

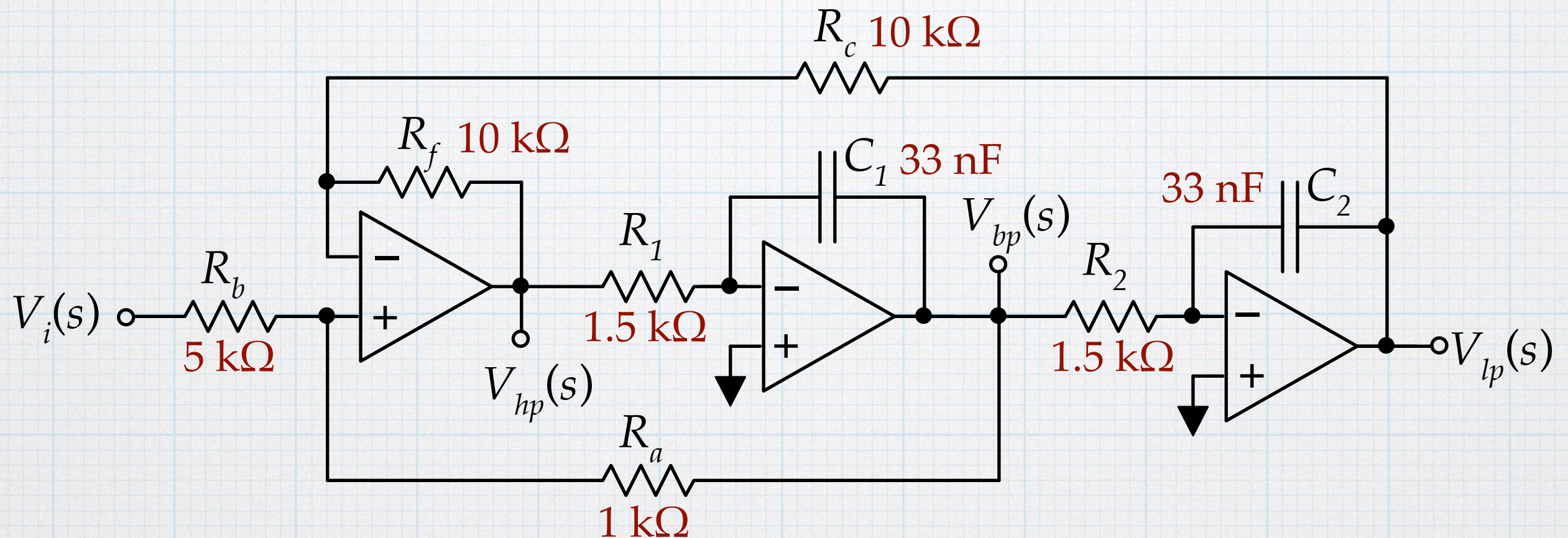
PCB layout tutorial – MultiSim/Ultiboard

The basic steps in designing a PCB

- Paper design and prototype of the basic circuit.
- Identify the parts — and the footprints — that will be used.
- Make a circuit schematic, with the correct parts identified.
- Transfer the schematic to the board layout view.
- Arrange the parts on the board.
- Make the interconnects on the board (autoroute, manual, or combo).
- Tweak the details.
- Design-rule check.
- Generate Gerber files.

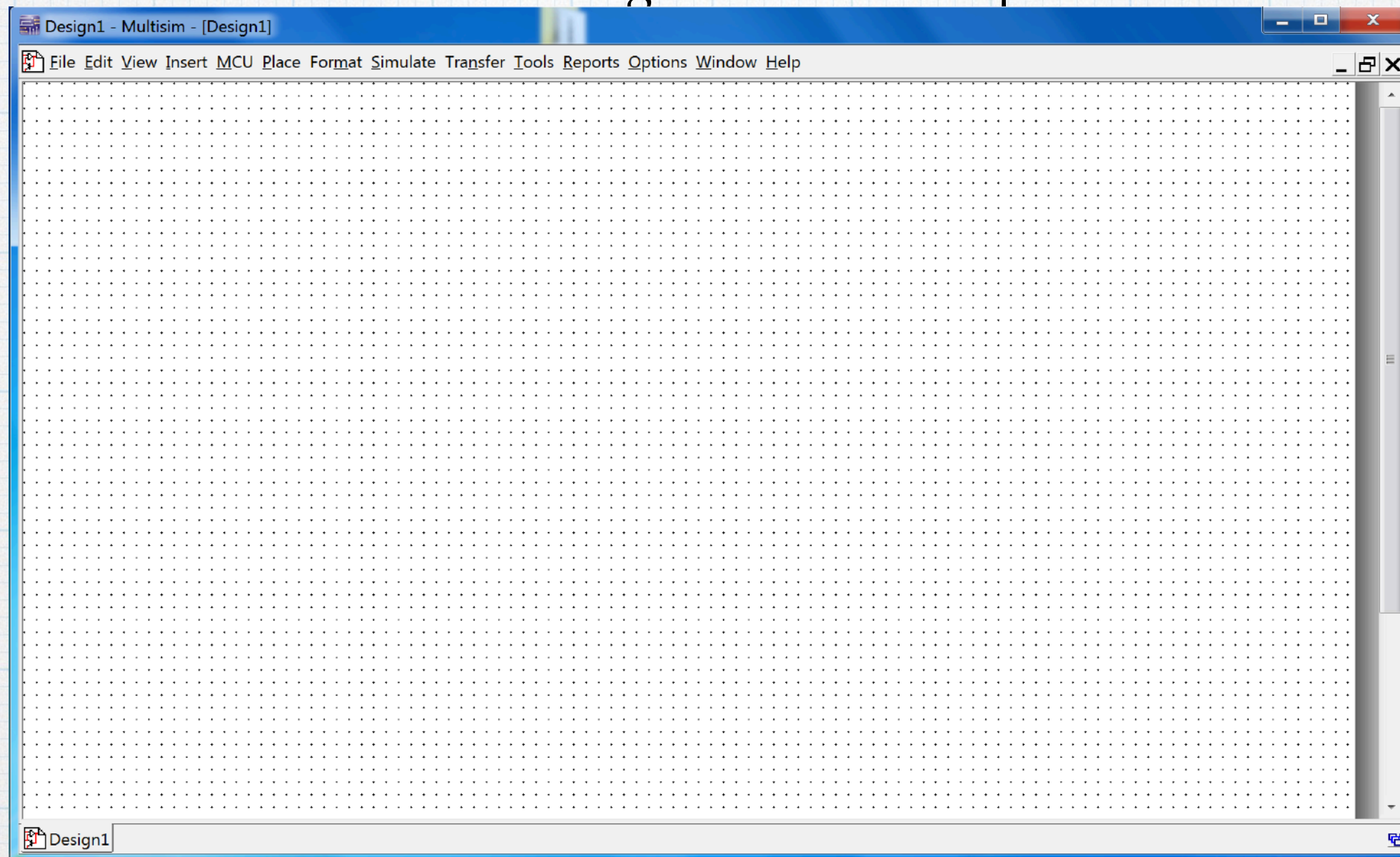
An example circuit

As a means for learning the basics of PCB layout, we will build a simple second-order filter circuit, shown below. This is a two-integrator loop type filter that was studied in EE 230 (maybe).



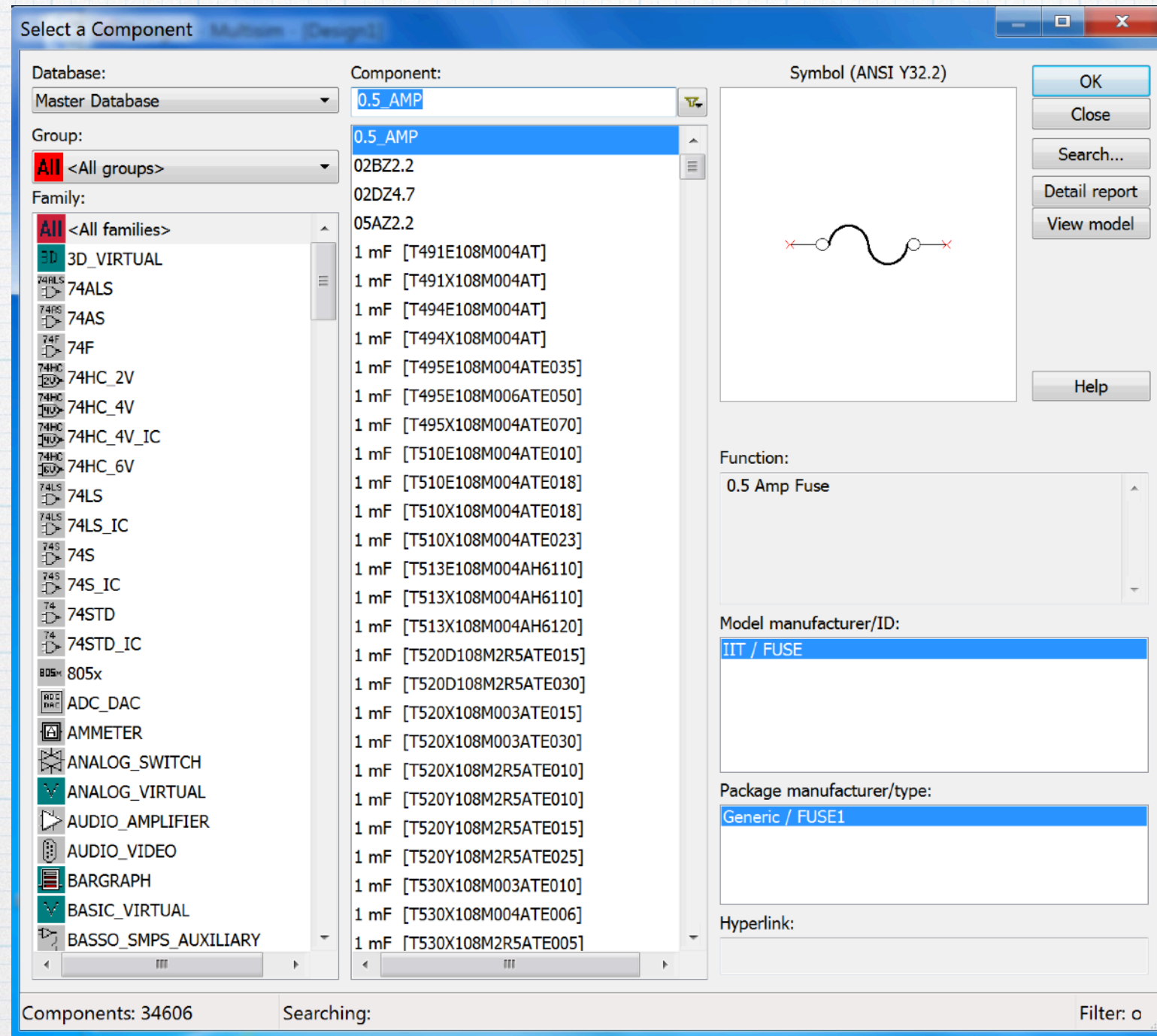
For the values listed, the expected center frequency will be $f_o = 3.2\text{ kHz}$, the quality factor will be $Q_p = 3$, and the gain at the center of the band will be $G_o = 5$.

On the Windows machine, start up MultiSim (All Programs»NI Multisim 14.0). A blank schematic drawing window will open.



There are a number of view options that can be invoked — ruler bar, status bar, design toolbox, etc. All are accessible through the view menu. In the slides that follow, all toolbars are turned off, which in my personal preference. Occasionally, I will be active specific items when I will be doing one sets of tasks repeatedly. Individual users should experiment with different view to see works best for them.

