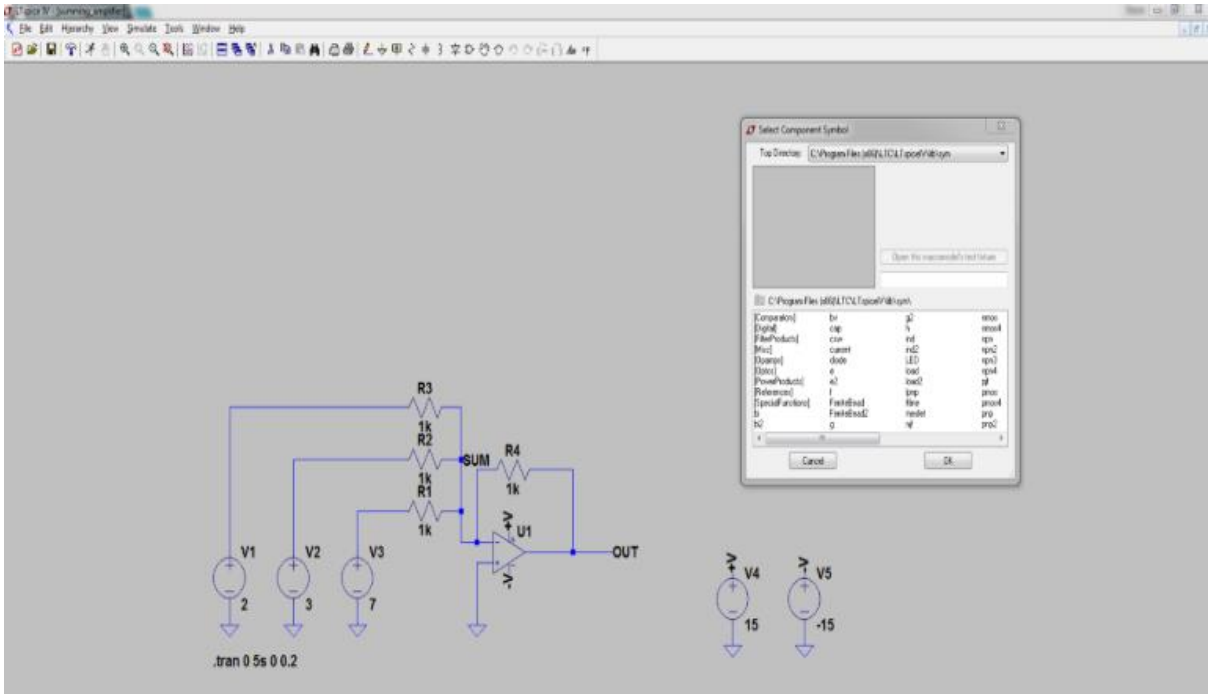
LTspice example: Simulating an Inverting Op-amp Summing Amplifier

Getting Started

We'll start by placing components and simulate a simple op-amp summing amplifier. Press F2 to access the component menu, which is where you'll find voltage sources and op-amp models. Check the screenshot below to see the pop-up component window (and the finished circuit schematic). You'll want to enter the "Opamps" subfolder and find the component called "UniversalOpamp2":



Place it anywhere on the schematic. Next you'll want to reopen the component menu and place a total of five voltage sources from the top level of the component menu (you can save time by using the copy hotkey after placing the first). Three of these should be to the left of the op-amp and two to the right. It's generally best to place all of your components and connect wires before editing values, so you'll also want to place three resistors in a configuration like the one shown again below. Note that your circuit should always have a ground symbol, or it will not function. If you want to rotate a component, remember to press CTRL+R while placing it.

Next you'll want to connect everything together by pressing F3 to draw wires. Once you've drawn a wire between two components, right click to end your draw. Regarding wires, clicking once will anchor the wire to the point you clicked and allow you to change direction (hold CTRL while pressing F3 to place wires that aren't at right angles), and you can draw wires directly through multiple components to connect them quickly. You'll also want to place a ground for every power supply and at the positive op-amp terminal.

Something you might've noticed at this point is that the +V and -V terminals on the op-amp and on two of the voltage sources in the above schematic don't have wires connecting them. By pressing F4 (label net), you can give a particular wire or port a name. Giving the same name to two objects connects them as if you had a wire between the two and is a good way to reduce clutter (as well as specify where you are interested in probing once you actually

simulate). Go ahead and use this feature to connect the terminals of your op-amp to the two right-most voltage sources.

