Printed circuit board layout with MultiSim and Ultiboard

The basic circuit – components and interconnections – is laid out in MultiSim. The process is very similar to the schematic layout procedure in PSPICE, with a few differences. (Some are better and some are worse than PSPICE). Once the circuit schematic is completed, you can perform SPICE simulations and/or begin the layout process in Ultiboard.

We will pick up the basics of using MultiSim and Ultiboard using a specific example – the simple Altoids amp that many of you have built on perfboard as part of Audio Club. (To see the complete circuit, see the schematic at the bottom of page 16.)

MultiSim and Ultiboard are Windows-only programs. All of the departmental Windows machines have fully licensed versions installed. You can also install a limited-time student version on your own computer. The trial period should be long enough to get you through the end of the semester. If you are Mac user (like me), you are not totally out of luck. I use Boot Camp on my laptop to boot into Windows.

There are many other PCB tools available. Eagle is a popular option – it is cross-platform and there is a free (but limited) version. KiCad is another free, multi-platform layout tool. In the future, we might develop tutorials for these other programs. However, because we need to start somewhere and because MultiSim and Ultiboard are fully supported by the department, we will use these initially.

Circuit schematic and component selection

1. Open Multisim by selecting All Programs»NI Multisim 14.0. The main schematic drawing window opens.
2. **Select a component**

From the menu, select **Place»Component**. The **Select a Component** – or Component Browser (CB) – window appears.

There are thousands of components available, everything from generic parts to very unique specialty parts. There are many form factors (footprints) available for the various items – this is important for the PCB layout process.

The components are collected in various databases. (We will use the Master database for all of our work here.) Within a database, components are arranged in **Groups**. Within each group are **Families**. From within a particular family, you can choose a specific component to place in the schematic.

Note: The toolbar near the top of the window has many clickable icons that will open the CB with a particular family chosen. Use these as short-cuts once you become more adept with MultiSim.
3. Choose the op amp
The first part we will place is the dual op amp used in the Altoids amp — the TL072 from Texas Instruments. Choose the Analog group. From the options, choose the OPAMP family. There are many op amps available. You can scroll through until you find the specific one or you can use the search box at the top of the component column. If you search for TL072, you see that there are still many options! Most of the options relate to different packages or specification details. If you plan to do SPICE simulations only, it probably isn’t important which TL072 option you use. However, if you are working towards a PCB layout, you need to pay attention to the package type (footprint). In this case, we are interested in a through-hole PCB design, so we will want a DIP (dual in-line) package. Choose the TL072ACP option, which comes in a standard DIP 8 package that is typical for dual op amps.
4. **Place the op amp.**

The TL072 chip has two op amps in one package, labeled A and B – you can use either or both in the schematic. For simulations, it may not matter which you select, but for a PCB layout, it is important to be specific about which amp you use.

When you are returned to the drawing window, you are presented with a small dialog box that allows you to pick the particular op amp. Choose A initially. The dialog box disappears and the cursor will have a “ghosted” view of the op amp attached to it. Move the cursor to a spot in the schematic space and click to place the op amp. (It can be moved later, if needed.)

The little “A or B” dialog for choosing op amps re-appears. Our stereo amp has two channels, so select “B” for U1 and place the second op amp in the drawing window. (Don’t choose either A or B in the “new” line – that will create a second TL072 chip.) Once the second op amp has been placed, you will be presented with the “A or B” dialog one more time. Since we are done with op amps, click cancel and you are returned to the CB window. (In MultiSim, you are always returned to the Component Browser, under the assumption that you will want to place more components.) To leave the CB and return to the schematic, click “Close”. You can always get back to the CB by using the Place menu.
5. **DC voltage sources**

The op amps will need power supplies, and we can add those now. Use the **Place->component** menu item bring up the CB. Choose the **Sources** group and select the **POWER_SOURCES** family. In the component list column, select the **DC_POWER** component. Click OK to close the CB and return to the schematic window.

Place the source off to the right side somewhere. Back in the CB, select a second DC source. Place the second DC source near the first.
6. **Ground**

This might be a good time to add the first ground connections. Go back to the CB. The ground symbol is also in the **Sources** group, part of the **POWER_SOURCES** family.

![Image of a circuit diagram with ground connections]

Put the ground somewhere near the voltage sources. Note that we will be adding many ground symbols into the circuit later. You can always go back to the CB each time, but copy-and-paste works well, too.
7. **Resistors**

Back in the component browser, select a 1-kΩ resistor. Resistors are in the **Basic** group and have their own family. You choose the specific resistor value from a ridiculously long list. (The value can be changed later, if needed.) Also, be sure to pick a footprint for the resistor. For this tutorial, use the one labeled RES1300-700x250. Drop the resistor into the schematic somewhere near the op amps.

After adding the first resistor, add three more 1-kΩ resistors. Then add two 15-kΩ resistors and two 100-kΩ resistors for the feedback networks.
A comment about resistor footprint notation

The clip below is from the website “http://www.resistorguide.com/resistor-sizes-and-packages/”.

Axial resistor size

The size of axial resistors is not as standardized as the SMD resistors and different manufacturers often use slightly different dimensions. Furthermore, the size of an axial resistor depends on the power rating and the type of resistor such as carbon composition, wirewound, carbon or metal film. The following drawing and table give an indication of the dimensions of common carbon film and metal film axial resistors. Whenever the exact size needs to be known, always check the manufacturer's datasheet of the component.

<table>
<thead>
<tr>
<th>Power rating</th>
<th>Body length (l)</th>
<th>Body diameter (d)</th>
<th>Lead length (a)</th>
<th>Lead diameter (da)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Watt</td>
<td>mm</td>
<td>mm</td>
<td>mm</td>
<td>mm</td>
</tr>
<tr>
<td>1/8 (0.125)</td>
<td>3.0 ± 0.3</td>
<td>1.8 ± 0.3</td>
<td>28 ± 3</td>
<td>0.45 ± 0.05</td>
</tr>
<tr>
<td>1/4 (0.25)</td>
<td>6.5 ± 0.5</td>
<td>2.5 ± 0.3</td>
<td>28 ± 3</td>
<td>0.6 ± 0.05</td>
</tr>
<tr>
<td>1/2 (0.5)</td>
<td>8.5 ± 0.5</td>
<td>3.2 ± 0.3</td>
<td>28 ± 3</td>
<td>0.6 ± 0.05</td>
</tr>
<tr>
<td>1</td>
<td>11 ± 1</td>
<td>5 ± 0.5</td>
<td>28 ± 3</td>
<td>0.8 ± 0.05</td>
</tr>
</tbody>
</table>

We see that a standard 1/4-W resistor is about 6.5 mm long and 2.5 mm wide, and we would like to choose a footprint that matches this size (at least approximately). The clip below is from the website “http://digital.ni.com/public.nsf/allkb/E81C051F276C5BE0862579E2006975D5” and explains the naming convention used in MultiSim for passive, through-hole components.

The IPC 2221A/2222 naming convention for the RLC components begin with RES, IND or CAP which stands for resistor, inductor and capacitor respectively. The unit is in millimeter (mm) and it has two significant digits. The first number represent the center-to-center pin distance, next is the body length and finally the body width. For example, the footprint name IND 1400-800X350 is an inductor with the following dimensions:
Thus we see that the resistor footprint denoted RES1300-700x250 corresponds to a resistor 7mm long by 2.5 mm wide, which is about right. The spacing between the through-holes where the leads will be soldered is 13 mm, which corresponds to about 5 “holes” in a standard perfboard or solder-less breadboard that has 0.1-inch hole spacing. (5 “holes” = 0.5 inches ≈ 12.5 mm.) This also sounds about right. If you are using bigger (i.e. higher power) resistors, you would will need to use correspondingly bigger footprints.

