

Section	
Bench No.	

# ECE110 Introduction to Electronics

## Pre-lab 5: Time-varying signal

Name:

南雅夫

Student ID:

3230112117

Instructor:

李斯

Date:

Oct. 29, 2023

Teammate/Student ID:

3230112155

This part is reserved for your instructor

Score	
Instructor Signature	
Date	

## Pre-lab 5: Time-varying signal

---

### Laboratory Outline

In experiment #3, we have already learned the simulation software: LTSpice, using simulation software to simulate circuits and study their characteristics. At this time, we will continue to learn how to use this software to simulate an oscillating circuit. By learning analog circuits, it can be more convenient for us to operate in experiment #5.

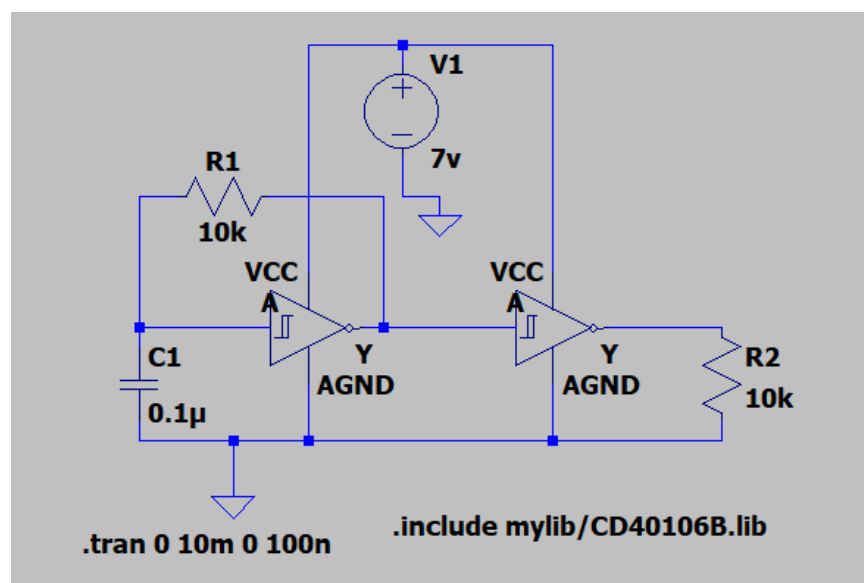
### Analog Circuit Design and Simulation

1. Add **CD40106B.lib** to the **library** of LTSpice:

Normally, the default path for LTSpice is:

C:\Users\username\AppData\Local\LTspice\lib\sub, suggest creating a subfolder **mylib** under sub and copying **CD40106B.lib** into it.

2. Copy **schmittti.asy** (custom schematic symbol) to the folder:  
C:\Users\username\AppData\Local\LTspice\lib\sym\AutoGenerated. Then the component can be searched in LTSpice.
3. Open LTSpice and draw the schematic diagram, as shown in the following figure (add an additional layer of buffer, and the load on the output terminal needs to be a bit larger. V1=7V R1=10K R2=10K C1=0.1uF is used here):

