GETTING STARTED GUIDE

NI Circuit Design Suite

This part contains the following step-by-step tutorial:

- *Multisim Tutorial*—*Multisim* is the schematic capture and simulation program designed for schematic entry, simulation, and feeding to downstage steps, such as PCB layout. This tutorial introduces you to Multisim and its many functions.

Some of the features described may not be available in your edition. Refer to `ni.com` for a list of the features available in your edition of Circuit Design Suite.

Contents Part 1

Instruktioner för TSTE93

- Skapa mappen TSTE93 under ditt hemkonto som "H\:TSTE93\" och spara filer i denna mapp
- Starta programmet med "Start/All Programs/NI Multisim 14.0"
- Se till att alla komponenter är väldefinierade enligt tabellen på på s. 9-10
  + Komponenter som har en storlek ("footprint") markeras blå och kan användas i Ultiboard
  + Komponenter utan definierad storlek är svarta och kommer inte med på kretskortet
- Exempelfilerna som omnämns i övningarna nås med "File/Open samples…/Getting Started/"
  + Gör helst egna nya filer då exemplen ligger i en gemensam mapp och kan vara ändrade
  + Annars kopierar du filerna innan du använder dem för att inte ändra i originalen
- Avsluta med att exportera konstruktionen till Ultiboard
  + Spara först konstruktionen med "File/Save"
  + Exportera sedan med "Transfer/Transfer to Ultiboard/Transfer to Ultiboard File…"

Multisim Tutorial

This section contains a tutorial that introduces you to Multisim and its many functions.

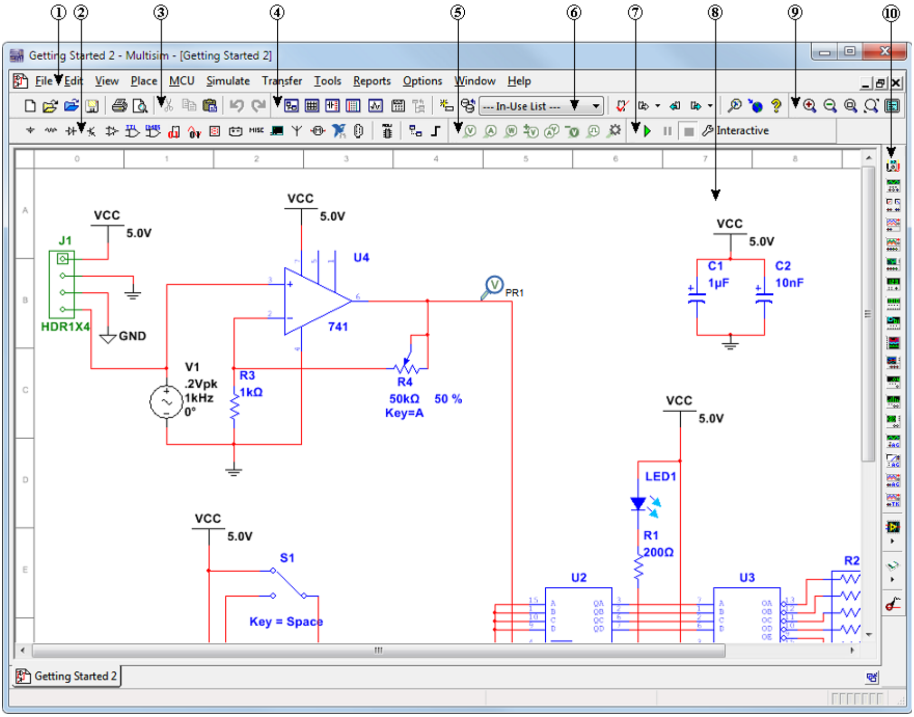
Introduction to the Multisim Interface

Multisim is the schematic capture and simulation application of Circuit Design Suite, a suite of EDA (Electronics Design Automation) tools that helps you carry out the major steps in the circuit design flow.

Multisim is designed for schematic entry, simulation, and exporting to downstage steps, such as PCB layout.

Multisim User Interface

The Multisim user interface includes the following elements:



1	Menu Bar	4	Main Toolbar	7	Simulation Toolbar	9	View Toolbar
2	Component Toolbar	5	Place Probe Toolbar	8	Workspace	10	Instruments Toolbar
3	Standard Toolbar	6	In-Use Toolbar				

Refer to the table below as needed:

	Element	Description
1	**Menu Bar**	Contains the commands for all functions.
2	**Component** toolbar	Contains buttons that you use to select components from the Multisim database for placement in your schematic.
3	**Standard** toolbar	Contains buttons for commonly-performed functions such as Save, Print, Cut, and Paste.
4	**Main** toolbar	Contains buttons for common Multisim functions.
5	**Place probe** toolbar	Contains buttons that you use to place various types of probes on the design. You can also access **Probe Settings** from here.

	Element	Description
6	**In-Use List**	Contains a list of all components used in the design.
7	**Simulation** toolbar	Contains buttons for starting, stopping and pausing simulation.
8	Workspace	This is where you build your designs.
9	**View** toolbar	Contains buttons for modifying the way the screen is displayed.
10	**Instruments** toolbar	Contains buttons for each instrument.

Overview

This tutorial leads you through the circuit design flow, from schematic capture to simulation. After completing the steps outlined on the following pages, you will have designed a circuit that samples a small analog signal, amplifies it and then counts the cycles on a simple digital counter.

If you wish to skip steps, or only complete specific sections of this tutorial, you can use these pre-made files, found in ...\Circuit Design Suite <version>\samples\Getting Started\:

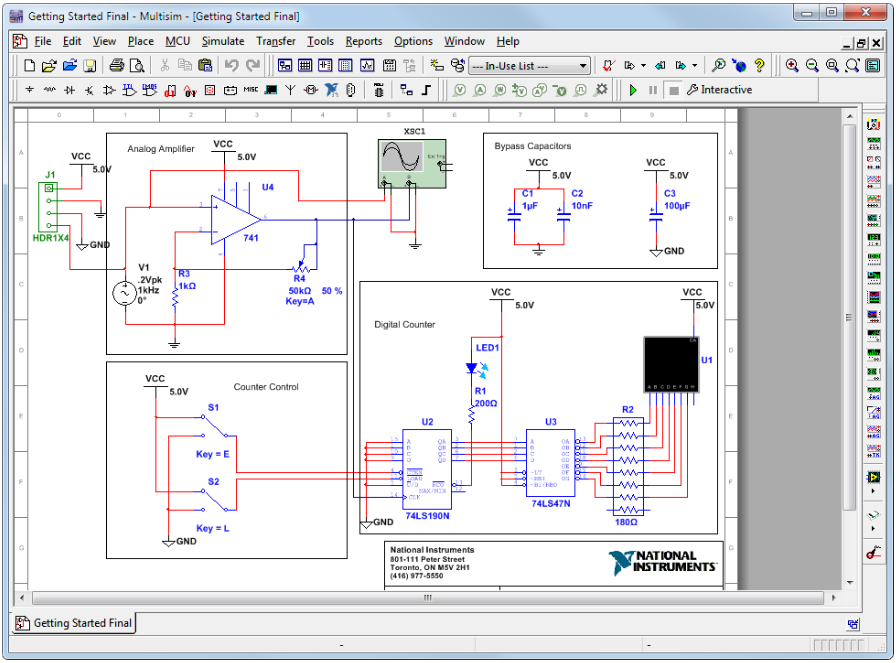
- Getting Started 1—The design with all components placed, ready to be wired. Use if you do not wish to place all of the components yourself.
- Getting Started 2—The wired design, without the oscilloscope.
- Getting Started Final—The ready-to-simulate design file.

Helpful tips are indicated by an icon in the left column, for example:

Tip You can access the online help at any time by pressing F1 on your keyboard, or by clicking the **Help** button in a dialog box.

Schematic Capture

In the following sections, you will place and wire the components in the design shown below.



Creating the File

Complete the following steps to create the design file:

1. Launch Multisim.

   A blank file called `Design1` opens on the workspace.

2. Select **File»Save as** to display a standard Windows Save dialog.
3. Navigate to the location where you wish to save the file, enter `MyGettingStarted` as the **File name**, and click the **Save** button.

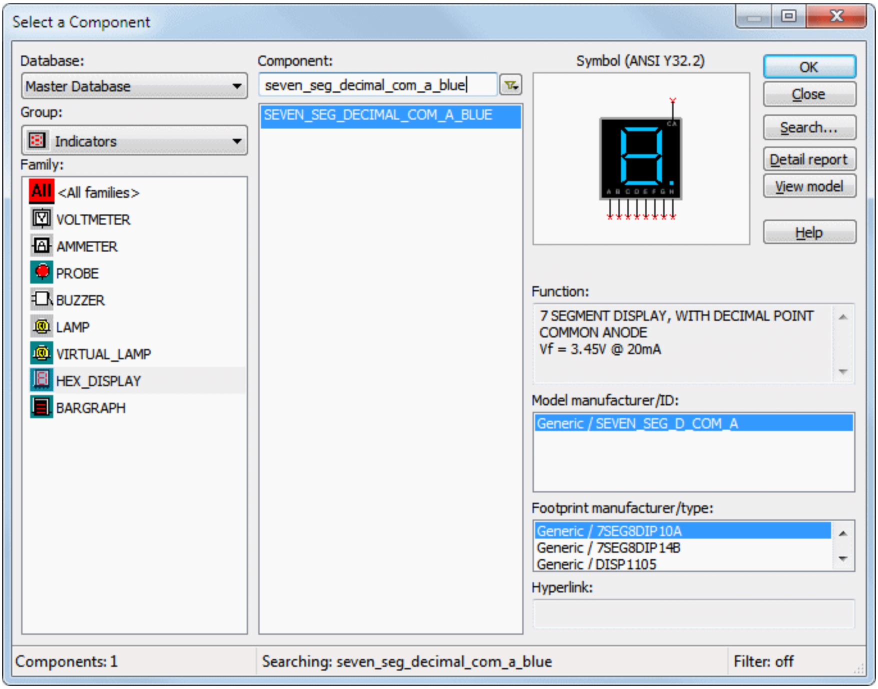
Tip To guard against accidental loss of data, set up a timed **Auto-backup** of the file in the **Save** tab of the **Global Options** dialog box.

Placing the Components

Complete the following steps to place the components on `MyGettingStarted`:

1. Select **Place»Component** to display the **Select a Component** dialog box.
2. Select the **Indicators** component **Group** and the **HEX_DISPLAY** component **Family**.

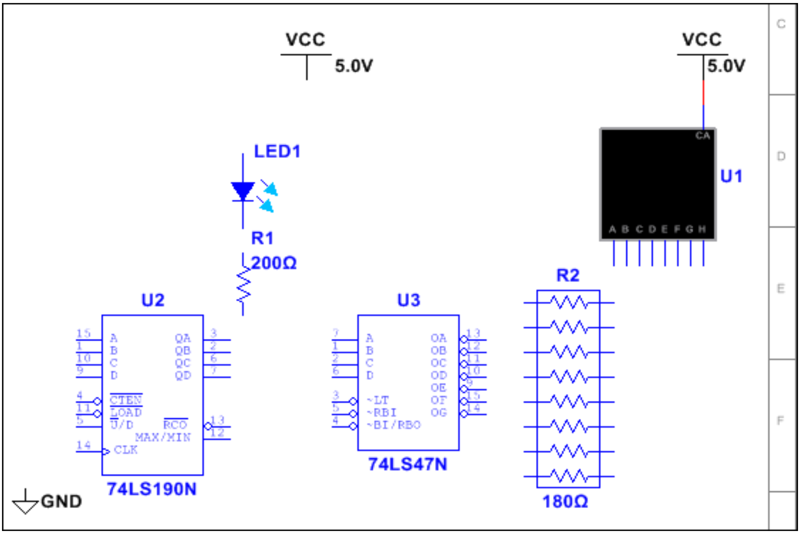
3. Select `SEVEN_SEG_DECIMAL_COM_A_BLUE` from the **Component** list and click **OK**.



The component appears as a "ghost" on the cursor.

4. Move the cursor to the bottom-right of the workspace and click to place the component. Note that the Reference Designator for this component is `U1`.

5. Place the remaining components in the Digital Counter area as shown below.



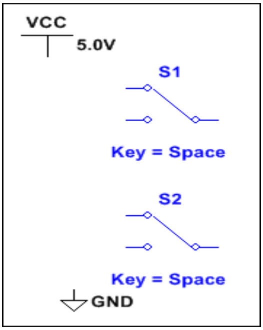
Tip While placing the 200 Ω resistor, press <Ctrl-R> to rotate it to a vertical orientation.

After placing R2, double-click on it and change the **Resistance** to 180 Ω.

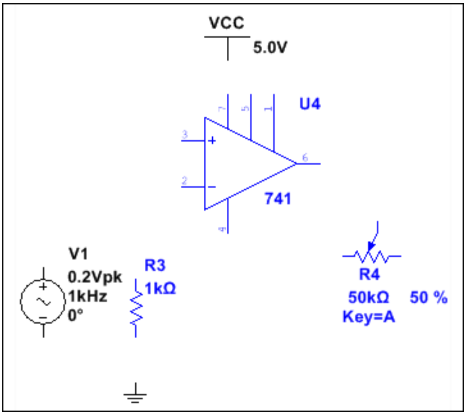
Tip The **Group** and **Family** location of each component is listed in the *Component Locations* section.

Note By default, resistors, capacitors and inductors have no footprint. However, to export components of this type to PCB layout in Ultiboard, you must select a footprint from the **Footprint manufacturer/type** field. Source and ground components have no available footprints and do not export to layout.

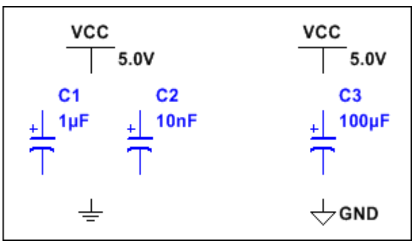
6. Place the components in the Counter Control section as shown below.
   Right-click on each SPDT switch and select **Flip horizontally**.



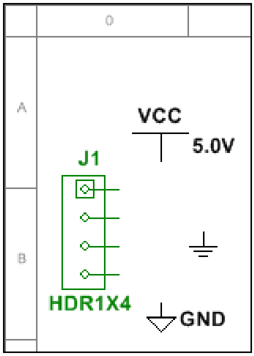
7. Place the components in the Analog Amplifier section as shown below, rotating as needed.
   Double-click on the AC voltage source (V1) and change **Voltage (Pk)** to 0.2 V.



8. Place the components in the Bypass Capacitors section as shown below.



9. Place the header and associated components as shown below.



Component Locations

The following shows you where to locate all components for this design in the **Select a Component** dialog box.

This tutorial only uses generic components from the Master database.

Tip Reference Designators (for example, U1, U2) are assigned in the order the components are placed. If you place components in a different order than in the original design, the numbering will differ. This will not affect the operation of the design in any way.

RefDes and Component	Group	Family	Footprint manufacturer/ type
LED1 - LED_blue	Diodes	LED	Ultiboard/ LED9R2_5vb
VCC GND - DGND GROUND	Sources	POWER_ SOURCES	—
U1 - SEVEN_SEG_DECIMAL_ COM_A_BLUE	Indicators	HEX_DISPLAY	Generic/ 7SEG8DIP10A
U2 - 74LS190N U3 - 74LS47N	TTL	74LS	IPC-2221A/2222 / NO16
R1 - 200 Ω	Basic	RESISTOR	IPC-7351/ Chip-R0805

RefDes and Component	Group	Family	Footprint manufacturer/ type
R2 - 8Line_Isolated	Basic	RPACK	IPC-2221A/2222/ DIP-16
R3 - 1k	Basic	RESISTOR	IPC-7351/ Chip-R0805
R4 - 50k	Basic	POTENTIOMETER	Generic/ LIN_POT
S1, S2 - SPDT	Basic	SWITCH	Generic/SPDT
U4 - 741	Analog	OPAMP	IPC-2221A/2222/ DIP-8
V1 - AC_VOLTAGE	Sources	SIGNAL_VOLTAGE_ SOURCES	—
C1 - 1 μF C2 - 10 nF C3 - 100 μF	Basic	CAP_ELECTROLIT	IPC-7351/ Chip-C1210
J1 - HDR1X4	Connectors	HEADERS_TEST	Generic/ HDR1X4

Note When placing resistors, inductors, or capacitors, the **Select a Component** dialog box has slightly different fields than for other components. When placing these, you can choose any combination of the component's value (for example, the resistance value), type (for example, carbon film), and so on. If you are placing a component that will be exported to PCB layout, the combination of values that you select must be available in a commercially available component.

Wiring the Design

All components have pins that you use to wire them to other components or instruments. As soon as your cursor is over a pin, the cursor changes to a crosshair, indicating you can start wiring.

Tip You can wire the design that you placed on the workspace or you can use `Getting Started 1` found in `...\Circuit Design Suite <version>\samples\Getting Started\`.

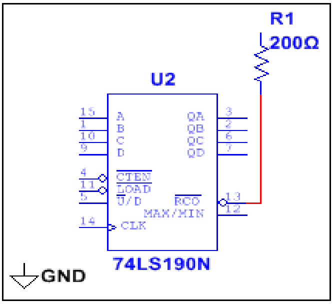
Complete the following steps to wire the design:

1. Click on a pin on a component to start the connection (your cursor turns into a crosshair) and move the mouse.

   A wire appears, attached to your cursor.

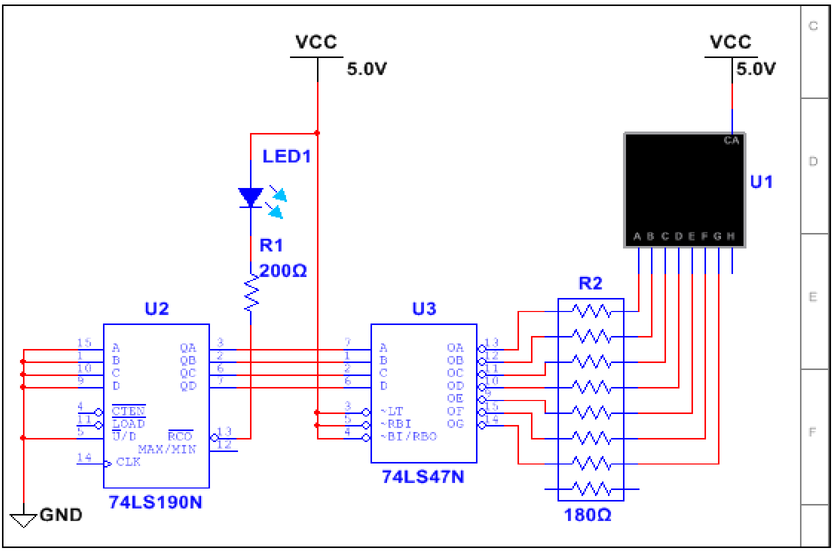
2. Click on a pin on the second component to finish the connection.

   Multisim automatically places the wire, which snaps to an appropriate configuration, as shown below.



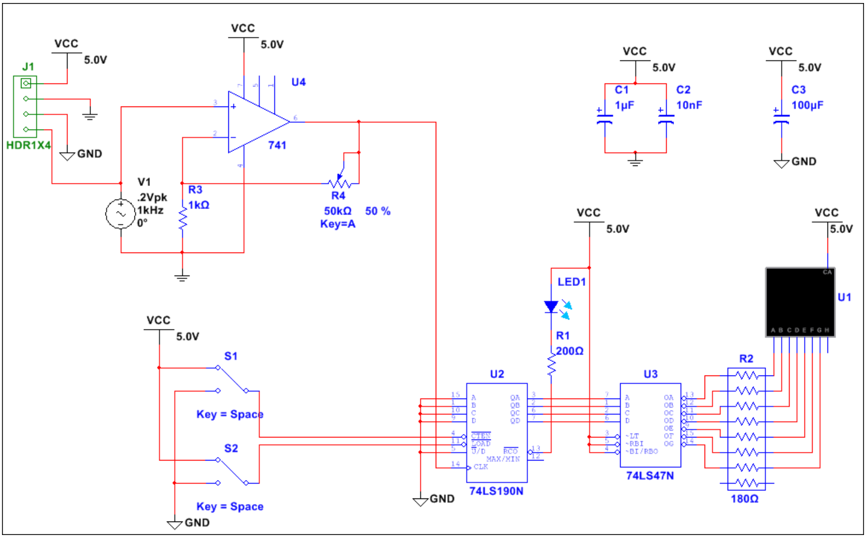
Tip You can also control the flow of the wire by clicking on points as you move the mouse. Each click "fixes" the wire to that point.

3. Finish wiring the Digital Counter section as shown below.



Tip **Virtual Wiring**—To avoid clutter, you can use virtual connections between the Counter Control and Digital Counter sections using on-page connectors. Refer to the *Multisim Help* for details.

4. Finish wiring the design as shown below.



Simulation

Simulating your designs with Multisim catches errors early in the design flow, saving time and money.

Virtual Instrumentation

In this section, you will simulate the design and view the results with the virtual oscilloscope.

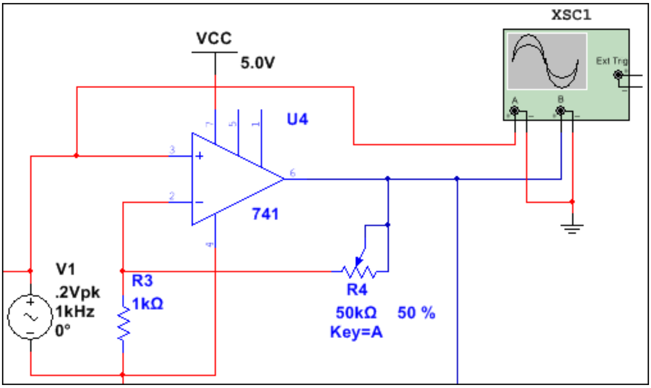
Tip You can also use `Getting Started 2` found in `...\Circuit Design Suite <version>\samples\Getting Started\`.

1. Set up the interactive keys for switches S1 and S2:
   a. Double-click on each and select the **Value** tab.
   b. Select “E” for S1 and “L” for S2 in the **Key for toggle** field.
2. Press <E> to enable the counter.

   Or

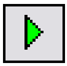
   Click on the widened switch arm that appears when you hover the cursor over S1.
3. Select **Simulate»Instruments»Oscilloscope** to place the oscilloscope on the workspace.

4. Wire the oscilloscope as shown below.



Tip To differentiate between traces on the oscilloscope, double-click on the wire connected to the scope's **B** input. Select a **Net color** that differs from the wire connected to the **A** input, for example blue.

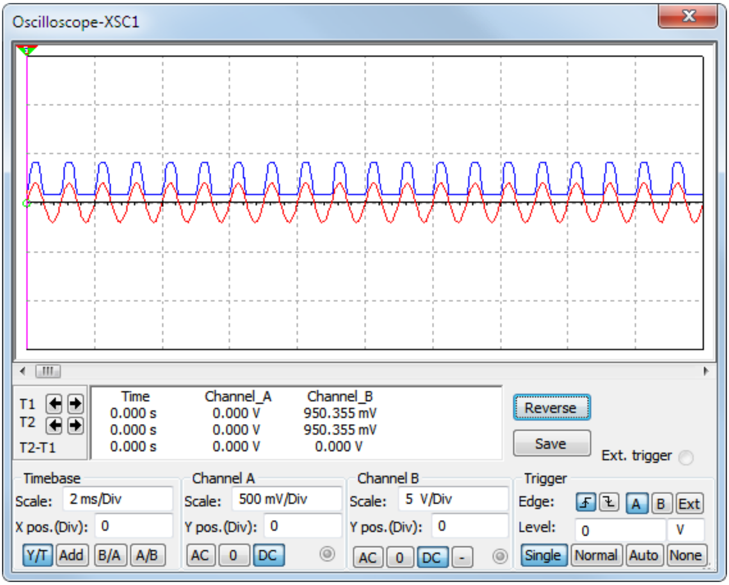
5. Double-click on the oscilloscope icon to show its front panel.
6. Select **Simulate»Run**.



The output of the opamp appears on the oscilloscope.

7. Adjust the **Timebase** to 2 ms/Div and Channel A's **Scale** to 500 mV/Div. Click **Reverse** to change the background to white.

   The following displays on the oscilloscope:



   As the design simulates, the 7-segment display counts up and the LED flashes at the end of each count cycle.

8. Do the following:

   a. Press <E> while the simulation is running to enable or disable the counter.

   b. Press <L> to load zeros into the counter.

   c. Press <Shift-A> to observe the effect of decreasing the potentiometer's setting. Repeat, pressing <A> to increase.

Tip Instead of pressing the above-mentioned keys, you can directly manipulate the interactive components on the schematic with your mouse.

Analysis

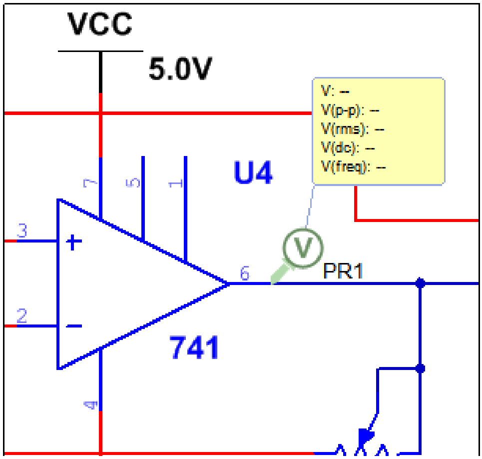
In this section, you will use **AC Sweep** to verify the frequency response of the amplifier.

Complete the following steps to perform an **AC Sweep** at the output of the op-amp:

1. Open `Getting Started Final` found in `...\Circuit Design Suite <version>\samples\Getting Started\`.

2. Click on the **Place voltage probe** button in the **Place probe** toolbar.

3. Click to place the voltage probe on the trace that is wired to pin 6 of the opamp.



Tip When a probe is placed on a wire, it will be green as shown above. If it is not placed on a wire, it will be greyed-out and inactive.

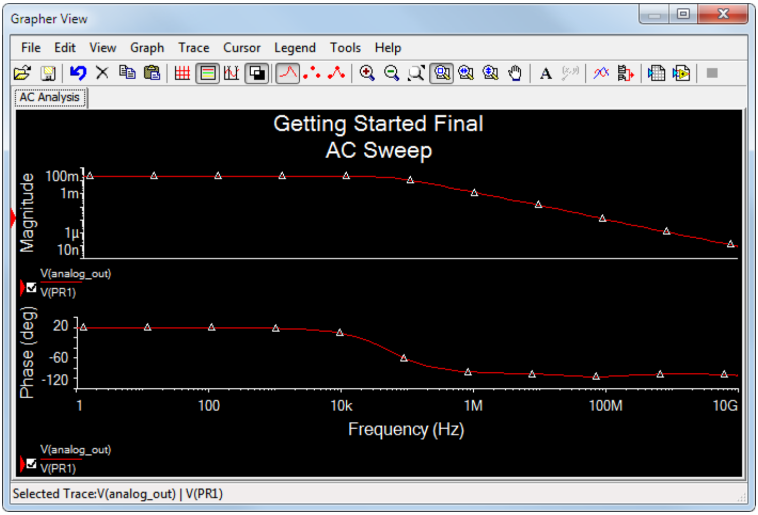
4. Select **Simulate»Analyses and simulation**.

   The **Analyses and Simulation** dialog box displays.

Tip Instead of using the menu, you can click **Interactive** in the **Simulation** toolbar.

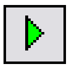
5. Select **AC Sweep** in the **Active Analysis** column and click **Run**.

   The **Analyses and Simulation** dialog box closes and the **Grapher** appears with the results of the analysis.



Tip The **Interactive** button in the **Simulation** toolbar has changed to **AC sweep** to reflect the new selection. You can also access the **Analyses and Simulation** dialog box from this button.

Tip Once the analysis has been selected, you can click **Run** to run it again.

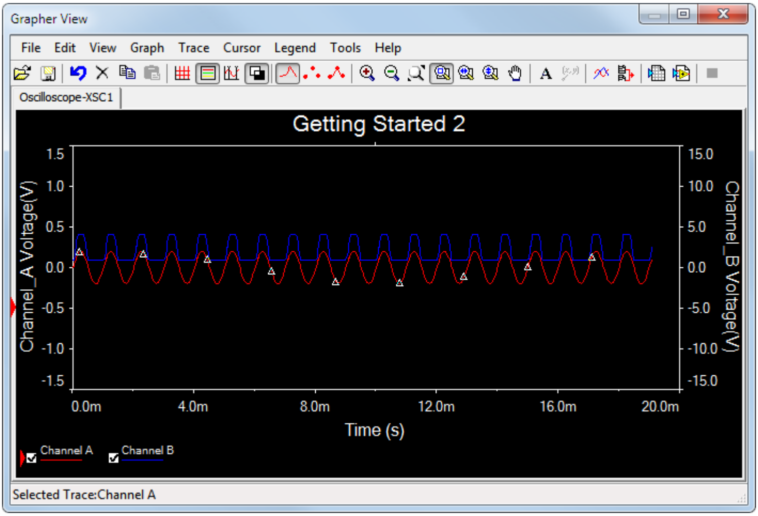


The Grapher

The **Grapher** is a multi-purpose display tool that lets you view, adjust, save and export graphs. It also displays graphs of traces for some instruments (for example, the oscilloscope).

Complete the following steps to view results of a simulation on the **Grapher**:

1. Confirm that **Interactive** is selected in the **Simulation** toolbar and run the simulation with the oscilloscope as described earlier.
2. Select **View»Grapher** if the **Grapher** is not displayed.



