NI Circuit Design Suite

Getting Started with NI Circuit Design Suite

August 2010
374482E

NATIONAL INSTRUMENTS™
ELECTRONICS WORKBENCH GROUP

Worldwide Technical Support and Product Information

ni.com

National Instruments Corporate Headquarters

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

Worldwide Offices

Australia 1800 300 800, Austria 43 662 457990-0, Belgium 32 (0) 2 757 0020, Brazil 55 11 3262 3599, Canada 800 433 3488, China 86 21 5050 9800, Czech Republic 420 224 235 774, Denmark 45 45 76 26 00, Finland 358 (0) 9 725 72511, France 01 57 66 24 24, Germany 49 89 7413130, India 91 80 41190000, Israel 972 3 6393737, Italy 39 02 41309277, Japan 0120-527196, Korea 82 02 3451 3400, Lebanon 961 (0) 1 33 28 28, Malaysia 1800 887710, Mexico 01 800 010 0793, Netherlands 31 (0) 348 433 466, New Zealand 0800 553 322, Norway 47 (0) 66 90 76 60, Poland 48 22 328 90 10, Portugal 351 210 311 210, Russia 7 495 783 6851, Singapore 1800 226 5886, Slovenia 386 3 425 42 00, South Africa 27 0 11 805 8197, Spain 34 91 640 0085, Sweden 46 (0) 8 587 895 00, Switzerland 41 56 2005151, Taiwan 886 02 2377 2222, Thailand 662 278 6777, Turkey 90 212 279 3031, United Kingdom 44 (0) 1635 523545

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

Important Information

Warranty

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

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

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

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

Copyright

Trademarks

Patents

WARNING REGARDING USE OF NATIONAL INSTRUMENTS PRODUCTS

Conventions

The following conventions are used in this manual:

» — The » symbol leads you through nested menu items and dialog box options to a final action. The sequence **Tools»Clear ERC Markers»Entire design** directs you to pull down the **Tools** menu, select the **Clear ERC Markers** item, and select **Entire design** from the resulting dialog box.

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

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

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

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

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

Contents

Chapter 4
Multisim MCU Tutorial

Appendix A
Technical Support and Professional Services

Index

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 the *NI Circuit Design Suite Release Notes* 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 circuit described in the Multisim Tutorial chapter. You will also learn how to autoplace parts and then autoroute them.
- *Multisim MCU Tutorial*—Leads you through the process of simulating and debugging a circuit that contains a microcontroller.

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

2

Multisim Tutorial

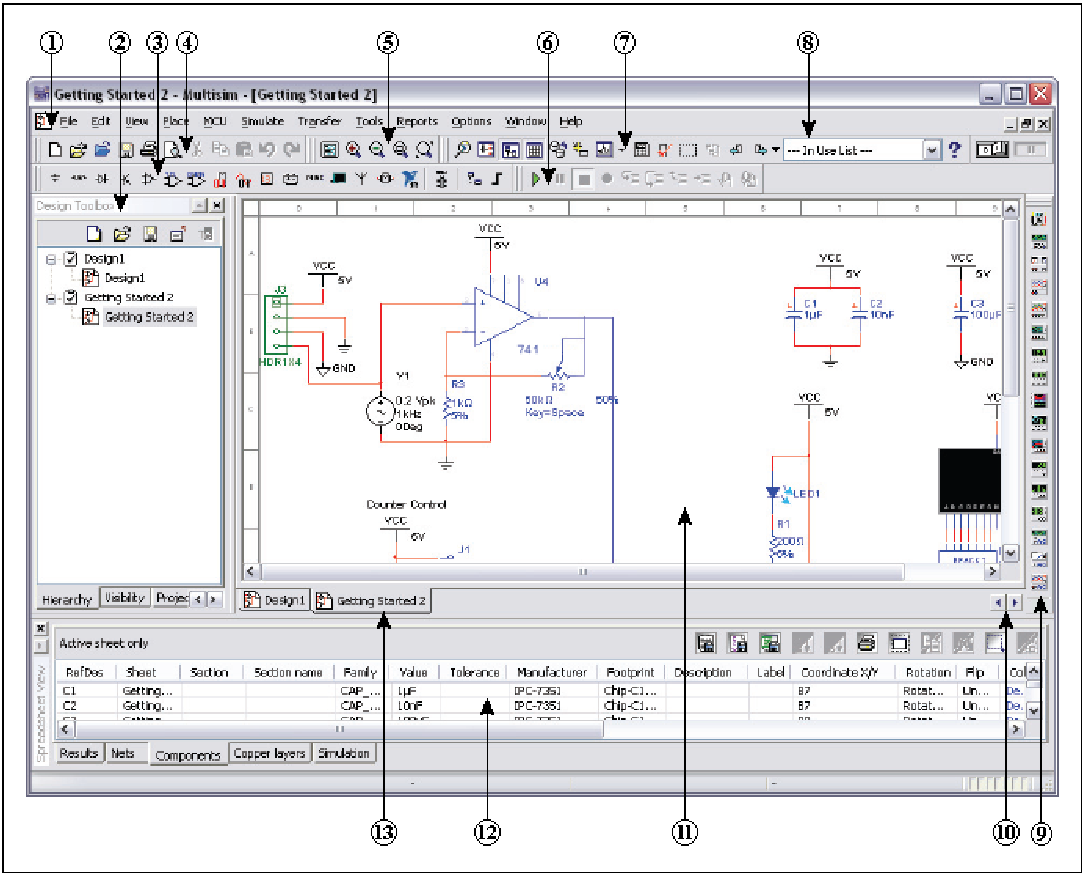
This chapter contains a tutorial that introduces you to Multisim and its many functions.

Some of the described features may not be available in your edition of Circuit Design Suite. Refer to the *NI Circuit Design Suite Release Notes* 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 assists you in carrying 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's user interface consists of the following basic elements:



1	Menu Bar	5	View Toolbar	8	In Use List	11	Circuit Window
2	Design Toolbox	6	Simulation Toolbar	9	Instruments Toolbar	12	Spreadsheet View
3	Component Toolbar	7	Main Toolbar	10	Scroll Left/Right	13	Active Tab
4	Standard Toolbar						

The **Menu Bar** is where you find commands for all functions.

The **Design Toolbox** is where you navigate through the different types of files in a project (schematics, PCBs, reports), view a schematic's hierarchy and show or hide different layers.

The **Component** toolbar contains buttons that you use to select components from the Multisim databases for placement in your schematic.

The **Standard** toolbar contains buttons for commonly-performed functions such as Save, Print, Cut, and Paste.

The **View** toolbar contains buttons for modifying the way the screen is displayed.

The **Simulation** toolbar contains buttons for starting, stopping, and other simulation functions.

The **Main** toolbar contains buttons for common Multisim functions.

The **In Use List** contains a list of all components used in the design.

The **Instruments** toolbar contains buttons for each instrument.

The **Circuit Window** (or workspace) is where you build your circuit designs.

The **Spreadsheet View** allows fast advanced viewing and editing of parameters including component details such as footprints, RefDes, attributes and design constraints. You can change parameters for some or all components in one step and perform a number of other functions.

Overview

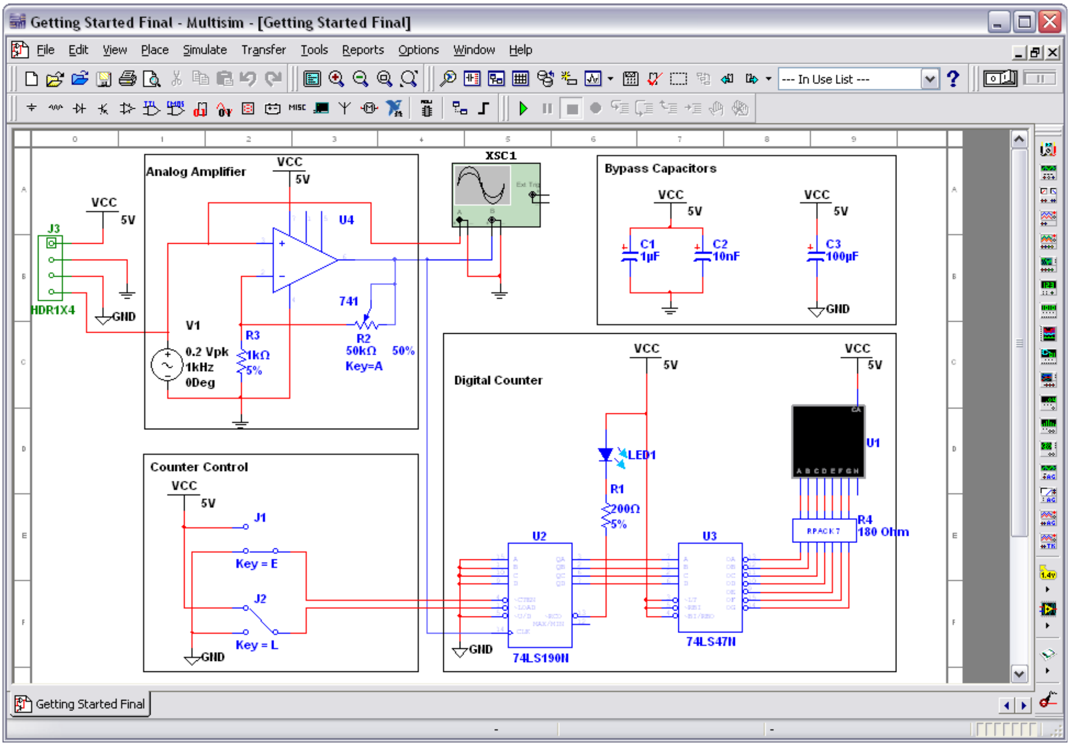
This tutorial leads you through the circuit design flow, from schematic capture, through simulation and analysis. After following the steps outlined on the following pages, you will have designed a circuit that samples a small analog signal, amplifies it and then counts the occurrences of the signal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

Helpful tips are indicated by the presence of 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 on the **Help** button in a dialog box.

Schematic Capture

In this section, you will place and wire the components in the circuit shown below.



Opening and Saving the File

Complete the following step to launch Multisim:

1. Select **Start»All Programs»National Instruments»Circuit Design Suite 11.0»Multisim 11.0**. A blank file opens on the workspace called `Design1`.

Complete the following steps to save the file with a new name:

1. Select **File»Save As** to display a standard Windows Save dialog.
2. Navigate to the location where you wish the file to reside, enter `MyGettingStarted` as the filename, 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 Preferences** dialog box.

Complete the following step to open an existing file:

1. Select **File»Open**, navigate to the location where the file resides, highlight the file, and click on the **Open** button.

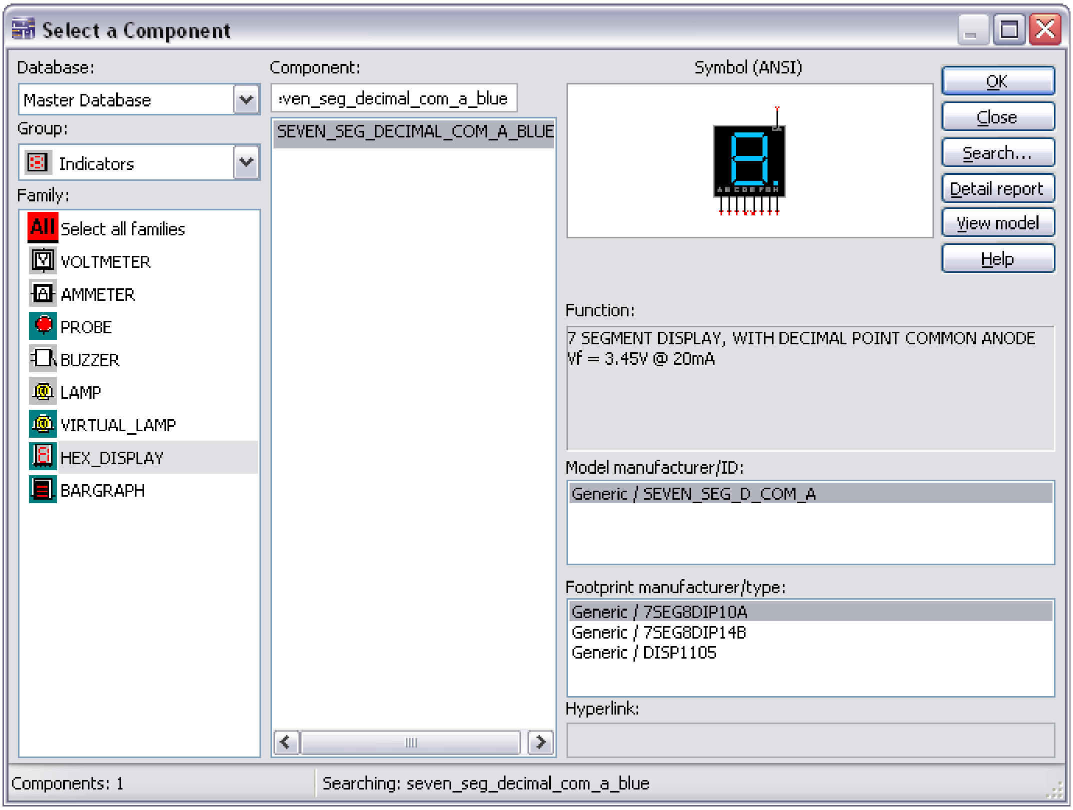
Tip To view files from earlier versions of Multisim, select the desired version in the **Files of Type** drop-down in the **Open** dialog.

