L-3, JT (Design)
Part B

Level 3 – Trade/Practical Test

Junior Technician (Design)

School of Interdisciplinary Design and Innovation (SIDI)

Instructions to the Candidates

A. Answer all questions provided in the question paper.

B. Each question carries 2 marks. The detailed distribution of marks is specified within each question.

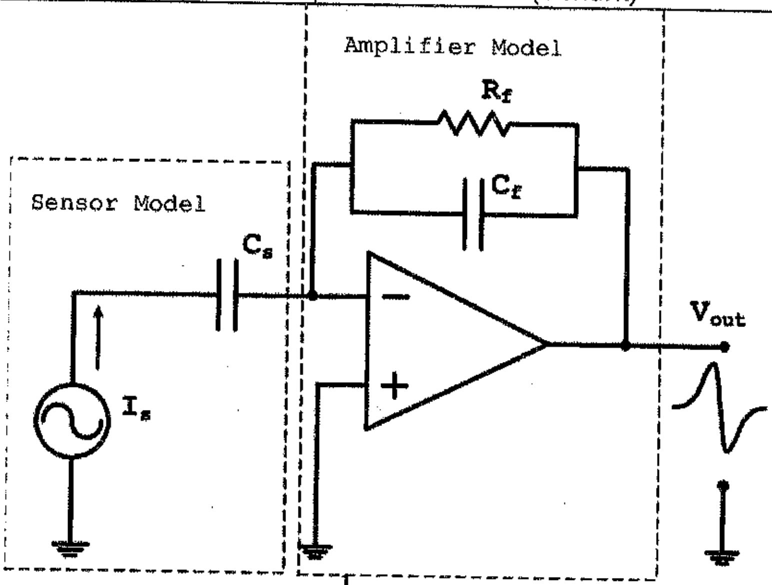
C. After completing each Practical Demonstration, the procedure and results must be verified and authenticated by the examiner.

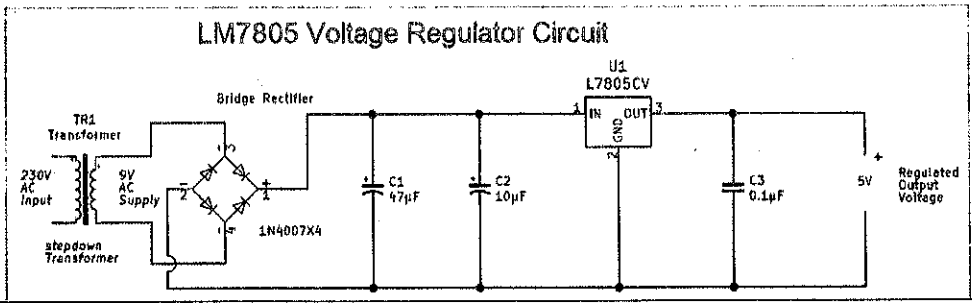
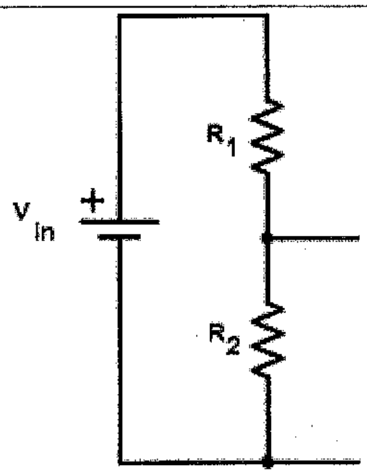
Total Marks: 20 X 2 = 20 marks **Total Time: 120 Minutes**

S.No.	Questions	Marks
1	Design and construct a circuit using an operational amplifier (Op-Amp) that produces an output signal which is 180 degrees out of phase with respect to an audio frequency input signal. (1 Mark) Demonstrate this phase inversion on a breadboard and measure both the input and output signals using an oscilloscope. (1 Mark)	2
Ans	To achieve a 180-degree phase difference between the input and output signals, use an Op-Amp in the inverting amplifier configuration. The circuit consists of an Op-Amp, two resistors (R1 and R2), and a DC power supply. The input signal is applied to the inverting terminal of the Op-Amp through resistor R1, while the non-inverting terminal is grounded. The feedback resistor R2 is connected from the output to the inverting input. The gain of the amplifier is given by ratio of -R2/R1 and the output signal is inverted (180 degrees out of phase) relative to the input signal. Steps for Demonstration • Select appropriate resistor values (e.g., R1 = 10 kΩ, R2 = 10 kΩ for unity gain inversion). • Connect the audio signal source to the inverting input via R1. • Ground the non-inverting input. • Connect the feedback resistor R2 from output to inverting input. • Power the Op-Amp with dual supply (e.g., ±12 V) or single supply with biasing as needed. • Use an oscilloscope to display both the input and output signals. • Observe the waveforms: the output should be a mirror image of the input, indicating a 180-degree phase shift.youtube+1 **Measurement and Observation** • Connect oscilloscope probes: one to the input signal and one to the output. • Adjust the oscilloscope to display both signals on the same screen. • Measure the phase difference by comparing the zero crossings of both waveforms; the time delay between corresponding zero crossings can be used to calculate the phase shift using the formula	

