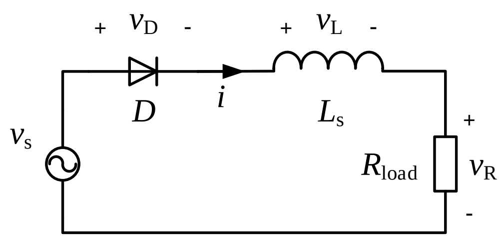
Assignment 1
Basic Circuits and Fourier Series

Observe that a short description of how to run the Cadence 'OrCAD Capture CIS Lite' (PSpice) software is given at the end of this document.

The questions marked HA are home assignments to be completed before the laboratory starts.

Task A – Basic Power Electronic Circuits



Nominal Values

Source voltage	$v_{s(AMP)} = 230$V
Source frequency	$f_s = 50$Hz
Inductance	$L_s = 10$mH
Resistance	$R_{load} = 5\Omega$

Source file: *assignment_1.opj* – Task_A

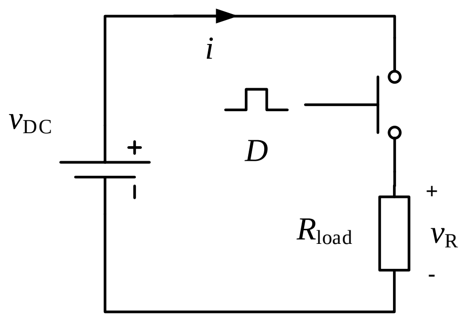
Tasks/Questions

1. Run the simulation and analyse the waveforms of v_s, v_D, v_L, v_R and i. Why does the current keep flowing after the supply voltage has gone to zero? What is the average value of the load voltage V_R? What is the peak reverse voltage of the diode D?

HA 1: Can the $j\omega$-method be used to calculate the amplitude of v_R? can you predict the average inductor voltage V_L in steady state? Confirm your answer using the simulation.

2. Now connect the 100V DC-source. Obtain the waveforms for v_s, v_D, v_L, v_R and i. When does the current peak occur and why? What is the RMS value of the inductor voltage? What is the peak reverse voltage of the diode D in this case?

Task B – Calculation of Fourier Coefficients



Nominal Values

Source voltage	$v_{DC} = 100$V
Pulse frequency	$f_{sw} = 100$kHz
Resistance	$R_{load} = 20\ \Omega$

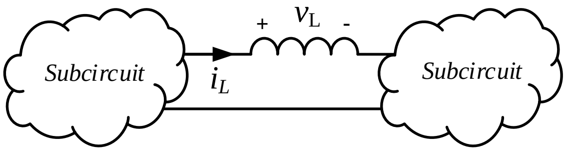
Source file: *assignment_1.opj* – Task_B

Tasks/Questions

1. Run the simulation and analyze the voltage over the load v_R. How does the average value relate to the duty cycle?
2. Obtain the first 5 components of the Fourier series of the load voltage (v_R) by using the method described under '**Obtaining mean DC-value and performing harmonic analysis'** below.

HA 2: Calculate the first 5 Fourier components for D = 0.5 and compare it with the simulation.

Task C – The Inductor



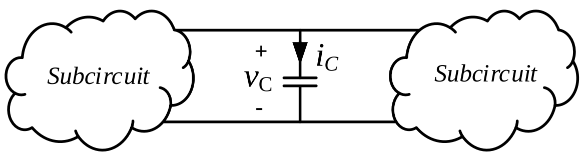
Nominal Values
Source frequency f_s = 500kHz
Inductance L = 10µH

Source file: *assignment_1.opj* – Task_C

Tasks/Questions

1. Run the simulation and analyse the waveforms of v_L and i_L. Explain the behaviour of the current and voltage. Is the circuit operating in steady state? Change the value in the field **'Implementation = SubCircuit1a' to 'SubCircuit1b'** and rerun the simulation. Is the circuit operating in steady state now and why?
2. Change the value in the field **'Implementation = SubCircuit1b' to 'SubCircuit1c'**. Run the simulation again and explain the behaviour of the currents and voltages in the circuit.

Task D – The Capacitor



Nominal Values
Source frequency f_s = 500kHz
Inductance C = 500nF

Source file: *assignment_1.opj* – Task_D

Tasks/Questions

1. Run the simulation and analyse the waveforms of v_C and i_C. Explain the behaviour of the current and voltage. Is the circuit operating in steady state? Change the value in the field **'Implementation = SubCircuit3a' to 'SubCircuit3b'** and rerun the simulation. Is the circuit operating in steady state now and why?
2. Change the value in the field **'Implementation = SubCircuit3b' to 'SubCircuit3c'**. Run the simulation again and explain the behaviour of the currents and voltages in the circuit.

Install OrCAD 16.6 Lite Software

You can download and install the PSpice software on your own computer. The demo version for the course can be requested here: http://www.orcad.com/buy/try-orcad-for-free. State that you are a student at Chalmers University of Technology in the registration in order to get the download link via email.

The "**OrCAD 16.6 Lite Software (Capture / PSpice only, download)**" package is enough to run the PSpice Assignments in the course and that is the version that is installed in the computers at Chalmers. But, you are free to choose the latest version as well.

Once the program is installed, a brief video introduction can be found here https://www.youtube.com/watch?v=dZUPBLNuaHk&app=desktop on how to start up the software and build a simple RLC circuit. More details on the software use can be obtained from the user manual http://www.seas.upenn.edu/~jan/spice/PSpice_CaptureGuideOrCAD.pdf.

In this course, the models will be prepared before each session to save time and the simulation files can be downloaded from the course home page. A brief introduction of how to run these simulation files and perform measurements and analysis is given below.

Brief description of running OrCAD Capture CIS Lite

General:
The program is a demo version of the full program with limited functionalities (a limited number of components and nodes can be used in the schematic). Like most programs, it is not extremely trivial; but with some patience, you will be able to simulate electronic circuits.

Files:
The P-Spice files can be directly downloaded from the course website. Create your own folder on your private account where you can save all simulation results. This is where you should store your files after each occasion.

Starting P-Spice:
Download and extract "Assignment_1.zip" from the course website. Start "OrCAD Capture CIS Lite" from the Cadence folder on the start menu.

Loading a drawing into P-Spice.
Open "assignment_1.opj" from "File/Open/Project…". Open "PAGE1" under "Design Resources/.\assignment_1.dsn /Task_A /"

The buttons you mostly need are "Current Marker", "Voltage/Level Marker", "Voltage Differential Marker(s)", "Run PSpice", "Edit Simulation Profile", "Place Part" and "Place Wire".

Simulating:
Make sure that the "Active Profile" is set according to the circuit that you want to run ("Task_A-diode" for "Task_A"), press the "Run PSpice" button and the program runs. If there is no error, a new window opens in which you can see the chosen voltages and current plots.

Obtaining mean Dc value and performing harmonic analysis:

Press the “PSpice/Edit Simulation Profile” button and click on “Output File Options…”. Click in the box “Perform Fourier Analysis” and set the centre frequency (use the fundamental frequency in the circuit) and the number of harmonics (use 25 for instance). As output values, you shall give the quantities you want to make a Fourier analysis on with the channels being separated with blank spaces. Look in the simulation results for the naming of the current and voltages given in the P-Spice. Rerun the simulation and click on the “View Simulation Output File” on the left side of the graphs when the results appear. If you scroll down you will see the result of the FFT performed on the selected quantities. To have a graphic view of the harmonics, you can also press the **FFT** button in the plot window.

Evaluate Measurements

Press “Trace/Evaluate Measurement…” and add a trace Expression. The functions “RMS()” and “AVG()” may be interesting to you. The cursors can be used for accurate reading of the values.

Printing results:

To save the simulation plots in a nice way: highlight all the curves, right click and choose “Trace Properties”. Under “Width” choose the fourth from the top and click “Ok”. From the “Window” menu choose “Copy to Clipboard...” and click “Ok“. In MS Word, you can now paste the figure where you like to make the presentation as convenient as possible