Register No.:   ....   Name: ....

SAINTGITS COLLEGE OF ENGINEERING (AUTONOMOUS)

(AFFILIATED TO APJ ABDUL KALAM TECHNOLOGICAL UNIVERSITY, THIRUVANANTHAPURAM)

FIFTH SEMESTER B.TECH DEGREE EXAMINATION (R), DECEMBER 2023
ELECTRICAL AND ELECTRONICS ENGINEERING

(2020 SCHEME)

Course Code: **20EET391**

Course Name: **Digital Simulation**

Max. Marks: **100**   **Duration: 3 Hours**

PART A
(Answer all questions. Each question carries 3 marks)

1. Describe the term sensitivity analysis in simulation problems with the help of an example.
2. Justify the need for nodal analysis in circuit simulation.
3. Define global error.
4. Describe the term stability in transient analysis using numerical techniques.
5. Describe a stiff system and its features.
6. Describe basic idea of adaptive/variable step size in circuit simulation.
7. Explain the '.dot' commands in PSpice for DC and AC simulation problems.
8. Discuss the PSpice commands for representation of indutcances and capacitances.
9. Describe the syntax and functions of any three arithmetic operations on arrays.
10. Explain the function of "while loop" with the help of an example.

PART B
(Answer one full question from each module, each question carries 14marks)
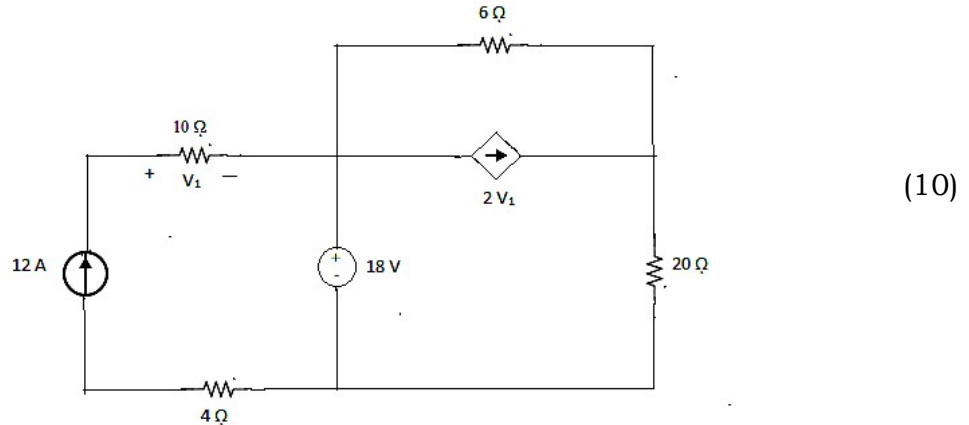### MODULE I

11. Obtain the Modified Nodal Analysis (MNA) matrix for the following circuit
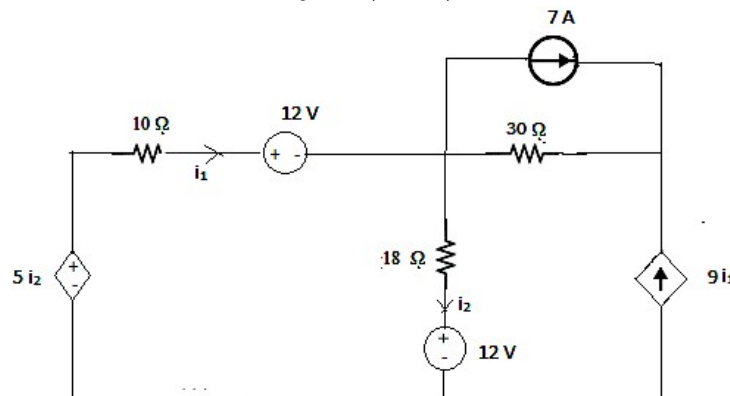
(14)

OR

12. a) Formulate the MNA matrix for the following circuits.

(10)

   b) Explain noise analysis in circuit simulation.   (4)

MODULE II

13. Describe the basic idea and steps in forward euler method for solving diferential equations with the help of an example.   (14)

