

Overview of Falstad

Falstad Circuit Simulator (developed by Paul Falstad and Iain Sharp) is a JavaScript-based circuit simulator which allows for circuit parameters (such as RLC characteristics and source parameters) to be adjusted during simulation by moving sliders.

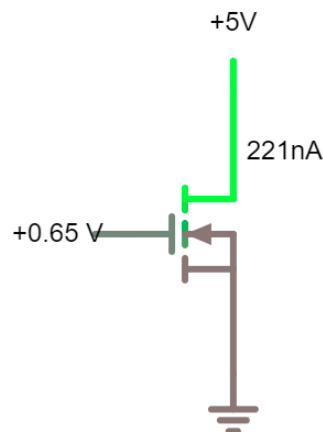
NOTE 1: The netlists read by Falstad are not interchangeable with SPICE netlists.

NOTE 2: Falstad is under the GNU Public License.

MOSFET Example

Creating the Circuit

As an example, consider the following MOSFET circuit:

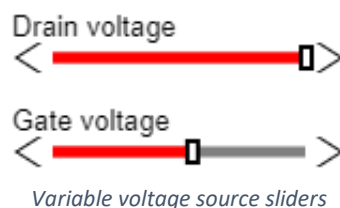


MOSFET circuit

This circuit consists of an n-channel MOSFET, two variable voltage sources and a ground node. The n-channel MOSFET can be found in **Draw => Active Components** and the variable voltage sources and ground can be found in **Draw => Inputs and Sources**. Simply select a component from the **Draw** menu and then “stretch” it on to the page.

Creating and Adjusting Sliders

Once the variable voltage sources are placed, you will see that sliders corresponding to them have appeared on the panel to the right side of the window:



Variable voltage source sliders

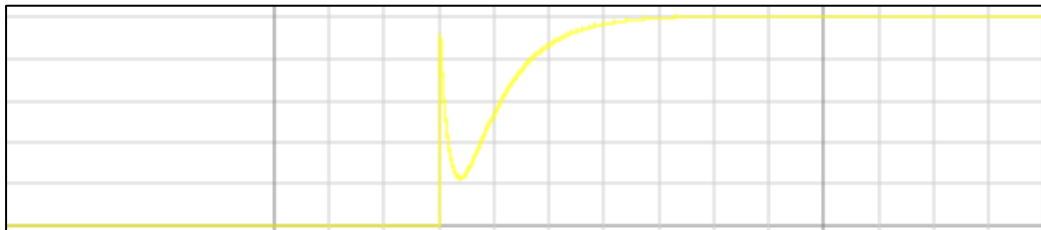
The slider label and voltage range for each voltage source can be adjusted by double-clicking on the source in question (or right-click => edit).

Not every component will automatically add sliders to this panel, but many components have characteristics that can be changed by manually adding sliders to this panel. To add a slider to the slider panel, right-click on a component (e.g., a resistor) and select **Sliders...**

Visualising Electrical Properties of a Circuit

To visualise the voltage across or current through a net or component, right-click on that net or component and select **View in Scope**. This will place a scope at the bottom of the window on which

voltage and current can be plotted. Cursors can be used to find the value of each characteristic at a given time:

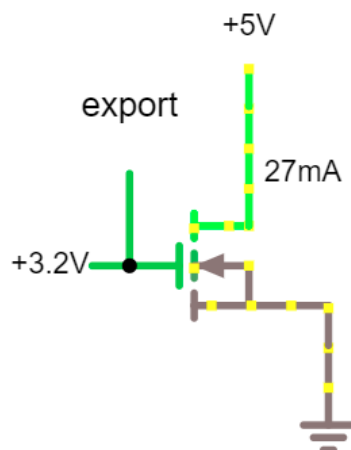


Current through a switch before and after it is closed. Note the bouncing behaviour

While this is useful to visualise the behaviour of a circuit, it is often more useful to create a text file containing the voltage of a net at each time step. To do this, we can export voltage data.

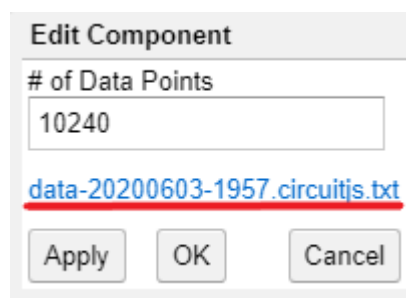
Exporting Voltage Data

1. Select **Draw => Outputs and Labels => Add Data Export**
2. Connect the Data Export net to the net of interest:



Export Data net connected to gate of MOSFET

3. Run the simulation for a period of time
4. Stop simulation
5. Double-click on the Data Export net
6. Click on the blue file download link:



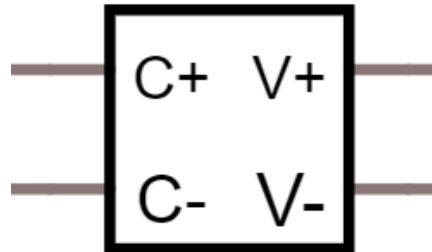
"Edit Component" window for Data Export net

7. Save the file as a .txt file
8. Open the file using a text editor. You should see line-separated values

Exporting Current Data

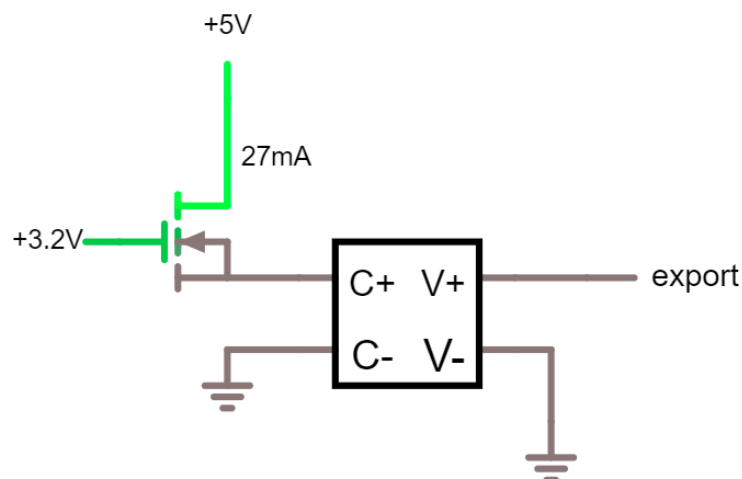
In practice, only voltage data can be exported. Current can, however, be directly converted to voltage by using a current-controlled voltage source (CCVS)

1. Place a CCVS (**Draw => Active Building Blocks => Add Current-Controlled Voltage Source**):



Current-Controlled Voltage Source

2. Double-click on the CCVS and change the output function to "i"
3. Connect the current port in series with the net of interest
4. Connect a Data Export net to the V+ terminal and a ground node to the V- terminal:



MOSFET circuit with CCVS for current data export

5. As before, double-click on the Data Export net and download the data file. The data in this file should be equal to the current through the net of interest at each time interval.

Useful Links

- <https://www.falstad.com/circuit/> - Online circuit simulator
- <https://www.falstad.com/circuit/directions.html> - Official directions
- https://www.bait-consulting.com/publications/circuit_simulator_manual.pdf - Manual by Kevin Berisso