The Postprocessor

Use the **Postprocessor** to manipulate the output from analyses and plot the results on a graph or chart. Types of mathematical operations that can be performed on analysis results include arithmetic, trigonometric, exponential, logarithmic, complex, vector and logic.

Reports

You can generate a number of reports in Multisim: **Bill of Materials** (BOM), **Component Detail Report**, **Netlist Report**, **Schematic Statistics**, **Spare Gates** and the **Cross Reference Report**.

The following section uses the **BOM** as an example for the tutorial design.

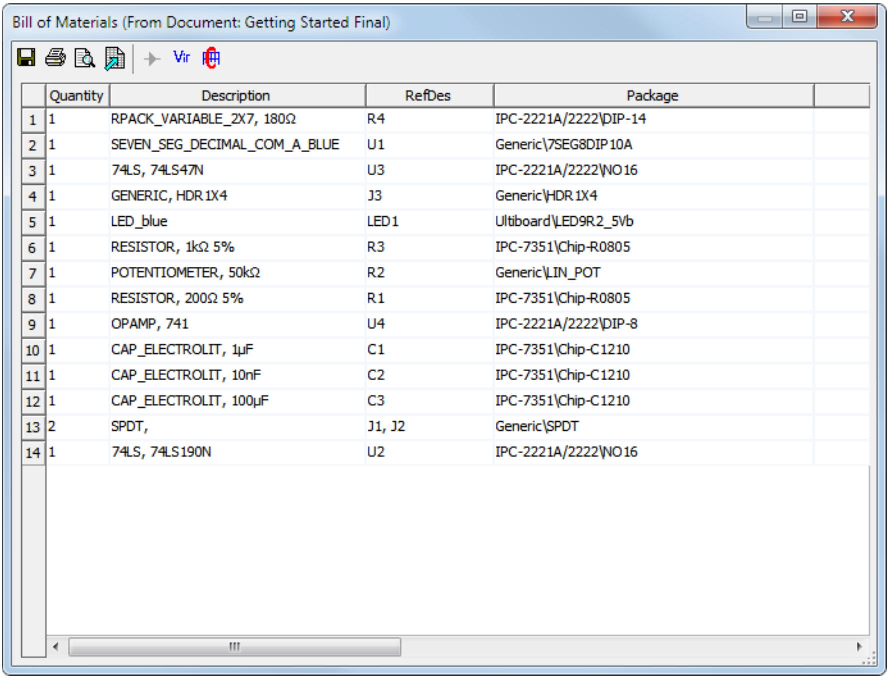
Bill of Materials

A bill of materials lists the components used in a design, providing a summary of the components needed to manufacture the circuit board.

Information provided for each component includes:

- quantity required.
- description, including the type of component (for example, resistor) and value (for example, 200 Ω).
- reference designator.
- package or footprint name.

Select **Reports»Bill of Materials** to display a report similar to this:



	Quantity	Description	RefDes	Package
1	1	RPACK_VARIABLE_2X7, 180Ω	R4	IPC-2221A/2222\DIP-14
2	1	SEVEN_SEG_DECIMAL_COM_A_BLUE	U1	Generic\7SEG8DIP10A
3	1	74LS, 74LS47N	U3	IPC-2221A/2222\NO16
4	1	GENERIC, HDR1X4	J3	Generic\HDR1X4
5	1	LED_blue	LED1	Ultiboard\LED9R2_5Vb
6	1	RESISTOR, 1kΩ 5%	R3	IPC-7351\Chip-R0805
7	1	POTENTIOMETER, 50kΩ	R2	Generic\LIN_POT
8	1	RESISTOR, 200Ω 5%	R1	IPC-7351\Chip-R0805
9	1	OPAMP, 741	U4	IPC-2221A/2222\DIP-8
10	1	CAP_ELECTROLIT, 1µF	C1	IPC-7351\Chip-C1210
11	1	CAP_ELECTROLIT, 10nF	C2	IPC-7351\Chip-C1210
12	1	CAP_ELECTROLIT, 100µF	C3	IPC-7351\Chip-C1210
13	2	SPDT,	J1, J2	Generic\SPDT
14	1	74LS, 74LS190N	U2	IPC-2221A/2222\NO16

…the tutorial continues in *Ultiboard.pdf*