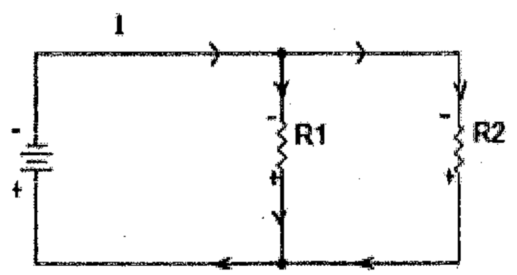
	• The result should confirm a 180-degree phase difference	
2	Design a signal conditioning circuit using an operational amplifier (Op-Amp) for a thermocouple that produces an output voltage of 40 μV per degree Celsius. Show the circuit diagram and explain the working principle Note: Temperature Range: 0 ˚C to 1200 ˚C Output voltage from Signal conditioning circuit: 0 to 5 V No need implements cold junction temperature junction.	2
Ans	The input thermocouple voltage is applied to the non-inverting (+) input of the Op-Amp. The amplifier outputs a voltage linearly proportional to the input, thus to the temperature. The output voltage ranges from 0 V at 0°C to 5 V at 1200°C. Since cold junction compensation is not required as per the note, the circuit focuses on amplification only. Summary The small thermocouple voltage (up to 48 mV) is amplified by a factor of ~104 using a precision Op-Amp to produce a 0–5 V output for 0–1200°C range. Use a low-offset, low-noise Op-Amp like OP07 or INA128 for accuracy.	
3	Design a circuit to convert the change in resistance of a strain gauge into a measurable voltage signal. Assume the strain gauge has a nominal resistance of 100 Ω. (1 Mark) Demonstrate the circuit using a decade resistance box as the strain gauge sensor. (1 Mark)	2
Ans	The most effective way to convert the resistance change of a strain gauge into a voltage signal is by using a Wheatstone bridge configuration. This circuit provides a differential output voltage that varies with changes in the strain gauge resistance. R1, R2, R3: Fixed resistors (100 Ω each, matching the strain gauge nominal value) Rg: Strain gauge (100 Ω nominal, replace with decade resistance box for demonstration) R4, R5: Optional, for bridge balancing (not required for basic demonstration) Vex: Stable DC excitation voltage (typically 5 V or 10 V) Vout: Voltage across the bridge, measured differentially Working Principle The Wheatstone bridge is balanced when all resistors are equal (R1=R2=R3=Rg=100 Ω), resulting in zero output voltage. When the strain gauge resistance changes due to strain, the bridge becomes unbalanced, producing a small differential voltage at the output.	

	This voltage is proportional to the change in resistance and can be amplified using an instrumentation amplifier for better measurement accuracy. For demonstration, use a decade resistance box as Rg and vary its resistance to simulate strain. Observe the corresponding change in output voltage using a voltmeter or oscilloscope. Practical Demonstration Connect the decade resistance box in place of the strain gauge. Apply a stable excitation voltage. Measure the output voltage as the resistance is varied. Observe that the output voltage changes linearly with the resistance change, confirming the conversion of resistance change to voltage signal. This circuit is widely used in strain measurement applications and provides a reliable method for converting small resistance changes into measurable voltage signals	
4	Explain purposes Zero and Span adjustment in a signal conditioning circuit. (1 Mark) How they are typically performed. (1 Mark)	2
Ans	Zero and Span adjustments are essential calibration steps in signal conditioning circuits to ensure accurate and linear measurement outputs. **Zero Adjustment** refers to setting the baseline output of the system when the input signal is at its minimum or zero level. For example, in a temperature measurement system, Zero adjustment ensures that the output reads exactly zero (or a defined baseline like 4 mA in a 4–20 mA system) when the temperature input is zero or at its lowest measurable point. This adjustment compensates for any offset errors in the circuit so that the system starts measuring from the correct reference point. **Span Adjustment**, also called gain adjustment, sets the maximum output of the system to the correct level when the input signal reaches its maximum specified value. It ensures that the output signal scales correctly across the entire input range. For example, in a 0 to 100°C temperature range with a 0–5 V output, Span adjustment will make sure that at 100°C the output is exactly 5 V, calibrating the slope of the input-output characteristic. Typically, zero adjustment is performed first by applying the lowest input and adjusting the output to the baseline value. Then, the span adjustment is done by applying the maximum input and tuning the output to the full-	

