Section	
Bench No.	

ECE110 Introduction to Electronics

Pre-lab 5: Time-varying signal

Name: _____

Student ID: _____

Instructor: _____

Date: _____

Teammate/Student ID: _____

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. If you are using Mac OS, the path is: /Users/*username*/Library/Application Support/LTspice/lib/sub.

    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. The path in Mac OS is:
    /Users*/username*/Library/Application Support/LTspice/lib/sym/AutoGenerated. If there is no AutoGenerated file, you can create one by yourself.

2. 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):
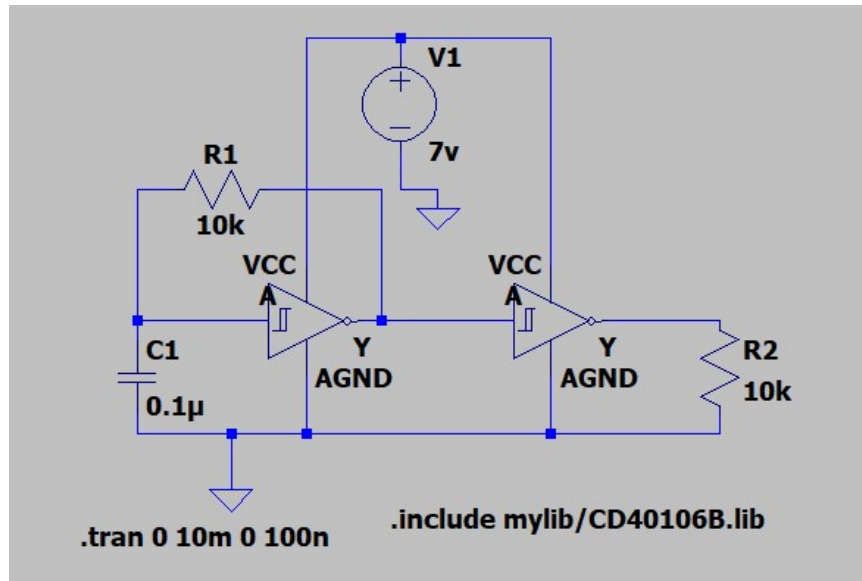
Figure 1: Schematic diagram of oscillator.

Select **CD40106B** components:

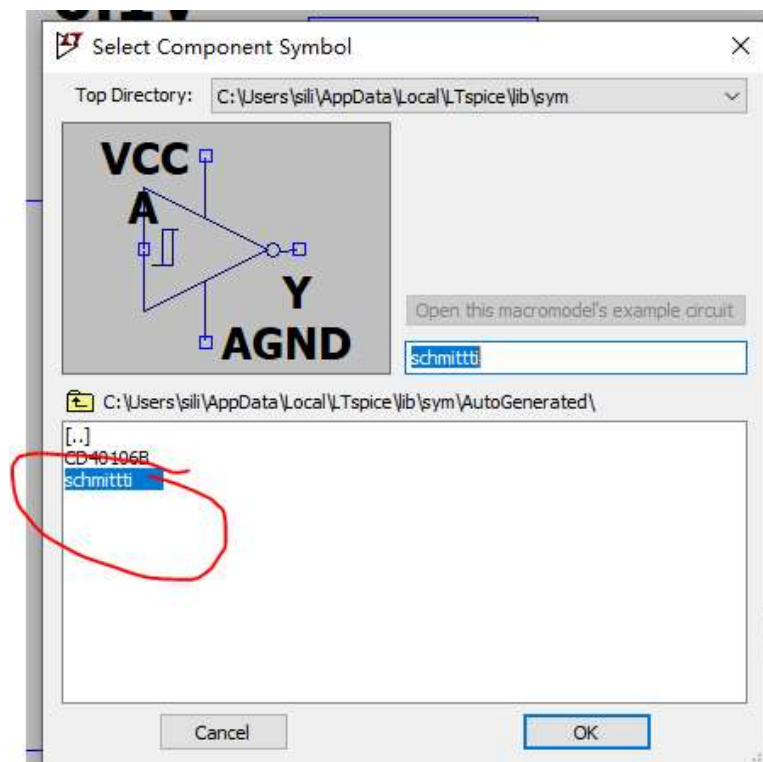(1) Select from the menu bar: Edit-Component

Input: schmitti

Figure 2: Insert schmitti icon.

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

Select from the menu bar: Edit-Spice ⌐ .ᵒp   Directive

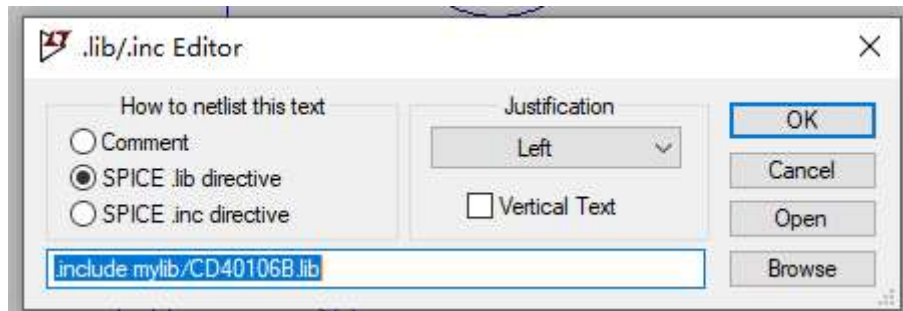Input commond: **.include mylib/CD40106B.lib**

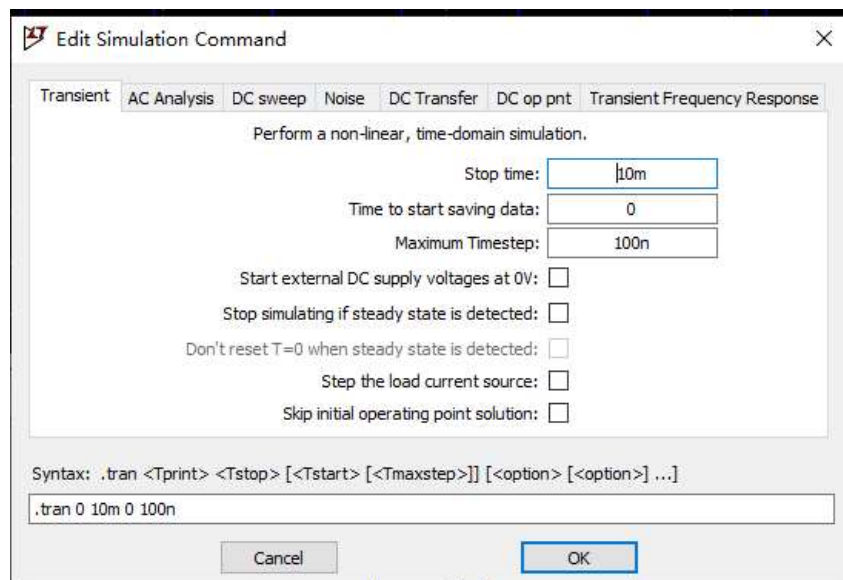*Figure 3: Reference library file.*

(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

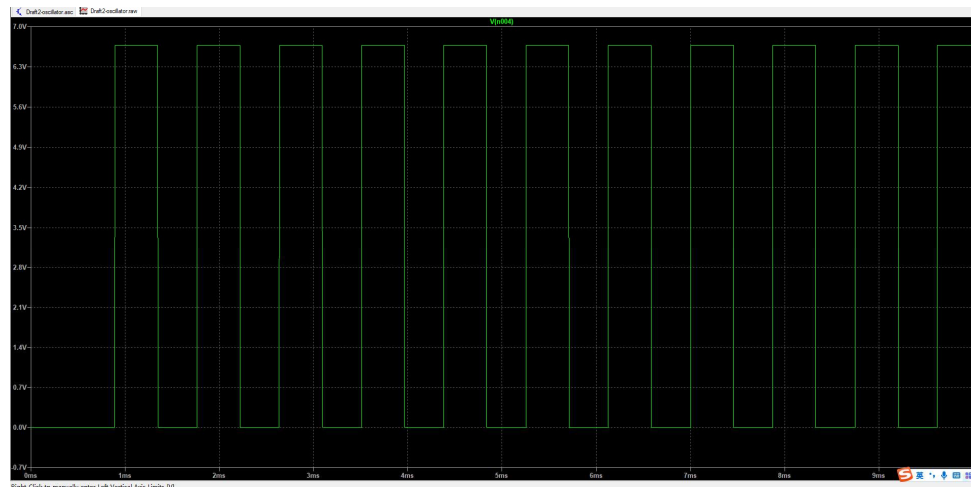simulation curve will appear as follows:

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.
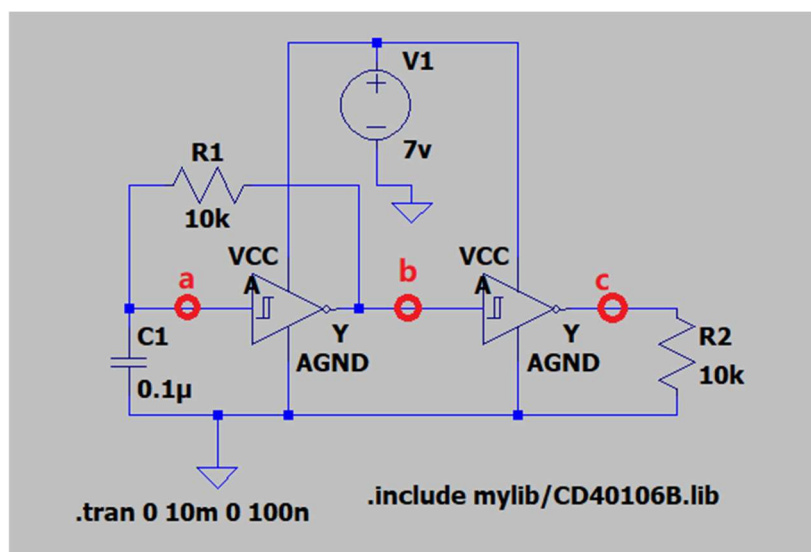

Figure 6: Schematic diagram of oscillator.

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: