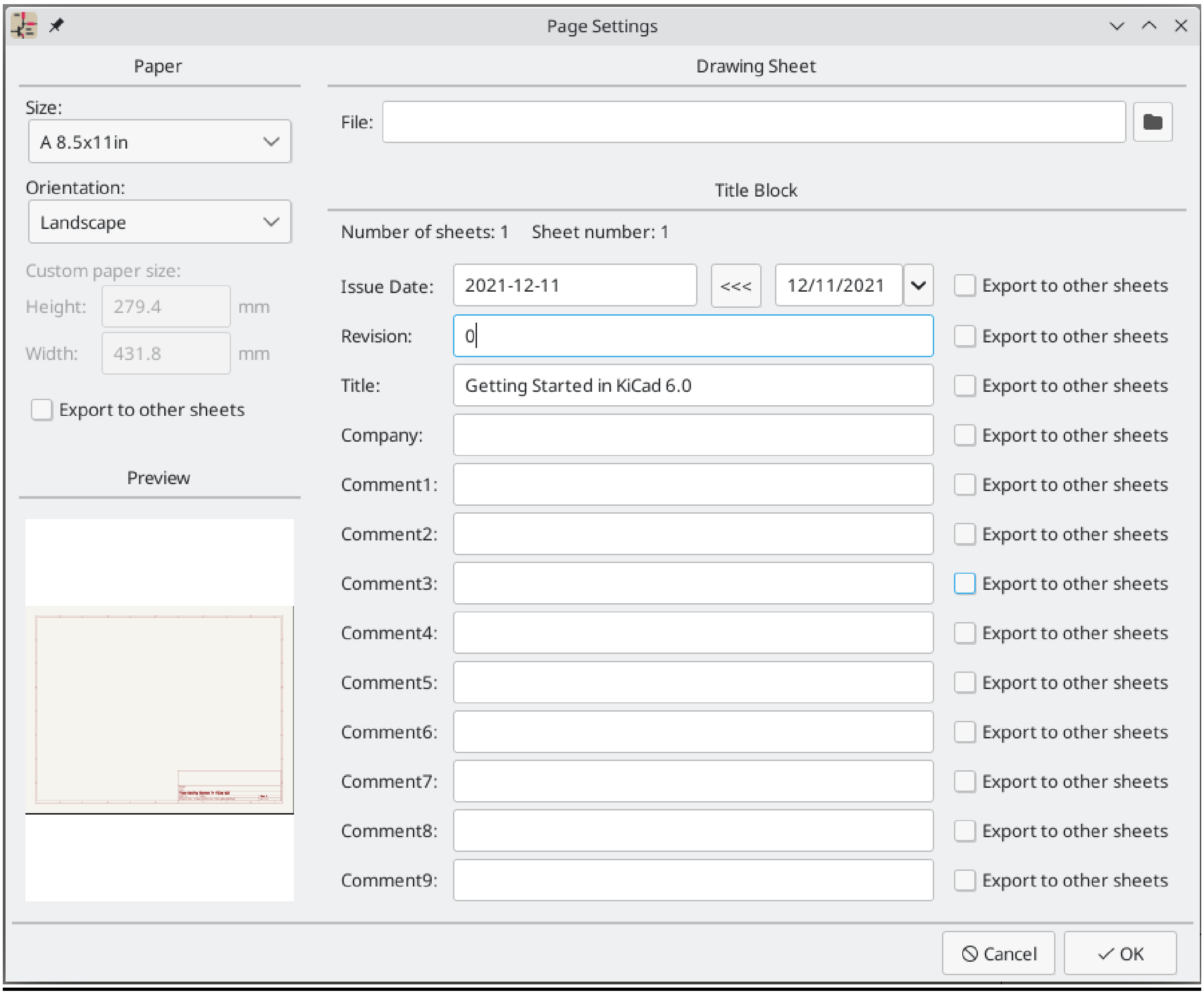
By default, KiCad enables a mouse setting called *Center and Warp Cursor on Zoom*. When this feature is enabled, the mouse cursor is automatically moved to the center of the screen when the user zooms in or out. This keeps the zoomed region centered at all times. This feature is unusual, but many users find it useful once they get used to it. Try zooming in and out with the mouse cursor in different areas of the canvas. If the default zoom behavior is uncomfortable, disable the feature in the Mouse and Touchpad preferences.



The toolbar at the left side of the screen contains basic display settings. The toolbar at the right side of the screen contains tools for editing the schematic.

Most tools in KiCad either have default hotkeys assigned, or can have custom hotkeys assigned. To view all hotkeys, go to **Help → List Hotkeys…**. Hotkeys can be changed in **Preferences → Preferences… → Hotkeys**.

Schematic Sheet Setup



Before drawing anything in the schematic, set up the schematic sheet itself. Click **File → Page Settings**. Give the schematic a title and date, and change the paper size if desired.

