NI Multisim™

for Education

May 2008
374484C-01

NATIONAL INSTRUMENTS™
ELECTRONICS WORKBENCH GROUP

Worldwide Technical Support and Product Information

`ni.com`

National Instruments Corporate Headquarters

11500 North Mopac Expressway   Austin, Texas 78759-3504   USA   Tel: 512 683 0100

Worldwide Offices

Australia 1800 300 800, Austria 43 662 457990-0, Belgium 32 (0) 2 757 0020, Brazil 55 11 3262 3599, Canada 800 433 3488, China 86 21 5050 9800, Czech Republic 420 224 235 774, Denmark 45 45 76 26 00, Finland 358 (0) 9 725 72511, France 01 57 66 24 24, Germany 49 89 7413130, India 91 80 41190000, Israel 972 3 6393737, Italy 39 02 41309277, Japan 0120-527196, Korea 82 02 3451 3400, Lebanon 961 (0) 1 33 28 28, Malaysia 1800 887710, Mexico 01 800 010 0793, Netherlands 31 (0) 348 433 466, New Zealand 0800 553 322, Norway 47 (0) 66 90 76 60, Poland 48 22 3390150, Portugal 351 210 311 210, Russia 7 495 783 6851, Singapore 1800 226 5886, Slovenia 386 3 425 42 00, South Africa 27 0 11 805 8197, Spain 34 91 640 0085, Sweden 46 (0) 8 587 895 00, Switzerland 41 56 2005151, Taiwan 886 02 2377 2222, Thailand 662 278 6777, Turkey 90 212 279 3031, United Kingdom 44 (0) 1635 523545

For further support information, refer to the *Technical Support and Professional Services* appendix. To comment on National Instruments documentation, refer to the National Instruments Web site at `ni.com/info` and enter the info code `feedback`.

Important Information

Warranty

The media on which you receive National Instruments software are warranted not to fail to execute programming instructions, due to defects in materials and workmanship, for a period of 90 days from date of shipment, as evidenced by receipts or other documentation. National Instruments will, at its option, repair or replace software media that do not execute programming instructions if National Instruments receives notice of such defects during the warranty period. National Instruments does not warrant that the operation of the software shall be uninterrupted or error free.

A Return Material Authorization (RMA) number must be obtained from the factory and clearly marked on the outside of the package before any equipment will be accepted for warranty work. National Instruments will pay the shipping costs of returning to the owner parts which are covered by warranty.

National Instruments believes that the information in this document is accurate. The document has been carefully reviewed for technical accuracy. In the event that technical or typographical errors exist, National Instruments reserves the right to make changes to subsequent editions of this document without prior notice to holders of this edition. The reader should consult National Instruments if errors are suspected. In no event shall National Instruments be liable for any damages arising out of or related to this document or the information contained in it.

EXCEPT AS SPECIFIED HEREIN, NATIONAL INSTRUMENTS MAKES NO WARRANTIES, EXPRESS OR IMPLIED, AND SPECIFICALLY DISCLAIMS ANY WARRANTY OF MERCHANTABILITY OR FITNESS FOR A PARTICULAR PURPOSE. CUSTOMER'S RIGHT TO RECOVER DAMAGES CAUSED BY FAULT OR NEGLIGENCE ON THE PART OF NATIONAL INSTRUMENTS SHALL BE LIMITED TO THE AMOUNT THERETOFORE PAID BY THE CUSTOMER. NATIONAL INSTRUMENTS WILL NOT BE LIABLE FOR DAMAGES RESULTING FROM LOSS OF DATA, PROFITS, USE OF PRODUCTS, OR INCIDENTAL OR CONSEQUENTIAL DAMAGES, EVEN IF ADVISED OF THE POSSIBILITY THEREOF. This limitation of the liability of National Instruments will apply regardless of the form of action, whether in contract or tort, including negligence. Any action against National Instruments must be brought within one year after the cause of action accrues. National Instruments shall not be liable for any delay in performance due to causes beyond its reasonable control. The warranty provided herein does not cover damages, defects, malfunctions, or service failures caused by owner's failure to follow the National Instruments installation, operation, or maintenance instructions; owner's modification of the product; owner's abuse, misuse, or negligent acts; and power failure or surges, fire, flood, accident, actions of third parties, or other events outside reasonable control.

Copyright

Trademarks

Patents

WARNING REGARDING USE OF NATIONAL INSTRUMENTS PRODUCTS

Conventions

The following conventions are used in this manual:

» The » symbol leads you through nested menu items and dialog box options to a final action. The sequence **File»Page Setup»Options** directs you to pull down the **File** menu, select the **Page Setup** item, and select **Options** from the last dialog box.

This icon denotes a tip, which alerts you to advisory information.

This icon denotes a note, which alerts you to important information.

bold Bold text denotes items that you must select or click in the software, such as menu items and dialog box options. Bold text also denotes parameter names.

italic Italic text denotes variables, emphasis, a cross-reference, or an introduction to a key concept. Italic text also denotes text that is a placeholder for a word or value that you must supply.

`monospace` Text in this font denotes text or characters that you should enter from the keyboard, sections of code, programming examples, and syntax examples. This font is also used for the proper names of disk drives, paths, directories, programs, subprograms, subroutines, device names, functions, operations, variables, filenames, and extensions.

Contents

Chapter 1
Educators' Guide

Chapter 2
Breadboarding

Chapter 3
Virtual NI ELVIS

Chapter 4
Ladder Diagrams

Appendix A
Technical Support and Professional Services

Index

1

Educators' Guide

Multisim is the schematic capture and simulation application of National Instruments Circuit Design Suite, a suite of EDA (Electronics Design Automation) tools that assists you in carrying out the major steps in the circuit design flow. Multisim is designed for schematic entry, simulation, and feeding to downstage steps, such as PCB layout.

In addition to the professional features that are detailed in the *NI Multisim User Manual*, there are a number of education-specific features that are outlined in this manual.

This chapter describes the tools that Multisim offers to let you exercise greater control over the program's interface and functionality when sharing circuits with students, as well as to set certain aspects of a circuit's behavior for instructional purposes.

Some of the described features may not be available in the Student edition of Multisim. Refer to the release notes for a description of the features available in your edition.

Circuit Creator's Name

Multisim provides a feature by which the name of the creator of each circuit is stored with that circuit. Educators can take advantage of this feature to identify the student who, for example, created the circuit being submitted as the answer to an assignment (provided that the student uses his/her own copy of the program to create the circuit). The name appears in the **Circuit Restrictions** dialog box, which you can view as long as no passwords have been set. Refer to the *Setting Circuit Restrictions* section for more information.

Using Restrictions

Restrictions are useful in a number of ways:

- when you are designing circuits for demonstration purposes and want to limit the functionality available to students
- when you are sharing circuits with students and want:
  - to prevent them from being able to edit the circuit in any way
  - to limit the types of modifications they can make to a circuit
  - to limit the types of analyses they can perform on it
  - to limit the information they can see about certain parts of the circuit (for example, the value of a resistor you want them to calculate).

You can set global-level restrictions, which become default Multisim settings, or circuit-level restrictions, which affect only specific circuits. To ensure that only you can set or modify restrictions, you use passwords which can protect both global and circuit restrictions. It is important that you set passwords immediately when using restrictions that you want to keep secure against any modification by students. The password for global restrictions is encrypted and stored in the Multisim program file. The password for circuit restrictions (for restricting only a particular circuit) is encrypted and stored in the circuit file.

Setting Global Restrictions

Use global restrictions to set the basic level of functionality of Multisim available to students in all circuits with which they will work. You can select a default path where circuits are to be saved, hide databases and the **In Use List**, and determine whether students may edit components or place instruments.

You can also hide complicated instruments and analysis options from the menus by using the simplified version. Refer to the *Simplified Version* section for more information.

Note Global restrictions are overridden by circuit restrictions if the circuit restrictions are saved with the circuit. Refer to the *Setting Circuit Restrictions* section for more information.

General Global Restrictions

Complete the following steps to set general global restrictions:

1. Choose **Options»Global Restrictions**. The **Password** dialog box appears.

Note The **Password** dialog also appears if you select **Options»Circuit Restrictions**, if you have previously set a password by clicking **Password** from the **Circuit restrictions** dialog box. Refer to the *Setting Circuit Restrictions* section for information about the **Circuit restrictions** dialog box.

2. Enter the default password "Rodney" (this is case sensitive) and click **OK**. The **Global restrictions** dialog box appears.

Note You can, and should, change the default password. Refer to the *Setting Passwords for Restrictions* section for more information.

3. Click the **General** tab.
4. Set the default path and location where students find and save files in the **Circuit Path** field.
5. Set the following as desired in the **Toolbars** box:
   - **Disable Instruments toolbar**—Makes instruments unavailable to be placed in the circuit.
   - **Disable In-Use List toolbar**—Hides the **In Use List**.
6. Set the following as desired in the **Databases** box:
   - **Disable database editing**—Ensures that students cannot edit components in the database.
   - **Disable Master DB component access**—Hides the Multisim Master database and parts groups and families from the interface.
   - **Disable Corporate DB component access**—Hides the corporate database and parts groups and families from the interface.
   - **Disable User DB component access**—Hides the "user" database and parts groups and families from the interface.
7. Click **OK**. Your options are immediately set for all circuits, unless you have set circuit restrictions. Refer to the *Setting Circuit Restrictions* section for more information.

Simplified Version

The simplified version restricts students to only certain instruments and analyses. The simplified version can also be locked, preventing students from turning it off with **Options»Simplified Version** and having access to all analyses and instruments.

Complete the following steps to set up the simplified version:

1. Display the **General** tab of the **Global Restrictions** dialog box.
2. Set your options by enabling one of the following options:
   - **Lock simplified**—Disables the **Options»Simplified Version** menu option.
   - **Simplified version**—Changes the interface display by hiding the more complex functions and restricting the available instruments and analyses. If the simplified version is restricted, it will be greyed out in the Options menu.
   - **Full version**—Displays the full default interface without restrictions.
3. Click **OK**.

Your options are immediately set for all circuits, unless you have set circuit restrictions. Refer to the *Setting Circuit Restrictions* section for more information.

Global Analyses Restrictions

Complete the following steps to set global analyses restrictions:

1. From the **Global Restrictions** dialog box, click the **Analysis** tab.
2. Enable the desired analyses by selecting their checkboxes and click **OK**. Only the analyses you check will be enabled in the **Simulate»Analyses** menu or when the student clicks the **Grapher/Analyses List** button in the **Main** toolbar.

Note Refer to the **Analysis** chapter in the *Multisim User Manual* for more information on analyses.

These options are immediately set for all circuits, unless you have set circuit restrictions. Refer to the *Setting Circuit Restrictions* section for more information.

Setting Circuit Restrictions

Use circuit restrictions to set restrictions on individual circuits. Circuit restrictions override global restrictions. They are saved with your circuit and invoked each time the circuit is loaded. In addition to hiding databases and setting available analyses, you can set a schematic to be read-only (not editable by students), you can hide components' values, faults and uses in analyses, and you can lock subcircuits to make them unavailable for opening by students.

Note Remember that circuit restrictions only apply to the current circuit; when you create a new circuit, only the global restrictions will apply. Refer to the *Setting Global Restrictions* section for more information. If you want circuit restrictions to apply to a new circuit, you will need to reset those restrictions each time you create a new circuit.

Complete the following steps to set general circuit restrictions:

1. Choose **Options»Circuit Restrictions**. If you have created a password, you will be prompted for it. Refer to the *Setting Passwords for Restrictions* section for more information. Enter your password in the **Password** dialog box, and click **OK**. The **Circuit Restrictions** dialog box appears.
2. Click the **General** tab and set the desired options by enabling the appropriate checkboxes. Select from the following options:
    - **Schematic read-only**—Prevents students from saving the circuit, and hides parts bins. Students will only be able to draw wires between instruments and an open pin on an existing connector. Also, they can only remove wires that are between an instrument and a connector.
    - **Circuit description read-only**—Prevents students from changing the contents of the Circuit Description box.
    - **Hide component values**—Marks the **Values** tab of components' properties dialog boxes with an "X" and hides values. You may wish to provide false values using labels.
    - **Hide component faults**—Marks the **Faults** tab of components' properties dialog boxes with an "X", and hides faults.
    - **Lock subcircuits**—Prevents students from opening subcircuits and seeing their contents. Students must measure the input and output of a hidden subcircuit to determine its contents.
    - **Disable Instruments toolbar**—Makes instruments unavailable to be placed on the circuit.

- **Disable In-Use List toolbar**—Disables the In-Use List for the current circuit.
- **Disable Multisim Master DB component access**—Hides the Multisim Master database and parts groups and families from the current circuit.
- **Disable User DB component access**—Hides the user database and parts groups and families from the current circuit.
- **Disable Corporate DB component access**—Hides the corporate database and parts groups and families from the interface.

Note The **Circuit Creator Name** is taken from the operating system.

3. Click **OK**. The options you select are immediately invoked in the circuit.
4. To have the restrictions apply each time the circuit is opened, choose **File»Save** to save the restrictions in the circuit file.

Complete the following steps to set circuit analyses restrictions:

1. From the **Circuit Restrictions** dialog box, click the **Analysis** tab.
2. Enable the desired analyses by selecting their checkboxes and click **OK**. Only the analyses you check will be enabled in the **Simulate»Analyses** menu or when the student clicks the **Grapher/Analyses List** button in the **Main** toolbar.

Note Refer to the **Analysis** chapter in the *Multisim User Manual* for more information on analyses.

3. To have these analyses apply each time the circuit is opened, choose **File»Save** to save the restrictions.

Complete the following steps to set circuit breadboard restrictions:

1. From the **Circuit Restrictions** dialog box, click the **Breadboard** tab.
2. Set the following as desired:
   - **Highlight Target Holes**—Disable if you do not wish to see where the targets for jumper wires are when placing them on the breadboard.
   - **Completion Feedback**—Disable if you do not wish components and wires on the schematic to change color as they are placed and wired on the breadboard.
3. Click **OK**.

Note For details on breadboarding, refer to Chapter 2, *Breadboarding*.

Setting Passwords for Restrictions

When using restrictions, you should create a password immediately to ensure that your settings are secure.

Complete the following steps to create/change a password:

1. For global restrictions, choose **Options»Global Restrictions**.

   Or

   For circuit restrictions, choose **Options»Circuit Restrictions**.
2. Enter a password if prompted to do so.

Note The default password for global restrictions is "Rodney" (this is case sensitive). Circuit restrictions *do not* have a default password.

3. From the restrictions dialog box that appears, click **Password**. The **Change password** dialog box appears.
4. If you are choosing a password for the first time, leave the **Old password** field blank.
5. If you are changing a password, enter the old password in the **Old password** field.
6. Enter your (new) password in the **New password** field.
7. Confirm your new password by entering it again in the **Confirm password** field.
8. Click **OK** to return to the dialog box, or **Cancel** to begin again.

Note If you want to change global or circuit restrictions, you will need to enter the respective password. Be sure to keep your passwords for both the **Global restrictions** and **Circuit restrictions** dialogs written down and in a safe place, as you will not be able to retrieve them from the program or circuit files, where they are stored in encrypted form.

Note A circuit password is not automatically transferred to a new circuit when you go to set circuit restrictions for it, so you will need to recreate the password every time you create circuit restrictions that you want to keep secure.

Link to Education Resources

Note This function is hidden when the simplified version option is selected. Refer to the *Simplified Version* section for more information.

To go to the National Instruments Academic website, click the **Educational Website** button or select **Tools»Education Webpage**.

2

Breadboarding

This chapter describes Multisim's breadboarding feature.

Some of the features described in this chapter may not be available in your edition of Multisim. Refer to the release notes for a description of the features available in your edition.

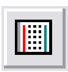
Breadboarding Overview

The **Breadboarding** feature provides a technical aid for educators who wish to illustrate breadboarding as a means of prototyping circuit designs. It also gives students exposure to the breadboarding process, and shows in 3D what the resulting breadboard will look like when completed.

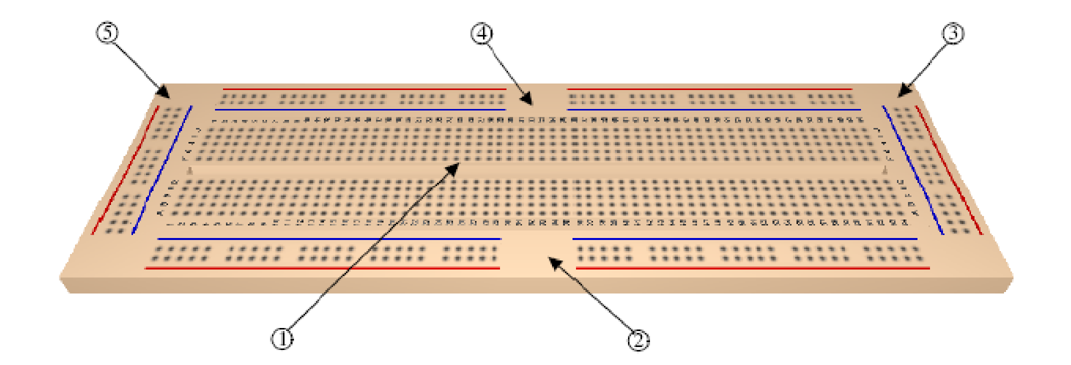
Breadboard Settings

The default breadboard is shown in the screen capture below. If you wish to change the default settings, use the following procedure.

Complete the following steps to change the breadboard's settings:

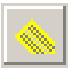


1. Select **Tools»Show Breadboard** from the main Multisim menu. The **Breadboard View** displays. The default breadboard appears as shown below.



1 One Slat with Rows	3 Bottom Strip	5 Top Strip
2 Left Strip	4 Right Strip	

As shown in the figure above, the default breadboard contains: one slat with two rows (1); one left strip (2); one bottom strip (3); one right strip (4); one top strip (5).



2. Select **Options»Breadboard Settings** to display the **Breadboard Settings** dialog box.
3. Enter the desired parameters for the breadboard and click **OK**. The view of the breadboard changes to reflect your changes.

3D Options

The 3D viewing options for the **Breadboard View** are set in the **3D Options** tab of the **Preferences** dialog box.

Complete the following steps to change the 3D options:

1. Select **Options»Global Preferences** and click on the **3D Options** tab.
2. Optionally, click on **Background color** to display a standard **Color** dialog box where you can adjust the background color as desired.
3. In the **Info Box** area:
   - **Info Box**—Disable this checkbox if you do not wish to see the box at the top of the **Breadboard View** that shows parts information.
   - **Left**—Places parts information box at top-left.
   - **Center**—Places parts information box at top-center.
   - **Right**—Places parts information box at top-right.

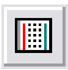
4. Disable the **Show Target Holes** checkbox if you do not wish to see where the targets for jumper wires are when placing them. Refer to the *Placing a Jumper* section for more information.
5. Disable the **Show Completion Feedback** checkbox if you do not wish components and wires on the schematic to change color as they are placed and wired on the breadboard.
6. In the **3D Performance** box:
   - Move the slider as desired to improve graphic performance. **More Details** will result in a slower screen refresh rate.
   - Enable the **User Defined** checkbox and disable the 3D features that you do not wish to see.

Tip Disabling **Show Breadboard Numbers** will result in a much quicker refresh rate.

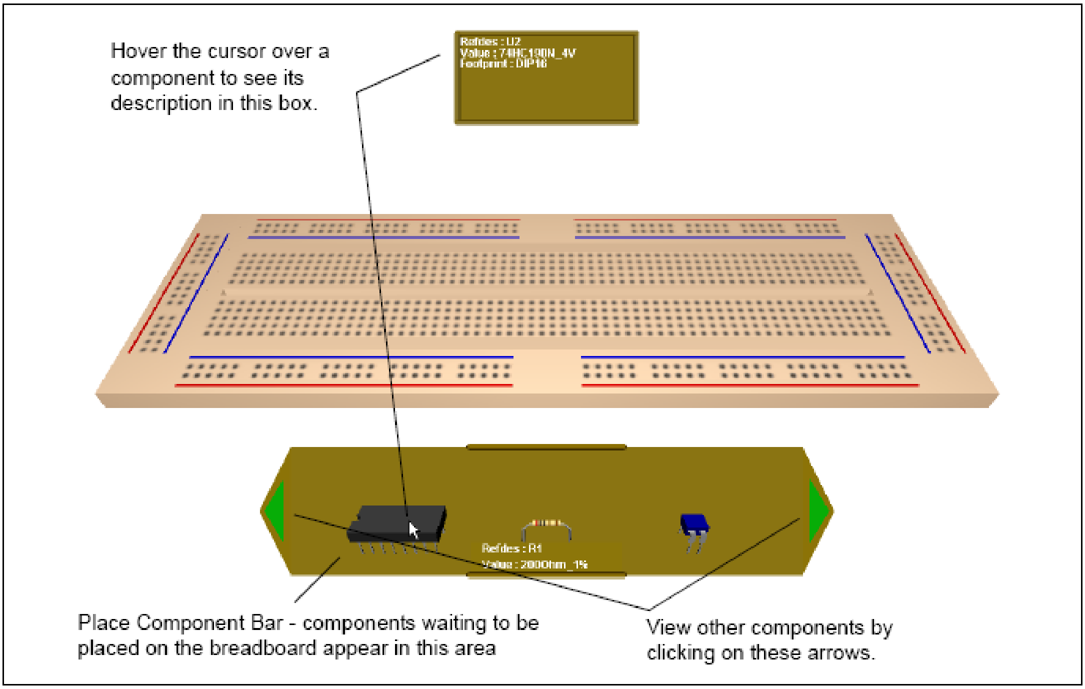
Placing Components on the Breadboard

Complete the following steps to place components on a breadboard:

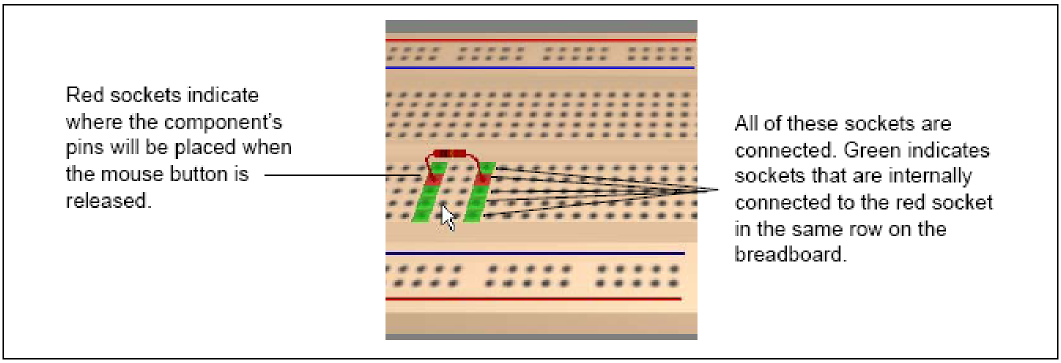
1. Create a schematic diagram of the desired circuit in the usual manner. (For details, on schematic capture, refer to the *Multisim User Manual*.)



