Reg No.:_______________ Name:__________________________

APJ ABDUL KALAM TECHNOLOGICAL UNIVERSITY

Fifth Semester B.Tech (Hons) Degree Examination December 2021 (2019 admission)

Course Code: EET393

Course Name: DIGITAL SIMULATION

Max. Marks: 100 Duration: 3 Hours

PART A

(Answer all questions; each question carries 3 marks)

		Marks
1	List any three types of simulation problems. Indicate briefly what each type intends to address.	3
2	Under what circumstances we use damping in Newton-Raphson method?	3
3	"Step-size is a significant parameter in transient simulation, with regard to its outcomes." Give three points to substantiate the above statement.	3
4	Identify the reasons that lead to global errorduring a numerical simulation.	3
5	Why is adaptive step-size used in circuit simulation?	3
6	For an Ordinary Differential Equation (ODE) defined by $x' = f(x,t)$, write the Backward Differentiation Formula 2 (BDF2) expression for $x_{n+1}$with h as the time step-size. Apply it on the equation $x' = -x$ and derive the expression for x_{n+1}. (Note: x' is $\frac{dx}{dt}$).	3
7	Explain the use of .NET directive in SPICE, with the help of one example.	3
8	When do you choose a .DC directive in SPICE simulation? Give a short example.	3

9	What does the following MATLAB code output?
	`A = [1 2 3 4; 5 6 7 8; 9 10 11 12; 13 14 15 16];`
	`A(4,3)`
	`A(12)`
	`A(5,3) = 20`
		3

10	Draw the simulation flow diagram to solve the following ODE: $4\ddot{y} + 4\dot{y} + 10 = 30t$; *(Here, $\ddot{y}$ indicates the second derivative and $\dot{y}$ indicates the first derivative of y, both with respect to time).*	3

PART B

(Answer one full question from each module, each question carries 14 marks)

Module -1

11 a) For the circuit in Fig. 1, form the Modified Nodal Analysis (MNA) equations and represent them in a matrix format. Assume, if necessary, dummy sourcesand corresponding additional nodes with respect tocontrolling currents. 10

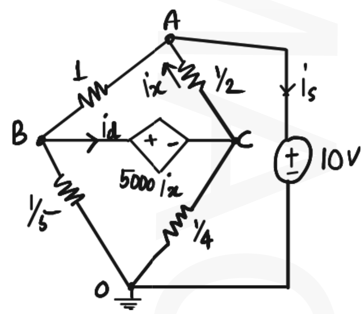

Figure 1: Circuit for Question 11 (a).

b) With suitable example, show anysituation where Newton-Raphson method for solution of nonlinear equations fails to converge. 4

12 For the following circuit in Fig. 2, formulate the Sparse Tableau Approach (STA) based equations.The node voltages are V_A, V_B and V_C, the branch voltages are V_1, V_2, V_3, V_4 and V_5, and the branch currents are i_1, i_2, i_3, i_4 and i_5. The STA matrix is in the format: $\begin{bmatrix} A & 0 & 0 \\ 0 & I & -A^T \\ M_{BCE} & & 0 \end{bmatrix}$. Identify the matrices A, I, M_{BCE} and the RHS vector from the nodal KCL equations, Branch Voltage Definition (BVD) equations and Branch Constitutive Equations (BCE). 14

(*Note:* M_{BCE} *is a* 5x10 *matrix*)

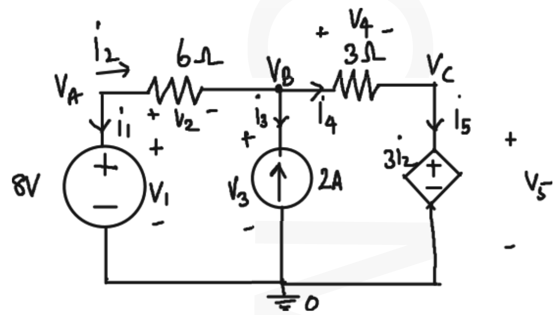

Figure 2: Circuit for STA Analysis, Question 12)