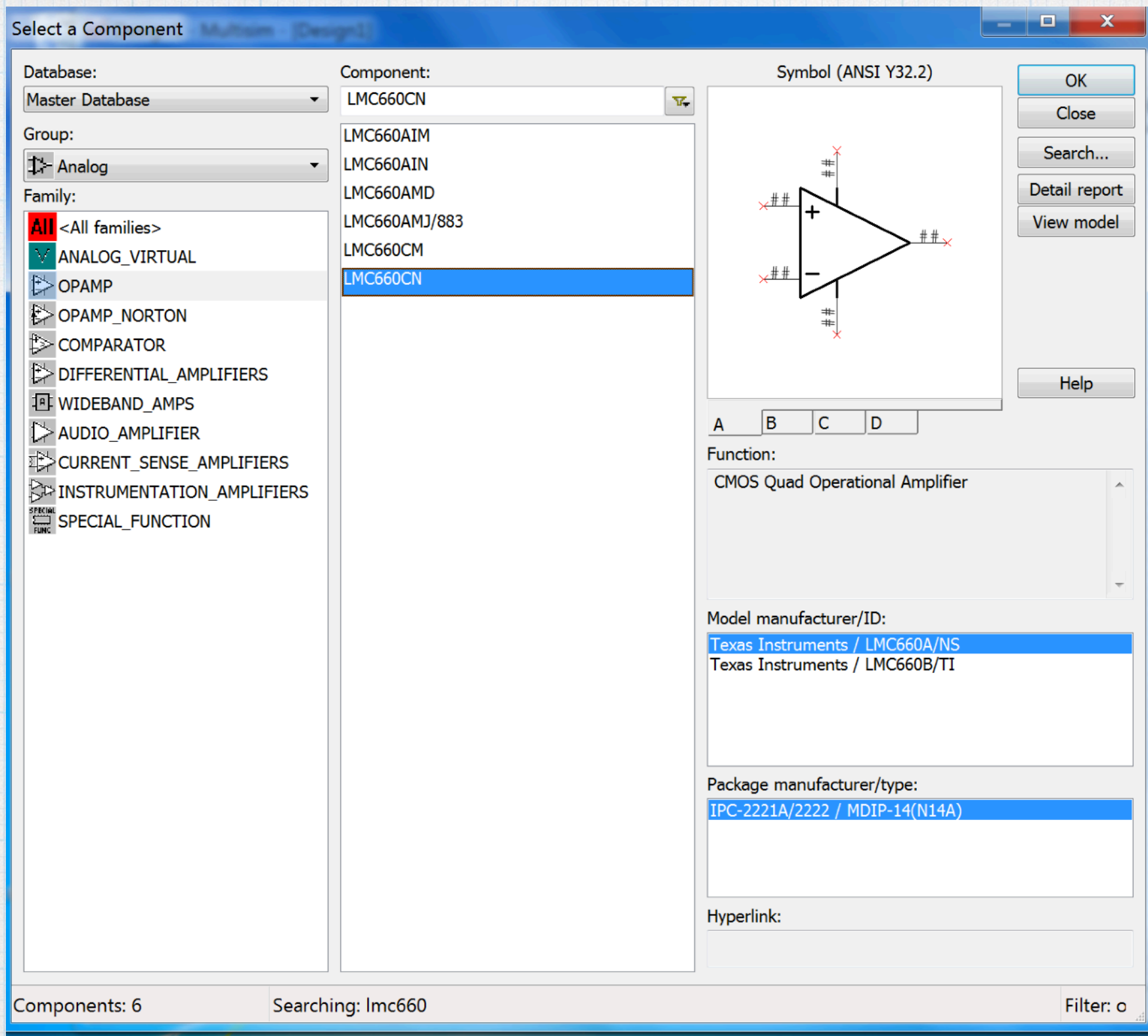
Selecting components. From the menu, select **Place»Component**. The **Select a Component** – or Component Browser (CB) – window appears.



There are thousands of components available — usually, we can find what we need. In addition, there may be different form factors (footprints) available for each item – pay attention!

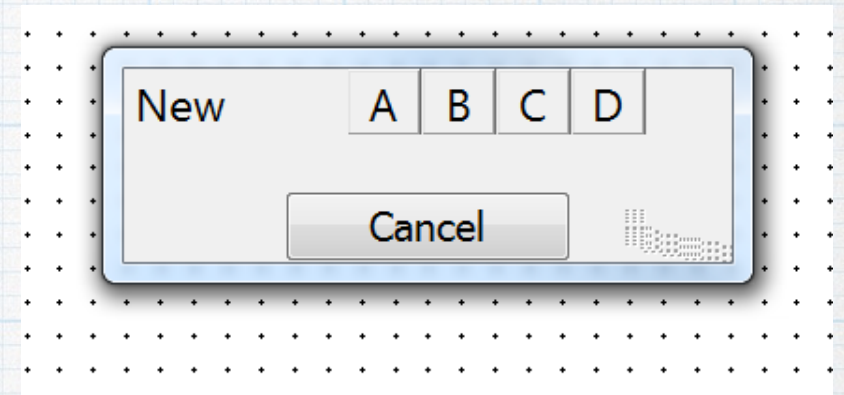
The components are collected in various databases. (We will use the Master database for all of our work here.) Within a database, components are arranged in Groups. Within each group are Families. From within a particular family, you can choose a specific component to place in the schematic.

Choose the op amp. The first part we will place is the “good” quad op amp chip from the EE 230 kit — the LMC660. In the CB window, choose the **Analog** group. From the options, choose the **OPAMP** family. There are *many* op amps available. We can scroll through until we find the specific one or we can just use the search box at the top of the component column. In searching for the LMC660 directly, we are still presented with several options. Some of the options are related specifically to package type. For SPICE simulations only, it probably isn't important which LMC660 option you use. However, in working towards a PCB layout, we *must* pay attention to the package type (footprint). Since we are planning in a through-hole PCB design, we will want a DIP (dual in-line) package. Choose the LMC660CN option, which comes in the standard 14-pin package that you recall from EE 230.

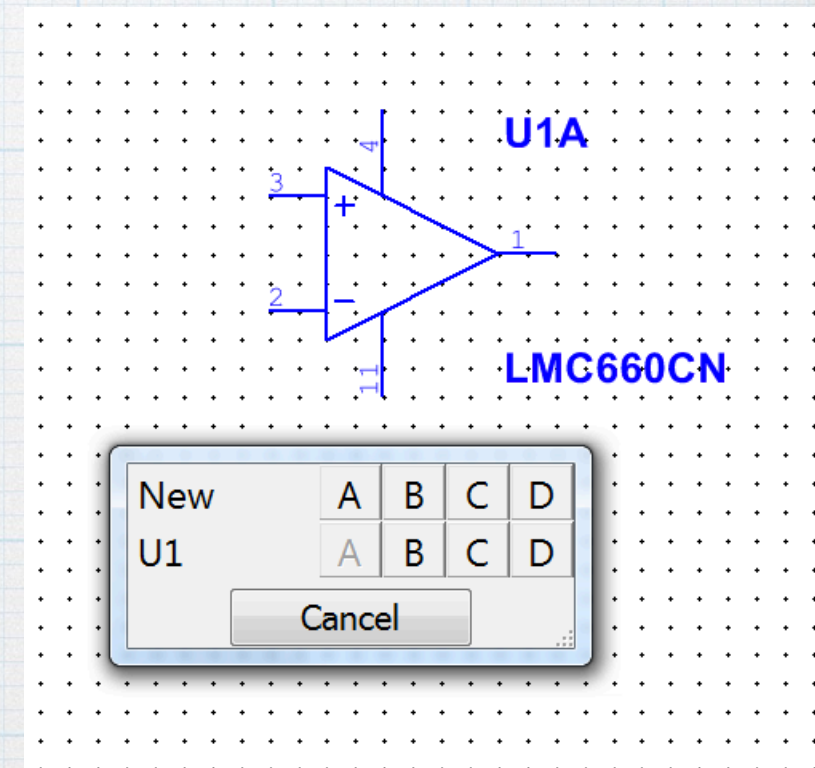


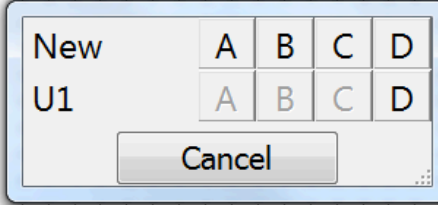
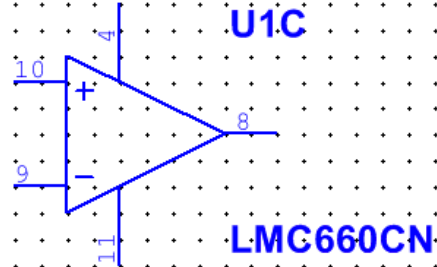
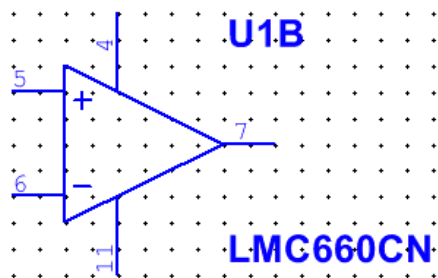
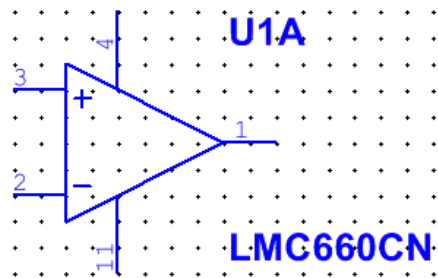
Note that there will 4 op amps in this one package — labeled A thru D. Of course, we need 3 of those for our circuit. Click OK to go back to the schematic window.

We are returned to the drawing window. There will be a small dialog box that allows us to select the specific op amps, A, B, C, or D. 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.)



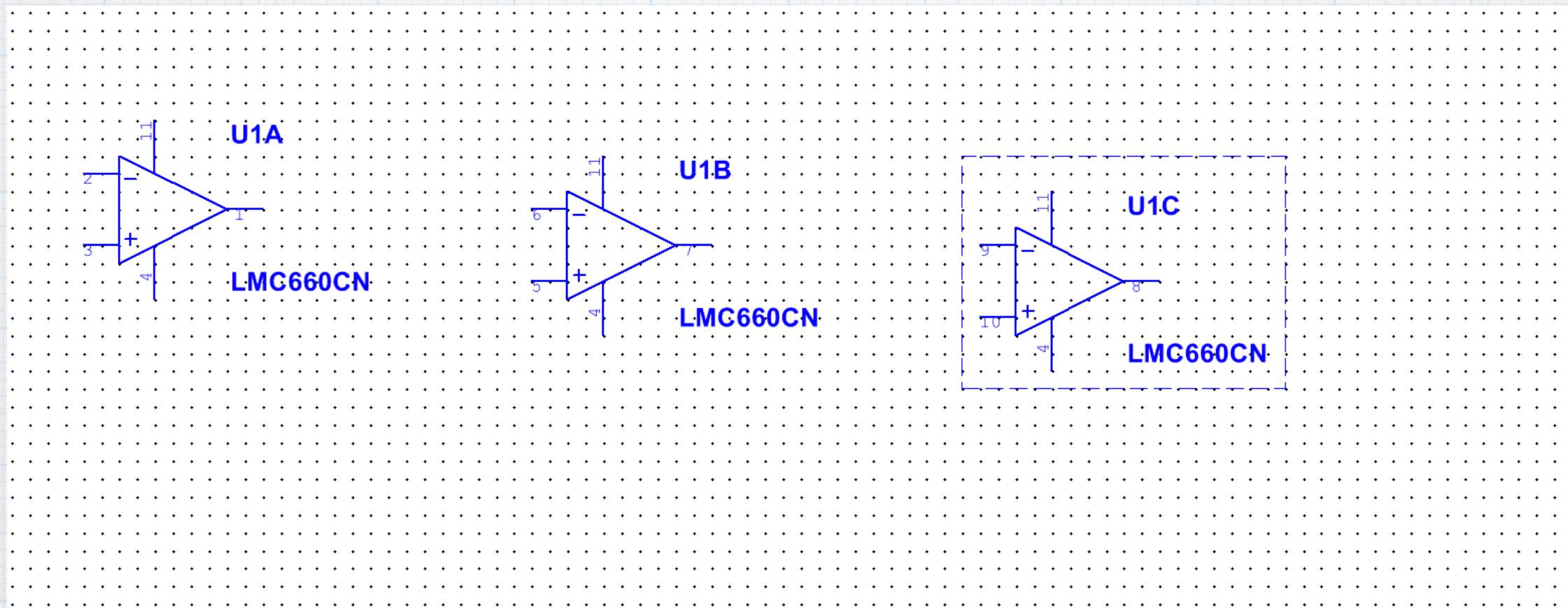
The little “A, B, C, D” dialog re-appears. There is a new line in the window, labeled “U1”, which is the default name of the op amp chip. Because we have already used the “A” op amp, you can no longer select it. To use the second op amp, click on “B” in the U1 line, and repeat the process for placing that op amp in the schematic. (Don’t choose any of the options in the “New “line – that would create a second LMC660 chip.) After placing the “B” op amp, repeat the process again to place the “C” op amp for U1.