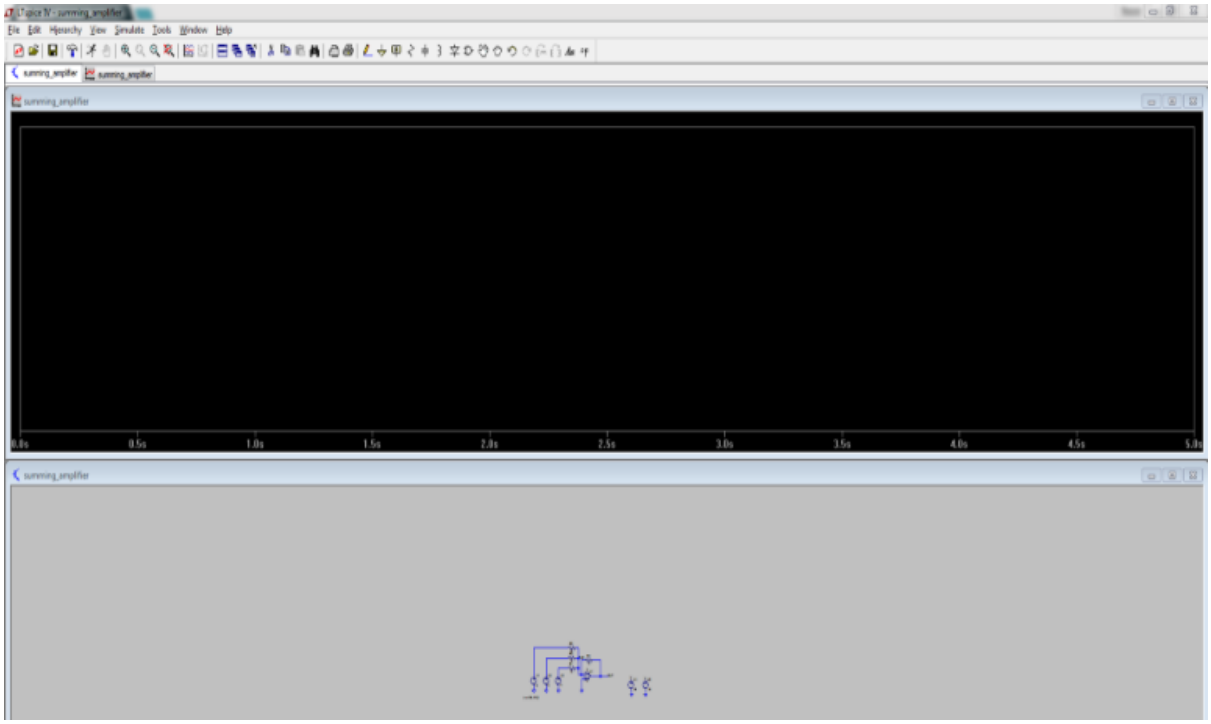
Now it's time to give components some values. By right-clicking on any component, you can edit its properties. You'll find that the resistors will be pretty straightforward, with boxes for resistance values, tolerance and power rating. The values you enter only require letters to specify order of magnitude (so you don't have to write "ohms" after every number), and it's important to note that in LTSpice, "m" and "M" mean "milli", while you use "MEG" (or something like 1000k) to specify mega-ohms/farads/henries/etc. In the example given, all four resistors have a value of 1k, so enter that for each. It's a bit slower than filling out one and copying, but in the future having your circuit placed before you edit components will help ensure you don't forget things.

You shouldn't need to change anything in the op-amp's properties, but looking at them will give you a hint of some of LTSpice's more advanced features. As for the voltage sources, for this circuit you'll want to simply enter DC values. The example above had input voltages of 2, 3, and 7 volts, so we should expect an inverted output of -12V when we simulate. Set the power supply voltages to +15 and -15.

One final thing before we get to simulation: you may have noticed that net labels are used to specify "SUM" and "OUTPUT". You can do this to make selecting traces easier, but remember that "SUM" should actually be around 0V.

Simulating our Summing Circuit

Once you've got the circuit set up like the screenshots above, you're ready to do your first simulation. LTSpice is capable of several types of simulation, but today we'll be covering just one: **.tran**, which stand for Transient analysis, respectively. This is one of the two most commonly-used simulation forms and garner some valuable info. Open the "Simulate" menu and go to "Edit Simulation Cmd". There are several tabs here, but this time we just need to use the Transient section. Fill it out with some reasonable values (I chose a stop time of 5s and a timestep of 0.2). Hit OK and you'll now have a final component to place, which contains the simulation parameters; put it anywhere you like. After that, either right-click the schematic and hit "Run", or press it in the Simulate menu. You should see a graph fill half of the LTSpice window, but there won't be anything in it yet:



In order to get a look at the circuit output, you'll need to choose what parts of the circuit you want to see. You can do this one of two ways:

Right-click on the black graph and hit "Add Trace", then choose the net you want to display.

(Easier) Click on the part of the circuit you want to see a trace for. Click on a wire to see a voltage, or click on a component to get a current (which we don't need for this example).

Go ahead and click on the wires above V1, V2 and V3, and again on the "OUT" label (or the wire if you didn't add the label). This'll plot four straight lines, each in a different color. You will see that the output is -12V as expected.

Congratulations! You've simulated your first circuit in LTSpice. It was pretty basic, but you should be familiar enough with the interface now that you can continue without being too lost. A few notes about traces when simulating:

If you ever want to plot signals relative to something other than ground, you can right-click on a node on your circuit and hit "Mark Reference". Until you set ground as the reference again, all plots you make of the circuit will be relative to that point.

You can view differential voltages by clicking one part of circuit with your probe and dragging to the second (negative) node.

If you want to see the current in a wire, click it while holding the ALT key. Doing this for a component will show you the part's instantaneous power.

Removing a trace can be done by right-clicking its name at the top of the output window, or by pressing F5 and clicking it.