

ECEN 3714----Network Analysis

Laboratory Manual

LAB 1:- Introduction to PSpice Software

Oklahoma State University
School of Electrical and Computer Engineering

1. Objective

This lab practices using PSpice software to sketch a circuit schematic diagram and to simulate its response to a given input. PSpice will be used in all future lab experiments too, especially for pre-lab works.

2. Introduction To PSpice Software

PSpice is a proprietary software package for simulation and verification of analog and mixed-signal circuits. Simulation is important as it will help validate a circuit design (e.g., delivering a specific frequency response) or discover bugs in the circuit design.

2.1. First step:-

Please specify a folder or construct a new folder for saving your files-----good housekeeping is important.

2.2. Opening SCHEMATIC Capture:- (depending upon the version, the interface may be slightly different)

- On the computers in ENDV 325, search with Capture CIS or Capture CIS Lite.
- If prompted with Studio Suite Selection, choose PCB design expert with Capture CIS (see Figure 1) from the drop-down menu. You might not see this window at all in some versions, and that's fine.

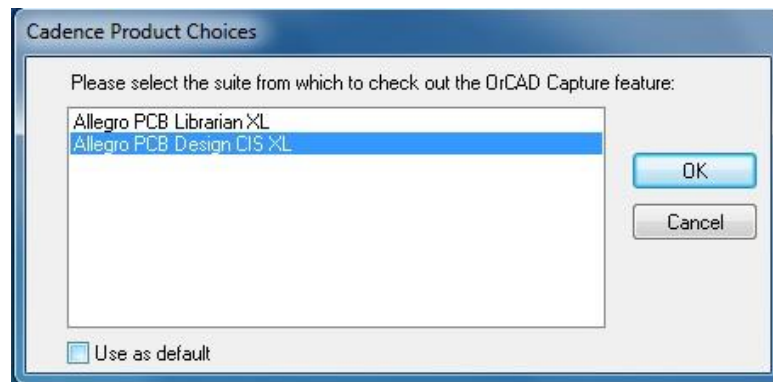


Figure 1 - Studio Suite Selection

- Select File -> New -> Project. From the New Project window, give your project a name, select "PSpice Analog or Mixed A/D", and select its location (see Figure 2). You're now ready to create your schematic.

Note: The directory you place your project must already exist to do this operation.

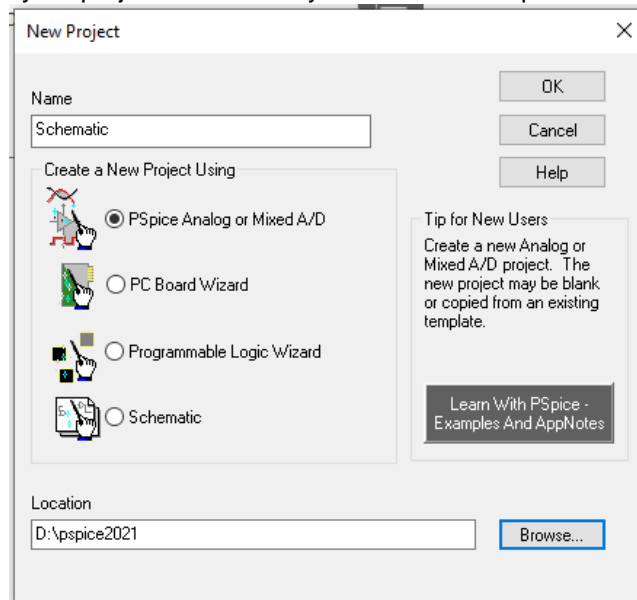


Figure 2 - New Project Window

d) In the create PSpice project window (if you are prompted with this), select “Create a blank project” as shown in Fig.3, and click “OK”. You will have the schematic window open where you can draw a new circuit.

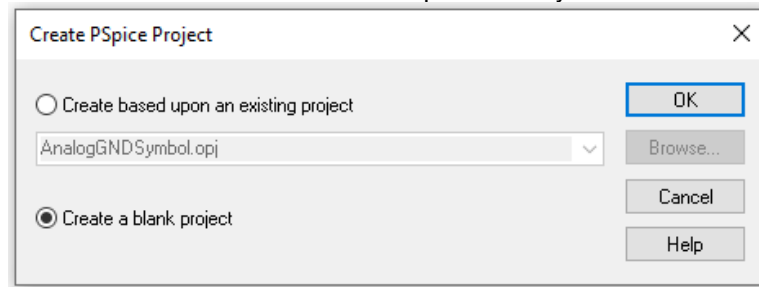


Figure 3 - Create PSpice Project Window

2.3. Creating Schematics with Capture CIS

a) The next step is drawing the desired schematic. To get parts, go to Place -> Part (see Fig. 4). Or at the right hand side of the schematic window, click on “Place Part” icon.

