Brunel University London

重庆邮电大学

CHONGQING UNIVERSITY OF POSTS AND TELECOMMUNICATIONS

OrCad Technical Report

Team	Autumn, 2022
Module	**EE1616 Electronics Workshop**
Class	34092102
Brunel ID	2161047
CQUPT ID	2021215069
Name	Xukang Liu
Tutor	Zhipeng Wang

Introduction and aims:

To familiar with PSPICE simulation software. And reinforce the understanding of some basic analogue circuits. PSPICE means simulation program with integrated circuit emphasis. It is a general-purpose circuit program that simulates electronic circuits. We need know how to work with the software. Simulate and calculate some circuits.

Task description:

1. Create a new project file. Add some components, such as resistance, source, and ground. Connect them with wire. Simulate it and calculate the total resistance.
2. Practice drawing AC circuits. Use VSIN, resistance and inductance. Connect them with series. Simulate it, find the voltage and the current. Find out the phase angle between the supply voltage and current.
3. Use inductance and capacitor to create RLC circuit. Find out the total impedance of the circuit and the phase angle between the total current and the supply voltage.
4. Create a peak detector. Use diode, capacitor, and resistance. Create a complex and interested circuit. Detect the input and output voltage. Devise the magnitude of the capacitor and observe the

difference phenomenon between the different magnitude of the capacitor.

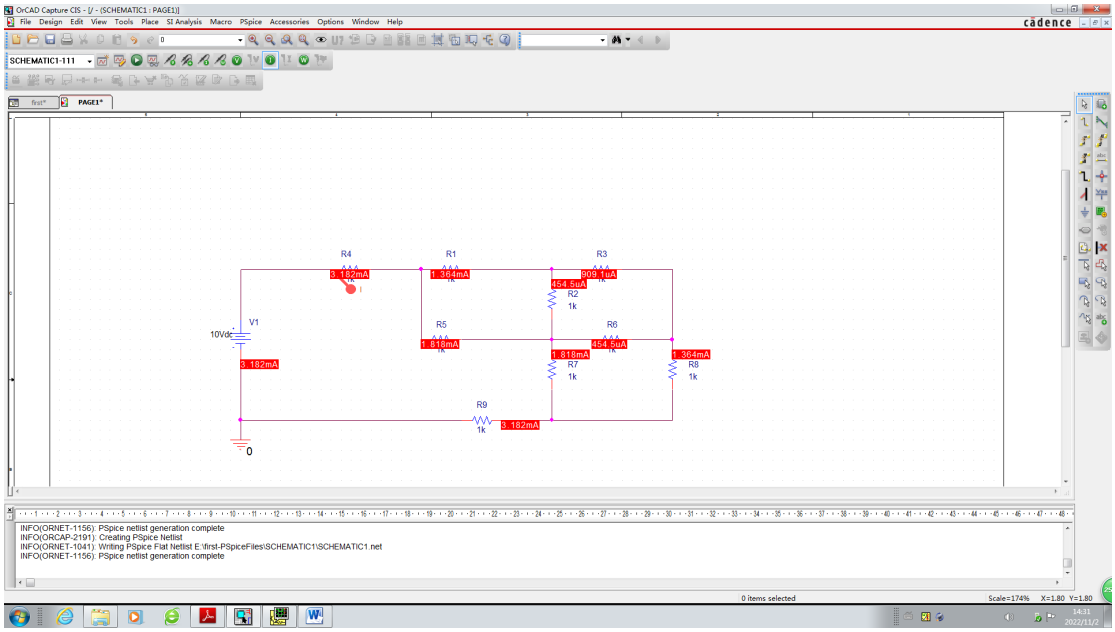
Experimental method:

1. Adjust some parameter to create a new project file.
2. Correct use of current marker and voltage marker. Start the circuit analysis again. Display the trace.
3. In some reactive components, the phase angle is not zero. Correctly use the relationship between period and abscissa difference. Calculate the total impedance and the phase angle.

