EEEE1028 Power and Energy

Coursework 1

Investigating Kirchhoff's Voltage Law and Resistor Combinations

The aim of this coursework is to provide you with a better understanding of voltage, potential, Ohm's law and Kirchhoff's voltage and current laws. You will investigate the behaviour of various circuits through simulation studies using PLECs modelling software. This document is divided into three sections:

1. Simulation work – familiarisation with PLECs and simulation of three circuits;
2. Requirements for the report to be submitted;
3. Details of downloading and running PLECs

1.0 Simulation Studies

See Appendix 1 for instructions for downloading and running PLECs. Relevant files are on the Moodle Website in the folder –

1.1 Introduction to Circuit Elements and Analysis Techniques

This part of the labsheet will provide you with a brief introduction to PLECs, a very useful simulation package that can be used to investigate the circuits we look at in lectures. There are some specific examples on the EEEE1028 Moodle website that relate to this particular piece of coursework (cw1a.plecs, cw1b.plecs, cw1c.plecs). However firstly we will look at the circuit we wish to simulate.

Circuit 1: Simple resistor circuit

Figure 1 shows the circuit we are going to simulate, comprising a DC voltage source and a resistor R1. We have added an ammeter and a voltmeter to the circuit. It is supplied from a 5V DC supply. The ammeter (circle with A in the middle) measures circuit current. It must be connected in series with the circuit or component in which you want to measure the current. The voltmeter (circle with V in) measures the voltage across two points (nodes) in a circuit: it must be connected in parallel to the component or circuit across which you want to measure voltage.

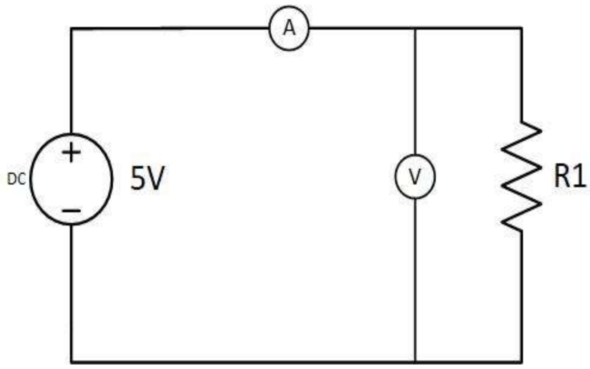


Figure 1: Circuit 1: simple "R" circuit

(Q1) What is the effective resistance of a) the ammeter and b) the voltmeter when connected in the circuit? (Remember they should not affect the circuit behaviour – they are only measurement devices!).

(Q2) Calculate the value of resistance you require for circuit 1 to have a supply current of 23mA.

Circuit 1: Simulation with PLECS

a. Consult Appendix 1 for instructions on downloading PLECs if it is not on your PC/Laptop. If PLECs has been downloaded, double click on the PLECs icon to start the programme.

b. Open File CWK1a.plecs (available on the Power and Energy Moodle page). The easiest way to do this is to download the file to your desktop, and then open it using the File tab in the PLECs library browser.

c. The circuit simulated comprises a 5V DC voltage source, a load comprising a 10Ω resistor, a voltmeter (Vm1 to look at voltage), and ammeter (Am1 to look at current) and a scope so that we can see these measured quantities.

d. Click on the "Simulation" tab at the top of the window and then click on "simulation parameters".

e. Check that the parameter "max Stepsize" = 0.001 and "Stop Time" = 0.06 – we will discuss these in later coursework.

f. Run the simulation (click the "Simulation" tab again and then click start). Open the scope block (double click on it) so that you can see the voltage and current.

g. You *may* see a simulation "start-up" transient – equivalent to switching the power supply on in the experiment – followed by a steady state (DC) period. It is the DC part that we will focus on.

(Q3) Record Vm1 and Im1 and confirm that these measurements follow Ohm's Law.

h. Play with the simulation – i.e. double click on the resistor and change its value and then re-run the simulation (e.g. use the value you calculated in Q2 and confirm that a current of 23mA flows in the circuit). Change the value of the voltage source. In all tests that you do, use your parameters and Ohm's Law to predict what the current should be and check that is what you get for the steady state value from your measurements. Record all calculations, predictions and measurements.