OR

14. Find the value of y(0.7) by Eulers method for the ODE given by
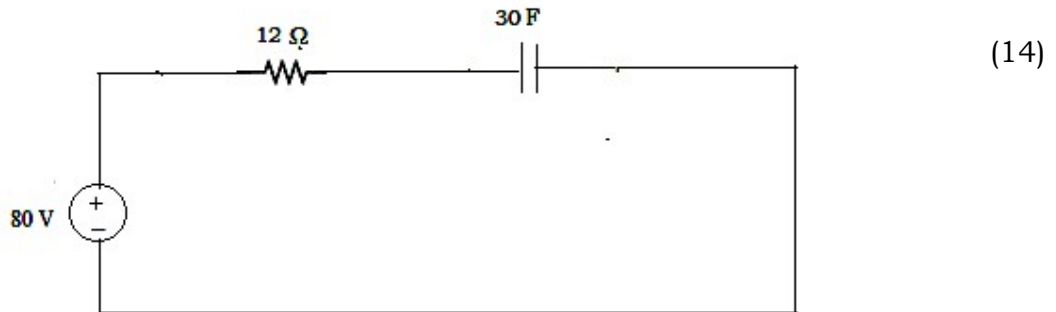$\frac{dy}{dx} + 2y = x^3 e^{-2x}$, y(0)=1. Use 7 iterations.   (14)

MODULE III

15. Find the voltage across the capacitor at t=1 sec in the following circuit by Euler's method. Use 5 iterations.

V_c=10 Volts at t = 0 sec, where V_c is the voltage across the capacitor.

Current through the capacitor is given by,

$$i_c(t) = 10.\frac{V}{R} + \frac{V_C}{R.C}e^{-5t}$$

(14)

OR

16. Explain the equivalent circuit approach in circuit simulation of electrical equipment. Describe the method for formulation of the equivalent circuit of any one electrical equipment/system with necessary equations.   (14)

MODULE IV

17. Formulate a PSpice program to simulate the voltages and currents acorss 30 kΩ and 100 kΩ resistors in the following circuit

(14)
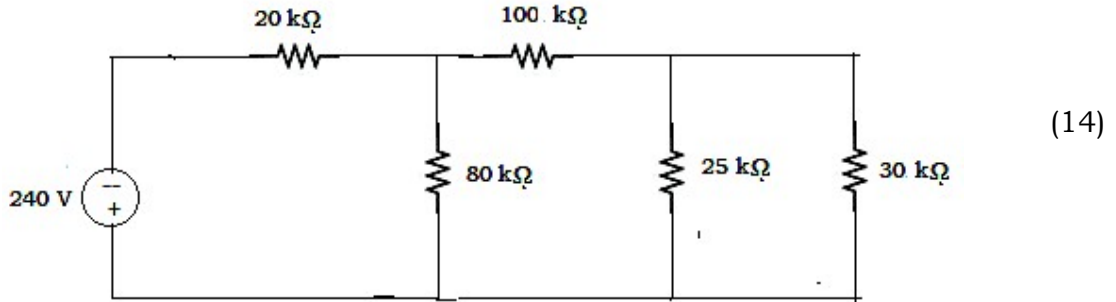
OR

18. Formulate a PSpice program to find the voltage and currents through all the circuit eleements.

(14)
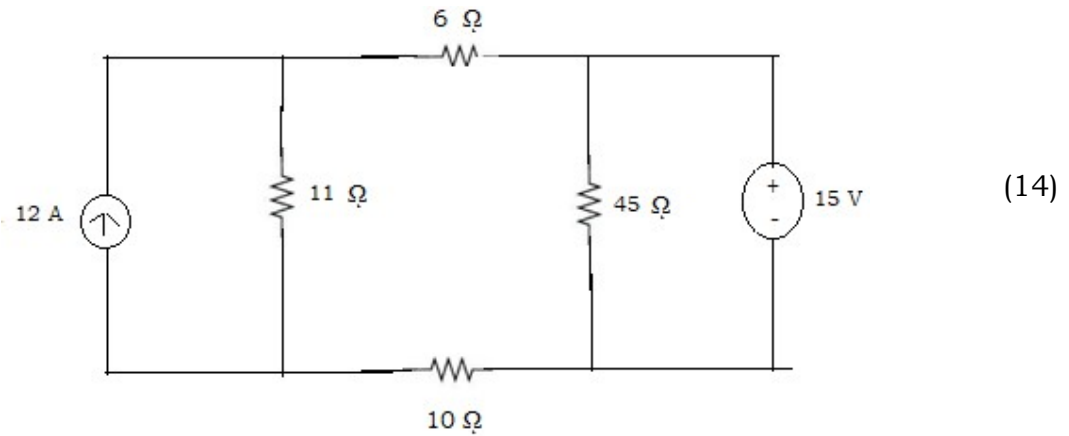
MODULE V

19. A series RLC circuit is excited by a 230 V, 50Hz AC supply. The circuit elements are R= 50 Ω, L= 2 H and C=40 µF. Formulate a MATLAB program to tune the value of a capacitor bank connected in series with the circuit to make the power factor unity in the circuit by measuring votages, current and power in the circuit during each step of tuning process. (14)

OR

20. Explain visual modelling scheme in MATLAB and explain the function of any 5 fundamental blocks. Describe the MATLAB Simulink block diagram of any second order system. (14)

****