# Pspice Practice Questions

ENEE236 – Analog Electronics

Instructor

Nasser Ismail

- Use Pspice student edition to enter and simulate the following circuits
- Make sure to learn how to set and perform the following analysis types:

1) Transient

2) DC Sweep

3) AC Sweep

4) Parametric analysis combined with

transient or AC sweep

# 1. Simulate the following circuit and plot Vo(t). use transient analysis

Use 1N4148 for the diodes D1 & D2

R1=1 k$\Omega$; for Vi(t) use Vsin with frequency =50Hz and magnitude =20V

*Note :*

*a. Make sure to set an appropriate final time in the transient analysis in order to be able to see few cycles of the waveform*

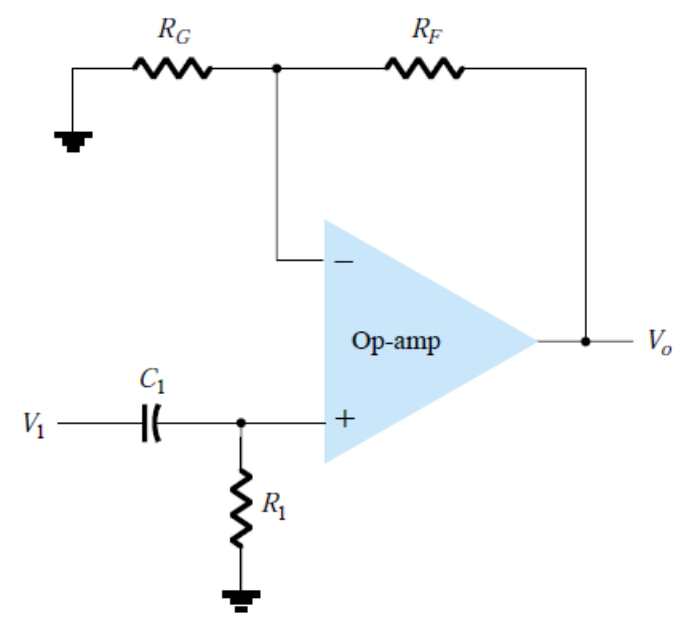*b. Consider setting a step ceiling in the analysis transient menu to get smooth waveforms.*



R1=1 kΩ

D1   D2

Vi(t) =20 sin ωt

5 V   5 V

Vo(t)

+

-
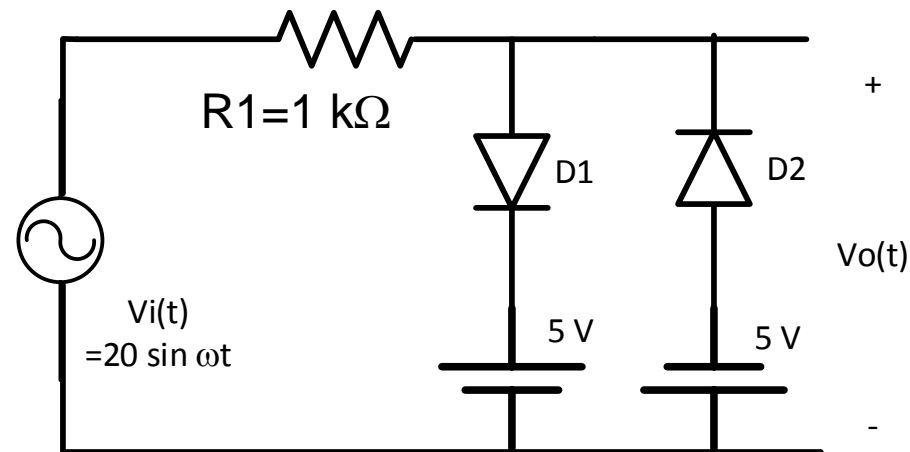
# 2. Simulate the following circuit using transient analysis type, use R=10kohm, C=1uF , Diode 1N4148
## Use Vpulse to create Vi as shown below