	scale value. These adjustments may be performed using potentiometers or digital calibration methods within the signal conditioning hardware. Together, Zero and Span adjustments enable precise measurement by aligning the system output with the actual physical input range, compensating for offsets and scaling errors.	
5	Draw a schematic of signal conditioning circuit to convert the nonlinear output produced by a capacitance sensor, caused by a change in the distance between parallel conductors, into a linear voltage output. (1 Mark) Derive the equation to prove the above (1 Mark)	2
Ans	 • Sensor impedance: $Z_s = \frac{1}{j\omega C_s}$ • Feedback impedance: $Z_f = R_f \parallel \frac{1}{j\omega C_f} = \frac{R_f}{1 + j\omega R_f C_f}$ Sensor current: $$I_s = \frac{V_{in}}{Z_s} = V_{in}\, j\omega C_s$$ Output voltage: $$V_o = -I_s Z_f = -V_{in}\, j\omega C_s \frac{R_f}{1 + j\omega R_f C_f}$$ So the transfer function using $j\omega$ is $$\boxed{\frac{V_o}{V_{in}} = -\frac{j\omega R_f C_s}{1 + j\omega R_f C_f}}$$	
6	Solder the LM7805-based regulated 5 V power supply circuit, as shown, on a single-sided universal PCB (prototyping board). After completing the soldering, measure and record the DC output voltage using a digital	2

	multimeter.  	
Ans	- The circuit consists of: - A 230 V AC to 9 V AC step-down transformer feeding a bridge rectifier made from four 1N4007 diodes. - A smoothing capacitor C1 (47 µF) after the rectifier to reduce ripple. - A second filter/decoupling capacitor C2 (10 µF) at the input of the 7805 regulator IC (U1) for better stability. - A 7805 (L7805CV) linear regulator providing a regulated 5 V DC output. - An output bypass capacitor C3 (0.1 µF) to improve transient response and reduce high-frequency noise. - When correctly soldered and powered from 230 V AC, the transformer and rectifier produce an unregulated DC voltage of about 12–14 V at the input of the 7805. The 7805 regulates this to a constant 5 V DC at its output as long as the input remains within its specified range and the load current is within its ratings. - To verify operation with a digital multimeter: - Set the DMM to measure DC voltage. - Connect the black probe to circuit ground and the red probe to the regulator output node. - A correctly built circuit should show approximately 5.0 V DC at the output under no-load or light-load conditions.	
7	 For the series circuit shown, with two resistors R1 and R2 connected in series across a DC supply Vin: R1 = R2 = 20 Ω, 5 W (or other values as specified by the examiner) Vin = 5 V (or the value specified by the examiner) 1. Measure the voltage across R1 and across R2 using a digital multimeter. (1 Mark)	2

	2. Compare the theoretical and measured values (1 Mark)	
Ans	Experiment: Measure Voltage Across Series Resistors Circuit Description Two resistors R1 and R2 are connected in series across a DC supply Vin: • R1 = 20 Ω, 5 W • R2 = 20 Ω, 5 W • Vin = 5 V --- 1. Theoretical Calculation of Voltages Step 1: Calculate Total Resistance Rtotal=R1+R2=20 Ω+20 Ω=40 Ω $R_{total}=R1+R2=20\ \Omega+20\ \Omega=40\ \Omega$ Step 2: Calculate Circuit Current Using Ohm's Law: I=VinRtotal=5 V40 Ω=0.125 A $I=\frac{V_{in}}{R_{total}}=\frac{5\ V}{40\ \Omega}=0.125\ A$ Step 3: Calculate Voltage Across R1 Using Ohm's Law: VR1=I×R1=0.125 A×20 Ω=2.5 V $V_{R1}=I\times R1=0.125\ A\times 20\ \Omega=2.5\ V$ Step 4: Calculate Voltage Across R2 Using Ohm's Law: VR2=I×R2=0.125 A×20 Ω=2.5 V $V_{R2}=I\times R2=0.125\ A\times 20\ \Omega=2.5\ V$ Theoretical Results • Voltage across R1 = 2.5 V • Voltage across R2 = 2.5 V • Total voltage = 2.5 V + 2.5 V = 5 V ✓ (Checks with Vin)	