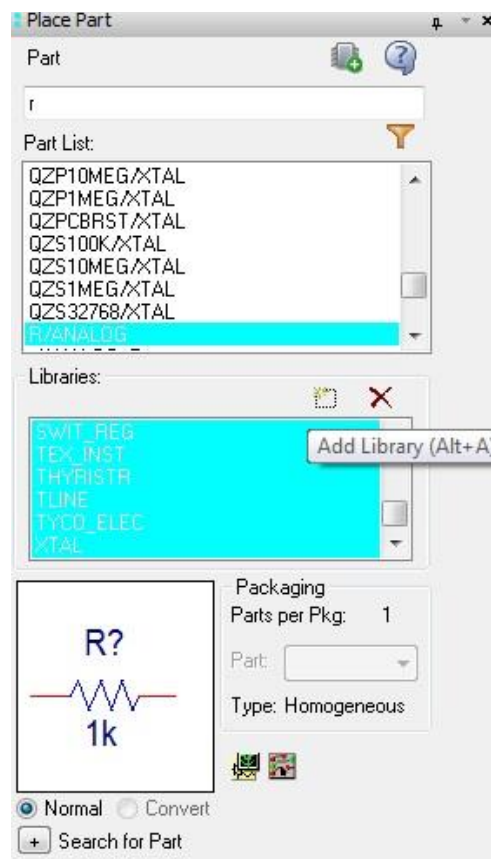


Figure 4 - Get Parts Dialog

Note: - If no parts are visible in the “Place Part” dialog, simply click on the “Add Library” button, and add Library files (.olb files) from these two directories:

.....Library/
.....Library/pspice

Click on any *.olb file and then press Ctrl+A to select all the files (or Ctrl+Arrow to select certain library files) and hit the “Open” button (Fig. 5).

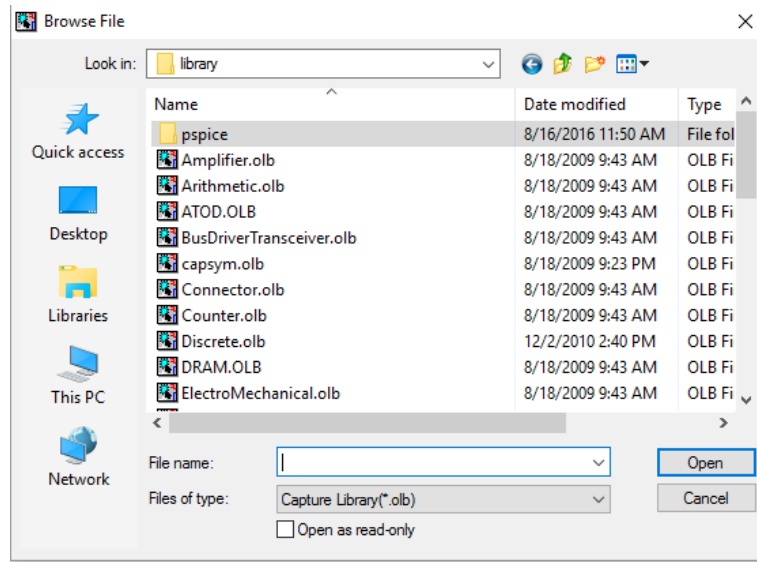


Figure 5 - Adding Libraries

b) Let's draw a simple RLC circuit. You will find the parts you need by typing the name of the part in the Part section and clicking "OK." So, let's draw the circuit below. For the schematic shown in Figure 6, you need one VPULSE, two R/ANALOGs, one C/ANALOG and one L/ANALOG. If you cannot find the voltage source in the "Part List", use "Add Library" button to add all the library files in the "advanls" directory, which is a sub-directory right below the "Pspice" directory (dependent upon the version of PSpice).

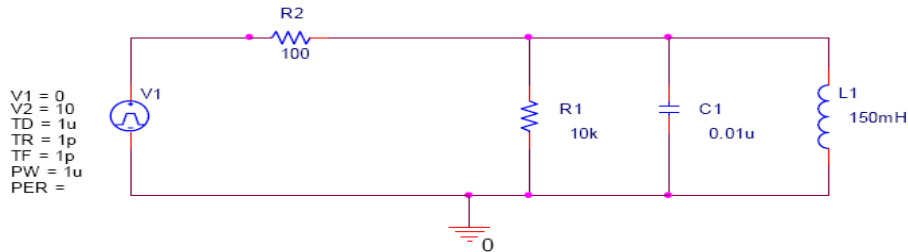




Figure 6 - Circuit diagram

c) Remember we always need a ground in the circuit, this can be found in the Place Ground menu, accessible by clicking the  button on the right side of the screen or go to menu Place / Ground. The ground you will use is the 0/CAPSYM identified by the symbol .

