UNIVERSITATEA POLITEHNICA DIN BUCUREŞTI

ŞCOALA DOCTORALĂ DE INGINERIE ELECTRICĂ

DOCTORAL THESIS SUMMARY

- summary -

CONTRIBUȚII PRIVIND SIMULAREA CIRCUITELOR NELINIARE COMPLEXE

CONTRIBUTIONS IN THE SIMULATION OF COMPLEX NONLINIAR CIRCUITS

Scientific Coordinator:

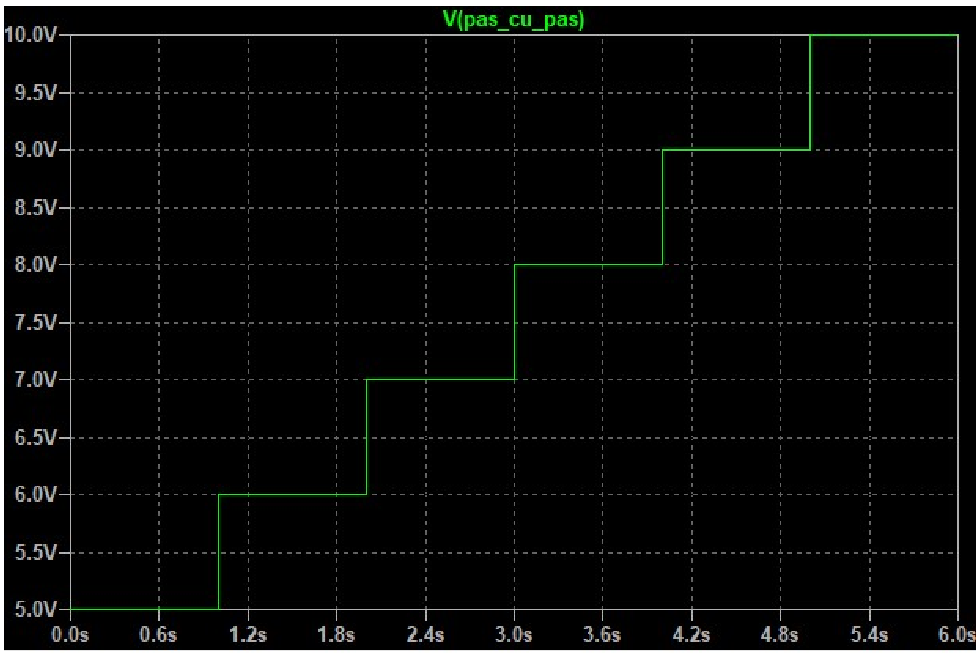
Prof. Dr. Eng. Mihai IORDACHE

PhD Student:

Eng. Marius-Florin STĂNILOIU

BUCUREŞTI

2023



Don't forget, increase every day, step by step.
@Marius Florin Staniloiu

Table of content

1 INTRODUCTION

1.1 Formulating the problem

SPICE is the most used program to simulate electronic components. Currently, almost all component manufacturers offer SPICE models for their products. SPICE is simple to use and quick to learn. Another advantage of this tool is that it is easy to create your own SPICE model when it is not available in the standard library or from the component manufacturer. But in addition to numerous advantages there are also disadvantages created especially by SPICE models. One disadvantage is the orientation of the simulation towards the nominal operation of the component, another disadvantage is the accuracy of the SPICE model depending on the program in which it runs or perhaps the limitations of the respective program (generally the operation of the program is not visible to us). In my experience, I have noticed that a SPICE model can perform differently depending on the simulation program used. A personal opinion would be that a simulation program using SPICE is better (operates closer to reality or catalog data) the higher the price.

1.2 Structure and content of the thesis

The doctoral thesis is structured in 6 chapters as follows:

In Chapter 1 – Introduction, general aspects about the idea of this thesis are presented. Where did the author's desire for this work come from, the objectives of this work, the structure of the thesis and the works published during the years of preparation for the doctorate.

In Chapter 2 – The current state of simulation in SPICE is oriented towards the current description of simulation in SPICE. Advantages and disadvantages of using simulation in SPICE.

In Chapter 3 – Contributions regarding the simulation of capacitors, coils and non-linear resistors in SPICE, new and high-performance SPICE models for the most used electronic components (capacitor, coil, resistor) are presented. These models are compared with already existing SPICE models and with the real behavior in various circuits of the studied component.

In Chapter 4 - Contributions regarding the simulation of diodes and transistors in SPICE, new equivalent models of diodes and transistors are presented, after which new and high-performance SPICE models are created for the most used electronic components (rectifier diode, stabilizer diode, bipolar transistor and MOS-FET transistor). These models are compared with already existing SPICE models and with the real behavior in various circuits of the studied component.

Chapter 5 - Conclusions presents the original contributions on the analysis of complex nonlinear circuits that can be analyzed in the SPICE programming environment and some future research directions.

In Chapter 6 – Bibliography.

1.3. Dissemination of results

There are 8 works published during the approximately 5 years of doctoral training, of which 4 as first author and 4 as co-author, as follows:

1. **Marius Florin Staniloiu**, Horatiu Samir Popescu, Bogdan Glod, Mihai Iordache, "*SPICE model of a real capacitor: Capacitive feature analysis with voltage variation*" (EPE2020), Iași ROMÂNIA, Date of Conferences: 22-23 October 2020, Iași România, Added to IEEE Xplore: 18 February 2021, DOI: 10.1109/EPE50722.2020.9305554, **INSPEC Accession Number:** 20470036, 978-1-7281-8126-4/20/$31.00 ©2020 European Union, Publisher: IEEE, pp. 333 – 338.
2. **Marius Florin Staniloiu**, Horatiu Samir Popescu, Bogdan Glod, Mihai Iordache, "*SPICE model of a Real Coil. Inductance feature analysis with current variation*" (EPE2020), Iași ROMÂNIA, Date of Conferences: 22-23 October 2020, Iași România, Added to IEEE Xplore: 18 February 2021, DOI: 10.1109/EPE50722.2020.9305677, **INSPEC Accession Number:** 20470072, 978-1-7281-8126-4/20/$31.00 ©2020 European Union, Publisher: IEEE, pp. 442 – 446.
3. Mihai Iordache, Horatiu Samir Popescu, Ionela Vlad, **Marius Florin Staniloiu**, "*ACAP – Analogic Circuit Analysis Program*" (Bucuresti 2021), Date of Conferences: 25-27 March 2021, 12th International Symposium on Advanced Topics in Electrical Engineering (ATEE), Added to IEEE Xplore: 12 May 2021, DOI: 10.1109/ATEE52255.2021.9425307, **INSPEC Accession Number:** 20691709, ISBN: 978-1-6654-1878-2/21/$31.00 ©2021 IEEE, **WOS:000676164800143,** Publisher: IEEE, 6 pages.
4. **Marius Florin Staniloiu**, Horatiu Samir Popescu, Georgiana Rezmerita, Mihai Iordache "*The Equivalent Circuits Thevenin and Norton*", Scientific Bulletin of the Electrical Engineering Faculty – Year 21 No.2 (45), Sciendo, ISSN 2286-2455, DOI: 10.2478/sbeef-2021-0021, pp. 40-48.
5. Victor Bucata, Mihai Iordache, Ionela Vlad, Horatiu Popescu, **Marius Florin Staniloiu** "*Wireless Power Transfer Systems: Thevenin Equivalent Circuits for Parallel-Series and Paralle-Parallel Magnetic Resonator Configurations*" (ICATE 2021), Craiova ROMÂNIA, Date of Conferences: 27-29 May 2021, Added to IEEE Xplore: 28 June 2021, DOI: 10.1109/ICATE49685.2021.9464974, **INSPEC Accession Number:** 20780269, 978-1-7281-8035-9/21/$31.00 ©2021 IEEE, **INSPEC Accession Number:** 20895674, Publisher: IEEE, 6 pages.
6. Victor Bucata, Mihai Iordache, Ionela Vlad, Horatiu Popescu, **Marius Florin Staniloiu** "*Thevenin Equivalent Circuits for Magnetic Coupling Rezonators (Series-Series, Series-Parallel) în Wireless Power Transfer System*" (ICATE 2021), Craiova ROMÂNIA, Date of Conferences: 27-29 May 2021, Added to IEEE Xplore: 28 June 2021, DOI: 10.1109/ICATE49685.2021.9464933, **INSPEC Accession Number:** 20895674, 978-1-7281-8035-9/21/$31.00 c2021 IEEE, **INSPEC Accession Number:** 20895674, Publisher: IEEE, 6 pages.
7. Mihaela Grib, Mihai Iordache, Alexandru Radu Grib, Horatiu Popescu, Ovidiu Laudatu, **Marius Staniloiu** "*The Use of Thevenin, Norton and Hybrid Equivalent Circuits în The Analysis and Polarization of Nonliniar Analog Circuits*" (EPE 2022), Iași România, Date of Conferences: 20-22 October 2022, Added to IEEE Xplore: 25 November 2022, DOI: 10.1109/EPE56121.2022.9959871, **INSPEC Accession Number:** 22330770, 978-1-6654-

8994-2/22/$31.00 ©2022 European Union, **WOS:000709089900011.** Publisher: IEEE, pp. 198-207.

8. **Marius Florin Staniloiu**, Horatiu Samir Popescu, Georgiana Rezmerita, Ionela Vlad, Mihai Iordache, "*SPICE model of a real Zener diode tested at room temperature*" (EPE2022), Iași România, Date of Conferences: 20-22 October 2022, Added to IEEE Xplore: 25 November 2022, DOI: 10.1109/EPE56121.2022.9959871, **INSPEC Accession Number:** 22330715, 978-1-6654-8994-2/22/$31.00 ©2022 European Union, Publisher: IEEE, **WOS:000709089900001,** pp. 182-186.

WOS – **W**orld **O**f **S**cience

2 CURRENT STATE OF SIMULATION IN SPICE

2.1 History of SPICE

Today, SPICE is a program that runs on a computer, it is designed to simulate analog electronic circuits. But it was originally designed to simulate integrated circuits, hence its name: "Simulation Program with Integrated Circuit Emphasis" - simulation program for integrated electronic circuits.

A first version was released into the public domain in May 1972. An improved version was later released in 1975.

A third version appeared in March 1985 which was a major improvement of SPICE (this was also available in the public domain). This was written in C and not FORTRAN (like the other two versions), the third version incorporated additional types of transistors (E.x.: MOSFET) and switching elements.

2.2 Fundamentals of SPICE Programming

Using a graphical interface, the user is allowed to draw a circuit diagram and then have the computer analyze that circuit, with the results displayed graphically.

This is a very fast and valuable analysis tool, but it is not perfect and has its shortcomings. First, it and other graphics programs like it tend to be unreliable when analyzing complex circuits, because the translation from image to computer code isn't quite the science we'd like it to be (or at least not yet). Second, due to its graphics requirements, it sometimes tends to need a significant amount of computing power to run and a computer operating system that supports graphics. Third, these graphics programs can be expensive.

Despite all these weaknesses, underneath the graphics lies a robust (and not least free) program called SPICE, which analyzes a circuit based on a text file description of the circuit's components and connections.

2.3 Advantages and disadvantages of using SPICE simulation

The advantages and disadvantages of using simulation in SPICE have been observed over a period of about 20 years of use. Using numerous programs based on SPICE simulation.
Benefits:

- fast simulation.
- large number of SPICE models available directly from electronic component developers.
- easy-to-use tools are already on the market.
- free availability of simulation tools with graphical interface.

Disadvantages:

- malfunction of the models depending on the temperature.
- many SPICE models do not take temperature into account.
- SPICE models focus on the nominal operation of components being more of a tool to see how an electronic circuit works. In most cases, the operation of electronic circuits must be found out towards the limits.
- linear components (resistor, capacitor, coil, ...) are considered as if they had 100% ideal operation.

3 CONTRIBUTIONS TO THE SIMULATION OF NONLINIAR CAPACITORS, COILS AND RESISTORS IN THE SPICE

Taking into account the disadvantages discovered over time in simulating circuits using SPICE, the author creates new SPICE models starting from existing models and the operation declared in the catalog data of electronic components. The result of the study was the development of new high-performance models that reflect the functioning of electronic components closer to reality. So after creating a new model, it will be tested with the existing model in the SPICE database and with the behavior of the component in real circuits.

3.1 SPICE model of a real capacitor

In this chapter, starting from the equivalent circuit of a real capacitor, a SPICE model is created in which the static capacitance varies with the voltage at the capacitor's terminals, with the temperature of the ambient environment in which the capacitor operates, and with its aging. Ideal voltage controlled current sources and ideal voltage sources are used in creating the SPICE model.

3.1.1 The equivalent circuit of a real capacitor

The equivalent circuit of a real capacitor is shown in figure 3.1.1.

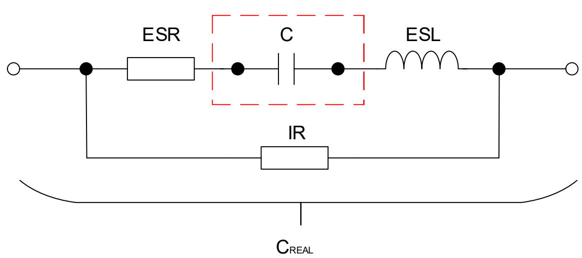


Fig. 3.1.1. The equivalent circuit of a real capacitor.

This circuit has the following components:

ESR - represents a parasitic resistor inserted in series with the capacitance component C, this represents losses due to dielectric substances, electrodes or other components;

ESL - is a parasitic coil inserted in series with the capacitance component C, it is due to the electrodes, cables and other components;

IR - represents a parasitic current drain resistor between the capacitor terminals;

C – is a capacitor that represents the static capacitance of the capacitor, which is dependent on voltage variation, temperature variation, and capacitor aging.

3.1.2 SPICE model of a nonlinear capacitor controlled in voltage and varying with temperature and aging

Given the real capacitor model and the capacitor equations, a SPICE model of the real nonlinear voltage-controlled capacitor will be created. In this model, the variation of static capacitance with voltage, temperature and aging is considered. Also, the variation of impedance and series resistance (ESR) with frequency is taken into account in this model (fig. 3.1.2.).

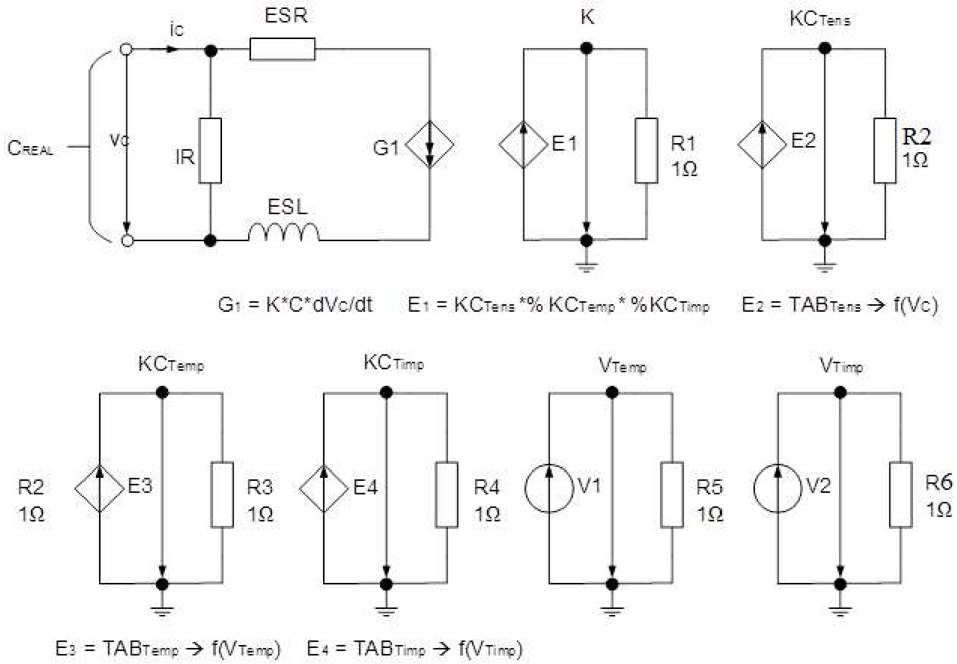


Fig. 3.1.2. SPICE model of a real nonlinear voltage-controlled capacitor.

In figure 3.1.2. G1 is a voltage-controlled current source, using the SPICE property of the source to vary the current output according to an integrated expression:

$$G_1 = KC\frac{dV_C}{dt} \tag{3.1.1}$$

The following parameters are found in this expression: K represents the percentage variation of the static capacity according to voltage, temperature and aging;
C represents the static capacity of the capacitor (the nominal capacity declared by the component manufacturer);
V_C represents the voltage at the capacitor terminals as a function of time;
E1 is a controlled voltage source, the output value depends on the variation of static capacitance with voltage, temperature and aging.

$$\boldsymbol{E_1 = KC_{Tens} \times \%KC_{Temp} \times \%KC_{Timp}} \tag{3.1.2}$$

KC_{Tens} represents the percentage variation of static capacity with voltage variation;
KC_{Temp} represents the percentage variation of static capacity with temperature variation;
KC_{Timp} represents the percentage variation of the static capacity with aging;
E_2 is a controlled voltage source, whose output value depends on the percentage variation of the static capacity declared by the component manufacturer with the voltage, the input of the source depending on the value of the voltage at the input of the capacitor.

$$E_2 = TAB_{Tens} \rightarrow f(V_C) \tag{3.1.3}$$

E_3 is a controlled voltage source, the output value depends on the component manufacturer's declared percentage variation of static capacitance with temperature, the source input depends on the component manufacturer's declared voltage value that varies with temperature (V_1).

$$E_3 = TAB_{Temp} \rightarrow f(V_{Temp}) \tag{3.1.4}$$

E_4 is a controlled voltage source, the output value depends on the component manufacturer's declared percent change in static capacitance with age, the source's input depends on the component manufacturer's declared age-varying voltage value (V_2) .

$$E_4 = TAB_{Timp} \rightarrow f(V_{Timp}) \tag{3.1.5}$$

3.1.3 Comparison of a non-linear capacitor model modeled in SPICE, of a capacitor from the standard library and the real capacitor at 25°C and an operating age of less than 10 hours

To verify the accuracy of the SPICE model a comparison of the SPICE model of the capacitor will be made with the behavior of the actual capacitor and with the behavior of the SPICE capacitor proposed by the component manufacturer for various circuits.

3.1.3.1 Charging and discharging the capacitor

Figure 3.1.3. shows the schematic of a circuit for charging and discharging a capacitor, which was practically realized. The charging and discharging voltage of the capacitor at the V_Real point was measured using an oscilloscope (Tektronix Model DPO 5104B).
The waveform was saved in .CSV format for later use in SPICE.

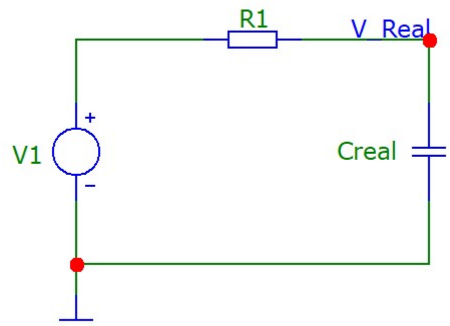


Fig. 3.1.3. Charging and discharging circuit of a practically made capacitor.

In figure 3.1.3. V1 is a step voltage source with a voltage increase from 0V to 10V (TDK-Lambda voltage source).
In figure 3.1.3. Creal is a 10µF capacitor (KEMET, Y5V Dielectric) and R1 a 180kOhm resistor.
Figure 3.1.4. shows a capacitor charging and discharging circuit modeled in SPICE.

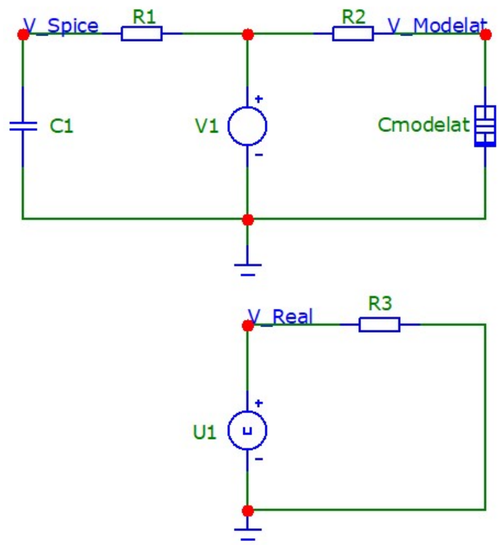


Fig. 3.1.4. Charge and discharge circuit of a capacitor made in SPICE.

V1 is a voltage source with a step-up from 0V to 10V.
C1 is a capacitor from the SPICE library, model proposed by the manufacturer of the 10 μF component.
Cmodelat is a real nonlinear capacitor modeled in SPICE with a variation of static capacitance as a function of voltage.
U1 is a user source from the SPICE library that allows voltage waveform loading in .USR format.
The voltage value is taken with an oscilloscope (Tektronix Model DPO 5104B), with which the charging and discharging of a real capacitor was measured.
Following the simulation of the circuit in figure 3.1.4 using SPICE, the graph in figure 3.1.5 was obtained, when the capacitors are charged and the graph in figure 3.1.6. when the capacitors discharge. R1, R2 and R3 are resistors from the standard SPICE library of 180kΩ.

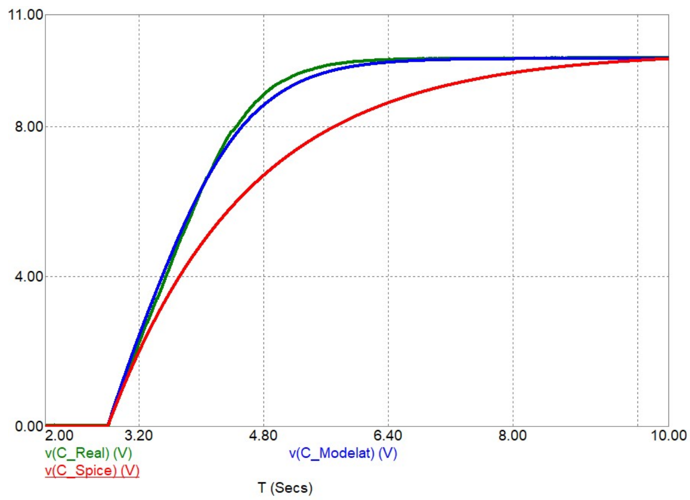


Fig. 3.1.5. Time variation of the charge of a real capacitor compared to a model modeled in SPICE and a SPICE model.

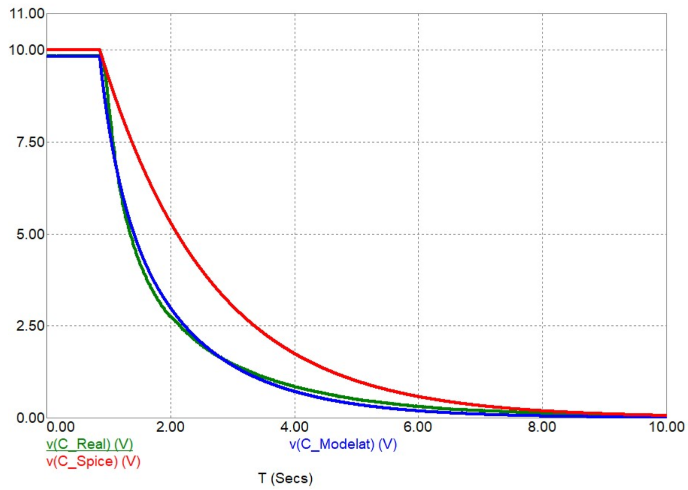


Fig. 3.1.6. Time variation of the discharge of a real capacitor compared to a model modeled in SPICE and a SPICE model.