8. Capacitors
Now, add two 0.1-µF capacitors for the input filters. Capacitors are also in the Basic group and have their own family. As with the resistors, you choose the specific value from the long, long list. Choose the capr250-500x250x550 footprint. Then add two 100-µF capacitors (footprint cappr750-1600x3150). These are the power supply bypass capacitors.
Regarding capacitor footprint and sizing:

The footprint notation for capacitors is similar to that for resistors with a couple of differences. First the capacitors include a “height” in addition to width and length, so that some definitions will have three dimensions. Typical ceramic and polymer capacitors will have this sort of notation and the dimensions will be quite small. (Recall the tiny caps in your 201 kits.) Typical electrolytic capacitors have a cylindrical shape with both leads coming out of one end (known as the “radial” configuration) and will usually be physically much larger than ceramic or polymer caps. Radial capacitors will probably be mounted so that they are standing “upright”. The cylindrical shape would be denoted by a diameter and a height, and the lead separation will probably be smaller than the diameter. Electrolytic capacitors can also be configured to have one lead coming out each end (the axial configuration), in which case the assumption would be that they will be “lying down” like a resistor. In that case, the lead separation will be much larger than the other dimensions. The axial configuration is less common than the radial, because “laying down” takes up more board area than “standing up”.

It is important to measure the dimensions of the capacitors (and inductors) that you plan to use in a circuit so that you can choose the correct footprint.

Finally, you can get some indication of the shape of the footprint by clicking the “Detail report” button in the upper right portion of the CB window. It will give you some information about the footprint and a drawing of the basic shape, although it is often difficult to discern the details of the figures because the graphics are not very good.
9. **LEDs**

Add two red LEDs for the power supply indicators. (If you prefer, you can use other colors, but the color makes no difference in either simulations or PCB layout.)
10. **Input Signal sources**

Add two sinusoidal voltage sources to serve as inputs to the amplifiers.
11. **Add more grounds**
Using the Component Browser or copy-and-paste in the schematic window, add seven more ground symbols.

![Schematic diagram showing additional ground symbols](image)

12. **Re-arrange components, re-naming components, changing values**
Everything needed for a simulation is in place. Before wiring components together, it is probably worthwhile to spend a few minutes to re-arrange components so that the circuit will be more tidy. By selecting and dragging, you can move the components anywhere within the schematic window. Also, the orientation of components can be changed by right-clicking and selecting the appropriate menu items – rotate 90° (CW or CCW), flip horizontally, flip vertically. You can change component names or values by double-clicking on the name and then entering a new name in the resulting dialog box.

![Schematic diagram showing re-arranged components](image)

Spend a couple of minutes to re-arrange and re-name the components as shown at right. (Or put them in whatever sort of pattern you prefer.)
13. **Wiring the circuit**

Interconnects can be completed at any time. We have waited until all the components were in place before wiring, but you can wire in each component as it is added. There is a **Wire** item in the **Place** menu, but it’s not necessary to use that. When you move the cursor near the end of a component connector, MultiSim “assumes” that you want to form interconnects and switches into wiring mode automatically. To wire the capacitor to the non-inverting terminal of op amp A, move the mouse to end of the right-hand capacitor lead, click on it, drag the wire to the op amp terminal, and click again to connect it.

Wire the remaining connections in the same manner. Leave the DC voltage source connection off for the time being. Note the changed names for many components and the changed voltages on the DC sources.

Note that you cannot connect a second wire to a component connector. For instance, resistor RA above cannot be connected directly to the non-inverting input of the op amp or the right-hand terminal of the capacitor. The resistor must be connected to the wire running between the op amp and the capacitor. See the figure below. This is an annoying aspect of MultiSim, but usually you can draw the interconnects in a way the works around this “feature”.

---

**Note:** The diagrams show the connections and labels for the circuit.
14. **Power supply connections**

It would seem that all that is remains is to run connections from the power supplies to the op amps and other points in the circuit. However, to keep things tidy in the circuit diagram, we will use virtual connectors. These can be used to connect any two nodes in a circuit, but are especially useful for power supplies.

From the the **Place -> Connectors -> on-page** menu item. The on-page connector window will open. Enter +VS in the connector name field and click OK to close the window. Place the connector symbol near to the positive terminal of the +9 supply and run a connection between the two.

Use the **Place -> Connectors -> on-page** menu item to open the on-page connector window again. This time, the already-created +VS connector name shows up in the window. Select it and click OK. A new virtual connector, already named +VS appears. Place near the positive supply terminal of the op amp, and run a wire from the connector symbol to the op amp supply lead. MultiSim (and later Ultiboard) treats those points as being the same node. The virtual connector works very much like the ground symbol — you can hook it up at many different points in the circuit, and all of those points will be treated as being the same node.
Add another +VS connector and attach it to the anode of the upper LED.

Then repeat the entire sequence to create -VS virtual connectors that join the lower terminal of the negative DC source, the negative supply terminal of the op-amp, and the cathode of the lower LED.

The circuit is complete! If we wanted, we could do SPICE simulations of the circuit. However, we will skip simulations in this tutorial and move directly to the PCB layout process.
PCB layout with Ultiboard

To begin the PCB layout, we need to remove the parts of the circuit diagram that are not actually part of the board. First, we remove the DC power supplies – in the Altoids amp, DC power is supplied by two 9-V batteries, which are not mounted on the board. Secondly, we remove the signal sources – in the real amp, the input signals come from signal source and are passed to the amplifier through a stereo audio jack.

1. In place of the power and signal voltage sources, we will insert two connectors into the circuit – one for DC power and one for the signal inputs. We also add a third connector for the audio outputs. The connecters are simple three-terminal rows of header pins. Fancier connectors are possible, but we will keep it simple for this design.

Header J1 connects to the amp inputs, J2 connects to the outputs, and J3 provides the DC power.
2. Once the circuit is ready, it can be transferred to Ultiboard for layout. Use the menu item: **Transfer to Ultiboard -> transfer to Ultiboard 14.0.** A window showing the Net list that describes the connections in the circuit shows up. Click OK to import the net list.

   The initial view shows a blank board with a default size of 2” x 6” and the components arranged just above it.

3. To facilitate manipulation of the various parts of the circuit, there is a cursor selection menu in the upper left corner of the program window. This menu allows you to enable or disable the various actions that you can do with the mouse.

   The icons represent (from left to right),
   - enable / disable selecting **parts**
   - enable / disable selecting **traces**
   - enable / disable selecting **copper areas**
   - enable / disable selecting **vias**
   - enable / disable selecting **THT** (through hole) pads
   - enable / disable selecting **SMD** (surface mount) pads
   - enable / disable selecting **attributes**
   - enable / disable selecting other objects.
4. **Change the board size**

Use the cursor selection menu to disable all actions except “selecting other objects”. Then use the mouse to double-click on the yellow board outline. In the dialog box that appears, change the board size by first changing the units to “inches” (from “mils”) and set the size to 2” x 2”. The board outline will be reduced accordingly. Close the dialog box.

5. **Move in the parts**

Change the cursor selection menu to disable everything except “selecting parts”. Start moving the parts lined at the top inside the board outline.

The yellow lines represents the connections between the terminals of the various components. The network is known as a “rat’s nest”, for obvious reasons. At first it is quite a mess. The goal of the layout will be to arrange the parts and draw traces (circuit board connections) between the terminals in place of the rat’s nest “wires”.
6. **Ground planes**
Generally, it is good practice to use ground planes – continuous metal planes over both surfaces – to form the ground node of the circuit.

Along the left side of the PCB window is a list of the various layers in the PCB. For now, the two most important are Copper Top and Copper Bottom.

To make a ground plane on the top copper layer, select “Copper Top” in the list – double-click to highlight it in the list. Then any actions we perform will affect only the top layer.

Under the **Place** menu, choose the **Place power plane** item. In the window that opens, select “Copper Top” and choose net “0”. (0 corresponds to the ground connection.) Click OK to close the window.

![Image of PCB window showing ground plane creation steps]

The entire surface of the board will turn bright green, meaning that it is completely covered in copper, with openings left for the through holes for the component pins.

All of the bright green makes it hard to see what is going on. The “copper” can be toggled on and off using the **View → copper area** menu item.

Once the green is turned off, you should note that many of the rat’s lines are gone, because the ground plane provides all of the ground connections – separate ground traces are not needed.

After completing the top layer, repeat the procedure to create a ground plane on the bottom layer. (The bottom layer is red, to distinguish it from the top.)
7. **Arranging the parts on the board**

Now comes the real work. You should spend some time to arrange the parts in an efficient manner. The goal is to keep the traces short and minimize points where wires might cross. When necessary, a connection can cross from the top copper to the lower copper using a “via”. (It turns out that we will not need any vias for this layout. But in general vias will be necessary part of a layout.)

There is no “right answer” for the arrangement of the parts and the routing of the traces. In a fairly short time, you can probably arrange the parts to get rid of many of the rat’s nest crossing and minimize trace lengths. But there is a law of diminishing returns, and lots of extra effort probably does not give much improvement. At some point, you have to say “Enough is enough”. Arranging the parts to form efficient patterns is as much art as it is science, and you will certainly get better at it with practice. Below is a reasonably efficient arrangement that took about 15 minutes of work.
8. Once you are satisfied with the component arrangement, you can route the traces to connect the various pins. Again, there is no single correct answer for routing the traces. You can route on either the top or the bottom, and can go back and forth using vias as vertical (through the board) interconnects.

To run a trace, choose **Line** from the **Place** menu. Choose the layer you want to use – Top Copper or Bottom Copper – by double-clicking on the item in the layer menu. Then draw the trace by clicking the mouse at the starting point and dragging to the other end. As you draw, the program will try to assist you by putting in 45° angles. (In general, because of current-crowding effects, you do not want 90° angles in the layout.) With a bit of practice, you can learn how to manipulate the mouse to arrange the angles where you want. You can also “tack down corners” by clicking the mouse at points where you want the trace to make a turn.

Below is one possible routing of traces for the component arrangement shown above. The power supply traces (in red) were routed using the bottom copper. The signal traces were all routed using the top copper. Again, this division between top and bottom layers is somewhat arbitrary.

The rat’s nest line will disappear when a trace has completed successfully. When all the rat’s next lines are gone, you are finished with routing.
9. **Changing trace widths**

The default line width is 5 mils (0.005 inches). This is quite small, particularly for the relatively simple circuits that will be built in EE 333x. You can draw all of the lines with the 5-mil width. However, they should be made wider before the board designs are sent off for manufacturing. There are online calculators that will determine the correct trace width given the expected current of the trace.

You can change the width of the traces at once at the end of the routing process. (It can also be done before routing by changing the default for all of the nets that define the circuit, but that is a bit tedious.) To change the trace width after they have been drawn, change the cursor selection menu to disable everything except “selecting traces”. Then click and drag the mouse across all the entire layout to select all of the traces. Then double-click on any one of the traces to bring up the properties window. In that window, you can change the trace width. In the little menu, change the selection from “Net List” to 5. Then change the 5 mils to something bigger. For larger, through-hole layouts, 20 or 24 mils should be fine. Any traces that carry larger current, like the power supply traces, can be bigger. In the design shown above, the green signal traces are 24 mils and the red power supply traces are 36 mils. In the same dialog window, you can change the trace clearance, which is the width of empty track (“ditch”) that surrounds each trace. For above design, the clearance was set at 20 mils.

When you are finished, you can view the top and bottom copper layers to see what top and bottom copper patterns will look like. (Toggle the **Copper area** item under the **View** menu.)

**Top copper**