

# LT Spice Documentation

## Introduction

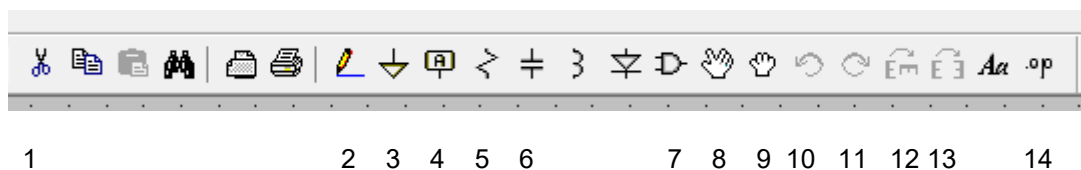
LT-Spice is a simulator package that allows you to easily draw out your circuit and then simulate its operation to check voltages, currents, etc. This allows you to easily check the effect of different component values on circuit behaviour, BEFORE you commit to put the circuit on breadboard and printed circuit board which is typically a much more time intensive task. This short document gives you an overview of how to use LT-Spice for the audio amplifier design in the 2<sup>nd</sup> year laboratory.

## Part 1: Drawing the Circuit

Analog Devices have a short video which introduces you to basic ideas of drawing your circuit here:

<https://www.analog.com/en/education/education-library/videos/5579254289001.html>

Please watch the video to get the basic ideas. Below in numerical order is a simple summary of useful **drawing icons** at the top of the screen:



1. **Scissors**: Delete Mode to click on wires or components for removal
2. **Yellow Pen**: Draw a wire connection
3. **Down Arrow**: Insert Ground Symbol
4. **Letter A**: Label a wire connection point
5. **Resistor**: Insert a resistor in your circuit
6. **Capacitor**: Insert a capacitor in your circuit
7. **And Gate**: Insert component (with search facility)
8. **Open Hand**: Move component in your circuit
9. **Closed Hand**: Drag component in your circuit
10. **Left Arrow**: Undo action
11. **Right Arrow**: Redo action
12. **E and M**: Rotate an component (e.g. resistor) you have just selected to place on your design – You can also press CTRL+R to do the same thing
13. **Two E's**: Mirror an component you have just selected
14. **.op**: Insert Spice commands to find libraries and run simulations

There are also some useful **zoom icons** as shown below:



The plus sign increases zoom, the minus sign reduces zoom and the red line icon will maximise the zoom of your circuit while showing it all in one window.

There are also some additional options you may find useful as you draw your circuit:

1. **Delete Key** – select delete mode to click on wires or components for removal from your diagram.
2. **Esc Key** – return from a special mode (e.g. delete, draw track, place component, etc) to normal mode (cross-hair mouse).
3. **Moving mouse over a component** will change the icon to a pointing finger. Right click the mouse to edit the properties of a component, such as the resistance, capacitance, etc.
4. **Moving mouse over a wire** and right-clicking allows you to select “Highlight Net”. This is a very useful option for debugging that shows you ALL parts of the circuit in yellow that are connected to the same voltage. This means you can check that everything is properly connected.

## Useful Circuit Components

Some of the main components you will need in your design are shown in numerical order in Figure 1 below:

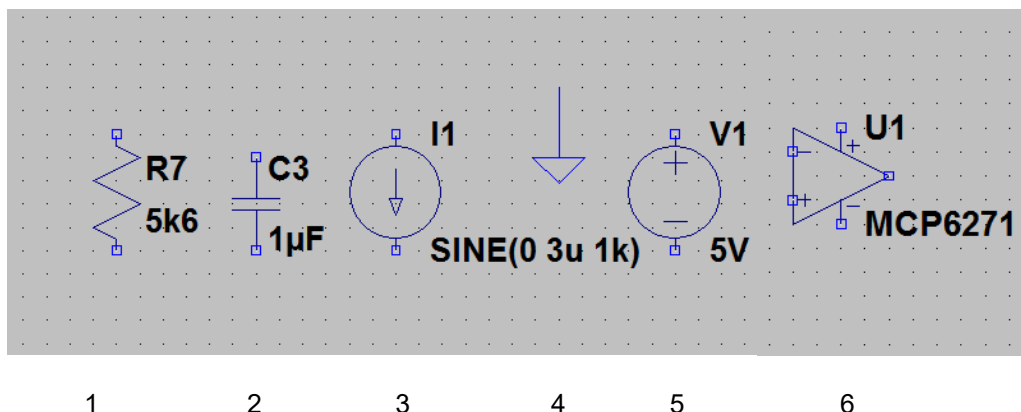


Figure 1: Useful components for the audio amplifier circuit

1. **Resistor**: Right-clicking on this component allows you to set the resistance, such as 5.6 kΩ (insert the value 5k6) as shown above.
2. **Capacitor**: Right-clicking on this component allows you to set the capacitance such as 1µF (insert the value 1uF) as shown above.
3. **Current Source**: You will need this to simulate a current source which is a suitable model for a microphone. Click on the component icon and search for “current” to find the current source. Once inserted, right click and hit the “advanced” button. From here, select the “sine” option on the left side and you can set the relevant parameters you need, such as DC current, AC current and AC frequency.

4. **Ground:** Insert this symbol wherever you need to, in order to connect relevant parts of the circuit to ground.
5. **Voltage Power Supply:** You will need this to provide a DV voltage to power your circuit. Click on the component icon and search for “voltage” to find the voltage source. Once inserted, right click and select the desired voltage.
6. **Operational Amplifier:** You will need this for the op-amp part of the circuit. Click on the component icon and search for “MCP6271” to find the component. The two connections on the left are the negative and positive op-amp inputs. The two connections top and bottom are the power supply connections (Vcc and 0V ground) and the right hand side is the op-amp output.

A full list of Spice shorthand for different component values is given in the Appendix of Spice Notation at the end of this document.

Now complete your circuit diagram. Use the label wire option to give sensible names to different nodes of your circuit.

Finally, you need to add some Spice commands to help LT-Spice find the libraries for your op-amp component. Click on the .op button (Drawing Icon No 14 on page 1) to add this line:

```
.LIB MCP6271.mod
```

When you press return, you can click somewhere on the drawing to add them – in Figure 2 below, the line has been placed at the bottom of the drawing.

## Part 2: Circuit Simulation

Once your circuit is complete, you should be ready to simulate the circuit. Linear Technology have a short video on circuit simulation here:

<https://www.analog.com/en/education/education-library/videos/5579252577001.html>

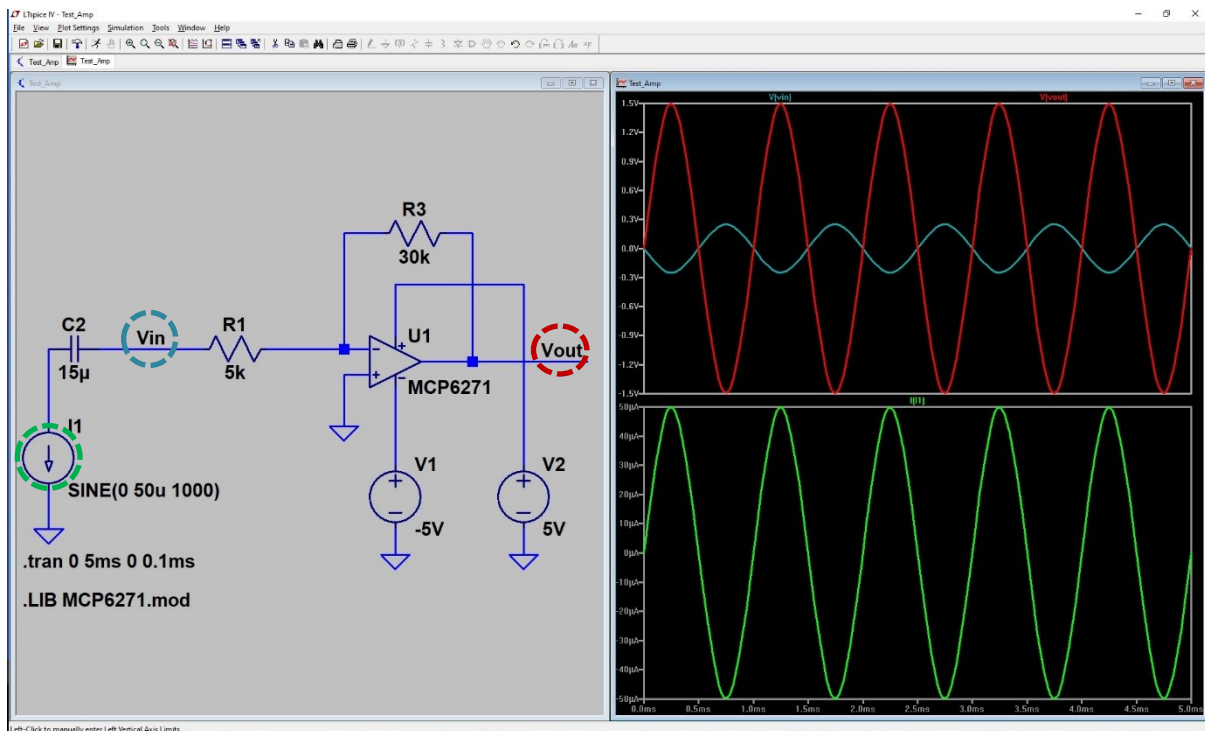
As an example consider the simple audio amplifier using an op-amp as shown in Figure 2. Note that this circuit contains a Norton source (current source with 5k resistor), but it is deliberately different from the one that you will design in this laboratory!

Click on “Simulate” menu then “Run” to bring up the Oscilloscope window shown in Figure 2 – initially it will be blank. You might also be asked to setup the simulation parameters – here the time period is set from 0-5ms with a minimum time step of 0.1ms. The default window arrangement is above/below – use the “Window” menu then “Tile Vertically” option to achieve the arrangement shown in Figure 2. You can then click on different parts of the circuit to plot the voltages on a wire or the current flowing through a component at specific places.

In Figure 2, at the top right we used the voltage probe to click the input voltage “Vin” (shown in a blue circle on the left) and “Vout” (Shown in a red circle on the right). The gain of the circuit is observed to be about 6 times. The voltage “Vin” is  $\pm 250\text{mV}$  or a peak-peak voltage of  $500\text{mV}$ . Similarly the voltage “Vout” is  $\pm 1.5\text{V}$  or a peak-peak voltage of  $3\text{V}$ .

When you are in the Oscilloscope Screen, you should see the “Plot Settings” menu (as in Figure 2) that allows you to add or delete traces. Right click on coloured labels to allow you to delete a trace if you don’t need it. In the menu, there is also an “Autorange Y-axis” option which can resize the plot when you have deleted traces and “Manual Limits” allows you to control both the X-axis and Y-axis ranges. You can also use the “Add Plot Pane” to give two plots, one above the other, as shown in Figure 2. The lower right plot in this figure shows the current I1 from the Norton source, which is circled in green on the left of the figure.

Finally at the bottom of Figure 2, note the .LIB commands as described above. The .tran command relates the simulation time and step size settings and is usually added automatically when you simulate.



**Figure 2: Example Circuit Simulation in LT-Spice**

For your own circuit you can measure the node “Vin” or its equivalent as the input voltage to your circuit. and you can also measure the output voltage from the op-amp to compute the circuit gain. Once this result is measured for a frequency of  $1\text{kHz}$ , then you can adjust the frequency of the current source to measure a few different frequencies. These should lie in the range from say  $100\text{ Hz}$  to  $100\text{ kHz}$  and will thus allow you to determine the frequency response or Bode plot of your circuit.

Selecting the “Simulate” menu then “Edit Simulation Cmd” allows you to change the time setting. In most cases the default of 5ms should be sufficient to see what is happening (unless you choose a very low AC frequency for the current source). You can also select different simulation types, such as “AC Analysis”, which is an alternative method to check the frequency response of your circuit.

## **Circuit Debugging**

There are a couple of useful options for checking that your circuit is drawn correctly. Firstly a reminder that you can right-click on a wire and use the “Highlight Net” option to check that everything is REALLY connected as it should be!

Selecting the “View” menu, then “Spice Netlist” gives a log of all components and the voltage nodes they are connected to (meaningful node names will help here!). Selecting the “View” menu then “Spice Error Log” will help to identify obvious problems such as floating component connections, etc.

## **Saving Circuit Diagram for Lab Book**

You may wish to save a copy of your circuit diagram for your notes or your lab book when finished. Selecting the “Tools” menu then “Copy Bitmap to Clipboard” will save your circuit diagram into the clipboard where you can then paste into Microsoft Word or other editors. Alternatively, if you have a PDF printer facility, you can print out the diagram using the “File” menu then “Print”.

## Appendix of Spice Notation

One of the commonest errors made in LT-Spice is to use the wrong units:

1p	stands for	1 pico	$10^{-12}$
1n	stands for	1 nano	$10^{-9}$
1u	stands for	1 micro	$10^{-6}$
1m	stands for	1 milli	$10^{-3}$
1k	stands for	1 kilo	$10^3$
1MEG	stands for	1 mega	$10^6$

Make sure you use the right notation for resistors, capacitors etc!

## Appendix of Formulas

Gain in dB is a logarithmic comparison between the input and output AC amplitudes  $V_i$  and  $V_o$ , given by:

$$\text{Gain(dB)} = 20 \times \log_{10} \left( \frac{V_o}{V_i} \right)$$

A 3dB reduction in the Gain value corresponds to the amplitude ratio being scaled by 0.707, or half the power.

The cutoff frequency (in Hz) of a resistor R + capacitor C is given by the formula:

$$f = \frac{1}{2\pi RC}$$

## Appendix: Installing LT-spice on Your Computer

You can download and install free Windows (and Mac OS versions) of the simulation software here:

<https://www.analog.com/en/design-center/design-tools-and-calculators/ltspice-simulator.html>

In order to simulate the amplifier circuit you will need to install some extra component files which are available in a Zip file on Learn.