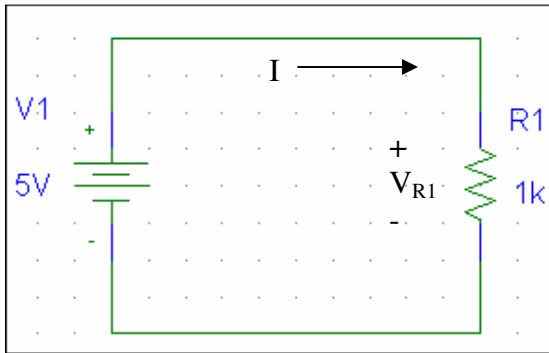


LAB #2: AN INTRODUCTION TO PSpICE

This lab is to give you an introduction in the OrCAD software Pspice that is powerful circuit simulation software. You will be using Pspice for most of the labs from now on for circuit design and implementation, so it is imperative to ask questions and to familiarize yourself with it during this lab. Given the following circuit, we will recreate and simulate it using PSpICE.



- 1) Open Pspice Schematics from the PROGRAMS Folder in the START Menu. This will give you a blank grid.
- 2) The figure below is a graphic of the Pspice toolbar and the identification of the objects you will encounter.
- 3) To place a part onto the schematic, click DRAW, then GET NEW PART (Ctrl+G). This will give you a menu where you can find a part.

Figure 1

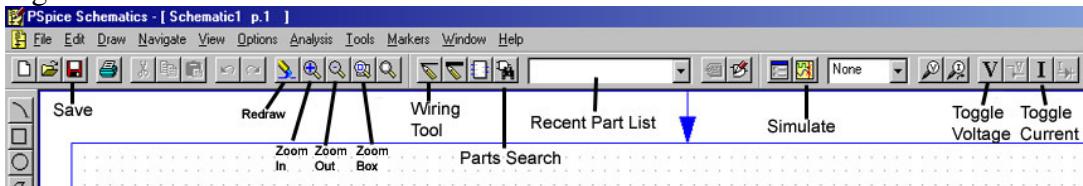
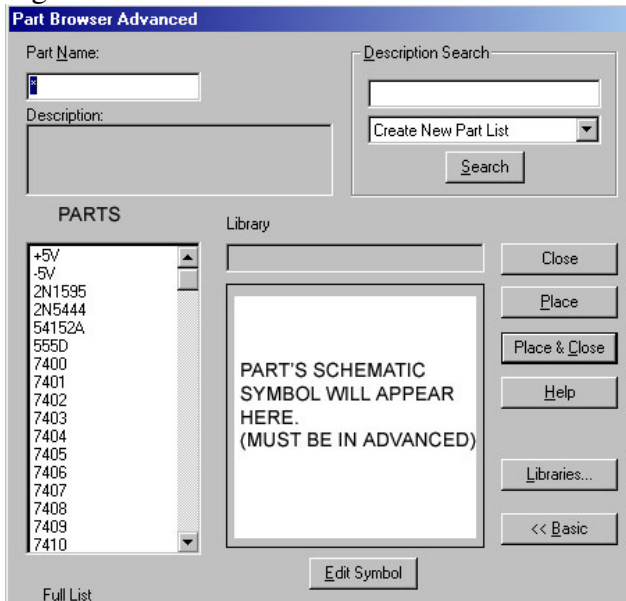


Figure 2



You can select a part by typing its part number or part identification (i.e. “R” for Resistor, “C” for Capacitor, etc.) in the “Part Name” field or you can select it by scrolling the list and clicking the part you want.

After you select the part click the “Place & Close” button to return to the schematic. You can repeat step 3 for every part.

Wire each component together using the wiring tool.

◆ To rotate a part, click on the part and press Ctrl+R, or select ROTATE from the EDIT menu.

MOST COMMON PARTS

Component	DC Power Supply	Resistor	Variable Resistor	Ground
Part Name	VDC	R	R_Var	Gnd_earth

Now that we have the basics, we will begin wiring the circuit we were given.

