# Experiment Number 3

Revised: Spring 2013

## Using PLECS to Simulate Diode Circuits

## Preface:

- Preliminary exercises are to be done and submitted individually
- Laboratory simulation exercises are to be done individually
- Laboratory hardware exercises are to be done in pairs (Not applicable for #3)
- Work done in notebooks will be submitted individually (Not applicable for #3)
- Notebook will be submitted immediately following the lab
- For this simulation based laboratory, students must submit the individual work done in their laboratory notebooks.
- The laboratory notebooks must include all settings, steps and observations in the exercises. All statements must be in complete sentences and all tables and figures must have captions.
- Review the guidelines for plagiarism to be aware of acceptable laboratory and classroom practices

## Objectives:

- Use PLECS tools to simulate the characteristics of the diode
- Use PLECS to model and simulate a half-wave rectifier
- Use PLECS to model and simulate a bridge rectifier
- While the laboratory notebook will not be turned in and graded for this lab, the specific content indicated in red and the questions should be recorded to add in completion of the Technical Memorandum.

## References:

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

## Background:

PLECS can be used to analyze the performance of nonlinear components. A diode is a two-terminal device that performs differently under forward and reverse bias. Rectifiers are one of the most common uses of diodes. They are used to convert AC waveforms to DC waveforms. This laboratory will simulate the operation of a half-wave rectifier and a bridge rectifier. The half-wave rectifier uses one diode and rectifies only the positive half of the wave. The bridge rectifier, or full-bridge rectifier, uses four diodes and rectifies both negative and positive halves of the wave. However, the half-wave rectifier is used when applicable because one diode uses less power and is more cost efficient than four.

## Preliminary:

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

- Consider a silicon(Si) pn step-junction at 300K. Let the p side be doped so that $N_A^- = 10^{16}$ $cm^{-3}$ and the n side doped so that $N_D^+ = 10^{16}cm^{-3}$. Note that ni(Si) = $1.5 \times 10^{10}$ $cm^{-3}$. Calculate the contact potential $V_0$ for this structure.
- For the diode circuit shown in figure 5, let the diode have an abrupt turn-on voltage of 0.7V and a reverse saturation current of ~0A and let the resistor be R=1.0KΩ. Show the load-line equation for this circuit with variables V and I. Calculate the voltage across the resistor using the graphical load-line approach for Vs = -10, -6, -2, +2, +6, +10V (show each calculation explicitly). Put the results in a table with columns for $V_S$ and $V_R$.