2. Select **Tools»Show Breadboard** from the main Multisim menu. The **Breadboard View** displays similar to the following example.

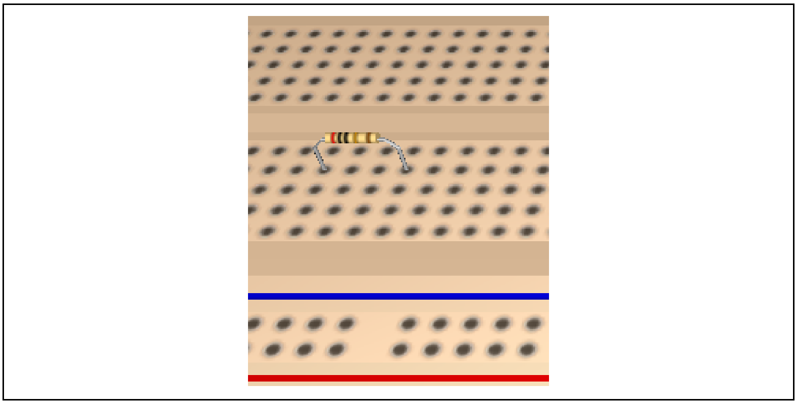


3. Click on a component in the **Place Component Bar** and drag it to the desired location on the breadboard. As the component passes over the breadboard, sockets change color as shown below.

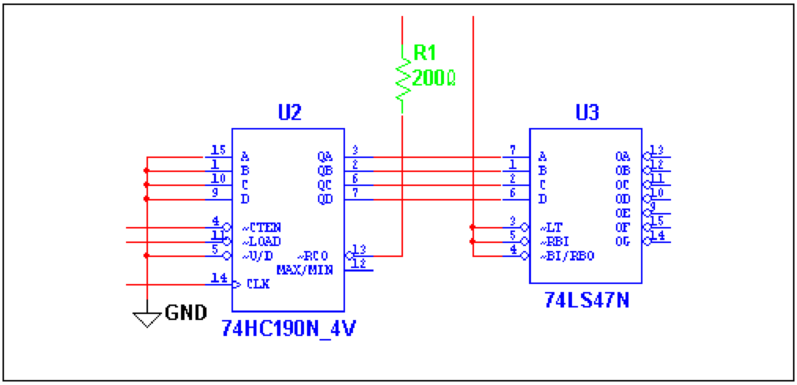


Tip Select <Ctrl-R> to rotate a selected component 90 degrees clockwise or <Ctrl-Shift-R> to rotate it 90 degrees counter-clockwise.

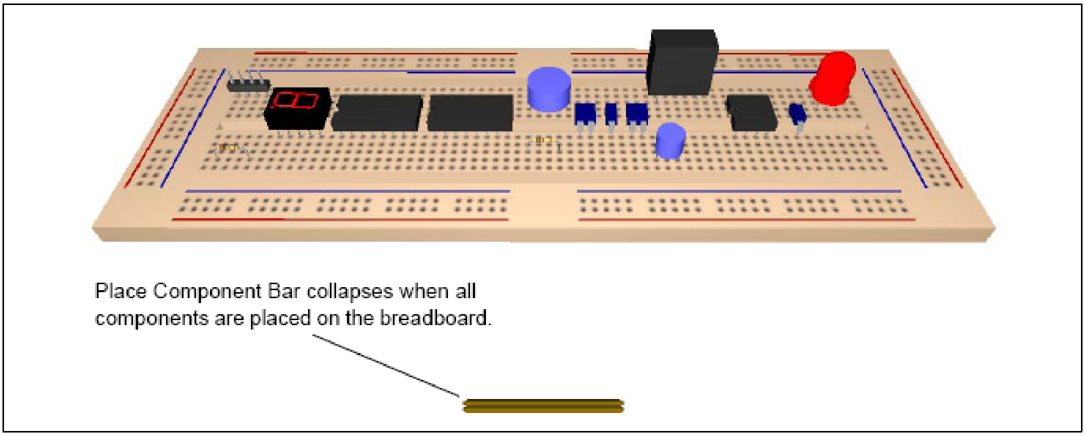
4. Release the mouse button to place the component. Notice that the colored (red and green) sockets on the breadboard no longer appear.



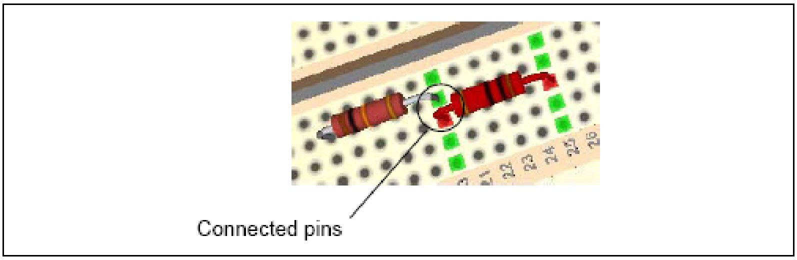
5. Return to the schematic view and note that the color of the placed component has changed as shown in the example below (R1).



6. Continue placing the circuit's components on the breadboard. When all the components have been placed, the **Place Component Bar** collapses as shown below.



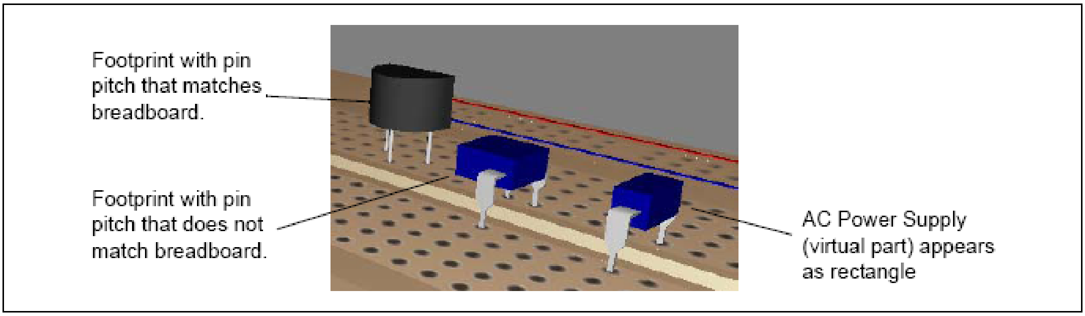
Tip Where pins of components are connected on the schematic, you can place them in connected sockets on the breadboard as shown below. This technique can reduce the number of jumper wires required. Refer to the *Placing a Jumper* section for more information about jumpers.



Appearance of 3D Components

The appearance of the 3D component is dependant on the footprint that is selected from the **Select a Component** browser during schematic capture in the **Footprint manuf./Type** list.

Some virtual parts have a default 3D view that appears as a blue 3D rectangle or cube. “Real” parts that have pin pitch (spacing) that does not fit the pin pitch on the breadboard will also appear as 3D rectangles or cubes, with properly spaced pins. (See below.)



Note Certain virtual components, including 3D parts also appear as 3D rectangles or cubes.

To view footprint information, hover the cursor over the desired component. Refer to the *Viewing Component Information* section for more information.

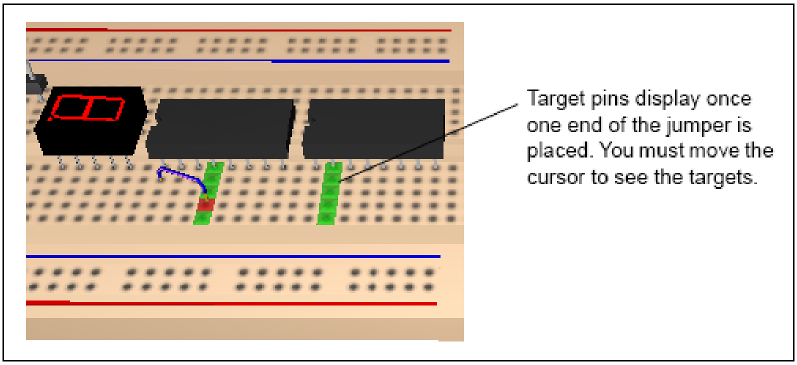
Wiring Placed Components

By placing component pins that are connected on the schematic into sockets that are internally connected, much of the “wiring” can be done at the same time components are placed. However, in most circuits, it will also be necessary to place jumpers to complete the wiring of the placed components.

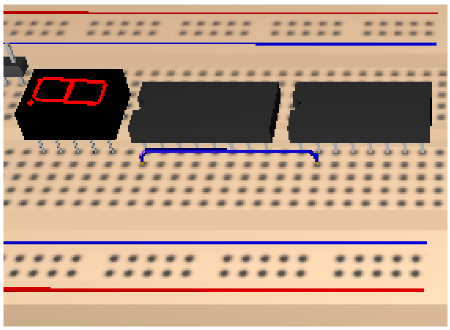
Placing a Jumper

Complete the following steps to place a jumper wire:

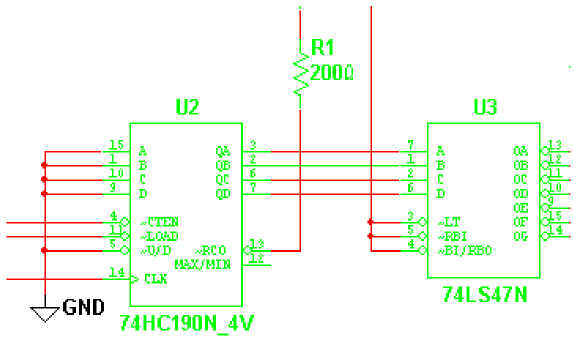
1. Click on a socket connected to the pin where you wish to start the jumper and begin moving the cursor. Legitimate “target” pins display as shown below.



2. Click to place the jumper in the desired socket.



3. Return to the schematic view and note that the color of the wire connecting the two pins has changed to green to indicate a connection has been made, as shown in the figure below, between pin 2 of U2, and pin 1 of U3.



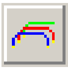
Note If a net contains more than two connections, all must be connected before any of the wires in the net change color.

4. Continue placing jumpers until all schematic connections have been made.

Tip Run a Design Rules and Connectivity Check to see if there are any errors in your breadboard. Refer to the *DRC and Connectivity Check* section for more information.

Changing Jumper Wire Color

Complete the following steps to change jumper wire color:



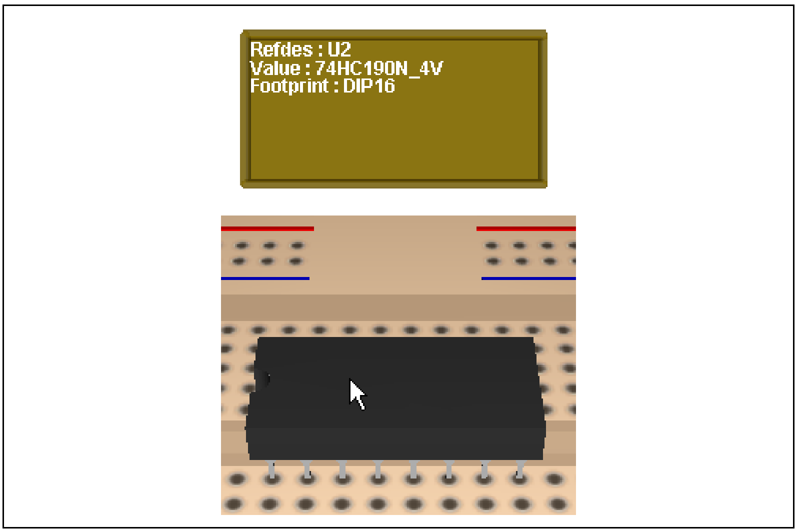
1. Select **Edit»Breadboard Wire Color**.
2. Select the desired color from the dialog box that appears.

Note The color of previously placed wires is not affected. The new color will be applied to any subsequently placed wires.

Viewing Component Information

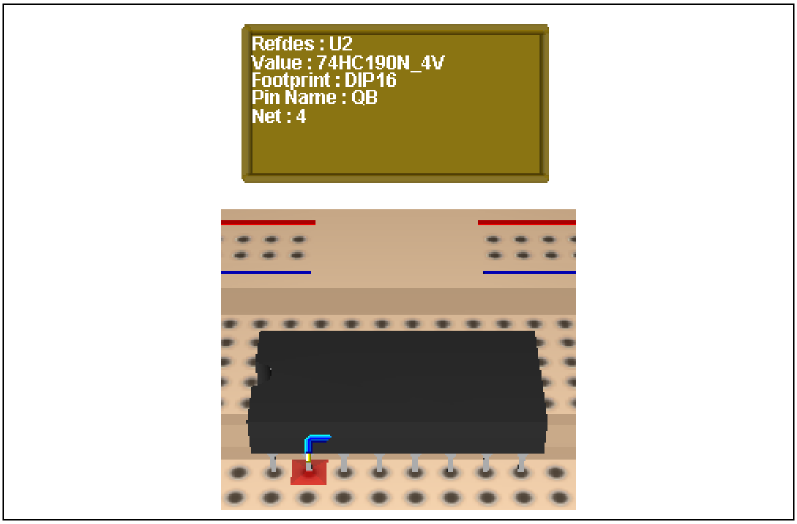
Complete the following steps to view information about a specific component:

1. Hover the cursor over the component. The information box is populated as shown below.



Complete the following steps to see pin information:

1. Hover the cursor over the "metal" part of the desired pin. The information box now includes the pin name and the schematic net to which the pin should be connected.

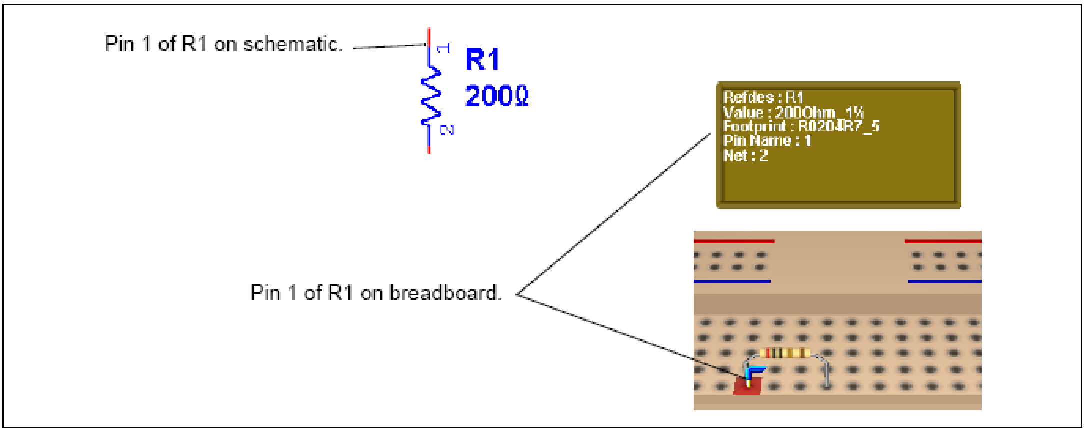


Two-terminal Components

Two-terminal non-directional components like resistors have symbol pin names (1 and 2) that will automatically swap if they are connected the "wrong way" according to the pin name that is on the schematic.

Complete the following steps to view the pin names for all devices on the schematic:

1. Select **Options»Sheet Properties** and click the **Circuit** tab of the **Sheet Properties** dialog box.
2. Click the **Symbol Pin Names** checkbox until the checkmark is a solid black color and click **OK** to close the dialog.



In the above example, if Pin 1 is connected to a pin that should be receiving Pin 2, the pin names will automatically swap. (Pin 1 will become Pin 2 and vice versa.)

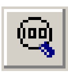
Manipulating the Breadboard View

You can manipulate the view of the breadboard in a number of ways.

To make the breadboard appear larger, select **View»Zoom in**.

To make the breadboard appear smaller, select **View»Zoom out**.

Tip Use your mouse's center wheel to zoom in or out. (This must be set up in the **General** tab of the **Preferences** dialog box. For details, refer to the *Multisim User Manual*.)



To view the entire breadboard, select **View»Zoom Full**.

To rotate the breadboard 180 degrees, select **View»Rotate 180 Degrees**.

Or

Press <Shift-R> on your keyboard.

Tip Rotate the breadboard in any direction by dragging the mouse from a blank area of the **Breadboard View**. You can also rotate the breadboard by using the arrow keys on your keyboard.

To pan the breadboard, hold down the Shift key on your keyboard and use any of the arrow keys.

Or

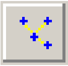
Press <Ctrl-Shift> and drag the mouse.

Or

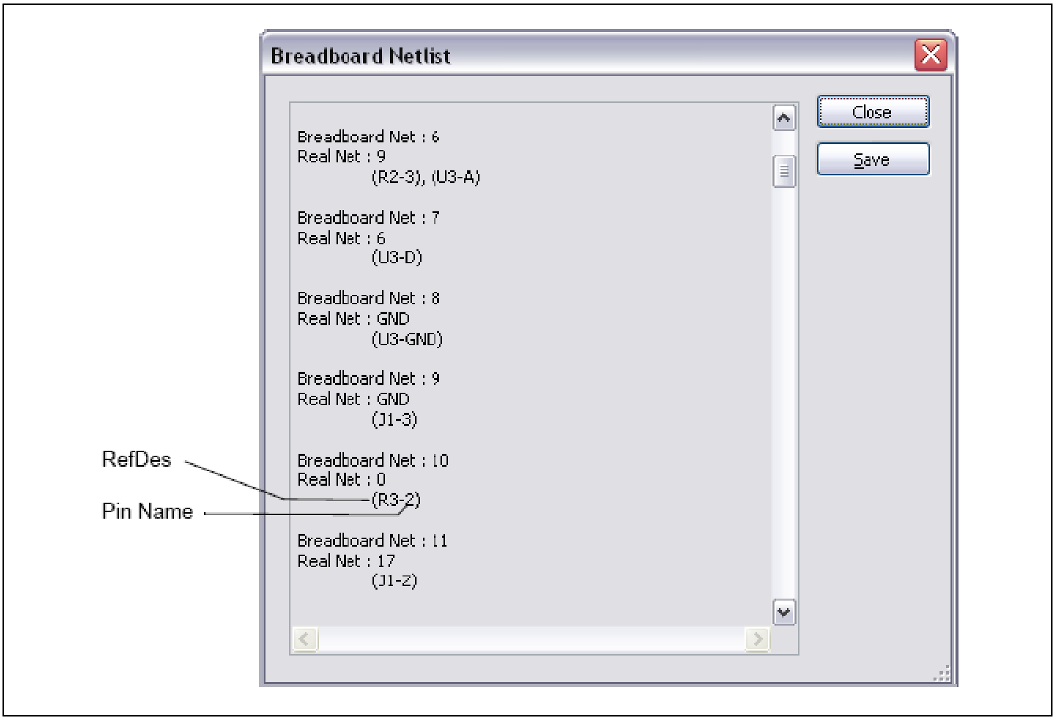
Hold down your mouse-wheel and drag the mouse.

Breadboard Netlist dialog box

Complete the following steps to display a netlist for the placed components and jumpers:



1. Select **Tools»Show Breadboard Netlist**. The **Breadboard Netlist** dialog box appears.



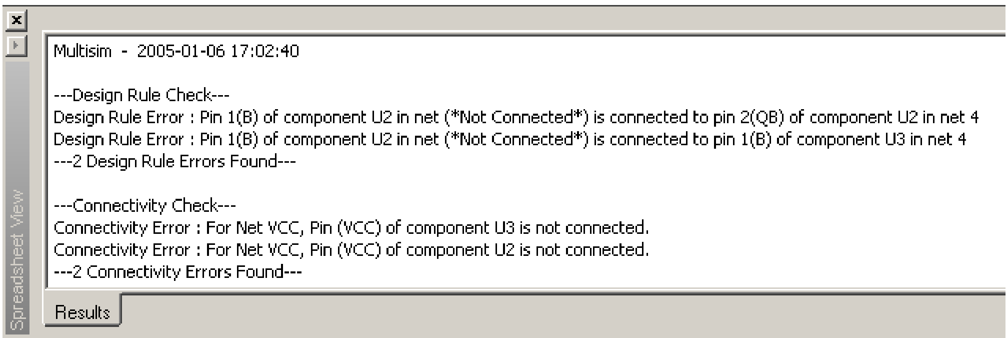
2. Optionally, click **Save** to save the breadboard netlist as a `.txt` or `.csv` file.

Note These nets are breadboard connections, and are not necessarily numbered in correspondence to the schematic nets.

DRC and Connectivity Check

You can run a **Design Rules and Connectivity Check** to see if there are any errors on your breadboard.

To run a DRC and Connectivity Check, select **Tools»DRC and Connectivity Check**. The results appear in the **Results** tab of the **Spreadsheet View** as in the example shown below.



Design Rule Errors—Indicate connections that are on the breadboard that are not on the schematic.

Connectivity Errors—Indicate component pins that are not connected to fully-completed schematic nets.

3

Virtual NI ELVIS

This chapter describes Multisim's virtual NI ELVIS feature.

Some of the features described in this chapter may not be available in your edition of Multisim. Refer to the release notes for a description of the features available in your edition.

Overview

Virtual NI ELVIS emulates much of the behavior of its real-world counter-part, the NI Educational Laboratory Virtual Instrumentation Suite (NI ELVIS). Planning, prototyping and testing of instructors' projects can be carried out on students' PCs before moving on to the real NI ELVIS I or NI ELVIS II workstation in the lab.

Multisim emulates both the original NI ELVIS I and NI ELVIS II. For details, refer to *The Virtual NI ELVIS I Schematic* and *The Virtual NI ELVIS II Schematic* sections.

