COLLEGE OF ENGINEERING AND MINES
DEPARTMENT OF ELECTRICAL AND COMPUTER ENGINEERING

COURSE CODE	EE F102 F01 (CRN: 34544)		

COURSE NAME	INTRODUCTION TO ELECTRICAL AND COMPUTER ENGINEERING		

SEMESTER	SPRING	YEAR	2022

LABORATORY LOCATION	ELIF 331 (ELECTRONICS LAB)		

LAB SESSION DATE AND TIME	MONDAY 31 JAN 2022		

TYPE OF SUBMISSION	LABORATORY REPORT	NUMBER OF SUBMISSION	2

TITLE OF SUBMISSION	MEASURING VOLTAGE AND CURRENT		

METHOD OF SUBMISSION	ONLINE TO: maher.albadri@alaska.edu		

DUE DATE OF SUBMISSION	MONDAY 07 FEB 2022	DUE TIME OF SUBMISSION	23:59

STUDENT NAME	

MAKE THIS FORM A "COVER PAGE" FOR YOUR REPORT SUBMISSION.

FOR THE TA USE ONLY
REMARKS:

Measuring Voltage and Current

Objective:
In this lab we will gain experience using a multimeter and power supply. We will learn how to measure voltage and current. We will learn the difference between node voltage and differential voltage. We will characterize the output voltage of a voltage regulator versus the input voltage and compare those measurements to simulation.

Equipment:
Agilent 34401A or 34410A Multimeter
Agilent E354xA Dual Output Power
Supply Protoboard
Parts kit

Background:
Two fundamental electrical parameters of all circuits are voltage and current. If you know the voltage across every branch in a circuit and the current through every branch of a circuit, then you know everything about the circuit. From these two measurements you can derive the resistance and the power dissipated in each branch of the circuit. You will also know how much energy the circuit uses to do useful work and how much energy is wasted and therefore the efficiency of the circuit.

Most of our measurement tools are based on devices that measure either voltage or current. For instance, analog multimeters and dial type meters such as your odometer and tachometer in your car use a current sensitive device called a galvanometer (see Figure 1). A galvanometer is created by attaching a needle to a coil and then placing that coil in a magnetic field. When a current passes through the coil it experiences a torque proportional to the current that will cause the needle to deflect. The coil's movement is opposed by a restoring spring such that the amount of deflection becomes proportional to the current passing through the coil. Devices created using galvanometers are sometimes called "moving coil meters".
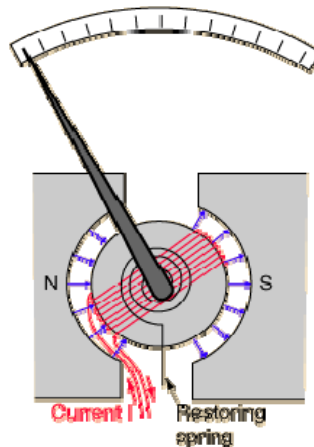

Figure 1: Galvanometer showing the flow of current and restoring spring.
(http://hyperphysics.phy-astr.gsu.edu)

A galvanometer is typically defined by the amount of current required to deflect the needle to full-scale, I_G , and the internal resistance of the galvanometer, R_G . In order to design an ammeter that will measure currents larger than I_G, a small resistor, R_P, is placed in parallel with the

galvanometer so as to shunt most of the input current around the galvanometer (see Figure 2[a]). In order to design a voltmeter, a large current limiting resistor, R_S, is placed in series with the galvanometer (see Figure 2[b]). The galvanometer is still sensing current. The voltage is inferred using Ohms law ($V = IR$) by measuring the current through a known resistor. We can also use a galvanometer to measure resistance. Again using Ohms law ($R = V / I$) we can design an ohmmeter by placing a known voltage source in series with the galvanometer (see Figure 2[c]). The voltage source injects a current through the unknown resistor which is then measured by the galvanometer. Note that in this case the current is inversely proportional to the resistance; so most analog meters have a non-linear scale for resistance.

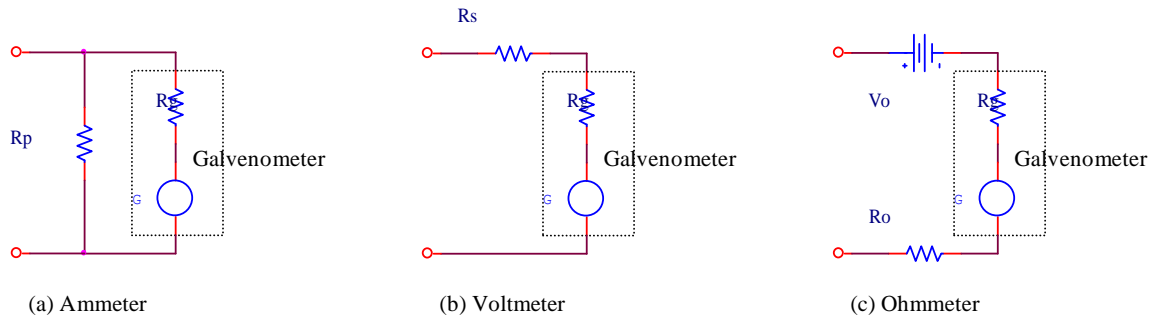(a) Ammeter   (b) Voltmeter   (c) Ohmmeter

Figure 2: Design of a single scale (a) ammeter, (b) voltmeter, and (c) ohmmeter.

A voltmeter measures the change in electric potential energy (volts) between two points and is therefore connected in parallel with the circuit to be measured (see Figure 3[a]). An ammeter measures the electric current (amperes) in a branch of a circuit and therefore must be placed in series with the measured branch so that all the current passing through the branch also passes through the meter (see Figure 3[b]). In order to not affect the circuit measurements, a voltmeter requires very high internal resistance, and an ammeter requires very low internal resistance.

