

# Power and Energy (EEEE1028)

## Introduction to PLECS and AC Circuit Elements

### (Preparation for Coursework 2)

#### Introduction

PLECS is a simulation package which allows us to simulate complicated electrical networks. It will be used throughout the course and its real advantages will become apparent when you use it to simulate power electronic systems in Year 2 of this course. However we all need to start somewhere when it comes to using simulation packages and this exercise will introduce you to the basics of PLECS and how you plot and analyse the results it gives you. The first section will show you some important features that you need to consider when using simulation packages. The second section will provide you with several ac electrical circuits to analyse.

#### Part 1 Introduction to PLECS for AC Systems

##### 1.1 Simulation Timestep

1. Consult the instructions in the Appendix of this document for running
2. Open File CWK2a.plecs (available on the Power and Energy Moodle page)

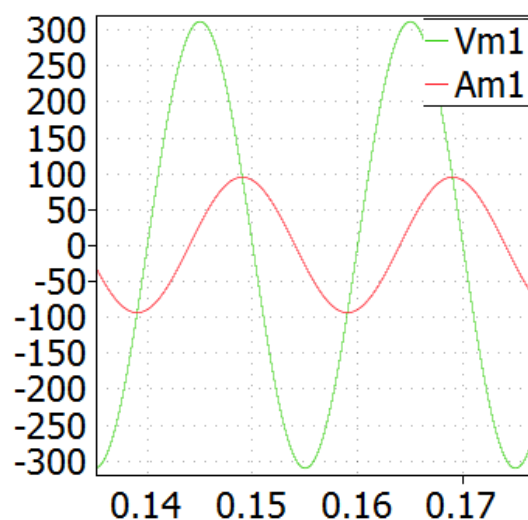
The circuit simulated comprises a 50Hz, 220V (rms) voltage source, a load comprising a 1Ω resistor and 10mH inductor, a voltmeter (Vm1 to look at voltage), and ammeter (Am1 to look at current) and a scope so that we can see the quantities. There is also a signal multiplexer block which allows us to put two signals into the scope – it is acting as a signal router.

3. Click on the “Simulation” tab at the top of the window and then click on “simulation parameters ...”.
4. Check that the parameter “max Stepsize” = 0.04 and “Stop Time” = 0.06.
5. 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.
6. Change the parameter “max simulation step” to 0.002 and sketch by hand what you now see.
7. Change the parameter “max simulation step” to 0.0001 and again sketch by hand what you see. You should now see something that looks like a sinewave.

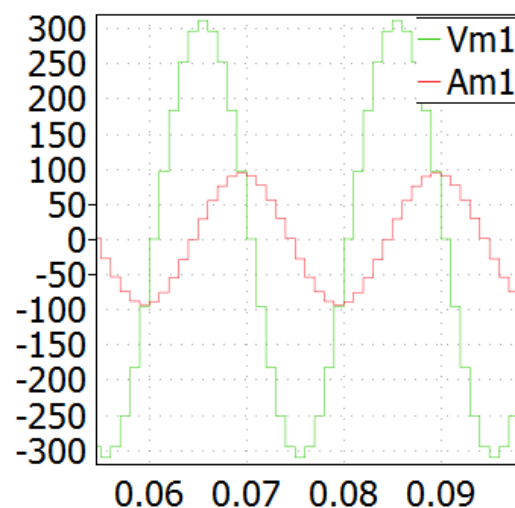
Simulation packages work by representing electrical (or other physical) systems by mathematical models i.e. they use equations to represent how the circuit behaves. These are usually in the form of differential equations, for example (1) is an example of a differential equation for a simple inductor circuit.

$$V = L \frac{di}{dt} \quad \text{or} \quad \frac{di}{dt} = \frac{V}{L} \quad (1)$$

The simulation package will generate the desired voltage from the voltage source defined and use (1) to calculate the inductor current. It therefore needs to integrate (1). However it will have to do this “numerically” rather than “algebraically” i.e. it samples the waveform at set intervals of time (separated by the timestep), and uses special algorithms to interpolate what the integral will be between these timesteps. The smaller the timestep, the more accurate the integration will be, BUT the simulation will take longer to run as the PC has many more calculations to do. The larger the timestep, the faster the simulation will run BUT the cruder the waveform that will be observed.



***Suitable Maximum Timestep***



***Unsuitable Maximum Timestep***

To help you, the simulation package tries to automatically optimise the timestep for best speed and it continually changes this as the simulation runs by looking at (for example) the gradients of the signals generated. However it needs your help. In particular the user needs to define what the maximum timestep should be using their expert judgement based on the circuit parameters they have input to the system. You will gradually develop experience to do this but for the remainder of this coursework we will set the maximum timestep to 0.00002s. As we are dealing with a 50Hz supply (which has a period of 20ms) this means that we will always have at least 100 points per cycle which is plenty to give is a good representation of our waveform. This is an important parameter you will need to set (based on your knowledge of the system being simulated) for any simulation that you work with.

## 1.2 Simulation Time

Another important parameter is the simulation time.

- *Open the “simulation parameters” menu; set the “max stepsize” parameter to 0.00002s and check that the parameter “Stop Time” = 1.0.*
- *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. Sketch by hand the waveforms that you actually see*

In this case the simulation is running for 50 periods ( $1s/20ms = 50$ ) and this clutters what you see on the scope so that you cannot really distinguish between any of the waveforms. You can use the “free

zoom” function of the scope (the magnifying glass icon on the scope’s menu) to zoom into the waveform (try this), or you can use the File Menu and Scope Parameters to change the x and y axis limits to achieve the same effect (again try this). However, for this simple case it is much more sensible to set a Simulation time that gives us only a few periods. As we have set our frequency to 50Hz, to see 5 periods we need a simulation time of 100ms.

- *Change the simulation time to 100ms. Close the scopes and reopen them – check that the scope parameters (in the scope’s “File” menu) for “Time Range” and Y limits” have been set to “auto”. Re-run the simulation. The simulation completes much more quickly and when you open the scope, you see waveforms which you can actually make measurements on.*

We need to be careful though. When the simulation starts at  $t=0$ , we are effectively putting a step change onto the system we are simulating and there will therefore be a transient component to the response we see. As we are focussing on steady state behaviour, we need to ignore this transient for the moment and focus on the last few cycles when making measurements. For the result you have just obtained, a good approximation is to say the transient has died out after 60ms and the system has reached steady state (but you must check this for every simulation you run).

Simulation Time is an important parameter you will need to set (based on your knowledge of the system being simulated) for any simulation that you work with.

### 1.3 Changing System Parameters

Open the file CWK2b.plecs and try experimenting with the circuit parameters. The initial values for the voltage are the same as in CWK2a.plecs, but the load has now been changed to a  $0.5\Omega$  resistor and a 10mH inductance. We need to find the amplitude of the current and its phase shift with respect to the voltage. However the start-up transient is now a bit more obvious.

- *Run the simulation and open the scope. Zoom into an area where you can see the peak of both waveforms (note this should be in the “steady state” part of the trace after about 100ms). Click on the menu button that looks like two vertical arrows ( $\uparrow\downarrow$  (cursors)). This should bring up two white lines in your scope – the cursors. You can left click on each cursor to drag it to a particular part of the waveform and you will see the time and the instantaneous value for that cursor location at the bottom of the scope. If you drag the left hand cursor to the peak of the voltage waveform (green) and the right hand cursor to the peak of the current waveform (red), you will be able to measure the time difference between the two waveforms ( $\Delta t$ ). Click on the cursor tab again and then click on “delta” to show the difference between the two cursor measurements. Calculate the phase difference between them, knowing the period (0.02s).*

$$\theta = \frac{\Delta t}{0.02} \cdot 360^\circ$$

- *Check that it matches the theoretical value, which can be calculated from the reactance and resistance of the circuit – ( $80.95^\circ$ ).*

- *Change the amplitude of the voltage source to 110V (double click on the voltage source block and change the value of the amplitude). What do you expect to see in the simulation (what will happen to the amplitude of the voltage and current and their phase difference?). Run the simulation and use the cursors to take measurements – does it confirm your predictions?*
- *Change the value of the inductance to 0.003H (double click on the inductor symbol and change the value of inductance to 0.003). What do you expect to see in the simulation (what will happen to the phase lag and amplitude of the current?). Run the simulation – does it confirm your predictions?*

An important note here. You should always try to predict what you will see in the simulation before you run it. Simulations may not give you the correct results – you may have accidentally entered the wrong circuit parameters, or the simulation parameters (e.g. max stepsize) may be unsuitable. You need to have an idea of what you expect to see to be able to identify if the simulation is not functioning correctly. This becomes more and more important as your simulation becomes more and more complicated.

#### 1.4 Measuring Power in AC Systems

Open the file CWK2c.plecs which has the same circuit as CWK2.b.plecs, but with some additional measurement channels. Run the simulation.

1. The scope labelled “V and I” simply shows the load voltage and current as you have seen before, and you can use this to measure the circuit phase angle (and then calculate power factor) using the cursors.
2. The scope labelled instantaneous power shows the instantaneous product of voltage and current, and you can see it has a frequency of 100Hz (two times the supply frequency) and a DC offset.
3. This instantaneous power is also averaged over one period (20ms in this case as the supply frequency is 50Hz) to calculate the average power.
4. Finally, the rms value of the supply voltage is calculated at the bottom using various maths functions- this only valid towards the end of the simulation when the model has reached steady state. You can compare the value displayed to the value of rms voltage you can calculate from the parameters of the voltage source V<sub>ac</sub>.
5. If you use a similar set of maths functions to calculate the rms current, you can then calculate the **apparent** power (S) delivered to the circuit ( $V_{rms}.I_{rms}$ ), and then the **power factor** and the **reactive** power. For this circuit:

$$S = 15195 \text{ VA}, \quad P = 2389 \text{ W}, \quad Q = 15006 \text{ VAR}, \quad PF = 0.159, \quad \theta = 80.8^\circ$$

(which do you this is the better method to calculate power factor and power factor – the method described in (6) or the method described in (1)).

As a final exercise open CWK2d.plecs. In this file you will see two ammeters but connected in opposite directions. Run the simulation and you will see the effect of the connection polarity in the scope: this will have a major effect (ie adds 180°) on the phase angle you measure if you get the polarity wrong!

The way I think of this – the arrow in the ammeter should point in the direction of the power flow i.e. away from the voltage source and towards the load

*You do not need to include any of the results from these experiments in your report.*

## Appendix A - Using PLECs

1. To add a voltage source, click on the “electrical” tab in the library browser, and then click on the “sources” tab. For example, 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.
2. 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.
3. 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.
4. 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.
5. 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).
6. 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.
7. 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 Stepsize” = 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