Reg. No. : | | | | | | | | | | |

Question Paper Code: 92052

M.E. DEGREE EXAMINATION, MAY 2014.

Elective

Power Electronics and Drives

01PPE505 - COMPUTER AIDED DESIGN OF POWER ELECTRONICS CIRCUITS

(Regulation 2013)

Duration: Three hours Maximum: 100 Marks

Answer ALL Questions.

PART A - (10 x 2 = 20 Marks)

1. What is PSpice? Mention the various analyses it performs.
2. Draw the small signal and static model of diode with reverse-biased condition in SPICE platform.
3. Write short note on decoupled systems.
4. What are the informations that can be obtained from steady state solution of a power electronic system through computer simulation?
5. List the minimum requirements to run a DC sweep analysis.
6. How transient analyses are carried out? List the dot commands pertaining to transient analysis?
7. What do you meant by schematic capture in circuit simulation?
8. State the significance of Monte Carlo analysis in the modelling of power electronic circuits.
9. Give the expression for calculating input power factor from the THD of diode rectifier input current.
10. Write short notes on voltage source inverters.

PART - B (5 x 14 = 70 Marks)

11. (a) (i) List the various limitations of PSpice. (7)

(ii) Draw the various models of BJT in PSpice. (7)

Or

(b) (i) Brief about various platforms of PSpice. (7)

(ii) Plot the output and transfer characteristics of IGBT in PSpice. (7)

12. (a) (i) Brief about analysis of power electronic systems in a sequential manner through simulation. (10)

(ii) Write a short note on future trends in computer simulation. (4)

Or

(b) (i) Elaborate any two algorithms for computing steady state solution in power electronic systems. (10)

(ii) Write shorts note on coupled and decoupled systems. (4)

13. (a) (i) Starting from minimum requirements for running AC sweep analysis, discuss how PSpice treats non linear devices with suitable example. (7)

(ii) How a MOSFET model is worked out in PSpice simulation? (7)

Or

(b) (i) List the minimum circuit design/program set up requirement to run a transient Analysis. Explain about time based stimulus generation. (7)

(ii) How the model of BJT is worked out in PSpice simulation? (7)

14. (a) (ii) Brief about PSpice default symbol libraries with their functions. (7)

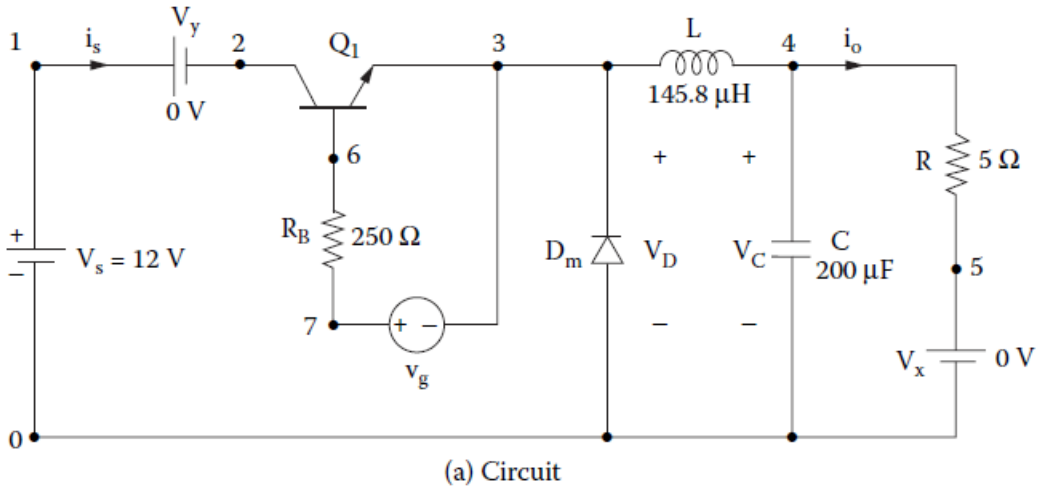
(ii) Brief the significance of fourier analysis in simulation of power electronic circuits with an example. (7)

Or

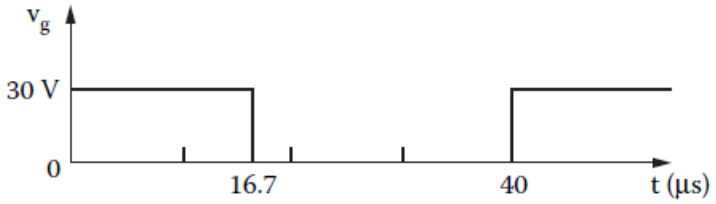
(b) (i) Brief the significance of Monte Carlo analysis in simulation of power electronic circuits with an example. (7)

