Introduction in simulation of passive electronic circuits

1. Laboratory scope:

To familiarize the students with Computer Aided Design (CAD) methods related to electronic schematic in order to simulate it’s operation. Introduction to virtual realization of an electronic schematic drawing.

2. Theoretical introduction

In order to minimize the time and cost it involves making an electronic circuit, and for testing, specials software programs were developed. These programs enable the simulation of circuit operation and permit the removal or correction of design mistakes, before the necessary components being really purchased. In this way, a schematic designs realized using computer, represents actually a virtual circuit.

OrCAD PSpice is a simulation program that models the behavior of circuits containing analog devices. OrCAD Capture is used as input for the schematic. It can be said that PSpice is a circuit “breadboard software” that you can use for testing and perfecting the schematic circuit before physically touching any part.

PSpice permits also analog behavioral modeling, so that we can describe the functional circuit blocks using mathematical expressions and functions.

PSpice simulates the circuit and calculates its parameters. Also, PSpice can transfer the data to the PROBE program for graphics visualization, so that the user can view various waveforms such as taking measurements with an oscilloscope.

The elements of the circuit (components) are represented by their usual standard electric symbols.

PSpice has numerous preset component models whose parameters can be varied to optimize a specific device.

PSpice has analog and digital libraries of standard components that make it useful in a wide range of applications.

Components are grouped into functional libraries such as ANALOG.OLB containing resistors, capacitors, inductors, etc. Other components whose behavior can be modeled and simulated in PSpice are respectively: diodes, bipolar transistors, MOSFETs, IGBTs, transmission lines, magnetic cores.

PSpice permits the changing of a large number of parameters (frequency, temperature, etc.) of components, not just the rated value of characteristic parameters.

To test the functionality of a schematic we need measuring devices such as ammeters, voltmeters, multimeters, oscilloscopes. The computer allows us to have access to such tools as virtual measurement equipment. In a real testing environment we need access to everything, from measurement tools, such as those in Figure 1, electronic components and support for their connection as shown in Figure 2. In comparison, when testing the circuit using
computer, everything is contained in the program dedicated to the simulation of circuit operation and the test environment will look like in Figure 3.

![Fig. 1 Measuring apparatus](image1)

- a. Oscilloscope
- b. digital multimeter
- c. panel mount voltmeter

![Fig. 2 Laboratory power supply](image2)

- a. Voltage source
- b. capacitors
- c. resistors
- d. printed circuit board (FR4)

![Fig. 3 Schematic drawing](image3)

- a. Schematic drawing and measurement point (voltage)
- b. Amplitude(Gain)-frequency characteristic

![Fig. 3 Virtual instruments](image4)

- a. Schematic drawing and (virtual) voltmeter
- b. (virtual) oscilloscope
3. Work procedure

Often in practical realization of electrical circuits appear some discrepancies between what is desired to be achieved and what has really been achieved. To eliminate these discrepancies is a good idea that the wiring diagrams (schematics) be tested and perfected before being made physically. In an initially stage, prototypes were realized and afterwards were tested and perfected, which method involves practical realization of physical circuit so certain material costs were involved. Subsequently, specialized software tools were developed, that can do the same procedure but without the use of components, test boards, measurement devices. This way, the risk to destroy the hardware elements is eliminated, in the event that some design flaws exist.

One of such programs is PSpice software. It allows the simulation and testing of components by electric schemes based on the models in its library.

For the physical realization of a circuit we need circuit schematic, circuit components, test board, soldering iron to connect the components together, measuring devices for testing functionality etc.

When testing the functionality of a circuit using computer we need the wiring diagram, specific software for schematic drawing and circuit simulation and some knowledge of how this program works.

The followings present the computer based simulation of some basic schematic circuits.

3.1. Simulation of a voltage divider using OrCAD Capture CIS-Lite

The Voltage Divider is a circuit that comprises two resistors connected in series and supplied with a voltage source. On each resistor falls a fraction of the supply voltage, depending on resistor value. The figure below illustrates a resistive voltage divider.

