TSTE19 Power Electronics

Lab 1

v 0.5, 2014

Kent Palmkvist

Introduction

The lab focus on simulation and evaluation of the bridge rectifier structure. Simulations should be analysed, using predefined models.

Initial setup

The design files used in the lab can be copied from /site/edu/ek/TSTE19/current/material/Lab1_files in Linux or from U:\ek\current\TSTE19\material\Lab1_files in Windows. Put the copied files into your home directory, for example in /edu/<userid>/TSTE19/ on Linux or H:\TSTE19 in Windows.
Please note that the software only works on Windows. You can thus only run the software in the Freja lab.

Starting the software

The software is started by Start ->All Programs ->National Instruments ->Circuit Design Suite 12.0 ->Multisim 12.0
Open the corresponding design project using File->Open and select the design file to use among the files you copied in the initial setup. The first design is in the project DBRECT1.ms12.

Notes on the schematics

Some of the schematics have more components added than shown in the tasks in the book. One reason for these (typically small resistances and inductances) are the computational properties of the simulation model. Without the extra components, the simulation calculations could become unsolvable.
Names and values of components can be changed by double-clicking on the value/name. Alternatively, the names and values can be changed by right-clicking on the component and select Properties.
Additional measures can be added by introducing the Measurement probe, which is a yellow symbol at the bottom of the column of symbols on the right side of the window. Place the cursor on top of the symbol to see the name of that particular symbol. Click on it and then click on a wire in the schematic to add a measurement probe to the schematic.
One useful variable for the following analysis step is a time variable. The Multisim environment does not directly support such a variable, but it can easily be added in one of many ways. The approach taken in the existing simulation models are the use of voltage source that increments its output voltage linearly at the rate of 1 V per second. The voltage value is then the same as the time value, as long as the maximum time is not exceeded.

Simulating the design

Simulation of the design can be done using different analysis configurations. The ones used in this lab will be transient and Fourier analysis.
The analysis (and simulation) of the design is started by selecting Simulate->Analysis->Transient analysis or Simulate->Analysis->Fourier analysis respectively. This will start the simulator, which will store the waveforms of all nodes in the circuit for future presentation. The simulator opens a new window named Grapher View, in which all waveforms are presented.

Existing simulation variables

All voltages and currents in the circuit are available after simulation. The voltages at individual nodes are accessed using the names V(1) etc. All nodes are either named explicitly or enumerated and shown as a red text or digit in the schematic. I(Rs) gives the current entering Rs. The currents usually are assuming the positive current entering the 1st pin of the symbol.
Plotting the voltage across a given component is then done by calculating the voltage difference between the node voltages the component is connected to.

Presentation and analysis of the result

The Grapher View window presents the waveforms of some selected voltages and currents. Individual traces can be disabled by deselecting the corresponding white box at the bottom of the window.

New traces can be added using Graph->Add trace(s) from latest simulation result. In the resulting dialog window can additional traces be added to the existing graph, or to a new graph. Select the trace of interest, press Copy variable to expression, then press Calculate.

Beside currents, node voltages and power traces, additional traces can be calculated using mathematical expression. Among the simplest examples of this is the calculation of the voltage across a component. E. g., if a component is connected between nodes 3 and 5 (assuming + on node 3), the voltage across that component is then calculated using the expression V(3) – V(5).

Other functions may also be used, such as RMS and AVG, which calculates the rms and average values respectively of a signal or expression. Note that these calculations is made on the calculated waveform, and is therefore different at different times, as the calculation is not performed on an infinite long waveform.

Arbitrary mathematical functions can also be plotted by use of a time variable (using the voltage of a triangle wave voltage source). A sinusiodal waveform of 10 V, 50 Hz with a phase shift of 45 degrees can be plotted using the trace entry 10*sin(2*pi*50*V(time)+45*pi/180). Note that the angles are always described in radians.

The Fourier transform can be calculated on signals and expressions. Select Simulate->Analysis->Fourier analysis. Set the fundamental frequency, number of harmonics, and stop time for sampling. Select the output tab, and add there the variables and expressions that will have their Fourier series coefficients calculated. Finally, press Simulate. The simulation is now run, and the simulation result is used to calculate the Fourier series coefficients and then present them together with details about DC component and distortion factor THD in the Grapher View window.

Waveform results can be copied using Edit->Copy graph to clipboard and then pasted into a LibreOffice or Wordpad document or Paint for editing.