# Using LTSPICE to Analyze an Op Amp Circuit.

Overview:

LTSPICE is circuit simulation software that automatically constructs circuit equations using circuit element models (built in or downloadable). In its modern form the software constructs those equations using a graphical user interface (the jargon is a schematic capture tool). Other SPICEs exist and have differing strengths.

You will construct your circuits on a schematic using menus to choose circuit elements from. Some common elements (wire resistor, capacitor, inductor, ground etc.) are on the bar at the top, others such as Transistors (npn, pnp, MOSFETs), Op Amps, voltage sources, current sources, etc. are available under the AND gate symbol ⎓⊐⎓ . Symbols that have not yet been placed can be rotated and transposed using bar tools.

Once you place the circuit elements you give them their values (e.g., resistance, capacitance, rating, precision) by right clicking on the element. For place value SPICE understands SI prefixes with some modification

mega=1E6=10^6

k=10^3

m=10^{-3}

u=10^{-6}

Units are a useful form of comments and as such are ignored by the program are a good idea. The simulation is a computer code and needs to have an explanation in the form of comments. These comments can be inserted using the bar Aa tool.

With a schematic drawn you can run a simulation, once you have configured and placed the simulation command. This is done under the Simulation menu by selecting Edit Simulation Command. The most common simulation we will do is a Transient analysis and minimally you need to select the duration of the simulation (Stop Time) and how much of the transient response to throw away (Time to Start Saving Data). The program constructs the command and clicking OK allows you to place the .tran command in the schematic window. You are now ready to run the simulation.

Running the simulation pops up a blank graph where you can display voltage and current traces. Hover the cursor over a wire or terminal and the program shows a voltage probe icon. Hover over a circuit element (e.g. a resistor) or a tap for an element and the program shows a current clamp. In either case a single click adds the trace to the graph and a double click makes that the only trace on the graph.

The tasks for today.

Task 1

In groups of three sketch on piece of paper the schematic for a non-inverting amplifier without any load and amplification of 10 (you will turn this in). Include your choice of resistor values.

Construct an LTSPICE schematic for a non-inverting amplifier without any load and amplification of 10. Power this with +/-15V rails and drive it with 4 different signals:

1. 1.0V DC
2. 2.0V DC
3. 0.1V AC at 100Hz
4. 0.1V AC at 100kHz

Describe the results of your simulation for each driving signal. Tell me about the output voltage in each case and anything else you think is important. I'm not looking for just a number but also a comparison to what you expected.

Task 2

Add a resistive load, R_L, to your circuit and check the voltage across three choices for R_L: 1kOhm, 100Ohm, 10Ohm using the 0.1V AC signal at 100Hz. From your simulations estimate the output impedance of the amplifier. Be sure to explain your results.

A hint for tidier circuits: rats nests obscure wiring mistakes and impede understanding. One thing you can do to simplify your schematic is to use names network ports to tie wires together. Ground is a special example of a network port but you can make your own for things like the rails (select an output port at the power supply and an input port at your circuit). All ports with the same name are connected.

Constructing Op Amp circuits

Task 1

Build an inverting amplifier circuit with an amplification of 100 using a NTE941 op amp:

Pin Connection Diagram

Offset Null	1	8 N.C.
Inverting Input	2	7 V (+)
Non–Inverting Input	3	6 Output
V (–)	4	5 Offset Null
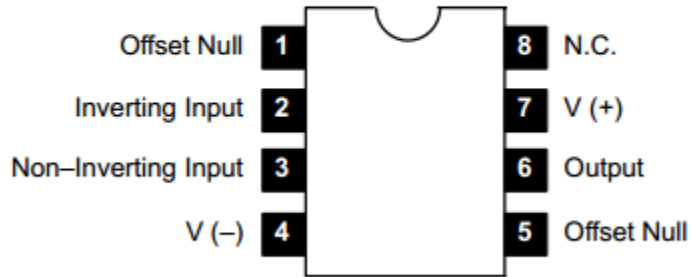
*from NTE's data sheet.

Test your circuit with a 0.1V at 100 Hz signal using only the Oscilloscope as a load. You should display both V_{in} and V_{out}.

Task 2

Explore the effect of the load resistance on V_{out}. A good assortment of resistances includes 100kΩ, 10 kΩ, 1 kΩ, and 100 Ω. Note any surprising behavior and how you fixed it.

Task 3

Explore the effect of signal frequency on amplification, A_v. Produce an amplitude vs frequency curve taking one to two data points per decade (more data points where A_v varies fastest with frequency). Make two graphs, one linear-linear the other log-log.

Optional

In addition to the NTE941 op amp that uses BPJ transistors, I've picked up a couple of NTE937 op amps that use JFETs at their inputs. You could compare the behavior of the NTE937 chip to the NTE941 chip, especially as a function of frequency.

Product:

Synthesize the notes your group has taken (on SPICE and Circuit construction of Op Amp circuits) into a 1-2 page document that you can use when designing and constructing op amp circuits. This should be at a level of detail that makes it useful to you when constructing your R&D project at the end of the semester.

It should briefly describe what Op Amps are good for and what their limitations are. It should also briefly describe what the various techniques you have at your disposal are good for and what their limitations are.

Punctuate this with examples using your data (simulation or experimental) as seems sensible. Each person will construct their own document.