On a practical note, resistors come in all shapes and sizes as shown in Figure 2, which shows three 330Ω resistors. They are physically different because they are for different applications and have to handle different voltages, currents and powers.

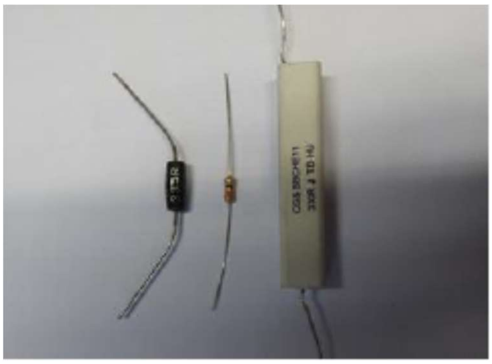


Figure 2: 330 Ω resistor for different electronic circuit board applications

(Q4) For the resistor you chose in (Q2) (to conduct 23mA when connected to a 5V supply), calculate the minimum required power rating of the resistor?

Circuit 2: Series–parallel Circuit

a. Close CWK1a.plecs and open CWK1b.plecs. The circuit here has the structure shown in Figure 3.

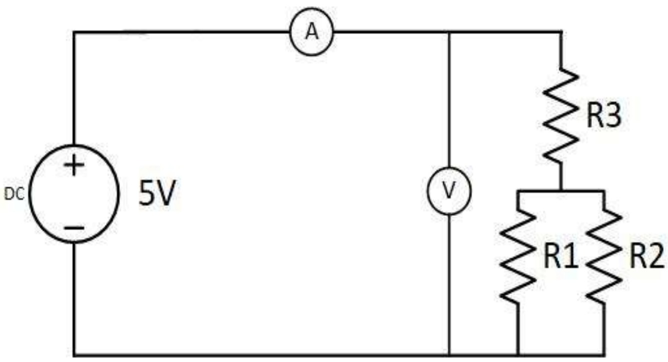


Figure 3: Series-parallel Circuit

(Q5) Check that the values of R1, R2 and R3 are 40Ω, 20Ω and 30Ω respectively and calculate the theoretical values for the currents I1, I2 and I3 – the currents in each of the resistors R1, R2 and R3 and the voltages across each of the resistors. Show your calculations.

b. Run the simulation and use the measurement blocks to record the steady state values of your currents and voltages.

(Q6) Do they match your calculated predictions? If no – why not?

Circuit 3: Series – Parallel – Series Circuit

c. Close CWK1b.plecs and open CWK1c.plecs. The circuit here has the structure shown in Figure 4. R1, R2 and R3 should have the same values as in section II.a (check them). The value of R4 is not correct, as you need to calculate it.

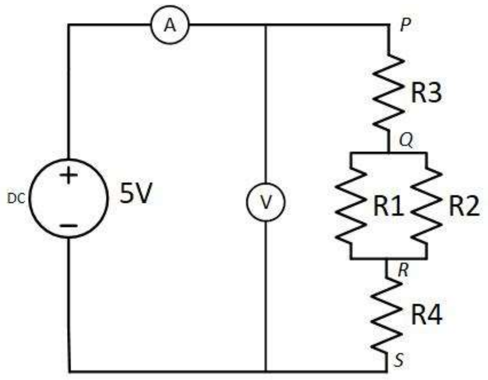


Figure 4: Series parallel series circuit

(Q7) Calculate the value of R4 required to reduce the supply current to 50mA. Calculate the values for the currents I_1 - I_4 – the currents in each of the resistors R1 - R4 (use the "current divider law" to obtain I_1 and I_2), and the voltages across each of the resistors. Show your calculations.

Change R4 in CWK1c.plecs. Run the simulation and use the measurement blocks to record the steady state values of your currents and voltages including the supply voltage (V_S), the voltage across R3 (V_{PQ}) and the voltage across the R2/R1 combination (V_{QR}) and the voltage across R4 (V_{RS}).

