

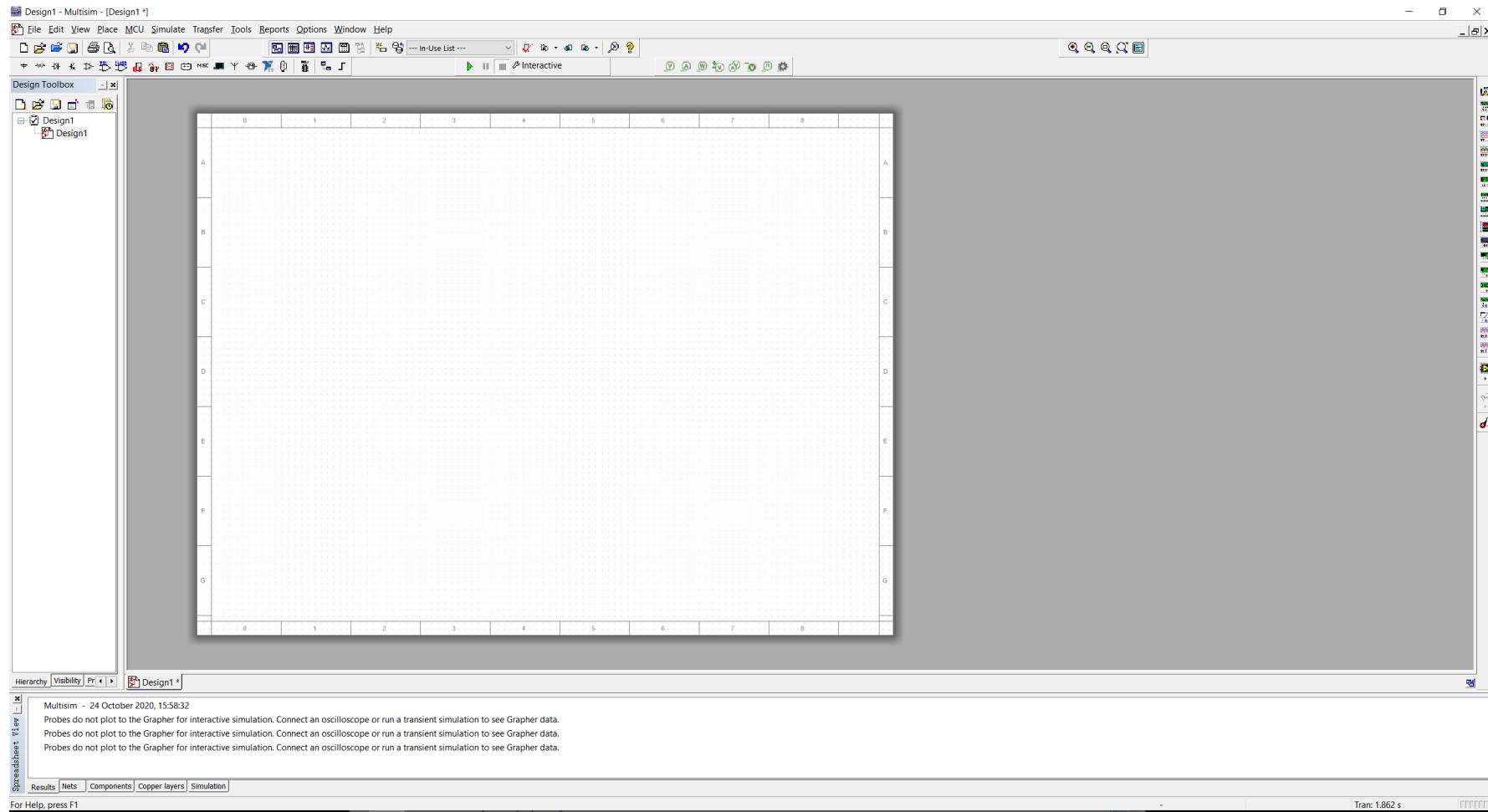
# Introduction to Multisim

SI100B Fall 2020 EE part Tutorial I

Yiming Gao, Fengxu Yang, Zirui Wang

# What is Multisim?

- Multisim is industry-standard SPICE simulation and circuit design software for **analog**, **digital**, and **power electronics** in education and research.



# Installation

Solutions ▾ Products ▾ Perspectives Support ▾ Community

About Contact Us



HOME / SHOP / ELECTRONIC TEST AND INSTRUMENTATION / APPLICATION SOFTWARE FOR ELECTRONIC TEST AND INSTRUMENTATION / MULTISIM™ / WHAT IS MULTISIM™ FOR EDUCATION

Are you a student who needs Multisim™ software?

## What Is Multisim™ for Education?

Multisim™ for Education is circuits teaching application software for analog, digital, and power electronics courses and laboratories. Visualize circuits and reinforce theory with simulated instruments, advanced analyses, and thousands of interactive components.

START FREE TRIAL    VIEW PRODUCT DETAILS

<https://www.ni.com/en-us/shop/electronic-test-instrumentation/application-software-for-electronic-test-and-instrumentation-category/what-is-multisim/multisim-education.html>