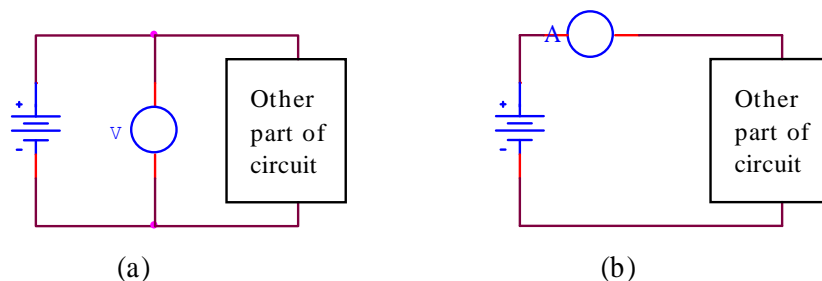(a)   (b)

Figure 3: Connection of meter to circuit to make (a) voltage and (b) current measurements

In lab we use two solid-state multimeter, Agilent 34401A or 34410A, which are built around a voltage sensitive device rather than a current sensitive device like the galvanometer. Equivalent circuits to those shown in Figure 2 can be constructed to illustrate how to measure current and resistance using a voltage sensitive device. The Agilent multimeters are standard DMM (digital multimeter) capable of a wide range of measurements. Figure 4 - Figure 6 shows how leads can be connected to the multimeter and which buttons to press to perform basic voltage, resistance, and current measurements.
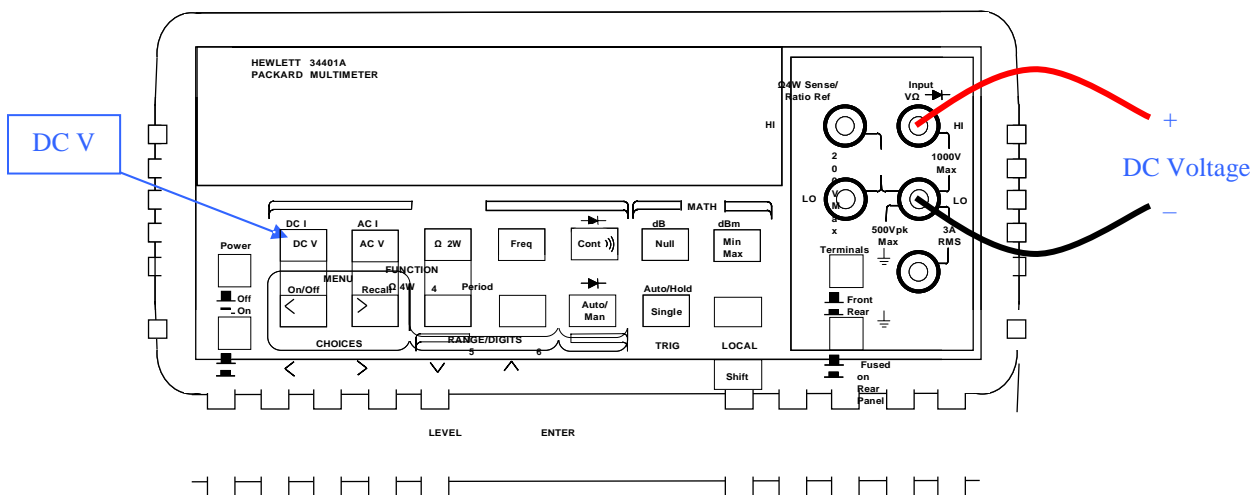
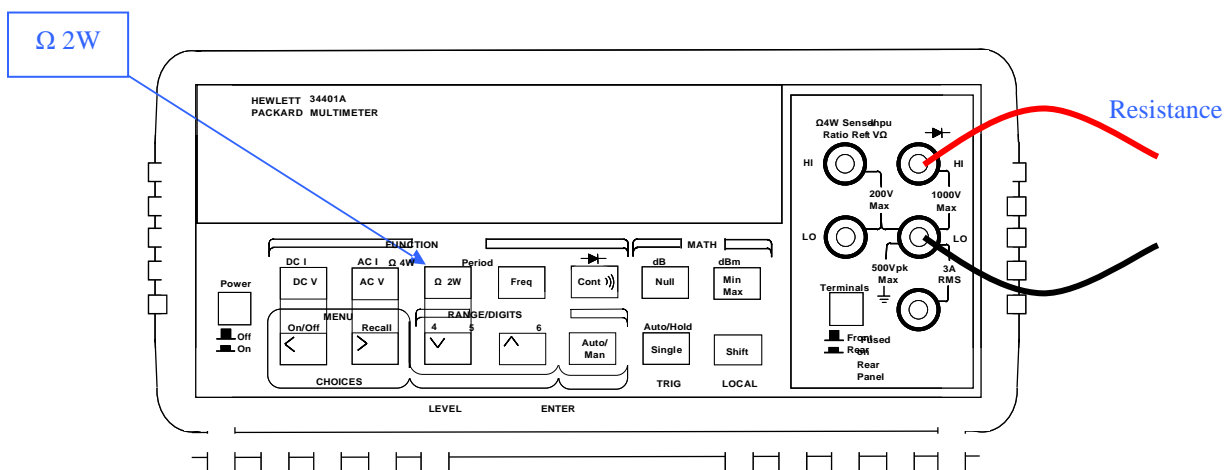Figure 4: Lead connections and selection button for DC voltage measurements.

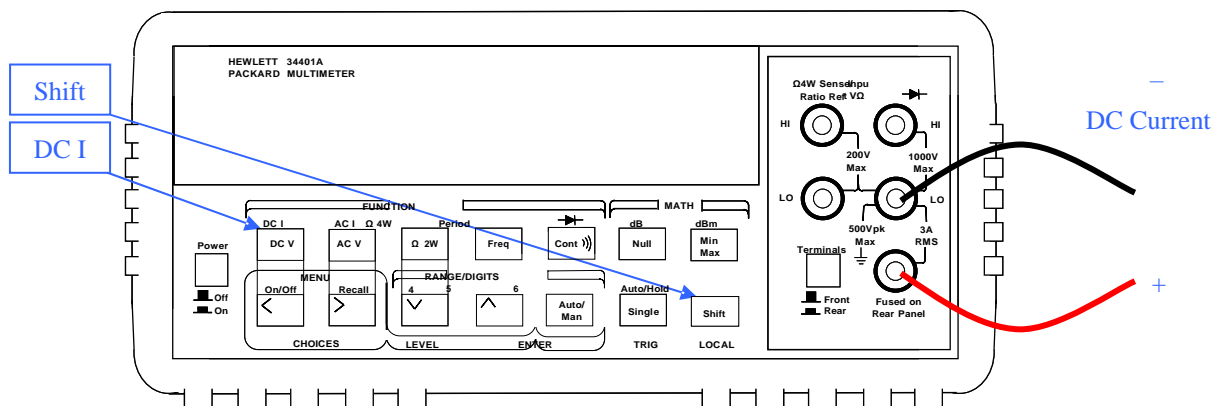Figure 5: Lead connections and selection button for resistance measurements.

Figure 6: Lead connections and selection button for DC current measurements.

Voltage Measurements

As previously mentioned, voltage is the measure of the difference in potential energy between two points. In other words, a voltage measurement is always a measure of the potential energy at one point relative to another point. The value shown on the DMM display is defined as being the potential energy on the red (V) lead relative to the potential energy on the black (common) lead.

All voltage measurements are **differential** voltage measurements. A special kind of differential voltage is called a **node** voltage, where the reference point is assumed to be circuit ground. In specifying a differential voltage between points A and B, where the potential at A is measured relative to the potential at B, one writes V. The node voltage at A (measured relative to circuit ground) is specified as V_A. As we will discuss later in class you can derive the differential voltage V_{AB} by simply subtracting the node voltages $V_A - V_B$.

Hints for using the multimeter:

The Null button allows you to make a relative measurement, i.e., result = reading – null. To null the test lead resistance for more accurate two-wire ohm measurements, short the ends of the leads together and then press NULL. This measurement will be stored in the NULL register. The null register is cleared when you change functions, turn null off, turn off the power.

Schematic Capture and PSpice Simulation Tutorial

You can watch a demonstration of this simulation via the following link:

PSpice Simulation Tutorial

In addition, the following text describes the simulation steps:

We use the schematic capture, simulation, and pcb layout tools provided by the CADENCE University program. These tools are licensed to run in ELIF 331 lab. They do not exist on any other computers at the university. For home computer use, you may download OrCAD from http://www.orcad.com/resources/orcad-downloads. The instructions require that you fill out a request form. After submitting the form, an email will be sent to your specified email address containing the link for downloading the software. The download file is very large. The OrCAD 16.6 is similar, *but not identical,* to the Design Entry CIS tool that we use in lab.

Create a folder under your user space and call it EE102_Project. We will use this folder for our design. The schematic capture, simulation, and pcb layout tools generate a considerable number of files. It is very useful to make sure that all those files are contained in a single design folder. If you wish to archive your design, or copy your design to another machine, save the entire folder so that all the necessary files will be contained in one place. Go to **CANVAS → LABORATORY → Cadence Libraries** and download *elab.lib* and *EE102_LIBRARY.OLB* to the folder that you just created.

This lab will get you started in using the schematic capture and simulation tools. We will explore the pcb layout tool in another lab.

To start the schematic capture program on ELIF 331 computer select:

   Start->Programs->Cadence->Release 16.6->Design Entry CIS

On your home computer select:

> *Start->Programs->Cadence->OrCAD Capture CIS Lite*

Make sure that Allegro PCB Design CIS L is selected when the Cadence Product Choices pop-up window appears; then click *OK*. If you get an error, contact me immediately so that I can fix it. After a bit of a delay the program window will appear. To create a new project, select:
> *New Project*

When the **New Project** dialog box appears type in your project name (e.g., VoltageRegulator), select *Analog or Mixed A/D*, and finally specify the location you wish to save your project by using the browse button and select the folder you previously created as the location to save your files. Select OK. Another pop-up window will appear. Select *Create based upon an existing project*, empty.opj should be listed in the field and click *OK*.

After some time, a schematic page should appear. If the schematic page does not appear but the file structure for the design appears then follow the directory structure under your *.dsn* until you find *Schematic1* and *Page1*. Select *Page1* to open your schematic page. When the schematic page is up, examine the icons along the right side of the window. These icons will allow you to place components, wire the components, add power and ground terminals, draw different shapes and add text. Examine the icons and menus along the top of the window. These menus will also allow you to place components, etc, as well as set-up and run a simulation.

Look on the l o w e r right hand side of your schematic page. Double click on *<Title>*. A *Display Properties* window will appear. Change the *Value:* field to an appropriate title. Double click on *<doc>* and put your name in the *Value:* field.

Click on the *Place Part* icon on the right hand side of your window. The previous file you downloaded, *EE102_LIBRARY.OLB,* is a library of all the parts needed for this project. You will also need the *SOURCE* library. Select *Add Library,* find your folder, select *EE102_LIBRARY.OLB*, and click Open. The *SOURCE* library is located in:

> *C:\Cadence\SPB_16.6\tools\capture\library\pspice\source.olb* or,
> *C:\OrCAD\OrCAD_16.6_Lite\tools\capture\library\pspice\source.olb*

Build the following schematic (see Figure 7) on your schematic page. You can select parts by clicking on the *Place Part* icon to the right.
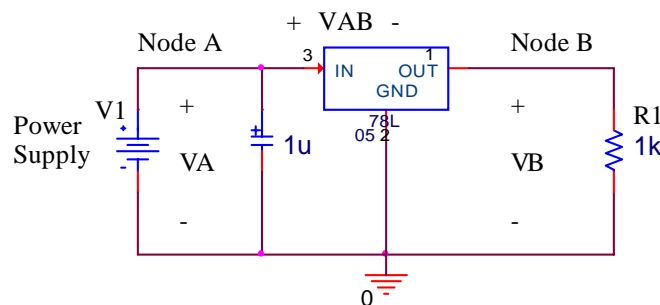

Figure 7: Voltage Regulator

You can find the parts by typing VDC for V1, and R for R1, in the part name box. You can rotate parts on your schematic page by pressing 'R' when the part is selected. You can change the value of the parts by double clicking on the value you wish to change and then changing that value when the *Display Property* box appears. The parts are connected by selecting the *Wire* icon to the right of the window.

The ground symbol is found by selecting the Ground icon to the right of the window and then selecting the 0/SOURCE ground. You must select the 0 ground in order for the simulations to run properly. This is a special ground that specifies where the 0 node or reference for the entire circuit will be. If the 0/SOURCE ground does not appear in the **Place Ground** window, select *Add Library* and browse the directory until you find *library/pspice/source.olb*. Select *source.olb* then click on OPEN. The 0/SOURCE should now appear in the **Place Ground** window. The simulation tool in PSpice performs all its calculations relative to this reference node.

The *voltage probe* icon is found at the top of the window. Your voltage probe will not turn color until after a simulation has been performed.

After you have created your schematic you need to specify what type of calculations you wish to perform. For this simulation we wish to examine how the output voltage changes for different input voltages.

