

GETTING STARTED GUIDE

NI Circuit Design Suite

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

- **Multisim Tutorial**—*Multisim* is the schematic capture and simulation program designed for schematic entry, simulation, and feeding to downstage steps, such as PCB layout. This tutorial introduces you to Multisim and its many functions.
- **Ultiboard Tutorial**—*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. This tutorial shows you how to place the components and traces for the design described in the Multisim Tutorial section. You will also learn how to autoplace parts and then autoroute them.

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

Contents

Multisim Tutorial	2
Introduction to the Multisim Interface	2
Multisim User Interface	3
Overview	4
Schematic Capture	5
Creating the File	5
Placing the Components	6
Wiring the Design	10
Simulation	12
Virtual Instrumentation	12
Analysis	15
The Grapher	17
The Postprocessor	17
Reports	18
Bill of Materials	18
Ultiboard Tutorial	19
Introduction to the Ultiboard Interface	19
Ultiboard User Interface	19
Overview	20
Opening the Tutorial	21
Creating a Board Outline	23
Placing Parts	24
Dragging Parts from Outside the Board Outline	24
Dragging Parts from the Parts Tab	25
Placing the Tutorial Parts	26

Placing Parts from the Database	26
Moving Parts.....	28
Placing Traces	29
Placing a Manual Trace	29
Placing a Follow-me Trace	31
Connection Machine Trace	31
Auto Part Placement	33
Autorouting Traces	33
Preparing for Manufacturing/Assembly	34
Cleaning up the Board	34
Adding Comments	34
Exporting a File	34
Viewing Designs in 3D	35

Multisim Tutorial

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

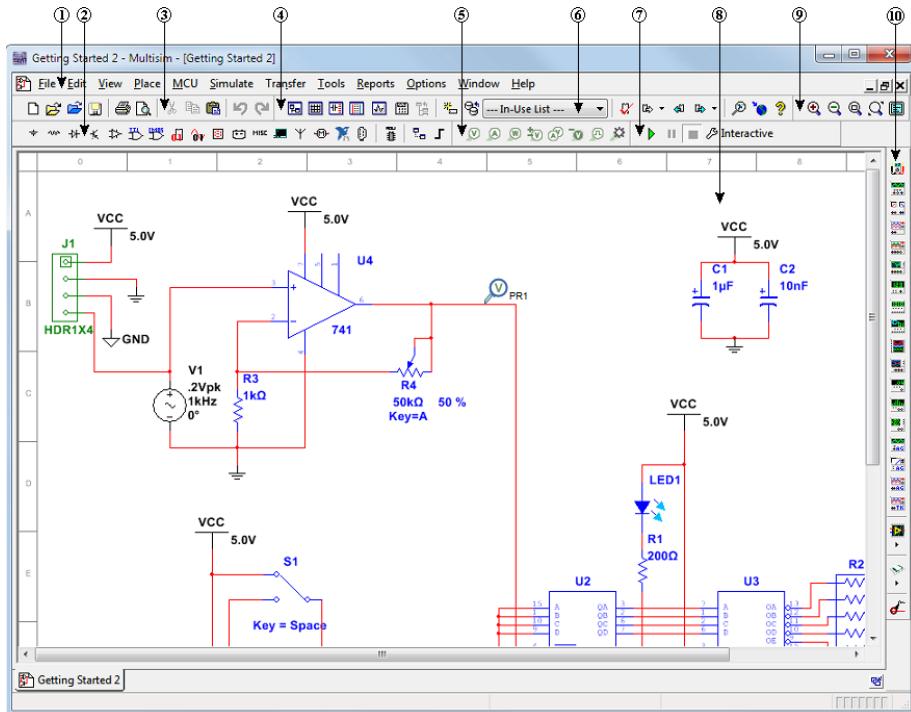
Introduction to the Multisim Interface

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

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

Multisim User Interface

The Multisim user interface includes the following elements:



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

Refer to the table below as needed:

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

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

Overview

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

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

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

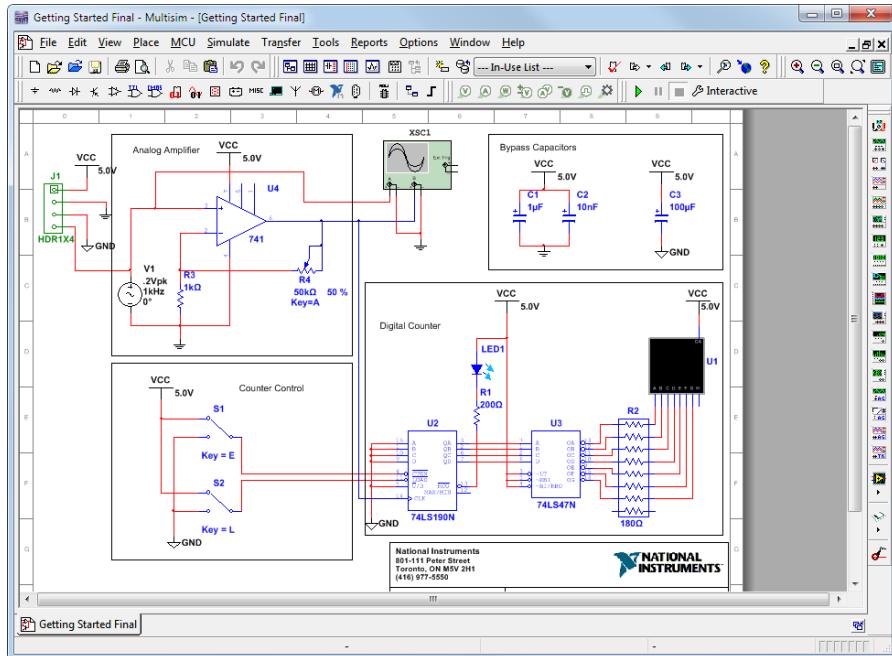
Helpful tips are indicated by an icon in the left column, for example:



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

Schematic Capture

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



Creating the File

Complete the following steps to create the design file:

1. Launch Multisim.
A blank file called Design1 opens on the workspace.
2. Select **File»Save as** to display a standard Windows Save dialog.
3. Navigate to the location where you wish to save the file, enter **MyGettingStarted** as the **File name**, and click the **Save** button.

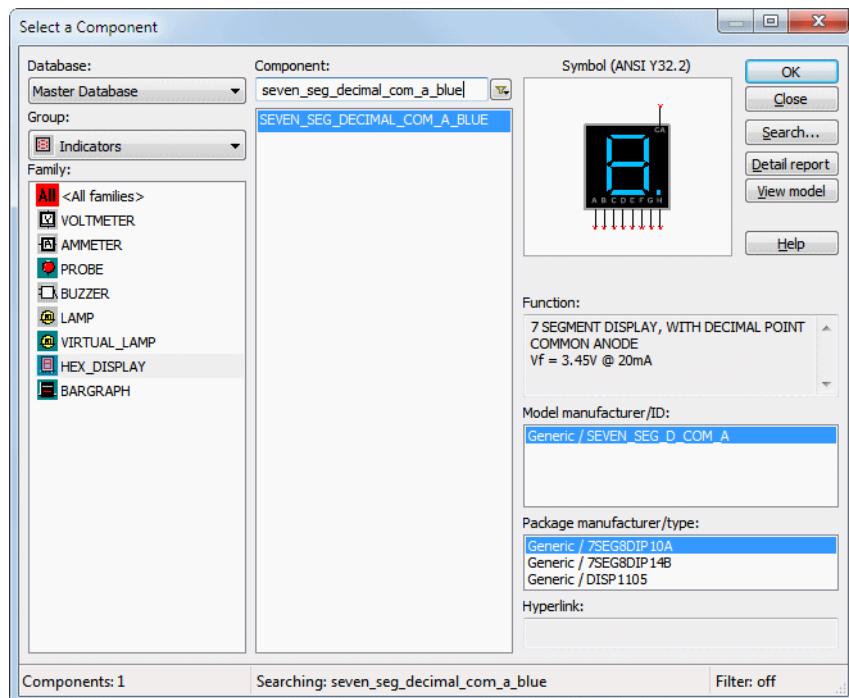


Tip To guard against accidental loss of data, set up a timed **Auto-backup** of the file in the **Save** tab of the **Global Options** dialog box.

Placing the Components

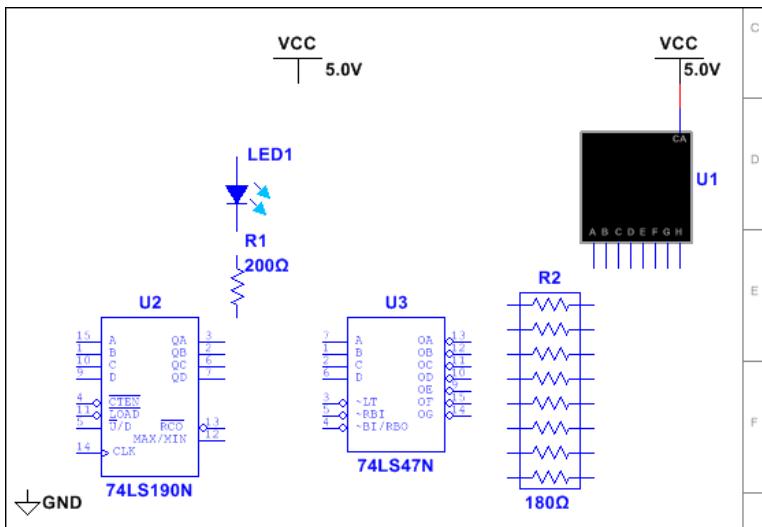
Complete the following steps to place the components on MyGettingStarted:

1. Select **Place»Component** to display the **Select a Component** dialog box.
2. Select the **Indicators** component **Group** and the **HEX_DISPLAY** component **Family**.
3. Select **SEVEN_SEG_DECIMAL_COM_A_BLUE** from the **Component** list and click **OK**.



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

1. 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.
2. Place the remaining components in the Digital Counter area as shown below.



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

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



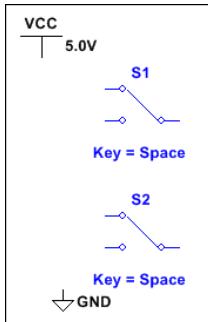
Tip The **Group** and **Family** location of each component is listed in the *Component Locations* section.



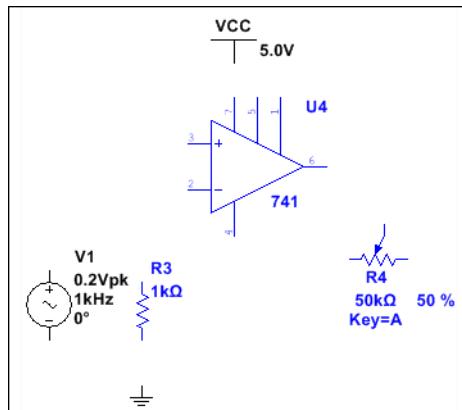
Note By default, resistors, capacitors and inductors have no package. However, to export components of this type to PCB layout in Ultiboard, you must select a package from the **Package manufacturer/type** field. Source and ground components have no available packages and do not export to layout.

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

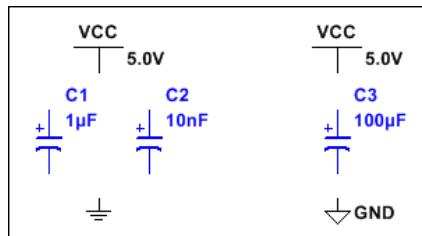
Right-click on each SPDT switch and select **Flip horizontally**.



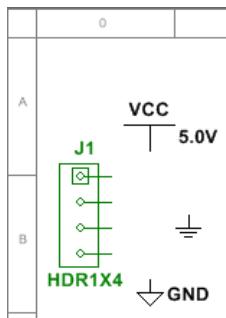
4. Place the components in the Analog Amplifier section as shown below, rotating as needed.
Double-click on the AC voltage source (V1) and change **Voltage (Pk)** to 0.2 V.



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



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



Component Locations

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

This tutorial only uses generic components from the Master database.



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

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

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



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

Wiring the Design

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



Tip You can wire the design that you placed on the workspace or you can use Getting Started 1 found in C:\Users\Public\Documents\National Instruments\Circuit Design Suite <version>\samples\Getting Started\.

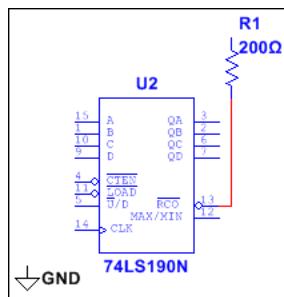
Complete the following steps to wire the design:

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

A wire appears, attached to your cursor.

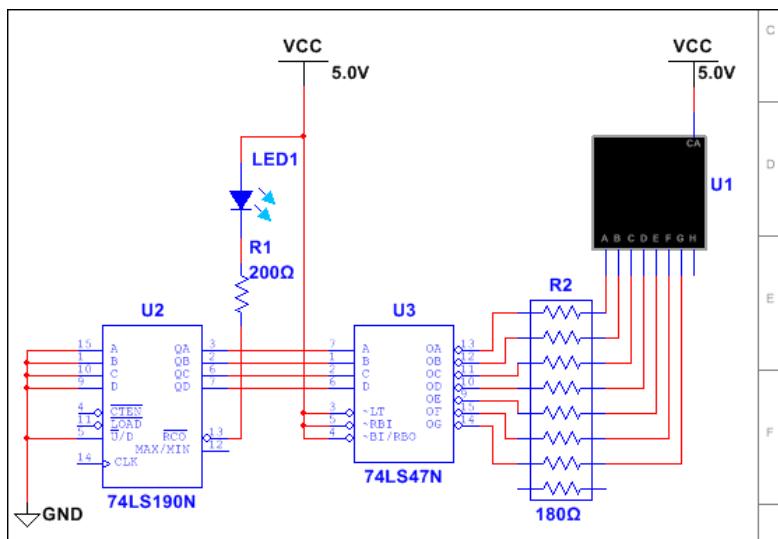
2. Click on a pin on the second component to finish the connection.

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



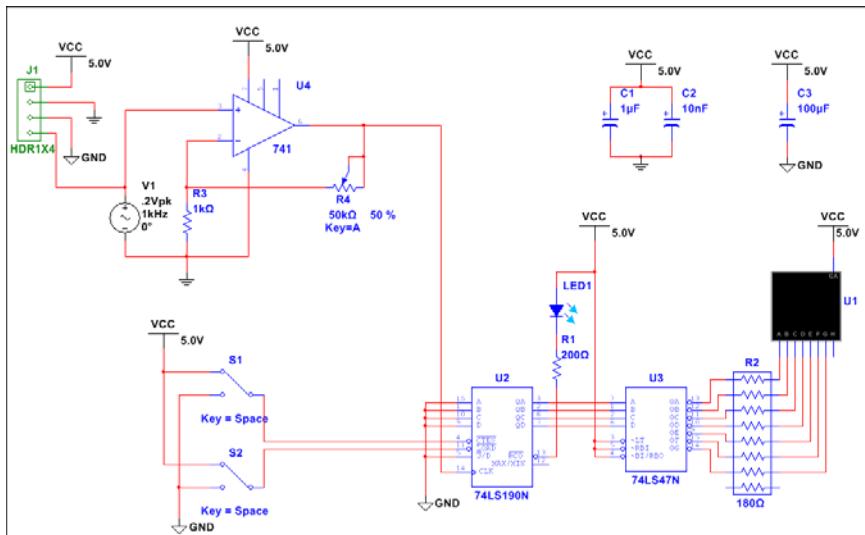
Tip You can also control the flow of the wire by clicking on points as you move the mouse. Each click “fixes” the wire to that point.

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



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

4. Finish wiring the design as shown below.



Simulation

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

Virtual Instrumentation

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



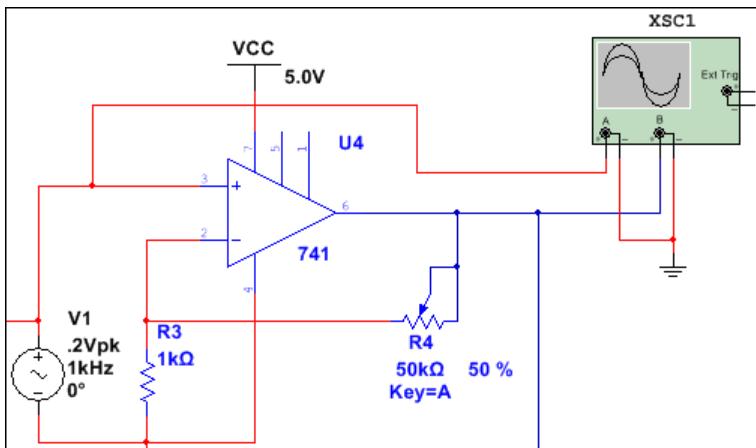
Tip You can also use Getting Started 2 found in C:\Users\Public\Documents\National Instruments\Circuit Design Suite <version>\samples\Getting Started.

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

Or

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

4. Wire the oscilloscope as shown below.



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

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



The output of the opamp appears on the oscilloscope.

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

The following displays on the oscilloscope:



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

8. Do the following:

- Press <E> while the simulation is running to enable or disable the counter.
- Press <L> to load zeros into the counter.
- Press <Shift-A> to observe the effect of decreasing the potentiometer's setting.
Repeat, pressing <A> to increase.



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

Analysis

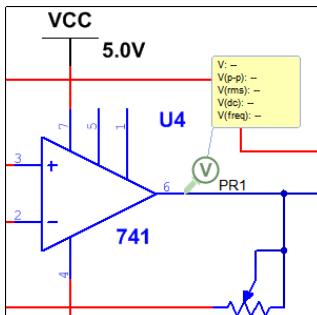
In this section, you will use **AC Sweep** to verify the frequency response of the amplifier.

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

1. Open Getting Started Final found in C:\Users\Public\Documents\National Instruments\Circuit Design Suite <version>\samples\Getting Started\.
2. Click on the **Place voltage probe** button in the **Place probe** toolbar.



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



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

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

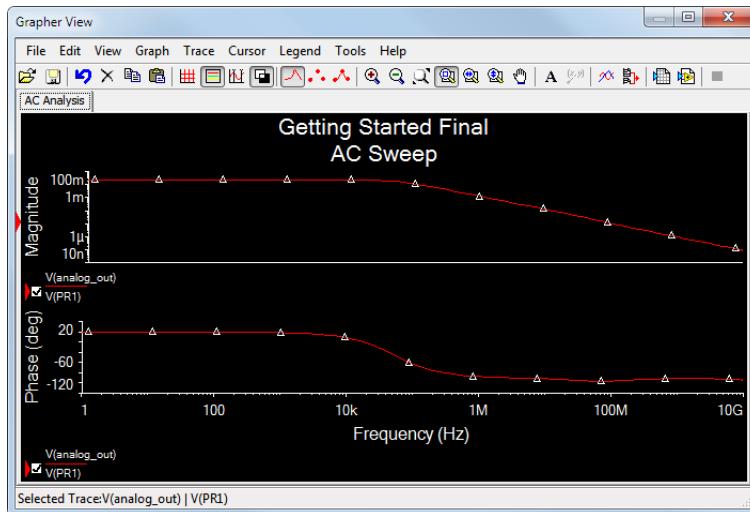
The **Analyses and Simulation** dialog box displays.



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

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

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



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



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

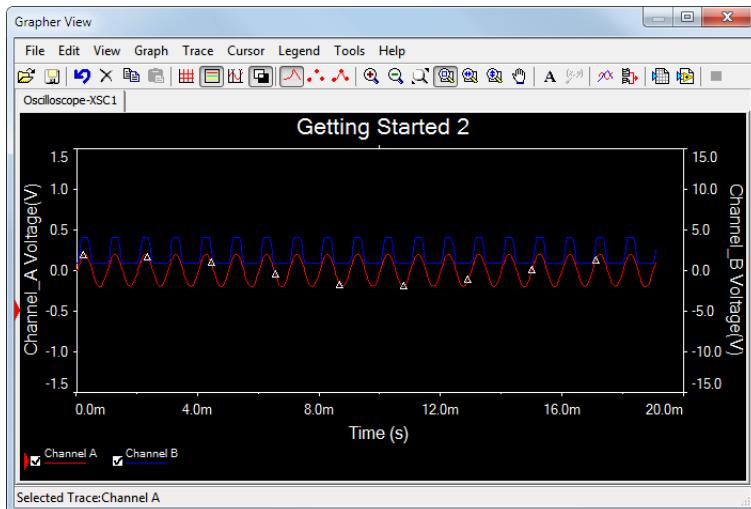


The Grapher

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

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

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



The Postprocessor

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

Reports

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

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

Bill of Materials

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

Information provided for each component includes:

- quantity required.
- description, including the type of component (for example, resistor) and value (for example, $200\ \Omega$).
- Reference designator.
- package name.

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

Bill of Materials (From Document: Getting Started Final)				
	Quantity	Description	RefDes	Package
1	1	RPACK_VARIABLE_2X7, 180Ω	R4	IPC-2221A/2222\DI ^P -14
2	1	SEVEN SEG_DECIMAL_COM_A_BLUE	U1	Generic\7SEG8DIP10A
3	1	74LS, 74LS47N	U3	IPC-2221A/2222\NO16
4	1	GENERIC, HDR 1X4	J3	Generic\HDR 1X4
5	1	LED_blue	LED1	Ultiboard\LED9R2_5Vb
6	1	RESISTOR, 1kΩ 5%	R3	IPC-7351\Chip-R0805
7	1	POTENTIOMETER, 50kΩ	R2	Generic\LIN_POT
8	1	RESISTOR, 200Ω 5%	R1	IPC-7351\Chip-R0805
9	1	OPAMP, 741	U4	IPC-2221A/2222\DI ^P -8
10	1	CAP_ELECTROLIT, 1μF	C1	IPC-7351\Chip-C1210
11	1	CAP_ELECTROLIT, 10nF	C2	IPC-7351\Chip-C1210
12	1	CAP_ELECTROLIT, 100μF	C3	IPC-7351\Chip-C1210
13	2	SPDT,	J1, J2	Generic\SPDT
14	1	74LS, 74LS190N	U2	IPC-2221A/2222\NO16

Ultiboard Tutorial

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

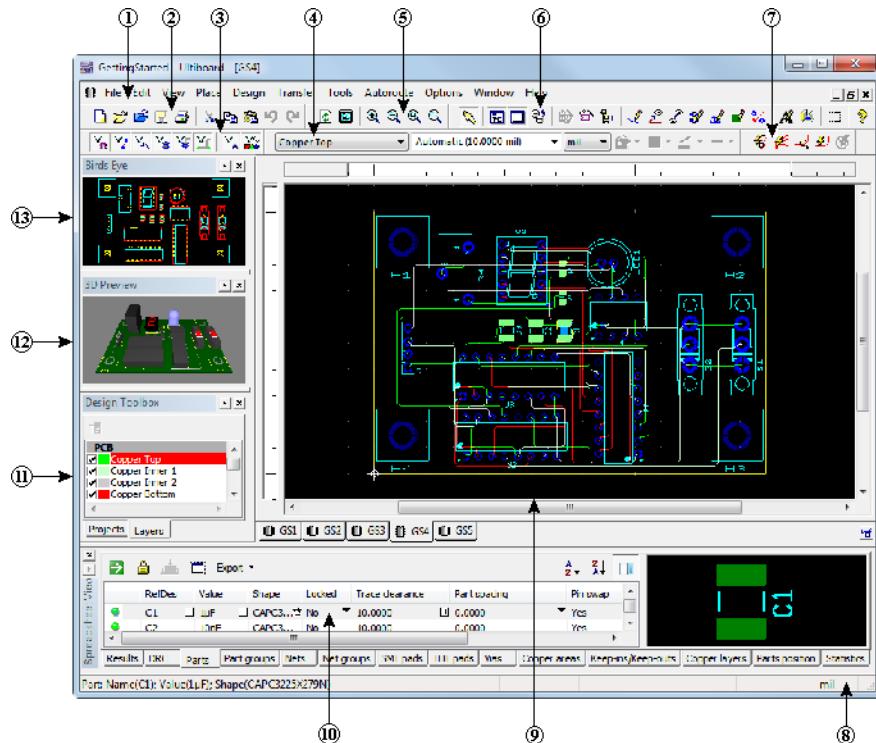
Introduction to the Ultiboard Interface

Ultiboard is the PCB layout application of 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

The Ultiboard user interface includes the following 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.

Overview

This tutorial shows you how to place the parts and traces for the circuit described in the Multisim Tutorial section.

If you wish to skip steps, or only complete specific sections of this tutorial, you can use the different tabs, found in C:\Users\Public\Documents\National Instruments\Circuit Design Suite <version>\samples\Getting Started\GettingStarted:

- Tab GS1—The design with all components imported from Multisim and placed outside of a default board outline. This is the starting point for this tutorial.

- Tab GS2—The design with the imported components placed on the correctly sized board.
- Tab GS3—The design with the imported components placed on the board, and four holes placed from the database.
- Tab GS4—The design with all of the parts placed, and all of the copper (traces) routed.
- Tab GS5—The design with some parts placed and locked. Locked parts are not moved during autoplacement.

