

2025

**ELECTRONICS — HONOURS**

**Paper : SEC-3**

**(Circuit Simulation with PSpice)**

**Full Marks : 75**

*The figures in the margin indicate full marks.*

*Candidates are required to give their answers in their own words  
as far as practicable.*

Answer **question no. 1** from **Group - I** and **any three** questions from **Group - II** and  
**any four** questions from **Group - III**.

**Group - I**

1. Answer **any ten** questions :

2×10

- (a) Which of the following is a graphical interface used with PSpice?
- |                   |                |
|-------------------|----------------|
| (i) OrCAD capture | (ii) Scilab    |
| (iii) Proteus     | (iv) Multisim. |
- (b) Which type of analysis is used in PSpice to study the behavior of a circuit with time-varying signals?
- |                           |                          |
|---------------------------|--------------------------|
| (i) DC analysis           | (ii) AC analysis         |
| (iii) Bias point analysis | (iv) Transient analysis. |
- (c) Which command is used to obtain the DC operating point of a circuit?
- |           |            |
|-----------|------------|
| (i) .DC   | (ii) .TRAN |
| (iii) .OP | (iv) .AC.  |
- (d) What file type is used to save a text-based circuit description (netlist) for PSpice analysis?
- |            |            |
|------------|------------|
| (i) .sch   | (ii) .txt  |
| (iii) .net | (iv) .cir. |
- (e) If you want to analyze how changing a specific resistor value affects the overall circuit performance, which analysis method would be most suitable?
- |                    |                           |
|--------------------|---------------------------|
| (i) Noise Analysis | (ii) Parameter Sweep      |
| (iii) AC Sweep     | (iv) Bias Point Analysis. |

**Please Turn Over**

**(4805)**

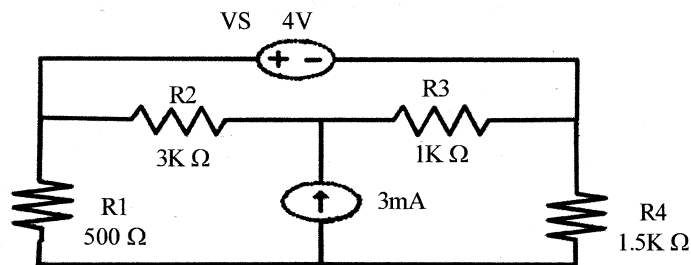
- (f) Which statement is used to define model parameters?
- (i) .PARAM (ii) .MODEL  
(iii) .PRINT (iv) .WIDTH.
- (g) In DC analysis, a capacitor behaves as
- (i) Short circuit (ii) Open circuit  
(iii) Voltage source (iv) Current source.
- (h) MOSFET I-V characteristics mainly describe
- (i) gate leakage (ii) operating regions of MOSFET  
(iii) power dissipation (iv) switching delay.
- (i) In statement '.TRAN 1 ns 100 ns UIC.OP 20 ns' operating point is calculated at t =
- (i) 1 ns (ii) 100 ns  
(iii) 2 ns (iv) 20 ns.
- (j) In PSpice, a small-signal input/output gain and impedance calculation is termed as
- (i) Transfer Function Analysis (ii) Transient Analysis  
(iii) Sensitivity Analysis (iv) Pole Zero Analysis.
- (k) The .PLOT command is mainly used to
- (i) display numerical output (ii) generate graphical output  
(iii) define sources (iv) control simulation time.
- (l) Which statement is true for .TF analysis?
- (i) It finds transient response (ii) It determines transfer function  
(iii) It performs Fourier analysis (iv) It plots frequency response.

### Group - II

2. Write a PSpice netlist to obtain the current through a PN diode in forward biased mode. Also plot the I-V characteristics. 5
3. (a) Explain different types of SPICE software and mention their areas of applications.  
(b) Describe the structure of an input circuit file in PSpice. 3+2
4. Write a PSpice netlist to obtain the output voltage of a series positive clipper using p-n diode. 5
5. Write a PSpice netlist to show the transient behavior of series LR circuit. 5
6. Write a PSpice netlist to obtain the current through a Zener diode in reverse biased mode. Also plot the I-V characteristics. 5

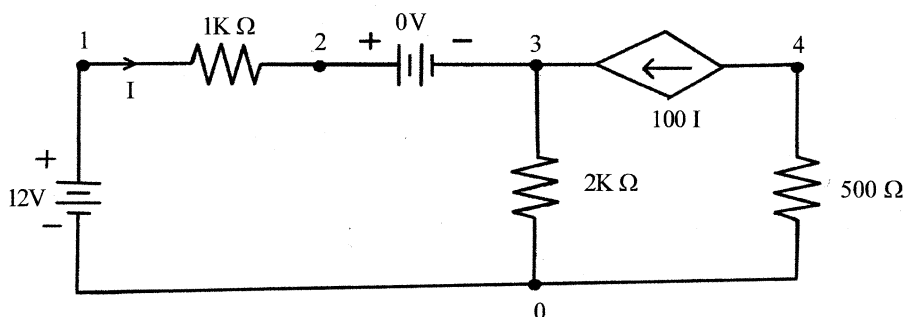
## Group - III

7. (a) Write a PSpice netlist to show the frequency response of a CE amplifier using BJT.  
 (b) Write a PSpice netlist to study the DC characteristics of an enhancement-mode NMOS transistor. 5+5
8. Write short notes on (*any two*) : 5×2
- (a) PSpice A/D  
 (b) OrCAD capture  
 (c) Circuit nodes  
 (d) PSpice output commands.
9. Write a PSpice netlist for the following circuit to calculate :  
 (a) All node voltages  
 (b) Current through R2 and R4 resistances.



4+(3+3)

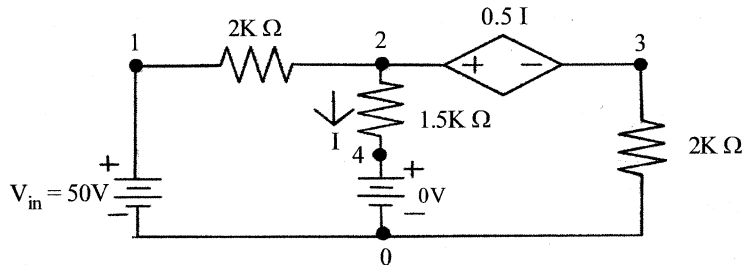
10. What is VCCS? How is it representing in PSpice? Write the PSpice netlist for the following circuit for DC analysis. Also find the Current (I) through the 500Ω Resistance and the voltages at nodes 3 and 4. 1+2+3+2+2



Please Turn Over

(4805)

11. Write the PSpice netlist for the following circuit to obtain the current I and Voltage at node 2. Vary  $V_{in}$  from 0 to 50V in steps 2V. 5+5



12. What is CCCS? How it is represented in PSpice? Write the PSpice netlist for the circuit to determine the voltages at node 2 and 3. 1+2+4+3

