School of Physical & Mathematical Sciences

Division of Physics & Applied Physics

(PH3399 Laboratory IIIB)

Experiment 8: Introduction to PSpice Circuit Designing, Coding and Analysis

Location: Computational Teaching Lab (SPMS-PAP-02-07A).
Note: Please bring along a thumbdrive to save your experimental data.

Software: PSpice 9.1 Student Version (Lite) can be downloaded here:
http://www.eng.auburn.edu/~troppel/91pspstu.exe

Recommended Readings and References

[1] PSpice User Guide (by Cadence Design Systems, Inc.)
[2] PSpice for basic circuit analysis (by Joseph G. Tront); ISBN: 0072939818

Background & Introduction

Simulation Program with Integrated Circuit Emphasis (SPICE) has been a predominant program used for circuit simulation and analysis. It was first created by the University of California, Berkeley in the 1970s. PSpice, created by Cadence Design Systems Inc, is a PC version of SPICE which allows the end user to perform numerous kinds of circuit analysis, i.e. DC, AC, transient and etc. In addition to the basic electrical components, PSpice has a wide digital and analog library of standard components such as logic gates and transistors. While the end user could draw circuits and set electrical parameters based on design, PSpice also allows the end user to code circuits to input their electrical parameters. This enhances the flexibility of the circuit architecture in PSpice.

Objectives

In this experiment, students would learn the basics of PSpice circuit simulation. Students would first learn to design circuits by drawing and integrating electrical components into circuits. Thereafter, they would learn to code the circuit, i.e. an alternative method to design circuits. After learning the basics, students would be tasked to design and code an RLC circuit with guidance. They would also be tasked to analyse graphical outputs from circuits and provide explanations to justify their analysis. At the final stage of the experiment, students would be tasked to design a half-wave rectifier circuit independently.

1. Designing basic PSpice circuits

In this session, students would explore the functions and parameters of PSpice and draw a simple circuit.

First, go to the "Start" menu and click on "Schematics". You would see the schematics window as shown in Fig. 1, a platform where you would be drawing/designing your PSpice circuits.
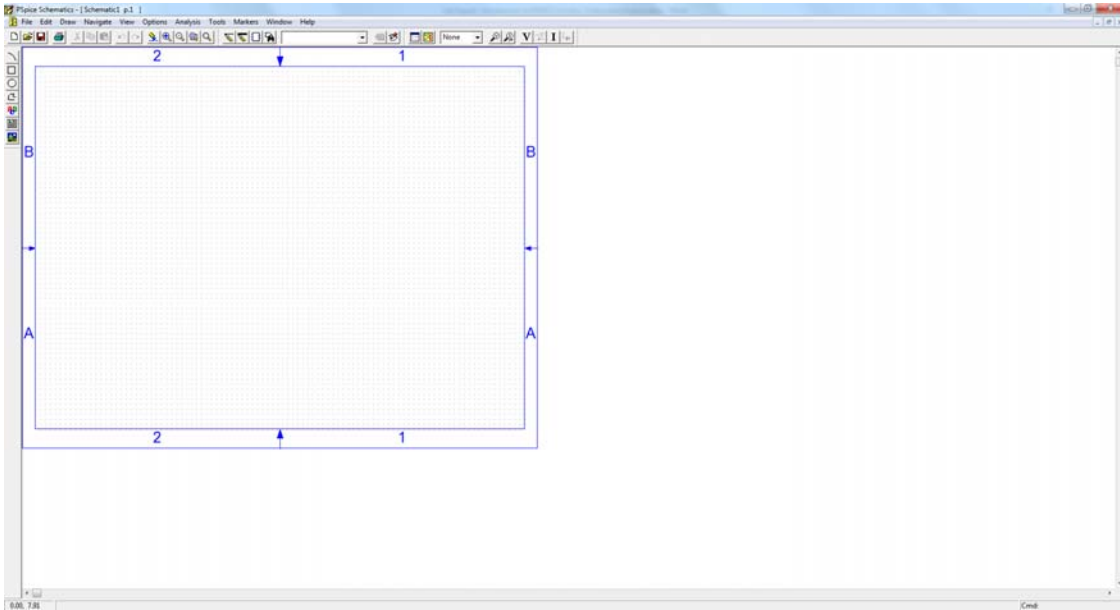
Fig. 1 PSpice schematics window

In this practice session, perform the following:

   (a) Explore the functions as shown at the top of the window (Fig. 2).

Fig. 2 PSpice functions

    (b) Draw a simple circuit, e.g. a simple circuit with a DC voltage source, resistor and ground.
    (c) Perform a transient analysis on the circuit and explore its parameters.
    (d) Plot Voltage vs. Time (V-t) & Current vs. Time (I-t) graphs.
    (e) Verify if these graphical outputs coincides with your concept and understanding of circuits.

2. Transient analysis on RC and RLC circuit designs

In this session, students would learn how to perform transient analysis on RC and RLC circuits using PSpice.