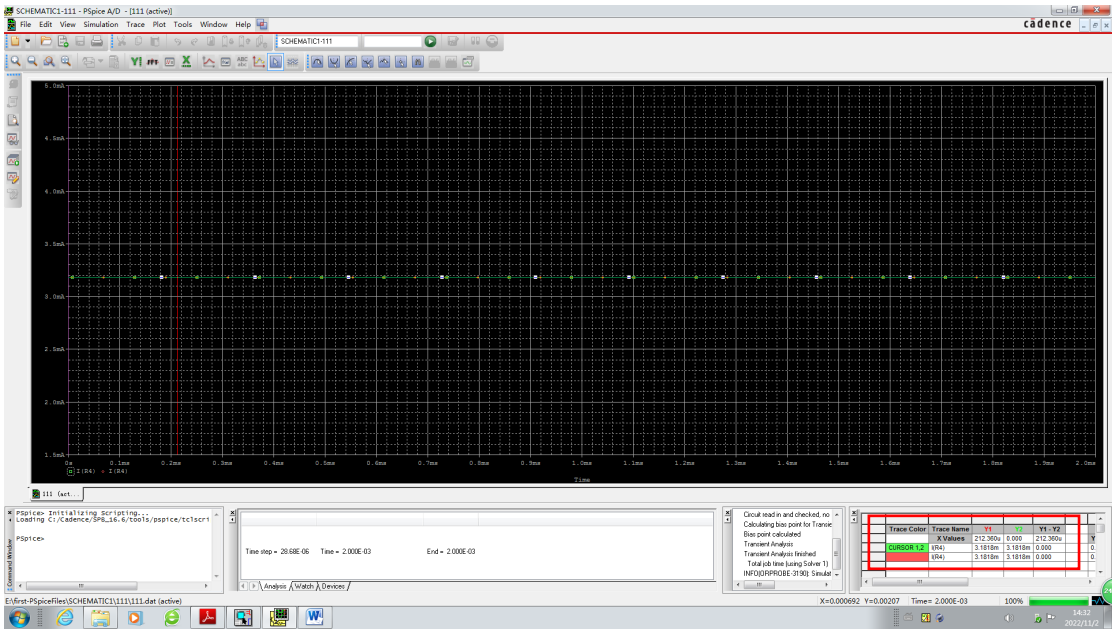
Result and observations:

Task1:

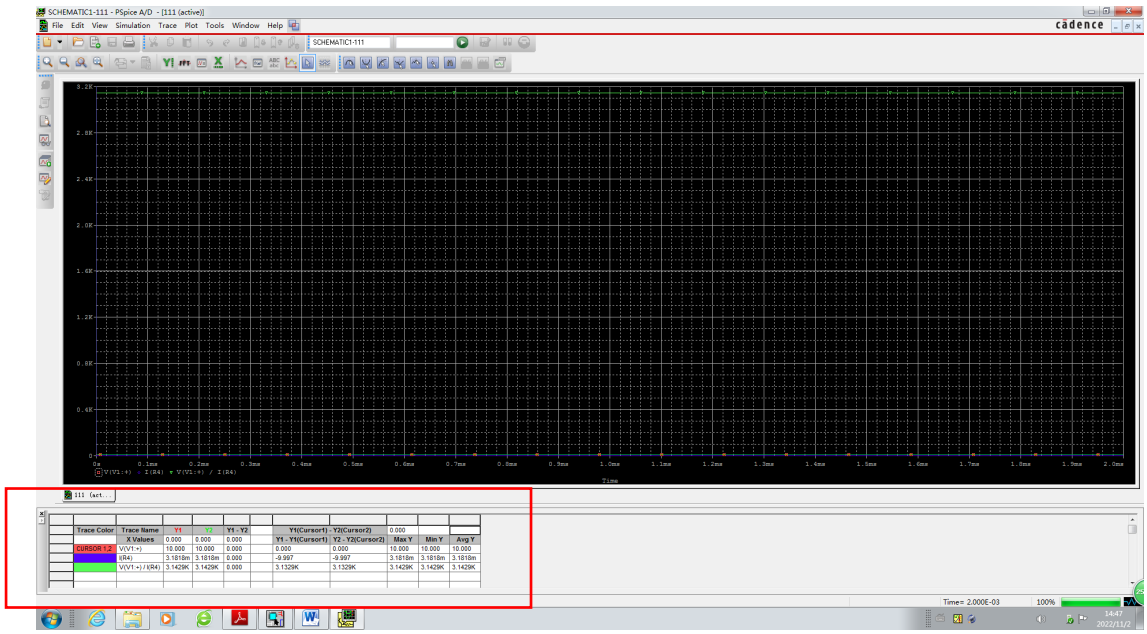
This is my circuit diagram.



The simulate result, and record the value:



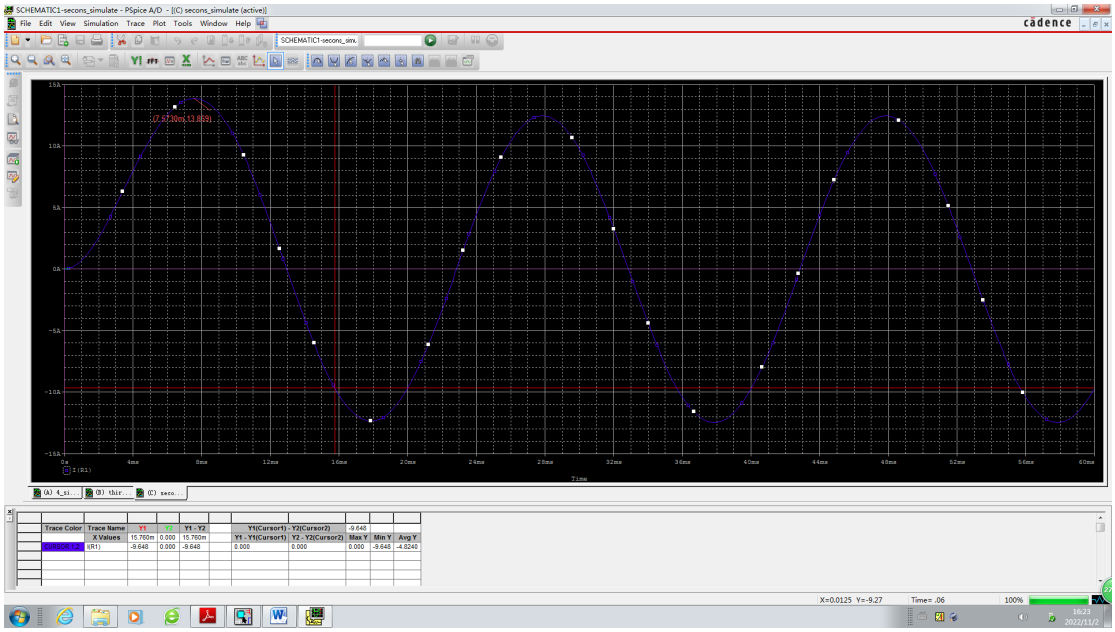
Use the ohm's law, I can calculate the total resistance: $R = \frac{E}{I}$.



