Simulation exercise 1: Boost converter

The simulation model of a boost converter will be created by using Matlab Simulink. The operation principle of the boost converter will be analyzed by using the created model.

1. Start Matlab and press from the left-up corner ***Home -> New -> Simulink Model***

   or by clicking 'Simulink Library' button in Matlab toolbar. Create a new simulation model ('File' -> 'New' -> 'Model'). The idea in Simulink is to simply drag and drop blocks from the library browser to the model window and then connect them together. After the model is complete, simulation is started by pressing the green 'play' button.

Option 1: Create the model by using SimSpace library

2. Create the Boost converter simulation model by using the conventional Simulink-blocks (constant, repeating sequence, relational operator and scope) as well as the SimSpace library (DC voltage source, series RLC branch, current measurement, ideal switch, diode, controlled current source, voltage measurement and powergui-block). Those are found by using ***Library browser***. One example is shown in Fig. 1.

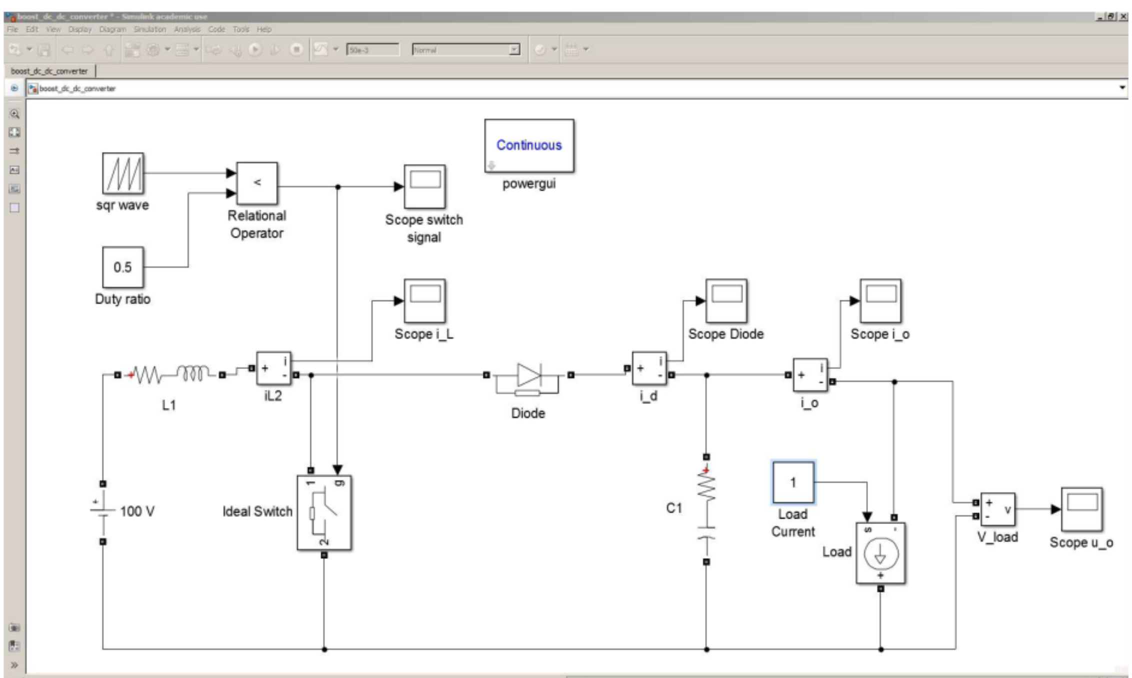

Fig. 1. Boost-converter model

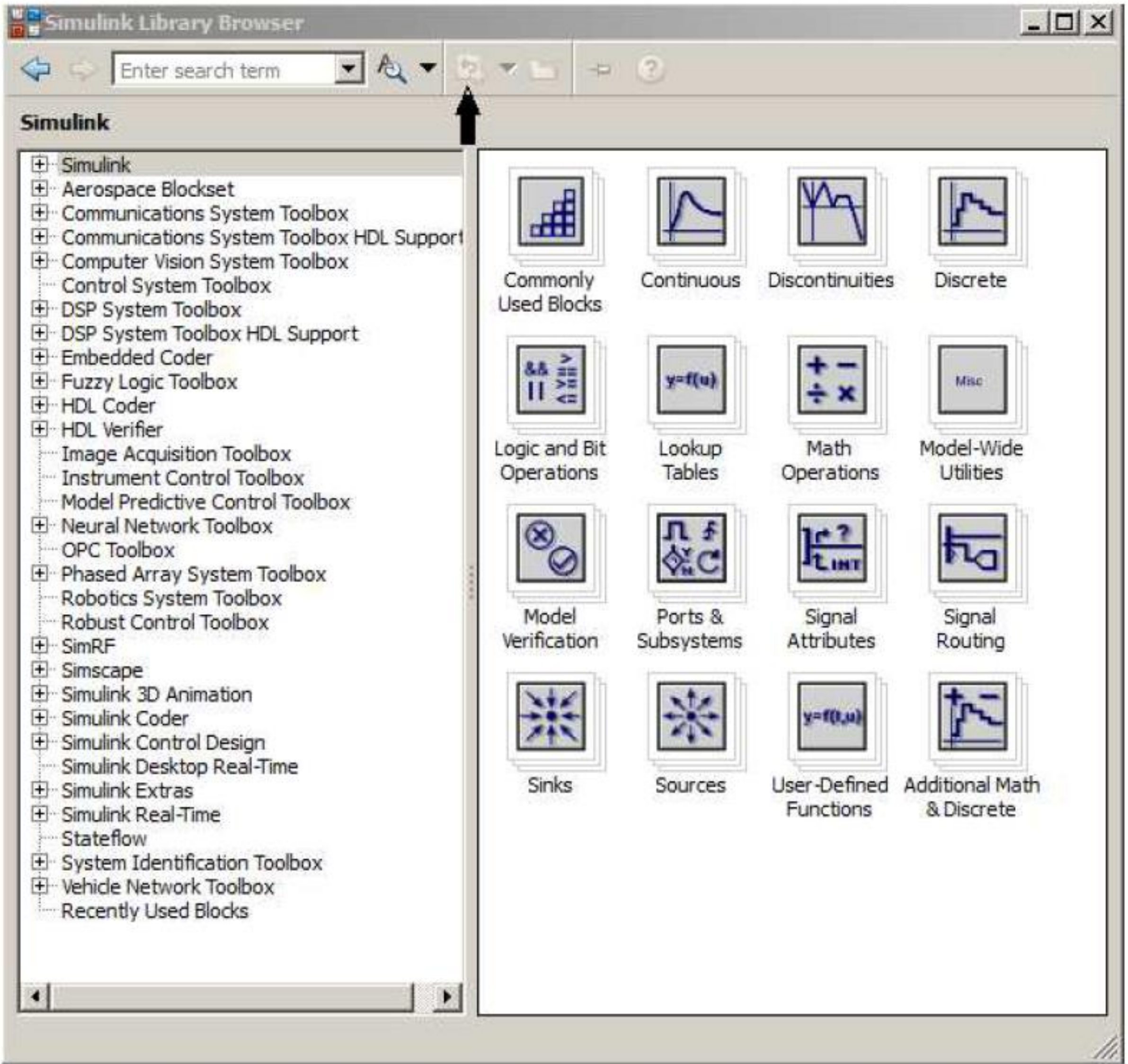

Fig. 2. Library browser

Option 2: Create the model by using only basic Simulink blocks

The equation relating inductor current and voltage is a first order differential equation

$$v_L = L\frac{di}{dt}$$

For simulation purposes, differential equations should be transformed to integral form. The reason is that the derivative block in Simulink does not work correctly when there is discontinuities in voltage and currents i.e. switching. The integral form of the previous equations is

$$i_L = \frac{1}{L}\int_0^t v_L dt + i_L(0)$$

Since we are only interested of the steady state, the initial inductor current can be chosen arbitrary. The inductor voltage equals

$$v_{in} = v_L$$

Thus the inductor current when $i_L(0) = 0$

$$i_L = \frac{1}{L}\int_0^t v_{in} dt$$

Simulink impelementation can be created by using ***Gain***-block and ***Integrator***-block (continuous, 1/s).

The output voltage depend on the state of the main switch. This switching action can be modelled using ***switch***-block as shown in Fig. 3. It is recommended to set the threshold in Switch block parameters to 0.5.

