

Lab 1 Power electronics

Contents

Introduction.....	1
Initial setup	2
Starting the software	2
Notes on the schematics.....	2
Simulating the design.....	2
Existing simulation variables	3
Extra measurement points	3
Presentation and analysis of the result	3
Lab1-1 Full-bridge diode rectifier.....	4
Lab 1-1a.....	4
DC-side filtering	5
Inrush current	7
Lab 1-1b Improve ripple of FB diode rectifier	7
Pre-calculation of circuit parameters	8
Lab 1-1b specification	8
Lab 1-1b results:.....	9
Reference.....	9
Lab 1-2 3-phase thyristor 6-pulse converter	10
Lab 1-2 tasks:.....	13
Reference:.....	14
Lab 1-3 Step-down (Buck) DC/DC-converter	15
Reference.....	15

Introduction

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

Tomas Jonsson

Initial setup

The design files used in the lab can be copied from **/site/edu/eks/TSTE19/current/material/Lab1_files** in Linux or from **U:\eks\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** on Windows.

Please note that the software only works on Windows. You can thus only run the software in the Freja and Transistorn labs.

Starting the software

The software is started by **Start ->All Programs ->National Instruments ->Circuit Design Suite 14.0 ->Multisim 14.0**

Open the corresponding design project using **File->Open** and select the design file to use among the files you copied in the initial setup.

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 Graph 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.

Extra measurement points

Additional measurements of voltage and current can be added through placing probes in the Multisim circuit by selecting

Place->Probe->Voltage (Current or Differential voltage).

OBS these probes cannot be used for harmonic Fourier analysis.

Presentation and analysis of the result

The Graph 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 additional traces can 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. Example: **RMS(V(1))**.

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 the use of a time variable (using

the voltage of a triangle wave voltage source). A sinusoidal waveform of 10 V, 50 Hz with a phase shift of 45 degrees can be plotted using the trace entry $10*\sin(2*\pi*50*V(\text{time})+45*\pi/180)$. 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 Graph 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.

Lab1-1 Full-bridge diode rectifier

In this task we are going to analyse and improve the performance of a full-bridge rectifier.

Lab 1-1a

The first task is based on the circuit shown below, where a resistive load is connected directly to the rectifier.

Load the file FB_RECT1.ms14

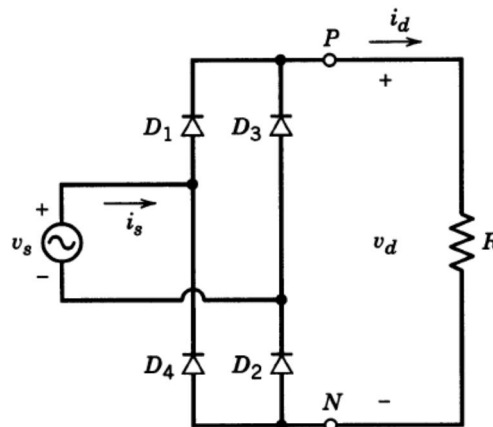


Figure 1

Make the following measurements as a reference to use for the continued work improving the rectifier performance:

1. Average dc-side voltage (create an output function (as discussed in "Presentation and analysis of the result"): $AVG(V(6))$).
2. Peak dc-side voltage
3. The first 9 harmonics in the dc-side voltage (Use the $V(6)$ signal)
4. The first 9 harmonics in the source current (Use the $I(VS)$ signal, OBS that the reference direction is into the source)

Calculate the following based on the measurements above:

- a) DPF.
- b) THD in the source current
- c) P_{s1} , the source side fundamental active power
- d) P_{dcav} , the dc-side average power
- e) $P_{dc2}, P_{dc4}, P_{dc6}, P_{dc8}$ the dc-side 2nd, 4th, 6th, and 8th harmonic power (Note that the given harmonic magnitudes are peak values)
- f) Total dc-side power

Plots to present:

1. DC-side voltage together with source voltage
2. Source current together with diode D1 and D3 currents

DC-side filtering

As found in task 1-1a, very high ripple voltage is obtained on the dc-side if a resistive load is connected directly to the rectifier. Practical single phase full bridge rectifiers normally need dc-side filtering to obtain acceptable ripple.

A common solution is to connect a capacitor as shown below, in order to obtain a sufficiently low voltage ripple.

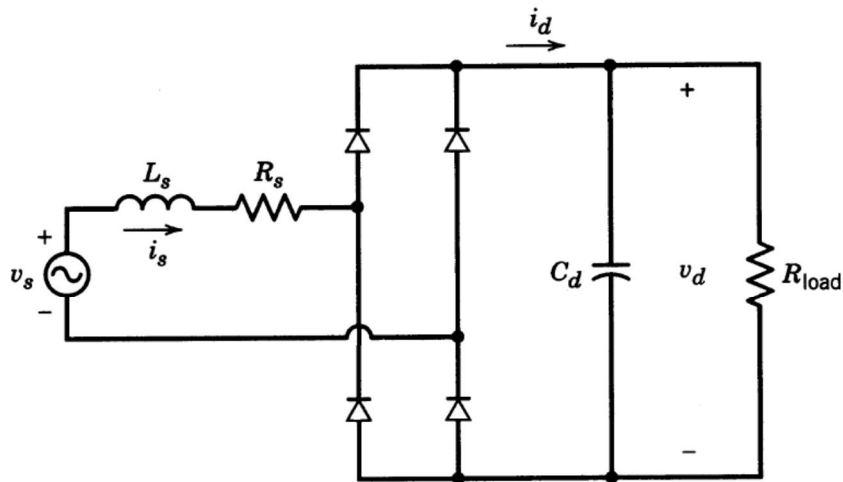


Figure 5-20 Practical diode-bridge rectifier with a filter capacitor.

This circuit is difficult to analyze analytically due to non-continuous current and the second order nature of the circuit related to both capacitance and inductance. Chapter 5-3-4-1 of the course book presents an equation setup which can be used for iterative solution. One half cycle of the operation can be represented by the equivalent circuit in Figure 5-21.

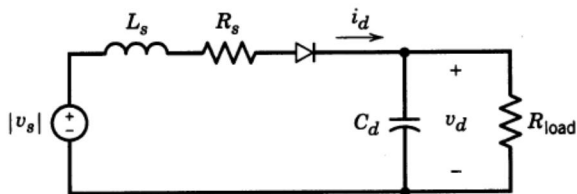


Figure 5-21 Equivalent circuit of Fig. 5-20.

The numerical analysis of this equivalent circuit gives voltage and current as shown in figure 5-22 below. OBS in this example $V_s=120$ Vrms, 60 Hz.

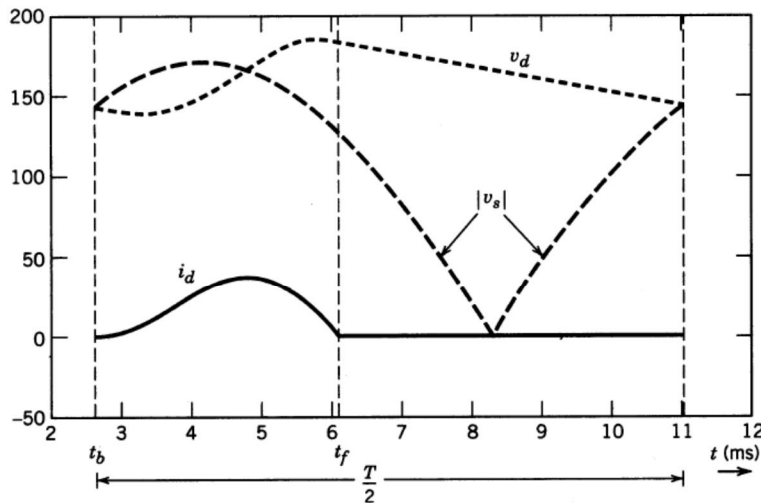


Figure 5-22 Waveforms in the circuit of Fig. 5-20, obtained in Example 5-1.

For a full-bridge rectifier with constant current load we have learnt that the average $U_d = 0.9V_s$, which in this case should give $U_d = 0.9 \cdot 120 = 108\text{V}$. The resulting average dc-voltage from the circuit is significantly higher as seen in Figure 5-22. The practical behavior is explained by the dc-capacitor is being charged to a level around the peak of the source voltage during the first half of the source voltage pulse. The second half, since the dc-voltage is higher than the source voltage, the diode will not conduct. Therefore the dc-voltage will drop as defined by discharge through the load resistance.

Inrush current

When connecting a dc-capacitor to the rectifier, the startup of the circuit when connected to the ac-source results in charging of this capacitor from zero. This charging current may be unacceptably high and due to the inductance provided by the transformer a LC circuit is obtained, as shown in the equivalent circuit in Figure 5-21, which could result in a transient over-voltage on the dc-side.

Lab 1-1b Improve ripple of FB diode rectifier

In this task you shall improve the full-bridge rectifier from Lab 1-1a in order to get lower dc-side ripple voltage.

Use the file from Lab 1-1 or load FB_RECT2.ms14.

The rectifier still supplies a resistive load of 40 ohm, 1kW. The rectifier is connected to 230Vrms 50Hz through a transformer. The transformer has a leakage inductance of 15mH and a resistance of 0.050 ohm.

The task is to modify the circuit in order to meet the specifications defined below. The following parameters may be changed:

- Dc-capacitance, C_d
- Dc-inductance, L_d
- Transformer turns ratio

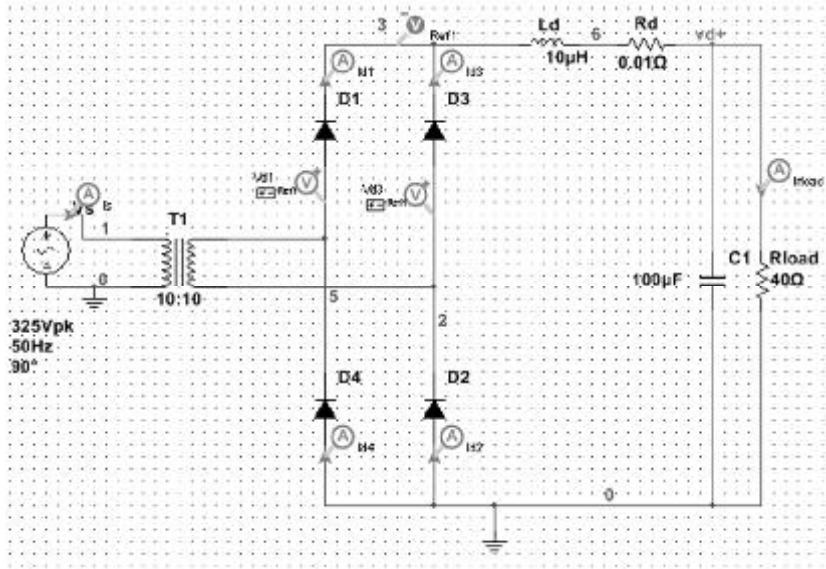


Figure 2

Lab 1-1b specification

The dc-voltage of the load shall be $200V \pm 2\%$ with a peak-peak ripple less than 5%. During startup, the peak dc-voltage must not exceed 230V and the peak inrush current shall be less than 30A. Startup shall be verified for arbitrary phase angles of the source voltage.

In order to obtain a practically realistic solution you should minimize the additional cost related to L_d and C_d . The cost is defined in relation to the stored magnetic and electric energy of L_d and C_d , respectively.

$$Cost(L_d) = 1000 \frac{kr}{J} = 1000 * 0.5LI_{rms}^2 :$$

$$Cost(C_d) = 10 \frac{kr}{J} = 10 * 0.5CU_{av}^2 :$$

Calculate and present the cost for L_d and C_d based on the steady state rms current and voltage which these components are seeing.

Pre-calculation of circuit parameters

The dc-voltage ripple is as found in task 1-1a mainly a 2nd harmonic voltage. The capacitance can be estimated by considering a source voltage of second harmonic defined by the value obtain in task 1-1a. Assume this voltage is driving a 2nd harmonic current through a series connection of L_s (here this is the transformer inductance) L_d and C_d .

$$I_{d2} = \frac{U_{d2}}{\omega L - \frac{1}{\omega C}} U_{c2} = I_{d2} \frac{1}{\omega C}$$

$$U_{c2} = I_{d2} \frac{1}{\omega C}$$

Consequently, the required capacitance C_d can be calculated based on the 2nd harmonic voltage from the rectifier, the total inductance of the circuit ($L_s + L_d$) and the specified ripple voltage ($U_{c2,max}$) of the capacitor voltage. Remember that ω shall be related to the 2nd harmonic.

$$C = \frac{1 + \frac{U_{d2}}{U_{c2,max}} T_O}{\omega^2 L}$$

$$\hat{I}_{LC} = \frac{\hat{V}_s}{\sqrt{L/C}}$$

Now we have two equations in order to solve for the two unknowns: L_d and C_d .

The unique solution which precisely satisfies the requirement on ripple voltage and inrush current is not necessarily the cheapest.

Lab 1-1b results:

Present the following output from the simulation:

1. Selected values for L_d and C_d
2. Transformer turns ratio.
3. The total cost of L_d and C_d based on the RMS current and voltage at the load. Let's see who presents the design with the lowest cost which meet the spec.
4. Graph of the dc-voltage from startup until steady state is reached.
5. Graph showing the inrush current to the rectifier
6. Graph with zoom in of the ripple in the dc-voltage

Reference

Section 5-3, 5-8 in Mohan Power Electronics

Lab 1-2 3-phase thyristor 6-pulse converter

This task is to analyse a thyristor converter in different operating modes.

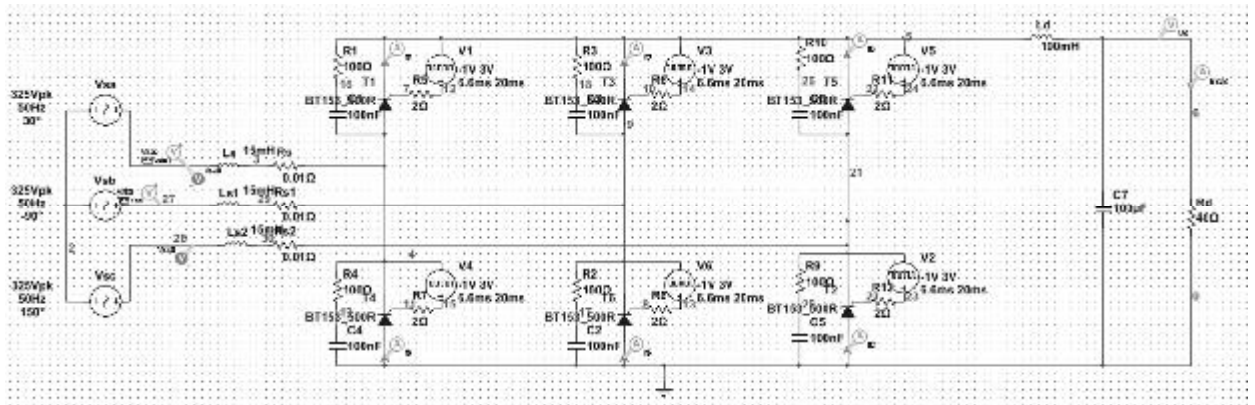


Figure 3

The converter is connected to a 3-phase ac-source with a line-line voltage of 400V, 50Hz. Phase a is shifted 30 deg in order to obtain the valve voltage of thyristor 1 in a phase position where the zero crossing starts at the beginning of a 20ms cycle as shown in Figure 4 below.

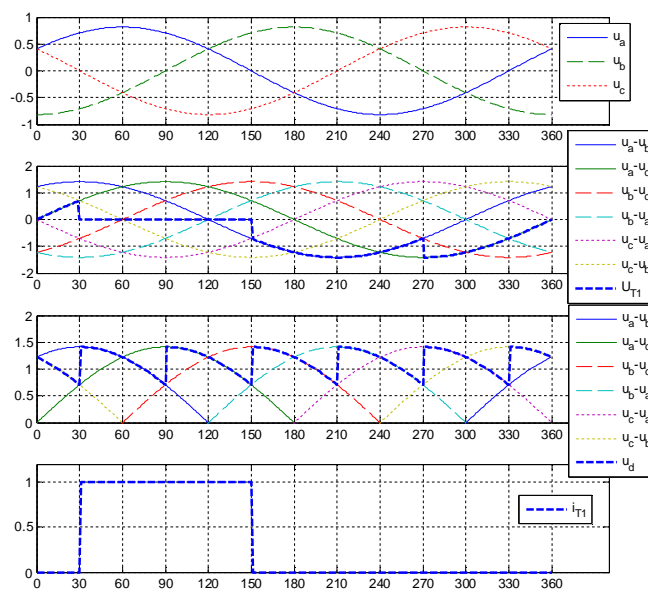


Figure 4

The voltage U_a - U_c which defines the turn-on of thyristor T1 here starts at $t=0$.

The circuit in Multisim shall be loaded from the file Thy_Rect_3ph.ms14

All thyristors are equipped with R-C snubbers in order to ensure the simulation to converge. Related to the high dV/dt at turn-on the simulation would otherwise stop for non-convergence.

Each gate is controlled by a voltage source with a fixed timing. The figure below shows the timing for $\alpha=15$ deg.

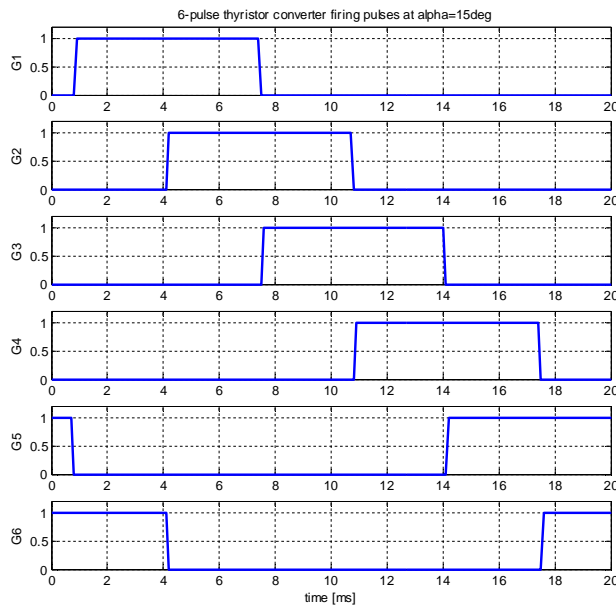


Figure 5

The gate pulses are defined by the Pulse_Voltage source, which has parameters according to Figure 6. Here the "delay time" defines the first instant of the pulse. "Pulse width" defines the duration, which is set to 6.6 ms corresponding to somewhat below 120 deg. Finally, the "period" is set to 20 ms in order to get a repetition frequency corresponding to the 50 Hz system voltage.

The "delay time" of thyristor 1 shall always be set to 20 ms plus the time corresponding to the firing angle. This will result in the converter to start at 20ms.

$$T_{\text{delay}}(T1) = 0.020 + \frac{\text{rad}(\alpha)}{\omega} \text{ [s]}$$

$$T_{\text{delay}}(T2) = 0.020 + \frac{\text{rad}(\alpha+60)}{\omega} \text{ [s]}$$

$$T_{\text{delay}}(T3) = 0.020 + \frac{\text{rad}(\alpha+120)}{\omega} \text{ [s]}$$

$$T_{\text{delay}}(T4) = 0.020 + \frac{\text{rad}(\alpha+180)}{\omega} \text{ [s]}$$

$$T_{\text{delay}}(T5) = 0.020 + \frac{\text{rad}(\alpha+240)}{\omega} \text{ [s]}$$

$$T_{\text{delay}}(T6) = 0.020 + \frac{\text{rad}(\alpha+300)}{\omega} \text{ [s]}$$

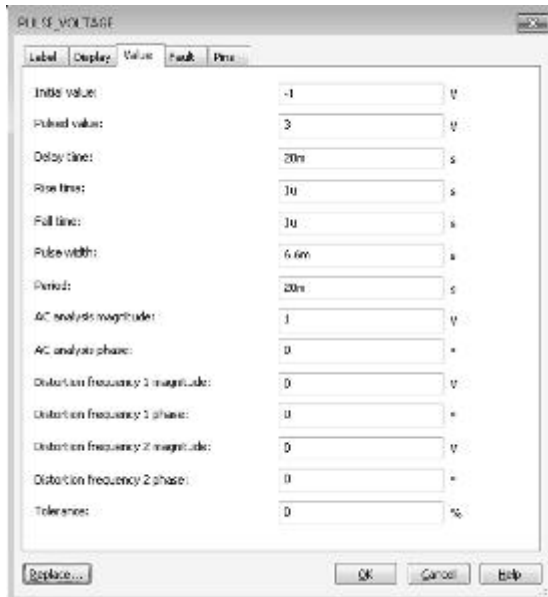


Figure 6

The pulse sources are initially setup for a firing angle of $\alpha=0$ deg.

Lab 1-2 tasks:

- Calculate the individual thyristor turn-on times for $\alpha=18$ deg. Save the simulation setup updated for $\alpha=18$ deg to different case to easily be able to switch between $\alpha=0$ deg and $\alpha=18$ deg.
- Make a plot for each case ($\alpha=0$ deg, $\alpha=18$ deg) showing one cycle under steady-state conditions:
 - the dc-voltage before and after L_d ,
 - thyristor 1 and 3 voltages
 - source voltages $U_{sa} - U_{sc}$ and $U_{sa} - U_{sb}$ in order to relate to the firing of thyristors 1 and 3.
- Make a separate plot for each case ($\alpha=0$ deg, $\alpha=18$ deg) in order to determine the phase angle between source voltage and current.
 - source voltages U_{sa} and the current I_{sa} .
- Use the plots to measure the quantities given by Table 1 below at $\alpha=0$ deg, $\alpha=18$ deg. Hint: the source current at 50Hz may be calculated as defined in Table 1.
- Why can't an inverter be studied using the Multisim setup in Figure 3 used above?
- Load case Thy_Inv_3ph.ms4 which defines a thyristor inverter.

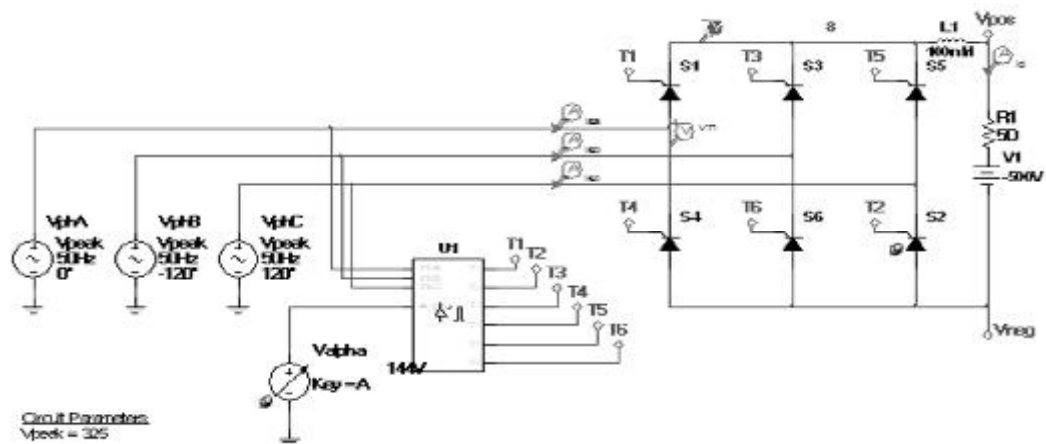


Figure 7

This setup has a dc-side voltage source with the voltage -500V with respect to the terminals $V_{pos} - V_{neg}$. Thereby power can be transferred from the dc-side to the ac-side. The thyristor firing control is handled by the block U1 which is synchronized to the ac-source voltages and has a reference V_{alpha} where the firing angle is defined in degrees.

This circuit is ideal in the sense it does not include any inductance on the ac-side.

- g) What does it mean for the commutation angle, u , not having any ac-side inductance?
- h) Use the setup in Figure 7 to study operation at $\alpha=90$ deg and $\alpha=144$ deg
- i) Make a plot for each case ($\alpha=90$ deg, $\alpha=144$ deg) showing one cycle under steady-state conditions:
- the dc-voltage before and after L_d ,
 - thyristor 1 and 3 voltages
 - source voltages $U_{sa} - U_{sc}$ and $U_{sa} - U_{sb}$.
- j) Make a separate plot for each case ($\alpha=90$ deg, $\alpha=144$ deg) in order to determine the phase angle between source voltage and current.
- source voltages U_{sa} and the current I_{sa} .
- k) Use the plots to measure the quantities given by Table 1 below at $\alpha=90$ deg, $\alpha=144$ deg.

Table 1

		$\alpha=0$ deg	$\alpha=18$ deg	$\alpha=90$ deg	$\alpha=144$ deg
U_d	V				
I_d	A				
$I_{s1}=0.78 \cdot I_d$	A				
DPF ($\cos(\phi_i)$)					
P_{s1}	W				
Q_{s1}	VAR				
u	deg				
γ	deg				

Reference:

Section 6-4 in Mohan Power Electronics

Lab 1-3 Step-down (Buck) DC/DC-converter

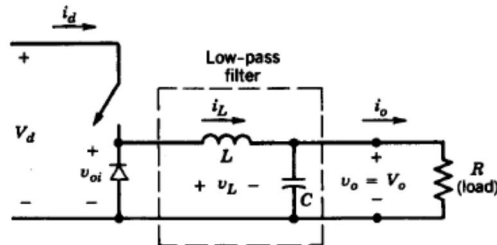


Figure 8

In this task you shall design a step down DC/DC-converter for supply of a 12V resistive load of 240W from a dc-source of 36V. The switching frequency is 10 kHz.

Load a base case for Multisim named: Buckconv.ms4

Perform the following design steps:

1. Determine the duty cycle of the switch
2. Determine the minimum L in order to get a continuous current at rated load.
3. Determine C in order to get a maximum peak-peak output voltage ripple of 10%

Hint: Make an excel sheet with the input parameters in separate cells and the formulas for calculation of L and C. Make also a case with fixed L and C in order to be able to calculate the operating conditions at varying voltage or load.

Run the simulation and make sure the results are as expected. Check the simulation time step is short enough in order to give reliable results. Thereafter, prepare the following:

1. Graph showing: Input current, inductor current, load current, switched voltage and DC-output voltage
2. What is the peak current in the switch and the diode?
3. If the input voltage drops to 24V, what will the new duty cycle be in order to maintain 12V at the output?
4. Graph showing: Input current, inductor current, load current, switched voltage and DC-output voltage.
5. Is the inductor current still continuous?
6. What is now the peak current in the converter?

Reference

Section 7-3 in Mohan Power electronics