EGB348 - Electronics

Studio 1 – Diodes

In this Studio, you will simulate many of the diode circuits studied in the Lecture and Tutorial.   This Studio also introduces LTSpice, and no prior knowledge of LTSpice is assumed.

Note that most screenshots in this Studio were taken with LTSpice IV.   It is expected that for the purposes of the EGB348 Studios, the differences with version XVII are minimal and will not affect the learning outcomes.

1. Pre Lab Research

Q 1.1. Find datasheets for the diodes used in this Studio, including the 1N4007, 1N4148 and 1N914 diodes, and 1N750 and TFZ10B Zener diodes.   What are some of their characteristics?

Q 1.2. What does the acronym SPICE stand for (in the context of circuit simulation)?   When was SPICE developed?

2. Library and File Setup

2.1 Files for this Studio

Download all files under **Studio 1/Studio 1 files**, and save to your working directory.

2.2 DVIEW Library

The DVIEW library is used to display digital waveforms with vertical separation, so it is easy to see how waveforms change relative to each other.   To make use of the DVIEW components, download **DVIEW.lib**, **DVIEW5.asy** and **DVIEW0.asy** from Blackboard and copy into your working directory.

3. <u>Getting Started with LTSpice</u>

This Studio uses LTSpice, originally developed by Linear Technology. The software can be downloaded from (http://www.analog.com/en/design-center/design-tools-and-calculators.html) and installed on your own laptop or home computer.

If using Windows, the software may be started by clicking the "Start" button, typing "LTSpice" in the search box, and then clicking the LTSpice XVII application. After LTSpice opens, click the **New Schematic** button to open a new, blank schematic.   Figure 3.1 shows the locations of some of the frequently used buttons in LTSpice.
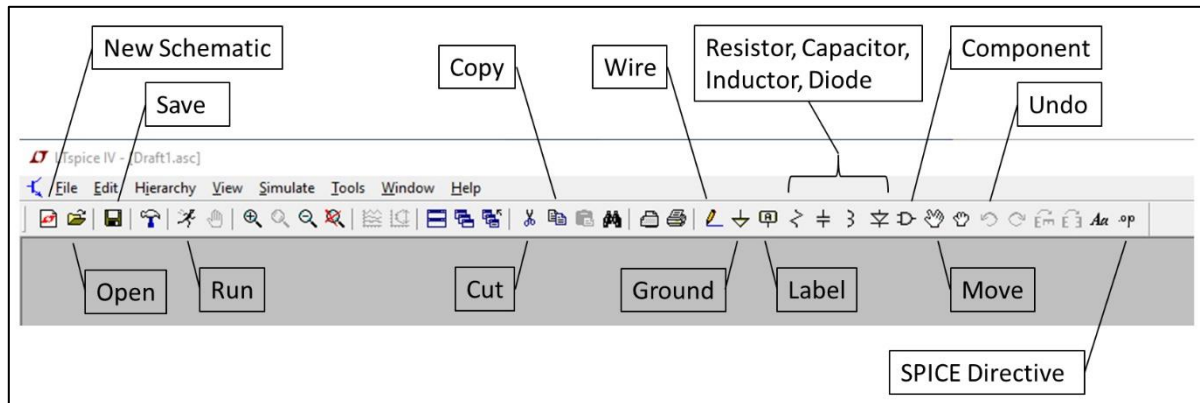

Figure 3.1: Portion of LTSpice window after selecting "New Schematic", showing toolbar and menus.

A circuit can now be constructed in the schematic window, by placing components and adding connecting wires.   Once the circuit schematic has been drawn, various types of simulations can be run, and the results displayed.

The buttons labelled in Figure 3.1 are discussed as follows:

- **New Schematic** – Open a new blank schematic.
- **Open** – Open a previously saved schematic.
- **Save** – Save the current schematic. Schematic files are saved with extension **.asc**.   They are actually text files and readable in a text editor.
- **Run** – Run the current simulation command.
- **Cut** – Delete one or more components. To delete a single component, left click on it. To delete multiple components, left click and drag a box around the components you wish to delete.   To exit from deleting components, right click or press ESC.
- **Copy** –   Copy one or more components. To copy a single component, left click on it. To copy multiple components, left click and drag a box around the components you wish to copy. To exit from copying components, right click or press ESC.
- **Wire** – Draw a wire to connect components.   To start drawing a wire, left click.   To set the end point of a section of wire, and turn a corner, left click.   To finish drawing a wire, right click.   To exit from drawing wires, press ESC.   Note that wires that cross are not connected by default - to make a connection between two wires that cross, double click on the wire being crossed. Also, LTSpice has a time saving feature, in that it is possible to draw a wire straight through a component, such as a resistor, and LTSpice will by default just connect the wire on either side.

- **Label** – Add a label to a wire in the circuit.   Labels can be useful to assign meaningful names to nodes in the circuit, which is helpful when choosing a value out of a list of node voltages or currents to graph. Two labels of the same name are considered to be electrically connected.   In addition, labels can be used to define input/ouput ports of a subcircuit (see Section 5).   To exit from placing labels, press ESC.
- **Component** – Allows a component to be chosen and placed in the circuit.   When this button is clicked, the window shown in Figure 3.2 appears. It is possible to choose between the default symbol location, and the current working directory, as places to look for component files.   If the 74HCT and DVIEW libraries were set up as discussed in Section 2.1, then the relevant components can be found in the locations indicated in Figure 3.2. Otherwise, they will be found in the current working directory. In addition to the **Component** button, there are also toolbar buttons that allow resistors, capacitors, inductors, diodes and ground to be quickly selected and placed.
- **Move** – Move one or more components. To move a single component, left click on it. To move multiple components, left click and drag a box around the components you wish to move. To exit from moving components, right click or press ESC.
- **Undo** – Undoes the last edit.
- **SPICE Directive** – Allows a SPICE directive to be placed on the schematic.

Some other LTSpice features useful for getting started are as follows:

- **Rotate and Flip** – **Ctrl-R** and **Ctrl-E** will respectively rotate and flip the selected component.
- **Edit Simulation Command** – Select **Simulation/Edit Simulation Cmd** to choose the type of simulation and parameters.
- **Drawing tools** - Select **Edit/Draw** to draw various shapes such as lines, rectangles and circles on your schematic.
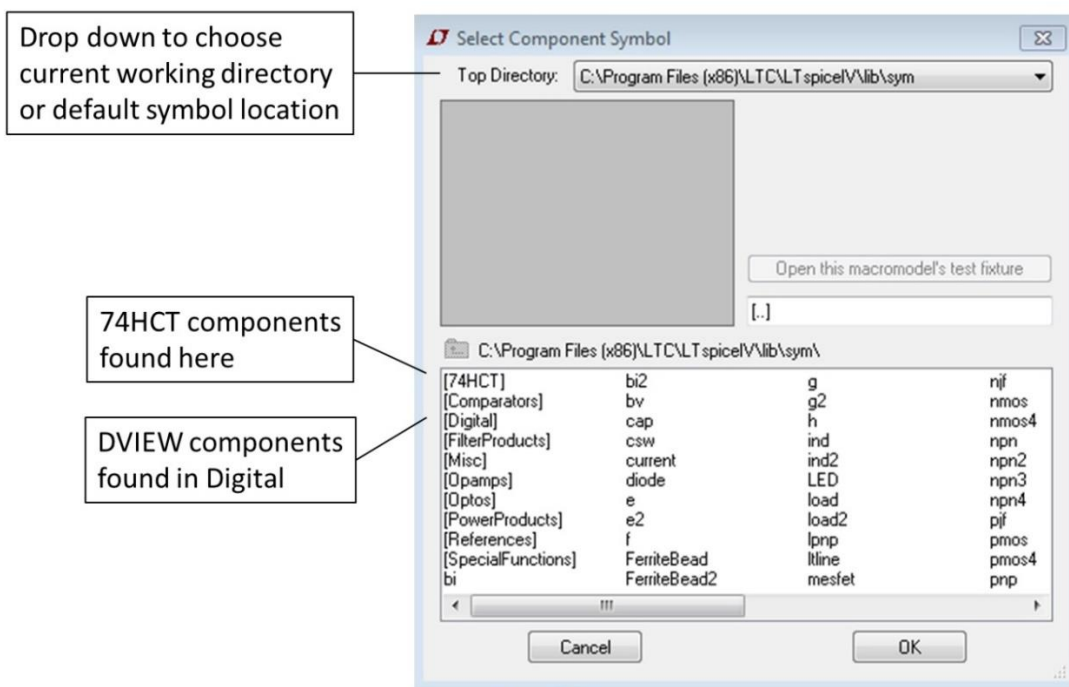

Figure 3.2: Window that pops up when the **Component** button is pressed.

Ex 3.1. Practice Exercise

Experiment with placing a few components in a schematic – for example resistors, capacitors, inductors and ground.   Experiment with flipping and rotating components, and drawing wires to connect them.

4. Diode Circuits

In this section, you will simulate some of the diode circuits in Tutorial 2 and the lecture notes.

4.1 Simple Diode Circuit

Here, you will draw the schematic and simulate the circuit from Tutorial 1, Q1.   Open LTSpice, and draw the schematic shown in Figure 3.1   You may use the file **simple_diode.asc** as a starting point.