	2. Experimental Measurement Procedure Step 1: Set Up the Multimeter 1. Set the digital multimeter (DMM) to DC Voltage (V=) range 2. Select an appropriate range (typically 10 V or 20 V scale) 3. Ensure the black probe is connected to the negative (common) terminal 4. Ensure the red probe is connected to the positive (V) terminal Step 2: Measure Voltage Across R1 1. Place the black probe at the negative end of the circuit (ground/negative terminal of Vin) 2. Place the red probe at the junction between R1 and R2 3. Record the reading (should be approximately 2.5 V) Step 3: Measure Voltage Across R2 1. Place the black probe at the negative terminal of Vin (ground) 2. Place the red probe at the negative end of R2 3. Record the reading (should be approximately 2.5 V) 3. Comparison of Theoretical and Measured Values Expected Results	
8	 For the circuit shown, two resistors R1 and R2 are connected in parallel across a DC supply Vin. R1 = R2 = 20 Ω, 5 W (or other values specified by the examiner) Vin = 5 V (or the value specified by the examiner)	2

<table>
<tr><td></td><td>1. Measure the current through R1 and through R2 using a digital multimeter.(1 Mark)
2.Compare the theoretical and measured values and comment on any differences.(1Mark)</td><td></td></tr>
<tr><td>Ans</td><td>For R1 and R2 in parallel, each resistor is directly across Vin, so each sees the full 5 V.

1. Theoretical current through each resistor

Total resistance of each branch is just the resistor value:

$$I_{R1} = \frac{V_{in}}{R_1} = \frac{5\ \text{V}}{20\ \Omega} = 0.25\ \text{A}$$
$$I_{R2} = \frac{V_{in}}{R_2} = \frac{5\ \text{V}}{20\ \Omega} = 0.25\ \text{A}$$

So, theoretically:

• Current through R1 = 0.25 A
• Current through R2 = 0.25 A

(If Vin or R values differ as per examiner, use I=V/RI = V/RI=V/R for each branch.)

2. Comparison with measured values

• When you measure with the DMM in series with each branch, you should obtain currents close to 0.25 A for R1 and for R2.
• Small differences from 0.25 A can be explained by:
  o Resistor tolerance (actual resistance not exactly 20 Ω).
  o Supply voltage not exactly 5 V under load.
  o Meter accuracy and contact resistance in connections.

If the measured currents are approximately equal and near the theoretical values, the experiment confirms Ohm's law and the behavior of parallel branches.</td><td></td></tr>
<tr><td>9</td><td>Describe the procedure used to identify open circuits and short circuits in PCB traces on a printed circuit board (PCB).</td><td>2</td></tr>
<tr><td>Ans</td><td>To identify opens and shorts on a PCB, always begin with power turned off and use both visual inspection and continuity testing with a multimeter.

1. Identifying open circuits (broken traces)

• Inspect the board under good lighting (and magnifier, if available) for cracked, burnt, or lifted copper traces and unsoldered or poorly soldered pads.</td><td></td></tr>
</table>

	• Using the circuit diagram, select two points that should be electrically connected by a trace (for example, from an IC pin to a resistor pad). • Set the multimeter to continuity or low-ohms mode and place the probes on these two points. • A beep or very low resistance indicates a good connection; no beep or very high resistance indicates an open circuit. • If an open is detected, move one probe step-by-step along accessible intermediate points on the trace to locate the exact position of the break, then repair it with re-soldering or a small wire jumper. 2. Identifying short circuits (unwanted connections) • Visually check for solder bridges between adjacent pads, excess solder, copper slivers, or conductive debris between tracks. • With the multimeter still in continuity or resistance mode, probe between nodes that must be isolated (for example, VCC to GND, or two adjacent signal tracks). • A beep or low resistance reading indicates a short. Systematically test around dense areas such as IC pins and connectors. • After locating the approximate shorted area, remove excess solder with a solder wick, clean residues, or carefully cut any unintended copper connection. 3. Verification after repair • Recheck the previously faulty nets with the multimeter to confirm that opens are now continuous, and shorts are cleared. Only after continuity is correct across all critical nets should the board be powered on and, if needed, further functional tests performed.	
10	Describe the procedure used to check whether a diode mounted on a PCB is functioning correctly.	2
Ans	For verify proper functioning of a diode on a PCB, combine visual inspection with electrical testing using a **digital multimeter (DMM).** 1. Visual inspection and preparation Ensure the PCB is powered off and any large capacitors are discharged to avoid damage to the meter or the board. Inspect the diode for obvious damage: cracks, discoloration, burnt marks, or lifted pads, and confirm the orientation (anode–cathode) matches the circuit markings. Physical damage or wrong orientation usually indicates the diode is faulty or misinstalled. 2. Testing with a DMM (diode / resistance mode) Set the DMM to "diode test" mode (preferred). If not available, use resistance (ohms) mode.	

