

# Experiment Number 2

Revised: Fall 2018

## PLECS RC, RL, and RLC Simulations

### Preface:

- Experiment number 2 will be held in CLC room 105, 106, or 107. Your TA will let you know
- **Preliminary** exercises are to be done and submitted individually and turned in **at the beginning of class**
- **Lab Report** to be done in Word Doc according to the format uploaded on CANVAS and submitted by individual students at the **beginning of the next class**
- Be sure to include answers to  $Q1$ ,  $Q2$ , and  $Q3$  in the **Lab Report**
- Tech Memo **not** required for this lab
- Laboratory simulation exercises are to be done individually
- Review the guidelines for plagiarism to be aware of acceptable laboratory and classroom practices

### Objectives:

- To learn how to create RC, RL, RLC circuit models in PLECS
- To learn how to analyze source-free circuits for transient analysis
- Questions in red should be answered in the lab notebook

### References:

- EE 151 and EE 153 text. Cunningham and Stuller, *Circuit Analysis*, 2<sup>nd</sup> Ed. (Houghton Mifflin Company, Boston, 1995)

### Background:

The PLECS software package can be used to simulate circuits. The simulation can be used to calculate transient voltages across components and transient currents through components. To help gain understanding with transient circuits, this laboratory will include building RC, RL, and RLC circuits.

### Preliminary:

Work on separate sheet of 8.5x11 printer paper and turn in at the beginning of the laboratory session.

- Sketch a circuit diagram with a resistor  $R$  and a capacitor  $C$ . No sources are included in this circuit. Label the voltage across  $C$  as  $V_1$  and the current through the resistor  $R$  as  $I_1$ . (A-1) Derive the theoretical first-order differential equation for voltage  $V_1$  and the associated characteristic equation. (A-2) Calculate the natural response for the transient voltage  $V_1$  for  $t > 0$  seconds in terms of  $A$  (constant to be found),  $R_1$ , and  $C$ . (A-3) Determine the theoretical expression for the constant  $A$  if the initial value of the voltage  $V_1$  at  $t=0$  seconds is  $V_0$ .

- Sketch a circuit diagram with a resistor R and an inductor L. No sources are included in this circuit. Label the voltage across L as  $V_L$  and the current through the resistor R as  $I_1$ . (B-1) Derive the theoretical first-order differential equation for voltage  $V_L$  and the associated characteristic equation. (B-2) Calculate the natural response for the transient voltage  $V_L$  for  $t > 0$  seconds in terms of A (constant to be found), R, and L. (B-3) Determine the theoretical expression for the constant A if the initial value of the voltage  $V_L$  at  $t=0$  seconds is  $V_0$ .
- The natural response of first-order circuits is of the form  $A \exp\left(-\frac{t}{\tau}\right)$  where  $\tau$  is the time constant. For  $t=\tau$ , the natural response has decreased by  $\exp(-1) = 0.3679$ . (C-1) Give an expression in terms of R and C for the  $\tau$  in the prior RC circuit. (C-2) Give an expression in terms of R and L for the  $\tau$  in the prior RL circuit.
- Sketch a circuit diagram with three parallel elements: a resistor R, a capacitor C, and an inductor L. No sources are included in the circuit. Label the voltage across C as  $V_C$  and the current through the inductor L as  $I_L$ . (D-1) Derive the theoretical second-order differential equation for voltage  $V_C$  and the associated characteristic equation. (D-2) For this case, the damping coefficient is  $\zeta = \left(\frac{1}{2R}\right) \sqrt{\frac{L}{C}}$ . Show the condition on R in terms of L and C for the circuit natural response to be under damped. (D-3) Sketch the under damped response.

## Equipment:

PLECS software package, an add-on to MATLAB Simulink program in the CLC

## Experimental Procedure:

(Record specifics in the Laboratory Notebook)

1. Start MATLAB
2. Open a new Simulink window by clicking on the Simulink button. The simulink button is in the top left of the MATLAB window. The Simulink icon is highlighted in Figure 1 shown below.

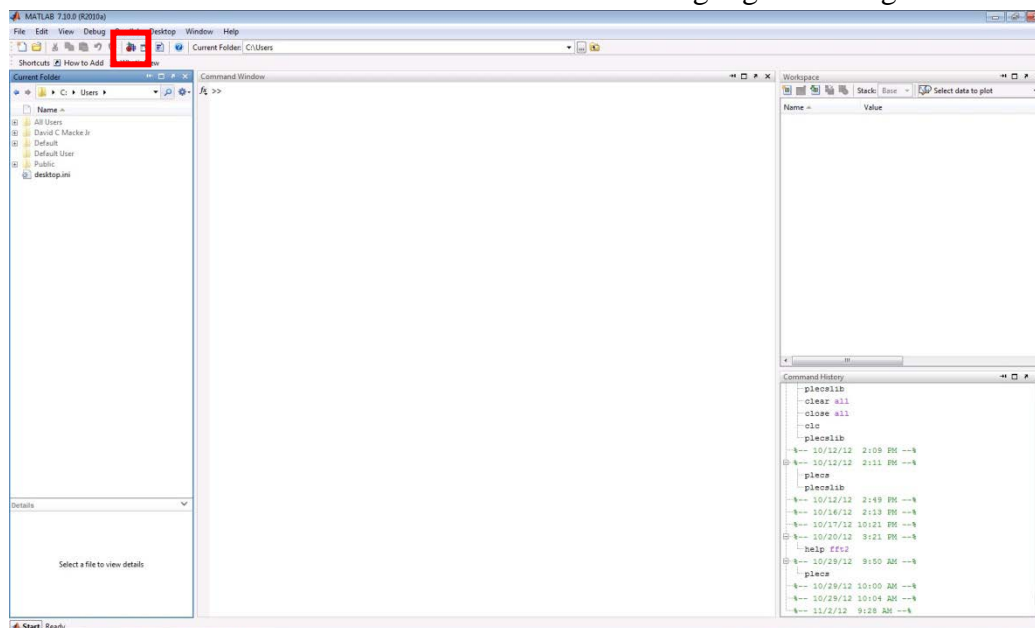


Figure 1:MATLAB Start Page

3. The window that appears is called the Simulink Library Browser. This window contains every component that can be placed into a simulation. The Library Browser window can be searched in two ways, with the search box at the top of the window or with the explorer window to the left of the window. The search bar, highlighted in Red, and the Explorer Window, highlighted in Green are shown in the image below. Create a New Model, click on the New Model Button. The New Model button is located in the top left of the Simulink Library Browser window. The New Model button, highlighted in Blue in Figure 2, is shown below.

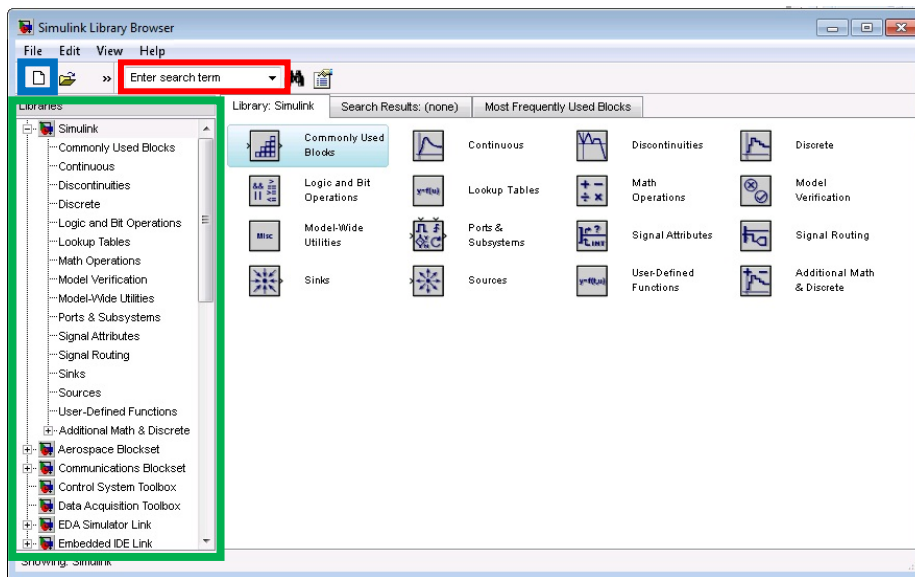


Figure 2: Simulink Library Browser

4. After the New Model window opens up, click on the Simulink Library Browser window. In the Library Browser, look at the Libraries window on the left. Scroll down until the PLECS Library appears. Click on the PLECS library, Drag and Drop the Circuit component into the New Model Window. In the Model window, double click on the PLECS Circuit name “Circuit” and change the name to “Part 1”.

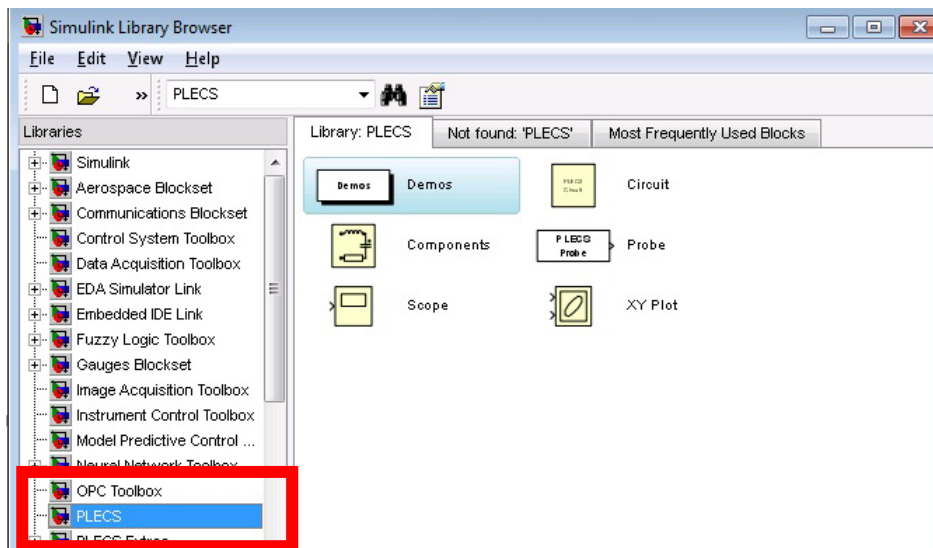


Figure 3: Simulink Library Browser - PLECS Library

## Part #1

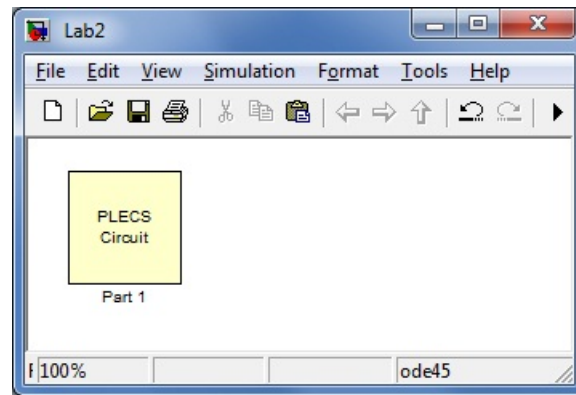


Figure 4: PLECS Circuit 1 in Simulink Model

5. Double click on the PLECS Circuit component. A new window with a tan background will pop up, the PLECS window. In this window hit Ctrl+L, that shortcut will bring up the Library Browser. In the library browse to the Electrical->Sources section. Drag and Drop a DC Voltage Source into the PLECS window. Next browse to the Electrical->Meters section, drag and drop one Voltmeter into the PLECS window. Next browse to the Electrical->Passive Components section, Drag and Drop one Resistor and one Capacitor into the PLECS window. To place the last component, browse to the System section, Drag and Drop a Scope into the PLECS window.

### Component Overview:

- DC Voltage Source: This component outputs a constant DC voltage. For this laboratory the DC voltage source will be set to 5 Volts. This DC source will charge the capacitor over the duration of the time constant.
- Scope: This component will be used to display the output from the voltmeters. For this part of the laboratory the scope will need two plots. To change the number of plots, double click on the scope block, then select File->Scope Parameters. In the Scope Parameters window, check to make sure the Number of Plots is One. Before clicking ok, verify the Limit Samples box is not checked.
- Voltmeter: This component functions like a digital multi meter in laboratory. The voltmeter is attached across one or many components in the circuit and outputs the measured voltage. The output from the meter is accessed through the green arrow on the component.
- Resistors: This component simulates the behavior of a Resistor in a circuit. For this part of the laboratory, you are using one resistor. Change the resistor name to 470 Ohms. Next change the resistance of the resistor to match the name.
- Capacitor: This component simulates the behavior of a Capacitor in a circuit. For this part of the laboratory, you will be using one capacitor. Change the capacitance from the default value to 10 $\mu$ F. Also verify that the initial charge on the capacitor is 0V. To change the capacitance, double click the icon and enter the new value.

6. After changing all the parameters of the components, connect the components to form the circuit shown below. To connect two components, click and drag from the dot at the end of one component to the end of the other component.

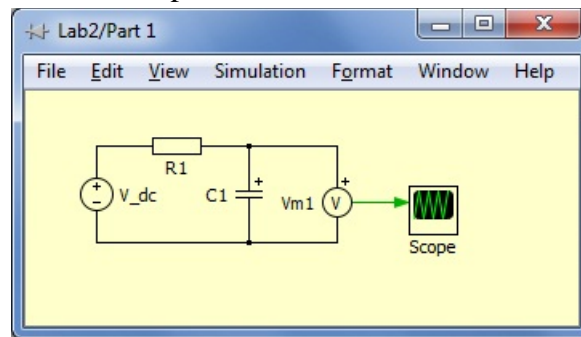


Figure 5: PLECS Circuit 1

7. Now that the circuit has been constructed, the simulation parameters need to be set. To set the simulation parameters, click on the PLECS window and browse to Simulation->Simulink Parameters, or hit Ctrl+E.

Simulation Parameters:

Start Time: 0 Seconds

End Time:  $6 * \text{Time Constant}$

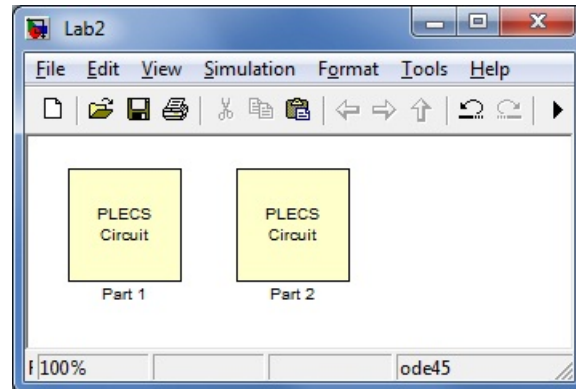
Max Step Size:  $(6 * \text{Time Constant}) / 1000$

8. Run the simulation, browse to Simulation->Start or hit Ctrl+T.

The Lab Report should list the steps, components used with the corresponding values, and include the PLECS schematic and Scope plots.

***Q1: Calculate the time constant with the given values for R and C using the theoretical expressions derived for the preliminary and compare it to the simulated value. Do they match?***

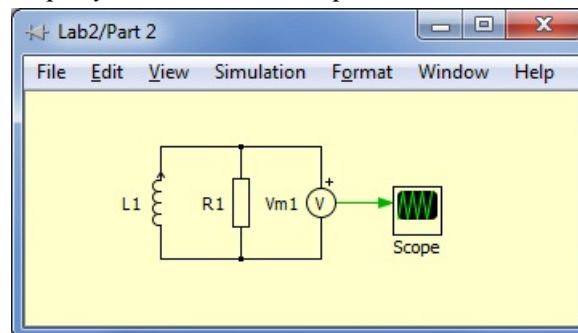
## Part #2



**Figure 6: PLECS Circuit 1 and 2 in Simulink Model**

9. For the next part of the laboratory, go back to the Model in Simulink, place a new PLECS Circuit in the Model. Rename the New PLECS block Part 2. Open the new PLECS block, in this PLECS block create a source free circuit. The circuit for part 2 needs one resistor, one inductor, and a voltage probe. The resistors should be set to  $330\Omega$  and the inductor set to  $0.1\text{H}$ . Change the initial condition on the inductor to  $1\text{mA}$ . Connect the voltage probe across the resistor. To place the voltage probe, browse to Electrical->Meters, and use the Voltmeter.

*Note, when you place the scope, you will need One plot.*



**Figure 7: PLECS Circuit 2**

The Lab Report should list all steps; give a table of the simulation values for all components and the corresponding initial conditions. Also include a printout of the PLECS schematic and results generated after simulation.

***Q2: Calculate the time constant with the given values for R and L using the theoretical expressions derived for the preliminary and compare it to the simulated value. Do they match?***

## Part #3

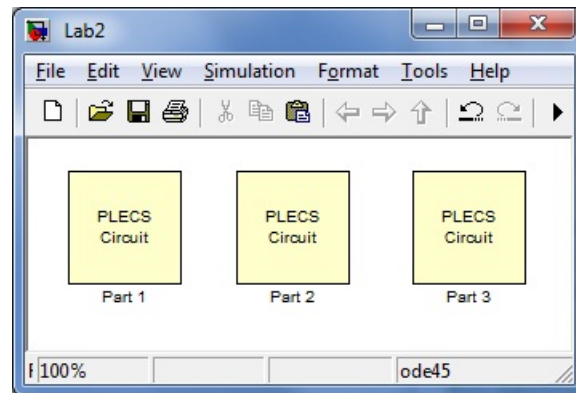


Figure 8: PLECS Circuit 1-3 on Simulink Model

10. Create the circuit shown in figure 9 in a new PLECS block. The circuit involves one Resistor, Capacitor, and Inductor in parallel. Set the resistor to  $330\Omega$ , the capacitor to  $10\mu\text{F}$ , and the inductor to  $0.1\text{H}$ . Inside the PLECS block, place a voltage probe across the capacitor. Set the initial value of the inductor to  $0\text{A}$  and the initial voltage on the capacitor to  $10\text{V}$ .

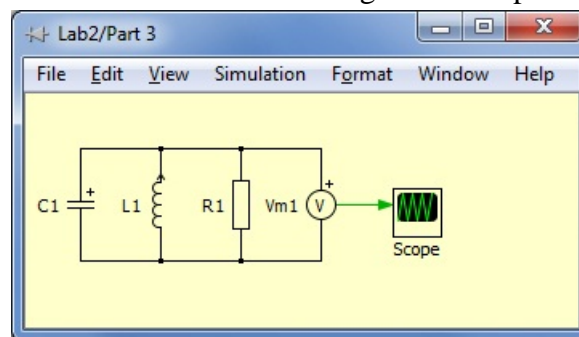
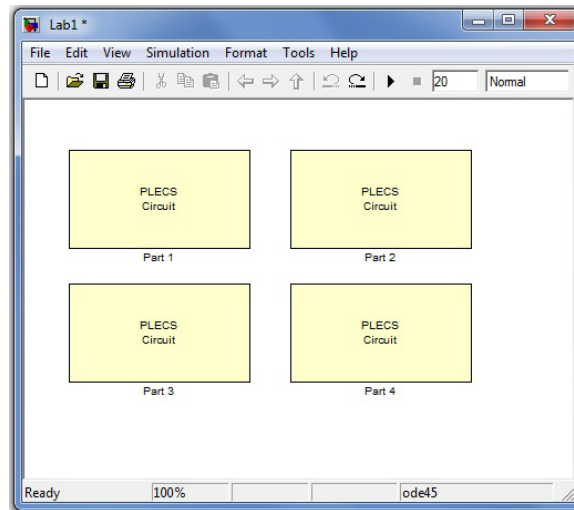


Figure 9: PLECS Circuit 3

The Lab Report should list all steps; give a table of the simulation values for all components and the corresponding initial conditions. Also include a printout of the PLECS schematic and results generated after simulation.

**Q3:** Calculate the damping coefficient with the given values for  $R$ ,  $C$ , and  $L$  using the theoretical expressions in the preliminary. Is the circuit underdamped or overdamped? Does the transient response behave as expected?

## Part #4



**Figure 10: PLECS Circuit 1-4 on Simulink Model**

11. In the main simulink file, copy and paste the PLECS block from part 3. Rename the copy Part 4. Within the part 4 PLECS block modify the RLC circuit model by changing the value of R (keep the same values of C and L). If the transient response in part 3 were overdamped, then use an R value giving an underdamped response. If the transient response in part 3 were underdamped, then use an R value giving an overdamped response. After changing the R value, simulate the circuit. Save the schematic and the transient response.

The Lab Report should list the steps, give the new value for R and include a printout of the PSPICE schematic and the transient response.