(Q8) Show that the voltages you have measured can be used to demonstrate Kirchhoff's Voltage Law. Also show the relationships between voltages VS, VPQ, VQR and VRS by using the voltage divider rule with your known values of resistances.

Voltage, Potential and Potential Difference

An important point to note here is that we can use three terms to define voltages within this circuit: voltage, potential difference and potential. Voltage and potential difference are essentially the same – they define the voltage between two specific points in the circuit and these must be specified eg V_{PQ} is the voltage (potential difference) across R3 or between nodes P and Q in Figure 4. Similarly V_{RS} is the voltage (potential difference) between nodes R and S. Note also the first stated node is usually defined as the positive node.

The term "potential" usually means that the voltage is measured with respect to the circuit ground point (in this case the negative terminal of the supply or Node S in Figure 4). We will explore this in the next section.

(Q9) From your measurements, state:

1) The potential of nodes P, Q, R and S

2) The voltage between nodes P and R

3) The potential difference between nodes S and Q.

(Q10) Show that VQR can be calculated knowing the potentials at node Q and Node R.

Circuit 5: Dual Supply Circuit

a. Close CWK1c.plecs and open CWK1d.plecs. The circuit here has two supplies connected in series so that one half of the circuit has a negative voltage with respect to ground as shown in Figure 5.

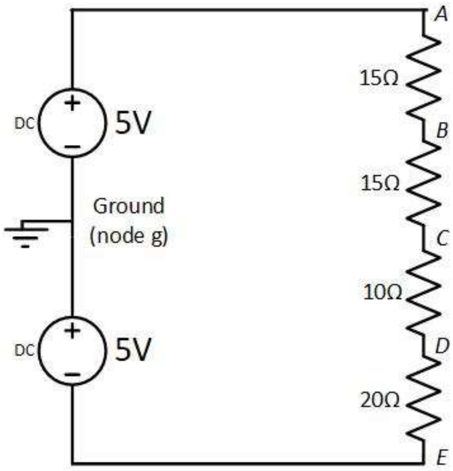


Figure 5: Circuit with bipolar supply

(Q11) Calculate the theoretical values for the voltages V_{AB} V_{BC} V_{CD} and V_{DE} where A,B,C, D and E relate to the node labelling in Figure 5. Show your working. Use these values to calculate V_{EC} noting that by convention, the first named node is considered to be the positive node.

b. Run the simulation and use the scope blocks to record the steady state values of your voltages.

(Q12) What are the potentials at nodes A-E (hint: think of the ground as node g). Use these potentials to calculate the values of V_{AB} and V_{CD} and confirm your calculations from (Q11).

c. Modify your simulation by adding scope blocks to measure the potentials at nodes A-D (it already contains a scope to measure the potential at node E). Run the simulation again and use these scope blocks to record the steady state values of your potentials.

(Q13) Do the measured potentials match the theoretical values calculated in (Q12)?

Circuit 6: The Limits of a Potential Divider

This section aims to help you learn to use PLECs to simulate your own simple circuits. You now need to build a model to simulate the circuit shown in Fig. 6. This circuit is formed of a potential divider (R1 and R2) supplying voltage Vout to the load resistor Rload. We will see that it is not always possible to use the potential divider rule.

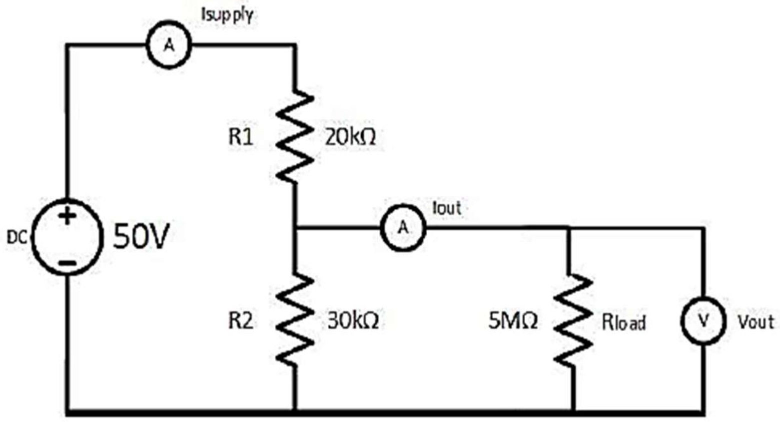


Figure 6

a. Open a new simulation file (click on "File" tab in the library browser, and then click on "new model". This opens the simulation window. Save this file using a suitable filename e.g. "test1".