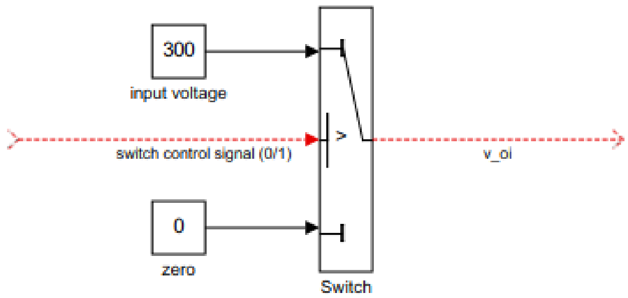

Fig. 3. Switch-block

The equation relating capacitor voltage and current in integral form is

$$i_C = C\frac{dv_C}{dt} \Rightarrow v_C = \frac{1}{C}\int_0^t i_C dt + v_C(0) = \frac{1}{C}\int_0^t (i_D - i_o) dt$$

After the capacitor voltage is known, the load current can be computed from

$$i_o = \frac{1}{R} v_o$$

3. Set the simulation parameters by opening ***Model Configuration Parameters***
   - ***Simulation stop time*** 5ms i.e. 5e-3
   - Solver type: Variable step
   - Additional options: ***Max step size*** 1e-6, ***Initial step size*** 1e-6 and ***Relative tolerance*** 1e-5
   - ***Diagnostics*** -> ***Connectivity*** -> ***Mux blocks used to create bus signals: error***

4. The component parameters

   Give the following parameters for the components
   - DC voltage source amplitude 10V
   - Inductor inductance 10mH and internal resistance 1Ω
   - Capacitor capacitance 100µF and internal resistance 20mΩ
   - MOSFET on-time resistance 0,01Ω and snubber resistance 1e10Ω
   - Diode on-time resistance 0,01Ω and snubber resistance 1e5Ω
   - Load resistor 470Ω

   PWM

   - Square wave time values [0 1e-4] and output values [0 1]
   - Duty ratio 0.5
   - Relational operator =<

In a conventional pulse-width modulation (PWM) scheme, a high frequency carrier signal is compared to a slowly varying or constant control signal. The output of the PWM process is digital, i.e. it is logic one when the magnitude of the control signal is higher than the carrier signal and vice versa. The output of the PWM circuit is always connected to a control input of a switch through a driver circuit. In the simulation model, an ideal switch is used and thus modeling of the gate drive circuit is not required.

A second example Simulink implementation of the pulse-width modulator is shown in Fig. 4.

The comparator is implemented using the 'sum' block ('math operations' library) and the relay block ('discontinuities' library).

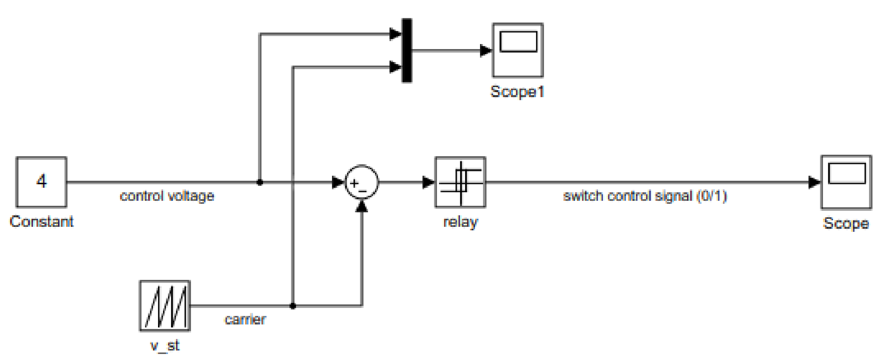

Fig. 4. Pulse-width modulator (naturally sampled trailing edge modulation)

A ***scope*** provides the most convenient way to view the simulation results. A ***mux***-block can be used to collect signals and view them in same window. By default, scope only stores limited amount of data points, but this feature can be turned off from scope parameter (history tab) by unchecking 'limit data points to last'.

Simulation exercises

1. What is the switching frequency of MOSFET?
2. Why the resistances are added to models of inductor and capacitor?
3. What is the output voltage value of boost converter when the input voltage is 10V and duty ratio D = 0.5? Is the value the same by simulations than by calculations? Why/why not?
4. Calculate the inductor current average value and current ripple by using the following parameters. Do you get the same values by using the simulation model. Why/why not?
5. Change the duty ratio to be D=0.3. What is now the output voltage value of boost converter? Is the value the same by simulations than by calculations? Why/why not?
6. Change the value of the passive components and the switching frequency of MOSFET. Analyze the effect on inductor current and output voltage ripple. What happens if the inductance value is very small (DCM operating mode)?
7. Why it is crucial to set the correct simulation step size? Test e.g the values of 1e-2 and 1e-7 to the maximum simulation step size. Test the operation of the simulation model with variable and constant simulation time step.

   The figures may look better if their color scheme is changed. Set e.g. figure color to white and reverse the axis colors (white background, black grid) from style tab of the scope parameters. Remember to zoom the waveforms so that only few switching periods are visible. In this exercise, of interest is the steady-state behavior, not e.g. the start-up transient.

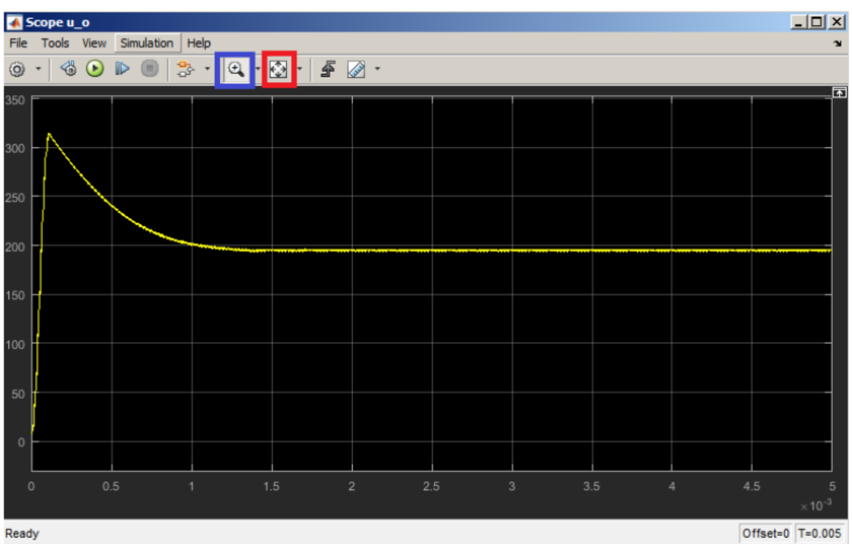

Fig. 5. Example figure in Scope

Report

Write a short report about the results

The report should include the following parts

1. Title of the report, name of the reporter and date

2. Introduction
   Describe shortly what have been done and why.

3. Theory

   Present shortly the theory, which is analysed in the simulation exercise. What kind of power converter is used? Why? What is the operation principle of the converter? Name some applications where the converter is used.

4. Simulation model

   Present the structure of the simulation model and the used parameters.

5. Results

   Present the simulated results and analyse the results, reply to questions of the exercises

Figures

- Use plot-commands in Matlab, not screenshots!
- Name figures as an example below. Refer to the figure in the text e.g. "the magnitude decrease as the frequency decrease as shown in Fig. 2".
- Name x and y-axis
- Scale the figure axis and size on that way, that the interesting phenomena is easy to recognize

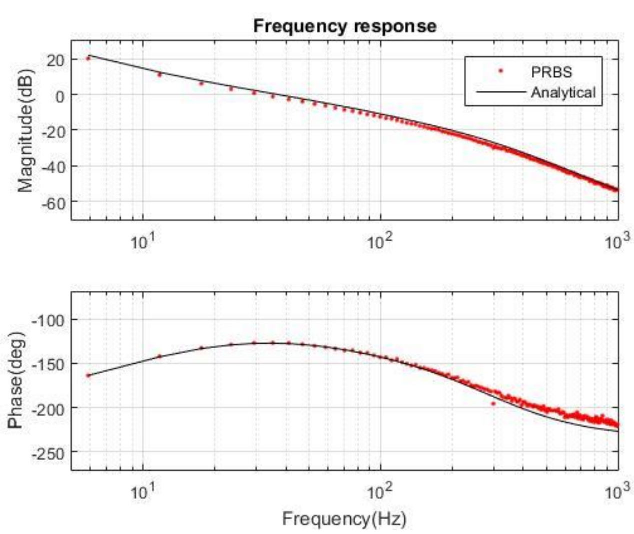

Fig. 2. *Analytical and measured frequency response of PLL loop gain*

Note, what means analyse! For example, “the red and black curve are almost the same, the red curve is a bit higher in the lower figure than the black curve” do not analyse Fig. 2. Instead, you should write e.g. “it can be shown from figure 2, that the frequency response measured by using PRBS methods gives highly accurate results. However, the measured phase slightly differs from the analytically calculated phase at high frequencies because of different operating point”.

6. Conclusions

Give conclusions of the work; what was done and how did you succeed, what did you learn.

7. Feedback

You can give some feedback about the exercises if you want. You can also give feedback anonymously through Google-feedback system.

8. References

If you used some references, mentioned them. Lecture slides do not need to mentioned.