

# Circuit Theory and Electronics Fundamentals 2020/2021

Integrated Masters in Aerospace Engineering, Técnico, University of Lisbon

First Laboratory Report

March 25, 2021

Rui Rodrigues, 95844  
Tiago Silva, 95850  
Tomás Ribeiro, 95854

## Contents

<b>1</b>	<b>Introduction</b>	<b>1</b>
<b>2</b>	<b>Theoretical Introduction</b>	<b>2</b>
<b>3</b>	<b>Theoretical Analysis</b>	<b>3</b>
3.1	Node Voltage Method Analysis . . . . .	3
<b>4</b>	<b>Simulation Analysis</b>	<b>5</b>
4.1	Operating Point Analysis . . . . .	5
4.2	Relative Error Analysis . . . . .	5
<b>5</b>	<b>Conclusion</b>	<b>5</b>

## 1 Introduction

The objective of this laboratory assignment is to study a circuit containing a voltage source  $V_A$ , a current-controlled voltage source  $V_C$ , a current source  $I_D$  and a voltage-controlled current source  $I_B$  connected to different fixed value resistors  $R_1, R_2, R_3, R_4, R_5, R_6$  and  $R_7$ . The circuit can be seen in Figure 1.

In Section 2, a theoretical introduction is made in order to contextualize all the main principles that sustain our analysis of the circuit. This circuit is carefully analysed according to two of the most efficient ways to solve a circuit: the Mesh Current Method and the Node Voltage Method, both presented in Section 3. In Section 4, the circuit is analysed by simulation and the results are compared to the theoretical results obtained in Section 3. The conclusions of this study are outlined in the final part of the report, in Section 5.

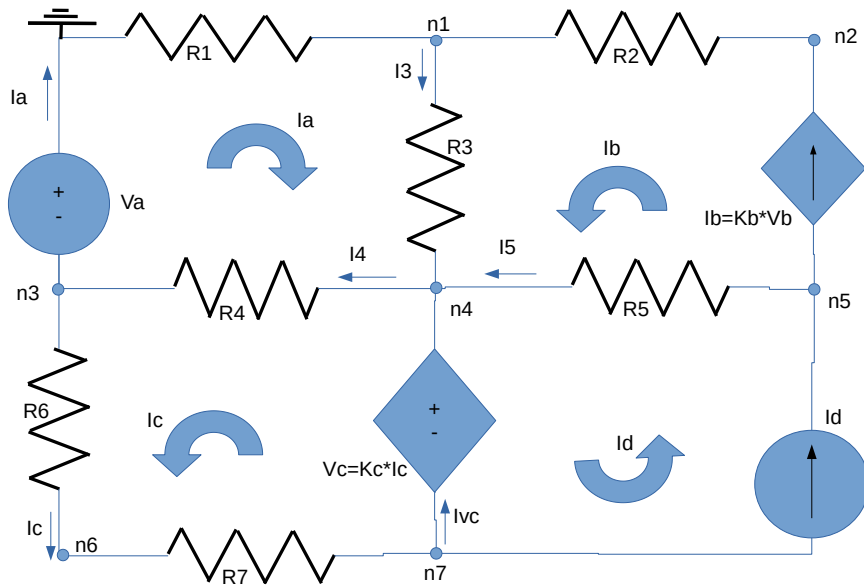


Figure 1: Second laboratory circuit.

## 2 Theoretical Introduction

The Mesh Current Method is a well-organized method for solving a circuit and is based on Kirchhoff's Voltage Law (KVL). To apply this method, we need to define what mesh current is. When we use the term mesh current, we are referring to an imagined current flowing around a loop. To apply this first step of this method, we first need to identify and distinguish a loop from a mesh. A loop corresponds to any closed path around the circuit and, to trace it, we start at any component terminal and trace a path through connected elements until we get back to the starting point. A loop is allowed to go through an element just one time. That leads us to the definition of a restricted kind of loop, a mesh, which contains no other loops.

The implementation of the Mesh Current Method to analyse the circuit was done following the common sequence of steps, summarized below.

- Identify the meshes.
- Assign a current variable to each mesh, using a consistent direction (clockwise or counterclockwise).
- Write Kirchhoff's Voltage Law equations around each mesh.
- Solve the resulting system of equations for all mesh currents.
- Solve for any element currents and voltages you want using Ohm's Law.

The Node Voltage Method is another way to analyze a circuit. This method is based on Kirchhoff's Current Law (KCL). To apply this method, we need to define what node voltage is. When we use the term node voltage, we are referring to the potential difference between two nodes of a circuit. We select one of the nodes in our circuit to be the reference node and, therefore, all the other node voltages are measured with respect to the referenced one. This reference node is called the ground node and, as it gets the ground symbol in Figure 1,

corresponds to the node between resistor  $R_1$  and voltage source  $V_A$ . The potential of the ground node is defined to be  $0V$  and the potentials of all the other nodes are measured relative to ground.