	Place the red probe on the anode and the black probe on the cathode (forward bias): A good silicon diode typically shows a forward voltage drop of about 0.6–0.7 V. In resistance mode, it should show a moderate resistance in one direction and very high resistance in the other. Reverse the probes (reverse bias): A good standard diode should show "OL" or very high resistance, indicating it blocks current. If it shows nearly 0 V or very low resistance in both directions, the diode is shorted. If it shows "OL" (open) in both directions, the diode is open-circuit. 3. In-circuit considerations In-circuit readings can be influenced by parallel paths (other components). If the readings are ambiguous, desolder one diode lead and repeat the test with that lead lifted. This isolates the diode from the rest of the circuit and gives a reliable result. After confirming the diode is good, resolder the lead and, if required, power the PCB and verify correct voltage levels across the diode during normal operation (e.g., correct rectified DC at a rectifier diode). Using these steps, proper diode operation on a PCB is confirmed when it conducts in the forward direction with a normal voltage drop and blocks current in the reverse direction, with no signs of physical damage or incorrect orientation.	
11	Describe the procedure used to check whether a MOSFET mounted on a PCB is functioning correctly.	2

Ans	To check whether a MOSFET on a PCB is functioning correctly, use this sequence with power initially OFF:
	
	1. Identify pins and inspect visually
	    o Find the MOSFET type (N-channel or P-channel) and its Gate (G), Drain (D), and Source (S) pins from the PCB silkscreen or datasheet.
	    o Look for obvious damage: cracks, burn marks, melted package, or lifted / bridged pads.
	2. Basic multimeter tests (power OFF)
	    o Set the digital multimeter (DMM) to diode-test or resistance mode.	

	o Measure Drain–Source both ways: place probes on D and S, note reading, then reverse probes. ▪ A good enhancement MOSFET usually shows high resistance in both directions (except for the intrinsic body diode in one direction for power MOSFETs). ▪ A very low resistance (near short) in both directions indicates a shorted device. o Measure Gate–Source and Gate–Drain resistance: both should be very high (typically megaohms). ▪ Any low resistance suggests gate-oxide damage; the MOSFET is faulty. 3. In-circuit functional check (power ON, with caution) o Power the PCB, reference the circuit schematic, and measure: ▪ Gate-to-Source voltage VGSV_{GS}VGS: it should move between "OFF" level (≈0 V for N-channel, near supply for P-channel) and a valid drive level (above threshold for N-channel, below for P-channel) when the control signal changes. ▪ Drain-to-Source voltage VDSV_{DS}VDS: ▪ In the ON state, VDSV_{DS}VDS should be low (a small voltage drop), showing the MOSFET is conducting. ▪ In the OFF state, VDSV_{DS}VDS should be close to the supply or expected blocking voltage, showing it is turning off properly. o If VGSV_{GS}VGS is correct but VDSV_{DS}VDS never goes low (or never goes high), the MOSFET is likely defective or incorrectly connected. 4. Isolating ambiguous cases o If parallel components on the PCB make readings unclear, desolder one lead (usually Source or Drain) and repeat the resistance/diode tests out of circuit. o Restore the connection if the MOSFET proves healthy; replace it if any tests indicate a short, open, or leaky junction. Using these steps—visual inspection, static resistance checks, and dynamic voltage measurements—the correct operation or failure of a MOSFET mounted on a PCB can be reliably determined.	
12	Describe the procedure for calibrating a Negative Temperature Coefficient (NTC) thermistor sensor over the temperature range 0 °C to 120 °C.	2
Ans	Calibration of an NTC thermistor establishes an accurate relationship between its resistance (or output voltage in a signal-conditioning circuit) and temperature over the required range. 1) Preparation - Mount the thermistor in its intended measurement configuration (e.g., in a voltage divider or bridge) and connect it to the measurement system (DMM or ADC).	