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



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

Opening the Tutorial

Complete the following steps to open the tutorial file:

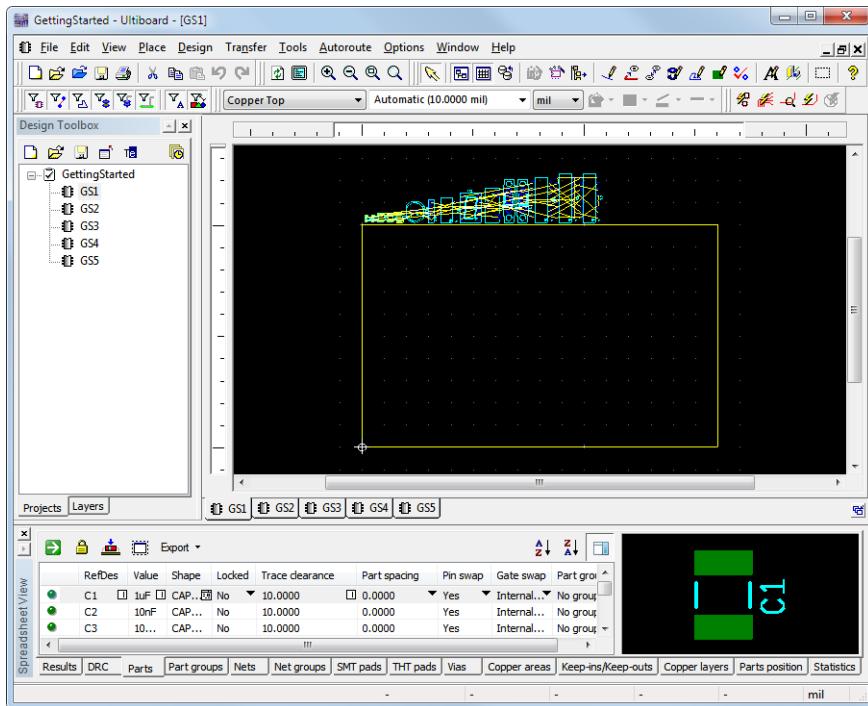
1. Launch Ultiboard, select **File»Open samples** and double-click on the **Getting Started** folder to open it.
2. Select **GettingStarted** and click **Open**.

The project file is loaded into Ultiboard.



Tip For instructions on transferring 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 a default 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. Select **Multi-layers constructed with double-sided boards (layer pairs)**, and click **Next**.
6. Change the number of **Layer pairs** to 2 and leave **Blind vias** unchecked.
7. 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.

8. 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 to move the board outline:

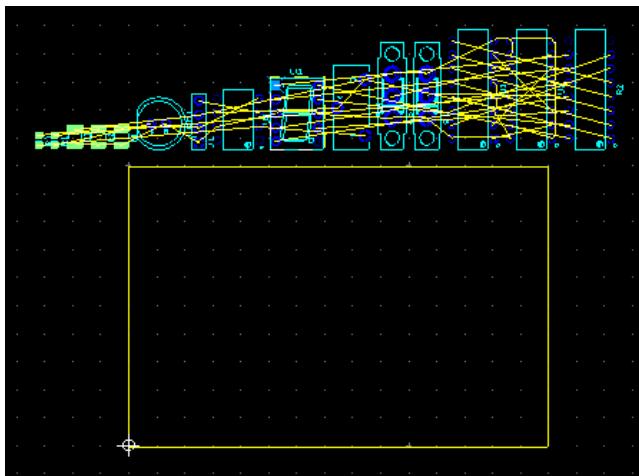
1. Click anywhere on the board outline in the workspace and drag the board to a location just below the row of parts.

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



Placing Parts

You can place parts on your GS1 board 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.

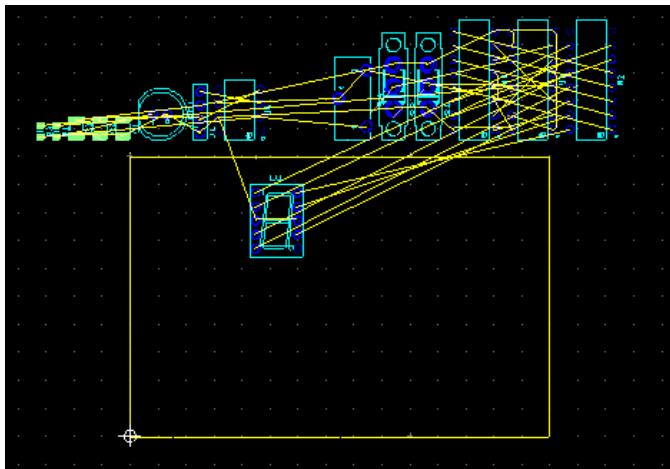
Complete the following steps to drag U1 from outside the board outline:

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.

- Click on U1 (the 7-segment display) and drag it to the location shown in the figure below. As you move the part, the ratsnest moves with it. A ratsnest is a straight line connection between pads, indicating their connectivity. It is not a trace. 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.



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

Notice that the green dot 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:

- In the **Parts** tab, scroll down to J1.

RefDes	Value	Shape	Locked	Trace clearance	Part spacing	Pin swap
C3	100uF CAPC3...	No	No	10.0000	0.0000	Yes
J1	HDR1x4	HDR1x4	No	0.0000	<input type="checkbox"/> Use design rule defaults	Yes
LED1	LED_blue	LED9R...	No	10.0000	<input type="checkbox"/> Use design rule defaults	Yes

- Click on the green dot beside J1 in the table and drag the component from the **Parts** tab onto the workspace.