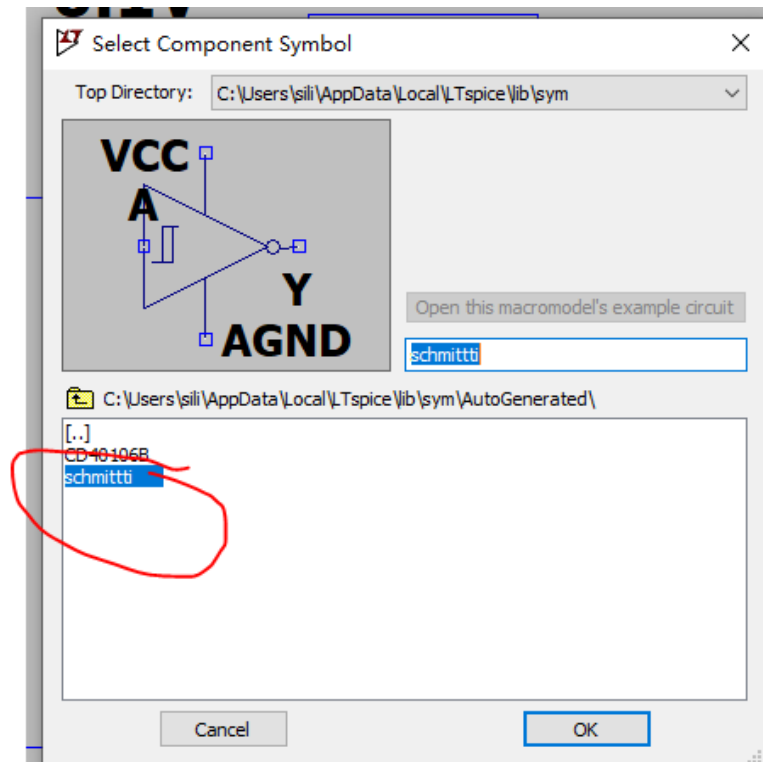


*Figure 1: Schematic diagram of oscillator.*

Select **CD40106B** components:

(1) Select from the menu bar: Edit-Component

Input: schmitti



*Figure 2: Insert schmitti icon.*

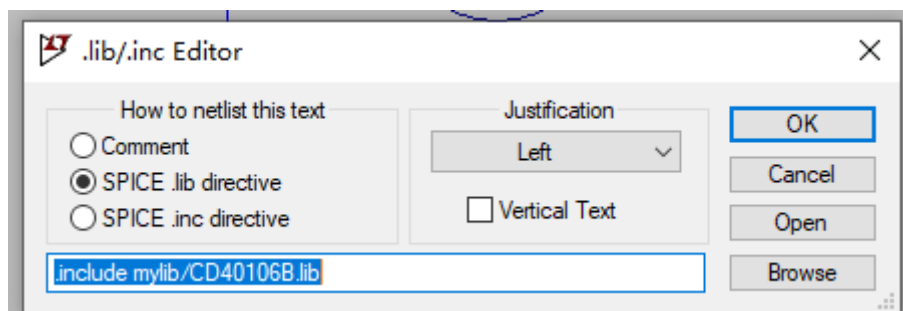
(2) Reference library file: **CD40106B.lib**

Select from the menu bar: Edit-Spice



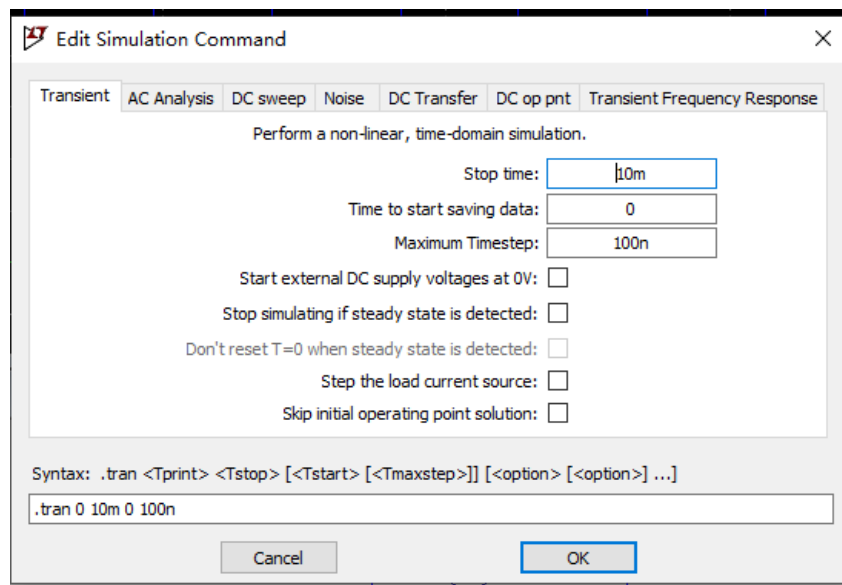
Directive

Input command: **.include mylib/CD40106B.lib**



*Figure 3: Reference library file.*

(3) Edit Simulation Command:

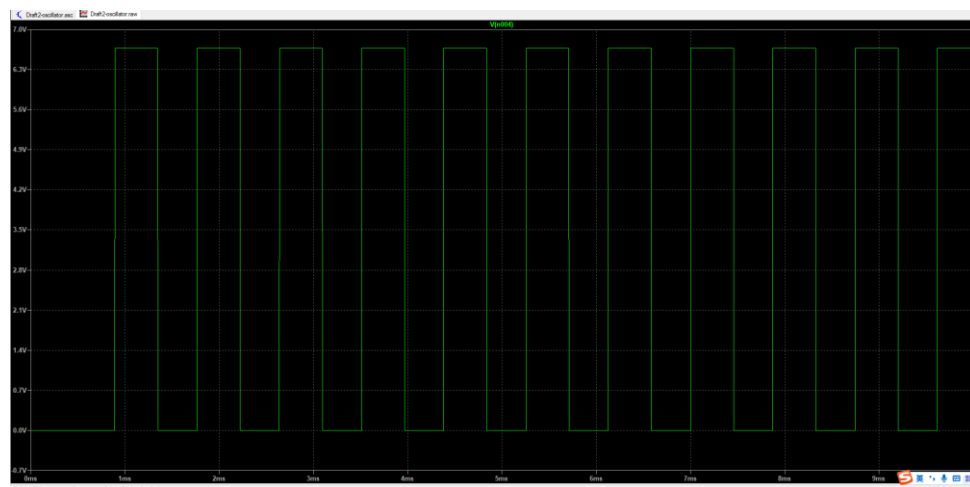


*Figure 4: Edit Simulation Command.*

(4) Run the simulation to obtain the image.

If the speed is slow after starting the simulation, press ESC multiple times to pop up the simulation image.

Click the desired point on the schematic diagram with the mouse, and the simulation curve will appear as follows:



*Figure 5: Simulation curve.*

Please simulate the oscillation circuit according to the above process. Please record the circuit diagram and the simulation curves of points a, b, and c separately, attached below.

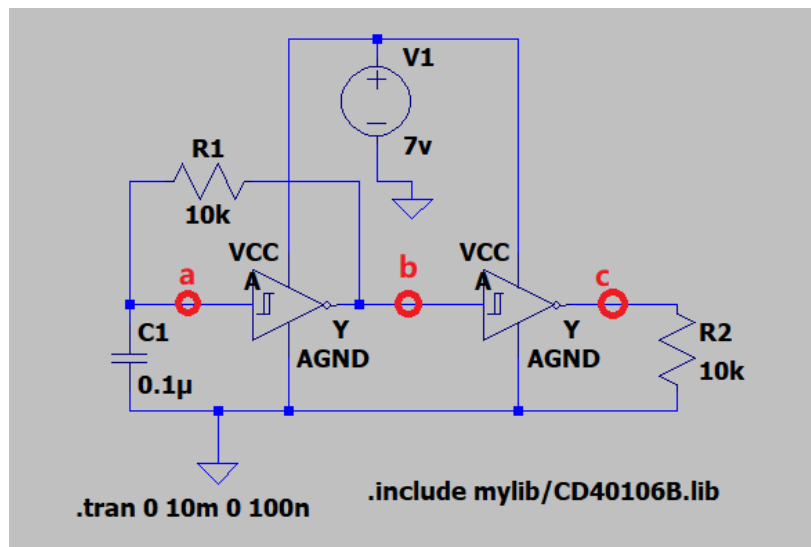
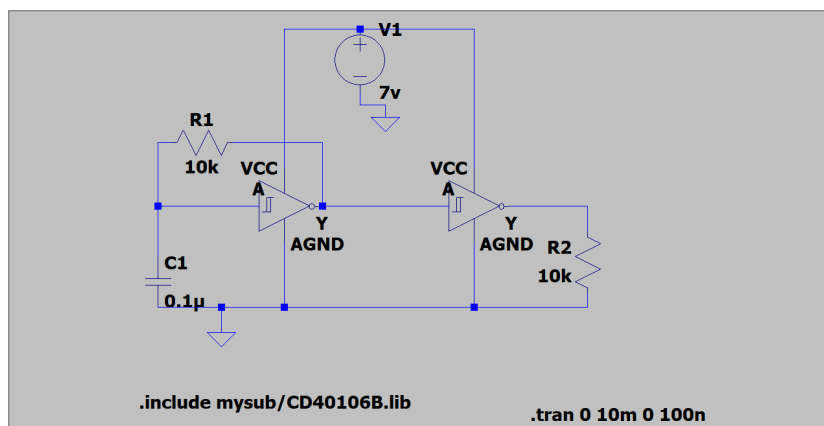
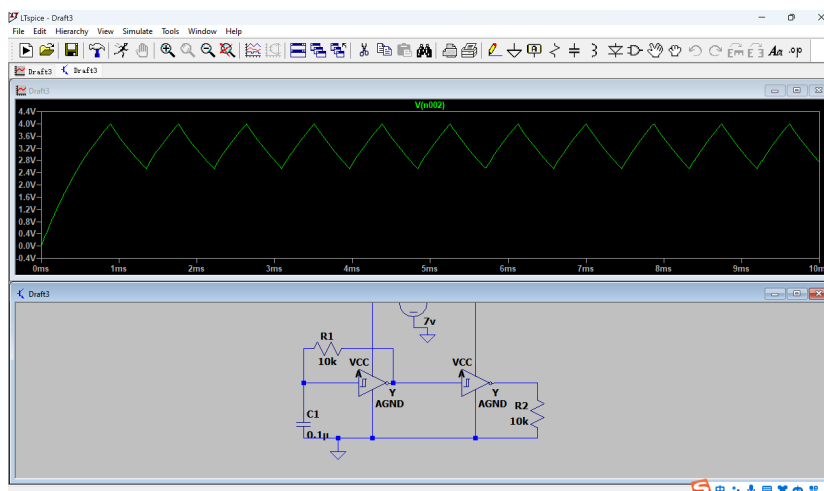


Figure 6: Schematic diagram of oscillator.

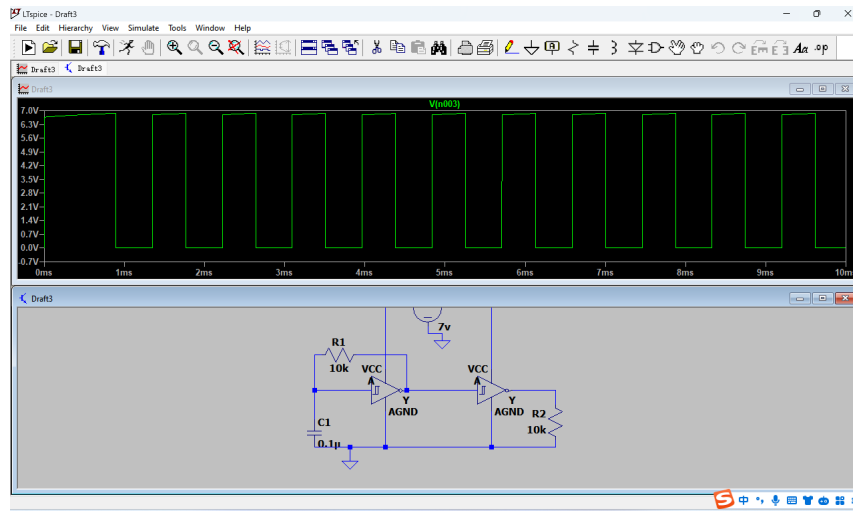
Q1: Please attach the circuit diagram you have built:



Q2: Please attach the simulation curves of point a:



Q3: Please attach the simulation curves of point b:



Q4: Please attach the simulation curves of point c:

