

# LELEC2811 - P2

## Quick start guide for LTSpice

The objectives of P2 are first to install LTSpice, a free Spice simulator for the simulation of electrical circuits (if not done yet). Then, to learn how to model a transducer - in this tutorial a light-dependent resistor (LDR) - based on its datasheet and to integrate it in LTSpice using a library file and a symbol.

### 1 Installation

- Download LTSpice, available both for Windows and Mac at the following link, and install it on your computer: <https://www.analog.com/en/design-center/design-tools-and-calculators/ltspice-simulator.html>
- If you are not familiar with Spice simulators, here is some interesting documentation to help you master this tool:
  - <https://cmosedu.com/cm0s1/ltspice/ltspice.htm>
  - [https://cmosedu.com/videos/ltspice/ltspice\\_videos.htm](https://cmosedu.com/videos/ltspice/ltspice_videos.htm)
  - <https://ecee.colorado.edu/~mathys/ecen1400/pdf/scad3.pdf>

### 2 Component modelling

If you need to use a specific transducer in LTSpice, it is unlikely that it is already part of the default component libraries. To use it, you will thus have to create a model and a symbol. The following example shows you how to create a photoresistor, also called light-dependent resistor, and how to simulate its transfer function from the light domain to the voltage domain using a voltage divider testbench.

The photoresistor considered here is a N5AC-501085 cadmium sulfide (CdS) LDR. Its transduction principle is fairly simple. When the device is kept in the dark, few electrons are present in the conduction band of the CdS semiconductor, hence the large resistance in the order of  $M\Omega$ . When exposed to light, photons excite electrons in the valence band, which consequently jump to the conduction band. This decreases the resistance as there are more free electrons in the CdS semiconductor. A more detailed explanation can be found in [1] and useful information about LDRs can be found in [2].

#### 2.1 Transducer model

Open the library file called "photoresistor.lib", provided to you on Moodle and whose content is shown in Fig. 1.

---

```
1 .SUBCKT photoresistor VP VM
2
3 .PARAM R_dark = 5Meg
4 .PARAM R_10_lux = 100k
5 .PARAM gamma = 0.8
6 .PARAM K = R_10_lux/pwr(10,-gamma)
7
8 .PARAM R_var = K*pwr(LUX,-gamma)
9
10 Rphoto VP VM {R_var}
11 Rdark VP VM {R_dark}
12
13 .ENDS photoresistor
```

---

Figure 1: Spice model of the N5AC-501085 CdS LDR.

This file contains the definition of a sub-circuit called "photoresistor", whose content is delimited by the `.SUBCKT` and `.ENDS` directives. This sub-circuit has two pins called VP and VM. It is composed of a main resistance  $R_{photo}$ , dependent on the illuminance expressed in lux and connected between VP and VM, in parallel with a fixed parasitic resistance  $R_{dark}$ , corresponding to the dark resistance. The values of this model are based on the datasheet of the N5AC-501085 CdS LDR available on Moodle. The model chosen for the light-dependent resistance is the following:

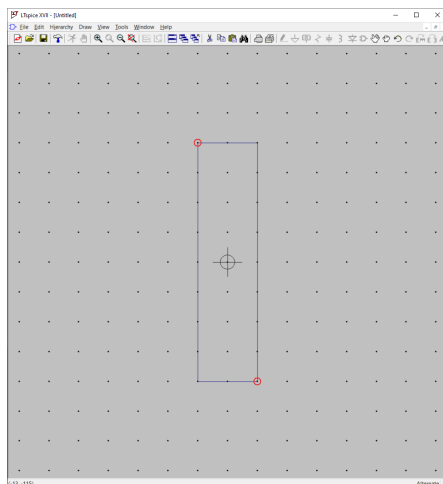
$$R(I) = R_0 I^{-\gamma} \quad (1)$$

where  $R$  denotes the light-dependent resistance in  $\Omega$ ,  $R_0$  a scale factor in  $\Omega \times \text{lx}^\gamma$ ,  $I$  the illuminance in lx and  $\gamma$  the slope of the resistance as a function of the illuminance in a log-log scale.