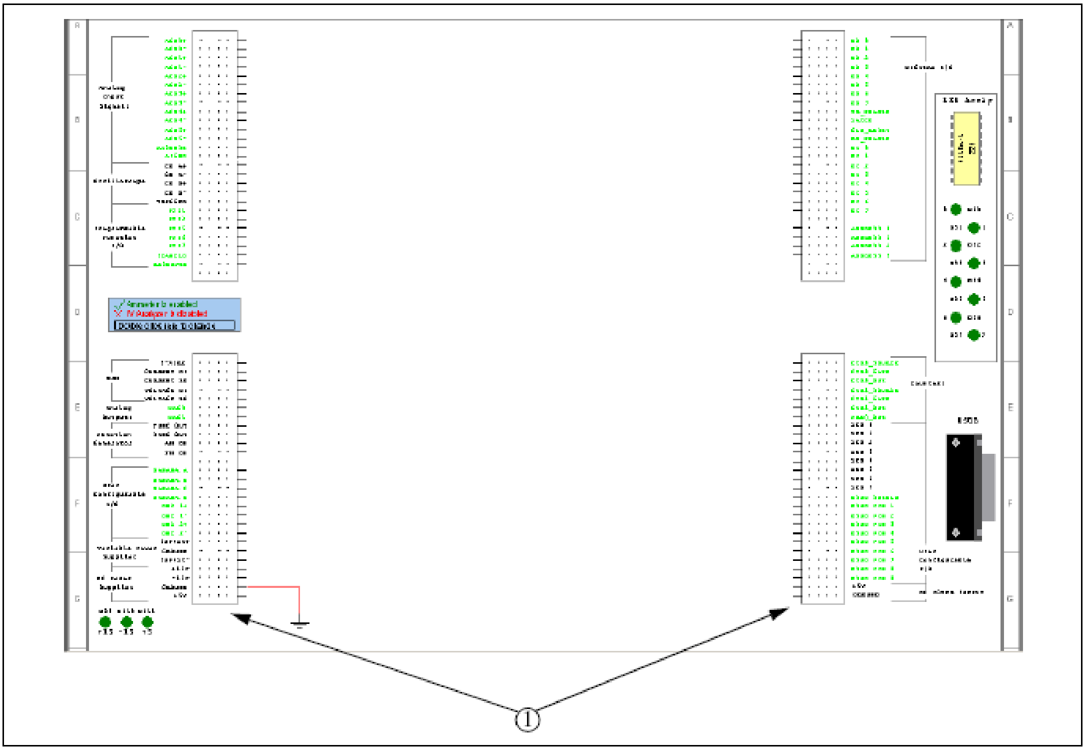
The Virtual NI ELVIS I Schematic

A virtual NI ELVIS I schematic contains a number of items that correspond to elements of the real-world NI ELVIS workstation. The connection and control of these elements is described in this section.

Note This section describes the behavior of Multisim's original NI ELVIS I schematic. Refer to *The Virtual NI ELVIS II Schematic* section for information on Multisim's NI ELVIS II functionality.

Complete the following steps to create a new virtual NI ELVIS I schematic:

1. Select **File»New»NI ELVIS I Schematic**. The schematic appears as shown below:



1 Rails

Note The ground connector that appears at the bottom left of the diagram is the reference point for measurements taken during simulation, and must not be removed.

2. Place and wire components in the virtual NI ELVIS I schematic in the same manner as other Multisim schematics. For details, refer to the *Multisim User Manual*.

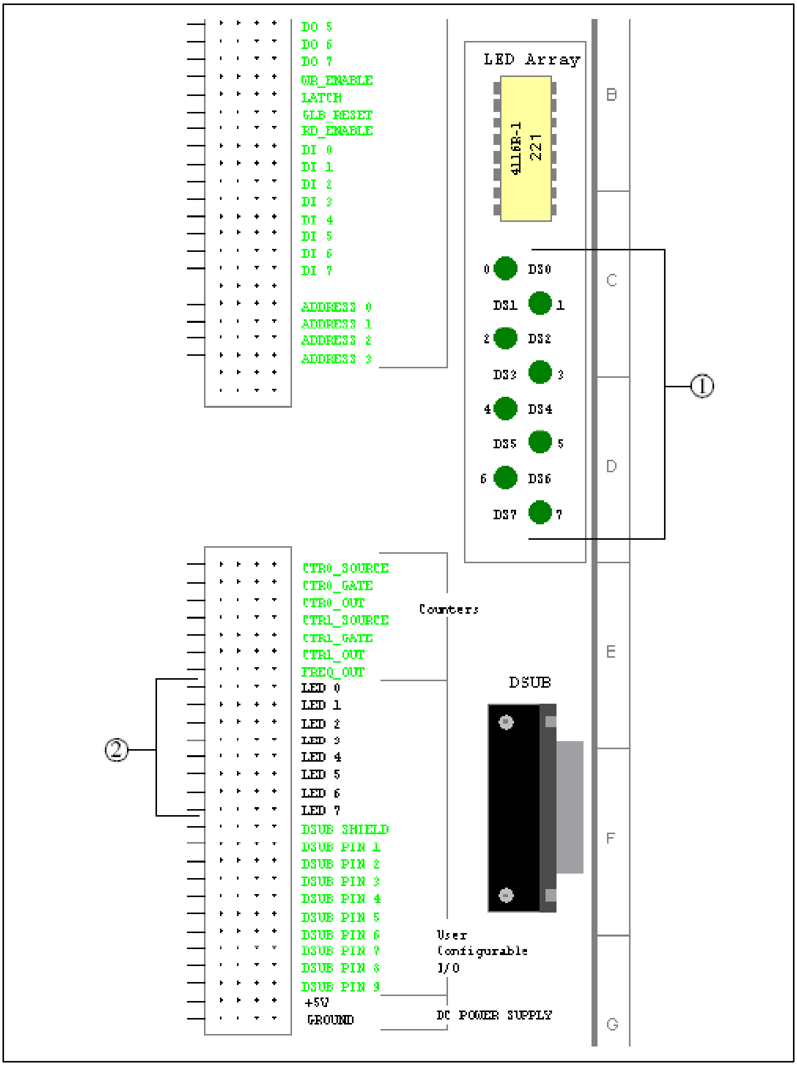
The prototyping board rails (see 1 in the figure above) found to the left and right of the main workspace correspond to rails on the prototyping board of the real-world version of NI ELVIS, and are labelled in the same manner.

Rows on the rails that are shown with green labels are not enabled for simulation in Multisim. However, they can be used for schematic capture and viewing of the completed virtual NI ELVIS I schematic in the 3D view.

Unlike other Multisim components, these rails cannot be moved to other places on the workspace.

LEDs

Connections to the eight LEDs on the right side of the NI ELVIS I schematic are found in the lower-right prototyping rail, as shown in the figure below (2). During simulation, any of these LEDs (1) that are correctly driven will light.

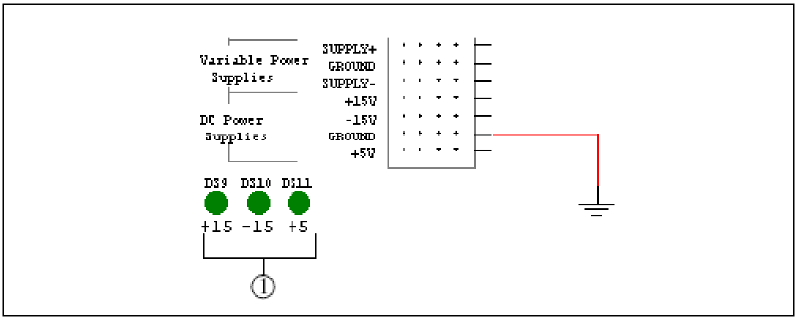


1 LEDs	2 LED Connections

Complete the following to connect to an LED:

1. Place a wire from one of the LED rows (**LED 0** through **LED 7**) to the desired point in your schematic.

There are also three power supply LEDs in the lower-left section of any virtual NI ELVIS schematic, as shown in the figure below (1).



1 Power Supply LEDs

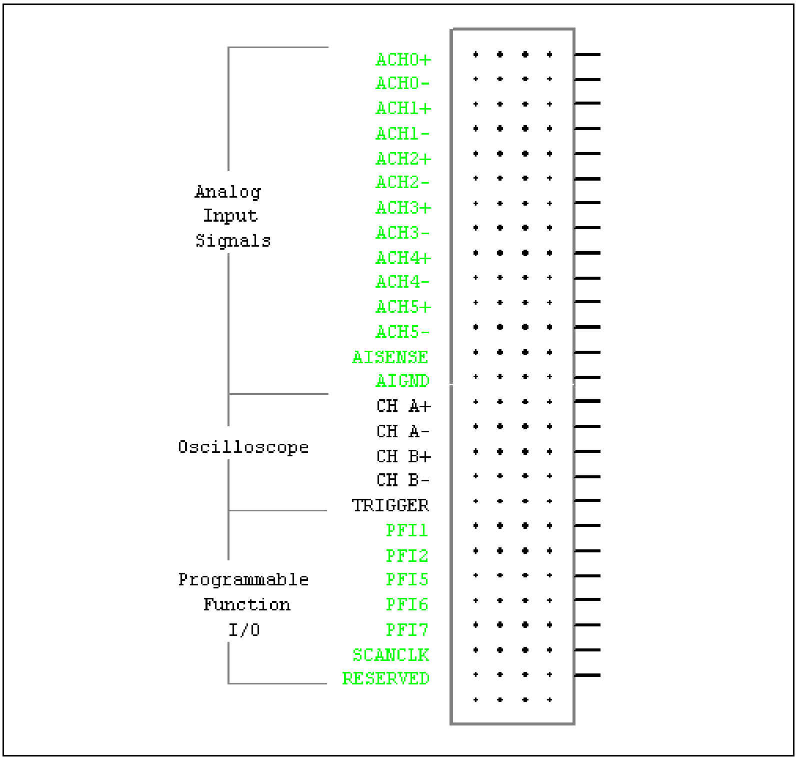
During simulation, these LEDs will light whether or not connections have been made to their corresponding pins in the prototyping rail. They indicate that power is available to the respective connections.

NI ELVIS I Instruments

One instance of each of the following NI ELVIS instruments is found in the virtual NI ELVIS I schematic:

- *Oscilloscope*—This instrument is a two-channel oscilloscope.
- *IV Analyzer and Multimeter*—This instrument can be enabled as either an IV analyzer, or a digital multimeter.
- *Function Generator*—This instrument generates sine, square or triangle waves.
- *Variable Power Supply*—This device is a variable power supply.

Oscilloscope



The connections to the virtual NI ELVIS oscilloscope are found in the upper-left prototyping rail.

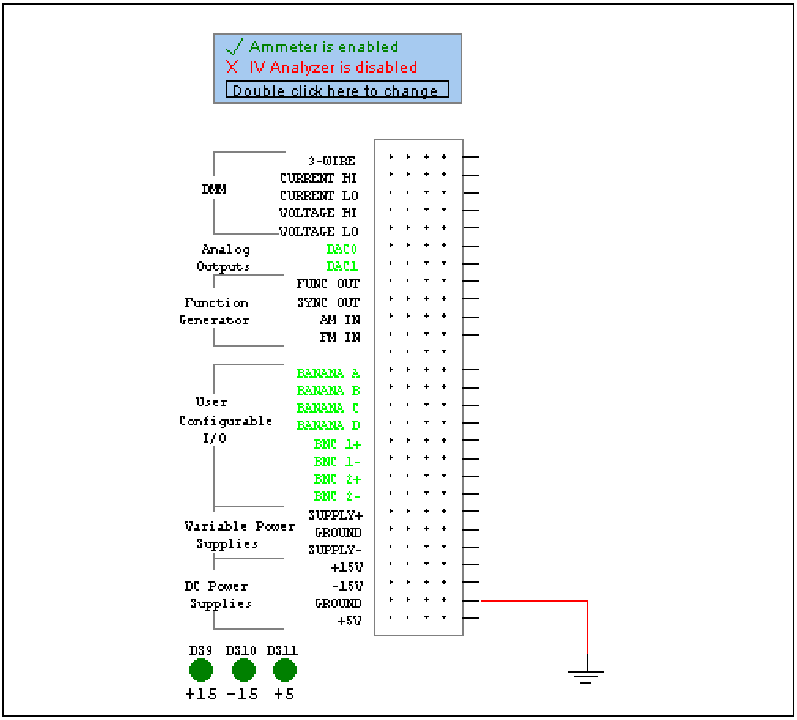
To connect the oscilloscope, place wires from the points in your schematic that you wish to measure with the oscilloscope to any of the pins on the **CH A+**, **CH A–**, **CH B+**, **CH B–** or **TRIGGER** rows beside "Oscilloscope". These rows correspond to the terminals of the oscilloscope.

- **CH A+** —Positive input of channel A.
- **CH A–** —Negative input of channel A.
- **CH B+** —Positive input of channel B.
- **CH B–** —Negative input of channel B.
- **TRIGGER**—Trigger input signal.

Complete the following steps to access the oscilloscope's controls:

1. Double-click on the the word "Oscilloscope" in the upper-left prototyping rail. The instrument face for the Multisim virtual oscilloscope displays.
2. Refer to the *Multisim User Manual* for details on the use of this instrument.

IV Analyzer and Multimeter



When a new virtual NI ELVIS I schematic is opened, the **IV Analyzer** is disabled, and the **Ammeter** is enabled as indicated above.

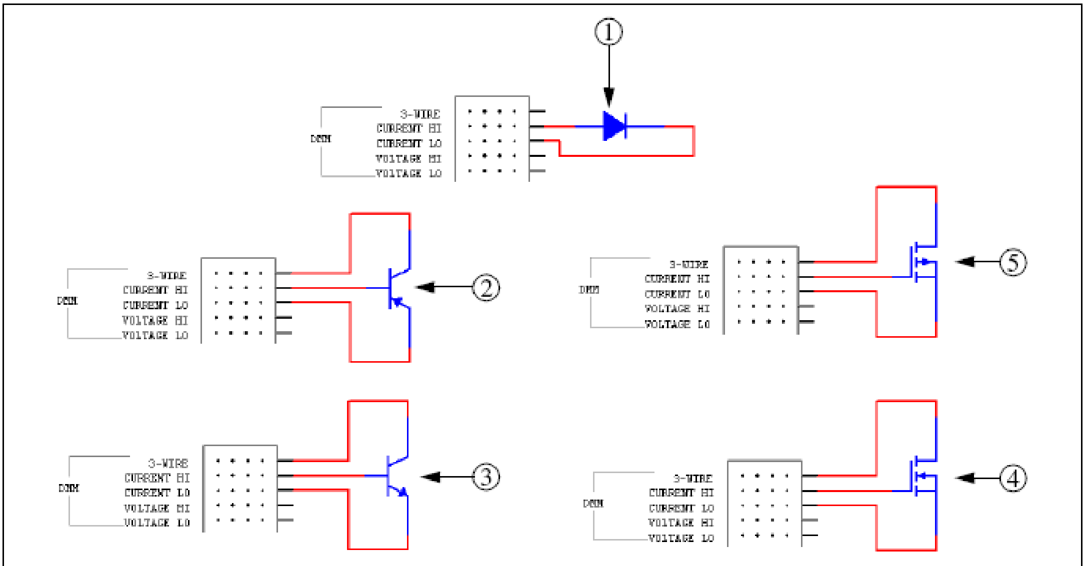
To disable the **Ammeter** and enable the **IV Analyzer**, double-click where indicated on the virtual NI ELVIS I schematic.

To disable the **IV Analyzer** and enable the **Ammeter**, double-click again.

Note When the **IV Analyzer** is enabled, there is a slight delay when simulation is started while a **DC Sweep** is performed. If the **Ammeter** is enabled, there is no delay.

Complete the following to connect the **IV Analyzer**:

1. Place wires from the component you wish to measure to the pins on the **3-WIRE**, **CURRENT HI** and **CURRENT LO** rows. These rows correspond to the inputs of the **IV Analyzer**. Refer to the figure below for connections for a diode (1); PNP BJT (2); NPN BJT (3); NMOS FET (4); PMOS FET (5).



1	Diode	3	NPN BJT	5	PMOS
2	PNP BJT	4	NMOS		

Complete the following to connect the **Ammeter**:

1. Place wires from the points in the circuit you wish to measure to the pins on the **CURRENT HI** and **CURRENT LO** rows.

   CURRENT HI corresponds to the **+** terminal of the ammeter and **CURRENT LO** corresponds to the **–** terminal.

Complete the following steps to access the controls for the enabled instrument:

1. Double-click just *above* the letters “DMM”. If you have enabled the **IV Analyzer** as described earlier, that instrument’s face appears. If you have enabled the **Ammeter**, an instrument containing the **Ammeter** function of a multimeter appears.
2. Refer to the *Multisim User Manual* for details on the use of these instruments.

Complete the following to connect the Volt/Ohmmeter:

1. Place wires from the points in the circuit you wish to measure to the pins on the **VOLTAGE HI** and **VOLTAGE LO** rows.

   VOLTAGE HI corresponds to the **+** terminal of the meter and **VOLTAGE LO** corresponds to the **–** terminal.

Complete the following steps to access the controls for the **Voltmeter** and **Ohmmeter**:

1. Double-click just *below* the letters “DMM”. An instrument containing the **Voltmeter** and **Ohmmeter** functions of a multimeter appears.
2. Refer to the *Multisim User Manual* for details on the use of this instrument.

Function Generator

Complete the following to connect the **Function Generator**:

1. Place wires from the pins on the **FUNC OUT**, **SYNC OUT**, **AM IN** and **FM IN** rows to the desired points in your schematic.
   - **FUNC OUT**—Output signal.
   - **SYNC OUT**—Outputs a TTL-compatible clock signal of the same frequency as the output waveform.
   - **AM IN**—A signal input here controls the amplitude of the signal at FUNC OUT.
   - **FM IN**—A signal input here controls the frequency of the signal at **FUNC OUT** and **SYNC OUT**.

Complete the following steps to access the controls for the **Function Generator**:

1. Double-click on **Function Generator**. The properties dialog for the **NI ELVIS Function Generator** appears.
2. Click on the **Value** tab and enter the desired output parameters.
3. Click **OK** to close the **Function Generator**’s properties dialog box.

Note Refer to the *Component Reference* help file for information about the user-settable parameters.

Note This instrument can be used for transient analysis only. This will not function for frequency-domain analyses. To run a frequency-domain analysis, use an AC source from the Master Multisim database. For more information on analyses, refer to the *Multisim User Manual*.

Variable Power Supply

The lower-left prototyping rail contains the following fixed DC power supplies:

- **+15 V**
- **–15 V**
- **+5 V** (also found in the lower-right prototyping rail.)

Variable Power Supplies are also available:

- **SUPPLY +** (+12 V max.)
- **SUPPLY –** (–12 V max.)

To connect to any of the power supplies, place wires from the pin on the corresponding row to the desired point in the circuit.

Complete the following steps to access the controls for the variable power supply:

1. Double-click on **Variable Power Supplies**. The properties dialog box for the NI ELVIS power supply appears.
2. Click on the **Value** tab and enter the desired parameters.
3. Click **OK** to close the properties dialog box.

Note Refer to the *Component Reference* help file for information about the user-settable parameters.

NI ELVIS II Instruments

The following NI ELVIS II instruments are available in Multisim:

- *Oscilloscope*—This instrument is a two-channel oscilloscope.
- *Dynamic Signal Analyzer*—This instrument computes and displays the RMS averaged power spectrum of a single channel.
- *Digital Multimeter*—This instrument is a digital multimeter.
- *Arbitrary Waveform Generator*—This instrument generates user-specified waveforms.
- *Function Generator*—This instrument generates sine, square or triangle waves.
- *Variable Power Supply*—This device is a variable DC power supply.
- *Digital Reader*—This instrument reads digital data.

- *Digital Writer*—This instrument updates the digital output on the NI ELVIS II Prototyping Board with user-specific digital patterns.

There are three ways to access NI ELVIS II instruments:

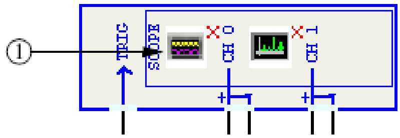
- Use the pre-placed icons found on the NI ELVIS II schematic rails and platform control panel sections.
- Place them from the menu onto any Multisim workspace using **Simulate»Instruments»NI ELVISmx Instruments»<instrument>**.
- Place them from the **NI ELVISmx Instruments** toolbar onto any Multisim workspace. Refer to the *NI ELVISmx Instruments Toolbar* section for more information.

An NI ELVIS II instrument soft front panel (SFP) is where you find that instrument's controls. To access any NI ELVIS II instrument's SFP, double-click on its icon.

Oscilloscope

This instrument is a two-channel oscilloscope.

In NI ELVIS II schematics, this instrument's pre-placed icon is located in the NI ELVIS II Platform Control Panel Scope Section of the NI ELVIS II schematic, as shown in the figure below (1).



1 Oscilloscope Icon

Complete the following step to connect the instrument:

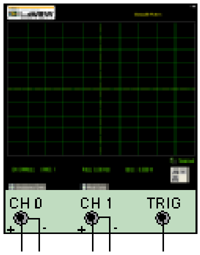
1. Place wires from the desired points in your schematic to the pins on the NI ELVIS II Front Control Panel Scope Section described below:
   - **TRIG**—Trigger input.
   - **CH 0 +** —Positive input of channel 0.
   - **CH 0 –** —Negative input of channel 0.
   - **CH 1+** —Positive input of channel 1.
   - **CH 1–** —Negative input of channel 1.

Complete the following steps to access the SFP:

1. Double-click the instrument's icon, shown in the figure above (1). The SFP appears.
2. Change the settings in the SFP as desired.

Complete the following steps to place this instrument directly onto *any* Multisim workspace:

1. Select **Simulate»Instruments»NI ELVISmx Instruments» NI ELVISmx Oscilloscope**.
2. Click to place the instrument in the desired location on the workspace. Its icon appears as shown below:



3. Wire in the same manner as any other Multisim instrument.

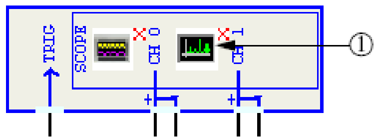
Note You can also use the **NI ELVISmx Instruments** toolbar to place this instrument. Refer to the *NI ELVISmx Instruments Toolbar* section for more information.

For information about using this instrument, click the **Help** button on its SFP to display the *NI ELVIS* help file.

Dynamic Signal Analyzer

The **NI ELVIS Dynamic Signal Analyzer** computes and displays the RMS averaged power spectrum of a single channel. A variety of windowing and averaging modes can be applied to the signal. It also detects the peak frequency component and estimates the actual frequency and power.

In NI ELVIS II schematics, this instrument's icon is located in the NI ELVIS II Platform Control Panel Scope Section of the NI ELVIS II schematic, as shown in the figure below (1).



1 Dynamic Signal Analyzer Icon

Complete the following step to connect the instrument:

1. Place wires from the desired points in your schematic to the pins on the NI ELVIS II Front Control Panel Scope Section described below:
   - **TRIG**—Trigger input.
   - **CH 0 +** —Positive input of channel 0.
   - **CH 0 –** —Negative input of channel 0.

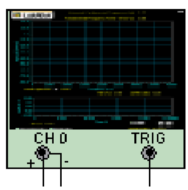
Note **CH 1 +** and **CH 1 –** are not used for this device.

Complete the following steps to access the SFP:

1. Double-click the instrument's icon, shown in the figure above (1). The SFP appears.
2. Change the settings in the SFP as desired.

