ECET 300

PSPICE PROBLEM MANUAL

Developed

by

Ronald H. Rockland, Ph.D.
## TABLE OF CONTENTS

EXPLANATION OF PROBLEMS ...................................................................................................................... 3

PROBLEM 1 .................................................................................................................................................... 4

PROBLEM 2 .................................................................................................................................................... 5

PROBLEM 3 .................................................................................................................................................... 5

PROBLEM 4 .................................................................................................................................................... 6

PROBLEM 5 .................................................................................................................................................... 6

PROBLEM 6 .................................................................................................................................................... 7

PROBLEM 7 .................................................................................................................................................... 7

LAPLACE TRANSFORM IN PSpICE ............................................................................................................ 8

USING THE LAPLACE COMPONENT ........................................................................................................ 9

PROBLEM 8 ................................................................................................................................................... 10

PROBLEM 9 ................................................................................................................................................... 11

PROBLEM 10 ............................................................................................................................................... 12

PROBLEM 11 ............................................................................................................................................... 13

FOURIER ANALYSIS ................................................................................................................................... 15

FOURIER ANALYSIS - PSpICE ................................................................................................................... 16

PROBLEM 12 ............................................................................................................................................... 18

PROBLEM 13 ............................................................................................................................................... 18
EXPLANATION OF PROBLEMS

This manual contains several problems based related material in the course. If you need to get information on PSpice, read the PSpice manual. Information specific to using special features in PSpice related to this course is also included in this manual. An assignment table will be given to you when the syllabus is handed out. Besides doing the required steps, you are encouraged to experiment by

- Changing values of components
- Adding/deleting components
- Changing the measuring points
- Changing the amplitude or type of input

You will be required to hand in a report based on the assigned problems. This report should include the following:

1. A title page with your name
2. An introductory section detailing what you think you will be accomplishing in this exercise, what new concepts you will be learning, and what if any additional items you tried.
3. Detailed calculations (similar to a pre-lab) which will demonstrate what you expect to see. This may involve mathematical proof of responses.
4. Printouts of the schematic(s) and of the responses. These printouts should be pasted into the Word document, as opposed to being printed out. If there is a problem doing this, see me during office hours.
5. A brief conclusion as to the observations, how close they came to the “pre-lab” calculations, and how do you think this exercise could have been changed.
PROBLEM 1

1- Place the following components on a new drawing page: 4 Resistors, VDC and AGND.
2- Move and rotate the components so they appear as in the figure below.

3- Wire the components together.
4- Change the values to the components as follows: \( V_1 = 100 \), \( R_1 = 7.5\,k \), \( R_2 = 5\,k \), \( R_3 = 3\,k \) and \( R_4 = 2\,k \). (Note that you do not have to put in \( \Omega \) for the resistors). You might have to move the value box away or nearer the part.
5- Add two VIEWPOINT objects to the drawing- one attached to the right of \( R_1 \), the other on top of \( R_4 \). You should have the figure below.

6- First calculate the voltage at the two VIEWPOINT objects. Show the calculations.
7- Save the drawing first, as CHAP3-1.
8- Run a simulation, and compare the values to those from item 6.
9- Change \( R_2 \) to 2\,k, and \( R_4 \) to 4\,k. Calculate first, and then run the simulation again.
10- Save this again and close out of this file.
PROBLEM 2

1- Create a waveform with the following characteristics, using the VPWL source: T1=0, V1=0, T2=1, V2=1, T3=2, V3=1, T4=4, V4=0.

2- Integrate and differentiate the waveform, both theoretically and using PSpice.

3- Create the figure shown in Fig. P2-60 (only the first two waveforms) using the VPWL source. Differentiate and integrate this waveform theoretically and with PSpice.

PROBLEM 3

1- Create a voltage waveform, using the VPWL source, with the following parameters: T1=0, V1=0; T2=3, V2=5; T3=6, V3=0. Place a 1k resistor across this source. Note that V and s are assumed units if they are not specified.

2- Place a BUBBLE part at the output (across the resistor), call that part Vout.

3- Use the Set Up button to run a Transient Analysis. Set the Print Step to 2ms and the Final Time to 10s. Run the simulation.

4- Add the trace Vout to the graph and observe the waveform.

5- Add another plot to this graph, and add a trace to that new plot with the function AVG(Vout).

6- Place a cursor on the upper graph, and move the cursor until it is at 6s. Read the value of the upper graph, and compare it to the calculated average for this triangular waveform.
PROBLEM 4

1- Place the following components on a new drawing page: 1 Resistor, VSIN, 1 Capacitor and AGND.

2- Move and rotate the components, and wire them together. Also add two voltage markers—one at the input, and one at the output.

3- Change the capacitor value to 1u.

4- Set the values of VSIN as follows: VOFF=0, VAMPL=10V, FREQ=1000.

