

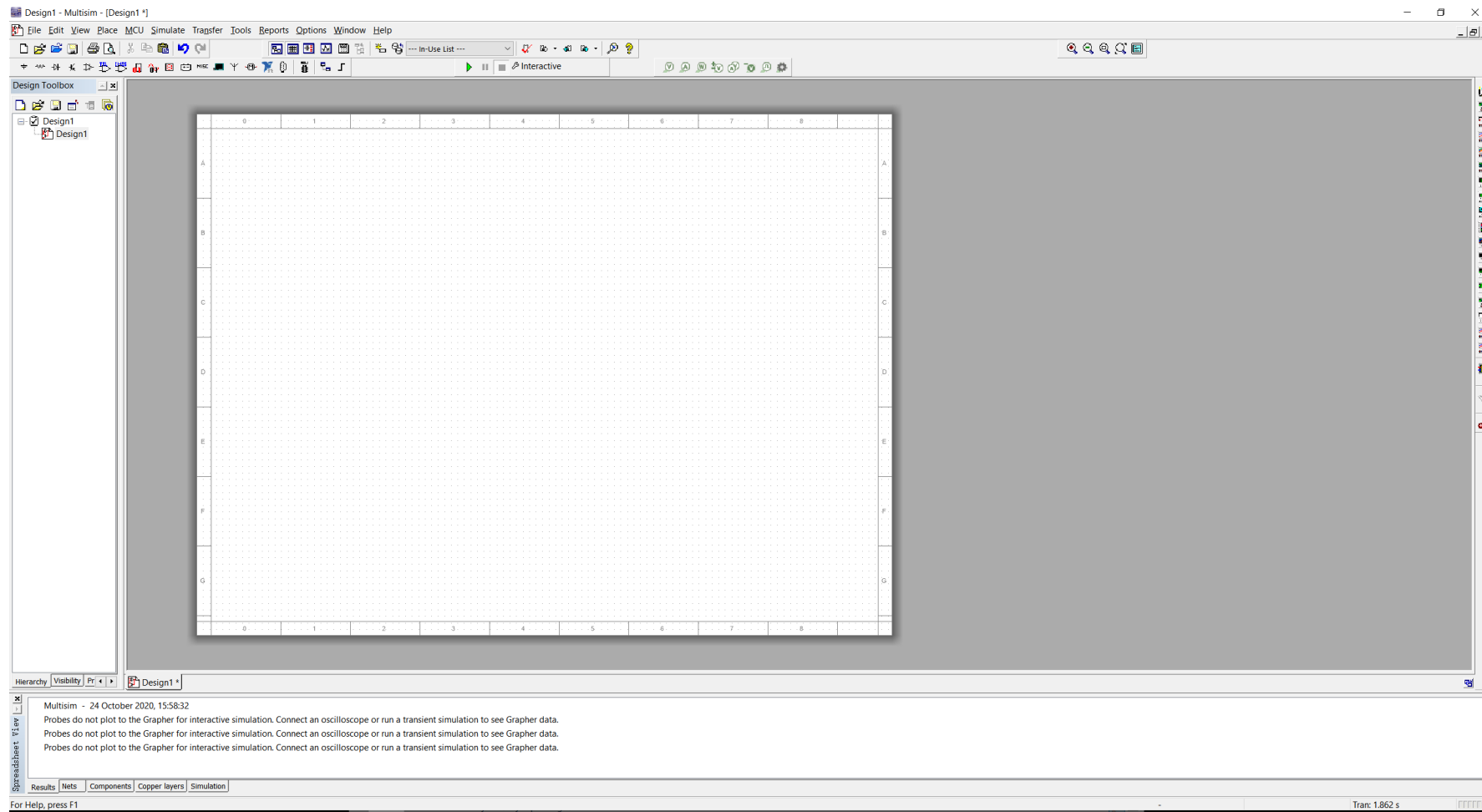
Introduction to Multisim

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?

GET STUDENT DOWNLOAD

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...

NI Circuit Design Suite 14.1 Education

ni.com/academic/multisim

NI Circuit Design Suite™ 14.1

Education Edition

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).

National Instruments Corporation is an authorized distributor of Microsoft Silverlight.

© 2001–2017 National Instruments. All rights reserved.

NATIONAL INSTRUMENTS
ELECTRONICS WORKBENCH GROUP

<< Back Next >> Cancel

1.41 MB

Installation

NI Circuit Design Suite 14.1 Education

User Information
Enter the following information.

Full Name:

Organization:

☐ Install this product using the following serial number

Serial Number:

☒ Install this product for evaluation

<< Back Next >> Cancel

NI Circuit Design Suite 14.1 Education

Product Notifications
Please read the following information about the configuration you have selected.

☐ 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](#)

<< Back Next >> Cancel

NI Circuit Design Suite 14.1 Education

Install Source

Install National Instruments ELVISmx software now?

Insert the National Instruments ELVISmx DVD to install now. If you do not wish to install the software now, you can install National Instruments ELVISmx later.

