VALENCIA COLLEGE

***Electrical & Computer Engineering Technology***

# ETP 4240C – POWER ELECTRONICS

# LABORATORY EXPERIMENTS

**BY**

**MASOOD EJAZ**

# ***EXPERIMENT # 1***

## ***SINGLE-PHASE FULL-WAVE RECTIFIERS***

**Note:** Make sure to finish your pre-lab that includes any theoretical derivations or calculations and PSpice simulations before you come to the lab.

**Purpose:** To design single-phase full-wave rectifiers including center-tapped and diode-bridge and to study their characteristics.

**PSpice Simulation:**

*(I)* *Center- Tapped Full-Wave Rectifier:*

1. Design the center-tapped rectifier as shown in *figure 1*. Center-tapped transformer is in *ANL_MISC* library (*XFRM_LIN/CT_SEC*). This model of center tapped transformer takes the ratio of primary-to-secondary winding as *primary peak voltage* to *secondary RMS voltage* as given by the following equation:

$$\frac{V_1(peak)}{V_2(rms)} = \frac{N_1}{N_2} = \sqrt{\frac{L_1}{L_2}}$$

For correct operation, go to the transformer properties and change primary inductance value to 2000H and secondary value to 20H. Once again, all of the waveforms that you see on the secondary side will correspond to the *RMS* values. To get peak value or peak-to-peak value, you will have to perform the necessary conversion.

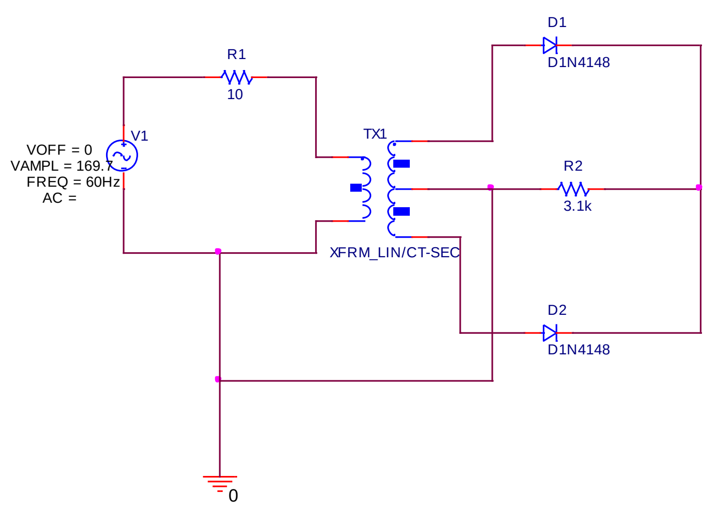

*Figure 1: Center-Tapped Full-Wave Rectifier*

2. Perform transient analysis for 50ms. Add four plots in the simulation window. From top to bottom, trace $v_p$, $v_{s1}$ & $-v_{s2}$ (in the same window), $v_{load}$, and reverse bias voltage across any diode. Using theoretical concepts, analyze your results for their validity.

3. Calculate the value of filter capacitor that will result in the ripple factor of 5%. Place the capacitor across the load and confirm your results from the simulation.

*(II)* *<u>Diode-Bridge Full-Wave Rectifier:</u>*

1. Design the diode-bridge rectifier as shown in *figure 2*. Transformer is in the *analog* library. Choose primary and secondary inductances as 2000H and 20H respectively, for the transformer according to the following relationship:

$$\frac{V_1(peak)}{V_2(peak)} = \frac{N_1}{N_2} = \sqrt{\frac{L_1}{L_2}}$$

where $V_1$ and $V_2$ represent primary and secondary *peak* voltages, $N_1$ and $N_2$ represent primary and secondary turns and $L_1$ and $L_2$ represent primary and secondary inductances, respectively. Secondary voltage of 12V(rms) is required.

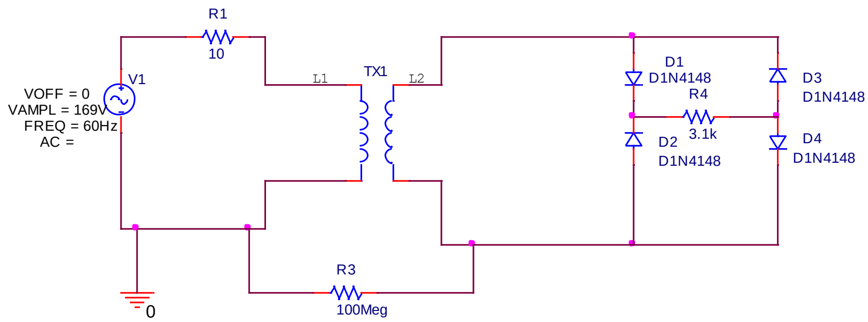


*Figure 2: Diode-Bridge Full-Wave Rectifier*

2. Perform transient analysis for 50ms and trace the same quantities as required in *step 3* for the center-tapped rectifier.
3. Calculate the value of filter capacitor for 5% ripple factor. Place the capacitor across the load and perform simulation to confirm your results.

**Lab Procedure:**

1. Using the transformer provided, build both the rectifier circuits and perform *steps 3 & 4* for the center-tapped rectifier, and steps *2 & 3* for the diode-bridge rectifier. Compare your results against the simulated ones.

Note: Make sure to save all of your waveforms from the oscilloscope for the lab report.

**Discussion:**

In your lab report, explain the operation of rectifiers in detail and relate your results accordingly. Also, discuss different advantages and disadvantages for the two circuits designed in the lab.

# EXPERIMENT # 2

# BOOST CONVERTERS

**Note:** Make sure to finish your pre-lab that includes any theoretical derivations or calculations and PSpice simulations before you come to the lab.

**Purpose:** To design a DC-DC Boost Converter and to study its characteristics.

**Prelab:**

1. Study the characteristics of *IRF150* N-channel Power MOSTFET from its datasheet and write down the following quantities:
   (i) Drain-to-Source voltage: ______________________________
   (ii) Drain-to-Source continuous current: ______________________
   (iii) Gate-to-Source voltage: ______________________________
   (iv) Maximum power dissipation: __________________________
   (v) Drain-to-Source ON resistance (typical value): _______________
   (vi) Total turn-on time ($t_d(ON)+t_r$): ________________________
   (vii) Total turn-off time ($t_d(OFF)+t_f$): ______________________

2. For the boost converter circuit shown in *figure 1*, required value for the average output voltage is 15V, switching frequency is 100kHz, required value for the peak-to-peak inductor current ($\Delta I_L$) is 6mA, and required value for the peak-to-peak output voltage ripple ($\Delta V_c$) is 28mV. Calculate the values for the following components and quantities and write down the expressions you are using to calculate the values.

| | Expression | Value |
|---|---|---|
| Duty Cycle $k$ | | |
| ON time $t_1$ | | |
| OFF time $t_1$ | | |
| Inductor | | |
| Capacitor | | |
| $I_{out}$ (avg) | | |
| $I_{in}$ (avg) | | |
| $I_L(max)$(approx.) | | |
| $I_L(min)$(approx.) | | |

[Note: You may consider writing a small script in MATLAB that takes values of *R*, *L*, *C*, *Vin*, *k*, and *f* and calculate $t_1$, $t_2$, $V_{out}$, $\Delta I_L$, $\Delta V_c$, $I_{out}$ (avg), $I_{in}$ (avg), $I_L(max)$(approx.), and $I_L(min)$(approx.) to confirm your results and to recalculate them later in the lab with different values of *L* and *C*.]

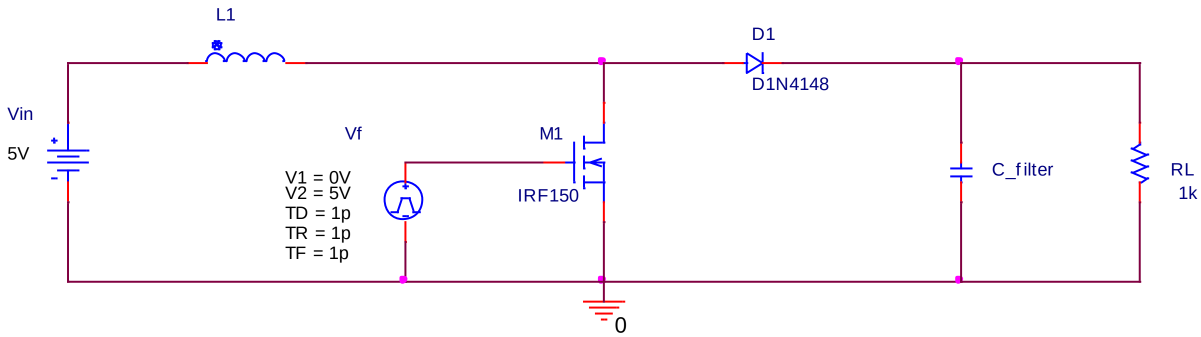


*Figure 1: Boost Converter*

3. Simulate the circuit and observe the output until it comes to steady-state. Observe the following quantities.

Note: If you encounter *convergence* problem with your simulation, go to *Simulation Settings*→*Options* → *Auto Converge* and check *Auto Converge* box.

| | Value |
|---|---|
| Duty Cycle $k$ | |
| ON time $t_1$ | |
| OFF time $t_1$ | |
| $\Delta V_c$ | |
| $\Delta I_L$ | |
| $V_{out}$ (avg) | |
| $I_{out}$ (avg) | |
| $I_{in}$ (avg) | |
| $I_L(max)$(approx.) | |
| $I_L(min)$(approx.) | |

## Lab Procedure:

Note: Make sure to save all of your waveforms from the oscilloscope for the lab report.

4. Design the circuit on the breadboard. Choose the closest values of inductor and capacitor that are available. Measure the following quantities.

| | Value |
|---|---|
| Duty Cycle $k$ | |
| ON time $t_1$ | |
| OFF time $t_1$ | |
| $\Delta V_c$ | |
| $\Delta I_L$ | |
| $V_{out}$ (avg) | |
| $I_{out}$ (avg) | |
| $I_{in}$ (avg) | |
| $I_L(max)$(approx.) | |
| $I_L(min)$(approx.) | |

5. Change the values for the inductor and capacitor in your theory and simulation to the ones that you used in your experiment and calculate and observe new values for the following quantities.

*Theory*

| | Value |
|---|---|
| Duty Cycle $k$ | |
| ON time $t_1$ | |
| OFF time $t_1$ | |
| Inductor | |
| Capacitor | |
| $I_{out}$ (avg) | |
| $I_{in}$ (avg) | |
| $I_L(max)$(approx.) | |
| $I_L(min)$(approx.) | |

*Simulation*

| | Value |
|---|---|
| Duty Cycle $k$ | |
| ON time $t_1$ | |
| OFF time $t_1$ | |
| $\Delta V_c$ | |
| $\Delta I_L$ | |
| $V_{out}$ (avg) | |
| $I_{out}$ (avg) | |
| $I_{in}$ (avg) | |
| $I_L(max)$(approx.) | |
| $I_L(min)$(approx.) | |

**Discussion:**

In your lab report, discuss the operation of boost converters in detail. Compare your theoretical, simulated and practical results and discuss possible logical reasons for any discrepancies that you have.

# ***EXPERIMENT # 3***

## ***SINGLE-PHASE FULL BRIDGE INVERTERS***

**Note:** Make sure to finish your pre-lab that includes any theoretical derivations or calculations and PSpice simulations before you come to the lab.

**Purpose:** To design and study the characteristics of a single-phase full bridge inverter with *R* and *RL* loads.

**Prelab:** (Note: switching frequency is 60Hz)

Theoretical Work:

1. For *circuit # 1*, draw the labeled output waveform. Assume switches to be ideal.

| *Output Waveform* |
|---|
| |

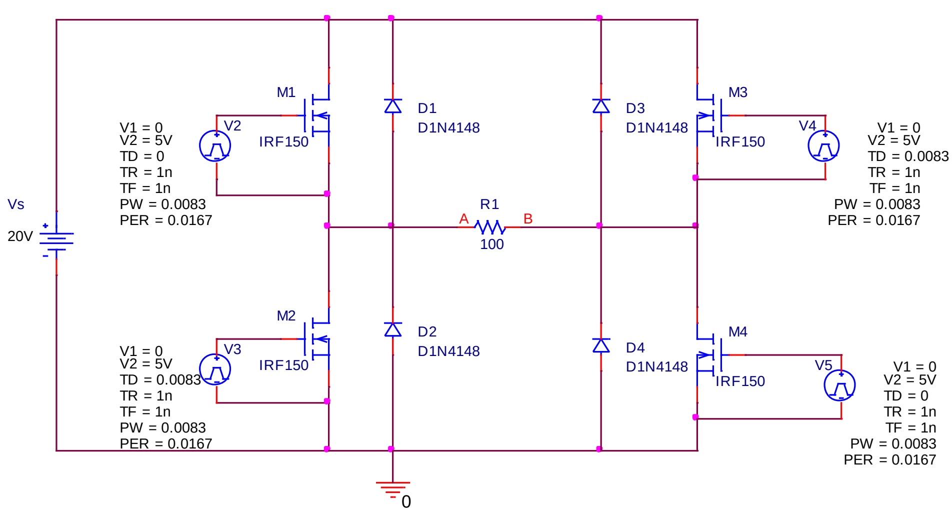


*Figure 1: Bridge Converter with Resistive Load*

2. Derive Fourier series for the output waveform and calculate the peak and *RMS* values of the fundamental component and the first seven harmonics.

*Fourier Series:*________________________________________

________________________________________________

| | *n* | *Peak* | *RMS* | | *n* | *Peak* | *RMS* |
|---|---|---|---|---|---|---|---|
| *Fundamental* | 1 | | | *Harmonic # 4* | 9 | | |
| *Harmonic # 1* | 3 | | | *Harmonic # 5* | 11 | | |
| *Harmonic # 2* | 5 | | | *Harmonic # 6* | 13 | | |
| *Harmonic # 3* | 7 | | | *Harmonic # 7* | 15 | | |

3. Derive the expression for the RMS value of the output voltage

*Output RMS voltage derivation*

4. For *circuit # 2*, derive the Fourier expression for the load current. Calculate the peak and RMS values of the fundamental component and the first seven harmonics of the load current.

*Fourier Series:*________________________________________

________________________________________________

| | *n* | *Peak* | *RMS* | | *n* | *Peak* | *RMS* |
|---|---|---|---|---|---|---|---|
| *Fundamental* | 1 | | | *Harmonic # 4* | 9 | | |
| *Harmonic # 1* | 3 | | | *Harmonic # 5* | 11 | | |
| *Harmonic # 2* | 5 | | | *Harmonic # 6* | 13 | | |
| *Harmonic # 3* | 7 | | | *Harmonic # 7* | 15 | | |

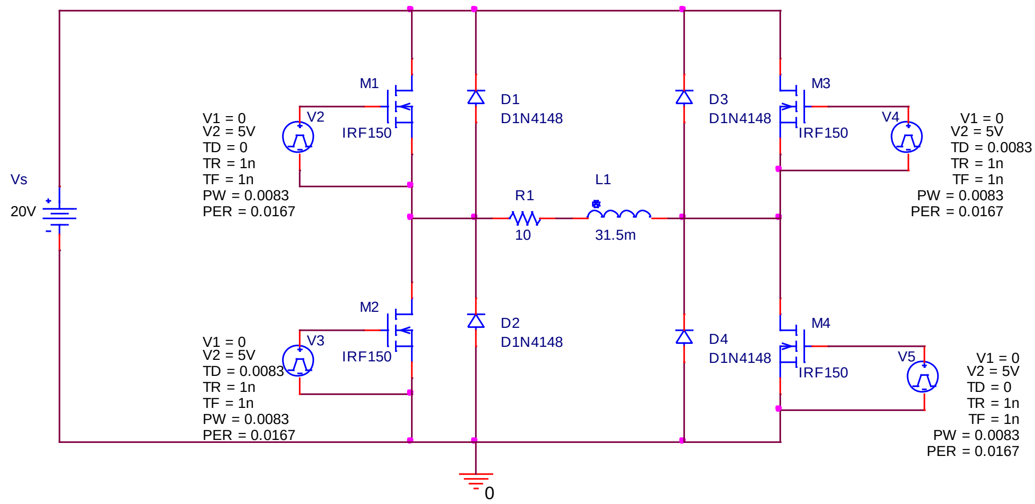


*Figure 2: Bridge Converter with RL Load*

5. Calculate the value of *Total Harmonic Distortion* for the load current using the following expression,

$$THD = \frac{\sqrt{I(peak)^2 - I_1(peak)^2}}{I_1(peak)} \tag{1}$$

where $I_1(peak)$ is the peak of the first harmonic and $I(peak)$ is the peak output current. Use fundamental and seven harmonics from *step* 4 to estimate the value of the peak output current.

$$I(peak) = \sqrt{{I_1}^2 + {I_3}^2 + \ \dots} \tag{2}$$

6. Calculate the approximate value of the output power ($I(rms)^2R$) and also the power corresponding to the fundamental component of the output current. Calculate the percentage of power carried by the fundamental component.

<u>PSpice Simulation:</u>

1. Design both the circuits and observe output voltages and currents.
2. Observe Fourier components in your simulation (*toolbar* → *FFT*) and compare values to your theoretical results.

**<u>Discussion:</u>**

In your lab report, discuss the operation of DC-AC inverter circuits in detail. Also, discuss any discrepancies in your theoretical and simulated results with logical reasoning. A good discussion shows your understanding about circuits and it carries most points in the report.

# EXPERIMENT # 4

# BASIC RESONANT INVERTER

**Note:** Make sure to finish your pre-lab that includes any theoretical derivations or calculations and PSpice simulations before you come to the lab.

**Purpose:** To design and study the characteristics of a basic resonant inverter with MOSFET switches.

## Prelab:

Theoretical Work:

1. For the circuit shown in *figure # 1*, calculate the *natural resonant frequency* ($\omega_d$) and resonant time period ($t_d$)
2. Since MOSFET switching is extremely fast comparing to thyristors, assume that output frequency is same as the natural resonant frequency of the *RLC* circuit and time period of the output is same as $t_d$; hence, both *M1* and *M2* will have switching period to be equal to $t_d$ and both will be ON for half of the time (not simultaneously though).
3. Use the expressions for the output current and capacitor voltage derived in the class and calculate the peak value of current and peak values of the capacitor voltage ($-V_c$ and $V_{c1}$).
4. Draw well-labeled diagrams for the output current and capacitor voltage in the space provided next to *figure # 1*.

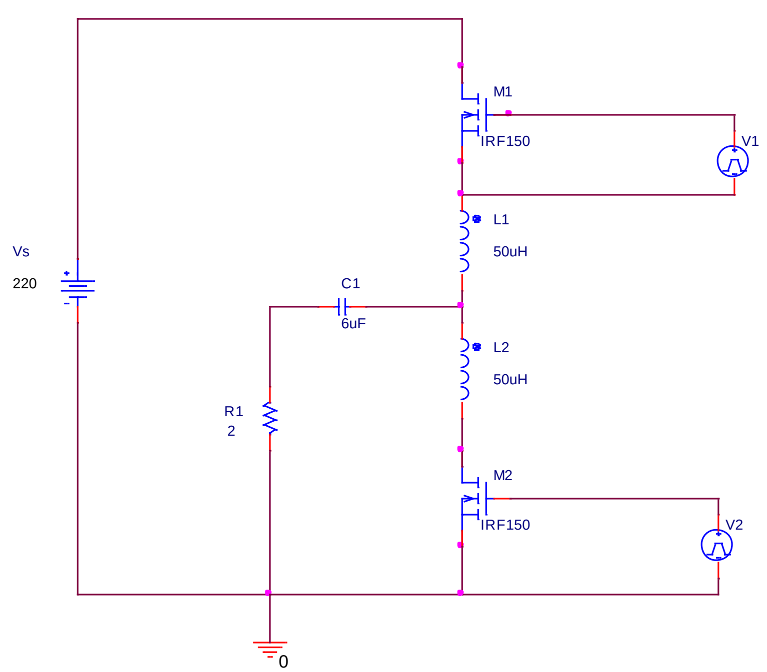


*Figure # 1: Basic Resonant Inverter*

PSpice Simulation:

1. Design the circuit in PSpice. For the square-wave, use 15V and 0V as high and low levels, respectively. Use $t_d$ as period and $t_d/2$ as pulse width. Delay *V2* by $t_d/2$ so that both the transistor switch ON and OFF alternatively.
2. Simulate the circuit and observe output current and capacitor voltage (when they get to the steady-state) and compare them against your theoretical results.

   $I_o(peak)$: ____________; *Vc*: _____________________; $Vc_1$: _________________________

3. Observe the Fourier series for the output current and calculate the total output power and power corresponding to the fundamental component.

$P_o$: ____________________________________; $P_{o1}$: _____________________________________

**Discussion:**

In your lab report, discuss the operation of basic resonant inverters in detail. Also, discuss any discrepancies in your results with possible and logical reasons.

# EXPERIMENT # 5

## HALF-WAVE CONTROLLED RECTIFIER

**Purpose:** To design and study different voltage and current characteristics of a half-wave controlled rectifier. The rectifier circuit will be analyzed for two different types of loads; *R* and *RL*. The effect of a freewheeling diode will also be observed for circuits with *RL* load.

**PSpice Design:**

1. From the data sheet of the thyristor 2N1597, observe the values of turn-off time (*tq*), gate trigger voltage, turn-on time, holding current and voltage drop when the thyristor is ON. This will help you when you will be comparing and discussing the theoretical and simulated results under the *Discussion* section.
2. Design the single-phase half-wave controlled rectifier as shown in *figure 1*. Calculate the time delay to be the time corresponding to the firing angle of 40$^o$ (convert into radians before the calculation of the corresponding firing time). Thyristor is in *thyristr* library

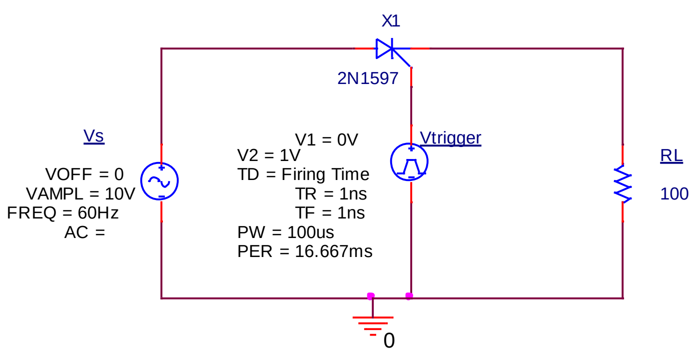


*Figure 1: Half-wave controlled rectifier with resistive load*

3. Split the simulation window in three plots. From top-to-bottom, trace input voltage, voltage across thrysitor, and output voltage. Keep width of the trace lines a little bold and choose different colors for each trace (*right click on the trace → trace property)*. Compare your waveforms against the one learned in the theory to make sure that they are in close proximation.
4. Observe average and RMS values of the output voltage in your simulation and compare them against the calculated values. To determine average and RMS values in PSpice, check out the video through the following link:

   *https://www.pspice.com/resources/video-library/evaluating-measurements-rms-value*

   Make sure that you should only have output voltage waveform when you are tracing the average and RMS voltage values as explained in the video. Note that when you are entering a trace information, as explained in the video, to calculate average or RMS voltage, use the equations from the theory and calculate all the constant values to write down the trace formula in terms of the maximum voltage and constant values. For example, a firing angle of $40^{o}$ will yield the following equation for the average output voltage of the rectifier,

$$v_o(avg) = \frac{Vm}{2\pi}(1+\cos(\alpha)) = \frac{Vm}{6.2832}(1+\cos(40^{\circ})) = \frac{1.7660Vm}{6.2832} \tag{1}$$

An example for the average value from the output waveform is shown in *figure 2*. Trace both average and RMS on the same plot.


*Figure 2: Tracing average value of the waveform (red trace)*

5. Next, replace the load for an *RL* load as shown in *figure 3*, keeping everything else to be the same. Split the simulation window in three plots. From top-to-bottom, trace input voltage, output current, and output voltage. Keep width of the trace lines a little bold and choose different colors for each trace (*right click on the trace → trace property)*. Compare your waveforms against the one learned in the theory to make sure that they are in close proximation.

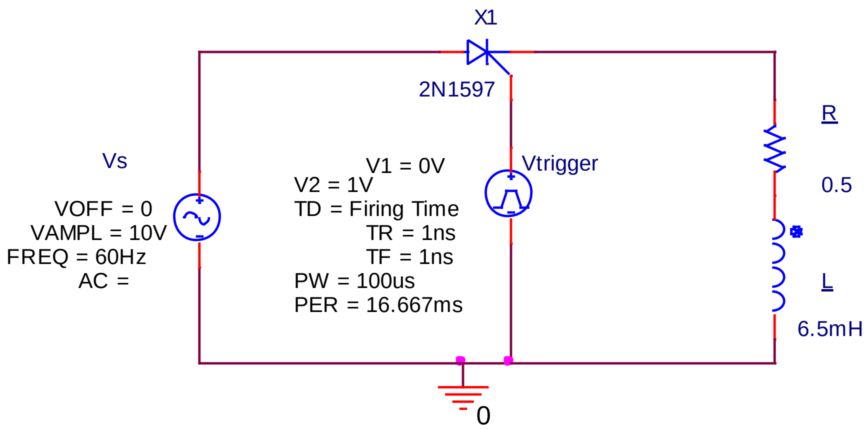


*Figure 3: Half-wave controlled rectifier with RL load*

6. Next, add a freewheeling diode parallel to the load, as shown in *figure 4*, and observe all three waveforms as you did in *step 5*. 1N5817 is a Schottkey diode with low forward voltage drop as compared to a regular silicon rectifier diode.

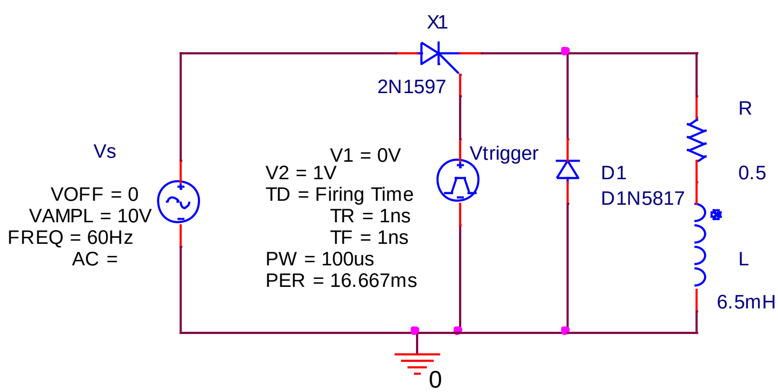


*Figure 4: Half-wave controlled rectifier with RL load and freewheeling diode*

## Discussion:

In your lab report, discuss the operation of each circuit in detail. Compare theoretical and simulated results and discuss sources of discrepancies. Your discussion should reflect your understanding of the subject matter.