	- Use a reliable reference thermometer with accuracy better than the required calibration accuracy. - Arrange a temperature-controlled environment that can produce stable points between 0 °C and 120 °C (ice bath for 0 °C, temperature bath or oven for higher points). 2) Selecting calibration points - Choose several temperature points spanning 0–120 °C, for example: 0 °C, 25 °C, 50 °C, 75 °C, 100 °C, and 120 °C. More points give better curve fitting. - At each point, allow sufficient time for thermal equilibrium between the thermistor and the reference (no drift in readings for a few minutes). 3) Data acquisition - At each calibration temperature: - Record the reference temperature. - Measure the thermistor resistance directly (ohmmeter) or compute it from the circuit (e.g., from divider voltage). - Build a table of (T, R) data pairs across the full range. 4) Generating the calibration curve - Use the collected data to derive a temperature–resistance relationship. Common approaches are: - Fit the Steinhart–Hart equation $$T = \frac{1}{A + B\ln(R) + C[\ln(R)]^3}$$ (T in kelvin), solving for constants A, B, C using three or more calibration points. - For simpler systems, create a piecewise linear or polynomial fit, or store a lookup table with interpolation in firmware. - Implement this relationship in the signal-conditioning or processing stage so that measured resistance (or voltage) is converted to calibrated temperature. 5) Verification - After computing the calibration equation or table, test the thermistor at a few intermediate temperatures (e.g., 10 °C, 60 °C, 110 °C). - Compare calculated temperatures with the reference; if errors exceed the allowable limit, repeat calibration with more points or refine the fitting method. This process yields a calibrated NTC thermistor whose output can be accurately converted to temperature over the 0–120 °C range.	
13	If the current to be measured is 1 mA, which current range should be selected on a digital multimeter, and why?	2

Ans	The appropriate range is the lowest milliampere range that is greater than 1 mA, typically the 2 mA or 20 mA DC current range (depending on the multimeter). This choice is justified because: The selected range must be higher than the expected current so the meter is not overloaded or its fuse blown. Using the smallest range that still exceeds the expected value gives the best resolution and accuracy; on a 2 mA or 20 mA range, a 1 mA reading will use a large portion of the display span, making the reading easier to see and reducing relative error compared with using a high range such as 200 mA or 10 A.	
14	Using the available function generator, generate a sine wave of frequency 2 kHz and amplitude 1 V_{PP}. Observe the waveform on the oscilloscope (without using any automatic measurement or autoset features). Compare the set (generated) waveform parameters with the values measured on the oscilloscope.	2

Ans	- Set the function generator to "Sine" output.
	- Adjust the frequency control to 2 kHz (use the frequency dial and range switches).
	- Set the amplitude so that the output is 1 VPP; if needed, use the oscilloscope as a reference while adjusting.
	- Connect the function generator output to Channel-1 of the oscilloscope using a BNC cable, and connect grounds properly.
	- On the oscilloscope:
	  - Select AC coupling for the input channel.
	  - Manually adjust the time base so that several cycles of the waveform (e.g., 0.1 ms/div for 2 kHz) are visible.
	  - Manually set the vertical scale so that the whole 1 VPP waveform occupies most of the screen (e.g., 0.2 V/div or 0.5 V/div depending on the scope).
	- Use the graticule to measure:
	  - Peak-to-peak voltage: count vertical divisions from top peak to bottom peak and multiply by volts/div.
	  - Period: count horizontal divisions for one full cycle and multiply by time/div, then compute frequency (f = 1/T).
	
	Comparison and Observation
	
	- The generator is set to 2 kHz and 1 V_{PP}.
	- The oscilloscope measurement should show:
	  - Frequency close to 2 kHz (period ≈ 0.5 ms per cycle).
	  - Amplitude close to 1 V_{PP}
	- Any small difference between set and measured values is due to instrument tolerances, calibration errors, and manual reading accuracy.
	- Note whether the waveform shape is a clean sine (smooth curves, no visible distortion). If the observed waveform is distorted or the amplitude varies with load,	

	students should comment on possible causes such as generator limitations, incorrect termination, or oscilloscope settings.	
15	A sine wave of frequency 1 kHz and amplitude 10 V_{PP} is generated using a function generator. The waveform is to be observed on an oscilloscope using a 10X probe oscilloscope (without using any automatic measurement or autoset features). Describe the steps to be followed on the oscilloscope to correctly observe this waveform.	2
Ans	1. **Connections and probe settings** • Connect the 10X probe tip to the function generator output and the probe ground lead to the generator ground. • Set the switch on the probe to "10X". • On the oscilloscope, set the input channel's probe attenuation to 10X so that the displayed voltage scale is correct. 2. **Basic channel setup** • Select the channel used (e.g., CH1) and set coupling to "DC". • Start with a vertical scale such as 1 V/div (with 10X probe this corresponds to 10 V/div at the signal), then reduce to 0.5 V/div or 0.2 V/div as needed so that the full 10 V_{PP} waveform occupies a good portion of the screen. • Set the vertical position so the waveform is centered vertically. 3. **Time-base and trigger** • Set the time base to show several cycles of a 1 kHz signal; a good starting point is 0.2 ms/div or 0.5 ms/div. • Set trigger source to the same channel, trigger mode to "Normal" or "Auto", and trigger edge to rising. Adjust the trigger level to intersect the middle of the waveform so the display is stable. 4. **Fine adjustment and verification** • Adjust volts/div and time/div manually (without autoset) so the waveform is clear and fills the screen appropriately. • Confirm that one period spans about 1 ms (≈1 kHz) and that the peak-to-peak height corresponds to 10 V (taking the 10X probe factor into account).	
16	What is meant by line regulation and load regulation in a linear regulated power supply?	2
Ans	Line regulation and load regulation are both usually expressed as a percentage change in output voltage. **Line regulation (LR)** Line regulation quantifies the change in output voltage when the input (line) voltage changes and load current is kept constant.	

	$$\text{Line Regulation (\%)} = \frac{V_{OUT,MAX\ LINE} - V_{OUT,MIN\ LINE}}{V_{OUT,NOM}} \times 100$$ where • $V_{OUT,MAX\ LINE}$ = output voltage at maximum specified input • $V_{OUT,MIN\ LINE}$ = output voltage at minimum specified input • $V_{OUT,NOM}$ = nominal output voltage. **Load regulation (LoR)** Load regulation quantifies the change in output voltage when the load current changes from minimum (or no-load) to maximum, with input voltage held constant. $$\text{Load Regulation (\%)} = \frac{V_{OUT,NO\ LOAD} - V_{OUT,FULL\ LOAD}}{V_{OUT,NOM}} \times 100$$ where • $V_{OUT,NO\ LOAD}$ = output at minimum load current • $V_{OUT,FULL\ LOAD}$ = output at maximum load current $V_{OUT,NOM}$ = nominal output voltage.	
17	Explain how a regulated DC power supply can be used as a constant current source of 0.5 A. Describe the circuit arrangement (using only a wire), show how to set the current, and outline the steps to demonstrate the constant-current operation.	2
Ans	Connect a wire in positive and negative terminal of power supply. Adjust the constant current knob up to 0.5 A Current observed on display or using multimeter	
18	What is Electrostatic Discharge (ESD), and how should integrated circuits (ICs) be handled correctly to avoid damage due to ESD?	2
ANs	Electrostatic Discharge (ESD) is the sudden flow of electric charge between two objects at different electrical potentials, usually caused by contact or close proximity after one or both objects have accumulated static electricity. A typical example is the small "shock" felt when touching a metal object after walking on a carpet. The voltage in such events can reach several kilovolts and can easily damage the thin oxide layers and junctions inside ICs, even when no visible spark is seen. To prevent ESD damage while handling ICs, follow these practices: 1) Personal grounding - Wear an antistatic wrist strap connected to a proper ground point so that any static charge on your body is safely dissipated. - Avoid synthetic clothing that generates static; prefer cotton garments in ESD-sensitive areas.[3][6][8]	

