

Multisim™

Multisim For Educators

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 0 662 45 79 90 0, Belgium 32 0 2 757 00 20, Brazil 55 11 3262 3599, Canada 800 433 3488, China 86 21 6555 7838, Czech Republic 420 224 235 774, Denmark 45 45 76 26 00, Finland 385 0 9 725 725 11, France 33 0 1 48 14 24 24, Germany 49 0 89 741 31 30, India 91 80 41190000, Israel 972 0 3 6393737, Italy 39 02 413091, Japan 81 3 5472 2970, 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 68 51, 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 56 200 51 51, Taiwan 886 02 2377 2222, Thailand 662 278 6777, United Kingdom 44 0 1635 523545

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

© 2007 National Instruments Corporation. All rights reserved.

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 THEREFORE 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

Under the copyright laws, this publication may not be reproduced or transmitted in any form, electronic or mechanical, including photocopying, recording, storing in an information retrieval system, or translating, in whole or in part, without the prior written consent of National Instruments Corporation.

National Instruments respects the intellectual property of others, and we ask our users to do the same. NI software is protected by copyright and other intellectual property laws. Where NI software may be used to reproduce software or other materials belonging to others, you may use NI software only to reproduce materials that you may reproduce in accordance with the terms of any applicable license or other legal restriction.

BSIM3 and BSIM4 are developed by the Device Research Group of the Department of Electrical Engineering and Computer Science, University of California, Berkeley and copyrighted by the University of California.

Trademarks

National Instruments, NI, ni.com, and LabVIEW are trademarks of National Instruments Corporation. Refer to the *Terms of Use* section on ni.com/legal for more information about National Instruments trademarks. Ultiboard is a registered trademark and Multisim and Electronics Workbench are trademarks of Electronics Workbench. Other product and company names mentioned herein are trademarks or trade names of their respective companies.

Members of the National Instruments Alliance Partner Program are business entities independent from National Instruments and have no agency, partnership, or joint-venture relationship with National Instruments.

Patents

For patents covering National Instruments products, refer to ni.com/patents.

Some portions of this product are protected under United States Patent No. 6,560,572.

WARNING REGARDING USE OF NATIONAL INSTRUMENTS PRODUCTS

(1) NATIONAL INSTRUMENTS PRODUCTS ARE NOT DESIGNED WITH COMPONENTS AND TESTING FOR A LEVEL OF RELIABILITY SUITABLE FOR USE IN OR IN CONNECTION WITH SURGICAL IMPLANTS OR AS CRITICAL COMPONENTS IN ANY LIFE SUPPORT SYSTEMS WHOSE FAILURE TO PERFORM CAN REASONABLY BE EXPECTED TO CAUSE SIGNIFICANT INJURY TO A HUMAN.

(2) IN ANY APPLICATION, INCLUDING THE ABOVE, RELIABILITY OF OPERATION OF THE SOFTWARE PRODUCTS CAN BE IMPAIRED BY ADVERSE FACTORS, INCLUDING BUT NOT LIMITED TO FLUCTUATIONS IN ELECTRICAL POWER SUPPLY, COMPUTER HARDWARE MALFUNCTIONS, COMPUTER OPERATING SYSTEM SOFTWARE FITNESS, FITNESS OF COMPILERS AND DEVELOPMENT SOFTWARE USED TO DEVELOP AN APPLICATION, INSTALLATION ERRORS, SOFTWARE AND HARDWARE COMPATIBILITY PROBLEMS, MALFUNCTIONS OR FAILURES OF ELECTRONIC MONITORING OR CONTROL DEVICES, TRANSIENT FAILURES OF ELECTRONIC SYSTEMS (HARDWARE AND/OR SOFTWARE), UNANTICIPATED USES OR MISUSES, OR ERRORS ON THE PART OF THE USER OR APPLICATIONS DESIGNER (ADVERSE FACTORS SUCH AS THESE ARE HEREAFTER COLLECTIVELY TERMED "SYSTEM FAILURES"). ANY APPLICATION WHERE A SYSTEM FAILURE WOULD CREATE A RISK OF HARM TO PROPERTY OR PERSONS (INCLUDING THE RISK OF BODILY INJURY AND DEATH) SHOULD NOT BE RELIANT SOLELY UPON ONE FORM OF ELECTRONIC SYSTEM DUE TO THE RISK OF SYSTEM FAILURE. TO AVOID DAMAGE, INJURY, OR DEATH, THE USER OR APPLICATION DESIGNER MUST TAKE REASONABLY PRUDENT STEPS TO PROTECT AGAINST SYSTEM FAILURES, INCLUDING BUT NOT LIMITED TO BACK-UP OR SHUT DOWN MECHANISMS. BECAUSE EACH END-USER SYSTEM IS CUSTOMIZED AND DIFFERS FROM NATIONAL INSTRUMENTS' TESTING PLATFORMS AND BECAUSE A USER OR APPLICATION DESIGNER MAY USE NATIONAL INSTRUMENTS PRODUCTS IN COMBINATION WITH OTHER PRODUCTS IN A MANNER NOT EVALUATED OR CONTEMPLATED BY NATIONAL INSTRUMENTS, THE USER OR APPLICATION DESIGNER IS ULTIMATELY RESPONSIBLE FOR VERIFYING AND VALIDATING THE SUITABILITY OF NATIONAL INSTRUMENTS PRODUCTS WHENEVER NATIONAL INSTRUMENTS PRODUCTS ARE INCORPORATED IN A SYSTEM OR APPLICATION, INCLUDING, WITHOUT LIMITATION, THE APPROPRIATE DESIGN, PROCESS AND SAFETY LEVEL OF SUCH SYSTEM OR APPLICATION.

Documentation Conventions

When Multisim guides refer to a toolbar button, an image of the button appears in the left column.

Multisim guides use the convention **Menu/Item** to indicate menu commands. For example, “**File/Open**” means choose the **Open** command from the **File** menu.

Multisim guides use the convention of an arrow (➤) to indicate the start of procedural information.

Multisim guides use the construction CTRL-KEY and ALT-KEY to indicate when you need to hold down the “Ctrl” or “Alt” key on your keyboard and press another key.

The Multisim Documentation Set

Multisim documentation consists of a *User Guide*, the *Component Reference Guide* and online help. All Multisim users receive PDF versions of the *User Guide* and the *Component Reference Guide*.

The *Multisim for Educators Guide* describes the functions specific to Education edition of Multisim.

You should also refer to *Getting Started with NI Circuit Design Suite*.

User Guide

The *User Guide* describes Multisim and its many functions in detail. It is organized based on the stages of circuit design, and explains all aspects of Multisim, in detail.

Online Help

Multisim offers a full helpfile system to support your use of the product.

Choose **Help/Multisim Help** to display the helpfile that explains the Multisim program in detail, or choose **Help/Component Reference** to display the helpfile that contains details on the components families provided with Multisim. Both are compiled HTML helpfiles, offering a table of contents and index.

In addition, you can display context-sensitive help by pressing F1 from any command or window, or by clicking the **Help** button on any dialog box that offers it.

Table of Contents

1. Educators' Guide

1.1	Circuit Creator's Name	1-1
1.2	Using Restrictions	1-2
1.2.1	Setting Global Restrictions	1-2
1.2.1.1	General Global Restrictions	1-2
1.2.1.2	Global Analyses Restrictions	1-4
1.2.2	Setting Circuit Restrictions	1-4
1.2.3	Setting Passwords for Restrictions	1-6
1.3	Link to Education Resources	1-6

2. Breadboarding

2.1	Breadboarding Overview	2-2
2.2	Breadboard Settings	2-2
2.3	3D Options	2-3
2.4	Placing Components on the Breadboard	2-3
2.5	Wiring Placed Components	2-6
2.5.1	Placing a Jumper	2-7
2.5.2	Changing Jumper Wire Color	2-8
2.6	Viewing Component Information	2-8
2.7	Manipulating the Breadboard View	2-10
2.8	Breadboard Netlist dialog box	2-11
2.9	DRC and Connectivity Check	2-12

3. Virtual NI ELVIS

3.1	Overview	3-1
3.2	The Virtual NI ELVIS Schematic	3-2
3.3	Placing Components on the Prototyping Board	3-8
3.4	Wiring Placed Components	3-10

4. Ladder Diagrams

4.1	Overview	4-1
4.2	Creating a Ladder Diagram	4-2
4.3	AND Rungs and OR Rungs	4-5
4.4	Sample Circuits	4-7
4.4.1	Holding Tank	4-7
4.4.2	Conveyor Belt	4-12
4.4.3	Traffic Light	4-17

Appendix A - Education Edition Parts

A.1	3D Virtual Parts	A-1
A.1.1	3D 555 Timer	A-1
A.1.2	3D BJT	A-2
A.1.3	3D Capacitors	A-2
A.1.4	3D 74LS160N Counter	A-3
A.1.5	3D Diode	A-3
A.1.6	3D Inductor	A-4
A.1.7	3D LED	A-4
A.1.8	3D MOSFET	A-5
A.1.9	3D DC Motor	A-5
A.1.10	3D Opamp	A-5
A.1.11	3D Potentiometer	A-6
A.1.12	3D AND Gate	A-6
A.1.13	3D Resistor	A-7
A.1.14	3D Shift Register	A-7
A.1.15	3D Switch	A-7
A.1.16	3D Components Toolbar	A-8
A.2	Ladder Diagram Parts	A-8
A.2.1	Ladder Rungs	A-8
A.2.1.1	L1 and L2	A-8
A.2.2	Ladder I/O Modules	A-9
A.2.2.1	Input Module	A-9
A.2.2.2	Output Module	A-9
A.2.3	Ladder Relay Coils	A-10
A.2.3.1	Relay Coil	A-10
A.2.3.2	Negated Relay Coil	A-10

A.2.3.3	Set Coil	A-10
A.2.3.4	Reset Coil	A-11
A.2.3.5	Pulsed Relay Coil	A-11
A.2.4	Ladder Contacts	A-11
A.2.4.1	Input Contact NC	A-11
A.2.4.2	Input Contact NO	A-12
A.2.4.3	Relay Contact NC	A-12
A.2.4.4	Relay Contact NO	A-13
A.2.5	Ladder Counters	A-13
A.2.5.1	Count Off	A-13
A.2.5.2	Count Off Hold	A-14
A.2.5.3	Count Off Reset	A-14
A.2.5.4	Count Off Up Down	A-15
A.2.5.5	Count On	A-15
A.2.5.6	Count On Hold	A-16
A.2.5.7	Count On Reset	A-16
A.2.5.8	Count On Up Down	A-17
A.2.6	Ladder Timers	A-17
A.2.6.1	Timer TOFF	A-17
A.2.6.2	Timer TON	A-18
A.2.6.3	Timer TON Retention	A-18
A.2.6.4	Timer TON Retention Reset	A-19
A.2.6.5	Timer TON Retention Hold Reset	A-19
A.2.7	Ladder Output Coils	A-20
A.2.7.1	Output Coil	A-20
A.2.7.2	Output Coil Negated	A-20
A.3	Miscellaneous Peripherals	A-21
A.3.1	Holding Tank	A-21
A.3.2	Conveyor Belt	A-22
A.3.3	Traffic Light	A-23
A.3.4	Single Traffic Light	A-23
A.4	Virtual NI ELVIS Components	A-24
A.4.1	NI ELVIS Function Generator	A-24
A.4.2	NI ELVIS Power Supply	A-25

Appendix B - Support and Services

B.1	Technical Support and Professional Services	B-1
-----	---	-----

Chapter 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 many features that have made Multisim a favorite with professional users, there are also a number of education-specific features that are outlined in this guide.

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 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.

The following are described in this chapter.

Subject	Page No.
Circuit Creator's Name	1-1
Using Restrictions	1-2
Link to Education Resources	1-6

1.1 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 on the **Circuit Restrictions** dialog box, which you can view as long as no passwords have been set — see “1.2.2 Setting Circuit Restrictions” on page 1-4 for more information.

1.2 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.

1.2.1 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. See “Simplified Version” on page 1-3.

Note Global restrictions are overridden by circuit restrictions if the circuit restrictions are saved with the circuit. See “1.2.2 Setting Circuit Restrictions” on page 1-4 for information.

1.2.1.1 General Global Restrictions

➤ 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. For details on the **Circuit restrictions** dialog box, see “1.2.2 Setting Circuit Restrictions” on page 1-4.

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. (See “1.2.3 Setting Passwords for Restrictions” on page 1-6 for more information).

3. Click the **General** tab and select from the following options:
 - **Circuit Path** — Sets the default path and location where students find and save files.
 - **Disable database editing** — Ensures that students cannot edit components in the database.
 - **Disable Instruments toolbar** — Makes instruments unavailable to be placed in the circuit.
 - **Disable In-Use List toolbar** — Hides the In Use List.
 - **Disable Master DB component access** — Hides the Multisim Master 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.
 - **Disable Corporate DB component access** — Hides the corporate database and parts groups and families from the interface.
4. Click **OK**. Your options are immediately set for all circuits, unless you have set circuit restrictions. (See “1.2.2 Setting Circuit Restrictions” on page 1-4.)

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.

- 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. (See “1.2.2 Setting Circuit Restrictions” on page 1-4.)

1.2.1.2 Global Analyses Restrictions

- 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 Guide* for more information on analyses.

These options are immediately set for all circuits, unless you have set circuit restrictions. (See “1.2.2 Setting Circuit Restrictions” on page 1-4.)

1.2.2 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 (see “1.2.1 Setting Global Restrictions” on page 1-2 for details). 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.

- To set general circuit restrictions:
 1. Choose **Options/Circuit Restrictions**. If you have created a password, you will be prompted for it. (See “1.2.3 Setting Passwords for Restrictions” on page 1-6 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.