Module –2

13 a) For the ODE $\frac{dx}{dt} = -10x$, apply Forward Euler (FE) method with a step size of 0.25 and initial value of $x_0 = 1$. Proceed for 5 steps and comment on the stability of the method.Is FE method suitable for circuit simulation? Give reasons. 8

b) Define Local Error (LE) and Local Truncation Error (LTE) for a numerical method. 6

14 For the ODE: $\frac{dx}{dt} = -x + 1 - t, x(0) = 1$, apply Backward Euler to solve for x, for t = 0 to t = 1, with steps of h = 0.1. The exact solution is $x = 2 - t - e^{-t}$. Show the exact and numerically computed values at each step in a table. 14

Module –3

15 For the circuit shown in Fig. 3, write the MNA equations. Apply Backward Euler method to discretize and obtain the resulting numerical equations. Express the equations in matrix format.Assume *h* is the step size. (No need to solve the equations).*(Hint: the unknowns are $V_{1,(n+1)}$, $V_{2,(n+1)}$, $V_{3,(n+1)}$ and $i_{s,(n+1)}$).* 14

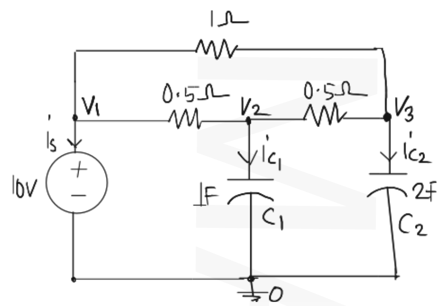

Figure 3: Figure for Question 15.

16 a) What are "Stiff" systems? Give an example of a stiff system. What are the difficulties in simulating stiff systems? 8

b) Explain some of the practical issues in numerical simulation regarding selection of algorithms for specific kinds of systems/circuits. 6

Module -4

17 a) A SPICE circuit file is shown below. Inspect the listing and draw the corresponding circuit diagram neatly. What kind of simulation is used here? 8

```
* TITLE LINE
R1 N001 N002 2K
R2 N001 N003 2K
R3 N002 N004 101K
R4 N003 N005 100K
C1 N003 N004 .01µ
C2 N005 N002 .01µ
V1 N001 0 5
```

```
Q1 N003 N005 0 0 2N3904
Q2 N002 N004 0 0 2N3904
.model NPN NPN
.lib standard.bjt
*standard.bjt contains the model 2N3904 for Q1 and Q2.
.tran 25m startup
.end
```

b) A SPICE frequency response analysis is to be set up for the parallel RLC circuit shown in Figure 4, with parameter value of C2 as10 pF, 100 pF and 2000 pF respectively. Develop the SPICE circuit description for this task. The source is a current source. Assume amplitude of AC exciting source to be 5 mA, and the frequency range to be from 10 Hz to 100 Megahertz. Assume 50 steps per decade. 6

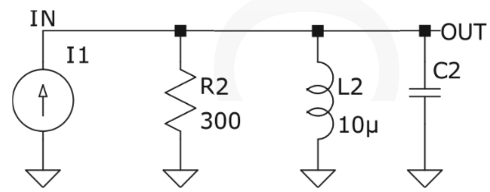

Figure 4: Frequency Response of Parallel RLC circuit (Question 17(b))

18 Figure 5 shows a filter circuit using opamps. Assume that the opamp model is described as a subcircuit in a file “opamp.sub”. The interface nodes for the opamp subcircuit are in the order: Inverting input, Non-inverting input and Output. Assign node numbers to the circuit given and develop the SPICE circuit file for obtaining the frequency response characteristics from 1 Hz to 10 MHz. A suitable number of steps per decade may be specified. Use 1 V as the amplitude of the exciting AC signal. It may be assumed that the opamp subcircuit need no special parameters to be specified. 14