![Voltage Divider Circuit Diagram](image)

**Fig. 4** The voltage divider.

In this circuit, the input voltage $V_1$ is divided by the two resistors $R_1$ and $R_2$, which form the resistive voltage divider. The output voltage $V_0$ can be calculated using the formula:

$$V_o = \frac{R_2}{R_1 + R_2} V_1$$

The following sections will be presented the steps to follow in order to simulate the circuit.
3.1.1. Creating a Project in OrCAD Capture CIS-Lite

1. Start OrCAD Capture CIS-Lite;

2. From the File menu of the program Capture, we choose the command New, and then Project:

3. We introduce a name for the newly created project;

4. We make a mouse click on Browse button to specify the location where the new project will be saved;

5. We select the option Analog or Mixed A/D (this is like an utility program used for PSpice simulations) and we click on OK.

6. In the following opened window we will tick Create a blank project.
7. We click on **Finish** button and the new project will be created (the work page will appear as in Fig. 8).

![Fig. 7. Choosing of project model](image)

3.1.2. **Designing the schematic of the circuit in the workpage**

The toolbar on the right side of the workspace is presented in figure 9. This toolbar is very useful when building circuits.
Building a voltage divider:

1. Choosing the passive components and the power source

From the Analog.olb library choose R/analog, and from the Source.olb choose VDC/VAC/VSIN depending on analysis type. The components are placed on the work area by clicking the desired component and then clicking on the spot where the component should be placed. To deselect a component press ESC.

*Obs.:* To rotate a component, first select it by left clicking on it and then press R.

To add a component, press the Part icon from the toolbar on the right (see fig. 9). To add a new library, press „Add Library” (see fig. 10).
2. To add a ground connection, click on **Ground** (from the toolbar on the right hand side of the workspace) (see fig. 9). A new window, similar to the one below, will open. You should select **0/CAPSYM**:

![Place Ground window](image)

Fig. 11 Placing a Ground symbol

3. After all the components have been placed they should be connected. To connect components, click on the **Wire** icon from the toolbar or simply click W, and then select the terminals of the components to be wired together. Press **ESC** to end mode.

4. To change the value of a component, select the displayed value, and in the Display Properties dialogue box change the value with the new one (only the value, without the measurement unit. For kilo ohms, the value should be followed by the letter „k“, and for mega ohms after the value we should add „meg“);

5. After saving the project, the voltage divider should be similar to the one in Fig.12:

![Voltage divider](image)

Fig.12 Schematic of a voltage divider in OrCAD Capture
**Obs.**: 1. For safety reasons, the project should be saved regularly!

2. If an „*“ appears next to the name of the project or next to the work page, then the project is not saved (see fig. 13).

![Unsaved project](image1)

Fig.13 Unsaved project

After all the components, including the ground symbol, were placed and connected, the circuit can be tested.

### 3.1.3. Setting simulation parameters in PSpice

To switch from Capture to PSpice, that is to create the simulation profile, the following procedure should be followed:

1. From the menu (upper side of the page) select **PSpice, New Simulation Profile**;

![Creating a simulation profile](image2)

Fig.14 Creating a simulation profile

2. Fill in the name of the simulation profile (same as project name) in the New Simulation window (see fig. 15):

![Naming the simulation profile](image3)

Fig. 15 Naming the simulation profile

3. Left click on **Create**;

4. In the Simulation Settings window, choose the desired analysis type from the drop down menu and fill in the relevant information;
5. After all the settings were introduced, left click **Apply** and then **OK**;

6. To edit an existing simulation profile, select **PSpice** from the upper menu and then choose **Edit Simulation Profile**. Here it is possible to modify an existing profile;

7. Markers are placed at the input or/and at the output of a circuit depending on the information needed. Markers can be found on the upper toolbar. The needed marker (V/I/W) is selected with a click and then placed with a left click on the schematic;

![Markers](image)

Fig. 17 Markers

8. To start a simulation, select **PSpice** and then **Run** from the menu, click the **Run** button from the upper toolbar or press **F11**.