        PSpice->New Simulation Profile

Give the simulation a name with the pop-up window appears. Several things need to be setup in the **Simulation Settings** dialog box. For this particular simulation we need to specify where the model of the 78L05 is located. Select the *Configurations File* tab and select *Library* in the *Catagory* window (see Figure 8). Now browse in your project folder until you find the *elab.lib* file you previously downloaded. Click *Add to Design* and *Apply*.
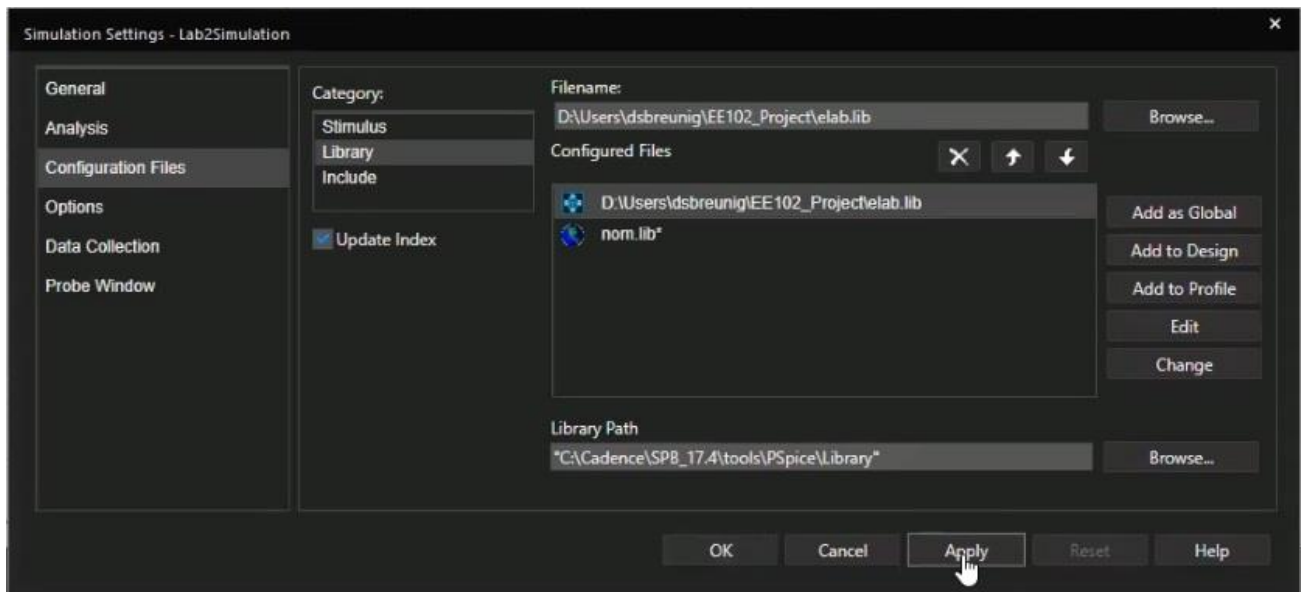

Figure 8: Setup window for 78L05 model library.

Now we need to specify the type of simulation we wish to perform and give the necessary parameters. Select the *Analysis* tab. We will be sweeping V1 from 0 to 9 V. Select the following parameters (see
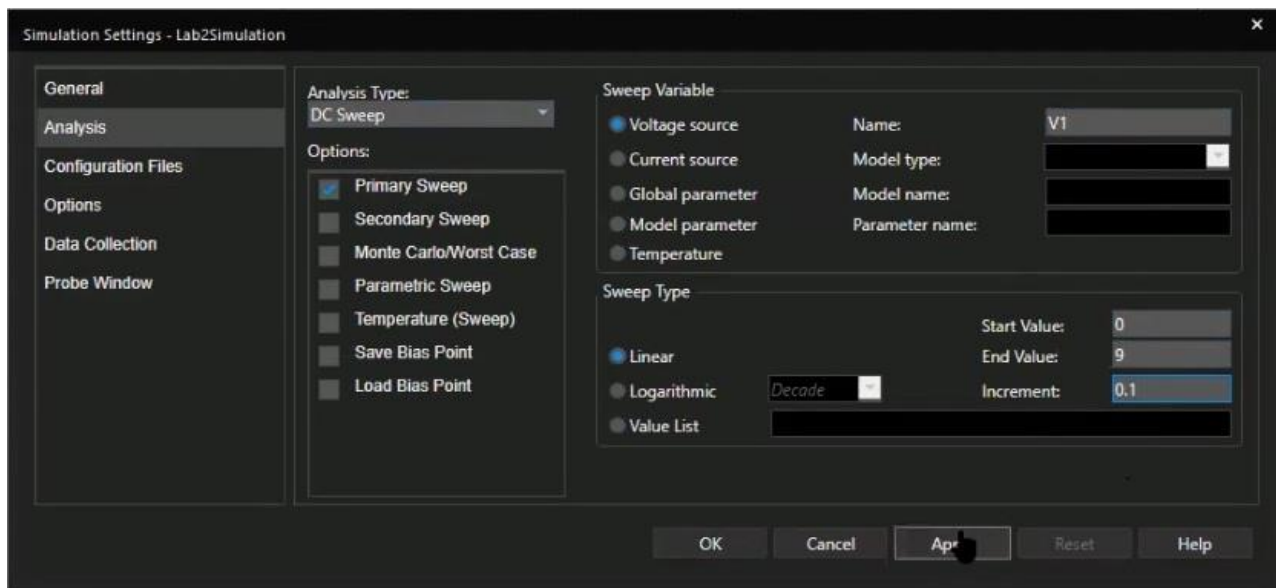
Figure 9) and click OK.

Figure 9: Example of the simulation setup for a voltage sweep.

Finally, we need to run the simulation. You can either select the *run simulation* icon at the top of the page or select:

PSpice->Run

After a while, a new window will appear showing the simulations progress. When the simulation is complete, a plot similar to Figure 10 will appear. If the plot does not appear, go back to your schematic and place a voltage probe (found in the top menu bar, looks like a scope probe with a V next to it) at the top of R1.
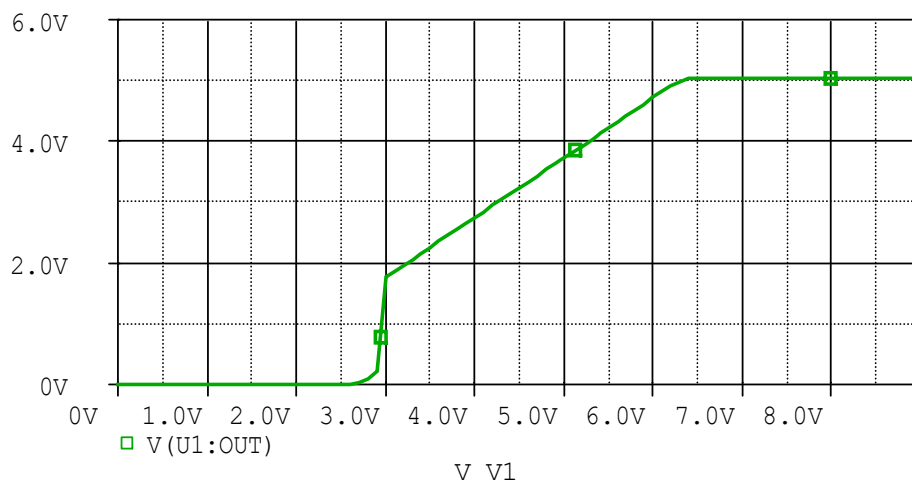
Figure 10: 78L05 output voltage versus input voltage simulation.