Placing the Components

Complete the following steps to start placing components:

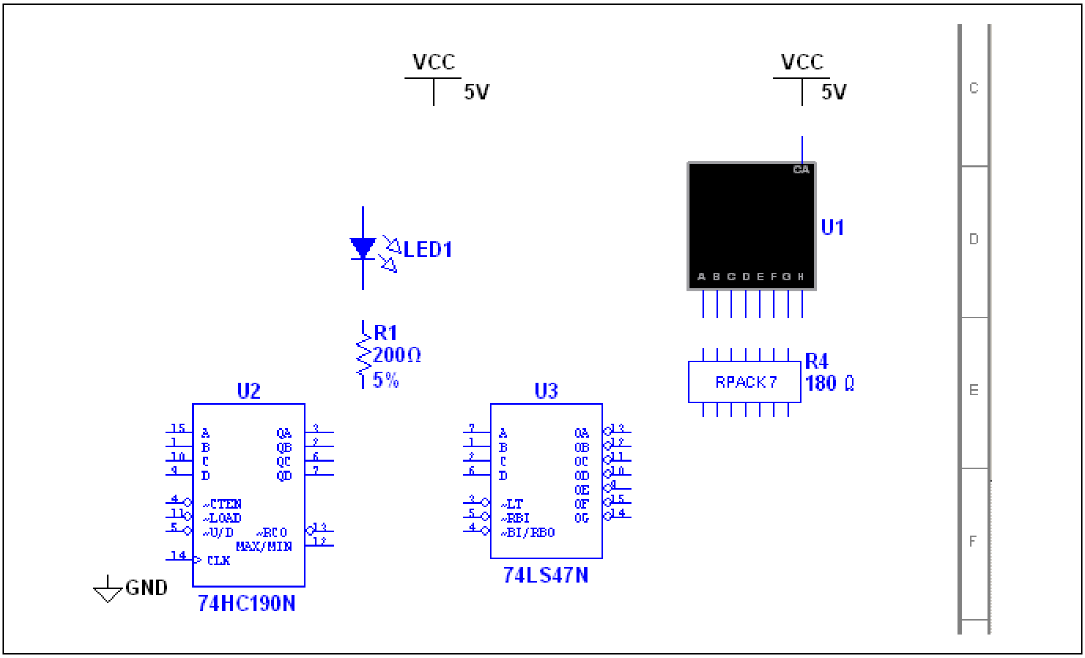
1. Open `MyGettingStarted.ms11` as described above.
2. Select **Place»Component** to display the **Select a Component** dialog box, navigate to the 7-segment LED display as shown below and click **OK**. The component appears as a "ghost" on the cursor.

Tip Once you have selected the desired **Group** and **Family**, start typing the component's name in the browser's **Component** field. As you type, the string appears in the **Searching** field at the bottom of the browser. In the example below, type `seven_seg_decimal_com_a_blue`. Matches are displayed as you type.



3. Move the cursor to the bottom-right of the workspace and left-click to place the component. Note that the Reference Designator for this component is “U1”.

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



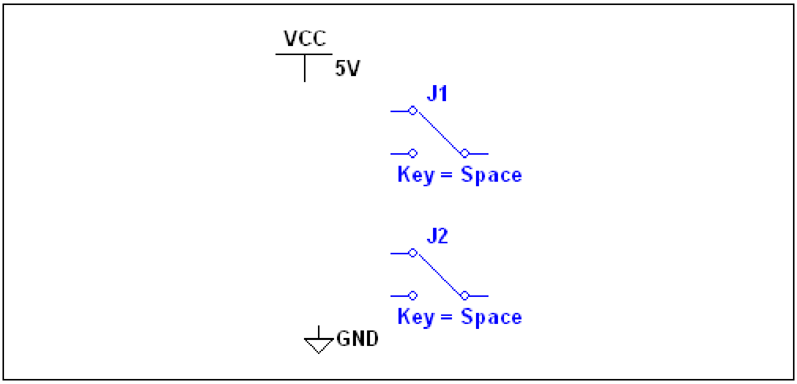
Note When placing resistors, inductors, or capacitors (RLC components), the **Select a Component** dialog box has slightly different fields than for other components. When placing these components, you can choose any combination of: the component's value (for example, the resistance value); type (for example, carbon film); tolerance; footprint and manufacturer. If you are placing a component that will be ultimately exported to PCB layout, and become part of a **Bill of Materials**, you must be careful that the combination of values that you select in the **Select a Component** dialog box are available in a real-world, purchaseable component.

Tip When placing RLC components, type the value of the device that you want to place in the field at the top of the **Component** list. The value does not need to appear in the list to be placed on the schematic.

Tip While placing the 200 Ω resistor, rotate it to a vertical orientation by pressing <Ctrl-R> on your keyboard.

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 circuit, the numbering will differ. This will not affect the operation of the circuit in any way.

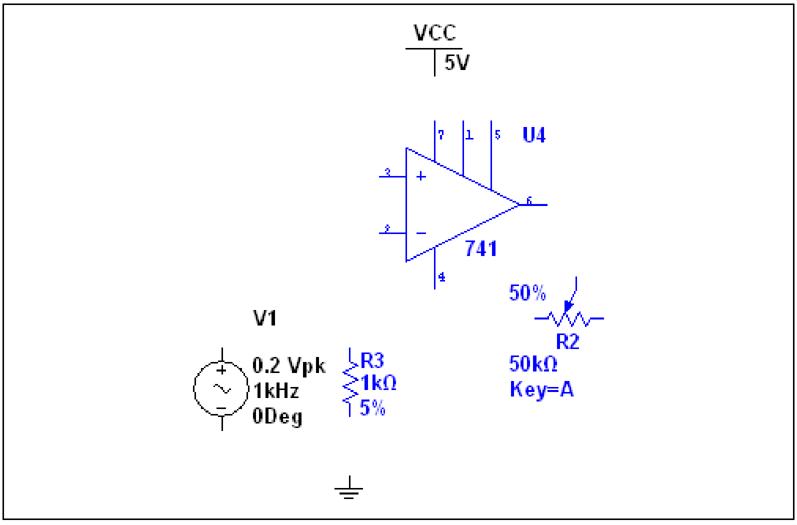
5. Place the components in the Counter Control section. After placement, right-click on each of the SPDT switches and select **Flip Horizontal**.



Tip The SPDT switches are in the **Basic** group, **Switch** family.

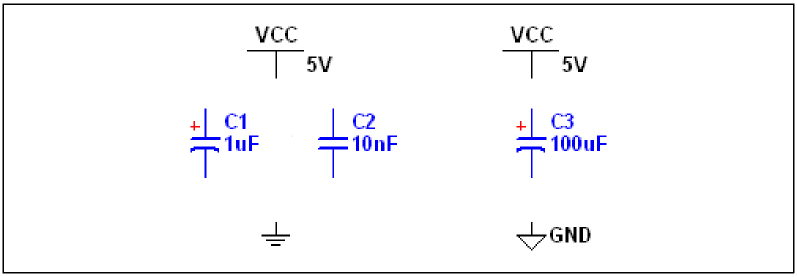
Tip When a component is on the workspace and you want to place the same component again, highlight it and select **Edit»Copy**, then **Edit»Paste**. You can also select it from the **In Use List** and click to place it on the workspace.

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

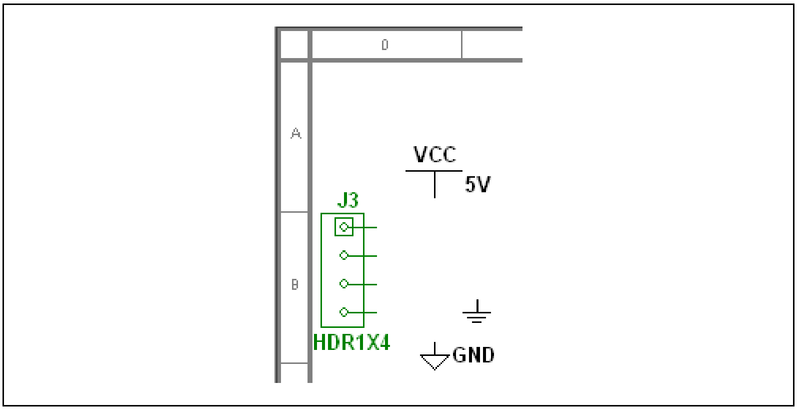


7. Double-click on the AC voltage signal source and change the **Voltage (Pk)** to 0.2 V and click **OK** to close the dialog.

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



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



Tip J3 is in the **Basic** group, **Connectors** family.

Tip Once you have wired a circuit, you can drop two-pinned components like resistors directly onto a wire. The connection is automatically made by Multisim.

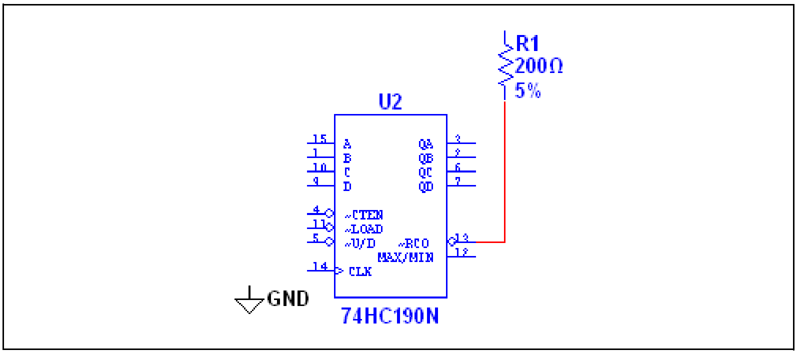
Wiring the Circuit

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 circuit that you placed on the workspace or you can use `Getting Started 1.ms11` from the `Getting Started` folder (found inside the `samples` folder).

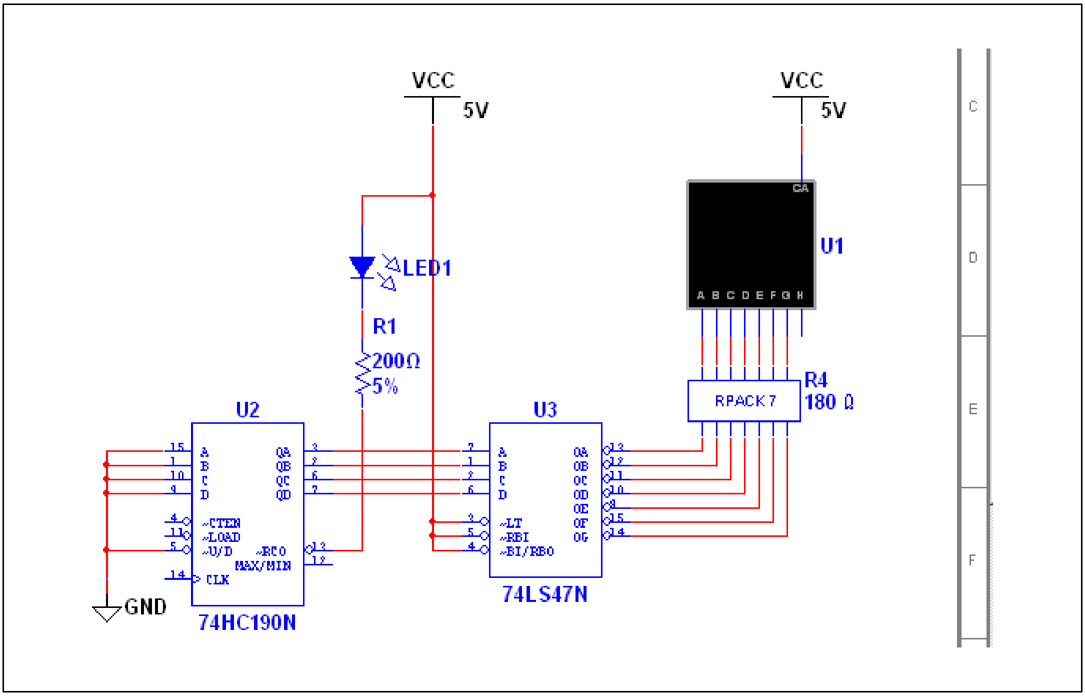
Complete the following steps to wire the circuit:

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. This feature saves a great deal of time when wiring large circuits.



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.

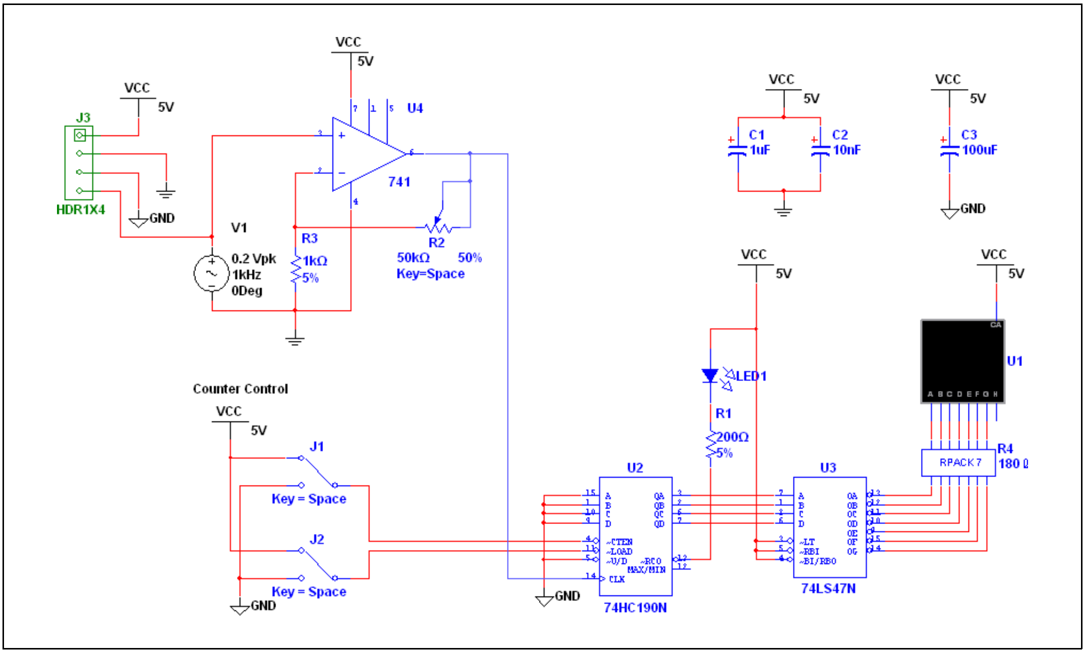
3. Finish wiring the Digital Counter section as shown below.



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.

4. Finish wiring the circuit as shown below.



Simulation

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

Virtual Instrumentation

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

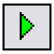
Tip You can also use `Getting Started 2.ms11` from the `Getting Started` folder (found inside the `samples` folder).

1. J1, J2 and R2 are interactive components.

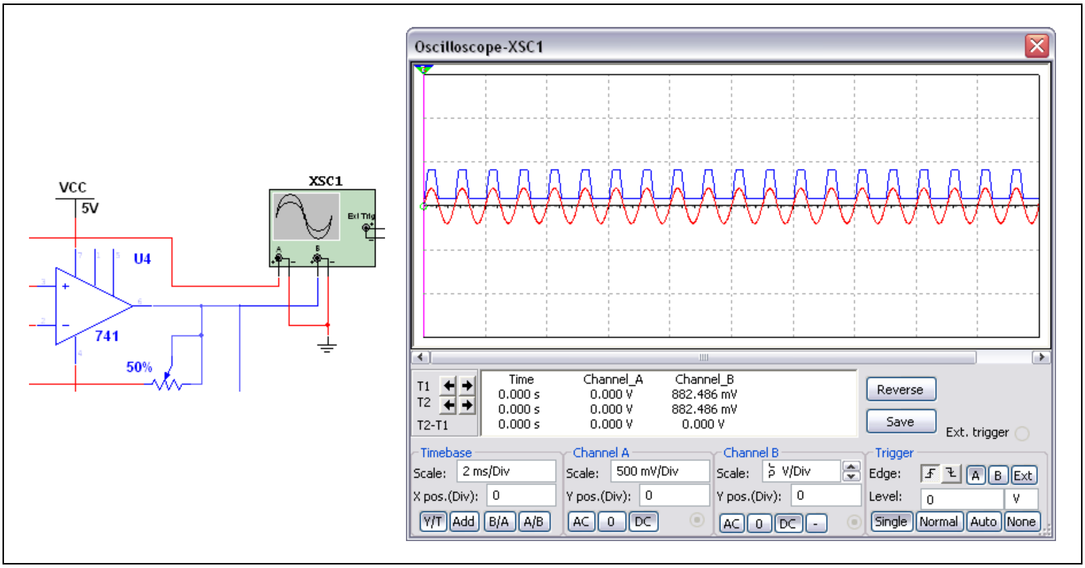
   Set up the interactive keys for J1, J2 and R2 by double-clicking on each and selecting the **Value** tab. In the **Key** field, enter "E" for J1, "L" for J2, and "A" for R2.

2. Press <E> on the keyboard to enable the counter, or just 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. Wire the instrument as shown in step 5.

Tip To easily differentiate between traces on the oscilloscope, right-click on the wire connected to the scope's "B" input and select **Color Segment** 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 any other editing function cannot be done while simulation is running.)



4. Double-click on the scope's icon to show the instrument face. Select **Simulate»Run**. The output of the opamp appears on the scope.
5. Adjust the Timebase to 2 ms/Div and Channel A's Scale to 500 mV/Div. You will see the following displayed on the scope.



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

6. Press <E> on your keyboard while the simulation is running to enable or disable the counter. Enable is Active Low.

   Press <L> to load zeros into the counter. Load is Active Low.

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

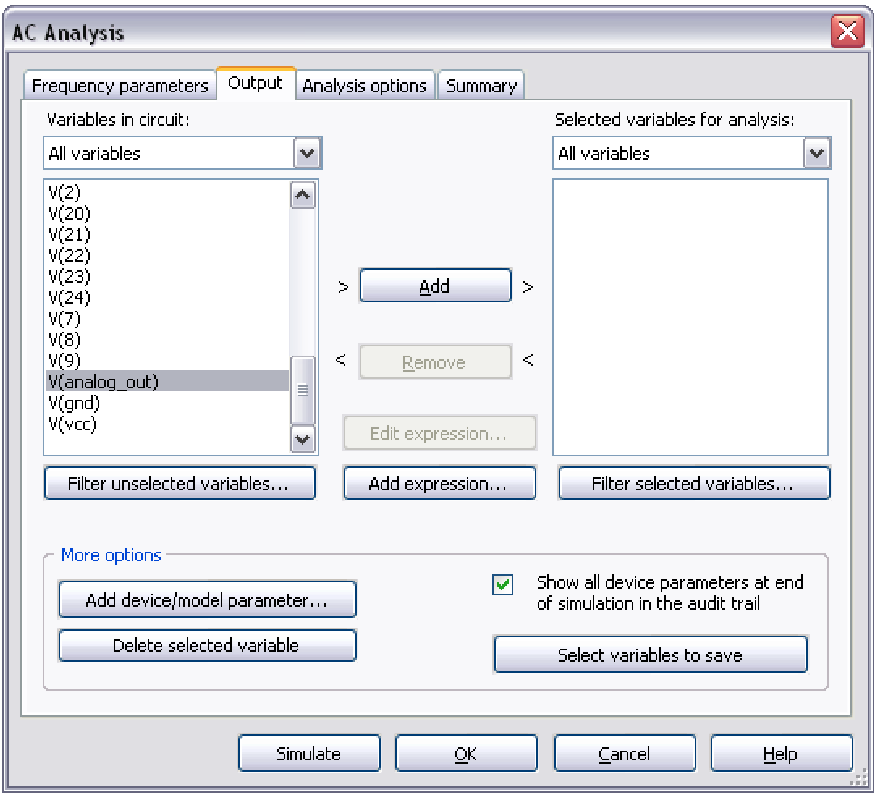
Tip Instead of pressing the above-mentioned keys, you can directly manipulate the interactive components on the schematic with your mouse.

Analysis

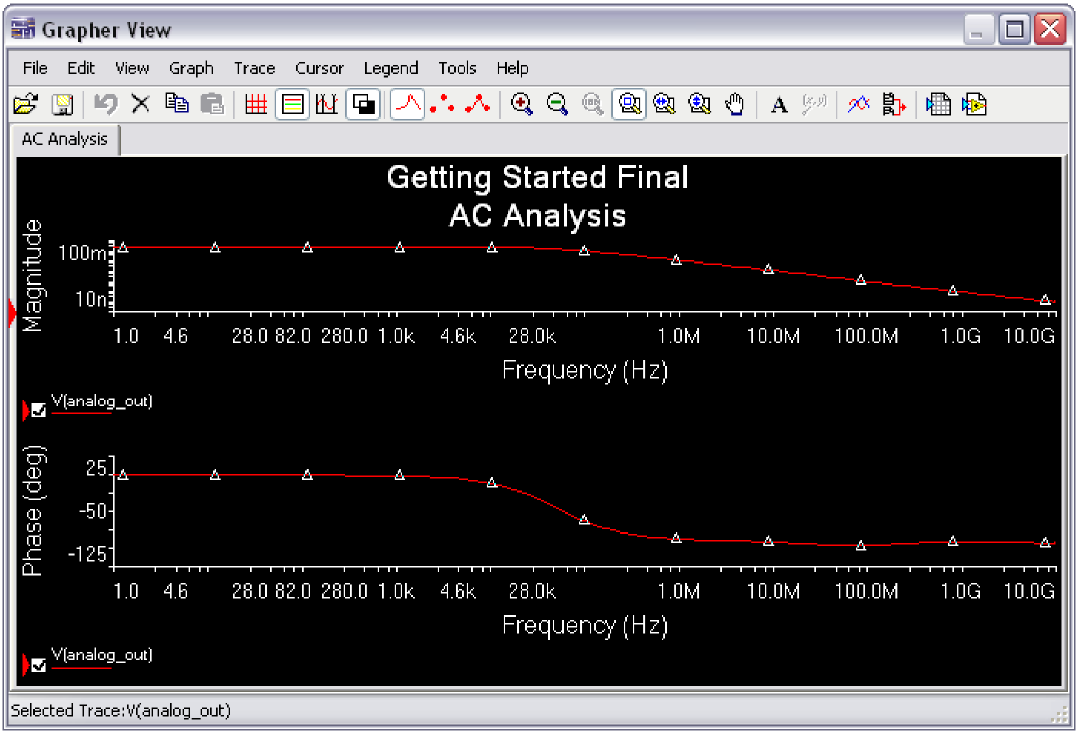
In this section, you will use **AC Analysis** to verify the frequency response of the amplifier.

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, and change the preferred net name to `analog_out` in the **Net Settings** dialog box.
2. Select **Simulate»Analyses»AC Analysis** and click on the **Output** tab.



3. Highlight `V(analog_out)` in the left column and click **Add**. `V(analog_out)` moves to the right column. This indicates the voltage at node `V(analog_out)` will be displayed after simulation.
4. Click **Simulate**. The results of the analysis appear in the **Grapher**.

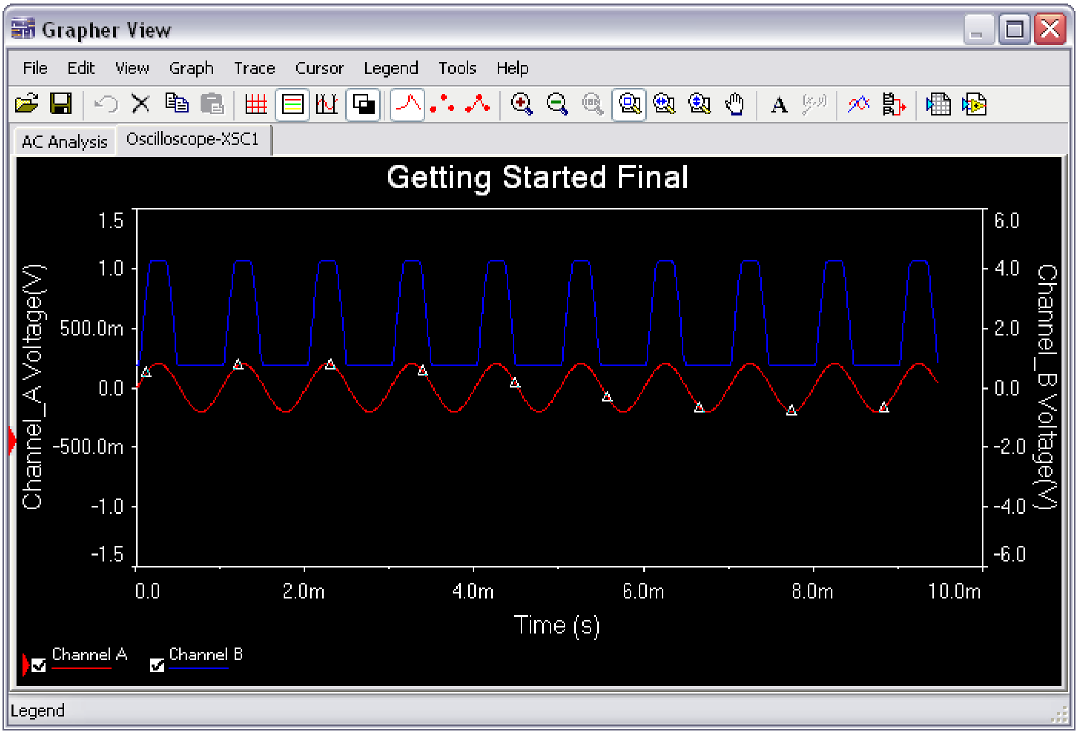


The Grapher

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 a graph of traces for some instruments (for example the results of 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. This section uses the **BOM** as an example for the tutorial circuit.

Bill of Materials

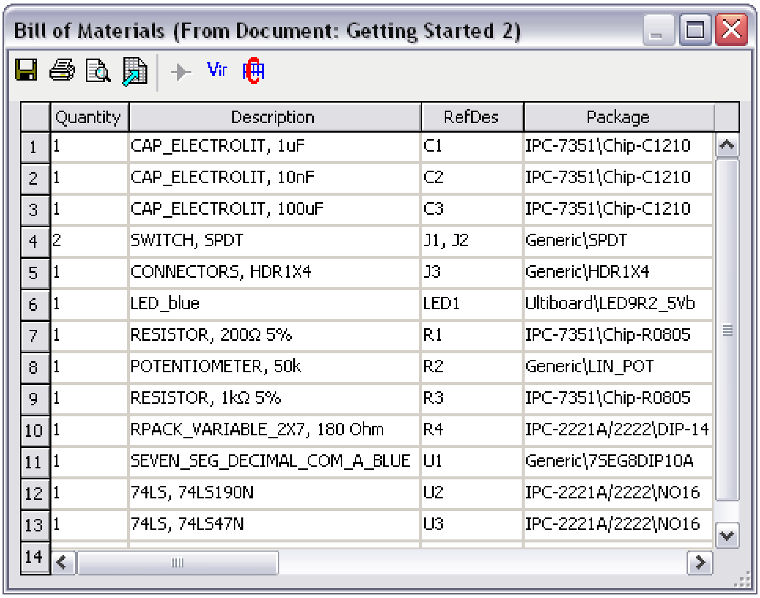
A bill of materials lists the components used in your design and therefore provides a summary of the components needed to manufacture the circuit board. Information provided includes:

- quantity of each component needed.
- description of each component, including the type of component (example: resistor) and value (example: 5.1 kohm).
- Reference Designator of each component.
- package or footprint of each component.

Complete the following step to create a **BOM** (bill of materials) for your circuit:

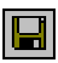
1. Select **Reports»Bill of Materials**.

   The report appears, looking similar to this:



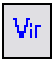
	Quantity	Description	RefDes	Package
1	1	CAP_ELECTROLIT, 1uF	C1	IPC-7351\Chip-C1210
2	1	CAP_ELECTROLIT, 10nF	C2	IPC-7351\Chip-C1210
3	1	CAP_ELECTROLIT, 100uF	C3	IPC-7351\Chip-C1210
4	2	SWITCH, SPDT	J1, J2	Generic\SPDT
5	1	CONNECTORS, HDR1X4	J3	Generic\HDR1X4
6	1	LED_blue	LED1	Ultiboard\LED9R2_5Vb
7	1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

To print the **Bill of Materials**, click the **Print** button. A standard Windows print dialog box appears, allowing you to choose the printer, number of copies, and so on.



To save the **Bill of Materials** to a file, click the **Save** button. 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 circuit that are not “real” components, click the **Virtual** button. A separate window appears, showing these components only.

Detailed information on this and other reports can be found in the *Multisim Help*.

3

Ultiboard Tutorial

The tutorial in this chapter places the parts and traces for the circuit described in Chapter 2, *Multisim Tutorial*.

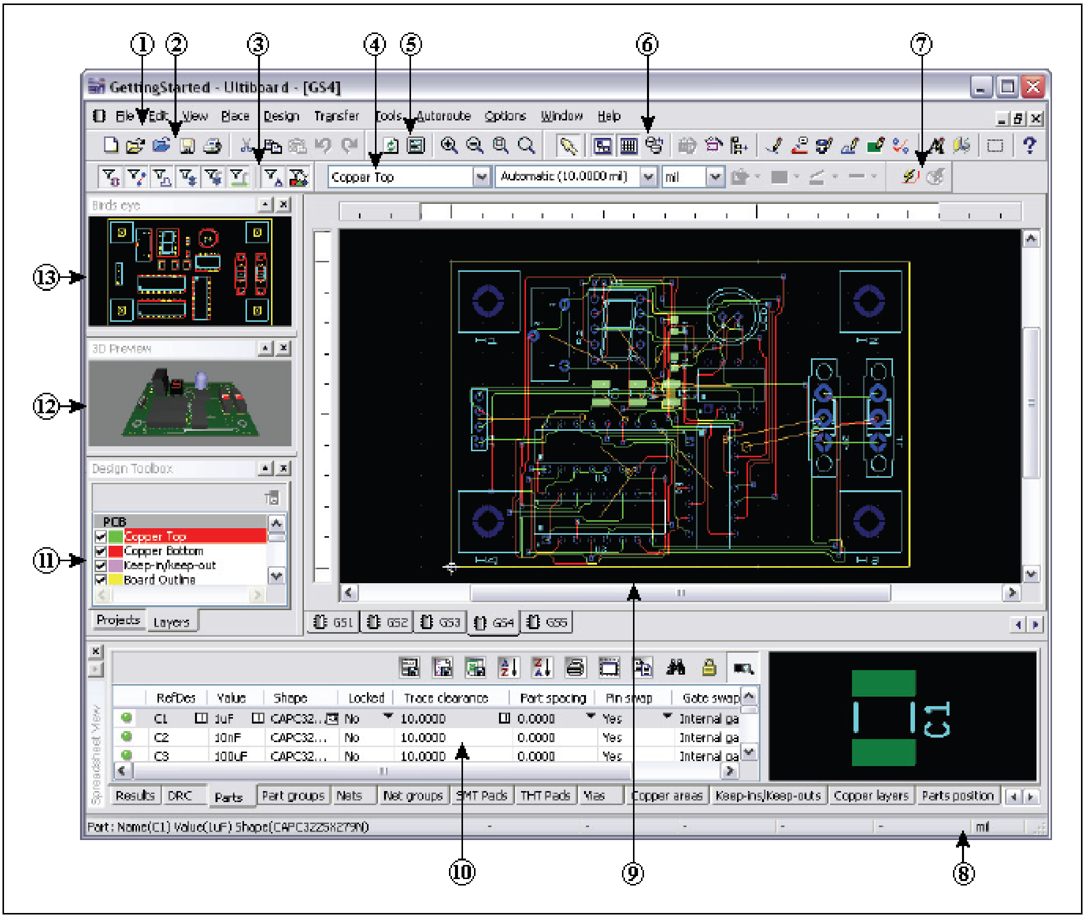
Some of the described features may not be available in your edition of Ultiboard. Refer to the *NI Circuit Design Suite Release Notes* for a list of the features available in your edition.

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 circuit 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'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			

The **Menu Bar** is where you find commands for all functions.

The **Standard** toolbar contains buttons for commonly-performed functions such as Save, Print, Cut, and Paste.

As you add more parts and traces to a board, it can become difficult to select only those which you want to use. The **Select** toolbar contains buttons used to control selections.

The **Draw Settings** toolbar 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.

The **View** toolbar contains buttons for modifying the way the screen is displayed.

The **Main** toolbar contains buttons for common board design functions.

The **Autoroute** toolbar contains autorouting and part placement functions.

The **Status Bar** displays useful and important information.

The **Workspace** is where you build your design.

The **Spreadsheet View** allows fast advanced viewing and editing of parameters including part details such as shapes, Reference Designators, attributes and design constraints.

The **Design Toolbox** lets you show, hide, or dim elements of your design.

The **3D Preview** shows you a three-dimensional preview of the board.

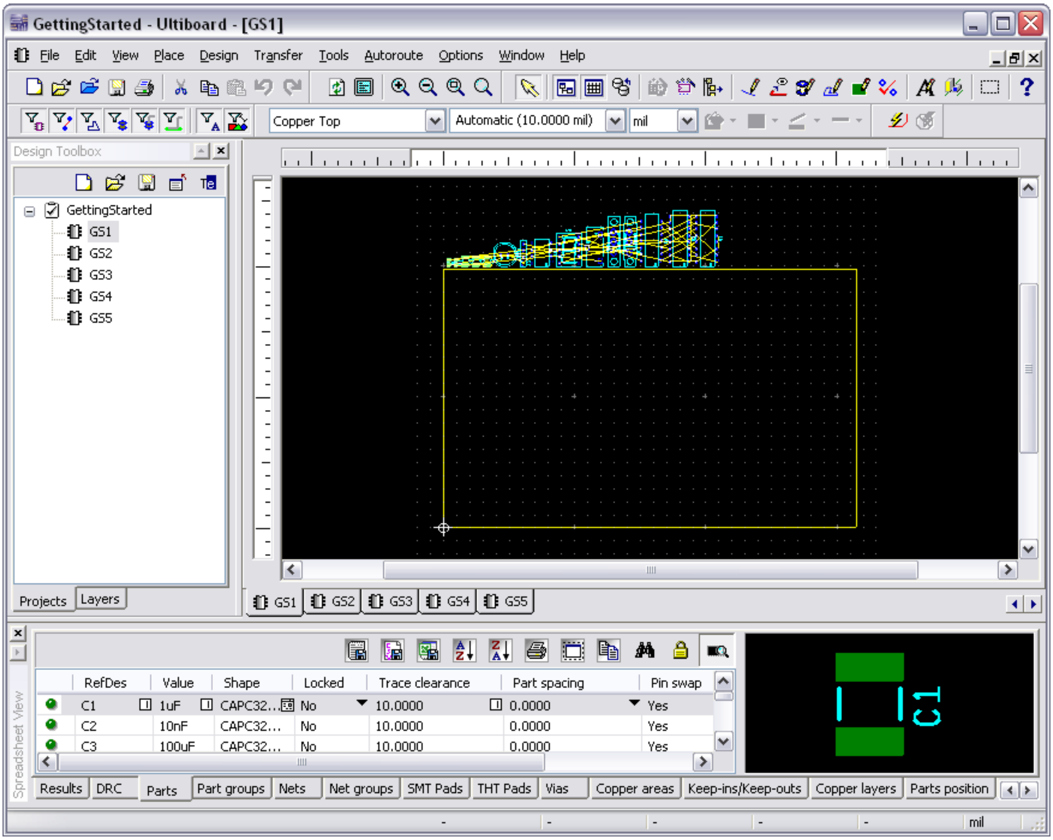
The **Birds Eye View** shows you the design at a glance and lets you easily navigate around the workspace.

Opening the Tutorial

Complete the following steps to open the tutorial file:

1. Select **Start»All Programs»National Instruments»Circuit Design Suite 11.0»Ultiboard 11.0** to launch Ultiboard.
2. Select **File»Open Samples** and double-click on the `Getting Started` folder to open it.
3. Select `Getting Started.ewprj` 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*.



4. To select a design from the project (for example, `GS1`), either 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**.

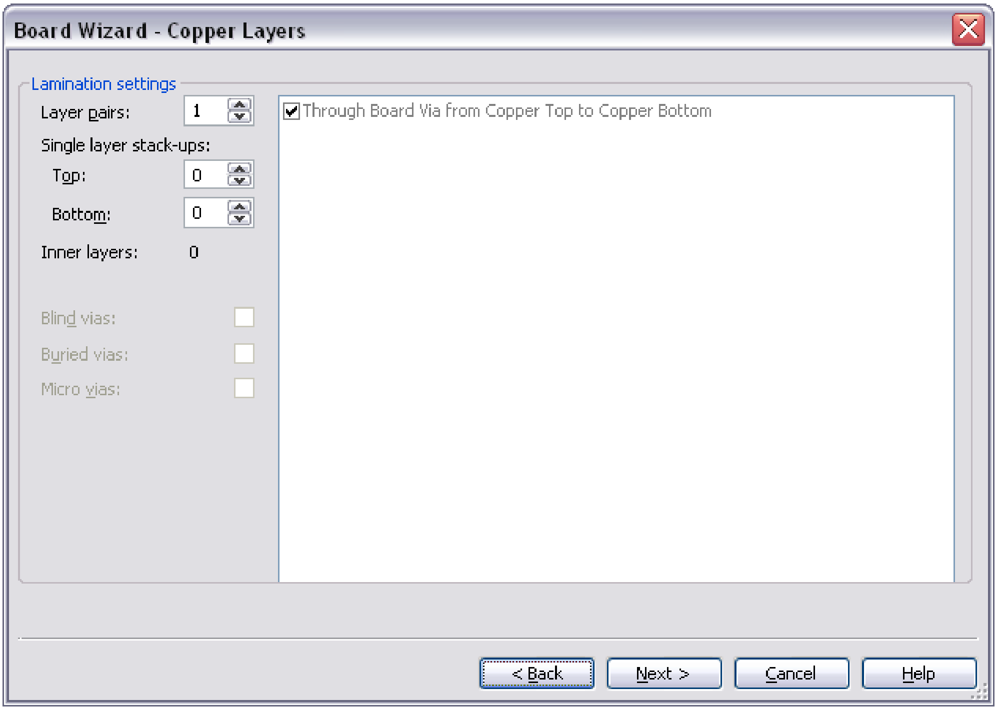
Complete the following steps to experiment with the **Board Wizard**:

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. Enable the **Change the layer technology** option 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 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:

   - make sure the **Reference Point** is set to **Bottom-left** for **Alignment**.
   - make sure the **Rectangular** option is selected for **Board shape and size**.
   - set the **Width** to 3000 and the **Height** to 2000 (a more suitable size for the parts in this design).
   - 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.

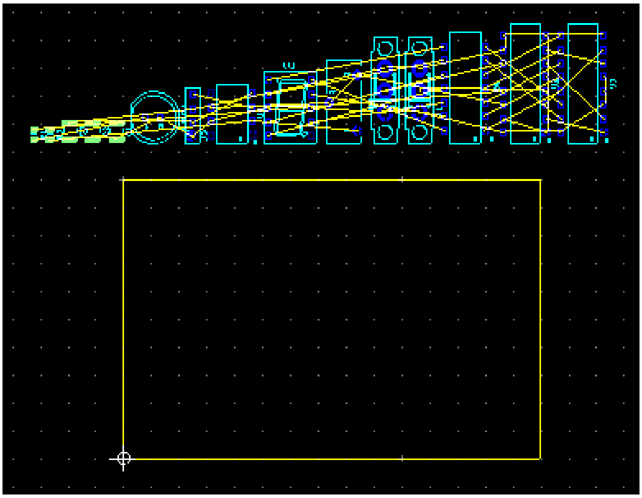
Note For complete details on the **Board Wizard**, refer to the *Ultiboard Help*.

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 file 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 quickly 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.

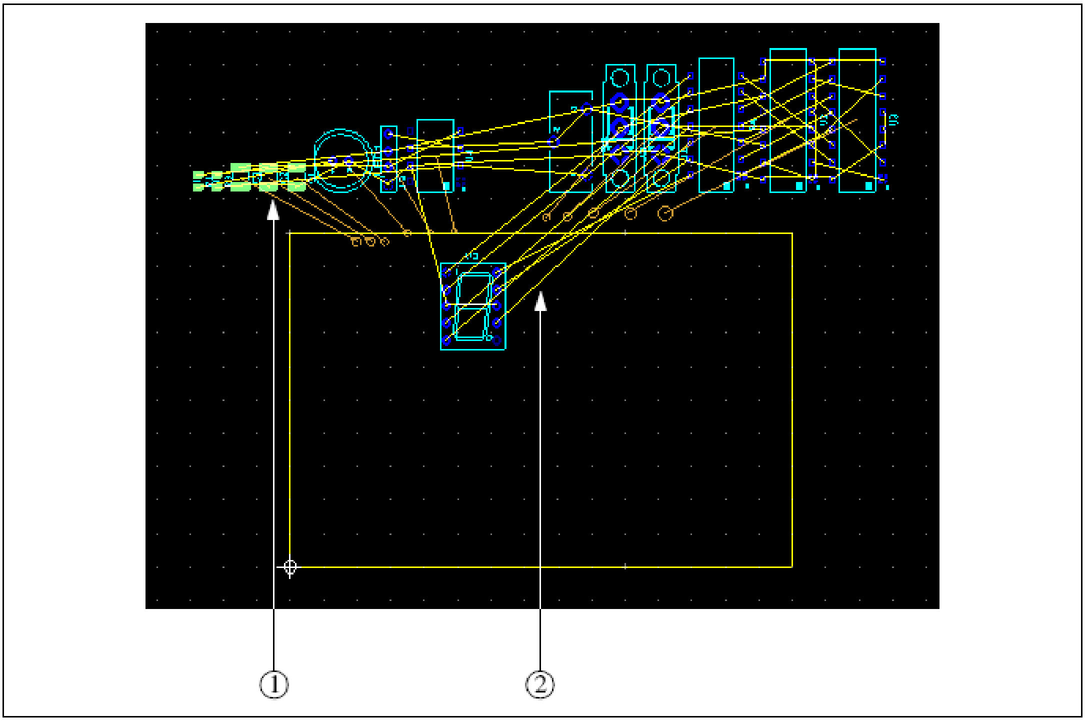
Before you begin, double-click the **Copper Top** layer in the **Design Toolbox** to make it the active layer.

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

1. 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.

2. 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)

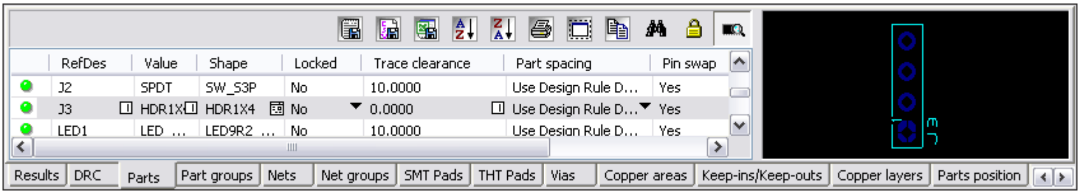
U1 remains selected. This is an important point for Ultiboard that holds throughout the application—you need to explicitly end any particular action. In this case, simply clicking somewhere else de-selects the part. Right-clicking also ends the current action.

3. Go to the **Parts** tab in the **Spreadsheet View** and scroll to U1. You will 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 until you see J3.

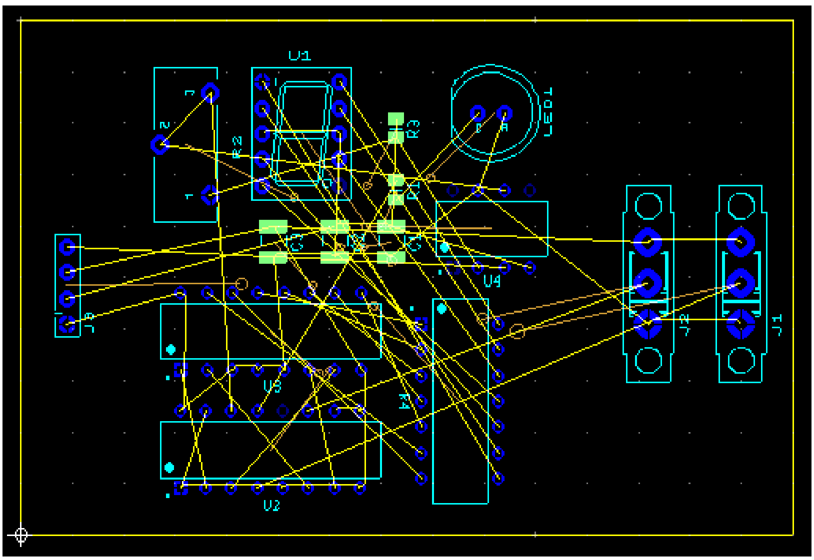


2. Click on J3 and drag it from the **Parts** tab onto the workspace. J3 is attached to your mouse pointer.
3. Drop J3 on the left edge of the board, roughly in the middle. As before, in the **Parts** tab J3's green light is slightly brighter, indicating that the part has been placed.

Placing the Tutorial Parts

Using any method or combination of methods, make your layout look like the illustration below. You can also simply open the next design file in the project, `GS2`, which has already been set up this way.

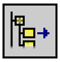
Your design should look like this:



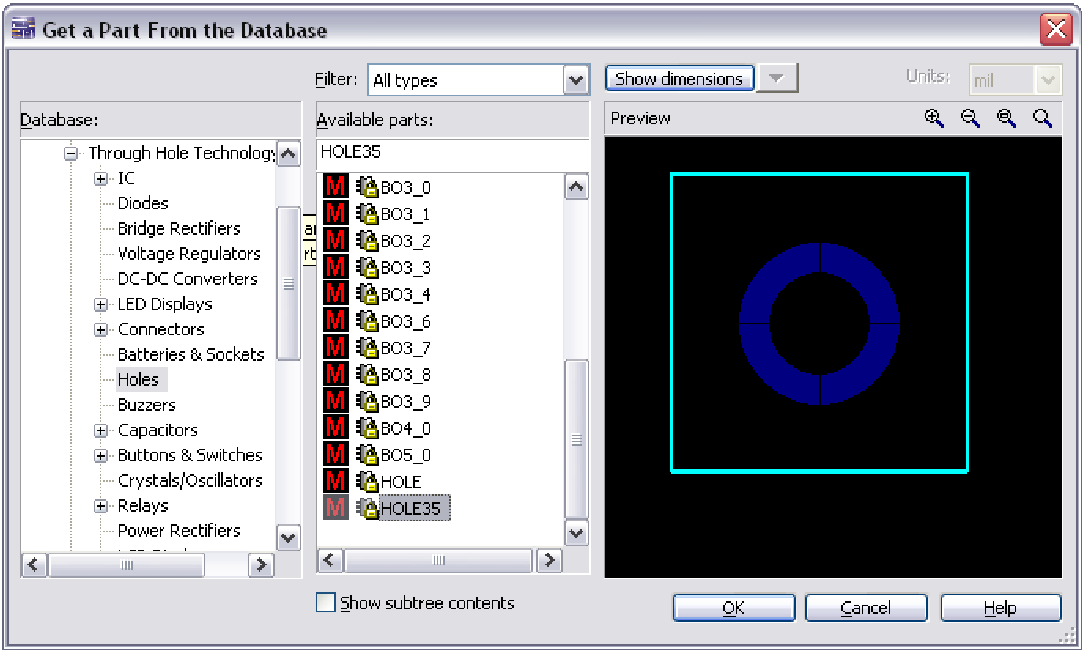
Placing Parts from the Database

In addition to placing parts imported as part of your design file, 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:

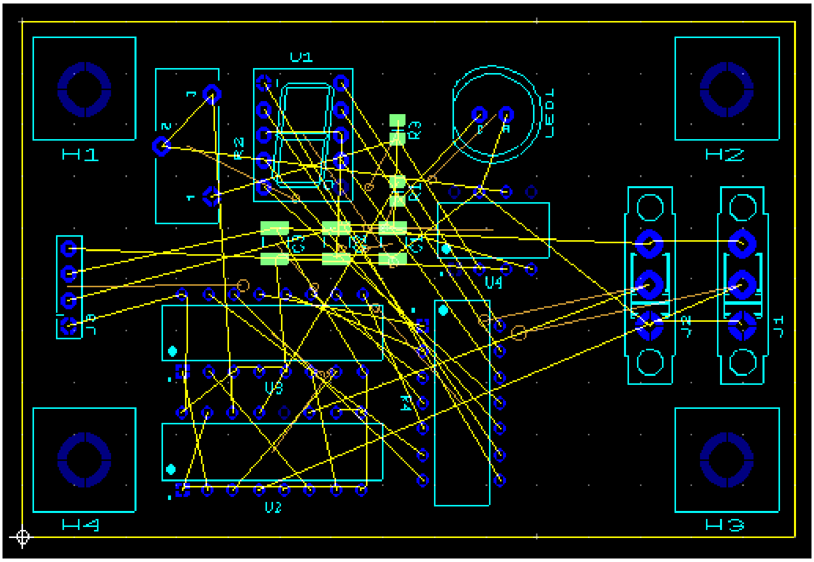


1. Choose **Place»From database**. The **Get a Part From the Database** dialog box opens.
2. In the **Database** panel, expand the **Through Hole Technology Parts** category and navigate to the **Holes** category. The parts appear in the **Available parts** panel.
3. In the **Available parts** panel, 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 you are prompted to enter the **RefDes** and **Value**.
5. Enter the hole's reference designator (`H1`) and value (`HOLE`) and click **OK**.
6. Move the pointer over the board. The part is attached to the pointer.

7. When the hole is in position in the top-left corner, click to drop it on the board.
8. The **Enter Reference Designation for Part** dialog box reappears, with the reference designator automatically incremented to H2.
9. Enter the value (`HOLE`) and click **OK** to place the next mounting hole in the top right corner, and 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 to close the **Get a Part From the Database** dialog box.



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, simply click on it. To specify the X/Y coordinates to which the selected part is to move, press the <*> key on the numeric keypad. Alternatively, in the **Parts** tab, select a placed part (indicated by a bright green light beside it) and drag it to a new location.

Tip The 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.

You can also select a group of parts and move them together. To do this, you can do one of the following:

- hold down the <Shift> key and click on more than one part.
- drag a box around several parts.

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

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**. For details, refer to the *Ultiboard Help*.

Another option for moving parts is to 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 down the <Shift> key 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 out. 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. When performing operations on traces, be sure to select either the appropriate segment or, if you wish, 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** should be highlighted in red in the **Layers** tab of the **Design Toolbox**.

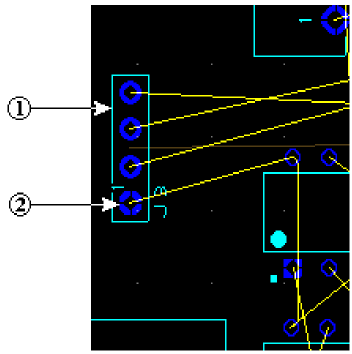
Tip If necessary, press <F7> to show the whole design.

Complete the following steps to place a trace manually:

1. Choose **Place»Line**.

Tip The **Line** command is used to create a line on any layer. The results differ depending on the layer selected. For example, if the selected layer is silkscreen, you will create a line 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. Find the start pin shown below:

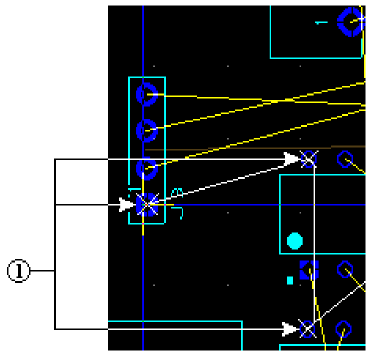


1 Part J3	2 Start Pin

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

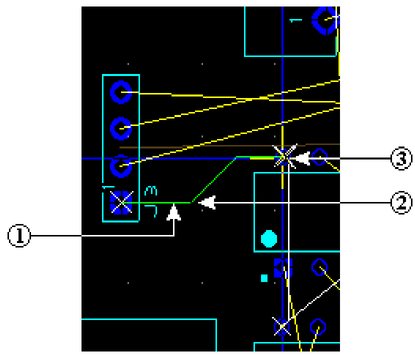
Tip If you have trouble locating the part, use the **Find** function of the **Parts** tab. Select the part in the **Parts** tab, then click the **Find and select the part** button. The part is shown in the workspace. If necessary, zoom in further using the mouse wheel.

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. (The color of the highlighting can be changed in the **Colors** tab of the **Global Preferences** dialog box.) This is how you know 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 on the destination pin.



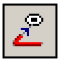
1	Trace	3	Destination Pin
2	Click to anchor trace		

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:

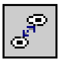


1. Choose **Place»Follow-me**.
2. Click on the top pin of J3.
3. Click on the second pin from the bottom on the left side of U4.
4. Ultiboard draws the connection for you.

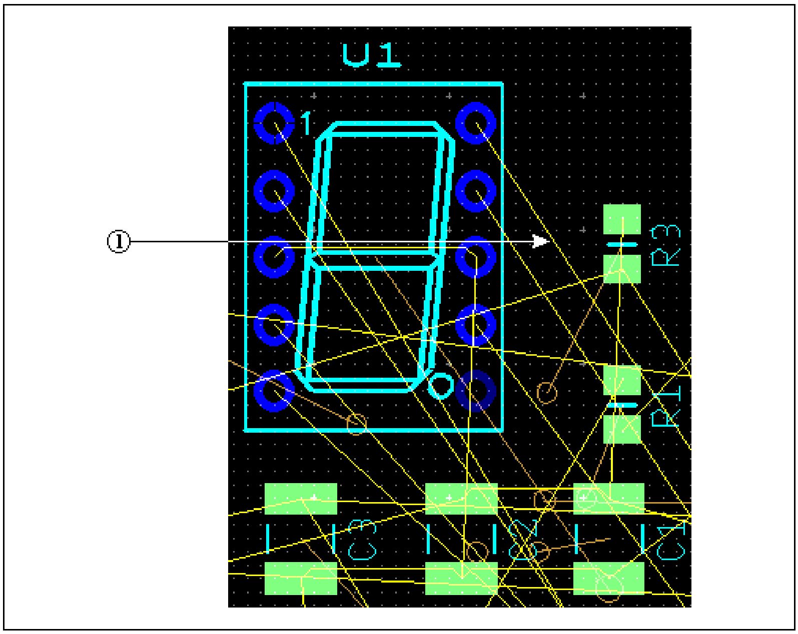
Tip You do not need to click exactly on a pin—you can also start by clicking on a ratsnest line.

Placing a Connection Machine Trace

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



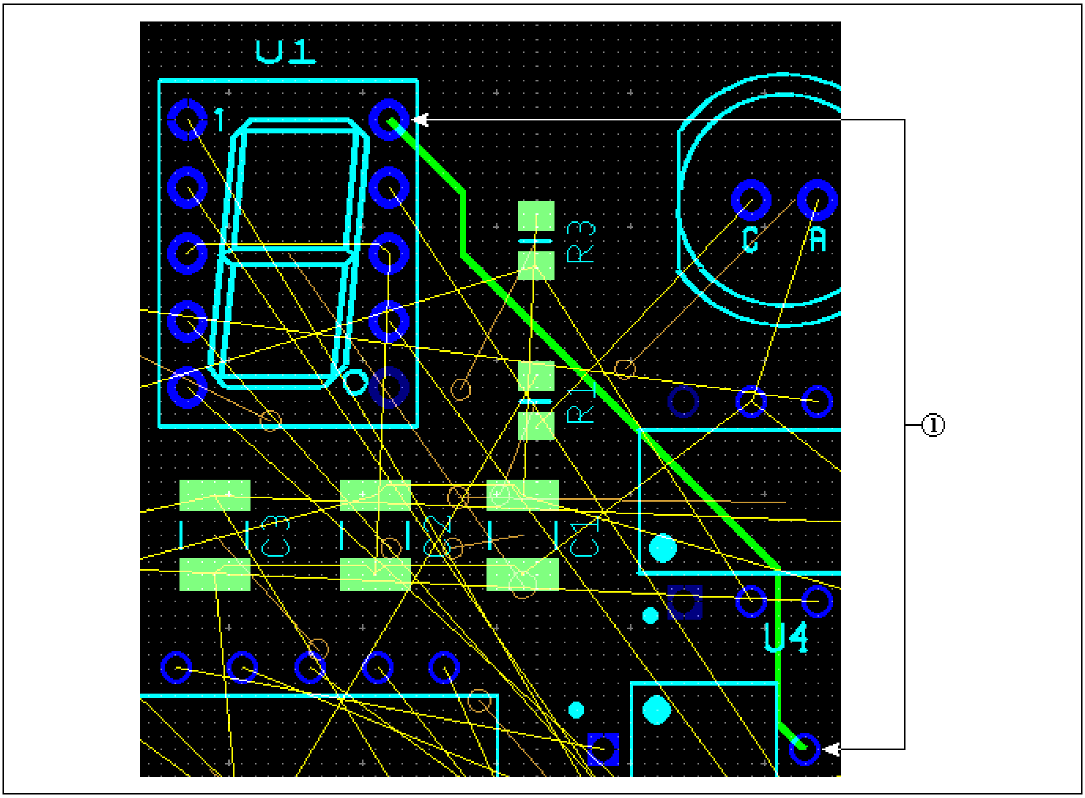
1. Choose **Place»Connection Machine**.
2. Click on the segment of the ratsnest indicated below.



1 Click Ratsnest

3. Move your cursor—Ultiboard suggests various trace placement options routed around obstacles.

4. When you see the route you want, click to fix the trace. You don't have 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.ewprj`:

1. Open the `GS5` design in Ultiboard.
2. Select **Autoroute»Start Autoplacement**. The parts are placed on the circuit board.

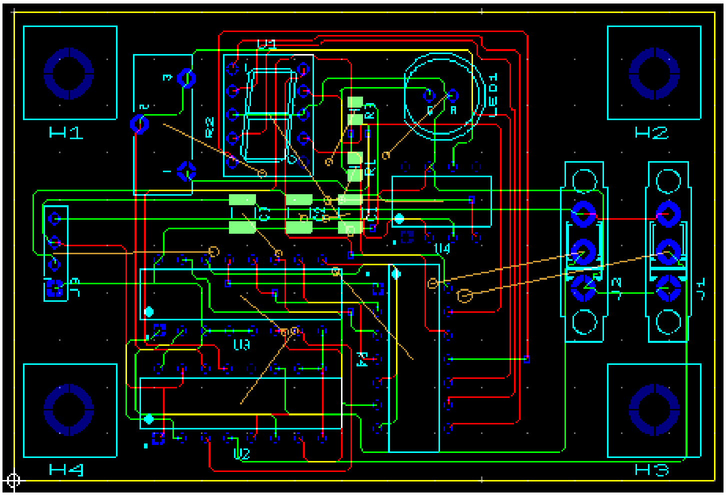
Autorouting Traces

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.ewprj`:

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. Optionally, select **Autoroute»Start Optimization** to optimize the placement of the traces.



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. This section explains 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 that have been left on the board.

To delete open trace ends, open the `GS4` design and choose **Edit»Copper Delete»Open Trace Ends**. This deletes all open trace ends in the design.

To delete any unused vias, make sure the design is open and choose **Design»Remove Unused Vias** to delete 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.

For details, refer to the *Ultiboard Help*.

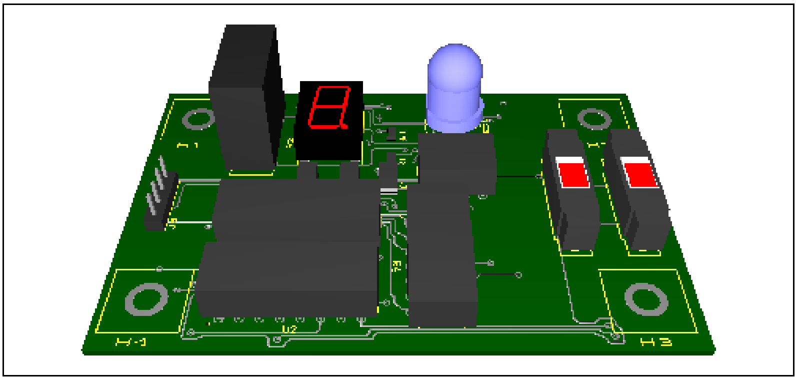
Exporting a File

An exported file contains complete information describing how a finished board is to be manufactured. Files that can be exported include Gerber RS-274X and RS-274D files.

For complete details, refer to the *Ultiboard Help*.

Viewing Designs in 3D

Ultiboard lets you see what the board looks like in three dimensions at any time during the design. For complete details, refer to the *Ultiboard Help*.



Tip You can use the **Internal View** to look between the layers of a multi-layer PCB. For details, refer to the *Ultiboard Help*.

4

Multisim MCU Tutorial

The tutorial in this chapter leads you through the process of simulating and debugging a circuit that contains a microcontroller.

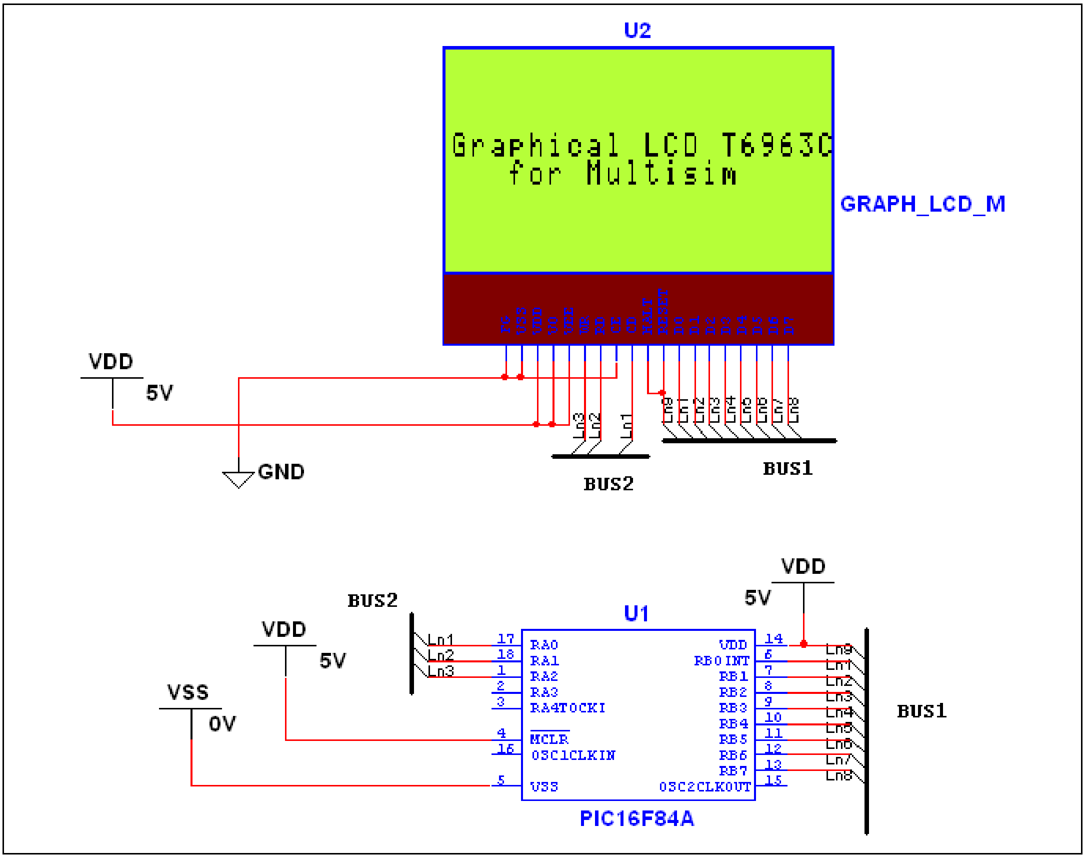
Some of the described features may not be available in your edition of Circuit Design Suite. Refer to the *NI Circuit Design Suite Release Notes* for a list of the features available in your edition.

Overview

The files used for this tutorial install with your NI Circuit Design Suite software at `...\samples\Getting Started`.

This tutorial uses `Getting Started MCU.ms11`, which accesses the contents of folder `LCDWorkspace` as required.

The LCD Graphical Display circuit example demonstrates the use of a PIC microcontroller to control a graphical LCD display component in Multisim based on a combination of the Toshiba T6963C controller and an external display RAM. To control the LCD display, the microcontroller sends signals to the LCD through the LCD's data and control lines. A software program written for the microcontroller determines the logic behind setting the lines on its pins to high or low to send commands and data to the LCD display.



About the Tutorial

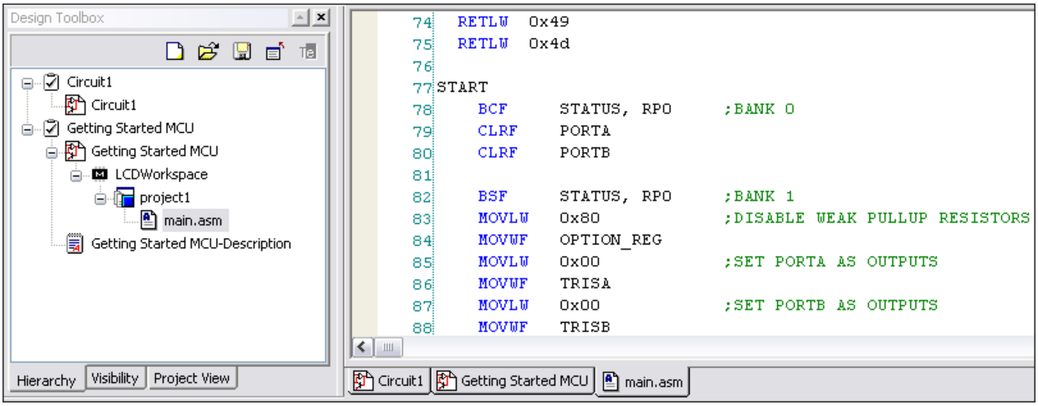
The data lines of LCD U2 are connected to pins RB0–RB7 on microcontroller U1. The control lines of the LCD are connected to RA0–RA2 on the microcontroller. The MCU U1 communicates with the LCD U2 via these wires. Data is sent to U2 in parallel and signals on the control lines determine the timing and type of data being sent (that is, address or data).

The LCD Graphical Display can operate in three modes: text mode, graphical mode and a combination text and graphical mode. This example demonstrates the controlling of the LCD Graphical Display in a combination text and graphical mode. The software that the MCU runs is contained in an MCU workspace that displays in the **Design Toolbox** as

`LCDWorkspace`. The workspace contains one project `project1` that consists of a single source code file `main.asm`.

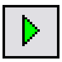
Complete the following steps to view the file:

1. Double-click on `main.asm` in the **Design Toolbox**. A tab appears in the schematic capture workspace called `main.asm` that displays the assembly program.



To display the line numbers, select **MCU»Show Line Numbers**.

Complete the following step to run this circuit:

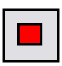


1. Select **Simulate»Run**. If you did not build your program beforehand, a dialog box displays stating that the configuration is out of date and asks if you would like to build it. Click **Yes**. The results of the build display in the **Results** tab of the **Spreadsheet View**.

   Switch back to the design tab. The program displays the line "Graphical LCD T6963C for Multisim" characters in text mode; the LCD then switches to graphical mode and draws an inverted "V" dot-by-dot on top of the text.

   Once the lines are drawn, the text scrolls right and then left. This is achieved by moving the start address of the text buffer of the LCD display. This also demonstrates that there are two buffers in the LCD, one for storing graphics and another for storing text. Other features of the LCD such as text flashing and erasing of characters are also demonstrated.

   The LCD display program continues to cycle through each of these effects.



To stop the simulation, select **Simulate»Stop**.

Understanding the Assembly Program

Constants and data

Switch back to `main.asm`.

To make the program easier to understand, the LCD display commands and temporary buffers for storing addresses and data in the MCU are predefined in constants at the start of the program:

```
DATA_BUFFER           EQU         0x20
DATA_BUFFER2          EQU         0x21
CMD_BUFFER            EQU         0x22
REF_BUFFER            EQU         0x24

ADDR_INDEX            EQU         0x25                ;STARTING ADDRESS IN EEPROM
ADDR_L                EQU         0x26                ;STARTING ADDRESS L
ADDR_H                EQU         0x27                ;STARTING ADDRESS H
COUNTER_INDEX         EQU         0x29                ;COUNTER
BIT_INDEX             EQU         0x2A                ;BIT INDEX

CMD_SET_CURSOR EQU    21H         ;SET CURSOR
CMD_TXHOME            EQU   40H   ;SET TXT HM ADD
CMD_TXAREA            EQU   41H   ;SET TXT AREA
CMD_GRHOME            EQU   42H   ;SET GR HM ADD
CMD_GRAREA            EQU   43H   ;SET GR AREA
CMD_OFFSET            EQU   22H   ;SET OFFSET ADD
CMD_ADPSET            EQU   24H   ;SET ADD PTR
CMD_SETDATA_INC       EQU   0C0H  ;WRITE DATA AND INCREASE ADP
CMD_AWRON             EQU   0B0H  ;SET AUTO WRITE MODE
CMD_AWROFF            EQU   0B2H  ;RESET AUTO WRITE MODE
```

The text to be displayed on the LCD display is stored in data tables for some microcontrollers, but there is no PIC assembly instruction that allows you to directly address a data value in the program memory space. Instead, you can load literal values into the W register so you can write a routine that returns a value in your string based on an index. The `RETLW` instruction loads a constant value into the W register and executes a `RETURN` in one instruction.

The `TXPRT` routine retrieves the text data to be displayed on the LCD display. The character codes for the LCD display are defined in the T6963C controller reference manual (for example, `0x27` is the code for the letter "G", `0x52` for "r", and so on):

```
;  DATA
DATA_NUM              EQU    23H
TXPRT                 ; Text data "Grapical LCD T6963C   for Multisim"
  ADDWF  PCL, 1
  RETLW  0x27
  RETLW  0x52
  RETLW  0x41
  RETLW  0x50
  RETLW  0x48
  RETLW  0x49
  RETLW  0x43
  RETLW  0x41
  RETLW  0x4c
  RETLW  0x00
  RETLW  0x2C
  …
```

Initialization

The initialization code begins at the `START` label as shown in the excerpt below. The pins in the microcontroller are set up as output pins, and the values are reset. The LCD display component is initialized by the microcontroller and set to graphical and text mode. The home addresses for the internal graphical and text buffers in the LCD display component are set to `0x0000` and `0x2941` respectively, which determines where on the display the LCD starts to display the buffer data. Finally, the control signals are set up for the proper read/write operation on the LCD display.

```
START
        BCF     STATUS, RP0             ;BANK 0
        CLRF    PORTA
        CLRF    PORTB

        BSF     STATUS, RP0             ;BANK 1
        MOVLW   0x80                    ;DISABLE WEAK PULLUP RESISTORS
        MOVWF   OPTION_REG
        MOVLW   0x00                    ;SET PORTA AS OUTPUTS
        MOVWF   TRISA
        MOVLW   0x00                    ;SET PORTB AS OUTPUTS
        MOVWF   TRISB

        BCF     STATUS, RP0             ;BANK 0
        MOVLW   0x0F                    ; 1111 no commands ready
        MOVWF    PORTA

;1 SET DISPLAY MODE to GRAPH + TEXT mode, cursor off
        MOVLW   0x9C
        MOVWF   CMD_BUFFER
        CALL    CMD
…
```

Drawing Text and Graphics

The rest of the program sends commands to the LCD graphical display via the control lines through MCU pins RA0 to RA2 and data through the data lines:

```
;5 write string
        MOVLW   0x7D
        MOVWF   DATA_BUFFER
        MOVLW   0x29
        MOVWF   DATA_BUFFER2   ; external CG start at: 1400h
        CALL    DT2
        MOVLW   CMD_ADPSET
        MOVWF   CMD_BUFFER
        CALL    CMD

        MOVLW   CMD_AWRON
        MOVWF   CMD_BUFFER
        CALL    CMD

        MOVLW   0x00                   ; Initial the counter
        MOVWF   ADDR_INDEX

LOOP_READ_DATA2
        MOVF    ADDR_INDEX,0   ; STARTING data ADDRESS
        CALL     TXPRT

        MOVWF   DATA_BUFFER            ; LOAD CHAR data TO W
        CALL    ADT

        INCF    ADDR_INDEX,1

        MOVF    ADDR_INDEX, 0
        SUBLW   DATA_NUM               ; 35 chars
        BTFSS   STATUS, Z
        GOTO    LOOP_READ_DATA2

        MOVLW   CMD_AWROFF
        MOVWF   CMD_BUFFER
        CALL    CMD
…
```

For example, the above excerpt from the main loop in the program sends the characters defined in the `TXPRT` subroutine to be displayed in text mode on the graphical LCD.

The following sets the LCD to auto write mode:

```
MOVLW    CMD_AWRON
MOVWF    CMD_BUFFER
CALL     CMD
```

At this point, the program starts counting, and executes through the loop `LOOP_READ_DATA2` 35 times. This loop calls `TXPRT` to retrieve the text data and load it into the W register. It then calls to the subroutine `ADT`, which calls `SEND_DATA`, which writes the values in the W register to port B, to be sent to the data lines of the LCD display. Once the data is sent, the proper value on port A of the microcontroller is sent to the control pins of the LCD display to let it know that the data is ready to be read. The subroutines all return at the end to the instruction just after the call to them and the same thing happens until all 35 characters have been transmitted. The final three instructions in the excerpt turn off the auto write mode in the LCD display after exiting the loop:

```
MOVLW        CMD_AWROFF
MOVWF        CMD_BUFFER
CALL         CMD
```

The next few instructions draw the horizontal and sloped lines in graphical mode:

```
;6 draw wave once
MOVF         ADDR_L, 0
BTFSC        STATUS, Z
CALL         DRAW_WAVE
```

Working with the MCU Debugging Features

This section provides a step-by-step walkthrough of Multisim's MCU debugging features. It is important to follow the steps exactly as scripted, otherwise, the descriptions will no longer apply. Once you understand the breakpoint and single stepping features you can explore the possibilities of advanced MCU debugging.

Debug View Overview

To write a program for a microcontroller either in C or assembly, you create source code files (`.asm`, `.inc`, `.c`, `.h`) as part of the MCU workspace, which can in turn be edited in the source code view.

Complete the following step to access the source code view:

1. Double-click on the file item (for example, `main.asm`) shown in the MCU workspace hierarchy in the **Design Toolbox**.

During simulation, additional debugging information displays to help you understand what is happening inside the MCU. For example, you can switch between viewing events happening in the high level source and at the assembly instruction level which also displays the actual opcodes for each instruction that are being executed by the MCU.

The source code view is not capable of displaying all this extra information. Instead, each MCU component in the circuit design has its own **Debug View** that displays debugging information.

