Using Multisim Live with NI ELVIS III

Publish Date: May 14, 2018

Overview

This document guides you through using Multisim Live and NI ELVIS III to simulate and build an active low pass filter. It will also show how to connect to the NI ELVIS III Bode Analyzer to measure real circuit and compare the real data with the simulation.

Table of Contents

1. Getting Started with Multisim Live

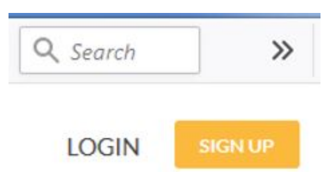
Multisim Live is an online SPICE circuit simulator which gives students the ability, to create, interactively simulate, learn, and share circuit all using a web browser. Using the same technology as Multisim (for desktop), Multisim Live adds to a student's circuit learning experience by giving the students the chance to learn on any device (cell phone, tablet, computer, etc) without installing any software and plugging students into a community of over 30,000 public circuits to help spur ideas and accelerate learning.

Registering and Login

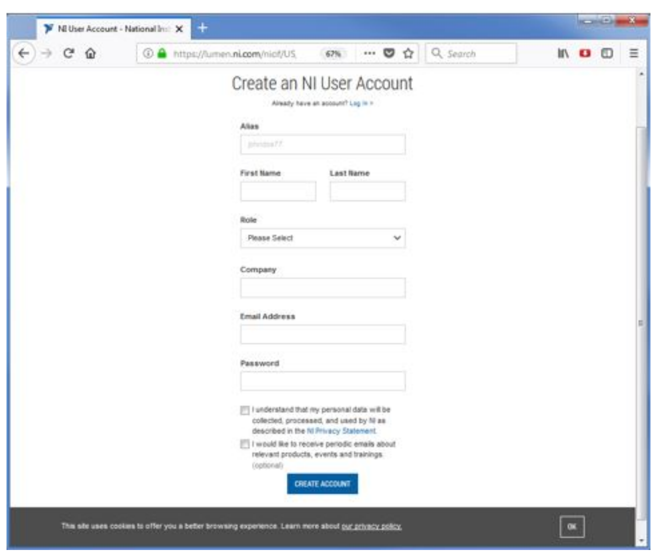
You can access Multisim Live at multisim.com (https://www.multisim.com/)

Note: This instruction was created using a laptop, if you are using a tablet or phone the images will be different.

- Before you can access Multisim Live, you must create a ni.com profile. Click the **Sign Up** button to create an account, if you already have one, go to step 7.



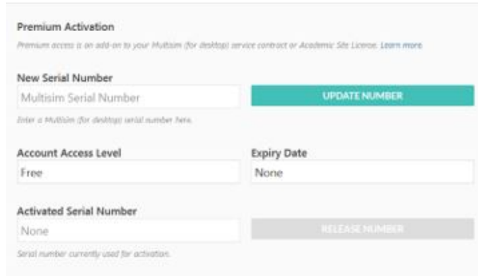
- Enter your personal information in the fields.
- Check the **NI Privacy Statement box.**
- Click **Create Account**.



- Log into the email account you used to create the profile. You should have received an email asking you to confirm the registration. Click on the link in the email to activate your NI account.
- Once you have activated your account, click on **Login** at multisim.com (https://www.multisim.com/) to enter the site.
- Enter your email address, and password then click the **Login** button.

Edit Your Profile

- If you already have a valid Multisim desktop serial number such as a Student, Education, or Professional Edition, enter your serial number into the New Serial Number textbox, this gives you premium access which has more features than the free access level. You can view the comparison between these levels by clicking on Pricing (https://www.multisim.com/pricing/) link.