Complete the following steps to place this instrument directly onto *any* Multisim workspace:

1. Select **Simulate»Instruments»NI ELVISmx Instruments» NI ELVISmx Dynamic Signal Analyzer**.
2. Click to place the instrument in the desired location on the workspace. Its icon appears as shown below:



3. Wire in the same manner as any other Multisim instrument.

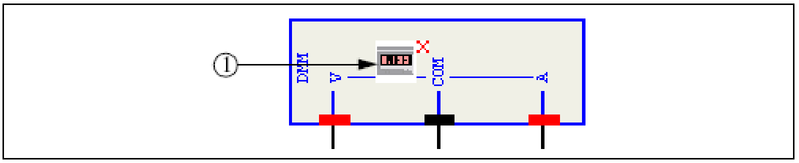
Note You can also use the **NI ELVISmx Instruments** toolbar to place this instrument. Refer to the *NI ELVISmx Instruments Toolbar* section for more information.

For information about using this instrument, click the **Help** button on its SFP to display the *NI ELVIS* help file.

Digital Multimeter

This instrument is a digital multimeter (DMM).

In NI ELVIS II schematics, this instrument's icon is located in the NI ELVIS II Platform Control Panel DMM Section of the NI ELVIS II schematic, as shown in the figure below (1).



1 Digital Multimeter Icon

Complete the following step to connect the instrument:

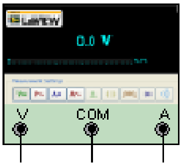
1. Place wires from the desired points in your schematic to the pins on the NI ELVIS II Front Control Panel DMM Section described below:
    - **V**—Voltmeter input.
    - **COM**—Common input.
    - **A**—Ammeter input.

Complete the following steps to access the SFP:

1. Double-click the instrument's icon, shown in the figure above (1). The SFP appears.
2. Change the settings in the SFP as desired.

Complete the following steps to place this instrument directly onto *any* Multisim workspace:

1. Select **Simulate»Instruments»NI ELVISmx Instruments» NI ELVISmx Digital Multimeter**.
2. Click to place the instrument in the desired location on the workspace. Its icon appears as shown below:



3. Wire in the same manner as any other Multisim instrument.

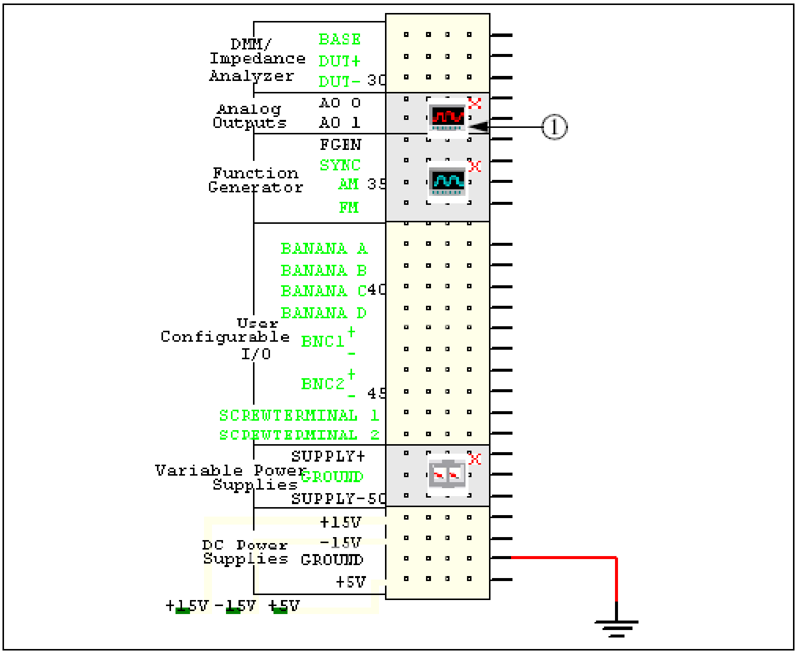
Note You can also use the **NI ELVISmx Instruments** toolbar to place this instrument. Refer to the *NI ELVISmx Instruments Toolbar* section for more information.

For information about using this instrument, click the **Help** button on its SFP to display the *NI ELVIS* help file.

Arbitrary Waveform Generator

This instrument generates user-specified waveforms.

In NI ELVIS II schematics, this instrument's icon is located in the NI ELVIS II Left-Bottom Rail Section of the NI ELVIS II schematic, as shown in the figure below (1).



1 Arbitrary Waveform Generator Icon

Complete the following step to connect the instrument:

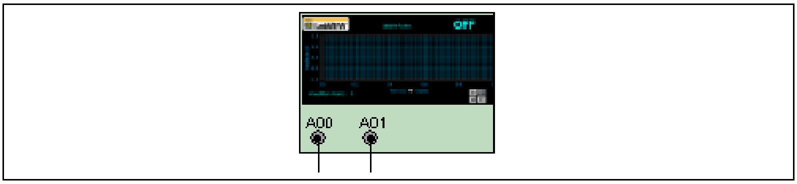
1. Place wires from the desired points in your schematic to the pins on the NI ELVIS II Left-Bottom Rail Section described below:
   - **AO 0**—Output pin 0.
   - **AO 1**—Output pin 1.

Complete the following steps to access the SFP:

1. Double-click the instrument's icon, shown in the figure above (1). The SFP appears.
2. Change the settings in the SFP as desired.

Complete the following steps to place this instrument directly onto *any* Multisim workspace:

1. Select **Simulate»Instruments»NI ELVISmx Instruments» NI ELVISmx Arbitrary Waveform Generator**.
2. Click to place the instrument in the desired location on the workspace. Its icon appears as shown below:



3. Wire in the same manner as any other Multisim instrument.

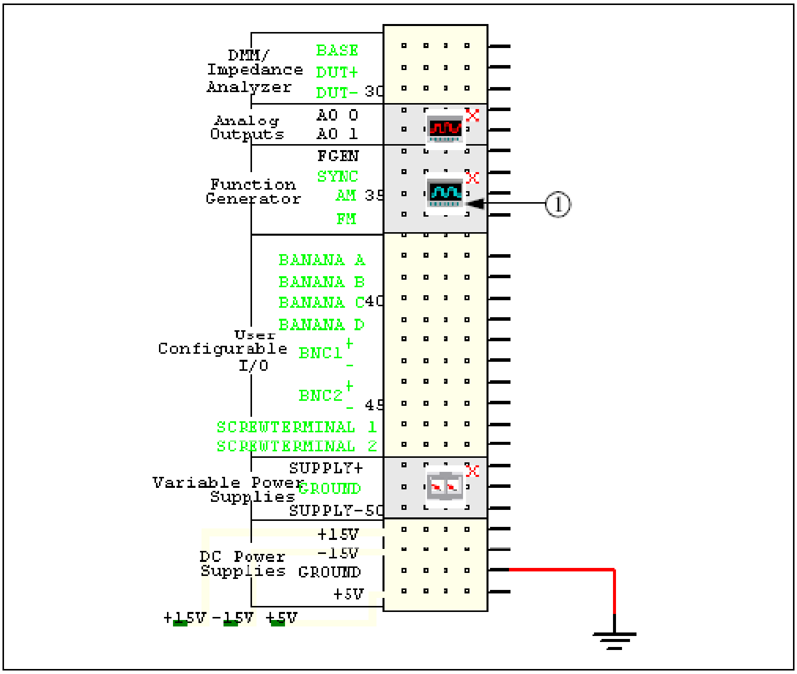
Note You can also use the **NI ELVISmx Instruments** toolbar to place this instrument. Refer to the *NI ELVISmx Instruments Toolbar* section for more information.

For information about using this instrument, click the **Help** button on its SFP to display the *NI ELVIS* help file.

Function Generator

This instrument generates sine, square or triangle waves.

In NI ELVIS II schematics, this instrument's icon is located in the NI ELVIS II Left-Bottom Rail Section of the NI ELVIS II schematic, as shown in the figure below (1).



1	Function Generator Icon

Complete the following step to connect the instrument:

1. Place wires from the desired points in your schematic to the pin on the NI ELVIS II Left-Bottom Rail Section described below:
   - **FGEN**—Output from the device.

Complete the following steps to access the SFP:

1. Double-click the instrument's icon, shown in the figure above (1). The SFP appears.
2. Change the settings in the SFP as desired.

Complete the following steps to place this instrument directly onto *any* Multisim workspace:

1. Select **Simulate»Instruments»NI ELVISmx Instruments» NI ELVISmx Function Generator**.
2. Click to place the instrument in the desired location on the workspace. Its icon appears as shown below:

3. Wire in the same manner as any other Multisim instrument.

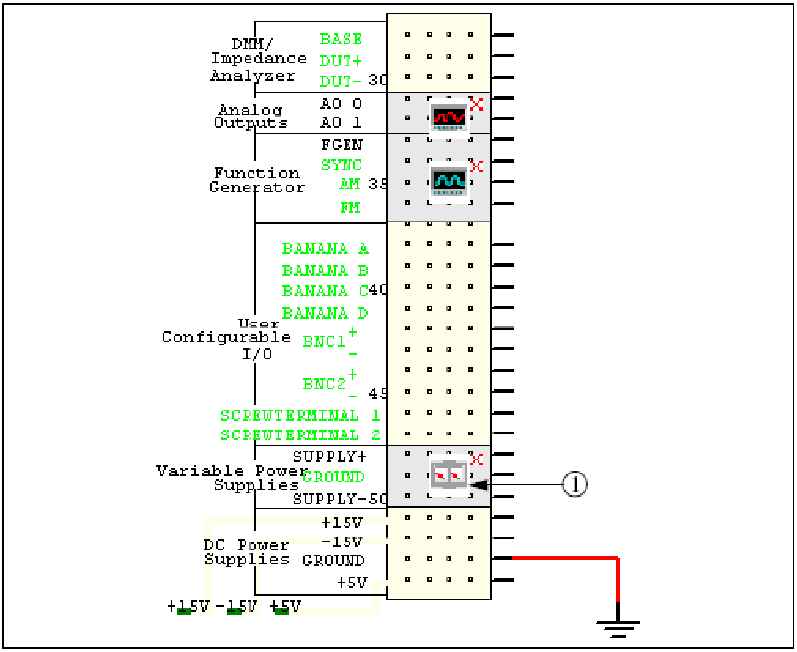
Note You can also use the **NI ELVISmx Instruments** toolbar to place this instrument. Refer to the *NI ELVISmx Instruments Toolbar* section for more information.

For information about using this instrument, click the **Help** button on its SFP to display the *NI ELVIS* help file.

Variable Power Supply

This device is a variable DC power supply.

In NI ELVIS II schematics, this instrument's icon is located in the NI ELVIS II Left-Bottom Rail Section of the NI ELVIS II schematic, as shown in the figure below (1).



1	Variable Power Supply Icon

Complete the following step to connect the device:

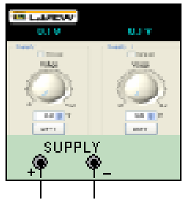
1. Place wires from the desired points in your schematic to the pins on the NI ELVIS II Left-Bottom Rail Section described below:
   - **SUPPLY +** —Positive output.
   - **SUPPLY –** —Negative output.

Complete the following steps to access the SFP:

1. Double-click the instrument's icon, shown in the figure above (1). The SFP appears.
2. Change the settings in the SFP as desired.

Complete the following steps to place this instrument directly onto *any* Multisim workspace:

1. Select **Simulate»Instruments»NI ELVISmx Instruments» NI ELVISmx Variable Power Supply**.
2. Click to place the instrument in the desired location on the workspace. Its icon appears as shown below:



3. Wire in the same manner as any other Multisim instrument.

Note You can also use the **NI ELVISmx Instruments** toolbar to place this instrument. Refer to the *NI ELVISmx Instruments Toolbar* section for more information.

For information about using this instrument, click the **Help** button on its SFP to display the *NI ELVIS* help file.

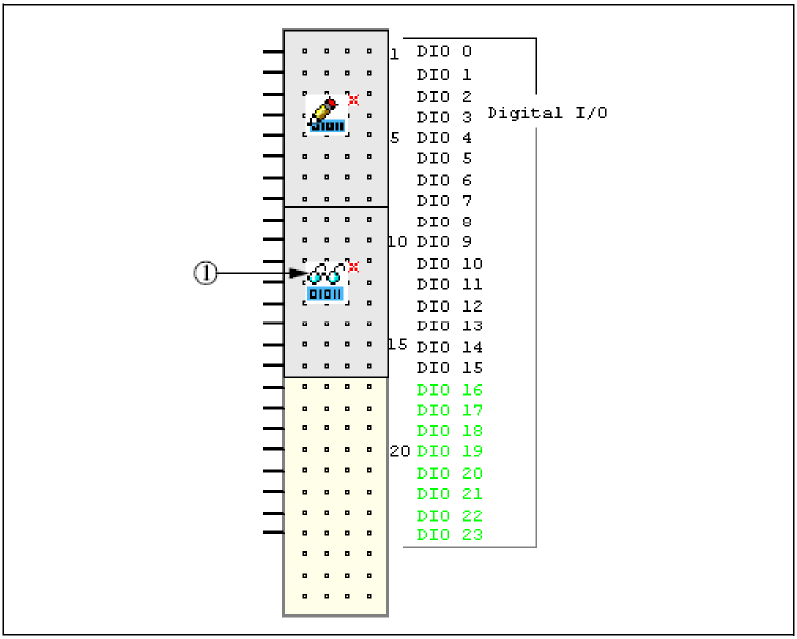
The left-bottom rail also contains the following fixed DC power supplies:

- **+15 V**
- **–15 V**
- **+5 V**

Digital Reader

This instrument reads digital data.

In NI ELVIS II schematics, this instrument's icon is located in the NI ELVIS II Right-Top Rail Section of the NI ELVIS II schematic, as shown in the figure below (1).



1	Digital Reader Icon

Complete the following step to connect the instrument:

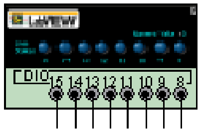
1. Place wires from the desired points in your schematic to the pins on the NI ELVIS II Right-Top Rail Section described below:
   - **DIO 8** - **DIO 15**—The inputs for the device.

Complete the following steps to access the SFP:

1. Double-click the instrument's icon, shown in the figure above (1). The SFP appears.
2. Change the settings in the SFP as desired.

Complete the following steps to place this instrument directly onto *any* Multisim workspace:

1. Select **Simulate»Instruments»NI ELVISmx Instruments» NI ELVISmx Digital Reader**.
2. Click to place the instrument in the desired location on the workspace. Its icon appears as shown below:



3. Wire in the same manner as any other Multisim instrument.

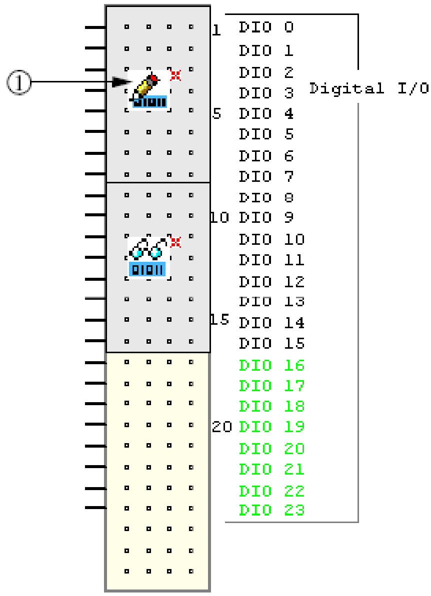
Note You can also use the **NI ELVISmx Instruments** toolbar to place this instrument. Refer to the *NI ELVISmx Instruments Toolbar* section for more information.

For information about using this instrument, click the **Help** button on its SFP to display the *NI ELVIS* help file.

Digital Writer

This instrument updates the digital output on the NI ELVIS Prototyping Board with user-specific digital patterns.

In NI ELVIS II schematics, this instrument's icon is located in the NI ELVIS II Right-Top Rail Section of the NI ELVIS II schematic, as shown in the figure below (1).



1 Digital Writer Icon

Complete the following step to connect the instrument:

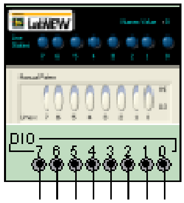
1. Place wires from the desired points in your schematic to the pins on the NI ELVIS II Right-Top Rail Section described below:
   - **DIO 0** - **DIO 7**—The outputs for the device.

Complete the following steps to access the SFP:

1. Double-click the instrument's icon, shown in the figure above (1). The SFP appears.
2. Change the settings in the SFP as desired.

Complete the following steps to place this instrument directly onto *any* Multisim workspace:

1. Select **Simulate»Instruments»NI ELVISmx Instruments» NI ELVISmx Digital Writer**.
2. Click to place the instrument in the desired location on the workspace. Its icon appears as shown below:



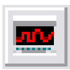
3. Wire in the same manner as any other Multisim instrument.

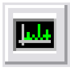
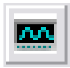
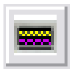
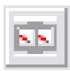
Note You can also use the **NI ELVISmx Instruments** toolbar to place this instrument. Refer to the *NI ELVISmx Instruments Toolbar* section for more information.

For information about using this instrument, click the **Help** button on its SFP to display the *NI ELVIS* help file.

NI ELVISmx Instruments Toolbar

The buttons in the **NI ELVISmx Instruments** toolbar are described below. In each case, the button places a specific NI ELVISmx instrument on the workspace.

Button	Description
	**NI ELVISmx Arbitrary Waveform Generator** button. Places an NI ELVISmx arbitrary waveform generator on the workspace. Refer to the *Arbitrary Waveform Generator* section for more information.
	**NI ELVISmx Digital Reader** button. Places an NI ELVISmx digital reader on the workspace. Refer to the *Digital Reader* section for more information.
	**NI ELVISmx Digital Writer** button. Places an NI ELVISmx digital writer on the workspace. Refer to the *Digital Writer* section for more information.

Button	Description
	**NI ELVISmx Digital Multimeter** button. Places an NI ELVISmx digital multimeter on the workspace. Refer to the *Digital Multimeter* section for more information.
	**NI ELVISmx Dynamic Signal Analyzer** button. Places an NI ELVISmx dynamic signal analyzer on the workspace. Refer to the *Dynamic Signal Analyzer* section for more information.
	**NI ELVISmx Function Generator** button. Places an NI ELVISmx function generator on the workspace. Refer to the *Function Generator* section for more information.
	**NI ELVISmx Oscilloscope** button. Places an NI ELVISmx oscilloscope on the workspace. Refer to the *Oscilloscope* section for more information.
	**NI ELVISmx Variable Power Supply** button. Places an NI ELVISmx variable power supply on the workspace. Refer to the *Variable Power Supply* section for more information.

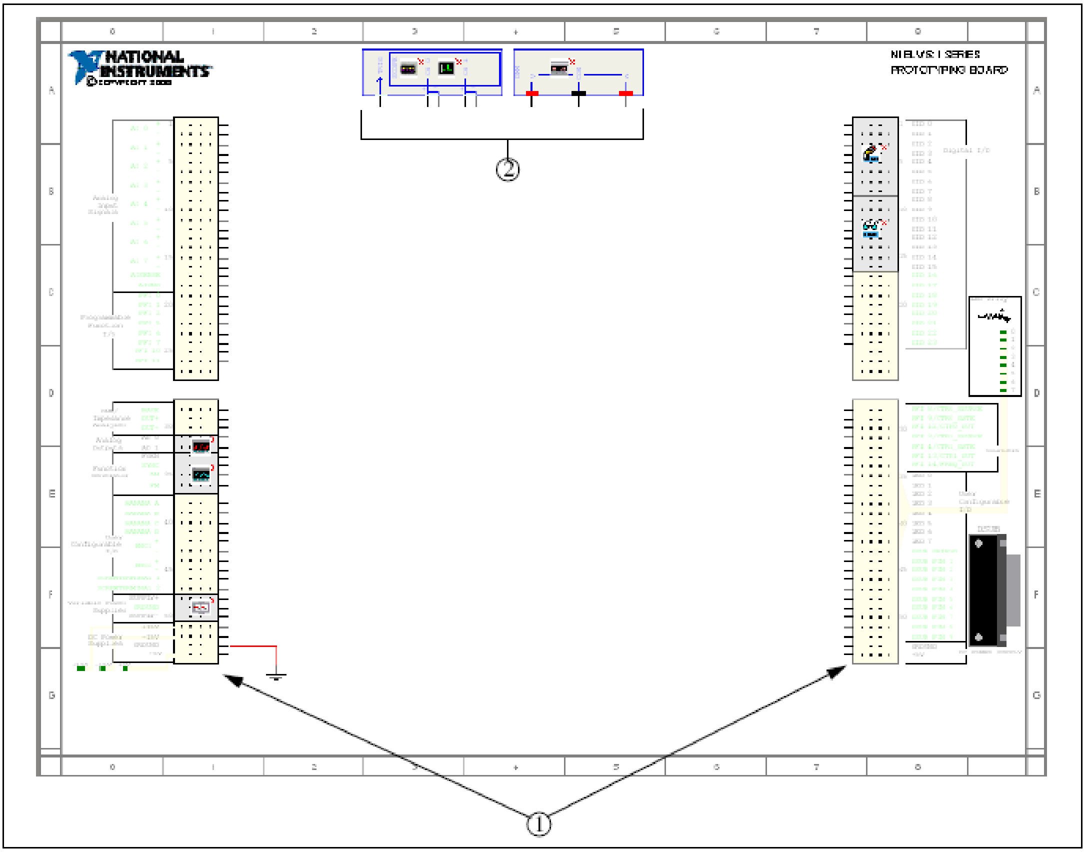
The Virtual NI ELVIS II Schematic

A virtual NI ELVIS II schematic contains a number of items that correspond to elements of the real-world NI ELVIS II workstation. The connection and control of these elements is described in this section.

Note This section describes the behavior of Multisim's NI ELVIS II schematic. Refer to *The Virtual NI ELVIS I Schematic* section for information on Multisim's original NI ELVIS I functionality.

Complete the following steps to create a new virtual NI ELVIS II schematic:

1. Select **File»New»NI ELVIS II Schematic**. When first opened, a virtual NI ELVIS II schematic appears as shown below.



1	Rails	2	Platform Control Panel

Note The ground connector that appears at the bottom left of the diagram is the reference point for measurements taken during simulation, and must not be removed.

2. Place and wire components in the virtual NI ELVIS II schematic in the same manner as other Multisim schematics. For details, refer to the *Multisim User Manual*.