b. Build the simulation model of Fig. 6. To add an element to the model, we basically "click", "drag" and "edit". For example,

i. To add a voltage source, click on the "electrical" tab in the library browser, and then click on the "sources" tab. Left click on the "Voltage Source DC" icon and then drag it into your simulation window. To edit the parameters associated with this component, double click on the icon; this opens up a parameters window, which you can edit and then save.

ii. To add a resistor, go again to the electrical tab in the library browser, click on "passive components" and use left click on the resistor icon to drag it to the simulation window. Edit its resistance by opening its parameter window.

iii. To connect components together, left click on the terminal of the device that is your starting point (terminals are denoted by small circles). A cross will appear, and if you move the mouse (still left clicking), a connecting line will appear. Move this line to the terminal of the device to which you are connecting.

iv. To measure a current, you need to add an ammeter. Go to the "electrical" tab again, and then meters and left click on ammeter. Drag the ammeter until it rests on the connecting line in which you want to measure the current. When you let go of the mouse button the ammeter will connect itself into the circuit. Alternatively, you can drag the ammeter to the simulation plane and connect to its terminals using procedure (3) above.

v. To view the output of the ammeter you will need to connect it to a "scope" or a "display". Drag a scope or display from the "system" tab into your simulation window and connect it to the triangular output of the ammeter using the process of (3).

vi. If you want to measure the voltage, you need a voltmeter. Drag a voltmeter (from the "electrical-meters" tab) into your simulation plane. Connect its terminals to the points in the circuit where you want to measure the voltage using the process of (3). Connect the voltmeter output to another scope or display. If you want to see current and voltage on the same scope, you will need to edit the scope by double clicking on it and then click on its "file" tab. Using "scope parameters" you can increase the number of plots that a scope can show by incrementing the value given. You will then have more than one port on the scope to connect to meters.

vii. Once your circuit is complete, prepare for simulation. Click on the "Simulation" tab at the top of the window and then click on "simulation parameters". Check that the parameter "max Step size" = 0.001 and "Stop Time" = 0.06. Run the simulation (click the "Simulation" tab again and then click start). Open the scope block (double click on it) so that you can see the voltage and current.

(Q14) Do your measurements demonstrate the "voltage divider" rule?

(Q15) Change R_{load} to 500kΩ and run the simulation. What happens to V_{out}?

(Q16) Change R_{load} to 50kΩ and run the simulation. What happens to V_{out}?

(Q17) What conclusions can you draw about the potential divider rule?

2.0 Report Structure

The report **MUST be typed-written with appropriate formatting**. The report should include your name, all parameters changed and measurements taken in the simulation studies, and answers to questions Q1-Q17. You DO NOT need to provide an Introduction or Conclusion and you do not need to repeat method statements or instructions given in this handout. A mark sheet for the coursework is available on the Power and Energy Moodle page.

2.1 Report Submission

The report should be **converted to pdf** and submitted through **Moodle** in the coursework section – "submit EEEE1028 Power and Energy Coursework 1 here".

2.2 Reminder

Remember that you are training to be professional engineers (or whatever profession you decide to follow) – and the work that you submit should have a professional quality. In the professional workplace, many of the reports I marked would have simply been handed back with an instruction to do them again. Your submission must have a uniform appearance with regard to text size and font – and that it needs to have appropriate titles, sections and numbering. You need to be able to typeset equations and introduce any needed figures etc. Note that for one piece of coursework, Moodle will expect one file. You will not be able to submit multiple files. **Some specific things which are not acceptable; i.e will not be marked (probably not a complete list):**

- A report with no author name in the report body (do not assume it is adequate to have your name within the name of the file)
- Anything which has been assembled as a collection of photographs – whether inserted into a typed report – or a hand-written one
- A report which is partially hand-written and partially word-processed – your submission must be word processed.
- A report in which the questions are not answered in the order the work is set. In an exam you can answer whatever you like and in any order, it will be found and marked. In a report there is no excuse for it not to be presented in the correct order – even if you did the work in a different order – you can present it in the correct order

Appendix 1

Running PLECs

The PLECS manual con be accessed through the PLECS help system or on the website http://www.plexim.com/sites/default/files/plecsmanual.pdf
It is useful to have a look at the first few chapters which introduce PLECS and the basic operations within PLECS in advance of going in the lab. There is also a range of demo programs within PLECS which you may find interesting.

PLECS Installation:

For installation of PLECS in your PC/laptop, go to PLECS webpage (https://www.plexim.com/), open license manager and request for license.

You will be directed to 2 options - Trial license (30 days) and Student license (for selected universities only).

Select Trial license, provide required particulars.

You can then proceed to install PLECS in your PC/laptop and start using it for your coursework.

Procedures:

1. Go to https://www.plexim.com/

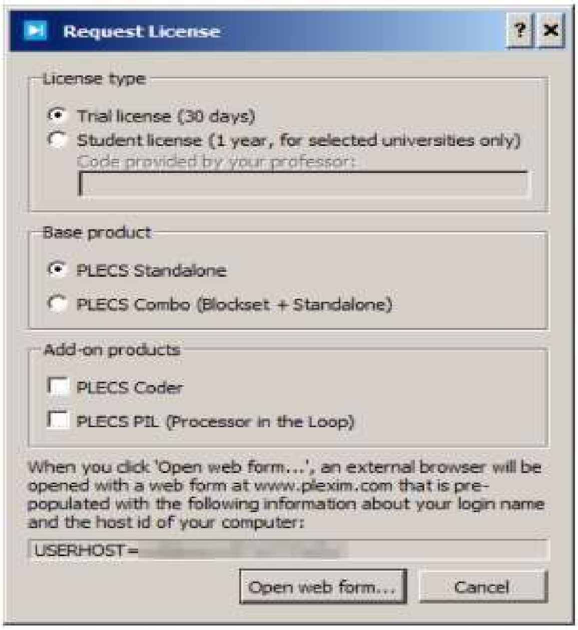
2. Select

3. Select ‘**PLECS Standalone**’.

4. Choose suitable package and install it according to the installation instructions.

> **Note** — You **do not** need administrator privileges in order to install PLECS.
>
> | Platform | File name |
> |---|---|
> | Microsoft Windows 32-bit | plecs-standalone-4-4-4_win32.exe (264630864 bytes) |
> | Microsoft Windows 64-bit | plecs-standalone-4-4-4_win64.exe (280691280 bytes) |
> | Mac / Intel 64-bit | plecs-standalone-4-4-4_maci64.dmg (228162266 bytes) |
> | Linux / Intel 64-bit | plecs-standalone-4-4-4_linux64.tar.gz (240134048 bytes) |
>
> The files were last updated on Wednesday, September 23, 2020 - 09:59

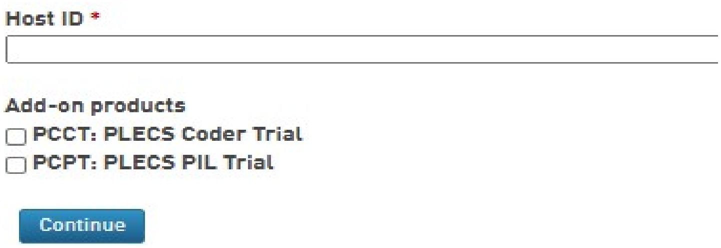
5. Start PLECS without a license file. In the dialogs that appear click Open license manager... and Request license.... Select a base product and optional add-on products and click Open web form....



This will take you back to a form on this web site that is pre-populated with your host ID information.

6. Once you have received your license file, restart PLECS and click on Open license manager..., then Manage license files... and Install... in order to install the license file.

If you cannot open the web form from within PLECS, copy the host ID information from the Request License dialog and enter it below.



Host ID *

Add-on products

PCCT: PLECS Coder Trial

PCPT: PLECS PIL Trial

Continue