Universidade de Aveiro
Departamento de Eletrónica, Telecomunicações e Informática

System Identification

Report – Work Assignment 1

Determination of Functional Parameters of a Transistor

Instructor: Telmo Cunha

MIEET

Authors: Rui Carapinha, 79970

Vadson Culanda, 79988

Date: 26/10/2019

1. Introduction and Objectives

In this work we intend to perform several tests on a transistor, using the configuration in Fig. 1, through the "TransistorTest_IS" function which allows us to polarize the transistor with a voltage V_{DS} (between 25V and 35V) and a voltage V_{GS} (between -1.2V and -1.4V) and apply voltages of $v_i(t)$ and $v_o(t)$. With these tests we intend to calculate the linear model that simulates the behavior for the small signal of the transistor of Fig. 2.

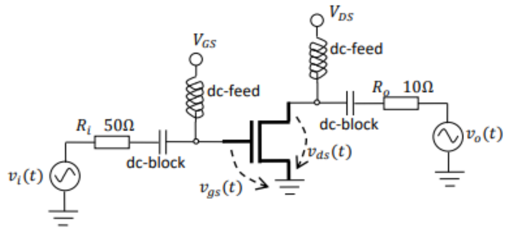

Figure 1 - Configuration for transistor testing

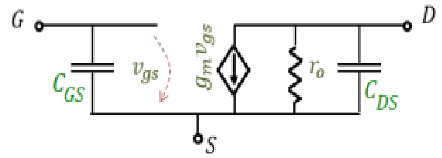

Figure 2 - Model of small-signal of the transistor

The specific objectives of the assignment are as follows:

- Determination of the 4 coefficients shown in Fig.2, namely, C_{GS}, r_o, C_{DS} and g_m.
- Validation of the obtained model and comparation with the test function.

2. Extraction of Modelling Parameters

The strategy adopted for the extraction of the parameters was based on an analytical analysis using Laplace transforms (to reach the necessary parameters), Z transforms (was only used to reach the best h (Sampling Period) which gives the best NMSE to each system) and, of course, MATLAB. As such, the circuit was analyzed for 2 different situations, which allowed the study of 3 different systems. To obtain the parameters we decide to use a step, although we could've used a ramp, parabolic or sinusoidal signal we decided to use a step, because all of these signals have different graphs and different effects, and we're interested in seeing the transient response of the system, and the step gives us the idea about how the system replies to an interruption and the system stability. He also defines the sudden change in properties of the actual signal.

2.1. System 1 and 2, $V_o = 0$:

The circuit was the following:

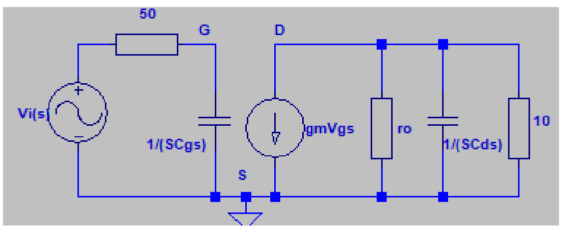

Figure 3- System 1 and 2 circuit

The next step is to obtain the circuit transfer functions. The circuit on the left can be regarded as a resistive divider where the voltage V_{gs} is given by:

$$V_{gs} = \left(\frac{\frac{1}{sC_{gs}}}{50 + \frac{1}{sC_{gs}}}\right) V_i \leftrightarrow \frac{V_{gs}}{V_i} = \frac{\frac{1}{sC_{gs}}}{50 + \frac{1}{sC_{gs}}} = \frac{\frac{1}{50C_{gs}}}{s + \frac{1}{50C_{gs}}}$$

This model is required to obtain the value of C_{gs}.

To obtain the right h we applied the Z transform (knowing that $\frac{1}{s+a} = \frac{1}{1-e^{-ah}Z^{-1}}$) and we got the following: $\frac{V_{gs}}{V_i} = \frac{\frac{1}{50C_{gs}}}{1-e^{-\frac{h}{50C_{gs}}}Z^{-1}}$, after this we just need to check where the pole is located (the best situation is at 0.5, so we tried to adjust our h to get the best possible solution). Using the script, we simulated the system with the right h and we got the value of the Numerator and we got that $N1 = \frac{1}{50C_{gs}} \leftrightarrow C_{gs} = \frac{1}{50*5.9939e8} = 33.367pF$

In the right circuit the voltage V_{ds} can be obtained using the Ohm's Law in the charge, as such we will have: $V_{ds} = \left(r_o// \frac{1}{SC_{ds}} // 10\right) g_m V_{gs} \leftrightarrow \frac{V_{ds}}{V_{gs}} = \left(r_o// \frac{1}{SC_{ds}} // 10\right) g_m$ The next step is to develop the transfer function and thus obtain: $\frac{V_{ds}}{V_{gs}} = \frac{-\frac{g_m}{C_{ds}}}{s+\frac{10+r_o}{10 r_o C_{ds}}}$.

As similarly to the previous system, using the script in annex, we simulated the system and we got that: $N2 = -\frac{g_m}{C_{ds}} \leftrightarrow g_m = -N2 * C_{ds}$, once that the value of C_{ds} is still unknown, it isn't possible to calculate g_m but both values will be calculated after analyze of the last system, system 3.

2.2. System 3, $V_i = 0$:

The circuit was the following:

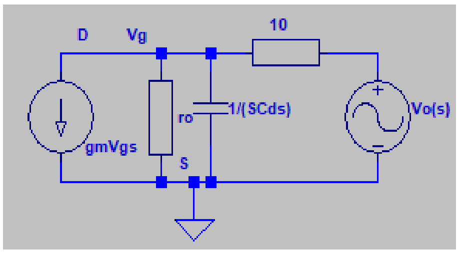

Figure 4- System 3 circuit

This circuit can also be seen as a resistive divider where the relationship between the voltages can be given by: $\frac{V_{ds}}{V_o} = \frac{r_o//\frac{1}{sC_{ds}}}{10+r_o//\frac{1}{sC_{ds}}} \leftrightarrow \frac{V_{ds}}{V_{gs}} = \frac{\frac{1}{10C_{ds}}}{s+\frac{10+r_o}{10r_oC_{ds}}}$

Similarly to the other processes we simulated the system corresponding to the transfer function V_{ds}/V_o and we got the value of the second component of the denominator (D3.2) and the numerator (N3) and we know

$$\text{that: } N3 = \frac{1}{10C_{ds}} \leftrightarrow C_{ds} = \frac{1}{10*2.0333e9} = 49.18pF \rightarrow g_m = -(-2.5895e10) * C_{ds} = 1.2735.$$

Knowing that:

$$D3.2 = \frac{10+r_o}{10r_oC_{ds}} \leftrightarrow r_o = \frac{10}{2.1018e9*10r_oC_{ds}-1} = 296.8687\Omega$$

3. Validation of the obtained model

In the previous point it was referred that the strategy adopted for the extraction of FET coefficients consists of the analysis of 3 systems with different test conditions. As such, the analysis of the accuracy of the model and the

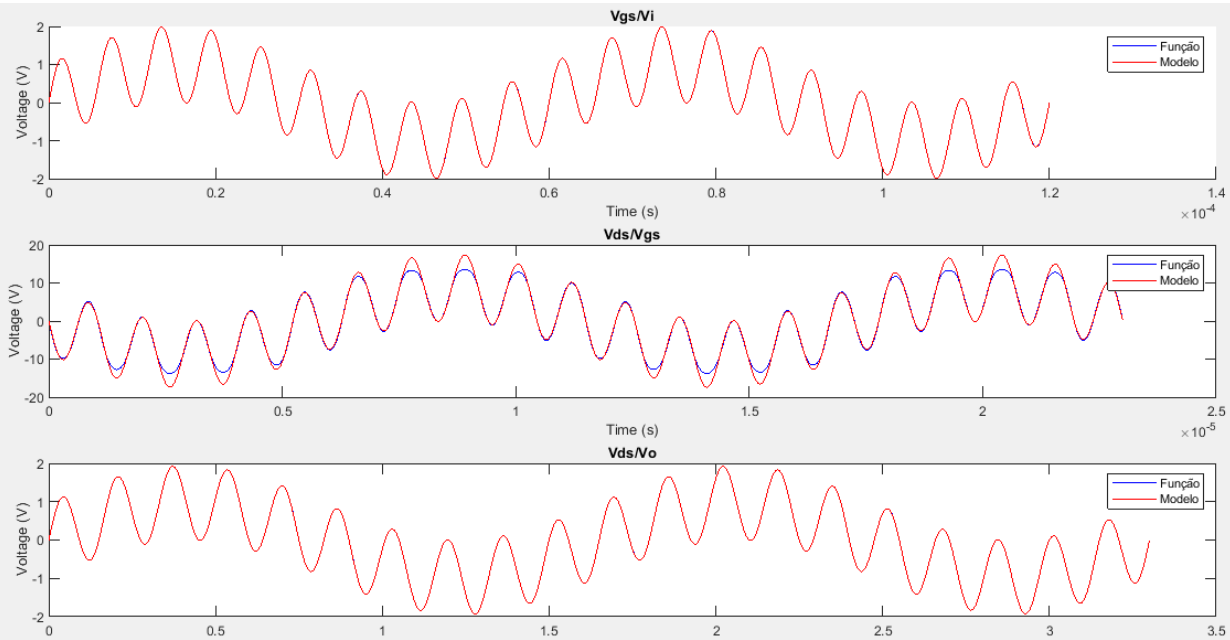

Figure 5 - Tests made to the model

comparison with the test function is made for each of these systems separately. Comparing the response of the model and the test function for the same input signal gives the following graphs:

Analyzing the charts, we can infer that the model looks well built, however these charts are not enough to ensure the accuracy of the model by itself. We can also analyze the error signals, which correspond to the difference between the output signal of the test function and the output signal of the model, when both are excited by the same input signal:

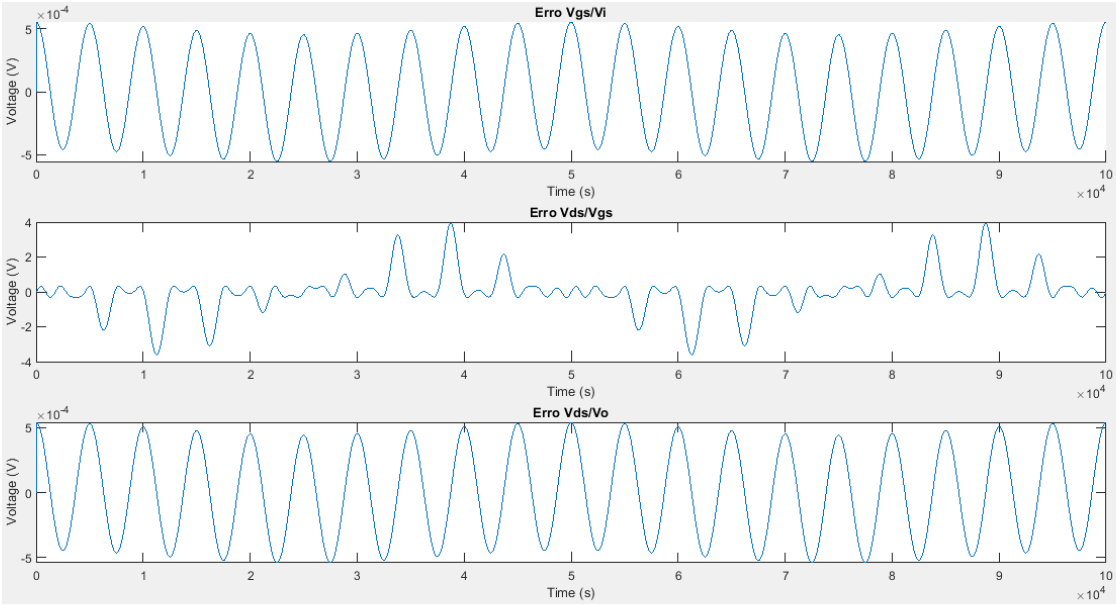

Figure 6 - Graphs of Error

There are several metrics used to characterize the accuracy of a model, however, we focused on only one of them, namely NMSE (Normalized Mean Squared Error) and the values obtained were NMSE_S1=-118.9dB, NMSE_S2=-86.89dB, NMSE_S3=-118.54dB and based on these values we can state that the systems are well modelled since low NMSE values mean that a good performance in space and time.

4. Conclusion

This work was important to us to get familiarize with the different types of systems modelling, namely, physical (white box modeling) and behavioral (black box modeling). A FET is modeled using grey box modeling, since its modeling is a mixture between the two types of modeling. The physical part is reflected in the topology of the model while the components are represented through behavioral models. In the first part of this assignment we had as objective the extraction of the parameters corresponding to the circuit that simulates the behavior for small signal of a FET, as such, we analyzed the circuit transfer function for 3 systems subject to different conditions and through them and with the help of MATLAB and the Z transform we were able to determine the unknown parameters. We noticed that we must be very careful when choosing the value of the sampling interval (h) for each of the systems since this value influences the calculation of the value of the extracted parameters (C_{GS}, r_o, C_{DS}, g_m). In the second part of the work we analyzed the accuracy based on the NMSE metrics and concluded that the model fits the behavior of the system since it presents a very low NMSE value below -40dB. In short, we conclude that in the modeling of a system we should take into account several factors, namely, the type of modeling (white, black or grey box), we should identify the parameters to calculate and arrange the appropriate strategy to proceed with their extraction and finally we should use the appropriate metric to validate the accuracy of the model.