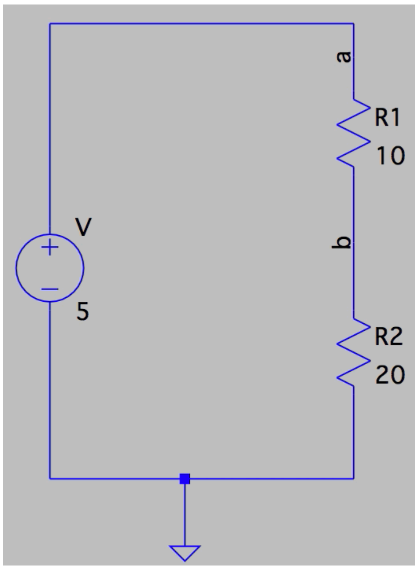
1.Introduction to LTSPICE[1]

This PDF completes the youtube video and its aim is not to substitute the video.

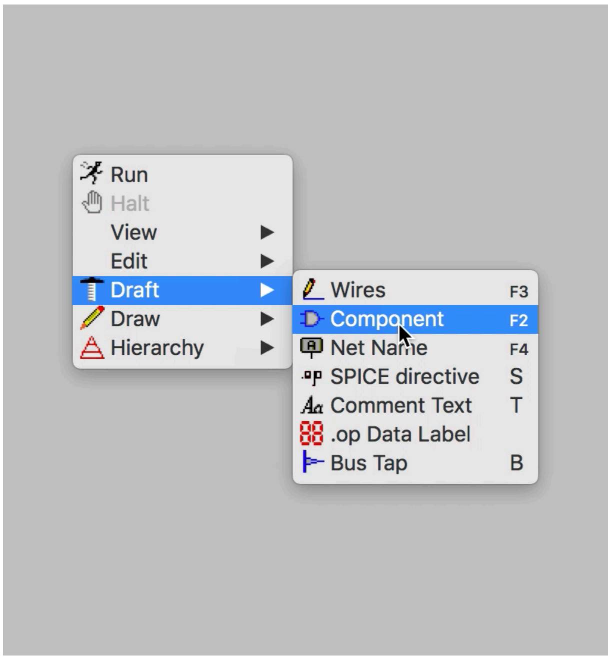
1.1 Build a resistive voltage divider circuit

When you open LTSpice, select « Start a new, blank Schematic » to create a circuit.

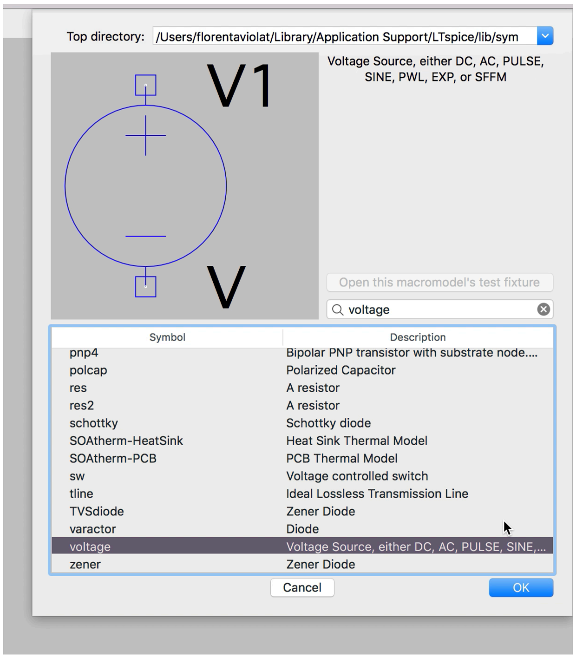


Resistive voltage divider circuit

To add a component, right-click → Draft → Component or F2 (or Fn-F2). On Windows, you can also use the toolbar. Firstly, select a voltage source (voltage).



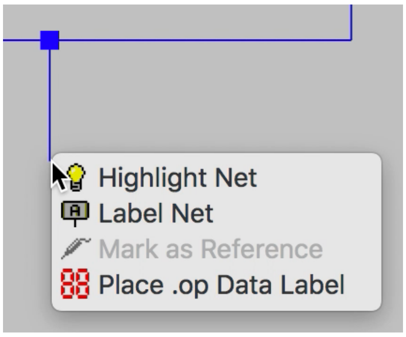
Add a component



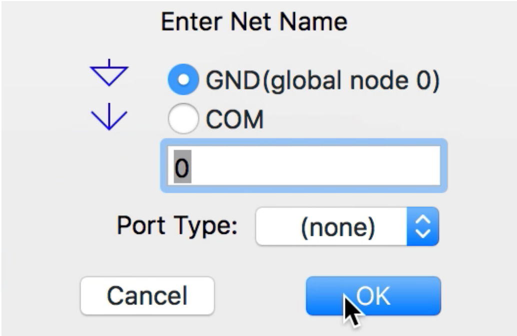
Add a voltage source

[1] The tutorial is realized on Mac with the version of the 18th of March 2020.

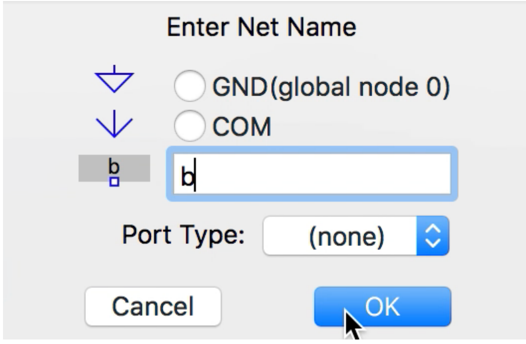
Then, add two resistors (res). If you want to rotate a component, press Ctrl+R.
To add wires, right-click → draft → Wires or F3 (or Fn-F3). Finally, we draw a wire that goes out of the loop. Right-click on this one, select Label Net and mark GND to connect this wire to the ground.



Label Net

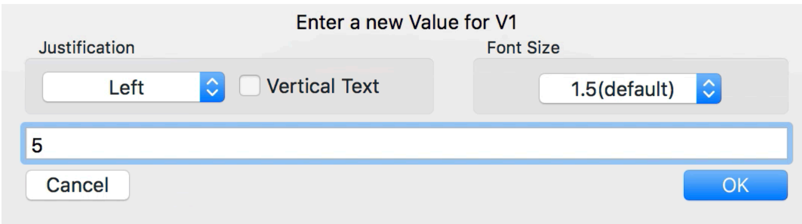


Label Net - GND



Label Net - Name a branch 'b'

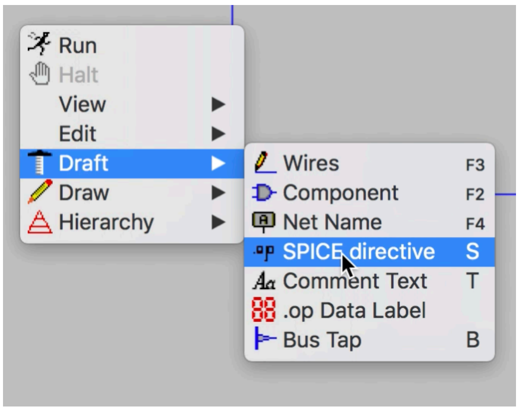
To modify the name of a component, right-click on its name.
To name a branch, right-click on the wire, select Label Net and enter the name.
To configure the value of a component, right-click on its value.
To edit a component, right-click on it.
Configure the names and values as on the picture of the circuit above.



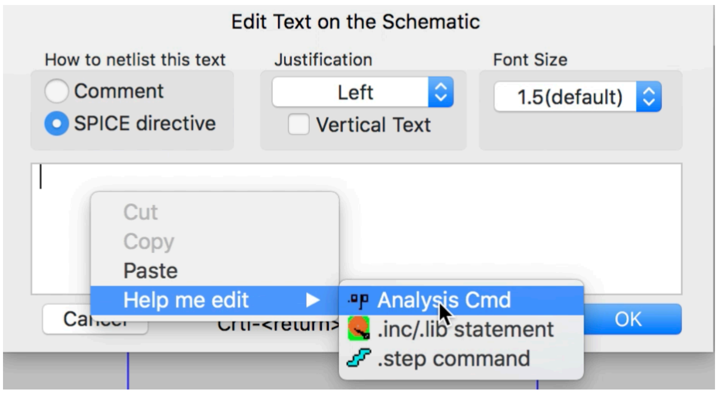
Enter a value for V1

1.2 Simulation : DC, .op

Enter the directive of the simulation, right-click → Draft → SPICE directive (or S). If you know how to write the directive, you can write it directly else, right-click → help me edit → Analysis Cmd. Then, select DC Biais point (.op). This directive

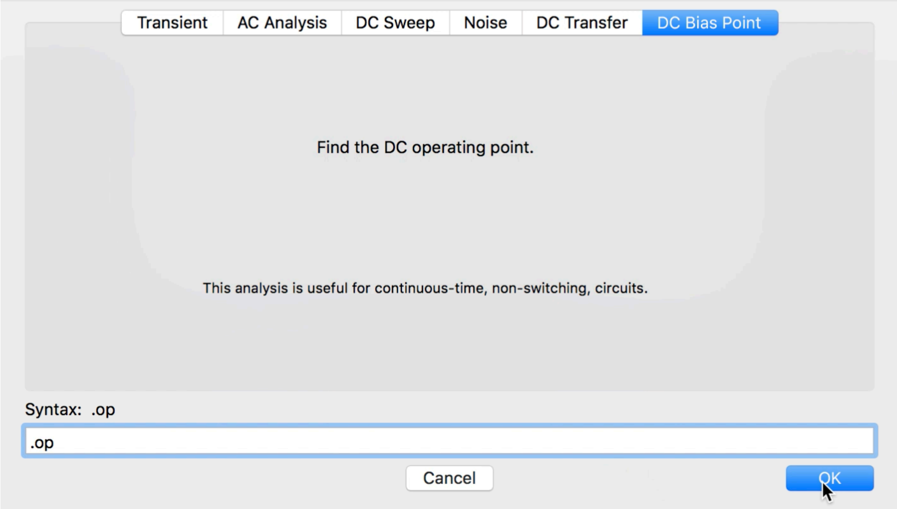


SPICE directive

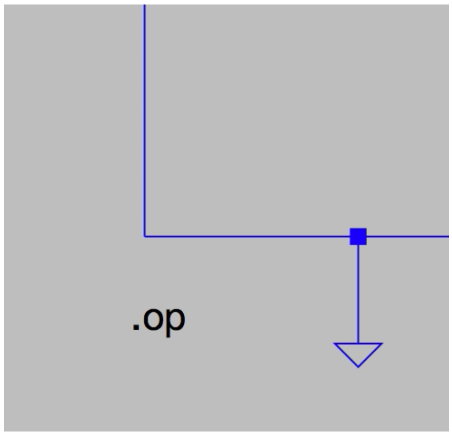


Help me edit

Professor : Farhad Rachidi Video and PDF : Aviolat Florent
* Original version in french, Video : Aviolat Florent / PDF : Herold Gaetan



DC Bias Point (.op)

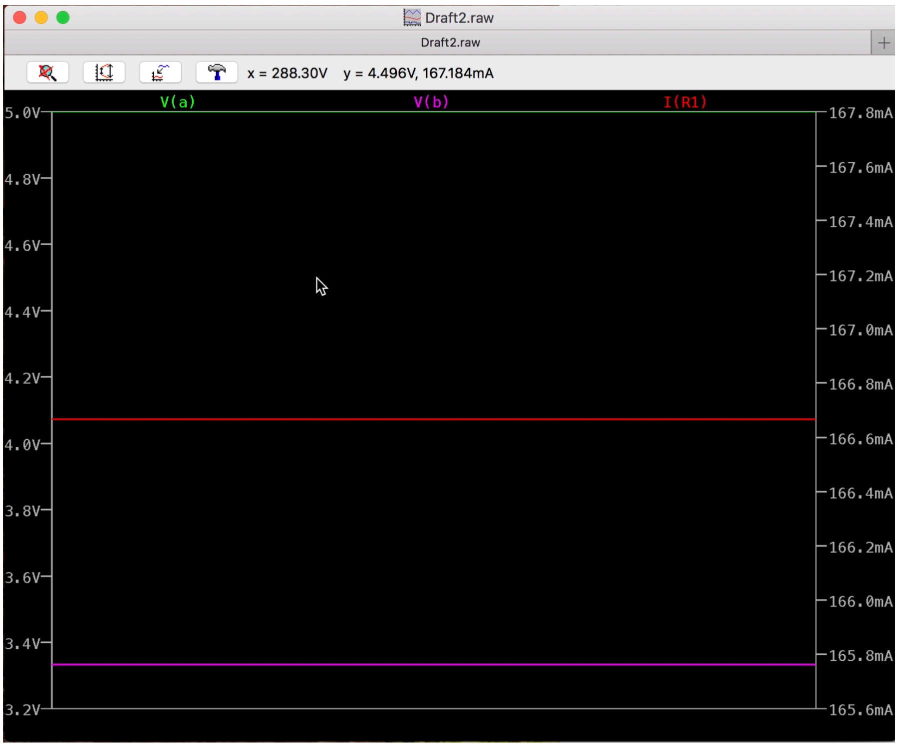


.op

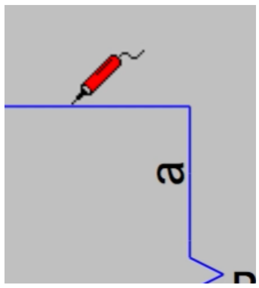
switches on the voltage source in DC. It is used to find the DC operating point of the elements of the circuit. You can then place the directive where you want on the sheet.

When the directive is set on the pane, the circuit is ready for the simulation. Click on Run. With .op, if you are on Mac, a black plot pane (.raw) should appear else if you are on Windows, a text file should appear. Even if the display is different with .op, the results are the same in either case. We will go on with the black pane, that is very used with the others directives.

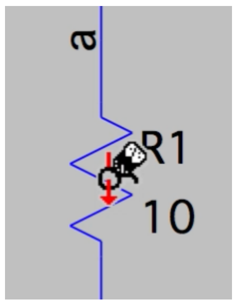
To measure the voltage of a branch, come back to the circuit (.asc). When you place the mouse on a branch, a red probe (cf. picture) appears. Click to measure.



Plot pane with .op



Voltage probe



Current probe

Professor : Farhad Rachidi Video and PDF : Aviolat Florent
* Original version in french, Video : Aviolat Florent / PDF : Herold Gaetan

To measure the current in a component, come back to the circuit (.asc). When you place the mouse on a component, a probe (cf. picture) appears. Click to measure. Be careful with the positive direction of the measurement indicated by the red cross.

Some options with the plot pane :

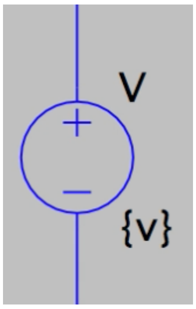
Add a grid : right-click → View → Grid
Modify an axis : right-click on the axis
Modify the color of a measurement : right-click on the measurement name

To read the measurements precisely, we can attach a cursor by right-clicking on the measurement name (more detailed later).

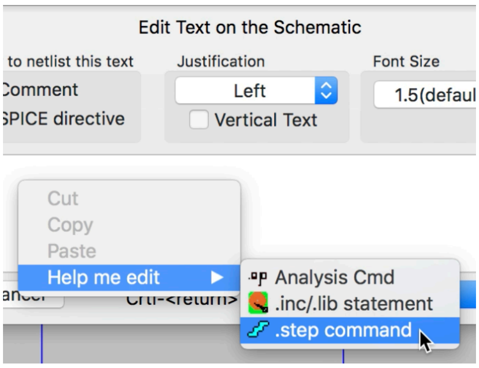
Finally, we check the obtained values with the theoretical expected values: $U_b = \frac{R_2}{R_1 + R_2} U_a = 3.3[V]$ and $I = \frac{U_a}{R_1 + R_2} = 0.166[A]$. These values match with our measurements on LTSpice, so our simulation and circuit seem to be correct.

1.3 Simulation : DC, .op and .step

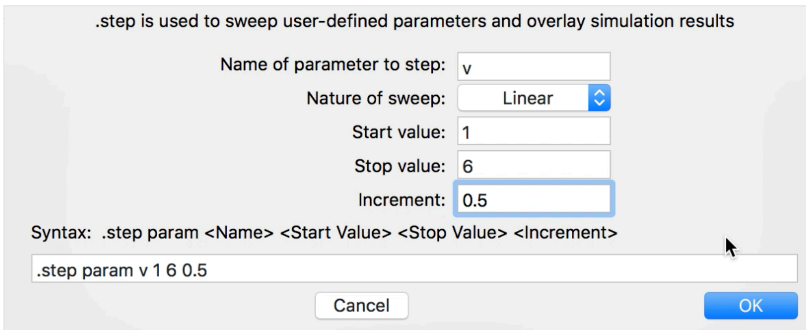
The aim here is to vary the voltage of the voltage source and to observe the circuit under different values of this voltage. As this value is now a variable, we have to change the value of the voltage of the source by the name of the variable between curly brackets (here {v}). Then, we have to add the command .step, right-click → SPICE directive, then right-click → help me edit → .step command. We configure the sweep of the variable v as written on the picture.



Variable v



Help me edit

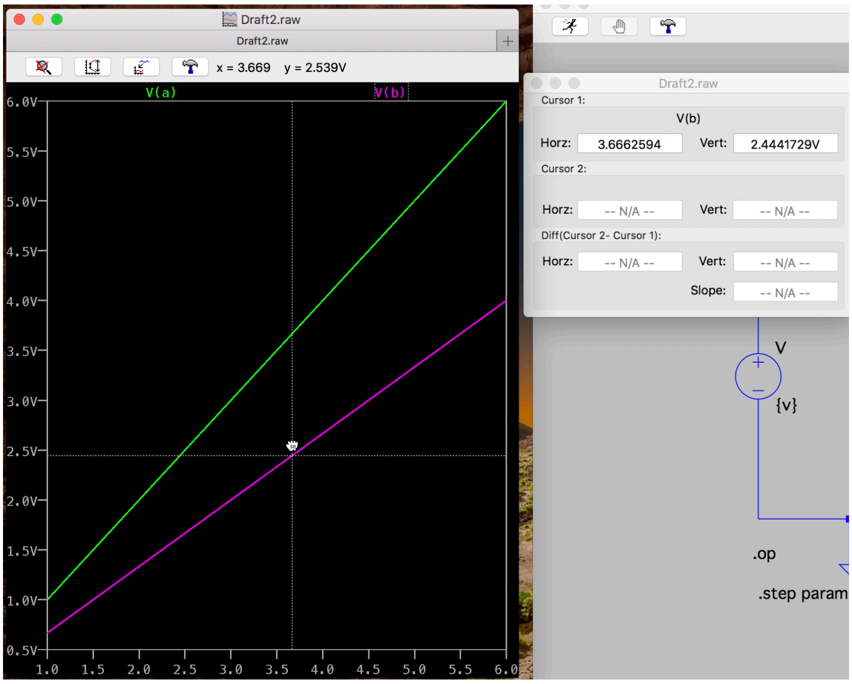


.step command

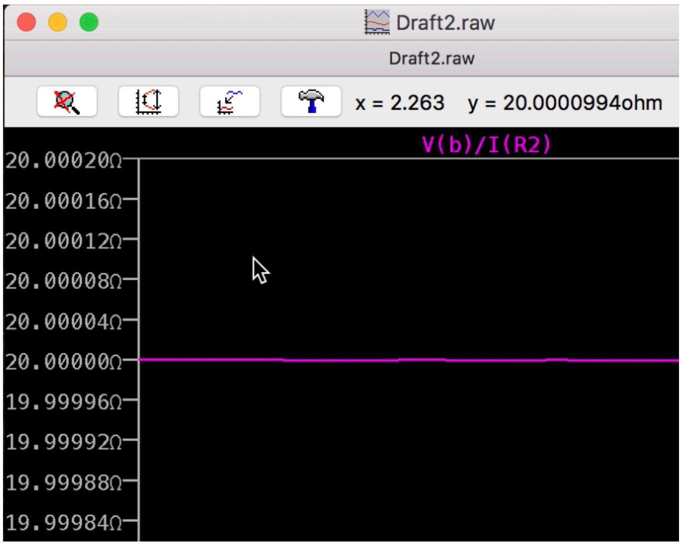
Afterward, we run the simulation. The x-axis is the variable v. We display V(a) and V(b) and to better read the result, we attach a cursor on V(b). We can move the cursor on the plot pane and have the values on another window.

Some options with the plot pane :

Display the data points : right-click → View → Data points
Add a new plot pane : right-click → Add Plot Pane
Add a new trace : right-click → Add Traces (or A)



Cursor

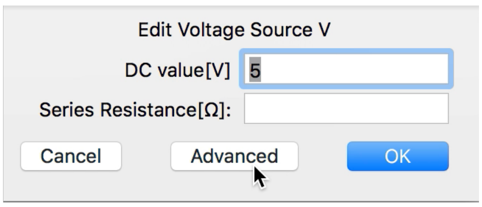


Resistance R2

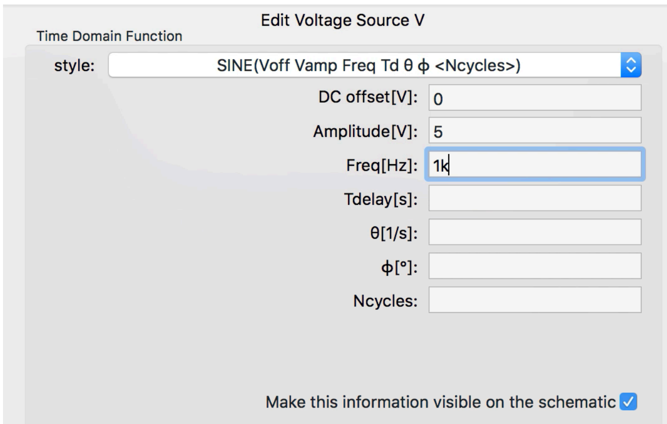
On a new plot pane, we want to calculate the resistance of R2 by using the Ohm law. Add a new pane, then select Add Traces and enter the expression V(b)/I(R2).

1.4 Simulation : AC, .tran

First thing, we configure the voltage source to have sinusoidal signal. Right-click on the source → advanced, and select the style : SINE. Then, we set the values as on

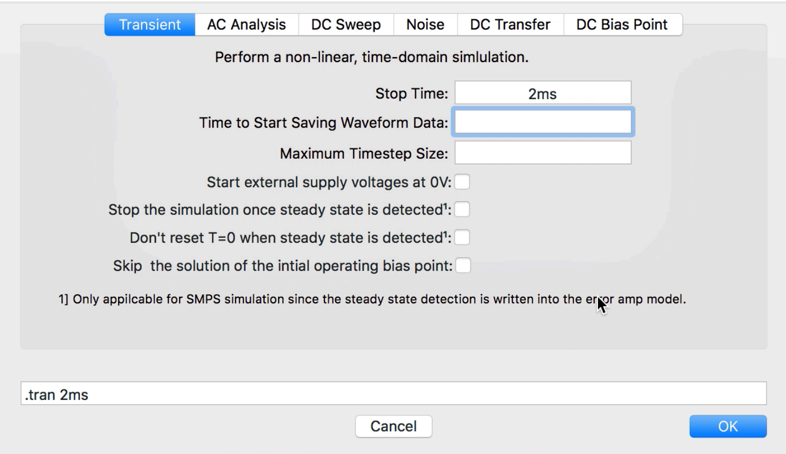


Configure the voltage source

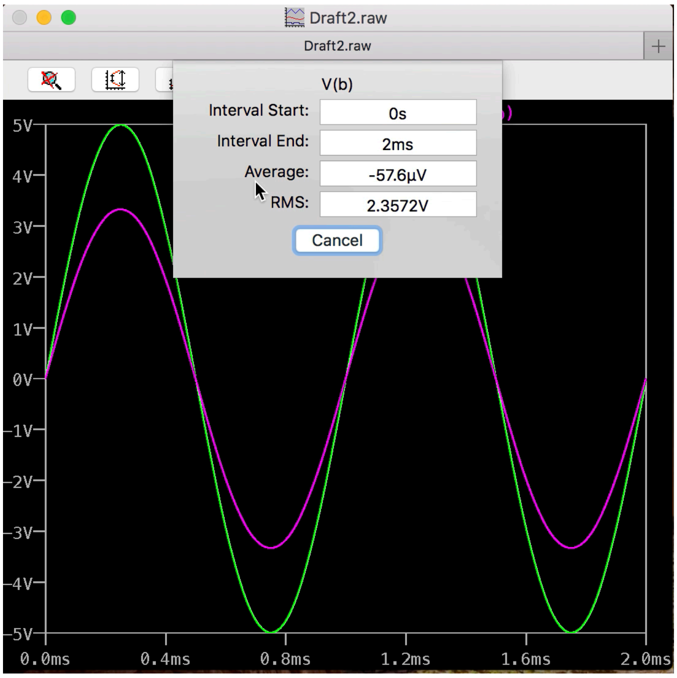


Configure the voltage source

the picture.
Now, we have to configure the new directive. As previously, select Analysis Cmd. Then, choose the directive Transient (.tran). With this, we can observe the circuit during a period of time. Ours will be between 0ms and 2ms (cf. picture).



.tran

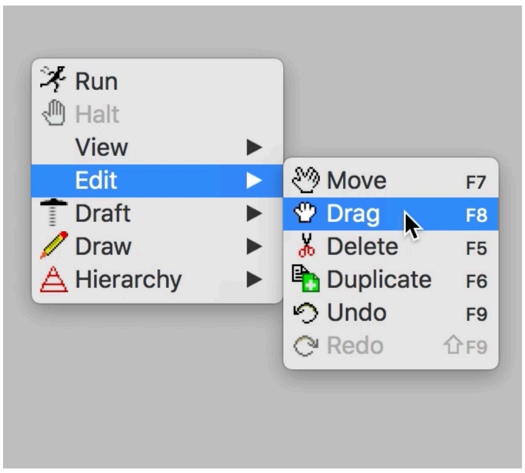


Plot pane with .tran

Then we can run the simulation. The x-axis of the plot pane is the time. We can see that the voltage divider is still valid. To have the Average and RMS value, press ctrl+left-click on the measurement name.

1.5 Extra notes

To move, drag, delete… , right-click → Edit,



.edit

Don't forget to use the LTSpice help if you don't know or don't remember how to use something with LTSpice.

Professor : Farhad Rachidi Video and PDF : Aviolat Florent
* Original version in french, Video : Aviolat Florent / PDF : Herold Gaetan