- We will start by Opening a NEW schematic by clicking on FILE and then NEW.
- Place the following parts onto the schematic: a Resistor (R), a DC power supply (VDC), and a ground (GND_EARTH).
- Using the wiring tool, connect each end of the components to the proper connection (**Refer to the circuit**).

BE SURE TO GROUND YOUR CIRCUIT OR ELSE IT WILL NOT SIMULATE!

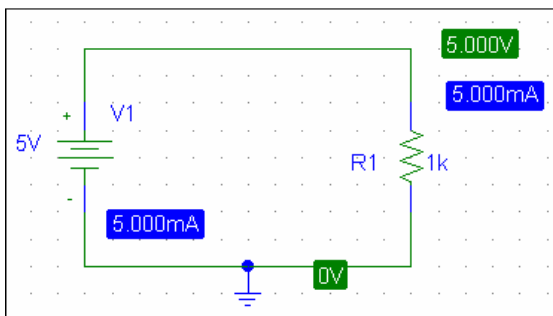
- Set the values of each component by double clicking on the value. A little box will pop up, set it to the appropriate value.

PREFIXES TO REMEMBER

KILO	MEGA	PICO	MICRO	NANO
k	Meg	p	u	n

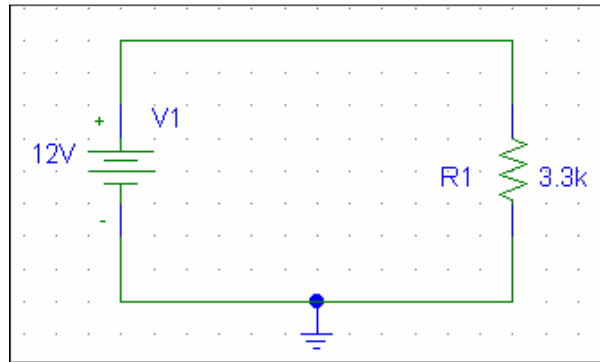
EX. 1 kΩ is 1k, 1.4 MΩ is 1.4M, and 4.5 μF is 4.5uF

- After all the values are set and the circuit resembles the circuit given, we are ready to simulate the circuit. To simulate a circuit, click ANALYSIS, then SIMULATE (F11) or the button on the toolbar. Pspice will prompt you to Save the schematic; Save the file as Lab 2.



To turn on the VOLTAGE and CURRENT values, click the “V” and “I” icons on the toolbar. You can see that Pspice evaluates the circuit for you. We have 5 mA of current flowing through the circuit and 5 V over the resistor R1. This is correct because: Ohms Law: $I = V/R = 5V/1k\Omega = 5 \text{ mA}$

Now that you have gone through an example of Pspice, perform analysis on the following circuit, write down the results of the simulation, and answer the questions at the end of the lab.



RESULTS (Don't forget the units) (10 pts. Each)

Voltage of R1 (V_{R1}) = _____

Current of the circuit (I) = _____

PRINT YOUR SCHEMATIC AND ATTACH TO YOUR LAB. (25 pts.)

Print by first selecting your circuit by holding the left mouse button and dragging the box that is created over your circuit. After the box is made, print your circuit by using the FILE menu and selecting PRINT or by using the Printer Icon on the toolbar.

QUESTIONS

1. How is a part placed in Pspice? (15 pts.)

2. Using the Part Browser (Get New Part Menu), draw the schematic symbol for the part name "D". (10 pts.)

3. Using the Part Browser (Get New Part Menu), draw the schematic symbol for the part name "uA741". (10 pts.)

4. Give two reasons on why Pspice is useful for circuit analysis. (10 pts.)

5. What was the objective of this lab? (10 pts.)

6. What was the equipment used for this lab? (5 pts.)

7. How confident are you in your ability to use Pspice? What areas do you still need help with? (5 pts.)