J1 is attached to the cursor.

- Drop J1 on the left edge of the board, approximately in the middle.

In the **Parts** tab, J1's green dot is now slightly brighter, indicating that the part has been placed.

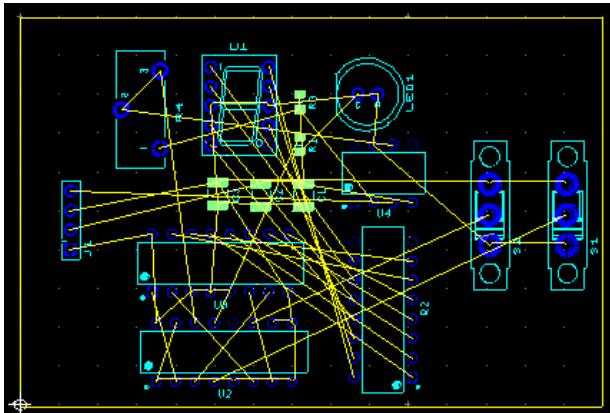
Placing the Tutorial Parts

Using any method or combination of methods, lay out the design as shown 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**.

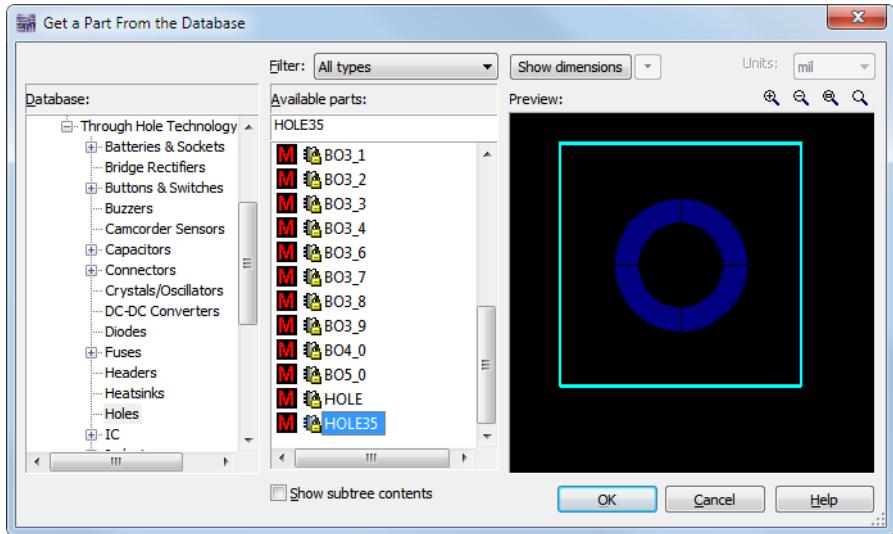


2. The **Get a Part From the Database** dialog box opens.
3. In the **Database** panel, expand the **Ultiboard Master»Through Hole Technology Parts** category and select the **Holes** category.

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

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

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



- Click **OK**.

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

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

- Move the cursor over the board.

The part is attached to the cursor.

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

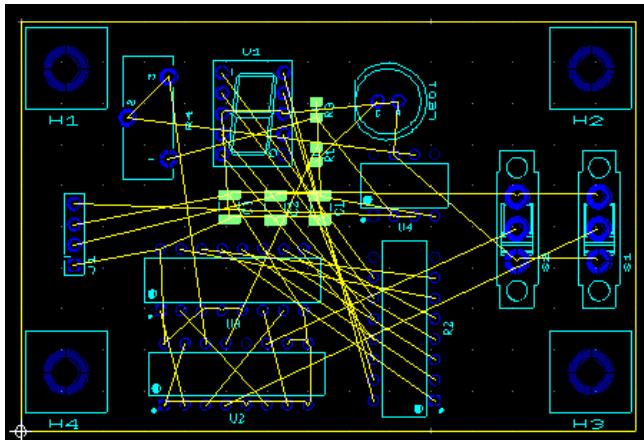
The **Enter Reference Designation for Part** dialog box reappears, with the **RefDes for shape HOLE35** automatically incremented to H2.

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

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

11. Click **Cancel** to stop, and click **Cancel** again.

The **Get a Part From the Database** dialog box closes.



Moving Parts

You can use similar 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 of the **Spreadsheet View** (indicated by a bright green dot 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 cursor.

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 to clear the selection, 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 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**.

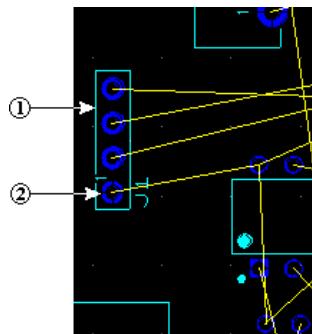
Complete the following steps to place a trace manually:

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 J1, toward the left-hand part of the board and find the start pin shown below:



1 Part J1

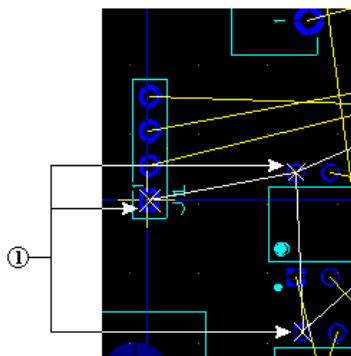
2 Start Pin



Tip If you have trouble locating the part, select the part in the **Parts** tab and click the **Find the selected part** button. The part is selected 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 and in the netlist.