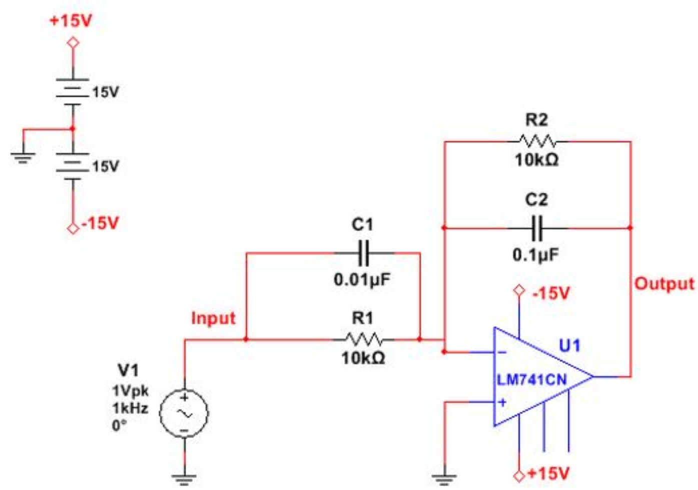


- Optionally, you can edit your Multisim Live User Account profile by adding images, biography and so on.

2. Creating a Circuit

Click the **Create Circuit** to enter the Multisim Live environment.

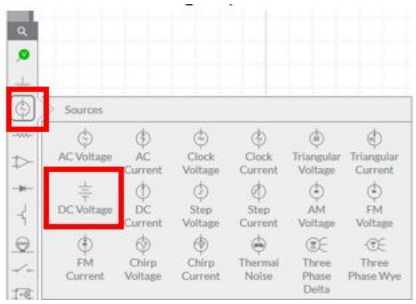
In this example, you will create a single stage low-pass filter as shown below.



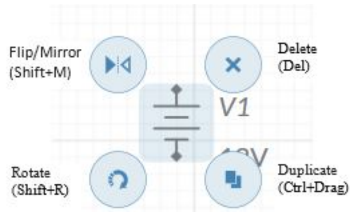
Placing Components

The Components toolbar is on the left side of the screen, you can access all components from this toolbar.

- Place a DC voltage source on the workspace.
  - Click the **Sources** group and then select **DC Voltage**.

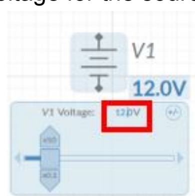


- Adjust the component properties.
  - When you click on a component, you should see four symbols around the component. This allows users using touch screen only devices to edit the component properties. The shortcut keys are for users that have access to a keyboard, it allows you to place parts faster.



- You can adjust the DC voltage by clicking on the **12V.**

- Use the slider or click on **12V** to enter a new DC voltage for the source, for this circuit enter 15V.



- Click the V1 DC source to select it, then click the **Duplicate** to make a copy.
- Click the 15V to change the second DC source to output negative voltage.

- Click the +/- symbol to set the voltage to -15V. Click on the empty space to close exit this mode.

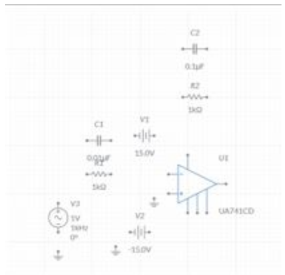
Note: An alternative method for changing component properties is to select the part on the workspace and then select the **Open configuration pane**. The pane gives you access to all the component properties and makes changing the multiple component properties faster in some cases.

- Place the remaining components from the following table on the workspace. If you are using a keyboard, use the shortcut keys to place the parts quickly.

Component (Keyboard shortcut)	Quantity	Library Location	Value(s)
Ground (G)	4	Schematic connectors	NA
Resistor (R)	2	Passive	10 K
Capacitor (C)	2	Passive	0.1 uF, 0.01 uF

Opamp	1	Analog	U741
AC Source (V)	1	Sources	NA

- Arrange the components to make wiring the circuit easy, remember, you can use the flip, mirror and rotate function to position the parts.

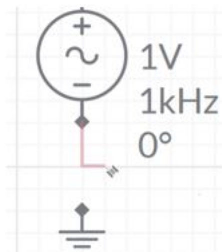


Wiring Components

There are several methods to connect components.

Method 1. Place a wire from pin-to-pin.

- Click the component pin to begin the wire mode. Your mouse cursor should look like a wire spool and there is a red wire attached to your mouse cursor.
- Place the mouse cursor on the other pin that you want to connect and click to complete the connection.



Method 2: Pin-to-pin connection

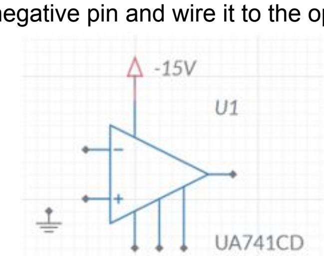
- Move the component so that the two pin endpoints touch each other, Multisim will automatically connect both parts.

Method 3: Virtual Connection.

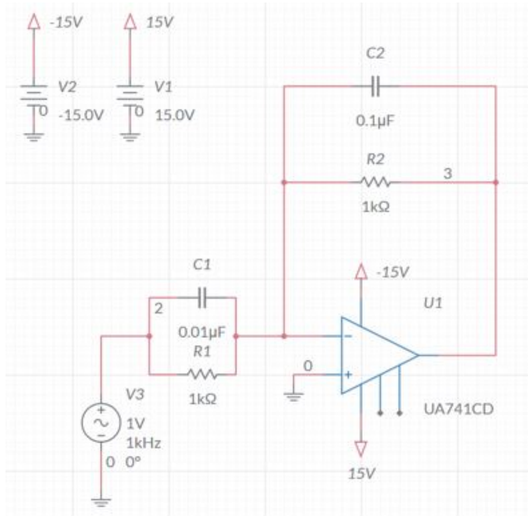
- Place a **Connector** (shortcut key is X) from the **Schematic connector** group on the workspace.

- Wire the connector to the -15V DC voltage source.
- Double click on **IO1**.

- In the **ID** textbox, enter **-15V**.
- Select the -15V connector and then click **Duplicate** to make a copy.
- Place the new connector above the opamp negative pin and wire it to the opamp.

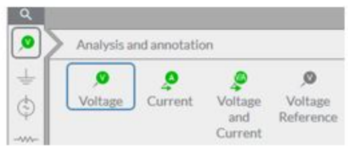


- Wire the remaining components. Your completed circuit should look like the schematic below.

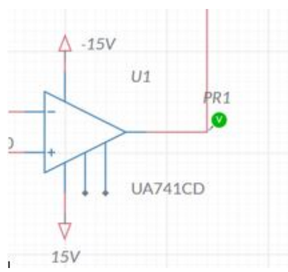


3. Simulating Circuits

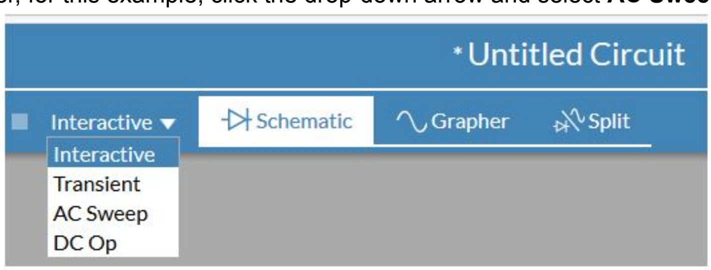
- Place a **Voltage** probe from the **Analysis and annotation** group.



- Place the probe on a wire at the output node.



- The circuit you just built is a filter, which means you should use the AC Sweep to view the circuit response. The default simulation mode is Interactive which is used when looking at the output vs time and will respond to real-time changes in component values; however, for this example, click the drop-down arrow and select **AC Sweep**.

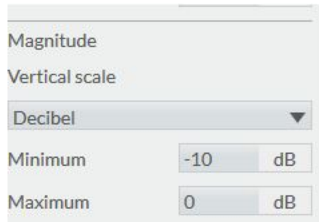


- Adjust the AC Sweep simulation settings.
  - In the **Configuration Pane** set the **Start frequency** to **1 Hz** and the **Stop frequency** to**1 kHz**.
  - Click the drop-down arrow in **Vertical Scale** and set the scale to **Decibel**.

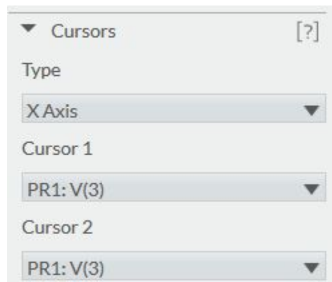
- Click the **Run** to begin the simulation.

- Adjust the Grapher properties.
  - Uncheck the **Phase: PR1 V(3)** to hide the phase plot.

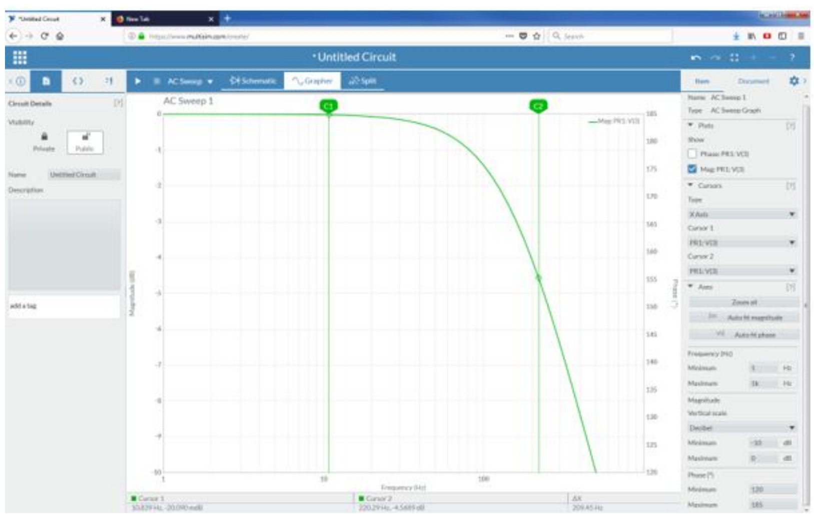
  - Change the **Maximum** to **0dB** in the **Vertical Scale** area.



  - Click the **Type** drop-down arrow in the **Cursors** area and select **X Axis**.



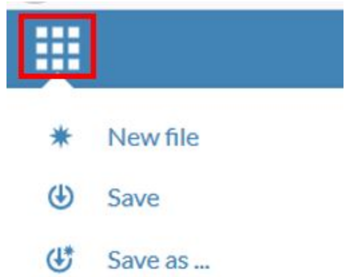
  - Slide the cursors **C1** and **C2** along the plot to find the -3dB point and see the drop-off rate. You can view the simulation data in the table at the bottom.



- Click **Schematic** or **Grapher** to display the schematic or simulation results, the Split option shows both in a split screen.

Save Your Circuit

- You can save your circuit to the web by selecting the **File navigation menu** and then **Save As**.

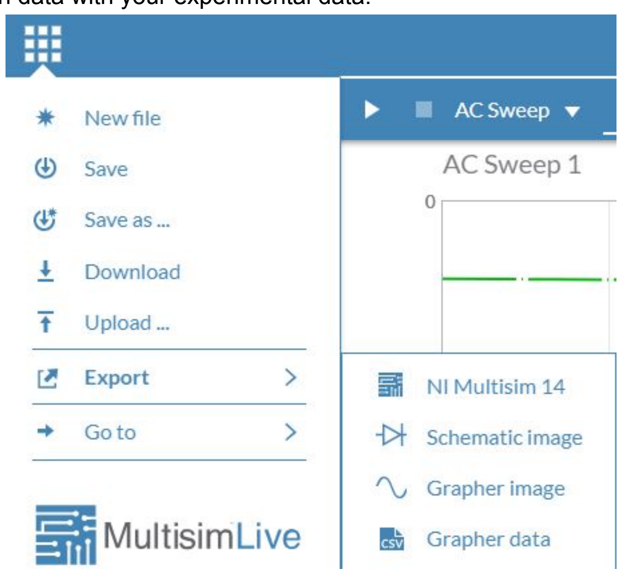


- In the **Name** textbox, enter the circuit name.
- You can add a description or tags to help other users find and learn about your circuit.

Note: If you have a premium access level, you can save the circuit as a **private,** and you can decide who can view it. All free access level users must save the circuit to **public** where everyone can see your circuit.

4. Export Simulation Data

- You can export the simulation data by selecting **File Navigation Menu»Export»Grapher Data**.This will be important later on to integrating your simulation data with your experimental data.



- Multisim saves the data as a .csv file and puts the file in your web browser default download path.

5. Getting Started with NI ELVIS III

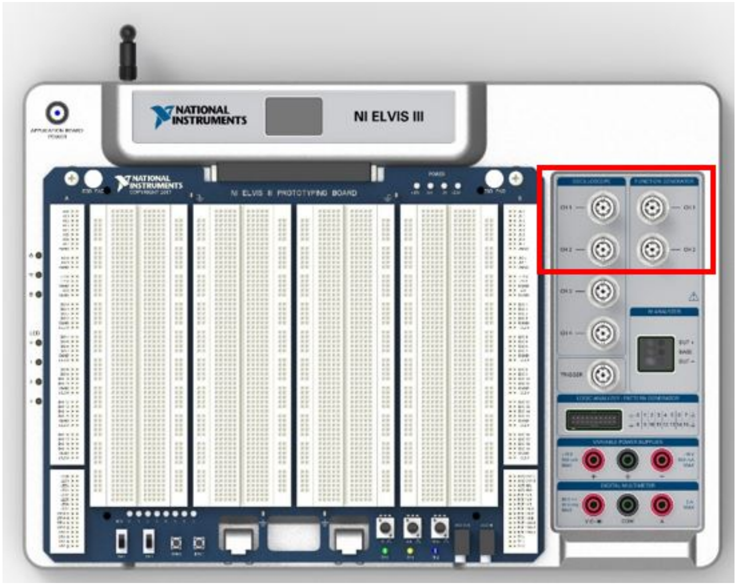
Go to measurementslive.ni.com (https://measurementslive.ni.com/) and follow the on-screen instructions. If you have not used your NI ELVIS III before, a helpful link will pop-up directing you to getting started instructions.

