Computer Studies

Zhetpisbayeva Aliya

2018-05-29

Contents

Chapter 1

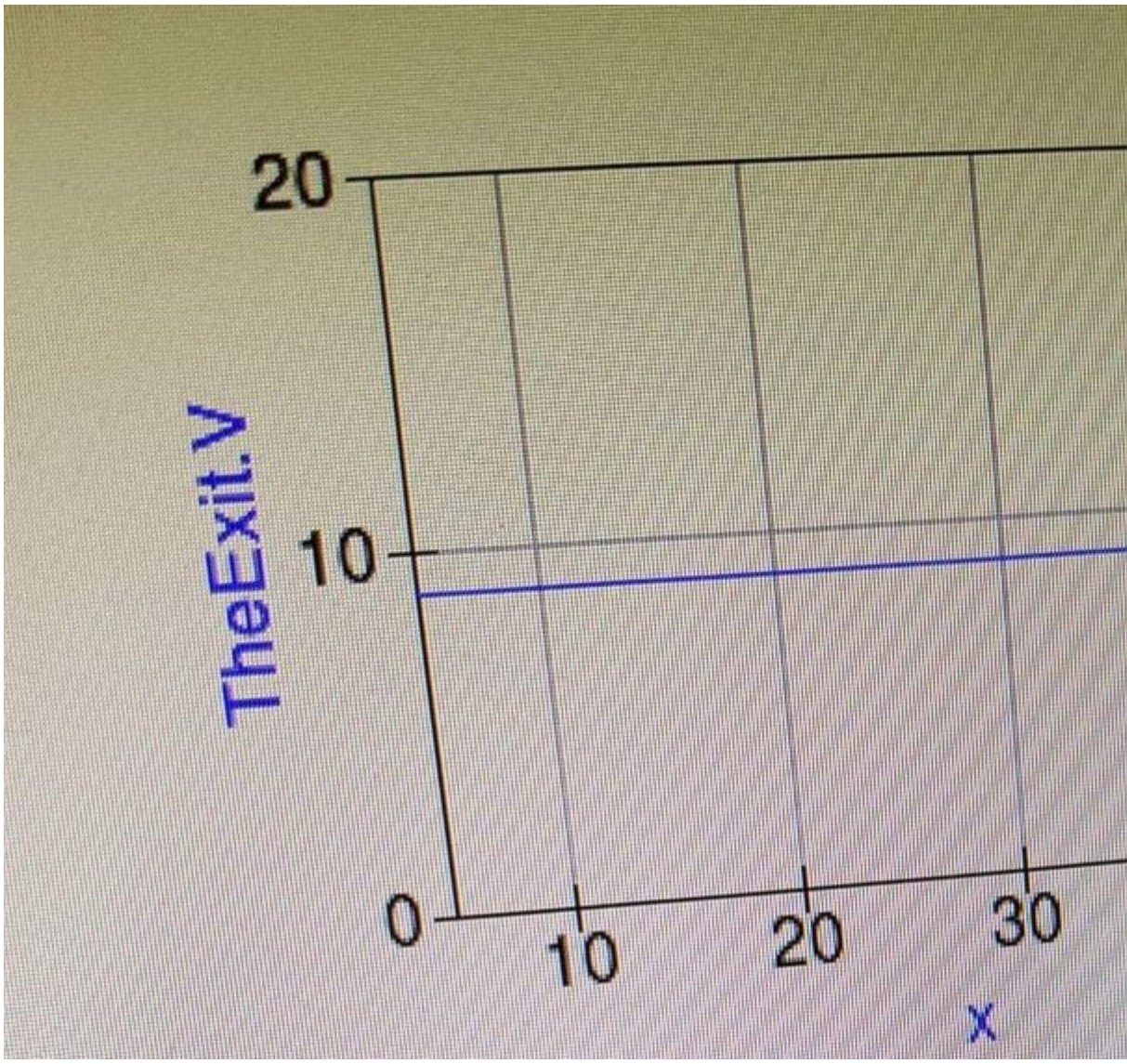
Theoretical part

1.1 Circuit calculation

Calculate the voltages on the resistors shown in the diagram 1. For voltage source V1 use DC voltage that is the last three digits of your student ID divided by 10. For example. 101REB123 means V1 = 12.3 (Volts)
R1 is the second digit of the last 3 digits of the student ID + 1. R2 is the last digit of the student ID number +1. For example, if your student ID number is 101REB123 then R1 = 3, R2 = 4. Take a picture of the calculation. The calculation process will be required at work P02. Additionally, the calculation will have to be added to the report you will complete at the end of the semester[1, 5]

1.2 Check modeling with gEDA

- Get into the students work area by using ssh on a training host with an IP address 213.175.92.37. For example: ssh -X x111REB ... @ 213.175.92.37
- Create a folder work and go to it. Create a folder P01 and go to it
- Launch the program gschem . Create the schematics shown in the source and resistor values according to the theoretical calculation made at point Save the schematics in file called 01.sch. Do not forget to add the value parameter to all elements. In addition, there must be a defined grounding point. This is done by assigning parameter netname = 0 to one of the connections (nets). See the lecture slides for the usage of the program. [1, 4]



generate the netlist file. For this purpose, run from the command line: gnetlist -g spice 01.sch -o 01.net .

- Using cat check whether the netlist file has been generated correctly.
- Make a simulation of the circuit 01.net using the program ngspice. For this purpose, run ngspice from the command line.
- Load the created netlist file using the command into ngspice: source 01.net

- Perform a simulation of the transient process (tran) from 0 to 5 seconds in 1 second step.
- Use plot to display the signal on the ***1*** connection. Using ***hardcopy*** button save the resulting image to 011.png or make screenshot.
- Using plot display the signal in the connection 2. Save the resulting image to ***012.png*** or make screenshot. All this will have to be used at work P02

Bold, Italic, Underline Some of the **greatest** discoveries in science were made by ***accident***

1.3 Advanced task: Modeling with QUCS

Start simulator QUCS. Locate the Project menu, then select Open Project point to the directory P01.

- Choose the tab Components on the left. From the palette, select the two components Resistor and the source of the component DC Voltage Source from the Sources menu. Put it all on the work surface as shown in the 2 image. Select the voltage source and resistor values according to the theoretical calculation at point 1.1. Make sure the visible component parameters (R1, R2, V1, etc.) are not overlapped and legible.
- Use CTRL + E to turn on wiring or connection mode and 0connect the components. Do not forget to add Ground to the scheme.
- From the menu, open the category simulations and add the DC simulation block to your schema. Without this block, QUCS will not know what needs to be done.
- Save the created scheme with the command sequence File-Save. Name the newly created file as 02. Qucs will add the extension (.sch) to the file itself. This will give you the file 02.sch.
- Perform elementary DC mode simulation with the F8 key, which results in calculations and determines the voltage on the resistor R2. The simulator variable that derives this value is designated R2.V.
- Add the schema to the simulation component Parameter sweep, which is selected from the category simulations (see Fig. 3). Evaluate the Sweep simulation attributes and their parameters displayed on the screen. [1, 3]

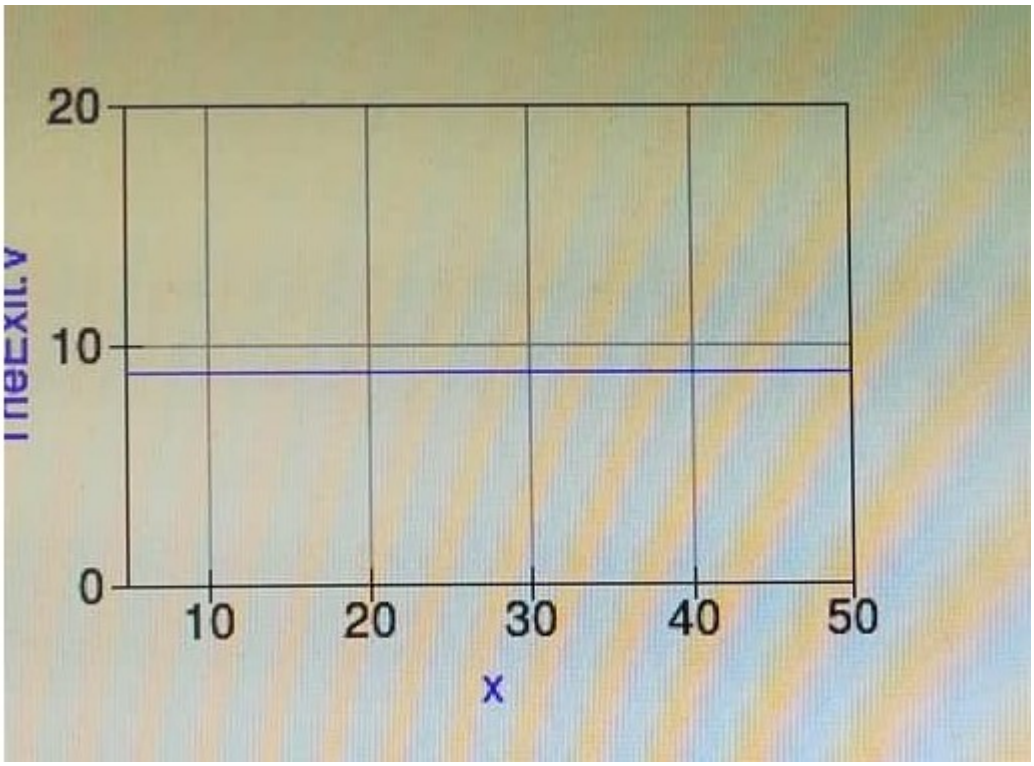


Figure 1.1: The QUCS schematics environment

1.4 Tabula creation:

I have created simple table which is required in practical work [1]

col1	col2
R1	2
R2	2
V1	1.1
UR1	3.3
UR2	4.7

Then you can use the environment wraptable which takes two parameters: The first one is the alignment that can be l, r, c, i or o for left, right, centre, inner and outer respectively. [1, 3]

1.5 Circuitikz - for adding the electrical circuit diagram

The symbols can also be used along a path, using the transistor-path-syntax(T in front of the shape name, see section 6.6). Dont forget to use parameter n to name the node and get acces to the anchors:
To then link them up with other components we would use the predefined node anchors.
For more information about all the components available and how you link components using node anchors, take a look at the documentation.[2, 3]

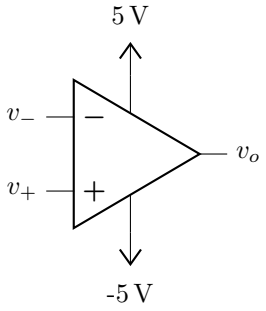


Figure 1.3: Logical ports

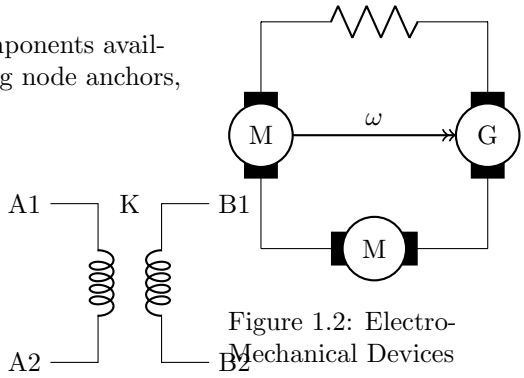


Figure 1.2: Electro-Mechanical Devices

Figure 1.4: Logical ports

Chapter 2

Practical part

2.1 Work with GEDA programs

2.1.1 Work with gschem

In this section, we will take a look the features of LaTeX with implementation[2] All details and explanations come from [1, 3]

2.1.2 Work with gnetlist

```
* Spice netlister for gnetlist
V1 2 0 1.2
R2 0 1 3
R1 2 1 2
.END
```

2.1.3 Work with ngspice

The second parameter is the width of the figure, in the example is 0.25 the width of the text. See the reference guide for a list of possible length units.[1, 3]

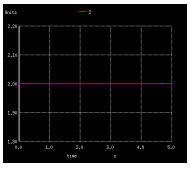


Figure 2.1: Gschem

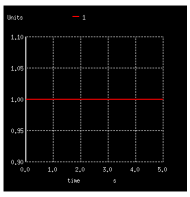
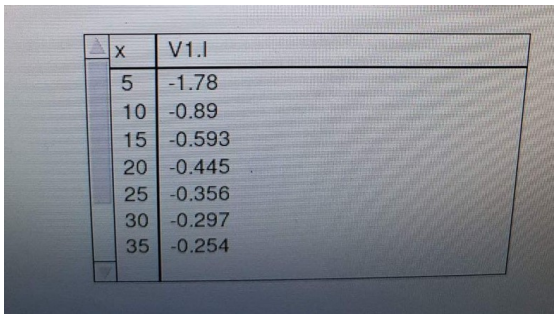


Figure 2.2: Gschem graph



x	V1.I
5	-1.78
10	-0.89
15	-0.593
20	-0.445
25	-0.356
30	-0.297
35	-0.254

Figure 2.3: Plots from the simulations of voltage on R1 and R2

2.2 Work with QUCS programs

- Image of the schematics
- Plot (curve) of DC simulation.
- Curve from Sweep simulation (advanced topic).
- Sweep simulation table (advanced topic).
- Explanations for each image and table (advanced topic).

Bibliography

[1] Documentation of ShareLetatex, Online *The LaTeX Companion*. Addison-Wesley, Reading, Massachusetts, 2003.

[2] Massimo A. Redaelli (m.redaelli@gmail.com) *CircuiTikZ*. (German) [*On the electrodynamics of moving bodies*]. Annalen der Physik, 322(10):891921, 2006.

[3] Latex Tutorial Online *http://www.latex-tutorial.com/tutorials*

[4] Latex Tutorial Online from Hackr *https://hackr.io/tutorials/learn-latex*

[5] Latex Tutorial Online from Lynda *https://www.lynda.com/LaTeX-kurs-tutorial/3141-0.html*