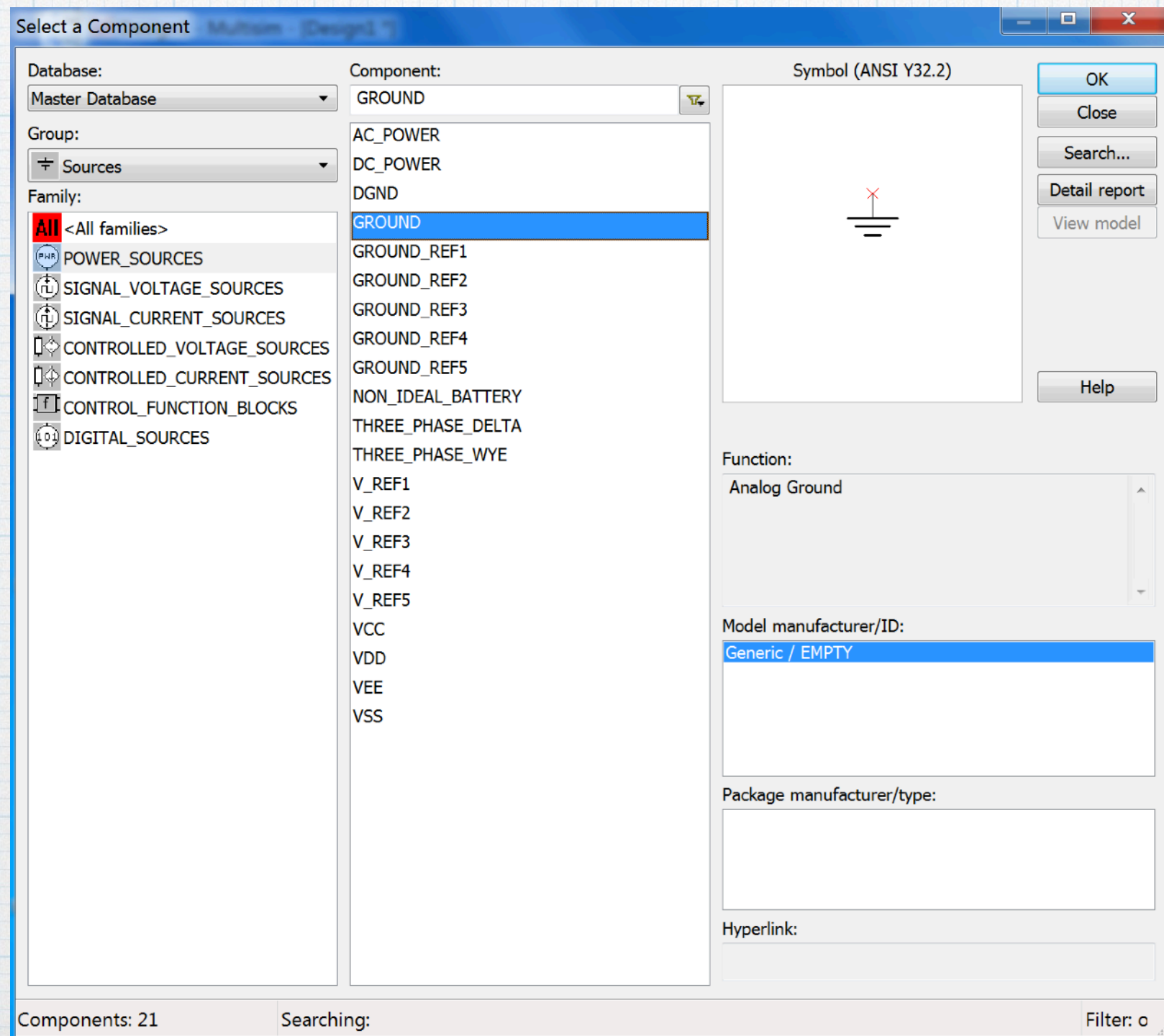


We are done with op amps, so click cancel to dispense with the small dialog box.

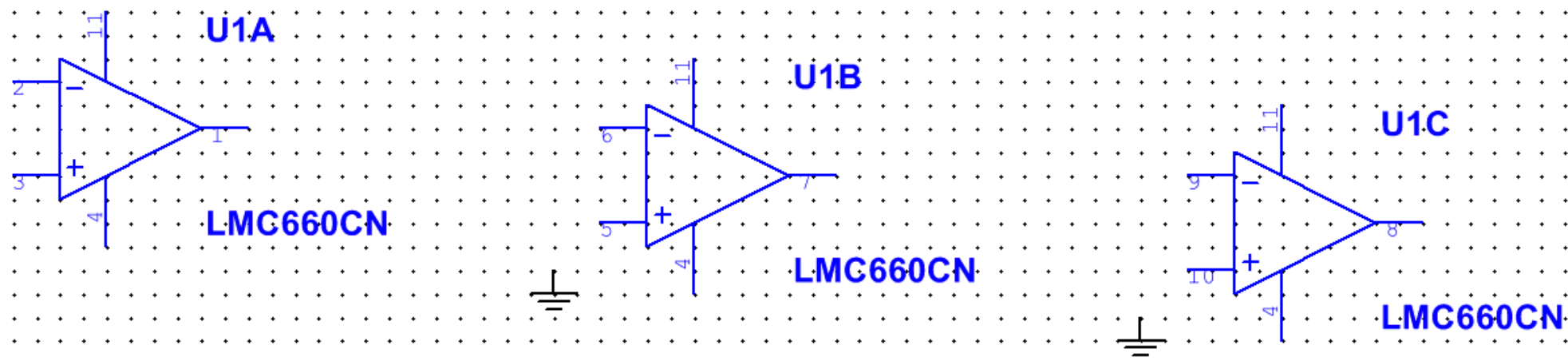
Components can always be moved and rotated, if necessary. Although it is certainly not a requirement, we will try to make our Multi-Sim schematic look similar to the original circuit. To that end, we should flip all of the op-amps so that the non-inverting terminal is down. For each amp, first select it by clicking on it (a surrounding box will appear), and then choose “Flip vertically” from the edit menu. Note that the power supply leads are flipped as well.



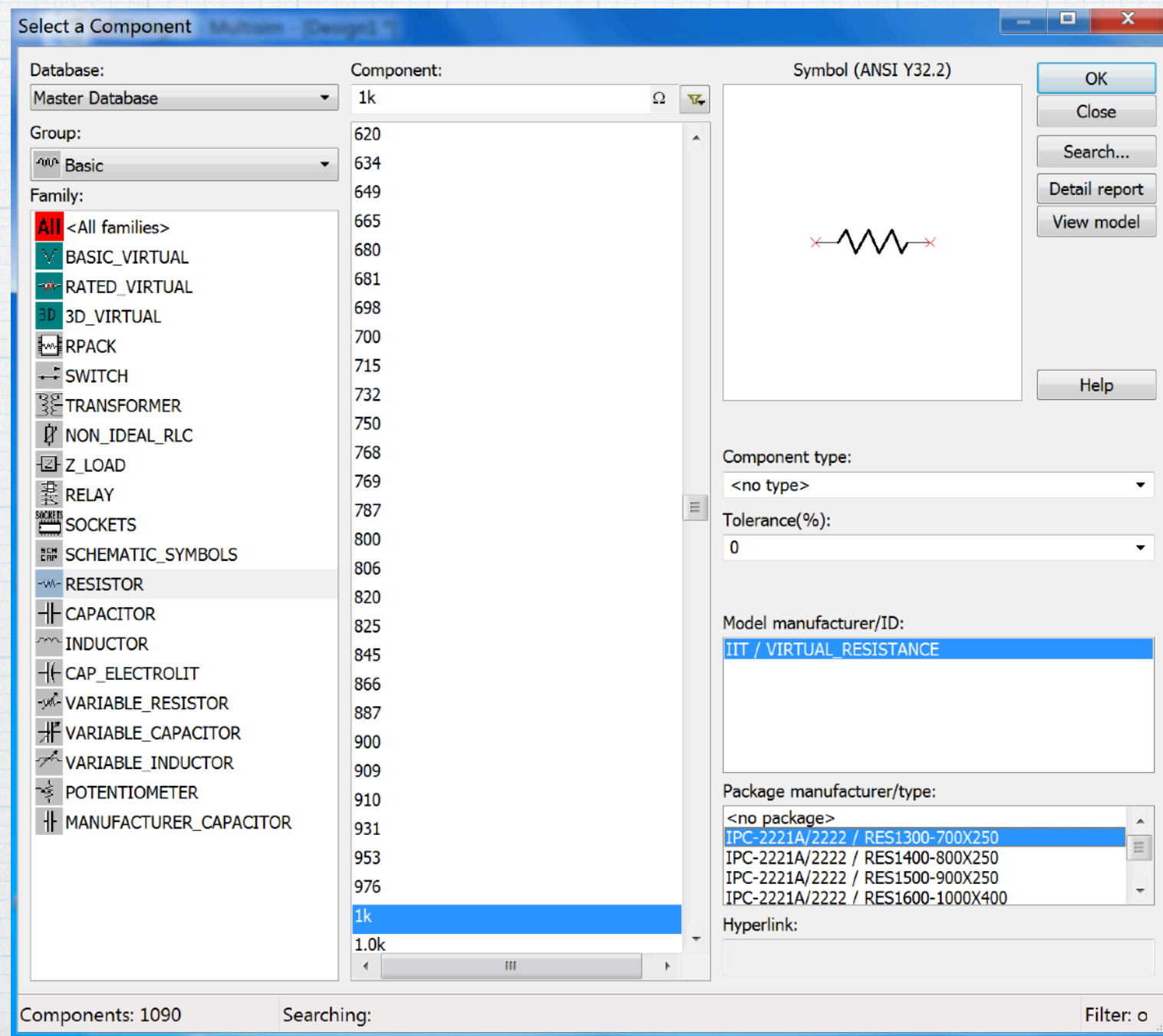
Ground connections. This is as good a time as any to place a ground into the circuit. Use the **Place->component** menu item bring up the CB. The ground symbol is in the **Sources** group, part of the **POWER_SOURCES** family. There are a number of options — just use the one called “ground”.