The resistor, diode and ground components can be quickly selected from buttons in the toolbar, as shown in Figure 3.2. Right-clicking on the Voltage and Resistor components will allow their values to be changed. The **DC Value** field of the Voltage source should be set to 9V.
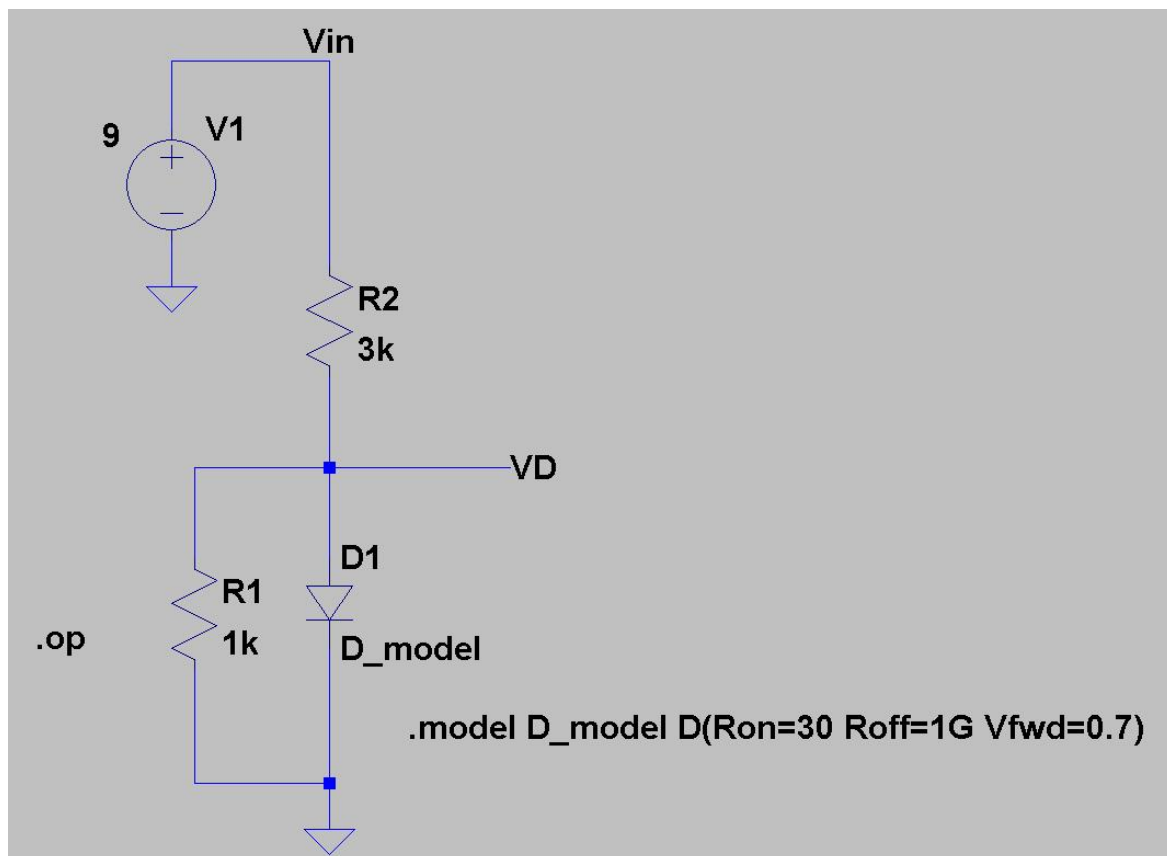
Figure 4.1: Schematic for the simple diode circuit.

In this example, the forward biased diode will be modelled as an offset voltage of 0.7V in series with a resistance of 30Ω. The reverse biased diode will be modelled as a very large resistance of 1GΩ. This model is implemented by adding the SPICE directive "**.model D_model D(Ron=30 Roff=1G Vfwd=0.7)**" to the schematic. The SPICE directive is added by clicking on the **SPICE directive** button shown in Figure 3.1. After being added, it may be edited by right clicking on the text in the schematic.

The diode model defined in the SPICE directive now needs to be associated with the diode in the schematic. This is done by holding down Ctrl and right-clicking on the diode. The **Component Attribute Editor** shown in Figure 4.2 will appear. As shown in Figure 4.2, the **Value** field should be changed to **D_Model**.
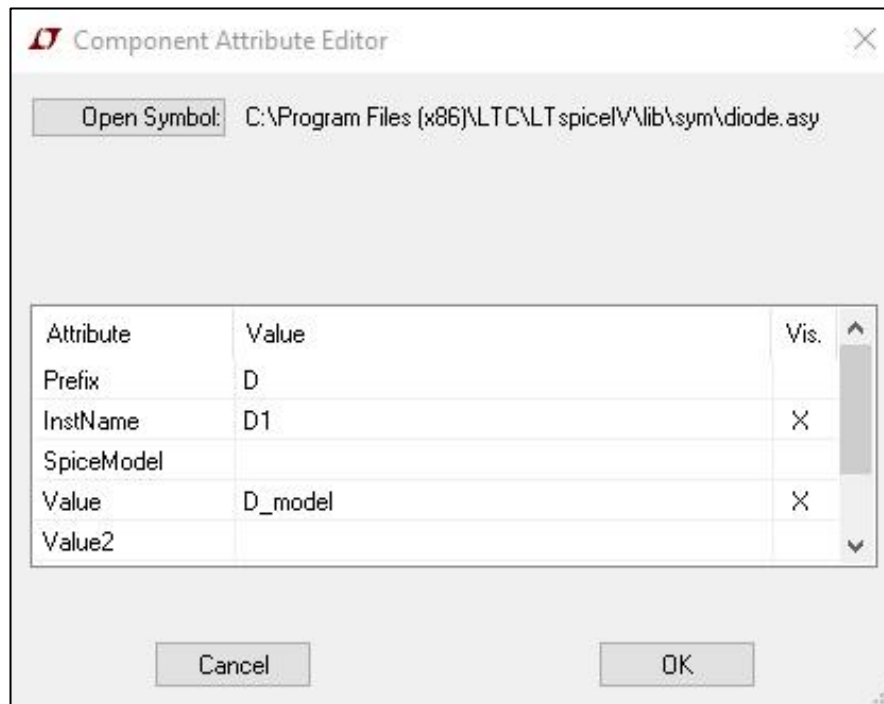

Figure 4.2: Component Attribute Editor for the diode.

To determine the DC voltages and currents in the circuit, operating point analysis will be done. This is chosen by clicking **Simulate/Edit Simulation Cmd**, then clicking on the **DC Op Pnt** tab. After clicking **OK**, the SPICE directive **.op** will appear on the schematic.

After clicking the **Run** button, a popup window will appear with the calculated values of voltage and current in the circuit, as shown in Figure 4.3.
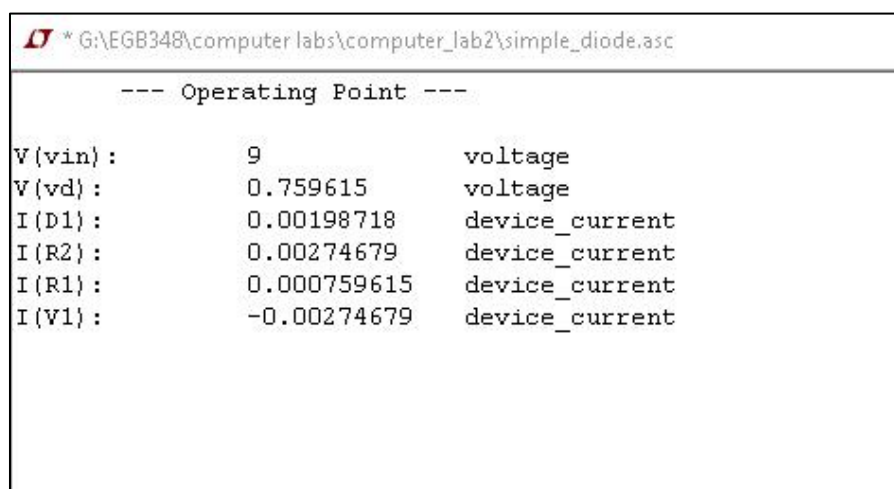
Figure 4.3: Results of operating point analysis of the simple diode circuit.

Q 4.1. How do the values of voltages and currents in the circuit compare to those calculated in Tutorial 1, Q1?

4.2 Diode Circuit

Here, you will draw the schematic and simulate the circuit from Tutorial 1, Q3. Draw the schematic shown in Figure 4.4   You may use the file **diode_circuit.asc** as a starting point.
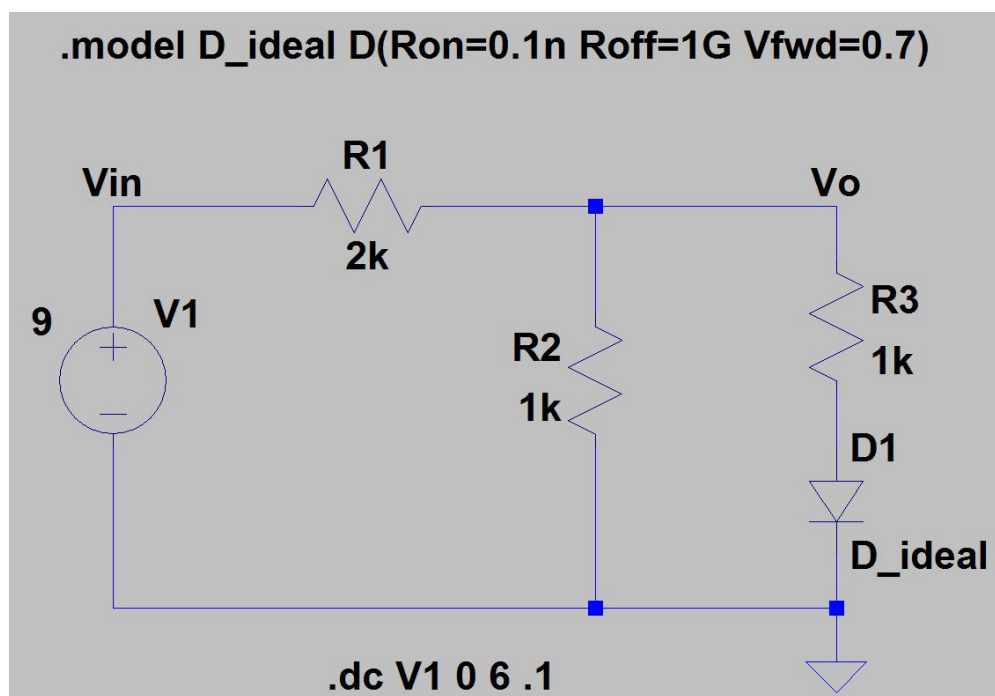

Figure 4.4: Diode circuit.