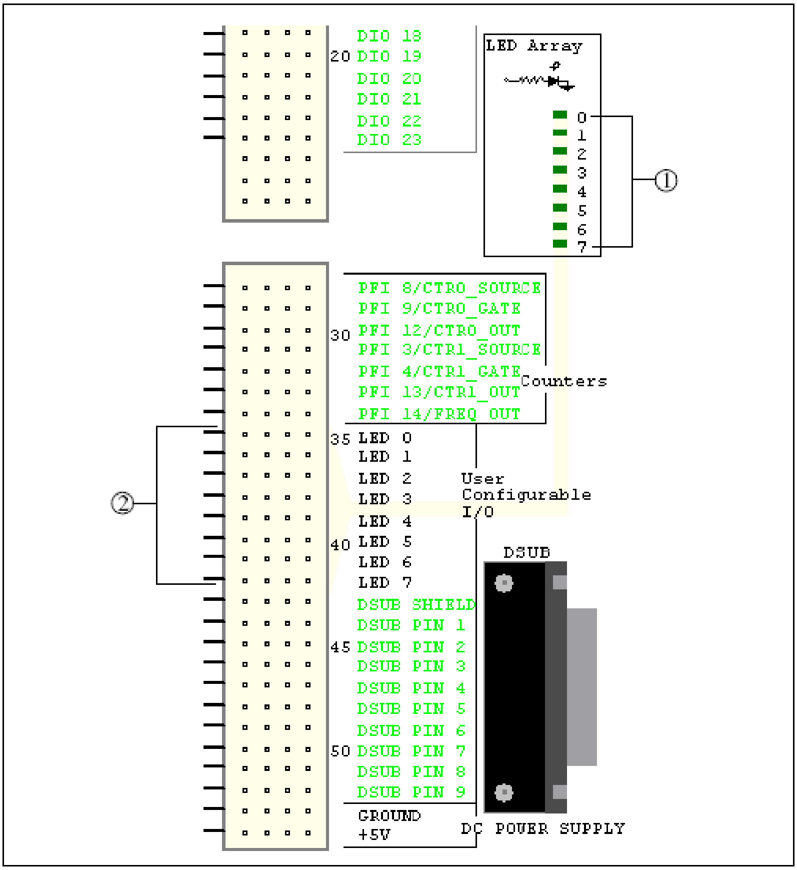
The prototyping board rails (see 1 in the figure above) found to the left and right of the main workspace correspond to rails on the prototyping board of the real-world version of NI ELVIS II, and are labelled in the same manner. At the top in the platform control panel (see 2 in the figure above), there are icons for connecting to the oscilloscope and digital multimeter. These instruments are isolated and are not available in the rails of the prototyping board. They are found on the main NI ELVIS II unit.

Rows on the rails that are shown with green labels are not enabled for simulation in Multisim. However, they can be used for schematic capture and viewing of the completed virtual NI ELVIS II schematic in the 3D view.

Unlike other Multisim components, these rails and instrument icons cannot be moved to other places on the workspace.

LEDs

Connections to the eight LEDs on the right side of the NI ELVIS II schematic are found in the NI ELVIS II Right-Bottom Rail Section, as shown in the figure below (2). During simulation, any of these LEDs (1) that are correctly driven will light.

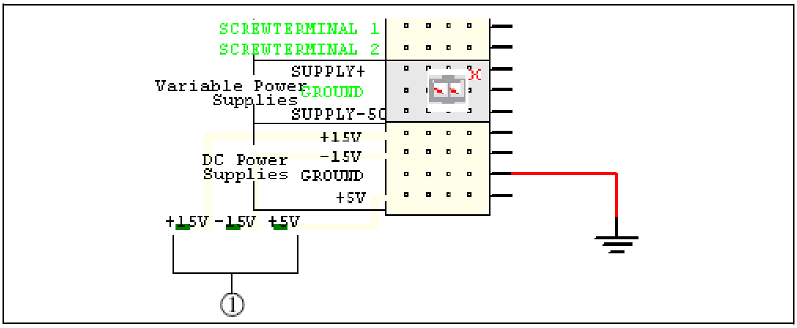


1 LEDs	2 LED Connections

Complete the following to connect to an LED:

1. Place a wire from one of the LED rows (**LED 0** through **LED 7**) to the desired point in your schematic.

There are also three power supply LEDs in the lower-left section of any virtual NI ELVIS II schematic, as shown in the figure below (1):



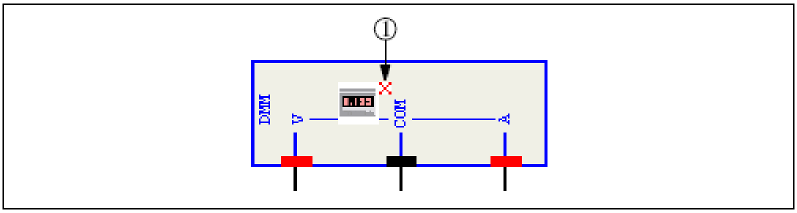
1	Power Supply LEDs

During simulation, these LEDs will light whether or not connections have been made to their corresponding pins in the prototyping rail. They indicate that power is available to the respective connections.

Enabling NI ELVIS II Schematic Instruments for Simulation

NI ELVISmx instruments on the NI ELVIS II schematic can be enabled or disabled for simulation on an individual basis. Each enabled instrument consumes system resources, so setting unused instruments to disabled will improve simulation speed.

When an NI ELVISmx instrument is disabled, a small red “X” appears next to the upper-right corner of the instrument icon on the schematic, as shown in the figure below (1). By default, all instruments in a new NI ELVIS II schematic begin as disabled.



1 Indicates Instrument is Disabled for Simulation

Note NI ELVISmx instruments that are placed directly onto a workspace from the **NI ELVISmx Instruments** toolbar cannot be disabled for simulation. Refer to the *NI ELVISmx Instruments Toolbar* section for more information.

NI ELVISmx instruments' enabled state for simulation may be modified in any one of three ways:

- Double-click on a disabled instrument to display its SFP. If simulation is running, a warning displays advising you to stop simulation before you enable the instrument.
- Right-click on the instrument to display a context menu that includes the item **NI ELVIS II Instrument Enabled in Simulation**. Select this menu item to toggle its check mark and switch the instrument from enabled to disabled and back again. This command is unavailable during simulation.
- Select **Simulate»NI ELVIS II Simulation Settings** to display the **NI ELVIS II Simulation Settings** dialog box. This dialog lists all NI ELVIS instruments on the schematic, with a check box next to each one. Check/uncheck the instrument name to enable/disable the instrument on the schematic. This menu item disabled during simulation.

Note This menu item is only active when an NI ELVIS II schematic is selected as the active workspace.

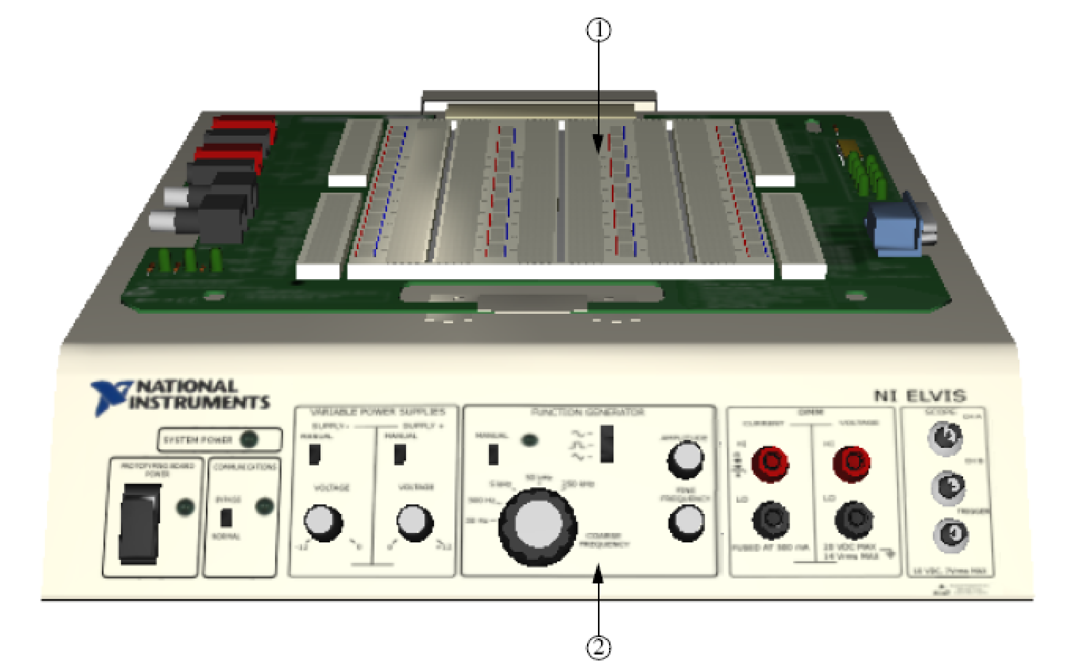
After enabling the desired instruments, run the simulation in the usual manner.

Note Refer to the *Multisim User Manual* for information about simulation.

NI ELVIS Prototyping Boards

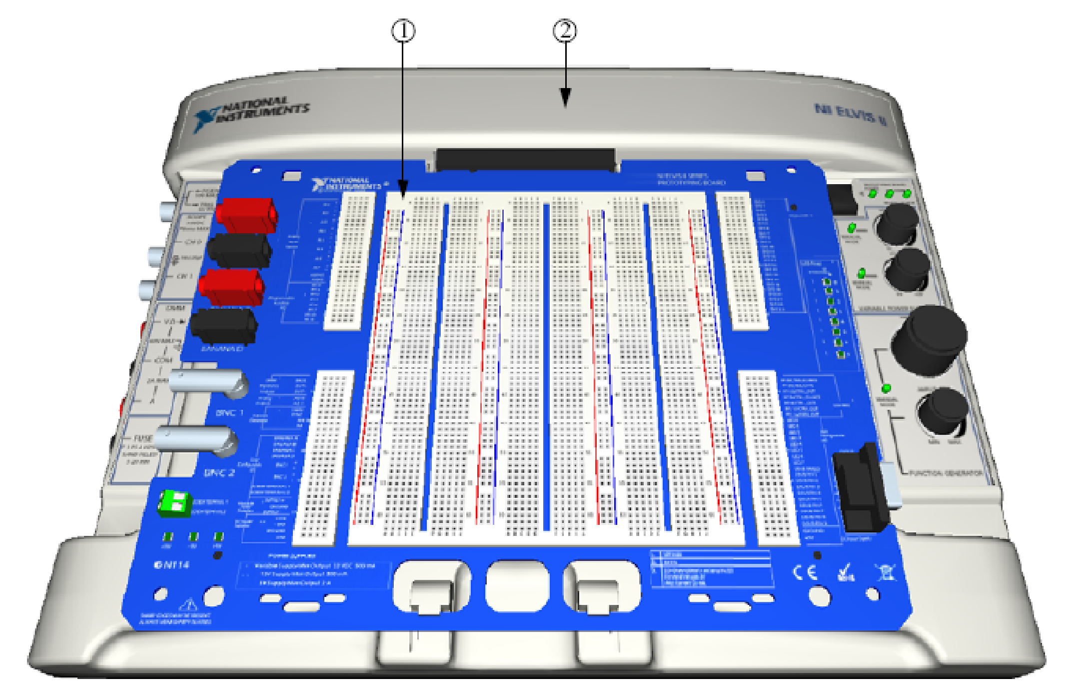
Once you have completed the virtual NI ELVIS I or NI ELVIS II schematic, you are ready to place the components on the 3D rendering of the prototyping board.

The figure below shows the virtual NI ELVIS I 3D prototyping board (1) and platform (2) with no placed components.



1 Prototyping Board	2 Platform

The figure below shows the virtual NI ELVIS II 3D prototyping board (1) and platform (2) with no placed components.

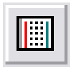


1	Prototyping Board	2	Platform

Note The controls that appear on the NI ELVIS I and the NI ELVIS II 3D platforms are inactive. Interactive simulation is accomplished via the schematic view. For details on simulation, refer to the *Multisim User Manual* or the *Multisim* help file.

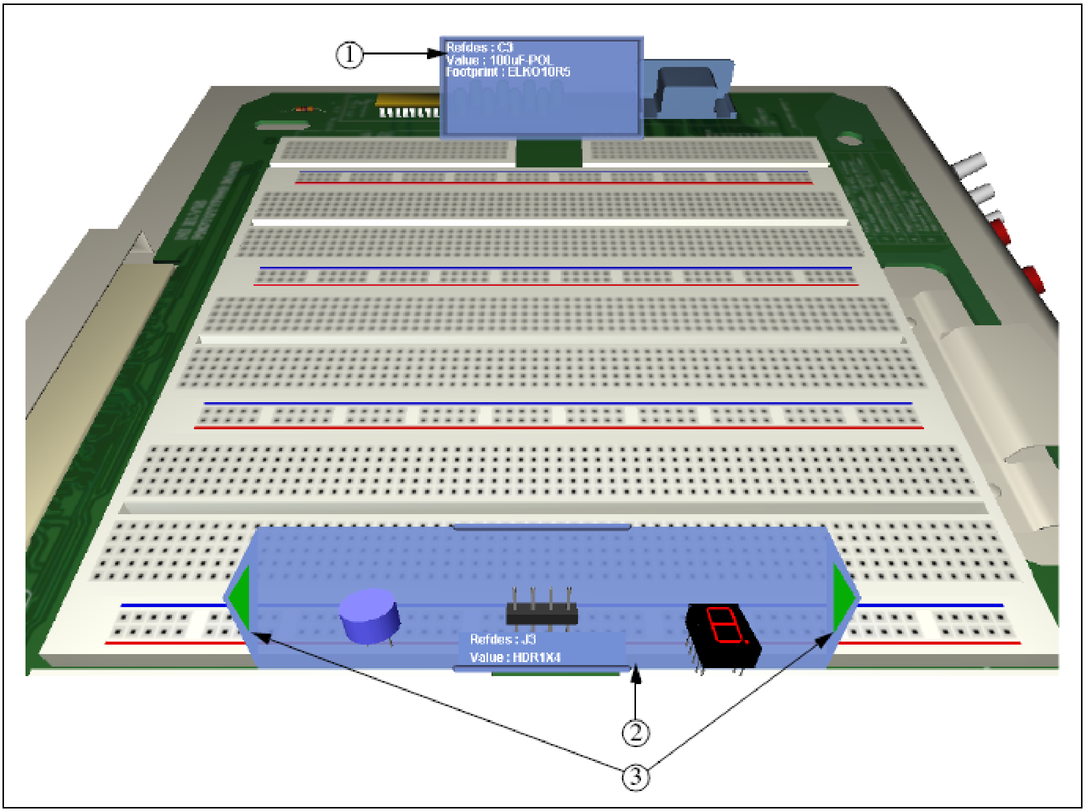
Tip For information about changing the 3D view, refer to the *Manipulating the Breadboard View* section of Chapter 2, *Breadboarding*.

Complete the following steps to place components on the 3D prototyping board:



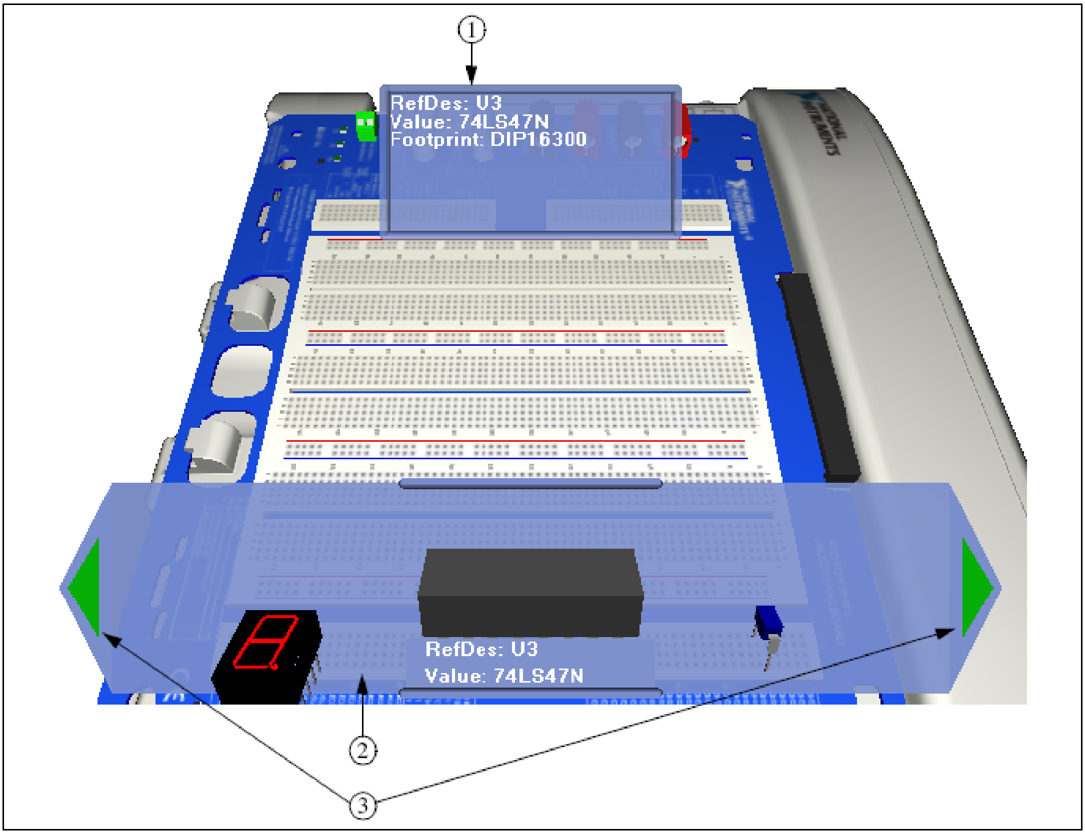
1. Select **Tools»Show Breadboard** from the main Multisim menu.

   For NI ELVIS I designs, the 3D prototyping board appears similar to the example shown in the figure below:



1	Component Description	2	Place Component Bar	3	Scroll Arrows

For NI ELVIS II designs, the 3D prototyping board appears similar to the example shown in the figure below:

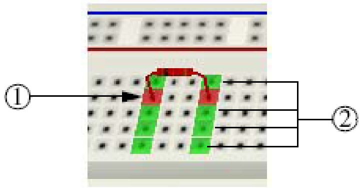


1	Component Description	2	Place Component Bar	3	Scroll Arrows

Note The **Place Component Bar**, shown in the figures above (2), is where components waiting to be placed on the prototyping board appear. To view other components, click the scroll arrows (3). To see a description of a component (for example, (1) in the above figure), hover the cursor over the component. 3D viewing options are set in the **Preferences** dialog box. Refer to the *3D Options* section of Chapter 2, *Breadboarding*, for more information.

2. Click on a component in the **Place Component Bar** and drag it to the desired location on the board. As the component passes over the board, sockets change color as shown in the figure below.

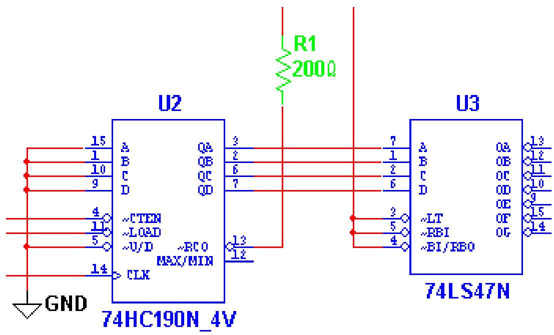
Red sockets (1) indicate where the component's pins will be placed when the mouse button is released. Green (2) indicates sockets that are internally connected to the red socket in the same row on the board.



1	Red Socket	2	Connected Green Sockets

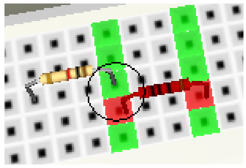
Tip Select <Ctrl-R> to rotate a selected component 90 degrees clockwise or <Ctrl-Shift-R> to rotate it 90 degrees counter-clockwise.

3. Release the mouse button to place the component. The colored (red and green) sockets on the board no longer appear.
4. Return to the schematic view and note that the color of the placed component has changed as shown in the example below (R1).



5. Continue placing the circuit's components on the board. When all the components have been placed, the **Place Component Bar** collapses.

Tip Where pins of components are connected on the schematic, you can place them in connected sockets as circled in the figure below. This technique can reduce the number of jumper wires required.



Note Refer to the *Appearance of 3D Components* section of Chapter 2, *Breadboarding*, for more information.

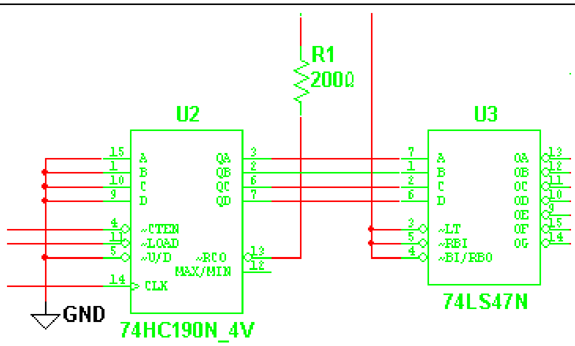
Wiring Placed Components in 3D Mode

Note This section applies to both NI ELVIS I and NI ELVIS II schematics.

By placing component pins that are connected on the schematic into sockets on the 3D prototyping board that are internally connected, much of the "wiring" can be done at the same time components are placed. However, in most circuits, it will also be necessary to place jumpers on the 3D prototyping board to complete the wiring of the placed components.

Complete the following steps to place a jumper wire:

1. Click on a socket connected to the pin where you wish to start the jumper and begin moving the cursor. Legitimate "target" pins (green) display as you move the cursor.
2. Click to place the jumper in the desired socket.
3. Return to the schematic view and note that the color of the wire connecting the two pins has changed to green to indicate a connection has been made, as shown in the figure below, between pin 2 of U2, and pin 1 of U3.



Note If a net contains more than two connections, all must be connected before any of the wires in the net change color.

4. Continue placing jumpers until all schematic connections have been made.

Tip Run a **Design Rules and Connectivity Check** from the 3D prototyping view to see if there are any errors in your board. Refer to the *DRC and Connectivity Check* section of Chapter 2, *Breadboarding*, for more information.

Note You may also wish to refer to the *Viewing Component Information* and *Breadboard Netlist dialog box* sections of Chapter 2, *Breadboarding*.

4

Ladder Diagrams

This chapter describes the ladder diagram functionality that Multisim contains.

Some of the features described in this chapter may not be available in your edition of Multisim. Refer to the release notes for a description of the features available in your edition.

Overview

The Education edition of Multisim lets you capture and simulate **Ladder Diagrams**. These diagrams are electrically based, as opposed to the binary/digital representations employed by ladder *logic*. Diagrams of this type are used extensively for industrial motor control circuits.

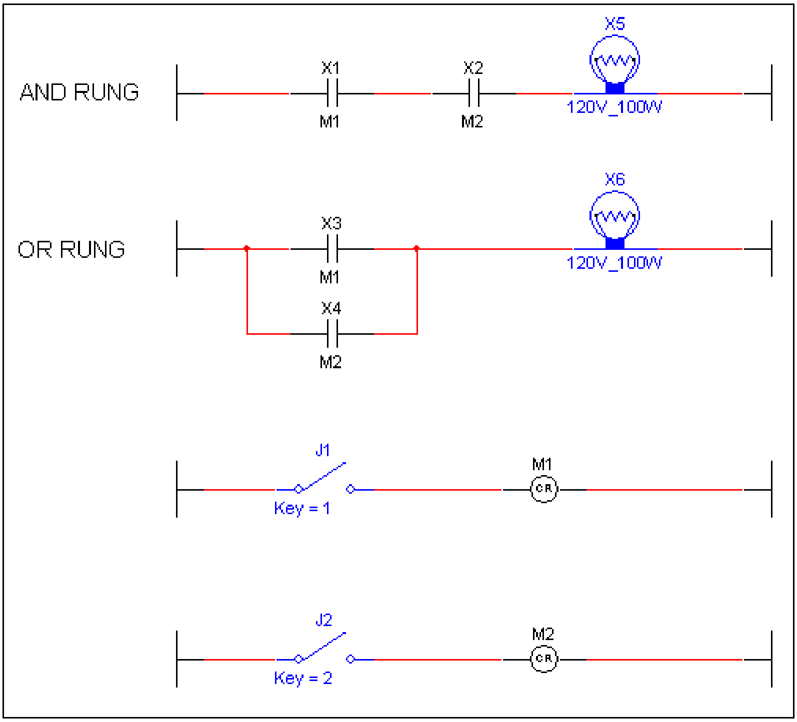
Ladder Diagrams are able to drive output devices or take input data from regular schematics and embed the instructions on how input states affect output states in either the same schematic or separate hierarchical blocks or subcircuits that contain the **Ladder Diagram**.

Note Refer to the *Multisim User Manual* for a complete description of hierarchical blocks and subcircuits.

Creating a Ladder Diagram

This section describes the steps required to make a simple **Ladder Diagram**. The concepts described here should be understood before reviewing the more complex circuits found in this chapter.

This section describes how to build the **Ladder Diagram** that is reviewed in the *AND Rungs and OR Rungs* section.

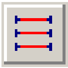


Note Refer to the *Component Reference* help file for details on ladder diagram components.

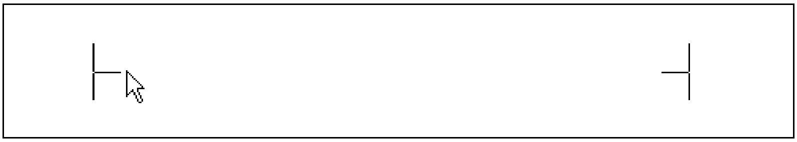
Circuit Notes:

- The relays (X1-X4) are normally open relays. When their controlling coils (M1or M2) are energized they close. The controlling coils are set in the **Value** tab of each relay's properties dialog box. Refer to the *Component Reference* help file for more information.
- Both X1 AND X2 must be closed for the lamp in the AND rung (X5) to light up.
- Either X3 OR X4 must be closed for the lamp in the OR rung (X6) to light up.
- Coil M1 controls the relays with M1 as their reference. (X1 and X3.)
- Coil M2 controls the relays with M2 as their reference. (X2 and X4.)
- Use keys 1 and 2 on your keyboard to open and close switches J1 and J2, or hover your cursor over the desired switch and click on the button that pops up.

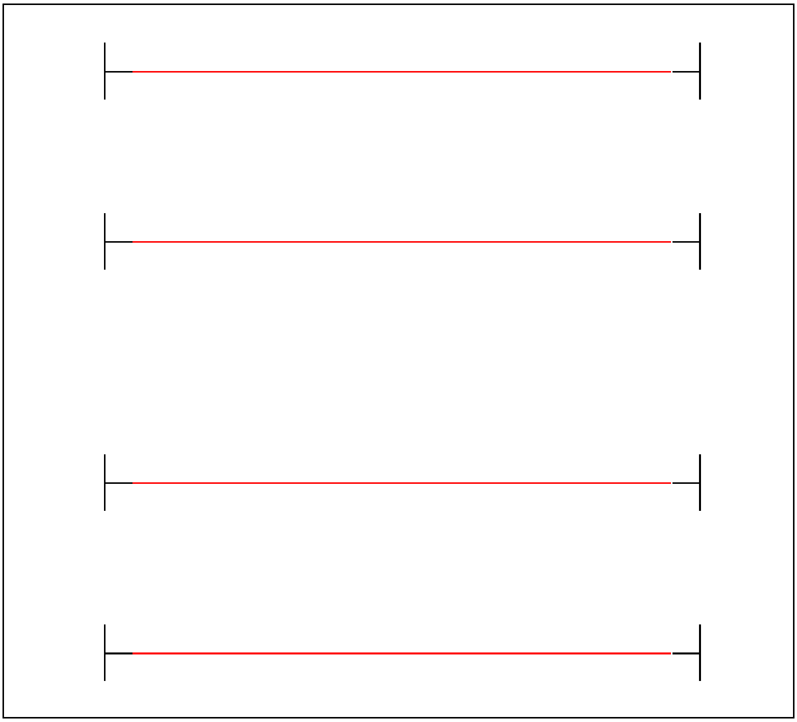
Complete the following steps to add the diagram's rungs:



1. Select **Place»Ladder Rungs**. The cursor appears with the rung's left and right terminators attached.



2. Click to place the first rung and continue clicking and placing until you have placed four rungs as shown below. Right-click to stop placing rungs.

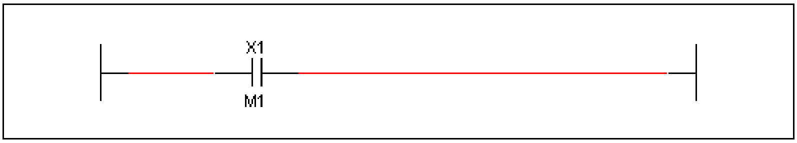


Complete the following steps to add components to the rungs:

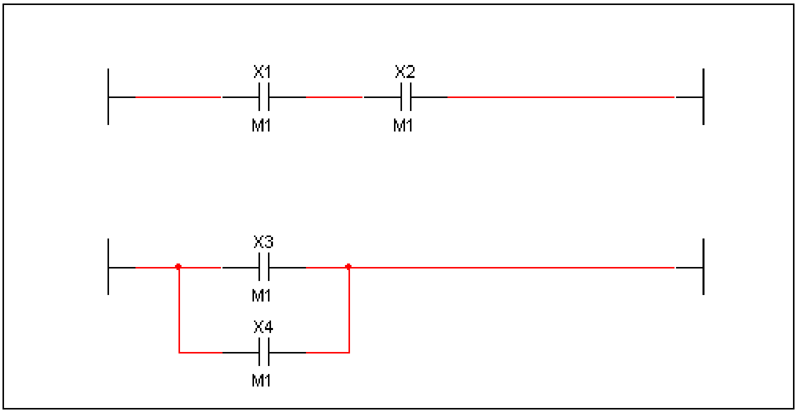
1. Select **Place»Component**, navigate to the Normally Open Relay Contact (RELAY_CONTACT_NO) and click **OK**.

Note This device is found in the Ladder Diagrams Group - Ladder Contacts Family.

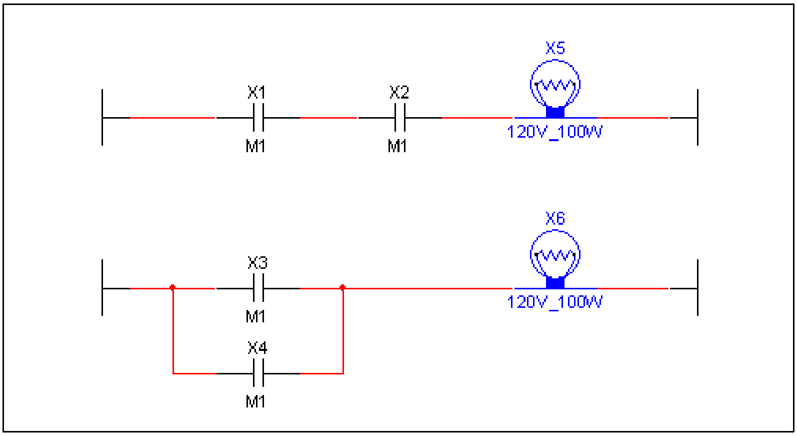
2. Drop the relay contact directly onto the first rung.



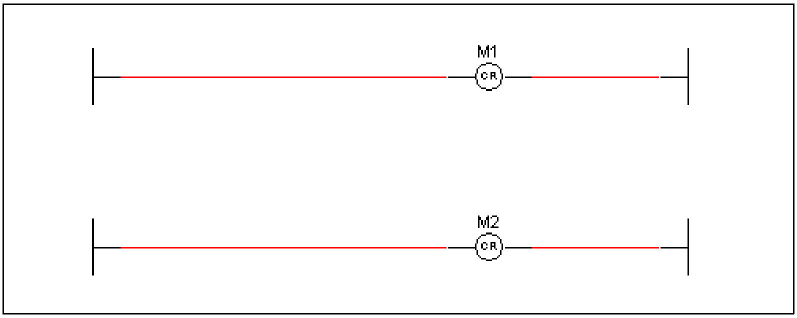
3. Continue in this manner until all relay contacts have been placed. (X4 must be placed and then wired separately.)



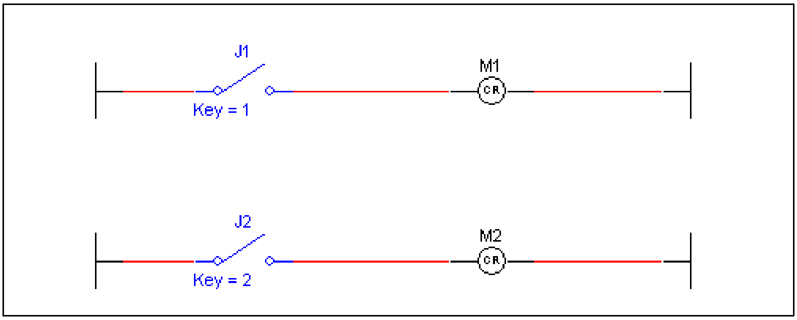
4. Place the lamps (**Group** - Indicators; **Family** - Lamp).



5. Place relay coils M1 and M2 on the third and fourth rungs (**Group** - Ladder Diagrams; **Family** - Ladder Relay Coils).



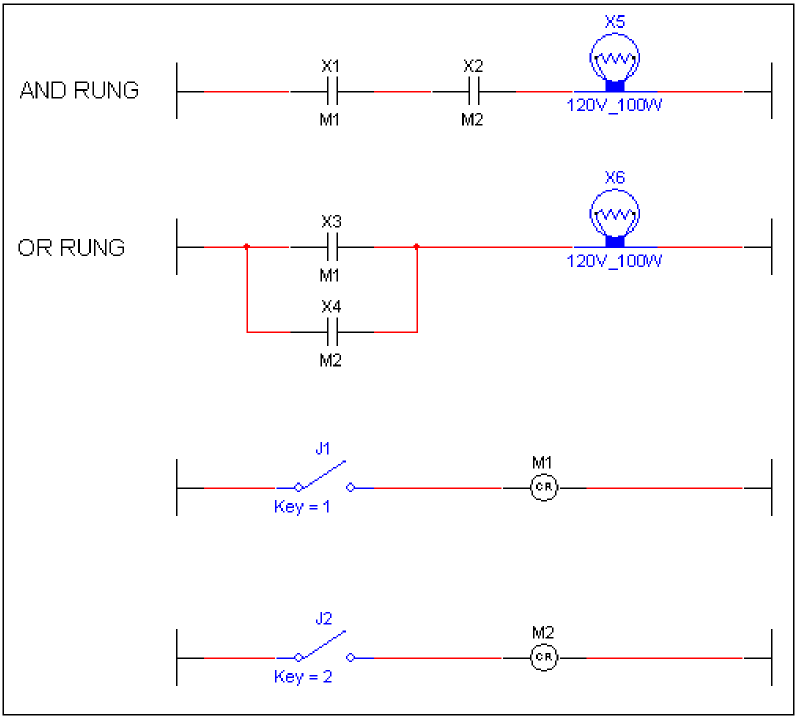
6. Place switches J1 and J2.
7. Double-click on each switch, select the **Value** tab, and change the key for J1 to 1 and the key for J2 to 2.



Complete the following steps to change the controlling device reference for X2 and X4:

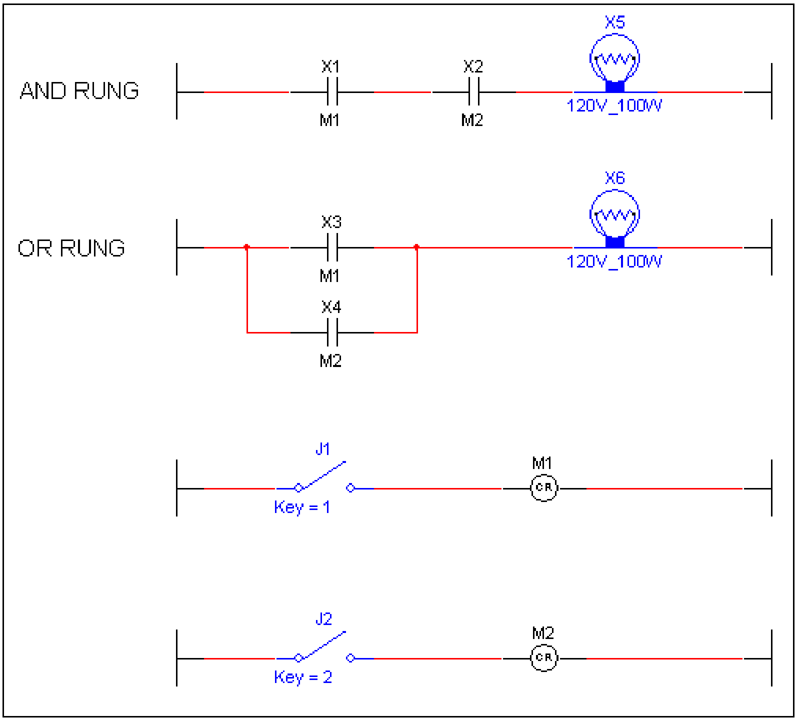
1. Double-click on X2 and click the **Value** tab.
2. Enter M2 in the **Controlling Device Reference** field and click **OK**.

Repeat for X4. The completed **Ladder Diagram** appears as shown below.



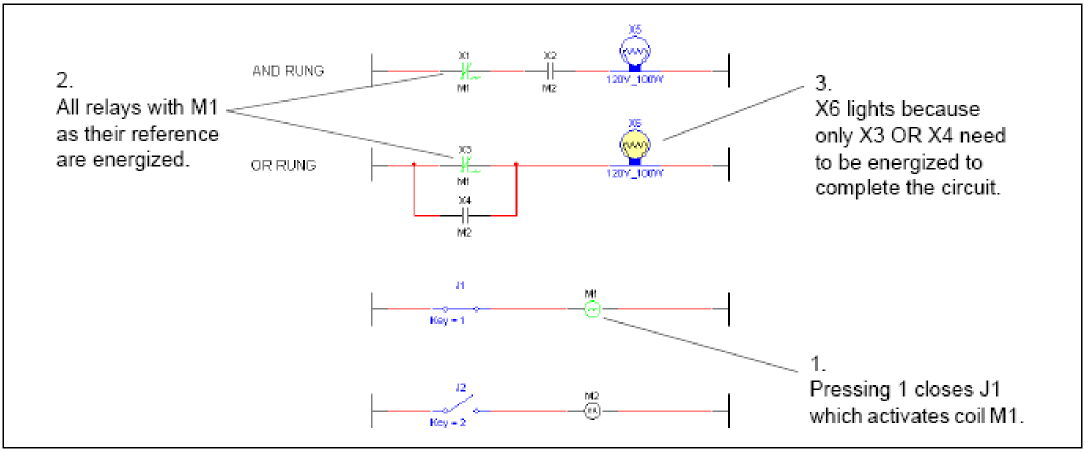
AND Rungs and OR Rungs

This section illustrates the difference between AND rungs and OR rungs that are found in **Ladder Diagrams**. The concepts described here should be understood before reviewing the more complex circuits found in this chapter.



Complete the following steps to activate the lamp in the OR rung:

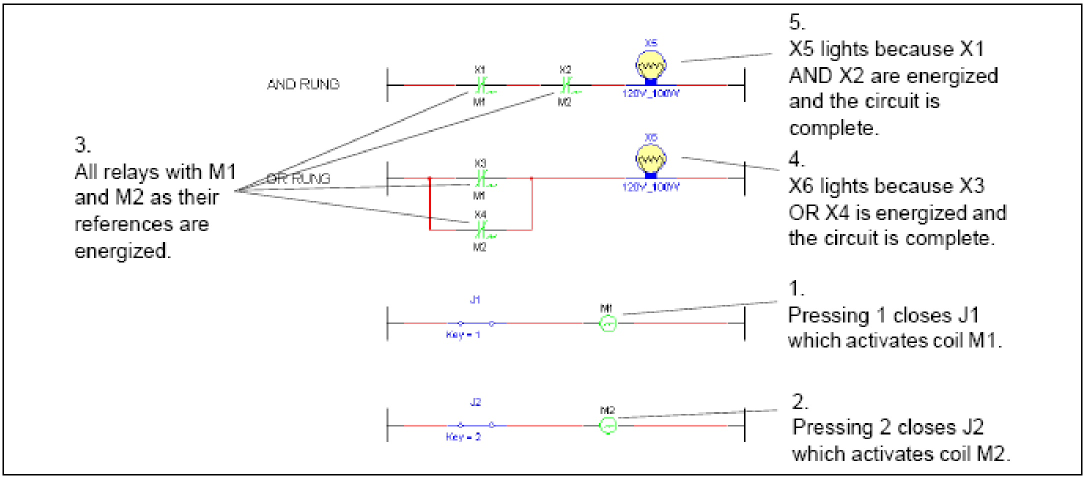
1. Select **Simulate»Run** to start simulation of the circuit.
2. Press <1> on your keyboard to close J1 (or hover your cursor over J1 and click the button that pops up). Lamp X6 lights as described below.



If you press <2> on your keyboard (or hover your cursor over J2 and click the button that pops up), J2 closes which activates coil M2. X6 lights because X4 is energized.

Complete the following steps to active the lamp in the AND rung:

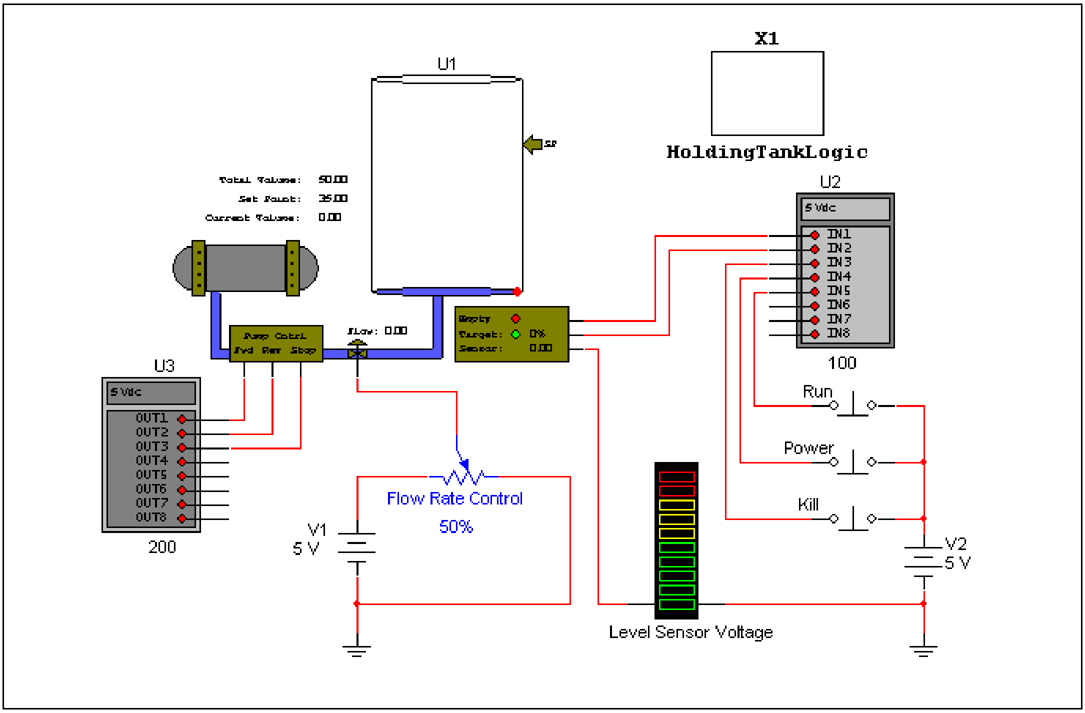
1. Select **Simulate»Run** to start simulation of the circuit.
2. Press <1> and <2> on your keyboard to close J1 and J2. Lamps X5 and X6 light as described below.



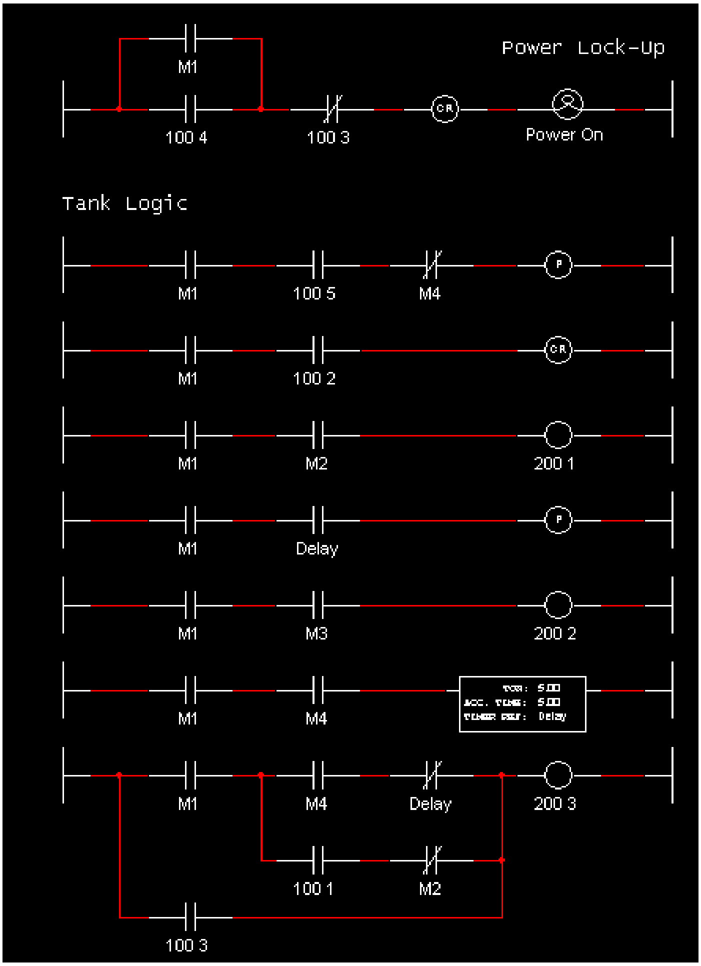
Sample Circuits

Holding Tank

This section contains an example of a logic diagram that drives a circuit that fills and then empties a fluid holding tank.