Taking Measurements

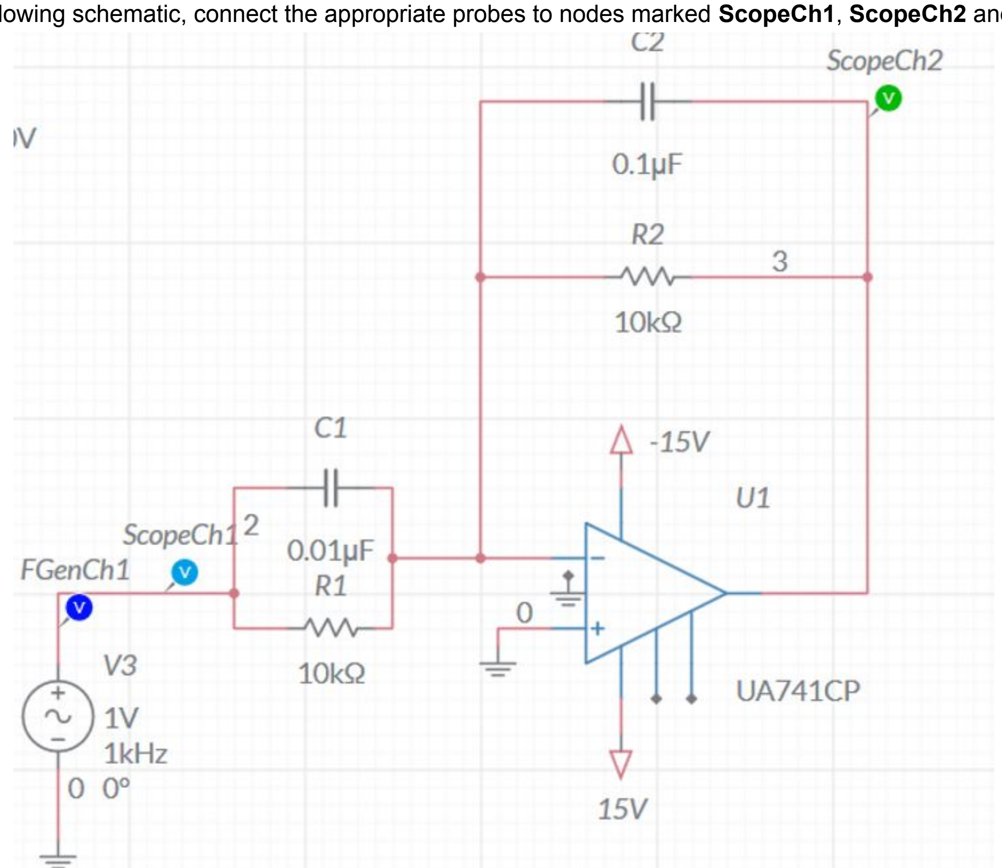
Build the low-pass filter circuit mentioned above on the breadboard. If you do not know how the breadboard a circuit, you can view the How to Breadboard with NI ELVIS III tutorial. (http://ni.com/tutorial/54749/en/)

Connect to the Instruments

- Connect a probe to the Function Generator **CH 1**.
- Connect two more probes, one to the Oscilloscope **Ch1,** and the other to **CH 2**.



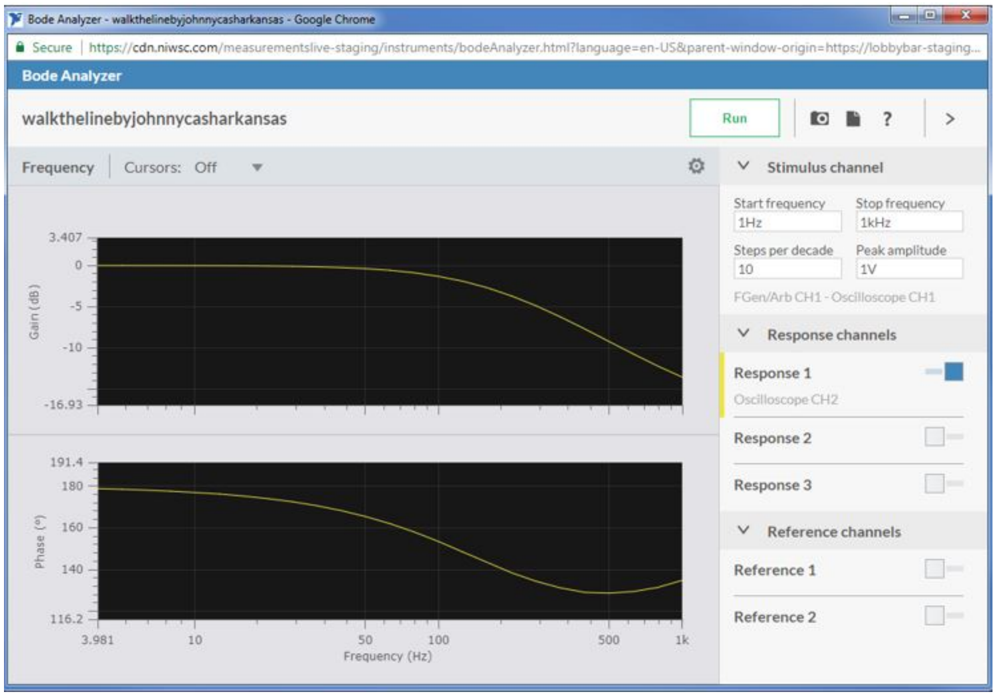
- Using the following schematic, connect the appropriate probes to nodes marked **ScopeCh1**, **ScopeCh2** and **FGenCH1.**