Adding Symbols to the Schematic

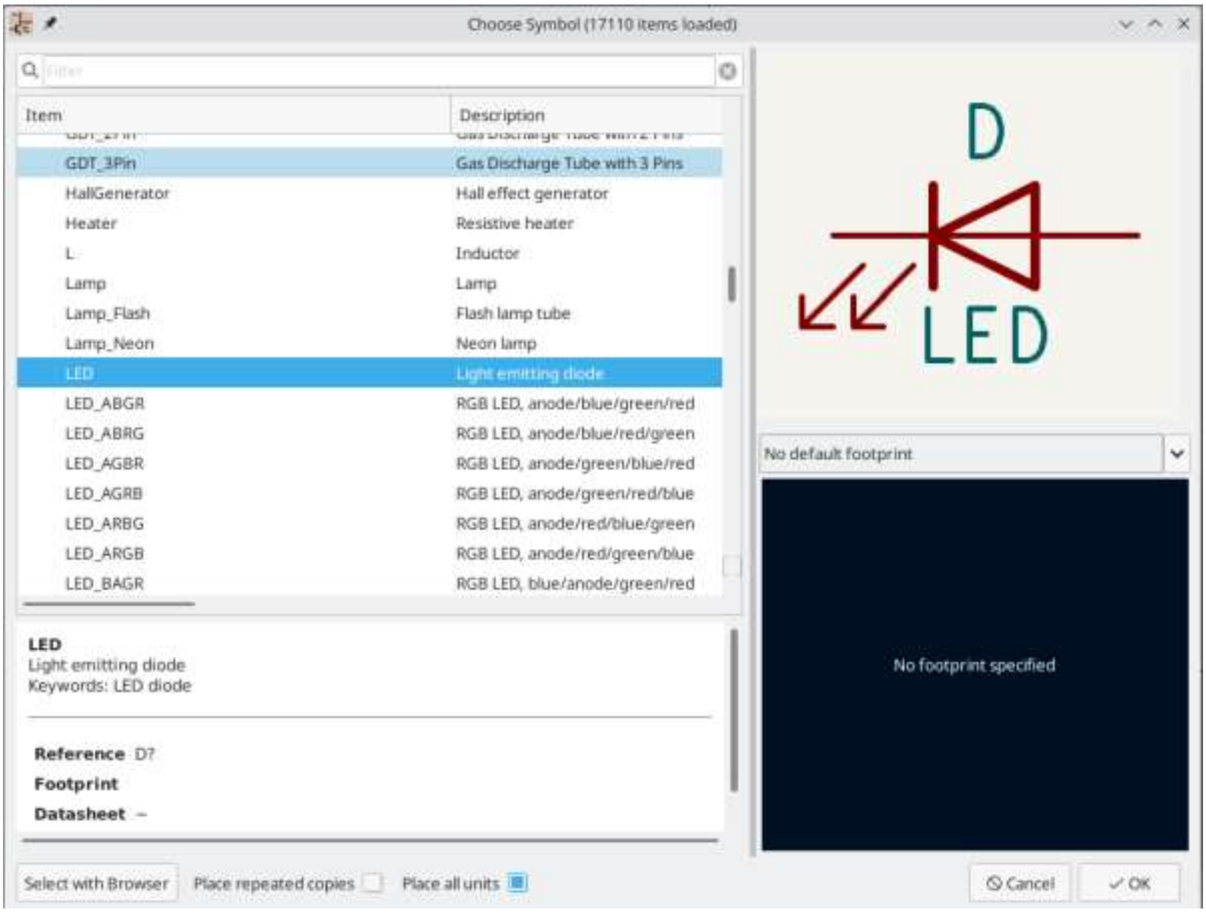
Start making the circuit by adding some symbols to the schematic. Open the Choose Symbol dialog by clicking the **Add a Symbol** button on the right side of the window or pressing A.

This action will trigger the Footprint Library Table Setup dialog. This dialog is equivalent to the Symbol Library Table Setup dialog explained earlier, but for footprints instead of symbols.

Again, select the default option: *Copy default global footprint library table (recommended)*. If this is option is disabled, select the second option, *Copy custom global footprint library table*. Click the folder button at the bottom, and browse to the location given in the symbol library table setup instructions. Select the fp-lib-table file and click **OK**.

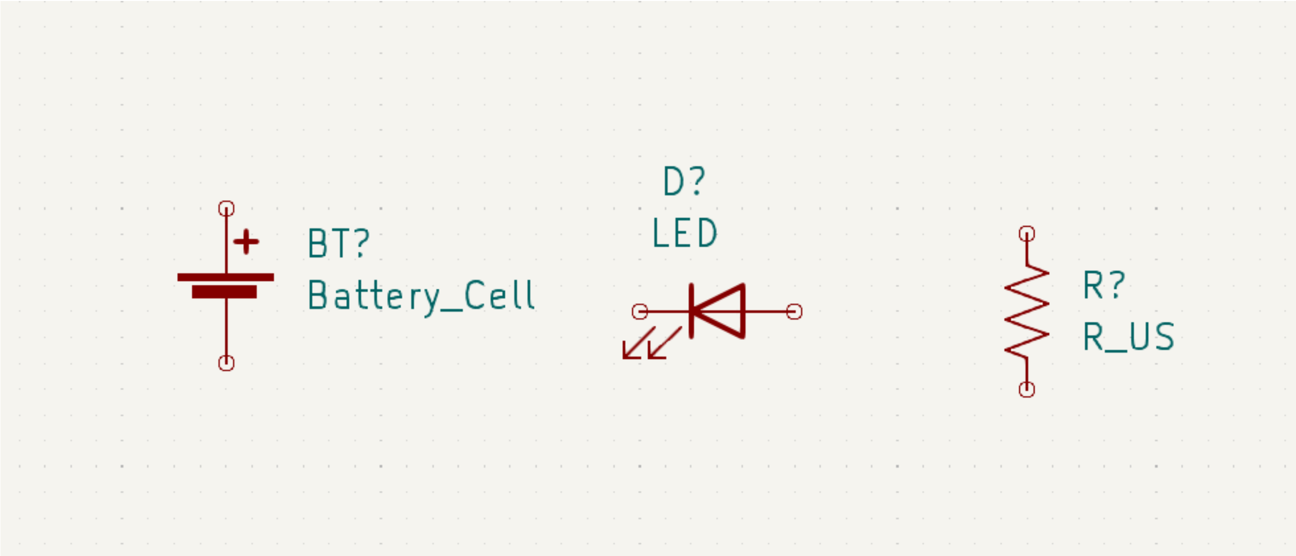
The Choose Symbol dialog lists the available symbol libraries and the component symbols contained within them. Basic devices like passive components, diodes, and other generic symbols are found in the Device library. Specific devices, such as a particular LED, may be found in other libraries.

Scroll down to the Device library, expand it, and select the LED symbol. Click **OK**, and click again to place the symbol in the schematic.



Next, add a current-limiting resistor. Go back to the symbol chooser, but this time try searching for a resistor by entering R in the filter box at the top. Again, it is found in the Device library. The R device is an IEC-style rectangular resistor symbol. An R_US symbol is also available for userswho prefer the ANSI-style zigzag symbol. Select a resistor symbol and add it to the schematic.

Finally, add a battery to power the LED. The Device library has a suitable Battery_Cell symbol.



Selecting and Moving Objects

Next, position the symbols correctly relative to each other, as shown in the screenshot. Do this by selecting, moving, and rotating the symbols.

In KiCad 6.0, objects are selected by clicking on them. Additional objects can be added to the selection with Shift +click, or removed with Ctrl + Shift +click (MacOS: Cmd + Shift +click). Drag selection is also possible; dragging from left-to-right selects objects that are entirely enclosed by the selection box, while dragging right-to-left also selects objects that are partially enclosed by the selection box. Shift and Ctrl + Shift / Cmd + Shift can also be used with drag selection to add or subtract from the selection, respectively.

Note that it is possible to select an entire symbol (by clicking on the symbol shape itself) or to select one text field in the symbol without selecting the rest of the symbol (by clicking the text). When just a text field is selected, any actions performed will act only on the selected text and not on the rest of the symbol.

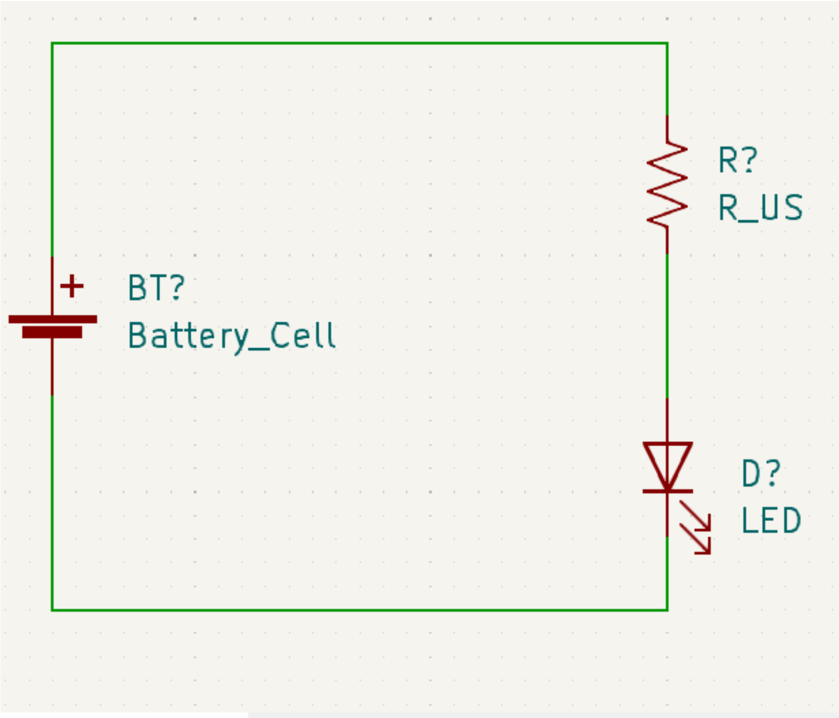
Selected objects are moved by pressing M and rotated by pressing R. The G hotkey (drag) can also be used to move objects. For moving unconnected symbols, G and behave identically, but

M

for symbols with wires attached, G moves the symbol and keeps the wires attached, while M moves the symbol and leaves the wires behind. Selected objects can be deleted with the Del key.

Wiring the Schematic

The symbol pins all have small circles on them, indicating that they are not connected. Fix that by drawing wires between symbol pins as shown in the screenshot. Click the **Add a Wire** button on the right-hand toolbar or use the W hotkey. Click to start drawing a wire, and finish drawing the wire by clicking on a symbol pin or double clicking anywhere. Pressing Escape will cancel drawing the wire.



Another convenient method of drawing wires is to hover over an unconnected pin. The mouse cursor will change to indicate that a wire can be drawn starting from that location. Clicking on the pin will then begin drawing a wire automatically.Next, add power and ground symbols to the schematic. While not strictly necessary in such a simple schematic, these make it easier to understand large schematics.

Annotation, Symbol Properties, and Footprints

Annotation

Each symbol needs a unique reference designator assigned to it. This process is also known as annotation. Click the **Fill in schematic reference designators** button in the top toolbar. There are some options available, but the defaults are fine. Click **Annotate** and then **Close**. The symbols are now annotated with reference designators R1, D1, and BT1.