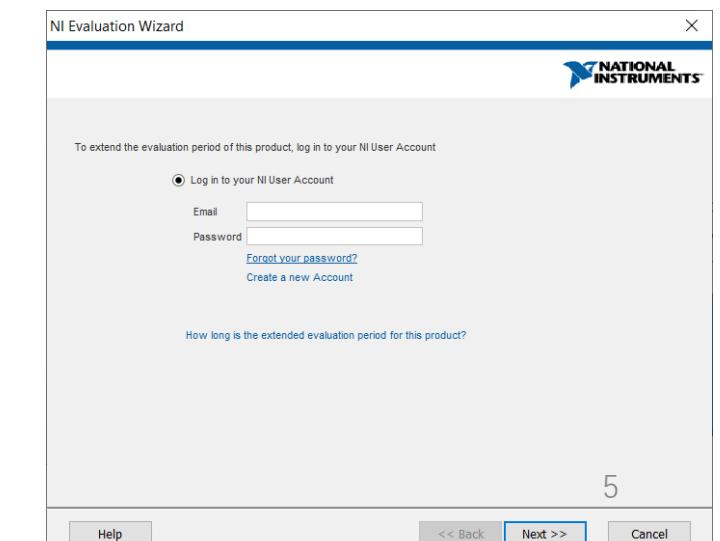
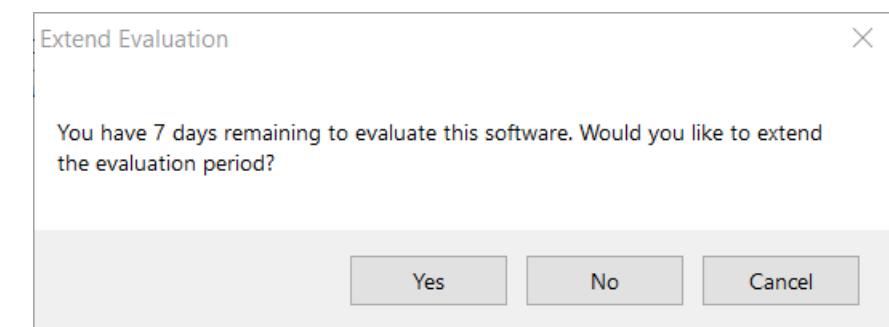
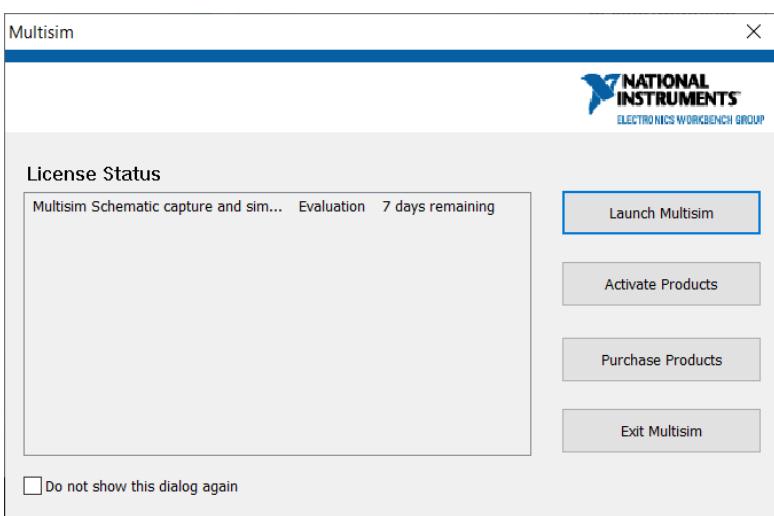
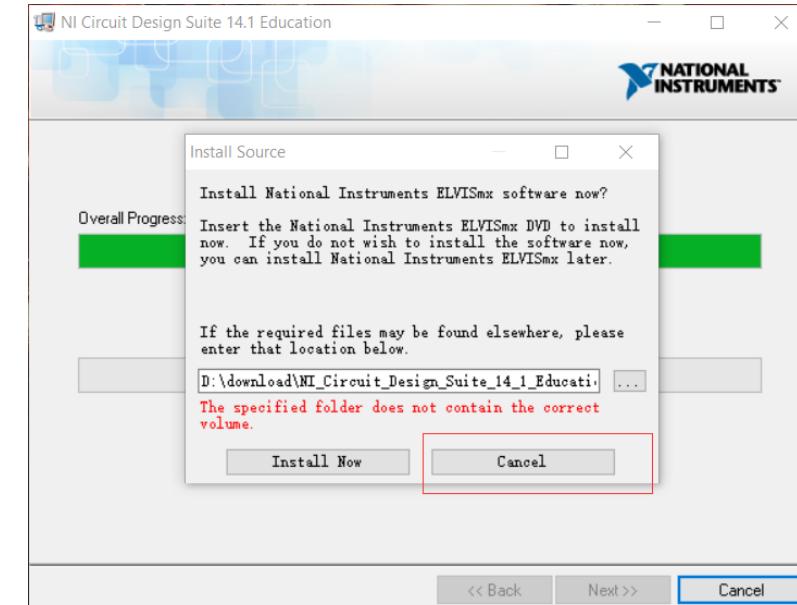
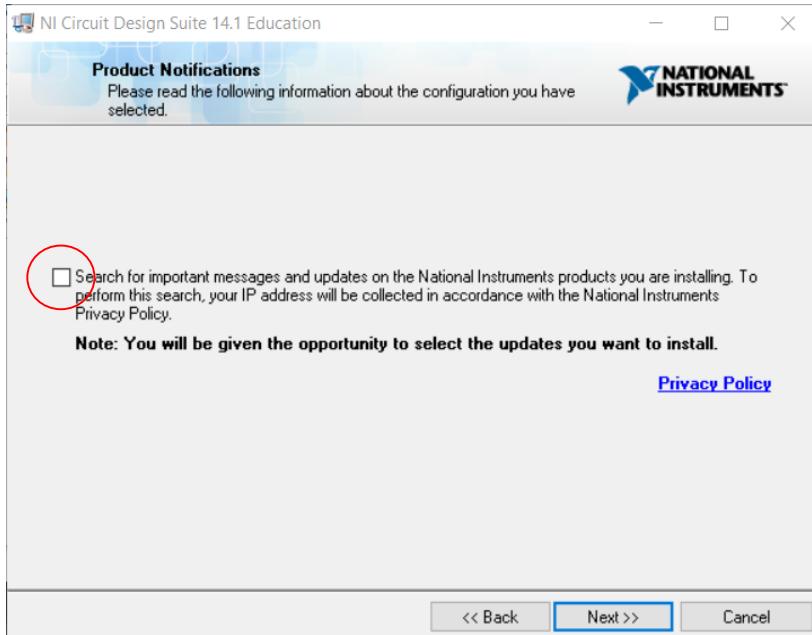
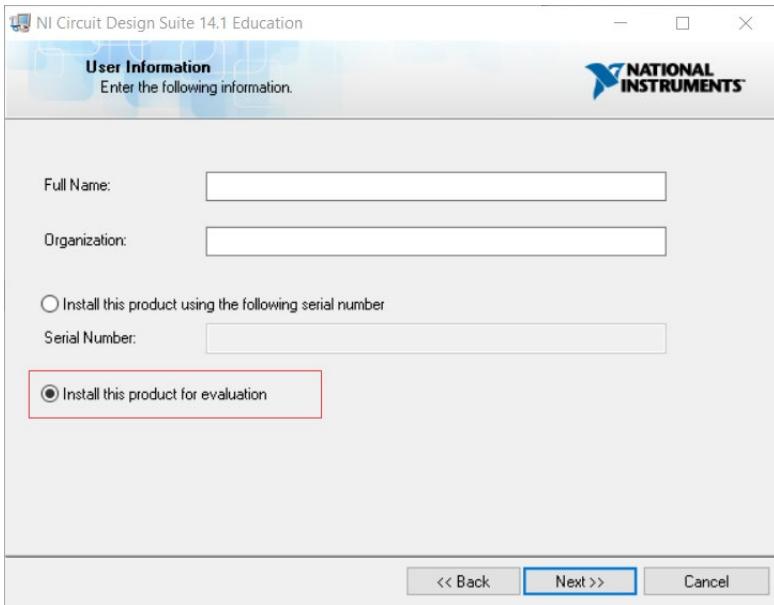
# Installation