Take a Measurement with the Bode Analyzer

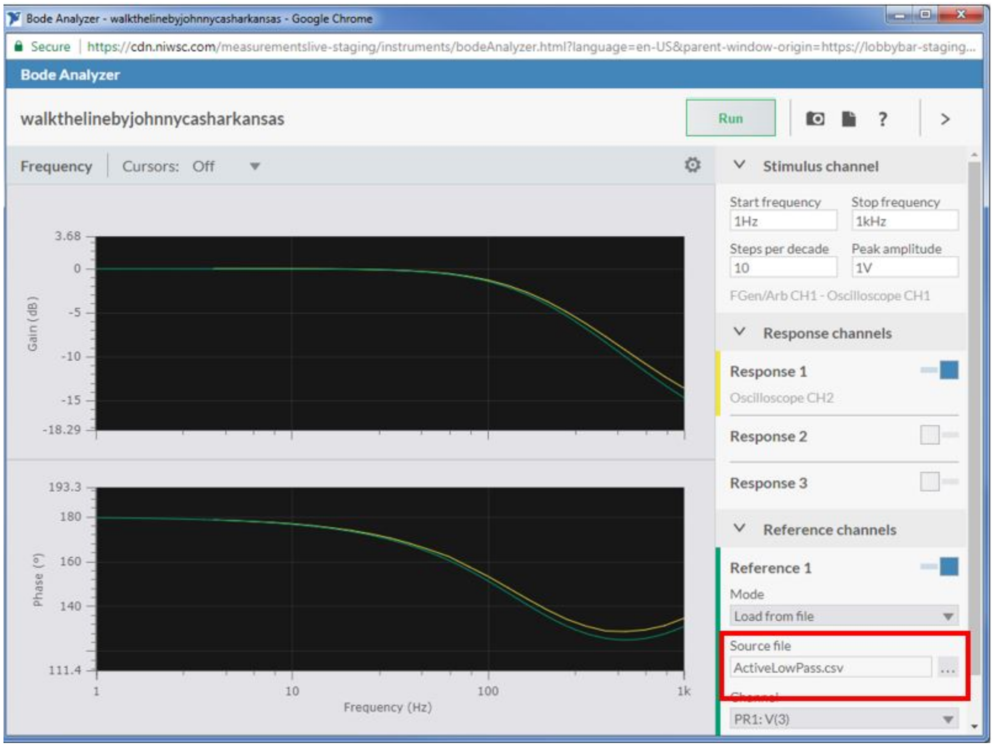
- From the MeasurementsLive click **Instruments** and select the **Bode Analyzer**.

- Set the sweep frequency range to match with the simulation.
  - Set the **Start frequency** to **1 Hz**.
  - Set the **Stop** frequency to 1 **kHz.**
  - Click **Run** to acquire the signal. The circuit response should look like the following image.



6. Compare a Multisim Live Simulation to an NI ELVIS III Experiment

- In the Bode Analyzer or any other soft front panel you are using, enable the **Reference 1** switch.
- Earlier, you exported the simulation data to a .csv file from Multisim Live, click the **Browse** button and navigate to that file.
- **Note: you must import data from a simulation that matches the type of experiment you are performing.** In this instance we did an AC Sweep to find the response of the circuit over different AC frequencies and so we must use the Bode Analyzer to get the same output from our real circuit.



The **green curve shows the simulation data** while the **yellow curve is the real measurement**. By comparing the simulation with real data, you can be confident that your circuit is working properly. If there is a problem with the real circuit, this feature can help you isolate the problem areas by comparing the simulation at different nodes in the circuit with the corresponding nodes in the real circuit.