Complete the following steps to access the **Debug View**:

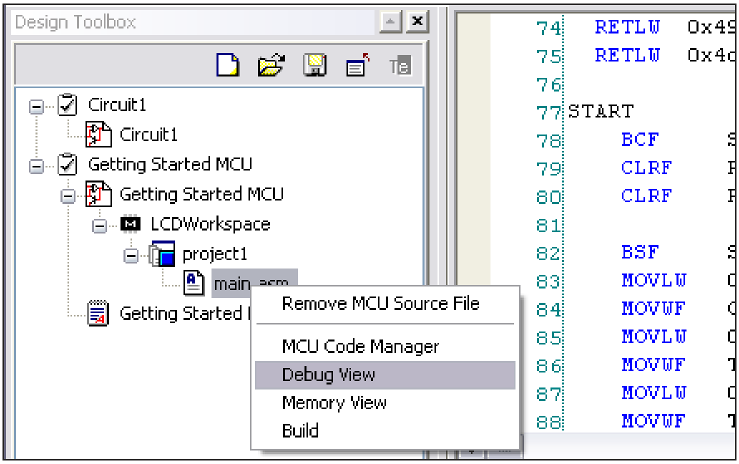
1. Select **MCU»MCU PIC 16F84A U1»Build**.

Note The **Debug View** is available only after you have successfully built your code, so the preceding step is only necessary once.

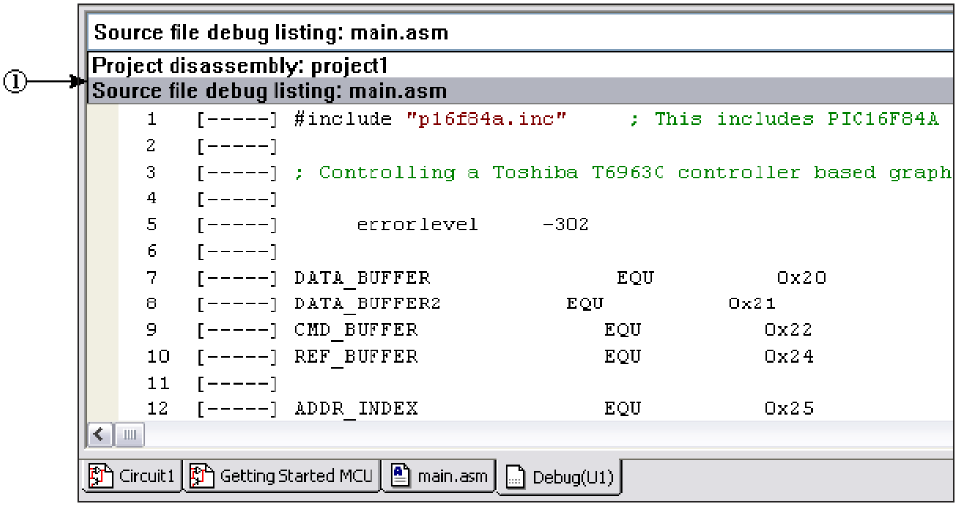
2. Select **MCU»MCU PIC 16F84A U1»Debug View**.

   Or

   Use the right-click context menu on an item in the MCU workspace of the **Design Toolbox**.



Another tab opens in the schematic capture workspace called **Debug(<reference designator of MCU>)**, in this case **Debug(U1)**.



1 Drop-down List

Use the drop-down list at the top of the **Debug View** to select between the disassembly instructions generated internally by Multisim and the listing file generated by the assembler or compiler (the format of the listing file is dependant on the tool that you choose to build your code).

In the LCD graphical display example, the code was written in assembly and built by the Microchip assembly tools. The Microchip assembler generates a listing file (`.lst`) that contains all of the opcodes generated for each assembly instruction. The debug listing view displays information from this listing file. Multisim generates the disassembly format using its internal disassembler to disassemble the opcode instructions into assembly instructions.

This format is not necessary for this example since the debug listing contains all of the information needed. In cases where an MCU project loads only the machine code (`.hex`) file, the disassembly view shows the disassembled opcode instructions so that you can see what's happening in the MCU. Since no listing file for MCU projects of this type is available, the disassembly view is very useful.

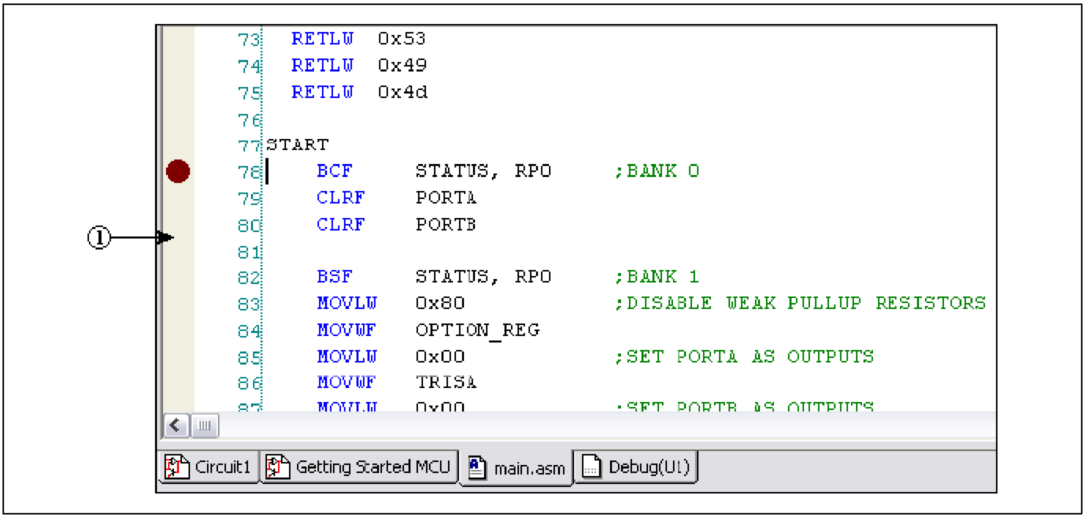
Adding a Breakpoint

You can add breakpoints in the source code view when simulation has stopped, as well as during simulation. You can add breakpoints to a microcontroller project in two ways.

One way is to add them in the source code view. In this example, the `main.asm` tab in the schematic capture workspace is the only source code view available.

Note If your MCU design contains more than one file, there will be a source code view for each of your source code files.

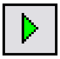
You can also set a breakpoint in the **Debug View** window. You can set breakpoints in the disassembly view or the debug listing view, but for this example, you will only use the debug listing view.



1 Grey Column

Complete the following steps to add a breakpoint in the source code view:

1. Open the **Debug View** for U1.
2. Double-click on `main.asm` in the **Design Toolbox**.
3. Scroll to the line just below the `START` label: `BCF STATUS, RP0`.
4. Double-click on the first (grey) column on the left side of the `main.asm` window next to the line `BCF STATUS, RP0`. A red circle appears at that location indicating that a breakpoint has been set at that line.

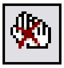


5. Select **Simulate»Run**. The simulation automatically pauses at the breakpoint that you have just set. The **Debug View** automatically displays with a yellow arrow showing where the MCU program execution is paused.

Complete the following step to remove the breakpoint:

1. Double-click on the breakpoint in the **Debug View** or the `main.asm` source code view.

   Or

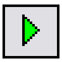


   Select **MCU»Remove all breakpoints** to remove all breakpoints.

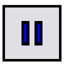
Note You can add and remove breakpoints in the **Debug View** in the same manner as the source code view.

Break and Step

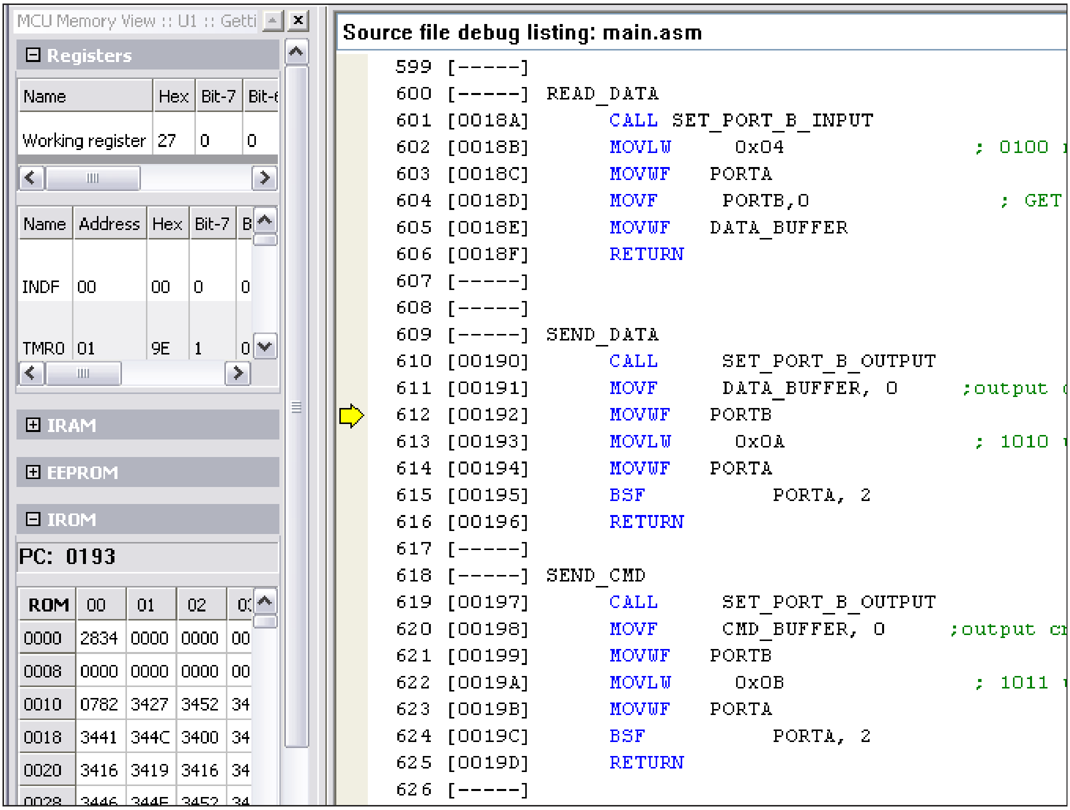
1. Select **MCU»Remove all breakpoints** to remove all breakpoints.



2. Go to the circuit design view (the **Getting Started MCU** tab) and select **Simulate»Run**. The words “Graphical LCD T6963CC for Multisim” start to display on the graphical LCD component.



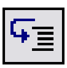
3. Select **Simulate»Pause**.
4. Go to the **Debug View** for U1 and notice that the line of code in the debug listing view where the MCU has stopped its execution is indicated by a yellow arrow in the left-most column.



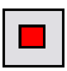
5. Select **MCU»MCU PIC16F84A U1»Memory View** to view the current state of the memory inside the microcontroller U1. Notice that the value of the program counter `PC` in the **IROM** section is one higher than the address value of the line the yellow arrow is pointing to. In the example in the above figure, the address in the **Debug View** is 192 and the PC value in the **Memory View** is 193.

Note If the MCU has not finished executing the current command when you pause the simulation, the value in the program counter will be the same as the address value.

You can also look at the other sections of the **Memory View** to see the values inside the other parts of memory in the microcontroller.



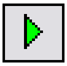
6. Click the **Step into** button in the **Simulation** tool bar.
7. The current instruction is executed and the simulation pauses at the next instruction.



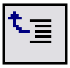
8. Select **Simulate»Stop**.

Break and Step Out

1. Place a breakpoint in the `SEND_DATA` subroutine at `MOVWF PORTB`.

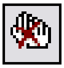


2. Select **Simulate»Run**. The simulation pauses at the breakpoint.

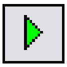


3. Click the **Step out** button in the **Simulation** toolbar to step out of the `SEND_DATA` subroutine.
4. The simulation executes all of the remaining instructions in the `SEND_DATA` subroutine and pauses at the first instruction after the call to the `SEND_DATA` subroutine.

Break and Step Into



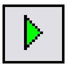
1. Select **MCU»Remove All Breakpoints**.
2. Place a breakpoint at the call to `SEND_DATA` where you had just stepped out of just above the yellow arrow.



3. Select **Simulate»Run**. The simulation pauses at breakpoint that you just placed.

4. Click the **Step Into** button on the **Simulation** toolbar. The simulation pauses inside the `SEND_DATA` subroutine.

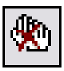
Break and Step Over



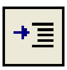
1. Select **Simulate»Run**. The simulation pauses at the same breakpoint that you set previously at the call to the subroutine `SEND_DATA`.

2. Click the **Step Over** button on the **Simulation** toolbar. The entire `SEND_DATA` subroutine is executed and the simulation pauses at the instruction after the `CALL SEND_DATA` instruction.

Run to Cursor



1. Select **MCU»Remove All Breakpoints**.
2. Click on a line inside the `SEND_DATA` subroutine since we know that this subroutine will be called again to send data to the LCD display.



3. Click the **Run to Cursor** button in the **Simulation** toolbar. The simulation runs until the MCU hits the instruction that you clicked on inside the `SEND_DATA` subroutine. It then pauses and places the yellow arrow next to that line.

A

Technical Support and Professional Services

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

- **Support**—Technical support 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 on demand training modules via the Services Resource Center. NI offers complementary membership for a full year after purchase, after which you may renew to continue your benefits.

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

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

Index