Numerical simulations of Electric Circuits using SimPower Systems in Matlab-Simulink environment

Module EE2304 – Circuit Analysis
Dr. Mahran Quraan

1. Introduction

This tutorial helps you to familiarize with some electric circuits using numerical simulations. These could be very useful for understanding the operating principles of circuits and their principal characteristics.

2. Learning objectives

In this laboratory exercise, you will examine some of the common blocks used in SimPower Systems building a simple simulator of different circuits.

Upon successful completion of the exercise, you should be able to:

- Construct a simple power circuit connecting together Simulink blocks;
- Construct a circuit with passive circuit elements and build a simulator of a series *RLC* circuit and a simple power system;
- Understand the step response of *RLC* circuit;
- Be able to understand the influence of adding shunt capacitor to the load on the transmission line losses.

3. Simulation 1 – Basic *RLC* circuit with AC supply (Practice)

Development of the circuit

The simulation of this circuit requires a power supply, an electric load, a control section and a measurement section.

From the SimPower system menu, select "Electrical Sources" and then "AC voltage source". Set the peak amplitude to 240*√2 V, the phase to 0 and the frequency to 50 Hz.

Select "Elements" and then "Series RLC branch". Set the branch type to RLC and put the value $R = 0.5\ \Omega$, $L = 2$ mH and $C = 100\ \mu$F.

Select "Measurements" and then the blocks "Current measurements" and "Voltage measurements". Connect the current measurement block in series with the RLC branch and the voltage measurement block in parallel. Connect each of the outputs of the measurement blocks to a scope (you can find it in "Simulink" -> "Sinks)

Connect the AC source to the *RLC* branch. Take care of having connected in the right way the Current and Voltage Measurement blocks.

Insert the "Powergui" and click on "Initial States Setting". Into the right section of the window, select "Force initial electrical states" to Zero.

Configuring simulation parameters

In the pull-down menu click "Simulation" and then "Configuration parameters". Select "ode 45" as a solver and set the maximum step to $1e^{-3}$. Select 0.1 as stop time.

Click on the scopes and uncheck "Limit data points to last" in the tab "History" inside "Parameters". In this way you should be able to see the whole waveform captured by the scope.

Running the simulation

You should be able to run the simulation and observe the shape of the waveforms of both voltages and currents. You probably noticed that the current has a transient before settling into a smooth sine wave. Change the value of the resistance to $R = 5\ \Omega$ and run the simulation again. Comment on the result obtained in comparison with the previous case.

4. Simulation 2 – Step response of series *RLC* circuit

A series *RLC* circuit has a DC source voltage Vs = 220 *V*, inductance *L* = 2 mH and resistance, *R* = 160 Ω.

a) Determine the capacitance values for each case (critical damped, under damped, and over damped).
b) Simulate the circuit using Matlab/Simulink.
c) Plot the waveforms $i(t)$, $v_c(t)$, and $v_L(t)$ of the circuit using Matlab/Simulink.

5. Simulation 3 – Power factor correction

Given the circuit in Fig. 1. The source electric frequency is 50 Hz.

a) Determine the input voltage V_S.
b) Simulate the circuit using Matlab/Simulink.
c) Plot the input voltage and current, the input power factor, the line power losses using Matlab/Simulink.
d) Determine the value of the capacitor that when placed in parallel with the load will change the input power factor to unity and then plot the input voltage and current, the input power factor, the line power losses using Matlab/Simulink.

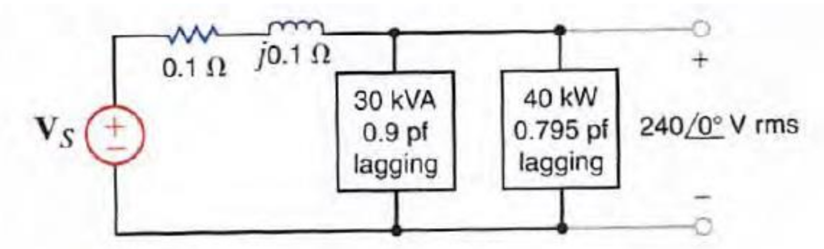

Fig. 1