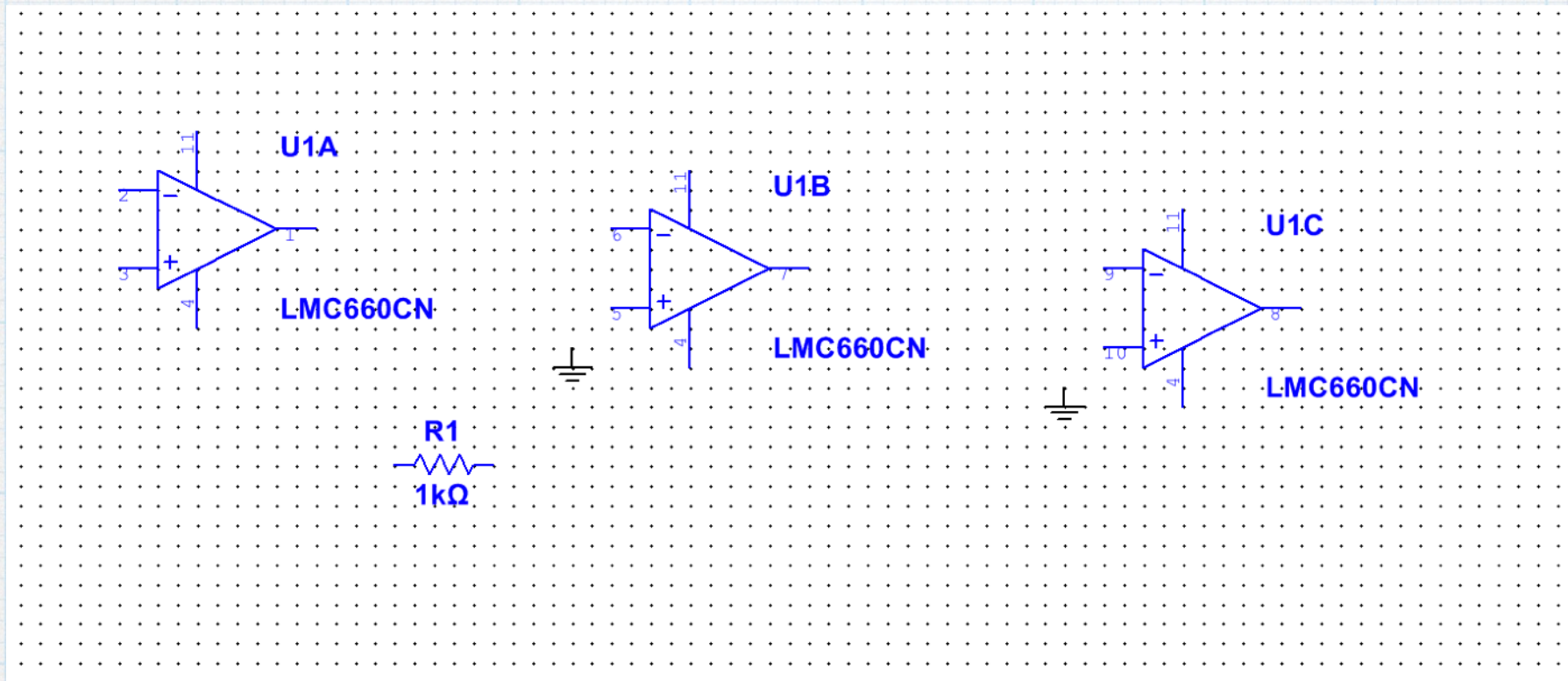
Place the ground somewhere near the inverting terminal of the middle op amp. Then go back to the CB to get another ground — or cut-and-paste, which is probably faster — to locate a second ground near the inverting terminal of the right-hand op amp. More ground symbols will be needed later, as well.



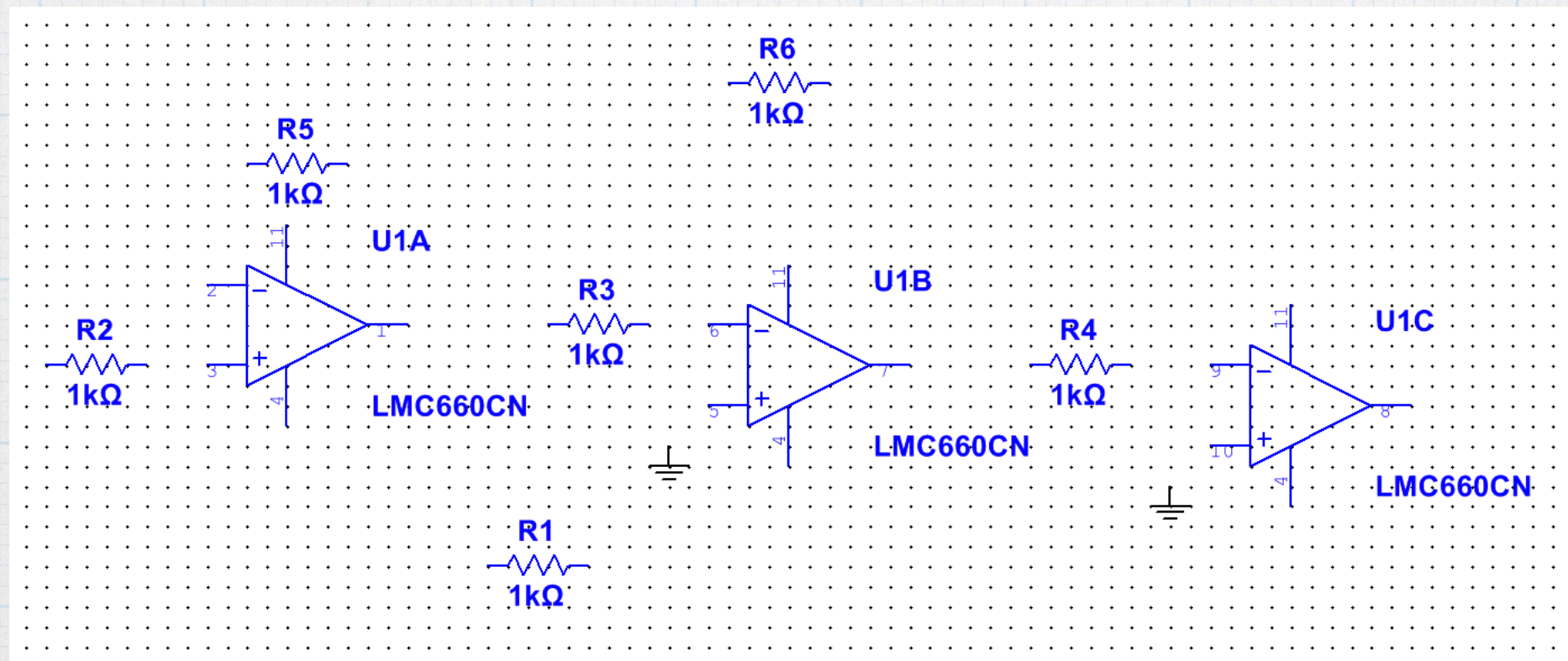
Resistors. Back in the component browser, select a 1-k Ω resistor. Resistors are in the **Basic** group and have their own family. You choose the specific resistor value from a ridiculously long list. (The value can be changed later, if needed.) Also, be sure to pick a footprint for the resistor. For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



Drop the resistor into the schematic somewhere below the op amps.



Then, use the component browser — or cut-and-paste — to place 5 more resistors in roughly the locations shown below.

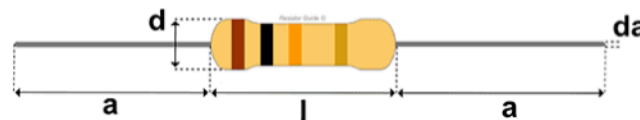


Through-hole resistor sizes

The clip below is from the web site "<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 datasheet of the component.

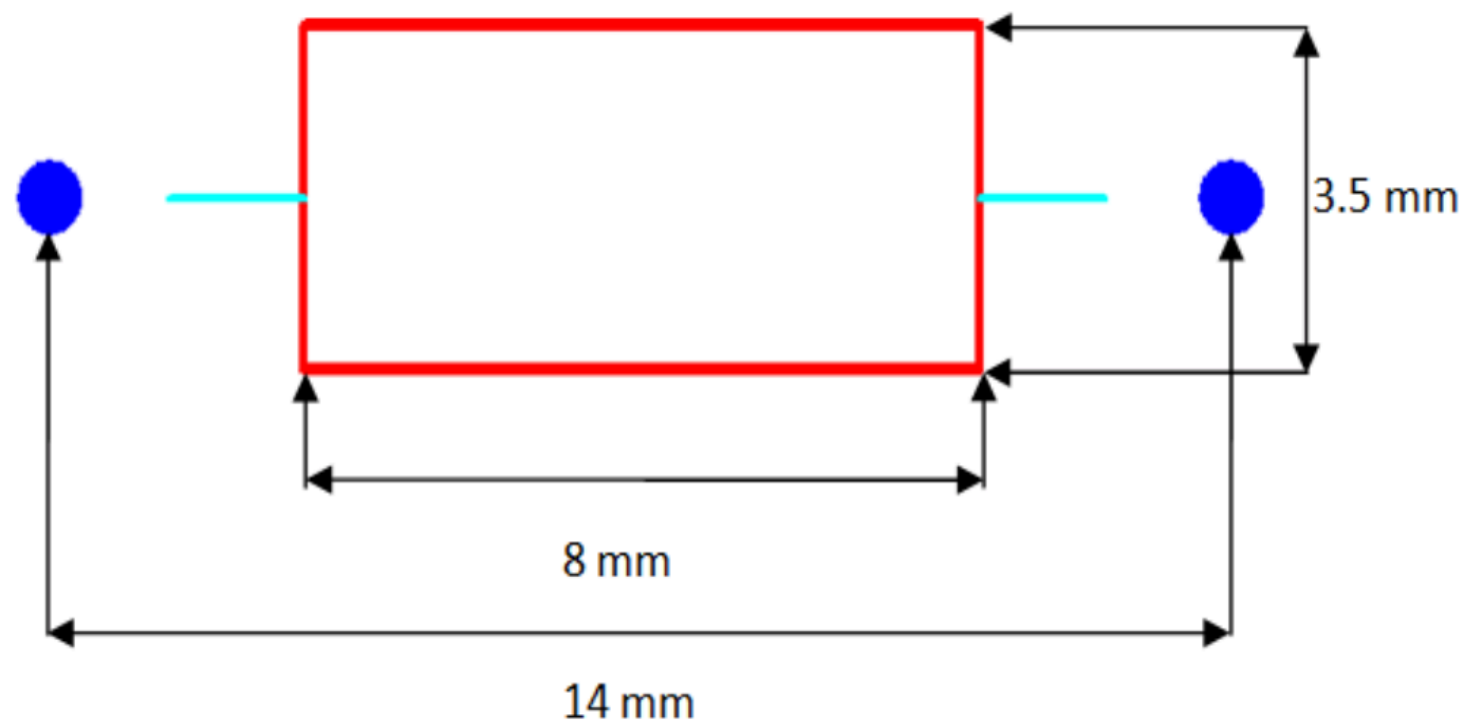


Power rating	Body length (l)	Body diameter (d)	Lead length (a)	Lead diameter (da)
Watt	mm	mm	mm	mm
1/8 (0.125)	3.0 ± 0.3	1.8 ± 0.3	28 ± 3	0.45 ± 0.05
1/4 (0.25)	6.5 ± 0.5	2.5 ± 0.3	28 ± 3	0.6 ± 0.05
1/2 (0.5)	8.5 ± 0.5	3.2 ± 0.3	28 ± 3	0.6 ± 0.05
1	11 ± 1	5 ± 0.5	28 ± 3	0.8 ± 0.05

So 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).