Symbol Properties

Next, fill in values for each component. Select the LED, right-click, and select Properties… (E). This project will use a red LED, so change the Value field to be red. In a real project, it might be better to write the LED manufacturer's part number here instead. Note that it's possible to edit reference designators individually in each symbol's properties.

This project will use a 3V lithium coin cell battery, so change the Value field of BT1 to 3V. Change the resistor's value to 1k.

Footprint Assignment

Finally, assign a footprint to each component. This defines how each component will attach to the PCB. Some symbols come with footprints pre-assigned, but for many components there are multiple possible footprints, so the user needs to select the appropriate one.

There are several ways to assign footprints, but one convenient way is to use the footprint assignment tool by clicking the button in the top toolbar.

The left pane of this window lists the available footprint libraries. The middle pane shows the symbols in the schematic. All of these symbols will get footprints assigned to them. The right pane shows the footprints that can be chosen for the symbol selected in the middle pane. Footprints can be viewed by right-clicking a footprint and selecting **View selected footprint**.

Many footprints are included with KiCad, so the footprint assignment tool offers several ways to filter out footprints that aren't relevant to the symbol in question.

- The leftmost button activates filters that can be defined in each symbol. For example, an opamp symbol might define filters that show only SOIC and DIP footprints. Sometimes those predefined filters are missing or too restrictive, so it can be useful to turn this filter off in some situations.
- The middle button filters by pin count, so that only 8-pad footprints are shown for 8-pin symbols. This filter is almost always useful.
- The right button filters by selected library. Libraries are selected in the left pane; footprints not in the selected library will be filtered out. This filter is useful as long as it is known which library contains the right footprint. Often it's best to use this filter or the symbol filters, but not both.
- The text box filters out footprints that don't match the text in the box. This filter is disabled when the box is empty.

Using the filters, find and assign the footprints shown in the screenshot above. Click **OK**.

There are other ways to assign footprints; one way is through the symbol properties window. For more information on assigning footprints, see the manual.

Electrical Rules Check

The last remaining thing to do in the schematic is to check for electrical errors. KiCad's Electrical Rules Checker (ERC) cannot make sure that the design in the schematic will work, but it can check for some common connection issues such as unconnected pins, two power outputs shorted together, or a power input that isn't powered by anything. It also checks for some other mistakes like symbols that aren't annotated and typos in net labels. To see the full list of electrical rules and to adjust their severity, go to **Schematic Setup → Electrical Rules → Violation Severity**. It is a good idea to run ERC before starting layout.Run an electrical rules check by clicking the **ERC** button in the top toolbar and then clicking Run ERC.Even in this simple schematic, KiCad has found two potential errors. The errors are listed in the ERC window, and arrows point to the violation locations in the schematic. Selecting a violation in the ERC window highlights the corresponding arrow.

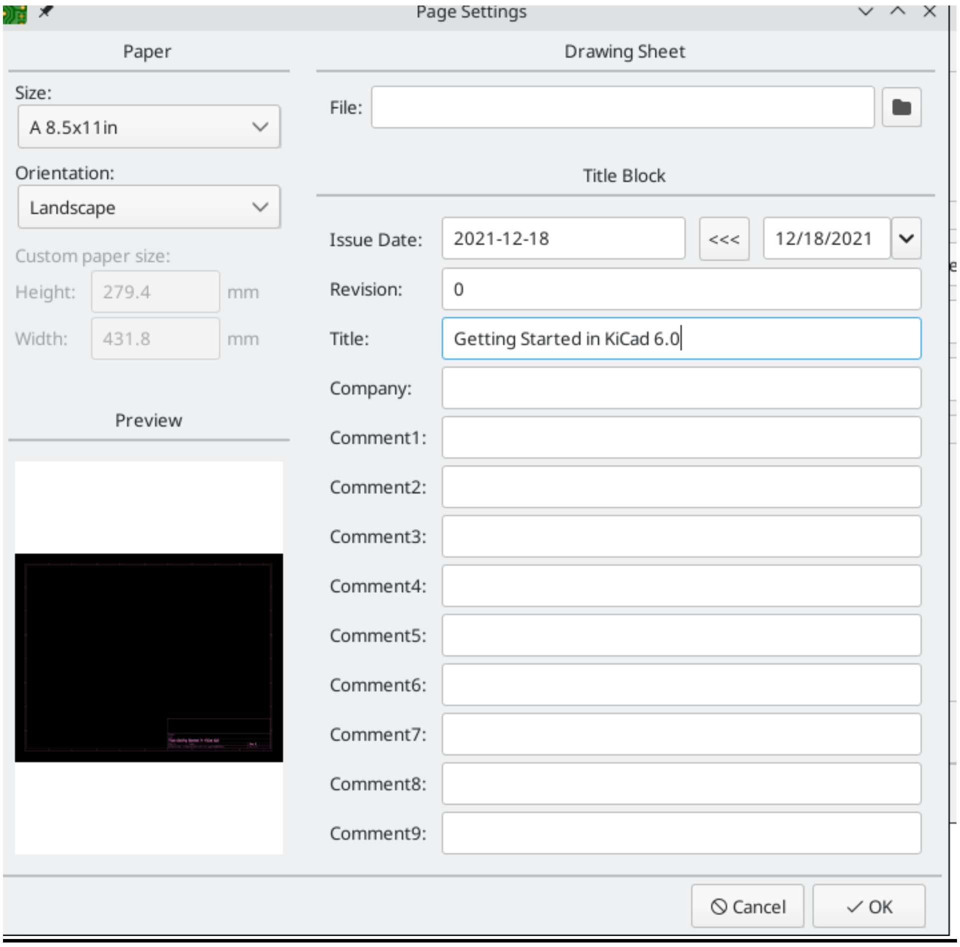
Tutorial Part 3: Circuit Board

With the schematic completed, go back to the Project Window and open the PCB Editor, either by clicking the PCB Editor button or opening the board file.