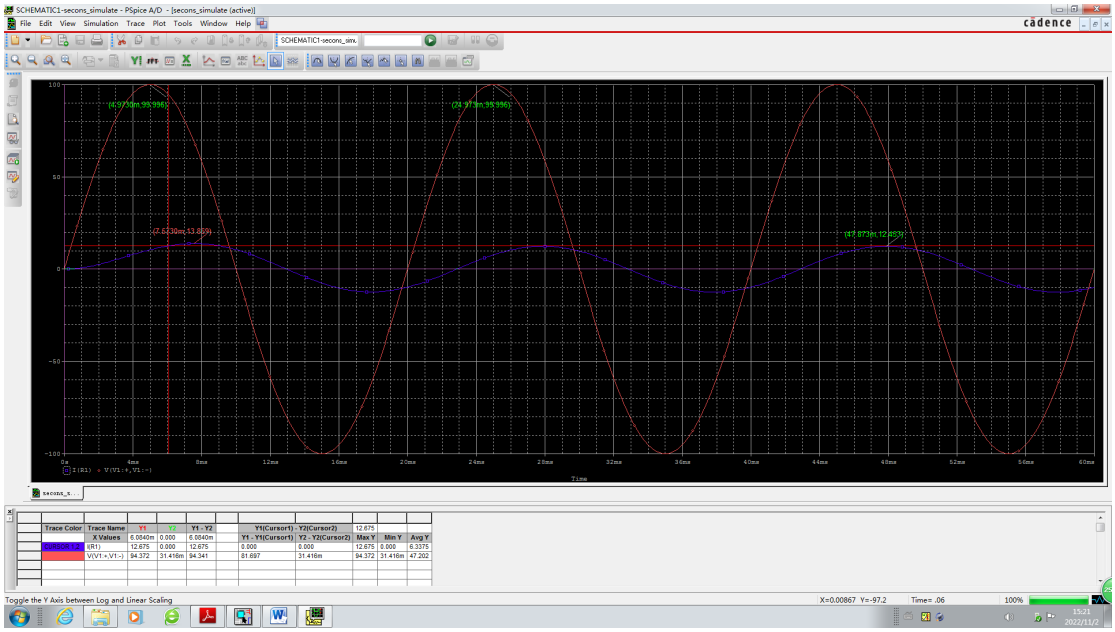
The current and the supply voltage I have measured in the last step. And then I add "tracestep" which is "V(V1:+)/I(R4)" to calculate the total resistance. Obviously, this result has been shown on this picture, the value is 3.1429kΩ.

Task2:

This is the circuit current diagram, I can find that the maximum current is 13.859A.



This diagram includes current and voltage, the voltage I set is 100v. In addition, the frequency which I set is 50HZ. Thus, the period $T = 20ms$.



$$\frac{7.5730ms - 4.9730ms}{20ms} = \frac{\theta}{2\pi}$$

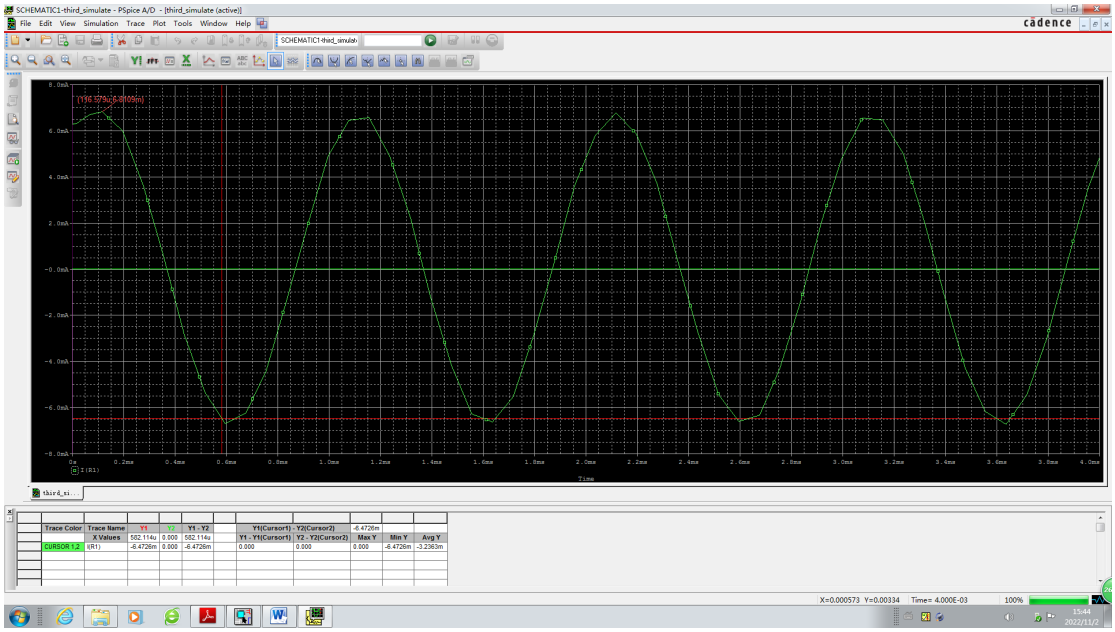
Thus $\theta = 0.26\pi = 46.8°$.

Thus, the phase angle is 46.8°.

Task3:

I set the supply voltage is 10v and the frequency is 1kHZ.

The current diagram is:



The voltage diagram is:

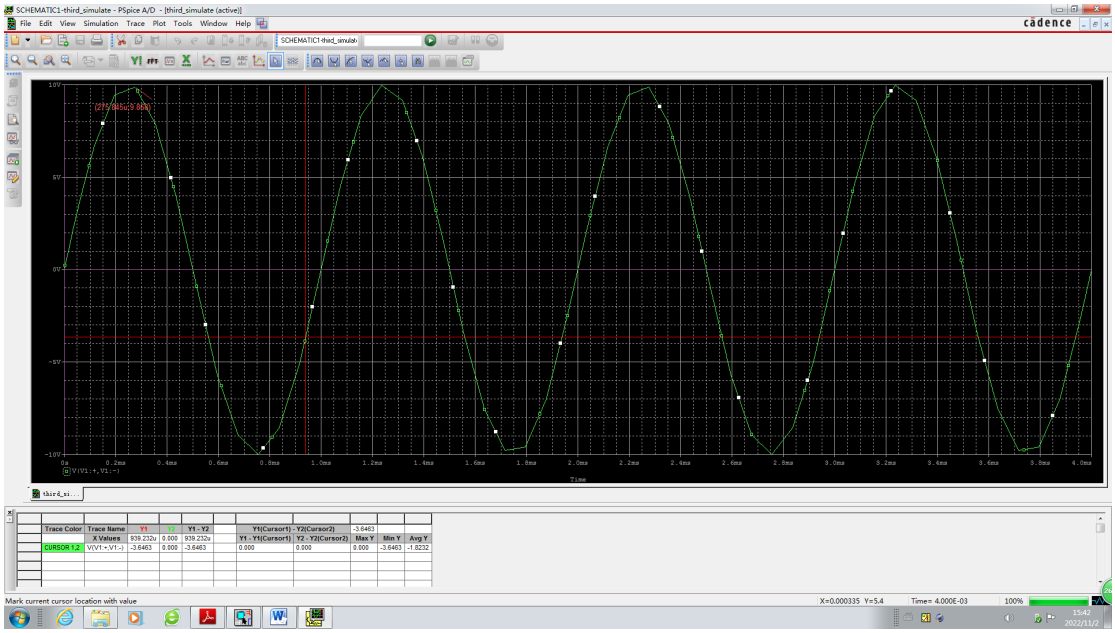
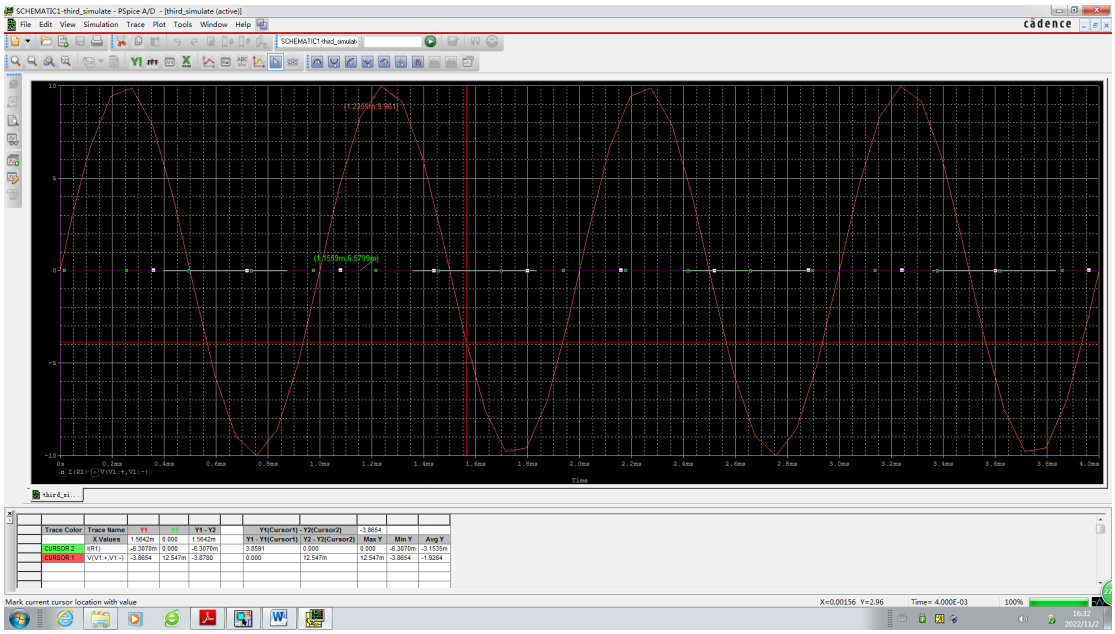


Image of voltage and current combination:



The maximum voltage is 10v and the maximum current is 6.8109mA.

The period is 1ms.

The **phase angle** is:

$$\frac{1.2359ms - 1.1559ms}{1ms} = \frac{\theta}{2\pi}$$

Thus $\theta = 0.16\pi = 28.8°$

Thus, the phase angle is 28.8°.

Because the phase angle of supply voltage is 0°. Thus, the phase angle of the current is 28.8°. $V = V_m\angle 0°$ $I = I_m\angle 28.8°$.

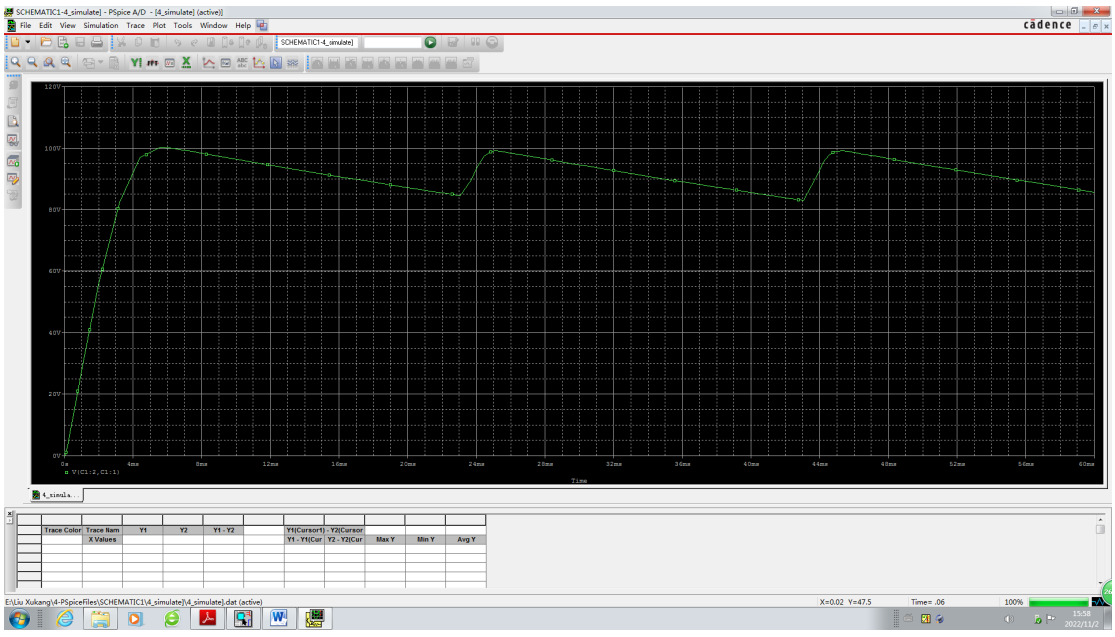
Thus, the **total impedance** is:

$$Z_T = \frac{V}{I} = \frac{V_m\angle 0°}{I_m\angle 28.8°} = 1.468 \times 10^3 \Omega\angle -28.8°$$

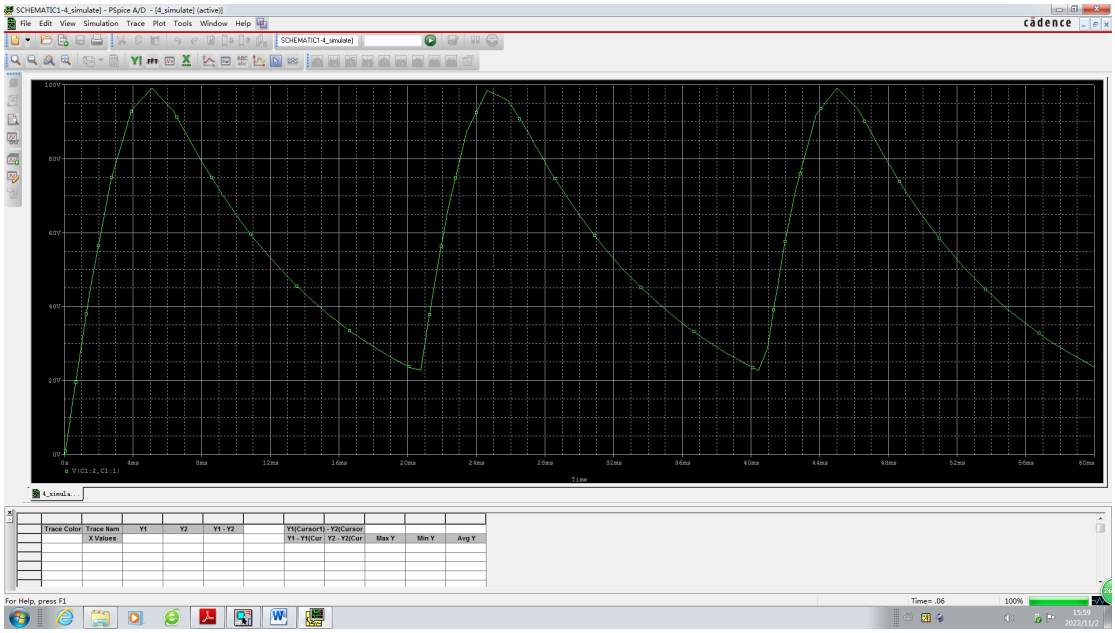
Task 4:

This task, I need create a peak detector.

If the capacitor is $100\mu F$, the diagram is:



If the capacitor is $10\mu F$, the diagram is:



Conclusion:

In the lecture, I have a preliminary understanding of the OrCad. The software has many powerful functions, it can simulate circuit so that can help me to understand some knowledges which I learned in EE1618. In the lecture, I confused about the calculation of the phase angle. After I discuss with other students, I get the solution.

Overall, the lecture helps me a lot, it can combine the theory we have learned with practice. It can help us better understand the circuit.