Name	Date modified	Type
Bin	25/10/2020 09:57	File folder
Licenses	25/10/2020 09:57	File folder
Products	25/10/2020 09:57	File folder
autorun.exe	20/06/2019 13:13	Application
autorun.inf	20/06/2019 13:13	Setup Information
nidist.id	20/06/2019 13:13	ID File
patents.txt	20/06/2019 13:13	Text Document
Readme_deu.html	20/06/2019 13:13	Chrome HTML Do...
Readme_eng.html	20/06/2019 13:13	Chrome HTML Do...
setup.exe	20/06/2019 13:13	Application
setup.ini	20/06/2019 13:13	Configuration setti...

1.41 MB

The screenshot shows the Windows file explorer interface on the left and the NI Circuit Design Suite setup window on the right. The setup window title bar reads 'NI Circuit Design Suite 14.1 Education'. It features a large central image of a computer monitor displaying a blue and white graphical user interface. Below the image, the text 'NI Circuit Design Suite™ 14.1' and 'Education Edition' is displayed. A note at the bottom of the window says: 'Exit all programs before running this Setup. Disabling virus scanning utilities may improve installation speed. This program is subject to the accompanying License Agreement(s.)'. Another note below it states: 'National Instruments Corporation is an authorized distributor of Microsoft Silverlight.' At the bottom of the window are three buttons: '<< Back', 'Next >>', and 'Cancel'. The National Instruments logo is located in the bottom right corner of the window.

# Installation



# Can I Install Multisim on Mac OS X or Linux?

## Issue Details

I want to use Multisim but my computer doesn't run Windows. Is there a Multisim installer available for the Mac OS X or Linux operating systems?

## Solution

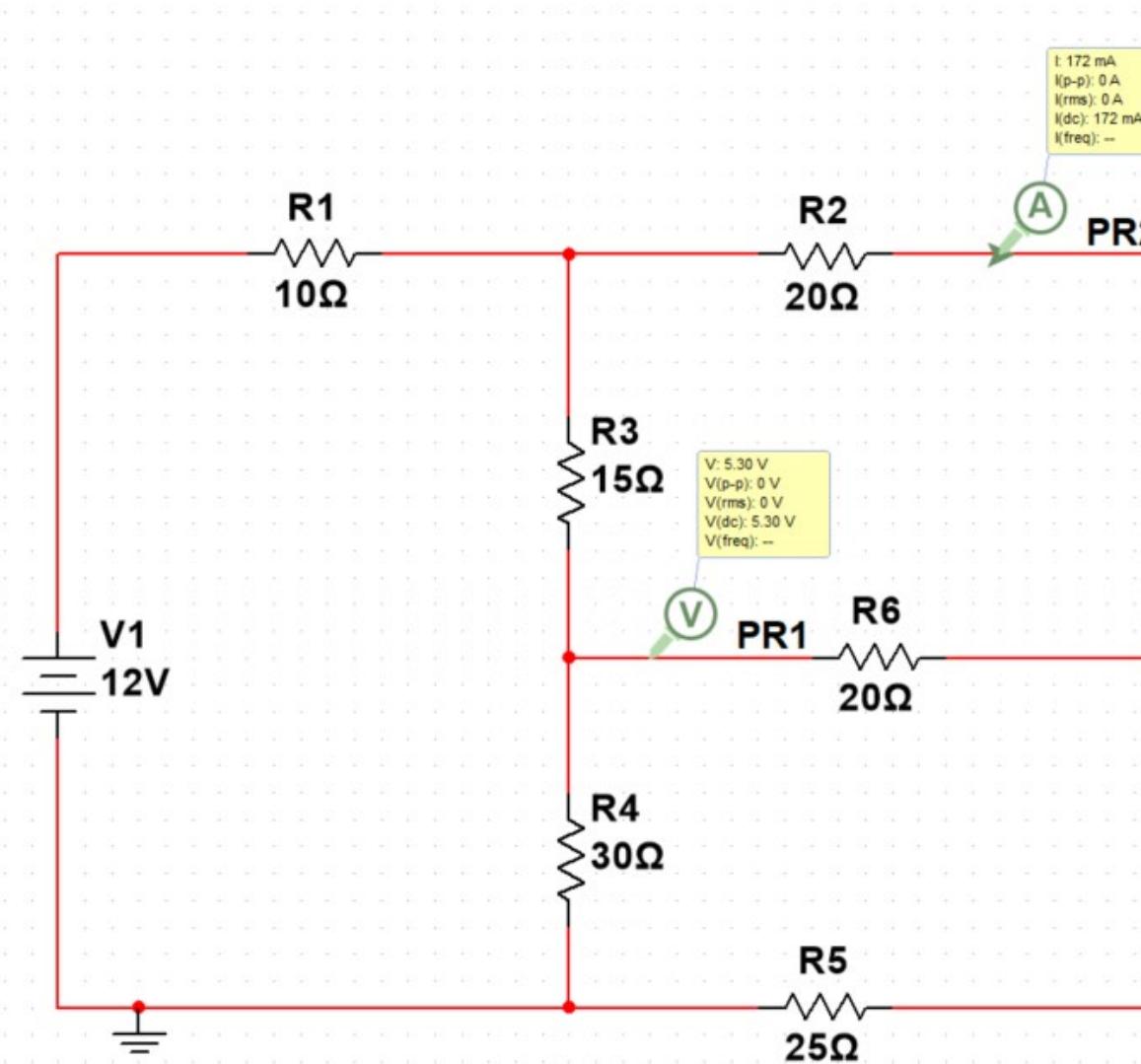
Multisim is currently only available for installation on Windows operating systems. Multisim can not be installed on Mac OS X or Linux.

If you have a Mac OS X or Linux based computer and you need to use Multisim, you have two options:

1. National Instruments' recommended method: Using [Multisim Live](#). Multisim Live is a web-based electronic schematic capture and simulation tool with built-in SPICE based analog simulation. It allows users to capture and simulate designs using 200 different components, anywhere with an Internet connection. Since Multisim Live is fully web-based and requires no installation, it can be used in Windows, Mac OS X, Linux, iOS and Android. For more information, please refer to the [Multisim Live FAQ](#) and the [Multisim Live Tutorial](#).
2. It is possible to use Multisim on a Mac OS X or Linux by means a Virtual Machine, where you are in fact installing on Windows. In Mac OS X you can also use Bootcamp. Please be aware that Bootcamp and Virtual Machine are not supported methods, and National Instruments does not recommend using them as a replacement for Multisim Live or for using a Windows based computer.

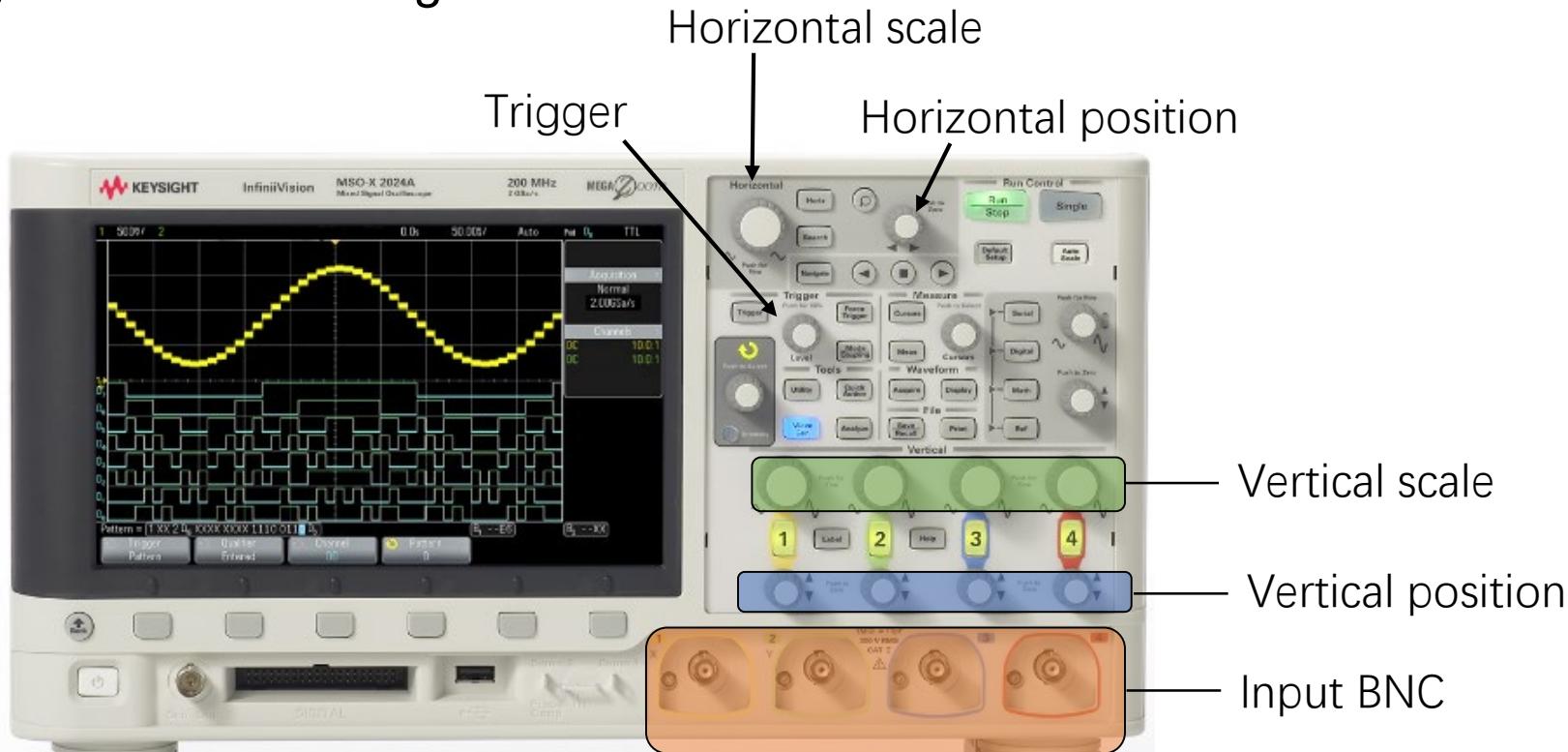
Multisim live: <https://www.multisim.com/create/>

# Demo 1: resistor network

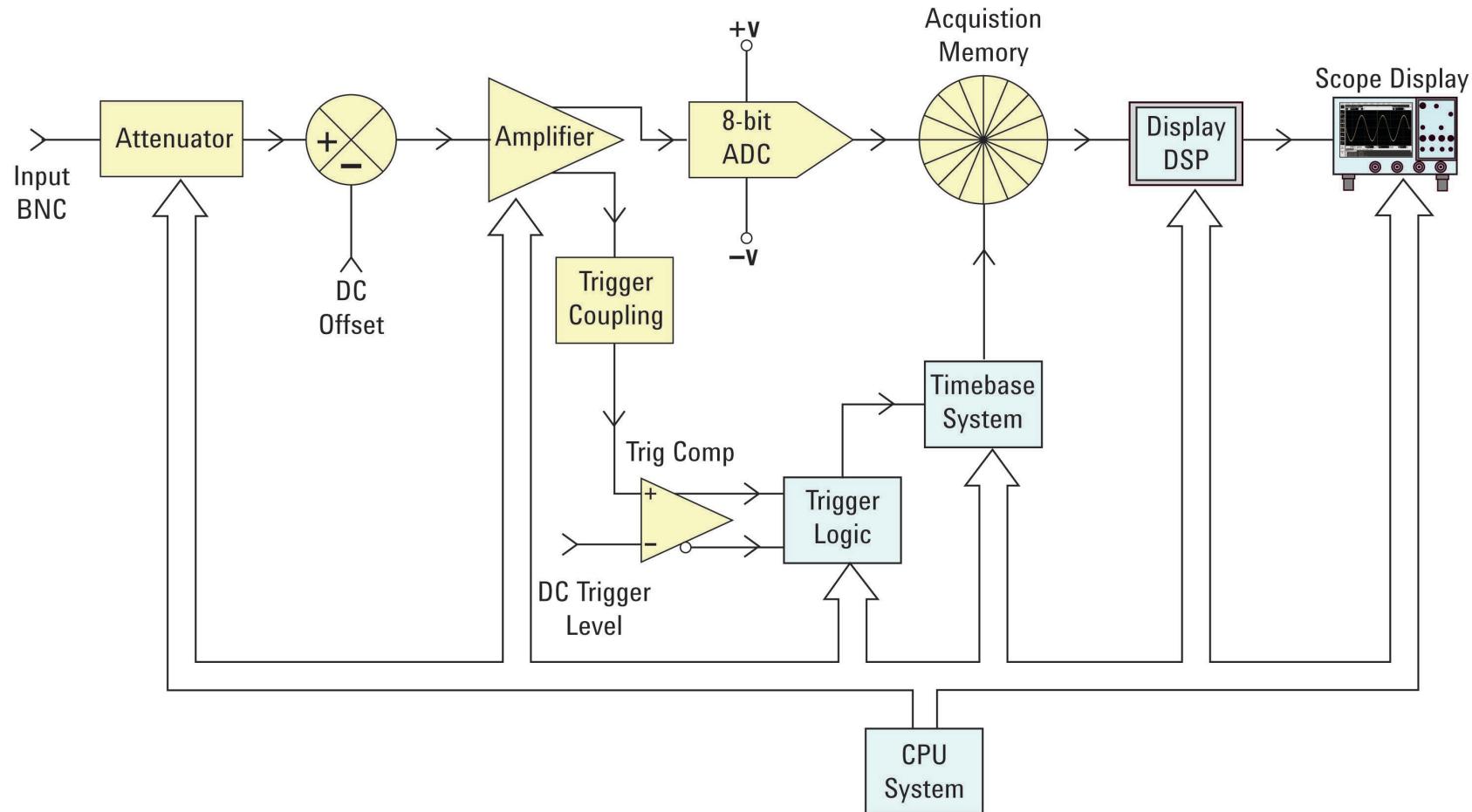


# Oscilloscope

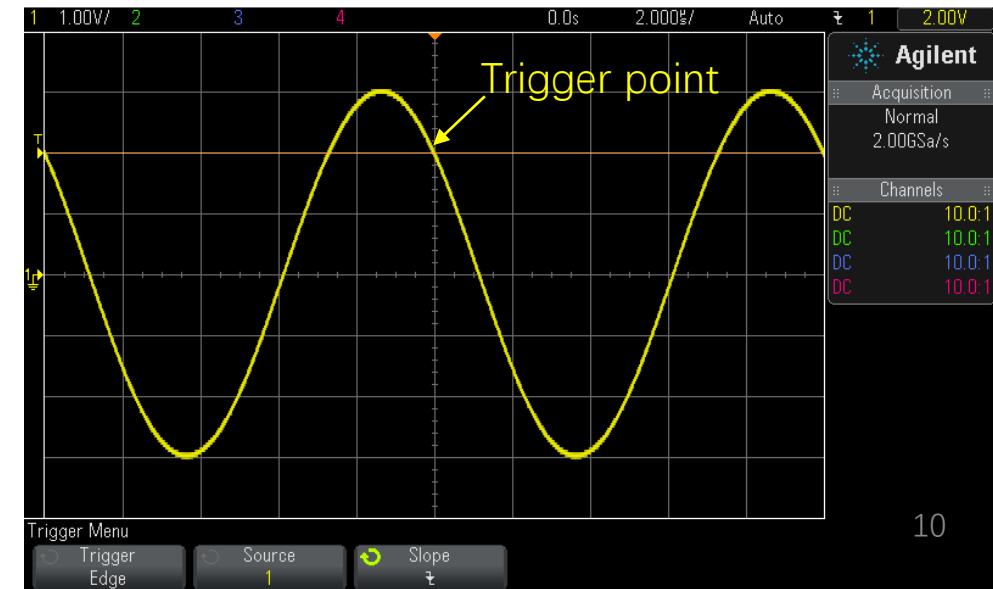
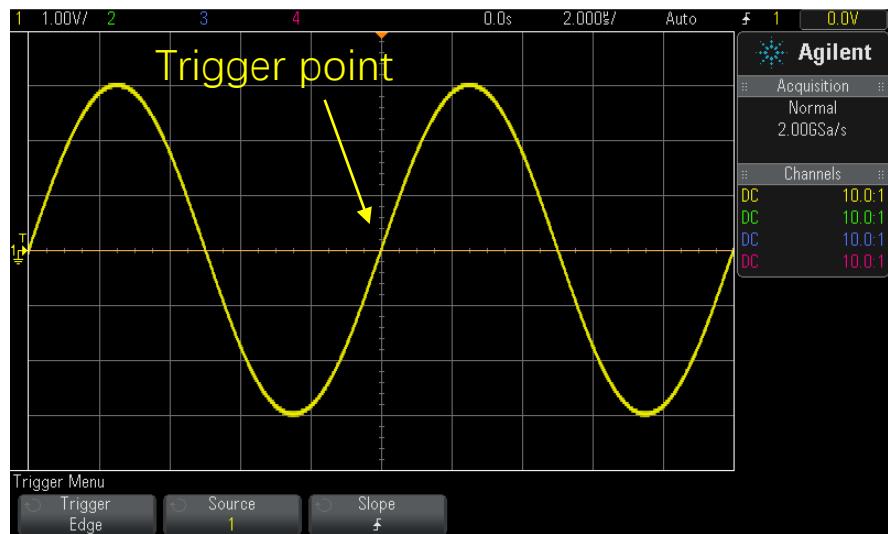
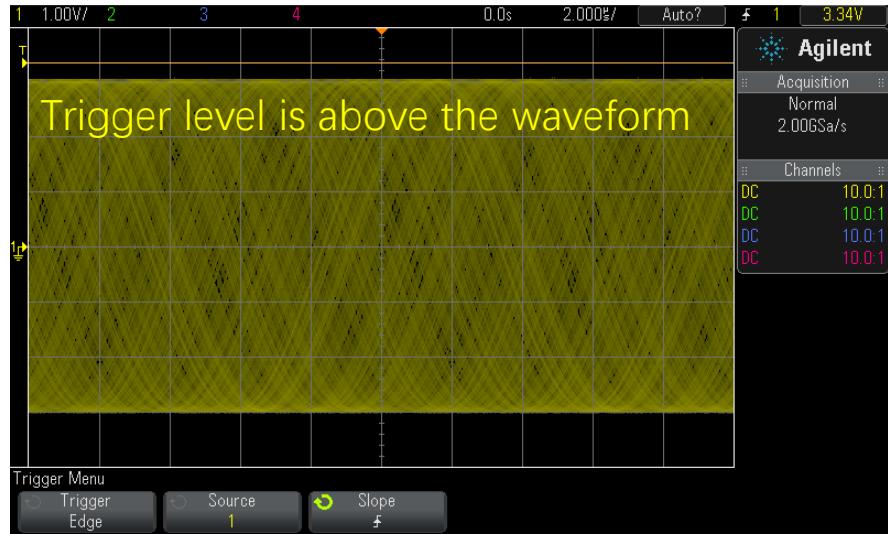
- Transfers the **electrical signals** to **light signal**.
- Maps electrical signals that vary over time in two dimensions, usually voltage and time
- Test, verify and debug electronic designs
  - The eyes of electrical engineer



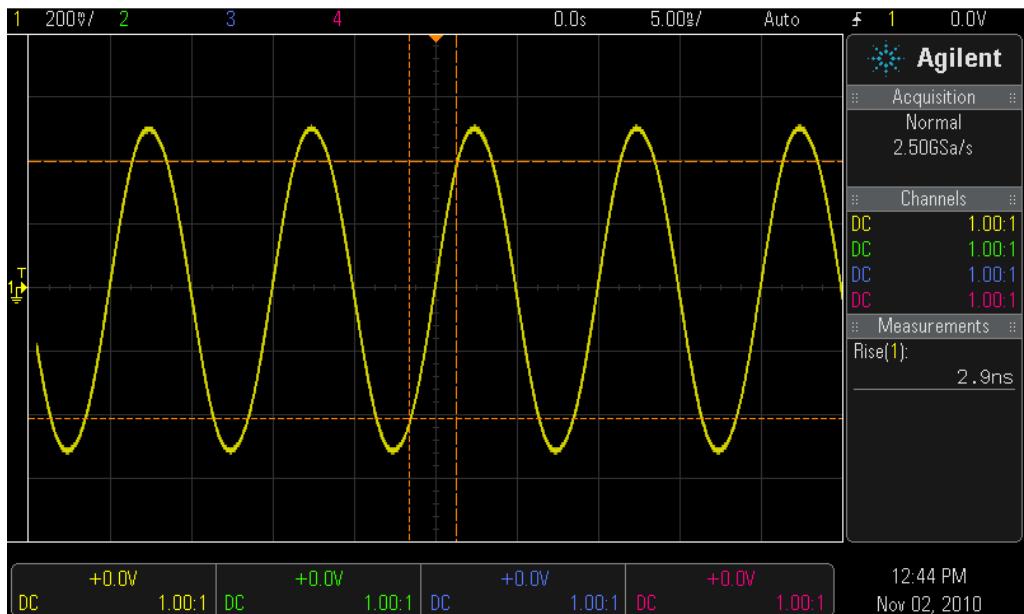
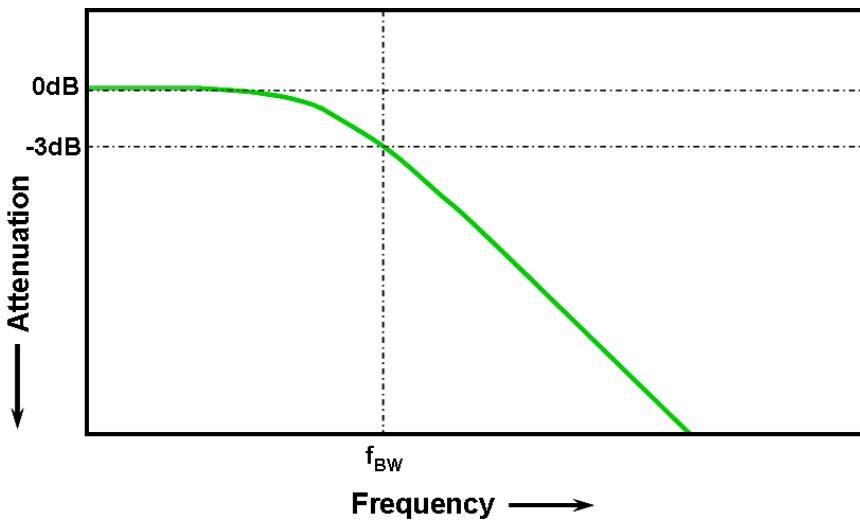
# Working principle



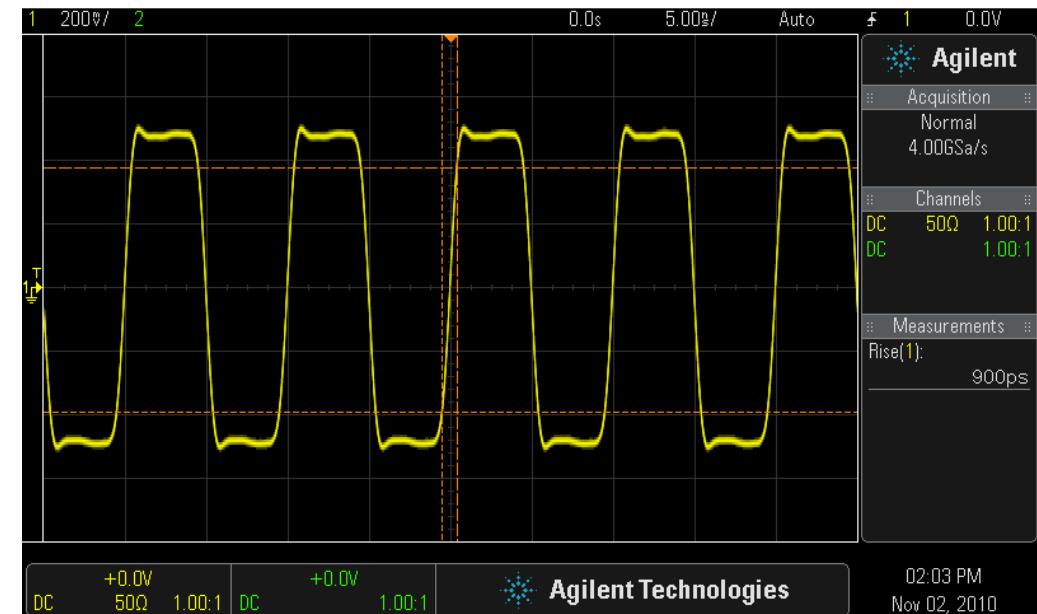
# Trigger



# Bandwidth

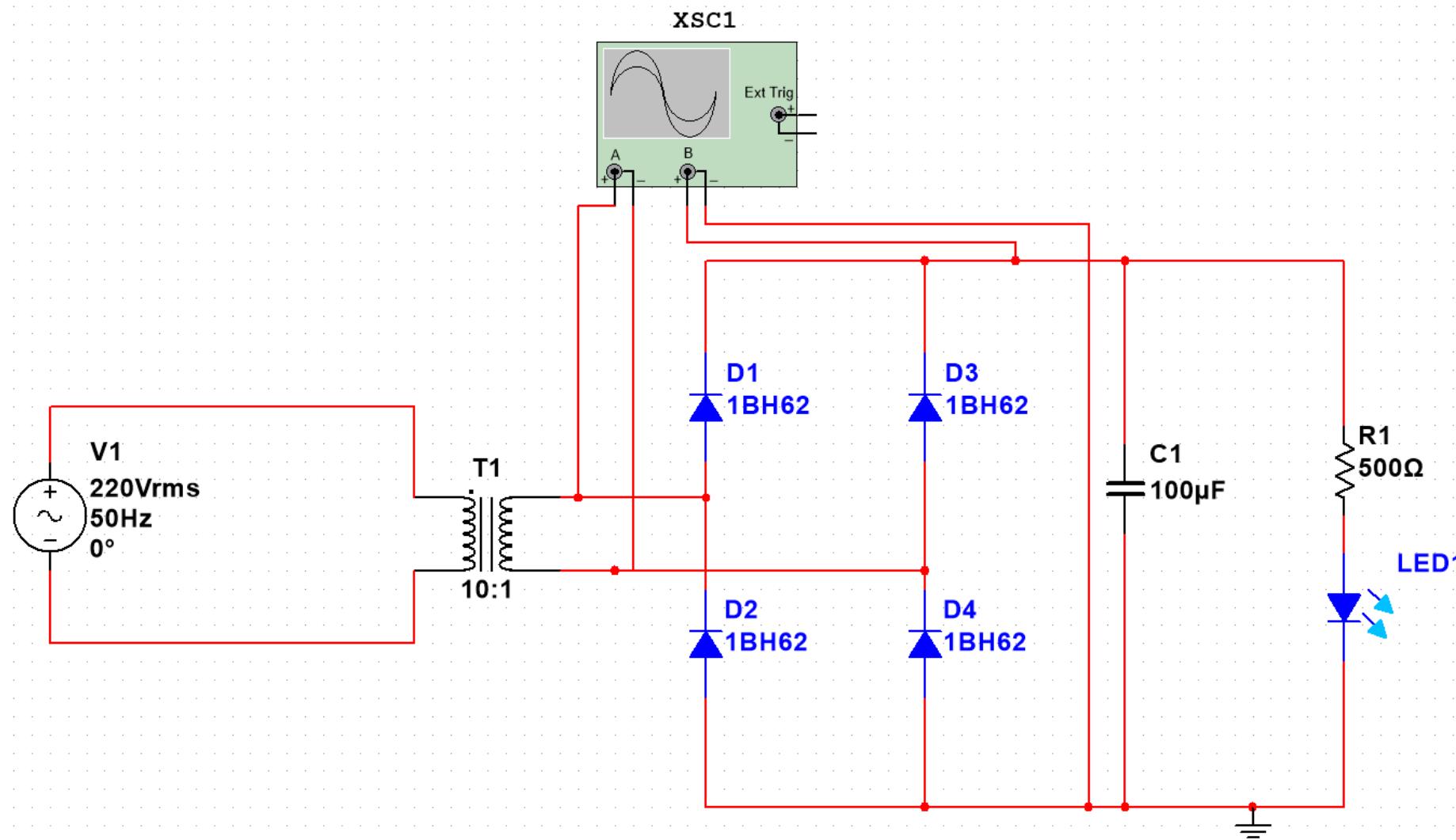


100-MHz