## 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.
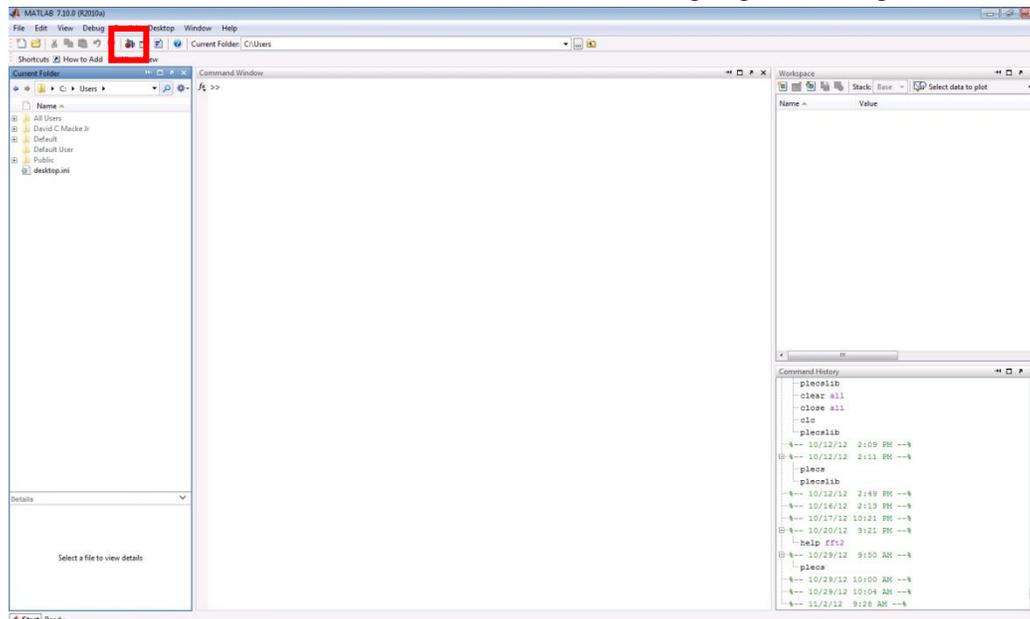


**Figure 1:MATLAB Start Page**

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.
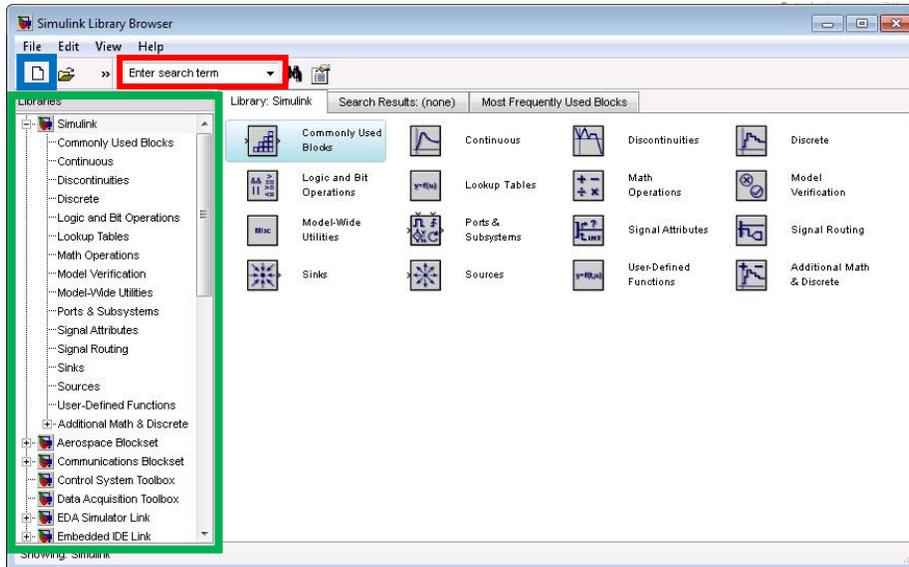
**Figure 2: Simulink Library Browser**

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".
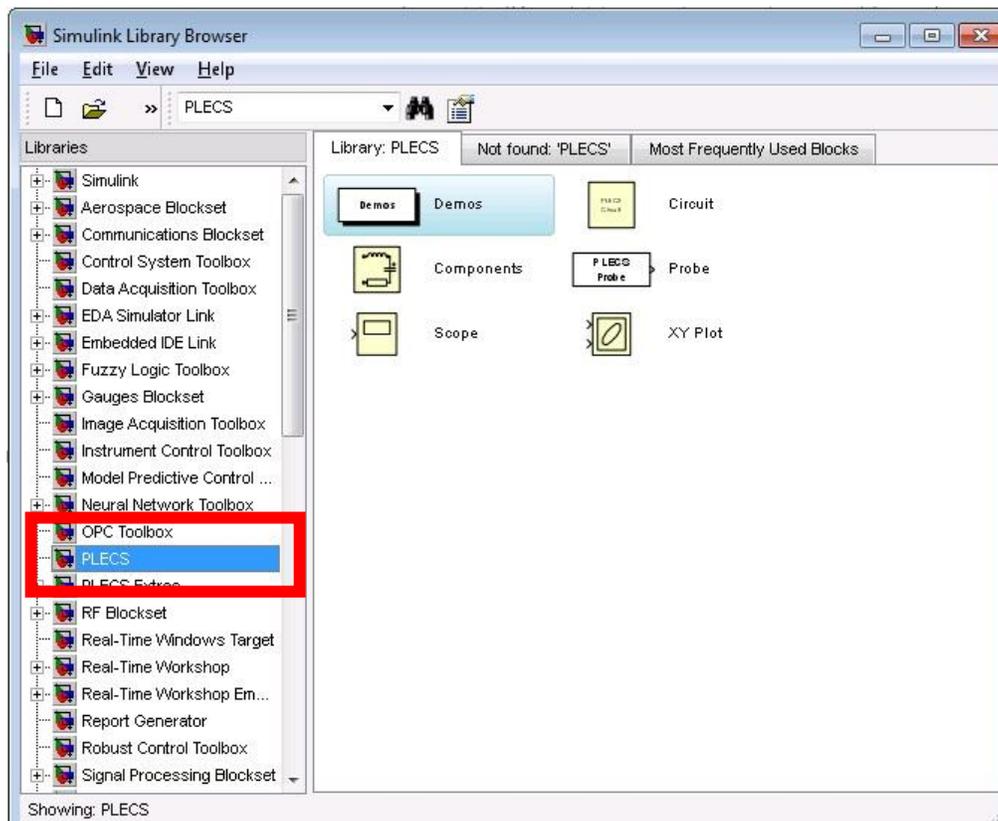