## Plot Vo ?
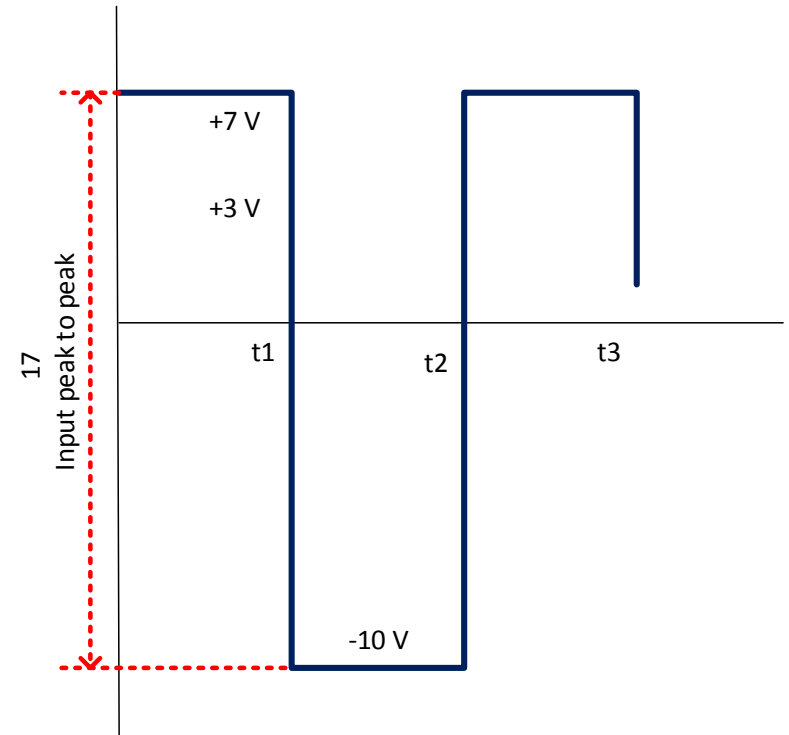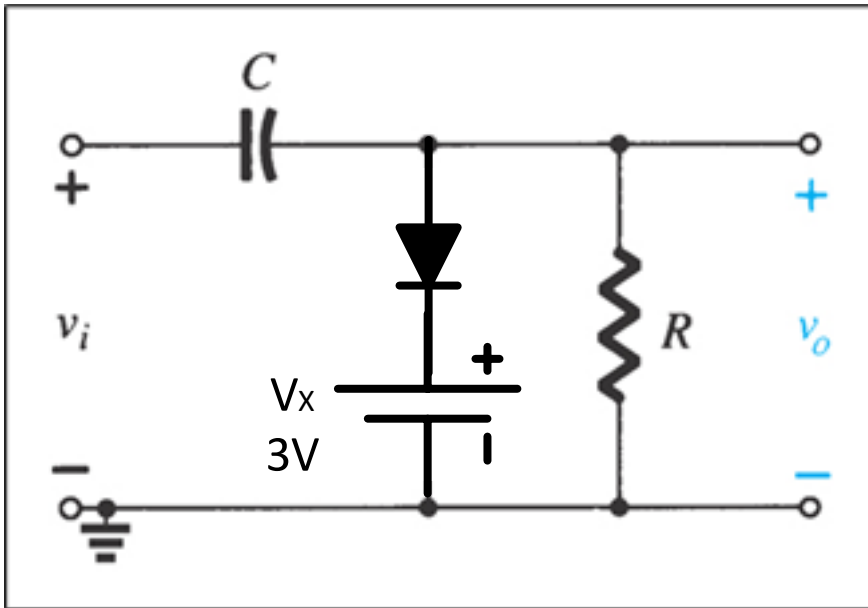
# Q3. Simulate the following circuit,



- RF= 8 kΩ; RG= 16 kΩ ;

- R1=10 kΩ; C1=10 pF; Opamp LM324

- +Vcc=15, -Vcc=-15

1) Replace V1 by a dc source =1V , find Vo using transient analysis?

2) Replace V1 by an ac source (Vac) with magnitude = 1V perform ac sweep and plot the magnitude frequency response in dBs

3) Find Midband gain and $\omega_c$

# Q4. Simulate the following circuit,

1) Replace Vi by an ac source (Vac) with magnitude = 1V, choose ac sweep analysis then plot the magnitude frequency response of Vo/Vi in DB

2) Based on the simulation results find the following quantities:

- mid band gain Av(mid)