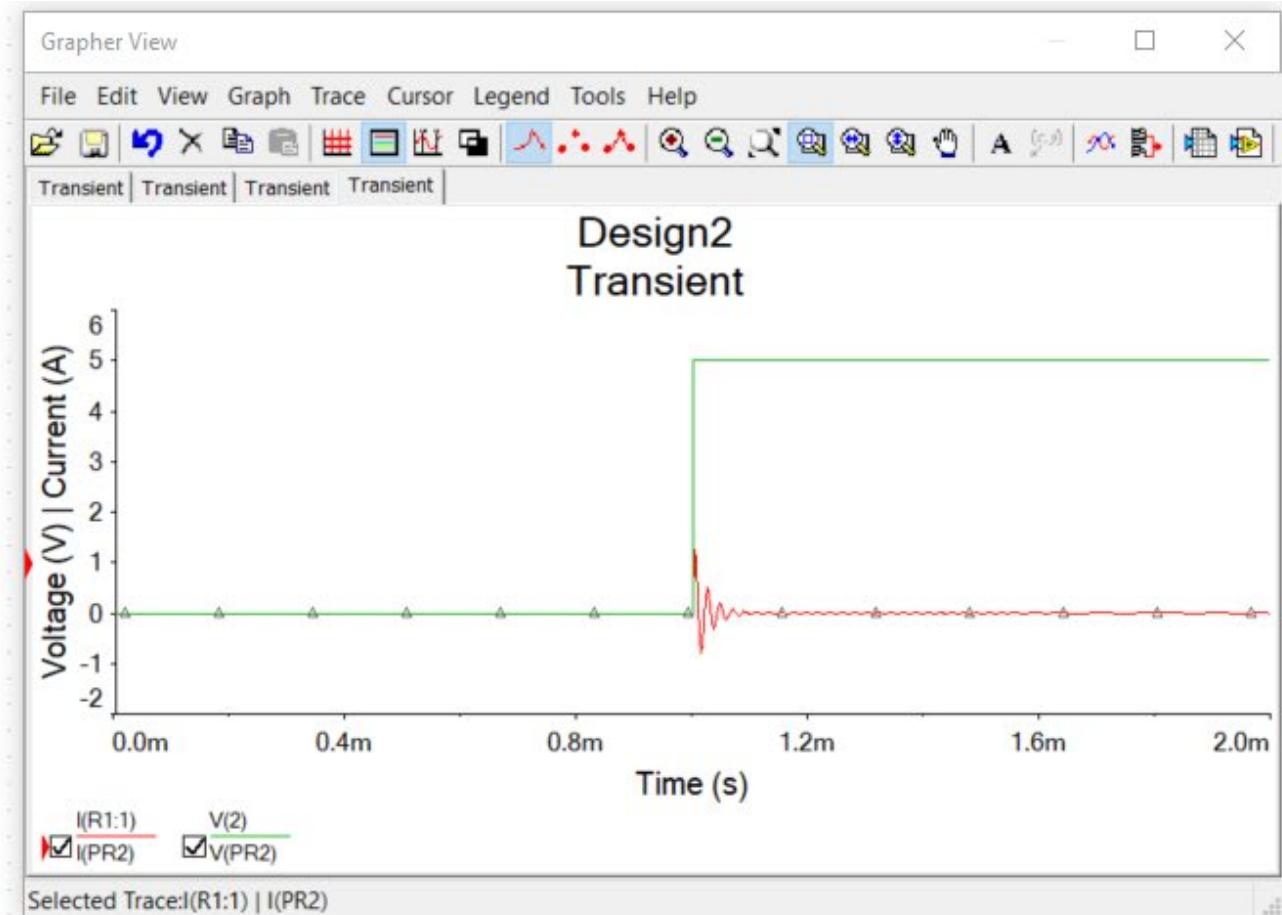
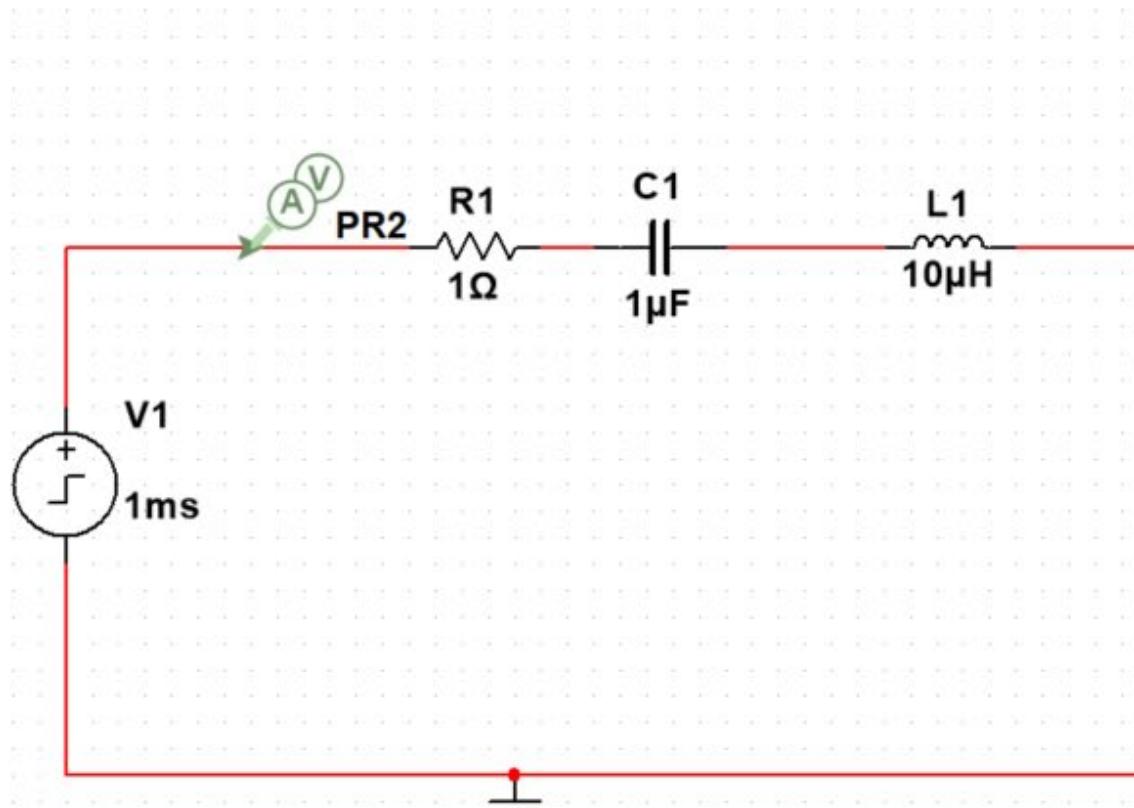


500-MHz

# Demo 2: AC-DC



# Demo 3: transient



# **Q&A**