Instruktioner för TSTE93

- Starta programmet med "Start/All Programs/NI Ultiboard 14.0"
- Exempelfilerna som nämns i övningarna nås med "File/Open samples…/Getting Started/"
  + Använd helst din egen Multisim-konstruktion då exemplen kan vara ändrade
  + Annars kopierar du filerna innan du använder dem för att inte ändra i originalen
- Tillvalen "Connection Machine" och "Autoplace" saknas i den aktuella installationen
  + Hoppa därför över övningarna "Connection Machine Trace" och "Auto Part Placement"

Contents Part 2

Ultiboard Tutorial

This tutorial will familiarize you with Ultiboard's basic functions.

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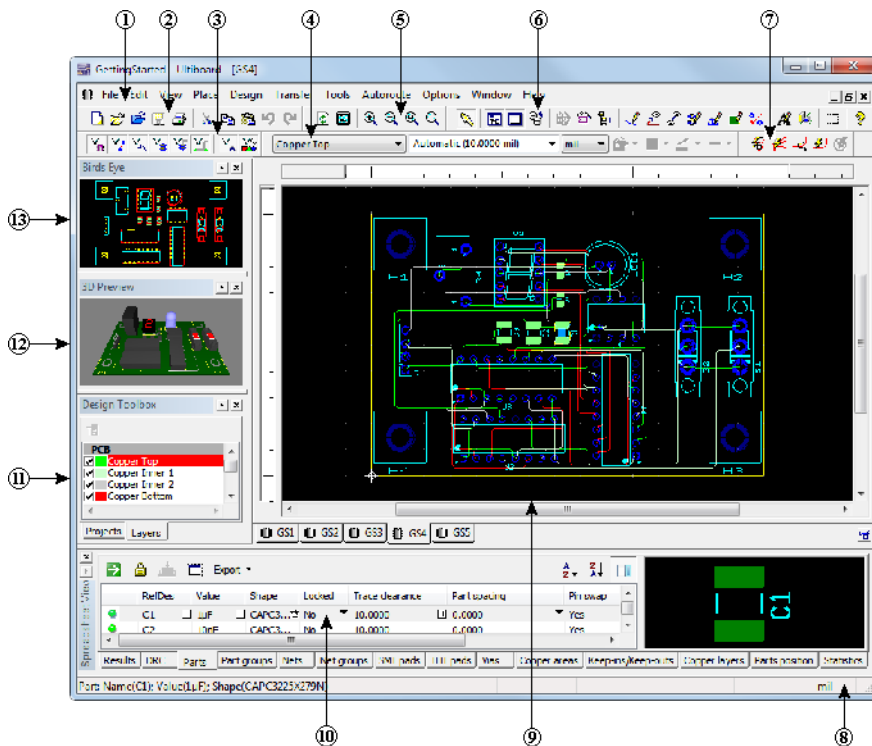
1	Menu Bar	5	View Toolbar	8	Status Bar	11	Design Toolbox
2	Standard Toolbar	6	Main Toolbar	9	Workspace	12	3D Preview
3	Select Toolbar	7	Autoroute Toolbar	10	Spreadsheet View	13	Birds Eye View
4	Draw Settings Toolbar						

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.

	Element	Description
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

The following sections show 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 ...\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.
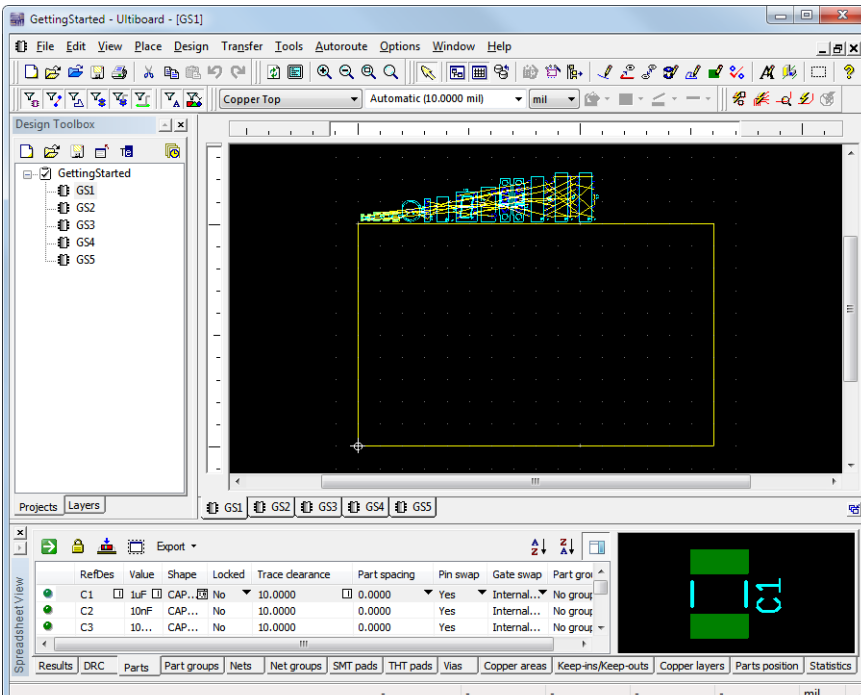
💡 **Tip** To select a design from the project (for example, GS1), click on its tab, or click on its name in the **Projects** tab of the **Design Toolbox**.

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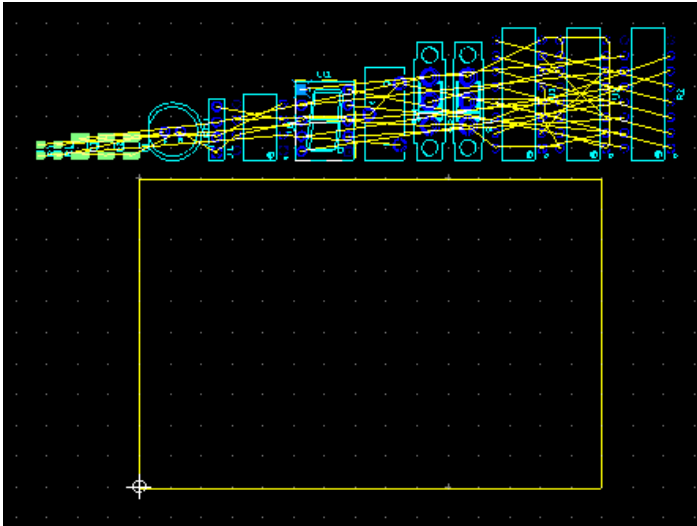
9. Click anywhere on the board outline in the workspace and drag the board to a location just below the row of parts.

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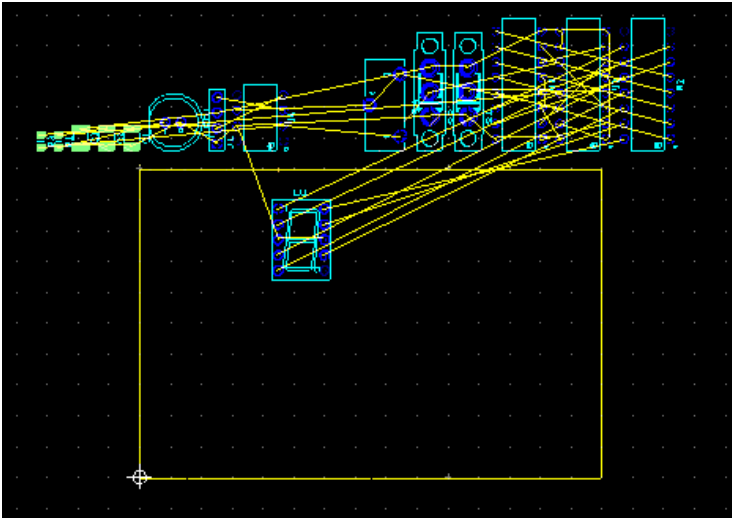
Complete the following steps to drag U1 from outside the board outline:

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.

3. 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.
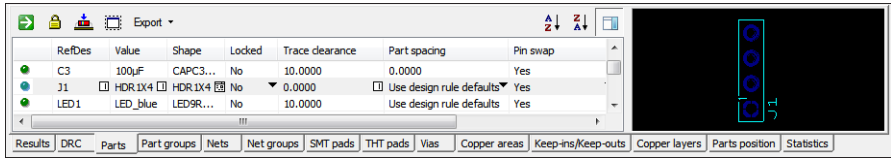

4. Go to the **Parts** tab in the **Spreadsheet View** and scroll to U1.

   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:

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

2. 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.

3. 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.
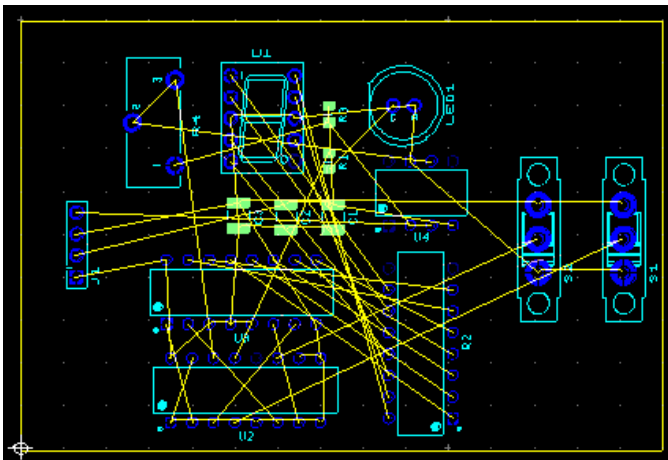
Placing the Tutorial Parts

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

    The part displays in the **Preview** panel.
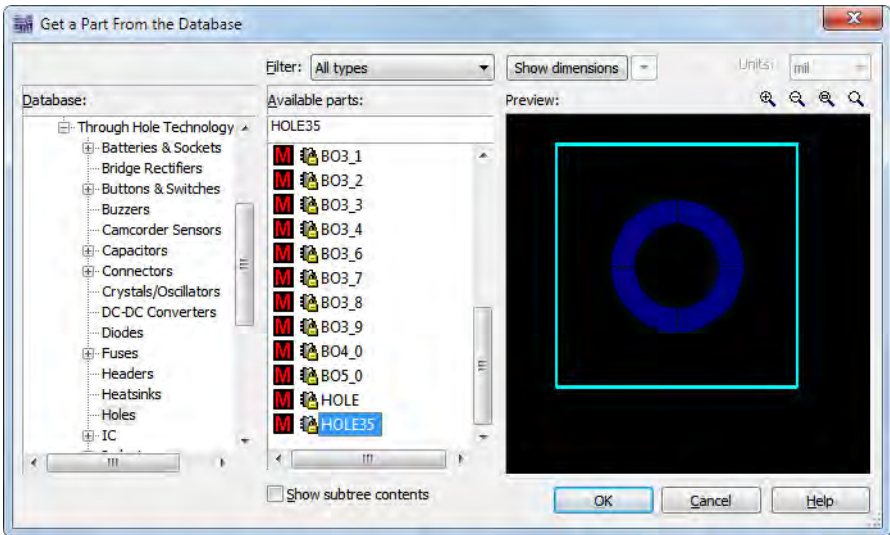
5. Click **OK**.

    The **Get a Part From the Database** dialog box disappears, and the **Enter Reference Designation for Part** dialog box displays.
6. Enter the **RefDes for shape HOLE35** (H1) and **Value** (HOLE) and click **OK**.
7. Move the cursor over the board.

    The part is attached to the cursor.

8. 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.

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

10. 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.
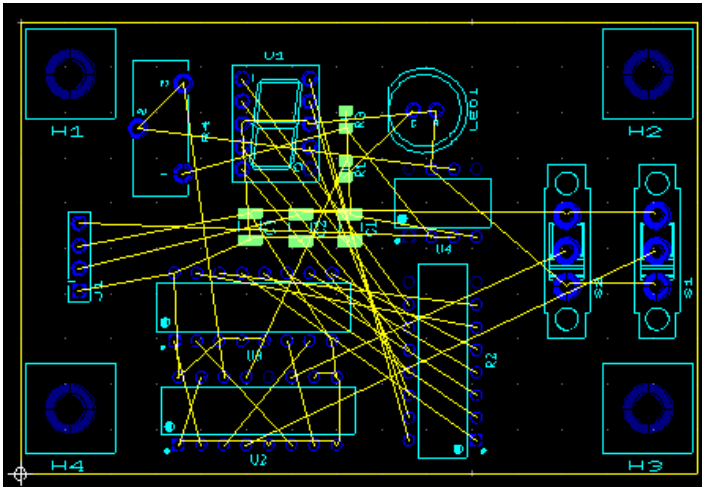
Moving Parts

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:
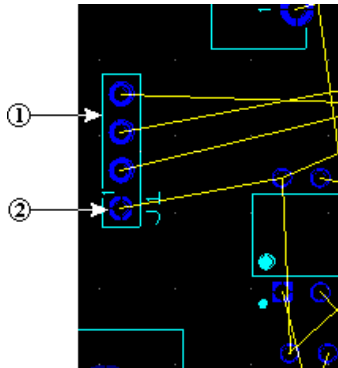
| 1 | Part J1 | 2 | Start Pin |

> 💡 **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.
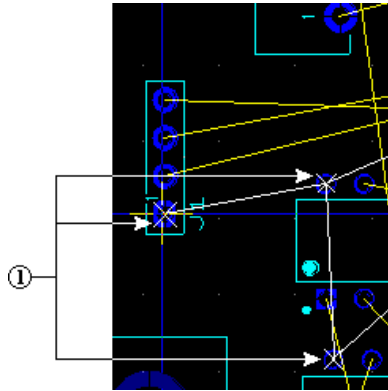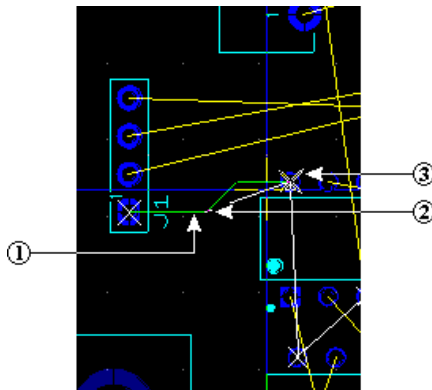
1	Pins in the Same Net

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.
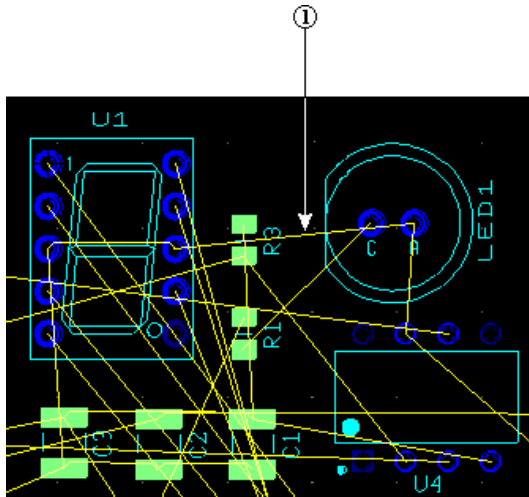4. Right-click and choose **Cancel** to stop placing traces.

Connection Machine Trace

Complete the following steps to place a **Connection Machine** trace:
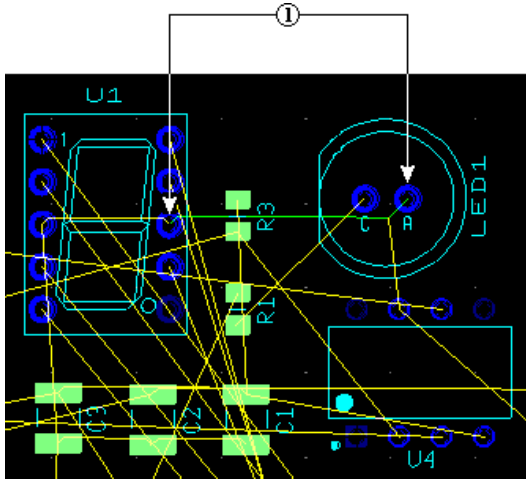
1. Choose **Place»Connection Machine**.

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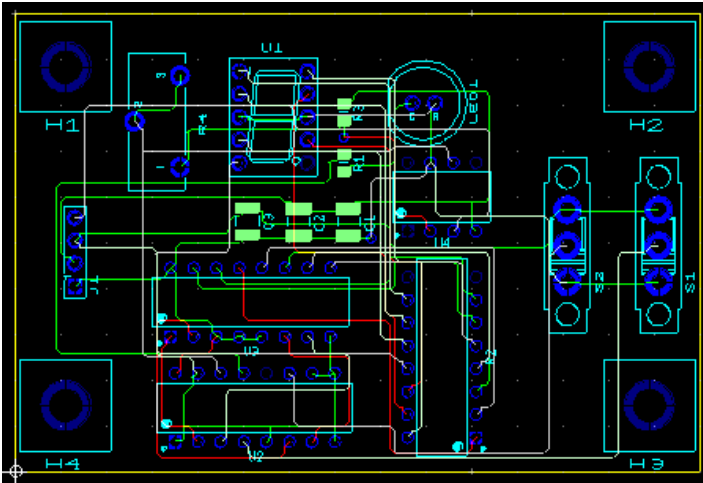
The autorouter can be stopped at any time and you can make manual changes as desired. When you restart the autorouter, it will continue with the changes you made. Remember to lock any traces that you have placed manually and do not wish to be moved by the autorouter.

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

Preparing for Manufacturing/Assembly

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

Cleaning up the Board

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

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

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

Adding Comments

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

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

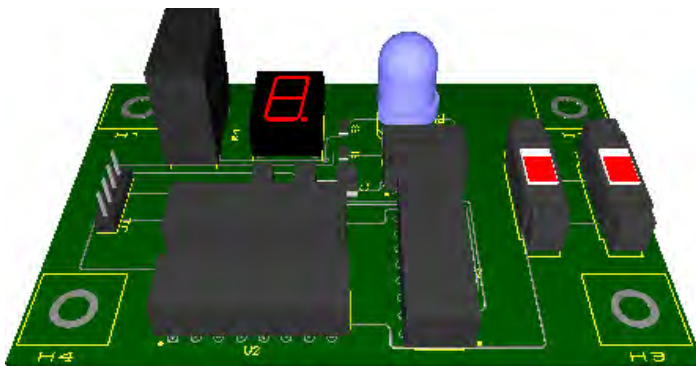Refer to the *Ultiboard Help* for details.

Exporting a File

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

Refer to the *Ultiboard Help* for details.

Viewing Designs in 3D

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

Refer to the *Ultiboard Help* for details.