NATIONAL UNIVERSITY OF SCIENCE AND TECHNOLOGY

FACULTY OF INDUSTRIAL TECHNOLOGY

DEPARTMENT OF ELECTRONIC ENGINEERING

CAD FOR ELECTRONIC ENGINEERS (EEE 1232)

Final Examination Paper

October 2024

Time Allowed	:	3 hours
Total Marks	:	100
Special Requirements	:	Computer loaded with Proteus and Microsoft Office
Examiner's Name	:	Mr B Dube, L Moyo

INSTRUCTIONS

1. This examination paper consists of five (5) printed pages and sections **A** and **B.**

2. Answer **ALL** the questions in section A and three (3) questions from section B.

3. Each question in **Section B** carries 20 marks.

4. Show any calculations and assumptions clearly (Use of calculators is permissible).

5. Start the answers for each question on a fresh page in section B.

6. Create a new folder on the desktop and name it
   "your_studentNumber_EEE1232_Exam2024".

7. All schematics and simulation results **MUST** be saved in an MS Word file named
   "your_studentNumber_Exam2024"

SECTION A [40 marks]

Answer ALL Questions and write your answers in the Answer book provided

A1. State and explain any two (2) uses of graphics to an Electronic Engineer. **[4]**

A2. State and explain two (2) specific advantages of using flow charts. **[4]**

A3. A computer program accepts two numbers entered by a user. It compares the two numbers. If the average of the numbers is greater than 12, it displays the average, if the average is less than 12, it displays the product of the two numbers and checks for input again. Draw a flow chart which shows the algorithm. **[6]**

A4. Design a poster inviting students for a public lecture on circuit analysis techniques using MS Word. Save your poster together with other answers in the file.
[8]

A5. The store-man in the department wants to assess and present to the rest of the department the movement of his stock in the previous semester. He finds out that he disbursed 100 resistors, 80 transistors, 65 logic gates, 45 capacitors and 90 operational amplifiers. He asks you to advise him on the best way to present this information to quickly show which components are in demand. Use an appropriate graphic to represent this information. **[5]**

A6. Draw a truth table and a logic diagram for a digital circuit which has four inputs A, B, C, D and one output Z. Use AND, OR, NAND, NOR gates to implement the following logical statement:
Z = (NOT A) AND (((B OR C) AND (NOT D)) OR (A AND C)). **[8]**

A7. SPICE simulation simplifies the design of integrated circuits. There is however a major difference between DC analysis and transient analysis.

a. What does SPICE mean? **[2]**

b. What is the difference between DC analysis and transient analysis? **[3]**

SECTION B [60 marks]

Answer three questions in this section. Show your design in the booklet and your design circuit in the file as given in the instructions. Every design should be simulated using proteus. (Ensure to submit your 'filename.DSN' and 'filename.PWI' files (or any related files) and a copy of your schematic and results on a MS word file as a backup)

B1. Draw the following circuit shown in Figure B1, using proteus and run the simulation of the circuit using AC analysis. **[6]**
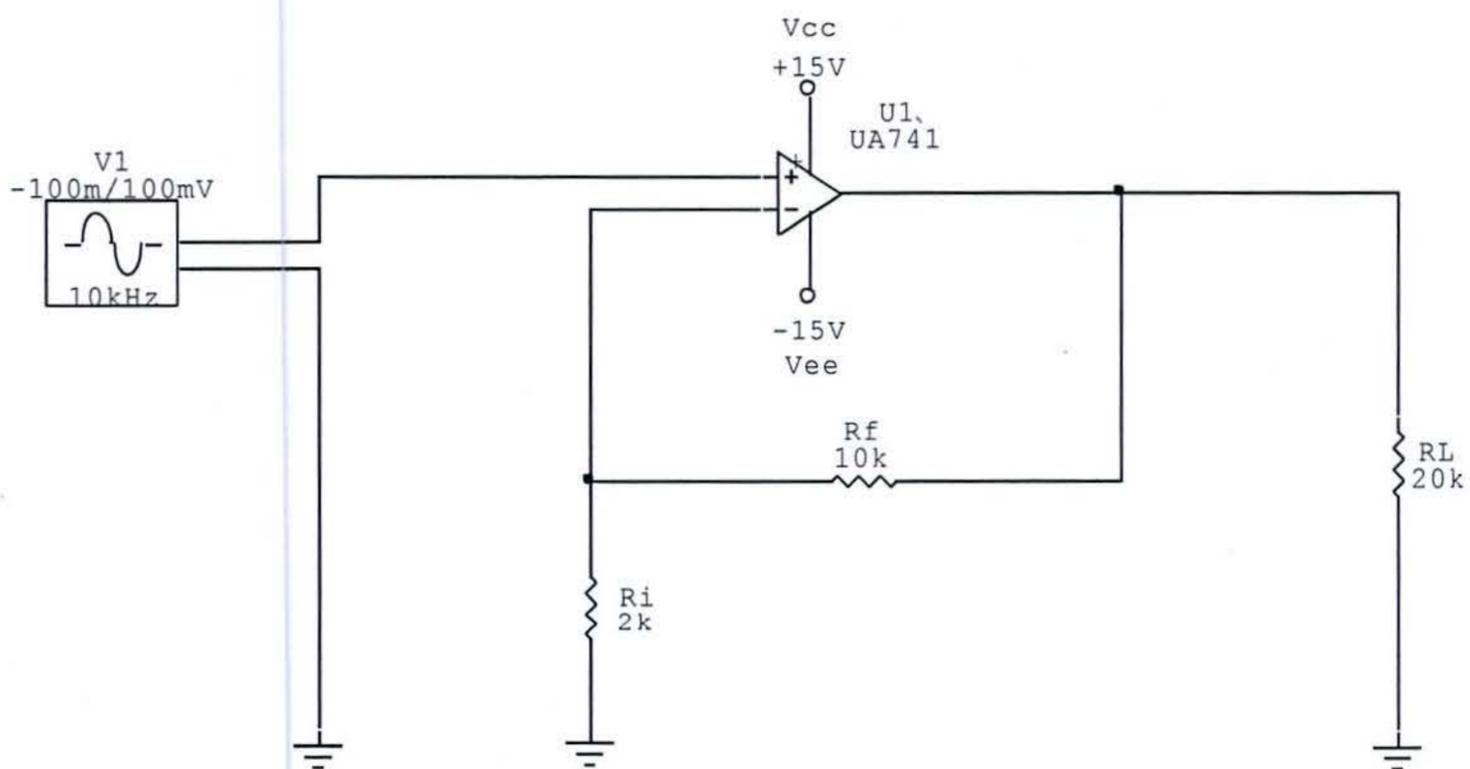
Figure B1