The forward biased diode will be modelled as an offset voltage of 0.7V in series with a very small resistance of 0.1nΩ. The reverse biased diode will be modelled as a very large resistance of 1GΩ. This model is implemented by adding the SPICE directive "**.model D_ideal D(Ron=0.1n Roff=1G Vfwd=0.7)**" to the schematic.

Holding down Ctrl and right-clicking on the diode will bring up the **Component Attribute Editor** (shown in Figure 4.2). In this case, the **Value** field should be changed to **D_Ideal**.

A **DC sweep** analysis will be done in this example. This allows V1 to be varied through a range of values. To enter the parameters for the DC sweep analysis, click **Simulate/Edit Simulation Cmd**, then click on the **DC DC sweep** tab, as shown in Figure 4.5.   Enter **Name of 1st Source to Sweep:** V1, **Type of Sweep:** Linear, **Start Value:** 0, **Stop Value:** 6, and **Increment:** 0.1. After clicking **OK**, the SPICE directive **.dc V1 0 6 .1** will appear on the schematic.

After clicking the Run button, an empty plot will appear.   To plot Vo, hover the mouse over the **Vo** label, and when the cursor changes to a red probe, left click. The plot of Figure 4.6, which shows how Vo changes with Vin, should appear.
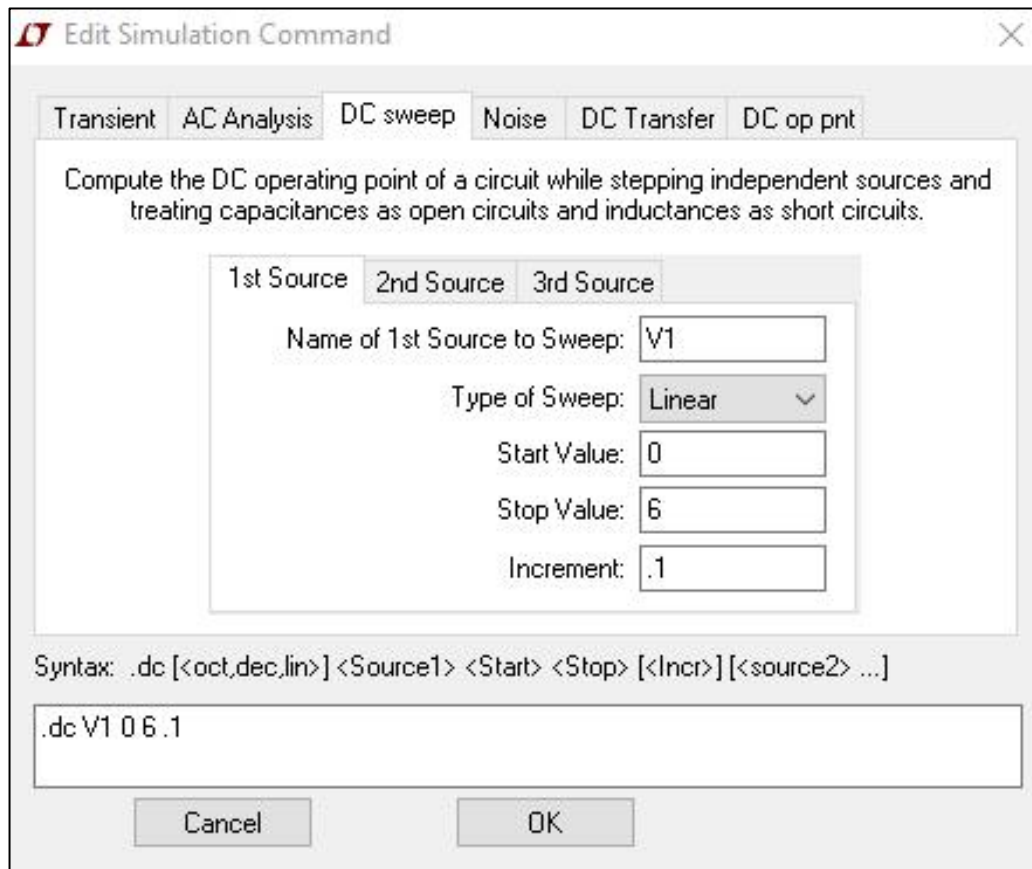

Figure 4.5: Diode circuit – DC Sweep simulation parameters.

Figure 3.7: Diode circuit – plot of Vo vs Vin.

Q 4.2. How does the plot of Figure 4.6 compare with your answer for Tutorial 2, Q3?

4.3 Clippers

Clipper circuits may be used to remove portions of a signal that are above or below a certain level.

4.3.1 Diode Clipper

Draw the clipper circuit schematic shown in Figure 4.7.   You may use **clipper1.asc** as a starting point.
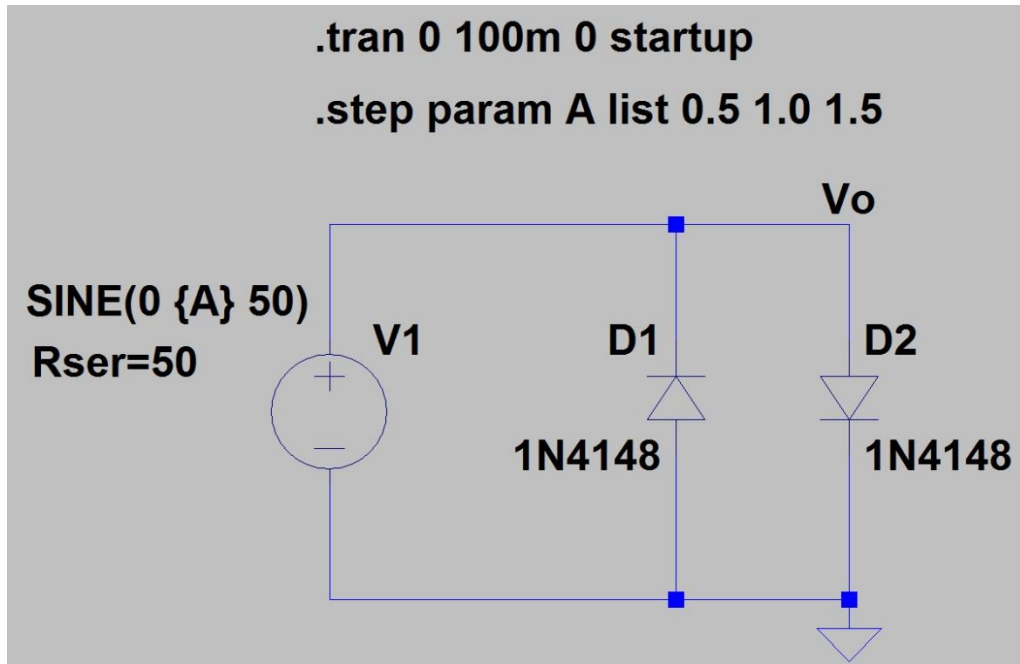
Figure 4.7: Diode clipper circuit 1.

In this example, we will use a diode model provided by LTSpice. Right click on the diode, and the diode properties will appear as shown in Figure 4.8. Click on **Pick New Diode**.   The **Diode Select** window will appear as shown in Figure 4.9. Scroll down to find the model for the **1N4148**, and click **OK**.
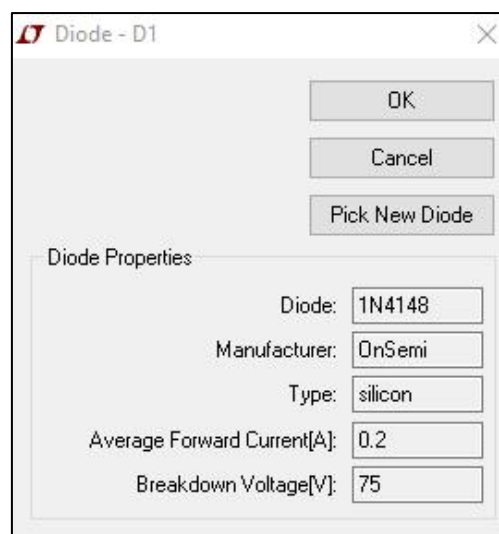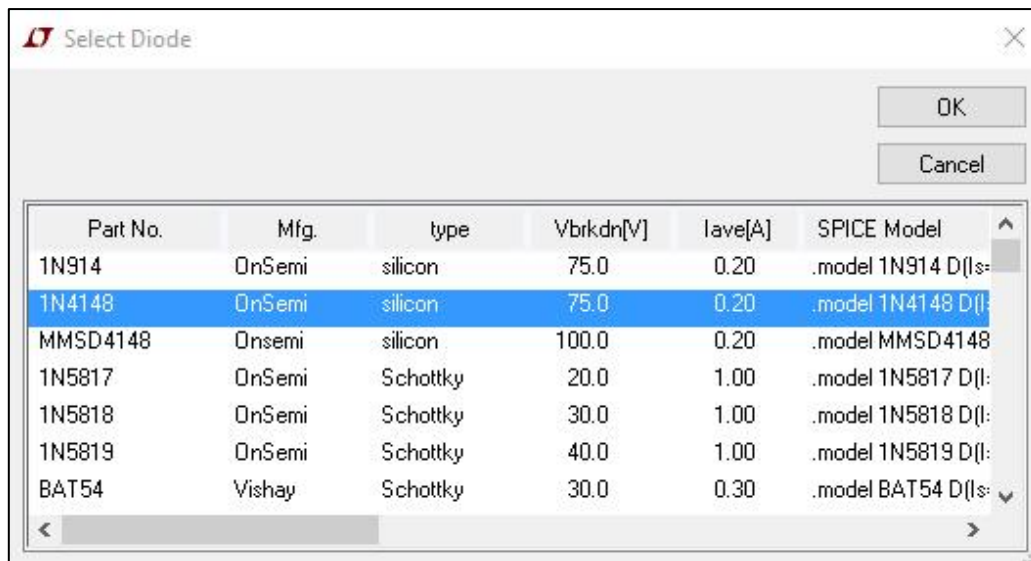

Figure 4.8: Diode properties.

Figure 4.9: Select diode window.

To view how circuit voltages change with time, transient analysis will be performed. The command for transient analysis may be edited by selecting **Simulate/Edit Simulation Cmd**, then clicking on the **Transient** tab.

In addition, the voltage source V1 has been defined with its amplitude as a parameter, {A}. To define values for this parameter, add the SPICE directive, **.step param A list 0.5 1.0 1.5** to the schematic using the **SPICE directive** button.   This **.step** SPICE directive will result in the transient analysis being run for each value of the parameter {A} in the list. Therefore, transient analysis will be performed for input sine waves of amplitude 0.5, 1.0 and 1.5V.

After clicking the **Run** button, an empty plot will appear.   To plot Vo, hover the mouse over the **Vo** label, and when the cursor changes to a red probe, left click. This will plot Vo for each amplitude of Vin, as shown in Figure 4.10.

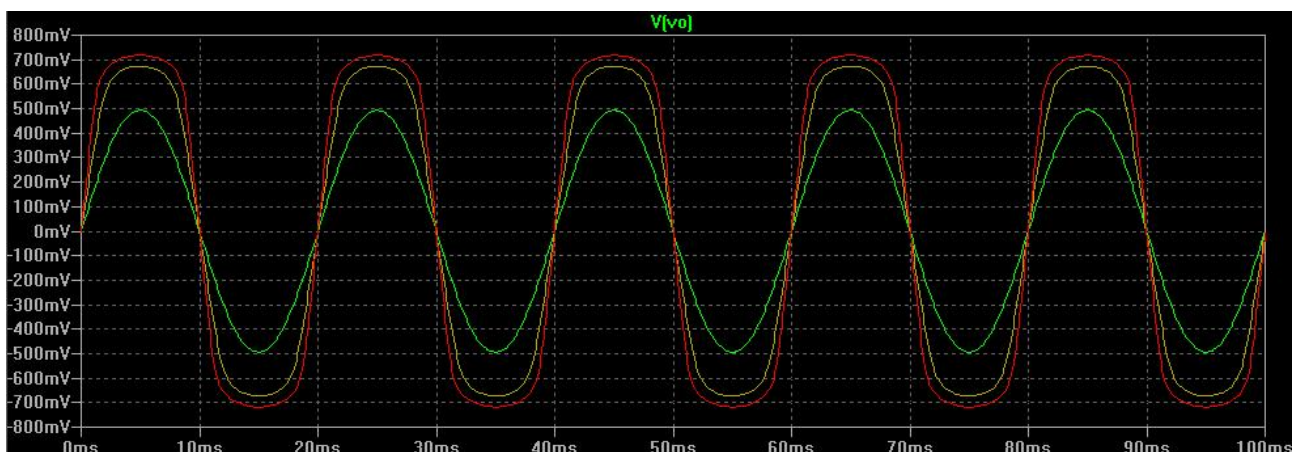Figure 4.10: Results of transient analysis, for input sine waves of amplitude 0.5, 1.0 and 1.5 V.

Q 4.3. Are the outputs of Vo for each amplitude of Vin as expected?

4.3.2 Zener Diode Clipper

Draw the clipper circuit schematic shown in Figure 4.11.   You may use **clipper2.asc** as a starting point.

A Zener diode part may be selected by clicking on the **Component** button, and choosing **zener** from the components listed. Again, we will use a diode model provided by LTSpice.   Right clicking on the Zener diode component in the schematic will display the diode properties. Click the **Pick New Diode** button, scroll down to find the **1N750**, and click **OK**.
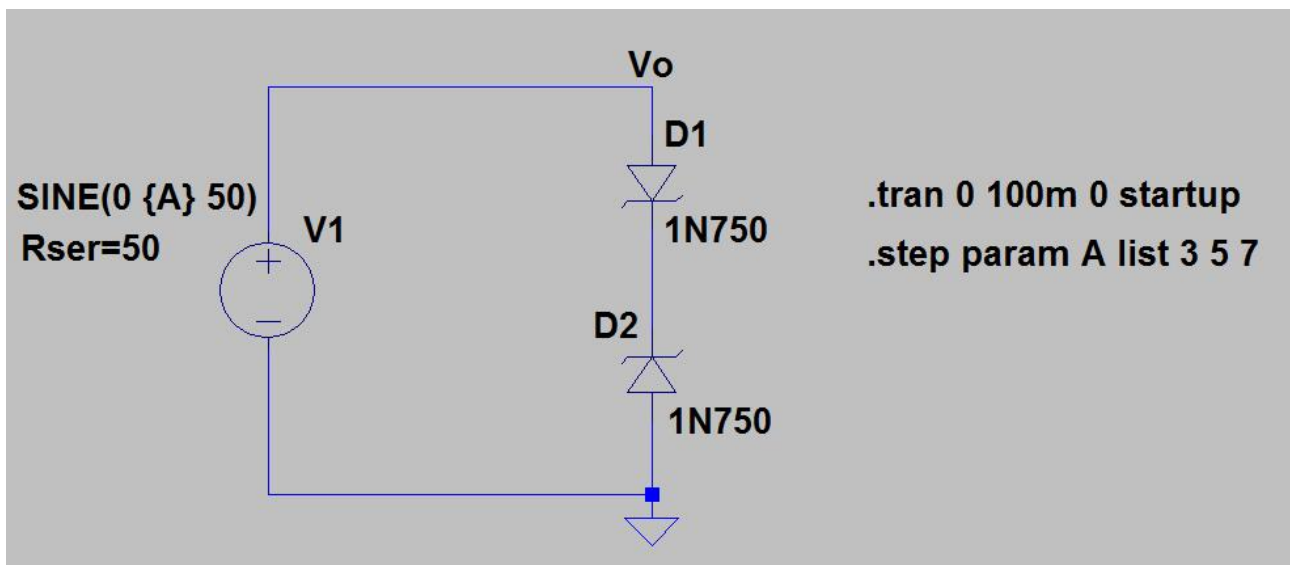
Figure 4.11: Diode clipper circuit 2.

Again, a parameter {A} has been used for the amplitude of the input sine wave (V1).   Add the SPICE directive **.step param A list 3 5 7**, to provide a list of values for this parameter. Click the **Run** button to perform transient analysis, and plot Vo.

Ex 4.1. Sketch your output plots for Vo below.   Are the output plots as expected?

4.4 Diode Logic

Here you will simulate diode logic circuits.   For convenience, the DVIEW part has been used to vertically view output waveforms in the same plot. It is easiest to copy the files for the DVIEW part to your working directory containing your circuit files.

4.4.1 Diode OR gate

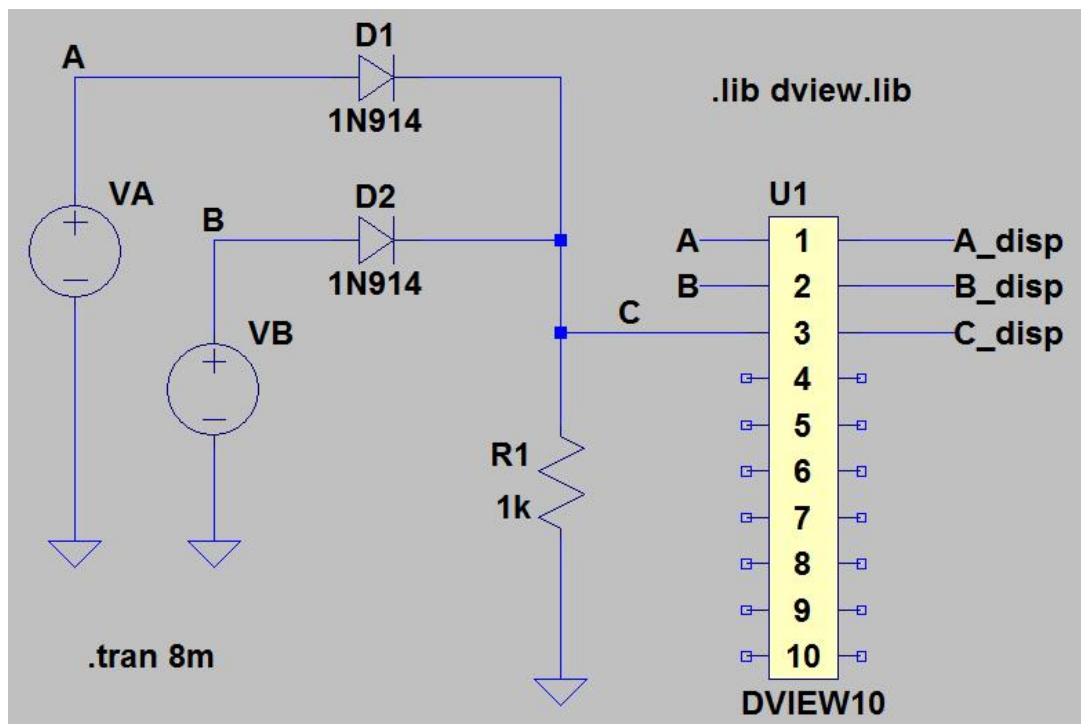Load the schematic **diode_or.asc**. The 1N914 diode model provided by LTSPice has been used in this example.

Figure 4.12: Diode OR gate.

The voltage sources have been set up as pulse functions, which will cycle through all possible input combinations of A and B.   Click the **Run** button, and an empty plot will appear.   Now hover the mouse over the A_disp, B_disp and C_disp outputs of the **DVIEW10**, and when the cursor changes to a red probe, left click. This will display the A, B and C waveforms separated vertically.

Ex 4.2. Sketch your output plots for A, B and C below.   Are the output plots as expected?

4.4.1 Diode AND gate

Modify your schematic so that it appears as below. You may save your work to a new file, for example **diode_and.asc**.   Run the simulation and plot A, B and C.
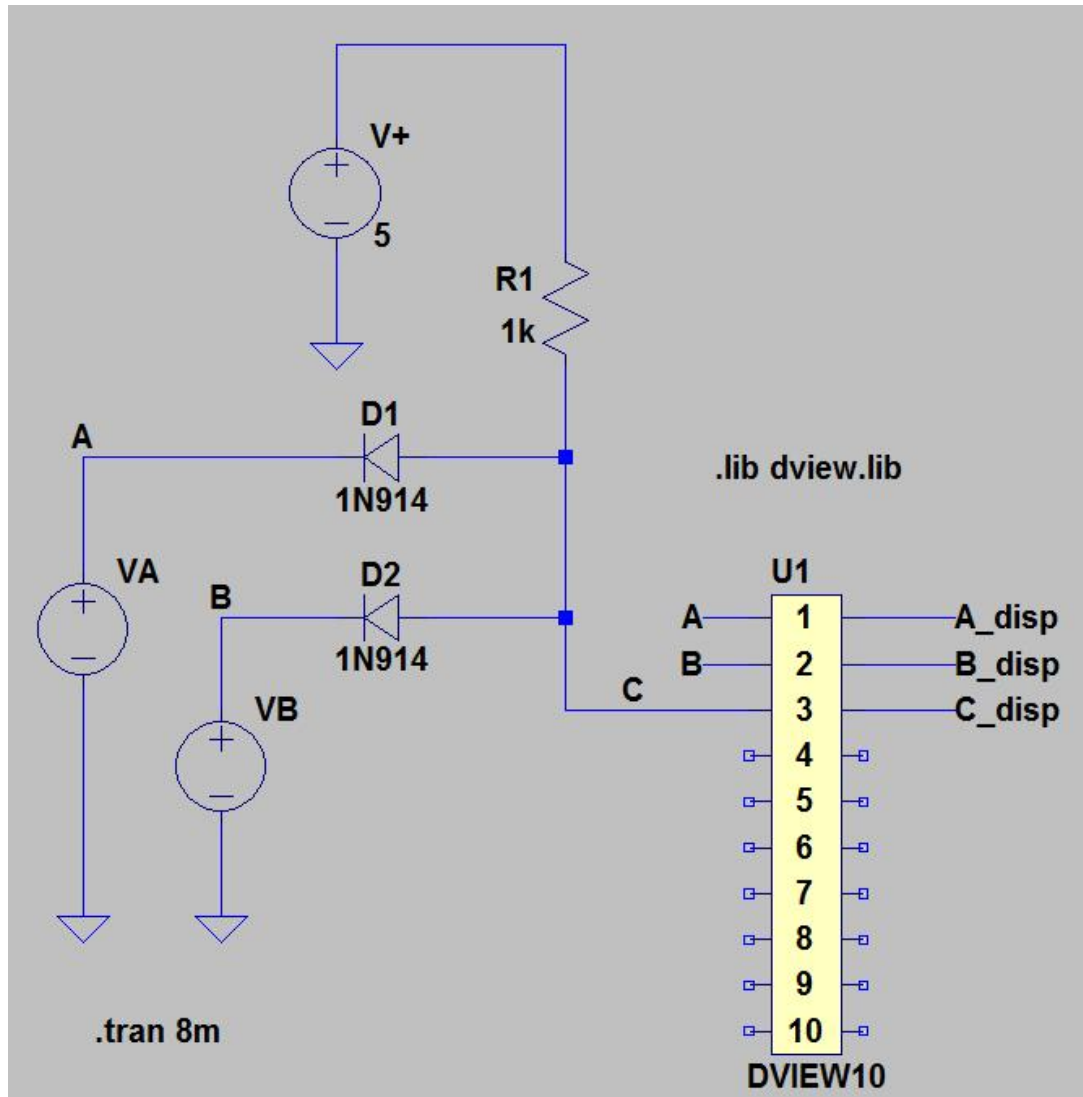

Figure 4.13: Diode AND gate.

Ex 4.3. Sketch your output plots for A, B and C below.   Are the output plots as expected?

5. Rectifiers

In this section, you will simulate some of the rectifier circuits from Tutorial 1 and the lecture notes.

5.1 Half Wave Rectifier

Draw the schematic of the half wave rectifier shown in Figure 5.1. You may use **rectifier_half_wave240.asc** as a starting point.

In this example, a third party diode model that is not provided by LTSpice will be used.   This third party model is given by the text file **1N4007.REV0.LIB**, as shown in Figure 5.2.   This is a text file, and it does not matter what the extension of the file is (in this case it is .lib, but it could be .txt or something else).
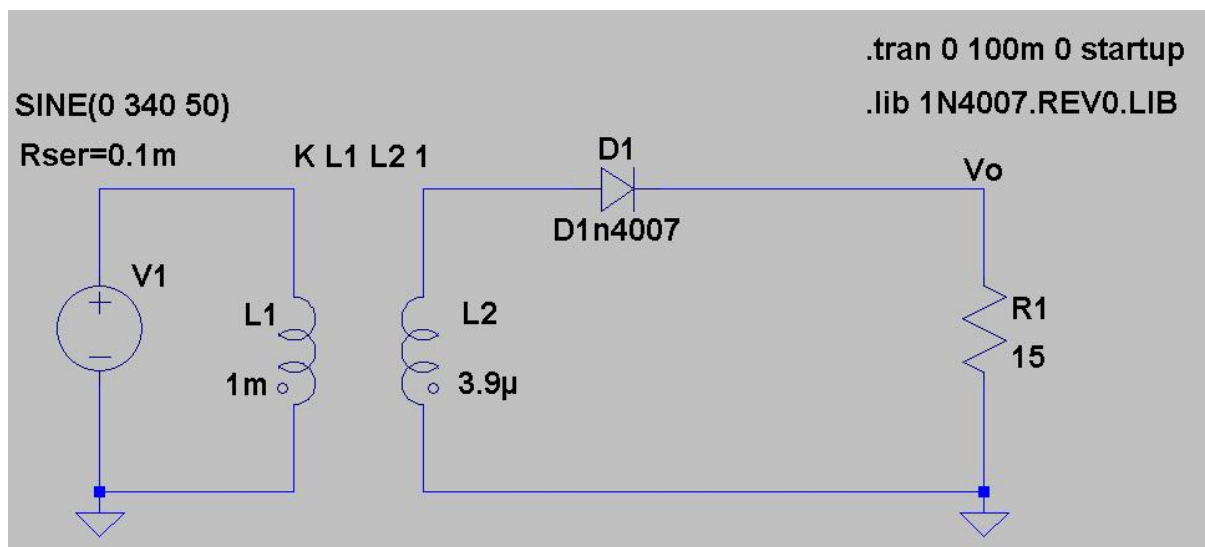

Figure 5.1: Half wave rectifier.

```
*************************************
*     Model Generated by MODPEX    *
*Copyright(c) Symmetry Design Systems*
*      All Rights Reserved      *
*   UNPUBLISHED LICENSED SOFTWARE   *
*  Contains Proprietary Information *
*    Which is The Property of     *
*    SYMMETRY OR ITS LICENSORS     *
*Commercial Use or Resale Restricted *
*  by Symmetry License Agreement   *
*************************************
* Model generated on May 30, 03
* MODEL FORMAT: PSpice
.MODEL D1n4007 d
+IS=7.02767e-09 RS=0.0341512 N=1.80803 EG=1.05743
+XTI=5 BV=1000 IBV=5e-08 CJO=1e-11
+VJ=0.7 M=0.5 FC=0.5 TT=1e-07
+KF=0 AF=1
```

Figure 5.2: Third party diode model 1N4007.REV0.LIB.

To associate this third party model with a diode, hold down Ctrl and right click on the diode part in the schematic.   The **Component Attribute Editor** window should appear.   Set the **Value** field to **D1n4007**, as shown in Figure 5.3.   This is the same as the name following the **.MODEL** statement in Figure 5.2.

The SPICE directive, **.lib 1N4007.REV0.LIB**, also needs to be added to the schematic, and the file **1N4007.REV0.LIB** should be in your working directory.
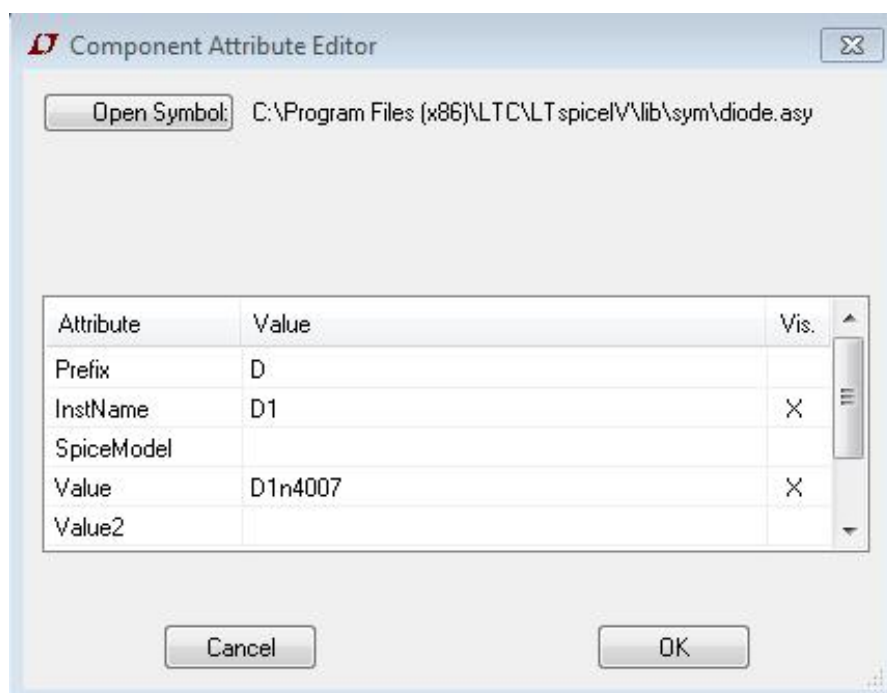

Figure 5.3: Component Attribute Editor.

The SPICE directive, **K L1 L2 1**, also called a **K statement**, indicates that L1 and L2 are a transformer. The last number, 1, indicates that there is perfect coupling between the inductors and there is no leakage inductance.

The transient analysis simulation command, **.tran 0 100m 0 startup** has been included on the schematic. This may be edited by right clicking on the text in the schematic, or by selecting **Simulate/Edit Simulation Cmd** from the menus. Click the **Run** button to perform the transient analysis simulation. Plot the output voltage Vo and input voltage V1, as shown in Figure 5.4.

Q 5.1. Are the plots of input and output voltages of the half wave rectifier as expected?
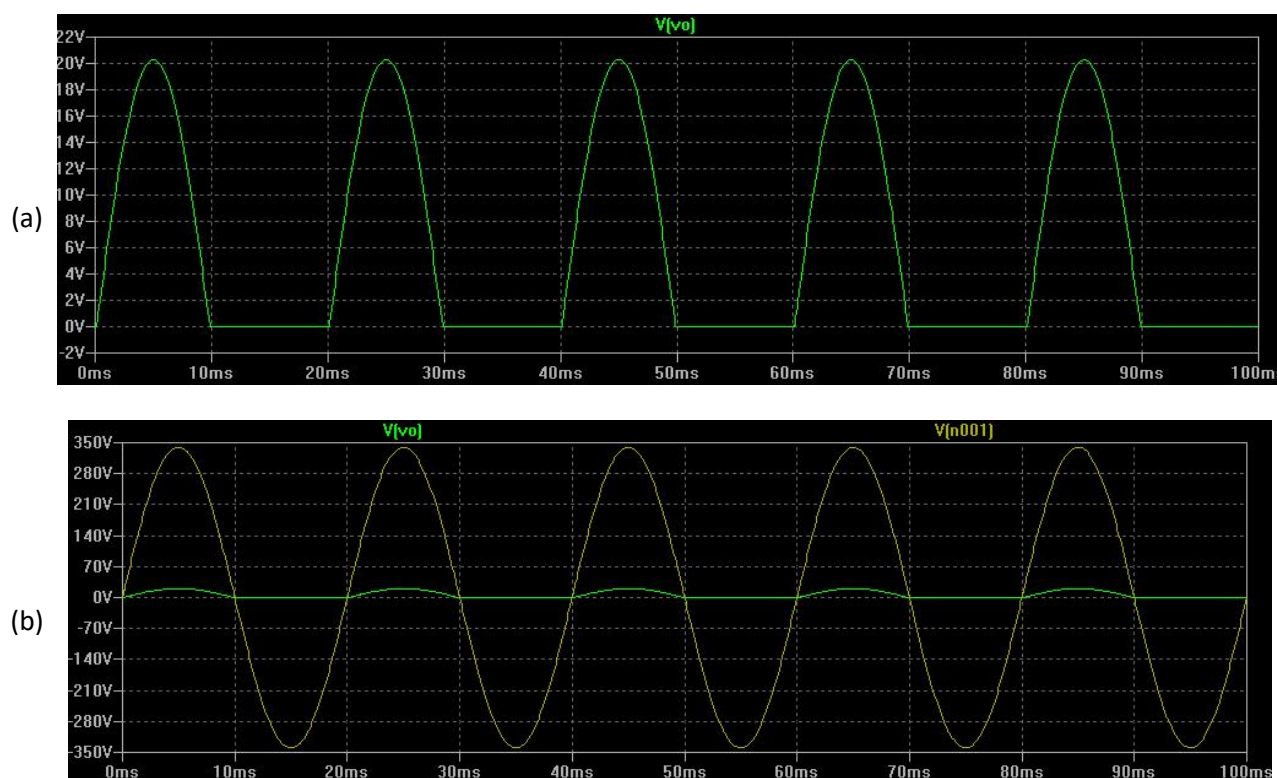
(a)

(b)

Figure 5.4: Half wave rectifier output (a) output Vo; (b) output Vo and input V1 on the same plot.

Next, add a capacitor across resistor R1, as shown in Figure 5.5. Perform the simulation again, and plot the waveforms for Vo, diode current (I_D), capacitor current (I_C) and resistor current (I_L).
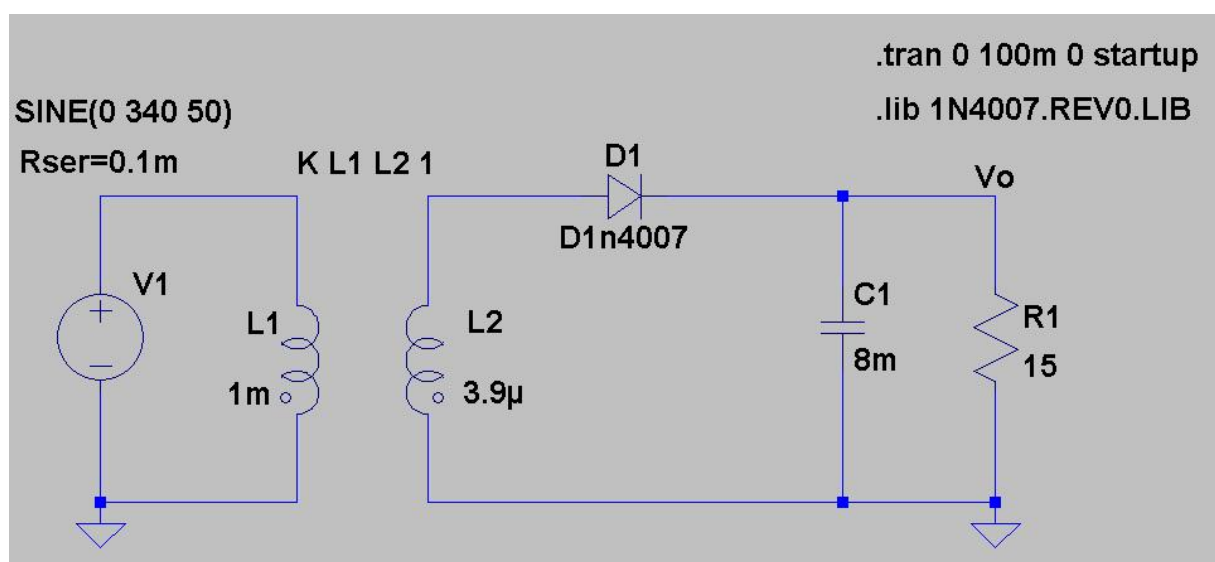

Figure 5.5: Half wave rectifier with smoothing capacitor.

Ex 5.1. Sketch your plots of circuit voltages and current below.   Are the simulation results as expected? How do the voltages and currents compare with theoretically calculated values?

5.2 Full Wave Bridge Rectifier

Draw the schematic of the full wave bridge rectifier shown in Figure 5.6. You may use **rectifier_full_wave_bridge240.asc** as a starting point.
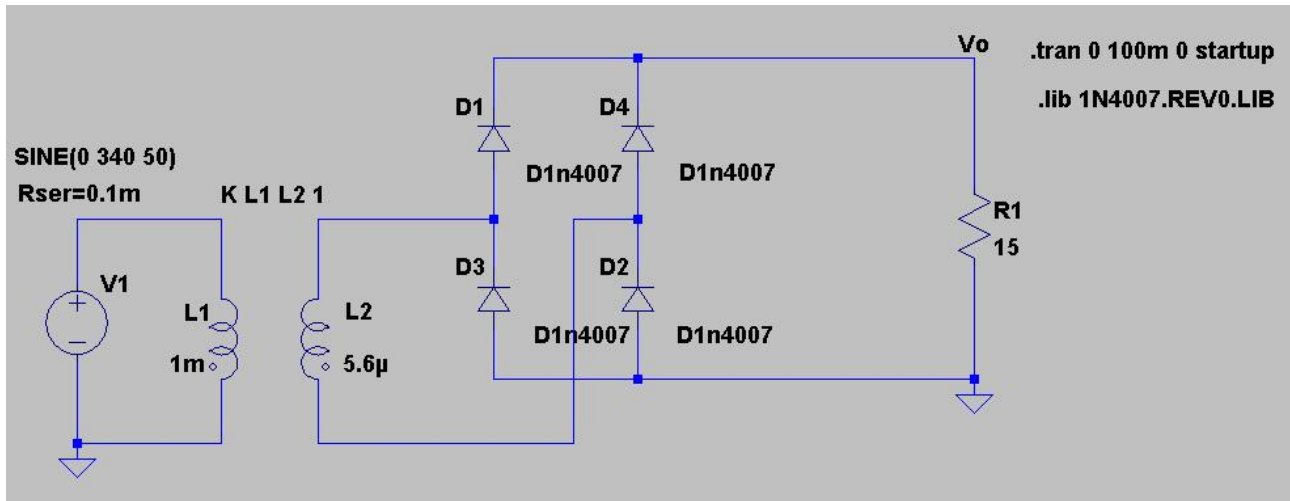

Figure 5.6: Full Wave Bridge Rectifier.

Run the simulation, and plot the output voltage Vo and input voltage V1.

Ex 5.2. Sketch your plots below. Are your waveforms for the full wave bridge rectifier as expected?

Next, add a capacitor across resistor R1, as shown in Figure 5.7. Perform the simulation again, and plot the waveforms for Vo, diode currents (I_{D12}, I_{D34}), capacitor current (I_C) and resistor current (I_L).
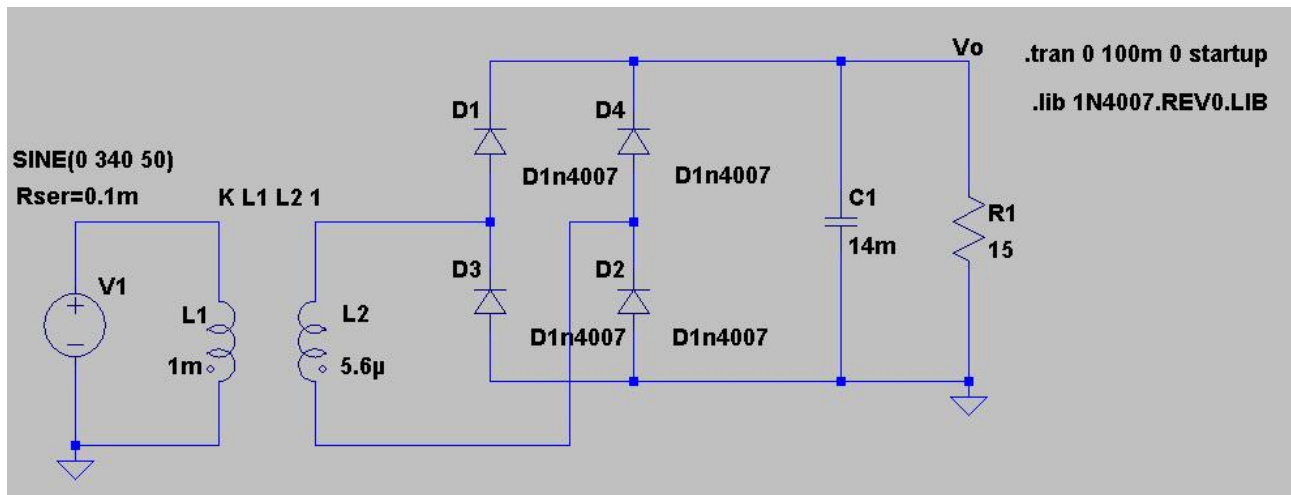

Figure 5.7: Full Wave Bridge Rectifier with smoothing capacitor.

Ex 5.3. Sketch your plots of circuit voltages and current below.   Are the simulation results as expected? How do the voltages and currents compare with theoretically calculated values?

6. Zener Diode Regulators

6.1 Unloaded Zener Regulator

Draw the schematic of the unloaded Zener regulator as shown in Figure 5.1. In this example, a model of the TFZ10B 10V Zener provided by LTSpice was used.
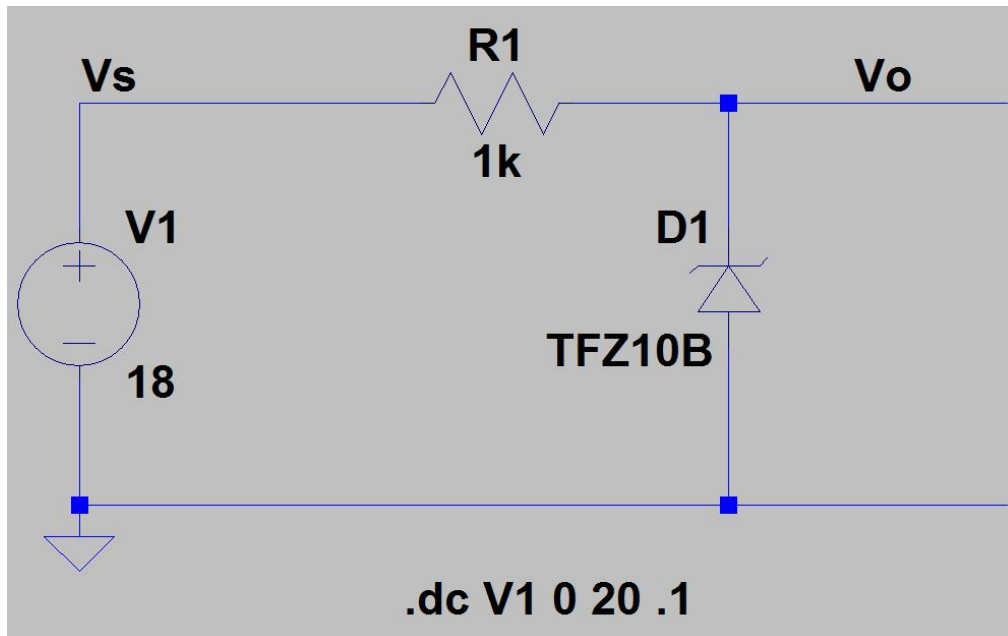

Figure 6.1: Unloaded Zener regulator.

Perform a DC Sweep analysis to plot Vo versus V1. The simulation command **.dc V1 0 20 .1** included in Figure 5.1 may be used.

Ex 6.1. Sketch your plot of Vo versus V1.

Q 6.1. At what value of input voltage (V1) does the circuit start behaving like a regulator?

Q 6.2. Can you estimate the Line Regulation ΔVo / ΔV1 from the plot?

6.2 Loaded Zener Regulator

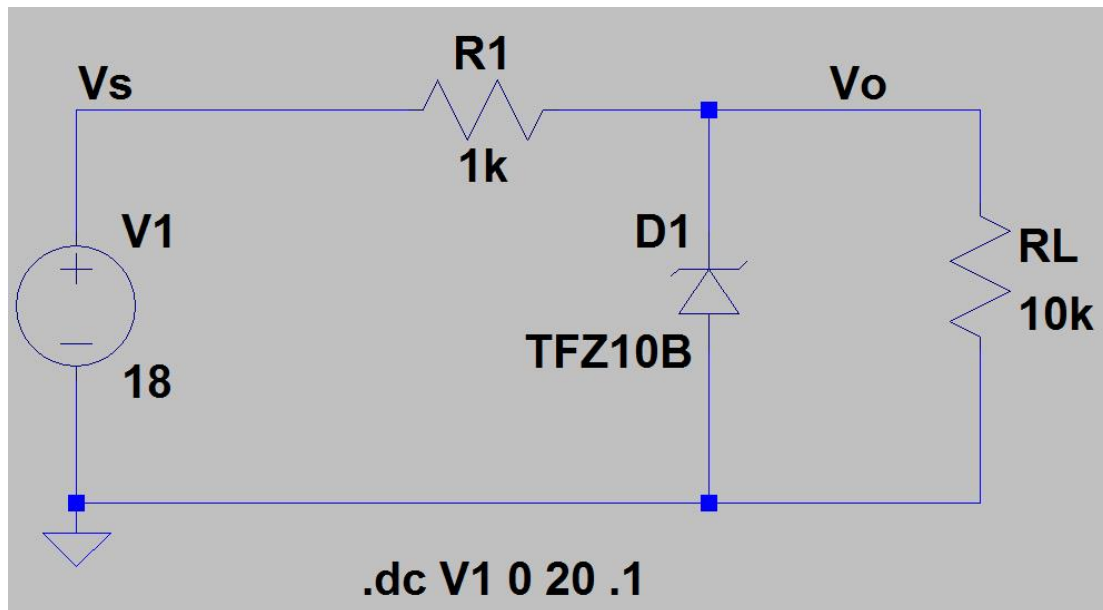Now add a load resistor, RL, at the output of the Zener regulator circuit, as shown in Figure 5.3.

Figure 6.3 Loaded Zener regulator.

Again, perform a DC Sweep analysis to plot Vo versus V1.

Ex 6.2. Sketch your plot of Vo versus V1.

Q 6.3. At what value of input voltage (V1) does the circuit start behaving like a regulator?

Q 6.4. Can you estimate the Line Regulation $\Delta Vo / \Delta V1$ from the plot?

One way to plot Vo versus I_L (where I_L is the current through RL), is to parameterise RL, and calculate Vo and I_L for each value of RL.   The **.step** SPICE directive included in Figure 5.4 steps the value of RL through a list of values. Operating point analysis is done for each value of RL, as specified by the **.op** simulation command in the schematic.
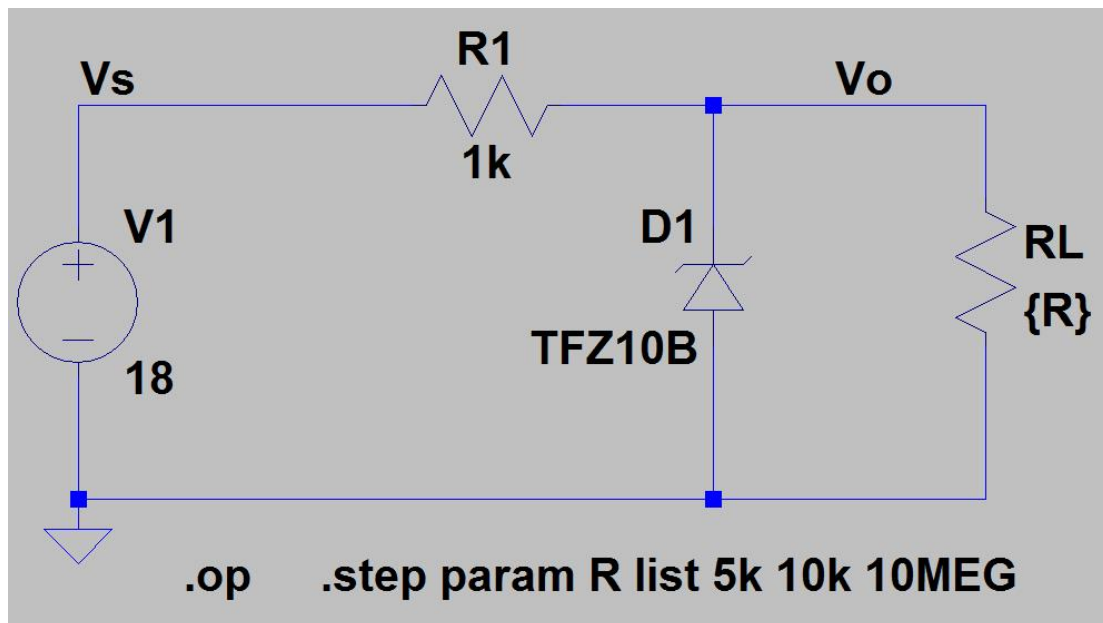

Figure 6.4: Loaded Zener regulator, with RL parameterised.

After clicking the **Run** button, the simulation runs and an empty plot appears. Left clicking the red probe on Vo will plot Vo versus R.   However, this is not the desired plot, as we wish to plot Vo versus I_L.

When the cursor hovers below the horizontal axis of the plot, it changes to a small ruler. Left clicking below the axis brings up the window of Figure 6.5. The **Quantity Plotted** field should be changed to **I(RL)**. After clicking OK, Vo should now be plotted versus IL, as shown in Figure 6.6.
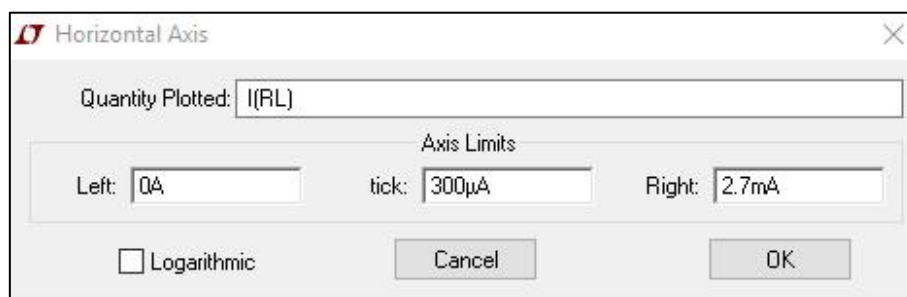

Figure 6.5: Horizontal Axis – change the quantity plotted to the current through RL.

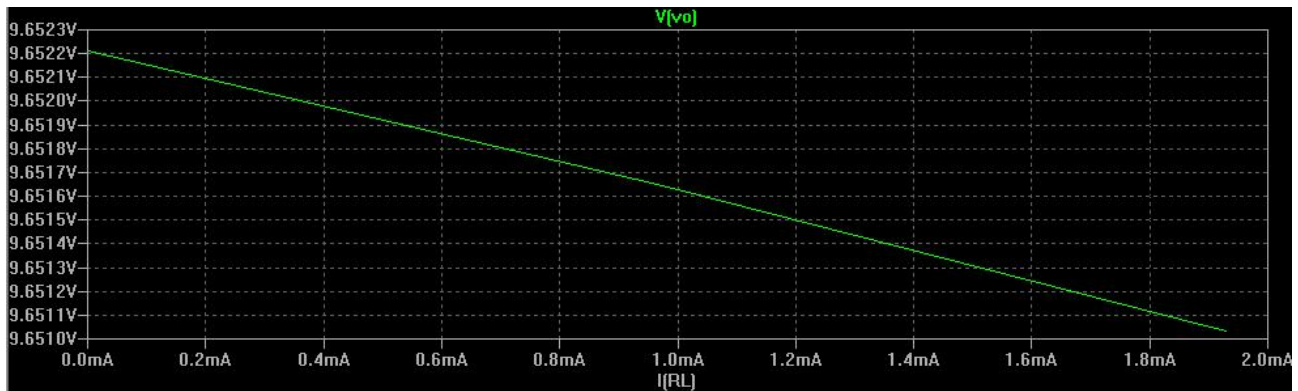Figure 6.6: Loaded Zener regulator, with RL parameterised – plot of V_O vs I_L.

Q 6.5. Can you estimate the Load Regulation ΔVo / ΔI_L from the plot?

Q 6.6. How does this compare to the theoretically calculated value?

7. Additional Practice Exercises

Ex 7.1. Draw the schematic for a centre tapped full wave rectifier, and simulate your circuit.

Studio 1 Workbook Section – Diodes

This section should be submitted as part of your Workbook. Working should be shown.

Student Name: _____

Student Number: _____

Hand Calculations

Part 1

Determine the reverse saturation current for the diode below, given that $V_S = 2.5V$, $R = 180\Omega$ produces a diode current of 10mA. Use $\eta = 1.5$ and $V_T = \frac{kT}{q} = 25mV$.

$I_S =$

Calculate the diode voltage (V_D) for diode currents (I_D) of 1mA, 5mA and 20mA. Fill your values in the table below. Use the values in the table to plot the diode characteristic and use a load line to graphically determine the quiescent diode current (I_{DQ}), quiescent diode voltage (V_{DQ}), and incremental resistance (r_d) at the operating point.

V_S

R

I_D

V_D

$0V$
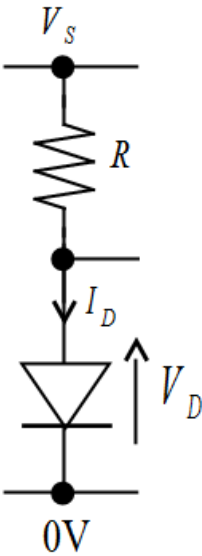
I_D(mA)	1	5	10	20
$V_D(V)$				

Include your plot here, or attach graph (graph paper provided):

Part 2

Use the quiescent diode current (I_{DQ}), obtained in Part 1 to theoretically calculate the incremental resistance (r_d) at the operating point.

$r_d =$

Computer Simulation

Part 1

Using LTSpice, model the full wave centre tapped rectifier below, and demonstrate its operation. Use an appropriate diode model. Choose appropriate values for the inductors, and you may assume no leakage inductance for your simulation. Include the input and output voltage waveforms as well as a screenshot of your circuit. Label important features on your graph.

Note: You will need to research how to implement the centre tapped rectifier in LTSpice. Some links to assist with this have been provided on Blackboard.
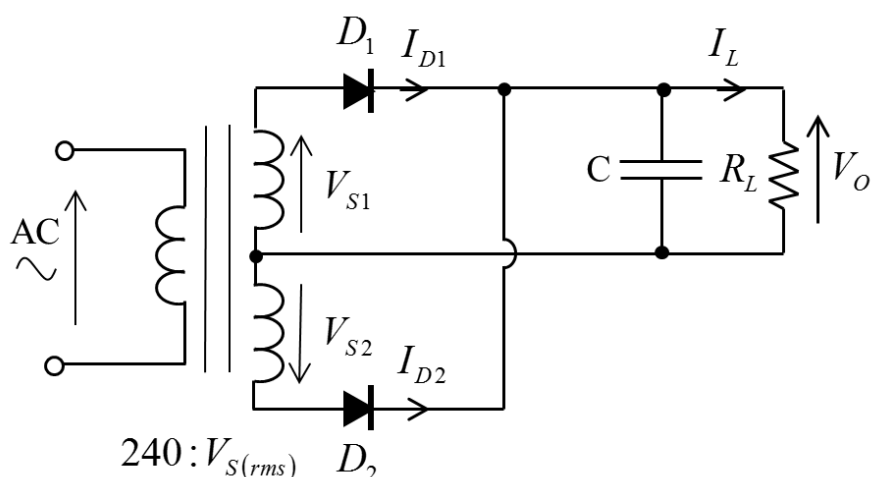
Use the following values:

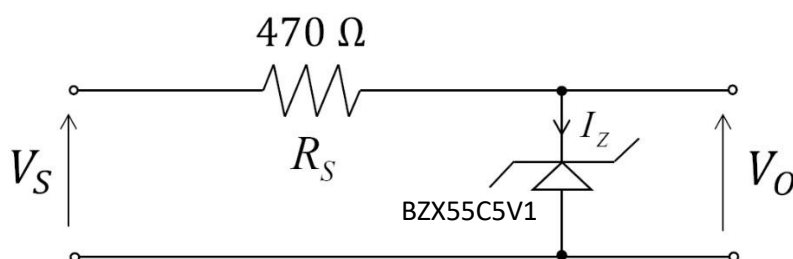$V_{S(rms)} = 1X$ V, where X is the last digit of your student number
$R_L = 15 \, \Omega$
$C = 1.5$ mF

Part 2

Using LTSpice, model the Zener diode circuit below. Use the provided diode model in bzx55c5v1.txt.

Note: Some links to assist with using a .subckt model in LTSpice have been provided on Blackboard.

i)   Plot V_O versus V_S for $V_S = 0 \dots 10V$
ii) At what value of V_S does the circuit start behaving like a regulator?
iii) Estimate the Line Regulation $\Delta V_O / \Delta V_S$ from the plot of V_O versus V_S.
iv) Plot V_O versus I_L, and use this to estimate the Load Regulation $\Delta V_O / \Delta I_L$.