NI Circuit Design Suite

Getting Started with NI Circuit Design Suite

July 2013
374482G

NATIONAL INSTRUMENTS

Worldwide Technical Support and Product Information

ni.com

Worldwide Offices

Visit 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.

National Instruments Corporate Headquarters

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

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

Important Information

Warranty

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

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

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

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

Copyright

End-User License Agreements and Third-Party Legal Notices

You can find end-user license agreements (EULAs) and third-party legal notices in the following locations:

- Notices are located in the `<National Instruments>\_Legal Information` and `<National Instruments>` directories.
- EULAs are located in the `<National Instruments>\Shared\MDF\Legal\license` directory.
- Review `<National Instruments>\_Legal Information.txt` for more information on including legal information in installers built with NI products.

Trademarks

Refer to the *NI Trademarks and Logo Guidelines* at `ni.com/trademarks` for more information on National Instruments trademarks.

ARM, Keil, and µVision are trademarks or registered of ARM Ltd or its subsidiaries.

LEGO, the LEGO logo, WEDO, and MINDSTORMS are trademarks of the LEGO Group. ©2013 The LEGO Group.

TETRIX by Pitsco is a trademark of Pitsco, Inc.©2013

FIELDBUS FOUNDATION™ and FOUNDATION™ are trademarks of the Fieldbus Foundation.

EtherCAT® is a registered trademark of and licensed by Beckhoff Automation GmbH.

CANopen® is a registered Community Trademark of CAN in Automation e.V.

DeviceNet™ and EtherNet/IP™ are trademarks of ODVA.

Go!, SensorDAQ, and Vernier are registered trademarks of Vernier Software & Technology. Vernier Software & Technology and `vernier.com` are trademarks or trade dress.

Xilinx is the registered trademark of Xilinx, Inc.

Taptite and Trilobular are registered trademarks of Research Engineering & Manufacturing Inc.

FireWire® is the registered trademark of Apple Inc.

Linux® is the registered trademark of Linus Torvalds in the U.S. and other countries.

Handle Graphics®, MATLAB®, Real-Time Workshop®, Simulink®, Stateflow®, and xPC TargetBox® are registered trademarks, and TargetBox™ and Target Language Compiler™ are trademarks of The MathWorks, Inc.

Patents

For patents covering National Instruments products/technology, refer to the appropriate location: **Help»Patents** in your software, the `patents.txt` file on your media, or the *National Instruments Patent Notice* at `ni.com/patents`.

Export Compliance Information

Refer to the *Export Compliance Information* at `ni.com/legal/export-compliance` for the National Instruments global trade compliance policy and how to obtain relevant HTS codes, ECCNs, and other import/export data.

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.

Contents

1

Introduction to NI Circuit Design Suite

Some of the described features may not be available in your edition of Circuit Design Suite.

Refer to `ni.com` for a list of the features available in your edition.

NI Circuit Design Suite Product Line

National Instruments Circuit Design Suite is a suite of EDA (Electronics Design Automation) tools that assists you in carrying out the major steps in the circuit design flow.

Multisim is the schematic capture and simulation program designed for schematic entry, simulation, and feeding to downstage steps, such as PCB layout. It also includes mixed analog/digital simulation capability, and microcontroller co-simulation.

Ultiboard is used to design printed circuit boards, perform certain basic mechanical CAD operations, and prepare them for manufacturing. It also provides automated parts placement and layout.

The Tutorials

This book contains the following step-by-step tutorials:

- *Multisim Tutorial*—Introduces you to Multisim and its many functions.
- *Ultiboard Tutorial*—Shows you how to place the components and traces for the design described in the Multisim Tutorial chapter. You will also learn how to autoplace parts and then autoroute them.

For more detailed information on the features discussed in these chapters, refer to the *Multisim Help* or the *Ultiboard Help*.

2

Multisim Tutorial

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

Some of the described features may not be available in your edition of Circuit Design Suite.

Refer to `ni.com` for a list of the features available in your edition.

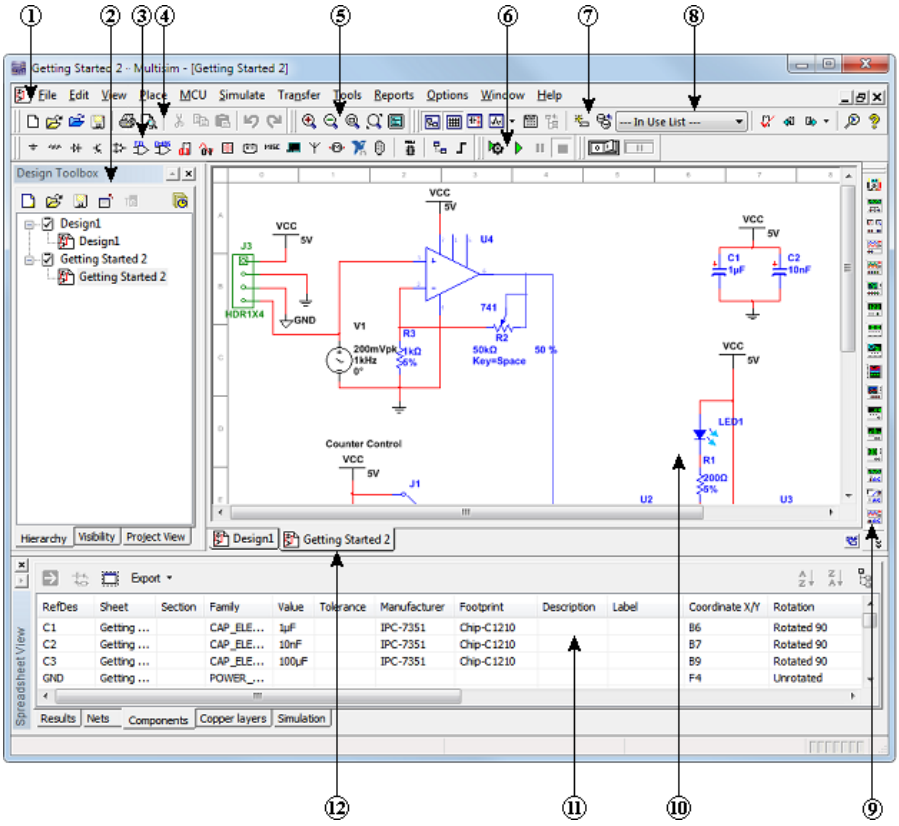
Introduction to the Multisim Interface

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

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

Multisim User Interface

Multisim's user interface includes the following elements:



1	Menu Bar	5	View Toolbar	9	Instruments Toolbar
2	Design Toolbox	6	Simulation Toolbar	10	Workspace
3	Component Toolbar	7	Main Toolbar	11	Spreadsheet View
4	Standard Toolbar	8	In Use List	12	Active Tab

Refer to the table below as needed:

	Element	Description
1	**Menu Bar**	Contains the commands for all functions.
2	**Design Toolbox**	Use to navigate through the different types of files in a project (schematics, PCBs, reports), view a schematic's hierarchy and show or hide different layers.
3	**Component** toolbar	Contains buttons that you use to select components from the Multisim databases for placement in your schematic.
4	**Standard** toolbar	Contains buttons for commonly-performed functions such as Save, Print, Cut, and Paste.
5	**View** toolbar	Contains buttons for modifying the way the screen is displayed.
6	**Simulation** toolbar	Contains buttons for starting, stopping, and other simulation functions.
7	**Main** toolbar	Contains buttons for common Multisim functions.
8	**In Use List**	Contains a list of all components used in the design.
9	**Instruments** toolbar	Contains buttons for each instrument.
10	Workspace	This is where you build your designs.
11	**Spreadsheet View**	Use for fast advanced viewing and editing of parameters including component details such as footprints, RefDes, attributes and design constraints.
12	Active tab	Indicates the design you are working on. Click another tab to switch.

Overview

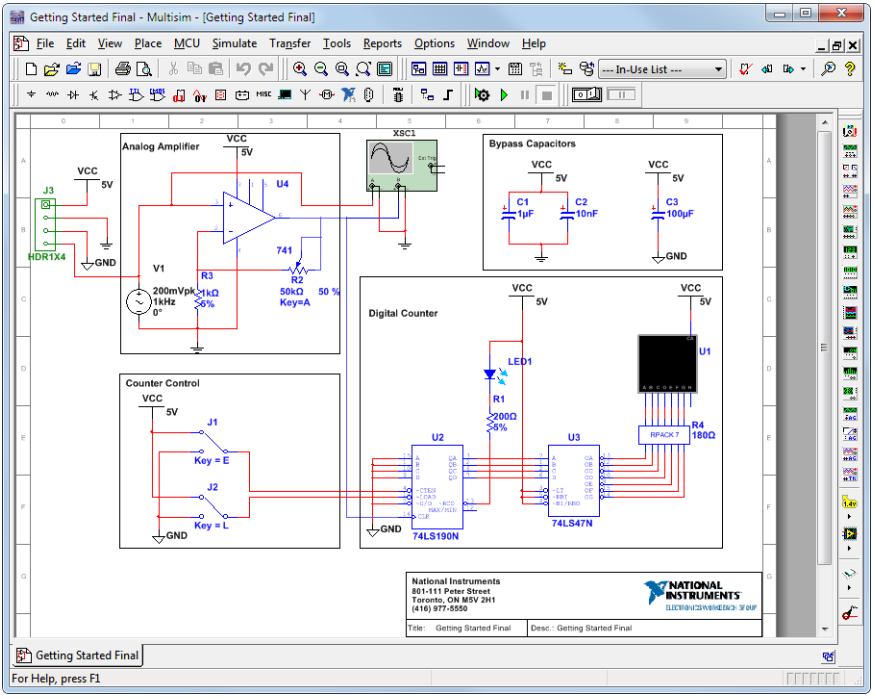
This tutorial leads you through the circuit design flow, from schematic capture, through simulation and analysis. After following the steps outlined on the following pages, you will have designed a circuit that samples a small analog signal, amplifies it and then counts the cycles on a simple digital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

Helpful tips are indicated by an icon in the left column, as in the tip below:

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

Schematic Capture

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



Creating the File

Complete the following steps to create the design file:

1. Launch Multisim.

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

2. Select **File»Save as** to display a standard Windows Save dialog.

3. Navigate to the location where you wish to save the file, enter `MyGettingStarted` as the **File name**, and click the **Save** button.

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

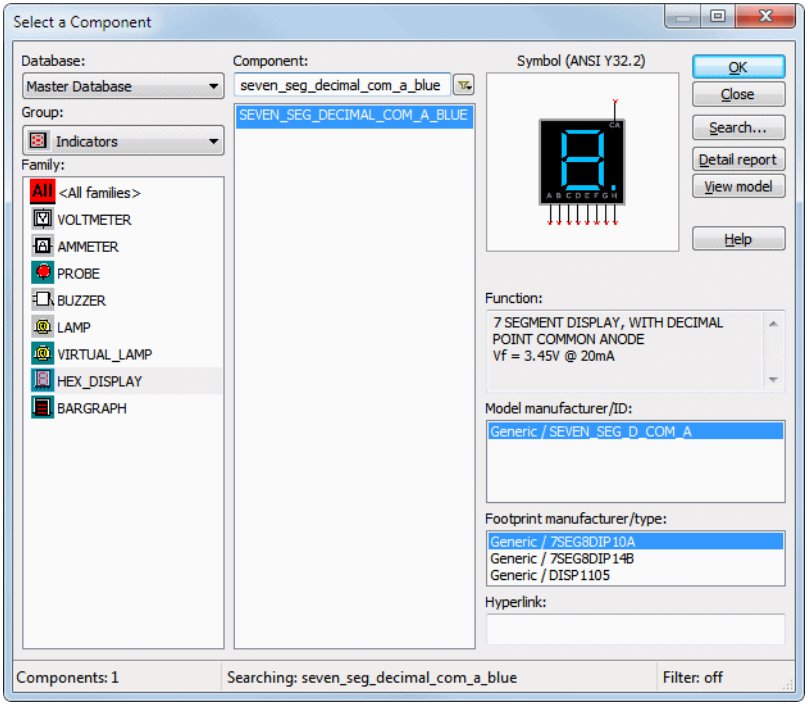
Placing the Components

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

1. Select **Place»Component** to display the **Select a Component** dialog box.
2. Select the **Indicators** component **Group** and the **HEX_DISPLAY** component **Family**.
3. Type `seven_seg_decimal_com_a_blue` in the **Component** field.

   As you type, the string appears in the **Searching** field at the bottom of the browser. Matches display in the **Component** list.
4. Click **OK** when the desired component displays as shown below.

   The component appears as a “ghost” on the cursor.

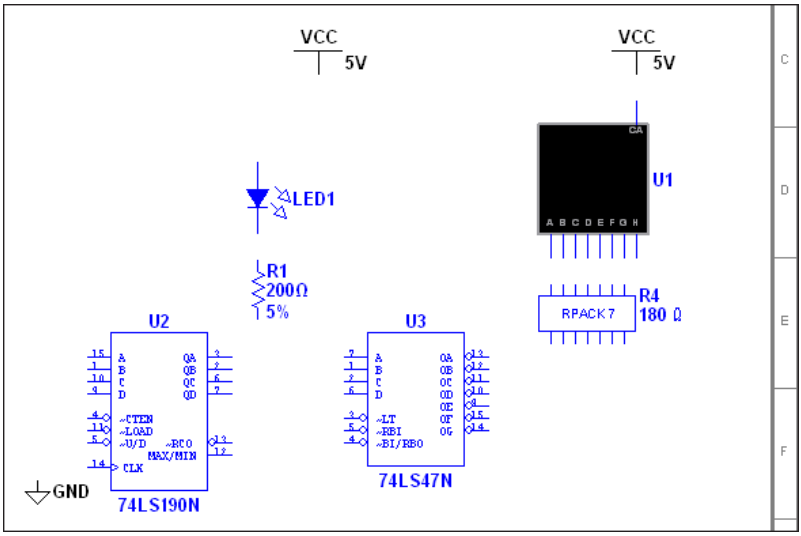


5. Move the cursor to the bottom-right of the workspace and click to place the component.

   Note that the Reference Designator for this component is `U1`.

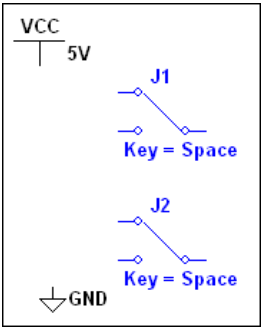
6. Place the remaining components in the Digital Counter area as shown below.

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



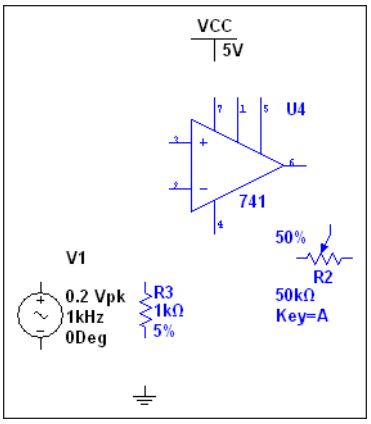
Tip While placing the 200 Ω resistor, press <Ctrl-R> to rotate it to a vertical orientation.

7. Place the components in the Counter Control section as shown below.

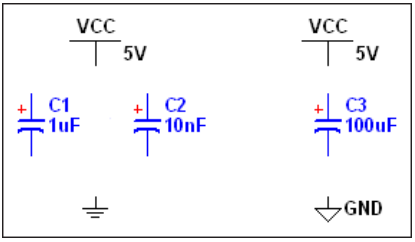


8. Right-click on each SPDT switch and select **Flip horizontally**.

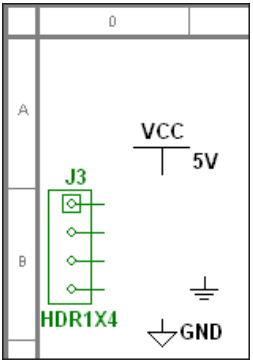
9. Place the components in the Analog Amplifier section as shown below, rotating as needed.



10. Double-click on the AC voltage source (V1), change **Voltage (Pk)** to 200 mV and click **OK** to close the dialog.
11. Place the components in the Bypass Capacitors section as shown below.



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



Component Locations

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

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

Component	Group	Family
VCC GND - DGND GROUND	Sources	POWER_SOURCES
LED1 - LED_blue	Diodes	DIODES_VIRTUAL
U1 - 7-segment display	Indicators	HEX_DISPLAY
U2 - 74LS190N U3 - 74LS47N	TTL	74LS
R1 - 200 Ω	Basic	RESISTOR
R2 - 1 k potentiometer	Basic	POTENTIOMETER
R3 - 1 k	Basic	RESISTOR
R4 - 10Line_Bussed	Basic	RPACK
J1, J2 - SPDT	Basic	SWITCH
U4 - 741	Analog	OPAMP
V1 - AC_VOLTAGE	Sources	SIGNAL_VOLTAGE_SOURCES
C1 - 1 µF C2 - 10 nF C3 - 100 µF	Basic	CAP_ELECTROLIT
J3 - HDR1X4	Connectors	HEADERS_TEST

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

Wiring the Design

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

Tip You can wire the design that you placed on the workspace or you can use `Getting Started 1` from the `Getting Started` folder (found inside the `samples` folder).

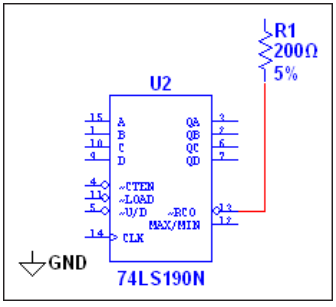
Complete the following steps to wire the design:

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

   A wire appears, attached to your cursor.

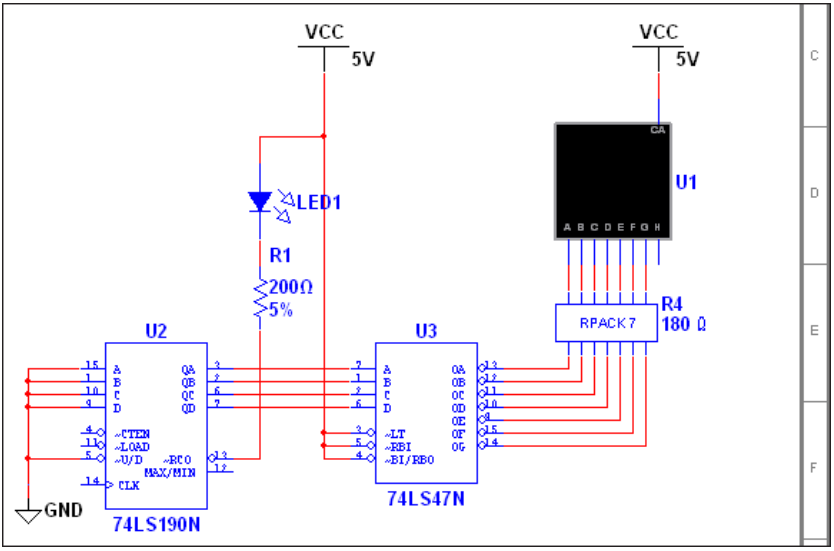
2. Click on a pin on the second component to finish the connection.

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



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

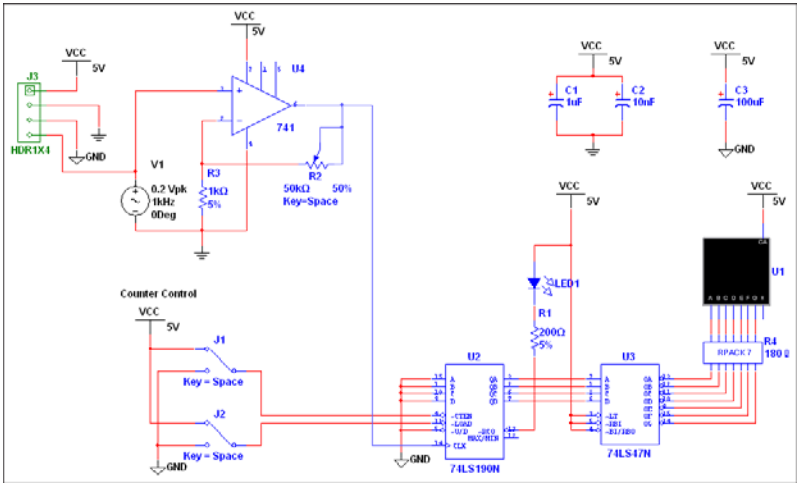
3. Finish wiring the Digital Counter section as shown below.



Tip Use **Bus Vector Connect** to wire multi-pinned devices like U3 and R4 together in a bus. Refer to the *Multisim Help* for details.

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

4. Finish wiring the design as shown below.



Simulation

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

Virtual Instrumentation

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

Tip You can also use `Getting Started 2` from the `Getting Started` folder (found in the `samples` folder).

1. Set up the interactive keys for J1, J2 and R2:
   a. Double-click on each and select the **Value** tab.
   b. Select “E” for J1 and “L” for J2 in the **Key for toggle** field.
   c. Select “A” in the **Key** field for R2.

Note J1, J2 and R2 are interactive components.

2. Press <E> to enable the counter.

   Or

   Click on the widened switch arm that appears when you hover the cursor over J1.

   Enable is Active Low.
3. Select **Simulate»Instruments»Oscilloscope** to place the oscilloscope on the workspace.

4. Wire the instrument as shown in step 7.

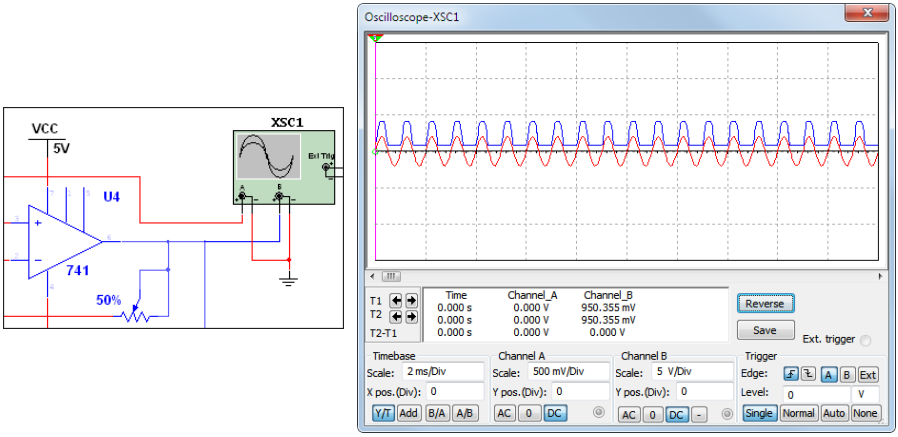
Tip To differentiate between traces on the oscilloscope, right-click on the wire connected to the scope's **B** input and select **Segment color** from the context menu that displays. Select a color that differs from the wire connected to the **A** input, for example blue. (Changing wire color or performing other editing functions cannot be done while simulation is running).

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

The output of the op-amp appears on the scope.

7. Adjust the **Timebase** to 2 ms/Div and Channel A's **Scale** to 500 mV/Div.

The following displays on the oscilloscope:



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

8. Do the following:
   a. Press <E> while the simulation is running to enable or disable the counter. Enable is Active Low.
   b. Press <L> to load zeros into the counter. Load is Active Low.
   c. Press <Shift-A> to observe the effect of decreasing the potentiometer's setting. Repeat, pressing <A> to increase.

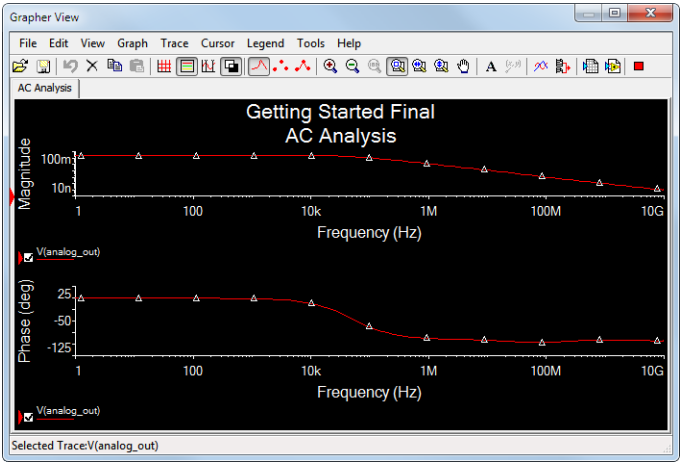
Tip Instead of pressing the above-mentioned keys, you can directly manipulate the interactive components on the schematic with your mouse.

Analysis

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

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

1. Double-click on the wire that is attached to pin 6 of the op-amp.
   The **Net Properties** dialog box displays.
2. Change the **Preferred net name** to `analog_out`.
3. Select **Simulate»Analyses»AC analysis»Output** tab.
4. Highlight `V(analog_out)` in the **Variables in circuit** (left) column and click **Add**.
   `V(analog_out)` moves to the **Selected variables for analysis** (right) column.
   This indicates that the voltage at node `V(analog_out)` will be displayed after simulation.
5. Click **Simulate**.
   The results of the analysis appear in the **Grapher**.

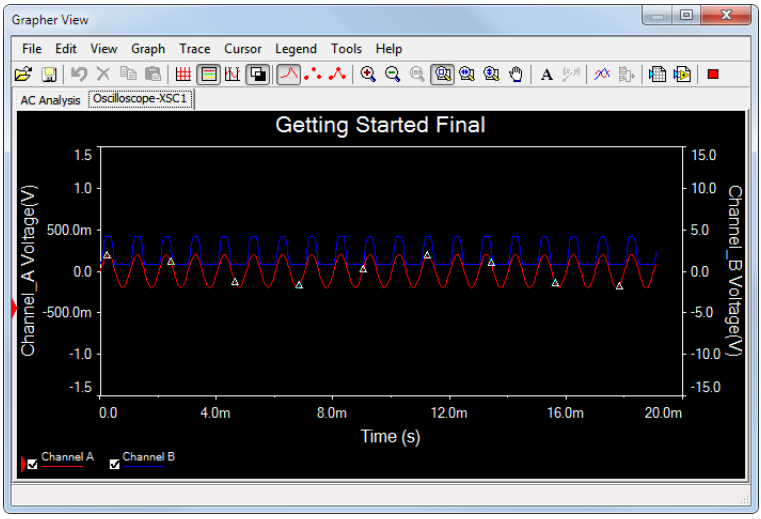


The Grapher

The **Grapher** is a multi-purpose display tool that lets you view, adjust, save and export graphs and charts. It is used to display the results of all Multisim analyses in graphs and charts, and graphs of traces for some instruments (for example, the oscilloscope).

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

1. Run the simulation with the oscilloscope as described earlier.
2. Select **View»Grapher**.



The Postprocessor

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

Reports

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

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

Bill of Materials

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

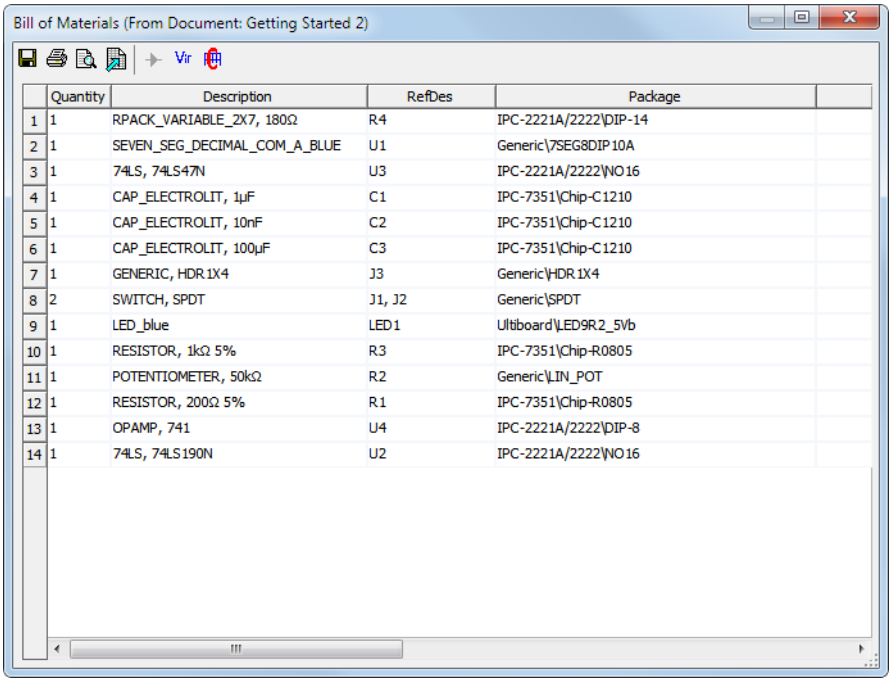
Information provided for each component includes:

- quantity required.
- description, including the type of component (example: resistor) and value (example: 5.1 kΩ).
- Reference Designator.
- package or footprint name.

Complete the following steps to create a BOM (bill of materials) for your design:

1. Select **Reports»Bill of Materials**.

   The report appears, similar to this:



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

Click the **Send to printer** button to print the Bill of Materials. A standard **Print** dialog box appears, where you choose the printer, number of copies, and so on.

Click the **Save to text file** button to save the Bill of Materials. A standard Windows file save dialog box appears, where you specify the path and file name.

Because the Bill of Materials is primarily intended to assist in procurement and manufacturing, it includes only “real” components—it excludes components that are not real or available for purchase, such as sources or virtual components. Components without assigned footprints do not appear in the Bill of Materials.

To see a list of components in your design that are not “real” components, click the **Show virtual components** button. A separate view appears, showing these components only.

Refer to the *Multisim Help* for detailed information on this and other reports.

3

Ultiboard Tutorial

The tutorial in this chapter places the parts and traces for the circuit described in the Multisim Tutorial chapter.

Some of the described features may not be available in your edition of Ultiboard.

Refer to `ni.com` for a list of the features available in your edition.

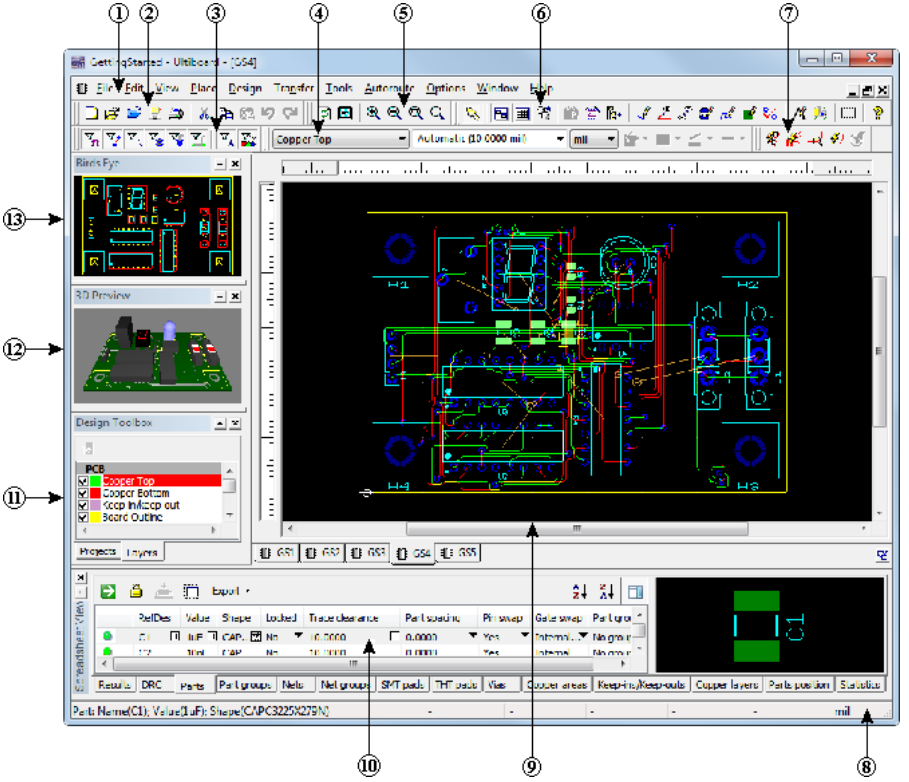
Introduction to the Ultiboard Interface

Ultiboard is the PCB layout 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 design flow.

Ultiboard is used to lay out and route printed circuit boards, perform certain basic mechanical CAD operations, and prepare boards for manufacturing. It also provides automated parts placement and wire routing.

Ultiboard User Interface

Ultiboard's user interface is made up of several elements.



1	Menu Bar	5	View Toolbar	8	Status Bar	11	Design Toolbox
2	Standard Toolbar	6	Main Toolbar	9	Workspace	12	3D Preview
3	Select Toolbar	7	Autoroute Toolbar	10	Spreadsheet View	13	Birds Eye View
4	Draw Settings Toolbar						

Refer to the table below as needed:

	Element	Description
1	**Menu Bar**	Contains the commands for all functions.
2	**Standard** toolbar	Contains buttons for commonly-performed functions such as Save, Print, Cut, and Paste.
3	**Select** toolbar	As you add more parts and traces to a board, it can become difficult to select only those which you want to use. This toolbar contains buttons used to control selections.
4	**Draw Settings** toolbar	This is where you select the layer, thickness and unit of measure of a line or object that is being drawn. It also contains buttons for functions that control the appearance of lines and shapes drawn on a layer.
5	**View** toolbar	Contains buttons for modifying the way the screen is displayed.
6	**Main** toolbar	Contains buttons for common board design functions.
7	**Autoroute** toolbar	Contains autorouting and part placement functions.
8	**Status Bar**	Displays useful and important information.
9	Workspace	This is where you lay out the PCB.
10	**Spreadsheet View**	This allows fast advanced viewing and editing of parameters including part details such as shapes, Reference Designators, attributes and design constraints.
11	**Design Toolbox**	Use to navigate through project files and show, hide or dim different elements of the design.
12	**3D Preview**	Displays a three-dimensional preview of the board.
13	**Birds Eye View**	Displays an “aerial view” of the design and lets you easily navigate around the workspace.

Opening the Tutorial

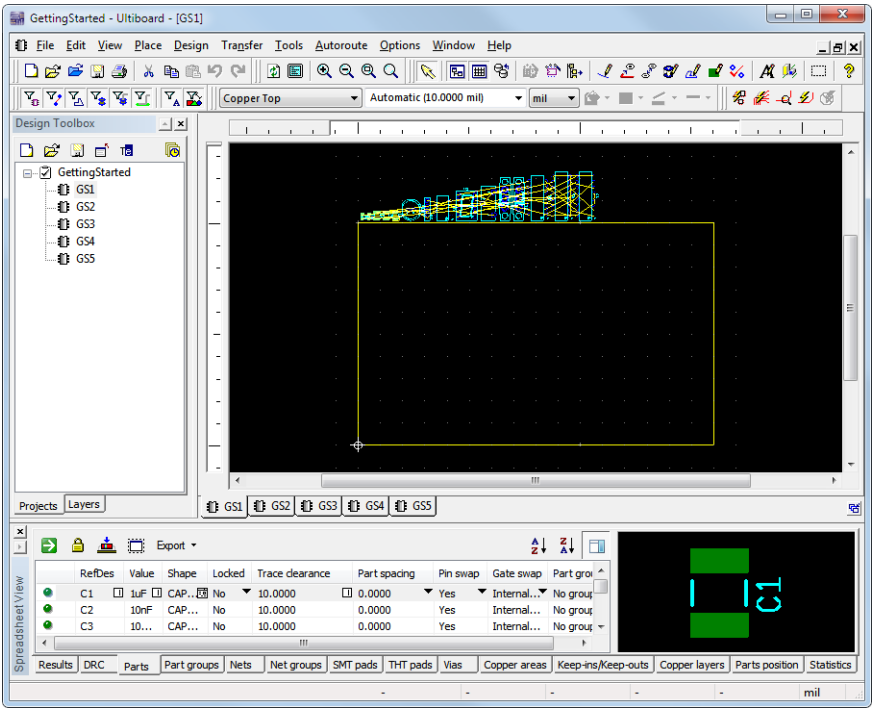
Complete the following steps to open the tutorial file:

1. Launch Ultiboard, select **File»Open samples** and double-click on the `GettingStarted` folder to open it.
2. Select `GettingStarted` and click **Open**.

   The project file is loaded into Ultiboard.

Tip For instructions on exporting a design from Multisim to Ultiboard, refer to the *Multisim Help* and the *Ultiboard Help*.

3. Select design `GS1`.



Tip To select a design from the project (for example, `GS1`), click on its tab, or click on its name in the **Projects** tab of the **Design Toolbox**.

Creating a Board Outline

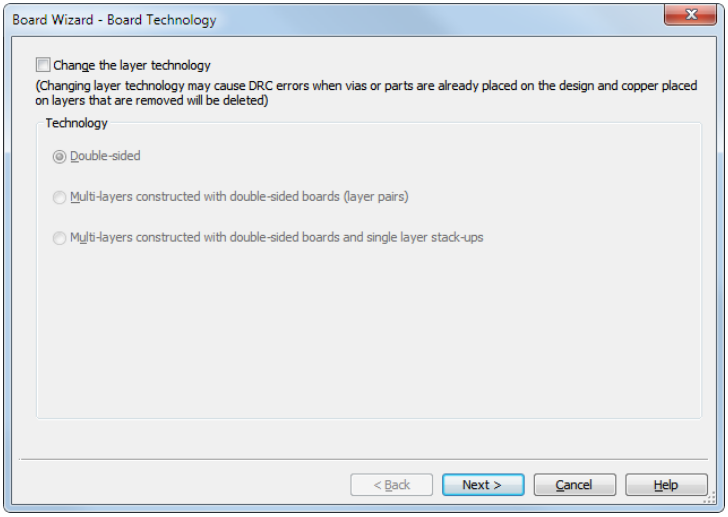
There is already a board outline, however, you can create one that is a more suitable size for the parts in this design in one of the following ways:

- Draw a board outline using the drawing tools.
- Import a DXF file.
- Use the **Board Wizard**.

Complete the following steps to experiment with the **Board Wizard**:

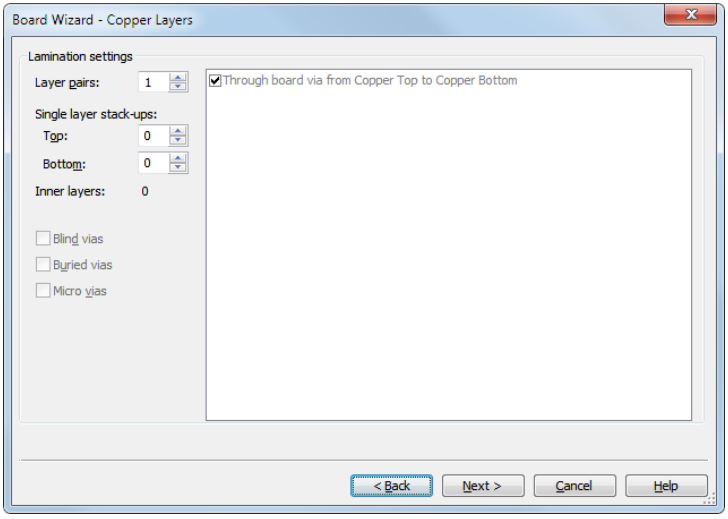
1. Double-click on **Board Outline** in the **Layers** tab to make it the active layer.
2. Click on the existing board outline in the `GS1` design and press `Delete` on the keyboard.

3. Choose **Tools»Board wizard**.



4. Select **Change the layer technology** to make the **Technology** options available.
5. Choose **Multi-layers constructed with double-sided boards and single layer stack-ups**, and click **Next**.

   The next dialog box (**Copper Layers**) is where you define the **Lamination settings** for the board. (For this tutorial you will not change settings).



6. Click **Next**.

   In the **Board Wizard - Shape of Board** dialog box:

   a. Confirm that **Units** is set to **mil**.

   b. Confirm that the **Reference point** is set to **Bottom-left** for **Alignment**.

   c. Confirm that the **Rectangular** option is selected in **Board shape and size**.

   d. Set the **Width** to 3000 and the **Height** to 2000.

   e. Set the **Clearance** to 5.00000.

      This is the distance from the edge of the board that is to be kept free of parts or any other elements.

7. Click **Finish**.

   The board outline is placed on your design.

Note For complete details on the **Board Wizard**, refer to the *Ultiboard Help*.

Complete the following steps to move the board outline:

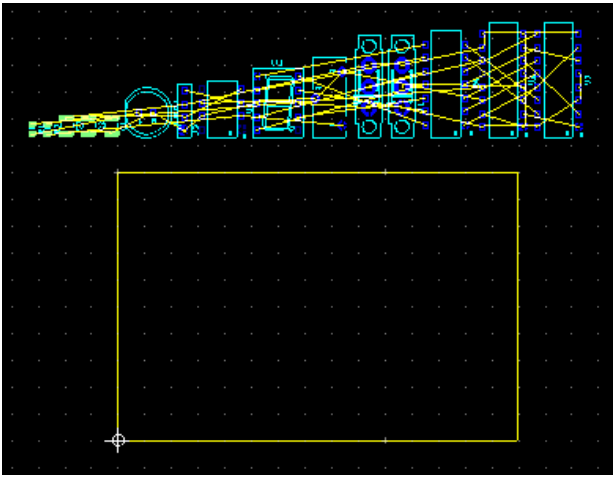
1. Double-click on **Board Outline** in the **Layers** tab.
2. Click anywhere on the board outline in the workspace and drag the board to a location just below the row of parts.

Complete the following steps to change the reference point:

1. Select **Design»Set reference point**.

   The reference point is attached to your cursor.

2. Move the cursor the the lower-left corner of the board outline and click to place it.



Placing Parts

You can place parts on your GS1 design in several different ways:

- Select one or more parts from outside the board outline and drag them into place.
- Use the **Parts** tab in the **Spreadsheet View** to locate parts and place them.
- Select and place parts from the database.

Tip Use the **Place»Unplace parts** command to remove all non-locked parts from the PCB and experiment with a different placement technique.

Dragging Parts from Outside the Board Outline

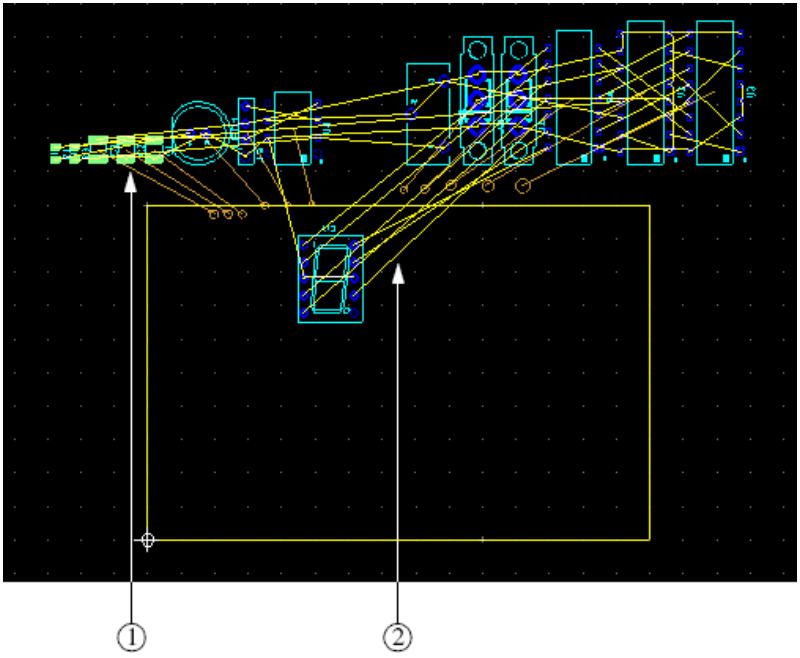
By default, parts are placed outside the board outline when you open a netlist from Multisim or another schematic capture program.

Complete the following steps to drag U1 from outside the board outline:

1. Double-click the **Copper Top** layer in the **Design Toolbox** to make it the active layer.
2. Find U1 in the collection of parts outside the board outline. To make this easier, use the mouse wheel to zoom in until you can see U1.

Tip You can also search for a part with the **Edit»Find** command. While this command works much like a Find function in other applications, it also allows you to search for a part by name, number, shape, value, or by all variables. Refer to the *Ultiboard Help* for details.

3. Click on U1 (the 7-segment display) and drag it to the location shown in the figure below.



1	Force Vector (orange line)	2	Ratsnest (yellow line)

U1 remains selected. This is an important point for Ultiboard that holds throughout the application—you need to explicitly end any particular action.

Click elsewhere on the workspace to de-select the part. Right-clicking also ends the current action.

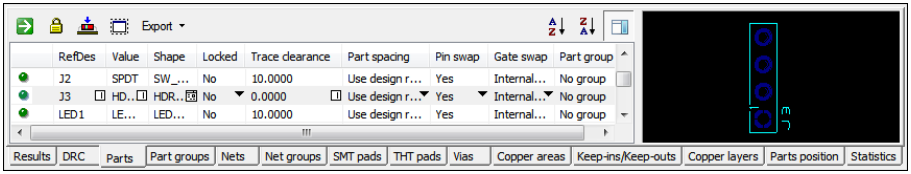
4. Go to the **Parts** tab in the **Spreadsheet View** and scroll to U1.

Notice that the green light beside the part is slightly brighter—this indicates that the part has been placed.

Dragging Parts from the Parts Tab

Complete the following steps to drag parts from the **Parts** tab:

1. In the **Parts** tab, scroll down to J3.



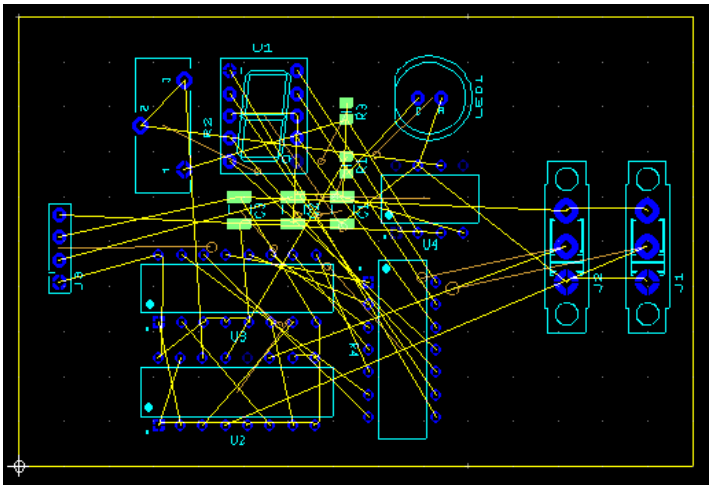
2. Click on J3 in the table and drag it from the **Parts** tab onto the workspace.
   J3 is attached to the mouse pointer.
3. Drop J3 on the left edge of the board, approximately in the middle.
   In the **Parts** tab J3's green light is slightly brighter, indicating that the part has been placed.

Placing the Tutorial Parts

Using any method or combination of methods, lay out the design like the illustration below.

Tip You can open the next design file in the project, `GS2`, which has already been set up this way.

The design should look like this:



Placing Parts from the Database

In addition to placing parts imported as part of your design, you can place parts directly from the database. The following uses this method to place the mounting holes.

Complete the following steps to place parts from the database:

1. Choose **Place»From database**.

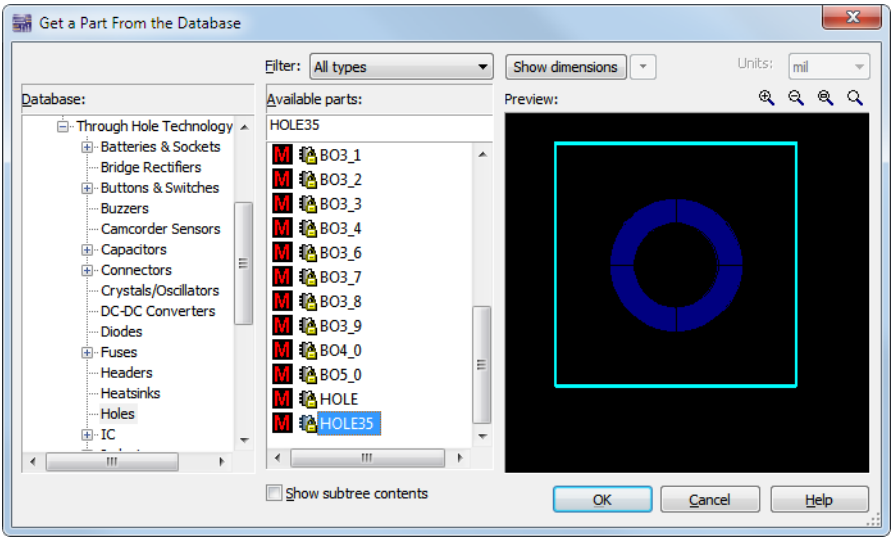
   

   The **Get a Part From the Database** dialog box opens.
2. In the **Database** panel, expand the **Ultiboard Master»Through Hole Technology Parts** category and navigate to the **Holes** category.

The parts appear in the **Available parts** panel.

3. In the **Available parts** panel, select the **HOLE35** part.

   The part displays in the **Preview** panel.



4. Click **OK**.

   The **Get a Part From the Database** dialog box disappears, and the **Enter Reference Designation for Part** dialog box displays.

5. Enter the **RefDes for shape HOLE35** (H1) and **Value** (HOLE) and click **OK**.

6. Move the pointer over the board.

   The part is attached to the pointer.

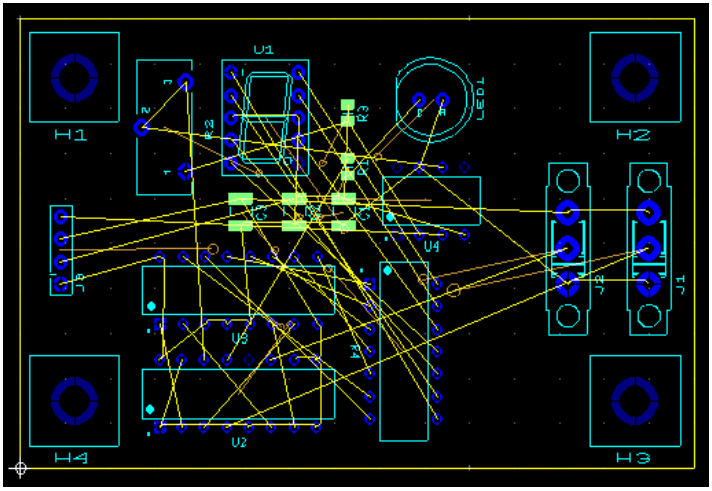
7. Move the hole to the top-left corner, and click to drop it on the board.

   The **Enter Reference Designation for Part** dialog box reappears, with the **RefDes for shape HOLE35** automatically incremented to H2.

8. Enter the value (HOLE) and click **OK** to place the next mounting hole in the top right corner.

9. Repeat to place H3 in the bottom right corner, and H4 in the bottom left corner.

10. Click **Cancel** to stop, and click **Cancel** again.
    The **Get a Part From the Database** dialog box closes.



Moving Parts

You can use the same methods for moving parts as you do for placing them.

To select a part already on the board, click on it.

To specify the X/Y coordinates to where the selected part is to move, press * on the numeric keypad.

Or, select a placed part in the **Parts** tab (indicated by a bright green light beside it) and drag it to a new location.

Tip A part's label and pads are separate elements from its shape. When selecting a part on the board, be sure to select the whole part, not just the label or pads. Use the **Selection Filters** to assist with this. Refer to the *Ultiboard Help* for more information.

Tip Once a part is selected, you can also move it around on the board by pressing the arrow keys on your keyboard.

To select a group of parts and move them together, do one of the following:

- Hold <Shift> and click on more than one part.
- Drag a box around several parts.

All the selected parts will move together when you drag the pointer.

Tip These are temporary groups—once you select another part, the group connection is lost. To make a group that remains until you remove it, you can use the **Group Editor**. Refer to the *Ultiboard Help* for details.

You can use the **Edit»Align** commands to align the edges of selected parts or to space them relative to each other.

Use the **Edit»Align** commands to align the mounting holes you just placed:

1. Select H1 and hold <Shift> to select H2.
2. Choose **Edit»Align»Align top**.
   If H2 was not originally placed exactly in line with H1, you will see it move.
3. Click on an empty space on the board, then select H2 and H3.
4. Choose **Edit»Align»Align right**.
5. Continue in this manner to align the bottoms of H3 and H4, and the left sides of H1 and H4.

Placing Traces

You have the following options for placing traces:

- Manual trace.
- Follow-me trace.
- Connection machine trace.

A manual trace is placed exactly as you specify, even running through a component or trace if that is the path you set.

A follow-me trace automatically draws a valid trace between the pins you select with your mouse movements—you can move from pin to pin, leaving a valid trace.

A connection machine trace automatically joins two pins by the most efficient route, though you have the option of changing it.

As you place a trace, and before you click to fix it in place, you can always remove a segment by backing up over it.

Each time you click while placing a manual trace, or each time a follow-me trace or connection machine trace changes direction, a separate segment of that trace is created.

Tip When performing operations on traces, be sure to select either the appropriate segment or the whole trace.

Placing a Manual Trace

You can continue with the design you have been working on, or open `GS3`.

Be sure you are on the **Copper Top** layer before beginning—**Copper Top** must be highlighted in red in the **Layers** tab of the **Design Toolbox**.

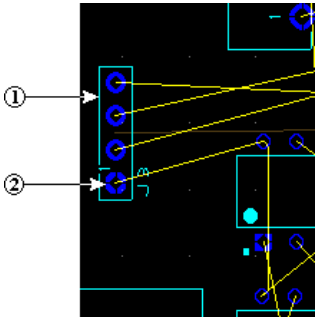
Tip If necessary, press <F7> to show the whole design.

Complete the following steps to place a trace manually:

1. Choose **Place»Line**.

Tip The **Line** command creates a line on any layer. The results differ depending on the layer selected. For example, if the selected layer is silkscreen, a line is created on the silkscreen layer of the PCB. If the selected layer is a copper layer, then the "line" is actually a trace.

2. Locate J3, toward the left-hand part of the board and find the start pin shown below:



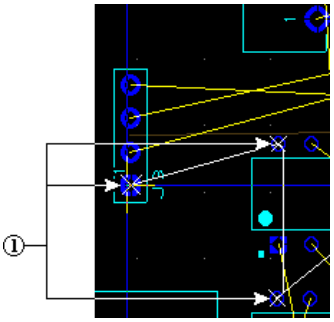
1	Part J3	2	Start Pin

Tip You can turn off or dim the **Force Vectors** to see the nets more clearly. Do this using the **Force Vectors** checkbox and color chooser in the **Layers** tab of the **Design Toolbox**. Refer to the *Ultiboard Help* for more information about **Force Vectors**.

Tip If you have trouble locating the part, use the **Find** function of the **Parts** tab. Select the part in the **Parts** tab, then click the **Find and select the part** button. The part is shown in the workspace. If necessary, zoom in further using the mouse wheel.

3. Click on the pin specified in the above step.

   Ultiboard highlights all the pins that are part of the same net as the pin you clicked on with an X. These indicate which pins to connect to match the connectivity from your schematic.

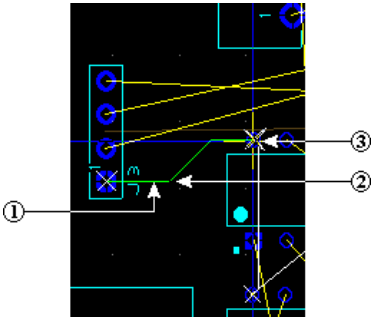


1	Pins in the Same Net

4. Move the cursor in any direction.

   A green line (the trace) is attached to the selected pin. Each time you click, you anchor the trace segment, as shown in the figure below (2).

5. Click to anchor the trace and on the destination pin as shown below.



1	Trace	2	Click to anchor trace	3	Destination Pin

6. Right-click and choose **Cancel** to stop placing traces.
7. Click the **Select** button on the **Main** toolbar to exit line-placing mode.

Placing a Follow-me Trace

Complete the following steps to place a follow-me trace:

1. Choose **Place»Follow-me**.

2. Click on the top pin of J3.
3. Click on the pin indicated by the “X” on U4.

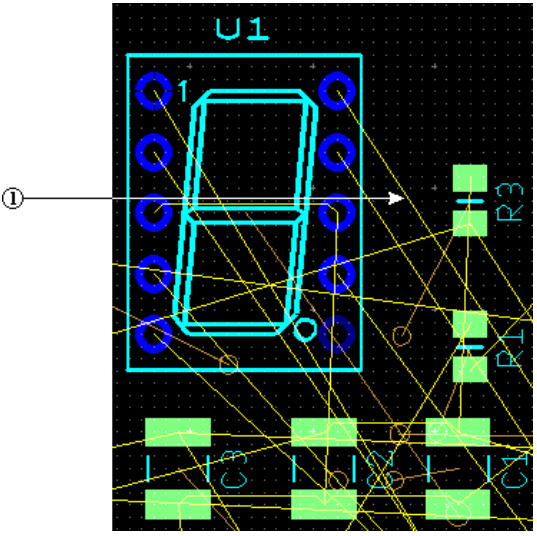
   Ultiboard draws the trace for you.