The red color graph indicates the voltage variation on the SPICE capacitor, the blue color graph indicates the voltage variation on the nonlinear capacitor modeled in SPICE and the green color graph indicates the voltage variation on the real capacitor (measured with the oscilloscope).
Comparing the two graphs, it can be seen that the model of the nonlinear capacitor modeled in SPICE is very close to the behavior of the real capacitor.

3.1.3.2 Oscillating circuit

Figure 3.1.7 shows the scheme of an oscillating circuit consisting of a coil, a resistor and a capacitor, which was practically realized and the variation of the voltage on the capacitor at the V_Real point was measured using an oscilloscope (Tektronix Model DPO 5104B). The waveform was saved in .CSV format for later use in SPICE. This circuit is used to see the variation of static capacitance in an oscillating circuit.

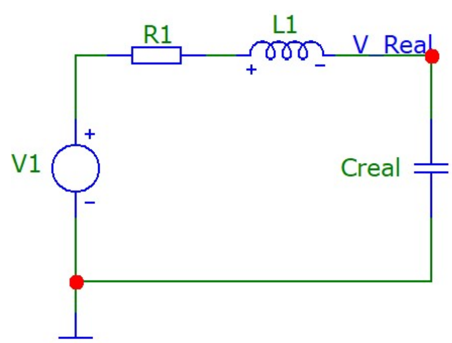


Fig. 3.1.7. Oscillating circuit made practically.

In figure 3.1.7. V1 is a voltage source with a step-up from 0V to 10V (TDK-Lambda Voltage Source). C1 is a capacitor (KEMET, Y5V Dielectric) with a capacitance of 10µF and L1 is a coil with an inductance of 33µH. R1 is a 1Ω resistor.

Figure 3.1.8 shows an oscillating circuit modeled in SPICE.

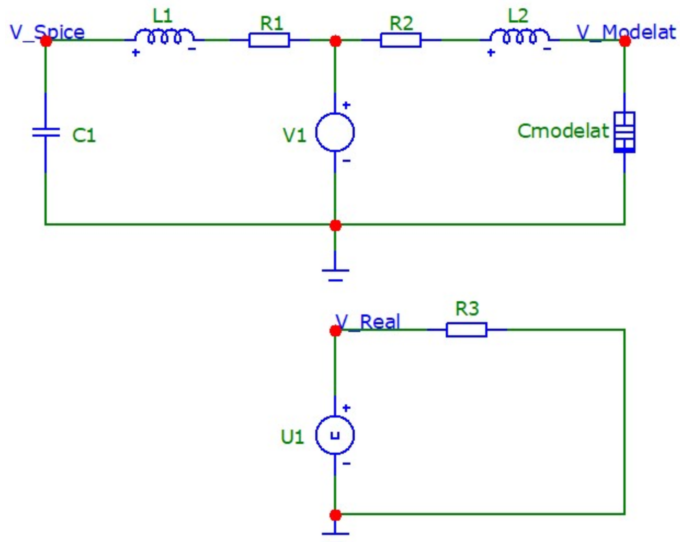


Fig. 3.1.8. Oscillating circuit made in SPICE.

V1 is a step voltage source with a rise from 0V to 10V, C1 is a capacitor from the SPICE library, model proposed by the component manufacturer of 10µF, C2 is a non-linear capacitor modeled in SPICE with a variation of static capacitance according to voltage, temperature and age, L1 and L2 are coils from the SPICE library with an inductance of 33µH , R1 and R2 are resistors from the SPICE library with a resistance of 1 Ω.
U1 is a user source from the SPICE library that allows voltage waveform loading in .USR format.
The voltage waveform is taken from an oscilloscope (Tektronix Model DPO 5104B), with which the voltage variation on the actual capacitor was measured.
After simulating the circuit in figure 3.1.8. from SPICE, the graph in figure 3.1.9 was obtained, where the voltage variation on the capacitors can be observed.

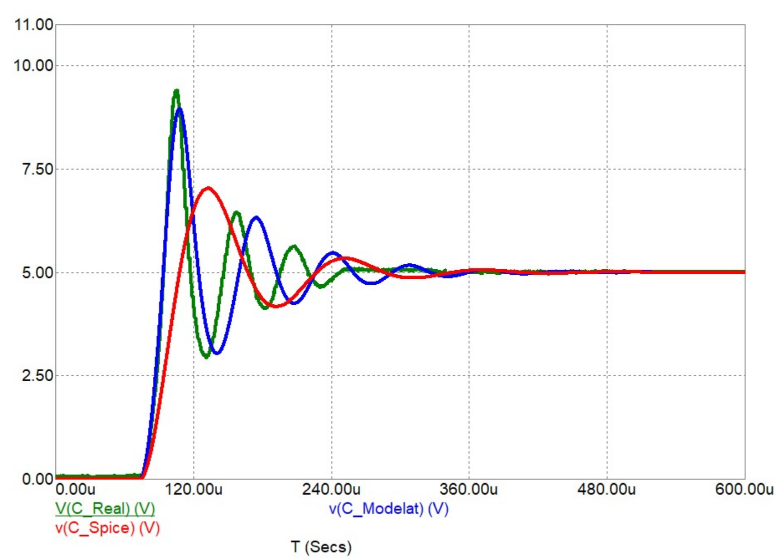


Fig. 3.1.9. Time variation of the voltage of a real capacitor compared to a nonlinear model modeled in SPICE and a SPICE model in an oscillating circuit.

The red color graph indicates the voltage variation on the SPICE capacitor, the blue color graph indicates the voltage variation on the nonlinear capacitor modeled in SPICE and the

green color graph indicates the voltage variation on the real capacitor (measured with the oscilloscope).
Comparing the signals on the graph, it can be seen that the model of the nonlinear capacitor modeled in SPICE is very close to the behavior of the real capacitor.

3.2 SPICE model of a real coil

In this chapter, starting from the equivalent circuit of a real coil, a SPICE model is created in which the inductance varies with the value of the current through the coil, with the temperature of the ambient environment in which the coil operates and with its aging.
In creating the SPICE model, ideal current-controlled voltage sources and ideal voltage sources are used.

3.2.1 The equivalent circuit of a real coil

The equivalent circuit of a real coil is shown in figure 3.2.1.

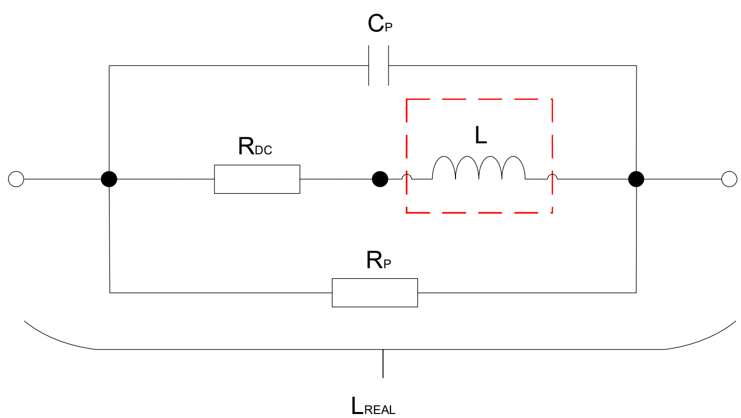


Fig. 3.2.1. The equivalent circuit of a real coil.

This circuit has the following components:

R_{DC} - represents a parasitic resistor inserted in series with the inductance component L, represents the losses due to the copper wire of the coil in direct current.

C_P - represents a parasitic capacitance in parallel with the inductance L, this is determined by the different potentials of the neighboring turns.

R_P - represents a parasitic resistor inserted in parallel with the inductance component L, represents the losses in the electromagnetic core.

L - represents the inductance of the coil, which depends on the current variation and the temperature variation.

3.2.2 SPICE of a nonlinear coil controlled in current and varying with temperature

Given the real coil model and the coil equations, a SPICE model of the real nonlinear current controlled coil will be created. In this model the variation of inductance with variation of current and temperature is taken into account. Also, in this model, the variation of inductance with frequency and resistance in parallel (RP) are taken into account (fig. 3.2.2).

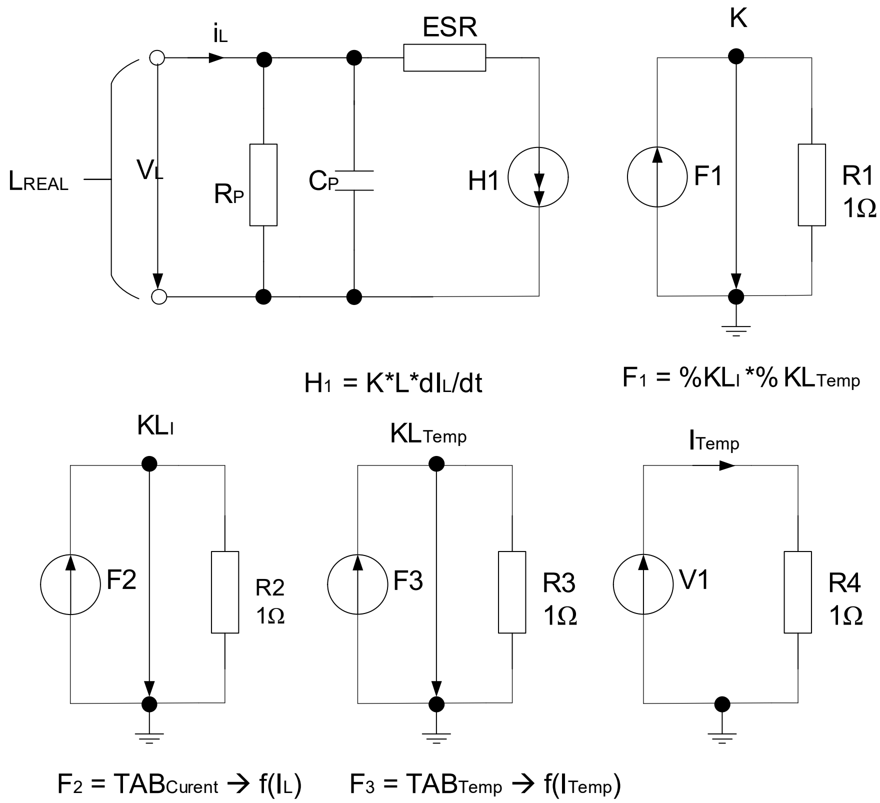


Fig. 3.2.2. SPICE model of a real nonlinear current-controlled coil.

In Figure 3.2.2 H1 is a current-controlled voltage source, using the SPICE property of the source to vary the output voltage according to an integrated expression:

$$H_1 = KL\frac{diL}{dt} \tag{3.2.1}$$

The following parameters can be found in the above expression: K represents the percentage variation of the inductance according to current and temperature. L represents the inductance of the coil (the nominal inductance declared by the component manufacturer). iL represents the current through the coil as a function of time.

F_1 is a controlled current source whose output value depends on the variation of inductance with variation of current and temperature.

$$F_1 = \%KL_I \times \%KL_{Temp} \tag{3.2.2}$$

KL_I represents the percentage variation of inductance with variation of current.

KL_{Temp} represents the percentage variation of inductance with temperature variation.

F_2 is a controlled current source, the output of which depends on the percentage variation in inductance declared by the component manufacturer with the current variation, the input of the source depending on the value of the current through the coil.

$$F_2 = TAB_{Curent} \rightarrow f(I_L) \tag{3.2.3}$$

F_3 is a controlled current source, whose output value depends on the percentage variation of inductance declared by the component manufacturer with temperature, the source input depending on the value of a current that varies with temperature declared by the component manufacturer.

3.2.3 Comparison of the Spice model of a non-linear coil, a standard library coil and the real coil at 25°C and an operating age of less than 10 hours

3.2.2.1 Oscillating circuit

To verify the accuracy of the modeled SPICE model a comparison of the SPICE model of the coil will be made with the behavior of the actual coil and with the behavior of the SPICE coil proposed by the component manufacturer.
Figure 3.2.3 shows the scheme of an oscillating circuit made with a coil, a resistor and a capacitor, which was studied experimentally.

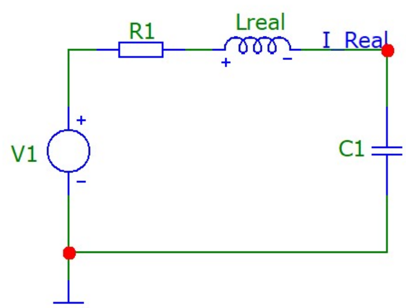


Fig. 3.2.3. Oscillating circuit made practically.

The variation of current through the circuit at the I_Real point was measured using an oscilloscope (Tektronix Model DPO 5104B) and a current probe (Model TCP0150). The waveform was saved in .CSV format for later use in SPICE.
In Figure 3.2.3 V1 is a voltage source with a step-type increase from 0V to 10V (TDK-Lambda Voltage Source).
In figure 3.2.3 C1 is a capacitor (C3225X7R1E106MTJYAN) with a capacity of 10uF and L1 is a coil with an inductance of 100uH from the manufacturer MURATA (LQH32PZ101MN0).
Figure 3.2.4 shows an oscillating circuit modeled in SPICE.

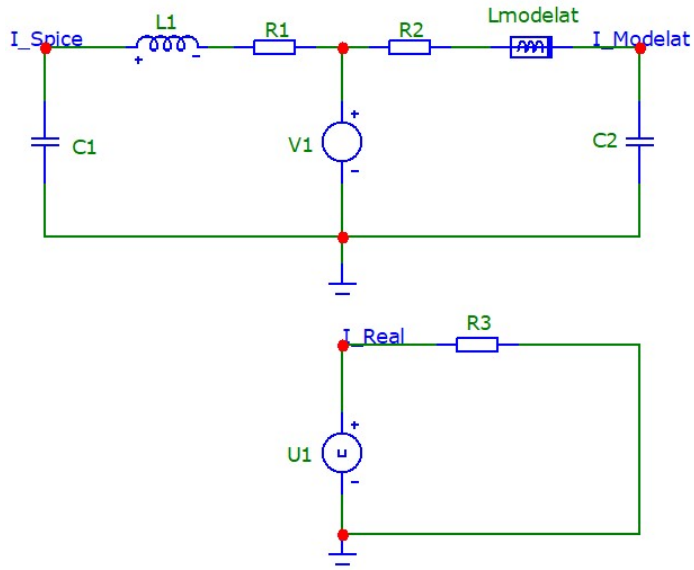


Fig. 3.2.4. Oscillating circuit made in SPICE.

V1 is a voltage source with a step-up from 0V to 10V.
C1 and C2 are capacitors from the SPICE library, model proposed by the manufacturer of the 10µF components.
L1 is a coil from the SPICE library, model proposed by the manufacturer of the 100µH component.
L2 is a real nonlinear coil modeled in SPICE with a variation of inductance as a function of current.
R1 and R2 are 1Ω SPICE library resistors.
U1 is a user source from the SPICE library that allows the voltage waveform to be loaded in .USR format.
The waveform acquired using an oscilloscope (Tektronix Model DPO 5104B) and a current probe (Model TCP0150) was used in the simulation.
Following the simulation of the circuit in figure 3.2.4 using SPICE, the graph in figure 3.2.5 was obtained, where the current variation can be observed.

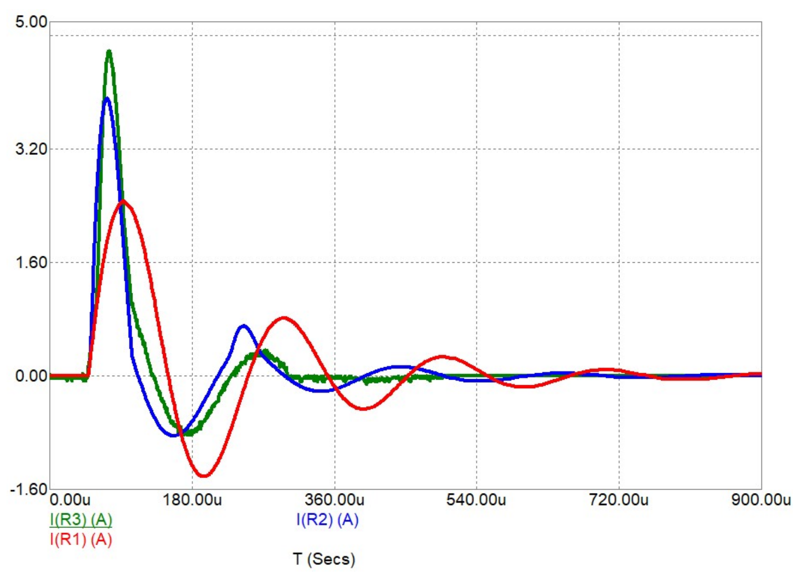


Fig. 3.2.5. Time variation of current through a real coil compared to a nonlinear model modeled in SPICE and a SPICE model in an oscillating circuit.

The red plot shows the variation of current through the SPICE coil, the blue plot shows the variation of current through the SPICE modeled nonlinear coil and the green plot shows the variation of current through the real coil (measured with the oscilloscope).
Comparing the graphs, it can be seen that the nonlinear coil model made in SPICE is very close to the behavior of the real coil.

3.3 SPICE model of a real resistor

In this paragraph, starting from the equivalent circuit of a real resistor, a SPICE model is created in which the resistance varies with the frequency at the resistor's terminals, with the temperature of the ambient environment in which the resistor operates, and with its aging.
Ideal voltage controlled current sources and ideal voltage sources are used in the creation of the SPICE model.

3.3.1 The equivalent circuit of a real resistor

The equivalent circuit of a real resistor is shown in figure 3.3.1.

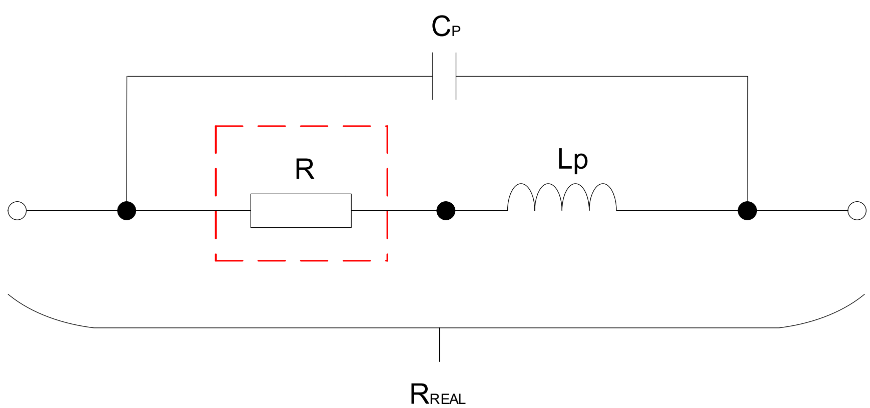


Fig. 3.3.1. The equivalent circuit of a real resistor.

This circuit has the following components:

C_P - represents a parasitic capacitor inserted in parallel with the resistance component R, it is due to the electrodes, cables and other components.

L_P - is a parasitic inductor in series with the resistance component R, it is due to the electrodes, cables and other components.

R – represents the electrical resistance of the resistor, which is dependent on frequency variation and temperature variation.

3.3.2 SPICE model of a nonlinear resistor raying with temperature and aging

Given the real resistor model and the resistor equations, a SPICE model of the real non-linear resistor will be created. The variation of resistance with temperature and aging is taken into account in this model. Also, in this model, the variation of the parasitic capacitance in parallel with the resistor and of the inductance in series with the resistor is taken into account (fig. 3.3.2).

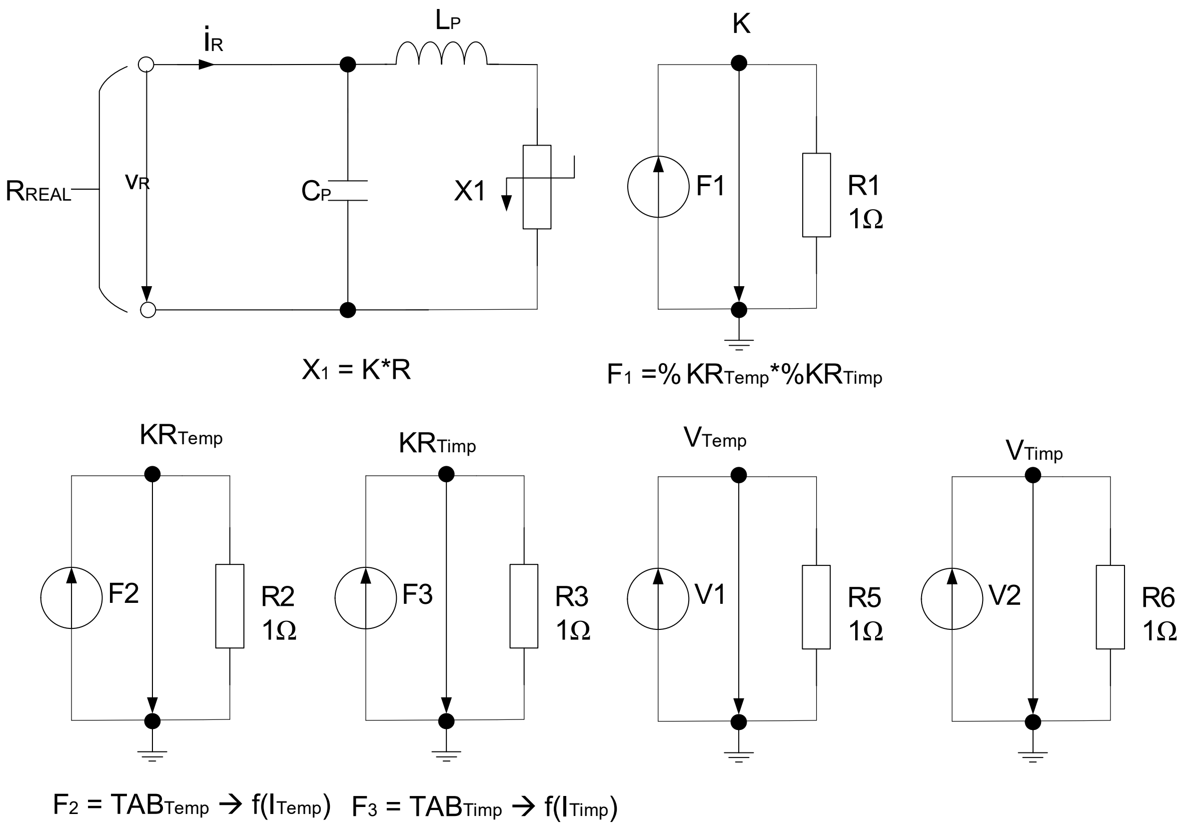


Fig. 3.3.2. SPICE model of a real non-linear resistor.

In Figure 3.3.2 X1 is a voltage-controlled variable resistor, using the SPICE source property to vary the resistance according to an integrated expression:

$$X_1 = KR \tag{3.3.1}$$

The following parameters are found in this expression: K represents the percentage variation of the resistance as a function of temperature and aging.

R represents the resistance of the resistor (the nominal resistance declared by the component manufacturer).
F_1 is a controlled voltage source, the output value depends on the resistance variation with temperature and aging.

$$\boldsymbol{F_1} = \%\boldsymbol{RC_{Temp}} \times \%\boldsymbol{RC_{Timp}} \tag{3.3.2}$$

KC_{Temp} represents the percentage variation in resistance with temperature variation.

KC_{Timp} represents the percentage change in resistance with aging.
F_2 is a controlled voltage source, the output value depends on the percentage variation of resistance with temperature declared by the component manufacturer, the source input depends on the value of a voltage that varies with temperature declared by the component manufacturer (V_1).

$$F_2 = TAB_{Temp} \to f(V_{Temp}) \tag{3.3.3}$$

F_3 is a controlled voltage source, the output value depends on the component manufacturer's declared percentage change in resistance with aging, the source's input depends on the component manufacturer's declared voltage-varying-with-aging value (V_2) .

$$E_3 = TAB_{Timp} \to f(V_{Timp}) \tag{3.3.4}$$

3.3.3 Comparison of the SPICE model of a non-linear resistor, a resistor from the standard library and the real resistor at an operating age of less than 10 hours

To verify the accuracy of the SPICE model a comparison of the SPICE model of the resistor will be made with the behavior of the actual resistor and with the behavior of the SPICE resistor proposed by the component manufacturer for various circuits.

3.3.3.1 Resistive divider at 25°C

Figure 3.3.3 shows the scheme of a resistive divider, which was practically made. The voltage was measured at the V_Real point using an oscilloscope (Tektronix Model DPO 5104B).
The waveform was saved in .CSV format for later use in SPICE.

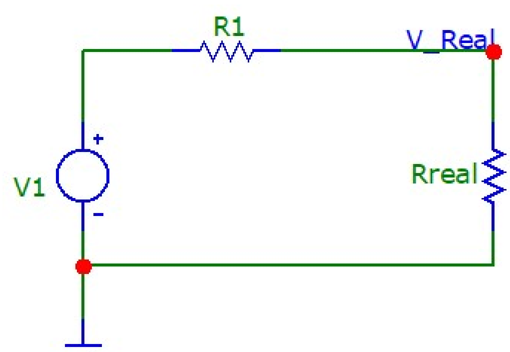


Fig. 3.3.3. Practically made resistive divider.

In figure 3.3.3 V1 is a step voltage source with a step from 0V to 10V (TDK-Lambda voltage source).
In figure 3.3.3 R1 is a 1 kΩ metal foil resistor. R2 is a 1kΩ metal glaze resistor (this model pattern was chosen because it has a large resistance variation with frequency and temperature)

Figure 3.3.4 shows a circuit modeled in SPICE.

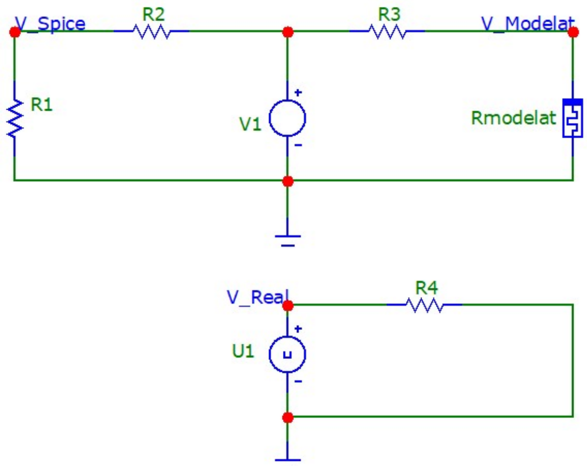


Fig. 3.3.4. Resistive divider made in SPICE.

V1 is a step voltage source with an increase from 0V to 10V or a sinusoidal voltage source with a frequency of 50MHz.
R1, R2, R3, R4 are resistors from the SPICE library with a value of 1kOhm.
Rmodelat is a real nonlinear resistor modeled in SPICE with a resistance variation as a function of temperature and frequency.
U1 is a user source from the SPICE library that allows voltage waveform loading in .USR format.
Voltage taken from an oscilloscope (Tektronix Model DPO 5104B), with which the voltage drop across a real resistor was measured.
Following the simulation of the circuit in figure 3.3.4 using SPICE, the graph in figure 3.3.5, figure 3.3.6 and figure 3.3.7 was obtained, where the voltage variation and the resistance value for the three studied resistors can be observed.

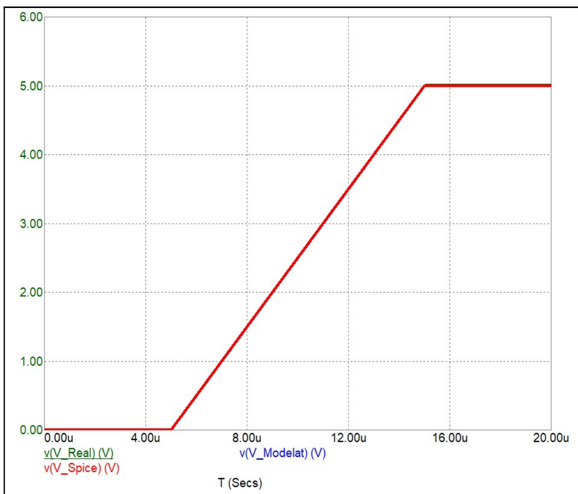


Fig. 3.3.5. Time variation of the voltage drop across a real resistor compared to a nonlinear model modeled in SPICE and a SPICE model to a ramping increase in supply voltage.

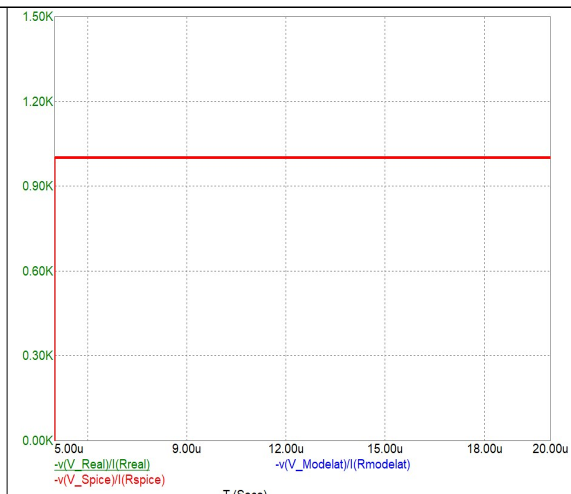


Fig. 3.3.6. The resistance value of a real resistor compared to a non-linear model modeled in SPICE and a SPICE model to a ramping increase in supply voltage.

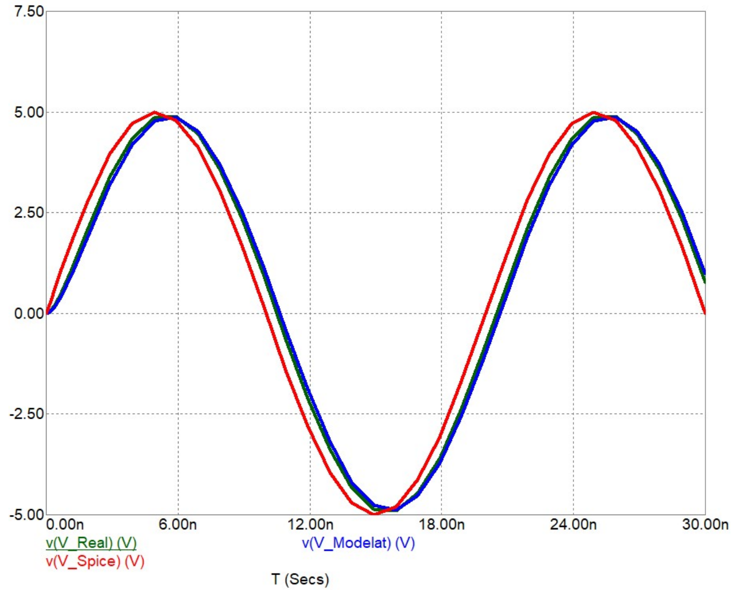


Fig. 3.3.7. Time variation of the voltage drop across a real resistor compared to a nonlinear model modeled in SPICE and a SPICE model at a frequency of 50MHz.

4 CONTRIBUTIONS ON THE SIMULATION OF DIODES AND TRANSISTORS IN SPICE

The author creates new equivalent models of diodes and transistors after which new and high-performance SPICE models are created for the most used electronic components (rectifier diode, stabilizer diode, bipolar transistor and MOS-FET transistor). These models are compared with already existing SPICE models and with the real behavior in various circuits of the studied component.

4.1 SPICE model of a real rectifier diode

In this paragraph, starting from the equivalent circuit of a real rectifier diode, a SPICE model is created in which the voltage drop across the diode varies with the value of the current through the diode and with the temperature of the ambient environment in which the diode operates.

In creating the SPICE model, ideal current-controlled voltage sources and ideal voltage sources are used

4.1.1 The equivalent circuit of a real rectifier diode

Taking into account these operating ranges and the fact that the diode is not ideal, figure 4.1.1 shows the model of a real rectifier diode.

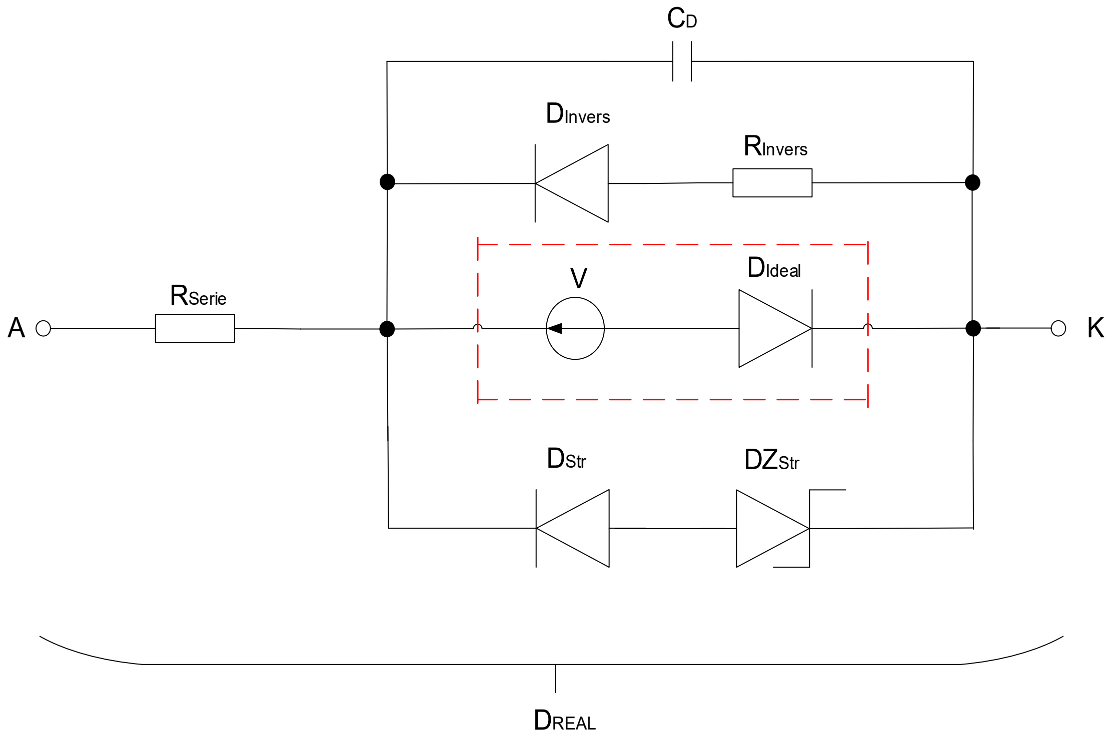


Fig. 4.1.1. The equivalent circuit of a real rectifier diode.

This circuit has the following components:

R_{Serie} – parasitic resistor in series with the ideal diode

C_D – parasitic resistor in series with the ideal diode

D_{Invers} – ideal rectifier diode to achieve reverse current

R_{Invers} – resistor to limit the reverse current

V – current-dependent voltage source through the diode (D_{Ideal}) and diode temperature (D_{Ideal})

D_{Str} – the ideal rectifier diode used to simulate the reverse voltage

DZ_{Str} – ideal Zener diode used to simulate the reverse voltage

4.1.2 SPICE model of a real rectifier diode

Given the model of the real diode next a SPICE model of the real diode will be created whose voltage drop varies with the variation of the current through the diode and with the temperature of the diode. Also, series resistance (R_{Serie}) and parasitic capacitance with frequency (C_D) are taken into account in this model (fig. 4.1.2).

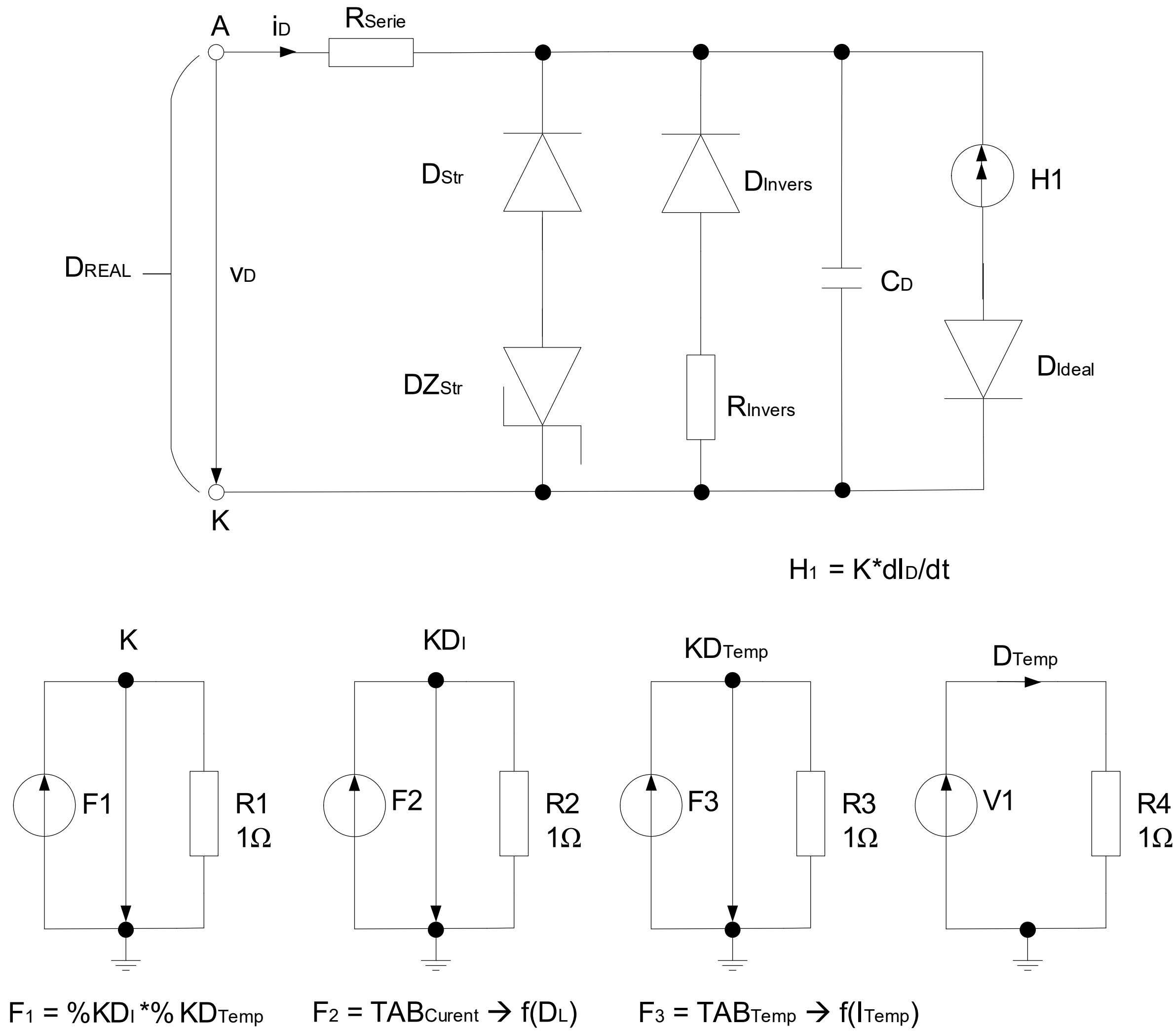


Fig. 4.1.2. SPICE model of a real rectifier diode.

In figure 4.1.2 H1 is a current controlled voltage source, using the SPICE property of the source to vary the voltage output according to an integrated expression:

$$H_1 = K\frac{dID}{dt} \tag{4.1.1}$$

In this expression there are the following parameters: K represents the percentage variation of voltage as a function of diode current and diode temperature. I_D represents the current through the rectifier diode.
F_1 is a controlled current source whose output value depends on the variation of the voltage as a function of the current through the diode and the temperature of the diode.

$$F_1 = \%KD_I \times \%KD_{Temp} \tag{4.1.2}$$

KD_I represents the percentage variation of the voltage drop with the variation of the current through the diode.
KD_{Temp} represents the percentage variation of voltage drop with temperature variation.

F_2 is a controlled current source, whose output value depends on the percentage variation of the voltage drop declared by the component manufacturer with the current intensity, the input of the source being a function of the current value through the diode.

$$F_2 = TAB_{Curent} \rightarrow f(D_I) \quad (4.1.3)$$

F_3 is a controlled current source whose output value depends on the percentage variation of the voltage drop declared by the manufacturer with temperature.

$$F_3 = TAB_{Temp} \rightarrow f(I_{Temp}) \quad (4.1.4)$$

4.1.3 SPICE model comparison of a SPICE-modeled rectifier diode, a standard library rectifier diode, and a real rectifier diode at 25°C and an operating age of less than 10 hours

To verify the accuracy of the SPICE model, a comparison of the SPICE model of the rectifier diode will be made with the behavior of the actual rectifier diode and with the behavior of the SPICE rectifier diode proposed by the component manufacturer in various circuits.

4.1.3.1 SPICE model validation – forward voltage (I~1mA)

Figure 4.1.3 represents the circuit diagram of a forward-conducting rectifier diode, which has been practically realized. The voltage drop at the V_Real point was measured using an oscilloscope (Tektronix Model DPO 5104B).

The waveform was saved in .CSV format for later use in SPICE.

In figure 4.1.3 V1 is a ramp voltage source with an increase from 0V to 10V (TDK-Lambda voltage source).

In figure 4.1.3 Dreal is a diode (BAV99) and R1 a 10kOhm resistor.

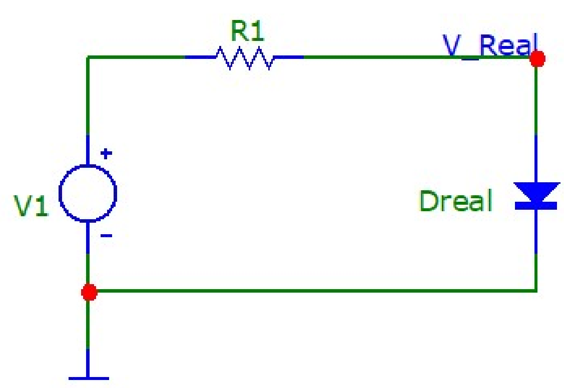


Fig. 4.1.3. Voltage drop across a real diode – practical circuit.

Figure 4.1.4 shows an operating circuit of a conducting rectifier diode modeled in SPICE.

V1 is a voltage source with a ramp-up from 0V to 10V.

D1 is a diode from the SPICE library, a model proposed by the component manufacturer.

Dmodelat is a real rectifier diode modeled in SPICE with a variation of voltage as a function of current through the diode.

R1, R2, R3 are 10kOhm resistors from the SPICE library.

U1 is a user source from the SPICE library that allows voltage waveform loading in .USR format.

Voltage taken from an oscilloscope (Tektronix Model DPO 5104B), with which the voltage drop across a real diode was measured.

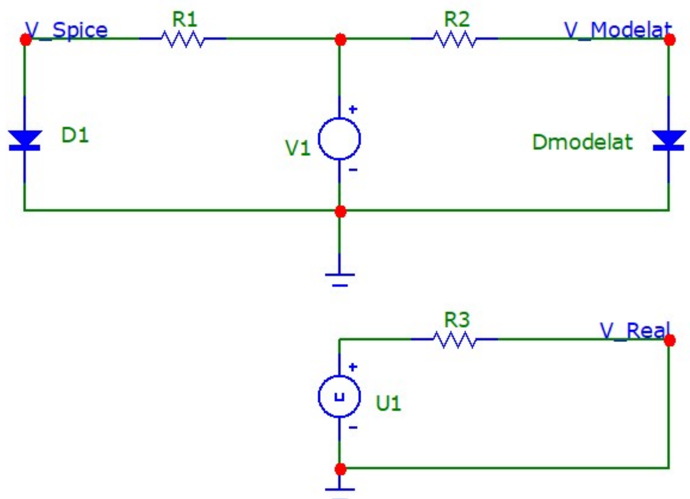


Fig. 4.1.4. Voltage drop across a diode (modeled in SPICE, SPICE, Real) – circuit made in SPICE.

After simulating the circuit in figure 4.1.4 using SPICE, the graph in figure 4.1.5 was obtained.

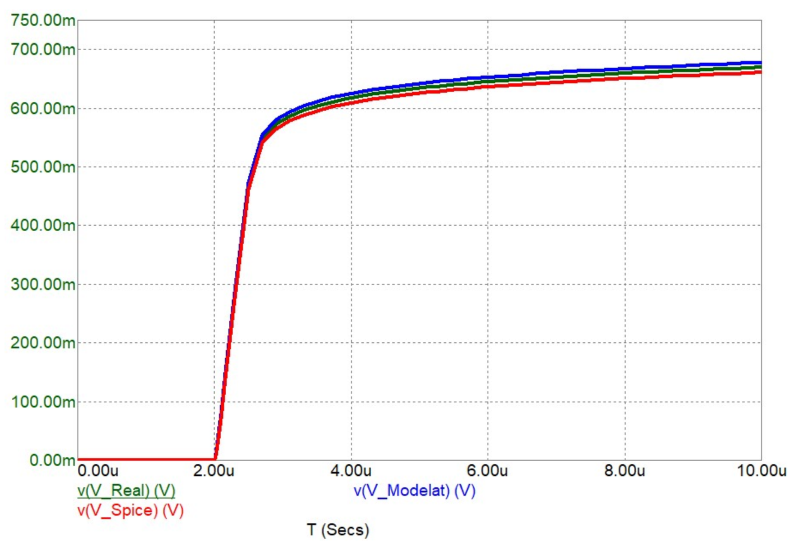


Fig. 4.1.5. Voltage drop across a real diode compared to a SPICE modeled nonlinear diode and a SPICE model diode.

The red graph indicates the voltage drop across the SPICE rectifier diode, the blue graph indicates the voltage drop across the SPICE modeled rectifier diode, and the green graph indicates the voltage drop across the real diode (measured with the oscilloscope).

Comparing the three graphs, it can be seen that the model of the real diode modeled in SPICE is very close to the behavior of the real diode and represents a worse case than the real diode.

4.1.3.2 SPICE model validation – forward voltage (I~10mA, I~100mA)

In figure 4.1.3 and in figure 4.1.4 respectively, the values of resistors R1, R2 and R3 will be changed so that we have a circuit current of 10mA and 100mA respectively. The simulation results are presented in figure 4.1.6 for a current of 10mA and in figure 4.1.7 for a current of 100mA.

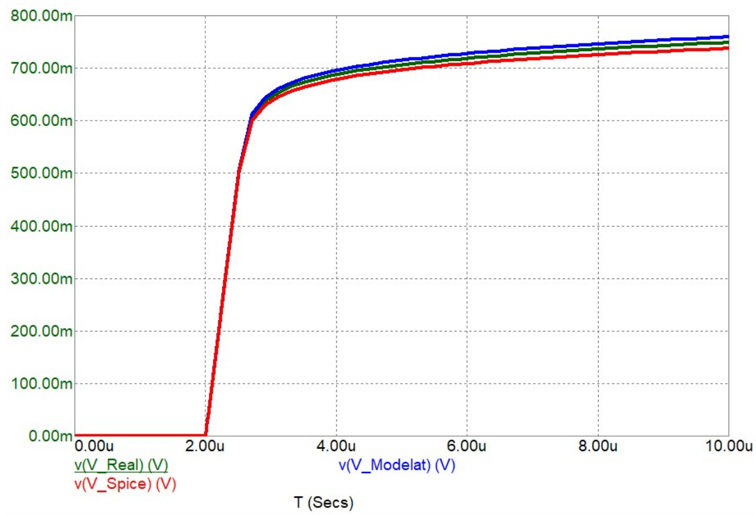


Fig. 4.1.6. Voltage drop across a real diode compared to a SPICE modeled nonlinear diode and a SPICE model diode (I=10mA).

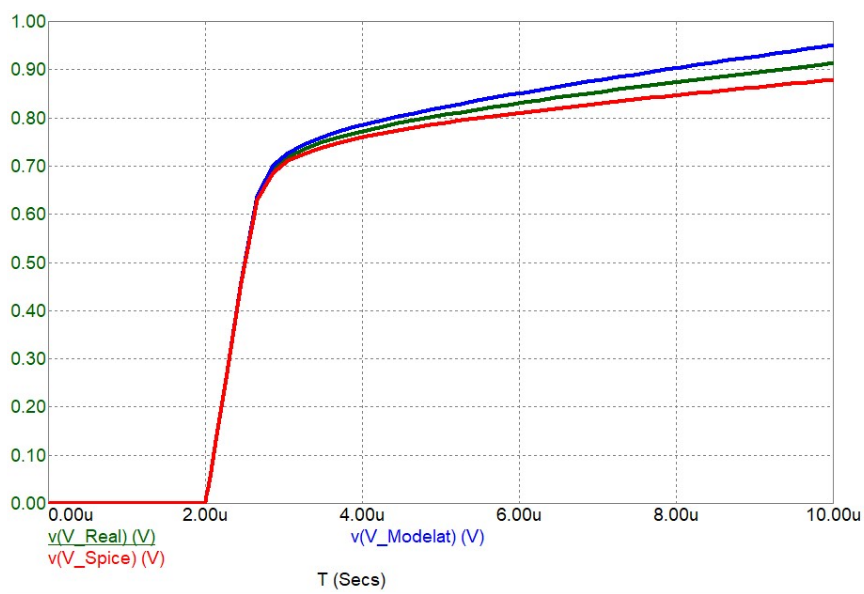


Fig. 4.1.7. Voltage drop across a real diode compared to a SPICE modeled diode and a SPICE model diode (I=100mA).

4.1.3.3 SPICE Model Validation – Inverse Voltage

Figure 4.1.8 represents the circuit diagram of the operation of a reverse-conduction rectifier diode, which was practically realized, and the variation of the current through the diode at the I_Real point was measured using an oscilloscope (Tektronix Model DPO 5104B) and a current probe (Model TCP0150). The waveform was saved in .CSV format for later use in SPICE.

In figure 4.1.8 V1 is a DC voltage source (TDK-Lambda voltage source). D_Real is a diode (BAV99) and R1 a 10kOhm resistor.

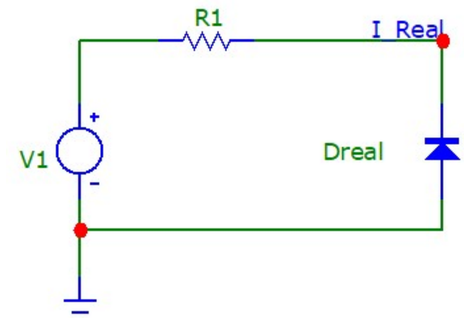


Fig. 4.1.81. Circuit to check the leakage current through a real diode - practical circuit.

Figure 4.1.9 shows a reverse conduction rectifier diode operating circuit modeled in SPICE, D1 is a diode from the SPICE library, a model proposed by the component manufacturer. V2 is a DC voltage source.

D_Modelat is a real rectifier diode modeled in SPICE with a voltage variation as a function of current.

U1 is a user source from the SPICE library that allows the current waveform to be loaded in .USR format.

The value of the electric current taken using an oscilloscope (Model Tektronix DPO 5104B) and a current probe (Model TCP0150), with which the current through a real diode was measured.

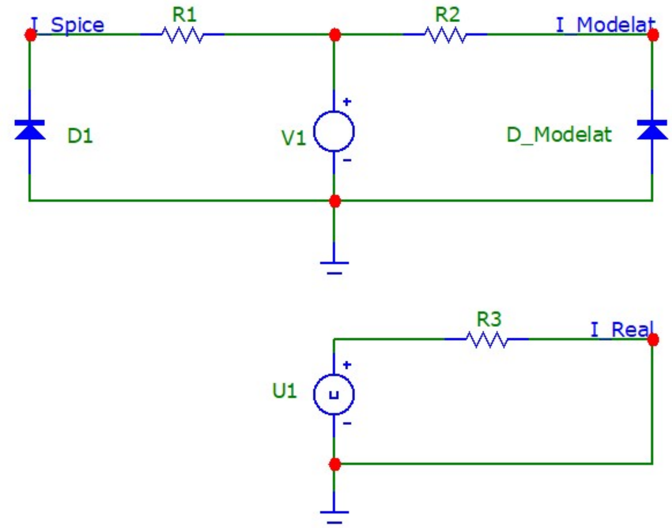


Fig. 4.1.9. Circuit to Check Current Through a Diode (Modeled in SPICE, SPICE, Real) – circuit made in SPICE.

After simulating the circuit in figure 4.1.9, using SPICE, the graph in figure 4.1.10 was obtained.

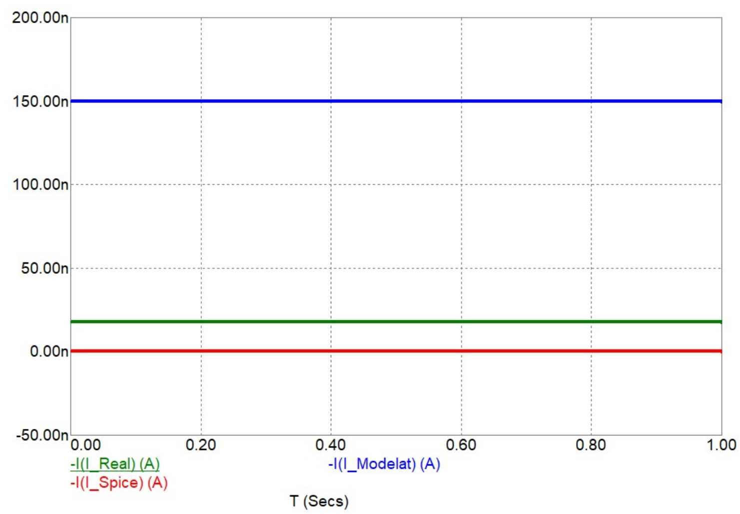


Fig. 4.1.10. Current through a real diode compared to a SPICE modeled diode and a SPICE model diode.

The red graph shows the current through a SPICE rectifier diode, the blue graph shows the current through a SPICE modeled rectifier diode and the green graph shows the current through a real diode (measured with the oscilloscope).

Comparing the three graphs, it can be seen that the diode model modeled in SPICE represents the worst case (where the loss current is 150nA, a value also found in the component's catalog data). This SPICE model helps to calculate the maximum leakage current in a complex circuit.

4.2 SPICE model of a real stabilizer diode

In this paragraph, starting from the equivalent circuit of a real stabilizer diode, a SPICE model is created in which the voltage drop on the diode varies with the value of the current through the diode and with the temperature of the ambient environment in which the diode operates.
In creating the SPICE model, ideal current-controlled voltage sources and ideal voltage sources are used.

4.2.1 The equivalent circuit of a real stabilizer diode

Taking into account these operating ranges and the fact that the diode is not ideal, figure 4.2.1 shows the model of a real stabilizer diode.

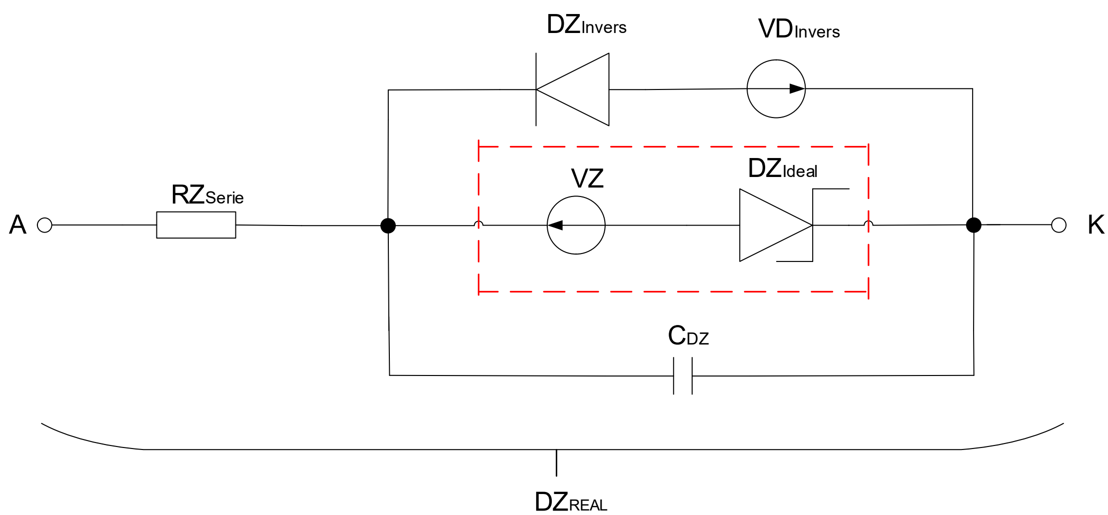


Fig. 4.2.1. The equivalent circuit of a real stabilizer diode.

This circuit has the following components:

R_{Serie} – parasitic resistor in series with the ideal Zener diode

C_{DZ} – parasitic capacitor in series with the ideal Zener diode

DZ_{Invers} – ideal diode to achieve reverse current

VD_{Invers} – voltage source dependent on diode current (D_{Ideal}) and diode temperature (D_{Ideal}) to achieve reverse current

VZ – voltage source dependent on diode current (DZ_{Ideal}) and diode temperature (DZ_{Ideal}) to realize diode current

DZ_{Ideal} – ideal Zener diode used to simulate the forward voltage

4.2.2 SPICE model of a real stabilizer diode

Given the model of the real diode next a SPICE model of the real stabilizer diode will be created whose voltage drop varies with the variation of the current through the diode and with the temperature of the diode. Also, in this model, series resistance (R_{Serie}) and parasitic capacitance with frequency (C_D) are taken into account (fig. 4.2.2).

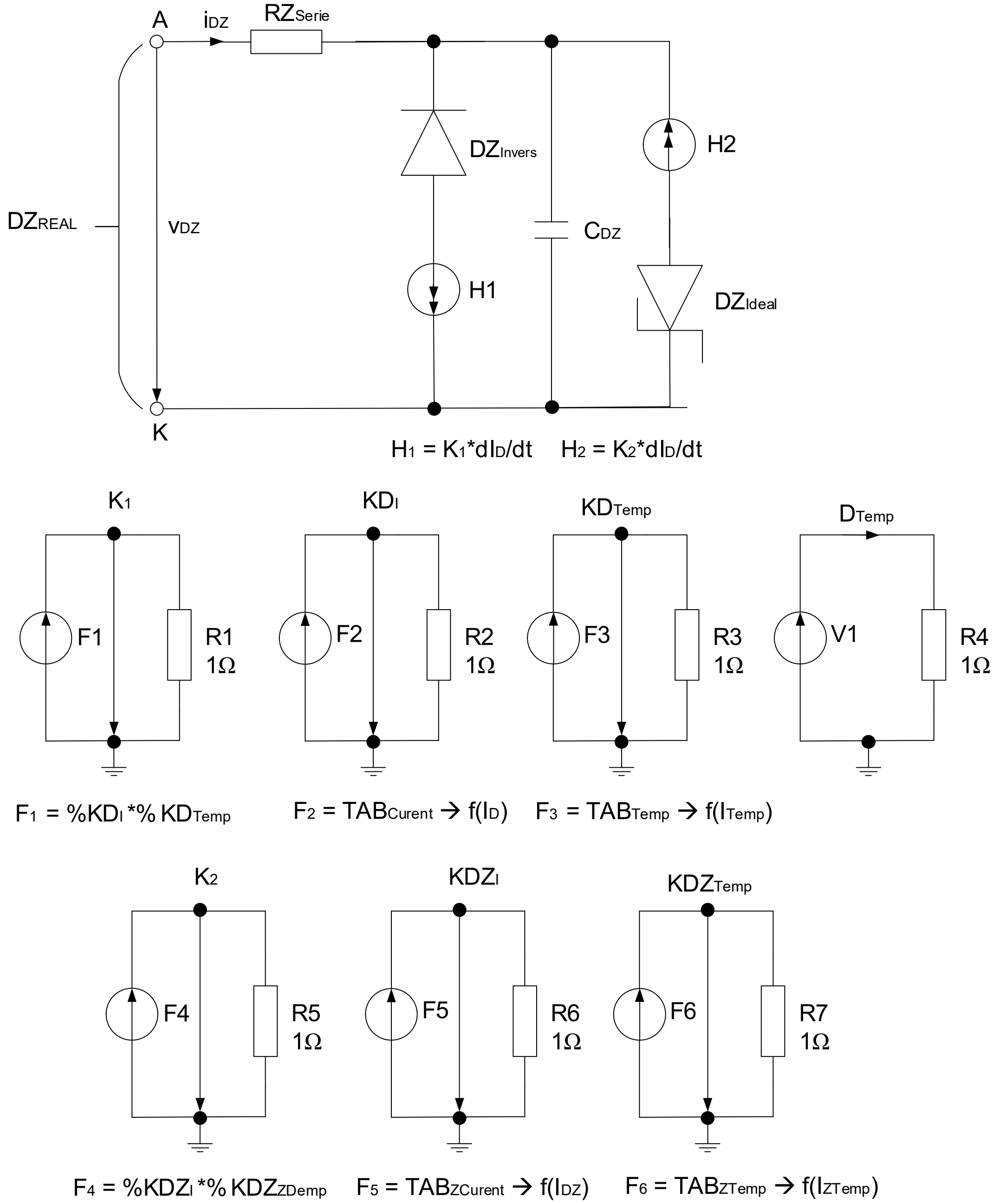


Fig. 4.2.2. SPICE model of a real stabilizer diode.

In Figure 4.2.2 H1 and H2 represent current controlled voltage sources, using the SPICE source property to vary the voltage output according to an integrated expression:

$$H_1 = K1\frac{dID}{dt} \tag{4.2.1}$$

$$H_{12} = K2\frac{dID}{dt} \tag{4.2.2}$$

In this expression there are the following parameters: K1 and K2 represent the percentage variation of voltage as a function of diode current and diode temperature. ID represents the current through the Zener diode.

F_1 and F_4 are controlled current sources, whose output value depends on the variation of voltage across the diode with variation of current and temperature.

$$F_1 = \%KD_I \times \%KD_{Temp} \tag{4.2.3}$$

$$F_4 = \%KDZ_I \times \%KDZ_{ZTemp} \tag{4.2.4}$$

KD_I and KDZ_I represents the percentage variation of the voltage drop with the variation of current through the diode (forward and reverse conduction).

KD_{Temp} represents the percentage variation of voltage drop with temperature variation.

F_2 and F_4 are controlled current sources, whose output value depends on the percentage variation of the voltage drop declared by the manufacturer of the component with the current intensity, the input of the sources being according to the value of the current through the diode (direct and reverse conduction).

$$F_2 = TAB_{Curent} \rightarrow f(D_I) \tag{4.2.5}$$

$$F_5 = TAB_{ZTCurent} \rightarrow f(D_{DZ}) \tag{4.2.6}$$

F_3 and F_6 are controlled current sources, whose output value depends on the percentage variation of the voltage drop declared by the manufacturer with temperature.

$$F_3 = TAB_{Temp} \rightarrow f(I_{Temp}) \tag{4.2.7}$$

$$F_6 = TAB_{Temp} \rightarrow f(I_{ZTTemp}) \tag{4.2.8}$$

4.2.3 Comparison of the proposed SPICE model of a stabilizer diode with a stabilizer diode from the standard library and with a real stabilizer diode at 25°C and an operating age of less than 10 hours

To verify the accuracy of the SPICE model a comparison of the SPICE model of the stabilizer diode will be made with the behavior of the actual stabilizer diode and with the SPICE behavior of the stabilizer diode proposed by the component manufacturer.

4.2.3.1 Spice model validation – forward voltage (I~5mA)

Figure 4.2.3 represents the circuit diagram of the operation of a stabilizer diode in direct conduction, which was practically realized. The voltage drop at the V_Real point was measured using an oscilloscope (Tektronix Model DPO 5104B).
The waveform was saved in .CSV format for later use in SPICE.
In figure 4.2.3 V1 is a 10V DC voltage source (TDK-Lambda voltage source).
In figure 4.2.3 D_Real is a stabilizer diode (BZX84_C5V1) and R1 is a 10kOhm resistor.

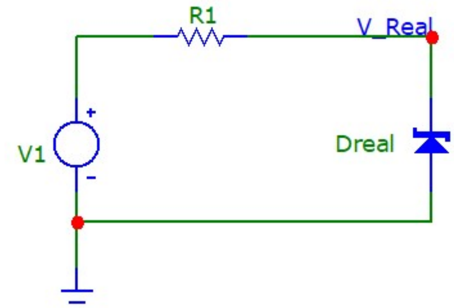


Fig. 4.2.3. Voltage drop across a real stabilizer diode - practical circuit.

Figure 4.2.4 shows an operating circuit of a direct conduction stabilizer diode modeled in SPICE.

V1 is a 10V DC voltage source.

D1 is a stabilizer diode from the SPICE library, a model proposed by the component manufacturer.

Dmodelat_max and Dmodelat_min are real stabilizer diodes modeled in SPICE with a voltage variation as a function of the current through them.

U1 is a user source from the SPICE library that allows the voltage waveform to be loaded in .USR format.

R1, R2, R3, R4 and R5 are resistors from the standard SPICE library with a resistance value of 1kΩ.

Voltage acquired using an oscilloscope (Tektronix Model DPO 5104B), with which the voltage drop across a real diode was measured.

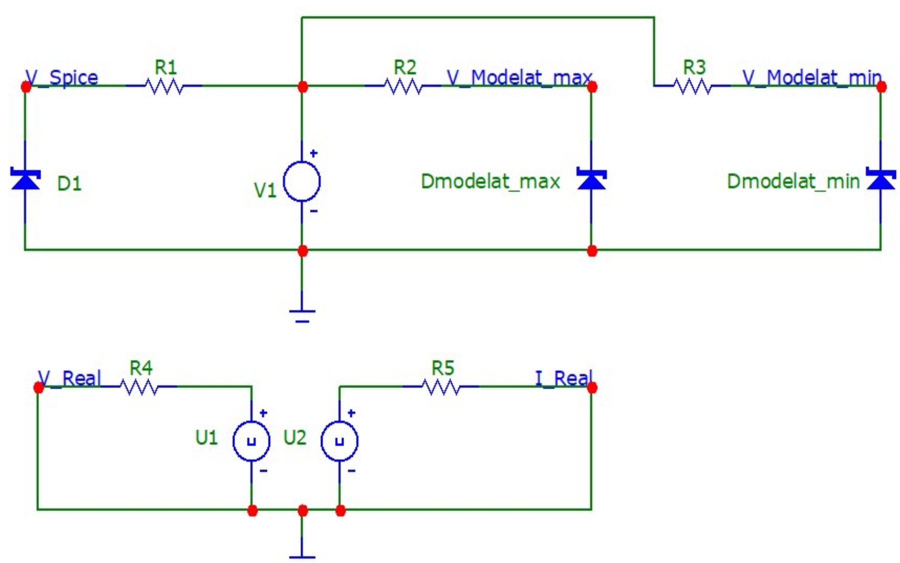


Fig. 4.2.4. Voltage drop across a stabilizer diode (modeled in SPICE, SPICE, Real) – circuit made in SPICE.

Following the simulation of the circuit in figure 4.2.4 using SPICE, the graph in figure 4.2.5 was obtained.

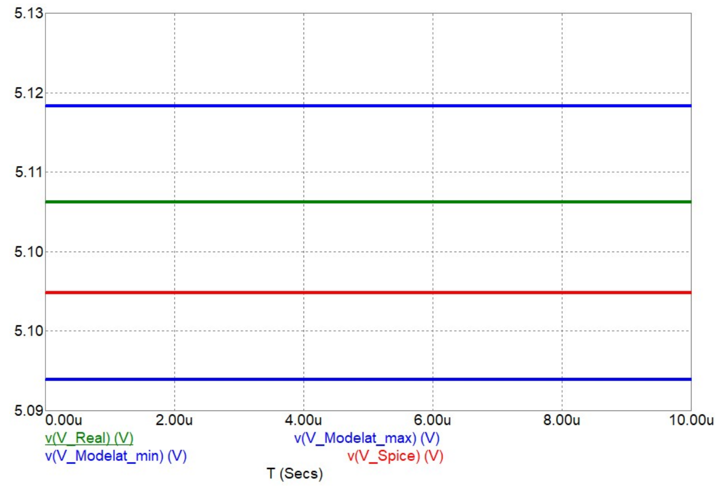


Fig. 4.2.5. Voltage drop across a real Zenner diode compared to a SPICE-modeled nonlinear Zenner diode and a SPICE-modeled Zenner diode.

The red graph shows the voltage drop on the SPICE stabilizer diode, the blue graph shows the voltage drop on the SPICE modeled stabilizer diode (minimum and maximum) and the green graph shows the voltage drop on the real diode (measured with the oscilloscope) .

Comparing the four graphs, it can be seen that the real diode model modeled in SPICE represents the worst cases (maximum and minimum).

4.2.3.2 SPICE model validation – reverse voltage (I~1mA)

Figure 4.2.6 represents the circuit diagram of a reverse conduction Zener diode, which was practically realized, and the variation of the diode voltage at the V_Real point was measured using an oscilloscope (Tektronix Model DPO 5104B). The waveform was saved in .CSV format for later use in SPICE.

In figure 4.2.6 V1 is a ramp voltage source from 0V to 10V (TDK-Lambda voltage source). Dreal is a diode (BZX84_C5V1) and R1 a 10kOhm resistor.

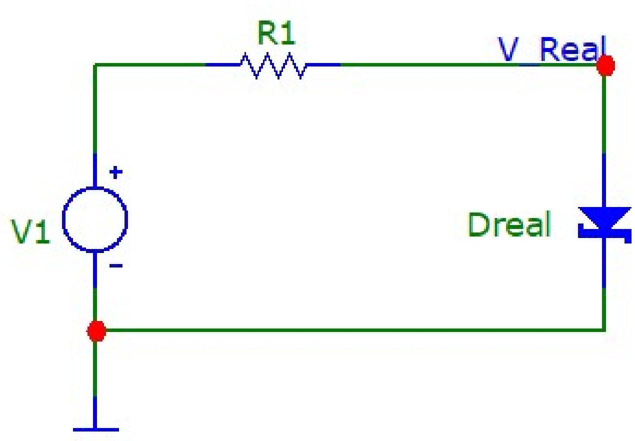


Fig. 4.2.6. Voltage drop across a real Zenner diode - practical circuit.

Figure 4.2.7 shows an operating circuit of a reverse conduction Zener diode modeled in SPICE, D1 is a diode from the SPICE library, a model proposed by the component manufacturer. V1 is a ramp voltage source from 0V to 10V.

Dmodelat is a real rectifier diode modeled in SPICE with a variation of voltage as a function of current.

U1 is a user source from the SPICE library that allows the voltage waveform to be loaded in .USR format.

Voltage taken from an oscilloscope (Tektronix Model DPO 5104B), with which the voltage drop across a real diode was measured.

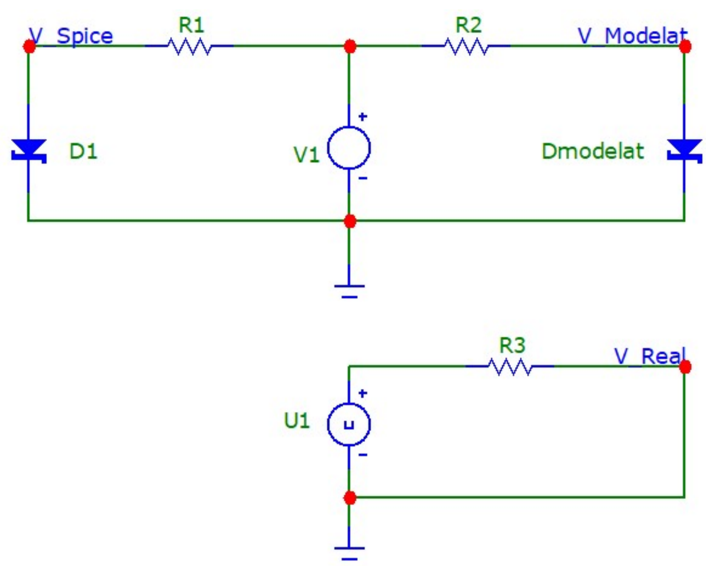


Fig. 4.2.7. Voltage drop across a stabilizer diode (modeled in SPICE, SPICE, Real) – circuit made in SPICE.

After simulating the circuit in figure 4.2.7 in SPICE, the graph in figure 4.2.8 was obtained, where you can see the voltage variation on the diodes.

The red graph shows the SPICE Zenner diode voltage drop, the blue graph shows the SPICE modeled Zenner diode voltage drop and the green graph shows the real Zenner diode voltage drop (measured with the oscilloscope).

Comparing the three graphs, it can be seen that the model of the real diode made in SPICE is very close to the behavior of the real diode representing a worse case than the diode in the SPICE library.

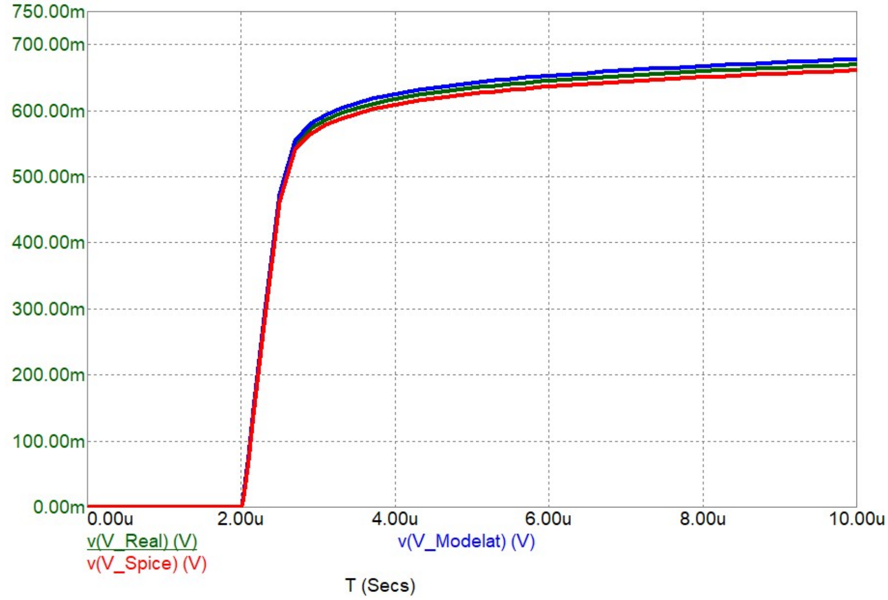


Fig. 4.2.8. Voltage drop across a real Zenner diode compared to a SPICE modeled Zenner diode and a SPICE model Zenner diode.

4.3 SPICE model of a NPN bipolar transistor

In this chapter, starting from a model already existing in the SPICE library of a NPN bipolar transistor, the parameters of this model are checked and some models for extracting these parameters from the catalog data are proposed. The validation of these parameters is done in SPICE simulations comparing the results with the information from the catalog data.

4.3.1 Circuit topology for checking and modifying the parameters of an NPN bipolar transistor

Figure 4.3.1 shows a circuit architecture that helps to manually check some electrical parameters of an NPN bipolar transistor such as: H_{FE}, h_{FE} FT, R_{BB}, R_{EE}, R_{CC}, ect. This data will be compared to the data published by the component manufacturer, then this data will be modified to see if a model is achieved that is closer to the behavior described in the catalog data of the electronic component.

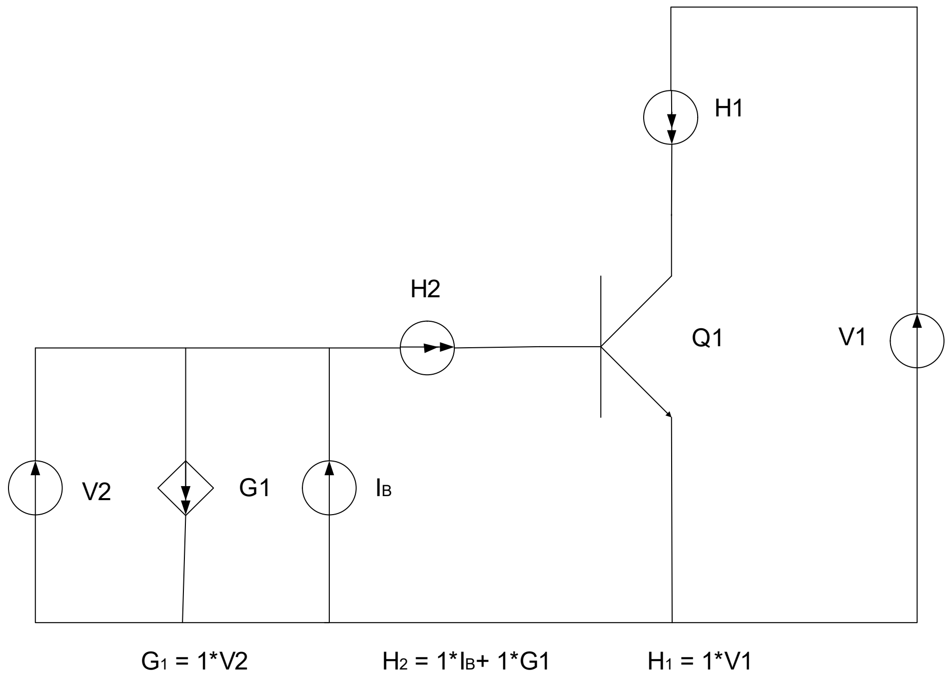


Fig. 4.3.1. Architecture of a circuit that helps to check the parameters of an NPN bipolar transistor.

Figure 4.3.1 uses ideal current-controlled voltage sources and ideal voltage-controlled current sources to monitor or inject test signals. Note that these ideal voltage and current sources are not in the real world but exist in the simulation environment. They allow studying a circuit without affecting its operation.

In figure 4.3.1 are the following components:

- V1 continuous voltage source.
- V2 alternating voltage source.
- G1 voltage controlled current source.
- I_B current source.
- H2 current controlled voltage source.
- H1 current controlled voltage source.
- Q1 bipolar transistor.

Also in this architecture certain points in the circuit are highlighted which help to approximate the parameters of the bipolar transistor, thus:

- ($H2=1*I_B+1*G1$) allows monitoring the base current and using it in future calculation formulas.
- ($G1=1*V2$) allows measuring the base voltage and using it in future calculation formulas.
- ($H1=1*V1$) allows monitoring the collector current, it is represented as a voltage with a ratio of 1 and its use in future calculation formulas.
- (V2) allows a current to be injected into the base of the transistor under study through an ideal voltage controlled current source to allow the current gain to be measured as a voltage ratio.

4.3.1.1 DC simulation of NPN bipolar transistor

Figure 4.3.2 shows the direct current gain (HFE) for the BC817 NPN bipolar transistor [32]. It can be seen that the maximum value of direct current gain (HFE) is about 270 and is constant from 0.1mA to about 100mA where it starts to decrease.

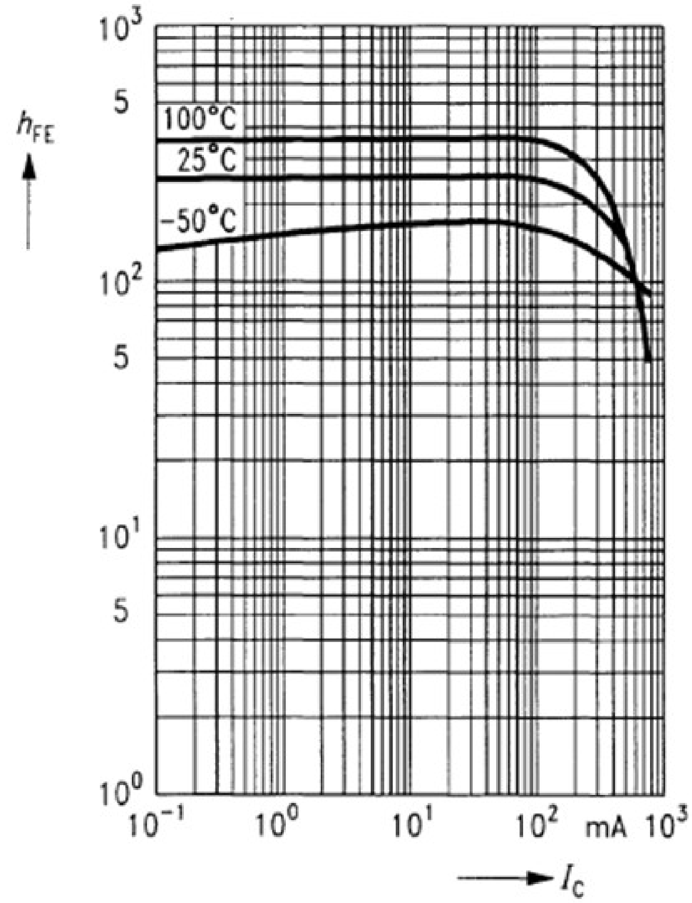


Fig. 4.3.2. Variation of the HFE amplification factor as a function of the collector current for a DC BC817 transistor [32].

The SPICE model proposed by the component manufacturer will be further studied. Thus, the electronic circuit proposed in figure 4.3.6 is made in SPICE. An NPN BC817 bipolar transistor is chosen to be studied. The direct current analysis of the SPICE simulator is run (result in fig. 4.3.3). The graph in Figure 4.3.3 is configured to show H_{FE} on the y-axis and I_C collector current in mA on the x-axis. It can be seen that the maximum value of direct current gain (H_{FE}) is about 300 and is constant from 0.1mA to about 20mA where it starts to decrease.

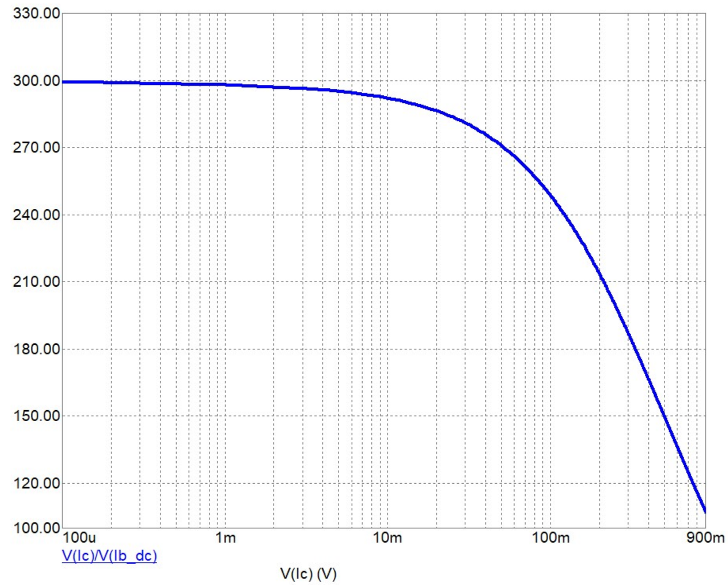


Fig. 4.3.3. DC analysis of H_{FE} gain factor as a function of collector current for a BC817 transistor [circuit simulated in SPICE].

Remarks:

- In the catalog data for the BC817 transistor, the maximum direct current gain (H_{FE}) value is about 270, but after simulation this gain is 300, resulting in a different amplification factor in the SPICE model than in the catalog.
- It is observed that the amplification factor starts to decrease much earlier than in the catalog data (it decreases from 20mA compared to the catalog information to about 100mA) resulting in the IKF parameter being different from the one in the catalog (the parameter responsible for changing the "high angle current").

Taking into account the two observations, the parameter BF (the maximum value of direct current gain in direct conduction) is changed to the value of 270 and the parameter IKF to the value of 900mA. Running the DC analysis of the SPICE simulator again yields the graph in figure 4.3.4.

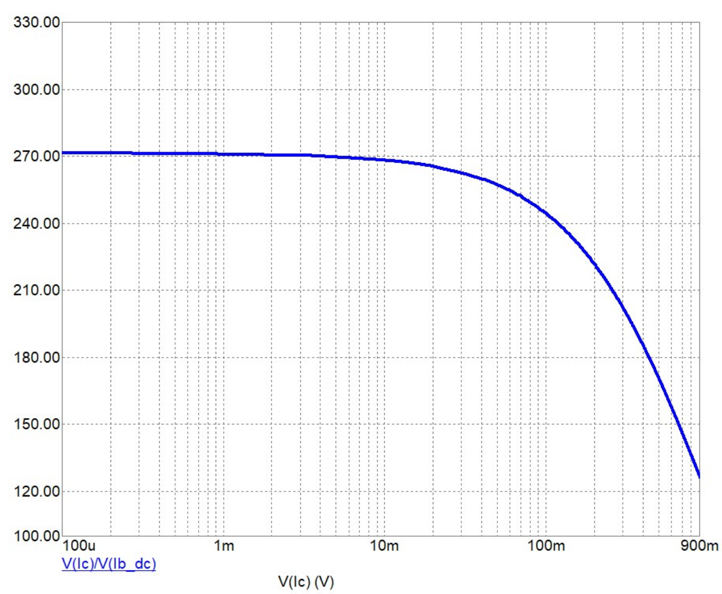


Fig. 4.3.4. DC analysis of HFE amplification factor as a function of collector current for a BC817 transistor [circuit made in SPICE, BF=270, IKF=900mA].

As can be seen in the graph in figure 4.3.4, the maximum value of direct current gain (H_{FE}) is about 270 but this amplification factor starts to decrease much earlier than the value in the catalog with increasing collector current (about 70mA). Thus some more tests are done, modifying this parameter (IKF), it is observed that at a value of approximately 1800mA, the graph looks the closest to the one in the catalog (but this contradicts the definition of this parameter). In the graph in figure 4.3.5 we have BF=270 and IKF=1800mA.

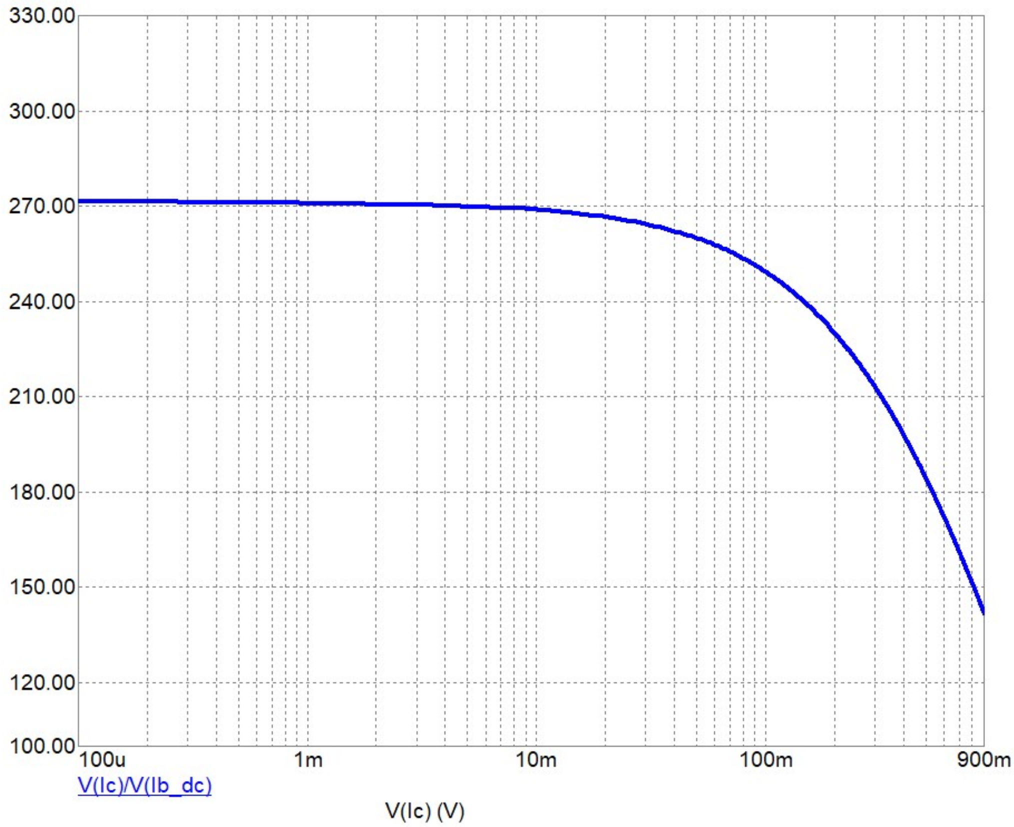


Fig. 4.3.5. DC analysis of HFE amplification factor as a function of collector current for a BC817 transistor [circuit made in SPICE, BF=270, IKF=1800mA].

4.3.1.2 Transient simulation of the NPN bipolar transistor

It is carried out in SPICE, the electronic circuit proposed in figure 4.3.1 The transient analysis of the SPICE simulator is run (the result in figure 4.3.6), it produces an output like an oscilloscope. We apply to the base of the BC817 NPN bipolar transistor a sinusoidal signal with a peak of 10µA and a frequency of 1kHz, corresponding to a period of 1ms, with the help of an ideal sinusoidal voltage source from the SPICE library. This signal is superimposed on a 10uA DC signal. Keep the values found in the previous chapter, BF=270, IKF=1800mA (blue graph.

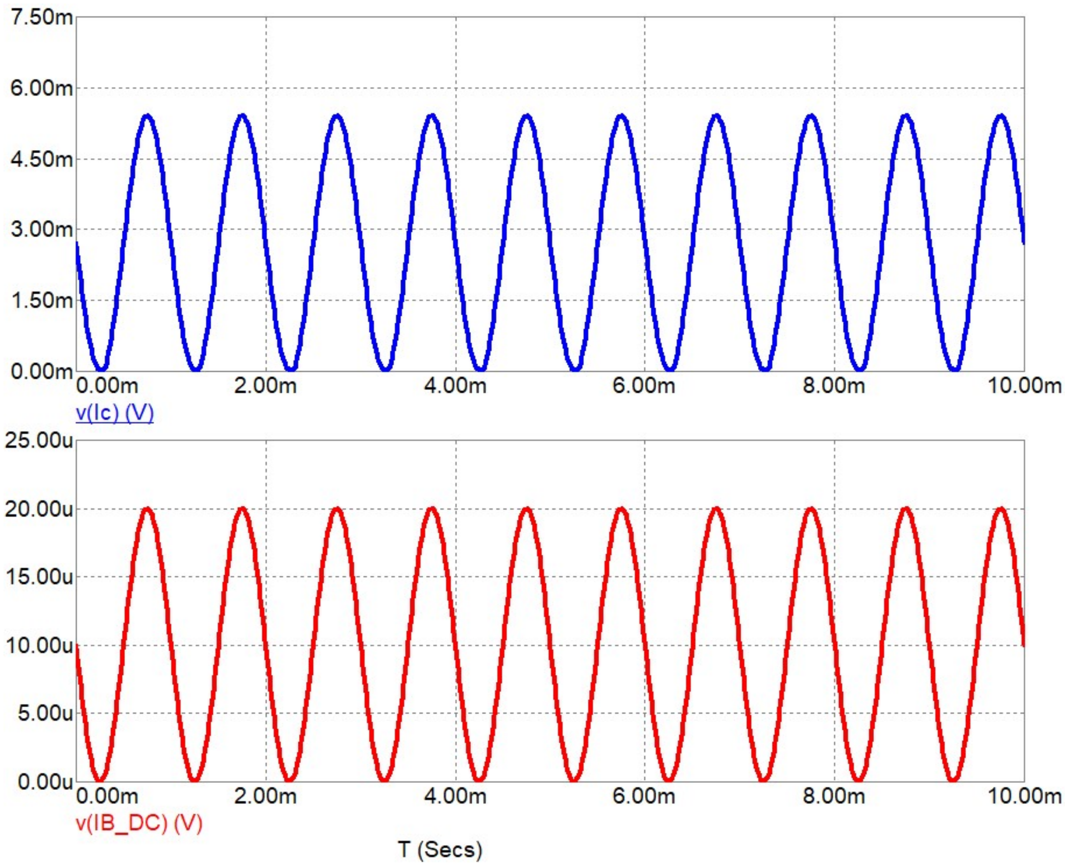


Fig. 4.3.6. Transient analysis for a BC817 transistor [circuit made in SPICE, BF=270, IKF=1800mA].

In the graph in figure 4.3.6, the base current of the bipolar NPN BC817 transistor (displayed in μA) is shown in red color, the collector current of the bipolar transistor is displayed in blue color (displayed in mA). It is observed that at a base current of 20μA we have a collector current of 5.4mA thus we have a transient amplification factor of:

$$h_{FE} = \frac{I_C}{I_B} = \frac{5.4mA}{20\mu A} = 270 \quad (4.3.1)$$

The one that corresponds to the value from the catalog data, respectively to the value of the amplification factor proposed in (BF=270). We have to take into account that the initial SPICE model value of the amplification factor was BF=300.

4.3.1.3 AC simulation of NPN bipolar transistor

It is done in SPICE, the electronic circuit proposed in figure 4.3.1 The AC Analysis of the SPICE simulator (AC Analysis) is run to display the h_{FE} using a variable frequency from 1kHz to 100MHz and a base current of 10uA.

In figure 4.3.7 we have the "Y" axis displayed in logarithmic scale and we can see a low frequency current gain (h_{FE}) of about 270, implicitly a collector current,

$$I_C = I_B h_{FE} = 10\mu * 270 = 2.7mA \quad (4.3.2)$$

It is also observed that h_{FE} =1 at a frequency of approximately 60MHz (FT=60MHz).

We have the drop point of the gain factor (h_{FE}) starting at FT = 170kHz which is quite reasonable. The "transit time" parameter in the SPICE model is TF= 738.663741ps, now if we calculate this time as a function of the drop point, we have:

$$TF = \frac{1}{4\pi FT} = \frac{1}{4*3.14159265*17*10^4} = 468.3402ps \quad (4.3.3)$$

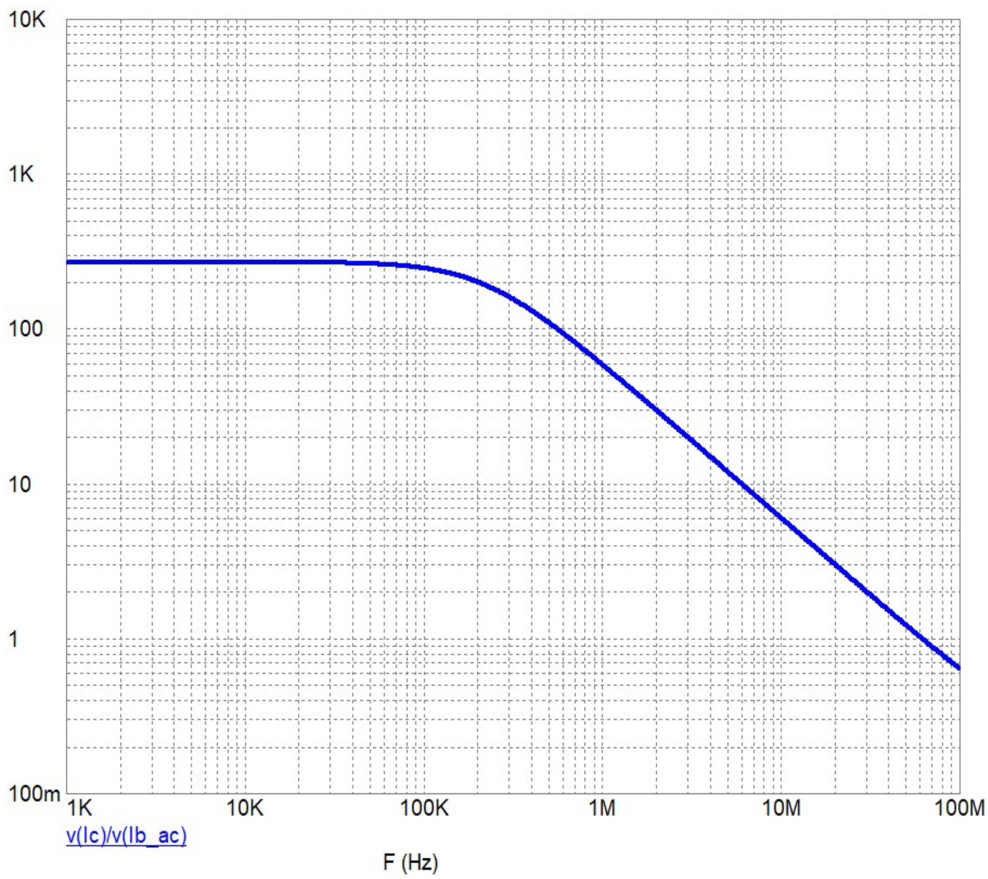


Fig. 4.3.7. AC analysis for a BC817 transistor [circuit made in SPICE, BF=270, IKF=1800mA].

This parameter is highly dependent on the batch of "silicon" from which the transistor is made.

Another important parameter in the small signal study of the transistor is the transconductance (Gm). Next, the variation of transconductance with frequency will be studied for the same bias current of 10uA.
Figure 4.3.8 shows the variation of transconductance (Gm) with frequency (the same variation is used as in the case of determining the h_{FE} parameter, from 1kHz to 100MHz), it should decrease at a frequency of 170kHz (FT = 170kHz) but this is seen to increase which means there is something wrong with the transistor design at a base current of 10µA.

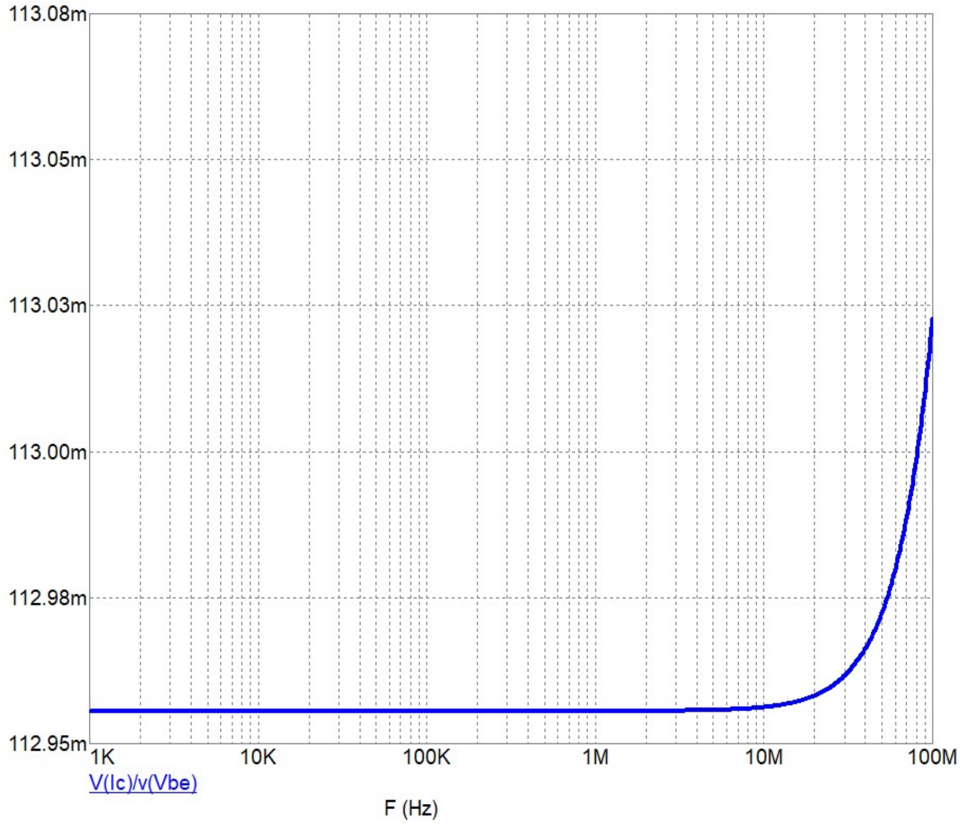


Fig. 4.3.8. AC analysis for a BC817 transistor to determine Gm [circuit made in SPICE, BF=270, IKF=1800mA].

Checking the SPICE model parameters they show that the base resistance (R_{BB}=0Ω), emitter resistance R_{EE}=401mΩ and collector resistance R_{CC}=0Ω. Considering that the transconductance (Gm) should decrease at 170kHz but it increases, it means that these parameters are not the correct ones. This means that the base resistance (R_{BB}) forms a low-pass filter with the base capacitance (which is obviously much larger than a value of "zero") and this causes the transconductance (Gm) to decrease. Taking into account the base current and the maximum possible current through the collector, the base resistance (R_{BB}) and the emitter resistance (R_{EE}) are chosen according to the calculation method presented previously, thus we have:

$$R_{BB}=1400\Omega$$

$$R_{EE}=10\Omega$$

Applying the new values (TF, R_{BB}, R_{EE}) in the SPICE model, we have the graph in figure 4.3.9, where the new values for current gain (h_{FE}) and transconductance (Gm) can be observed.
Note that changing the base resistance (R_{BB}) also changes the current gain (h_{FE}) but the transistor operation is closer to reality considering that the transconductance starts to drop at

about 170kHz. It is important to take into account this parameter (R_{BB}) because it affects the power gain at high frequency.

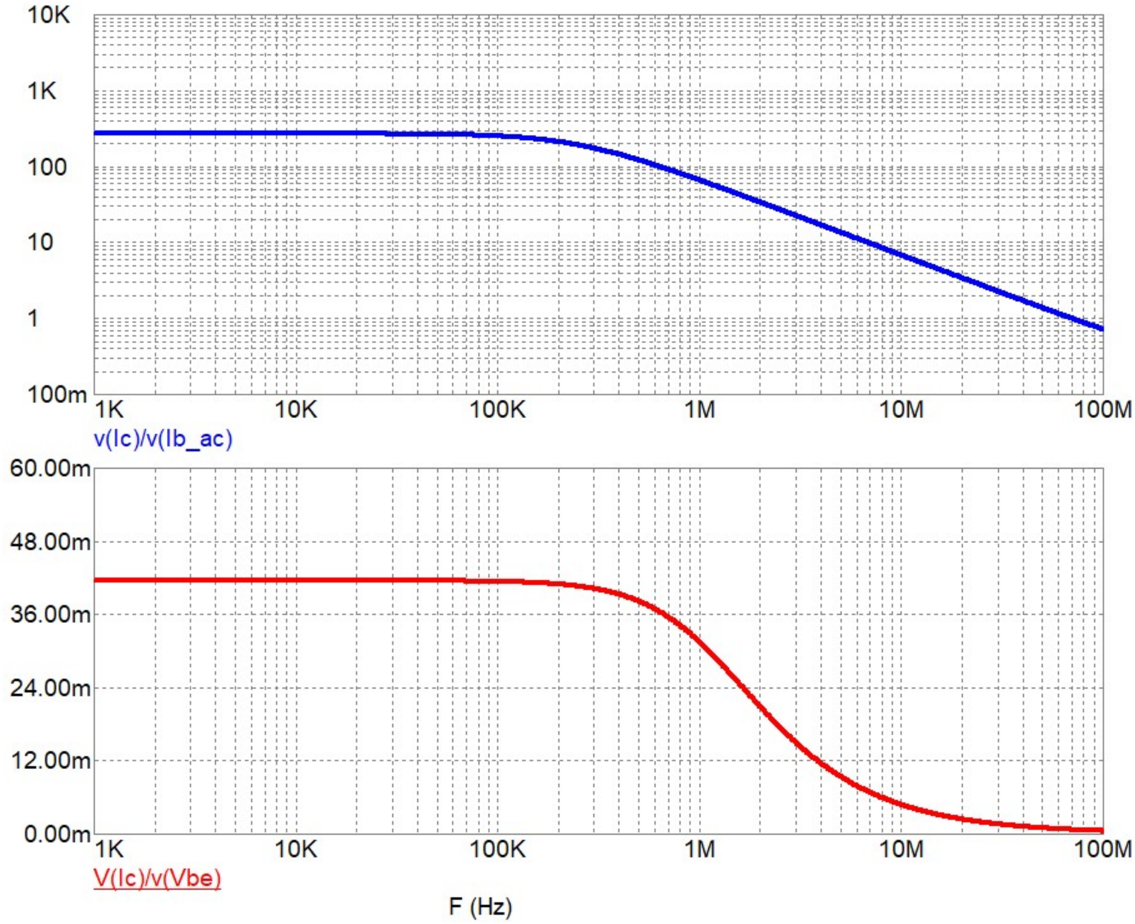


Fig. 4.3.9. AC analysis for a BC817 transistor to determine h_{FE}, Gm [circuit made in SPICE, BF=270, IKF=1800mA, R_{BB}=1400Ω, R_{EE}=10 Ω, TF=467ps].

4.4 SPICE model of a n-channel MOS-FET transistor

In this chapter, starting from the equivalent circuit of a real n-channel MOS-FET transistor, a SPICE model is created in which the drain-source resistance of the transistor varies with the change in the base-source voltage and with the temperature of the ambient environment in which the transistor operates.
Creating the SPICE model uses ideal current-controlled voltage sources, ideal components, and ideal voltage sources.

4.4.1 The equivalent circuit of a real n-channel MOS-FET transistor

Taking into account these operating regions, the variation of parameters with temperature and that the n-channel MOS-FET is not ideal, a model of a real n-channel MOS-FET is created in figure 4.4.1.

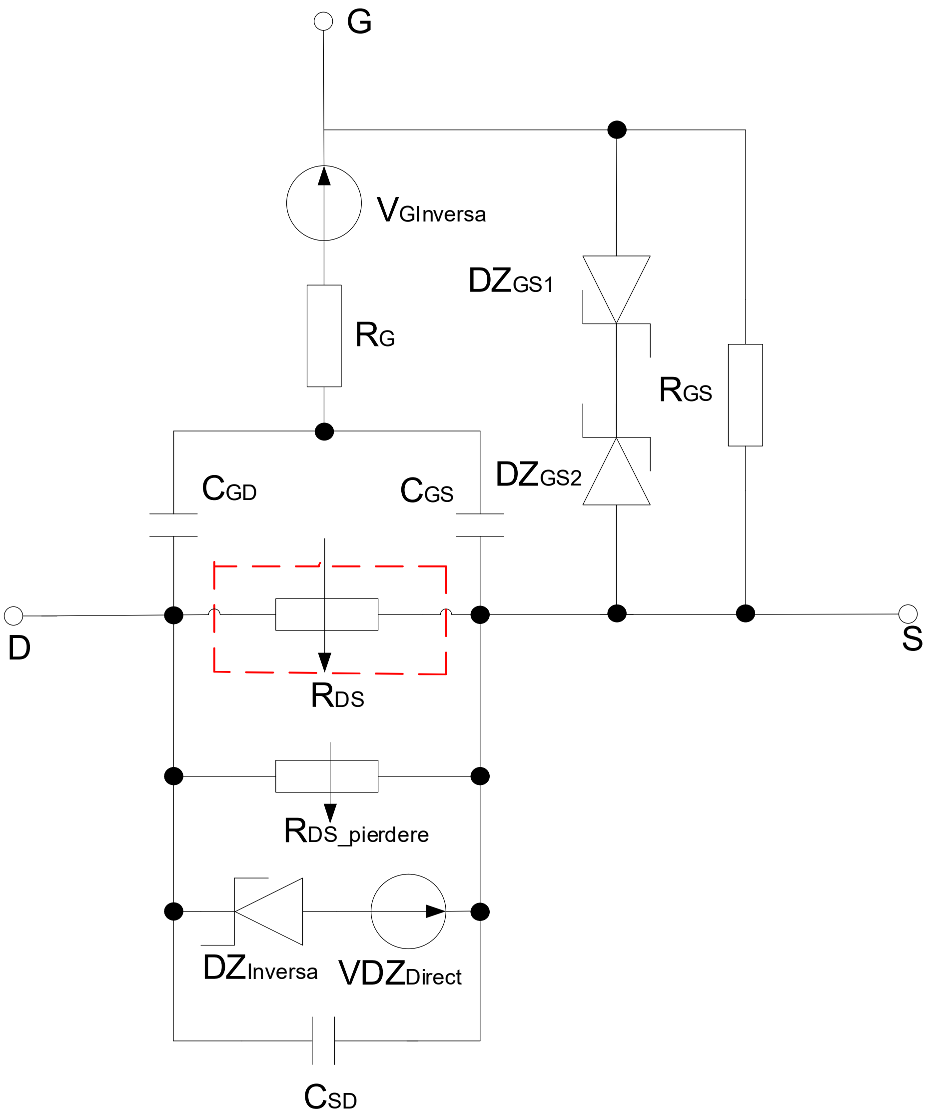


Fig. 4.4.1. The equivalent circuit of a real MOS-FET transistor.

This circuit has the following components:

$V_{GInversa}$ – ideal voltage source to achieve the threshold voltage
R_G – parasitic resistor in the grid of the transistor
R_{GS} – parasitic resistor between the grid and source of the transistor
DZ_{GS1}, DZ_{GS2}– Zener diodes between the grip and the source of the transistor
C_{GS} – parasitic capacitor in the grid of the transistor
$R_{DS\_pierdere}$ – resistor to ensure drain current between drain and source
$DZ_{Inversa}$ – Zener diode internal to the transistor
VDZ_{Direct} – ideal voltage source to provide the source-drain voltage.
C_{SD} – parasitic capacitor between the drain and the source of the transistor
R_{DS} – drain-source variable resistor

4.4.2 SPICE model of a real n-channel MOS-FET transistor

Given the model of the real n-channel MOS-FET next a SPICE model of the real n-channel MOS-FET whose drain-source resistance varies with the value of the drain-source voltage, the value of the voltage of the grid, the value of the drain current and the variation of these parameters with the temperature of the transistor is taken into account (fig. 4.4.2).

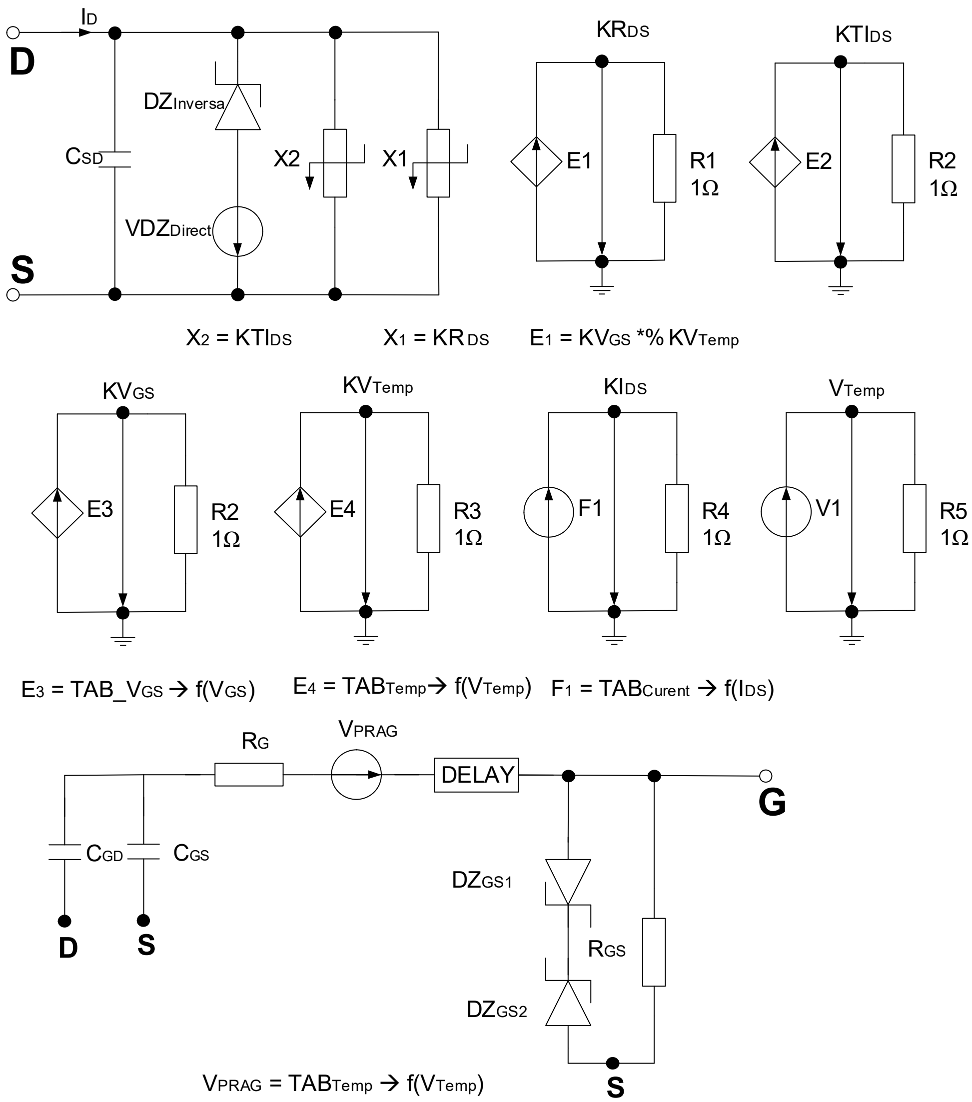


Fig. 4.4.2. SPICE model of a 'n' channel MOSFET transistor.

In figure 4.4.2 X_1 and X_2 are voltage controlled variable resistors, using the SPICE source property to vary the resistance according to an integrated expression:

$$X_1 = KR_{DS} \tag{4.4.1}$$

$$X_2 = KTI_{DS} \tag{4.4.2}$$

In this expression there are the following parameters: K and KT which represent the variation of the drain-source resistance depending on the drain-source voltage and the temperature of the transistor respectively the variation of the drain-source loss current depending on the temperature of the transistor. R_{DS} represents the nominal drain-source resistance. IDS represents the nominal leakage current.

E_1 is a controlled voltage source, the output value depends on the variation of drain-source resistance as a function of variation of grid-source voltage and variation of temperature.

$$E_1 = KV_{GS} \times \%KV_{Temp} \tag{4.4.3}$$

E_2 is a controlled voltage source, the output value depends on the drain-source current variation and the temperature variation.

$$E_2 = KI_{DS} \times \%KV_{Temp} \tag{4.4.4}$$

KV_{GS} represents the percentage variation of the drain-source resistance with the variation of the grid-source voltage.
KV_{Temp} represents the percentage variation of drain-source resistance with temperature variation.
KI_{DS} represents the percentage variation of drain-source leakage current with temperature variation.
E_3 is a controlled voltage source, whose output value depends on the variation of the drain-source resistance declared by the component manufacturer with the variation of the grid-source voltage.

$$E_3 = TAB\_V_{GS} \rightarrow f(V_{GS}) \tag{4.4.5}$$

E_4 is a controlled voltage source, whose output value depends on the variation of the drain-source resistance declared by the component manufacturer with temperature variation (V_1).

$$E_4 = TAB_{Temp} \rightarrow f(V_{Temp}) \tag{4.4.6}$$

F_1 is a controlled current source, whose output value depends on the percentage variation of the drain-source loss current with temperature variation (V_1).

$$F_1 = TAB_{Curent} \rightarrow f(I_{DS}) \tag{4.4.7}$$

V_{PRAG} – ideal voltage source to achieve the threshold voltage

R_G – parasitic resistor in the transistor grid

R_{GS} – parasitic resistor between the grid and the source of the transistor

DZ_{GS1}, DZ_{GS2}– Zener diodes between the grip and the source of the transistor

C_{GS} – parasitic capacitor between the grid and the source of the transistor

C_{GD} – parasitic capacitor between the grid and the drain of the transistor

4.4.3 Comparison of the proposed SPICE model of a n-channel MOS-FET with a standard library transistor and a real transistor at 25°C and an operating age of less than 10 hours

To verify the accuracy of the SPICE model, a comparison of the SPICE model of an n-channel MOS-FET with the behavior of the actual MOS-FET and with the SPICE behavior of the MOS-FET proposed by the component manufacturer will be made.

4.4.3.1 SPICE model validation

Figure 4.4.3 represents the circuit diagram of an n-channel MOS-FET transistor, which has been practically realized. The voltage drop across V_Drain and V_Grid was measured using an oscilloscope (Tektronix Model DPO 5104B).

The waveform was saved in .CSV format for later use in SPICE.

In figure 4.4.3 V1 is a voltage source (TDK-Lambda voltage source). Xreal is a "n" channel MOS-FET transistor (IRFH3707). R1, R2, R3 are resistors having various electrical resistance values (these resistors change depending on the application, the electrical resistance values will be presented next). V2 DC voltage source (TDK-Lambda voltage source).

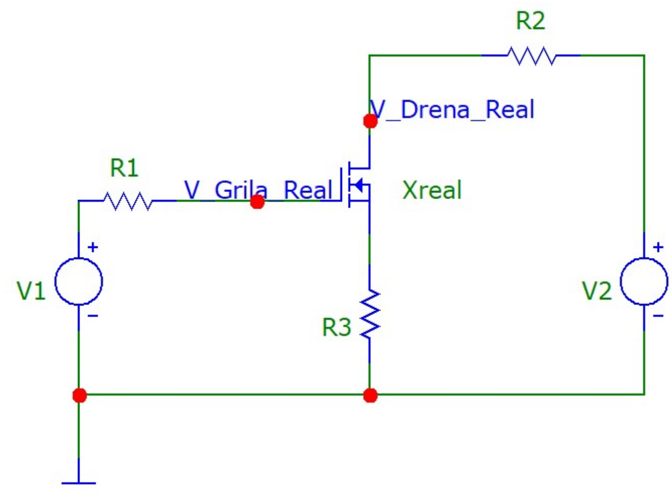


Fig. 4.4.3. Practical circuit for the study of the "n" channel MOS-FET transistor (IRFH3707).

Figure 4.4.4 shows an operation circuit of a n-channel MOS-FET transistor modeled in SPICE, a n-channel MOS-FET transistor from the SPICE library, and the results of a practically studied transistor (IRFH3707).

V1 is a voltage source (depending on the studied parameter of the MOS-FET transistor it can be: step, pulse, sinusoidal).

V2 DC voltage source.

X1 is an IRFH3707 MOS-FET transistor from the SPICE library, a model proposed by the component manufacturer.

Xmodelat is a real MOS-FET transistor modeled in SPICE.

U1, U2 are user sources from the SPICE library which allows voltage waveform to be loaded in .USR format.

R1, R2, R3, R4, R5, R6, R7 R8 are ideal resistors from the SPICE library.

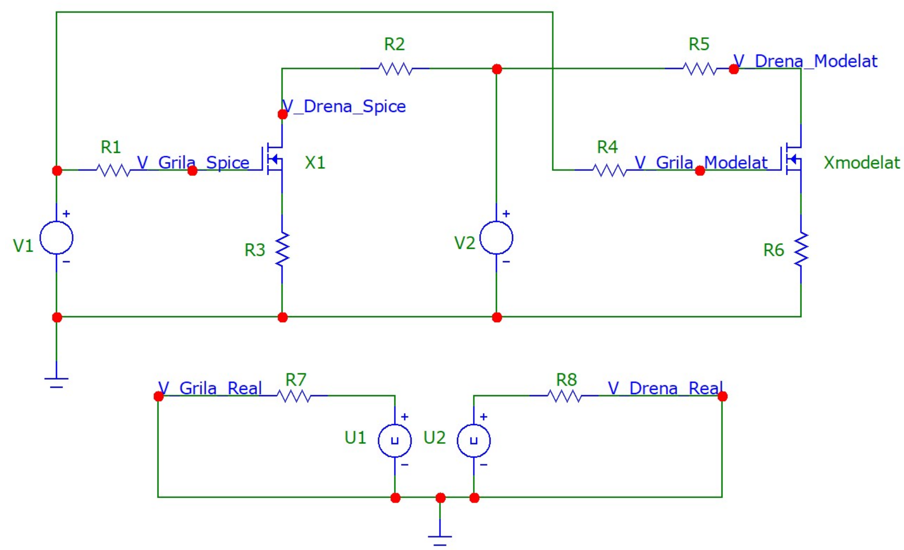


Fig. 4.4.4. Circuit made in SPICE for the study of the "n" channel MOS-FET transistor (IRFH3707) (Modeled, SPICE, Real).

4.4.3.2 Validation of the SPICE model – stray capacitance from the grid (drain in "air")

Using the circuit in figure 4.4.3, the grid-source capacity of a real MOS-FET transistor (IRFH3707) is studied, under the following conditions:

- The drain is in "air", R2 does not exist in the circuit.
- The resistance value of resistor R1 is 100Ω.
- The resistance value of resistor R3 is zero Ω.
- V1 is a pulse type voltage source (Initial=0V, V1=5V, TD=50ns, Tr=0ns, Tf=0ns, PW=800ns, PER=2000ns)
- V2 is a 10V DC voltage source
- Xreal is a transistor MOS-FET (IRFH3707)
- The voltage at point V_Grila was measured using an oscilloscope (Model Tektronix DPO 5104B). The waveform was saved in .CSV format for later use in SPICE.

Using the circuit in figure 4.4.4, the grid-source capacitance of a real MOS-FET transistor (IRFH3707) is studied in comparison with a transistor from the SPICE library and a transistor modeled in SPICE under the following conditions:

- The drain of transistor X1 and transistor Xmodeled are in "air", R2 and R5 do not exist in the circuit
- The resistance value of resistors R1 and R4 is 100Ω.
- The resistance value of resistors R3 and R6 is zero Ω.
- V1 is a pulse type voltage source from the SPICE library (Vinitial=0V, V1=5V, TD=50ns, Tr=0ns, Tf=0ns, PW=800ns, PER=2000ns).
- V2 is a 10V DC voltage source.
- U1 is a user source from the SPICE library that allows loading the voltage waveform in .USR format (the voltage acquired with an oscilloscope (Tektronix Model DPO 5104B), with which the grid-source voltage was measured on a real transistor).

After simulating the circuit in figure 4.4.4 using SPICE, the graph in figure 4.4.5 was obtained.

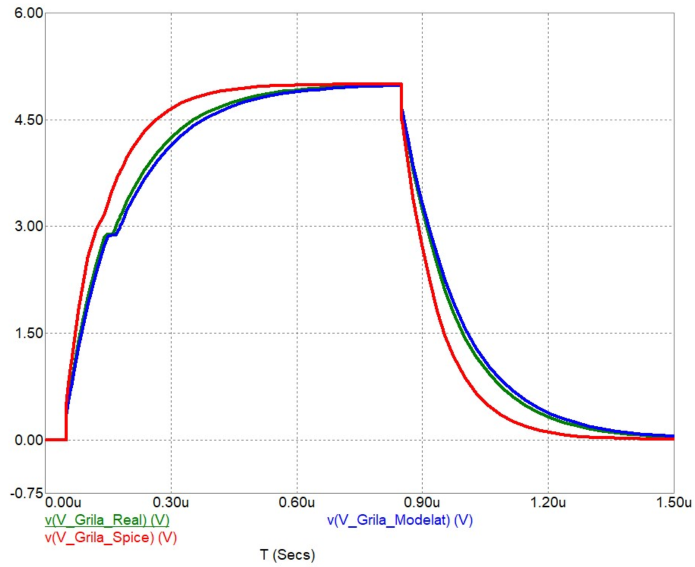


Fig. 4.4.5. Grid-source voltage of a real MOS-FET compared to a SPICE modeled MOS-FET and a SPICE-model drain to “air” MOS-FET.

The red graph indicates the grid-source voltage of the SPICE transistor, the blue graph indicates the grid-source voltage of the modeled transistor in SPICE, and the green graph indicates the grid-source voltage of the real transistor (measured with the oscilloscope).

Comparing the three graphs, it can be seen that the actual transistor model modeled in SPICE represents the worst case and the result is closer to the real one.

4.4.3.3 SPICE model validation – drain-source leakage current

Using the circuit in figure 4.4.3, the drain-source loss current of a real MOS-FET transistor (IRFH3707) is studied, under the following conditions (imposed in the catalog data):

- The voltage in the drain is 24V.
- Grid-source voltage is 0V.
- Grid resistance is 1.3Ω.

Thus in the circuit in figure 4.4.3 we have:

- The resistance value of resistor R2 is 10Ω.
- The grid is in "air", R1 does not exist in the circuit.
- The resistance value of resistor R3 is zero Ω.
- V2 is a 24V DC voltage source.
- Xreal is a MOS-FET transistor (IRFH3707).

The current through resistor R2 was measured using an oscilloscope (Tektronix DPO 5104B Model). The waveform was saved in .CSV format for later use in SPICE.

Using the circuit in figure 4.4.4, the drain-source leakage current of a real MOS-FET transistor (IRFH3707) is studied in comparison with a transistor from the SPICE library and a transistor modeled in SPICE under the following conditions:

- The resistance value of resistors R2 and R5 is 10Ω.
- The resistance value of resistors R1 and R4 is 1.3Ω.
- The grids of the transistors are in "air", R1 and R4 do not exist in the circuit
- The resistance value of resistors R3 and R6 is zero Ω.
- U2 is a user source from the SPICE library that allows loading the voltage waveform in .USR format (the voltage acquired with an oscilloscope (Tektronix

Model DPO 5104B), with which the grid-source voltage was measured on a real transistor).

After simulating the circuit in figure 4.4.4 using SPICE, the graph in figure 4.4.6 was obtained.

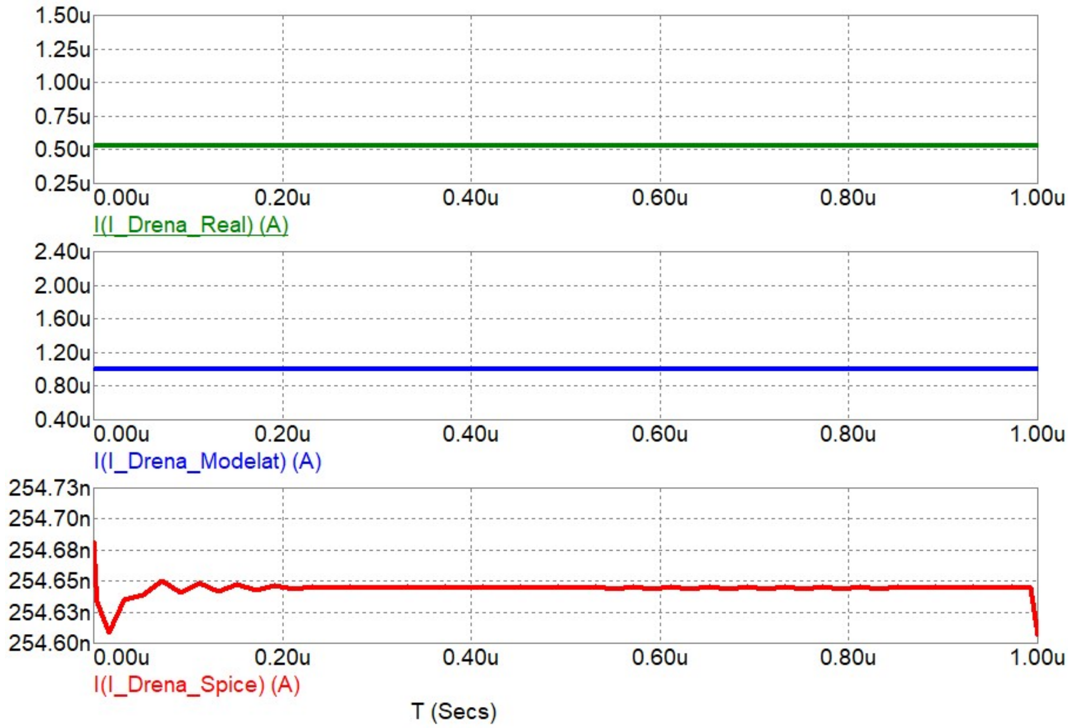


Fig. 4.4.6. Drain-source leakage current of a real MOS-FET compared to a SPICE-modeled MOS-FET and a SPICE MOS-FET.

The red graph indicates the drain-source current of the SPICE transistor, the blue graph indicates the drain-source current of the transistor modeled in SPICE and the green graph indicates the drain-source current of the real transistor (measured with the oscilloscope). It can be seen that the value of the drain current for a real transistor is about 550nA, that of the SPICE library transistor is 254nA, and that of the nonlinear transistor is 1µA (value also found in the component catalog data). Comparing the three values, it can be seen that the real transistor model modeled in SPICE represents the worst case. This helps in calculating the maximum leakage current in a complex scheme.

4.4.3.4 Validation of the SPICE model – drain-source breakdown voltage

Using the circuit in figure 4.4.3, the drain-source breakdown voltage of a real MOS-FET transistor (IRFH3707) is studied, under the following conditions (imposed in the catalog data):

- Drain-source voltage greater than 30V.
- The drain resistance is 120kΩ (ID=120µA for a drain-source voltage of 30V).

Thus in the circuit in figure 4.4.3 we have:

- The resistance value of resistor R2 is 120kΩ.
- The grid is in "air", R1 does not exist in the circuit.
- The resistance value of resistor R3 is zero Ω.

- V2 is a step-type DC voltage source from 0V to 80V (Initial=0V, V1=80V, TD=100u, TR=400u, TF=0, PW==500u, PER=1m).
- Xreal is a MOS-FET transistor (IRFH3707).

- The voltage at the V_Drena_Real point was measured using an oscilloscope (Tektronix DPO 5104B Model). The waveform was saved in .CSV format for later use in SPICE.

Using the circuit in figure 4.4.4, the breakdown voltage of a real MOS-FET transistor (IRFH3707) is studied in comparison with a transistor from the SPICE library and a transistor modeled in SPICE under the following conditions:

- The resistance value of resistors R2 and R5 is 120kΩ.
- Grid is in "air", R1 and R4 are in circuit.
- The resistance value of resistors R3 and R6 is zero Ω.
- V1 is a pulse-type voltage source from the SPICE library (Vinitial=0V, V1=80V, TD=100u, TR=400u, TF=0, PW==500u, PER=1m).
- U1 is a user source from the SPICE library that allows loading the voltage waveform in .USR format (voltage acquired with an oscilloscope (Tektronix Model DPO 5104B), with which the grid-source voltage was measured on a real transistor).

After simulating the circuit in figure 4.4.4 using SPICE, the graph in figure 4.4.7 was obtained for the drain-source breakdown voltage.

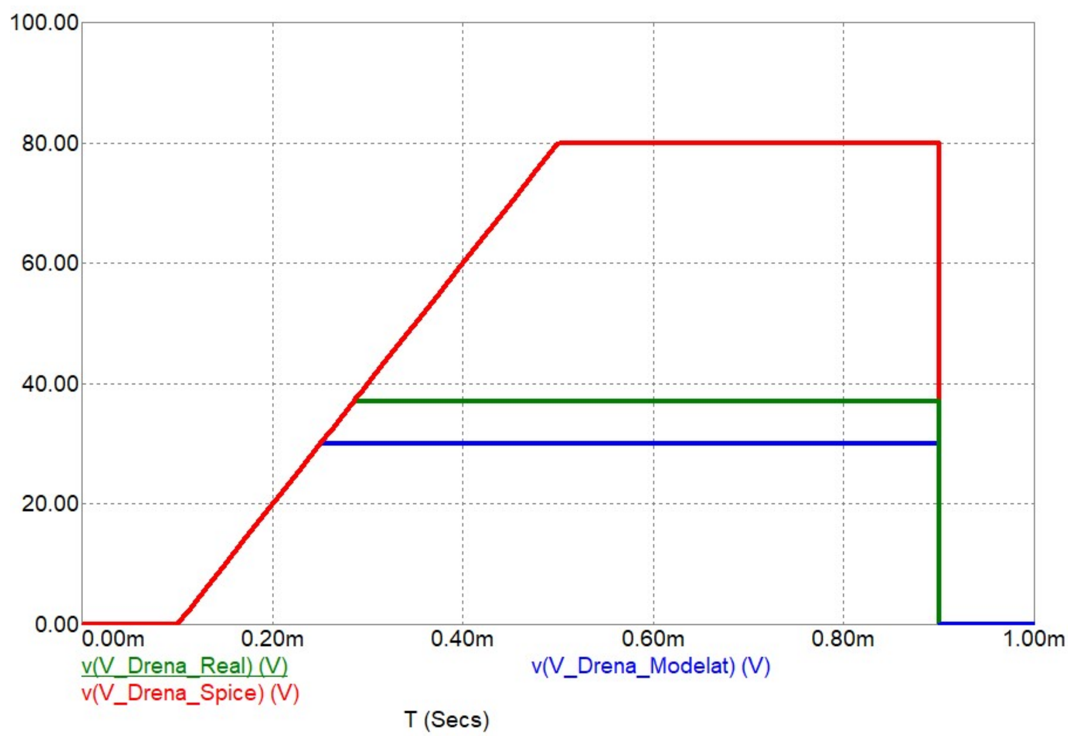


Fig. 4.4.7. Drain-source breakdown voltage of a real MOSFET compared to a SPICE-modeled MOS-FET and a SPICE MOS-FET.

The red graph indicates the drain-source voltage of the SPICE transistor, the blue graph indicates the drain-source voltage of the transistor modeled in SPICE and the green graph indicates the drain-source voltage of the real transistor (measured with the oscilloscope).

When applying a ramp voltage (with an increase from 0V to 80V) in the drain of the transistors, it can be observed that the real transistor cuts the voltage around 37V, the modeled one at 30V (value specified in the catalog data) and the transistor in the SPICE library does not cut this voltage until reaching maximum values (80V). Also studied a higher voltage (up to 1000V) on the transistor in the SPICE library and the result is the same.

4.4.3.5 Validation of the SPICE model – grid-source breakdown voltage

Using the circuit in figure 4.4.3, the grid-source breakdown voltage of a real MOS-FET transistor (IRFH3707) is studied, under the following conditions (imposed in the catalog data):
- Grid-source voltage greater than 20V.

Thus in the circuit in figure 4.4.3 we have:
- The drain is in "air", R2 does not exist in the circuit.
- The resistance value of resistor R1 is 1.3Ω.
- The resistance value of resistor R3 is zero Ω.
- V1 is a pulse type continuous voltage source from 0V to 80V (Initial=0V, V1=80V, TD=100u, TR=400u, TF=0, PW==500u, PER=1m).
- Xreal is a MOS-FET transistor (IRFH3707).

The voltage at the V_Grila_Real point was measured using an oscilloscope (Tektronix DPO 5104B model). The waveform was saved in .CSV format for later use in SPICE.

Using the circuit in figure 4.4.4, the breakdown voltage of a real MOS-FET transistor (IRFH3707) is studied in comparison with a transistor from the SPICE library and a transistor modeled in SPICE under the following conditions:
- The drain is in "air", R2 and R5 do not exist in the circuit.
- The resistance value of resistors R1 and R4 is 1.3 Ω.
- The resistance value of resistors R3 and R6 is zero Ω.
- V1 is a pulse-type voltage source from the SPICE library (Vinitial=0V, V1=80V, TD=100u, TR=400u, TF=0, PW==500u, PER=1m)
- U1 is a user source from the SPICE library that allows loading the voltage waveform in .USR format (voltage acquired with an oscilloscope (Tektronix Model DPO 5104B), with which the grid-source voltage was measured on a real transistor).

After simulating the circuit in figure 4.4.4 using SPICE, the graph in figure 4.4.8 was obtained for the grid-source breakdown voltage.

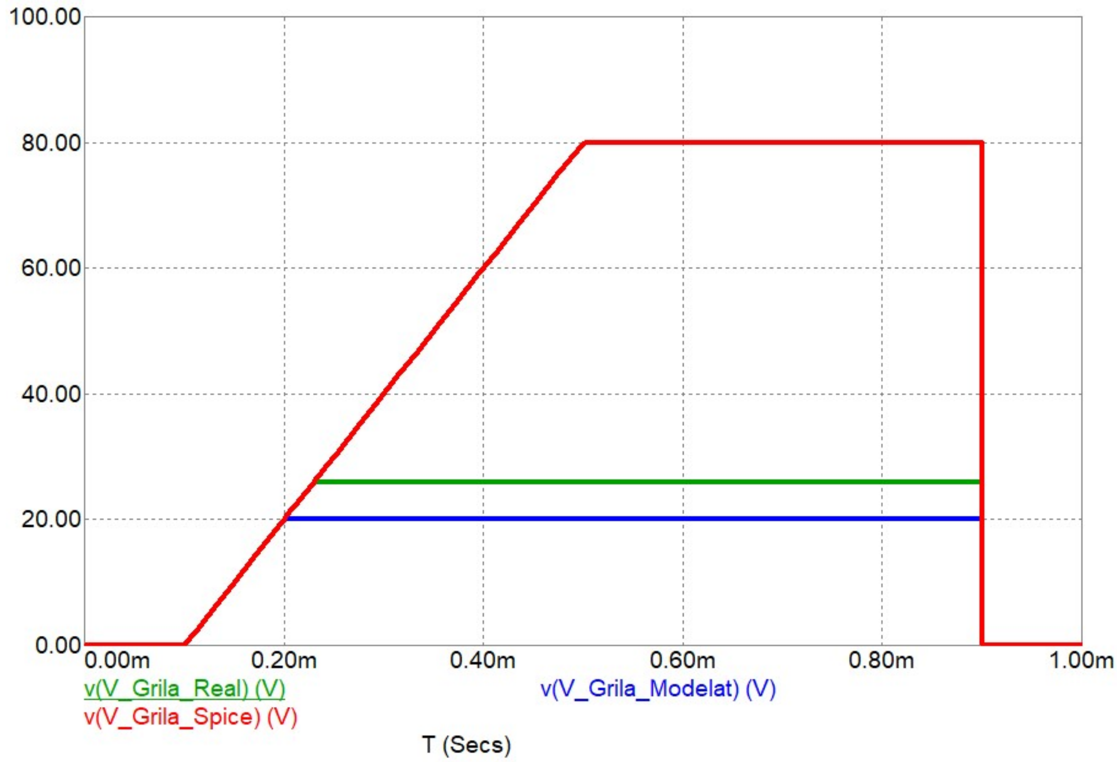


Fig. 4.4.8. Grid-source breakdown voltage of a real MOS-FET compared to a SPICE-modeled MOS-FET and a SPICE MOS-FET.

The red graph indicates the grid-source voltage of the SPICE transistor, the blue graph indicates the grid-source voltage of the modeled transistor in SPICE, and the green graph indicates the grid-source voltage of the real transistor (measured with the oscilloscope).
When applying a pulse voltage (with an increase from 0V to 80V) in the transistor grid, it can be observed that the real transistor cuts the voltage around 26V, the modeled one at 20V (value specified in the catalog data) and the transistor in the SPICE library does not cut this voltage until reaching maximum values (80V). A higher voltage (up to 1000V) was also studied on the transistor in the SPICE library and the result is the same.

4.4.3.6 SPICE model validation – drain-source resistance (25°C)

Ambient temperature 25°C

Using the circuit in figure 4.4.3, the variation of the drain-source resistance of a real MOS-FET transistor (IRFH3707) is studied according to the variation of the grid-source voltage and the ambient temperature, under the following conditions (imposed in the catalog data):

- The drain current is 12A (ID=12A).
- The ambient temperature is 25°C.

Thus in the circuit in figure 4.4.3. we have:

- The resistance value of resistor R2 is 1Ω
- The resistance value of resistor R1 is 1.3Ω
- The resistance value of resistor R3 is zero Ω.
- V1 is a continuous voltage source (V1=0V□10V)
- V2 is a continuous voltage source (V2=12V)
- Xreal is a MOS-FET transistor (IRFH3707)

- The voltage at the V_Grila_Real point was measured using an oscilloscope (Tektronix DPO 5104B model). The waveform was saved in .CSV format for later use in SPICE.
- The ambient temperature is 25°C (the circuit is inserted in a climatic chamber).

Measure and save (in .CSV format) the voltage at the point V_Grila_Real at a variation of the grid voltage from 0V to 10V with a step of one volt.

Using the circuit in figure 4.4.4. the drain-source resistance variation of a real MOS-FET transistor (IRFH3707) is studied compared to a transistor from the SPICE library and a transistor modeled in SPICE under the following conditions:

- The resistance value of resistors R2 and R5 is 1Ω.
- The resistance value of resistors R1 and R4 is 1.3 Ω.
- The resistance value of resistors R3 and R6 is zero Ω.
- V1 is a continuous voltage source (V1=0V□10V)
- V2 is a continuous voltage source (V2=12V)
- U1 is a user source from the SPICE library that allows loading the voltage waveform in .USR format (the voltage acquired with an oscilloscope (Tektronix Model DPO 5104B), with which the grid-source voltage was measured on a real transistor).

After simulating the circuit in figure 4.4.4 using SPICE's "Transient" and "Stepping" functions, the graph in figure 4.4.18 was obtained for the drain-source resistance variation. Figure 4.4.9 uses a 0mΩ to 30mΩ scale on the 'Y' axis to display most drain-source resistances. For the real transistor and for the modeled one, a great closeness of the drain-source resistance values is observed, in addition, there are several values of the drain-source resistance below 2mΩ. For the transistor in the SPICE library, the drain-source resistance variation is between 12mΩ and 17mΩ (the nominal value declared in the catalog data is 9.4mΩ).

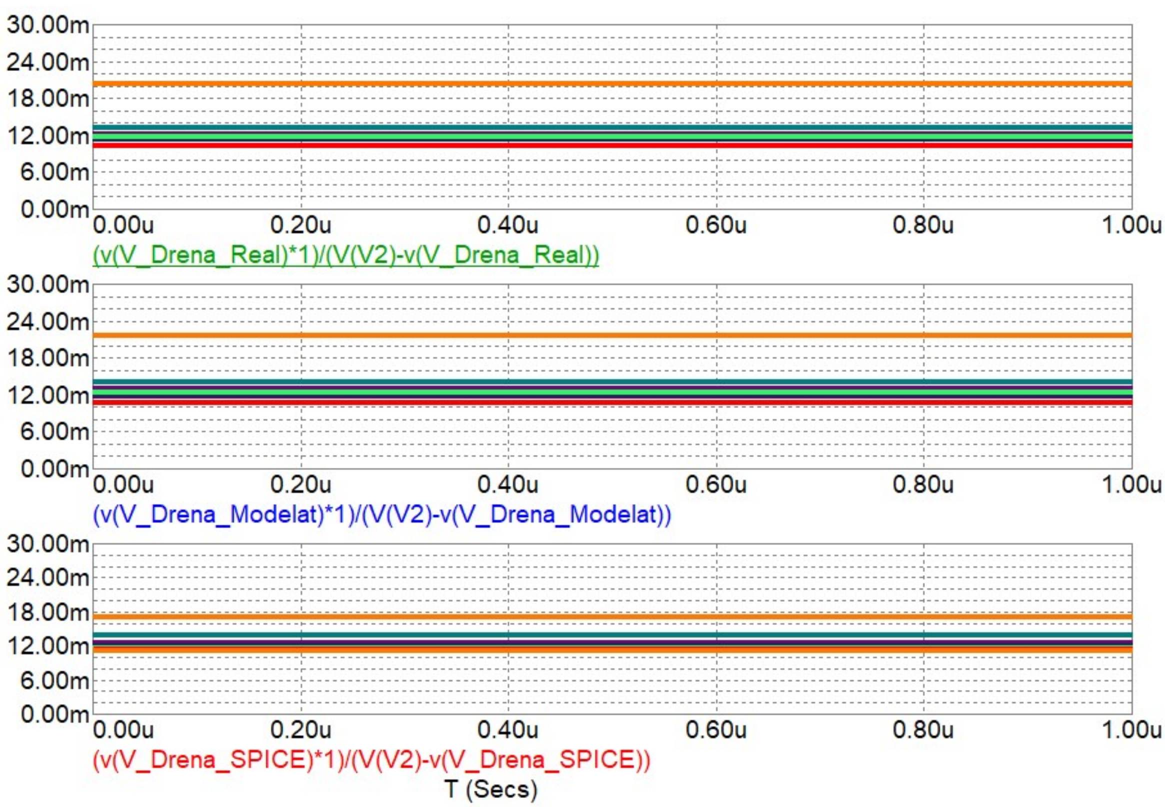


Fig. 4.4.9. Variation of drain-source resistance of a real MOS-FET compared to a SPICE-modeled MOS-FET and a SPICE MOS-FET as a function of grid-source voltage (ambient temperature is 25°C).

5 CONCLUSIONS

In this doctoral thesis, high-performance procedures for the simulation and analysis of complex nonlinear electronic circuits were presented.

The world's most used SPICE circuit simulators have a large library of electrical and electronic components, in addition, major electronic component manufacturers provide a SPICE model for their components. But from the tests carried out in this paper, these proposed SPICE models do not seem to be complete because the simulators are focused on operating in the nominal area of the components. Another weak point observed in this study is the variation of component parameters with temperature. They are SPICE models that do not have a parameter variation with temperature or this variation is not similar to the variation tested or declared in the component catalog data.

SPICE models proposed in this paper can be widely used in the automotive industry, where the performance of components at the limit is much more important than in the nominal mode of operation.

In the automotive field, circuit operation towards limits (upper and lower) is used to calculate circuit protections. Due to the degree of safety required in the automotive field, most circuits are protected when operating towards the limits.

New algorithms and computational programs dedicated to the analysis of nonlinear analog circuits have been developed, based on original SPICE models with a high degree of generality, and which can be widely used in the automotive industry, where the operation of boundary components is much more important than in nominal operating mode.

5.1 The original contributions made by the author in this doctoral thesis

In the following, the main original contributions made by the author in this doctoral thesis are presented:

- The paper begins with carefully selected and up-to-date documentation on the operating principles and performance of SPICE models used in the automotive industry. The main advantages and disadvantages of using SPICE programs are presented.

- • In paragraph 3.1, starting from the equivalent circuit of a real capacitor, a new SPICE model of the nonlinear voltage-controlled capacitor is created. In this model, the variation of static capacitance with voltage, temperature and aging is considered. The accuracy of the SPICE model created is done by comparing it to the behavior of the actual capacitor and to the behavior of the SPICE capacitor proposed by the component manufacturer for various circuits. It is tested in a charging, discharging and oscillating circuit at a temperature of 25°C considering a component under 10 hours of operation The following conclusions are observed after the tests:
    - At a temperature of 25°C the operation of the created SPICE model is closer to the operation of the real capacitor than the model available in the SPICE library.

- In paragraph 3.2, starting from the equivalent circuit of a real coil, a new SPICE model of the real nonlinear current-controlled coil is created. In this model, the variation of inductance with the variation of current through the coil and with temperature is considered. The accuracy of the SPICE model created is done by comparing it to the behavior of the actual coil and to the behavior of the SPICE coil proposed by the component manufacturer for various circuits. It is tested in an oscillating circuit at a temperature of 25°C considering a component under 10 hours of operation. Following the tests, the following conclusions can be observed:

  - At a temperature of 25°C, the operation of the created SPICE model is closer to the operation of the real coil than the model available in the SPICE library.

- In paragraph 3.3, starting from the equivalent circuit of a real resistor, a new SPICE model of the non-linear resistor is created. The accuracy of the created SPICE model is done by comparing it to the behavior of the real resistor and to the behavior of the SPICE resistor in the component library for various circuits. The resistive divider is tested in a circuit at a step voltage increase and at a sinusoidal voltage at a temperature of 25°C considering a component under 10 hours of operation. Following the tests, the following conclusions can be observed:
  - At a temperature of 25°C and a step increase in supply voltage the behavior of the created SPICE model and the model available in the SPICE library is identical to the behavior of the real resistor.
  - At a temperature of 25°C and a supply with a sinusoidal voltage of 50MHz the operation of the created SPICE model is closer to the operation of the real resistor than the model available in the SPICE library and is a worse case of it.

- In paragraph 4.1, starting from the working principle of a rectifier diode and considering the information from the catalog data [31] of a rectifier diode, a new equivalent model of a real rectifier diode was created. Based on this model, a new SPICE model is created that contains all the parameters of the real diode model as well as their variation with the variation of the current through the diode and with the temperature of the diode. The accuracy of the SPICE model created is done by comparing it to the behavior of the actual rectifier diode and to the diode behavior in the SPICE library for various circuits. The SPICE model is validated in direct conduction by applying a constant current and in forward conduction at a temperature of 25°C. Following the tests, the following conclusions can be observed:
  - The diode is tested in direct conduction at a current of approximately 1mA, 10mA and 100mA. In all three situations the voltage drop across the diode created in SPICE represents a worse case than the voltage drop across the real diode and the diode in the SPICE library. This model is particularly helpful in the automotive industry where operating limits are a major factor in design. Of course, the diode can be designed to have the lower limits as output (if they are available in the catalog data).
  - In reverse conduction it is observed that the value of the current through the diode in the SPICE library is zero, the value of the current through the actual diode is indeed of small value and much closer to zero than the value declared in the catalog data. Of course the value of the current through the diode modeled in SPICE has exactly the value declared in the catalog data which in this case is much higher than the value of the current through the real diode. But even in this case, the ability of the created model to be used in SPICE simulations in the automotive industry to simulate the maximum loss current of an electronic circuit is observed.

- In paragraph 4.2, starting from the working principle of a Zenner diode and considering information from the catalog data [30] of a Zenner diode, a new equivalent model of a real Zenner diode was created. Based on this model, a new SPICE model is created that contains all the parameters of the real diode model as well as their variation with the variation of the current through the diode and with the temperature of the diode. The accuracy of the created SPICE model is done by comparing it to the behavior of the real Zenner diode and to the diode behavior in the SPICE library for various circuits. The SPICE model is validated in direct and indirect conduction by applying a constant current at a temperature of 25°C. Following the tests, the following conclusions can be observed:
    - In direct conduction, the SPICE model created at the lower and upper limits is tested, thus having two results. It is observed that the SPICE model indicates the extremes of the Zenner diode behavior. Thus, in the given situation, we obtain the worst possible lower and higher case.
    - The diode is tested in indirect conduction at a current of approximately 1mA. In this situation the voltage drop across the diode created in SPICE represents a worse case than the voltage drop across the actual diode and the diode from the SPICE library. This model is particularly helpful in the automotive industry where operating limits are a major factor in design. Of course the diode can be designed to have the lower limits as output (if they are available in the catalog data).

- In paragraph 4.3, starting from a model already existing in the SPICE library, the model parameters are verified with a newly created test model and a method is proposed for extracting the main parameters of the bipolar transistor from the catalog data. Following the tests, the following conclusions can be observed:
    - The collector and emitter resistances depend on the value of the collector current. In the SPICE model these values are fixed. Thus if we know the collector current we can modify these resistances creating a SPICE model closer to reality.
    - Base resistance has a large variation with input frequency and is important to account for especially at high frequency.
    - New values for BF and IKF parameters have been identified in DC analysis and transient analysis that better respect the declared values in the catalog data.
    - New values for R_{BB}, R_{EE} and TF parameters were identified in the AC analysis that better respect the declared values in the catalog data.

- In paragraph 4.4, starting from the operating principle of an "n" channel MOS-FET transistor and considering the information from the catalog data [35] of a MOS-FET transistor, a new equivalent model of a MOS-FET transistor was created FET real. Based on this model, a new SPICE model is created that contains all the parameters of the real MOS-FET transistor model as well as their variation with the variation of the current through the transistor and with the temperature of the transistor. Following the tests, the following conclusions can be observed:
    - The grid-source voltage shape of the created SPICE model is very close to the real transistor voltage shape and represents a worse case. The transistor in the SPICE library having a grid-source voltage with a more ideal ramp.
    - The value of the drain-source leakage current at a temperature of 25°C for a real transistor is about 550nA, the transistor in the SPICE library is 254nA and the transistor modeled in SPICE is 1µA (value also found in the component catalog data) . It can be seen that the transistor created in SPICE represents the

worst case. This helps in calculating the maximum leakage current in a complex scheme.

- The drain-source breakdown voltage of a real transistor is around 37V, the one created in SPICE is 30V (value specified in the catalog data) and the transistor in the SPICE library is infinite. In this situation we cannot see in the simulated circuit a possible overvoltage that could affect the circuit.
- The grid-source breakdown voltage of a real transistor is around 26V, the one created in SPICE is 20V (value specified in the catalog data) and the transistor in the SPICE library is infinite. In this situation we cannot see in the simulated circuit a possible overvoltage that could affect the circuit.
- When testing the grid-source resistance variation, a clustering of transistor resistance values from the SPICE library is observed regardless of the applied grid-source voltage. Noting that the smallest transistor resistance value in the SPICE library is greater than the nominal value declared in the catalog data. That's assuming a large simulation error using this SPICE model.

5.2 Future research directions

Considering what has already been studied in this thesis, several future directions for further research can be identified:

- Create and study a SPICE model of the real suppressor diode. Diode frequently used in the automotive industry.
- Creation of a method for extracting the main parameters used in SPICE for a PNP bipolar transistor.
- Create and study a SPICE model for an NPN bipolar transistor.
- Create and study a SPICE model for a PNP bipolar transistor.
- Create and study a SPICE model for a "p" channel MOS-FET transistor.
- Testing of components at a temperature of -60°C, -40°C, 125°C, 150°C depending on the minimum or maximum operating temperature of each component.
- Creation of a tool to extract the operating limits of a circuit automatically using the proposed SPICE models.

6 BIBLIOGRAPHY

[1] Nagel, L. W, and Pederson, D. O., SPICE (Simulation Program with Integrated Circuit Emphasis), Memorandum No. ERL-M382, University of California, Berkeley, Apr. 1973.

[2] *Nagel, L. W.; Rohrer, R. A. (August 1971). "Computer Analysis of Nonlinear Circuits, Excluding Radiation", IEEE Journal of Solid-State Circuits, vol. 6, no. 7, pp. 166-184, 1971.*

[3] M. Iordache, L. Dumitru, “Simularea asistată de calculator a circuitelor analogice. Algoritmi şi tehnici de calcul” [Computer aided simulation of analog circuits] , Editura POLITEHNICA Press, Bucureşti, 2014.

[4] M. Iordache, L. Mandache, “Analiza asistată de calculator a circuitelor analogice neliniare” [Computer aided analysis of nonlinear analog circuits], Editura POLITEHNICA, Bucureşti, 2004.

[5] Charles K. Alexander, Matthew N. O. Sadiku, “Fundamentals of electric circuits”, Published by McGraw-Hill, New York, 2012.

[6] LTspice Tutorials; Mike Engelhardt, October 2011.

[7] Voltage characteristics of electronics capacitance https://article.murata.com/en-us/article/voltage-characteristics-of-electrostatic-capacitance

[8] The temperature characteristics of electrostatic capacitance https://article.murata.com/en-sg/article/temperature-characteristics-electrostatic-capacitance

[9] Capacitance change with aging https://www.digikey.com/en/articles/what-is-the-capacitance-of-this-capacitor

[10] https://www.murata.com/en-eu/products/emc/emifil/knowhow/basic/chapter06-p6

[11] Impedance and ESR vs frequency characteristics for a capacitor https://article.murata.com/en-us/article/impedance-esr-frequency-characteristics-in-capacitors

[12] KEM_C1005_Y5V_SMD.pdf https://content.kemet.com/datasheets/KEM_C1005_Y5V_SMD.pdf

[13] SPICE model for KEMET Capacitor, http://ksim.kemet.com/Plots/SpicePlots.aspx

[14] Different Type of Inductors https://www.agilemagco.com/wpcontent/uploads/Different-Types-of-Inductors.pdf

[15] Frequency impedance characteristics of inductor https://www.murata.com/en-u/products/emc/emifil/knowhow/basic/chapter06-p6

[16] Characteristics of compact metal alloy power inductors https://article.murata.com/en-us/article/characteristics-of-compact-metal-alloy-power-inductors

[17] Bobine; http://ep.etc.tuiasi.ro/files/MCCP/Mccp11-L.pdf

[18] LQH32PZ101MN0.pdf https://www.murata.com/en-us/api/pdfdownloadapi?cate=cgInductors&partno=LQH32PZ101MN0%23

[19] SPICE model for MURATA Inductor, https://ds.murata.co.jp/simsurfing/powerinductor.html?lcid=en-us

[20] TDK-ceramicSMD.pdf https://pccomponents.com/datasheets/TDK-ceramicSMD.pdf

[21] https://www.researchgate.net/publication/323507493_Diode_Models

[22] https://www.diodes.com/design/tools/spice-models/

[23] https://www.coursera.org/lecture/electronics/4-2-models-of-diode-behavior-tfHbH

[24] https://www.eeeguide.com/ac-equivalent-circuit-of-semiconductor-diode/

[25] https://www.researchgate.net/figure/Equivalent-circuit-for-the-rectifier_fig5_2982972

[26] http://fourier.eng.hmc.edu/e84/lectures/ch4/node8.html

[27] https://www.researchgate.net/figure/Nonlinear-transistor-equivalent-circuit-with-capacitor-and-transconductance_fig1_3130846

[28] https://www.daenotes.com/electronics/devices-circuits/hybrid-equivalent-transistor

[29] https://www.afahc.ro/ro/facultate/cursuri/dispo_electro.pdf

[30] BZX84_SERIES (NXP, 2003 Apr 10).

[31] bav99series (Infineon, 2007-09-19).

[32] BC817 (SIEMENS 07.94).

[33] BC817 (NXP Semiconductors, 17 November 2009).

[34] BC817 (ONSEMI, August 2009).

[35] https://www.infineon.com/dgdl/Infineon-IRFH3707-DataSheet-v01_01-EN.pdf?fileId=5546d462533600a40153561a2e0b1e78

[36] https://incompliancemag.com/article/estimating-the-parasitics-of-passive-circuit-components/

[37] **Marius Florin Staniloiu**, Horatiu Samir Popescu, Bogdan Glod, Mihai Iordache, "*SPICE model of a real capacitor: Capacitive feature analysis with voltage variation*" (EPE2020), Iași ROMÂNIA, Date of Conferences: 22-23 October 2020, Iași România, Added to IEEE Xplore: 18 February 2021, DOI: 10.1109/EPE50722.2020.9305554, **INSPEC Accession Number:** 20470036, 978-1-7281-8126-4/20/$31.00 ©2020 European Union, Publisher: IEEE, pp. 333 – 338.

[38] **Marius Florin Staniloiu**, Horatiu Samir Popescu, Bogdan Glod, Mihai Iordache, "*SPICE model of a Real Coil. Inductance feature analysis with current variation*" (EPE2020), Iași ROMÂNIA, Date of Conferences: 22-23 October 2020, Iași România, Added to IEEE Xplore: 18 February 2021, DOI: 10.1109/EPE50722.2020.9305677, **INSPEC Accession Number:** 20470072, 978-1-7281-8126-4/20/$31.00 ©2020 European Union, Publisher: IEEE, pp. 442 – 446.

[39] Mihai Iordache, Horatiu Samir Popescu, Ionela Vlad, **Marius Florin Staniloiu**, "*ACAP – Analogic Circuit Analysis Program*" (Bucuresti 2021), Date of Conferences: 25-27 March 2021, 12th International Symposium on Advanced Topics in Electrical Engineering (ATEE), Added to IEEE Xplore: 12 May 2021, DOI: 10.1109/ATEE52255.2021.9425307, **INSPEC Accession Number:** 20691709, ISBN: 978-1-6654-1878-2/21/$31.00 ©2021 IEEE, **WOS:000676164800143,** Publisher: IEEE, 6 pages.

[40] **Marius Florin Staniloiu**, Horatiu Samir Popescu, Georgiana Rezmerita, Mihai Iordache "*The Equivalent Circuits Thevenin and Norton*", Scientific Bulletin of the Electrical Engineering

Faculty – Year 21 No.2 (45), Sciendo, ISSN 2286-2455, DOI: 10.2478/sbeef-2021-0021, pp. 40-48.

[41] Victor Bucata, Mihai Iordache, Ionela Vlad, Horatiu Popescu, **Marius Florin Staniloiu** "*Wireless Power Transfer Systems: Thevenin Equivalent Circuits for Parallel-Series and Paralle-Parallel Magnetic Resonator Configurations*" (ICATE 2021), Craiova ROMÂNIA, Date of Conferences: 27-29 May 2021, Added to IEEE Xplore: 28 June 2021, DOI: 10.1109/ICATE49685.2021.9464974, **INSPEC Accession Number:** 20780269, 978-1-7281-8035-9/21/$31.00 ©2021 IEEE, **INSPEC Accession Number:** 20895674, Publisher: IEEE, 6 pages.

[42] Victor Bucata, Mihai Iordache, Ionela Vlad, Horatiu Popescu, **Marius Florin Staniloiu** "*Thevenin Equivalent Circuits for Magnetic Coupling Rezonators (Series-Series, Series-Parallel) in Wireless Power Transfer System*" (ICATE 2021), Craiova ROMÂNIA, Date of Conferences: 27-29 May 2021, Added to IEEE Xplore: 28 June 2021, DOI: 10.1109/ICATE49685.2021.9464933, **INSPEC Accession Number:** 20895674, 978-1-7281-8035-9/21/$31.00 c2021 IEEE, **INSPEC Accession Number:** 20895674, Publisher: IEEE, 6 pages.

[43] Mihaela Grib, Mihai Iordache, Alexandru Radu Grib, Horatiu Popescu, Ovidiu Laudatu, **Marius Staniloiu** "*The Use of Thevenin, Norton and Hybrid Equivalent Circuits in The Analysis and Polarization of Nonliniar Analog Circuits*" (EPE 2022), Iași România, Date of Conferences: 20-22 October 2022, Added to IEEE Xplore: 25 November 2022, DOI: 10.1109/EPE56121.2022.9959871, **INSPEC Accession Number:** 22330770, 978-1-6654-8994-2/22/$31.00 ©2022 European Union, **WOS:000709089900011.** Publisher: IEEE, pp. 198-207.

[44] **Marius Florin Staniloiu**, Horatiu Samir Popescu, Georgiana Rezmerita, Ionela Vlad, Mihai Iordache, "*SPICE model of a real Zener diode tested at room temperature*" (EPE2022), Iași România, Date of Conferences: 20-22 October 2022, Added to IEEE Xplore: 25 November 2022, DOI: 10.1109/EPE56121.2022.9959871, **INSPEC Accession Number:** 22330715, 978-1-6654-8994-2/22/$31.00 ©2022 European Union, Publisher: IEEE, **WOS:000709089900001,** pp. 182-186.