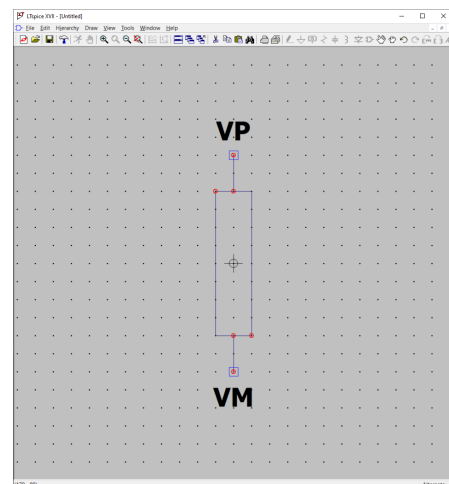
## 2.2 Testbench

**Create a symbol for the model** There are two possibilities to create a symbol for your new component. Either to generate it automatically from the `.lib` file or to draw it yourself. In this tutorial, we choose to do the latter. To create the symbol, the following steps must be followed.

- In LTSpice, open the "File" tab and click on "New Symbol". An empty window to create your symbol will open.
- Draw a rectangle to represent your component, as shown in Fig. 2a, either by going into the "Draw" tab and clicking on "Rect 'R'", or by using the shortcut 'R'.
- Add the pins VP and VM either by going into the "Edit" tab and then "Add Pin/Port 'P'", or by using the shortcut 'P'. Pay attention that the "Netlist Order" field must agree with your model, i.e. VP has netlist order 1 and VM, netlist order 2. Connect them to your rectangle by drawing some lines. After doing that, you should obtain something similar to Fig. 2b.



(a)



(b)

Figure 2: Creation of the symbol (a) A rectangle shape represents the photoresistor. (b) Pins are added for VP and VM.

- Save the symbol as `photoresistor.asy` in your working directory.
- The symbol must now be linked to the model. To do so, open the "Edit" tab and go to "Attributes -> Edit Attributes" or "Ctrl+A". In the Symbol Attribute Editor, the "Prefix" attribute must be changed to "x", because the name of a sub-circuit has to start with "x", and the "SpiceModel" attribute must be changed to the name of your sub-circuit, in this case "photoresistor". The filled window should look like Fig. 3.

**Create the testbench schematic** To convert the resistance value into a voltage, the LDR is interfaced with a simple voltage divider. It is then possible to simulate the transfer function from the light domain to the voltage domain by reproducing the circuit depicted in Fig. 4a. To include the component you have just created, you have to respect the following points:

- When creating your schematic, make sure to save it in your working directory before adding the custom component. It is easier to use the **same working directory** for the model (`.lib` file), the symbol (`.asy` file) and the testbench schematic (`.asc` file).

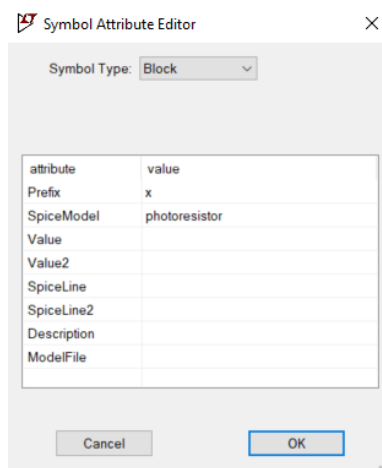
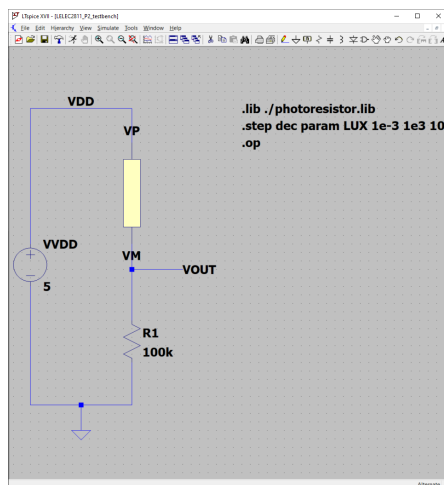
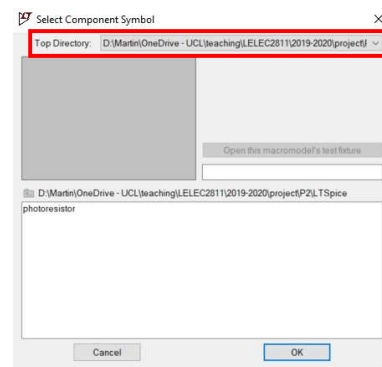


Figure 3

- To include the LDR model, the `.lib` Spice directive is used, as shown in Fig. 4a. In some cases, it is possible that LTSpice is not able to find your library file, likely due to a problem of path. In such case, just copy-paste the content of the `.lib` file directly inside the testbench.
- To add your custom component, click on the "Component" icon in the toolbar and change the "Top Directory" to your working directory, as depicted in Fig. 4b. You should now be able to add the "photoresistor" symbol to your schematic. To do so, make sure that you have saved the schematic to your working directory!



(a)



(b)

Figure 4: Creation of the testbench schematic (a) Testbench schematic. (b) Addition of the photoresistor.

Finally, in Fig. 4a, two Spice directives are used to define the analysis to be performed. The `.op` directive computes the (DC) operating point of the circuit and the `.step dec` directive sweeps the parameter `LUX` logarithmically, from 1 mlx to 1 klx with 10 points per decade.

### 2.3 Results analysis

Based on your testbench, you should be able to answer the following questions:

- Why is the transfer function non-linear? To answer this question, you might find helpful to write  $V_{OUT}$  as a function of the illuminance  $I$ , based on the model given in Eq. (1).
- Based on the same voltage divider configuration, is it possible to make the transfer function more linear by changing one of the circuit parameters? If yes, is there a drawback to it?

### 3 Parameterized components

From now on, you used only pre-made models for your components, but of course you can also create them by yourself. If the model is very simple, you can use existing components and make them vary with a given parameter.

Metal detectors are based on the changing of inductance of a coil depending on the proximity with metallic elements. We will create an inductor whose value depends on a given parameter  $d$ , representing the position of a metal object on the axis of the coil.

$d$	-0.2	-0.18	-0.16	-0.14	-0.12	-0.1	-0.08	-0.06	-0.04	-0.02	0	0.02	0.04	0.06	0.08	0.1	0.12	0.14	0.16	0.18	0.2
$L$	0.63	0.64	0.65	0.68	0.71	0.76	0.82	0.88	0.92	0.95	0.96	0.95	0.92	0.88	0.82	0.76	0.71	0.68	0.65	0.64	0.63

Figure 5: Inductance value  $L$  for different position of metallic object on the axial axis of the inductor.  $L$  values were obtained by CST simulations [3].

Least Square Regression is performed on data from Figure 5 to obtain analytical expression of the inductance variation with  $d$ . Regression is performed on a polynomial and a gaussian expression :

#### Polynomial

$$L(d) = L_0.(a_0 + a_1.d + a_2.d^2 + a_3.d^3 + a_4.d^4) \quad (2)$$

with  $(a_0, a_1, a_2, a_3, a_4) = (0.99452, -1.3469e^{-15}, -20.0989, 8.4902e^{-14}, 311.1909)$

#### Gaussian

$$L(d) = L_0.(a_0.e^{-\left(\frac{d-a_1}{a_2}\right)^2} + a_3) \quad (3)$$

with  $(a_0, a_1, a_2, a_3) = (0.3434, -0.5605e^{-12}, 0.105, 0.62)$

For both expressions,  $L_0 = 1e^{-3}$  [H].

To perform the regression, you can use MATLAB's 'polyfit' and 'polyval' functions for polynomial regression. For the gaussian, you can use the 'fit' function with the gaussian model 'gauss1'. You will notice that the gaussian fitting does not perform as intended, can you identify the problem and correct it ?

The 'polyfit' and 'polyval' also exist in python. It is possible to fit the gaussian without regression, by extracting the parameters of the distribution such as mean and standard deviation and use the 'curve\_fit' function.

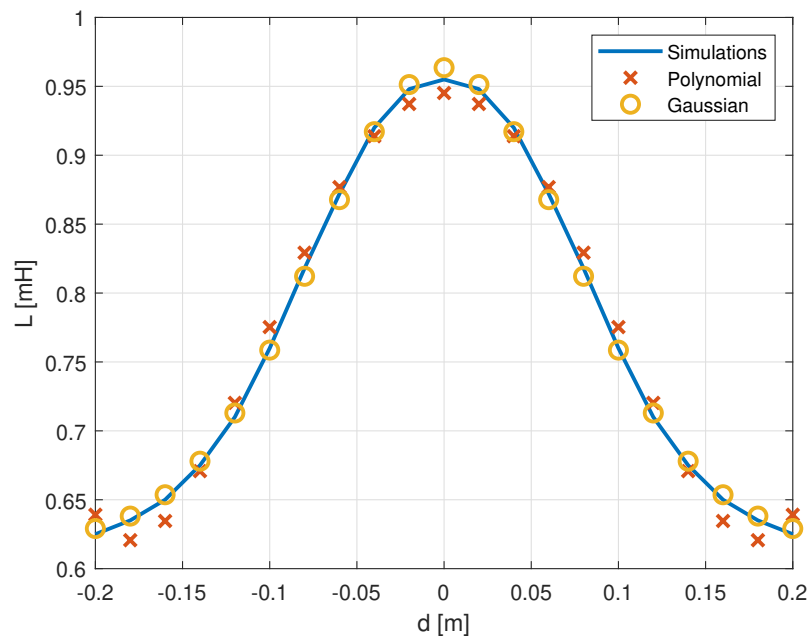


Figure 6: Representation of data from Figure 5 (Simulations), plus comparison with Polynomial and Gaussian Least-Square regression from equations 2 (Polynomial) and 3 (Gaussian).