(a)   (b)
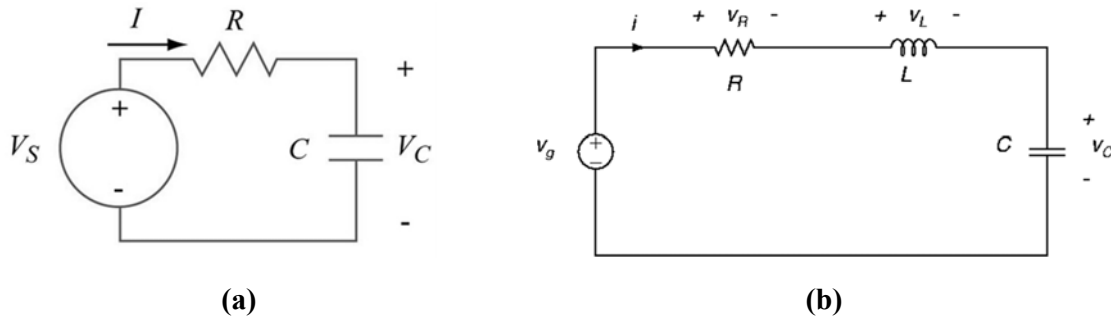
Fig. 3(a) RC circuit, **(b)** RLC circuit

Using what you have learnt in the previous session,

(a) Draw RC and RLC circuits as shown in Fig. 3.
(b) Set appropriate values for all the electrical components, use a DC voltage source for simplicity.
(c) Perform a transient analysis on the circuits and plot the I-t and V-t graphs of each electrical component.
(d) Label the graphs properly and explain the differences between the respective RC and RLC circuit graphs.
(e) Perform graphical analysis and error analysis (if any).
(f) Save the circuits into separate files in a self-created folder in C: drive.

3. Coding RC and RLC circuits

In this session, students will learn how to code PSpice netlists. Netlists are used in conjunction with circuits to set the parameters of electrical components. It offers the user an alternative to code rather than input the parameters directly into electrical components. Fig. 4 shows a PSpice sample netlist.

```
R1      node1  node2 value1
C1      node2  node3 value2
V1      node1  node3 value3
V2      node3  node4
+PULSE (V1 V2 TD TR TF PW PER)
V3      node4  node1
+PWL (T1 V1 <T2 V2 T3 V3 T4 V4 ...>)
```

Fig. 4 PSpice netlist format

First, study and understand the parameters of these voltage sources:

- Pulse: PULSE(V1 V2 TD TR TF PW PER)
- Sinusoidal: SIN(VO VA FREQ TD THETA)
- Exponential: EXP(V1 V2 TD1 TAU1 TD2 TAU2)
- Piecewise linear: PWL(T1 V1 <T2 V2 T3 V3 T4 V4 ...>)
- Single frequency FM: SFFM(VO VA FC MDI FS)

Code the circuits using the RC and RLC circuits from the previous session and perform a transient analysis on the circuits:

(a) Replace the DC voltage sources by coding the voltage pulse and piecewise linear sources for RC and RLC circuits respectively.
(b) Perform a transient analysis on the circuits and plot the I-t and V-t graphs of each electrical component.
(c) Label the graphs properly and explain the differences between the respective RC and RLC circuit graphs.
(d) Perform graphical analysis and error analysis (if any).
(e) Save the circuits with a different name in the same C: drive folder.

4. Designing a Half-Wave Rectifier

In this session, students would design a half-wave rectifier independently. Fig. 5(a) and (b) exhibit the half-wave rectifier circuit and graphical plot respectively.

(a)   (b)
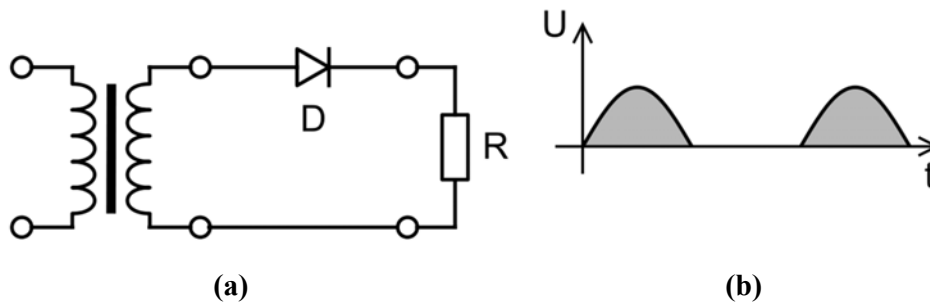
Fig 5. Half-wave rectifier **(a)** circuit and **(b)** graphical plot

The step down transformer has turn ratios 15:1. The load has resistance of 10kΩ. The AC voltage source has amplitude of 150V and frequency 50Hz. The diode model used is D1N4002.

Using equation (1) and the above information, determine appropriate inductances for L_1 and L_2.

$$\frac{L_1}{L_2} = \left(\frac{N_1}{N_2}\right)^2 \qquad (1)$$

(a) Perform a transient analysis on the circuits and plot the I-t and V-t graphs of each electrical component.
(b) Perform graphical analysis and error analysis (if any).
(c) Find out the voltage amplitude of the half-wave rectifier.

(d) Thereafter, add a 50uF capacitor in parallel to the load. Repeat steps (a) to (c) for the new circuit and find out its estimated average DC output voltage.

(e) Label the graphs properly and explain the differences between the respective half-wave rectifier graphs.

Bonus: Write a netlist that represents the half-wave rectifier circuit in Fig. 5(a).

5. Additional Information

- Duration of experiment: 6 weeks (half-semester)

Table 1. Experimental Schedule

	1st Half Semester	2nd Half Semester	Tasks
Session	Week	Week	
1	2	8	• Designing basic PSpice circuits
2	3	9	• Transient analysis on RC and RLC circuit designs
3	4	10	• Coding switches in RC & RLC circuits
4	5	11	• Designing a Half-Wave Rectifier
5	6	12	• Lab report preparation • Viva preparation
6	7	13	• Consultation • Viva (contact Assoc. Prof. Lew Wen Siang for viva session)

Table 2. Contact Personnel Particulars

Professor-in-Charge	Assoc. Prof. Lew Wen Siang
Office Address	SPMS-PAP-03-04
Contact No.	6316 2963
Email	wensiang@ntu.edu.sg
Student-in-Charge	Mr. Desmond Loy Jia Jun
Office Address	SPMS-PAP-02-07A (Computational Teaching Lab)
Contact No.	9117 4936
Email	dloy002@e.ntu.edu.sg