M.Sc.

2011

4th Semester Examination

ELECTRONICS

PAPER—EL-2211

(PRACTICAL)

Full Marks : 50

Time : 3 Hours

The figures in the right-hand margin indicate full marks.

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

Illustrate the answers wherever necessary.

Answer any *one* question from either *PSPICE* or *Digital Electronics*.

(PSPICE)

In each of the following questions, you have to save the design file by "your roll no., –design" and then save the plot in a file "your roll no. –plot".

1. Design an inverting amplifier circuit using PSPICE taking input resistor R_1 = 1kΩ, feedback resistor R_2 = 6.8 kΩ and load resistor R_3 = 10kΩ. Apply sine wave as an input signal with suitable amplitude and frequency of your choice. Simulate the circuit and plot the input voltage V_{in} versus time and output voltage V_o versus time in same graph. Also verify the gain of the amplifier with its theoretical value.

2. Design a second order low pass Butterworth active filter with cut-off frequency 1KHZ in PSPICE. Simulate the circuit and plot the gain versus frequency curve. Also verify the cut-off frequency with its given value.

3. Design an astable multivibrator with frequency 1KHZ and duty cycle 66.67% using IC 555 in PSPICE. Simulate the circuit and plot the amplitude versus time curve. Also verify the frequency and duty cycle with its given value.

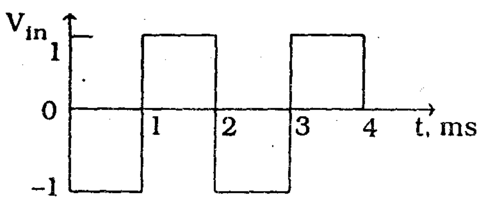
4. Create a PSPICE model of the non-inverting amplifier taking input resistor R_1 = 1kΩ, feedback resistor R_2 = 4.7 kΩ, and load resistor R_3 = 10kΩ. Apply sine wave as an input signal with suitable amplitude and frequency of your choice. Simulate the circuit and plot the input voltage V_{in} versus time and output voltage V_o versus time in the same graph. Also verify the gain of the amplifier with its theoretical value.

5. Design a second order high pass Butterworth active filter with cut-off frequency 3KHZ in PSPICE. Simulate the circuit and plot the gain versus frequency curve. Also verify the cut-off frequency with its given value.

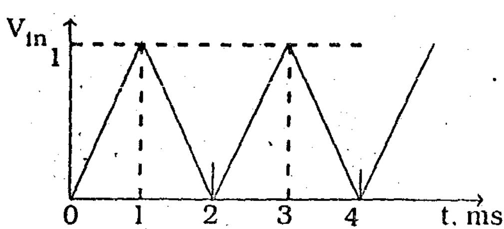
6. Design an inverting amplifier circuit using PSPICE taking input resistor R_1 = 1kΩ, feedback resistor R_2 = 10kΩ and load resistor R_3 = 10kΩ. Apply sine wave as an input signal with suitable amplitude and frequency of your choice. Simulate the circuit and plot the input

voltage V_{in} versus time and output voltage V_o versus time in same graph. Also verify the gain of the amplifier with its theoretical value.

7. Design an integrator circuit using op-amp taking input resistor R_1 = 2.5kΩ, feedback resistor R_2 = 1MΩ, feedback capacitor C_2 = 0.1 μF and load resistor R_3 = 100kΩ. Simulate the circuit using PSPICE and plot the transient response of the output voltage for a duration of 0 to 4 ms in steps of 50 μs for the input voltage given below.



8. Design a differentiator circuit using op-amp taking input Resistor R_1 = 100Ω, input capacitor C_1 = 0.4μF, feedback resistor R_2 = 10kΩ and load resistor R_3 = 100kΩ. Simulate the circuit using PSPICE and plot the transient response of the output voltage for a duration of 0 to 4 ms in steps of 10μs for the input voltage given below.



9. Design an astable multivibrator with frequency 10 KHZ and duty cycle 75% using 555IC. Simulate the circuit using PSPICE and plot the amplitude vs. time curve. Also verify the frequency and duty cycle with its given value.

Marks Distribution (For PSPICE)

Theory — 07
Circuit Design — 10
Simulation — 10
Verification and Accuracy — 05
Discussion — 03
Viva — 10
Laboratory Note Book — 05

(Digital Electronics)

10. Design a 3-bit synchronous even counter using JK flip-flop. Verify the count sequence by LED display.

11. Design a 3-bit synchronous odd counter using JK flip-flops. Verify the count sequence by LED display.

12. Design a mod-5 synchronous down counter using JK FFs. Verify the count sequence by LED display.

Marks Distribution (For Digital)

Theory — 05
Circuit Design — 15
Implementation — 07
Experimental Result — 05
Discussion — 03
Viva — 10
Laboratory Note Book — 05