To simulate this inductance variation in LTSpice, instead of creating a model like for the photoresistor, the parametric equations 2 and 3, can be directly used to set the value of an inductor. For this, the inductor value is replaced by the expression between brackets, see Figure 7. The schematic consists in a simple AC source connected to the inductor. The ".ac lin 100 1k 4k" performs a frequency sweep between 1k and 4k Hz with 100 steps for the AC analysis and the ".step param d -0.2 0.2 0.01" create the sweep on d variable, between -0.2 and 0.2 with a step size of 0.01.

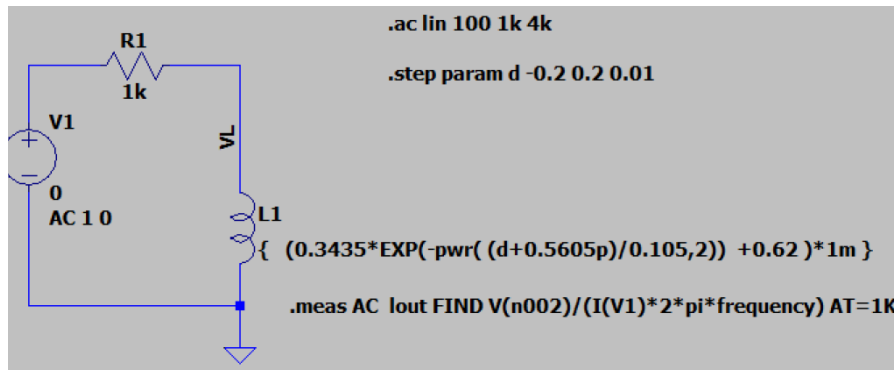


Figure 7: Spice schematic to simulate the varying inductor.

The inductance value can be extracted by probing the voltage and current at the inductor level. With the VL label on the schematic, the voltage at the inductor level is "V(vl)" and the current through the L1 inductor is "I(L1)", the inductance value is :  $V(vl)/(I(L1) * 2 * \pi * frequency)$ . To plot the result of this equation, go to right click on the plot, go to "add traces" and enter this equation in the "Expression(s) to add :" field.

### 3.1 export data in log file

You might find useful to export some data instead of plotting them directly in LTSpice. Especially in this case as it is a plot of L with frequency as a function of d. To export data and plot them by yourself, the .meas command can be added, as in Figure 7. "AC" refers to the AC analysis, "lout" is the name of the variable that will be exported, "FIND Y" followed by "AT X" is a statement that returns the value of Y expression at the X abscissa. In this case, the value of our equation at 1 KHz is exported. Any frequency is suitable as L does not vary with frequency, 1 KHz is arbitrary here. More complex statement can be used. More information to use .meas : [https://ltwiki.org/LTspiceHelp/LTspiceHelp/\\_MEASURE\\_Evaluate\\_User\\_Defined\\_Electrical\\_Quantities.htm](https://ltwiki.org/LTspiceHelp/LTspiceHelp/_MEASURE_Evaluate_User_Defined_Electrical_Quantities.htm)

The .log file containing the data will be in your LTSpice folder, with the same name than the Spice simulation file. You can therefore plot L as a function of d and verify that the simulated inductance takes the required values.

1) The exported data will be in dB, what transformation would you do to obtain the linear data and how (and where) would you implement it ?

2) is it possible to measure the inductance value differently ?

## References

- [1] EEPower, "Photoresistor". [Online]. Available: <http://www.resistorguide.com/photoresistor/>. [Accessed Oct. 16, 2019].
- [2] AdaFruit, "CdS Cells, Photoresistors and Light Dependent Resistors (LDR)". [Online]. Available: <https://learn.adafruit.com/photocells/overview>. [Accessed Oct. 16, 2019].
- [3] Zhu Yuanzhe, Chen Baichao, Luo Yao, Zhu Runhang (2019), *Inductance calculations for coils with an iron core of arbitrary axial position*, Electromagnetics, 39, 1-21, DOI:[10.1080/02726343.2019.1577426](https://doi.org/10.1080/02726343.2019.1577426).