PCB Editor Basics

Navigation in the PCB Editor is the same as the Schematic editor: pan by dragging with the middle mouse button or right mouse button, and zoom with the scrollwheel or F1 / F2 .At far right is the Appearance Panel and Selection Filter. The Appearance panel is used to change visibility, colors, and opacity of PCB layers, objects, and nets. The active layer is changed by clicking on the name of a layer.Below the Appearance Panel is the Selection Filter, which enables and disables selection of various types of PCB objects. This is useful to select specific items in a crowded layout.

Board Setup and Stackup



Before designing the board, set the page size and add information to the title block. Click **File → Page Settings…**, then choose an appropriate paper size and enter a date, revision, andtitle. Next, go to **File → Board Setup…** to define how the PCB will be manufactured. The most important settings are the stackup, i.e. what copper and dielectric layers the PCB will have (and their thicknesses), and the design rules, e.g. sizes and spacing for tracks and vias.

To set the stackup, open the **Board Stackup → Physical Stackup** page of the Board Setup window. For this guide, leave the number of copper layers at 2, but more complicated projects might require more layers. Next, go to the **Design Rules → Constraints** page. The settings on this page specify the overriding design rules for everything in the board design. For the purposes of this guide, the defaults are fine. However, for a real project these should be set according to the capabilities of the PCB fab house so that the PCB design is manufacturable. Finally, open the **Design Rules → Net Classes** page. A net class is a set of design rules associated with a specific group of nets. This page lists the design rules for each net class in the design and allows assigning nets to each net class. Trackwidth and spacing can be managed manually by the designer during layout, but net classes are recommended because they provide an automatic way to manage and check design rules.In this design, all nets will belong to the Default net class, and the default design rules for this net class areacceptable. Other designs may have multiple net classes, each with different design rules. For example a board might have a High Current netclass with wide traces, or a 50 Ohm netclass with specific width and clearance rules for 50 Ohm controlled-impedance traces.

Drawing A Board Outline

Now the three components have been placed, but the board itself has not been defined. The board is defined by drawing a board outline on the Edge.Cuts layer.

It's often useful to draw the board outline with a coarse grid, which makes it easy to get round numbers for the board size. Switch to a coarse grid by selecting 1mm in the Grid dropdown menu above the canvas. To draw on the Edge.Cuts layer, click **Edge.Cuts** in the Layers tab of the Appearance panel at right. Choose the rectangle tool in the right toolbar, and use it to draw a rectangle roughly surrounding the three footprints. The other graphic tools (line , arc , circle , or polygon) could also be used to define the board outline; the only requirement is that the outline is asingle closed shape that doesn't intersect itself.

Placing Footprints

The next step in the layout process is to arrange the footprints on the board. In general, there are a several considerations for positioning footprints:

- Some footprints may have exact requirements for their locations, such as connectors, indicators, or buttons and switches.

- Some components may need to be placed according to electrical considerations. Bypass capacitors should be close to the power pins of the associated IC and sensitive analog components should be far from digital interference.
- Almost all components have a "Courtyard" (or two if both Front and Back are defined). Generally Courtyards should not intersect.
- Otherwise components should be positioned for ease of routing. Connected components should generally be close together, and arranged to minimize routing complexity. The ratsnest (the thin lines indicating connections between pads) is useful for determining how best position footprints relative to other footprints.

For the purposes of this guide, the only placement goal is to make the routing process as simple as possible.Start by moving the battery holder BT1 onto the back side of the board. Click it to select it, then press M to move it. Press F to flip it to the opposite side; it now appears mirrored and its pads have changed from red to blue.

All PCB layers are viewed from front side of the board. Footprints on the bottom of the board are therefore upside down and appear mirrored.

Each PCB layer has a unique color, which is shown by the swatches in the Layers tab of the Appearance panel. In the default color scheme, items on the F.Cu (Front Copper) layer are red, while items on the B.Cu (Back Copper) are blue.

Now place the other two components. One at a time, select each component, then move and rotate it with M and R. Watch the ratsnest lines between each pad to choose the simplest arrangement of components; a good arrangement will leave the lines untangled. One possible arrangement is shown in the screenshot below.

EAGLE is available from Autodesk either as part of the Fusion 360 software package or you can a free version of just EAGLE here. Grab the version that matches your operating system (the software is available for Windows, Mac and Linux). It's a relatively light download -- about 125MB.

EAGLE installs just like most programs. The download is an executable file; open it and follow the installation instructions.

Licensing EAGLE

One of our favorite things about EAGLE is that it can be used for **free**! The free download is limited version for hobbyists. The free download is a Personal Learning License that may be used by individuals for *personal*, *non-commercial* use. There are a few limitations to be aware of when using the free version:

- Your PCB design is limited to a maximum size of $80cm^2$ ($12.4in^2$) of PCB real estate, which is still pretty darn big. Even if you're designing a big 'ol Arduino shield, you'll still be well under the maximum size.
- Only two signal layers allowed. If you need more layers look into upgrading to the single-user Fusion 360 license.
- Only two sheets in your schematic editor.
- The Personal Learning License is limited to *personal*, *non-commercial* use by individuals. If you're going to go out and sell your design, you'll want to check out the single-user EAGLE license included with a Fusion 360 License.

Those limitations still make EAGLE an amazing piece of software. Engineers here at SparkFun could design 99% of our boards using the freeware version, if not for that pesky non-profit stipulation. You still have access to all phases of the EAGLE software, including the Autorouter.

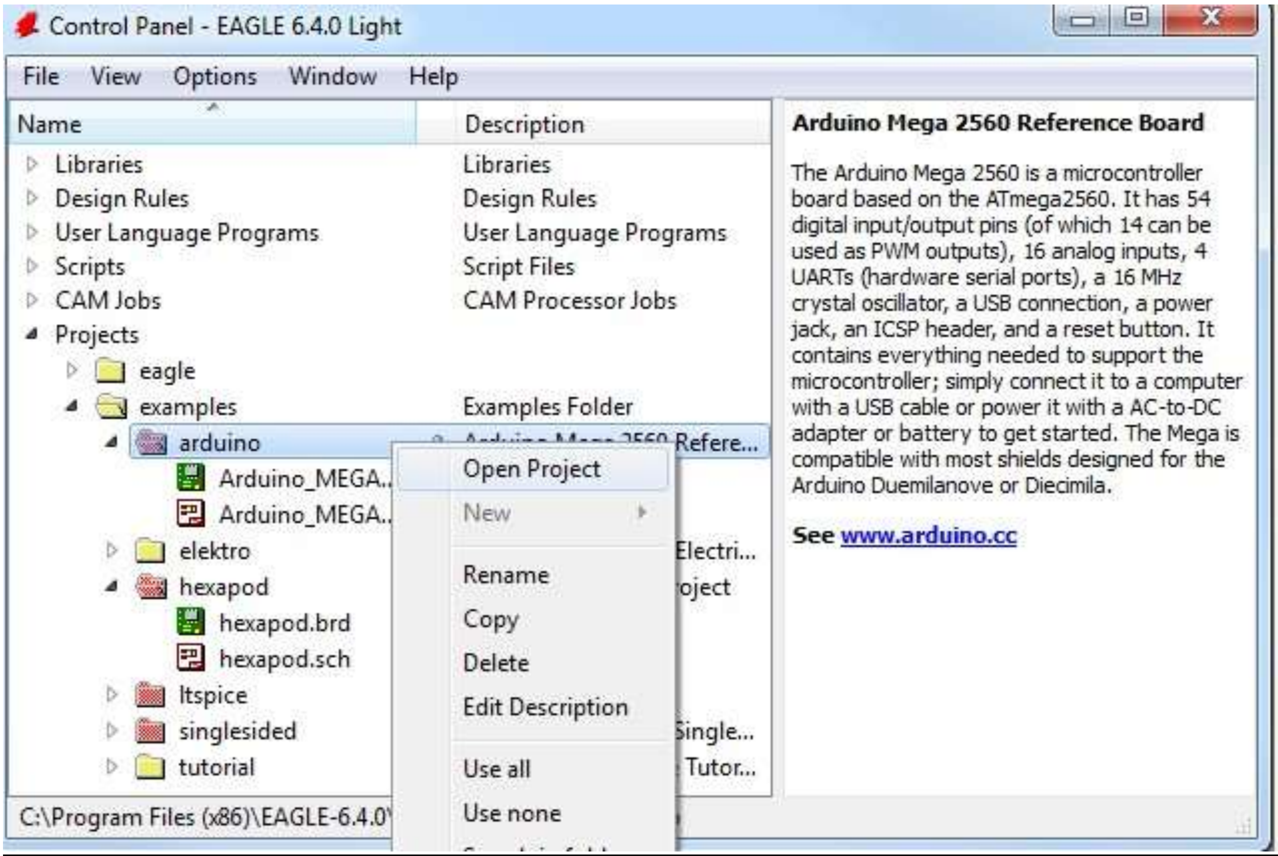
Step 1: Download the SparkFun Libraries

Step 2: Updating the Directories Window

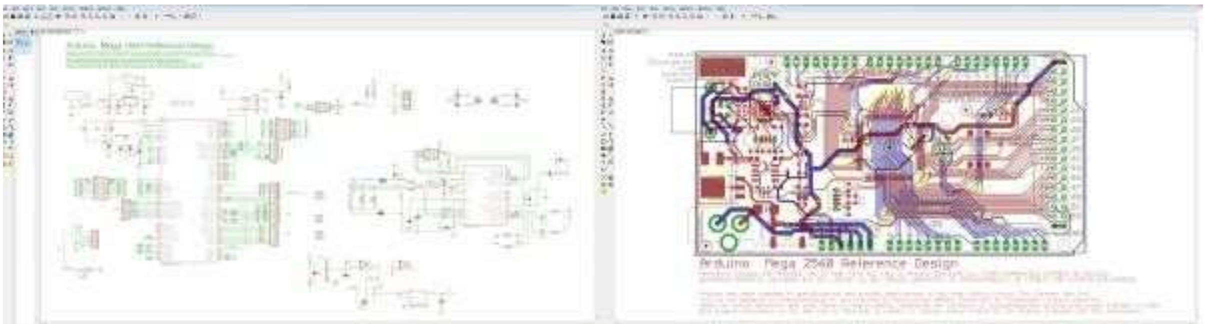
Step 3: "Using" Libraries

Opening a Project and Explore

EAGLE is packaged with a handful of nifty example PCB designs. Open one up by expanding the "Projects" tree. From there, under the "examples" folder open up the "arduino" project by double-clicking the red folder (or right-clicking and selecting "Open project"). Note that, in this view, project folders are red and regular folders are the standard yellow.



Opening the project should cause two more EAGLE windows to spawn: the board and schematic editors. These are the yin and the yang of EAGLE. They should be used together to create the finished product that is a functional PCB design.



The **schematic editor** (on the left above) is a collection of red circuit symbols which are interconnected with green nets (or wires). A project's schematic is like the comments in a program's code. It helps tell the story of what the board design actually does, but it doesn't have much influence on the end product. Parts in a schematic aren't precisely measured, they're laid out and connected in a way that's easy to read, to help you and others understand what's going on with the board design.

The **board editor** is where the real magic happens. Here colorful layers overlap and intersect to create a precisely measured PCB design. Two copper layers -- red on top, blue on the bottom -- are strategically routed to make sure different signals don't intersect and short out. Yellow circles (on this design, but they're more often green) called "vias" pass a signal from one side to the other. Bigger vias allow for through-hole parts to be inserted and soldered to the board. Other, currently hidden, layers expose copper so components can be soldered to it.

Keep Both Windows Open!

Both of these windows work hand-in-hand. Any changes made to the schematic are automatically reflected in the board editor. Whenever you're modifying a design it's important to **keep both windows open at all times**.

If, for instance, you closed the board window of a design, but continued to modify a schematic. The changes you made to the schematic wouldn't be reflected in the board design. This is bad. The schematic and board design should always be consistent. It's really painful to backtrack anychanges in an effort to reattain consistency. **Always keep both windows open**!

There are a few ways to tell if you don't have consistency between windows. First, there's a "dot" in the lower-right hand corner of both windows. If the dot is green, everything is groovy. If the dot is magenta, a window's probably closed that shouldn't be. Second, and more obvious, if you close either of the two windows a big, huge warning should pop up.

Navigating the View

This is a subject that's usually glazed over, but it's important to know how to navigate around both of these windows.

To move around within an editor window, a **mouse with a scroll wheel** comes in very handy. You can zoom in and out by rotating the wheel forward and backward. Pressing the wheel down, and moving the mouse allows you to drag the screen around.

Configuring the UI

EAGLE's user interface is highly customizable. Anything from the background color, to layer colors, to key bindings can be modified to fit your preference. Better tailoring your interface can make designing a PCB much easier. On this page we'll talk about how we at SparkFun prefer to customize our UI. None of these steps are required. Customize your UI as you see fit. These are just the settings that we've grown accustomed to.

Setting the Background Color

The first adjustment we always make to the UI is the background color of the board editor. The standard white background doesn't always meld very well with the array of colored layers required for board design. Instead, we usually opt for a black background.

To change the background color, go up to the "Options" menu and select "User interface".

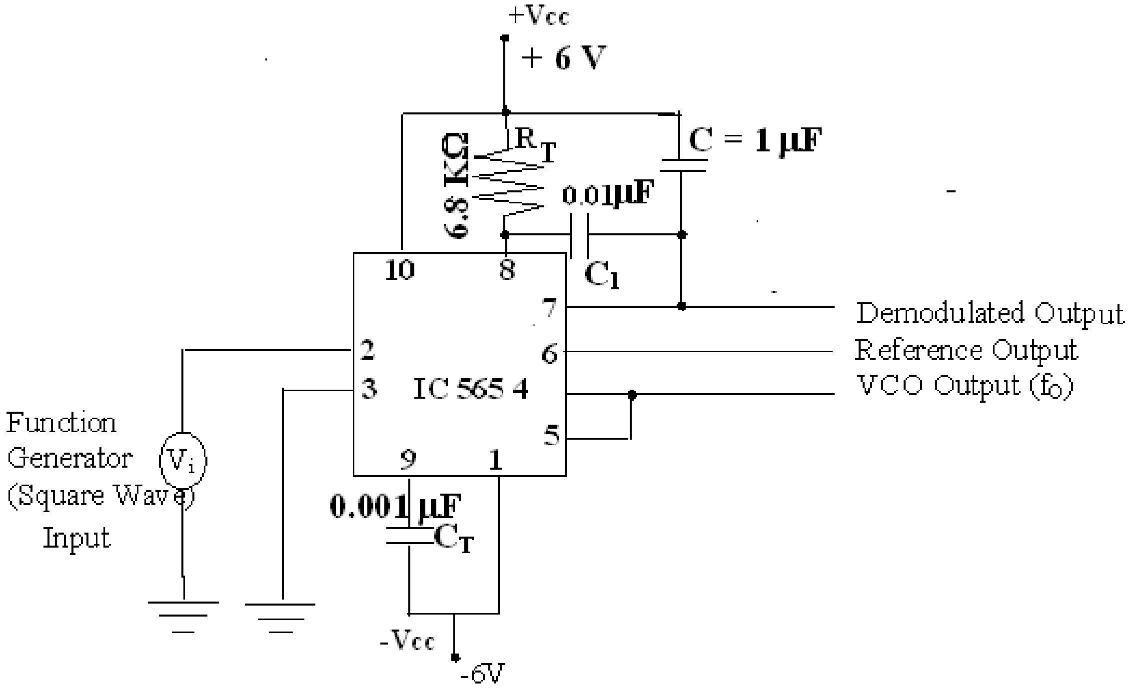
Inside the "Layout" box you can set the background to black, white, or a specific color.

RESULT:

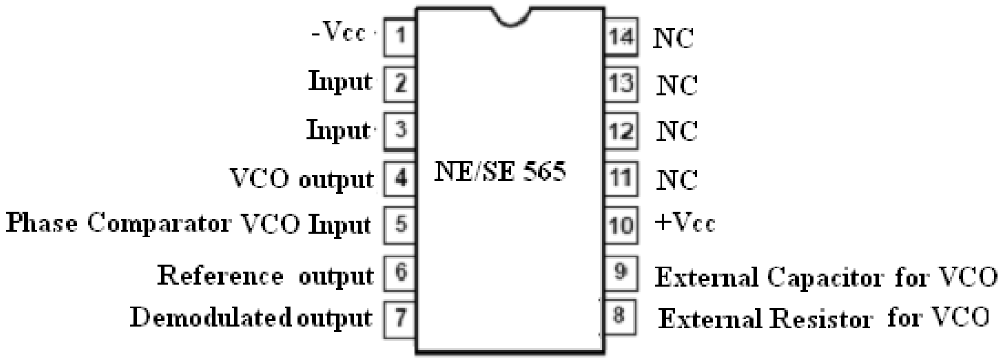
Thus PCB design software (open source) was studied using KiCad, and Eagle.

CIRCUIT DIAGRAM:

PLL



PIN DIAGRAM



EX.NO:13 PHASE LOCKED LOOP CHARACTERISTICS

AIM:

To construct and study the operation of Phase Locked Loop(PLL) IC 565 and determine its Characteristics.

APPARATUS REQUIRED:

S.No	ITEM	RANGE	QTY
1	IC 565	-	1
2	Resistors	6.8 KΩ	1
3	Capacitors	0.001 μF 0.1 μF, 1 μF	1 each
4	Function Generator (Digital)	1 Hz – 2 MHz	1
5	C.R.O	-	1
6	Dual Power Supply	0- 30 V	1

PRE LAB QUESTIONS:

1. What is a VCO?
2. What is PLL?
3. List the features of PLL.
4. Mention The applications of PLL

FORMULA

Free running frequency $f_o = 0.25/(R_T C_T)$

Lock – in range $\Delta f_L = \pm 7.8\ fo / V$

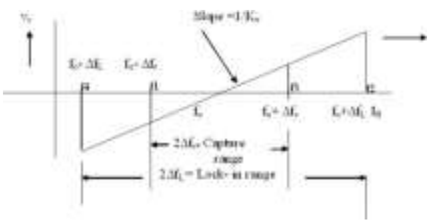
Where , $V = +V_{cc} - (-V_{cc})$

Capture Range $\Delta f_c = \pm (\Delta f_L / (2\pi)(3.6)(10^3) C)^{1/2}$

THEORY:

The PLL consists of Phase detector, Low Pass Filter, Error Amplifierand VCO. The VCO is a free running Multivibrator operates at f0called *free running frequency*. The frequency deviation is directlyproportional to the DC control voltage and hence it is called as VCO.If an input signal Vs of frequency fs is applied to the PLL , the phasedetector compares the phaseand frequency of the incoming signal tothat of the output V0 of the VCO, if the two signals differs infrequency or phase an error voltage is generated, then the phasedetector is a multiplier and produces fs±f0 , the high frequencycomponent fs+f0 is removed by LPF and the Low frequencycomponent is amplified and applied as a control voltage to VCO. Vcshifts the VCO frequency to reduce the frequency difference between fs and f0, once this action starts the signals is in the capture range. The VCO continues to change the frequency till its output frequency is same as the input frequency then it is said to be a locked range.

MODEL GRAPH:



PROCEDURE:

1. The connections are given as per the circuit diagram.
2. Measure the free running frequency of VCO at pin 4, with the input signal V_i set equal to zero. Compare it with the calculated value of f_o= 0.25 / (R_T C_T).
3. Now apply the input signal of 1 V_{PP} square wave at a 1 KHz to pin 2. Connect one channel of the scope to pin 2 and display this signal on the scope.
4. Gradually increase the input frequency till the PLL is locked to the input frequency. This frequency f1 gives the lower end of the capture range. Go on increasing the input frequency, till PLL tracks the input signal, say, to a frequency f2.This frequency f2 gives the upper end of the lock range. If input frequency is increased further, the loop will get unlocked.
5. Now gradually decrease the input frequency till the PLL is again locked. This is the frequency f3, the upper end of the capture range. Keep on decreasing the input frequency until the loop is unlocked. This frequency f4 gives the lower end of the lock range.
6. The lock range Δf_L= (f2 – f4).Compare it with the theoretical value of Δf_L.The capture range is Δf_c = (f3 – f1).Compare it with the theoretical value of capture range

TABULATION

Amplitude of input and output signal -

S.No.	Input Frequency (kHz)	Output Frequency (kHz)

POST LAB QUESTIONS:

1. Define Phase Locked Loop.
2. What are the building blocks of a PLL?
3. What is capture range in PLL?
4. What is lock –in range in PLL?
5. What is free running frequency in PLL?

RESULT:

Thus the PLL circuit is constructed and its characteristics are determined and the frequency multiplier circuit using PLLis constructed and studied.