- $\omega_L$

- $\omega_H$
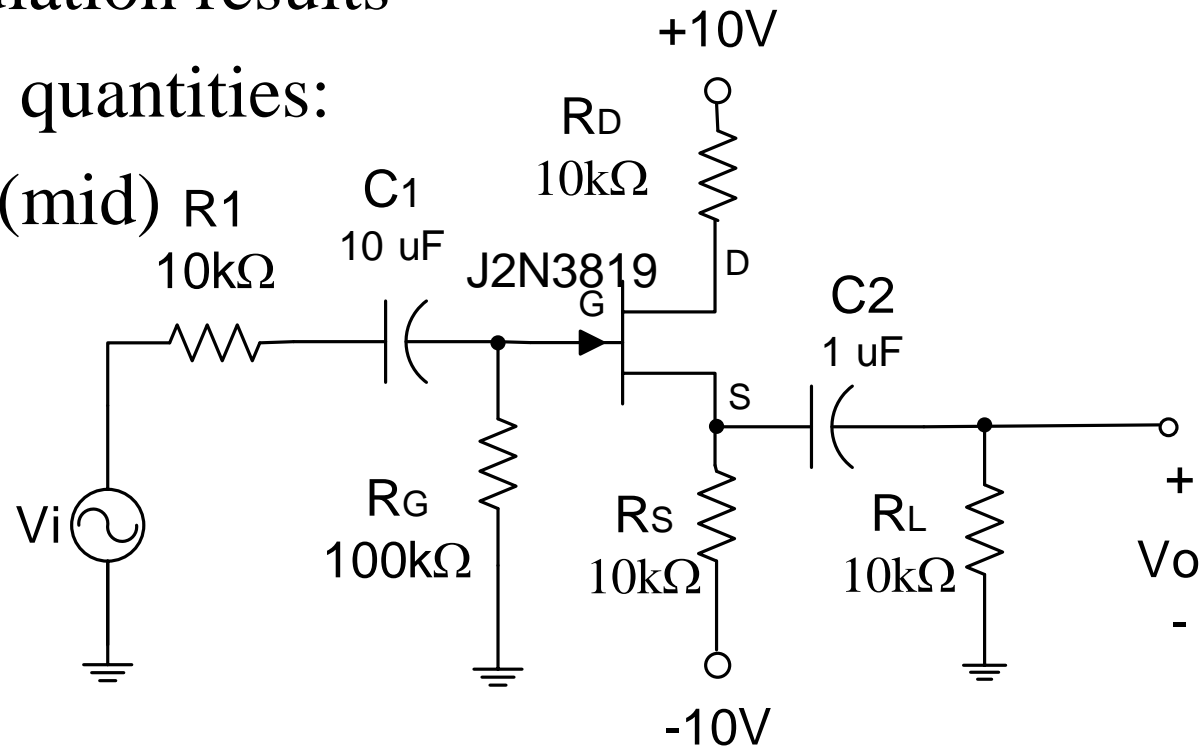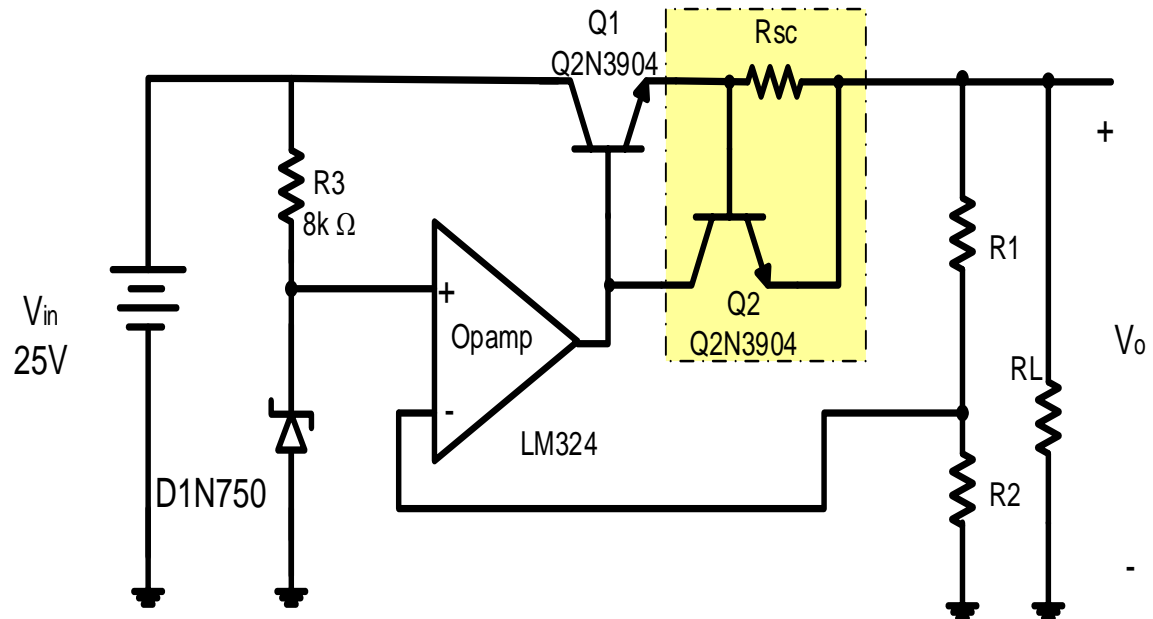
# Q5. Voltage Regulator example

Simulate the following series voltage regulator ,

- R3= 8 kΩ ;R2=R1=10 kΩ;Zener diode  1N750;Opamp LM324
- Q1 & Q2 : 2N3904;Rsc=0.7 Ω;RL=1kΩ;Vin=25 V

1) Find: Vz, Vo, Ic, Ib, Ie, Iz

2) Use parametric analysis to vary the  resistor RL=1Ω, 100Ω;1 kΩ and find Vo in each case , also find the maximum allowable load current $I_{RL}(max)$

# Q6. DC sweep,

1) Enter only the dc equivalent circuit in pspice

2) Use Rs as a parameter, choose PARAM from the new parts menu, enter the name and any initial value for Rs,

3) Replace Rs value by {Rs}

4) Perform dc sweep for a range of Rs from 1k to 100k

5) Plot Vgs, Id and Vds as a function of Rs