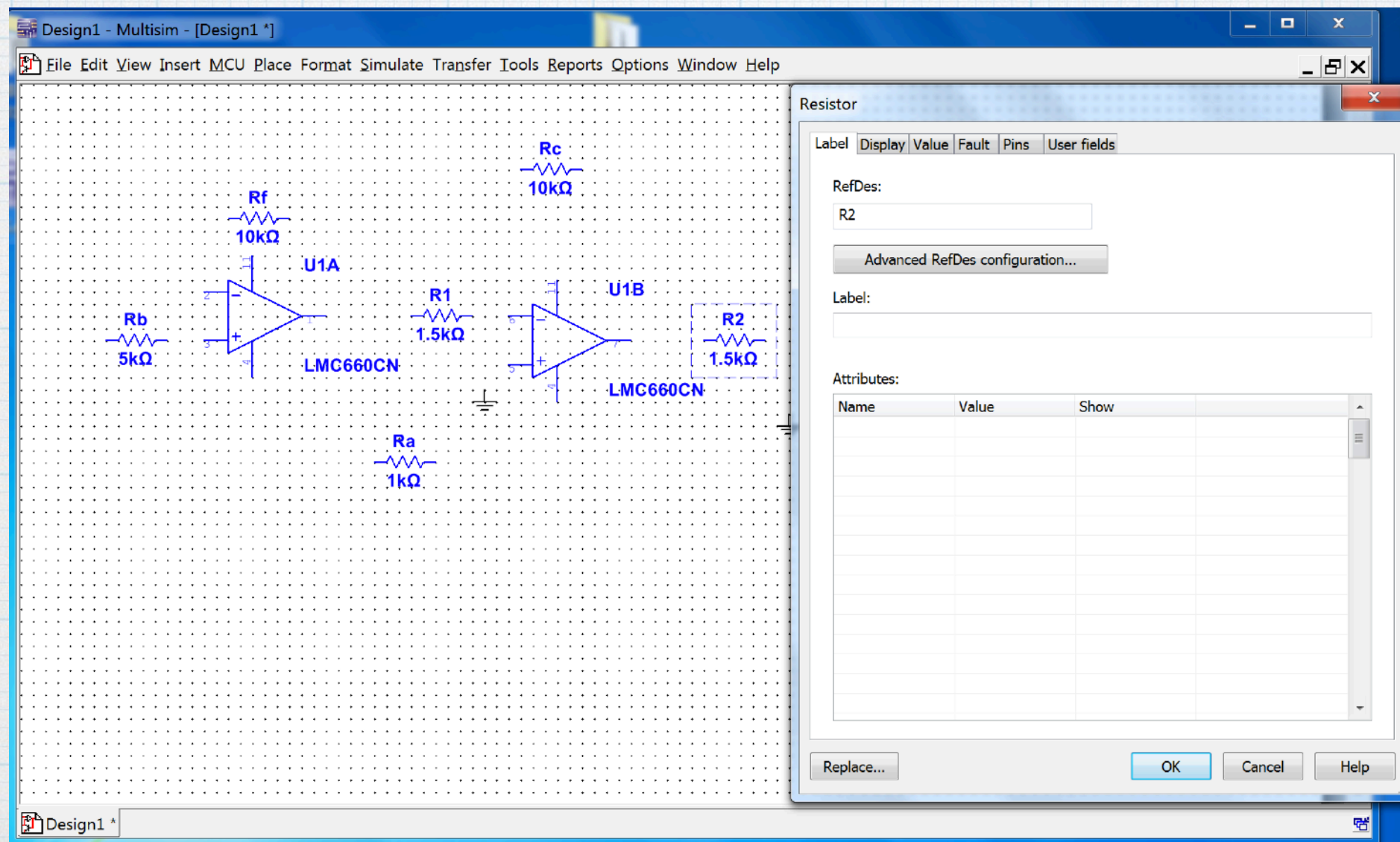
The clip below is from the web site "<http://digital.ni.com/public.nsf/allkb/E81C051F276C5BE0862579E2006975D5>" and explains the naming convention used in MultiSim for passive, through-hole components.

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:

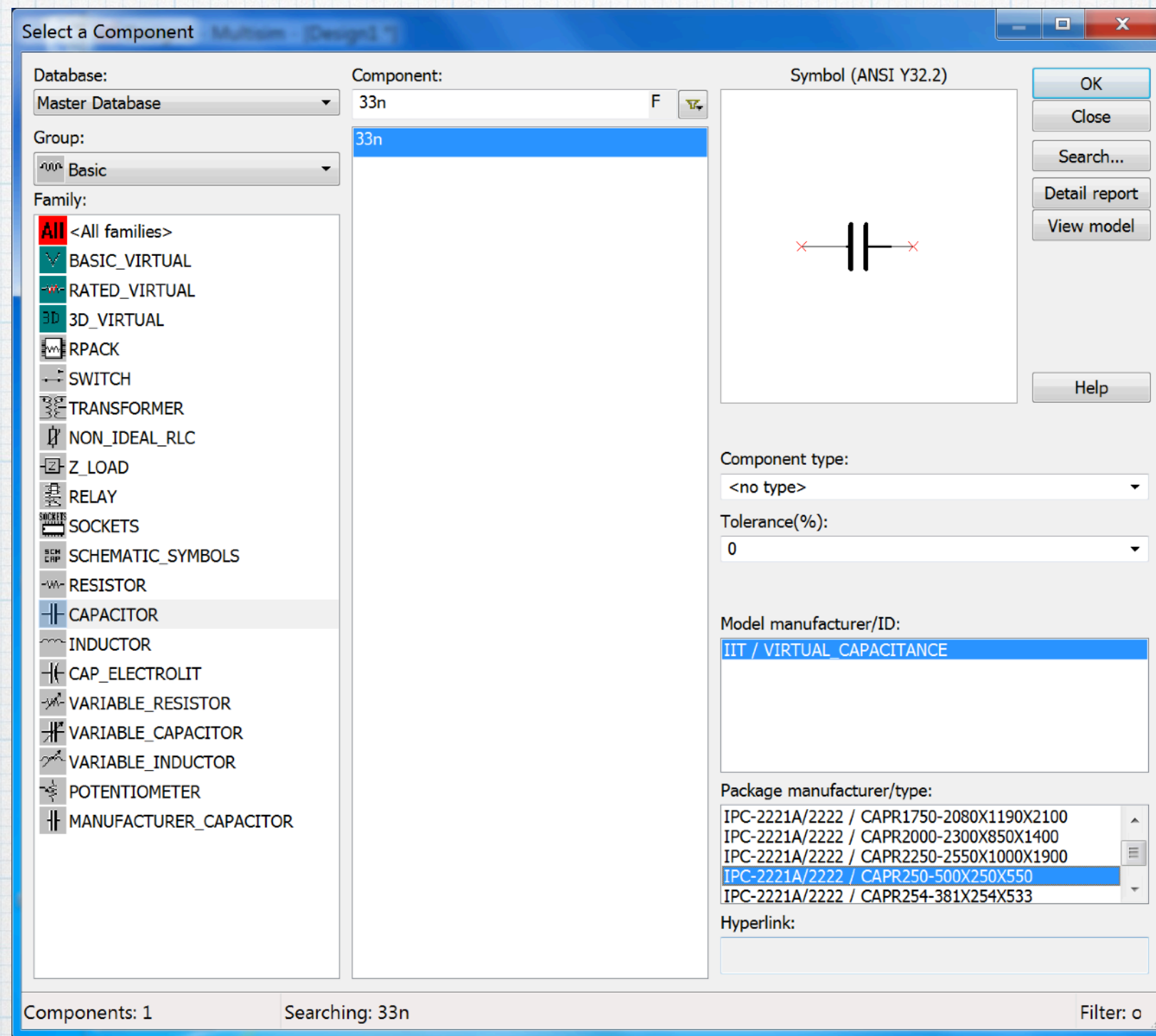


Thus we see that the resistor footprint denoted RES1300-700x250 corresponds to a resistor 7mm long by 2.5 mm wide, which is about right. The spacing between the through-holes where the leads will be soldered is 13 mm, which corresponds to about 5 “holes” in a standard perfboard or solder-less breadboard that has 0.1-inch hole spacing. (5 “holes” = 0.5 inches \approx 12.5 mm.) This also sounds about right.

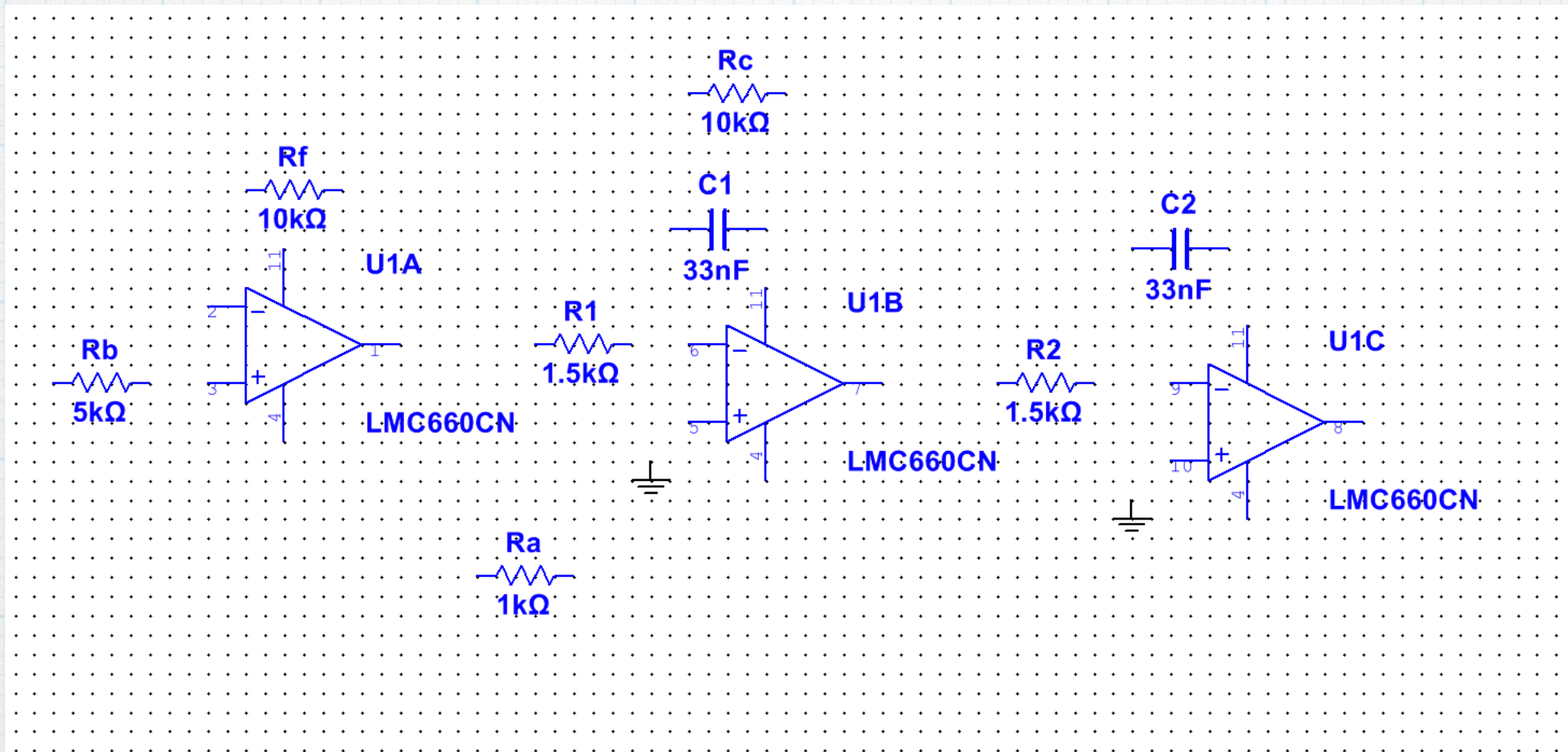
For making a PCB, the resistor values really don't matter — only the physical size is important. In principle, we could leave all the values at 1 k Ω , and we could also leave the names as R1 thru R6. However, we will go ahead and change the names to match the original circuit and list the correct values. Component parameters can be edited by double clicking on the part and changing the values and names in the resulting dialog. Note: Double-click on the part symbol, and then you can change both name and value in the dialog. You can also double-click on just the name or the value, and then you can change just that particular thing.



Capacitors. Now, add two 33-nF capacitors. 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.



Place the capacitor above the middle op amp. Then place a second, identical, capacitor above the right op amp.



Through-hole capacitor 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.

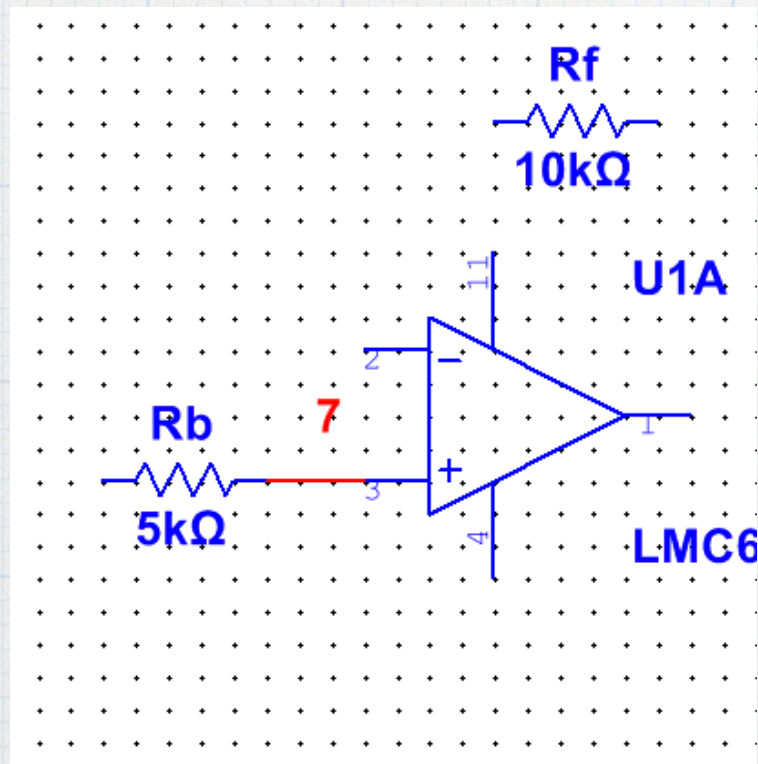
Finally, you can get some indication of the shape of the footprint by clicking the “Detail report” button in the upper right portion of the CB window. It will give you some information about the footprint and a drawing of the basic shape, although it is often difficult to discern the details of the figures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

Decoding the resistor size, capr250-500x250x550:

- cap - obvious
- r - radial (leads both come out the “bottom” — both in the same direction)
- 250 - lead spacing of 2.50 mm (\approx 0.1 inch, standard for through-hole)
- 500 - 5.00 mm body length
- 250 - 2.5 mm body width
- 550 - 5.5 mm body height.

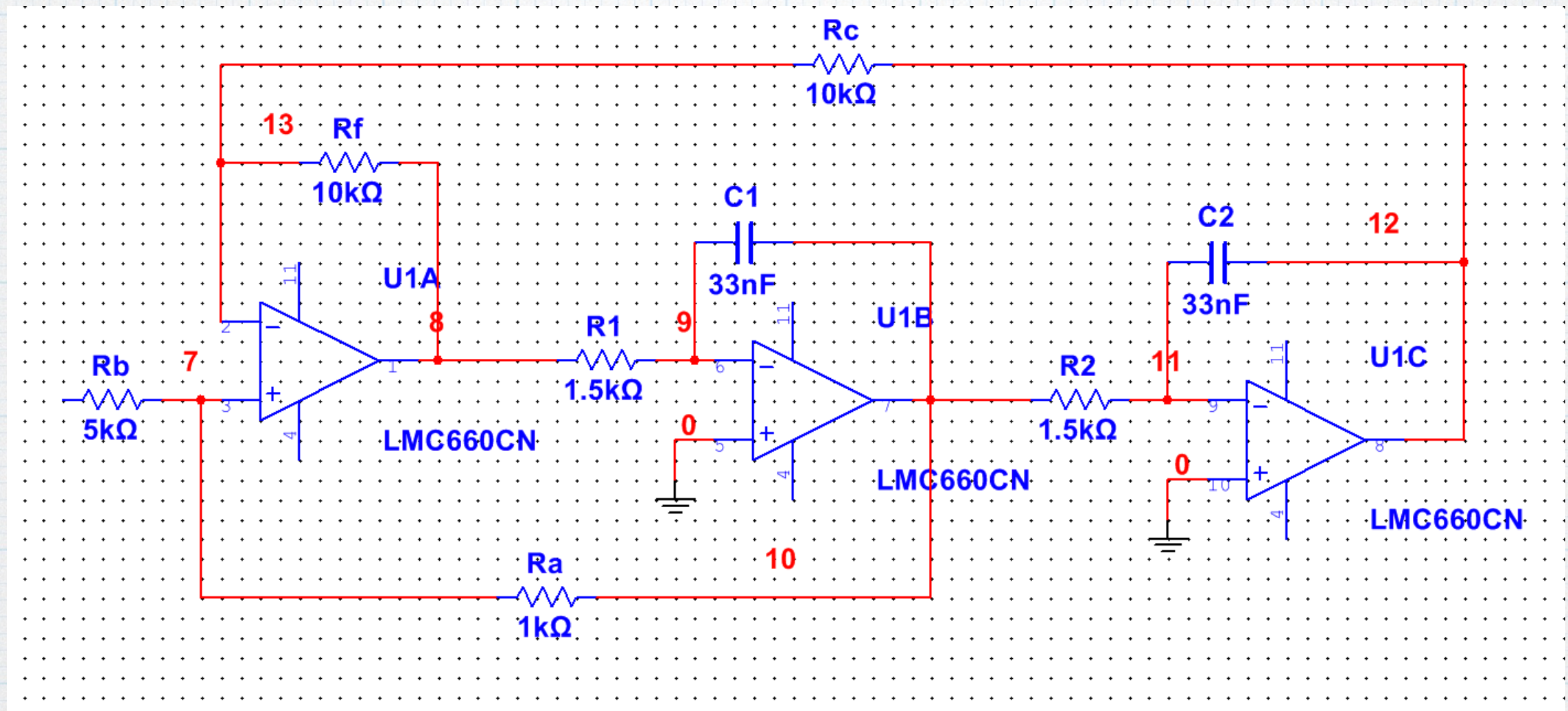
The lead spacing is the most important.

Wiring connections. Interconnects can be completed at any time. We have waited until all the components were in place before wiring, but we could have been wiring in each component as it was 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 R_b to the non-inverting terminal of op amp A, move the mouse to end of the lead on the right side of the resistor, click on it, drag the wire to the op amp terminal, and click again to connect it.



A node number may show up.
We can ignore these for now.

Wire the remaining connections in the same manner. Do not connect anything to the op amp power supplies yet.



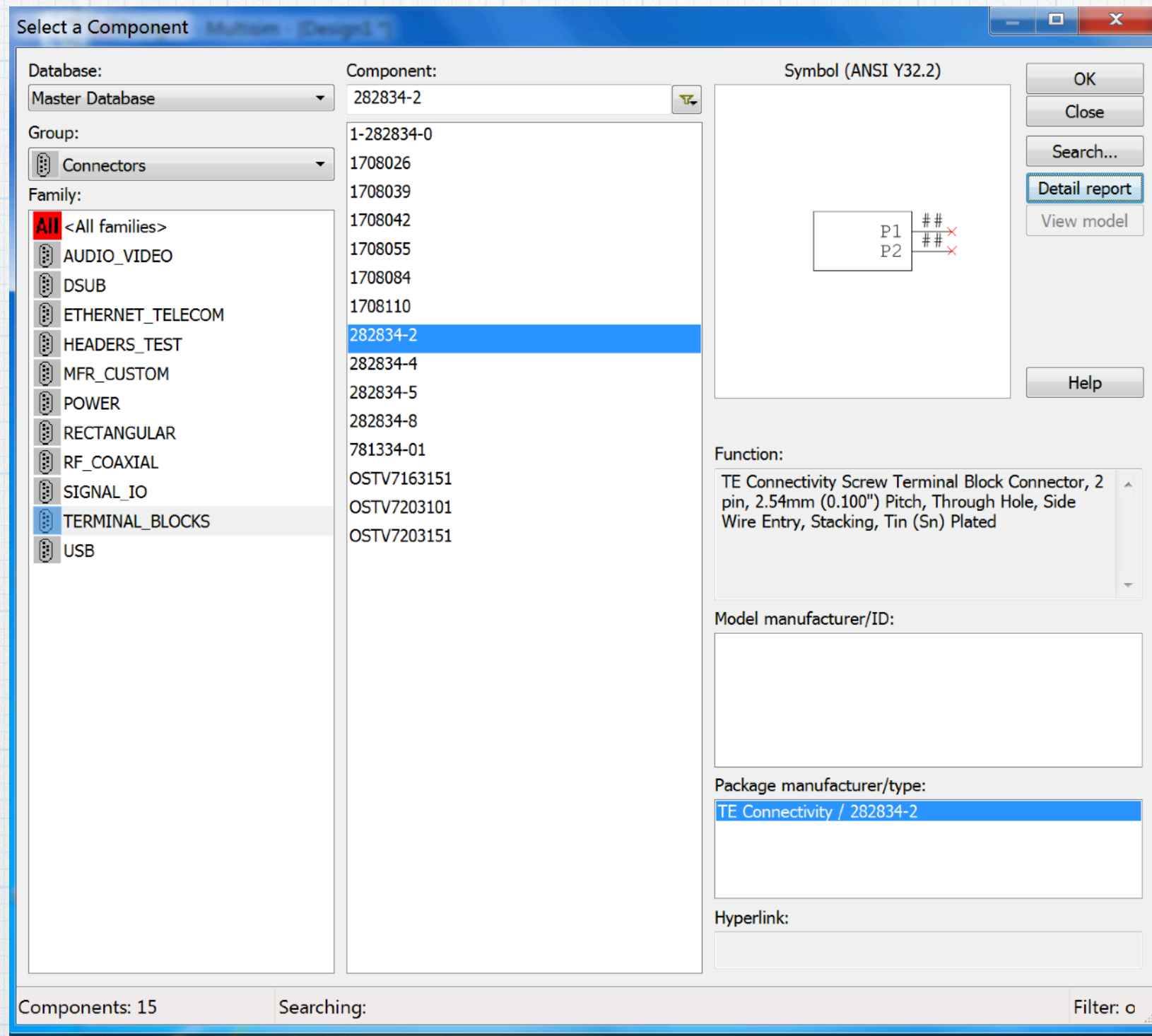
(For some reason, node numbers are included. These are used in a SPICE simulation, but are not needed for PCB layout. These numbers will not transfer to the PCB.)

External connections. The power for the op amps and the signal inputs and outputs must be connected to external supplies and meters. We could just leave some long wires dangling, but that is not very tidy and definitely not reliable. Instead, we will provide connectors for external components.