a). Show the input and the output signals using oscilloscope on the same window. **[8]**

b). Describe the relationship between the input and the output. **[2]**

c). From your simulation, what is the amplitude of the output? What is the

ratio of the output to the input? **[2]**

d). Express the relationship of this ratio to Rf and Ri? **[2]**

B2. Design a logic circuit using two terminal logic gates to implement the following equation. Q = (((A OR B) AND (C AND D)) AND ((/A AND /B) OR (/B AND /D))) AND (A OR D)

where A, B, C, D are inputs and Q is output. A slash "/" means that a value is complimented (that is NOT operation is performed on it).

Draw the truth table and the logic circuit diagram in your booklet, Redraw the circuit diagram using proteus and add active logic state sources to all inputs to simulate the circuit and transfer your results to MS word. **[20]**

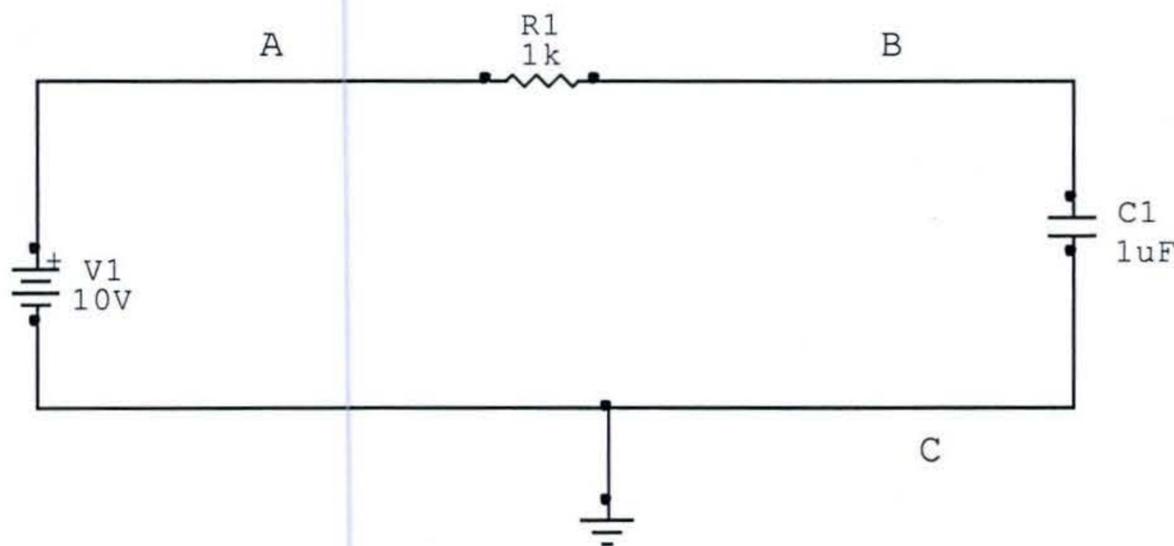**B3.** Shown below is an RC circuit

Figure B3

    i.   Explain how the initial capacitor voltage is initialized to zero (discharged) **[2]**

    ii.   Explain why the capacitor must be initialized to zero? **[2]**

    iii.   Run the simulation and using an oscilloscope, show the transient response at points A, B and C (preferably in one plot). **[4]**

    iv.   Stop the simulation, change R1 to 100 Ω and C1 to 0.001µF. **[3]**

    v.   Then run the simulation again and show the transient response at point B. **[3]**

vi.   How long does it take for the capacitor to charge up to 10v?   **[2]**

vii.   Remove the battery for the discharging process, show the transient response using an oscilloscope at points A, B and C.   **[4]**

B4. a) Draw a voltage divider circuit using 4 series resistors with values R1 = 20 ohms, R2 = 47 ohms, R3= 10 ohms, R4 = 15 ohms

    i)   Using Vin = 9V DC, use DC voltmeters to measure and show the potential differences across each resistor and transfer your results to MS word.

    ii)   Using Vin = 110V AC, use an oscilloscope to measure and show the voltage waveforms across each resistor, accurately calibrate the oscilloscope and deduce the frequency and amplitude of each waveform, transfer your results to MS word.

b) Draw a current divider circuit using 3 parallel resistors with values R1 = 15 ohms, R2 = 25 ohms, R3 = 10 ohms, Using Vin = 24V DC, use DC ammeters to measure and show the currents drawn by each resistor and transfer your results to MS word.

c) Draw an inverting three input summing amplifier circuit using 741 operational amplifier with R1 = 10k, R2 = 5K, R3 = 20K, Rf = 47k, V1 = 3V2 = 2V3 = 5V DC. Measure the output voltage and the currents through each input resistor and transfer your results to MS word.

[20]