HOW TO MULTISIM

Multisim is circuit simulation software. In this course, you will be tasked with building and simulating some circuits using Multisim as part of your homework.
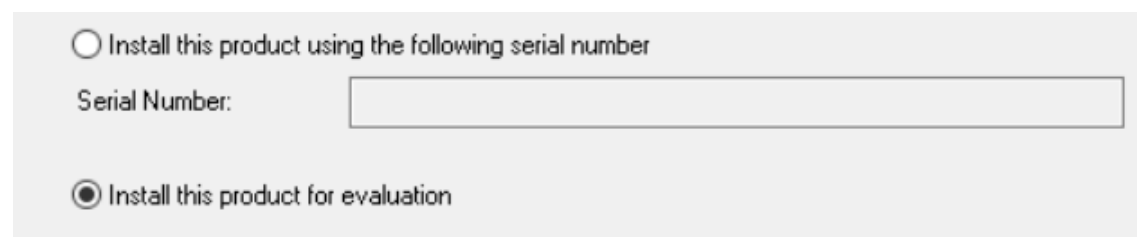
Installation

Go to https://www.ni.com/zh-cn/shop/electronic-test-instrumentation/application-software-for-electronic-test-and-instrumentation-category/what-is-multisim/multisim-education.html

 And click on "下载教学版软件"

Unzip the downloaded zip archive and double-click setup.exe to install the simulation software. During the installation process, choose "Install this product for evaluation". Uncheck the update searching function is recommended. You also don't need to install ELVISmx. You can leave everything else untouched.

Search for important messages and updates on the National Instruments products you are installing. To perform this search, your IP address will be collected in accordance with the National Instruments Privacy Policy.

Note: You will be given the opportunity to select the updates you want to install.

Privacy Policy
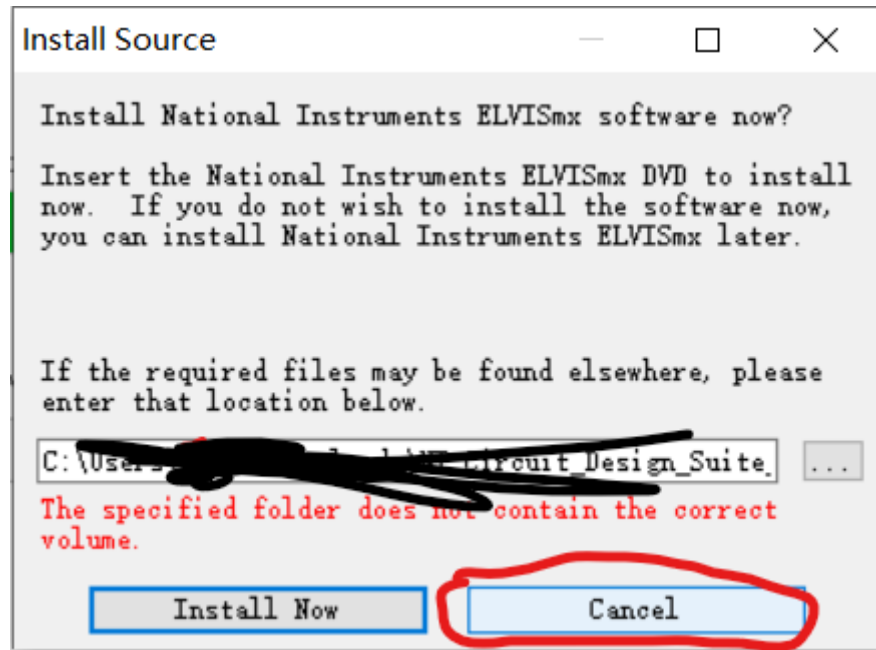
After installation is complete, you can restart your computer and run the software. Note that the setup wizard does not create a desktop shortcut on your computer, so you need to find the software in your start menu or on your computer. The default location of the software is C:\Program Files (x86)\National Instruments\Circuit Design Suite 14.1\multisim.exe

When you first run the software, it will ask you to extend the evaluation period. Click yes.
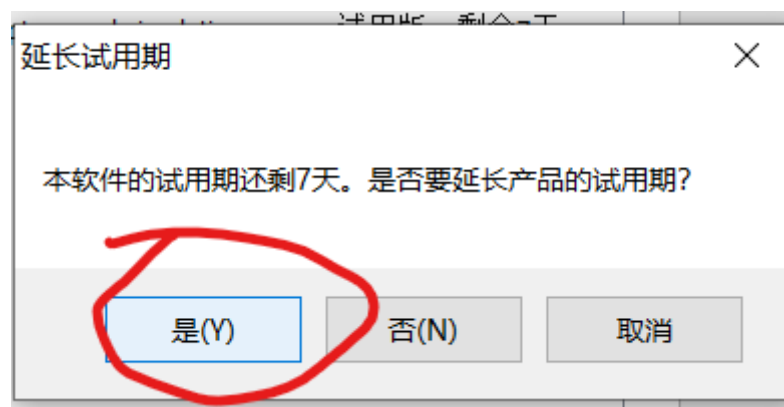
Then you can extend the evaluation period according to the guide. If you can't sign up or log in to your account in the software, simply click on "下一步" until you reach the page shown below. Then you can go to the website to extend the evaluation period.

The website will ask you to register an NI account. You may want to use your qq email or other email services rather than your Shanghaitech email, as our email system will block the confirmation email sent by NI.

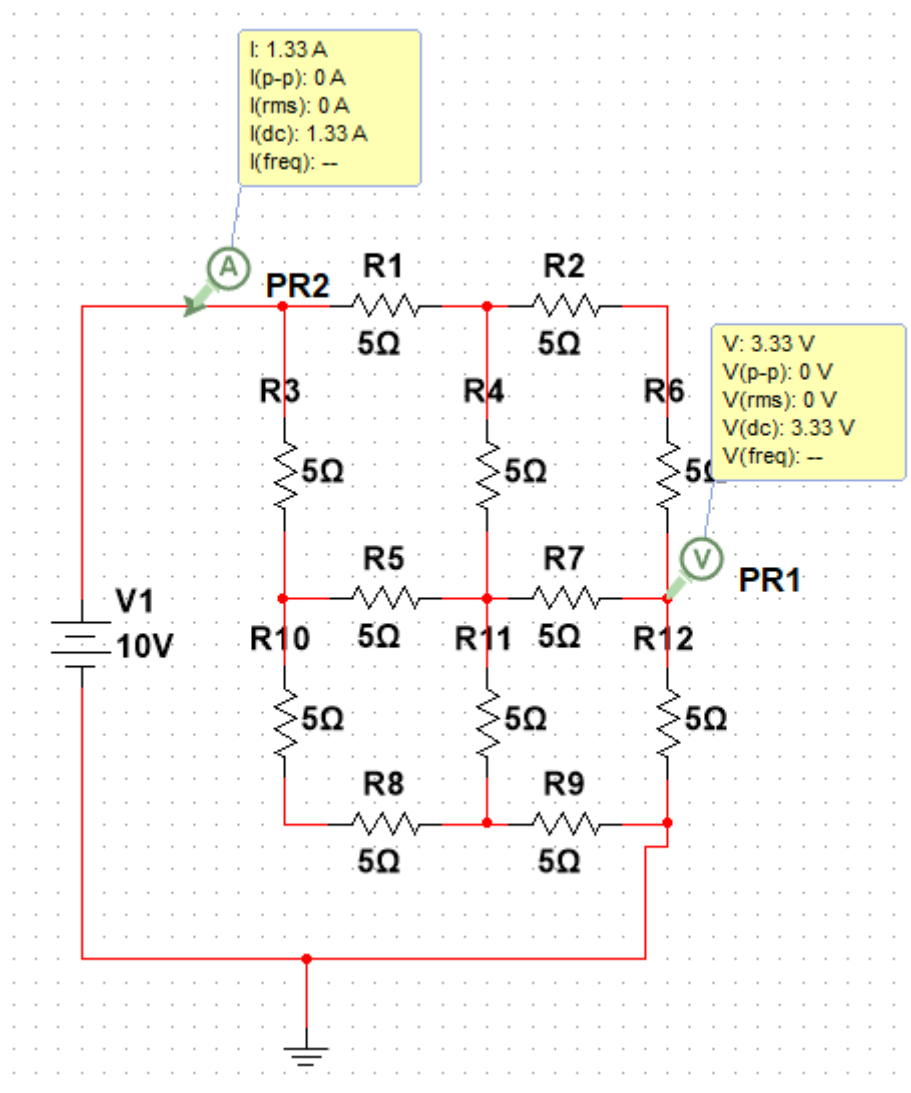Make sure you get the product information right.

You will get a 45-day extended evaluation period, which will be more than enough for the purpose of this course.

For MAC or Linux users, please install windows by either using virtual machines like parallel desktop or multi-booting.

How to simulate a circuit

In this section, some examples will help you to get familiar with Multisim.

EXAMPLE 1, How to find the voltage and current of a circuit: Please check out *HOW_TO_MULTISIM_EXAMPLE_1.mp4* on piazza.

I: 1.33 A
I(p-p): 0 A
I(rms): 0 A
I(dc): 1.33 A
I(freq): --

V: 3.33 V
V(p-p): 0 V
V(rms): 0 V
V(dc): 3.33 V
V(freq): --

Building the circuit

Press Ctrl+W to place circuit components. You can select <All groups> and <All families>, then type in the name of the component you want to place. Choose the right component and click OK to place the components.
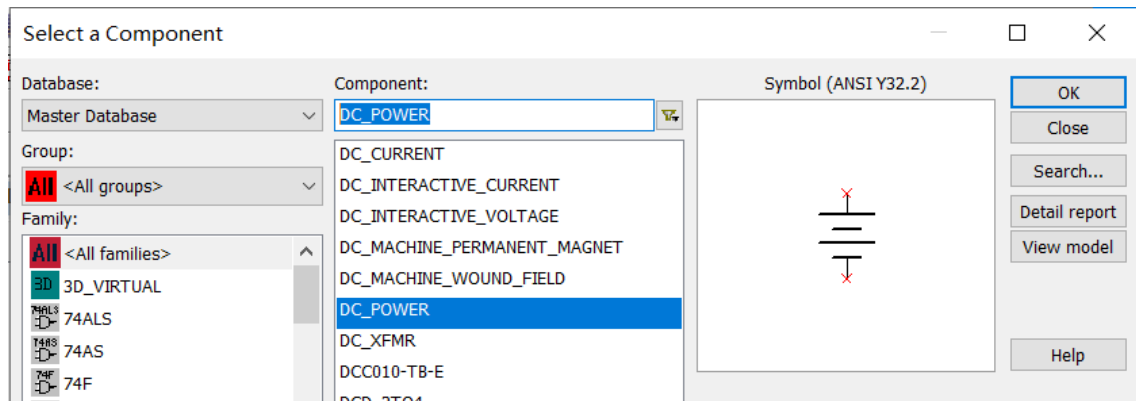
Recommended components:

Resistors, capacitors, and inductors: RESISTOR_RATED CAPACITOR_RATED INDUCTOR_RATED

Voltage and current sources: DC_POWER DC_CURRENT GROUND

If you want to know the simulated results, you will need probes. Just like in the real world, you will use multimeter or oscilloscope to measure voltages or currents. In Multisim, the probes are located in the upper right corner.
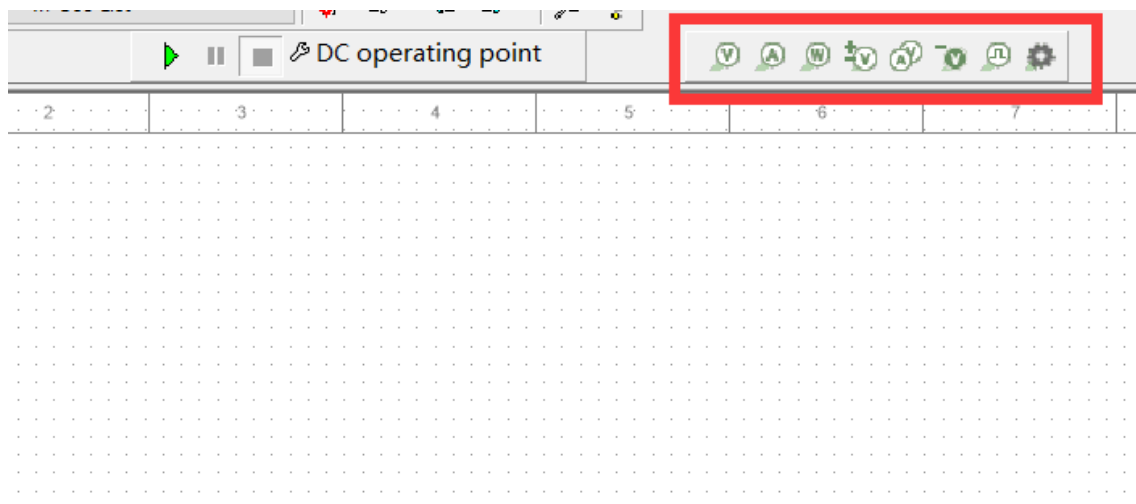

Voltage probes measure the voltage of a node with respect to ground. So always place a ground node for your circuit. Differential voltage probes will measure the voltage difference between two nodes, like a voltmeter.

Choose the correct simulation type

There are many types of simulation in Multisim, but for now, you only need to get familiar with three of them.

In interactive simulation, the software continuously calculates the circuit's voltage and current, as if the circuit is running in the real world. You can place voltage and current probes to measure the voltage of any node or the current of any wire. Differential probes can measure the voltage between two terminals. For example, this is the simple lightbulb circuit simulated in interactive simulation.
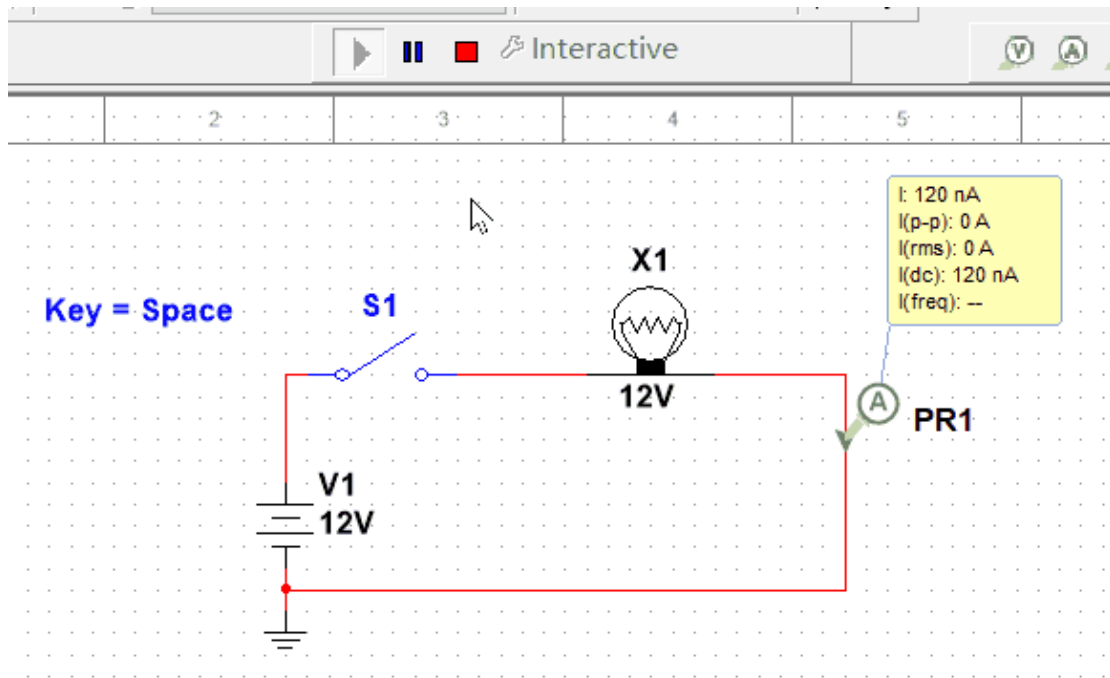
In DC operating point, the software will calculate the steady state of the circuit. If your circuit is purely linear, the result will be the same as interactive simulation. The following figure shows a simulation results of DC operating point.
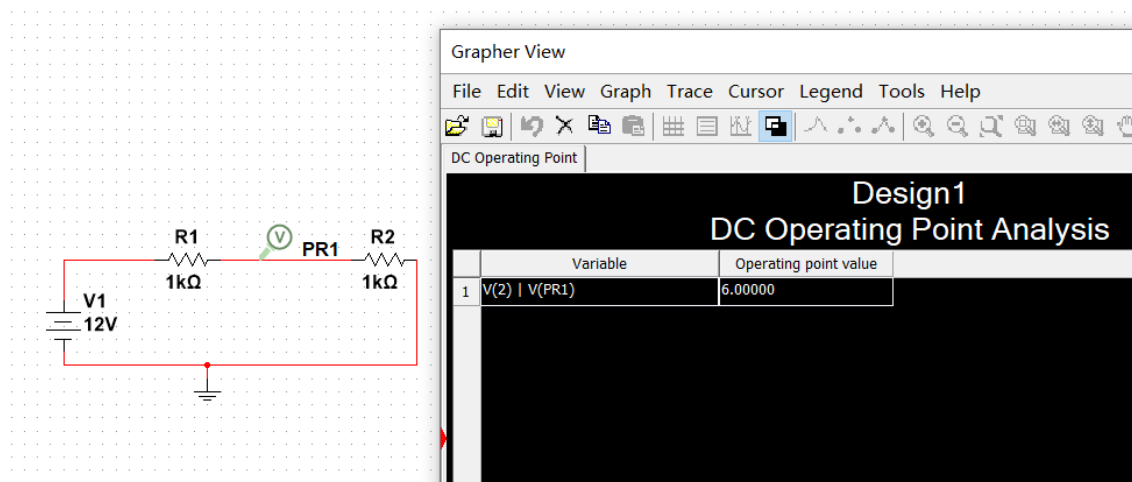
In DC sweep, the DC operation point will be calculated based on the given parameters, then the parameters will change according to the user's settings. If you wish to know how the circuit will change according to some parameters, you should use this simulation type. In example 2, we use DC sweep to find out when the base current of a transistor changes, how will the current of its collector change. Please check out *HOW_TO_MULTISIM_EXAMPLE_2.mp4* on piazza.

In the simulation settings, you will need to define the output of the simulation. The output is the data you desire. For example, in EXAMPLE 2 we wish to know the current flowing into the collector of the transistor, so we set the output to be the current of V1, which is also the current flowing into the collector. All the data of probes are automatically added to output.

But in general all you need to do is place a probe at the node that you are interested

in.