If the required files may be found elsewhere, please enter that location below.

D:\download\NI_Circuit_Design_Suite_14_1_Educati...

The specified folder does not contain the correct volume.

Install Now Cancel

<< Back Next >> Cancel

Multisim

NATIONAL INSTRUMENTS
ELECTRONICS WORKBENCH GROUP

License Status

Multisim Schematic capture and sim... Evaluation 7 days remaining

Launch Multisim

Activate Products

Purchase Products

Exit Multisim

☐ Do not show this dialog again

Extend Evaluation

You have 7 days remaining to evaluate this software. Would you like to extend the evaluation period?

Yes No Cancel

NI Evaluation Wizard

NATIONAL INSTRUMENTS

To extend the evaluation period of this product, log in to your NI User Account

☒ Log in to your NI User Account

Email

Password

[Forgot your password?](#)

[Create a new Account](#)

How long is the extended evaluation period for this product?

5

Help << Back Next >> Cancel

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

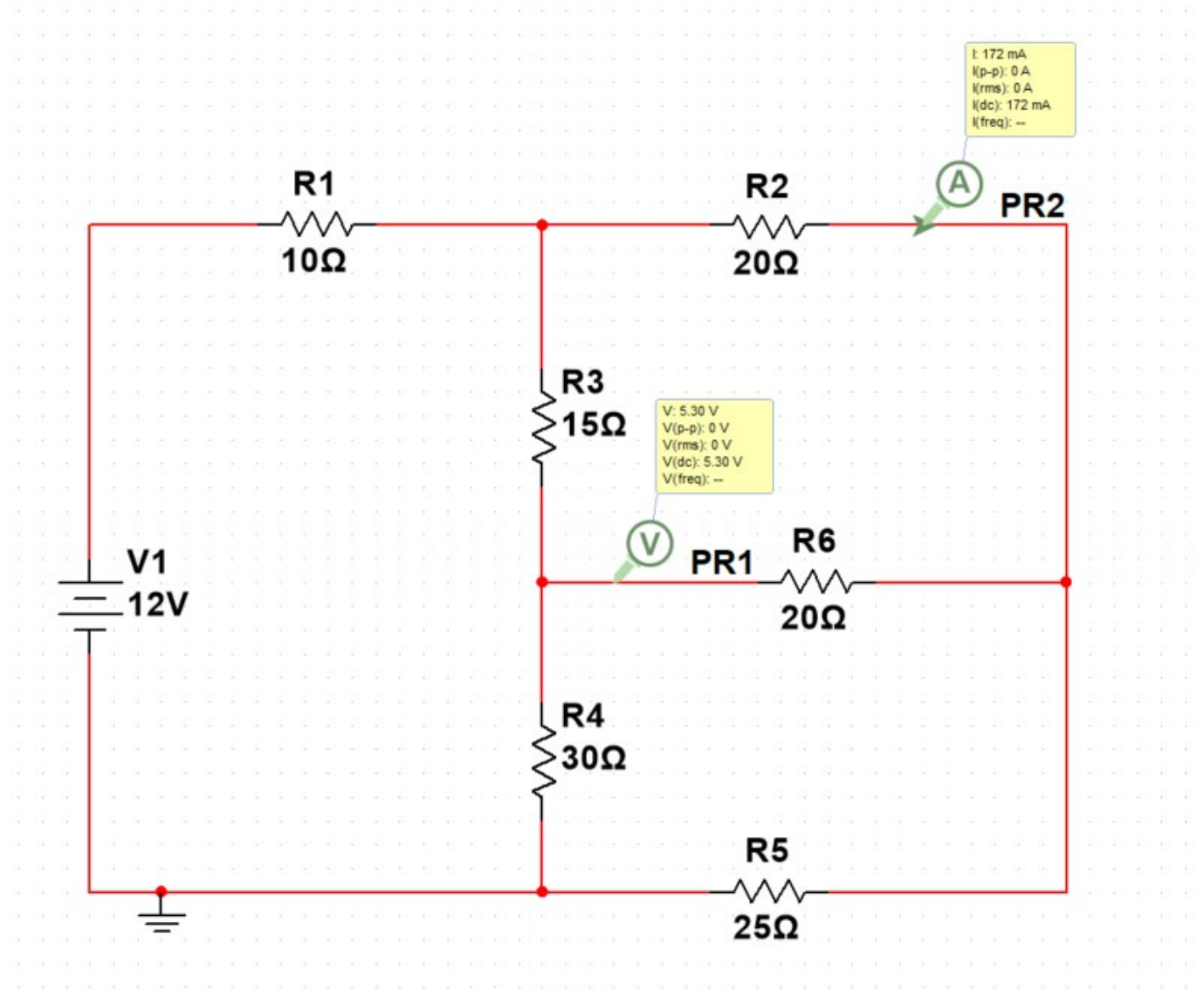
Multisim is currently only available for installation on Windows operating systems. Multisim can not be installed on Mac OS X or Linux.

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.

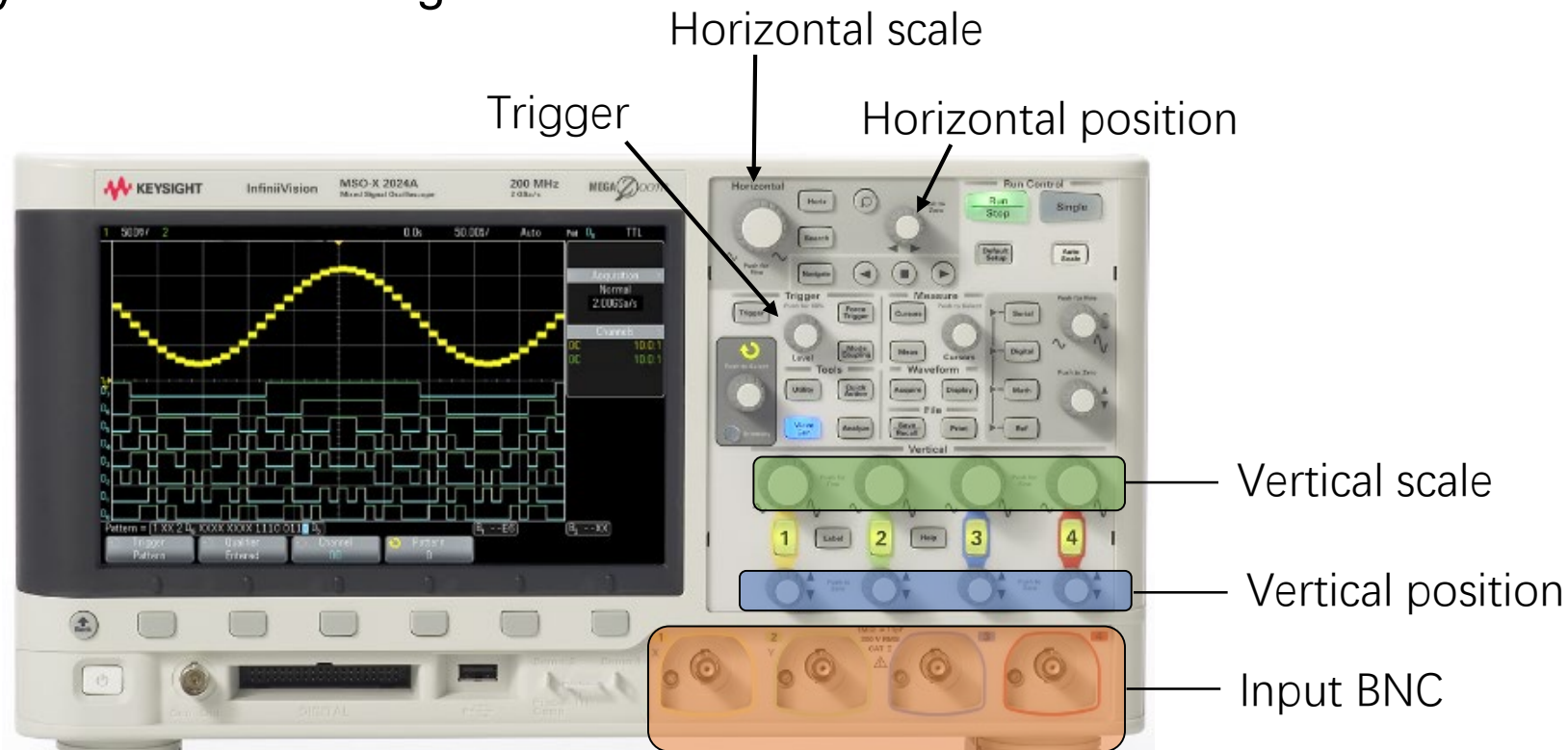
Multisim live: <https://www.multisim.com/create/>

Demo 1: resistor network

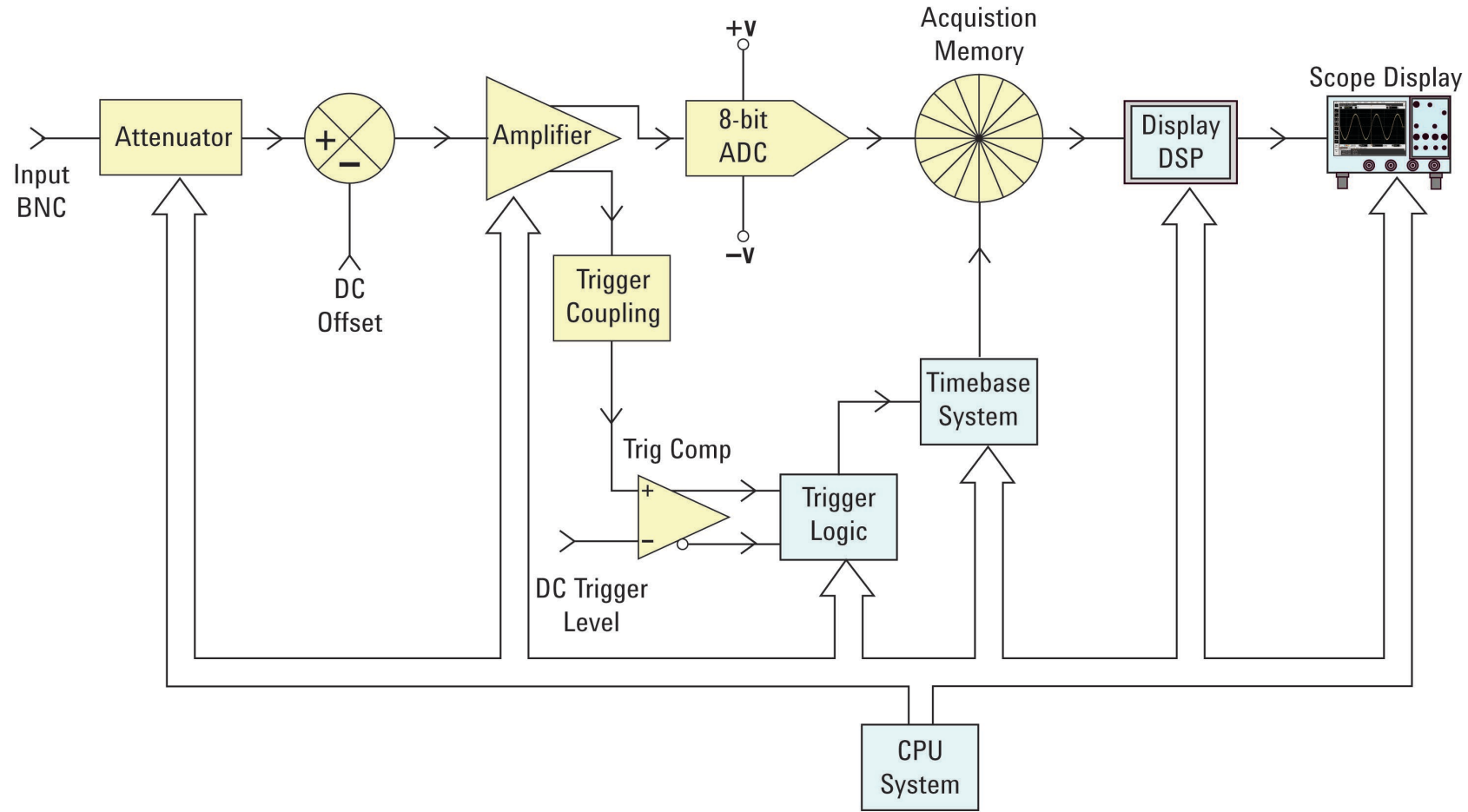


Oscilloscope

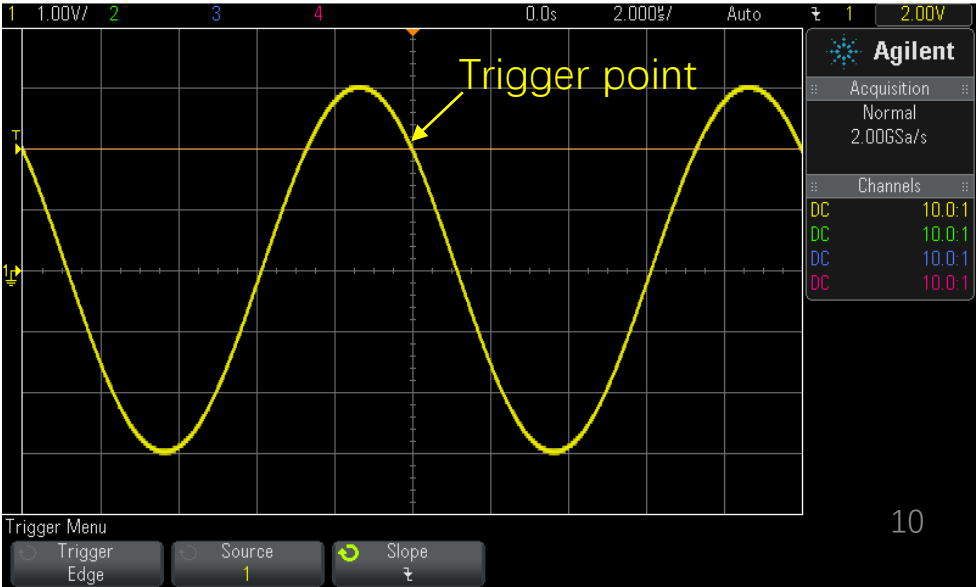
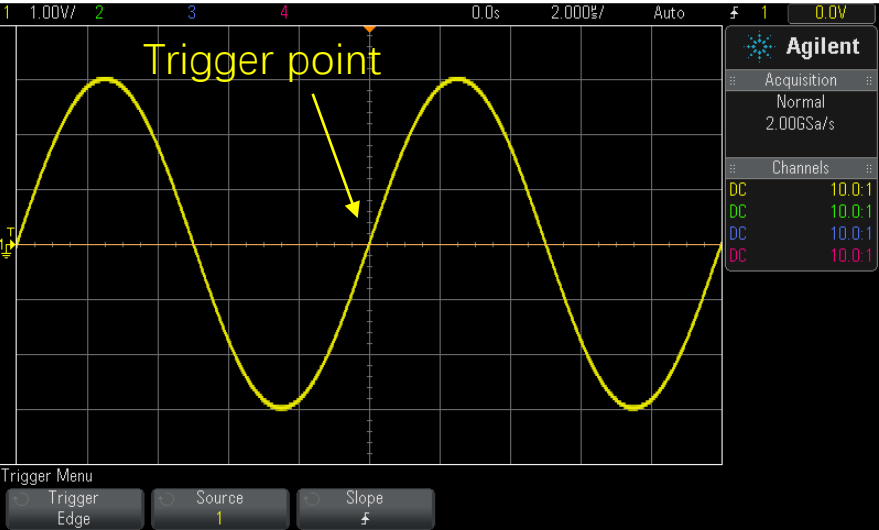
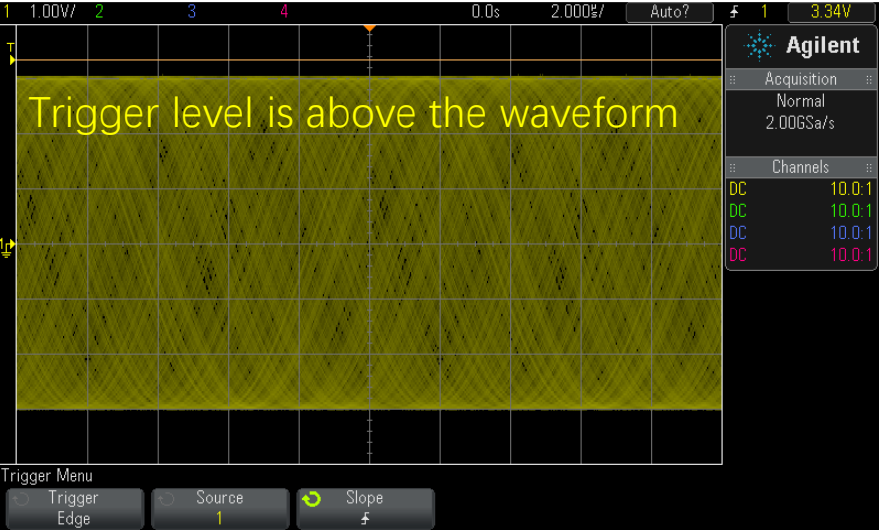
- Transfers the **electrical signals** to **light signal**.
