UNIVERSITY OF SANTO TOMAS
Faculty of Engineering
ELECTRONICS ENGINEERING DEPARTMENT
First Term, AY 2021 – 2022

EE2315 Lab: Industrial Electronics

Experiment 2: Gate Control Circuits for Triggering SCRs in AC Circuits

Intended Learning Outcomes

1. Be familiarized with the .meas SPICE directive
2. Be familiarized with the operation of different SCR-based rectifier circuits.
3. Be able to plot and measure the voltage across the load.
4. Be able to compute the power delivered to the load at various triggering voltages.

Discussion

An SCR is typically used in controlled rectifier circuits. Fig. 2.1(a) shows the typical circuit configuration for a controlled half wave rectifier circuit. As observed in the figure, the circuit is similar to a typical half wave rectifier circuit, however, an SCR is used for rectification instead of a diode. Unlike in a diode which conducts during the entire positive half cycle, the conduction angle of a controlled rectifier can be adjusted by proper controls circuits.

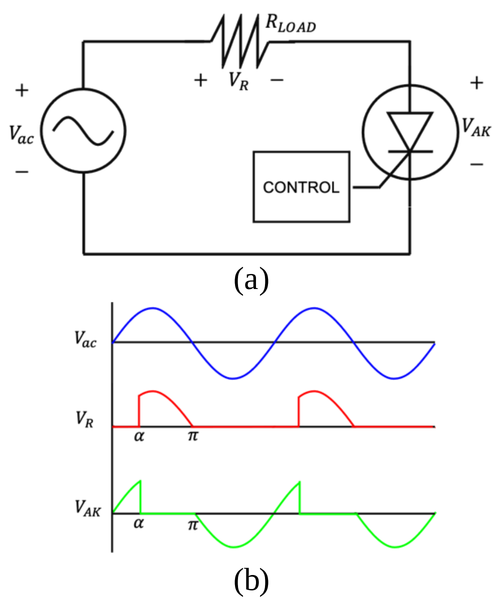

Fig. 2.1 (a) Controlled half wave rectifier and (b) the corresponding voltage waveforms.

By varying the conduction angle of the SCR, the RMS voltage at the load can be varied. Recalling, the RMS value of any periodic voltage $v(t)$ is defined as

$$V_{RMS} = \sqrt{\frac{1}{T}\int_{\tau}^{\tau+T} v^2(t) \cdot dt}$$

and the power delivered to the load is defined as

$$P_{LOAD} = \frac{V_{LOAD_{RMS}}^2}{R_{LOAD}}.$$

This formula shows that, the conduction angle directly affects the RMS voltage the power delivered to the load. In this experiment, three different control circuits will be evaluated.

Materials

LTSpice Simulator

Pre-Experiment Simulation: Introducing the .measure directive.

For this experiment, we need to compute the power delivered to the load. In order to do that, we first need to compute or measure the effective voltage (or the RMS voltage) across the load. To measure the RMS voltage, we need to use a new SPICE directive, the `.measure` (or `.meas`). The .meas directive is used to evaluate user-defined electrical quantities. For example, you can measure a voltage of an AC signal at a specific time. Another example is the measurement of the average and RMS of a signal over a time interval.

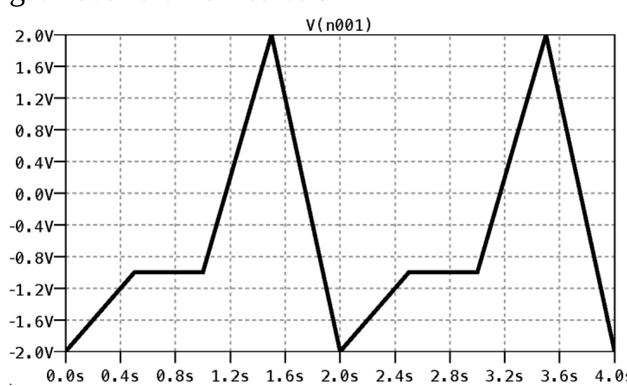

Fig. 2.2 V(n001) Voltage Signal

For this pre-experiment simulation, we will measure the average and RMS values of the signal shown on Fig. 2.2. The signal shown has a period of 2s. The syntax for measuring the average value in LTSpice is

```
.MEAS TRAN <NAME OF MEASUREMENT> AVG <VOLTAGE EXPRESSION>,
```

while for the RMS value, the syntax is

```
.MEAS TRAN <NAME OF MEASUREMENT> RMS <VOLTAGE EXPRESSION>.
```

The syntax shown above will measure the average and RMS value over a time interval of 4 seconds (from t=0s to t=4s). To measure in a specific time interval, an additional information should be added at the end of the syntax.

1. Add a voltage source on your workspace.
2. Right click the voltage source to edit the said component.
3. Under Time Domain Functions, choose PWL(t0, V0, t1, V1…), click on the “Additional PWL points” and enter the values shown on Fig. 2.3.

Time[s]	Value[V]
0	-2
0.5	-1
1	-1
1.5	2
2	-2
2.5	-1
3	-1
3.5	2
4	-2

Fig. 2.3 PWL points of V1

4. Run a 4-second transient simulation using the SPICE directive

```
.TRAN 4
```

and plot the voltage generated by V1. It should be the same as the plot shown on Fig. 2.2.

5. To measure the average and RMS voltage for the whole time interval of the signal (from t=0s to t=4s), add the following SPICE directives and re-run the transient simulation.

```
.MEAS TRAN V_AVE AVG V(n001)
.MEAS TRAN V_RMS RMS V(n001)
```

Interpretation of the above SPICE directive:

Measure the average value of V(n001) and name the result V_AVE; Measure the RMS value of V(n001) and name the result V_RMS.

6. To view the measurement result, use the keyboard shortcut CTRL+L (CMD+L for MacOS) to open the circuit’s .log file.

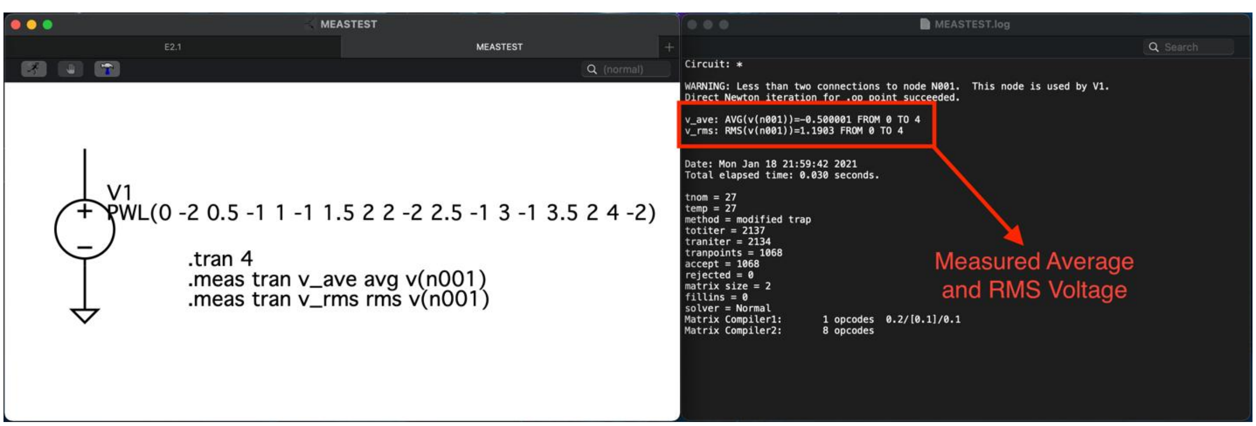

Fig. 2.4 The .log file

7. An example of .meas directive shown below will be used for measuring the average and RMS values of V(n001) at a specific interval.

```
.meas tran v_ave avg v(n001) trig v(n001) val=0 cross=1 targ v(n001) val=0 cross=2
.meas tran v_rms rms v(n001) trig v(n001) val=0 cross=1 targ v(n001) val=0 cross=2
```

Interpretation of the above SPICE directive:

Measure the average value of V(n001) from the time when V(n001) is **crossing 0V** for the **first time** until the time when V(n001) is **crossing 0V** for the **second** time and name the result V_AVE. For .meas directive, TRIG indicates the start of measurement and TARG indicates the end of measurement.

8. Again, the measurement result will be in the .log file.

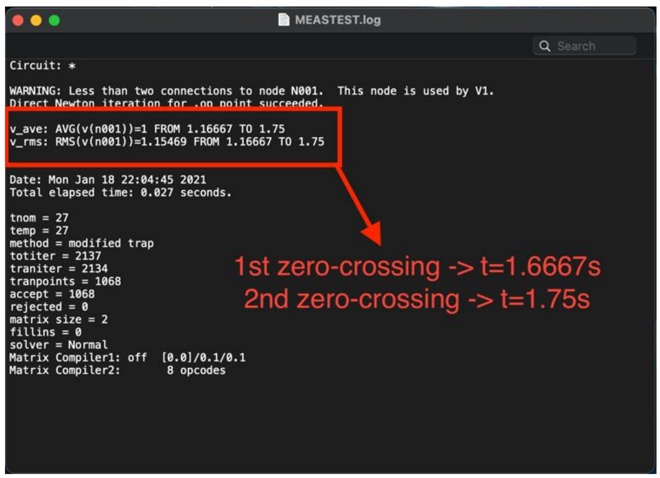

Fig. 2.5 The .log file using the directive from step 7

9. To verify this time interval, the 1st and 2nd zero-crossing occurrences of the signal are measured in the plot using cursor.

Fig. 2.6 1st and 2nd zero-crossing occurrences

Experiment Proper

Part A.

1. Construct the experimental circuit shown in Fig. 2.7. Label the terminals across the resistor R_LOAD as `LOAD+ and LOAD-`. Also, take note that the values of R4 and R3 are parameters `{500k-RX-1f}` and `{RX+1f}`, respectively. With this arrangement, R4 and R3 will simulate the sweeping action of a $500k\Omega$ potentiometer.
   *Do not forget to include the model file of the SCR.

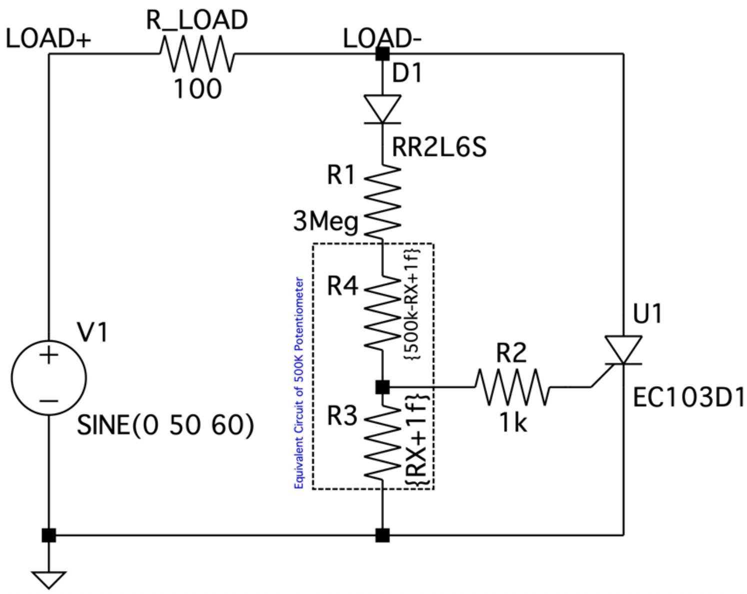

Fig. 2.7 Experimental circuit for Part A

2. Edit the voltage V1 by right clicking the component. Under Time Domain Function, select “SINE(…)” function and set the **amplitude to 50V** and **frequency to 60Hz**.
3. Run a 100ms transient simulation while sweeping the parameter RX from **0 to 500kΩ with a step size of 50kΩ** using the SPICE directive shown.

```
.TRAN 100m
.STEP PARAM RX 0 500K 50K
```

4. **Using separate plot panes**, plot the voltages V(LOAD+,LOAD-), V(LOAD+), and V(LOAD-). These waveforms are the load voltage, supply voltage, and anode to cathode voltage, respectively.
5. Use the SPICE directive shown to measure the RMS voltage of the load per step and re-run the transient simulation.

```
.MEAS TRAN V_LOAD RMS V(LOAD+,LOAD-) TRIG V(LOAD+) VAL=0 CROSS=7 TARG V(LOAD+) VAL=0 CROSS=9
```

6. The measurement result will be located in the .log file. You can access this file using the keyboard shortcut CTRL+L (CMD+L). Record the RMS voltage per step in Table 2.1.
7. Compute and record for the power delivered to the load using the formula

$$P_L = \frac{V_{RMS}^2}{R_L}.$$

8. Measure the firing delay angle (α) by including an additional SPICE directive show and re-running the transient simulation.

```
.MEAS TRAN T_DELAY TIME TRIG V(LOAD+) VAL=0 CROSS=7 TARG V(LOAD+,LOAD-) VAL=1.5 CROSS=7
```

9. Convert the measured T_DELAY into an angle using the formula

$$\alpha = \frac{t_{delay}}{T} \cdot 360°$$

For the given circuit conditions,

$$\boxed{\alpha = t_{delay} \cdot 60Hz \cdot 360°}$$

10. Write your observations for Part A.

Part B.

1. Construct the experimental circuit shown in Fig. 2.7. The value of R2 is a parameter `{RX+1n}`. R2 will simulate a 400kΩ potentiometer. Set the voltage source V1 to be SINE(…) with an amplitude of 50V and frequency of 60Hz.
   *Do not forget to include the model file of the SCR.

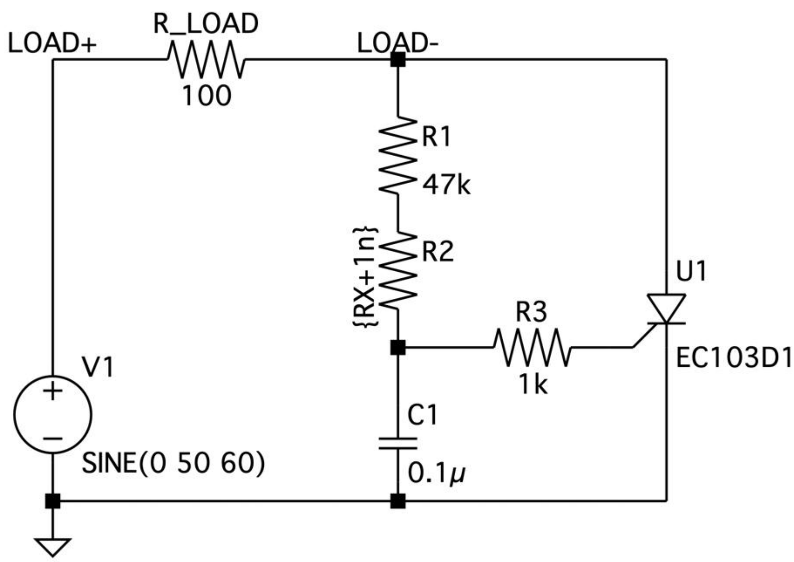

Fig. 2.8 Experimental circuit for Part B

2. Run a **100ms transient simulation** while sweeping the parameter RX from **0 to 400kΩ with a step size of 50kΩ** using the SPICE directive shown.

```
.TRAN 100m
.STEP PARAM RX 0 400K 50K
```

3. Using separate plot panes, plot the voltages V(LOAD+,LOAD-), V(LOAD+), and V(LOAD-). These waveforms are the load voltage, supply voltage, and anode to cathode voltage, respectively.
4. Use the SPICE directive shown to measure the RMS voltage of the load per step and re-run the transient simulation.

```
.MEAS TRAN V_LOAD RMS V(LOAD+,LOAD-) TRIG V(LOAD+) VAL=0 CROSS=7 TARG V(LOAD+) VAL=0 CROSS=9
```

5. The measurement result will be located in the .log file. You can access this file using the keyboard shortcut CTRL+L (CMD+L). Record the RMS voltage per step in Table 2.2.
6. Compute and record for the power delivered to the load using the formula

$$P_L = \frac{V_{RMS}^2}{R_L}.$$

7. Measure the firing delay angle (α) by including an additional SPICE directive show and re-running the transient simulation.

```
.MEAS TRAN T_DELAY TIME TRIG V(LOAD+) VAL=0 CROSS=7 TARG V(LOAD+,LOAD-) VAL=1.5 CROSS=7
```

8. Convert the measured T_DELAY into angle using the formula

$$\alpha = \frac{t_{delay}}{T} \cdot 360°$$

For the given circuit conditions,

$\boxed{\alpha = t_{delay} \cdot 60Hz \cdot 360°}$

9. Write your observations for Part B.

Part C.

1. Construct the experimental circuit shown in Fig. 2.8. The value of R2 is a parameter `{RX+1n}`. R2 will simulate a 100kΩ potentiometer. Set the voltage source V1 to be SINE(…) with an amplitude of 50V and frequency of 60Hz.
   *Do not forget to include the model file of the SCR.

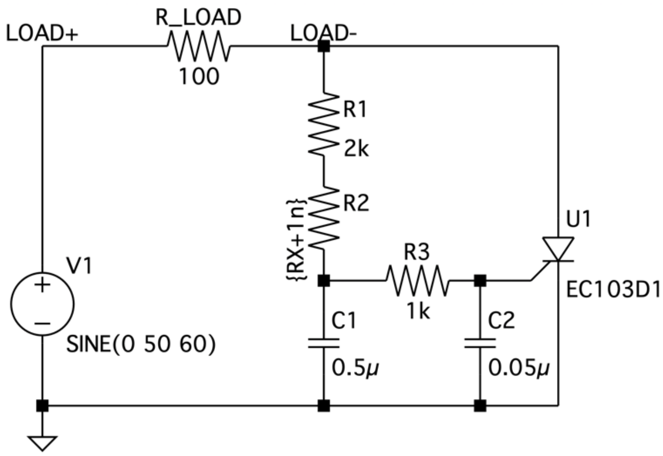

Fig. 2.8 Experimental circuit for Part C

2. Run a 100ms transient simulation while sweeping the parameter RX from **0 to 100kΩ with a step size of 10kΩ** using the SPICE directive shown.

```
.TRAN 100m
.STEP PARAM RX 0 100K 10K
```

3. Using separate plot panes, plot the voltages V(LOAD+,LOAD-), V(LOAD+), and V(LOAD-). These waveforms are the load voltage, supply voltage, and anode to cathode voltage, respectively.
4. Use the SPICE directive shown to measure the RMS voltage of the load per step and re-run the transient simulation.

```
.MEAS TRAN V_LOAD RMS V(LOAD+,LOAD-) TRIG V(LOAD+) VAL=0 CROSS=7 TARG V(LOAD+) VAL=0 CROSS=9
```

5. The measurement result will be located in the .log file. You can access this file using the keyboard shortcut CTRL+L (CMD+L). Record the RMS voltage per step in Table 2.3.
6. Compute and record for the power delivered to the load using the formula

$$P_L = \frac{V_{RMS}^2}{R_L}.$$

7. Measure the firing delay angle (α) by including an additional SPICE directive show and re-running the transient simulation.

```
.MEAS TRAN T_DELAY TIME TRIG V(LOAD+) VAL=0 CROSS=7 TARG V(LOAD+,LOAD-) VAL=1.5 CROSS=7
```

8. Convert the measured T_DELAY into an angle using the formula

$$\alpha = \frac{t_{delay}}{T} \cdot 360°$$

For the given circuit conditions,

$\boxed{\alpha = t_{delay} \cdot 60Hz \cdot 360°}$

9. Write your observations for Part A.

· End of Experiment 2 ·

Group Report Format:

Group No.: ____________________
Group Members: ________________
____
____

Part A:
Plot of Part A Simulations (Step 4):

<Insert Plot Here>

Table 2.1

Step	RX	V_{RMS}	P_L	t_{delay}	α
1	0				
2	50k				
3	100k				
4	150k				
5	200k				
6	250k				
7	300k				
8	350k				
9	400k				
10	450k				
11	500k				

Comment on the results for part A.

<Insert Comments Here>

Part B:

Plot of Part A Simulations (Step 4):

<Insert Plot Here>

Table 2.2

Step	RX	V_{RMS}	P_L	t_{delay}	α
1	0				
2	50k				
3	100k				
4	150k				
5	200k				
6	250k				
7	300k				
8	350k				
9	400k				

Comment on the results of Part B:

<Insert Comments Here>

Part C:
Plot of Part A Simulations (Step 4):

<Insert Plot Here>

Table 2.3

Step	RX	V_{RMS}	P_L	t_{delay}	α
1	0				
2	10k				
3	20k				
4	30k				
5	40k				
6	50k				
7	60k				
8	70k				
9	80k				
10	90k				
11	100k				

Comment on the results of Part C:

<Insert Comments Here>

Conclusion:

<Inset Conclusion Here>