	2) ESD-safe work area - Work only on an antistatic (ESD) mat connected to ground, and ensure the soldering iron is ESD-safe and grounded. - Keep humidity at a moderate level where possible, since very dry air increases static buildup. - Clearly mark and use ESD Protected Areas (EPAs) for handling sensitive ICs.[6][8] 3) Handling and storage of ICs - Always store and transport ICs in ESD-protective packaging such as antistatic bags, conductive tubes, or trays. - When picking up ICs, hold them by the package body and avoid touching the pins directly. - Before removing an IC from its package or inserting it into a socket/PCB, briefly touch a grounded metal part to discharge any residual static.[2][5][6] 4) PCB and assembly precautions - Do not place ICs or PCBs directly on plastic, paper, or other insulating surfaces that can hold static charge. - When passing boards or ICs between people, ensure both are grounded or touch a common grounded object first. - After assembly, keep finished boards in ESD-safe containers until they are installed in their final, grounded equipment.[7][6] By combining proper grounding, ESD-safe workstations, protective packaging, and careful handling of IC leads and PCBs, the risk of ESD-induced failures in integrated circuits is greatly reduced	
19	What precautions (at least 2) should be taken while connecting an electrolytic capacitor in a circuit?	2
Ans	• **Observe polarity.** Electrolytic capacitors are polarized; the positive terminal must be connected to the higher potential and the negative terminal to the lower potential (usually ground). Reversing polarity can cause excessive leakage current, heating, venting, or explosion. • **Respect voltage rating.** Ensure the applied DC voltage (including ripple and surges) never exceeds the capacitor's rated voltage. Using a capacitor with insufficient voltage rating accelerates degradation and can lead to dielectric breakdown and rupture. • **Consider ripple current and temperature.** Use the capacitor within its specified ripple-current and temperature limits, and keep it away from heat sources such as power resistors or heatsinks. Excess ripple current or high ambient temperature shortens life and may cause bulging or leakage. • **Discharge before handling and avoid shorts.** Always power off the circuit and discharge the capacitor safely through a resistor before touching or reworking the circuit, and never short the leads	

directly with tools. This prevents electric shock, sparks, and damage to PCB tracks or other components.

- **Mounting and mechanical care.**
  Do not bend, twist, or stress the leads close to the can, and avoid touching or covering the safety vent on the top of the capacitor. Mechanical stress or blocking the vent can prevent safe pressure relief if the capacitor fails.

20 The figure shows a portion of the LM741 operational amplifier datasheet giving its absolute maximum ratings. Based on this information, what precautions should be taken during PCB assembly of the LM741 — 2

6 Specifications

6.1 Absolute Maximum Ratings

over operating free-air temperature range (unless otherwise noted)(1)(2)(3)

		MIN	MAX	UI
Supply voltage	LM741, LM741A		±22	
	LM741C		±18	
Power dissipation (4)			500	m
Differential input voltage			±30	
Input voltage (5)			±15	
Output short circuit duration		Continuous		
Operating temperature	LM741, LM741A	–50	125	
	LM741C	0	70	
Junction temperature	LM741, LM741A		150	
	LM741C		100	
Soldering information	PDIP package (10 seconds)		260	
	CDIP or TO-99 package (10 seconds)		300	
Storage temperature, T_{stg}		–65	150	

Ans

1. Respect supply-voltage and input limits during testing
   - Do not apply more than ±22 V (LM741/LM741A) or ±18 V (LM741C) between the supply pins when powering the assembled PCB.
   - Ensure the differential input voltage does not exceed ±30 V and that each input pin remains within the specified input-voltage range (typically ±15 V) with respect to ground or supply rails during bring-up and functional tests. Exceeding these limits can permanently damage the input stage.

2. Observe power-dissipation and temperature ratings
   - Keep total package power dissipation below 500 mW by choosing appropriate supply voltages, output loading, and ambient conditions; avoid driving heavy loads directly from the op-amp.
   - Ensure the device operates only within its specified ambient temperature range (for example, 0 to 70 °C for LM741C) and that nearby power components or inadequate ventilation do not cause junction temperature to exceed its limit. Use adequate copper area or airflow if necessary.

3. Follow soldering temperature and time limits
   - **During PCB soldering, respect the maximum soldering temperature and duration given in the datasheet (for example, 260 °C for 10 s for PDIP and 300 °C for 10 s for some metal-can packages).**
   - Avoid prolonged contact with the soldering iron and prevent repeated rework on the same pins to avoid internal bond-wire or package damage.

	4. Protect against output short circuits and ESD - Although the datasheet may specify "continuous" output short-circuit capability, avoid intentional or prolonged shorts to ground or supply rails during assembly and testing to prevent excessive package heating. - Handle the IC in an ESD-safe manner (grounded wrist strap, antistatic mat and packaging) to prevent latent damage to internal junctions. 5. Storage and handling - **Store LM741 devices within the specified storage-temperature range (for example, −65 °C to 150 °C) and avoid exposure to high humidity or corrosive atmospheres before assembly.** - Do not mechanically stress or bend the leads close to the package more than necessary; insert and remove the IC vertically from sockets or PCBs to prevent lead or package cracking.	