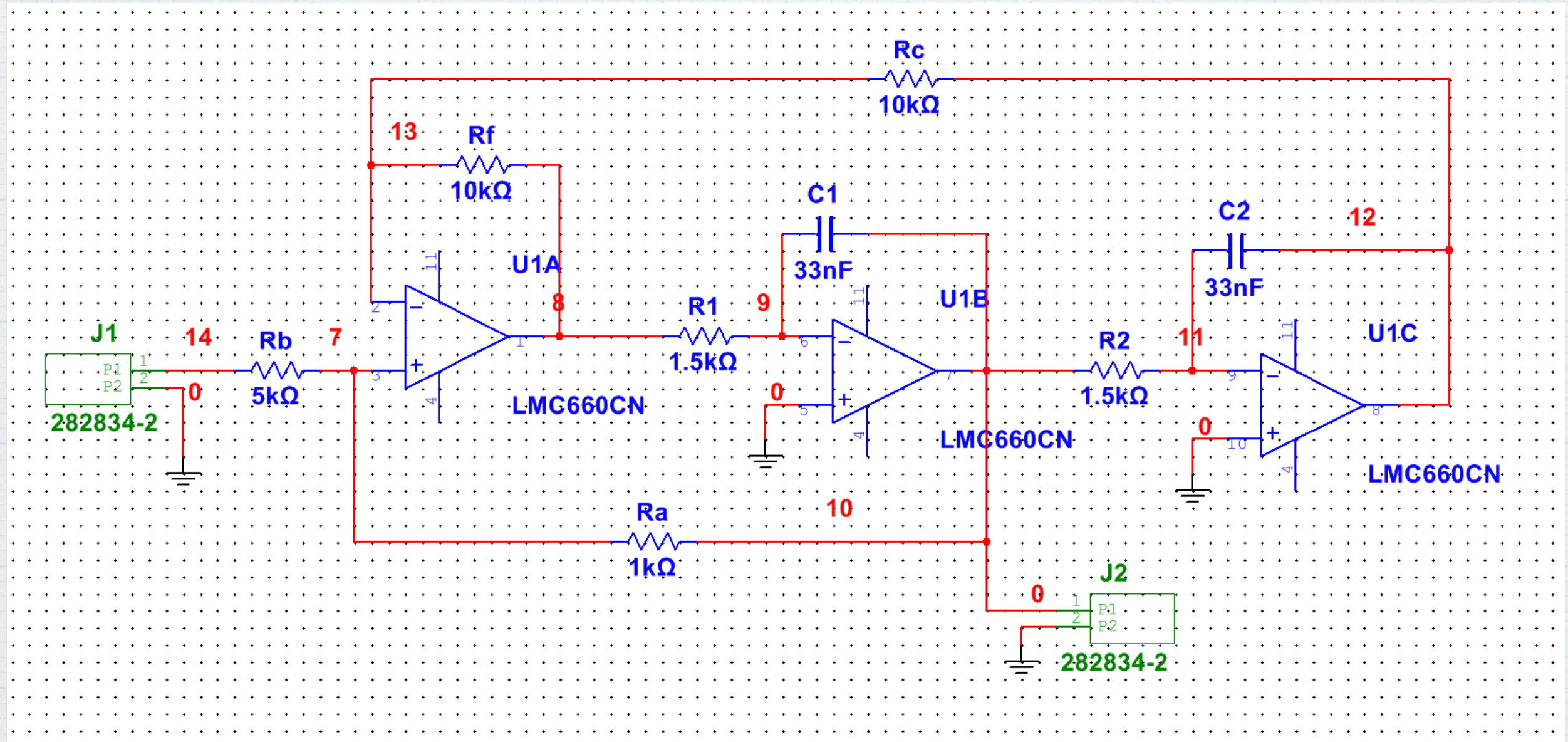
For the DC power, we need three connections positive DC, negative DC, and ground. For the input signal, we need two connections, one of the source voltage and one for ground. Similarly, we need two connections for the signal output. We could conceivably use a three-wire header that combines the input and output into one connector, but it is probably and simply and cleaner to use two separate connectors for input and output.

For the signal inputs and outputs, we will use a “terminal block” connection. Open the component browser (Place Part), select the “Connectors” group and the “TERMINAL_BLOCKS” family.

For the signal inputs and outputs, we will use a “terminal block” connection. Open the component browser (Place Part), select the “Connectors” group and the “TERMINAL_BLOCKS” family. Choose the two-pin part, as shown.



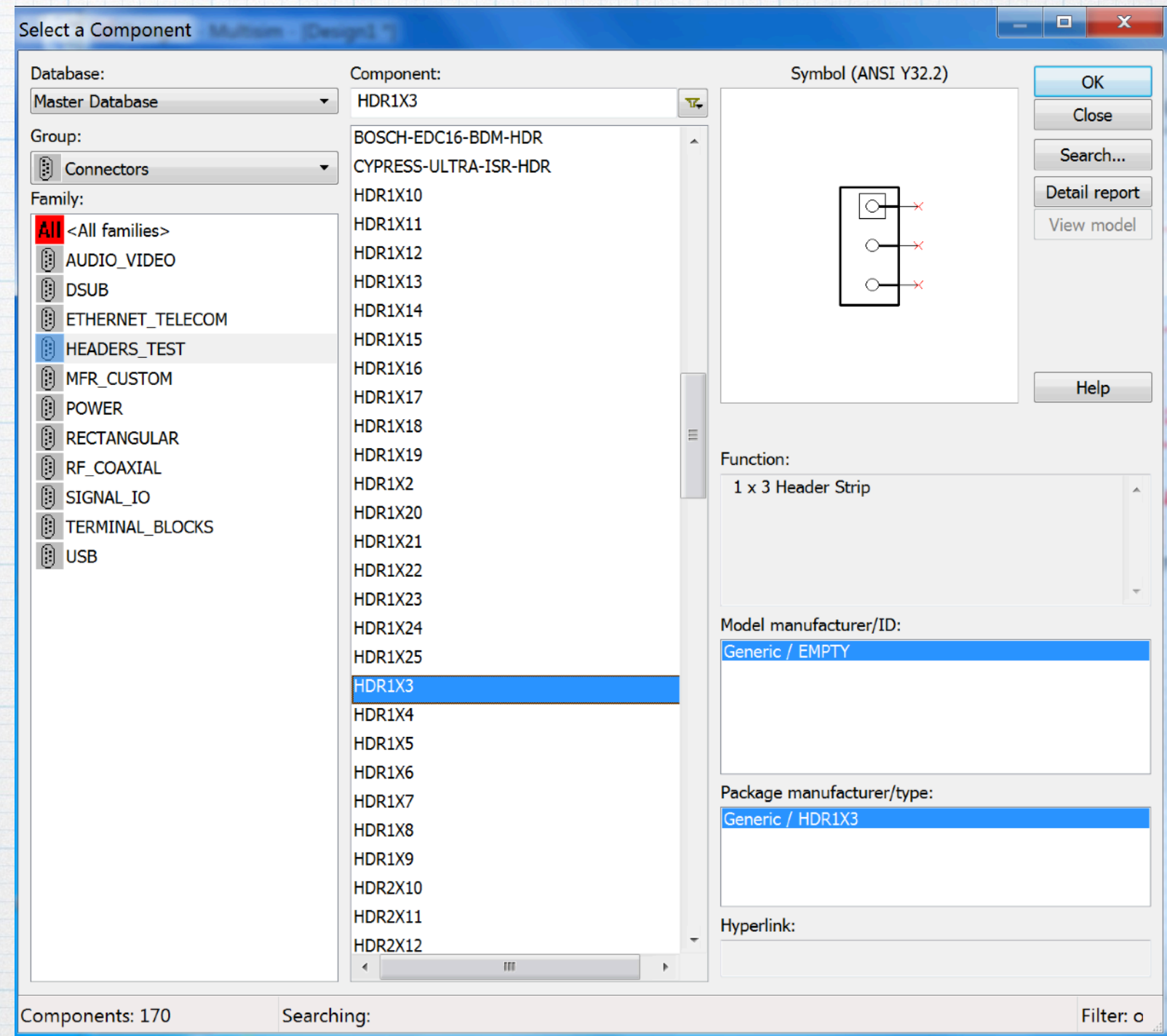
Place one terminal block near the input. Then, place a second one near the output (output of op amp B) — flip it horizontally to make the connections easier. Copy and paste ground symbols near each of the connectors. Wire one side of each connector to ground and the other side to the input or output.



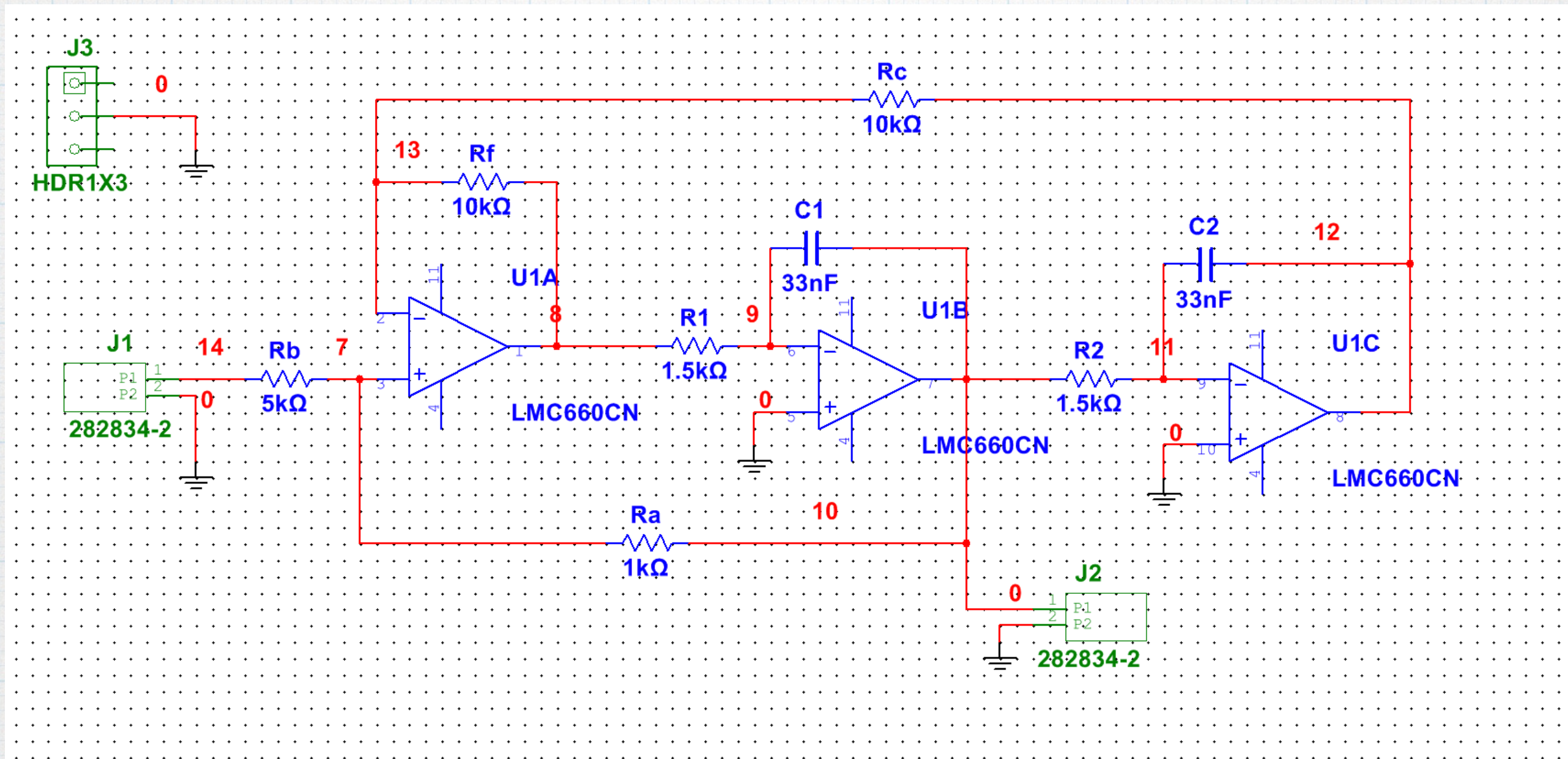
Finally, we will make connections for the power supplies. Ideally, we would use a 3-pin terminal similar to the 2-pin versions used for the signals. Unfortunately, there is no 3-pin version in Multi-Sim. (Go figure.) Instead, we will use a 3-pin “header”. Normally headers are used for test points or for jumpers in the circuit.

However, all we really need are three holes in the board that will match the pins of the real, existing 3-pin terminal blocks that we have in our lab kits. The header pins are spaced exactly the same as the pins for the terminal blocks.

Use the “HDR1x3” part from the “HEADERS_TEST” family.

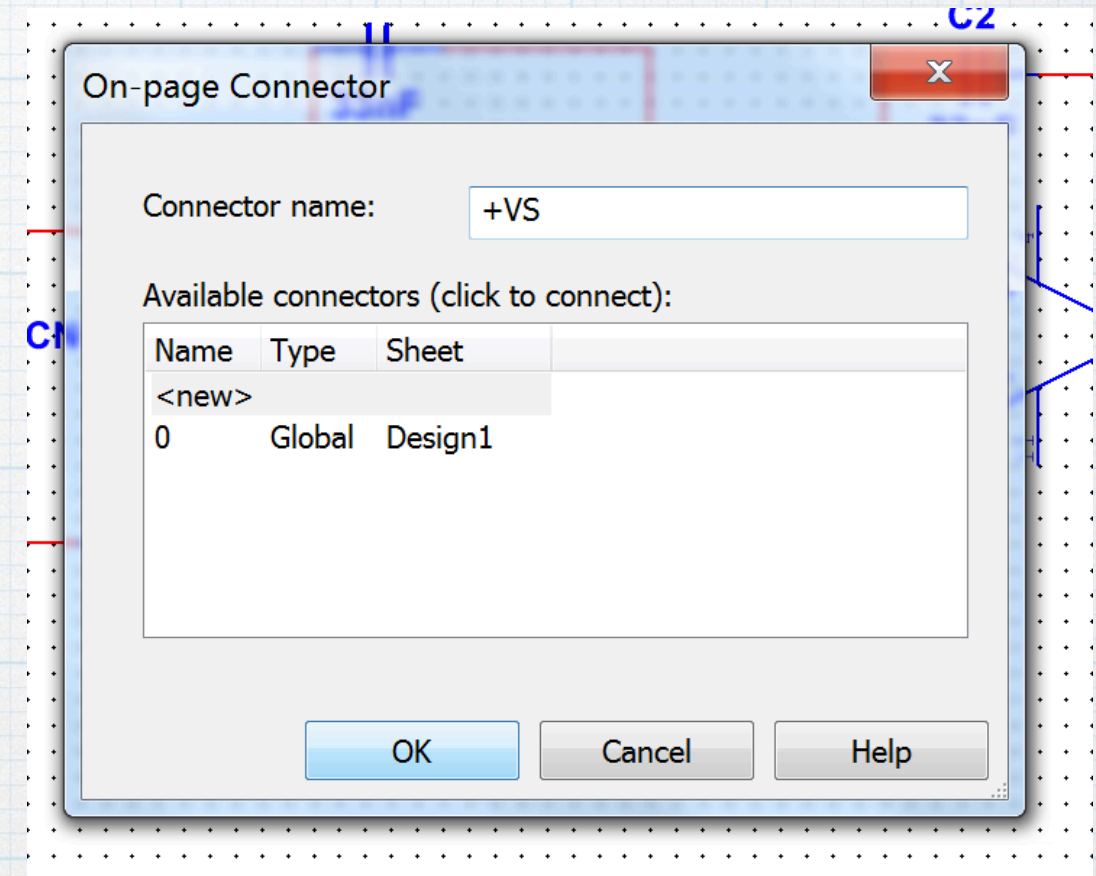


Place the three-pin header off to the side somewhere. Place a ground symbol near the header and connect it to the center pin.



We could run wires from the power supply pins to the op amp power connections. However, to keep things tidy in the circuit diagram, we will use virtual connectors. These can be used to connect any two points in a circuit to create a node. They are especially useful for power supplies. (Note that the ground symbol is a particular instance of a virtual connector.)

Select the the **Place -> Connectors -> on-page** menu item. Move the small connector symbol near the upper pin of the header and click to place it. The on-page connector window will open. Enter **+VS** in the connector name field and click OK to close the window. Wire from the on-page connector to the upper pin.

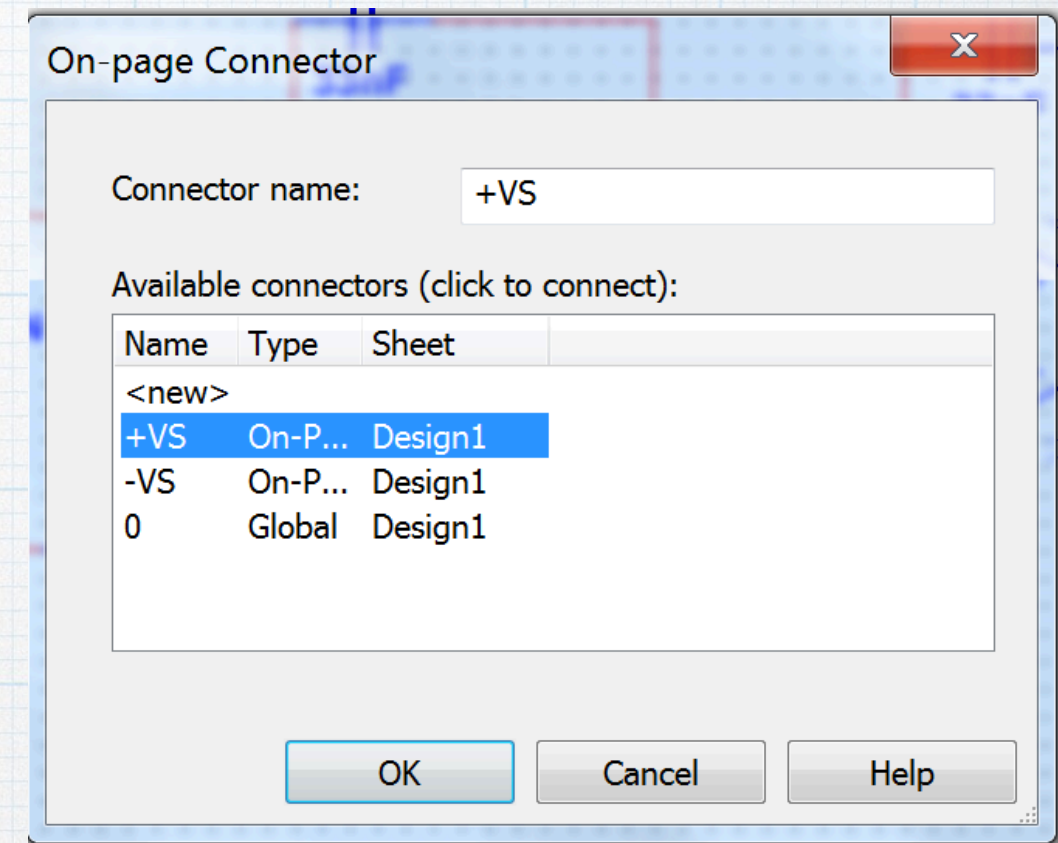


Then repeat the process to make a -VS on-page connector and wire to the lower pin of the header. (Flipping it vertically will make the wires slightly cleaner. Also, for some reason, there might be two name labels for each on-page symbol.)

Next, place another on-page connector near the terminal-4 pin of one of the op amps. It doesn't matter which one. However, this time, when the dialog appears, don't create another new name. Instead, select the +VS connection that we created initially.

Click OK and then run a wire from the new on-page connector to pin 4. This tells the layout editor that pin 4 is connected to the positive power supply.

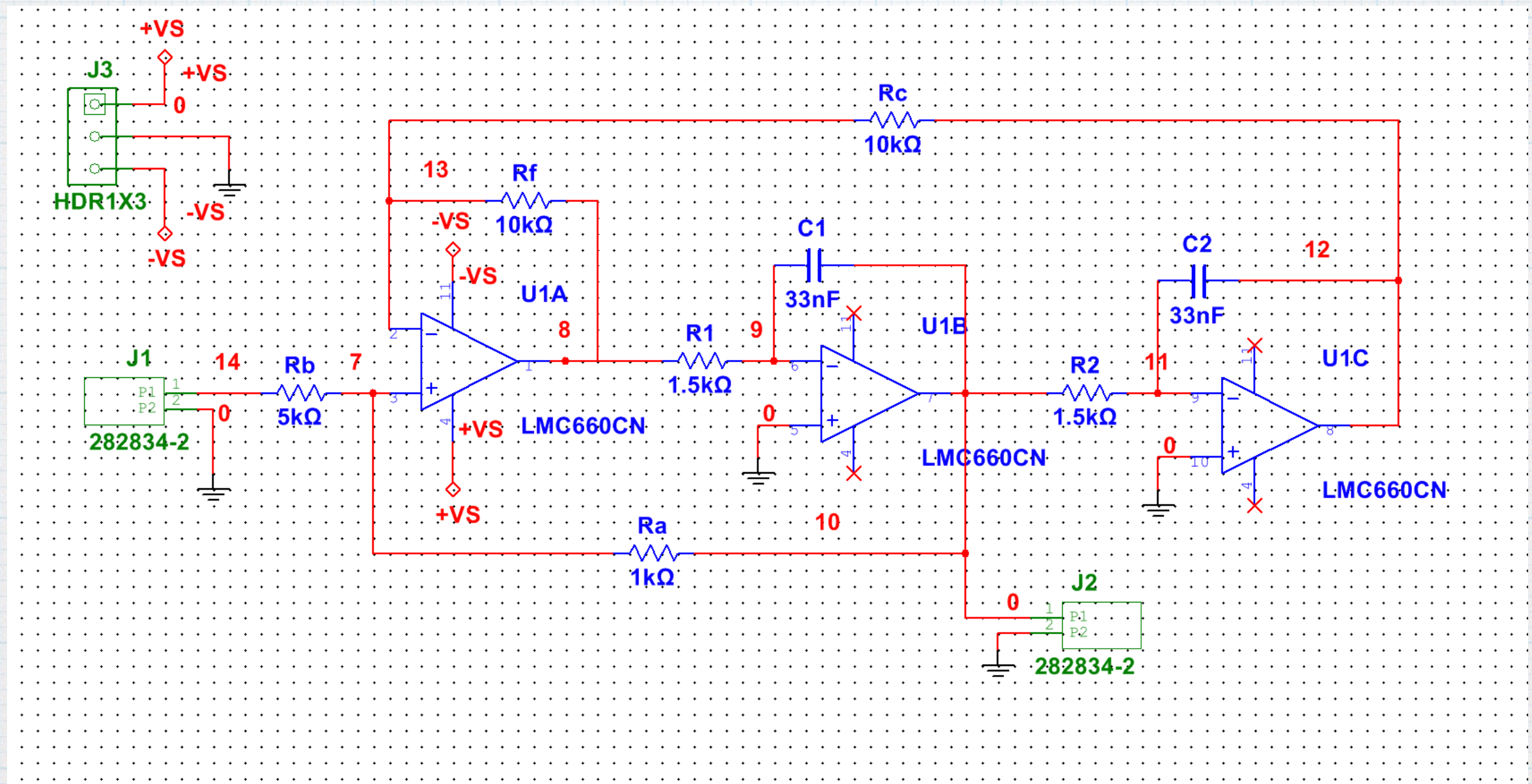
If there were other points in the circuit that should be connected to the positive supply, we could use this same process.



Repeat the process to create a -VS on-page connector that is wired to pin 11. Again, it doesn't matter which op amp is used.

Note that the power connectors on the other op amps now terminated with x's, because all the op amps in the chip share the same power connections — no more can be added.

And there we have it — the schematic layout is complete. We are ready to move over to the PCB layout procedure.



Of course, we should save this design file. We can give it whatever name we like — it will be saved as an .ms14 file type.