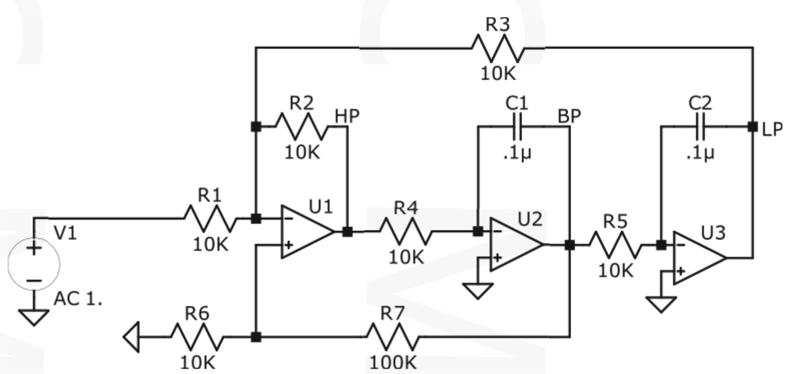

Figure5: A filter circuit (Question18))

Module -5

19 a) Consider anODE of the form $\dot{x} = f(t, x)$, where $f(t, x) = -4\sin(2t) + 6\cos(2t)$, in the range $0 < t < 10$. The initial value $x(0) = x_0 = 3$.Write a MATLAB script to solve the ODE for $x$using Forward Euler method, without using dedicated ode functions in MATLAB. Use a user-input value for the time-step. 4

b) For the two – loop circuit shown in figure 6, assume mesh currents i_1 and i_2 and capacitor voltages V_{C1} and V_{C2}. Write the dynamic equations for the meshes in terms of these variables and develop the simulation diagram using basic blocks (summer, gain, integrator etc.). Assume V_{in} as the input constant dc excitation and the two capacitor voltages and the inductor currents as the output variables. 10

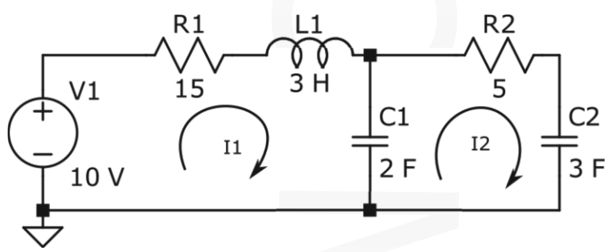

Figure 6: A two loop dynamic circuit (Question 19 (b))

20 a) Write a MATLAB function ‘`fsum`’ that computes the following sum by receiving the value of x and n as arguments. 4

$$\sum_{i=1}^{n} \left(\frac{2}{x}\right)^{i}$$

To call the function, the user should use the syntax: `y = fsum(x, n).`

b) The following equations define the dynamic operation of a separately excited DC motor with constant field excitation: 10

1. Electrical equation: $V_a = i_a r_a + L_a \frac{di_a}{dt} + e_b$; Where V_a is the armature voltage applied (V), i_a is the armature current (A), r_a is the armature resistance (Ω), L_a is the armature inductance (H), and e_b is the back emf (V).
2. Electromechanical equations: (a).$e_b = k_b\omega$, Where k_b is the back emf constant (V/rad/s), and ω is the angular speed in rad/s.

   (b). $T_e = k_t i_a$, where T_e is the electromagnetic torque developed by the motor (N-m) and k_t is the torque constant (N-m/A).

3. Mechanical equation: $T_e = J\frac{d\omega}{dt} + B\omega + T_L$, where, J is the rotational moment of inertia, $B$is viscous friction constant and T_L is the load torque.

It is required to develop a simulation model for the separately excited dc motor with constant field excitation, based on the above equations. Develop the model with armature voltage V_a and load torque T_L as input/excitation variables and angular speed ω and armature current i_a as output variables, using basic blocks such as summers, gain blocks, integrators etc.

***