The implementation of the Node Voltage Method to analyse the circuit was done following the common sequence of steps, summarized below.

- Assign a reference node.
- Assign node voltage names to the remaining nodes.
- Solve the easy nodes first, the ones with a voltage source connected to the reference node.
- Write Kirchhoff's Current Law for each node.
- Solve the resulting system of equations for all node voltages.
- Solve for any currents you want to know using Ohm's Law.

### 3 Theoretical Analysis

\*\*\*INTRO HERE\*\*\*

#### 3.1 Node Voltage Method Analysis

We start to apply the Node Voltage Method by starting to identify the nodes of the circuit. In this case, our circuit has 8 nodes and each one has a node voltage designated  $V_1, V_2, V_3, V_4, V_5, V_6, V_7$  and  $V_8$ , according to the related node. Then, we apply the KCL to each one of the nodes.

$$\left\{ \begin{array}{l} \text{Node1 : } V_1 = V_S \\ \text{Node2 : } I_1 + I_2 = I_3 \\ \text{Node3 : } I_B = I_2 \\ \text{Node4 : } V_4 = 0 \\ \text{Node5 : } V_5 - V_8 = V_D \\ \text{Node6 : } I_C + I_B + I_5 = 0 \\ \text{Node7 : } I_D = I_7 \\ \text{Node8 : } V_5 - V_8 = V_D \end{array} \right.$$

After that, we rewrite it into an equivalent system defining the currents in terms of node voltages.

$$\left\{ \begin{array}{l} \text{Node1 : } V_1 = V_S \\ \text{Node2 : } \frac{V_1 - V_2}{R_1} + \frac{V_3 - V_2}{R_2} = \frac{V_2 - V_5}{R_3} \\ \text{Node3 : } I_B = \frac{V_3 - V_2}{R_2} \\ \text{Node4 : } V_4 = 0 \\ \text{Node5 : } V_5 - V_8 = K_D * I_D \\ \text{Node6 : } I_C + I_B + \frac{V_6 - V_5}{R_5} = 0 \\ \text{Node7 : } I_D = \frac{V_7 - V_8}{R_7} \\ \text{Node8 : } V_5 - V_8 = V_D \end{array} \right.$$

Now, we have a system of 6 equations (note that applying KCL to node 5 and node 8 generate the same exact equation). Between these two nodes, the circuit presents a voltage source  $V_D$ . Therefore, to obtain another equation to add to the system, we need to consider a supernode that includes both node 5 and node 8 and, after that, apply KCL to the supernode we just created.

We obtain another equation:  $I_3 + I_5 + I_C + I_D = I_4$ .

Once again, we define the currents in terms of node voltages and obtain an equivalent equation:  $\frac{V_2 - V_5}{R_3} + \frac{V_6 - V_5}{R_5} + I_C + I_D = \frac{V_5}{R_4}$ .

At this point, we have a system with 7 equations. However, besides the node voltages  $V_1, V_2, V_3, V_5, V_6, V_7$  and  $V_8$ , we still have two more variables to determine its value,  $I_B$  and  $I_D$  (note that  $I_C = 0V$  when  $t < 0$ ). So, we need to find two more equations to complete our system. We have to define the missing value currents in terms of node voltages and get the two extra equations.

$$\begin{cases} I_B = (V_2 - V_5) * K_B \\ I_D = \frac{0 - V_7}{R_6} \end{cases}$$

This takes us to our final system of equations, with 10 equations to find the values of 10 variables.

$$\left\{ \begin{array}{l} V_1 = V_S \\ \frac{1}{R_1}V_1 + (-\frac{1}{R_1} - \frac{1}{R_2} - \frac{1}{R_3})V_2 + \frac{1}{R_2}V_3 + \frac{1}{R_3}V_5 = 0 \\ I_B - \frac{1}{R_2}V_3 + \frac{1}{R_2}V_2 = 0 \\ V_5 - V_8 - K_D * I_D = 0 \\ I_B + \frac{1}{R_5}V_6 - \frac{1}{R_5}V_5 = 0 \\ I_D - \frac{1}{R_7}V_7 + \frac{1}{R_7}V_8 = 0 \\ I_D + \frac{V_2 - V_5}{R_3} + \frac{V_6 - V_5}{R_5} - \frac{V_5}{R_4} = 0 \\ I_D + \frac{1}{R_3}V_2 + (-\frac{1}{R_3} - \frac{1}{R_4} - \frac{1}{R_5})V_5 + \frac{1}{R_5}V_6 = 0 \\ I_B - -K_B * V_2 + K_B * V_5 = 0 \\ I_D + \frac{V_7}{R_6} = 0 \end{array} \right.$$

We can transform this system of equations and put it into a matrix form, ready to be solved in GNU Octave.

$$\begin{bmatrix} 1 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ \frac{1}{R_1} & -\frac{1}{R_1} - \frac{1}{R_2} - \frac{1}{R_3} & \frac{1}{R_2} & 0 & \frac{1}{R_3} & 0 & 0 & 0 & 0 & 0 \\ 0 & \frac{1}{R_2} & -\frac{1}{R_2} & 0 & 0 & 0 & 0 & 1 & 0 & 0 \\ 0 & 0 & 0 & 1 & 0 & 0 & -1 & 0 & -K_D & 0 \\ 0 & 0 & 0 & -\frac{1}{R_5} & \frac{1}{R_5} & 0 & 0 & 1 & 0 & 0 \\ 0 & 0 & 1 & 0 & 0 & 0 & 0 & -\frac{1}{R_7} & 1 & 0 \\ 0 & -\frac{1}{K_B} & 0 & \frac{1}{K_B} & 0 & 0 & 0 & 1 & 0 & 0 \\ 0 & \frac{1}{R_3} & 0 & -\frac{1}{R_3} - \frac{1}{R_4} - \frac{1}{R_5} & \frac{1}{R_5} & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & \frac{1}{R_6} & 0 & 0 & 0 & 0 \end{bmatrix} \begin{bmatrix} V_1 \\ V_2 \\ V_3 \\ V_5 \\ V_6 \\ V_7 \\ V_8 \\ I_B \\ I_D \end{bmatrix} = \begin{bmatrix} V_S \\ 0 \\ 0 \\ 0 \\ 0 \\ 0 \\ 0 \\ 0 \\ 0 \end{bmatrix}$$

These are the final values obtained with the application of the Node Voltage Method:

Nodal Analysis Voltages [in Volts]	
Node Voltage 1	5.02924600001e+00
Node Voltage 2	5.05316219884e+00
Node Voltage 3	3.96088725636e+00
Node Voltage 5	5.12810185377e+00
Node Voltage 6	6.80901710066e+00
Node Voltage 7	-1.42995796839e+00
Node Voltage 8	-7.23032834928e-01

## 4 Simulation Analysis

### 4.1 Operating Point Analysis

Ngspice is a circuit-simulation program that makes it possible to have an accurate representation of how the circuit would behave if it was actually assembled. Table ?? shows the simulated operating point results for the circuit under analysis.

As it can be seen, the table shows the values of the voltage in all nodes, the currents in all of the branches and also the currents in independent voltage sources. When looking at the simulation results there is one very interesting detail which is very important: Ngspice operates with the idea that the positive current flows from the positive pole to the negative pole in all components, sources included. This explains why, for example, in the the very same branch where resistor  $R_1$  and voltage source  $V_A$  are, the current given by ngspice in the whole branch (@r1[i]) is the symetric of the one given specifically in the voltage source  $V_A$ . The same logic applies to  $I_{Vc}$ .

Another important detail about this table is the node  $N_A$  and the voltage source  $V_{AB} = 0V$ , which are absent from the circuit's picture. This is because the Current-Controlled Voltage Source is dependent on current  $I_C$ , but Ngspice requires a voltage source where this current flows through, which made necessary the use of an auxiliary voltage source in series with  $R_6$  and, therefore, an auxiliary node.

### 4.2 Relative Error Analysis

When the simulation results given by Ngspice are compared to the theoretical ones obtained in Section 3, it is possible to highlight the fact that these are, in reality, extremely close to each other. As it can be seen, the highest error in percentual value is in the order of  $10^{-4}$ , which is negligible. Such result can be explained by the fact that this is a very simple circuit with very simple components, thus not having a lot of chances to differ greatly.

For reasons explained in Subsection 4.1, the value considered for  $I_A$  was the one given by the resistor  $R_1$  and the value considered for  $I_{Vc}$  was the symetric of the one given by Ngspice.

## 5 Conclusion

In this first laboratory assignment, all the major goals of the project were achieved. We concluded with success our first interaction with a new software (Ubuntu), with a simulation platform (Ngspice), with a computational language program (GNU Octave) and with a text report

editor (LaTeX). The analysis of the circuit was also finished with success through simulation and theoretical interpretation.

The main objective of the report was completed with the study of a circuit containing a voltage source  $V_A$ , a current-controlled voltage source  $V_C$ , a current source  $I_D$  and a voltage-controlled current source  $I_B$  connected to different fixed value resistors  $R_1$ ,  $R_2$ ,  $R_3$ ,  $R_4$ ,  $R_5$ ,  $R_6$  and  $R_7$ . Mesh currents and node voltage were analysed both theoretically, using the Octave maths tool, and by circuit simulation, using the Ngspice tool. The simulation results matched the theoretical results precisely. This accuracy was confirmed by the mathematical calculation of relative errors, which were proved to be really small. The reason for this perfect match is the fact that this is a straightforward circuit containing only simple and linear components, so the theoretical and simulation models cannot differ. For more complex components, the theoretical and simulation models could differ. However, this is not the case of the analysis of this report, where the results are obtained successfully and with notorious precision.