



DJJ-003-015204

Seat No. \_\_\_\_\_

**M. Sc. (Electronics) (Sem. II) (CBCS) Examination**

May / June – 2015

**Paper - 8 : Introduction to PSPICE**

**Faculty Code : 003**

**Subject Code : 015204**

Time : Hours]

[Total Marks : 70

**Q.1 Answer the following: TRUE or FALSE [any seven] [14]**

1. SPICE is a general purpose circuit program that simulates electronics circuits.
2. PSPICE can perform only a time-domain response analysis.
3. SPICE is available only on mainframe computers.
4. AC analysis is used for small-signal analysis of circuits with sources of variable frequencies.
5. In ORCAD the file containing simulation has extension of .cir
6. .OP stands for 'opening parameters'.
7. .PRINT commands print the value in output file.
8. The simulation of dc circuits with passive elements requires the modeling of Resistors.
9. The simulation of dc circuits with passive elements requires the modeling of AC Sources.
10. The results are available for graphical displays through .PROBE command.

**Q.2 Answer following: (Any Two) [14]**

1. Write notes on Types of Analysis
2. The circuit of Fig. 1, calculate and print the voltage at 4, the current  $I_R$  and  $I_{R3}$  for  $V_S=5V, 10V$  and  $30 V$ .

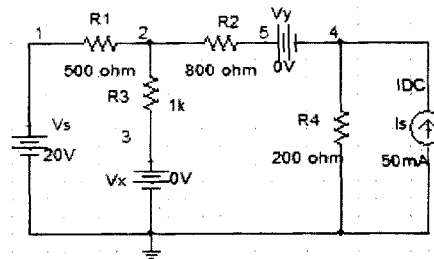


Fig. 1

3. The circuit of Fig 1, calculate and print the voltage at 4, the current  $I_R$  and  $I_{R3}$  for  $V_S=5V$  and  $30 V$  for each value of  $I_S = 50 mA$  and  $150 mA$  for the following circuit. The current  $I_{R2}$  is to be plotted.

**Q.3 Answer following: [14]**

1. An RLC circuit is shown in Fig. 2. Write various currents and voltages in forms that are allowed by PSpice.

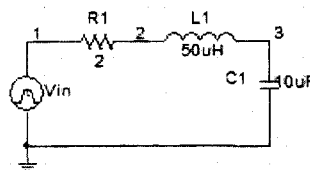


Fig. 2

2. Write note on Limitations of PSPICE.

OR

**Q.3 Answer following:**

[14]

1. Write note on Modeling of Elements.
2. Repeat calculation of Fig 1 with the values of R1 and R2 increased by +5% and those of R3 and R4 decreased by 10%.

**Q.4 Answer following: (Any Two)**

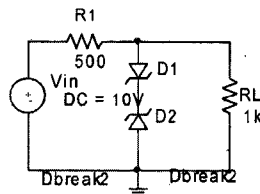
[14]

1. Write note on Transient Analysis
2. Write note on Switches
3. List AC output variables

**Q.5 Answer following: (Any Two)**

[14]

1. List the Advanced PSPICE Commands and Analysis.
2. A Zener voltage regulator is shown in below fig. Plot the dc transfer characteristic if the input voltage is varied from -15V to 15V with an increment of 0.5V. The Zener voltages of the diodes are the same and  $V_z = 5.2V$ ; the current at the Zener breakdown is  $I_z = 0.5\mu A$ . The model parameters are  $I_S = 0.5\mu A$ ,  $R_S = 1$ ,  $BV = 5.20$  and  $IBV = 0.5\mu A$ . The operating temperature is 500C.  $V_{in}$  has a normal voltage of 10V(dc).



3. A bipolar transistor circuit is shown below, where the output is taken from Node 4. Calculate and print the sensitivity of the collector current with respect to all parameters. Print the details of the bias point. The equivalent circuit for transistor Q1 is also shown in next figure.