- Maps electrical signals that vary over time in two dimensions, usually voltage and time
- Test, verify and debug electronic designs
 - The eyes of electrical engineer



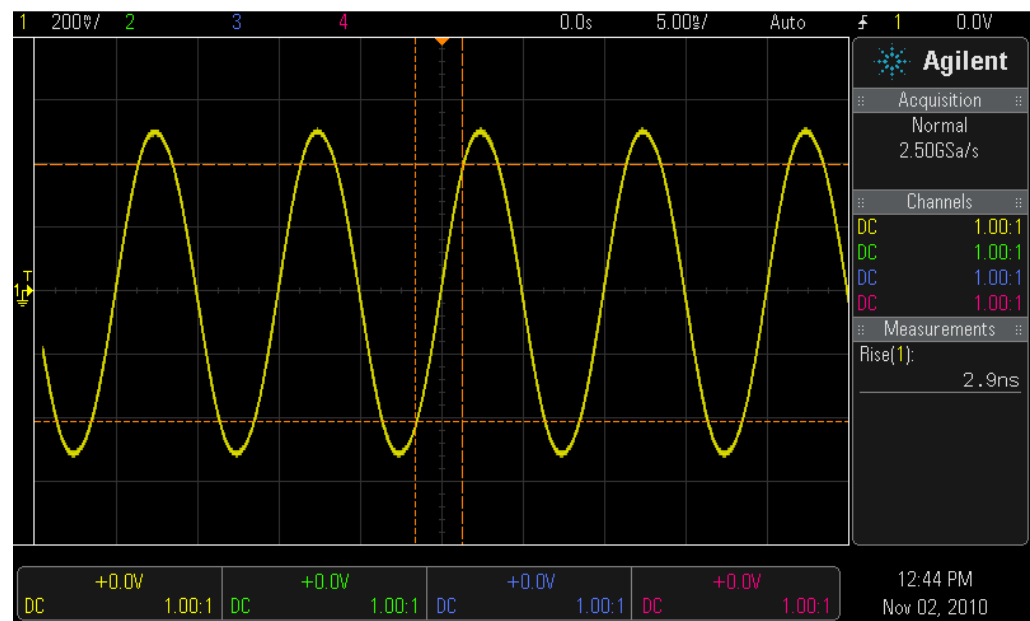
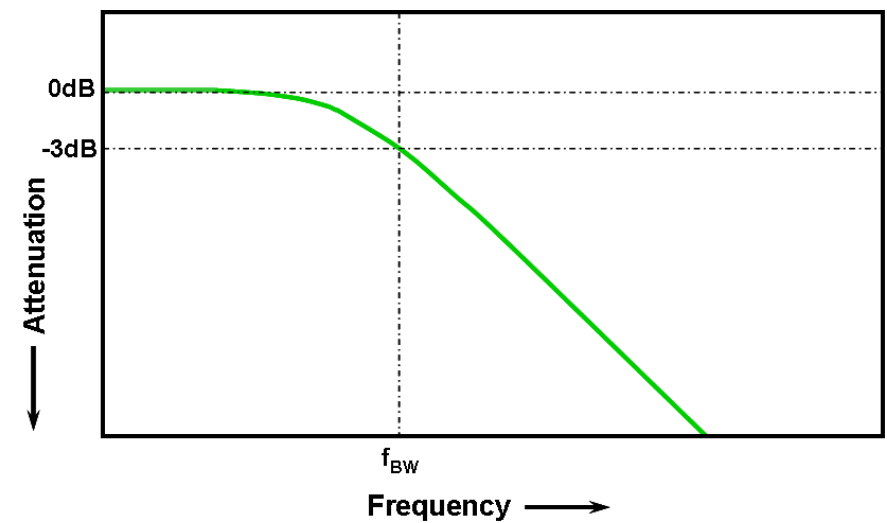
Working principle



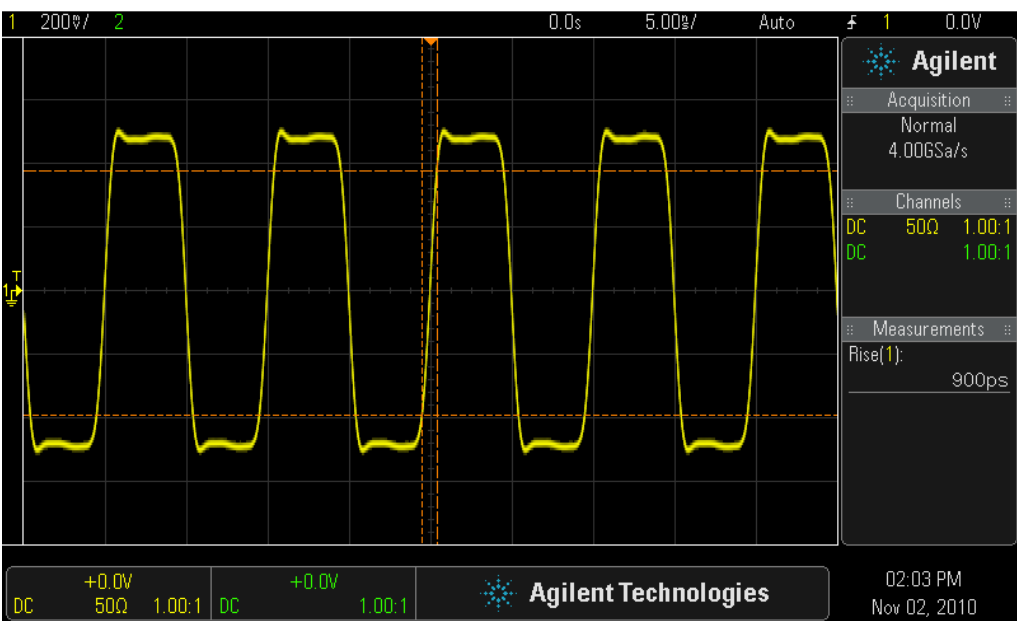
Trigger



Bandwidth

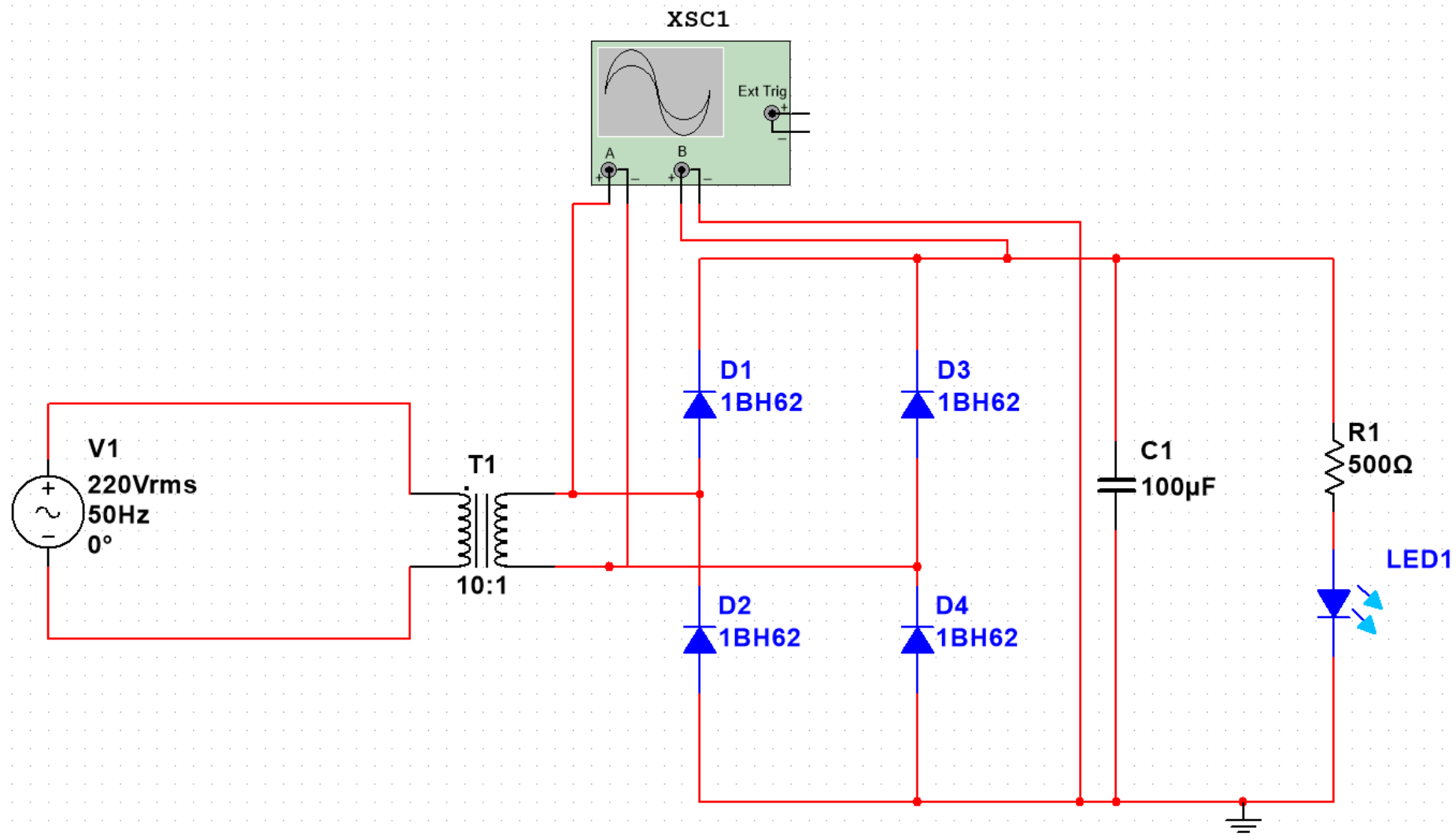


100-MHz

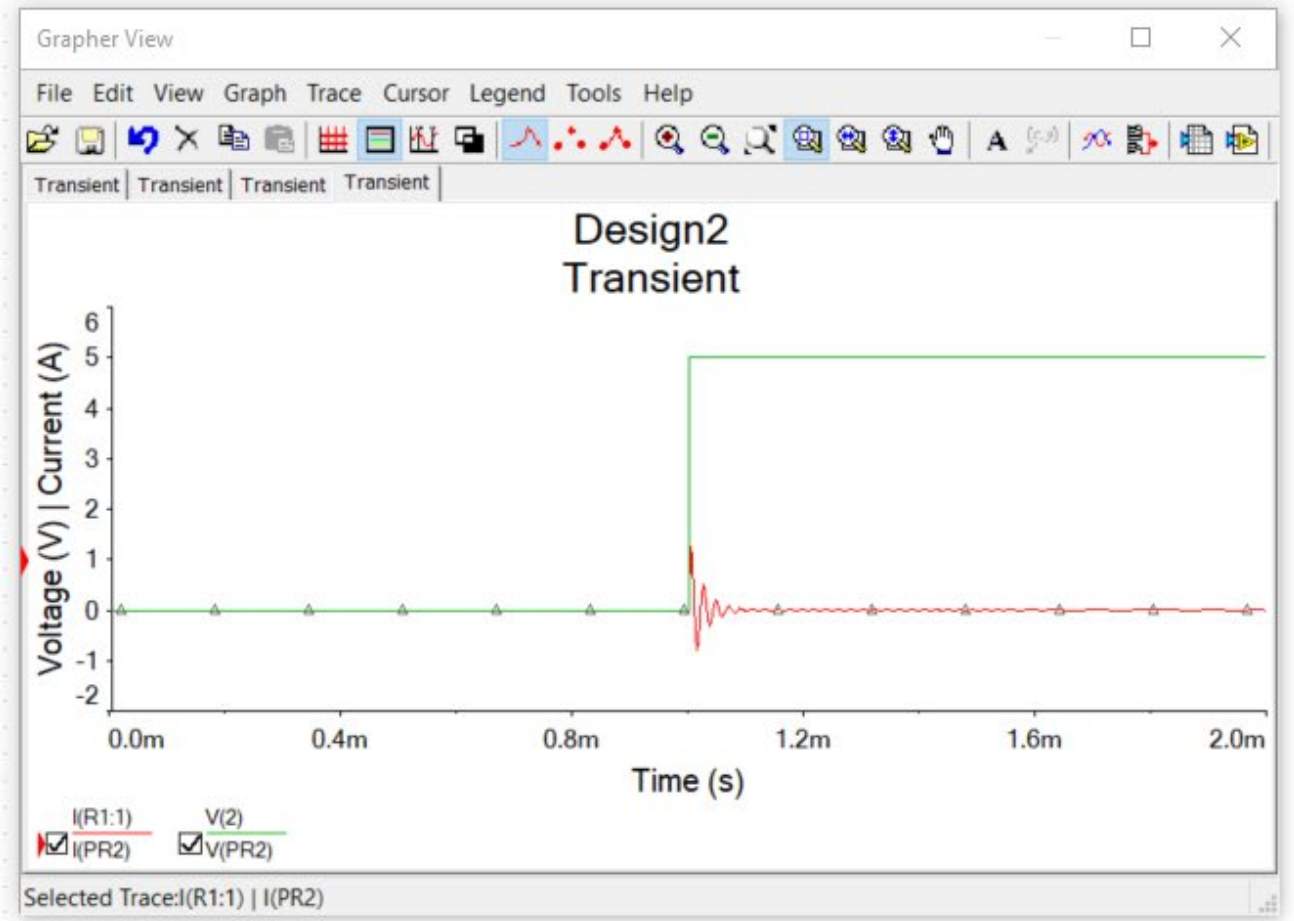
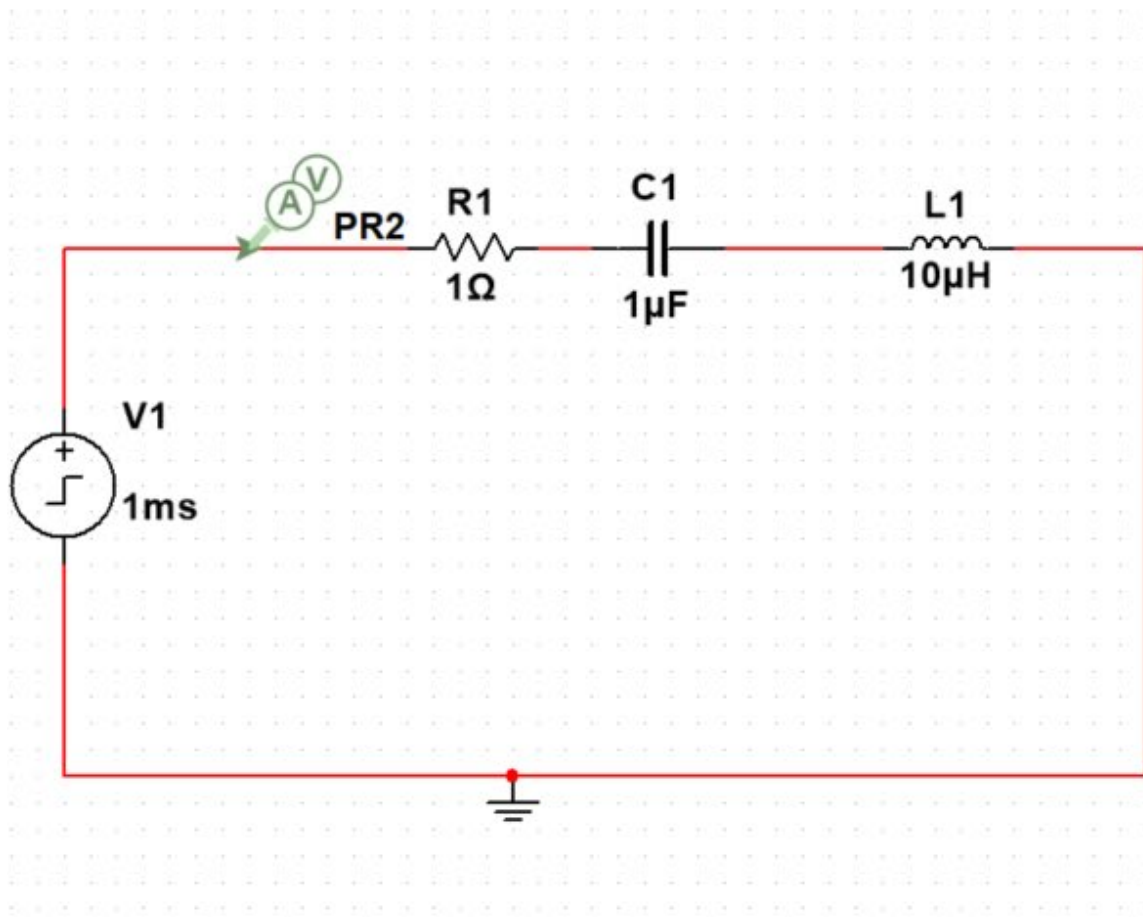


500-MHz

Demo 2: AC-DC



Demo 3: transient



Q&A