For the voltage divider, a PSpice simulation profile will be created for the 4 Analysis Types (Bias Point, DC Sweep, AC Sweep/Noise, Time Domain) respecting the specifications for each analysis type:
1. Bias point:

- To select this analysis simply choose **Bias Point** from the **Analysis type** drop-down menu (see fig. 18);

![Fig. 18 Selecting a Bias Point analysis](image)

- If under **Options**, the option **Temperature (Sweep)** is selected, and if the box next to **Run the simulation at temperature** is ticked, then it is possible to fill in the temperature at which the analysis will be run (for example 50°C). If the **Temperature (Sweep)** option is not marked, the simulation will run at 25°C—considered room temperature;

- Click on **Apply**, and then **OK**;

- After running the simulation (by clicking **Run**), to display the values for the voltages, currents and powers (to determine the operating point) select the **V**, **I**, and **W** from the upper toolbar;

![Fig. 19 Selecting the button for determining the operating point](image)

- After running a **Bias Point** analysis, the circuit should be similar to that in figure 20:
Fig. 20 **Bias Point** analysis of a voltage divider ($R_1=100\Omega, R_2=100\Omega, V_1=10V$)

**Obs.** This analysis is used to compute direct voltages between the nodes of the circuit and ground and the direct currents that flow through the branches of the circuit.

This analysis can be run only if there is a DC power supply in the circuit (such as the direct voltage source $V_{DC}$).

This type of analysis corresponds to measuring voltages and currents with a multimeter. The values are displayed directly on the circuit.

2. **DC sweep:**

- To select this analysis, choose **DC Sweep** from the **Analysis type** drop-down menu. This type of analysis can be performed only if there is a DC power supply in the circuit (such as the direct voltage source $V_{DC}$);

![DC Sweep analysis](image)

Fig. 21 Selecting a **DC Sweep** analysis

- The parameters for this analysis are the following:
  - Under **Sweep variable**, choose **Voltage source** and fill in the name of the source whose value will be varied, exactly as it appears in the circuit. In our
example, there is a DC source called V1 in the circuit. If we wish to analyse the behaviour of the circuit for different values of V1, then in the field Name we should fill in V1.

- Under **Sweep type**, we choose the way in which the parameter selected before is varied (linear, logarithmic, value list). If we choose linear, in the fields **Start value**, **End value** and **Increment**, we input the range within which the value will vary and with which increment respectively. In our example, V1 will have a linear variation between 0 and 10V with a 0.1V increment. This way we can compute the circuit values for current, voltage and power at any of the predetermined values of V1.

![Simulation Settings - diagram de ternaie](image)

**Fig. 22** Adding the parameters for the **DC Sweep** analysis

- Press the **Apply** button, and then **OK**;
- Place voltage markers at the input and output of the circuit;
- Start the simulation by pressing **Run**;
- After running the simulation, the plot for the input and output voltages should look like this:
Fig. 23 Plot of the input and output voltages of the voltage divider generated by PSpice after running a **DC Sweep** analysis.

**Obs.:** This type of analysis is used to determine the **variation** of an electrical quantity (like voltage, current, power) for a circuit as determined by the **variation of the values of the source**.

This type of analysis is useful in determining the operating characteristics of circuit devices.

3. **AC Sweep/Noise:**

- To run this analysis, we should replace VDC with a small signal VAC source. VAC can be found in the **Source.olb** library. VAC sources have two distinct fields:
  - **Vac** – the amplitude of the variable voltage;
  - **Vdc** – mean value of the variable voltage.

- The modified voltage divider should look like the one below:

  ![Voltage divider with a small signal source](image)

  Fig. 24 Voltage divider with a small signal source

- To select this analysis, choose **AC Sweep/Noise** from the **Analysis type** drop-down menu. This type of analysis can be performed only if there is a small signal source in the circuit (such as the signal source **VAC**).
The parameters for this analysis are the following:
- in the field **AC Sweep Type**, set the sweep type for the signal source (in this case, Logarithmic was selected) and in the fields **Start frequency**, **End frequency**, and **Points/Decade** respectively, input the range for the sweep and the number of points/decade (decade = the value range between two consecutive powers of 10). In our example, the variation range for the signal source V1 is [0.1Hz÷10MHZ], and the number of points per decade=10.

- click **Apply**, and then **OK** ;
place a voltage marker at the output and start the simulation by pressing **Run**. After
the simulation process finishes, the plot for the output voltage should look like the one
in fig. 27:

![Plot of output voltage waveform](image)

**Fig. 27** The waveform at the output of the voltage divider generated by PSpice after an **AC Sweep/Noise** analysis

**Obs.**: This type of analysis is used to determine the frequency variation of an electrical
quantity (like voltage, current, power) for a circuit.
This type of analysis is used to determine the frequency response of a circuit.