5- Set the Transient values to 1us for the Print Step, and 2 ms for the Final time.

6- Run the simulation and observe the two curves. Which voltage (input or output) leads, and approximately by how much. How do you calculate phase?

7- Replace the position of the resistors and capacitors. What happens to the phase relationship between the input and output voltage.

PROBLEM 5

1- Solve problem 4-17 for the voltage and current waveforms, using PSpice. Estimate theoretically what the initial and steady state condition should be. Does it conform to the Probe waveform?
PROBLEM 6

1- Draw the circuit in Figure 6-49 in PSpice.

2- Derive the output v(t) theoretically, using Laplace transforms. Do not use MATLAB for the transforms (do it by hand).

3- From the output v(t) you derived theoretically, use MATLAB to plot this voltage, and paste this plot in Word.

4- Use a VPULSE generator in place of the 10V source and switch with the following parameters:
   V1=0 and V2=10
   TR and TF = .01us
   PW=10 and PER=20

5- Run the simulation with the Setup dialog box as Transient, Print Step= 20ms, and Final Time=7000ms. Copy this plot to word

6- Observe what happens and compare it to the theoretical analysis from step 3.

PROBLEM 7

1- Analyze, both theoretically and with PSpice, problem 6-25.

2- Try to do problem 6-26. See if you can determine how to do it with PSpice (you should be able to do it theoretically, and use MATLAB to plot the final output voltage).
LAPLACE TRANSFORM IN PSPICE

You will Learn:

✓ How to use the Laplace component in a circuit
USING THE LAPLACE COMPONENT

You can analyze the effect of a system with a specific Laplace transform by utilizing the Laplace component. This component allows you to specify both a numerator and denominator. To analyze the effect of the transform, you need to have an input (either a pulse or sine wave), the setup (either an AC sweep or transient), a resistive load on the output, and a ground.

The resistive load will not be part of the circuit in analyzing the response. Therefore, the response you will observe will be strictly due to the Laplace component, and not as a result of voltage division between the resistor and the Laplace component.

Using the Laplace Component

1- Set up a circuit with a VPULSE source, a LAPLACE component in series, a resistor in series with the LAPLACE component, and a ground (AGND).

2- Double click on the VPULSE component and set up the parameters (see Lesson 2).

Note: You want to set up the pulse width parameters based on the time constants of the transform. For example, if the transform is \( \frac{1}{s + 1} \), then you want to specify the PW (pulse width) of the VPULSE component as 1s (1 second), if you want to see the entire response.

3- Double click on the LAPLACE component and specify the numerator and denominator. If you want higher order, type in s*s to represent \( s^2 \), and s*s*s to represent \( s^3 \).

4- Place a voltage marker on the input, and another one on the output.

5- Type in the appropriate values for the Print Step and Final Time in the Transient setup. Make sure that the Final Time is no more than 1000-2000 times the Print Step (otherwise the simulation will take a long time).

6- Run the Simulation. You should see the output in the Probe window that represents the Laplace transform.

Do not include the resistor in the analysis of the circuit (i.e. use the resistor as a voltage divider). Only consider the Laplace transform block.
PROBLEM 8

1- Setup up the circuit as shown below, with the LAPLACE component set to the default of $\frac{1}{s+1}$, and the resistor set to 1, and the VPULSE component set to 1us for the rise and fall time, 1s for the PW, and 10s for the PER. Make sure that you use the AGND component. Also, set V1 to 0, V2 to 5.

2- Place two voltage markers (one on the input, and one on the output).

3- In the Transient setup, set the PRINT STEP to 10ms, and the FINAL TIME to 5s.

4- Run the simulation. The graph should appear as shown below. Verify theoretically that this is the correct answer, by plotting your theoretical answer with MATLAB, and comparing to PSpice.
PROBLEM 9

1. Set the LAPLACE component to $\frac{1}{s + 2}$, and run the simulation. Observe the graph, and describe the differences between Exercise 1.

2. Develop the theoretical solution to transform $\frac{1}{s + 2} \left( \frac{5}{s} \right)$, and compare to the graph in step 1.

3. Put another LAPLACE component in series, and set it to the same value as in step 1 of Exercise 2. Run the simulation, observe the graph, develop the theoretical solution to the transform, and compare. Make sure you have voltage markers at the input and at each output of the LAPLACE component.

4. Change the second LAPLACE component to $\frac{1}{s + 1}$, run the simulation, observe the graph, and develop the theoretical solution.

5. Change the denominator in the first LAPLACE component to $s$, run the simulation, observe the graph, and develop the solution. Comment on why you are seeing the specific graph.

6. Repeat items 2-4 with a VSIN component. Set the frequency to 1 Hz (see page Error! Bookmark not defined. for additional information on the VSIN component).

You should do the theoretical work as well as the PSpice examples, and plot the theoretical results with MATLAB.
PROBLEM 10

1- Setup an RC circuit, as shown below. Set the frequency to 10Hz, the VAMPL to 5 V, the Final time to 40 ms, and run the simulation (make sure to save this file first). Measure the ratio of the peak output waveform to the peak input waveform, and record it. Measure the phase difference between the input and output.

Note that to measure phase differences, 360° represents the time interval between two consecutive peaks for the input waveform (period of the input). Therefore, to measure the phase differences between output and input, measure the time differences between the two peaks, and divide 360° by that number, and then multiply that result by the period. That will be the phase shift at that frequency. It might also be easier to change the value of the X-axis setting to 20 ms.

2- Change the frequency to 100Hz, and repeat step 1. Do the same thing for 250Hz, 500Hz, 1000Hz, and 5000Hz.

3- Take the values of the amplitude ratio (output/input), convert those values to decibels, and plot the result vs. frequency on semi-log paper.

4- Plot the phase measurements vs. frequency on semi-log paper.
5- Try repeating the same thing with the circuit as shown below (save this file with a different name)

![Circuit Diagram]

6- Go back to the original file in item 1, and leave that open.

**PROBLEM 11**

1- Develop a theoretical Bode plot (amplitude and phase) for the RC circuit that is open from the previous exercise. Use breakpoint analysis. What type of filter is this?

![Bode Plot]

2- Determine the break frequency from this plot. How would you do it, and how does it compare to the calculated break frequency?
3- Replace the VSIN in the schematic with a VPULSE (set PW equal to 1 ms, PER to 5 ms). What happens to the output waveform, and how does it compare to the frequency response of this circuit? Determine the output theoretically using Laplace transforms

4- Reverse the capacitor and resistor, and develop a new Bode plot for this circuit. What type of filter is this? Repeat items 3 & 4 for this new circuit. In all cases, do a theoretical Bode plot as well.
FOURIER ANALYSIS

You will Learn:

✓ How to do a Fourier analysis with PSpice
FOURIER ANALYSIS - PSpICE

You can perform Fourier analysis on any waveform, whether you have generated it using a simple source or the waveform is the output from a circuit.

Fourier Analysis of A Simple Waveform

1- Set up the circuit with a source, a resistive load, a ground and a voltage marker.
2- Set up the transient analysis.

If you set up the combination of pulse width, period and Final Time in the analysis so that you only see one waveform, you will see a Fourier Transform.

If you change the parameters (most likely the Final Time) so that there are repetitive waveforms in the plot, you will be approaching a Fourier series.

3- Simulate the circuit, and generate a plot.
4- Choose Trace, Fourier or click on the Fourier button in the toolbar.
5- To view the lower harmonics, change the X-axis settings.

You can also look at the Fourier analysis of any circuit using the same procedure. If you set up the voltage markers to show the input and outputs, you can then observe how the circuit is affecting the frequency content of the input signal.

EXERCISE

1- Set up a circuit with a VPULSE part, a resistor (use the default value) and an AGND part.
2- Set the VPULSE part to the following parameters: V1=0, V2=5, PW=1, PER=5.
3- Run a transient analysis with a Final Time of 5 s.
4- Change the plot to a Fourier analysis, and change the X-axis setting to user defined, 0 to 40Hz.
5- Change the X axis settings to a maximum of 10 Hz, and observe the difference in the graph.

6- Derive theoretically and compare.

7- Change the PER to 2, and change the Final Time in the transient analysis to 20.

8- Generate the waveform, and then generate the Fourier plot. Because you see many waveforms on the plot, the Fourier plot will appear more like a line plot, or more like the Fourier series.

9- Add a 100 uF capacitor in series with the resistor, and place a voltage probe at the input and output. Change the PER back to 5, change the Final Time back to 5, and run the transient analysis. Change the plot to a Fourier analysis, and compare the input and output spectra.
PROBLEM 12

1- Generate a square wave, PW=1s and PER=10s.

2- Set the Final Time to 2s, and generate a Fourier plot. How much less is the 5th harmonic from the fundamental.

3- Decrease both PW and Final Time by 1/10, and generate a new Fourier plot. Compare the two (you might want to look at only the first 10 harmonics).

4- Repeat item 3 once more.

5- Repeat items 1-4 with a triangular waveform and a sawtooth pulse (see pg. 414). Compare the three waveforms, both theoretical and from the PSpice experiment.

6- Change the PER to 2s, and the Final Time to 10s for the square wave. Generate a Fourier plot. How is it different from the one in item 2?

7- Change the Final Time to 40s for the square wave, and repeat item 6. What is happening?

PROBLEM 13

1- Generate plots for both the input and output for the circuits in Examples 6-49 and 6-50 (page 257). Make sure that you set the Final Time so that you have only one waveform appear on the plot.

2- Generate a Fourier plot, with the input and output superimposed. Compare the input and output plots for both circuits. What does the Fourier transform show, and is it correct?