(ii) Perform worst case analysis for RLC circuit. (7)

15. (a) (i) Find the performance of a step-down dc–dc converter with a BJT switch for the following example: A BJT buck chopper and its control voltage is shown in the following figure (a) and (b) respectively. The DC input voltage is V_s = 12 V. The load resistance R is 5 Ω. The filter inductance is L = 145.84 μH, and the filter capacitance is C = 200 μF. The chopping frequency is f_c = 25 kHz, and the duty cycle of the chopper is k = 42%. Use PSpice to (a) plot the instantaneous load current io, the input current i_s, the diode voltage v_D, and the output voltage v_C and (b) calculate the Fourier coefficients of the input current i_s. Plot the frequency response of the converter output voltage from 10 kHz to 10 MHz and find the resonant frequency. (7)

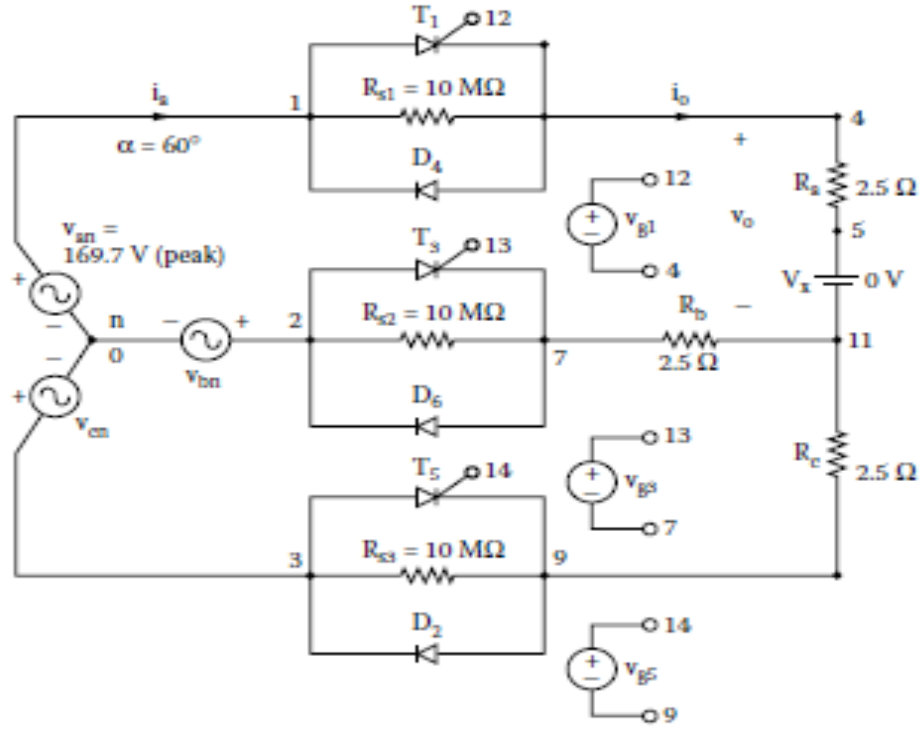


(a) Circuit

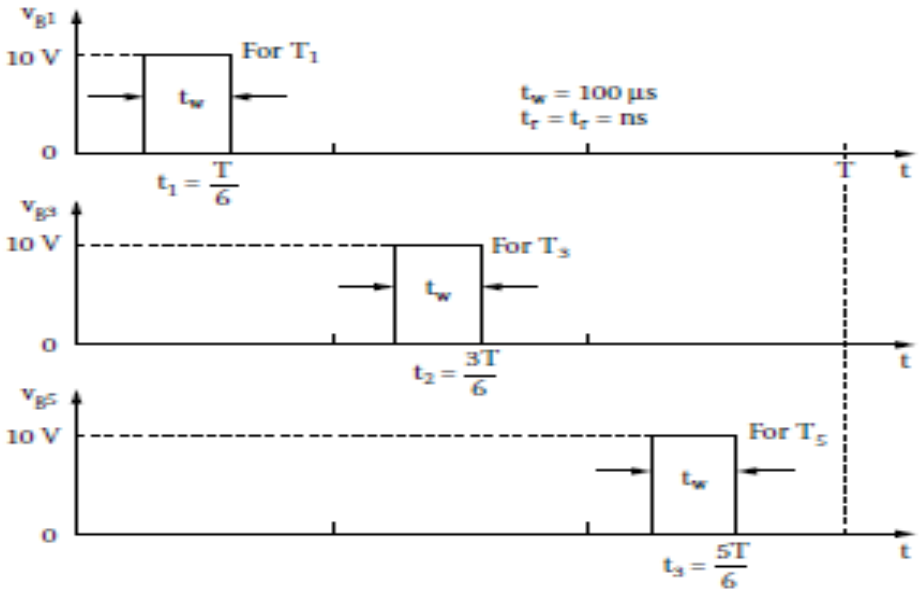


(b) Control voltage

(ii) A single-phase half-wave AC voltage controller is supplied from a three-phase wye connected supply and its gate voltage is shown in following figure. The input phase voltage has a peak 169.7 V, 60 Hz. The load resistance per phase is R = 2.5 Ω. The delay angle is α = 60°. Use PSpice to (a) plot the instantaneous output phase voltage v_o and (b) calculate the Fourier coefficients of the input phase current i_s and the input power factor PF. (7)