4. **Time Domain (Transient):**

- To run this analysis, we should replace **VAC** with a **VSIN** sinusoidal source. **VSIN**
can be found in the **Source.olb** library. **VSIN** has three distinct fields:
  - **VOFF** – is the DC offset;
  - **VAMPL** – is the amplitude of the sine wave voltage;
  - **FREQ** – is the frequency of the sine wave (for the voltage divider, we have chosen 1 kHz).

- The modified voltage divider should now look like the one in fig. 28:

![Modified voltage divider](image)

**Fig. 28** The voltage divider with a sine wave source
To select this analysis type, choose **Time Domain (Transient)** under **Analysis type**. This type of analysis can be performed only if there is a signal source (voltage/current) in the circuit with a specified waveform (for example a sine wave source - **VSIN**, or a square wave source - **VPULSE**, etc.).

![Simulation Settings](image)

**Fig. 29** Selecting the **Time Domain** analysis

- The parameters for this type of analysis:
  - In the **Run to time** field, the value of the parameter that controls the number of periods for the signal to be plotted is given. It is computed using the formula: 
    \[ n \times T \]
    where \( n \) = number of plotted periods;
    \( T \) = value of the period of the signal source used in the circuit.
    To determine \( T \):
    \[ T[\text{seconds}] = \frac{1}{FREQ[\text{hertz}]} \]
    In our case, since the frequency of the sine wave signal is 1kHz, \( T \) will be 1ms. We will plot 5 periods so \( n=5 \). It follows that the parameter **Run to time**=5ms (5 milliseconds).
  - In the **Maximum step size** field, the value of the parameter that controls the accuracy of the simulation is given; this parameter should have a much smaller value that the value introduced next to **Run to time**. In our case this parameter will be calculated using:
    \[ \text{Maximum\_step\_size} = \frac{\text{Run\_to\_time}}{100} \]
Click Apply, and then OK;

Place voltage markers at the input and output of the circuit and start the simulation by pressing Run. After the Time Domain simulation finishes, the plot of the input and output voltage generated by PSpice should resemble the one in fig. 31:

---

**Obs:** This type of analysis is used to determine the time variation of an electrical quantity (like voltage, current, power) for a circuit.

This type of analysis is useful to view the variable voltages/currents – to view the waveforms for these electrical quantities; it is like using in practice an oscilloscope.
3.2. Analysing the Frequency response of an RC circuit using PSpice

A high pass filter allows only signals above a given frequency, called the cut-off frequency, undamped or slightly damped. Around the cut-off frequency, the amplitude of the output signal is around 0.707 of the amplitude of the input signal. A high pass filter can be seen in Fig. 31.

The cut-off frequency is computed using:

\[ f_c = \frac{1}{2\pi RC} \]

The RC circuit schematic is drawn in OrCAD (fig. 32):

![Fig. 32 The RC filter](image)

The chosen source is VAC (the parameters have the values from the figure above) from the Source.olb library. The capacitor, similar to the resistor, is taken from the Analog.olb library.

We create a simulation profile: PSpice > New Simulation Profile > input a name for the simulation profile > click Create > Select the analysis type: AC Sweep > Select under AC Sweep Type: Logarithmic > Select Start Frequency, End Frequency and Points/Decade (fig. 32).

![Fig. 33 Simulation Settings for the RC Filter](image)
Place a voltage marker and then click **RUN** (fig. 34).

**Fig. 34** The RC filter **Frequency Response**

We will set a logarithmic scale for the y axis: **Plot > Axis Settings** (fig. 35) > **RUN** (fig. 36).

**Fig. 35** Setting the parameters for the y-axis
3.3. Time domain (Transient) response of an RC filter

The transient response of the RC circuit from below will be studied.

Draw the schematic from figure 37.

For this type of analysis, a VDC source has to be used.

Create a simulation profile: PSpice > New Simulation Profile > input a name for the simulation profile > click Create > Select Analysis Type: Time Domain and replicate the settings from fig. 38.
Place a voltage marker at the output of the circuit. Some initial settings for the capacitor: it charges from 0: right click on the capacitor and select Edit > Edit Properties > select IC (initial condition) and input 0, then close the window (fig. 39) > RUN (fig. 40).

3.3. Simple RLC circuit

Draw the schematic of the parallel RLC circuit from figure 41. The source is a VPULSE with the parameters from the figure below.
Create a simulation profile: **PSpice > New Simulation Profile** > input a name for the simulation profile > click **Create** > Select **Analysis Type: Time Domain** with the settings from figure 43.

Place a voltage marker and click **RUN** (fig. 44).
3.4. Being given the resistive network from the figure below:

![Resistive network](resistive_network.png)

A voltage $U_A=5\text{V}$ (a \textbf{VDC} source) is connected between pins 1 and 8 of the resistive network, pin 8 is considered ground.

The schematic in OrCAD should look similar to that in fig. 46. Using this schematic, the equivalent resistance between pins 1 and 8 will be computed (using voltage and current markers or the Bias point analysis).

![Resistive network with the source connected between pins 1 and 8](resistive_network_with_source.png)
The equivalent resistance between pins 1-2, 2-3, 2-4, and 2-8 respectively will be computed.

**Observation:** To determine the equivalent resistance between pins 1 and 2 the schematic has to be modified. This statement holds true for 2-3, 2-4, and 2-8 respectively.

The determined values will be written in table 1.

<table>
<thead>
<tr>
<th>Measurement pins</th>
<th>1-8</th>
<th>1-2</th>
<th>2-3</th>
<th>2-4</th>
<th>2-8</th>
</tr>
</thead>
<tbody>
<tr>
<td>$R_{computed}$</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Table 1

The electrical powers for each component of the circuit are displayed using the Bias Point analysis and the values should be written in table 2.

<table>
<thead>
<tr>
<th>P(W)</th>
<th>$P_{R1}$</th>
<th>$P_{R2}$</th>
<th>$P_{R3}$</th>
<th>$P_{R4}$</th>
<th>$P_{R5}$</th>
<th>$P_{R6}$</th>
<th>$P_{R7}$</th>
<th>$P_{R8}$</th>
<th>$P_{R9}$</th>
<th>$P_{R10}$</th>
<th>$P_{R11}$</th>
<th>$P_{R12}$</th>
</tr>
</thead>
</table>

Table 2

3.5. For the circuit in fig. 46 (a resistor cube), determine the value of the equivalent resistance in OrCAD using Ohm’s law and current markers.

The supply voltage will be connected between points A and B and will be a 10 V **VDC**. All resistors have a resistance of 1kΩ.

![Resistor cube](image)

**Fig. 47 Resistor cube**

### 4. Observations and Conclusions

In conclusion, a circuit can be tested using the computer, but this only becomes possible by using models for real-life components. This can be an advantage especially when designing complex circuits, since, in this case, testing is time consuming and also financially burdening.
5. Questions

1. What is the purpose of simulating a circuit’s functionality?
2. What is the name of the library that includes passive components? What about the library containing sources?
3. How do we change the value of a component?
4. What are the benefits of using virtual measurement instruments? (e.g. ampermeter)
5. What types of sources can be used for each analysis type?
6. In the RC circuit, interchange R and C and perform an AC Sweep analysis. What do you observe?
7. For a voltage divider with $R_1=100\,\Omega$ and $R_2=400\,\Omega$, run a Bias Point analysis.
8. For the circuit in fig. 46, determine the value of the equivalent resistance (for intermediate stages represent the equivalent circuits) knowing that the power supply is connected between points A and B.

6. Content of the laboratory report:

- Bias Point and Time Domain analyses of the voltage divider
- AC Sweep analysis of the presented high pass filter
- Time Domain analysis of the high pass filter and of the RLC circuit
- OrCAD analysis of the resistive network
- Determining the equivalent resistance for the resistor cube and schematic representation in OrCAD
- Answers to questions

Glossary

The voltmeter is a device used to measure voltages in electric circuits.
The ammeter is a device used to measure the current that flows through a conductor or an electric circuit.
The resistor is a passive component characterized by a resistance.
The oscilloscope is a complex device used to display and analyse time variable electric signals. By electric signal, we mean a voltage or a current that varies in time.

Symbols of the components used: