2017

M.Sc. 4th Semester Examination

ELECTRONICS

PAPER—ELC-405

(Practical)

Full Marks : 50

Time : 3 Hours

The figures in the margin indicate full marks.

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

Illustrate the answers wherever necessary.

(*Advanced Electronics Lab.*)

Answer *one* question selecting it by a lucky draw.

1. Design a first order high pass active filter with cut-off frequency 3 kHz in PSPICE. Simulate the circuit and plot the gain vs. frequency curve. Also verify the cut-off frequency of the plot with given value.

2. Design a second order active low pass Butterworth filter with cut-off frequency 2 kHz. Simulate the circuit using PSPICE and plot the gain vs. frequency curve. Also verify the cut-off frequency with the given value.

3. Design an inverting amplifier circuit using OP.AMP taking input resistor R_1 = 1 kΩ, feedback resistor R_2 = 10 kΩ and load resistor R_3 = 10 kΩ. Apply sine wave as an input signal with suitable amplitude and frequency of your choice. Simulate the circuit using PSPICE and plot the input voltage (V_{in}) vs. time and output voltage (V_0) vs. time in the same graph. Also verify the gain of the amplifier with the given value.

4. Design an astable multivibrator circuit with frequency 2 kHz and duty cycle 66.67% using IC 555. Simulate circuit using PSPICE and plot the output voltage vs. time curve. Also verify the output frequency and duty cycle of the plot with their given values.

5. Design a first order low pass active filter with cut-off frequency 5 kHz in PSPICE. Simulate the circuit and plot the gain vs. frequency curve. Also verify the cut-off frequency of the plot with its given value.

6. Design a differentiator circuit using OP-AMP taking input resistor R_1 = 1 kΩ, input capacitor C_1 = 0.4 μF, feedback resistor R_2 = 6.8 kΩ and load resistor R_3 = 10 kΩ. Simulate the circuit using PSPICE and plot the transient response of the output voltage for a suitable input voltage.

7. Design a second order active high pass Butterworth filter with cut-off frequency 1 kHz. Simulate the circuit using PSPICE and plot the gain vs. frequency curve. Also verify the cut-off frequency with the given value.

8. Design an astable multivibrator circuit with frequency 1

kHz and duty cycle 80% using IC-555. Simulate the circuit using PSPICE and plot the output voltage vs. time curve. Also verify the output frequency and duty cycle of the plot with their given values.

9. Design a non-inverting amplifier circuit using OP-AMP taking input resistor R_1 = 1 kΩ, feedback resistor R_2 = 8.2 kΩ and load resistor R_3 = 10 kΩ. Apply sine wave as an input signal with suitable amplitude and frequency of your choice. Simulate the circuit using PSPICE and plot the input voltage (V_{in}) vs. time (t) and output voltage (V_0) vs. time (t) in same graph. Also verify the gain of the amplifier with given value.

10. Design a integrator circuit using OP-AMP taking input resistor R_1 = 1 kΩ feedback resistor R_2 = 6.8 kΩ, feedback capacitor C_2 = 0.1 μF and load resister R_3 = 10 kΩ. Simulate the circuit using PSPICE and plot the transient response of the output voltage for a suitable input voltage.

11. Design AND, OR and NOT gates using MOSFETs. Also verify their truth tables.

12. Design a 3-bit synchronous MOD-5 counter using J-K flip-flop. Verify the count sequence by LED display.

Marks Distribution (For PSPICE)

For Question Nos. 1 to 10 :		*For Question Nos. 11 to 12 :*	
Theory :	07 Marks	Theory :	05 Marks
Circuit Design :	10 Marks	Circuit Design :	15 Marks
Simulation :	10 Marks	Implementation :	07 Marks
Verification and Accuracy :	05 Marks	Experimental Result :	05 Marks
Discussion :	03 Marks	Discussion :	03 Marks
Viva-voce :	10 Marks	Viva-voce :	10 Marks
Lab. Note Book :	05 Marks	Lab. Note Book :	05 Marks
Total	50 Marks	Total	50 Marks