Note Refer to the *Component Reference* help file for information about the user-settable parameters for the **Holding Tank**, **Input Module** and **Output Module**.

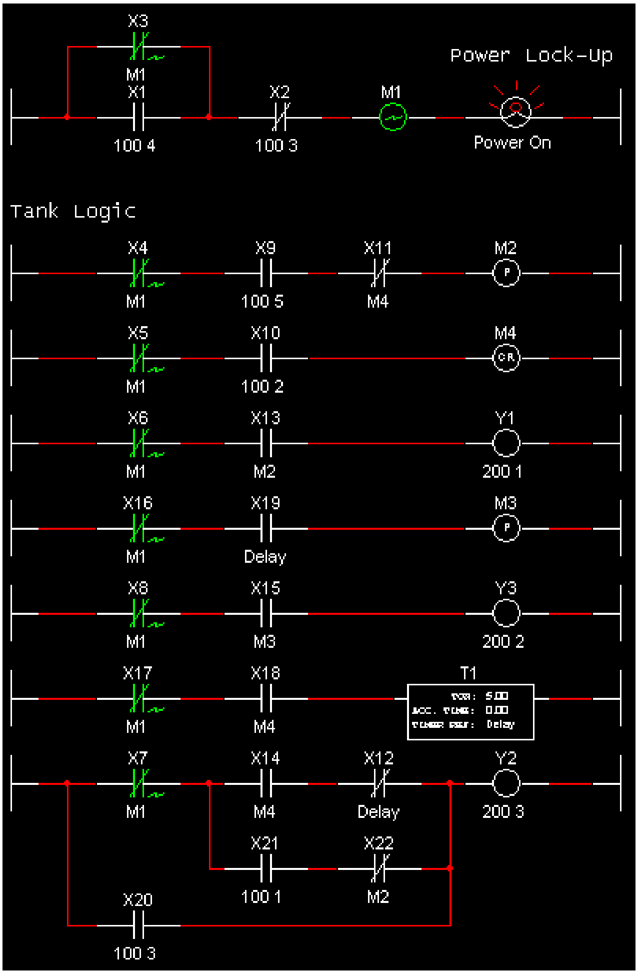


Note The Ladder Diagram is contained in a separate Hierarchical Block called `HoldingTankLogic`. For details on hierarchical blocks, refer to the *Multisim User Manual*.

Complete the following steps to activate this circuit:

1. Select **Simulate»Run** to begin simulation.
2. Press <P> on your keyboard to activate the Power temporary switch (or hover your cursor over the Power switch and click the button that pops up). This sends 5 V to pin IN4 of Input Module U2 (**Input Module Base Address** = 100) which in turn energizes Input Contact X1 in the Power Lock-up Rung of the ladder diagram. Relay Coil M1 is

energized, causing all Relay Contacts with **Relay Device Reference** = M1 to energize.

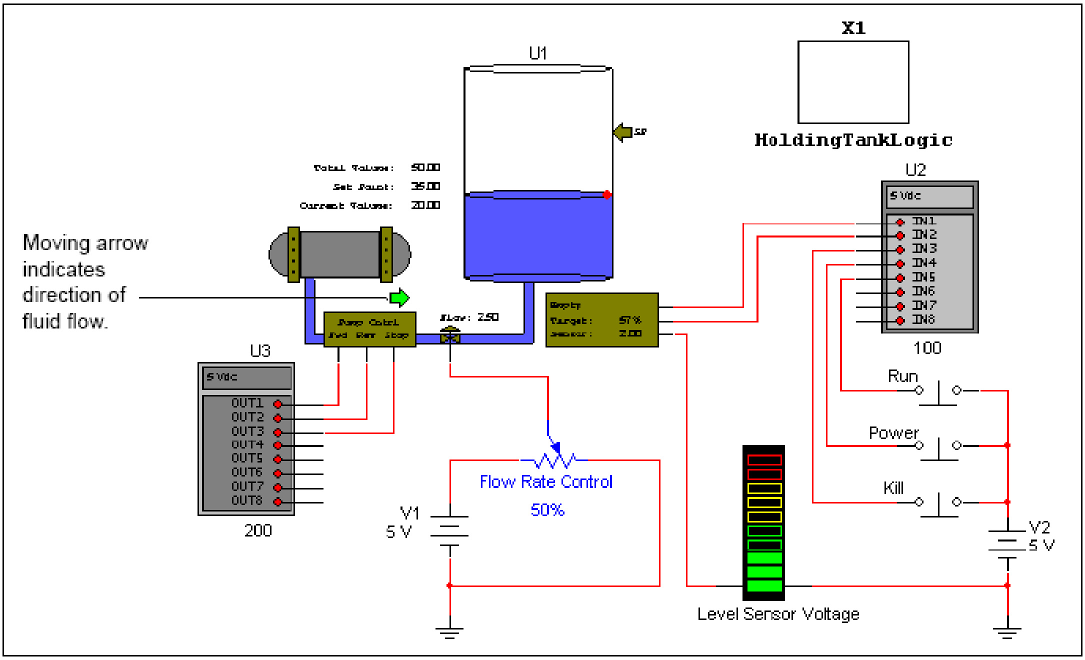


Complete the following steps to run the holding tank circuit:

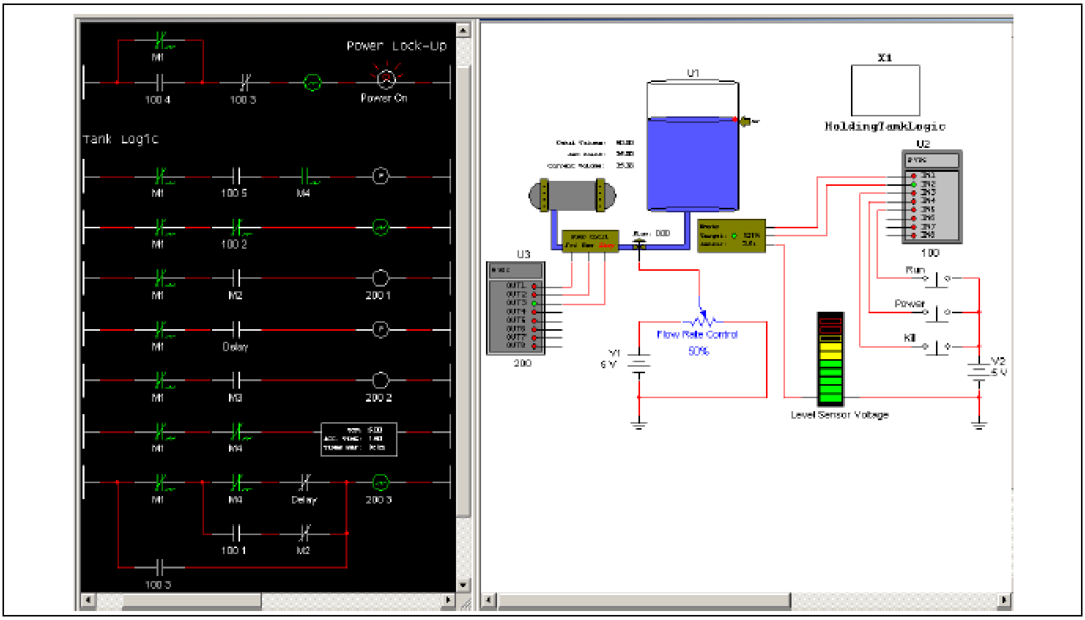
1. Activate the circuit as described above.
2. Press <R> on your keyboard (or hover your cursor over the Run switch and click the button that pops up) to activate the Run temporary switch.

Tip Select **Window»Tile Vertical** to view the ladder diagram and the circuit at the same time. Observe the interaction between the ladder diagram and the circuit as the simulation proceeds.

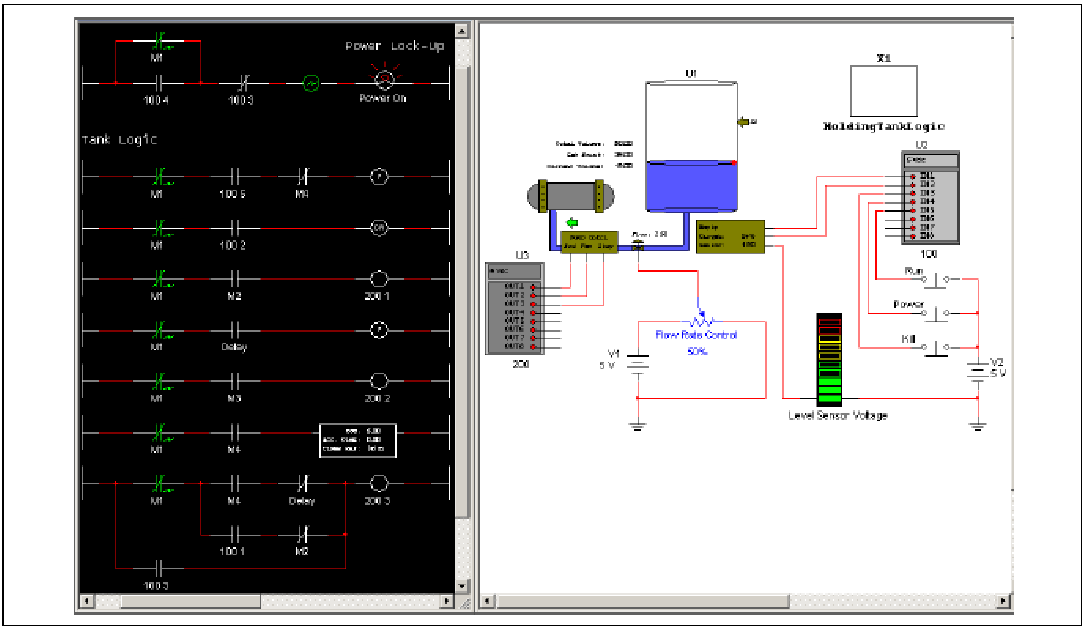
3. As the simulation proceeds, the tank begins to fill.



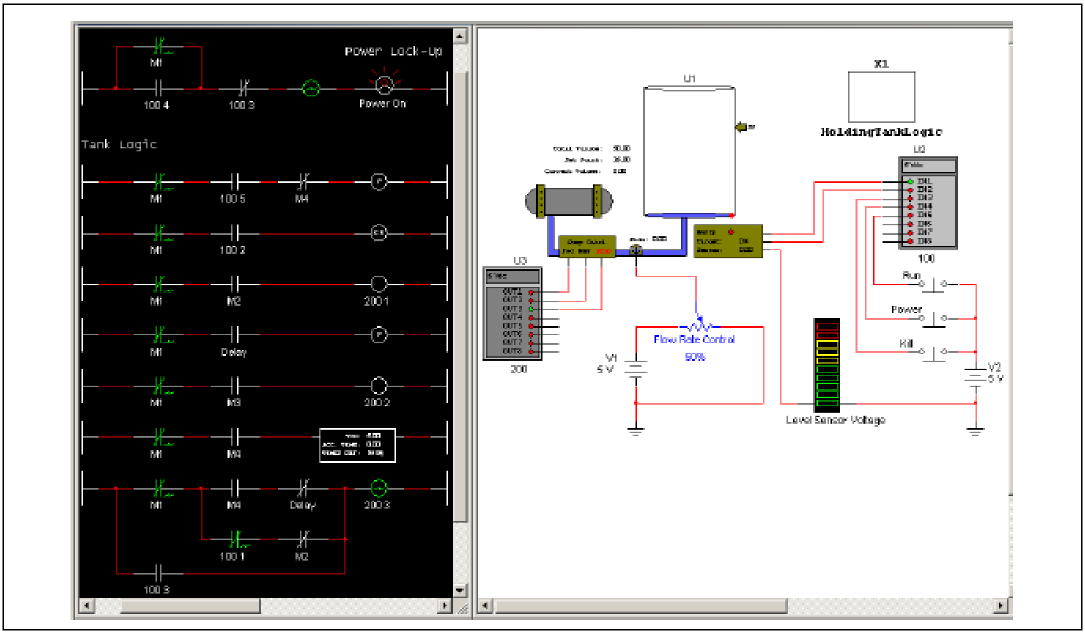
4. When the level of the fluid in the tank gets to the **Set Point**, fluid stops being pumped.



5. After a delay of five seconds, the tank begins to empty.



6. When the tank is empty, the flow stops.



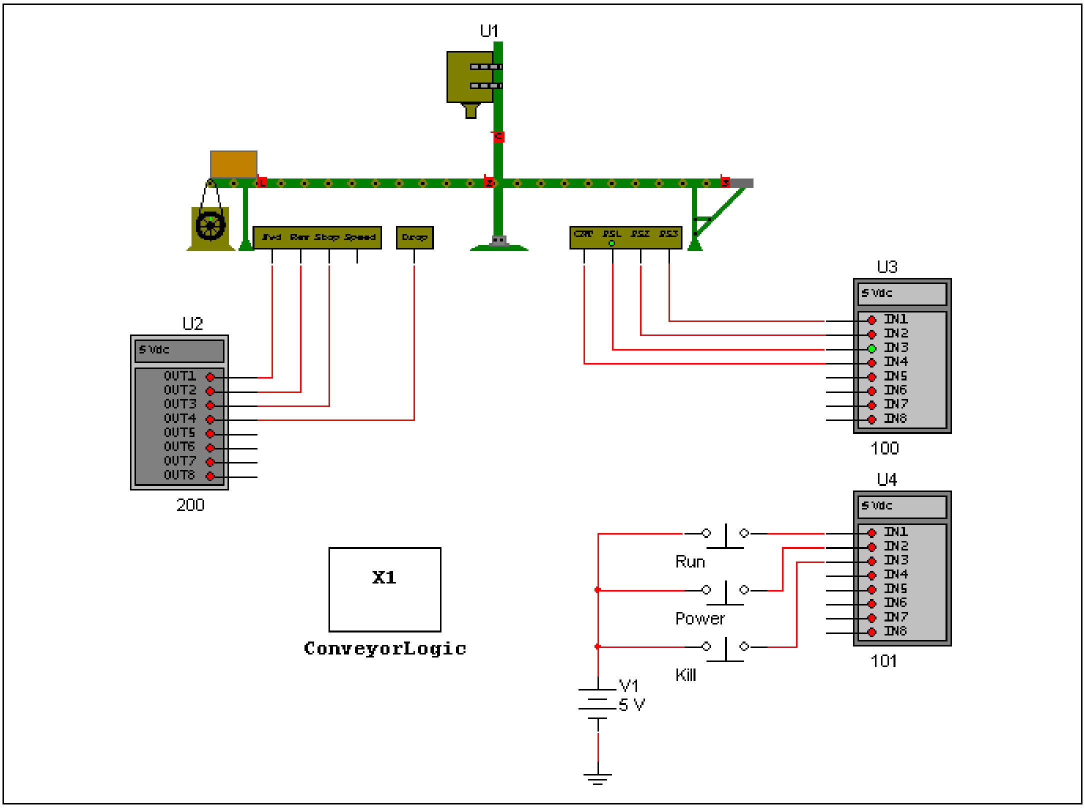
Complete the following to turn off the power at any point in the simulation:

1. Press <K> on your keyboard (or hover your cursor over the Kill switch and click the button that pops up) to activate the Kill temporary switch. This sends 5 V to pin IN3 of Input Module U2 (**Input Module Base Address** = 100) which in turn energizes Input Contact X2 (the contact opens). The continuity in the Power Lock-up Rung is broken and Relay Coil M1 is de-energized, which in turn switches off all Relay Contacts with **Relay Device Reference** = M1.

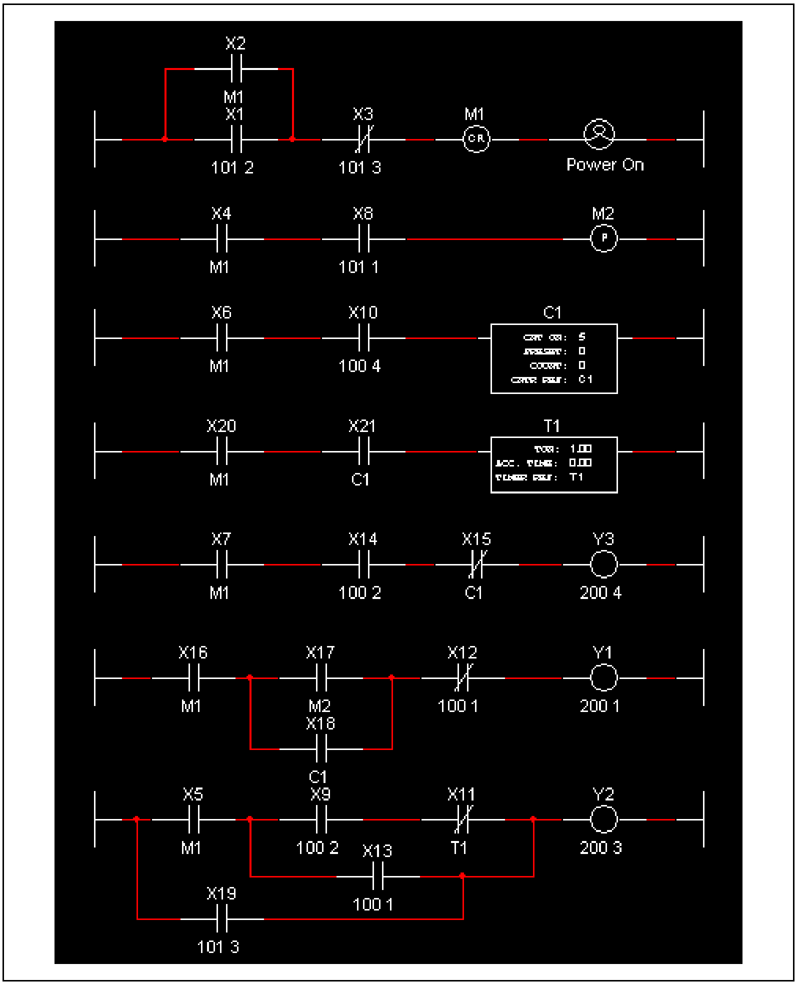
   When you press <K>, X20 is also temporarily energized, which in turn temporarily energizes Output Coil Y2, which sends a pulse to pin Out3 of Output Module U3. This is wired to the **Stop** pin of the holding tank, so the tank stops filling or emptying (depending on which is currently occuring).

Conveyor Belt

This section contains an example of a **Ladder Diagram** that drives a conveyor belt.



Note Refer to the *Component Reference* help file for information about the user-settable parameters for the Conveyor Belt, Input Module and Output Module.

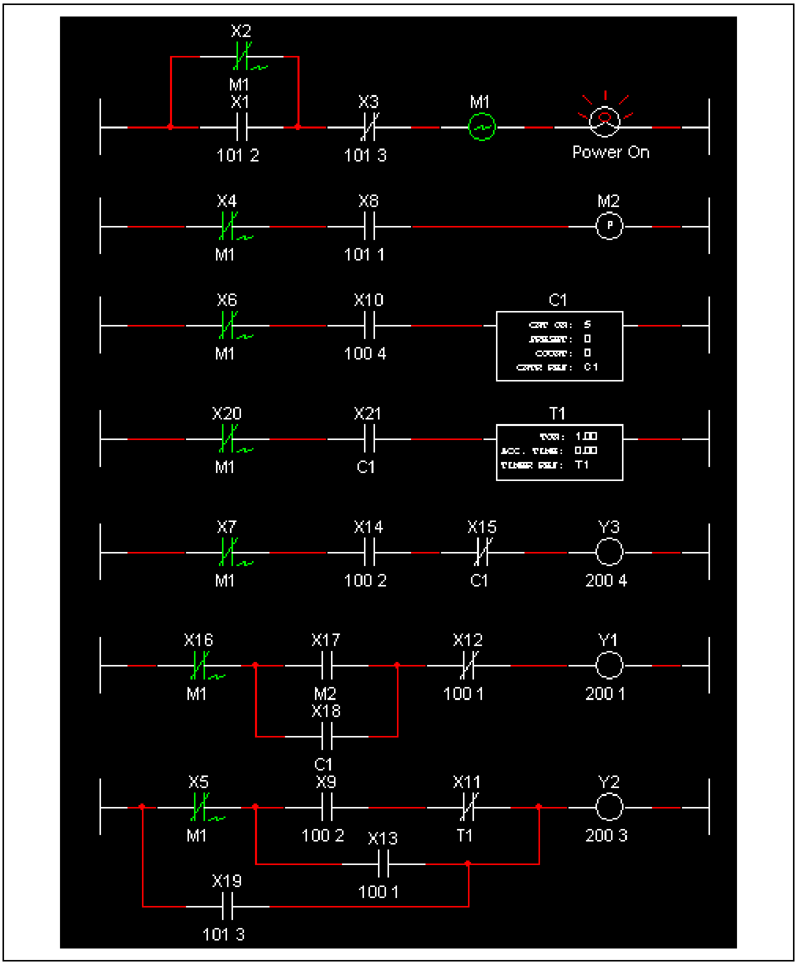


Note The ladder diagram is contained in a separate hierarchical block called `ConveyorLogic`. For details on hierarchical blocks, refer to the *Multisim User Manual*.

Complete the following steps to activate this circuit:

1. Select **Simulate»Run** to begin simulation.
2. Press <P> on your keyboard (or hover your cursor over the Power switch and click the button that pops up) to activate the Power temporary switch. This sends 5 V to pin IN2 of Input Module U4 (**Input Module Base Address** = 101) which in turn energizes Input Contact X1 in the Power Lock-up Rung of the ladder diagram. Relay

Coil M1 is energized, causing all Relay Contacts with **Relay Device Reference** = M1 to energize.