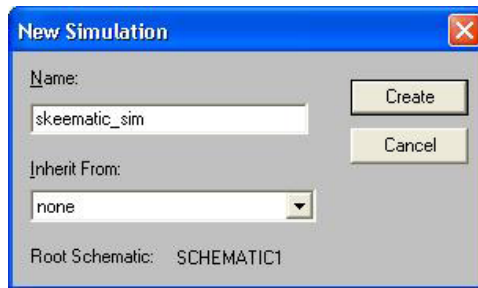


Figure 8 - Naming Simulation

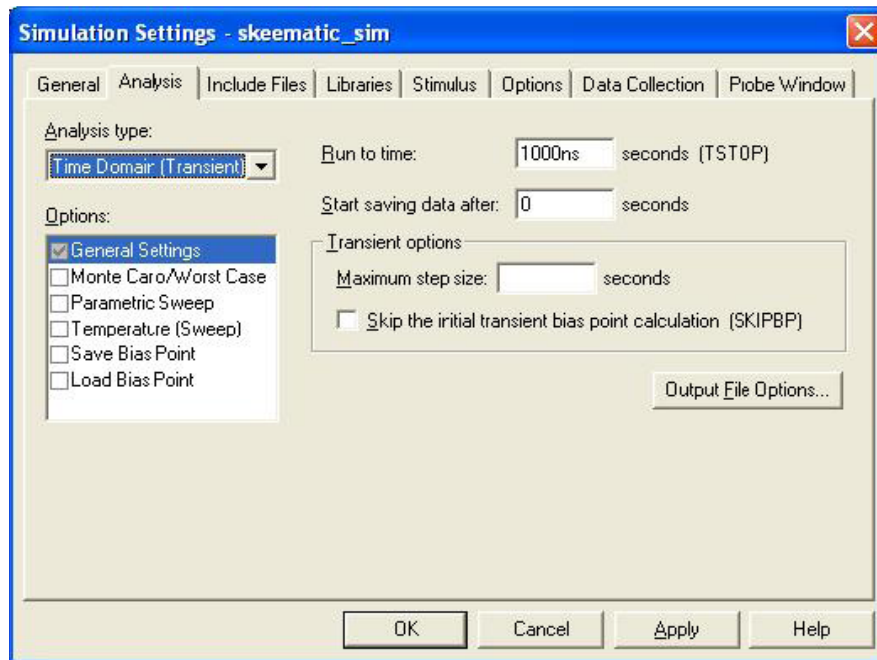


Figure 9 - Simulation Settings

b) Select the correct analysis type for desired output. Then further change the options for a more specific analysis. Once desired options are selected, click "OK." To run the simulation, select PSpice -> Run.

2.5. Types of Stimulus and Analysis

Each type of source is paired with a certain type of analysis, and in most cases with only one. The primary sources you will use in this class are VPulse, Vsin, and Vac. These are available in the standard PSpice libraries you added when you created the schematic.

----- **VPulse:** - It is used to measure the transient response of a circuit. For Vpulse, you control all aspects of a generated pseudo-square wave. Factors directly affecting the produced Vpulse function are visible by the parameter which is described in the following table.

Name	Description	Symbol
Start voltage	low voltage value	V1
Second voltage	high voltage value	V2
Time delay	Voltage will remain at V1 until the specified time.	TD
Rise time	the rise time of the pulses	TR
Fall time	the fall time of the pulses	TF
Pulse width	Amount of time pulse remains at V2	PW
Period of pulse	How often pulse actually repeats itself	PER

Table 1

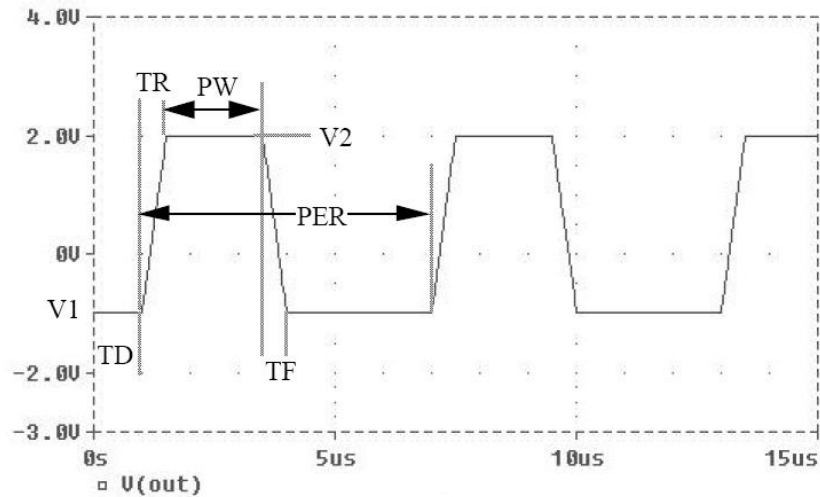



Figure 10 - value description diagram

For a Square wave the Vpulse value V1 represents the low voltage value of the square wave, V2 represents the high voltage value of the square wave, TD is the delay time to wait during the square wave's period before going high, TR is the rise time of the pulses, TF is the fall time of the pulses, PW is the pulse width for the high voltage and PER is the square wave's period. A square wave of 5Volts may be generated by setting V1=0, V2=5, TD=0, TR=0.5p, TF=0.5p, PW=the width of the wave, PER=the period of the wave.

VPulse is combined with the time-domain (transient) analysis: Let's take the following example of an RLC network. First draw the following circuit in Fig. 11. Now, it is time to change the Simulation Settings to the desired values. You can access the settings by clicking on the  button. So, you will find that value and in turn set the Run time to approximately 8 us. As can be seen from the Time Domain simulation below in Fig 12, the output is trying to follow the input and at the end, the output reaches the steady state value of 0 Volts after the input becomes zero.

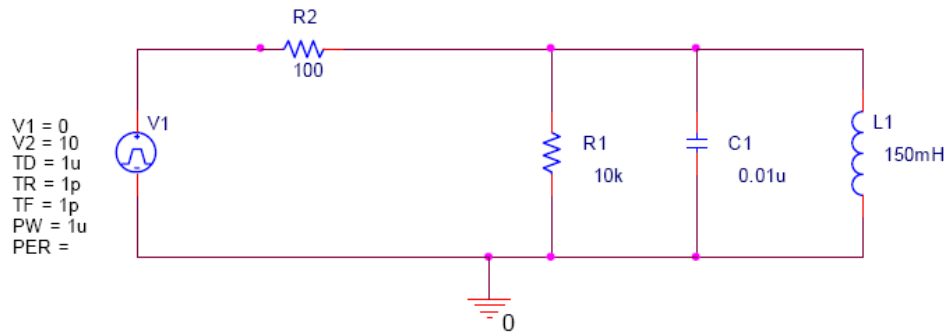


Figure 11 -Circuit diagram

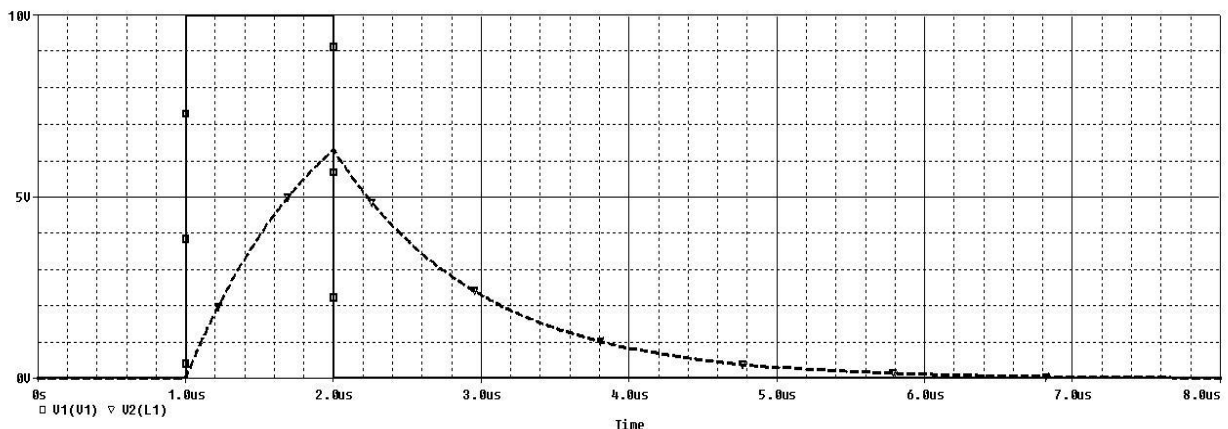


Figure 12 -Output diagram

----- **Vsin**:-This source yields the time response of a circuit to an input of constant frequency. Vsin has the parameters of: Voff (Offset Voltage), Vamp (Amplitude), and Freq (Frequency).

- a) Let's take the following example. Once drawn, you will again use the Time Domain (transient) sweep as you did in the section above. Select a running time of approximately 60 ms, just so you can see beyond the point where the sine wave crosses 0 volts.

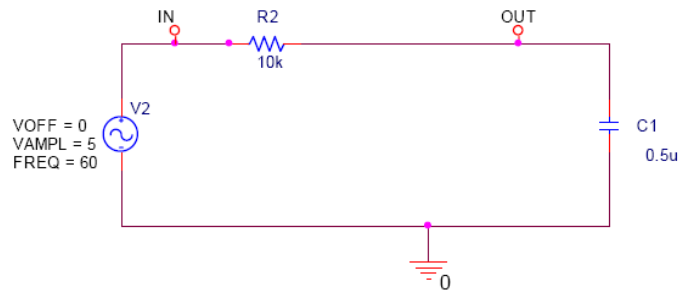



Figure 13- circuit diagram

- b) But wait a second, what are those funny little bubble things? They are a trick to name the wires that they are attached to. Simply find the “Place Ground” button on the right of the screen. Once you are in the Place Ground

menu, find VCC/CAPSYM identified by . Now, let's just change their names.

c) Since you already have the desired simulation settings, let's simulate by clicking the  button, or by pressing F11. PSpice A/D will appear as before but nothing will be present. So you will need to find the appropriate waveforms to add.

d) Click on the “Add Trace” button in PSpice A/D window. Once the Add Trace dialog is open, find V(IN) and V(OUT). Multiple traces can be added (when separated by commas, as visible in Figure 14).

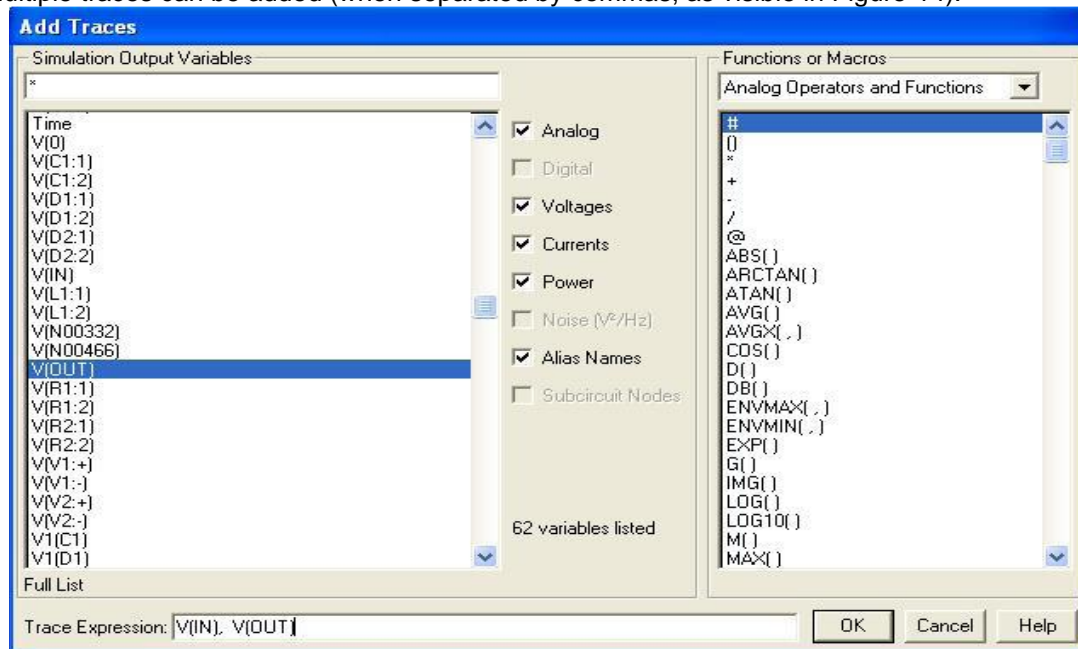


Figure 14 - Add Trace Dialog

Our output is now visible as seen in Figure 15, below. To have the smooth curves, change the default maximum step size equal to 10u seconds in the Simulation Settings. Observe the curve, is it smooth or not? If not then reduce the step size more. If simulations are taking a very long time, then increase the step size.

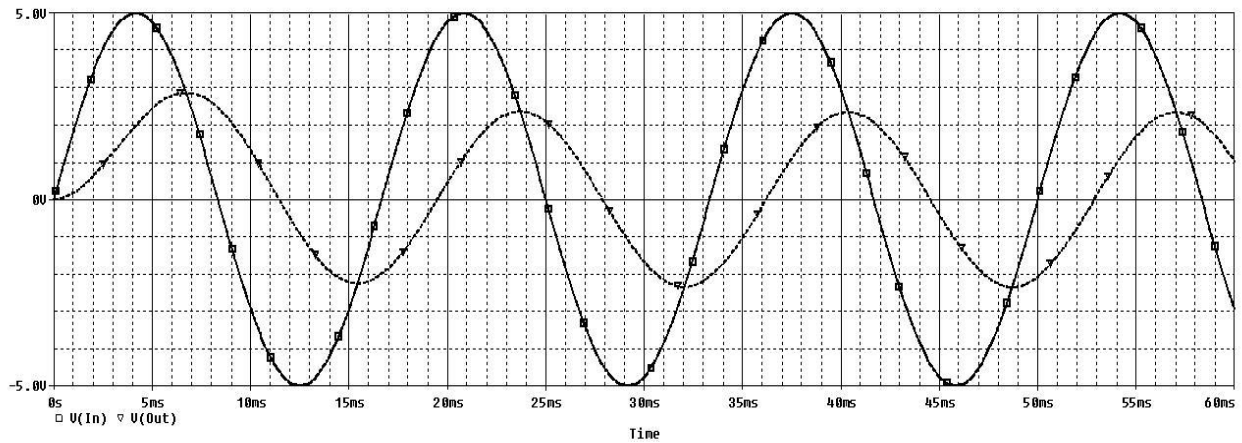


Figure 15 - Vsin Transient Sweep

2.6. Output Display options

You now know how to get the desired results based on a circuit's response to either a change in time or frequency. Let's take what you've learned and find some ways to transform your output data into something else that allow you to gather exactly the information you need from the circuit.

----- **Gain/Attenuation:** - Gain or attenuation is a measure of how much of a signal actually passes through a circuit. Gain is defined by V_{out}/V_{in} . Let's see if you can find the gain for a simple RLC circuit.

a) Let's draw the circuit below and find the smooth output.

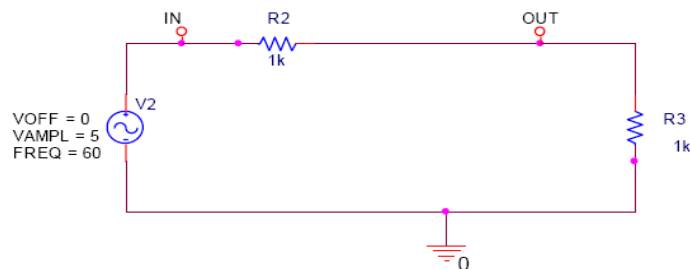


Figure 16 - Circuit diagram

b) Remember that you want to use the Time Domain analysis with our Vac source; otherwise, you might not get any results. Are these the results you expected?

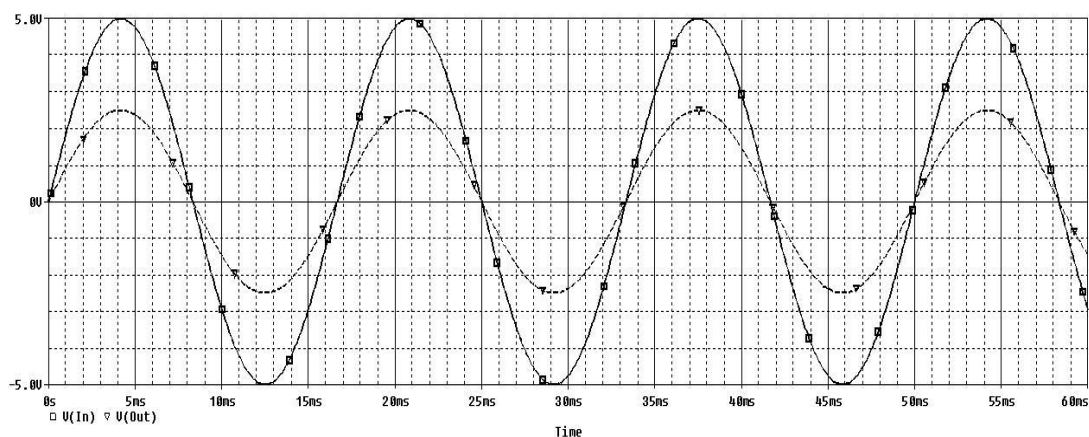


Figure 17 - output gain diagram

c) What is the relationship between the output and input? Let's use the above definition of gain and find the exact gain value for this circuit. First, you will keep both of these traces and go back to the Add Trace button. In the

Trace Expression window, you will divide one waveform by the other as shown. Next you just have to click "OK." Did you expect the graph below? What kind of information does this give to you?

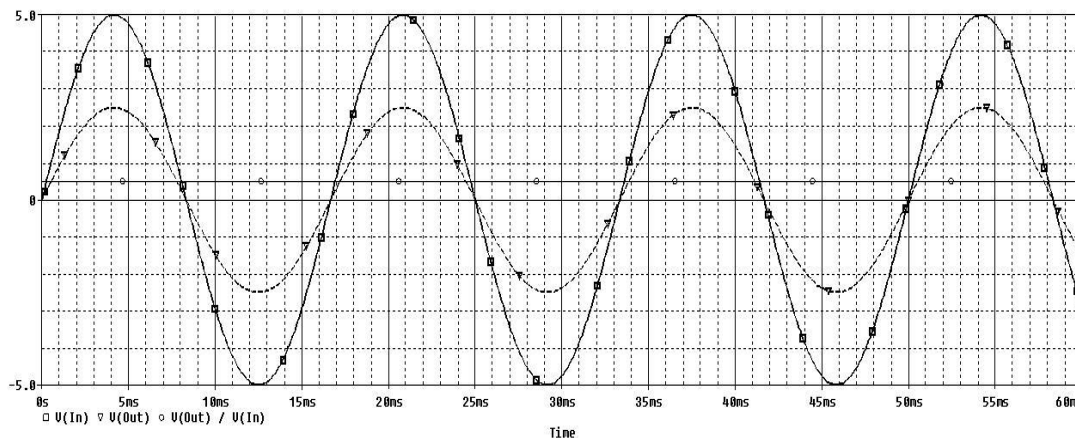


Figure 18 - output gain diagram

d) Another useful way to illustrate gain or attenuation is with the use of decibels. Decibels are best defined as the level of relative strength of a signal. From the Functions or Macros section of the Add Trace window select Analog Operators and Functions in the drop-down menu. Now select DB() from the list, and make sure the following

appears in the Trace Expression Window: `Trace Expression: DB(V(OUT)/V(IN))` then just click on "OK." What did you observe?

3. Laboratory Assignments

Guidelines to follow:

- Please complete below assignments individually using PSpice and include the results (circuit diagrams, graphs etc.) in your report. No groupwork is allowed in this lab.
- For all the following assignments, the input and output wave forms should be displayed in the same graph (to show any differences in amplitude and phase).
- The circuit diagrams must be drawn in PSpice and it must display the parameter values of the parts used.
- The square wave voltage inputs will have 50% duty cycle (ON for 50% time and OFF for the rest 50% time in each time-period), unless stated otherwise.
- Do not print a schematic or a graph to have a blacked-out background. Change the background to white before you paste those in your report. To change the background color, go to tools → options → color settings tab in the PSpice A/D window and select background to be white and foreground to be black.
- Make all the graphs clearly visible by increasing their thickness and choosing contrasting colors, if required. To change these properties, right click on graph -> trace property in the PSpice A/D window.

Assignment 1:

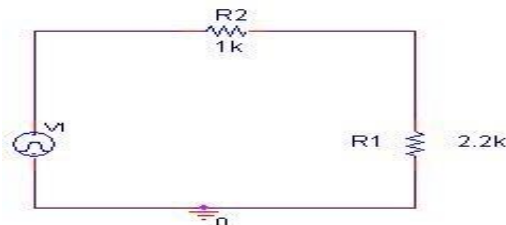


Figure A1

- Draw the circuit as shown in Figure A1.
- Find the voltages across the resistor R1 when the input (V1) is a **square wave** at the frequency of 1kHz having a peak-to-peak voltage of 20V (-10V to 10V) with zero DC offset.
- Find the voltage across the resistor R1 when the input is a **sine wave** at the frequency of 1kHz having a peak-to-peak voltage of 20V (-10V to 10V). Show the input and the R1 output wave form within the same graph.

(d) For (c), calculate the ratio of the maximum of R1 output with respect to the maximum of the input at that frequency. (Use $\text{Max}(V_{\text{out}}) / \text{Max}(V_{\text{in}})$).

(e) What will be the ratio $\text{Max}(V_{\text{out}}) / \text{Max}(V_{\text{in}})$ when the input frequency changes to 10Hz and 100Hz?