**Figure 3: Simulink Library Browser - PLECS Library**
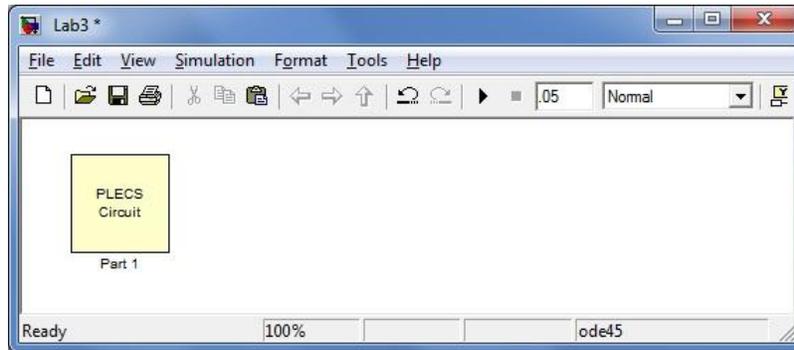
## Part #1



**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 <u>Voltage Source (Controlled)</u> into the PLECS window. Next browse to the Electrical->Meters section, drag and drop two <u>Voltmeters</u> into the PLECS window. Next browse to the Electrical->Power Semiconductors section, Drag and Drop one <u>Diode</u> into the PLECS window. Then browse to the Control->Sources section and place a <u>Ramp</u> function. To place the last component, browse to the System section, Drag and Drop a <u>Scope</u> into the PLECS window.

Component Overview:
- Ramp: This component is a control component. It is used to supply control signals to a variety of components. For this lab, the ramp will have a slope of 1/second, an initial output of -10, and a final output of 10. To change the parameters, double click on the component block in the PLECS window and change the necessary values.
- Voltage Source (Controlled): This component outputs a controlled voltage. The control signal is fed through the green input on the component. For this lab the control signal will be from the Ramp component. The ramp will vary from -10 volts to +10 volts over 20 seconds. By varying the voltage supplied by the source, the circuit will be DC swept.
- Scope: This component will be used to display the output from the voltmeters. For this part of the lab 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 Two. Before clicking ok, verify the Limit Samples box is not checked.
- Voltmeter: This component functions like a digital multi meter in lab. 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. For this lab the output of the voltmeter will go into the Scope.
- Diode: This component simulates the behavior of a Diode in a circuit. For this part of the lab, you are using one diode. Change the parameters of the diode to the following. The forward

voltage, Vf, should be set to 0.7V and the On-resistance, Ron, should be set to 0.001Ω. To change the parameters of the component, double click the logo, change the two parameters in the window that pops up, and then click ok.

- Resistor: This component simulates the behavior of a resistor in a circuit. For this part of the lab, set the resistance to 1Ω.

6. After changing 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.
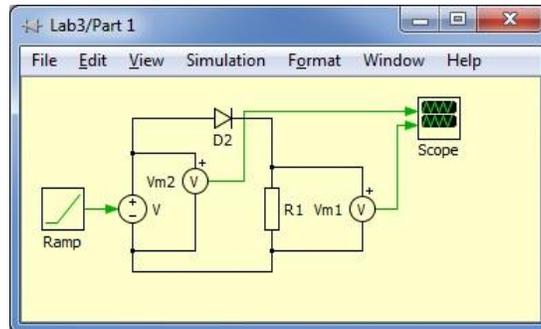


**Figure 5:PLECS Circuit 1**

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: 20 Seconds
Max Step Size: 1e-4
8. Run the simulation, browse to Simulation->Start or hit Ctrl+T.

(Your laboratory notebook should list the steps, components used with the corresponding values, and include a Sketch of the PLECS schematic and Scope plots.)

*Q1: What is the current through the diode at 1.0V? At 0.02V?*
*Q2: At what voltage does the current begin to increase very quickly?*
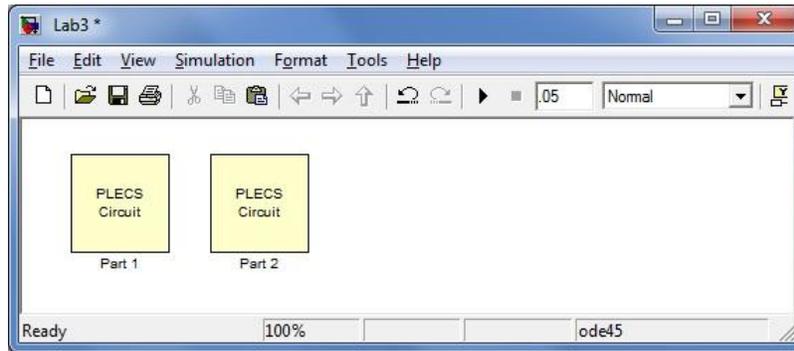
# Part #2



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

9. For the next part of the lab, 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 the same circuit as part 1 with two changes, an additional resistor and change the Voltage Source(Controlled) to an AC Source. The resistor should be set to 1.0KΩ. Change the AC Source Amplitude to 10 and the frequency to 2*pi*60. Connect the voltage probe across the resistor instead of across the diode. Simulate the model and record the simulated waveform.
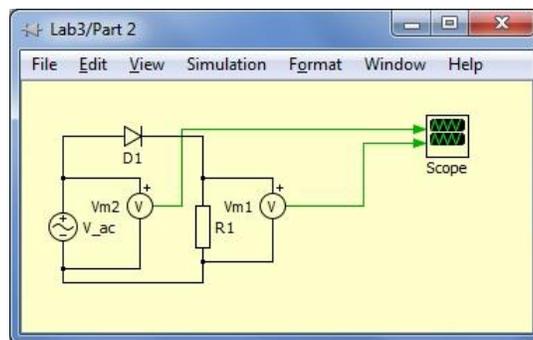


**Figure 7:PLECS Circuit 2**

(Your laboratory notebook 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.)

Simulation Parameters:
Start Time: 0 Seconds
End Time: 0.05 Seconds
Max Step Size: 1e-4

*Q3: Explain why the output signal is different from the input signal.*
*Q4: Sketch the output signal that would result if the diode direction were reversed.*
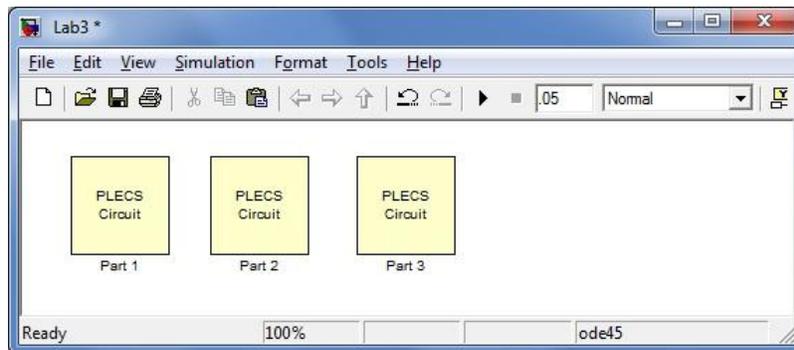
## Part #3



**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 and Capacitor in parallel. Set the resistor to 1.0KΩ and the capacitor to 30μF. Inside the PLECS block, place a voltage probe across the capacitor. Set the initial value of the on the capacitor to 0V. Simulate the model and record the simulated waveform.
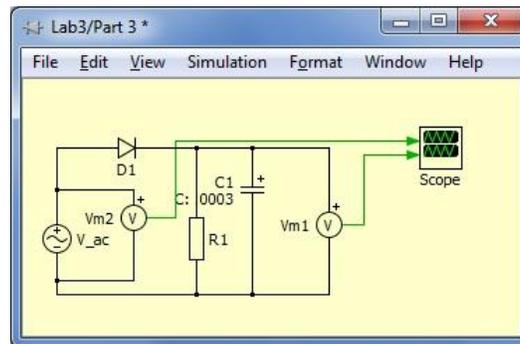


**Figure 9:PLECS Circuit 3**

(Your laboratory notebook 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.)

*Q5: By comparing these results to those in step 2, what effect did the capacitor have on the output signal.*
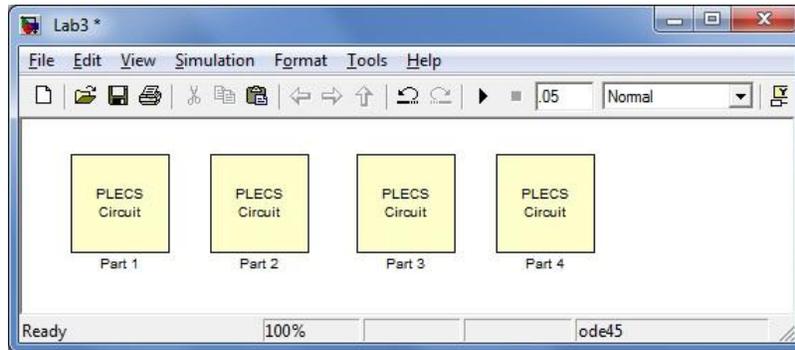
# Part #4



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

11. Create the circuit shown in figure 11 in a new PLECS block. The circuit for part 4 is a full-wave rectifier. Each of the diodes will need to have a Forward voltage of 0.7V and an On-resistance of 0.1Ω. After building the full-wave rectifier, add a 1Ω load resistor across the output of the full-wave rectifier.  Place a volt meter across the AC source and the load resistor. Simulate the model and record the simulated waveform.
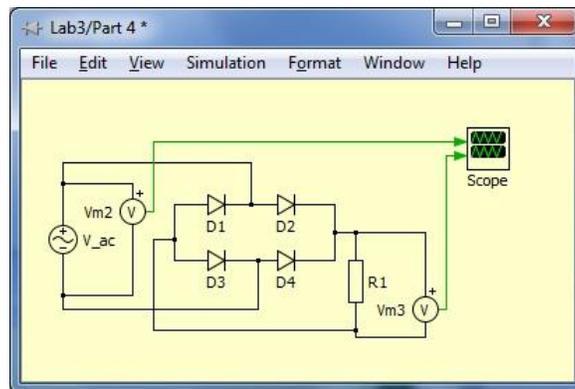


**Figure 11:PLECS Circuit 4**

(Your laboratory notebook should list the steps and include a printout of the PSPICE schematic and the transient response.)

*Q6: By comparing these results to those in step 2, what effect did the added diodes have on the output signal?*
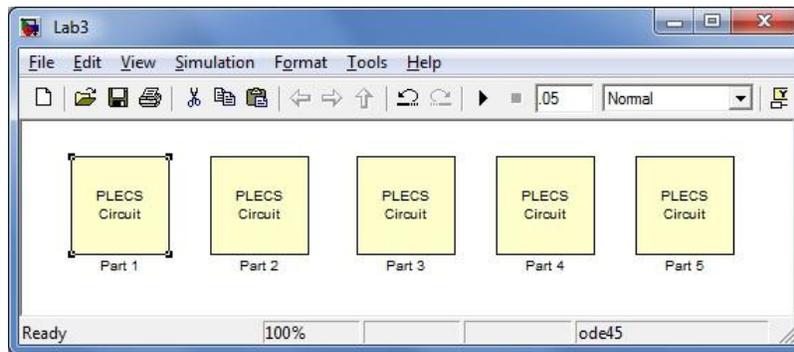
## Part #5



**Figure 12:PLECS Circuit 1-5 on Simulink Model**

11. Create the same circuit from Part #4. Add a capacitor in parallel with the load resistor. Set the capacitor to a value of 0.05F. Set the initial voltage to 0V. Simulate the model and record the simulated waveform.
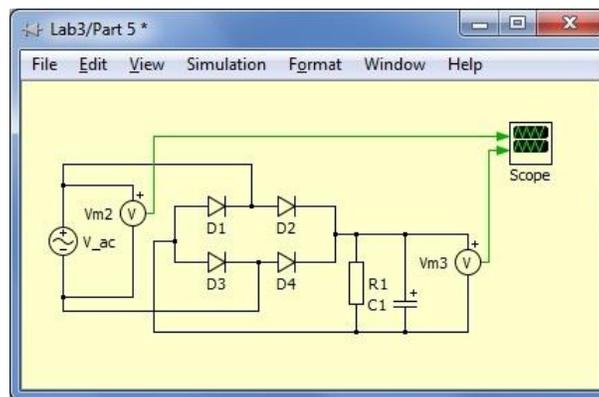


**Figure 13:PLECS Circuit 5**

(Your laboratory notebook should list the steps and include a printout of the PSPICE schematic and the transient response.)

*Q7: Compare these results to those seen in Part #3.*

## Technical Memorandum:
A Technical Memorandum is required for this laboratory experiment.
   *Memorandum Discussion:*
(1) What is the current through the diode of figure 5 at 1.0V? Compare the value to the theoretical values.
(2) Compare the output of figure 5 and figure 11. What were the advantages and disadvantages of each circuit.
(3) When building a rectifier circuit, what are the two primary forces that compete when designing the rectifier?
   Appendix:
Appendix 1: Sketch the output of Figure 7 if the diode were reversed
Appendix 2 Sketch the output of Figure 7 if a capacitor were add in parallel with the resistor