DIY detector simulation guide.

August 1, 2022

1 Draw the schematic

1.1 Drawing the schematic

Before starting to build the detector and solder component on the PCB you will draw the schematic and simulate it with LTspice. LTspice is an analog electronic circuit simulator produced by Analog Devices. Download and install it. For Linux users, simply download the windows version and install with Wine.
All the information given bellow are based on LTspice running on Ubuntu. If you are on Windows or mac it may differ.

1.1.1 Detector circuit

Start by drawing the detector schematic from (careful: electron detector / 4-diode version! read schematic thoroughly) DIY particle detector schematic v1-2.pdf .From the menu you can find common component such as resistor, capacitor, diode ... To find more component click on the Component button.

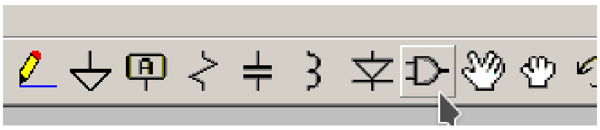


Figure 1.1: Open the component library.

From there you can search for voltage or current sources but also operational amplifier. The opamp used for this project does not appear in the library. The next section explains how to add the model.

1.1.2 Add model

The detector is built around the dual operational amplifier TLE2072 which is not initially available in the LTspice library which only includes Linear or very common components. Almost every component on the market have a SPICE model that can be found on the website of the manufacturer.
Search and download the SPICE model of the TLE2072. The archive contains two file, TLE2072.301 and TLE2072.302. Open one of them with LTspice.
Select the component name, right click and choose Create Symbol as shown in fig. 1.2 To add the TLE2072 to the circuit, open the Component library and search for it. As you can see, it does not look like the usual op-amp because the model was only added but it was not associated to an LTspice intrinsic symbol. It also has only 5 pins that correspond to one half of the dual op amp. For the detector circuit the symbol needs to be added twice. The file is

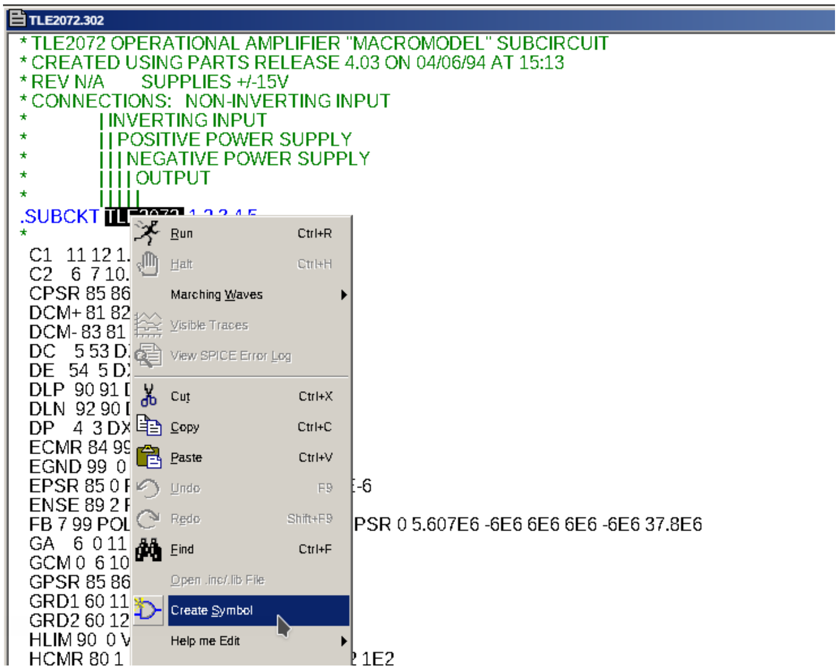
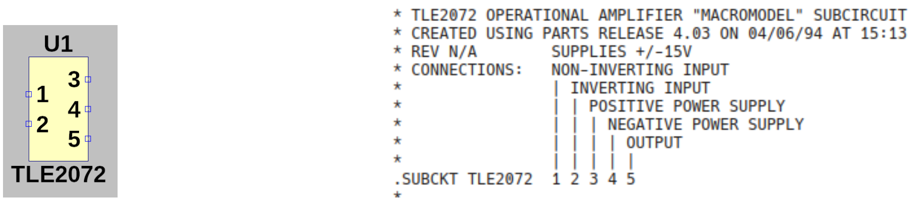


Figure 1.2: Create Symbol.



(a) (b)

Figure 1.3: TLE2072 symbol and pin assignment

not shown properly on LTspice, therefore a clearer view is shown in fig. 1.3a. From the pin assignment you can now draw the schematic.

1.1.3 Use opamp symbol

You can now decide to keep using the imported model as it is or, for clear design you can give the standard op amp symbol to the model.
To do so, click on the SPICE directive button as shown bellow. Type in the following command:

Figure 1.4: Create Symbol.

```
.include /model_location/TLE2072.302
```

From the component library search for opamp2 which is just the symbol of an op amp but can

not be simulated.
Right click on the added symbol and change the field Value for the name of the model as it appears in fig. 1.2.

1.1.4 Final schematic

Finish drawing the schematic. In order to keep it clean you can use labels such as vcc or test1 and test2, as shown in fig. 1.5.

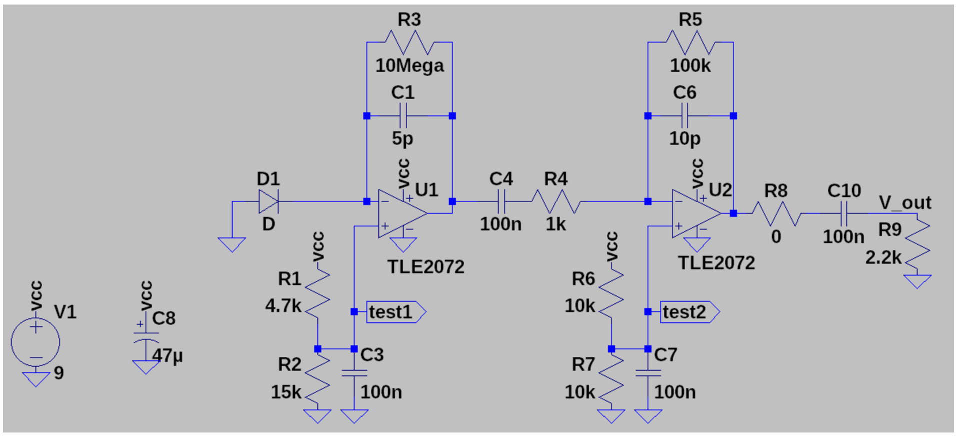


Figure 1.5: Full schematic.

Calculate the test1 and test2 voltages (hint: voltage divider), in the next section you will be able to read those values by simulating the circuit.

2 Simulation

2.1 Simulate circuit

There are various ways of simulating a circuit with LTspice depending on the needs of the designer. For this project we will focus only on the transient, or time-domain simulation of the circuit. From the menu select Simulate → Edit Simulation Cmd as shown in fig. 2.1.

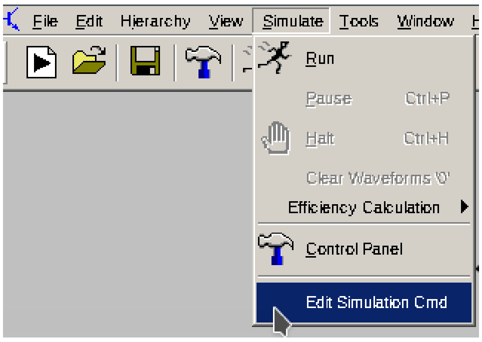


Figure 2.1: Open simulation settings.

In the edit simulation window there are various simulation types but for this project, select Transient which performs a time domain analysis of the circuit. AC analysis or DC sweep can be used to vary a source frequency or voltage for small or large signal analysis.

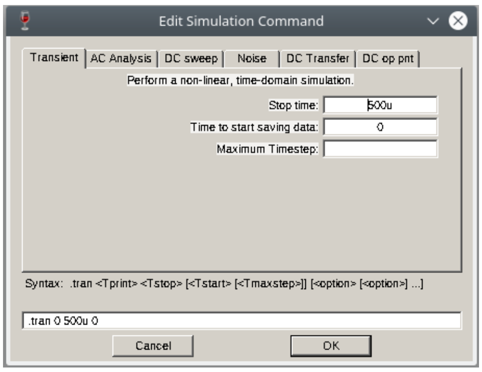


Figure 2.2: Edit Simulation Command.

Choose a desired stop time according to the circuit you are simulating. Considering the length of pulse generated by the detector, 500 μs is suited.
Leave the start time at 0. For longer simulation or more complicated circuit the maximum time-step can be changed. With the current settings it can be left blank. Click OK.

The simulation command is now shown on the schematic. Here `.tran 0 500u 0`, which corresponds to the parameters previously chosen.

To run the simulation, from the menu, select Simulate → Run as shown in fig. 2.3

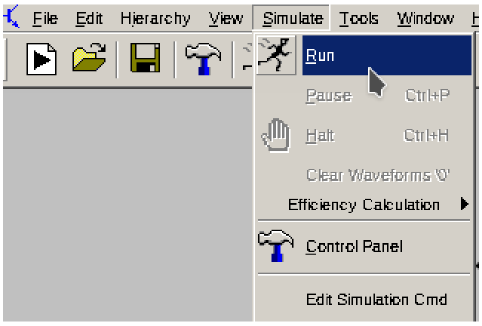


Figure 2.3: Run simulation.

If there is no error in the circuit design, the simulation is performed and a new window opens. The abscissa shows the time from 0 to 500 μs.
To visualize a node on your circuit and move the mouse to the point you want to visualize. The cursor then appears as an oscilloscope probe.
The time analysis of the selected node appears on the waveform viewer.
By clicking somewhere else on the circuit the waveform will be added. Double click will show the selected node and remove all the others from the viewer.
To ease visualization, additional pane can be added. Right click on the viewer and select Add Plot Pane.
Initially only voltages are shown but you can also choose to see the currents. Right click and select Add Traces and choose the current passing trough the component you want to see.

Select the node test1 and test2 introduced in section 1.1.4 and make sure you read the voltages you calculated. Once the circuit will be built you will also make the measurement.

2.1.1 Generate an input

You have now drawn the entire circuit of the detector with LTspice and were able to measure voltages at specific nodes with simulation and directly on your PCB.
To investigate the response of the entire circuit, the signal from the diodes must be emulated. The input will be a charge pulse. **Think of a possibility** how to implement that before flipping to the next page. Where does the charge pulse have to be applied in the circuit?
Hint: Voltage signals (like pulse generators producing e.g. step/box/saw-tooth/triangle/sinus functions) can easily be included. How could a charge pulse be generated from that?

Answer: A voltage step (box function) and a capacitor in series connecting to the input node (inverting input of first stage) generate a pulse of defined charge. Next: which charge is required?

Expected signal (calculate!)

What is the expected signal for a MIP? You can estimate the thickness of the depletion region based on the bias voltage, and you can assume that a MIP generates between 60 and 80 electron/hole pairs per micron in silicon.
How much charge do you need to inject to approximate one minimum ionizing particle? Adjust the input parameters accordingly.
Hint: calculate the amplification of the first amplifier stage (in U/Q [V/C]) and of the second stage (in U/U [V/V] / unitless).
In reality you will observe voltage pulses of about 200 mV. Which input charge does this correspond to and what kind of voltage step needs to be generated to inject a test charge of the same size via a 10 pF capacitor? Set fitting parameters to the voltage pulse, choose a rise time short compared to the expected pulse length and check if the output looks like the expected pulse (next paragraph).

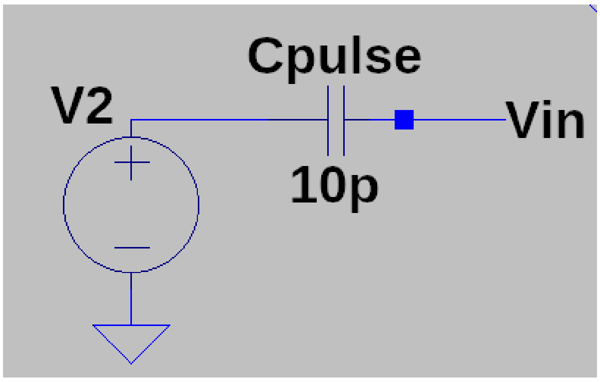


Figure 2.4: Test-charge injection circuit

2.2 Run the full simulation

With the diode signal emulated you can now run the entire simulation. By selecting the output of the detector you should observe similar results as the one presented in fig. 2.5

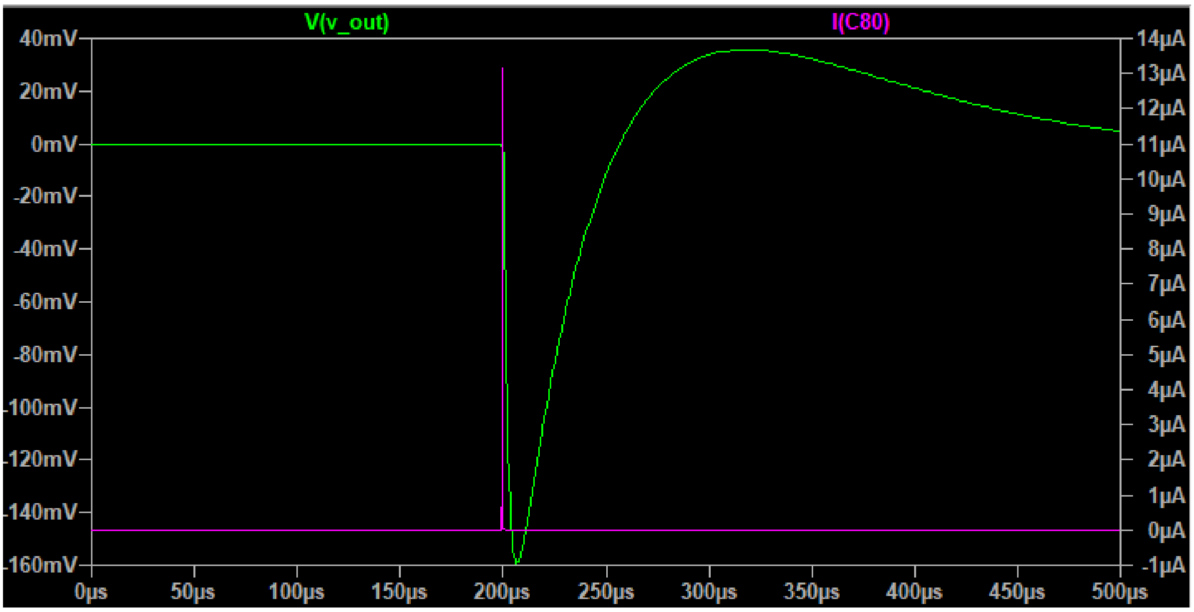


Figure 2.5: Output signal.

Nice, now you are ready to build the detector!