If necessary, change the axis of the plot by selecting *Plot->Axis Settings*. Choose *Y Axis tab*, *User Defined Data Range* and change the range to 0 to 9V.

Since we wish to compare the simulation with our measurements we need to acquire the actual data values in a form that can be imported into excel. Select:

   File->Export->Text

When the **Export Text Data** window appears select the browse button (…) find your folder and specify a name for the file. This will then write data to an ascii file that can be imported into Excel. (See Excel Help for information on importing data into Excel.)

You can make measurements by selecting the *Evaluate Measurements* icon at the top of the simulation window. We would like to know what the maximum output of the voltage regulator is. Select *Max(1)* from the *Functions or Macros* window and then select your simulation output variable (mine was *V(U1:OUT)*). If you do not know what your simulation output variable is look on the bottom left side of your plot. The figure will show you what variable is plotted. My result was 5.03408 V.

DC Voltage and Current Measurements
In lab:
1. Connect the circuit shown in Figure 7_{AB} One possible layout in shown in **Figure 11**. See the datasheet for the 78L05 to determine which pins corresponds to the input, output, and gnd connections. Make sure that you use the 78L05! You also have a transistor that looks similar. Measure the node voltages V_A and V_B (i.e., the voltage from nodes to ground) and the differential voltage V_{AB} while adjusting the power supply voltage from 0 V to 9 V. Be sure to take enough measurements to show all relevant changes in output voltage versus input voltage. Measure the current, I, through the 1 kΩ resistor. Determine the input voltage for which the voltage regulator just turns on (i.e. output voltage becomes greater than zero) and the input voltage for which the voltage regulator falls out of regulation (i.e. output voltage becomes less than 5 V). Record all these values in a table similar to that shown in **Table 1**.
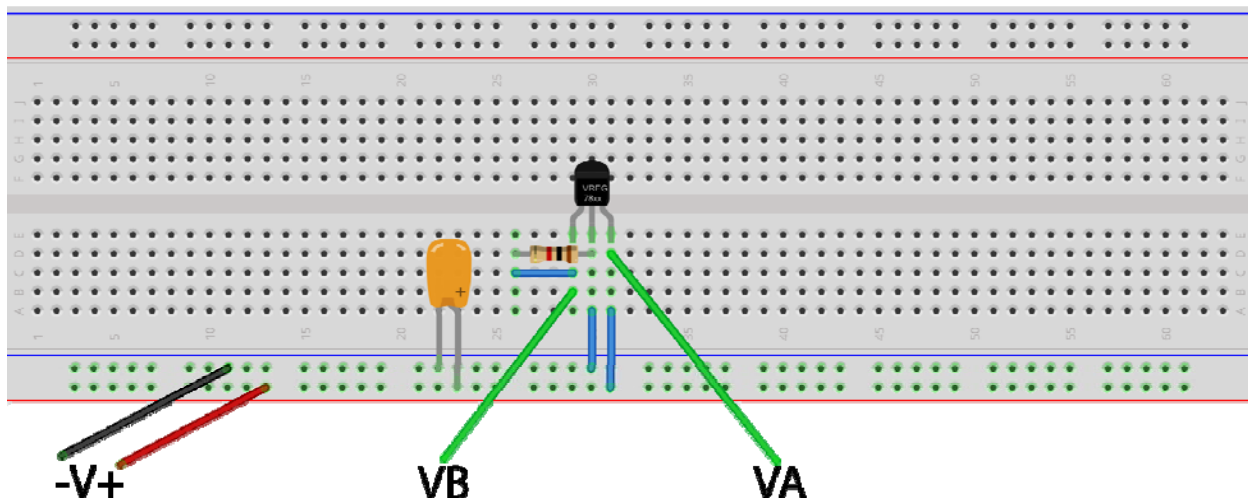

Figure 11. One possible layout for the voltage regulator prototype.

Node Voltage Measured		Differential Voltage Measured V_{AB}	Current Measured Through 1 kΩ	Differential Voltage Calculated V_A-V_B	Calculated Resistance
V_A	V_B				

2. Upload your Node Voltage measurements (V_A=V_{in}, V_B=V_{out}) to the class data input form that can be accessed via the following link:

Lab 2 Measurements

3. Create a simulation of the voltage regulator in Cadence. Acquire the simulated V_{in} versus V_{out} values for upload into Excel. Determine the input voltage for which the voltage regulator just turns on (i.e. output voltage becomes greater than zero) and the input voltage for which the voltage regulator falls out of regulation (i.e. output voltage becomes less than 5 V).

Out of lab:

1. Calculate the differential voltage V_{AB} from the measured node voltage using the equation:

$$V_{AB}=V_A-V_B \quad\quad\quad\quad (6)$$

Record these calculations in your table.

2. Using Ohms Law, V=IR, calculate the resistance for each of your voltage-current measurements at the output (i.e., R=V_B/I).

3. Using Excel, create a plot showing your simulated measurements V_A (V_{out}) versus V_B (V_{in}). Download the class's measurements and overlay the class's measurements on your simulated measurements plot. This plot will contain 30+ lines! Highlight the line which represents your simulated values and your specific measurements (i.e. by color and/or width of line). Copy this plot to your lab report and provide an appropriate caption.

Lab 2 report should contain:

1. Objective(s) of the lab (you should have written this in Lab 0).
2. Take the appropriate measurements and record the values in appropriate table.
3. Include a schematic showing where the measurements were made.
4. Create appropriate captions for each table and figure.
5. Perform the appropriate calculations and record the results in tables.
6. Provide a plot of the voltage regulator output voltage versus input voltage. This plot should contain your simulated values, your specific measured values, and the measured values obtained by the entire class. Your simulation and measurements should be highlighted in the plot.
7. Provide appropriate discussion of the measurements, figures, and tables (using cross-references) including equations used in calculations. Note, only complex equations (like the thermistor equation you typeset in Lab 1) need to be formatted on a separate line with an equation number. Your discussion should include comments on any similarities and/or differences between the simulated and the measured values and comments on any variation of plots across the class.

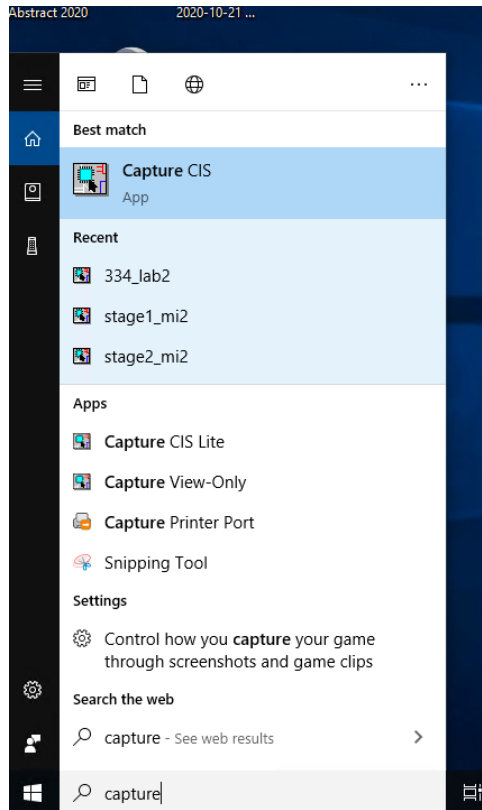**Schematic Capture and PSpice simulation Tutorial, visuals**

1) Create a folder in your user space and call it "EE102_Project"
   a. To create a folder, right click on the desktop. The following screens will appear
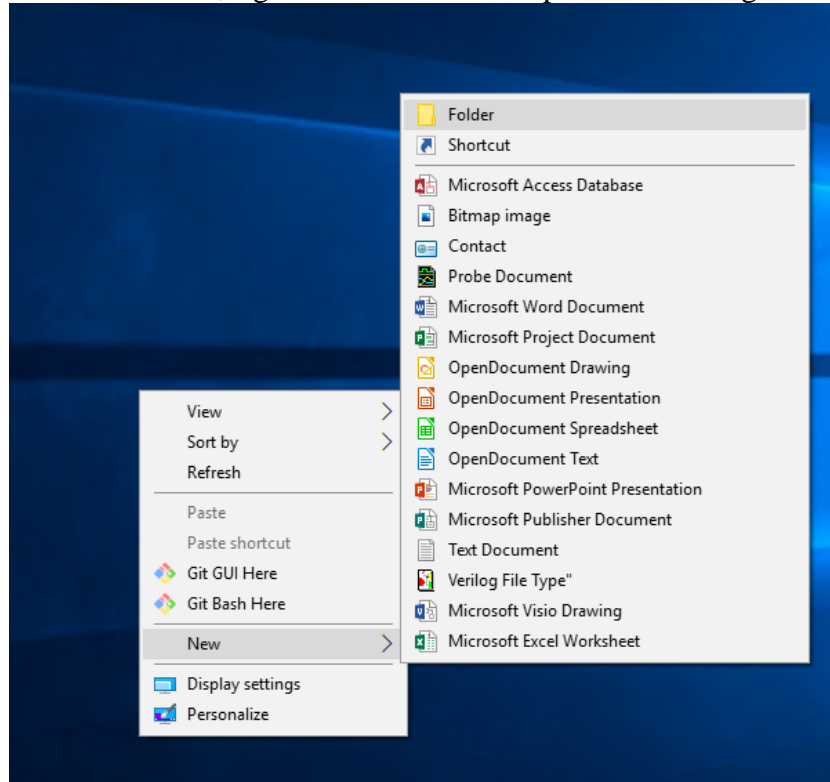
2) Download "elab.lib" and "EE102_LIBRARY" from Blackboard under the "lab instructions" tab. Move it to the folder you created.

3) Open up cadence on your computer. It's called "Capture CIS"

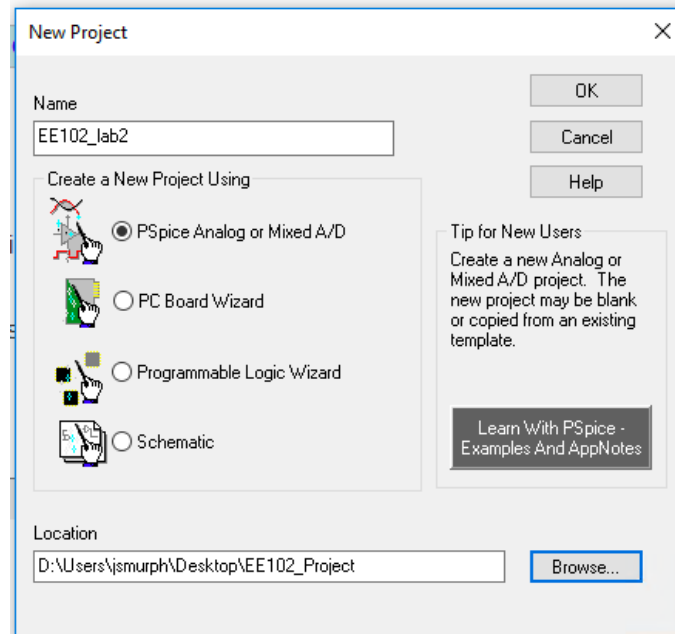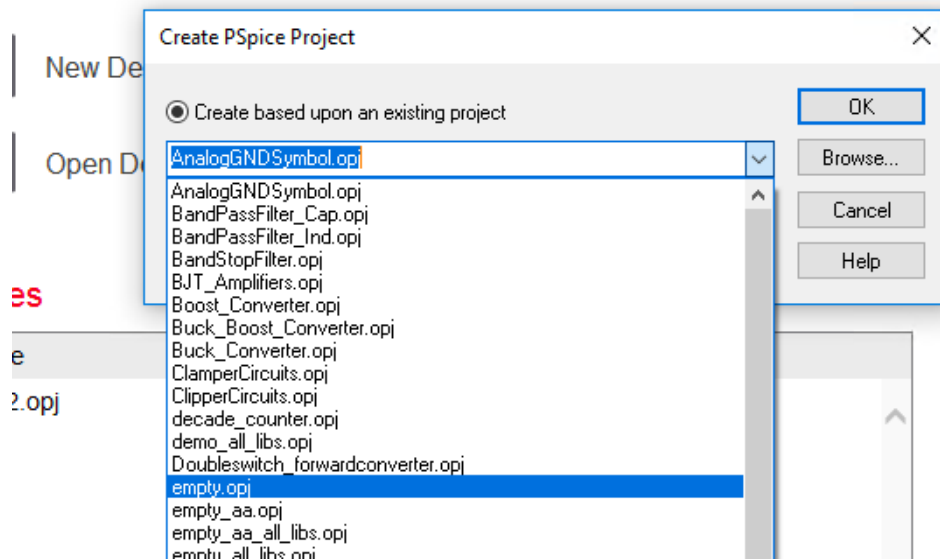4) Select "New Project" from the popup window

5) Fill out the "new project" window
   a. Name your file "EE102_lab2"
   b. Select "PSpice Analog or Mixed A/D"
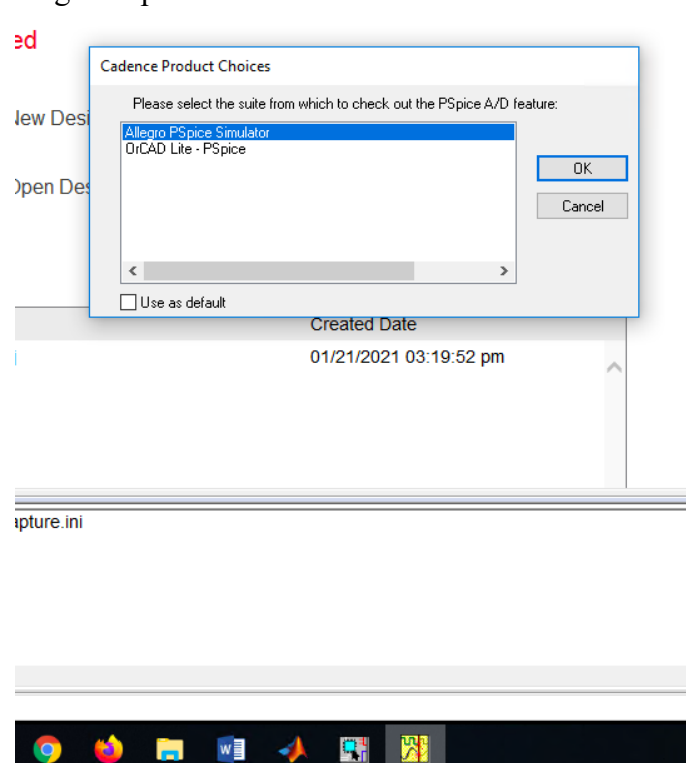   c. Save to the "EE102_project" folder you created
   d. Click "OK"

6) "Create PSpice project" window should open up
   a. Select "Create based upon an existing project"
   b. Select "empty.opj" from the dropdown bar
   c. Click "OK"

arted

New De

Create PSpice Project   ✕

◉ Create based upon an existing project   OK

Open D | AnalogGNDSymbol.opj   Browse...

AnalogGNDSymbol.opj
BandPassFilter_Cap.opj   Cancel
BandPassFilter_Ind.opj
BandStopFilter.opj   Help
BJT_Amplifiers.opj
es   Boost_Converter.opj
Buck_Boost_Converter.opj
e   Buck_Converter.opj
ClamperCircuits.opj
2.opj   ClipperCircuits.opj
decade_counter.opj
demo_all_libs.opj
Doubleswitch_forwardconverter.opj
empty.opj
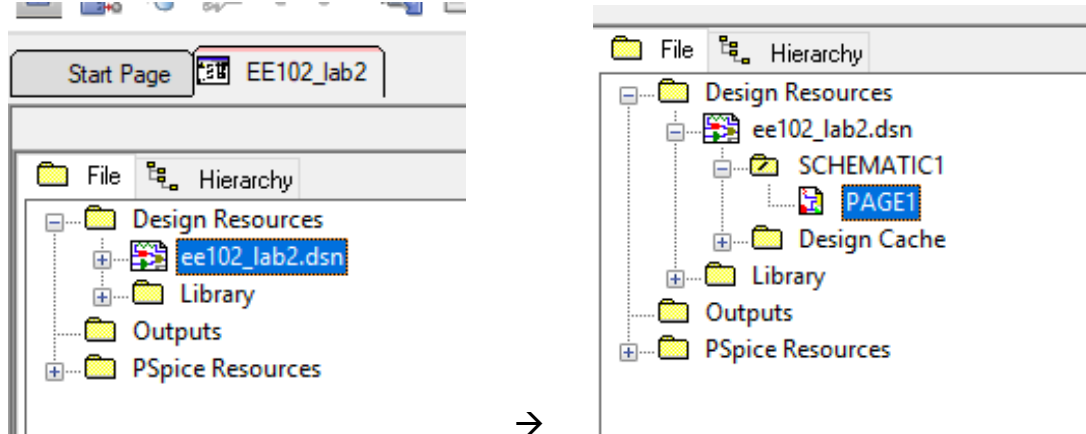empty_aa.opj
empty_aa_all_libs.opj
empty_all_libs.opj

7) A yellow box will open at the bottom of your screen, click on it so the "Cadence Product Choices" window pops up
   a. Select "Allegro PSpice simulator" and select "OK"

ed

Cadence Product Choices

lew Desi   Please select the suite from which to check out the PSpice A/D feature:

Allegro PSpice Simulator
OrCAD Lite - PSpice   OK

)pen Des   Cancel

☐ Use as default
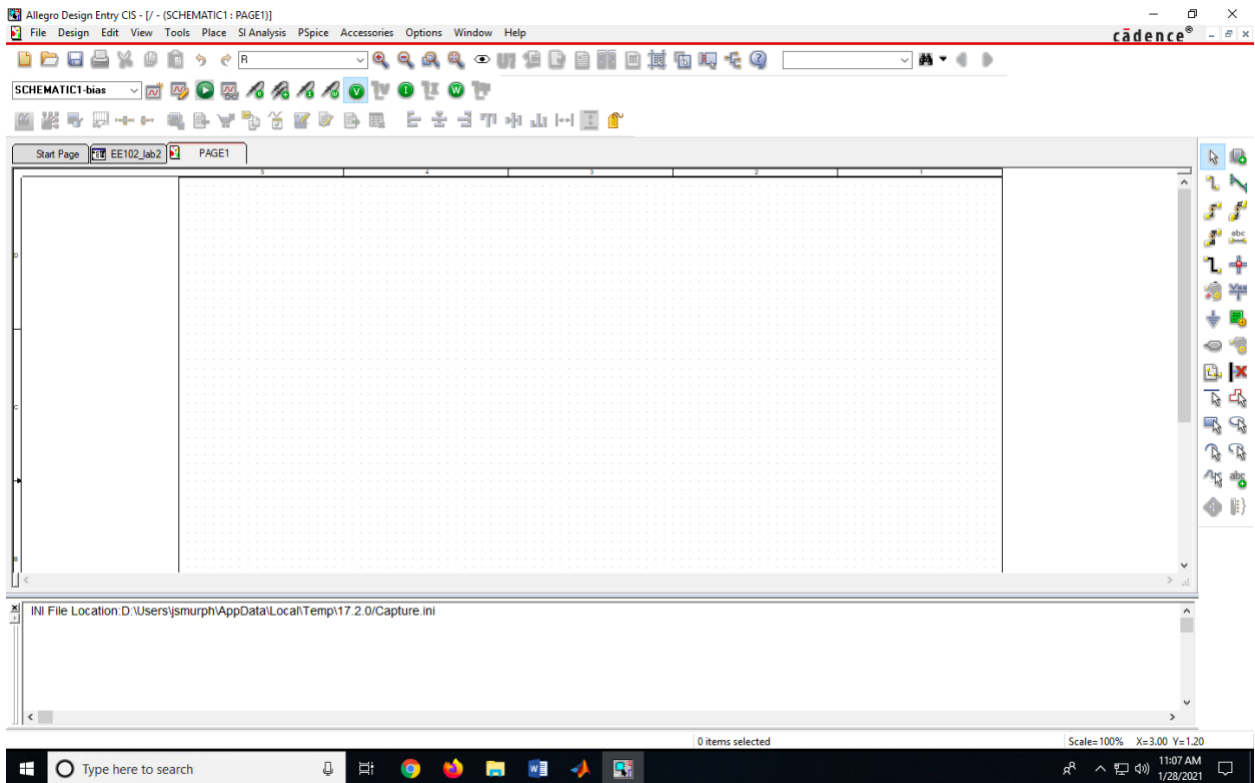
Created Date
01/21/2021 03:19:52 pm

apture.ini

8) Double click on the file you created, then double click up "SCHEMATIC", and open "PAGE1"

→

9) An empty schematic page should open up that looks like this:

10) To add a DC power supply select "Place", "PSpice Component", "Source", "Voltage Source", "DC". The figure on the left shows these buttons. The figure on the right shows what the component will look like. Place it somewhere on your schematic.
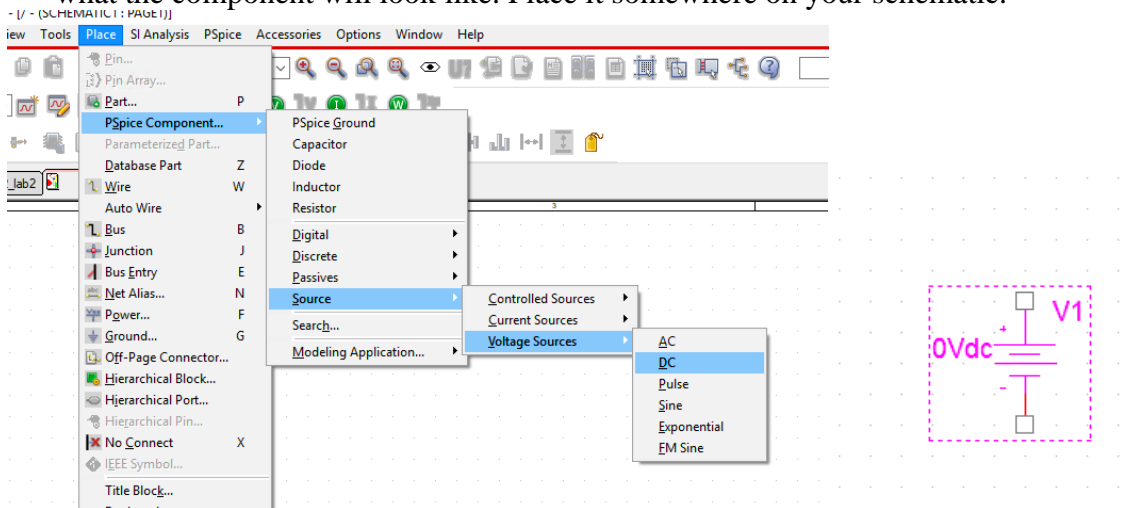
11) You will need to add a library for some of your other components. Select "Place", "Part" and the window shown on the right should appear:
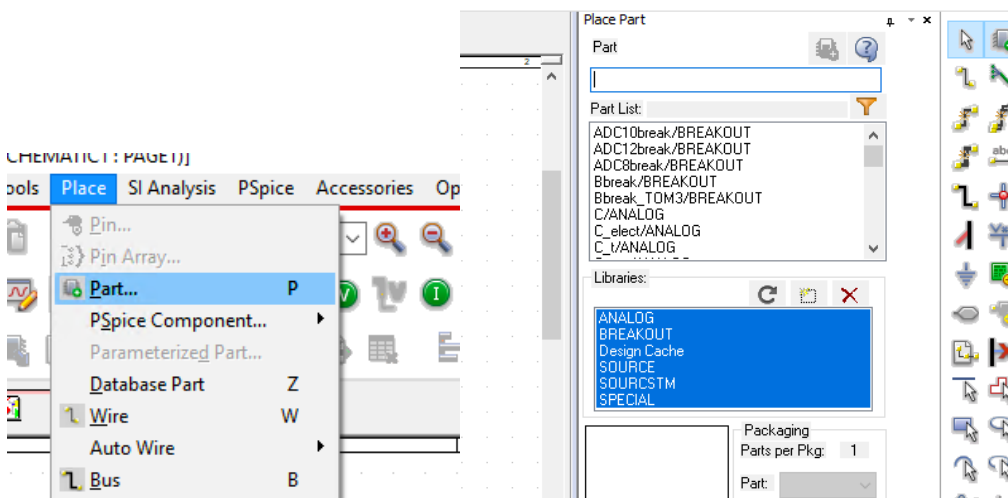
12) Select the add library icon as shown below
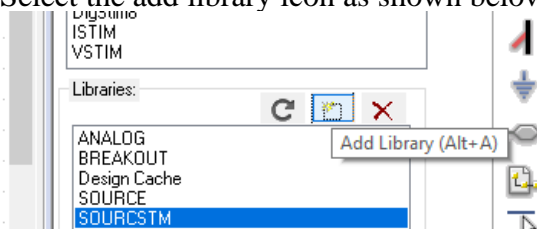
13) Find your folder from the window that pops up and select "EE102_LIBRARY.OLB". If this isn't in your folder, then it needs to be downloaded from the lab introduction in blackboard