Assignment 2:

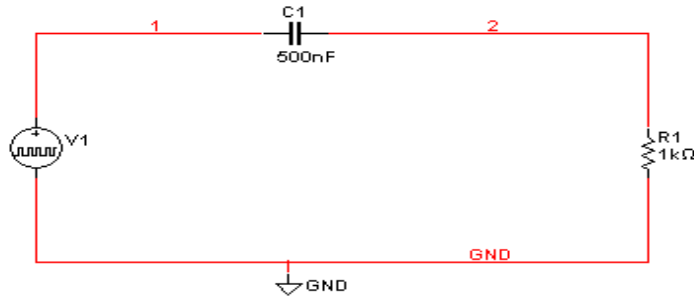


Figure A2

(a) Draw the circuit as shown in Figure A2.

(b) Find the voltage across R1 if the input is a square wave of a frequency of 10Hz having a peak-to-peak voltage of 10V (-5V to 5V) with zero DC offset.

(c) Find the voltage across R1 if the input is a square wave of a frequency of 100Hz having a peak-to-peak voltage of 10V (-5V to 5V) with zero DC offset.

(d) Find the voltage across R1 if the input is a square wave of a frequency of 1kHz having a peak-to-peak voltage of 10V (-5V to 5V) with zero DC offset.

Assignment 3:

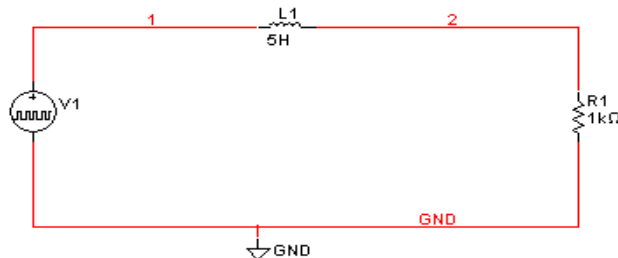


Figure A3

(a) Draw the circuit as shown in Figure A3.

(b) Find the response of the circuit to an input of a square wave of a frequency of 10Hz having a peak-to-peak voltage of 10V (-5V to 5V) with zero DC offset. Show the input and output (across R1) wave forms within the same graph.

(c) Find the response of the circuit to an input of a square wave of a frequency of 100Hz having peak-to-peak voltage of 10V (-5V to 5V) with zero DC offset.

(d) Find the response of the circuit to an input of a square wave at a frequency of 1kHz having a peak-to-peak voltage of 10V (-5V to 5V) with zero DC offset.

Assignment 4:

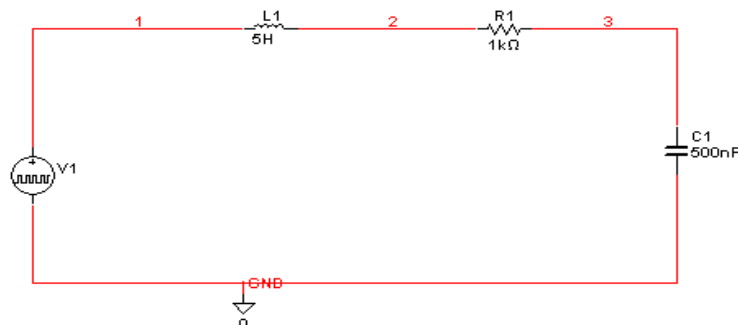


Figure A4

- (a) Draw the circuit as shown in Figure A4.
- (b) Find the voltage across the capacitor C1 if the input is a square wave of a frequency of 200Hz having a peak-to-peak voltage of 10V (-5V to 5V) with zero DC offset.
- (c) Find the voltage across the capacitor C1 if the input is a sine wave of a frequency of 200Hz having peak-to-peak voltage of 10V (-5V to 5V) with zero DC offset.
- (d) Find the voltage across the capacitor C1 if the input is a sine wave of a frequency of 1kHz having peak-to-peak voltage of 10V (-5V to 5V) with zero DC offset.

3.1 Requirements for Laboratory Report

The lab report must be completed individually, and due by the same lab session at the following week. It is expected that comprehensive technical writing is demonstrated in your laboratory report. The following contents constitute the minimum amount of materials that should be included in your report. There will be no pre-lab for this lab.

- (a) **Cover Page:** minus 5% points for not having one.
- (b) **Introduction:** Describe the objectives of this lab, in your own words. (5% points)
- (c) **Assignments:**
For each assignment, present the results corresponding to each bullet items (i.e., (a)---(e))
- **Assignment 1:** schematic and graphs and the measurements required (15%)
 - **Assignment 2:** schematic and graphs (20%)
 - **Assignment 3:** schematic and graphs (20%)
 - **Assignment 4:** schematic and graphs (30%)
- (d) **Discussion:** Discuss your observations and learnings from this lab. (10% points)
- (e) **Reference:** If any.