➤ 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 Guide* for more information on analyses.

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

➤ 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".

1.2.3 Setting Passwords for Restrictions

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

➤ To create/change a password:

1. For global restrictions, choose **Options/Global restrictions**. For circuit restrictions, choose **Options/Circuit restrictions**. 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.

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

1.3 Link to Education Resources

Note This function is hidden when the simplified version option is selected. For details, see “Simplified Version” on page 1-3.

➤ To go to the Electronics Workbench Education website:



1. Click on the **Educational Website** button or select **Tools/Education Webpage**.

Chapter 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.

The following are described in this chapter.

Subject	Page No.
Breadboarding Overview	2-2
Breadboard Settings	2-2
3D Options	2-3
Placing Components on the Breadboard	2-3
Wiring Placed Components	2-6
Placing a Jumper	2-7
Changing Jumper Wire Color	2-8
Viewing Component Information	2-8
Manipulating the Breadboard View	2-10
Breadboard Netlist dialog box	2-11
DRC and Connectivity Check	2-12

2.1 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.

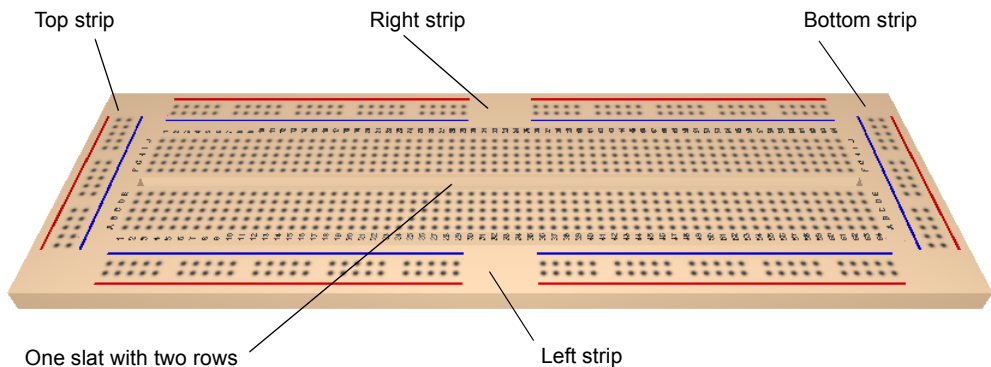
2.2 Breadboard Settings

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

- 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.



The default breadboard contains: one slat with two rows; one left strip; one right strip; one bottom strip; one top strip.



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.

2.3 3D Options

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

- To change the 3D options:
 1. Select **Options/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. (For details, see “2.5.1 Placing a Jumper” on page 2-7).
 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.

2.4 Placing Components on the Breadboard

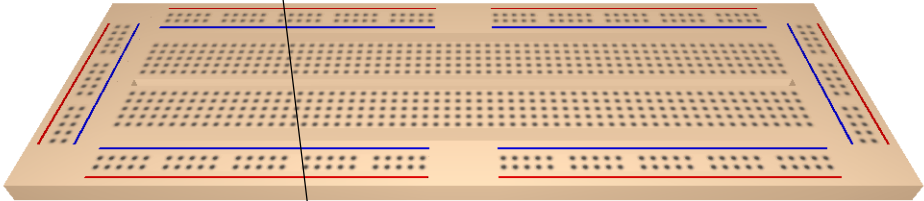
- To place components on a breadboard:
 1. Create a schematic diagram of the desired circuit in the usual manner. (For details, on schematic capture, refer to the *Multisim User Guide*).



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

Hover the cursor over a component to see its description in this box.

Refdes : U2
Value : 74HC190N_4V
Footprint : DIP16

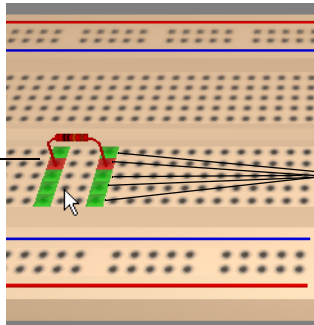


Place Component Bar - components waiting to be placed on the breadboard appear in this area

View other components by clicking on these arrows.

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.

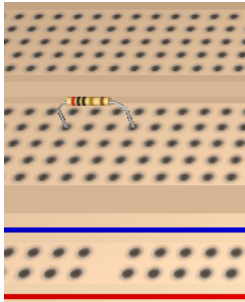
Red sockets indicate where the component's pins will be placed when the mouse button is released.



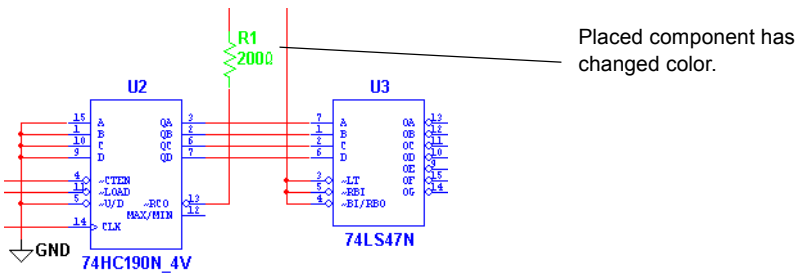
All of these sockets are connected. Green indicates sockets that are internally connected to the red socket in the same row on the breadboard.

Tip Select CTRL-R to rotate a selected component 90 degrees clockwise or CTRL-SHIFT-R to rotate it 90 degrees counter-clockwise.

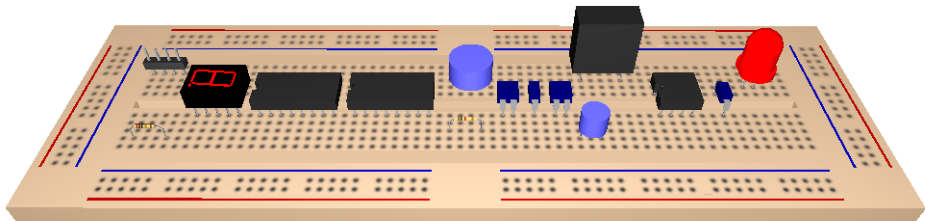
- Release the mouse button to place the component. Notice that the colored (red and green) sockets on the breadboard no longer appear.



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



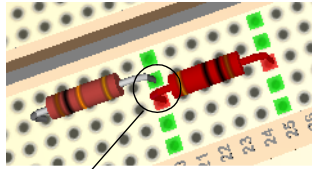
- Continue placing the circuit's components on the breadboard. When all the components have been placed, the **Place Component Bar** collapses as shown below.



Place Component Bar collapses when all components are placed on the breadboard.



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. For details on jumpers, see “2.5.1 Placing a Jumper” on page 2-7.

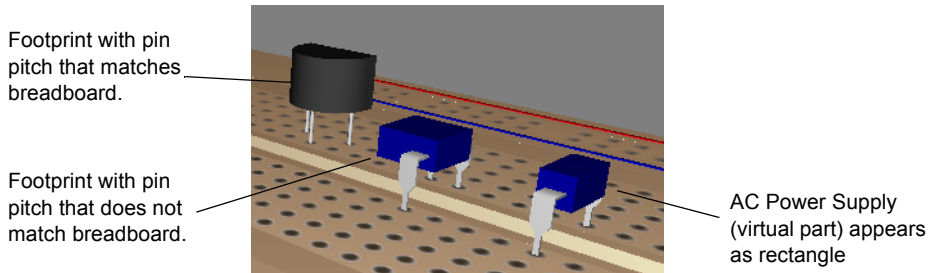


Connected pins

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.

Note To view footprint information, hover the cursor over the desired component. For details, see “2.6 Viewing Component Information” on page 2-8.

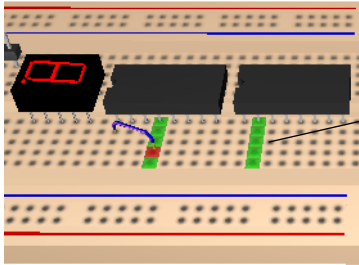
2.5 Wiring Placed Components

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.

2.5.1 Placing a Jumper

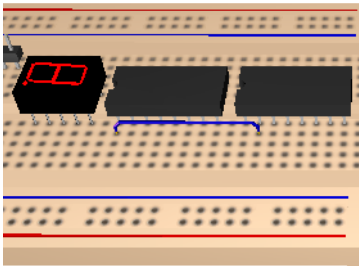
➤ To place a jumper wire:

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.

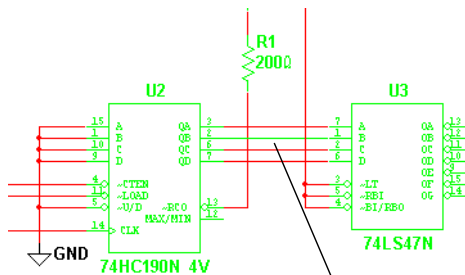


Target pins display once one end of the jumper is placed. You must move the cursor to see the targets.

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.



Green wire shows connection on breadboard

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 “2.9 DRC and Connectivity Check” on page 2-12.

2.5.2 Changing Jumper Wire Color

- To change jumper wire color:

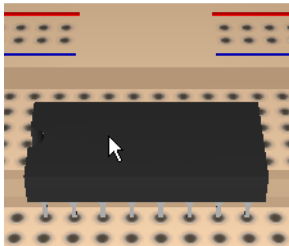
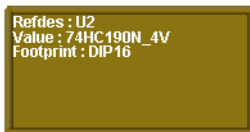


1. Select **Edit/Breadboard Wire Color**.
2. Select the desired color from the dialog box that appears.

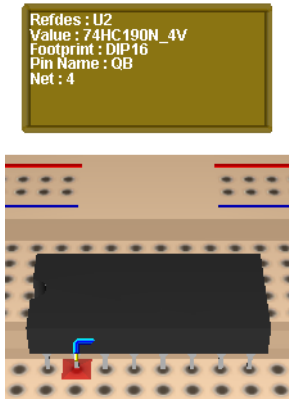
Note The color of previously placed wires is not affected. The new color will be applied to any subsequently placed wires.

2.6 Viewing Component Information

- To view information about a specific component:
1. Hover the cursor over the component. The information box is populated as shown below.



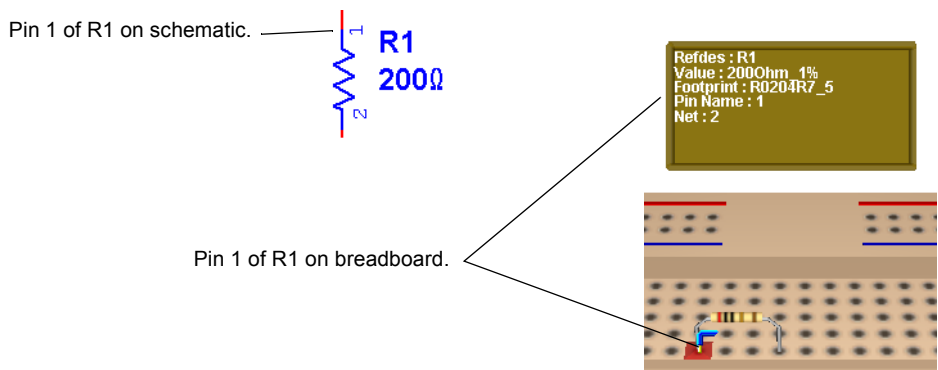
- To see pin information:
 1. Hover the cursor over the “metal” part of the desired pin. The information box now includes the pin name and the schematic net to which the pin should be connected.



Two-terminal Components

Two-terminal non-directional components like resistors have 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.

- 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 **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).

2.7 Manipulating the Breadboard View

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

- To make the breadboard appear larger:



1. Select **View/Zoom in**.

- To make the breadboard appear smaller:



1. 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 Guide*).

- To view the entire breadboard:



1. Select **View/Zoom Full**.

- To rotate the breadboard 180 degrees:



1. 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:

1. Hold down the SHIFT key on your keyboard and use any of the arrow keys.

Or

Press CTRL-SHIFT and drag the mouse.

Or

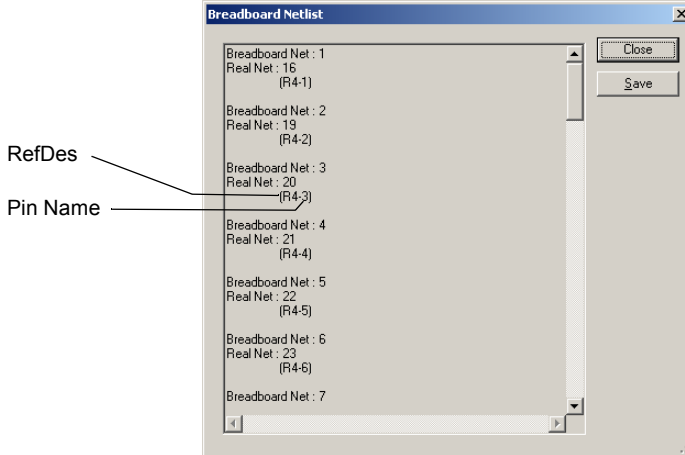
Hold down your mouse-wheel and drag the mouse.

2.8 Breadboard Netlist dialog box

- To display a netlist for the placed components and jumpers:



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



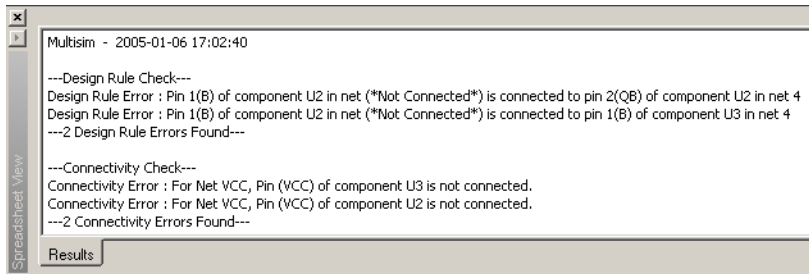
2. Optionally, click **Save** to save the breadboard netlist as a `.txt` or `.csv` file.

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

2.9 DRC and Connectivity Check

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

- To run a DRC and Connectivity Check:
 1. Select **Tools/DRC and Connectivity Check**. The results appear in the **Results** tab of the **Spreadsheet View**.



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.

Chapter 3

Virtual NI ELVIS

This chapter describes the Multisim 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.

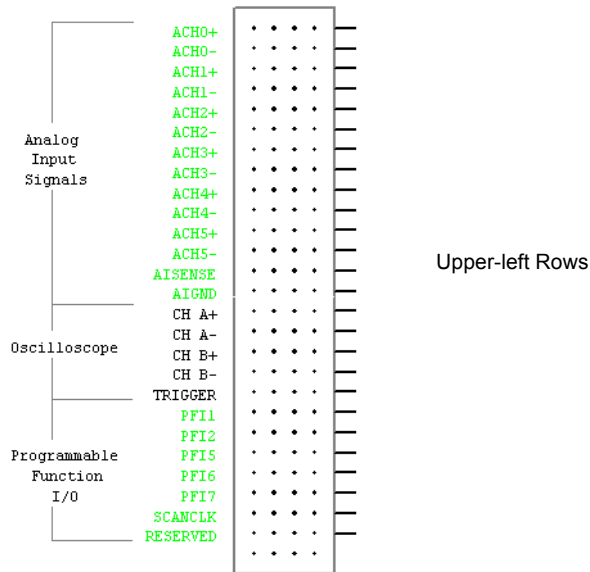
The following are described in this chapter.

Subject	Page No.
Overview	3-1
The Virtual NI ELVIS Schematic	3-2
Placing Components on the Prototyping Board	3-8
Wiring Placed Components	3-10

3.1 Overview

Electronic Workbench's 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 instructor's projects can be carried out on the student's PC before moving on to the real NI ELVIS workstation in the lab.

Oscilloscope



The connections to Virtual NI ELVIS’s oscilloscope are found in the upper-left prototyping rows.

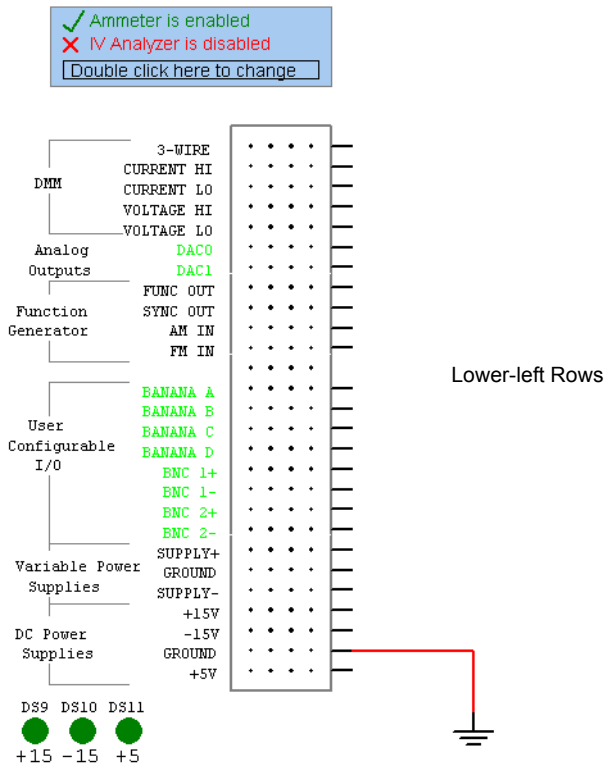
➤ To connect the oscilloscope:

- 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.

➤ To access the oscilloscope’s controls:

- Double-click on the the word “Oscilloscope” in the upper-left prototyping rows. The instrument face for Multisim’s virtual oscilloscope displays.
- Refer to the *Multisim User Guide* for details on the use of this instrument.

IV Analyzer and Multimeter



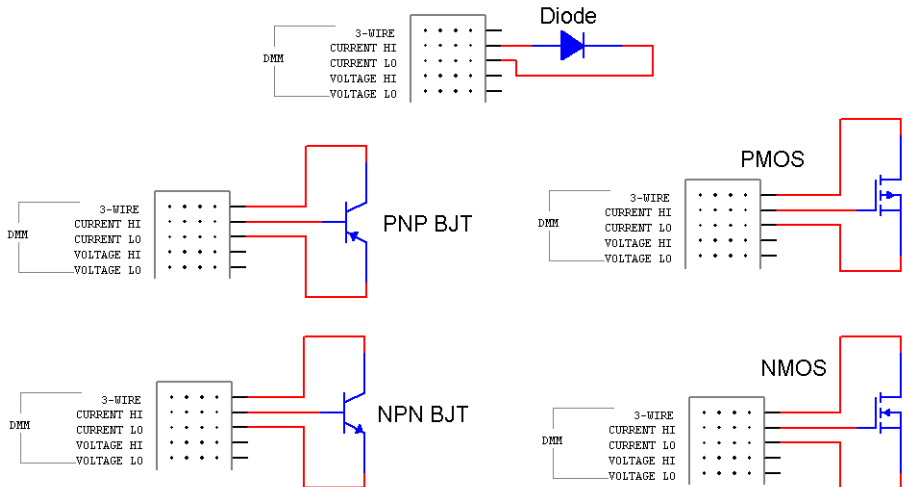
When a new Virtual NI ELVIS schematic is opened, the IV Analyzer is disabled, and the Ammeter is enabled as indicated above.

- To disable the Ammeter and enable the IV Analyzer, double-click where indicated on the Virtual NI ELVIS 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.

➤ 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. See below for connections to specific component types.



➤ 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.

➤ 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 Guide* for details on the use of these instruments.

➤ 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.

- 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 Guide* for details on the use of this instrument.

Function Generator

- 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.
- 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 For details on the user-settable parameters, see “A.4.1 NI ELVIS Function Generator” on page A-24.

Power Supplies

The lower-left prototyping rows contain the following fixed DC Power Supplies:

- +15 V
- -15 V
- +5 V (also found in the lower-right prototyping rows).

Variable Power Supplies are also available:

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

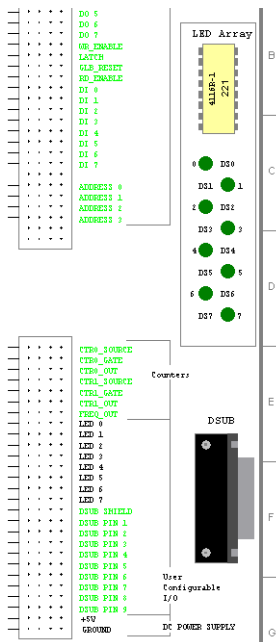
- To connect to any of the power supplies, place wires from the pin on the corresponding row to the desired point in the circuit.

- To access the controls for the variable power supplies:
 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 For details on the user-settable parameters, see “A.4.2 NI ELVIS Power Supply” on page A-25.

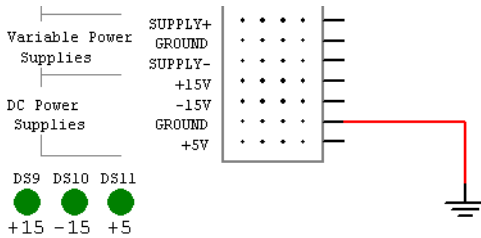
LEDs

Connections to the seven LEDs on the right side of the NI ELVIS schematic are found in the lower-right prototyping rows. During simulation, any of these LEDs that are correctly connected will light.



- 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.

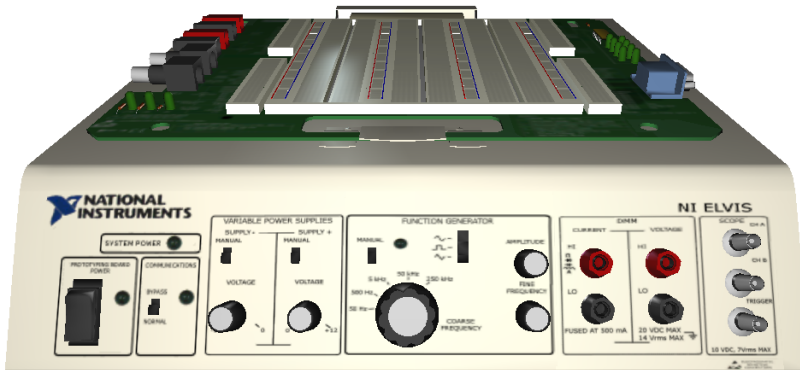
There are also three LEDs in the lower-left section of any Virtual NI ELVIS schematic.



During simulation, these LEDs will light whether or not connections have been made to their corresponding pins in the prototyping rows.

3.3 Placing Components on the Prototyping Board

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



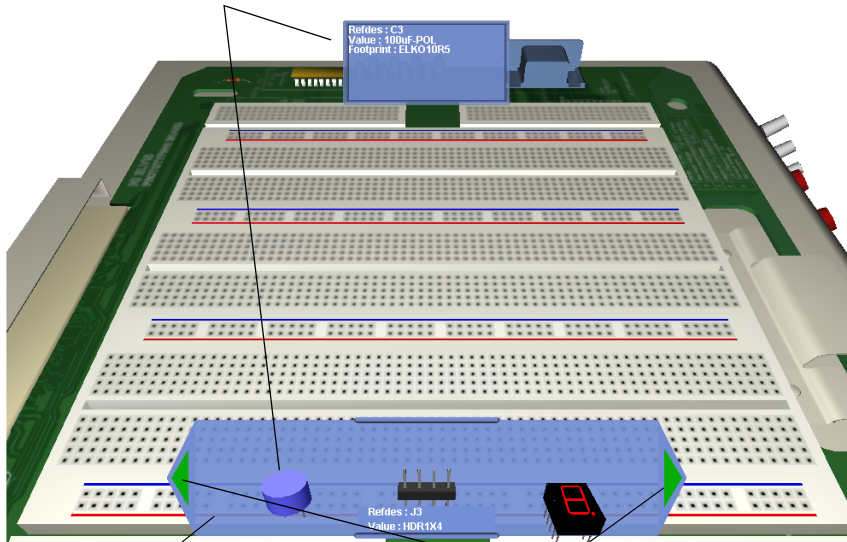
Note The controls that appear on the front of the 3D Virtual NI ELVIS are inactive. Interactive simulation of Virtual NI ELVIS is accomplished via the schematic view. For details on simulation, refer to the *Multisim User Guide* or the Multisim helpfile.

➤ To place components on the 3D prototyping board:



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

Hover the cursor over a component to see its description in this box.



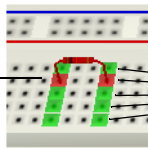
Place Component Bar - components waiting to be placed on the breadboard appear in this area

View other components by clicking on these arrows.

Note 3D viewing options are set in the **Preferences** dialog box. For details, see “2.3 3D Options” on page 2-3. See also, “2.7 Manipulating the Breadboard View” on page 2-10 for useful viewing 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 indicated below:

Red sockets indicate where the component's pins will be placed when the mouse button is released.

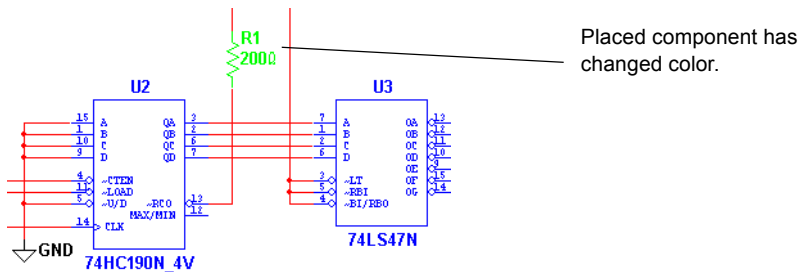


All of these sockets are connected. Green indicates sockets that are internally connected to the red socket in the same row on the board.

Tip Select CTRL-R to rotate a selected component 90 degrees clockwise or CTRL-SHIFT-R to rotate it 90 degrees counter-clockwise.

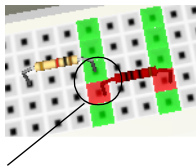
3. Release the mouse button to place the component. The colored (red and green) sockets on the board no longer appear.

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



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 shown below. This technique can reduce the number of jumper wires required.



Connected pins

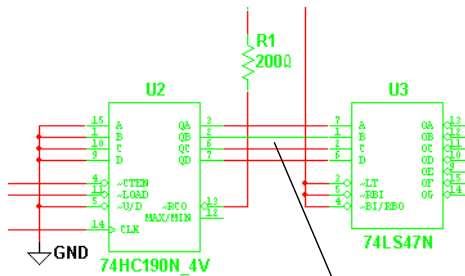
Note See also, “Appearance of 3D Components” on page 2-6.

3.4 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.

- 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.

- 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.



Green wire shows connection done in 3D view

- Note** If a net contains more than two connections, all must be connected before any of the wires in the net change color.
- 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 board. Refer to “2.9 DRC and Connectivity Check” on page 2-12.
- Note** See also “2.6 Viewing Component Information” on page 2-8 and “2.8 Breadboard Netlist dialog box” on page 2-11.

Chapter 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.

The following are described in this chapter.

Subject	Page No.
Overview	4-1
Creating a Ladder Diagram	4-2
AND Rungs and OR Rungs	4-5
Sample Circuits	4-7
Holding Tank	4-7
Conveyor Belt	4-12
Traffic Light	4-17

4.1 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.

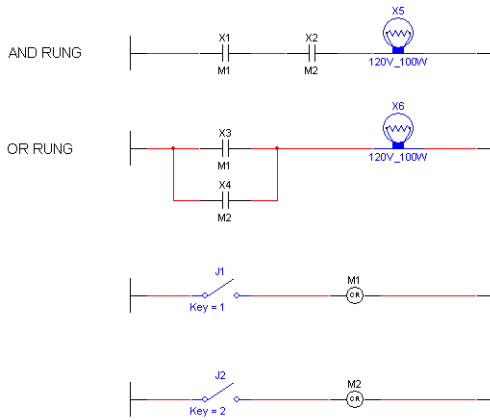
Ladder Diagrams are able to drive output devices or take input data from regular schematics and embed the instructions on how input states affect output states in either the same schematic or separate hierarchical blocks or subcircuits that contain the **Ladder Diagram**.

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

4.2 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 “4.3 AND Rungs and OR Rungs” on page 4-5.



Note For details on all ladder diagram components, refer to “A.2 Ladder Diagram Parts” on page A-8.

Circuit Notes:

- The relays (X1-X4) are normally open relays. When their controlling coils (M1 or M2) are energized they close. (The controlling coils are set in the **Value** tab of each relay’s properties dialog box. For details, see “A.2.4.4 Relay Contact NO” on page A-13).
- 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.

- 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.

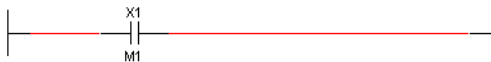


- To add components to the rungs:

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

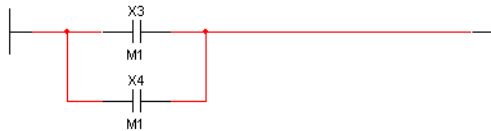
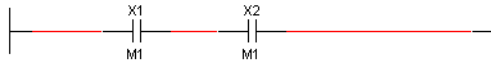
Note This device is found in the Ladder Diagrams Group - Ladder Contacts Family.

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

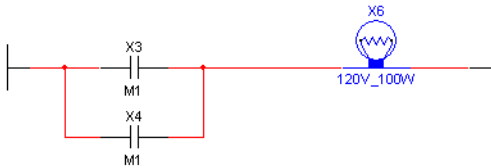
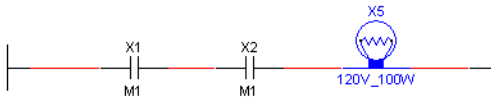


Ladder Diagrams

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).

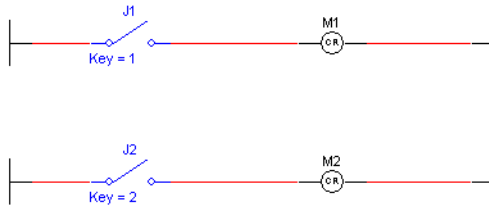


5. Place relay coils M1 and M2 on the third and fourth rungs (**Group** - Ladder Diagrams; **Family** - Ladder Relay Coils).

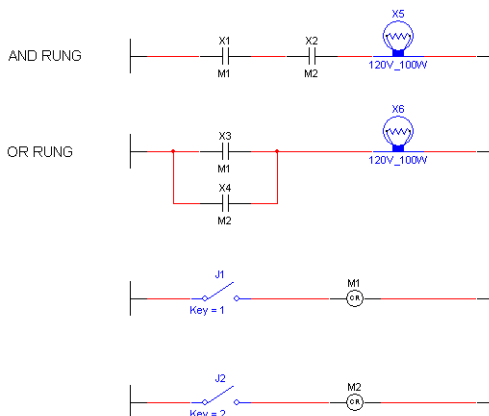


6. Place switches J1 and J2.

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.



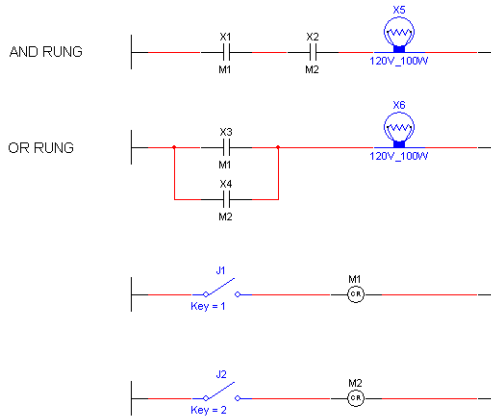
- To change the controlling device reference for X2 and X4:
 1. Double-click on X2 and click the **Value** tab.
 2. Enter M2 in the **Controlling Device Reference** field and click **OK**.
 Repeat for X4. The completed **Ladder Diagram** appears as shown below.



4.3 AND Rungs and OR Rungs

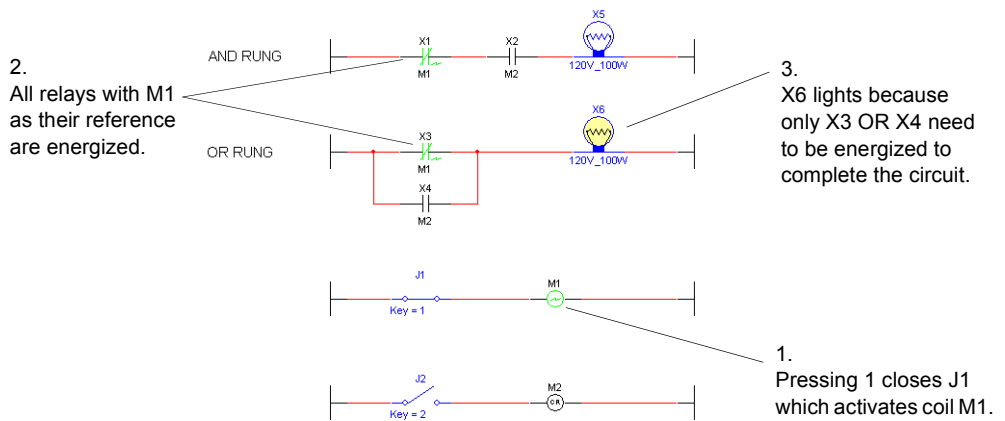
This section illustrates the difference between AND rungs and OR rungs that are found in **Ladder Diagrams**. The concepts described here should be understood before reviewing the more complex circuits found in this chapter.

Ladder Diagrams



➤ 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.



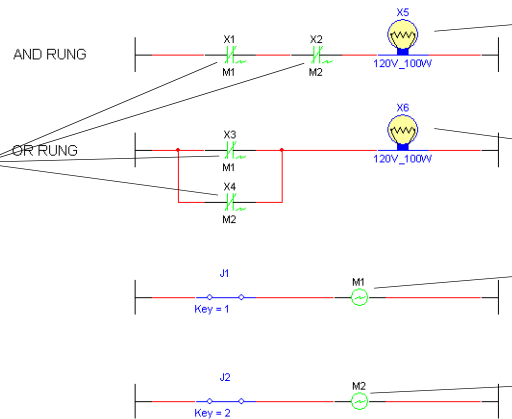
If you press 2 on your keyboard (or hover your cursor over J2 and click the button that pops up), J2 closes which activates coil M2. X6 lights because X4 is energized.

➤ To active the lamp in the AND rung:

1. Select **Simulate/Run** to start simulation of the circuit.

- Press 1 and 2 on your keyboard to close J1 and J2. Lamps X5 and X6 light as described below.

- All relays with M1 and M2 as their references are energized.

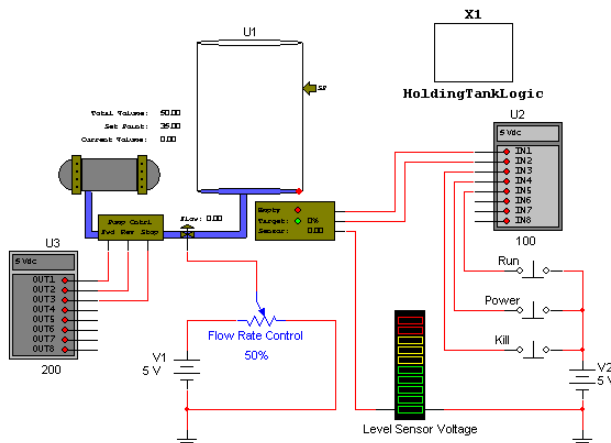


- X5 lights because X1 AND X2 are energized and the circuit is complete.
- X6 lights because X3 OR X4 is energized and the circuit is complete.
- Pressing 1 closes J1 which activates coil M1.
- Pressing 2 closes J2 which activates coil M2.

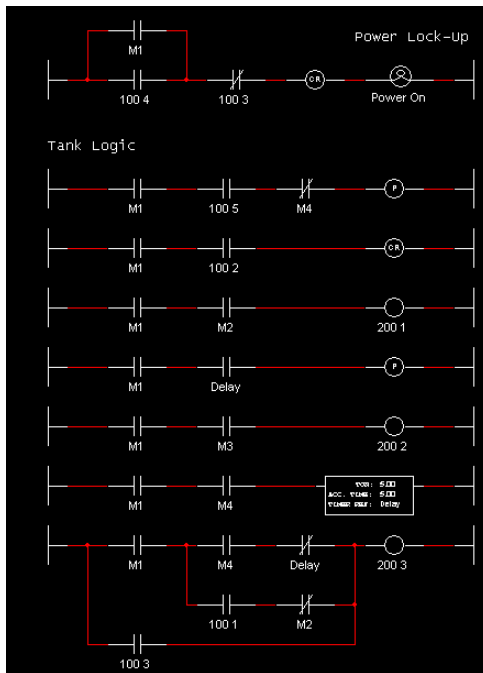
4.4 Sample Circuits

4.4.1 Holding Tank

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



For details on the user-settable parameters for the Holding Tank, Input Module and Output Module see "A.2 Ladder Diagram Parts" on page A-8

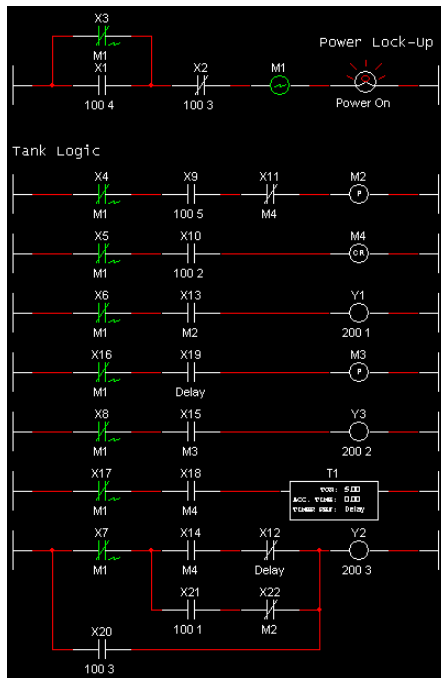


The Ladder Diagram is contained in a separate Hierarchical Block called HoldingTankLogic. For details on hierarchical blocks, refer to the Multisim User Guide.

➤ 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.

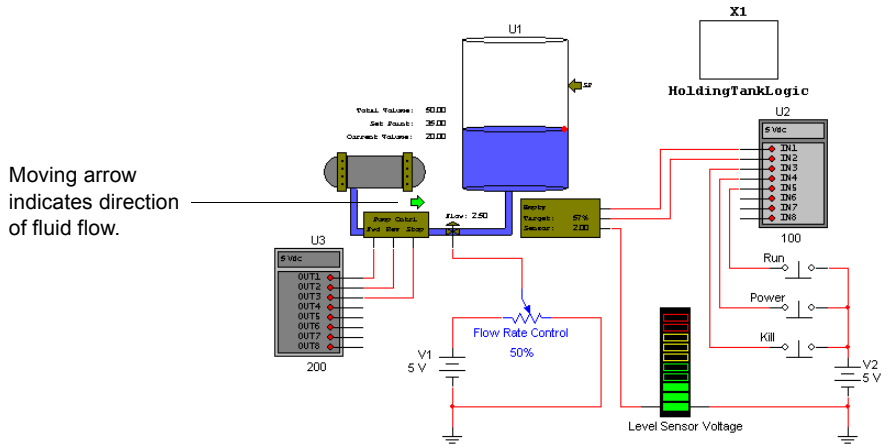


➤ To run the holding tank circuit:

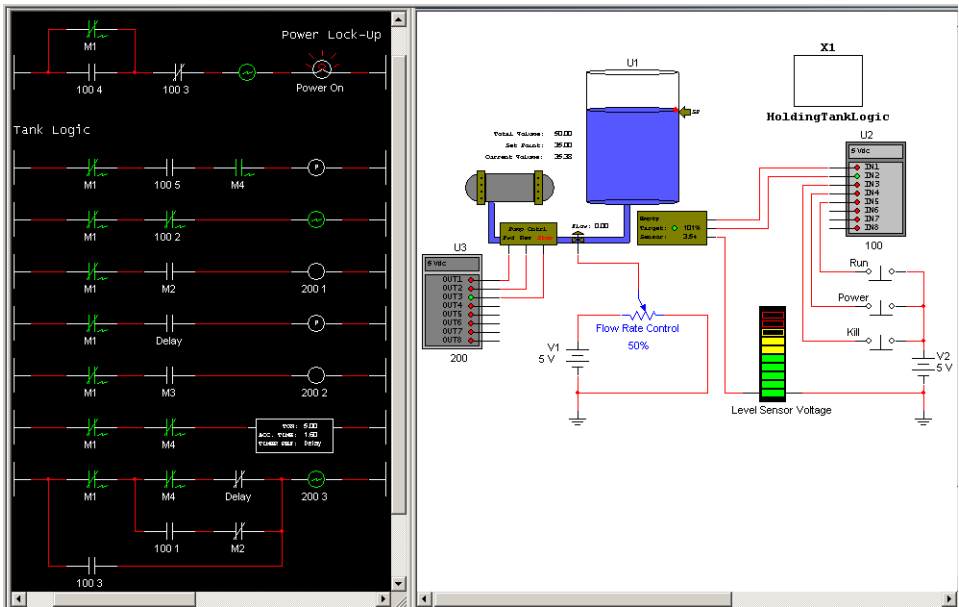
1. Activate the circuit as described above.
2. Press R on your keyboard (or hover your cursor over the Run switch and click the button that pops up) to activate the Run temporary switch.

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.

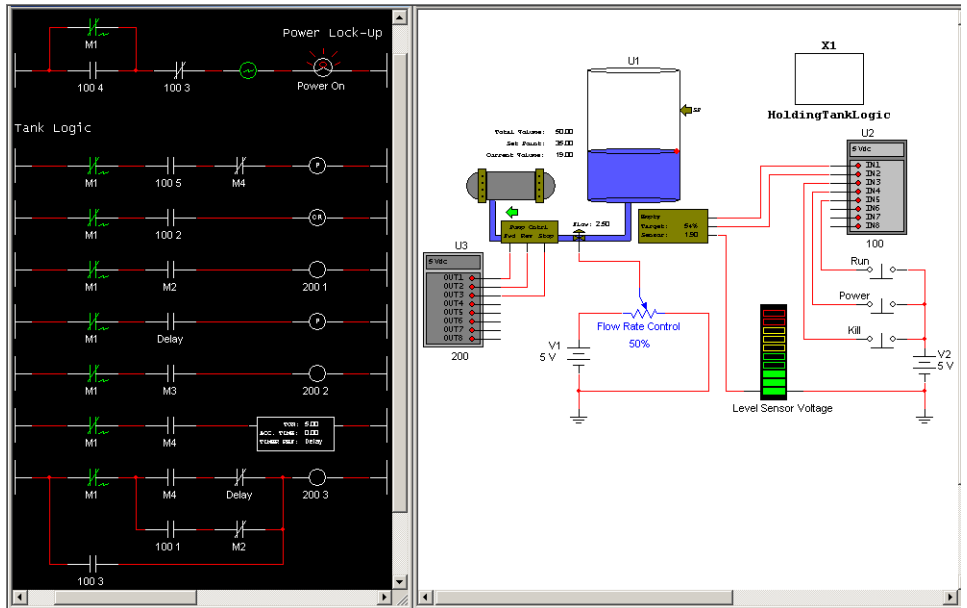
- As the simulation proceeds, the tank begins to fill.



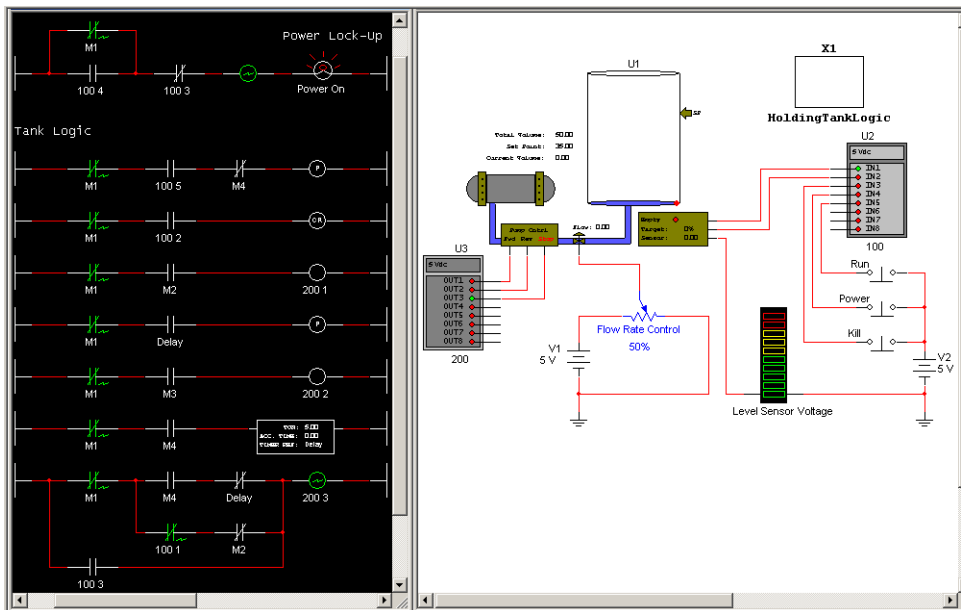
- When the level of the fluid in the tank gets to the Set Point, fluid stops being pumped.



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



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

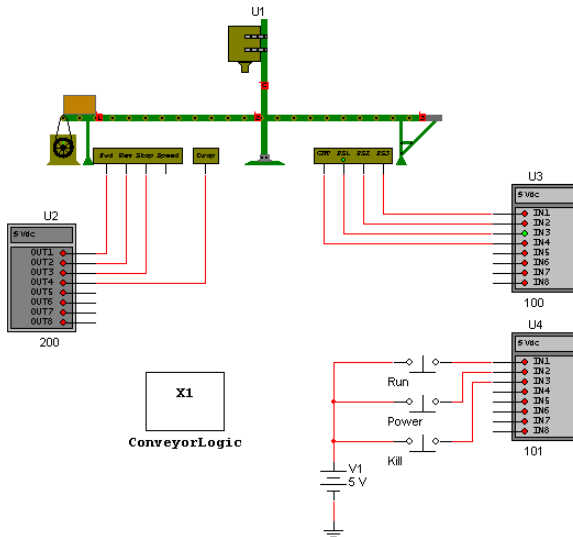


- To turn off the power at any point in the simulation:
 1. Press K on your keyboard (or hover your cursor over the Kill switch and click the button that pops up) to activate the Kill temporary switch. This sends 5 V to pin IN3 of Input Module U2 (**Input Module Base Address = 100**) which in turn energizes 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**.

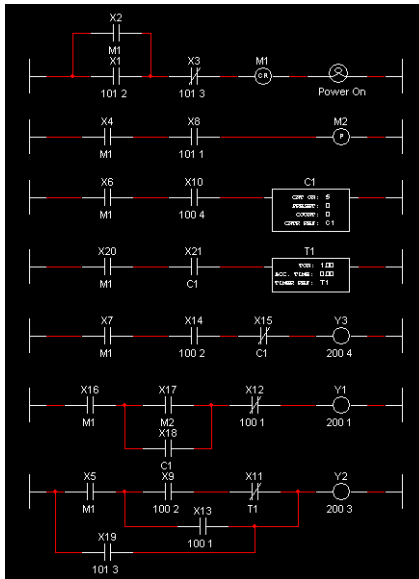
When you press K, X20 is also temporarily energized, which in turn temporarily energizes Output Coil Y2, which sends a pulse to pin Out3 of Output Module U3. This is wired to the **Stop** pin of the holding tank, so the tank stops filling or emptying (depending on which is currently occurring).

4.4.2 Conveyor Belt

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



For details on the user-settable parameters for the Conveyor Belt, Input Module and Output Module, see “A.2 Ladder Diagram Parts” on page A-8

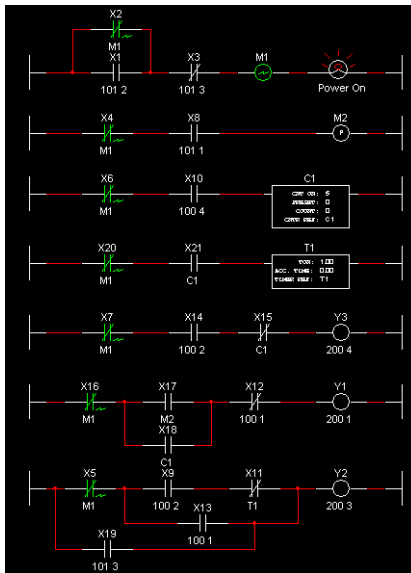


The Ladder Diagram is contained in a separate Hierarchical Block called ConveyorLogic. For details on hierarchical blocks, refer to the Multisim User Guide.

➤ To activate this circuit:

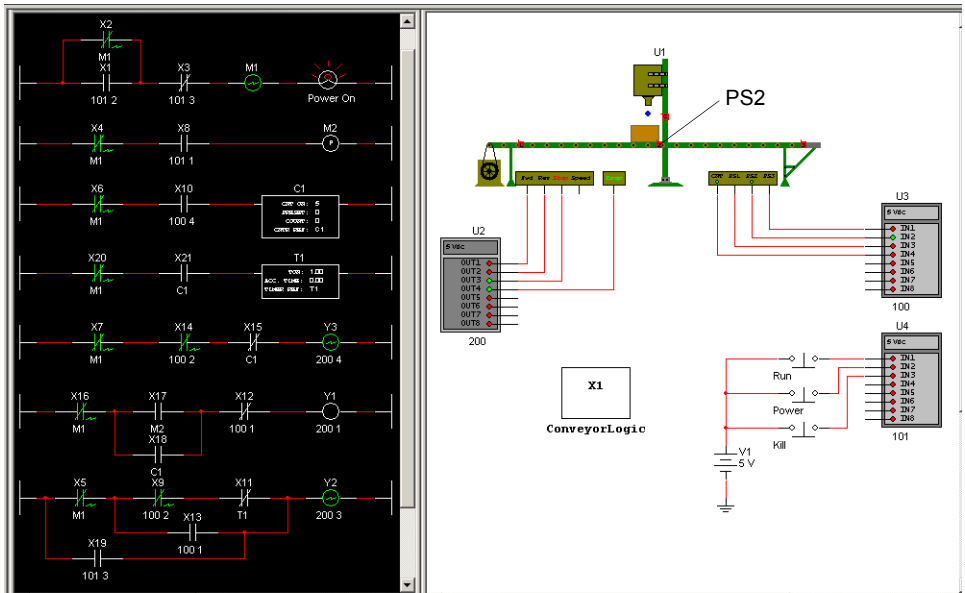
1. Select **Simulate/Run** to begin simulation.
2. Press P on your keyboard (or hover your cursor over the Power switch and click the button that pops up) to activate the Power temporary switch. This sends 5 V to pin IN2 of Input Module U4 (**Input Module Base Address = 101**) which in turn energizes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

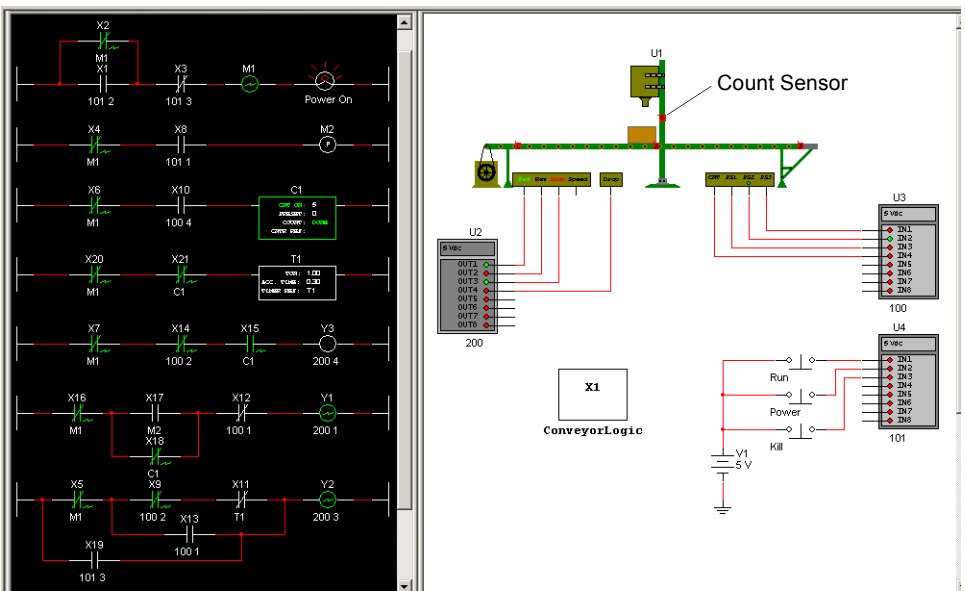


- To run the conveyor belt:
 1. Activate the circuit as described earlier.
 2. Press R on your keyboard (or hover your cursor over the Run switch and click the button that pops up) to activate the Run temporary switch.
- Tip** Select **Window/Tile Vertical** to view the ladder diagram and the circuit at the same time. Observe the interaction between the ladder diagram and the circuit as the simulation proceeds.

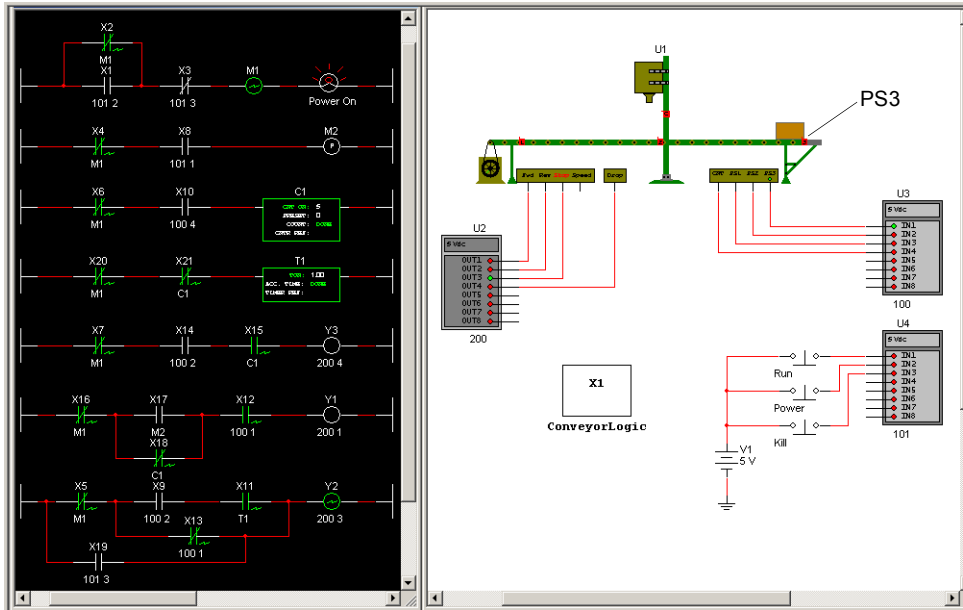
- As the simulation proceeds, the box moves along the conveyor belt to Position Sensor 2 (PS2). The box stops moving and balls begin dropping from the hopper into the box.



- 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).

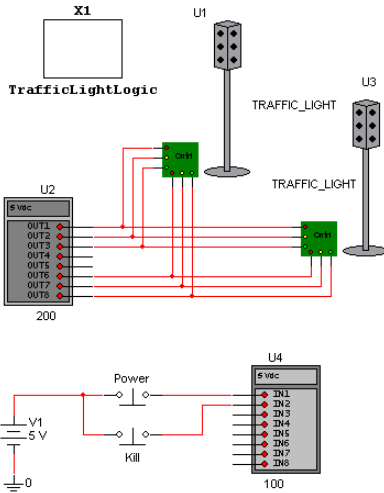


- To turn off the power at any point in the simulation:
 1. Press K on your keyboard (or hover your cursor over the Kill switch that pops up) to activate the Kill temporary switch. This sends 5 V to pin IN3 of Input Module U4 (**Input Module Base Address** = 101) which in turn energizes Input Contact 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.

4.4.3 Traffic Light

The ladder diagram in this section runs two traffic lights.



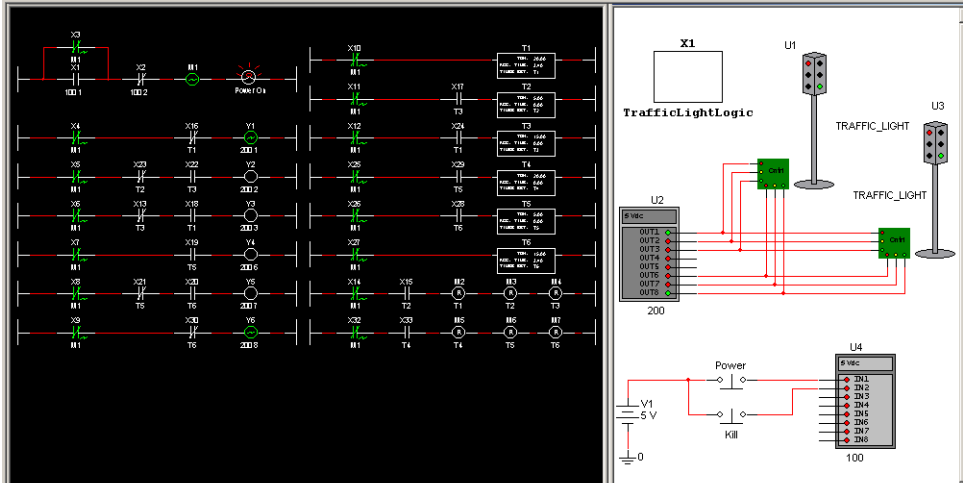
The Ladder Diagram is contained in a separate Hierarchical Block called TrafficLightLogic. For details on hierarchical blocks, refer to the Multisim User Guide.



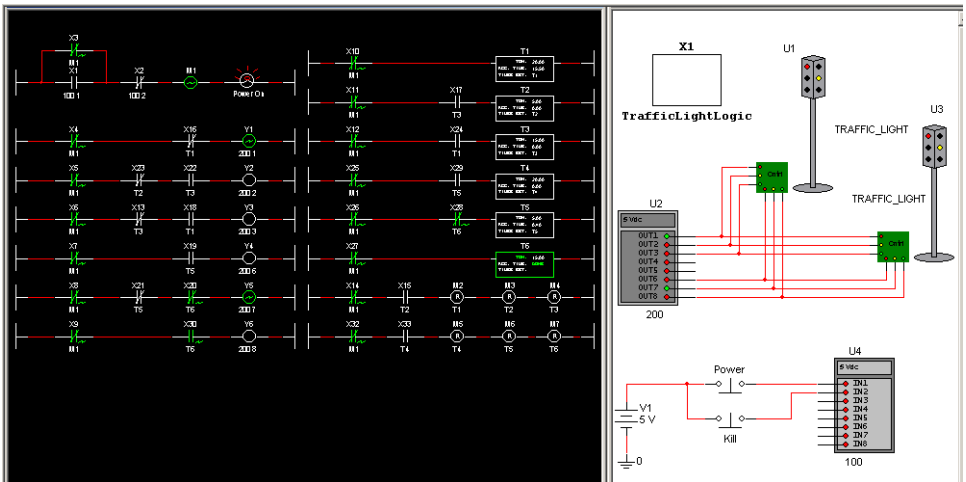
- To run the traffic lights:
 1. Select **Simulate/Run**.
 2. Press P on your keyboard (or hover your cursor over the Power switch and click the button that pops up) to activate the Power momentary switch.

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.

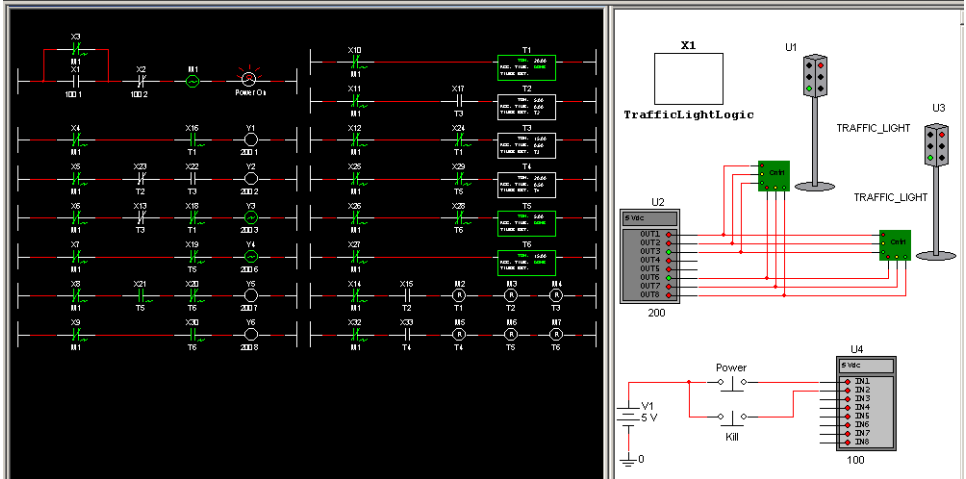
3. The red and green lights in traffic lights U1 and U3 light as shown below.



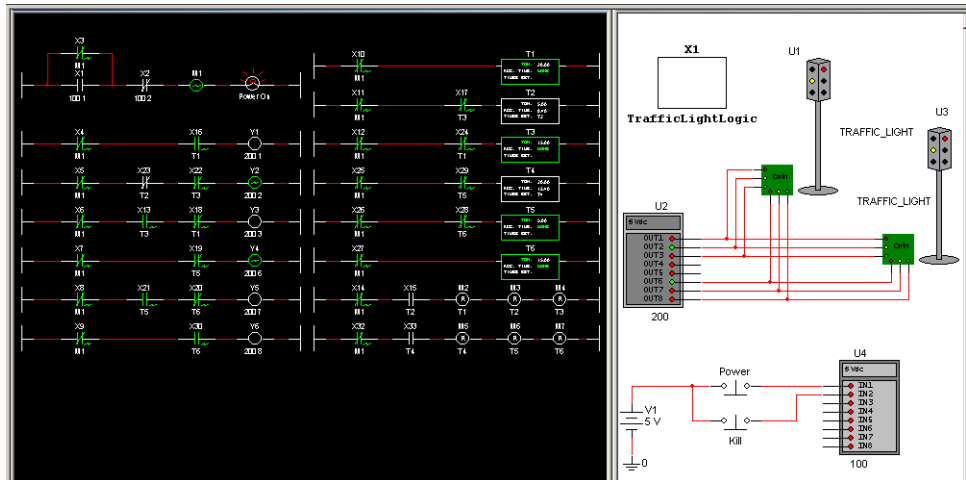
4. After 15 seconds, the green lights turn amber.



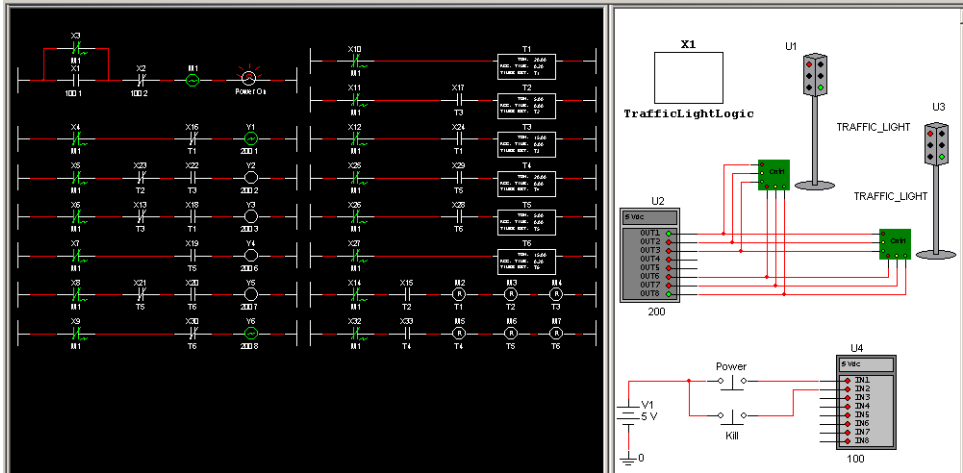
5. After 5 more seconds, the amber lights turn red and the red lights turn green.



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.

Appendix A - Education Edition Parts

This appendix contains a description of the parts that are unique to the Education edition of Multisim.

A.1 3D Virtual Parts

This family contains a number of 3D parts. These components function normally when the circuit is simulated, but appear like the real component on the circuit schematic.

The 3D virtual components are found in the **Basic** group in the **3D Rated Virtual** family.

A.1.1 3D 555 Timer



555_Timer

The 555 timer is an IC chip that is commonly used as an astable multivibrator, a monostable multivibrator or a voltage-controlled oscillator. The 555 timer consists basically of two comparators, a resistive voltage divider, a flip-flop and a discharge transistor. It is a two-state device whose output voltage level can be either high or low. The state of the output can be controlled by proper input signals and time-delay elements connected externally to the 555 timer.

Note Refer to the *Component Reference Guide* for a more detailed discussion of the 555 timer.

A.1.2 3D BJT



A bipolar junction transistor, or BJT, is a current-based valve used for controlling electronic current. BJTs are operated in three different modes, depending on which element is common to input and output: common base, common emitter or common collector. The three modes have different input and output impedances and different current gains, offering individual advantages to a designer.

Note Refer to the *Component Reference Guide* for a more detailed discussion of BJTs.

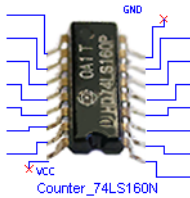
A.1.3 3D Capacitors



A capacitor stores electrical energy in the form of an electrostatic field. Capacitors are widely used to filter or remove AC signals from a variety of circuits. In a DC circuit, they can be used to block the flow of direct current while allowing AC signals to pass.

Note Refer to the *Component Reference Guide* for a more detailed discussion of capacitors.

A.1.4 3D 74LS160N Counter



This synchronous, presettable decade counter features an internal carry look-ahead for fast counting.

Note Refer to the *Component Reference Guide* for a more detailed discussion of the 74LS160N counter.

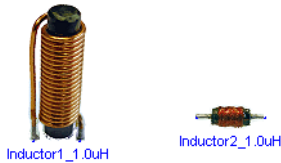
A.1.5 3D Diode



Diodes allow current to flow in only one direction and can therefore be used as simple solid-state switches in AC circuits, being either open (not conducting) or closed (conducting). Terminal A is called the anode and terminal K is called the cathode.

Note Refer to the *Component Reference Guide* for a more detailed discussion of diodes.

A.1.6 3D Inductor



An inductor stores energy in an electromagnetic field created by changes in current through it. Its ability to oppose a change in current flow is called inductance, L , and is measured in Henries.

Note Refer to the *Component Reference Guide* for a more detailed discussion of inductors.

A.1.7 3D LED



This diode emits visible light when forward current through it, I_d , exceeds the turn-on current, I_{on} . The electrical model of the LED is the same as the diode model.

Note Refer to the *Component Reference Guide* for a more detailed discussion of LEDs.

A.1.8 3D MOSFET



A MOSFET is a Metal-Oxide-Semiconductor FET. This transistor is a type of FET that uses an induced electrical field to control current through the device. Either negative or positive gate voltages can be applied to control the current.

Note Refer to the *Component Reference Guide* for a more detailed discussion of MOSFETs.

A.1.9 3D DC Motor



The component is a universal model of an ideal DC motor which can be used to model the behavior of a DC motor excited in parallel, in series or separately.

Note Refer to the *Component Reference Guide* for a more detailed discussion of DC motors.

A.1.10 3D Opamp



An ideal operational amplifier (Opamp) is an amplifier with infinite gain, infinite input impedance and zero output impedance. With the application of negative feedback, Opamps can be used to implement functions such as addition, subtraction, differentiation, integration, averaging and amplification.

Note Refer to the *Component Reference Guide* for a more detailed discussion of opamps.

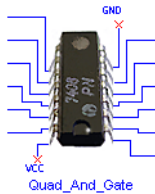
A.1.11 3D Potentiometer



This component acts much like a regular resistor, except that you can adjust its resistance.

Note Refer to the *Component Reference Guide* for a more detailed discussion of potentiometers.

A.1.12 3D AND Gate



This device contains four independent 2-input AND gates.

Note Refer to the *Component Reference Guide* for a more detailed discussion of the 74LS08 Quad AND gate.

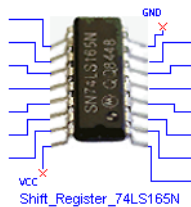
A.1.13 3D Resistor



Resistors come in a variety of sizes, depending on the power they can safely dissipate. A resistor's resistance, R , is measured in ohms.

Note Refer to the *Component Reference Guide* for a more detailed discussion of resistors.

A.1.14 3D Shift Register



This serial shift-register shifts the data in the direction of QA toward QH when clocked. To load the data at the 8-inputs into the device, apply a low level at the shift/load input. This register is equipped with a complementary output at the eighth bit.

Note Refer to the *Component Reference Guide* for a more detailed discussion of the 74LS165N shift register.

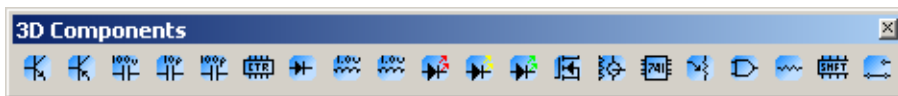
A.1.15 3D Switch



This component is a SPST (single-pole, single-throw) switch.

Note Refer to the *Component Reference Guide* for a more detailed discussion of switches.

A.1.16 3D Components Toolbar



Some of the more commonly-used 3D components can be placed using the **3D Components** toolbar.

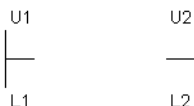
- To display the **3D Components** toolbar, click the **Show 3D Family** button in the **Virtual** toolbar (refer to the *Multisim User Guide* for **Virtual** toolbar information).

The buttons (from left to right) in the **3D Components** toolbar place the following virtual components: NPN transistor; PNP transistor; 100 uF capacitor; 10 pF capacitor; 100 pF capacitor; 74LS160N counter; Diode; 1 uH inductor; 1 uH inductor; Red LED; Yellow LED; Green LED; MOSFET; DC motor; 741 OPAMP; Potentiometer; Quad AND gate; 1.1 k Resistor; Shift Register 74LS165N; Switch.

A.2 Ladder Diagram Parts

A.2.1 Ladder Rungs

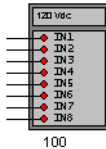
A.2.1.1 L1 and L2



L1 is a Rung Starter and L2 is a Rung Terminator. Continuity between L1 and L2 is required to activate/energize devices placed between them.

A.2.2 Ladder I/O Modules

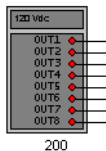
A.2.2.1 Input Module



This device is an input module for ladder diagrams and is available in various input voltages. It is used to bring external stimulus into a ladder diagram.

- To set the Input Module's base address:
 1. Double-click on the module and select the **Value** tab.
 2. Enter the desired value in the **Input Module Base Address** field. The default value is 100, as shown in the above diagram.

A.2.2.2 Output Module



This device is an output module for ladder diagrams and is available in various output voltages. It allows external circuit elements to be manipulated by the logic contained in a ladder diagram.

- To set the Output Module's base address:
 1. Double-click on the module and select the **Value** tab.
 2. Enter the desired value in the **Output Module Base Address** field. The default value is 200, as shown in the above diagram.

A.2.3 Ladder Relay Coils

A.2.3.1 Relay Coil



This device is a relay coil for ladder diagrams. When this coil is energized, corresponding contact(s) which reference this device will change their state. (For example, a normally open contact will close).

- To set the coil reference for this device:
 1. Double-click on the relay coil and select the **Value** tab.
 2. Enter the desired value in the **Coil Reference** field.

A.2.3.2 Negated Relay Coil



This device is a negated relay coil for ladder diagrams.

- To set the coil reference for this device:
 1. Double-click on the relay coil and select the **Value** tab.
 2. Enter the desired value in the **Coil Reference** field.

A.2.3.3 Set Coil



This device is a latch relay coil for ladder diagrams (requires a reset coil to operate).

- To set the coil reference for this device:
 1. Double-click on the relay coil and select the **Value** tab.
 2. Enter the desired value in the **Coil Reference** field.

A.2.3.4 Reset Coil



This device is an unlatch relay coil for ladder diagrams, used to reset timers, counters and set coils.

- To set the target device for this coil:
 1. Double-click on the set coil and select the **Value** tab.
 2. Enter the desired value in the **Target Device Reference** field.

A.2.3.5 Pulsed Relay Coil



This device is a positive pulse relay coil, which produces a positive pulse of user-definable duration.

- To set the parameters for this coil:
 1. Double-click on the positive pulse relay coil and select the **Value** tab.
 2. Enter the desired values in the **Coil Reference** and **Pulse Duration** fields.

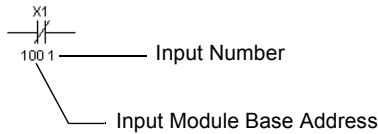
A.2.4 Ladder Contacts

A.2.4.1 Input Contact NC

This device is a normally closed input contact for ladder diagrams. It responds to the status at the specified input on the referenced input module.

- To set the parameters for this device:
 1. Double-click on the input contact and select the **Value** tab.
 2. Enter the address of the input module associated with this contact (see “A.2.2.1 Input Module” on page A-9) in the **Input Module Base Address** field.

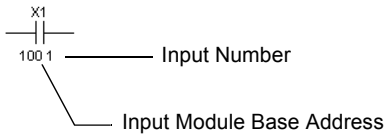
-
3. Enter the number of the input on the input module which will control this input contact in the **Input Number** field.



A.2.4.2 Input Contact NO

This device is a normally open input contact for ladder diagrams. It responds to the status at the specified input on the referenced input module.

- To set the parameters for this device:
 1. Double-click on the input contact and select the **Value** tab.
 2. Enter the address of the input module associated with this contact (see “A.2.2.1 Input Module” on page A-9) in the **Input Module Base Address** field.
 3. Enter the number of the input on the input module which will control this input contact in the **Input Number** field.



A.2.4.3 Relay Contact NC



This device is a normally closed relay contact for ladder diagrams. This contact will open when the controlling device (coil, counter or timer) is energized.

- To set the parameters for this device:
 1. Double-click on the NC contact and select the **Value** tab.
 2. Enter the desired value in the **Controlling Device Reference** field.

A.2.4.4 Relay Contact NO

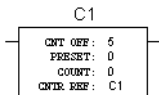


This device is a normally open relay contact for ladder diagrams.

- To set the parameters for this device:
 1. Double-click on the relay contact and select the **Value** tab.
 2. Enter the desired value in the **Controlling Device Reference** field.

A.2.5 Ladder Counters

A.2.5.1 Count Off

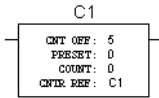


This device is a count OFF counter with preset, for ladder diagrams. The contact associated with this device will be energized at simulation start. The contact will be de-energized when the counter “Set Value” is reached.

This device does not retain its count out state and will automatically reset after count out occurs on the next action.

- To set the parameters for this device:
 1. Double-click on the counter and select the **Value** tab.
 2. Enter the following parameters:
 - **Set Value** — the counter’s count out value.
 - **Preset Value** — the count starting point.
 - **Counter Reference** — set to the device’s **RefDes** by default. You can enter any other identifying string.

A.2.5.2 Count Off Hold

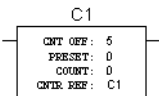


This device is a count OFF and hold counter with preset, for ladder diagrams. The contact associated with this device will be energized at simulation start. The contact will be de-energized when the counter “Set Value” is reached.

Once the count out state is reached, the counter status is held until simulation is restarted.

- To set the parameters for this device:
 1. Double-click on the counter and select the **Value** tab.
 2. Enter the following parameters:
 - **Set Value** — the counter’s count out value.
 - **Preset Value** — the count starting point.
 - **Counter Reference** — set to the device’s **RefDes** by default. You can enter any other identifying string.

A.2.5.3 Count Off Reset

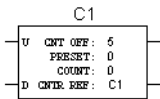


This device is a count OFF and hold until reset counter with preset, for ladder diagrams. The contact associated with this device will be energized at simulation start. The contact will be de-energized when the counter “Set Value” is reached.

You can reset this counter using a reset coil at any time during simulation, regardless of the current counter state.

- To set the parameters for this device:
 1. Double-click on the counter and select the **Value** tab.
 2. Enter the following parameters:
 - **Set Value** — the counter’s count out value.
 - **Preset Value** — the count starting point.
 - **Counter Reference** — set to the device’s **RefDes** by default. You can enter any other identifying string.

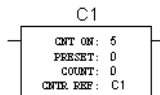
A.2.5.4 Count Off Up Down



This device is a resettable up/down count OFF and hold until reset counter with preset, for ladder diagrams. This bi-directional counter increments the count via the “U” input, and decrements the count via the “D” input.

- To set the parameters for this device:
 1. Double-click on the counter and select the **Value** tab.
 2. Enter the following parameters:
 - **Set Value** — the counter’s count out value.
 - **Preset Value** — the count starting point.
 - **Counter Reference** — set to the device’s **RefDes** by default. You can enter any other identifying string.

A.2.5.5 Count On

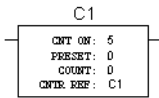


This device is a count ON counter with preset, for ladder diagrams. The contact associated with this device will be de-energized at simulation start. The contact will be re-energized when the counter “Set Value” is reached.

This device does not retain its count out state and will automatically reset after count out occurs on the next action.

- To set the parameters for this device:
 1. Double-click on the counter and select the **Value** tab.
 2. Enter the following parameters:
 - **Set Value** — the counter’s count out value.
 - **Preset Value** — the count starting point.
 - **Counter Reference** — set to the device’s **RefDes** by default. You can enter any other identifying string.

A.2.5.6 Count On Hold

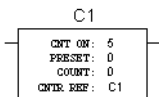


This device is a count ON and hold counter with preset, for ladder diagrams. The contact associated with this device will be de-energized at simulation start. The contact will be energized when the counter “Set Value” is reached.

Once the count out state is reached, the counter status is held until simulation is restarted.

- To set the parameters for this device:
 1. Double-click on the counter and select the **Value** tab.
 2. Enter the following parameters:
 - **Set Value** — the counter’s count out value.
 - **Preset Value** — the count starting point.
 - **Counter Reference** — set to the device’s **RefDes** by default. You can enter any other identifying string.

A.2.5.7 Count On Reset

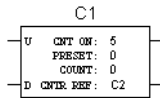


This device is a count ON and hold until reset counter with preset, for ladder diagrams. The contact associated with this device will be de-energized at simulation start. The contact will be energized when the counter “Set Value” is reached.

You can reset this counter using a reset coil at any time during simulation, regardless of the current counter state.

- To set the parameters for this device:
 1. Double-click on the counter and select the **Value** tab.
 2. Enter the following parameters:
 - **Set Value** — the counter’s count out value.
 - **Preset Value** — the count starting point.
 - **Counter Reference** — set to the device’s **RefDes** by default. You can enter any other identifying string.

A.2.5.8 Count On Up Down

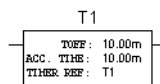


This device is a resettable up/down count ON and hold until reset counter with preset, for ladder diagrams. This bi-directional counter increments the count via the “U” input, and decrements the count via the “D” input.

- To set the parameters for this device:
 1. Double-click on the counter and select the **Value** tab.
 2. Enter the following parameters:
 - **Set Value** — the counter’s count out value.
 - **Preset Value** — the count starting point.
 - **Counter Reference** — set to the device’s **RefDes** by default. You can enter any other identifying string.

A.2.6 Ladder Timers

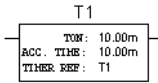
A.2.6.1 Timer TOFF



This device is a TOFF timer for ladder diagrams. The associated contact is energized at the start of simulation, and is de-energized when this device “times out”. Any interruption in continuity on the rung in which this timer is located causes the time value to be reset to zero. (This occurs whether or not the timer has timed out).

- To set the parameters for this device:
 1. Double-click on the timer and select the **Value** tab.
 2. Enter the following parameters:
 - **Delay Time** — the timer’s time out value.
 - **Timer Reference** — set to the device’s **RefDes** by default. You can enter any other identifying string.

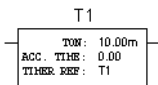
A.2.6.2 Timer TON



This device is a TON timer for ladder diagrams. The associated contact is de-energized at the start of simulation, and is energized when this device “times out”. Any interruption in continuity on the rung in which this timer is located causes the time value to be reset to zero. (This occurs whether or not the timer has timed out).

- To set the parameters for this device:
 1. Double-click on the timer and select the **Value** tab.
 2. Enter the following parameters:
 - **Delay Time** — the timer’s time out value.
 - **Timer Reference** — set to the device’s **RefDes** by default. You can enter any other identifying string.

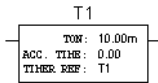
A.2.6.3 Timer TON Retention



This device is a TON timer with retention capabilities, for ladder diagrams. Once continuity is established, the timer starts. If continuity is broken, the timer retains the accumulated time and will continue from this time when continuity is re-established, until time out. Once timed out, the timer will reset if continuity is broken, and will start accumulated time at zero.

- To set the parameters for this device:
 1. Double-click on the timer and select the **Value** tab.
 2. Enter the following parameters:
 - **Delay Time** — the timer’s time out value.
 - **Timer Reference** — set to the device’s **RefDes** by default. You can enter any other identifying string.

A.2.6.4 Timer TON Retention Reset



This device is a TON timer with retention and reset capabilities, for ladder diagrams. Once continuity is established, the timer starts. If continuity is broken, the timer retains the accumulated time and will continue from this time when continuity is re-established, until time out. Once timed out, the timer will reset automatically if continuity is removed. This device can be reset by a reset coil at any time before time out.

- To set the parameters for this device:
 1. Double-click on the timer and select the **Value** tab.
 2. Enter the following parameters:
 - **Delay Time** — the timer's time out value.
 - **Timer Reference** — set to the device's **RefDes** by default. You can enter any other identifying string.

A.2.6.5 Timer TON Retention Hold Reset

This device is a TON timer with retention, hold and reset capabilities, for ladder diagrams. Once continuity is established, the timer starts. If continuity is broken, the timer retains the accumulated time and will continue from this time when continuity is re-established, until time out. Once timed out, the timer will not reset automatically when continuity is removed. This device can be reset by a reset coil at any time.

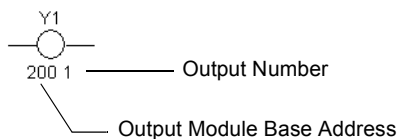
- To set the parameters for this device:
 1. Double-click on the timer and select the **Value** tab.
 2. Enter the following parameters:
 - **Delay Time** — the timer's time out value.
 - **Timer Reference** — set to the device's **RefDes** by default. You can enter any other identifying string.

A.2.7 Ladder Output Coils

A.2.7.1 Output Coil

This device is an output coil for ladder diagrams.

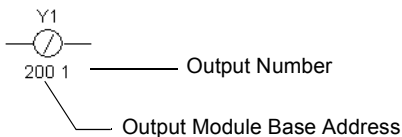
- To set the parameters for this device:
 1. Double-click on the output coil and select the **Value** tab.
 2. Enter the address for the output module associated with this device in the **Output Module Base Address** field.
 3. Enter the number of the output on the output module that this device will drive in the **Output Number** field.



A.2.7.2 Output Coil Negated

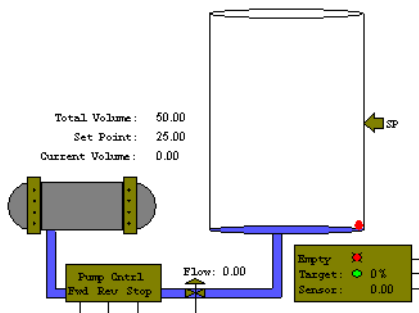
This device is a negated output coil for ladder diagrams.

- To set the parameters for this device:
 1. Double-click on the output coil and select the **Value** tab.
 2. Enter the address for the output module associated with this device in the **Output Module Base Address** field.
 3. Enter the number of the output on the output module that this device will drive in the **Output Number** field.



A.3 Miscellaneous Peripherals

A.3.1 Holding Tank

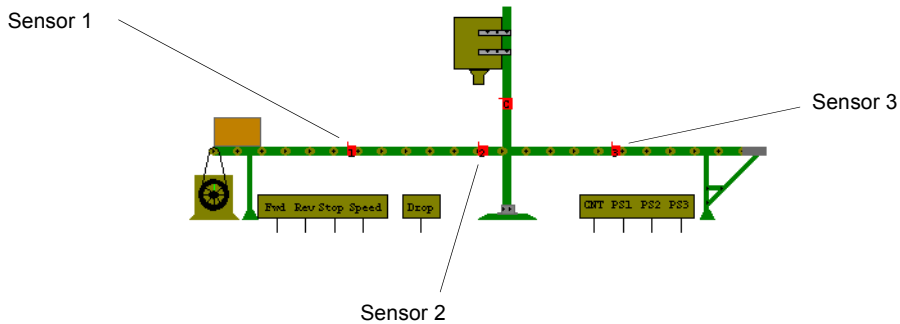


This device is a liquid holding tank for use in ladder diagrams.

- To set the parameters for this device:
 1. Double-click on the relay coil and select the **Value** tab.
 2. Set the following parameters as desired:
 - **Tank Volume (litres)** — capacity of the tank in litres.
 - **Level Detector Set Point (litres)** — the level of the **SP** (set point) marker (see above screen capture).
 - **Maximum Pump Flow Rate (litres/second)** — the maximum speed at which the liquid is pumped into the holding tank. If the **Flow** pin of the holding tank (see above screen capture) is unconnected, the liquid only flows at this speed.
 - **Flow Rate Full Scale Voltage** — if the **Flow** pin of the holding tank is connected, this is the voltage required at the **Flow** pin to make the liquid move at the maximum speed set in the **Maximum Pump Flow Rate** field. For example, if you enter 5 V in this field, and then apply 5 V to the **Flow** pin, the liquid will flow at 1 litre/second if the value in the **Maximum Pump Flow Rate** field is 1. If you apply 2.5 V to the **Flow** pin, the liquid will flow at half that speed (0.5 litre/second).
 - **Sensor Full Scale Voltage** — the sensor voltage that equates to a full tank. As the tank fills, the voltage shown in the **Sensor** field of the holding tank (see above screen capture) increases.

Note For an example of this component in a complete circuit, refer to “4.4.1 Holding Tank” on page 4-7.

A.3.2 Conveyor Belt



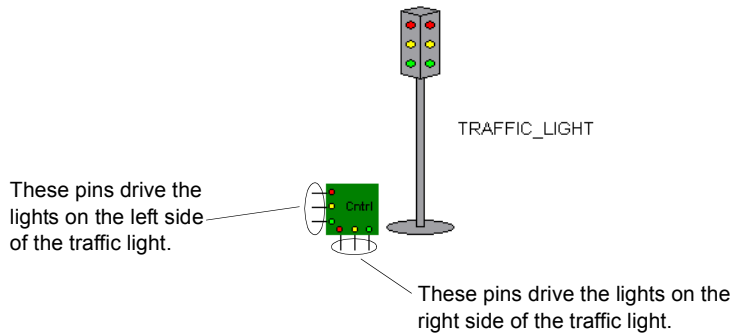
This device is a conveyor belt for use in ladder diagrams.

➤ To set the parameters for this device:

1. Double-click on the relay coil and select the **Value** tab.
2. Set the following parameters as desired:
 - **Belt Length (meters)** — the length of the conveyor belt in meters.
 - **Max. Belt Speed (meters/sec)** — the maximum speed the belt moves. If the **Speed** pin of the conveyor belt (see above screen capture) is unconnected, the belt only moves at this speed.
 - **Speed Control Full Scale Voltage** — if the **Speed** pin of the conveyor belt is connected, this is the voltage required at the **Speed** pin to make the belt move at the maximum speed set in the **Max. Belt Speed** field. For example, if you enter 5 V in this field, and then apply 5 V to the **Speed** pin, the belt will move at 0.5 meters/second if the value in the **Max. Belt Speed** field is 0.5. If you apply 2.5 V to the **Speed** pin, the belt will move at half that speed (0.25 meters per second).
 - **Sensor 1 Position (meters)** — the position of Sensor 1 from the left edge of the belt.
 - **Sensor 2 Position (meters)** — the position of Sensor 2 from the left edge of the belt.
 - **Sensor 3 Position (meters)** — the position of Sensor 3 from the left edge of the belt. Do not set greater than the **Belt Length** value.

Note For an example of this component in a complete circuit, see “4.4.2 Conveyor Belt” on page 4-12.

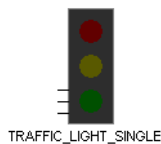
A.3.3 Traffic Light



This component has no parameters that need to be set up in the **Value** tab.

Note For an example of this component in a complete circuit, see “4.4.3 Traffic Light” on page 4-17.

A.3.4 Single Traffic Light



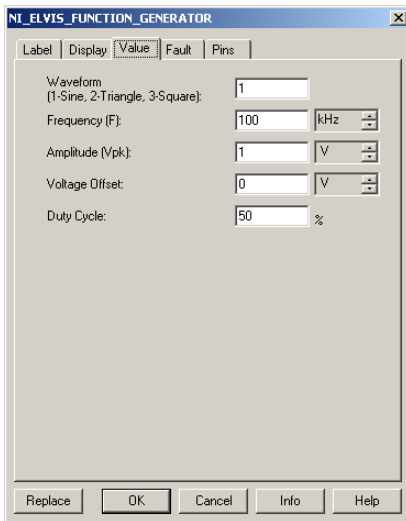
This device is similar to the double traffic light (see “A.3.3 Traffic Light” on page A-23) except that it contains a single bank of lights.

A.4 Virtual NI ELVIS Components

Virtual NI ELVIS components are special devices that are only accessible from special rows on the Virtual NI ELVIS prototyping schematic.

A.4.1 NI ELVIS Function Generator

This component emulates the behavior of a function generator, and is accessed from the left row of the Virtual NI ELVIS prototyping schematic.

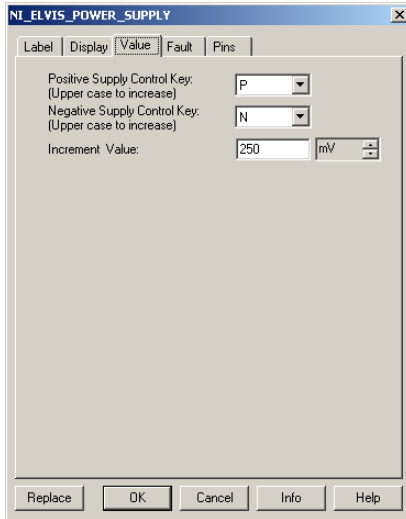


- To set the parameters for the function generator:
 1. Double-click on “Function Generator” in the Virtual NI ELVIS schematic. The properties dialog for the NI ELVIS Function Generator appears.
 2. Click on the **Value** tab and set the following parameters as desired:
 - **Waveform** — choose 1 for a sine wave; 2 for a triangle wave; 3 for a square wave.
 - **Frequency** — the frequency and unit of measure of the function generator output.
 - **Amplitude** — the peak voltage and its unit of measure.
 - **Voltage Offset** — controls the DC level about which the alternating signal varies. An offset of 0 positions the waveform along the oscilloscope's x-axis (provided its Y POS setting is O). A positive value shifts the DC level upward, while a negative value shifts it downward. Offset uses the units set for Amplitude.
 - **Duty Cycle** — the ratio of on-period to off-period. A sine wave is not affected by this setting.

Note See also, “3.2 The Virtual NI ELVIS Schematic” on page 3-2.

A.4.2 NI ELVIS Power Supply

This component is accessed from the left row of the Virtual NI ELVIS prototyping schematic.



- To set the parameters for this device:
 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 set the following parameters as desired:
 - **Positive Supply Control Key** — select a key to press on the keyboard to increase or decrease the output of the positive variable power supply by the amount set in the **Increment Value** field. Maximum output is +12 VDC.
 - **Negative Supply Control Key** — select a key to press on the keyboard to increase or decrease the output of the negative variable power supply by the amount set in the **Increment Value** field. Maximum output is -12 VDC.
 - **Increment Value** — the size of steps by which the power supply’s output is increased or decreased as the control key is pressed on the keyboard.

Note See also, “3.2 The Virtual NI ELVIS Schematic” on page 3-2.

Appendix B - Support and Services

B.1 Technical Support and Professional Services

Visit the following sections of the National Instruments web site at ni.com for technical support and professional services:

- **Support** — online technical support resources at ni.com/support include the following:
 - **Self-Help Resources** — For answers and solutions, visit the award-winning National Instruments web site 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.
 - **Free Technical Support** — All registered users receive free Basic Service, which includes access to hundreds of Application Engineers worldwide in the NI Discussion Forums at ni.com/forums. National Instruments Application Engineers make sure every question receives an answer. 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 in the front of this manual. You can also 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

Numerics

- 3D Components
 - 555 timer A-1
 - 74LS160N counter A-3
 - BJT A-2
 - capacitor A-2
 - DC motor A-5
 - diode A-3
 - inductor A-4
 - LED A-4
 - MOSFET A-5
 - opamp A-5
 - potentiometer A-6
 - resistor A-7
 - shift register A-7
- 3D Components toolbar A-8

B

- Breadboard Netlist dialog box 2-11
- Breadboard Settings 2-2

C

- circuit restrictions
 - setting 1-4
- Conveyor Belt A-22
- Count Off A-13
- Count Off Hold A-14
- Count Off Reset A-14
- Count Off Up Down A-15
- Count On A-15
- Count On Hold A-16
- Count On Reset A-16
- Count On Up Down A-17

G

- global restrictions
 - passwords 1-6
 - setting 1-2

I

- Input Module A-9

L

- Liquid Holding Tank A-21

N

- Negated Output Coil A-20
- Negated Relay Coil A-10
- NI ELVIS Function Generator A-24
- NI ELVIS Power Supply A-25
- Normally Closed Input Contact A-11
- Normally Closed Relay Contact A-12
- Normally Open Input Contact A-12
- Normally Open Relay Contact A-13

O

- Output Coil A-20
- Output Module A-9

P

- passwords, creating/changing 1-6
- Pulsed Relay Coil A-11

R

- Relay Coil A-10
- Reset Coil A-11
- restrictions
 - about 1-2
 - setting circuit 1-4
 - setting global 1-2

S

- Set Coil A-10
- setting circuit restrictions 1-4
- setting global restrictions 1-2
- Simplified Version 1-2

T

TOFF timer A-17
TON timer A-18
TON timer with retention A-18
TON timer with retention and reset A-19
TON timer with retention, hold and reset A-19
Traffic Light A-23

V

Virtual NI ELVIS
 3D prototyping board 3-8
 ammeter 3-4
 function generator 3-6
 IV Analyzer 3-4
 oscilloscope 3-3
 overview 3-1
 power supplies 3-6
 prototyping board LEDs 3-7
 schematic 3-2
 Volt/Ohmmeter 3-4