(a) Circuit

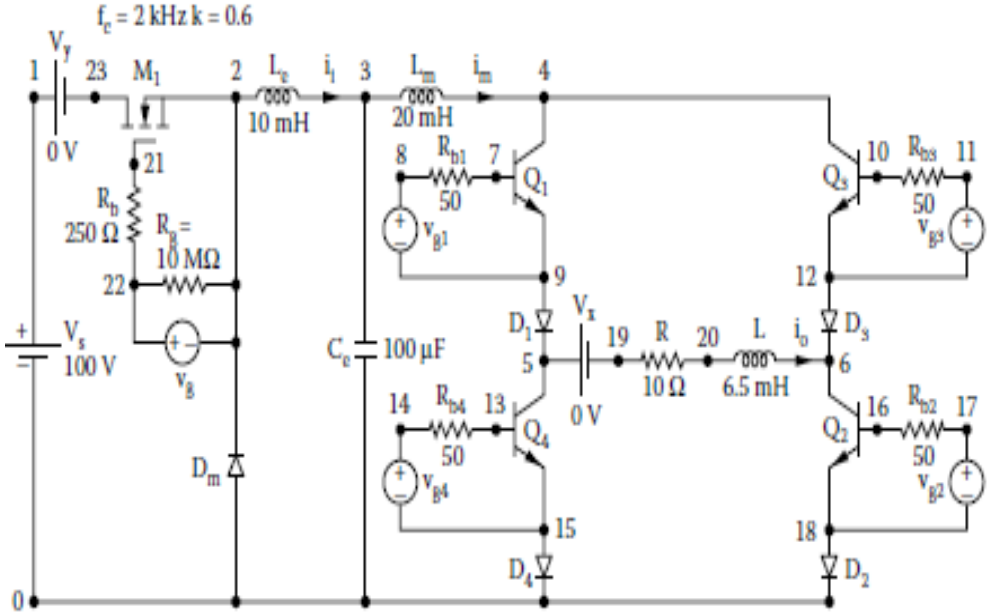


(b) Gate voltages

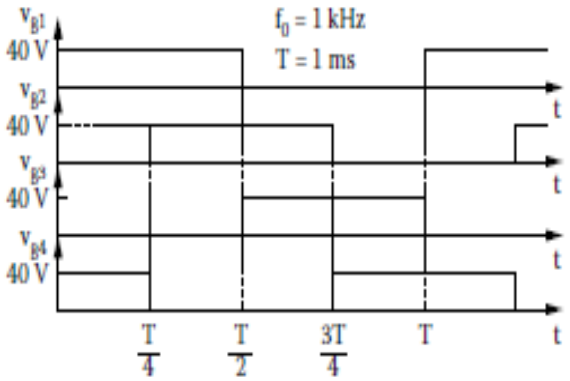
Or

(b) (i) A single-phase current-source inverter and its control voltages are shown in following figures (a) & (b). The DC input voltage is 100 V. The output frequency is f_o = 1 kHz. The chopping frequency is f_s = 2 kHz, and its duty cycle is k = 0.6. The load resistance is R = 10 Ω, and the load inductance is L = 6.5 mH. Use PSpice to (a) plot the instantaneous output current i_o, the instantaneous source current i_s, and the instantaneous current i_1 through inductor L_e and (b) calculate the Fourier coefficients of the output current i_o. The model parameters of the BJTs are IS = 2.33E27, BF =

13, CJE = 1PF, CJC = 607.3PF, and TF = 26.5NS. The model parameters of the MOSFETs are VTO = 2.83, KP = 31.2U, L = 1U, W = 3.0M, CGDO = 1.359N, and CGSO = 2.032N. (7)

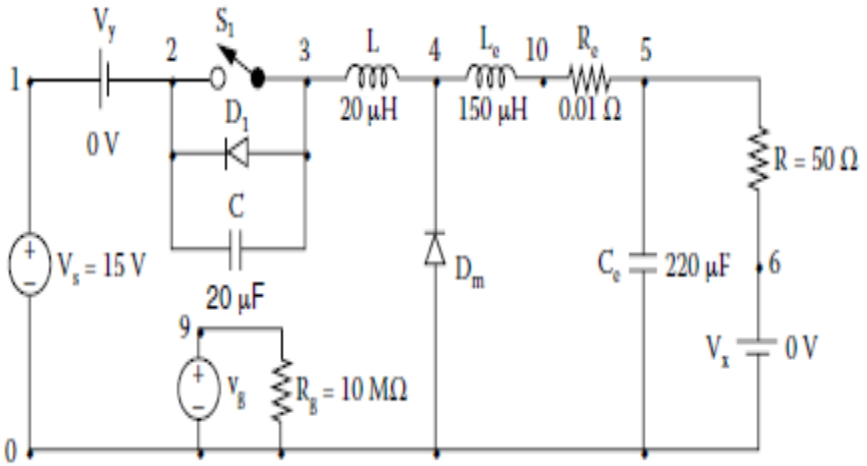


(a) Circuit

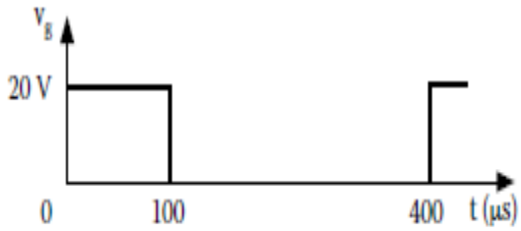


(b) Control voltages

(ii) A Zero Voltage Switching Converter and its control voltage is shown in following figures. The DC input voltage is V_s = 15 V. The switching frequency is f_s = 2.5 kHz. Use PSpice to plot the instantaneous capacitor voltage v_c, the inductor current i_L, the diode current v_{Dm}, and the load voltage v_L. Use a voltage-controlled switch to perform the switching action. (7)



(a) Circuit



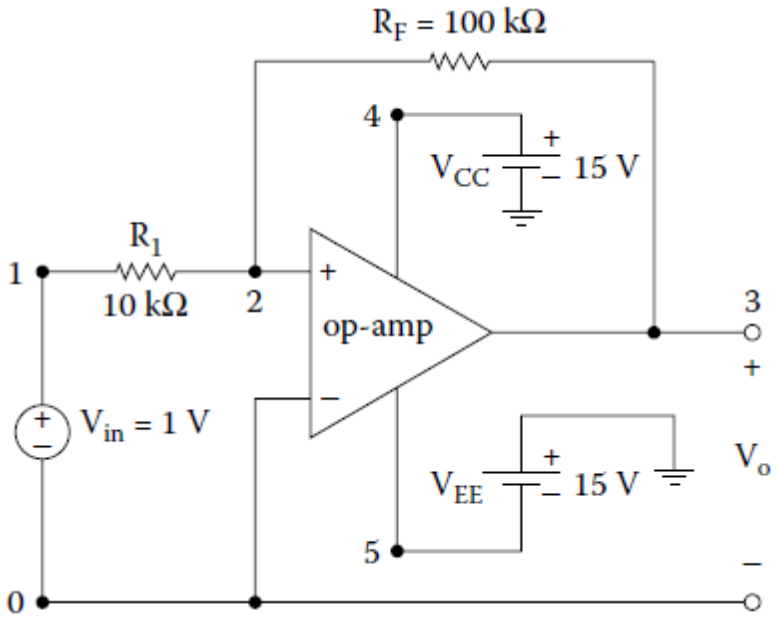
(b) Control voltage

PART - C (1 x 10 = 10 Marks)

16. (a) Design a circuit of your own specification with the help of PSpice simulation software for a transient pulse response of RLC circuit with the effect of resistors in PSpice, sinusoidal input voltage source and frequency response. (10)

Or

(b) Describe about DC/AC linear and non linear macro models of OP- AMP circuits. Find its performance through these models for the given inverting amplifier shown in following figure. Use PSpice to plot the DC transfer characteristic if the input is varied from −1 to +1 V with an 0.2 -V increment. (a) Use DC op – amp model as a sub circuit: its parameters are $A_0 = 2 \times 10^5$, $R_i = 2$ MΩ, $R_o = 75$ Ω. (b) Use AC op – amp model as a sub circuit: its parameters are $R_i = 2$ MΩ, $R_o = 75$ Ω. $C_1 = 1.5619$ µF and $R_1 = 10$ kΩ. (c) Use macro model for UA 741. (10)


R_F = 100 kΩ
4
V_{CC} 15 V
R_1
10 kΩ
1
2
op-amp
3
V_{in} = 1 V
V_{EE} 15 V
V_o
5
0