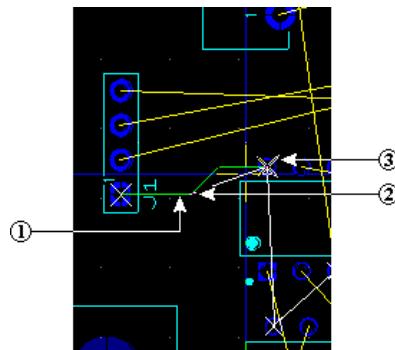


1 Pins in the Same Net

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 J1.
3. Click on the pin indicated by the "X" on U4.
Ultiboard draws the trace for you.
4. Right-click and choose **Cancel** to stop placing traces.

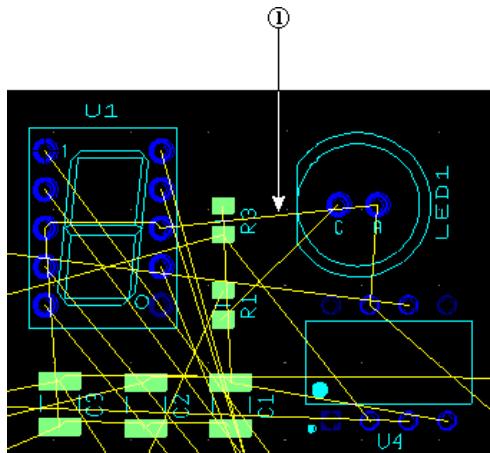
Connection Machine Trace

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.

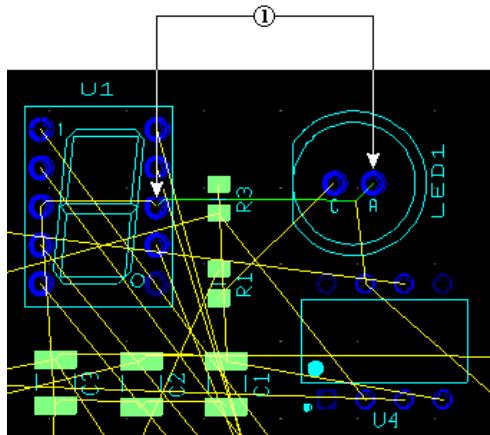


1 Click Ratsnest

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 section, you can use Ultiboard's 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, S1, S2, 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

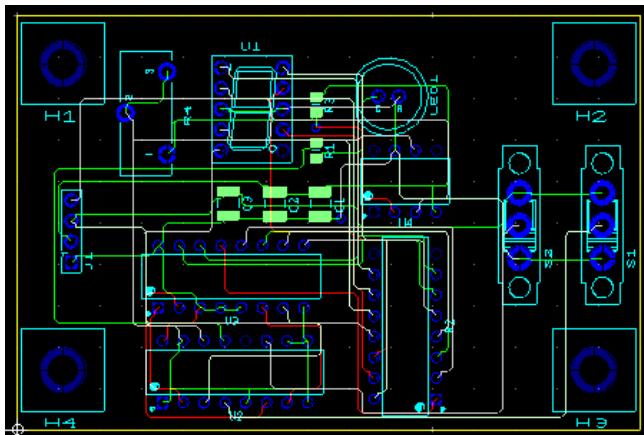
You can place traces in Ultiboard using the methods described earlier in this section, or automatically route the traces as described below.

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.



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 autoplace 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

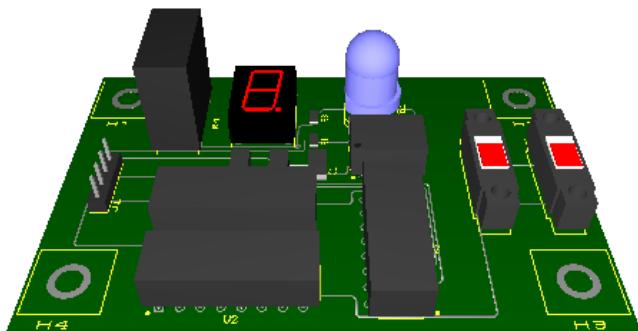
You can export a number of file types, including Gerber. An exported file contains complete information describing how a finished board is to be manufactured.

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.



Refer to the *NI Trademarks and Logo Guidelines* at ni.com/trademarks for more information on National Instruments trademarks. Other product and company names mentioned herein are trademarks or trade names of their respective companies. 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 Patents Notice* at ni.com/patents. You can find information about end-user license agreements (EULAs) and third-party legal notices in the readme file for your NI product. 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. NI MAKES NO EXPRESS OR IMPLIED WARRANTIES AS TO THE ACCURACY OF THE INFORMATION CONTAINED HEREIN AND SHALL NOT BE LIABLE FOR ANY ERRORS. U.S. Government Customers: The data contained in this manual was developed at private expense and is subject to the applicable limited rights and restricted data rights as set forth in FAR 52.227-14, DFAR 252.227-7014, and DFAR 252.227-7015.

© 2006–2016 National Instruments. All rights reserved.