Placing a Connection Machine Trace

Complete the following steps to place a **Connection Machine** trace:

1. Choose **Place»Connection Machine**.

2. Click on the segment of the ratsnest indicated below.

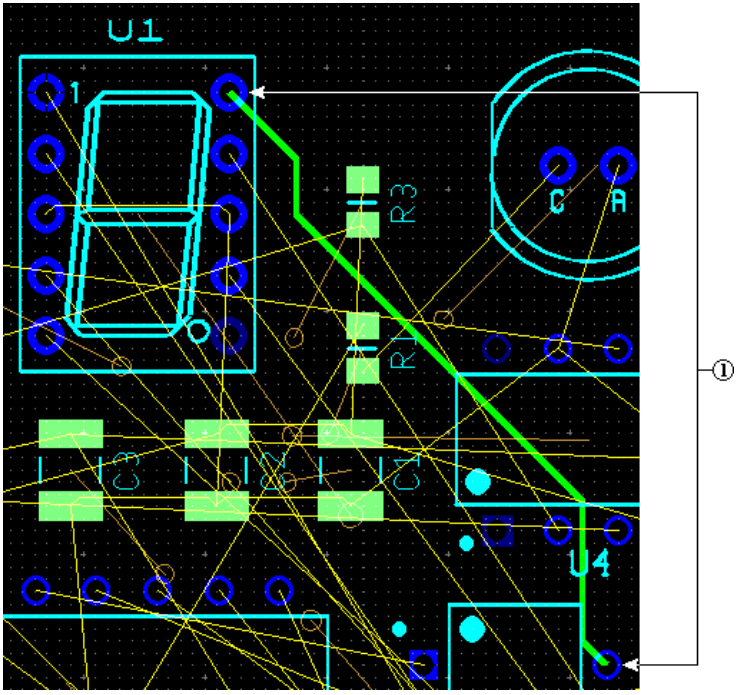


1	Click Ratsnest

3. Move your cursor.

   As the cursor moves, Ultiboard displays various trace placement options.

4. Click to fix the trace when you see the route you want. There is no need to click on the ratsnest or the destination pin.



1 Trace Segments Appear Between Pins

5. Right-click to end trace placement.

Auto Part Placement

As well as placing parts as described earlier in this chapter, you can use Ultiboard's advanced automatic part placement functionality.

Tip Before autoplacing parts, pre-place and lock any parts that you do not wish to be moved during the autoplacement process. (The mounting holes, and U1, J1, J2, J3, and LED 1 in `GS5` have been pre-placed and locked). For details on locking parts, refer to the *Ultiboard Help*.

Complete the following steps to autoplace the parts in `Getting Started`:

1. Open the `GS5` design in Ultiboard.
2. Select **Autoroute»Autoplace parts**.

   The parts are placed on the circuit board.

Autorouting Traces

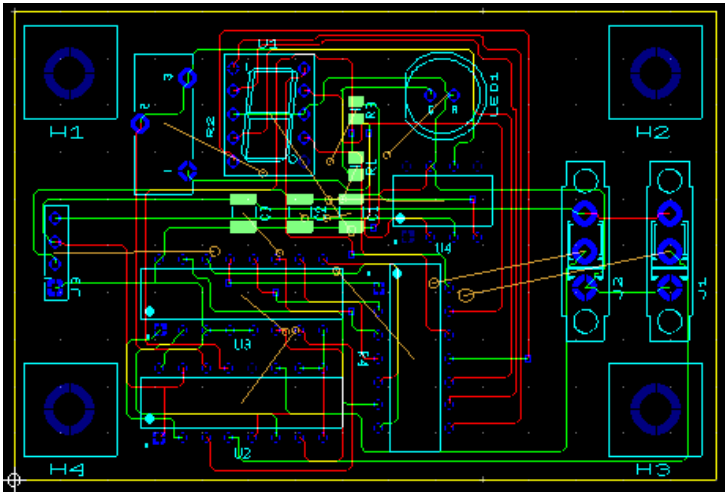
You can place traces in Ultiboard using the methods described earlier in this chapter, or automatically route the traces as described below.

Complete the following steps to autoroute the traces in `Getting Started`:

1. Open the `GS3` design in Ultiboard.
2. Select **Autoroute»Start/resume autorouter**.

   The workspace switches to **Autorouter Mode** and trace autorouting begins.

   As autorouting proceeds, you will see traces being placed on the board. When autorouting is complete, **Autorouter Mode** closes and you are returned to the workspace.
3. Select **Autoroute»Optimize routing** to optimize trace placement.



The autorouter can be stopped at any time and you can make manual changes as desired. When you restart the autorouter, it will continue with the changes you made. Remember to lock any traces that you have placed manually and do not wish to be moved by the autorouter.

Tip Use the **Routing Options** dialog box to modify autoplacement and autorouting options. Refer to the *Ultiboard Help* for details.

Preparing for Manufacturing/Assembly

Ultiboard can produce many different output formats to support your production and manufacturing needs. The following sections explain the functions performed to output your board for production and documentation purposes.

Cleaning up the Board

Before sending the board for manufacturing, you should clean up any open trace ends (trace segments that do not have any terminating connections in the design) and unused vias.

To delete any open trace ends, open the `GS4` design and choose **Edit»Copper delete»Open trace ends**.

To delete any unused vias, select **Design»Remove unused vias**. This deletes all vias that do not have any trace segments or copper areas connected to them.

Adding Comments

Comments can be used to show engineering change orders, to facilitate collaborative work among team members, or to allow background information to be attached to a design.

You can pin a comment to the workspace, or directly to a part. When a part with an attached comment is moved, the comment also moves.

Refer to the *Ultiboard Help* for details.

Exporting a File

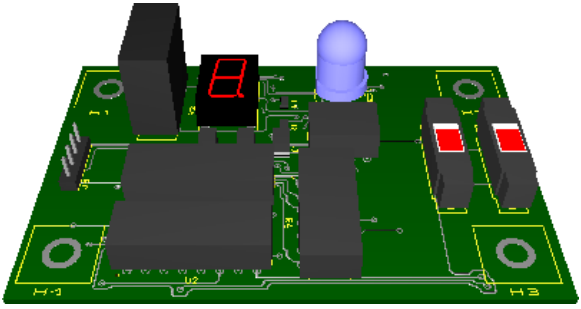
You can export a number of file types, including Gerber. An exported file contains complete information describing how a finished board is to be manufactured.

Refer to the *Ultiboard Help* for details.

Viewing Designs in 3D

You can use the **3D View** to see what the board looks like in three dimensions at any time during the design.

Refer to the *Ultiboard Help* for details.



A

Technical Support and Professional Services

Log in to your National Instruments ni.com User Profile to get personalized access to your services. Visit the following sections of ni.com for technical support and professional services:

- **Support**—Technical support at ni.com/support includes the following resources:
  - **Self-Help Technical Resources**—For answers and solutions, visit ni.com/support for software drivers and updates, a searchable KnowledgeBase, product manuals, step-by-step troubleshooting wizards, thousands of example programs, tutorials, application notes, instrument drivers, and so on. Registered users also receive access to the NI Discussion Forums at ni.com/forums. NI Applications Engineers make sure every question submitted online receives an answer.
  - **Standard Service Program Membership**—This program entitles members to direct access to NI Applications Engineers via phone and email for one-to-one technical support, as well as exclusive access to self-paced online training modules at ni.com/self-paced-training. All customers automatically receive a one-year membership in the Standard Service Program (SSP) with the purchase of most software products and bundles including NI Developer Suite. NI also offers flexible extended contract options that guarantee your SSP benefits are available without interruption for as long as you need them. Visit ni.com/ssp for more information.

    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 training and certification program information. You can also 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.

You also can visit the Worldwide Offices section of ni.com/niglobal to access the branch office Web sites, which provide up-to-date contact information, support phone numbers, email addresses, and current events.

Index

T

U

V

W