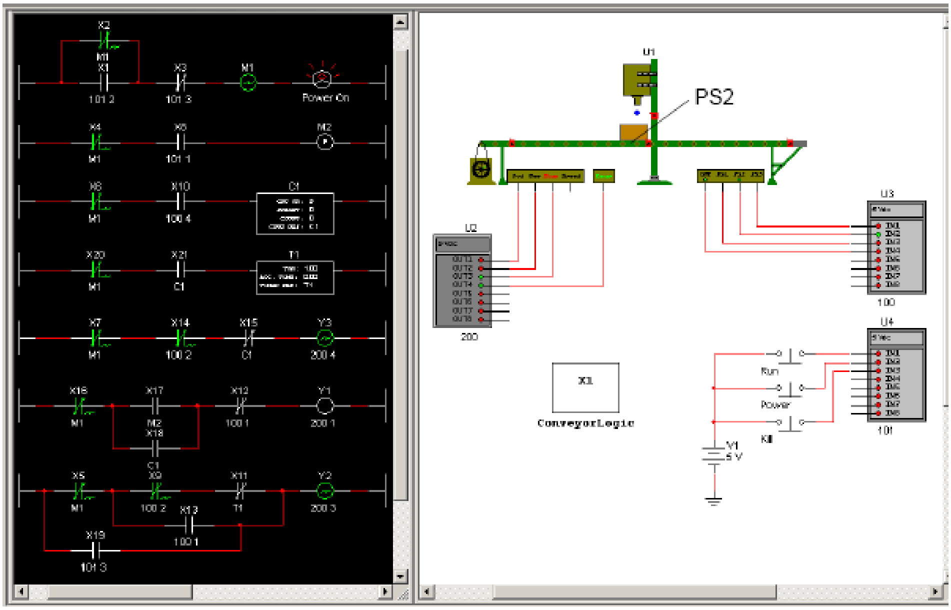


Complete the following steps to run the conveyor belt:

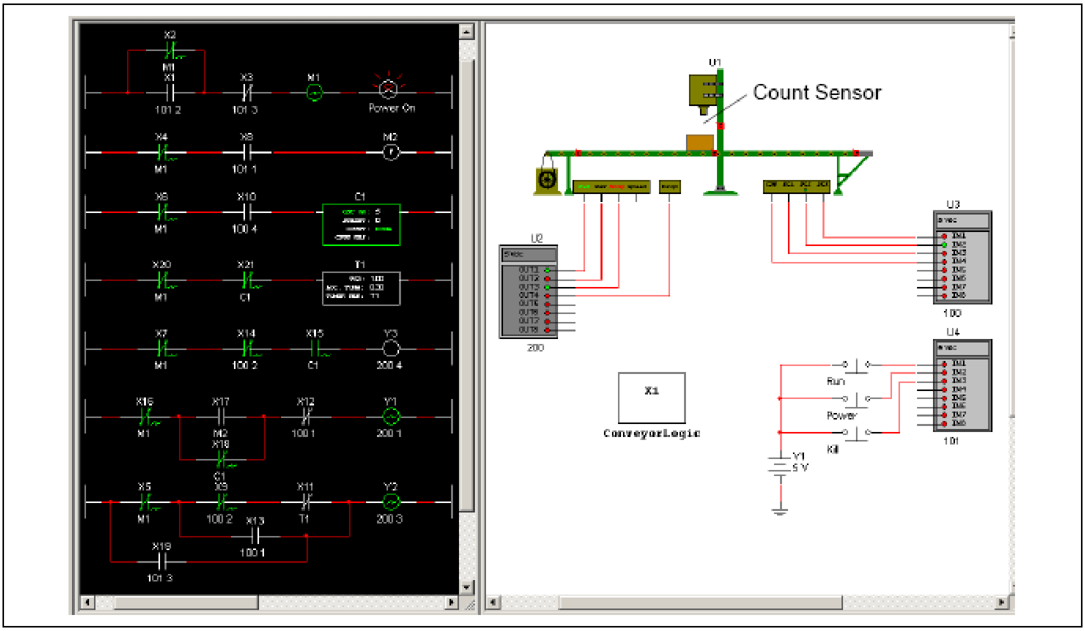
1. Activate the circuit as described earlier.
2. Press <R> on your keyboard (or hover your cursor over the Run switch and click the button that pops up) to activate the Run temporary switch.

Tip Select **Window»Tile Vertical** to view the ladder diagram and the circuit at the same time. Observe the interaction between the ladder diagram and the circuit as the simulation proceeds.

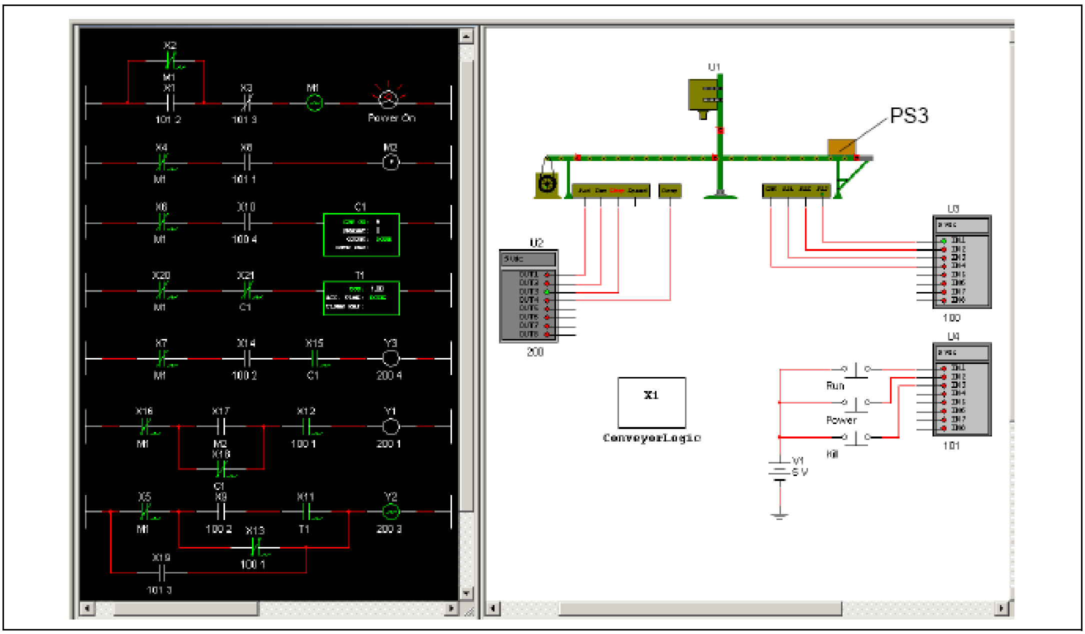
3. As the simulation proceeds, the box moves along the conveyor belt to Position Sensor 2 (PS2). The box stops moving and balls begin dropping from the hopper into the box.



4. When five balls have dropped into the box (counted by Count sensor and C1), the hopper stops dropping balls.



5. The conveyor continues moving and stops when the box gets to Position Sensor 3 (PS3).



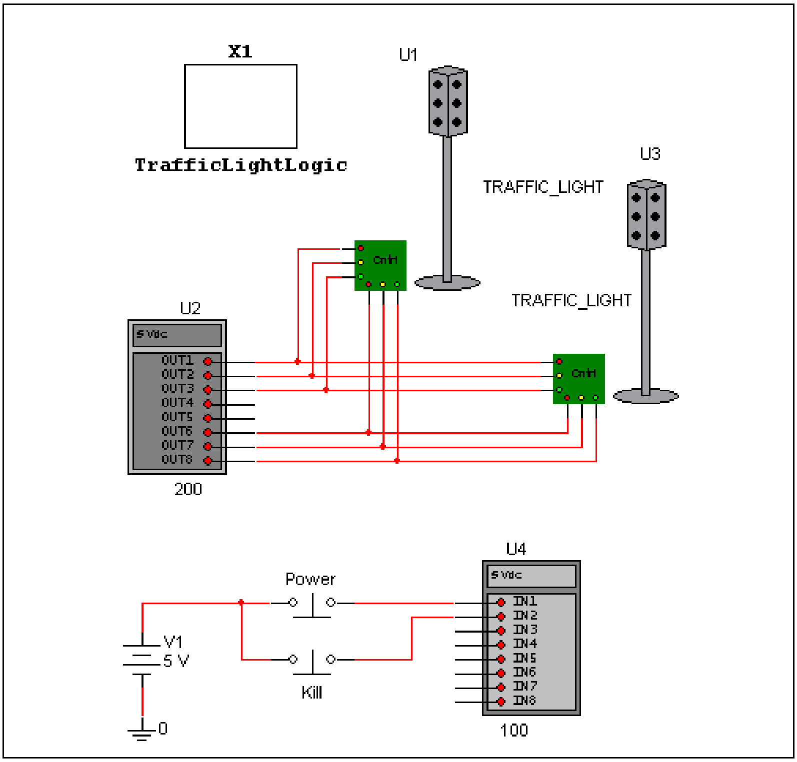
Complete the following to turn off the power at any point in the simulation:

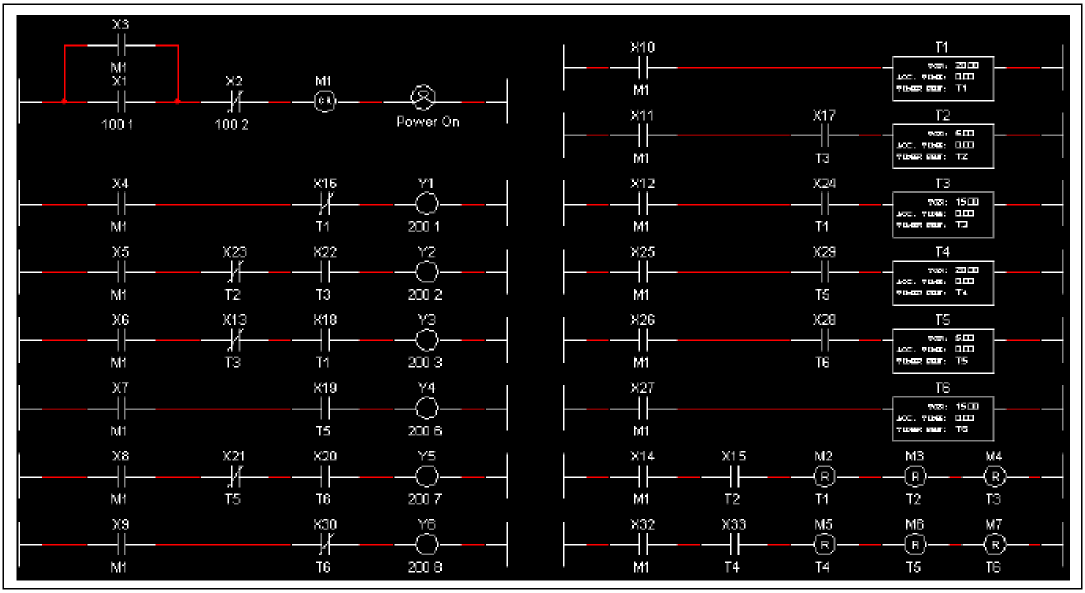
1. Press <K> on your keyboard (or hover your cursor over the Kill switch and click the button that pops up) to activate the Kill temporary switch. This sends 5 V to pin IN3 of Input Module U4 (**Input Module Base Address** = 101) which in turn energizes Input Contact X3 (the contact opens). The continuity in the Power Lock-up Rung is broken and Relay Coil M1 is de-energized, which in turn switches off all Relay Contacts with **Relay Device Reference** = M1.

   When you press <K>, X19 is also temporarily energized, which in turn temporarily energizes Output Coil Y2, which sends a pulse to pin Out3 of Output Module U2. This is wired to the **Stop** pin of the conveyor belt, so the belt stops.

Traffic Light

The ladder diagram in this section runs two traffic lights.





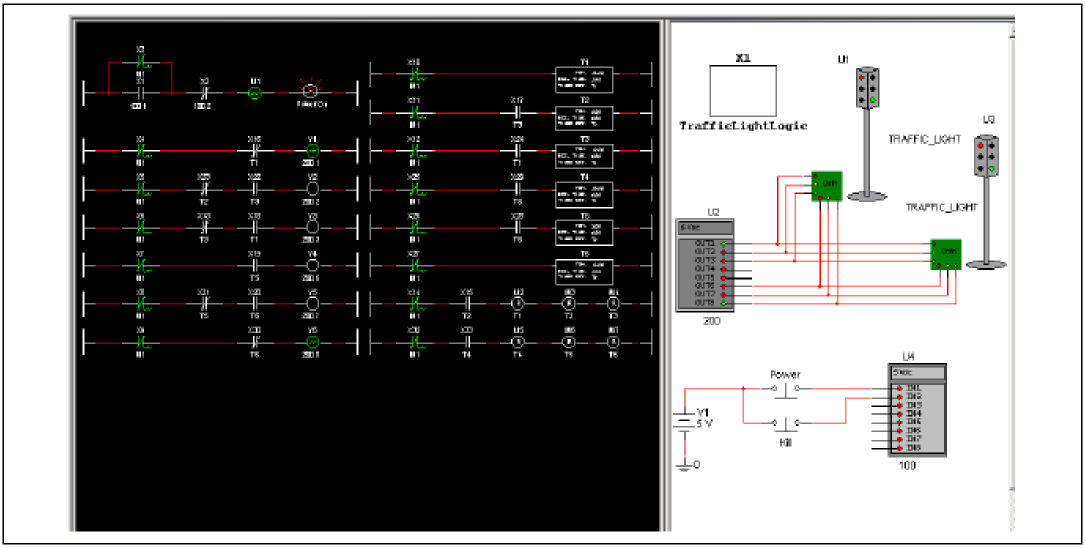
Note The ladder diagram is contained in a separate hierarchical block called `TrafficLightLogic`. For details on hierarchical blocks, refer to the *Multisim User Manual*.

Complete the following steps to run the traffic lights:

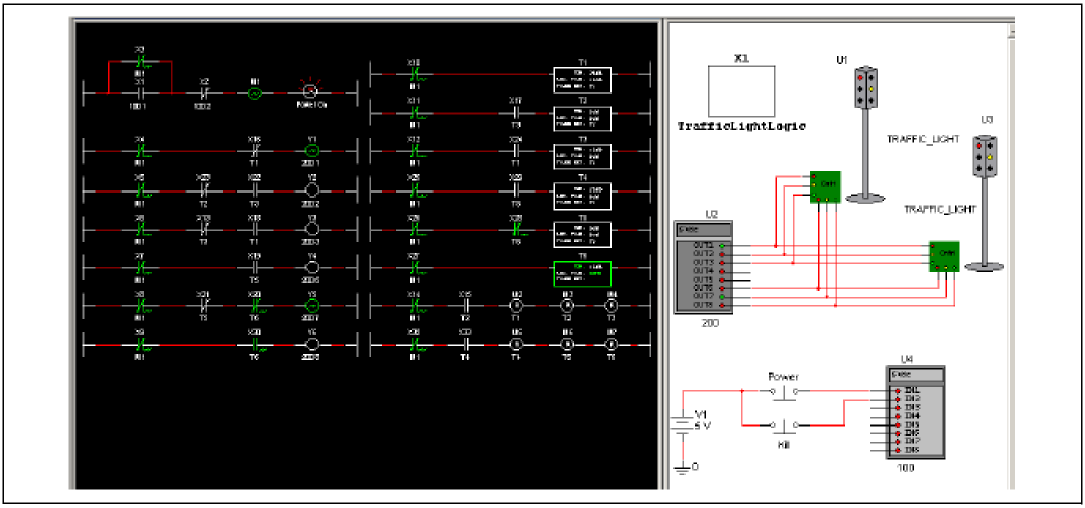
1. Select **Simulate»Run**.
2. Press <P> on your keyboard (or hover your cursor over the Power switch and click the button that pops up) to activate the Power momentary switch.

Tip Select **Window»Tile Vertical** to view the ladder diagram and the circuit at the same time. Observe the interaction between the ladder diagram and the circuit as the simulation proceeds.

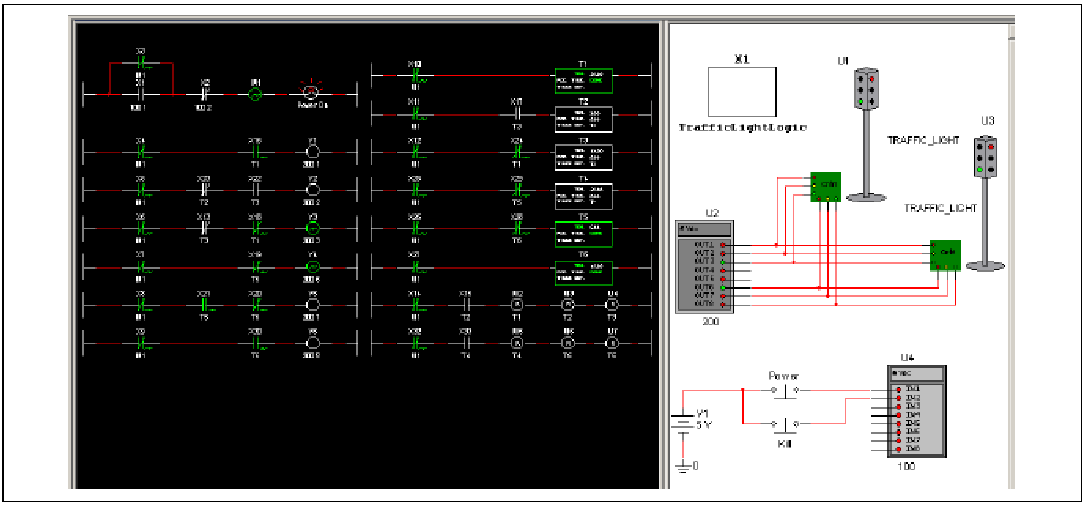
3. The red and green lights in traffic lights U1 and U3 light as shown below.



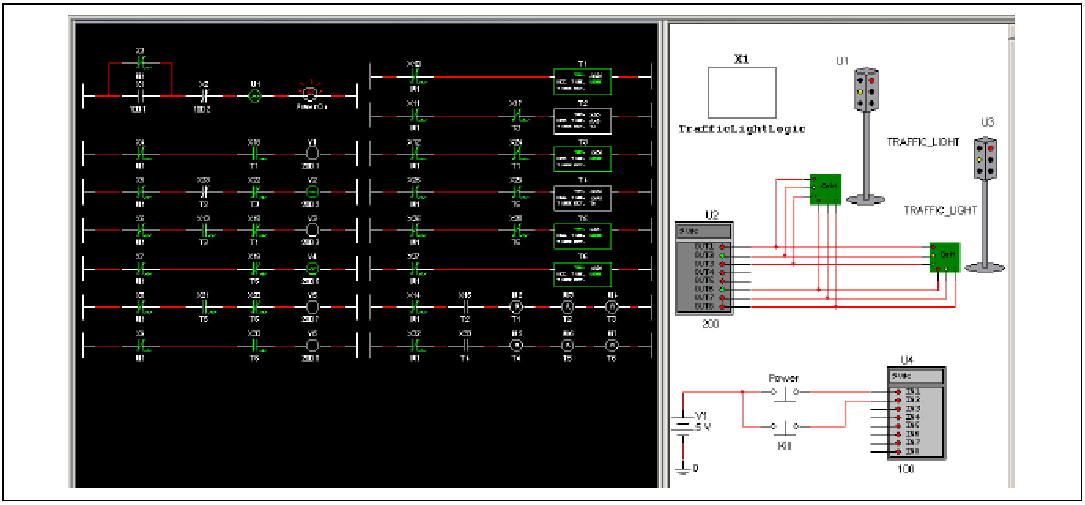
4. After 15 seconds, the green lights turn amber.



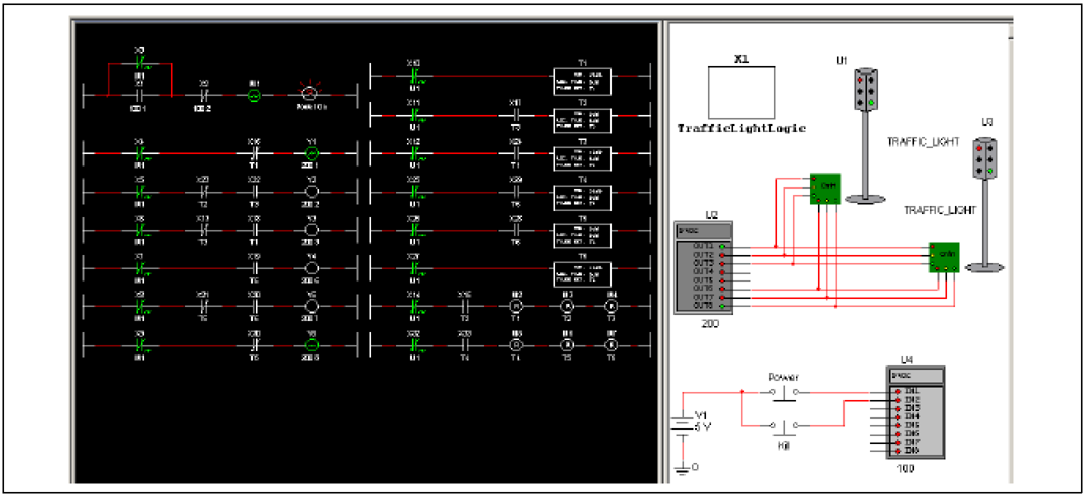
5. After 5 more seconds, the amber lights turn red and the red lights turn green.



6. After 15 seconds, the green lights turn amber.



7. After 5 more seconds, the amber lights turn red and the red lights turn green.



8. The cycle continues in this way until you stop the simulation, or press <K> (or hover your cursor over the Kill switch and click the button that pops up) to activate the Kill momentary switch.

A

Technical Support and Professional Services

Visit the following sections of the award-winning National Instruments Web site at `ni.com` for technical support and professional services:

- **Support**—Technical support resources at `ni.com/support` include the following:
  - **Self-Help Technical Resources**—For answers and solutions, visit `ni.com/support` for software drivers and updates, a searchable KnowledgeBase, product manuals, step-by-step troubleshooting wizards, thousands of example programs, tutorials, application notes, instrument drivers, and so on. Registered users also receive access to the NI Discussion Forums at `ni.com/forums`. NI Applications Engineers make sure every question submitted online receives an answer.
  - **Standard Service Program Membership**—This program entitles members to direct access to NI Applications Engineers via phone and email for one-to-one technical support as well as exclusive access to on demand training modules via the Services Resource Center. NI offers complementary membership for a full year after purchase, after which you may renew to continue your benefits.

    For information about other technical support options in your area, visit `ni.com/services`, or contact your local office at `ni.com/contact`.
- **Training and Certification**—Visit `ni.com/training` for self-paced training, eLearning virtual classrooms, interactive CDs, and Certification program information. You also can register for instructor-led, hands-on courses at locations around the world.
- **System Integration**—If you have time constraints, limited in-house technical resources, or other project challenges, National Instruments Alliance Partner members can help. To learn more, call your local NI office or visit `ni.com/alliance`.

If you searched `ni.com` and could not find the answers you need, contact your local office or NI corporate headquarters. Phone numbers for our worldwide offices are listed at the front of this manual. You also can visit the Worldwide Offices section of `ni.com/niglobal` to access the branch office Web sites, which provide up-to-date contact information, support phone numbers, email addresses, and current events.

Index

T

V

W