Jacobs University Bremen

CO-520-B
Signals and Systems Lab

Fall Semester 2021

Course Signals and Systems Lab – CO-520-B

Instructors	- Uwe Pagel, Res.I Room 37	Tel.: +49 421 200 3114
	- u.pagel (at) jacobs-university.de	
	- Prof. Dr.Ing. Werner Henkel	Tel.: +49 421 200 3157
	- w.henkel (at) jacobs-university.de	
Website	- http://www.faculty.jacobs-university.de/upagel	

September 9, 2021

Contents

Part I

General remarks on the course

1. Experiments and Schedule

1. 1.Day - Group 1 - Time & Date TBD.
   Introduction to the Lab
   RLC-Circuits - Transient Response

   1.Day - Group 2 - Time & Date TBD.
   Introduction to the Lab
   RLC-Circuits - Transient Response

2. 2.Day - Group 1 - Time & Date TBD.
   Fourier Series and Fourier Transform

   2.Day - Group 2 - Time & Date TBD.
   Fourier Series and Fourier Transform

3. 3.Day - Group 1 - Time & Date TBD.
   AM Modulation

   3.Day - Group 1 - Time & Date TBD.
   AM Modulation

2. Lab Guidelines

2.1 Grading Scheme

1. All grades are collected in percent according to the Jacobs grading scheme.
2. The lab is a part of the module CO-520-B and counts 30%. The grade is collected by writing lab reports.
3. Distribution of the grades:

Action	Involved	Grade
5 Prelabs (Exp.2-6) each 8%	Individual	40%
3 Lab Reports each 20%	Individual	60%

2.2 Attendance

Attendance to the course is mandatory. Missing an experiment without valid excuse will subtract 1/3 from the grade.

2.3 Prelab

Each student has to be prepared before attending the lab. All students have to be familiar with the subject, the objectives of the experiment and have to be able to answer questions related to the experiment handout and prelab. The prelab has to be prepared in a written form individually by every student. Without prelab the student will be excluded from the lab for this specific experiment!

2.4 Lab Report

It is mandatory to write three lab reports. It is also mandatory to deliver an evaluation for all prelabs. The general structure of a report is known from the first year. Now it should become like this:

- Cover Sheet - as before
- Introduction - prelab belonging to the experiment
- Experimental Set-up and Results - as before
- Evaluation
  Conclusion - is now a combined summary including error discussion
- References - as before
- **Additional:** The prelab of one of the other experiments!!

The submission of the report should be by email to **u.pagel@jacobs-university.de**. Of course in case of computer problems hand written reports are also accepted. Deadline is the Sunday after the experiment at 23:59. If you miss it, the experiment will be downgraded or even reduced to 0%!!! As already stated before lab reports are individual work.

2.5 Cheating & Copying

In case of cheating or plagiarism (marked citations are allowed but no complete copies from a source) we will follow **'The Code of Academic Integrity'**. The report will be counted as **not submitted** 0%.
Note that there can be more consequences of a disciplinary nature depending on the circumstances.

2.6 Supplies

All equipment, cabling and component you need should be in your work area. If you cannot find it, ask your lab instructor or teaching assistant, do not take it from another group. Before leaving the lab, put everything back, where you found it! Please bring your notebook so that you can record the readout the oscilloscope.

2.7 Safety

Recall the Safety Session from first semester!

3. Manual Guideline

The manual and the course web-site contains all necessary information around the course. Beside this the manual includes a description of all experiments. Every experiment is divided in the Objective section and one (or more) sub section(s) with Preparation, Execution, and Evaluation.

The Objective Section should give an introduction to the problem. In some cases it also contains theory not completely covered in the lecture.

The Preparation Section describes the electrical setup.

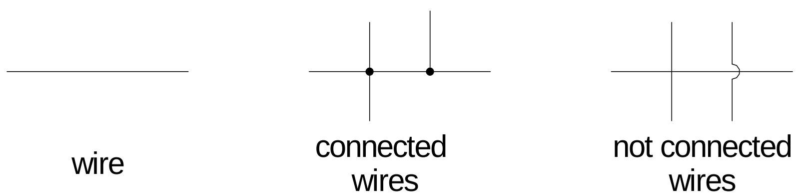
The Execution Section is a detailed description on what to do and how and what to measure.

The Evaluation Section should deepen the understanding of the topic. There are questions about the experiment. You should solve these with help of the taken data and compare the results to theory.
Before you start working on a (sub)section read **-the whole-** section carefully. Try to understand the problem. If something is not clear read again and/or ask the TA or instructor. Follow the preparation carefully to have the right setup and not to destroy any components. Take care that you record **-ALL-** requested data. You may have problems to write a report otherwise!!

3.1 Circuit Diagrams

Next is an overview about the used symbols in circuit diagrams.

Connections

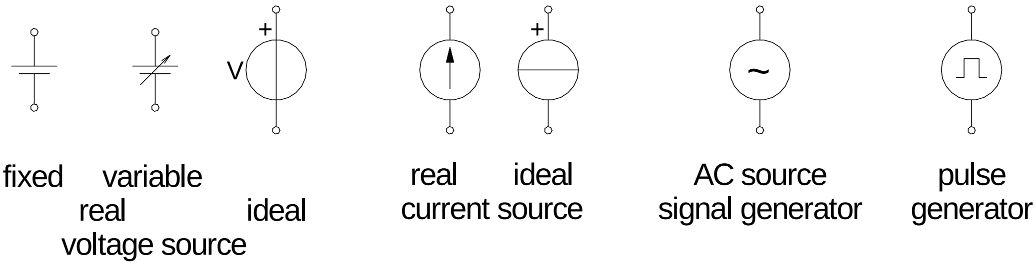


Connection are usually made using 1 or 0.5m flexible lab wires to connect the setup to an instrument or voltage source and short solid copper wires one the breadboard. In most of our experiments we consider these connections as ideal, i.e. a wire is a real short with no 'Impedance'. In the following semesters you will see that this is not true.

Instruments

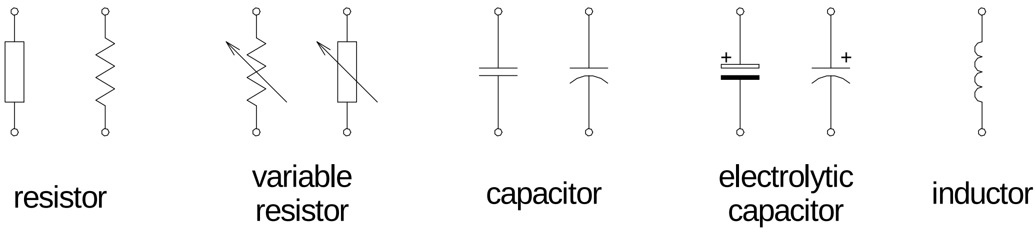
Since we have 'Multimeters' this symbol tells you how to connect and configure the instrument. Take care of the polarity. Be careful, in worst case you blow it!!!

Voltage/Current Sources



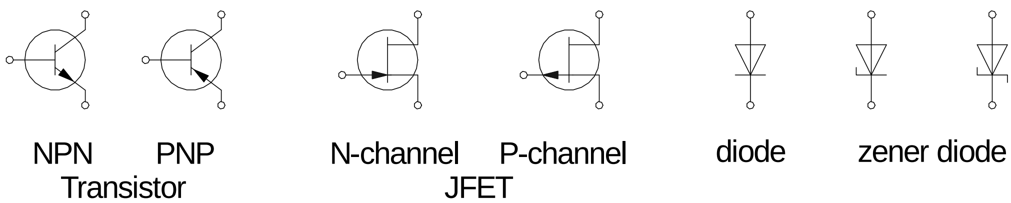
These are the symbols used in the manual. If you check the web and look into different books there are also other symbols in use!

Lumped Circuit Elements



There is a different symbol for every lumped circuit element. Depending which standard is used (DIN or IEC).

Semiconductors



Same as with the symbols before you may find different representations for every component!

3.2 Values in Circuit Diagrams

As you will see in the lab, we use resistors with colored rings. These rings represent numbers or a multiplier. Most of the resistors have five rings. Three digits for the value, one multiplier for the dimension, and one for the tolerance. In the circuit diagrams we have a similar scheme. There are three digits and a dimension. The letter of the dimension also acts as the comma i.e.:

1R00, 10R0, 100R	for	$1\,\Omega$, $10\,\Omega$, $100\,\Omega$	$(= Value * 10^0)$
1K20, 10K0, 100K	for	$1.2\,\mathrm{K}\Omega$, $10\,\mathrm{K}\Omega$, $100\,\mathrm{K}\Omega$	$(= Value * 10^3)$
1M00, 10M0	for	$1\,\mathrm{M}\Omega$, $10\,\mathrm{M}\Omega$	$(= Value * 10^6)$

The numbering for capacitors in the circuit diagram is similar. Only the dimension differs. Instead R, K, M (Ω, KΩ, MΩ) we have μ, n, or p (μF, nF, pF) (i.e. 1n5 means 1.5nF). The value is printed as number on the component.

3.3 Reading before the first Lab Session

As preparation for the first lab session read the description of the workbench, especially the parts about the power supply and the multimeter. You will find the document on the course Web page in 'GeneralEELab I & II Files' *'Instruments used for the Experiments'*.

Part II

Experiments

4. Experiment 1 : RLC-Circuits - Transient Response

4.1 Introduction to the experiment

4.1.1 Objectives of the experiment

The goal of the first experiment of the Signals and Systems Lab, is to study the transient response of second-order systems. Particularly, we will in study the behavior of second-order electrical systems. A typical second-order system in electrical engineering is an RLC circuit. As part of the prelab the transient response of such systems will be studied using Matlab. As part of the experiment, different RLC circuit configurations will be implemented and tested. The experimental and the simulation results will be compared and the differences will be discussed.

4.1.2 Introduction

Second-order systems are very common in nature. They are named second-order systems, as the highest power of derivative in the differential equation describing the system is two.
In electrical engineering, circuits consisting of two energy storage elements, capacitors and inductors, for example RLC circuits, can be described as second-order electrical system. These circuits are frequently used to select or attenuate particular frequency ranges, as in tuning a radio or rejecting noise from the AC power lines.
The handout is divided into two parts. Throughout the first part of the handout the following topics will be discussed.

1. The equation describing a second-order system in its general form.

2. The complete solution for a second-order differential equation (D.E.) representing the complete response of the system. The complete solution consists of two responses.

   (a) The transient response
   In transient response, depending on the circuit parameters the circuit operates under,

      i. Over-damped condition
      ii. Critically damped condition
      iii. Under-damped condition

   (b) The steady-state response
   The steady state response due to a constant input signal (DC source).

The second part of the handout describes the practical part of the experiment. It will explain how to develop a second-order differential equation describing a series RLC circuit configuration and how the differential equation can be solved in order to have a complete solution consisting of the transient response and steady-state response.

4.1.3 Differential equations describing second-order systems

Circuits with two energy storage elements as the RLC circuits are described by a second-order ordinary D.E.

$$a_2 \frac{d^2y(t)}{dt^2} + a_1 \frac{dy(t)}{dt} + a_0 y(t) = x(t) \tag{4.1}$$

where,

$y(t)$	is the response of the system to an applied input $x(t)$.
a_0, a_1 and a_2	are the system parameters.

In the context of the response of second-order systems, it is more useful to rewrite Eq. (4.1) in the form of a linear constant coefficient non-homogeneous differential equation

$$\frac{d^2y(t)}{dt^2} + 2\zeta\omega_n \frac{dy(t)}{dt} + \omega_n^2 y(t) = K\omega_n^2 x(t) \tag{4.2}$$

Thus, the system parameters become

$$\omega_n = \sqrt{\frac{a_0}{a_2}} \tag{4.3}$$

$$\zeta = \frac{a_1}{2\sqrt{a_0 a_2}} \tag{4.4}$$

$$K = \frac{1}{a_0} \tag{4.5}$$

where,

ω_n	is the natural frequency.
ζ	is the damping ratio.
K	is the gain of the system.

4.1.4 The complete solution for a second-order D.E.

The complete solution of the second-order non-homogeneous differential equation, Eq. (4.2), is given by the following sum

$$y = y_h + y_f \tag{4.6}$$

The solution consists of two terms, where y_h is the homogeneous solution of the second-order non-homogeneous differential equation, Eq. (4.2), where the input

$x(t) = 0$. This solution satisfies the initial condition of the system. Therefore, the homogenous solution of the second-order non-homogeneous differential equation describes the transient response of the system. The term y_f is the forced solution of the second-order non-homogeneous differential equation, Eq. (4.2), where an external input signal $x(t) \neq 0$ is applied to the system. Therefore, the forced solution of the second-order non-homogeneous differential equation describes the steady-state response of the system. In the following we will derive in details the homogenous solution and the forced solution for a constant input signal (DC source).

a. The homogeneous solution

The homogeneous solution of the second-order non-homogeneous differential equation can be found by rewriting Eq. (4.2), where the applied input $x(t)$ is equal to 0, so that the non-homogeneous D.E is reduced to a homogeneous equation with constant coefficients

$$\frac{d^2y(t)}{dt^2} + 2\zeta\omega_n\frac{dy(t)}{dt} + \omega_n^2 y(t) = 0 \tag{4.7}$$

This equation has a solution of the form

$$y(t) = Ce^{\lambda t} \tag{4.8}$$

By substituting $y(t)$ from Eq. (4.8) in Eq. (4.7), we get

$$Ce^{\lambda t}(\lambda^2 + 2\zeta\omega_n\lambda + \omega_n^2) = 0 \tag{4.9}$$

From Eq. (4.9), we get

$$\lambda^2 + 2\zeta\omega_n\lambda + \omega_n^2 = 0 \tag{4.10}$$

The above equation is called the characteristic equation. To find the homogeneous solution, we need to solve the characteristic equation. The characteristic equation has two roots (solutions)

$$\lambda_1 = -\zeta\omega_n + \omega_n\sqrt{\zeta^2 - 1} \tag{4.11a}$$

$$\lambda_2 = -\zeta\omega_n - \omega_n\sqrt{\zeta^2 - 1} \tag{4.11b}$$

By substituting in Eq. (4.8), we get

$$y_1(t) = C_1 \exp\left((-\zeta + \sqrt{\zeta^2 - 1})\omega_n t\right) \tag{4.12a}$$

$$y_2(t) = C_2 \exp\left((-\zeta - \sqrt{\zeta^2 - 1})\omega_n t\right) \tag{4.12b}$$

Each root λ_1 and λ_2 contributes a term to the homogeneous solution. The homogeneous solution is

$$y_h = C_1 e^{\lambda_1 t} + C_2 e^{\lambda_2 t} \tag{4.13}$$

where,

C_1 and C_2	are unknown coefficients determined by the initial conditions.
λ_1 and λ_2	are unknown constants determined by the coefficients of the D.E. (depends on the circuit parameters R, L and C).

The time-dependent response of the circuit depends upon the relative values of the damping ratio ζ and the undamped natural frequency ω_n (radians/sec). According to the relative values of ζ and ω_n, we can classify the transient response into three cases:

1. Under-damped case : $0 < \zeta < 1$, λ_1 and λ_2 are complex numbers
2. Critically damped case : $\zeta = 1$, λ_1 and λ_2 are real and equal
3. Over-damped case : $\zeta > 1$, λ_1 and λ_2 are real and unequal

Now, let us describe each case separately.

1. **Under-damped Case:** $0 < \zeta < 1$
   For $0 < \zeta < 1$ the homogeneous solution of the second-order homogeneous differential equation exhibits a damped oscillatory behavior.

   $$y(t) = \exp(-\zeta\omega_n t)(C_1 \cos(\omega_n\sqrt{(1-\zeta^2)}\,t) + C_2 \sin(\omega_n\sqrt{(1-\zeta^2)}\,t)) \tag{4.14}$$

   where,
   C_1 and C_2 are unknown coefficients derived from the initial conditions.
   On defining ω_d, the damped natural frequency is given as

   $$\omega_d = \omega_n\sqrt{1-\zeta^2} \tag{4.15}$$

   Eq. (4.14) can be written as

   $$y(t) = \exp(-\zeta\omega_n t)(C_1 \cos(\omega_d t) + C_2 \sin(\omega_d t)) \tag{4.16}$$

   Thus, the under-damped response is an exponentially damped sinusoid whose rate of decay depends on the factor ζ. The terms $\pm\exp(-\zeta\omega_n t)$ define what is called the envelope of the response. A step response of an under-damped system is shown in Fig. 4.1. The oscillations of decreasing amplitude, exhibited by the waveform are called ringing.

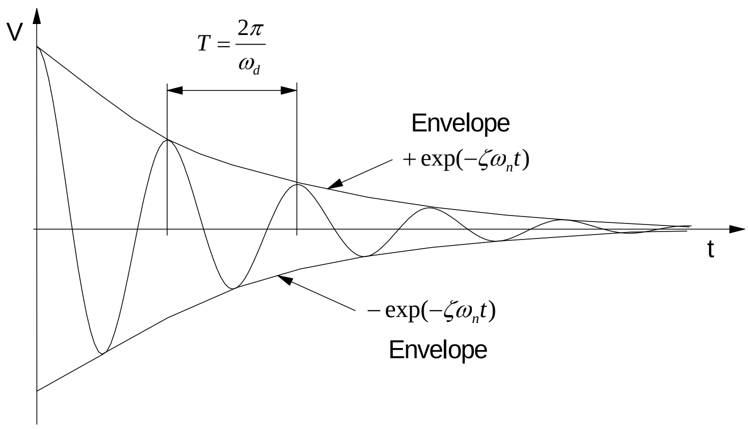


Figure 4.1: Under-damped 2nd order homogenous D.E.

2. **Critically damped Case:** $\zeta = 1$
   When the damping ratio is equal to one, the general solution is of the form

   $$y(t) = C_1 \exp(-\zeta\omega_n t) + C_2 t \exp(-\zeta\omega_n t) \tag{4.17}$$

   where, C_1 and C_2 are unknown coefficients derived from the initial conditions. Thus for a critically damped system, the response is not oscillatory. It approaches equilibrium as quickly as possible.

3. **Over-damped Case:** $\zeta > 1$
   When the damping ratio is greater than one, the general solution to the homogenous equation is

$$y(t) = C_1 \exp\left((-\zeta + \sqrt{\zeta^2 - 1})\omega_n t\right) + C_2 \exp\left((-\zeta - \sqrt{\zeta^2 - 1})\omega_n t\right) \qquad (4.18)$$

   where, C_1 and C_2 are unknown coefficients derived from the initial conditions. This indicated that the response is the sum of two decaying exponentials.

The total solution of a second-order system for the previously discussed cases can be summarized as shown in Fig. 4.2 for $\zeta = 0.1, 0.3, 1, 2$ and 3.

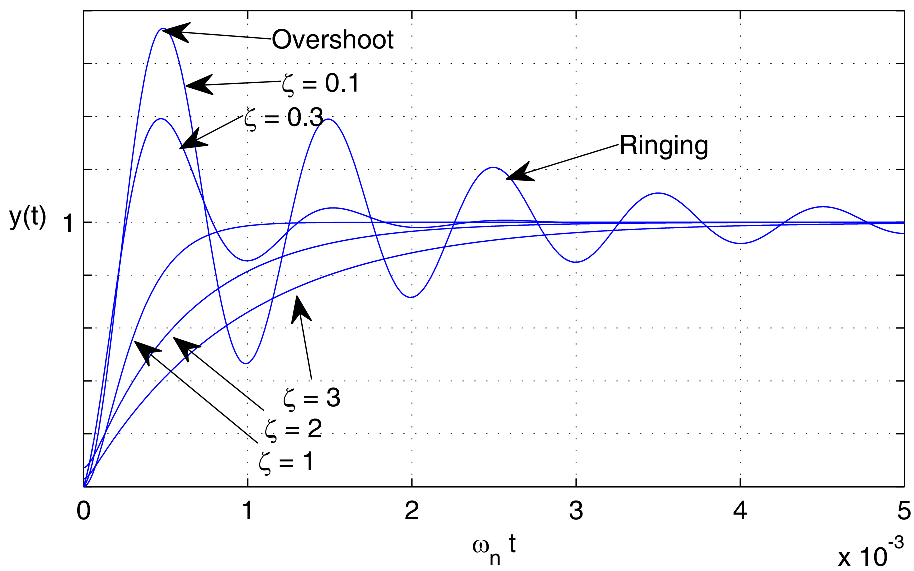


Figure 4.2: Solution of a second-order homogenous system

Figure 4.2 shows the normalized step response, where A is the amplitude of the constant input signal. Generally, the aim of normalizing (scaling) is to compare the output (the response) to a reference value. In our case, the reference value is the step function and A represents the input voltage (DC source).
Note: When $\zeta = 0$, the response becomes undamped and oscillations continue indefinitely at frequency ω_n.

b. The forced solution

The response to a forcing function will be of the same form as the forcing function. The forced solution of the non-homogeneous second-order differential equation is usually given by a weighted sum of the input signal $x(t)$ and its first and second derivatives. If the input $x(t)$ is constant, then the forced response y_f is constant as well. If $x(t)$ is sinusoidal, then y_f is sinusoidal.
However, we will not discuss the forced solutions of second-order differential equations as part of this lab. Here we will deal only with constant input signals (DC sources). The necessary steps for determining the steady-state response of RLC circuits with DC sources will be described later in the handout.

4.1.5 Solving a second-order differential system

The behavior of a series RLC circuit shown in Fig. 4.3 can be determined from a simple circuit analysis.

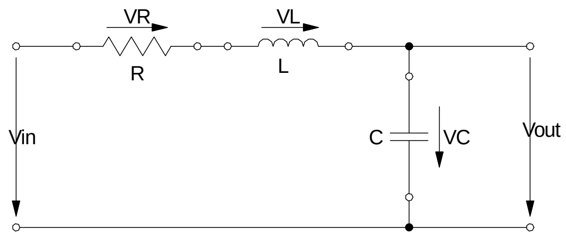


Figure 4.3: Second order system based on a serial RLC circuit

The input voltage is the sum of the output voltage and voltage drops across the inductor and resistor

$$V_{in} = V_R + V_L + V_{out} \tag{4.19}$$

The current i is related to the current flowing through the capacitor

$$i = i_C = C\frac{dV_{out}}{dt} \tag{4.20}$$

The voltage drop across the resistor is given by

$$V_R = iR = RC\frac{dV_{out}}{dt} \tag{4.21}$$

the voltage drop across the inductor is

$$V_L = L\frac{di}{dt} = L\frac{d}{dt}(C\frac{dV_{out}}{dt}) = LC\frac{d^2V_{out}}{dt^2} \tag{4.22}$$

Substituting Eq. (4.21) and (4.22) into (4.19) yields

$$LC\frac{d^2V_{out}}{dt^2} + RC\frac{dV_{out}}{dt} + V_{out} = V_{in} \tag{4.23}$$

Thus, from Eq. (4.3), (4.4) and (4.5), the undamped natural frequency (radians/sec), the damping ratio and the gain of the circuit are:

$$\omega_n = \frac{1}{\sqrt{LC}} \tag{4.24}$$

$$\zeta = \frac{R}{2}\sqrt{\frac{C}{L}} \tag{4.25}$$

$$K = 1 \tag{4.26}$$

Example:
Let's assume that the components in Fig. 4.3 have the values $R = 50\Omega$, $C = 1\mu F$ and $L = 50mH$. By substituting in Eq. (4.24) and (4.25)

$$\omega_n = \frac{1}{\sqrt{LC}} = 4472rad/sec$$

$$\zeta = \frac{R}{2}\sqrt{\frac{C}{L}} = 0.1 \quad \text{(under-damped case)}$$

4.1.6 Definitions and Practical Hints

Step response

The step response is the response of a system upon applying an input signal in the form of a step function.

The steady-state value

The steady-state value is the magnitude of the voltage, or current, after the system has reached stability.

Ringing

Ringing is the oscillation phenomenon that occurs if the system is under-damped.

Overshoot

An overshoot is observed if the transient signal exceeds the final steady state value. The overshoot is often represented by a percentage of the final value of the step response. The percentage overshoot is

$$\text{Percentage overshoot} = \frac{V_{max} - V_{SteadyState}}{V_{SteadyState}} * 100\%$$

The undamped natural frequency, ω_n

It is the frequency of oscillation of the system without damping.

The Step Response

For the step response of an under-damped system shown in Fig. 4.4, the transient response specifications are:

- **Peak time,** T_p
  It is the time required for the response to reach the peak of the overshoot.
- **Rise time,** T_r
  It is the time required for the step response to rise from 10% to 90% of its final value for critical and over-damped cases, and from 0% to 100% for under-damped cases.
- **Settling time,** T_s
  It is the time required for the step response to settle within a certain percentage of its final value. The percentage can be chosen to be 2% or 5%.

Note: Not all these specifications apply to all cases of system response. For example, for an over-damped system, the terms ringing, peak time and maximum overshoot do not apply.

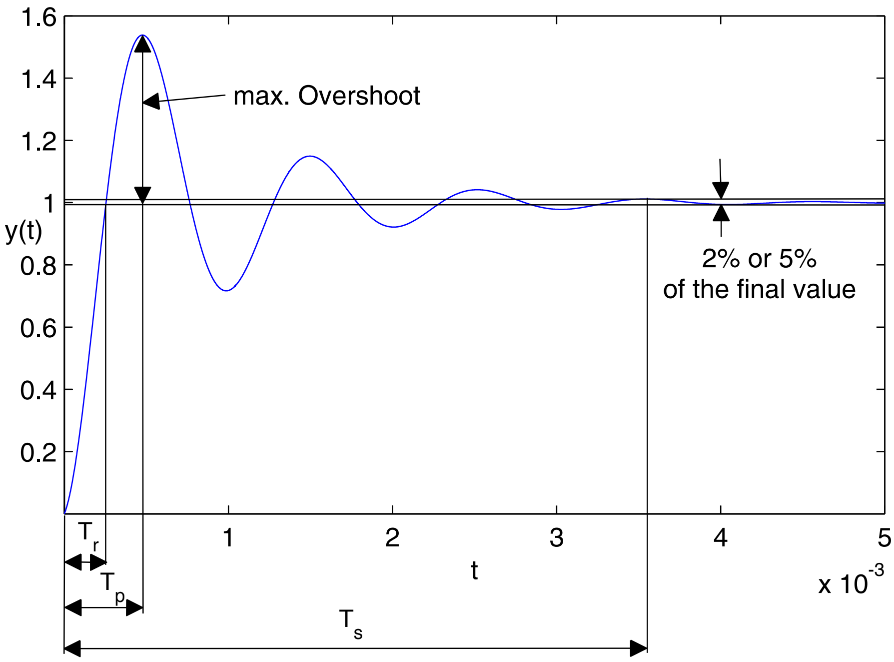


Figure 4.4: Under-damped second-order system

4.1.7 Initial conditions of switched circuits

The switched circuit is a circuit with one or more switches that open or close at a certain point in time. We are interested in the change of the current and voltage of energy storage elements (L and C) after the switch changed from open to close or vice versa. So if the time of the switch is $t = 0$, we want to determine the current through the inductor and voltage across the capacitor at $t = 0^-$ and $t = 0^+$ immediately before and after the switching. These values together with the sources determine the behavior of the circuit for $t > 0$. Before continuing with a solved example two important notes should be mentioned which are:

a. The current through an inductor cannot change instantaneously, whereas the voltage drop across an inductor can change instantaneously.

b. The voltage drop across a capacitor cannot change instantaneously, whereas the current flow through a capacitor can change instantaneously.

Example:

Consider the circuit shown in Fig. 4.5. The switch has been closed and steady state conditions were reached. In order to find $v_C(0^-)$ and $i_L(0^-)$ the capacitor is replaced by an open circuit and the inductor by a short circuit as shown in Fig. 4.6. It can be easily calculated that

$$i_L(0^-) = \frac{V_{in}}{R_1 + R_2} \tag{4.27}$$

$$v_C(0^-) = V_{in}\frac{R_2}{R_1 + R_2} \tag{4.28}$$

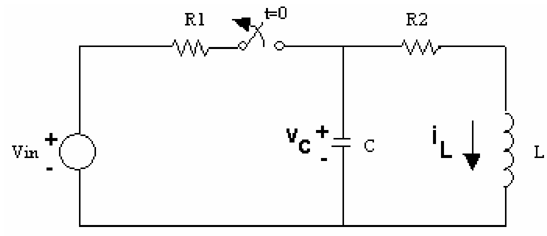


Figure 4.5: Switched circuit

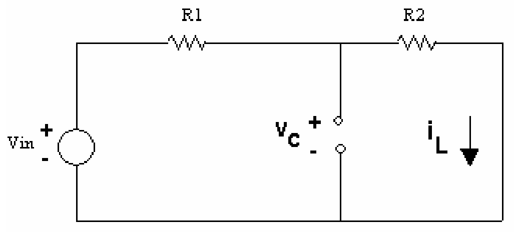


Figure 4.6: Steady-state conditions

Since the current through an inductor cannot change instantaneously and the voltage across a capacitor cannot change instantaneously, therefore,

$$i_L(0^-) = i_L(0^+) = \frac{V_{in}}{R_1 + R_2} \tag{4.29}$$

$$v_C(0^-) = v_C(0^+) = V_{in}\frac{R_2}{R_1 + R_2} \tag{4.30}$$

4.1.8 The Complete Response

The necessary steps to determine the complete response of a second-order system based on RLC network with DC sources are:

- For transient response
  1. Using Ohm's law, KVL, and/or KCL, obtain a second order differential nonhomogeneous equation. Another way is to obtain two first-order differential equations, and then combine them to a second order differential non-homogeneous equation.
  2. Solve the homogeneous equation corresponding to the obtained second order differential non-homogeneous equation.
  3. Obtain the complete solution by adding the forced solution to the homogeneous solution. The complete solution still contains unknown coefficients C_1 and C_2.
  4. Use the initial conditions to determine the value of C_1 and C_2.
- For DC steady state response
  1. Replace all capacitances with open circuits.
  2. Replace all inductances with short circuits.
- Solve the remaining circuit.

4.1.9 References

1. A. V. Oppenheim, A. S. Willsky, S. H. Nawab, "Signals and Systems", Prentice Hall, Second Edition (1997)

2. Sarma, M.S., "Introduction to Electrical Engineering", Oxford University Press, 2001.

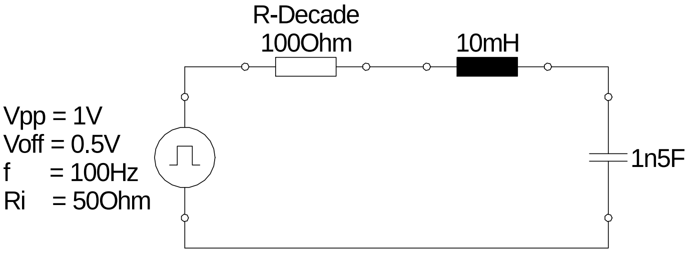
3. R.A. DeCarlo, P-M. Lin, Linear Circuit Analysis, Oxford press, 2nd edition.

4. Allan R. Hambley ,"Electrical Engineering: Principles and Applications", Prentice Hall,Second Edition.

4.2 Execution Transient response of RLC-Circuits

4.2.1 Problem : Design of an RLC circuit

Implement the RLC circuit shown below on the breadboard.



1. Set the function generator to produce a 100 Hz square wave with an amplitude of 0.5 V and an offset of 0.5 V. Check with the oscilloscope if the signal modulates between 0 V and 1 V. Set the R-decade to 100 Ω. Connect the oscilloscope in parallel to the capacitor.

2. Measure the damped frequency f_d. The frequency f_d can be determined by measuring the time or frequency of the exponentially damped sinusoid. Take a hardcopy of one signal period and one focusing on the ringing phenomenon.

3. Calculate the damped radian frequency ω_d. In your calculation, consider the internal resistance of the function generator to be 50 Ω. Compare the calculated value with the measured value in step (2). If they are consistent, proceed with the next steps.

4. Calculate the resistance so that the circuit is critically damped. Display the signal and take a hardcopy.

5. Check if the practical signal is critically damped. Vary the the R-decade value and take a hardcopy of the final result.

6. Set the R-decade to $30k\,\Omega$, so that the circuit is over-damped. Display the transient voltage across the capacitor and take a hardcopy.

4.3 Evaluation

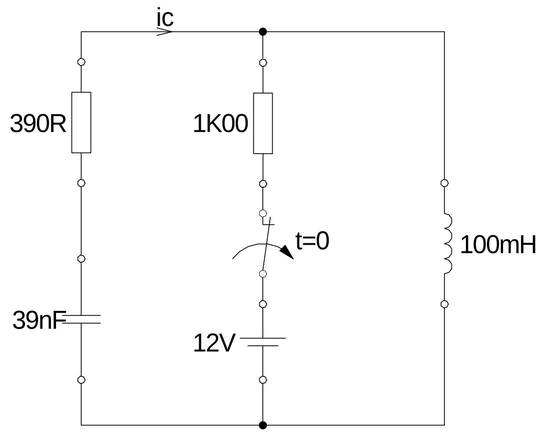
1. Use the circuit from the experiment and obtain the differential equation for the voltage $v_c(t)$ across the capacitor when $R = 100\,\Omega$, identify the damping nature of the circuit and determine the values for the coefficients C_1 and C_2.

2. Plot the voltage $v_c(t)$ using Matlab.

3. Calculate the resistor value to obtain a critically damped case and obtain the corresponding equation describing the voltage $v_c(t)$ including the values for C_1 and C_2. Plot the voltage $v_c(t)$ using Matlab.

4. Compare the experimental results obtained in the lab with the calculations. Provide a detailed explanation if the experimental results deviate. Discuss the origin of the deviation.

5. Solve the following problem:
   The switch in the circuit below is opened at $t = 0$.



   (a) Obtain the differential equation for the current $i_c(t)$ through the capacitor, identify the damping nature of the circuit and determine the values for the coefficients C_1 and C_2.

   (b) Plot the current $i_c(t)$ using Matlab.

5. Theory 2 : RLC-Circuits - Frequency Response

5.1 Introduction to the experiment

5.1.1 Objective of the experiment

The goal of the experiment is to study the frequency response of RLC circuits and their application as analog filters and resonators. As part of the prelab, the frequency response of different RLC circuit configurations will be studied using Matlab. As part of the experimental procedure, different RLC circuit configurations will be implement and tested. The experimental and the simulation results will be compared and the differences will be discussed.

Introduction

In the experiment "RLC Transient Response", we studied the time dependent response of RLC circuits including the transient response and the steady-state response for a constant DC input signal. In the second experiment, we will only explore the steady-state response of an RLC circuit for a periodic sinusoidal input signal. As the frequency of the periodic sinusoidal input signal changes the circuit response changes, that is why the second experiment is called "RLC frequency response".
In electronics, resonating circuits are often used to select or attenuate particular frequency ranges, as in tuning a radio. In its easiest form, a resonator can be realized using a resistor, inductor, and a capacitor. Therefore, the circuit consists of at least two different energy storage devices.

Series and parallel RLC configurations

In this experiment, we will in particular study the series and the parallel configuration of the RLC resonating circuits shown in Fig. 5.1 and (5.2).
First, we will write down the impedance of the series RLC circuit in Fig. 5.1.

$$\underline{Z_S} = R_S + j\omega L + \frac{1}{j\omega C} \tag{5.1}$$

The admittance of the parallel RLC circuit in Fig. 5.2 can be expressed in an analog way

$$\underline{Y_P} = G_P + j\omega C + \frac{1}{j\omega L} \tag{5.2}$$

The absolute value of the impedance and the admittance is given by

$$|\underline{Z_S}| = \sqrt{R_S^2 + \left(\omega L - \frac{1}{\omega C}\right)^2} \tag{5.3a}$$

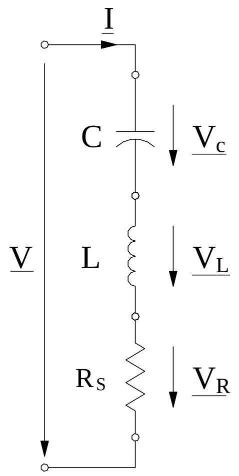


Figure 5.1: Series resonator based on a serial RLC circuit.

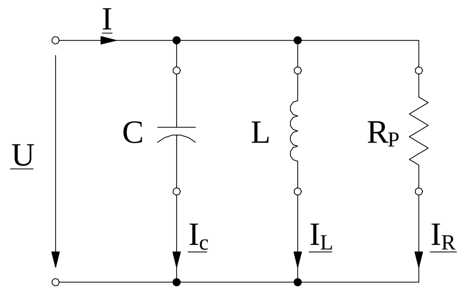


Figure 5.2: Parallel resonator based on a parallel RLC circuit.

$$|\underline{Y_P}| = \sqrt{G_P^2 + \left(\omega C - \frac{1}{\omega L}\right)^2} \tag{5.3b}$$

The phase of the complex impedance and admittance can be expressed by

$$\varphi = \arctan\left(\frac{\omega L - 1/\omega C}{R_S}\right) \tag{5.4a}$$

$$\varphi = \arctan\left(\frac{\omega C - 1/\omega L}{G_P}\right) \tag{5.4b}$$

Based on the phasor plot of the series and parallel resonators in Fig. 5.3 and Fig. 5.4, the amplitude and the phase of the complex impedance and admittance can be determined for a given frequency.
In general, the impedance and the admittance can be written as

$$\underline{Z}(\omega) = R + jX(\omega) \tag{5.5a}$$

$$\underline{Y}(\omega) = G + jB(\omega) \tag{5.5b}$$

In the phasor plot, the tip of the vector corresponds to the impedance $\underline{Z}(\omega)$ or the admittance $\underline{Y}(\omega)$ of the circuit. The following information can be extracted from the phasor plot.

- The circuit is in resonance if the oscillation parameter is maximized. In the case of a series resonator the oscillation parameter is the current, whereas for a parallel resonator the oscillation parameter is the voltage.

- The oscillation parameter is maximized and the resonance point is reached, when the phasor intersects with the real axis of the graph. In this case the frequency ω is getting equal to the resonance frequency ω_0.

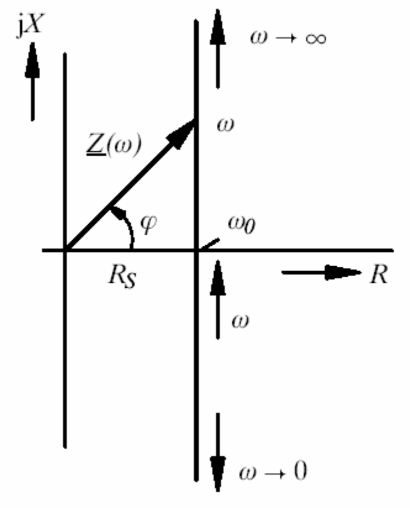


Figure 5.3: Phasor diagram of a series resonator based on a RLC circuit. In this case, the impedance of a series resonator is plotted [3].

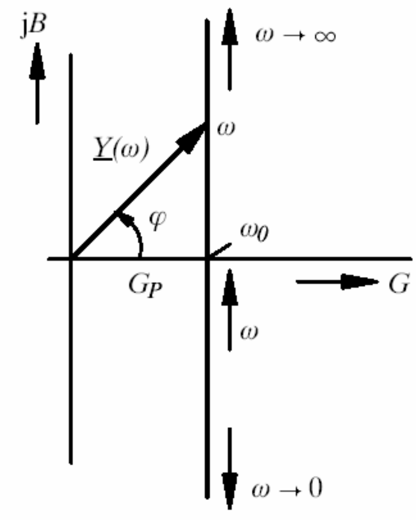


Figure 5.4: Phasor diagram of a parallel resonator based on a RLC circuit. In this case, the admittance of a parallel resonator is plotted [3].

The frequency ω_0 is the frequency associated with the resonance point of the circuit. Consequently, the impedance results in

$$Im(\underline{Z}(\omega = \omega_0)) = 0 \tag{5.6}$$

and the admittance results in

$$Im(\underline{Y}(\omega = \omega_0)) = 0 \tag{5.7}$$

In both cases, the phase gets zero

$$\varphi = 0 \tag{5.8}$$

Furthermore, the following circumstances apply for the resonance points of a series resonator:

$$Z_S(\omega = \omega_0)) = Re(\underline{Z_S}(\omega)) = R_S \tag{5.9}$$

If $\omega = \omega_0$, the impedance is minimized and $|\underline{Z_S}(\omega_0)| = min$.
The parallel resonator exhibits an analog behavior.

$$Y_P(\omega = \omega_0)) = Re(\underline{Y_P}(\omega)) = G_P = 1/R_P \tag{5.10}$$

Due to the opposite characteristics of an inductive reactance (the inductive reactance increases as the frequency is increased) and a capacitive reactance (capacitive reactance decreases with higher frequencies) the reactance X_L equals X_C for the resonance frequency. Consequently, the inductive and the capacitive impedances compensate each other, so that the reactive power becomes zero, which means that only effective (real) power is consumed by the circuit.

5.1.2 Application of series and parallel RLC circuits

Reactive power compensation

The reactive power is minimized under the following conditions:

- The reactive power gets minimized for a **series resonator** if the circuit is driven by a current source. In such a case the input current is constant ($|\underline{I}| = const.$) and the impedance determines the voltage drop across the circuit. As the impedance is minimized $\underline{Z}(\omega = \omega_0) \rightarrow min.$ it follows that the voltage drop is minimized as well $|\underline{V}(\omega_0)| = |\underline{I}| * |\underline{Z}(\omega_o)| \rightarrow min.$!

- The reactive power gets minimized for a **parallel resonator** if the circuit is driven by a voltage source. In such a case the input voltage is constant ($|\underline{V}| = const.$). As the admittance is minimized $\underline{Y}(\omega = \omega_0) \rightarrow min.$ it follows that the current is minimized as well $|\underline{I}(\omega_0)| = |\underline{V}| * |\underline{Y}(\omega_o)| \rightarrow min.$!

The reactive power is maximized under the following conditions:

- The reactive power gets maximized for a **series resonator** if the circuit is driven by a voltage source. In such a case the input voltage is constant ($|\underline{V}| = const.$) and the impedance determines the current flow. As the impedance is minimized $\underline{Z}(\omega = \omega_0) \rightarrow min.$, it follows that the current flow through the circuit is maximized $|\underline{I}(\omega_0)| = |\underline{V}|/|\underline{Z}(\omega_o)| \rightarrow max.$!

- The reactive power gets maximized for a **parallel resonator** if the circuit is driven by a current source. In such a case the input current is constant ($|\underline{I}| = const.$). As the admittance is minimized $\underline{Y}(\omega = \omega_0) \rightarrow min.$ it follows that the voltage drop across the circuit is maximized as well $|\underline{V}(\omega_0)| = |\underline{I}|/|\underline{Y}(\omega_o)| \rightarrow max.$!

Therefore, resonators can be applied for reactive power compensation. Very often electrical consumers have an "inductive character". This is the case if several inductive consumers like motors, pumps or heaters are in operation. The reactive power consumption of a load can be reduced or compensated by using a reactive load, which has the opposite reactive impedance. For example: A capacitive load can be used to compensate an inductive load. As a consequence the impedance of the whole system is getting reduced, which means the impedance is getting nearly real.

Filters

Analog Filters can be constructed based on RLC circuits, which transmit certain frequencies (resonance frequency) in an optimized fashion. Other frequencies (higher and lower frequencies) can be attenuated or blocked. By combining RLC circuits with slightly different resonance frequencies band-pass filters and band-stop filters can be designed.

RLC filter design The frequency response of a RLC circuit can be represented by a magnitude and a phase diagram. Two magnitude and phase plots for a series and a parallel resonator are shown in figures 5.5 and 5.6. The upper graphs show the magnitude diagram and the lower graphs the phase diagram of the series and the parallel resonator.
The magnitude of the signal is normalized to simplify visualization and facilitate a comparison of different magnitude plots. Furthermore, the frequency is normalized in all graphs by the resonance frequency ω_0. We know from our previous discussion that the current is maximized for a series resonator in resonance, whereas for a parallel resonator the voltage is maximized. Consequently, Fig. 5.5 exhibits the normalized current for the series resonator and Fig. 5.6 shows the normalized voltage for a parallel resonator.
The magnitude and the phase for the two circuits in figures 5.5 and 5.6 were calculated for two different resistors. The dashed lines correspond to the resistors R_{s1} and R_{p1}, whereas the solid lines correspond to the resistors R_{s2} and R_{p2}. With increasing resistance of the resistor R_s the width of the magnitude for the series resonator is enhanced. The opposite behavior is observed for a parallel resonator. With increasing resistance of the parallel resistor R_p the width of the magnitude is reduced. It follows that, $R_{s1} < R_{s2}$ and $R_{p1} > R_{p2}$.
The phase diagram exhibits a corresponding behavior. With increasing series resistance the transition region from $-\pi/2$ to $\pi/2$ is widened, whereas for increasing parallel resistance the transition region is getting narrower. Therefore, the series and the parallel resistance have a distinct influence on the bandwidth of the resonators.

Bandwidth and quality factor The bandwidth is a measure of the frequency selectivity of a resonating circuit. The bandwidth B of the resonators can directly be extracted either from the magnitude or the phase plot in Fig. 5.5 and Fig. 5.6. In the magnitude plot, the bandwidth corresponds to the full width of the curve at half maximum, which means that the magnitude is dropped by a factor $\sqrt{2}$. From the phase plot, the bandwidth B can be extracted by taking the difference of the phase between $+45° \equiv +\pi/4$ and $-45° \equiv -\pi/4$). Furthermore, the bandwidth can be determined by mathematical means.

In the following, we will focus on a series resonator In the case of a phase difference of 45° the real part and the imaginary part of the impedance are equal. It follows,

$$Im(\underline{Z}_S(\omega_1)) = Im(\underline{Z}_S(\omega_2)) = R_S \tag{5.11}$$

The equation can be expressed in different terms for negative values of the imaginary part,

$$|Im(\underline{Z}(\omega_1))| = \frac{1}{\omega_1 C} - \omega_1 L = R_S \tag{5.12}$$

So the frequency ω_1 can be determined by:

$$\omega_1 = -\frac{R_S}{2L} + \sqrt{\left(\frac{R_S}{2L}\right)^2 + \frac{1}{LC}} \tag{5.13}$$

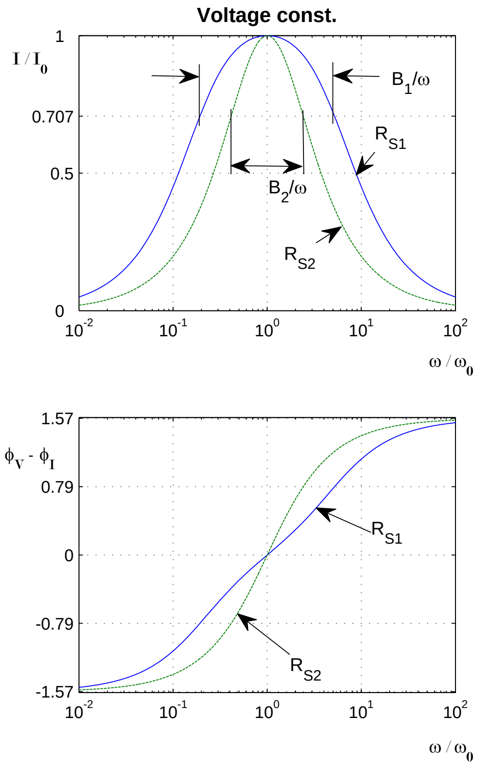


Figure 5.5: Magnitude and phase plot of a series resonator (RLC circuit) driven by a voltage source ($|\underline{V}| = const.$). The normalized magnitude and the phase are shown as a function of the normalized frequency. R_s is the series resistor of the RLC circuit. R_{s1} is the larger and R_{s2} the smaller series resistor: $R_{s1} > R_{s2}$ [3].

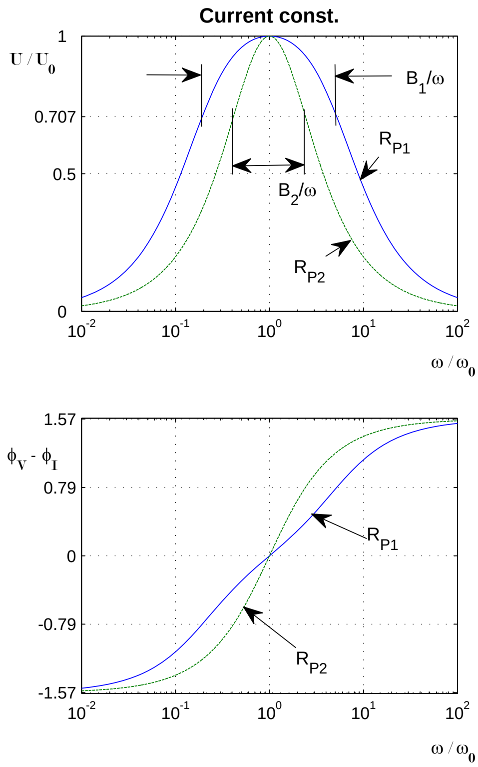


Figure 5.6: Magnitude and phase plot of a parallel resonator (RLC circuit) driven by a current source ($|\underline{I}| = const.$). The normalized magnitude and the phase are shown as a function of the normalized frequency. R_p is the parallel resistor of the RLC circuit. R_{p1} is the larger and R_{p2} the smaller series resistor: $R_{p1} > R_{p2}$ [3].

For the positive imaginary part the following equation applies:

$$|Im(\underline{Z}(\omega_2))| = \omega_2 L - \frac{1}{\omega_2 C} = R_S \tag{5.14}$$

So the frequency ω_2 can be determined by:

$$\omega_2 = \frac{R_S}{2L} + \sqrt{\left(\frac{R_S}{2L}\right)^2 + \frac{1}{LC}} \tag{5.15}$$

Based on the two resonance frequencies the bandwidth B can be determined as:

$$B = \omega_2 - \omega_1 = \frac{R_S}{L} \tag{5.16}$$

Besides the bandwidth, the quality-factor (Q-factor) is also an important measure of the frequency selectivity. For example, filters with high Q-factors are important

and necessary for applications in wireless communications to separate or filter out closely spaced channels/bands. The quality factor is a representation of the width of the resonance peak (the larger the Q value, the narrower the peak). Hence, there is a relation between the Q-factor and the bandwidth, where high-Q circuit has a small bandwidth and low-Q circuit has a large bandwidth.
In terms of energy consumption, the Q-factor is a measure for the ratio between the reactive power (inductive or capacitive) and the real power of a resonator. In other word, it is a measure of the energy-storage in relation to the energy dissipation of the circuit. The Q-factor can be determined by:

$$Q = \frac{Q_C}{P_R} = \frac{Q_L}{P_R} \tag{5.17}$$

where,

Q	is the Q-factor of the resonator.
Q_C and Q_L	are the reactive power of the capacitor or inductor.
P_R	is the real (effective) power.

The quality factor of a series resonator can be expressed by:

$$Q_S = \frac{X_0}{R_S} = \frac{\omega_0}{\omega_2 - \omega_1} \tag{5.18}$$

X_0 is the reactive resistance of series RLC circuit under resonance conditions.

As the circuit is driven in resonance frequency, the following equation applies:

$$X_0 = \omega_0 L = \frac{1}{\omega_0 C} \tag{5.19}$$

or it can be expressed by the following terms,

$$\omega_0 = \frac{1}{\sqrt{LC}} \tag{5.20}$$

and

$$X_0 = \sqrt{\frac{L}{C}} \tag{5.21}$$

It can be seen from Eq. (5.17)-(5.21) that the resonance frequency is defined by the inductance and capacitance of the circuit, whereas the series resistance of the circuit determines the Q-factor.

5.1.3 Practical hints

Phasor diagram

Generally, the phasor diagram represents a complex variable $G(j\omega)$ by a rotating phasor as ω varies from zero to infinity. where,

$$G(j\omega) = Re[G(j\omega)] + Im[G(j\omega)] = R(\omega) + jX(\omega) = |G(j\omega)|\measuredangle G(j\omega)$$

$$|G(j\omega)|^2 = [R(\omega)]^2 + [jX(\omega)]^2$$

$$\measuredangle G(j\omega) = \varphi = \arctan\left(\frac{X(\omega)}{R(\Omega)}\right)$$

Each point on the curve represents the complex value of the variable $G(j\omega)$ for a given frequency. In the phasor diagram provides information about the locus of the complex variable and the circulation of the phasor as the frequency increases. The projections of $G(j\omega)$ on the axis are the real and imaginary components at that frequency.

Magnitude and phase plot of frequency response

Magnitude and phase plots are used to represent the frequency response of any given circuit. Both plots are usually shown together and called Bode diagram.

a. Plot of the magnitude of $|G(j\omega)|$, indicating the amplitude change imposed on a sinusoidal input signal as a function of frequency.

b. Plot of the phase angle $\measuredangle G(j\omega)$, indicating the phase change imposed on a sinusoidal input signal as a function of frequency.

Both graphs are plotted against the frequency ω *rad/sec.*

Lissajou figure

An oscilloscope can be used to determine the phase difference between two periodic signals. However, an oscilloscope is only able to measure a voltage signal. A current can be measured by creating a voltage drop over a resistor. The phase difference can be extracted by applying both of these signals to the input channels of the oscilloscope.
However, for small phase differences it is difficult to extract the phase difference directly from the screen of the oscilloscope. As an alternative, a Lissajou figure can be used to determine the phase difference of two signals. A Lissajou figure is more accurate if it comes to smaller phase differences. The concept of a Lissajou figure goes back to the "old days" when people were using cathode ray tubes (CRT) based oscilloscopes. The position of a spot on the screen of a CRT oscilloscope is controlled by the voltages applied to the x and y deflection capacitors of the cathode ray tube. In normal operation (a voltage is shown as a function of time), a triangular voltage is applied to the x-deflection capacitor so that the spot moves from the left to the right side of the screen. In the case of a Lissajou figure we directly apply the second signal to the x-deflection capacitor.
The x-deflection of the signal can be described by:

$$V_x(t) = V_{pp} * \sin(\omega t) \tag{5.22}$$

whereas the y-deflection can be described by:

$$V_y(t) = c * I_{pp} * \sin(\omega t + \varphi) \tag{5.23}$$

V_{pp} is the peak amplitude of the voltage, c is a constant factor (resistance) and I_{pp} is the peak current.

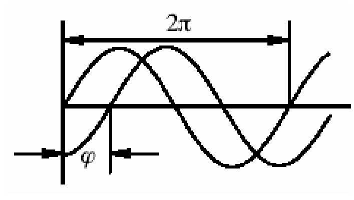


Figure 5.7: Extraction of the phase difference between two sinusoidal signals.

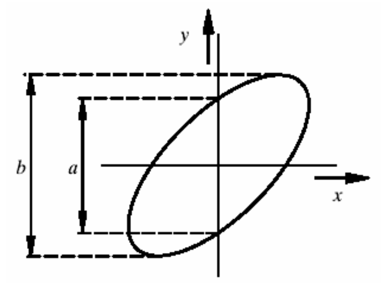


Figure 5.8: Extraction of the phase difference by using a Lissajou figure.

For $\omega t = 0$, we can extract: $2 * c * I_{pp} * \sin\varphi = a$
Furthermore, we can determine $2 * y_{max} = 2 * c * I_{pp} = b$.
As a consequence we can deduce the phase difference from the parameters a and b by the following simple expression

$$\varphi = \arcsin\left(\frac{a}{b}\right) \tag{5.24}$$

The accuracy of this method is distinctly higher than a direct comparison of the signals. This is in particular true for small phase differences.

5.2 Handling of the function generator and the oscilloscope

Throughout the procedure of the lab the frequency response of several circuits has to be taken. In order to measure the frequency response of a circuit the sweep mode of the function generator can be used. In this case the function generator changes its frequency over time.
Using and setting up the sweep mode:

a. **Settings of the Function generator:**
Select the "Sweep Menu" from the function generator and set:

Sweep frequency from $100\,Hz$	(1: START F)
Sweep frequency to $100\,KHz$	(2: STOP F)
Sweep time to $500\,msec$	(3: SWP TIME)
Sweep mode to logarithmic	(4: SWP MODE)

b. **Settings of the Oscilloscope:**
To see the full sweep at the oscilloscope you have to set the time base to $500\,\mathrm{ms}/10\,\mathrm{div} = 50\,\mathrm{ms}/\,\mathrm{div}$.

c. **Synchronization of the function generator and the oscilloscope:**
In order to measure a frequency response with the oscilloscope the function generator and the oscilloscope have to be synchronized. To do so, the synchronization signal of the signal generator has to be connected with the trigger input of the oscilloscope and the oscilloscope has to be set to "external trigger".

d. **Grounding of the function generator and the oscilloscope:**
The grounds of the function generator and the oscilloscope have to be connected together throughout all measurements in order to have the correct output on the oscilloscope screen. You have to construct the circuit in this way that one of the terminals of the component under test is always on ground while measuring the voltage across the component.

5.3 References

1. M.S. Sarma, Introduction to Electrical Engineering, Oxford Series in Electrical and Computer Engineering, 2000.
2. J. Keown, ORCAD PSpice and Circuit Analysis, Prentice Hall Press (2001).
3. F. Hohls, Lab experiments, Resonators, University Hannover, Spring 2003.
4. R.A. DeCarlo, P-M. Lin, Linear Circuit Analysis, Oxford Press, 2nd edition.

5.4 Prelab RLC Circuits - Frequency response

5.4.1 Problem: RLC resonator

Given is a series RLC resonant circuit with $R = 390\Omega$, $C = 270\,\mathrm{nF}$ and $L = 10\,\mathrm{mH}$.

1. Show the Bode magnitude plot across the resistor, the capacitor, the inductor, and across both the capacitor and the inductor. Use a 5 V amplitude and vary the frequency starting at 100 Hz up to 100 KHz.
   Develop a Matlab script to show all four characteristic in one plot. Attach the script to the prelab!

2. Taking the magnitude across the resistance represents a band-pass filter. Extract the bandwidth from the Matlab plot and calculate the Q-factor.

3. Name the remaining filter characteristic measured over the different components, component combinations.

6. Experiment 3 : Fourier Series and Fourier Transform

6.1 Introduction to the experiment

6.1.1 Objectives of the experiment

The goal of the second experiment of the Signals and Systems Lab is to study different signals in terms of their Fourier coefficients and to get a deeper understanding of the Fourier transform. The handout will provide the basic theory and describes the various variables and concepts involved. A more detailed description of the theory can be found in reference [1]. The prelab and the experimental procedure will concentrate on the simulation and implementation of the Fast Fourier Transform (FFT) rather than the detailed mathematical description. The experimental and the simulation results will be compared and differences will be discussed.

6.1.2 Introduction

A signal can be represented in the time domain or in the frequency domain. The frequency domain representation is also called the spectrum of the signal. The Fourier analysis is the technique that is used to decompose the signal into its constituent sinusoidal waves, i.e. any time-varying signal can be constructed by superimposing sinusoidal waves of appropriate frequency, amplitude, and phase. The knowledge of the frequency content of a signal can be very useful. For example, the frequency content of human speech can be filtered, the quality of transmitted signals can be improved and noise can be removed. The Fourier transform is used to transform a signal from the time domain to the frequency domain. For certain signals, Fourier transform can be performed analytically with calculus. For arbitrary signals, the signal must first be digitized, and a Discrete Fourier Transform (DFT) is performed. On the other hand, the inverse Fourier transform is used to transform a signal from the frequency domain to the time domain. The handout is divided into two parts. The first part introduces Fourier series representation and Fourier transform for periodic continuous-time signals. The second part describes Fourier series representation and Fourier transform for periodic discrete-time signals.

6.1.3 Part I: Continuous time signals

A. Fourier series representation

A continuous-time periodic signal can be described by the sum of basic signals, i.e. the sum of sine or cosine waves.

Periodic signals

A signal is defined as periodic, if for some positive value of T, the signal can be described by Eq. (6.1),

$$x(t) = x(t + nT) \tag{6.1}$$

This must hold for all t. The fundamental period is the minimum positive, nonzero value of T for which the above equation is satisfied. The value $\omega_0 = 2\pi/T$ is referred to as the fundamental frequency.

Determination of Fourier series coefficients

Given a function $x(t)$, its Fourier series coefficients c_k can be obtained by using the following equation.

$$c_k = \frac{1}{T}\int_T x(t)e^{-jk\omega_0 t}dt \tag{6.2}$$

Thus, $x(t)$ can be expressed in terms of c_k as follows,

$$x(t) = \sum_{k=-\infty}^{+\infty} c_k e^{jk\omega_0 t} \tag{6.3}$$

On substituting $k = 0$ into Eq. (6.2), the DC or the constant component of the signal $x(t)$ is obtained,

$$c_0 = \frac{1}{T}\int_T x(t)dt \tag{6.4}$$

So far we expressed the signal $x(t)$ as a sum of superimposed complex exponential functions. However, a number of other ways can be used to represent $x(t)$. As example, Eq. (6.3) can be rewritten as,

$$x(t) = \frac{a_0}{2} + \sum_{k=1}^{\infty}\Big[A_k \cos(k\omega_0 t) + B_k \sin(k\omega_0 t)\Big] \tag{6.5}$$

In such a case, each continuous time periodic function can be described by the sum of superimposing sine and cosine functions. The required Fourier coefficients are a_0 (DC component), A_k and B_k.
In the following section, the Fourier series coefficients and the Fourier series for a continuous time periodic square wave will be obtained analytically.

Continuous time periodic square wave

A continuous time periodic square wave is given in Fig. 6.1 with a period T and a pulse width of $2T_1$.

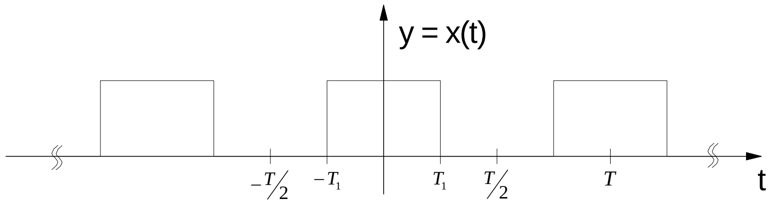


Figure 6.1: Continuous periodic square wave

The signal is described as follows:

$$x(t) = \begin{cases} 1, & |t| \leqslant T_1 \\ 0, & T_1 < |t| < T/2 \end{cases} \tag{6.6}$$

The Fourier series coefficients are obtained using,

$$c_k = \frac{1}{T}\int_T x(t)e^{-k\omega_0 t}dt \tag{6.7}$$

To calculate the constant c_0, use $k = 0$ and perform the integration over any interval of length T,

$$c_0 = \frac{1}{T}\int_{-T/2}^{T/2} x(t)\,dt = \frac{1}{T}\int_{-T/4}^{T/4} 1\,dt = \frac{1}{T}\left(\frac{T}{2}\right) = \frac{1}{2} \tag{6.8}$$

Due of the symmetry of $x(t)$ about $t = 0$, it is suitable to integrate from $-T/2 \leqslant t < T/2$. The signal is only equal to 1 in the range of $-T/4 \leqslant t < T/4$, so that Eq. (6.7) results to,

$$\begin{aligned} c_k &= \frac{1}{T}\int_T x(t)e^{-jk\omega_0 t}dt = \frac{1}{T}\int_{-T_1}^{T_1} x(t)e^{-jk\omega_0 t}dt \\ &= -\frac{1}{jk\omega_0 T}e^{-jk\omega_0 t}\Bigg|_{-T_1}^{T_1} = \frac{2}{k\omega_0 T}\left(\frac{e^{jk\omega_0 T_1} - e^{-jk\omega_0 T_1}}{2j}\right) \end{aligned} \tag{6.9}$$

Making use of the trigonometric identity,

$$\sin(x) = \frac{e^{jx} - e^{-jx}}{2j} \tag{6.10}$$

Eq. (6.9) becomes

$$c_k = \frac{2}{k\omega_0 T}\sin(k\omega_0 T_1) = \frac{1}{k\pi}\sin(k\omega_0 T_1) \tag{6.11}$$

where, T is the period of the signal and T_1 is the width of the periodic pulses.
For more demonstration, Matlab was used to plot the scaled Fourier coefficient Ta_k given by Eq. (6.11) for $k = -50$ to $k = 50$. The width of the pulses were kept constant at $T1$, and the period of the signal was varied from $T = 4T_1$, $T = 8T_1$, and $T = 20T_1$ as shown in Fig. 6.2, Fig. 6.3 and Fig. 6.4, respectively.

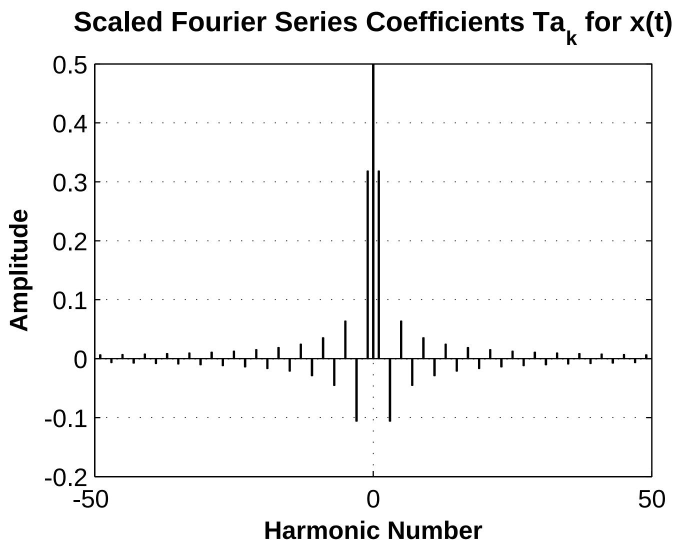


Figure 6.2: Scaled Fourier coefficients Ta_k for T_1 fixed and $T = 4T_1$

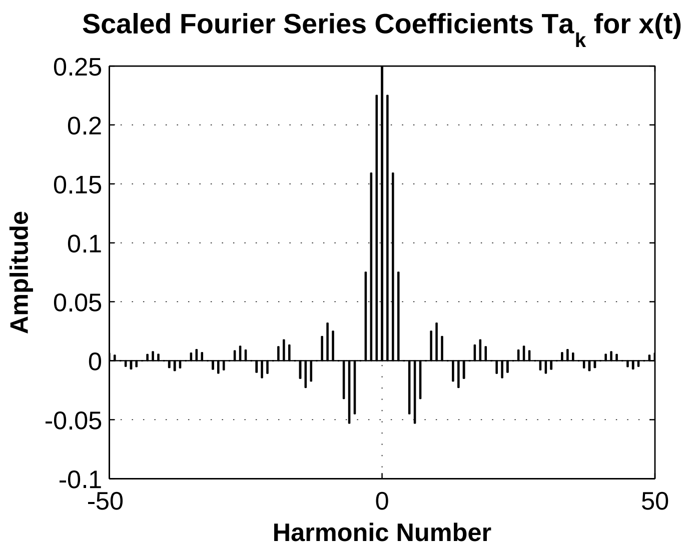


Figure 6.3: Scaled Fourier coefficients Ta_k for T_1 fixed and $T = 8T_1$

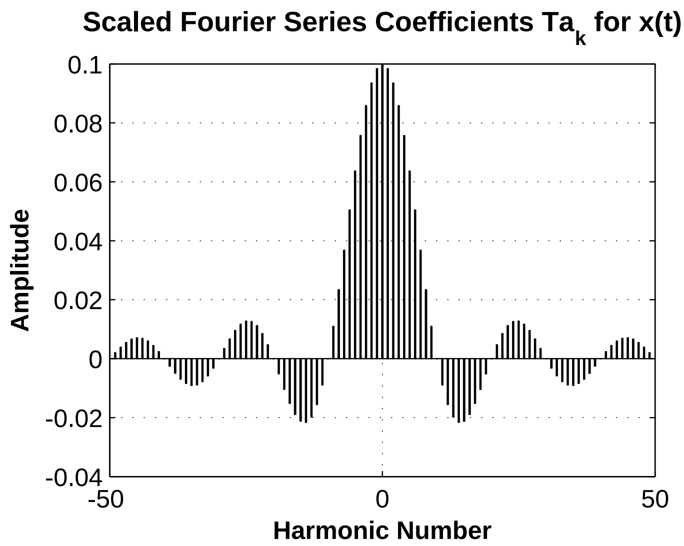


Figure 6.4: Scaled Fourier coefficients Ta_k for T_1 fixed and $T = 20T_1$

Now, we will continue to describe $x(t)$ in terms of sine and cosine functions as described by Eq. (6.5). The coefficient a_0 becomes $a_0 = 2c_0$. The coefficients A_k and B_k can be determined by

$$A_k = 2\,\Re e\,\{c_k\} \tag{6.12}$$
$$B_k = -2\,\Im m\,\{c_k\} \tag{6.13}$$

Using Eq. (6.11), for $T = 4T_1$.

$$A_k = \frac{2}{k\pi}\sin{(k\frac{\pi}{2})} = \begin{cases} \frac{2}{k\pi} & \text{k} = 1,\ 3,\ 5,\ ...\text{(odd)} \\ 0 & \text{k} = 2,\ 4,\ 6,\ ...\ \text{(even)} \end{cases} \tag{6.14}$$

$$B_k = 0 \tag{6.15}$$

Substituting into Eq. (6.5),

$$x(t) = \frac{1}{2} + \frac{2}{\pi}\sum_{k=1,3,5,...}^{\infty}\sin{(k\frac{\pi}{2})} * \frac{\cos{(k\omega_0 t)}}{k} \tag{6.16}$$

For more demonstration, Matlab was used to plot the first three harmonics as shown in Fig. 6.5.

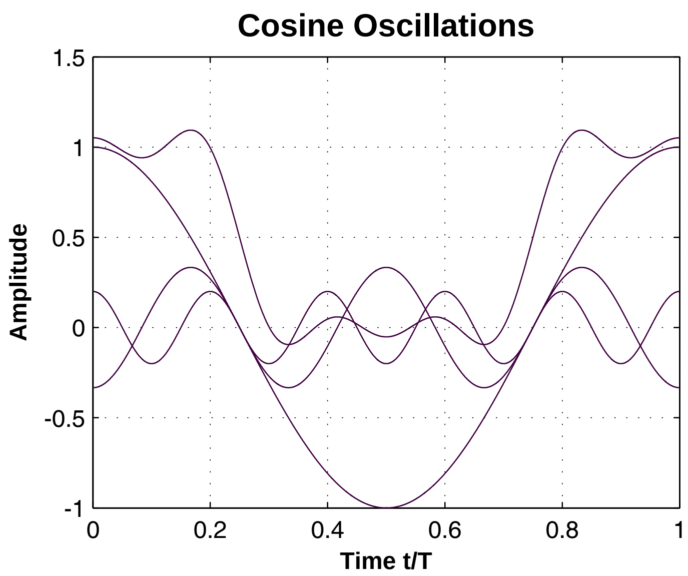


Figure 6.5: The first three harmonics of a square wave. The time axis is normalized. The time $t = 1$ corresponds to $t = T$

Figure 6.6 shows the first 50 harmonics (terms in Eq. (6.16)) and the corresponding square wave after the summation.

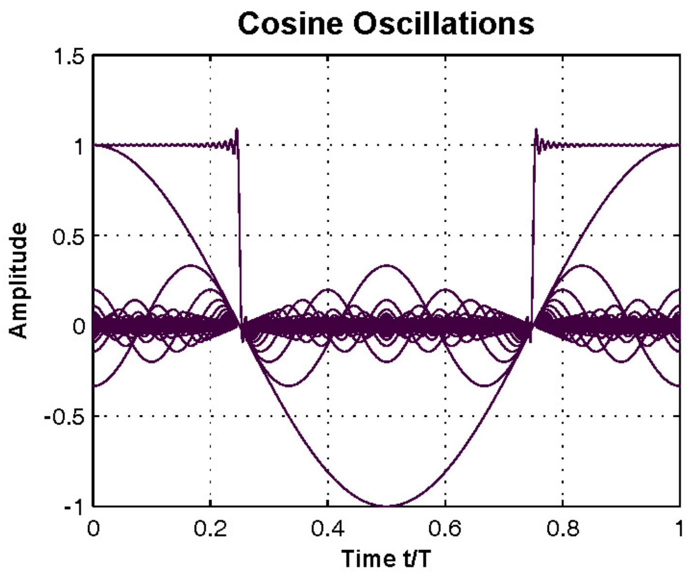


Figure 6.6: The first 50 harmonics of a square wave given in Eq. (6.16) and the sum of the harmonics. The time axis is normalized. Time $t = 1$ corresponds to $t = T$.

Figure 6.7 shows the first 500 harmonics (terms in Eq. (6.16)) and the corresponding square wave after the summation.
A comparison of Fig. 6.6 and Fig. 6.7 indicates that the shape of the reconstructed square wave can already be recognized after the summation of the first 50 harmonics. However, the reconstructed signal exhibits a lot of 'ringing' at each step change

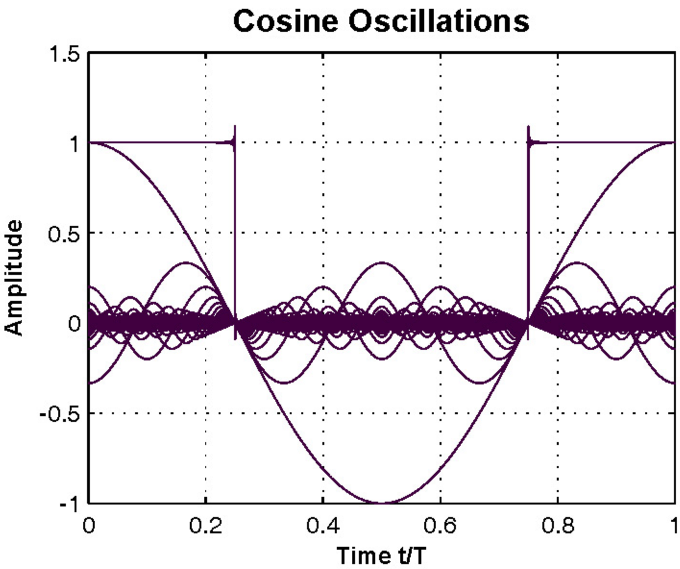


Figure 6.7: The first 500 harmonics of a square wave given in Eq. (6.16) and the sum of the harmonics. The time axis is normalized. Time $t = 1$ corresponds to $t = T$.

in the square wave, i.e. the Fourier series exhibits a peak followed by rapid oscillations. The phenomenon is called Gibbs effect. With increased number of terms, e.g.1000 harmonics, reconstructed signal is getting closer to the original signal. Generally, this phenomenon is due to the discontinuities in the square wave and many high frequency components are required to construct the signal accurately. More information about Gibbs effect can be found in reference [1].

B. Continuous time Fourier transform

The continuous time Fourier transform is a generalization of the Fourier series. It can be also called Fourier series representation for continuous time aperiodic signals. It only applies to continuous time aperiodic signals. The Fourier transform of a given signal $x(t)$ is defined as

$$X(j\omega) = \int_{\infty}^{\infty} x(t)e^{-j\omega t}dt \tag{6.17}$$

Using the inverse Fourier Transform the original signal can be obtained using the following equation,

$$x(t) = \frac{1}{2\pi}\int_{\infty}^{\infty} X(j\omega)e^{j\omega t}d\omega \tag{6.18}$$

As an example, we will calculate the Fourier transform of the square pulse shown in Fig. 6.8.

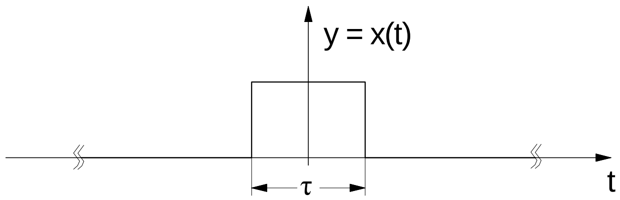


Figure 6.8: Square pulse with a pulse width of τ

The signal in Fig. 6.8 is 0 everywhere except in the range $-\tau/2 \leqslant t < \tau/2$, where the signal is equal to 1. After calculating the Fourier transform we get:

$$X(j\omega) = \int_{-\tau/2}^{\tau/2} e^{-j\omega t} dt = \left. \frac{e^{-j\omega t}}{j\omega} \right|_{-\tau/2}^{\tau/2} = \left(\frac{e^{j\omega\tau/2} - e^{-j\omega\tau/2}}{j\omega} \right) \tag{6.19}$$

Again making use of the trigonometric identity (Eq. (6.10)),

$$X(j\omega) = \tau * \left[\frac{\sin\left(\frac{\omega\tau}{2}\right)}{\frac{\omega\tau}{2}} \right] = \tau \, sinc\left(\frac{\omega\tau}{2}\right) \tag{6.20}$$

NOTE

Fourier declared that an aperiodic signal could be viewed as a periodic signal with an infinite period.
As an example, we studied the Fourier series of a square wave and the Fourier transform of a rectangular pulse. As the period becomes infinite, the periodic square wave approaches the Fourier transform of the rectangular pulse.

6.1.4 Part II: Discrete time signals

A. Fourier series representation

In this part, we will discuss the Fourier series representation for discrete time signals. The discussion will closely follow the discussion in the first part.

Periodic signals

A signal is defined as periodic, with period N if,

$$x[n] = x[n+N] \tag{6.21}$$

This must hold for all n. The fundamental period is the smallest positive integer N for which the above equation holds. The parameter $\omega_0 = \frac{2\pi}{N}$ is referred to as the fundamental frequency.

Determination of Fourier series coefficients

Given a function $x[n]$, its Fourier series coefficients a_k can be obtained using the following equation.

$$a_k = \frac{1}{N} \sum_{n=(N)} a[n] e^{-jk\omega_0 n} \tag{6.22}$$

Thus x[n] can be expressed in terms of a_k as follows,

$$x[n] = \frac{1}{N} \sum_{n=(N)} a_k e^{jk\omega_0 n} \tag{6.23}$$

In the following section, the Fourier series coefficients for a discrete time periodic square wave will be obtained analytically.

Discrete time periodic square wave

Given a discrete time periodic square wave as shown in Fig. 6.9 with a period N and a pulse width of $2N_1$. The signal can be described as follows:

$$x[n] = 1 \quad \text{for} \quad -N_1 \leqslant n \leqslant N_1 \tag{6.24}$$

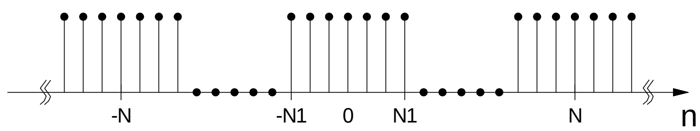


Figure 6.9: Discrete periodic square wave with a period of N and a pulse width of $2N_1$

The Fourier series coefficients can be obtained using

$$a_k = \frac{1}{N} \sum_{n=-N_1}^{N_1} e^{-jk(2\pi/N)n} \tag{6.25}$$

where, $m = n + N_1$, so that the Fourier coefficients can be described as

$$a_k = \frac{1}{N} \sum_{m=0}^{2N_1} e^{-jk(2\pi/N)(m-N_1)} = \frac{1}{N} e^{jk(2\pi/N)N_1} \sum_{m=0}^{2N_1} e^{-jk(2\pi/N)m} \tag{6.26}$$

The summation term is a geometric series. On using the equation for the sum of a geometric series

$$a_k = \sum_{n=0}^{N-1} \alpha^n = \begin{cases} N & \alpha = 1 \\ \frac{1-\alpha^n}{1-\alpha} & \alpha \neq 1 \end{cases} \tag{6.27}$$

The following equation is obtained

$$
\begin{aligned}
a_k &= \frac{1}{N} e^{-jk(2\pi/N)N_1} \left(\frac{1 - e^{-jk2\pi(2N_1+1)/N}}{1 - e^{-jk(2\pi/N)}} \right) \\
&= \frac{1}{N} \left(\frac{e^{-jk(2\pi/2N)} \left(e^{jk2\pi(N_1+1/2)/N} - e^{-jk2\pi(N_1+1/2)/N} \right)}{e^{-jk(2\pi/2N)} \left(e^{jk(2\pi/2N)} - e^{-jk(2\pi/2N)} \right)} \right)
\end{aligned}
\tag{6.28}
$$

The equation can be rewritten as

$$
a_k = \begin{cases} \frac{1}{N} \frac{\sin\left[\frac{2\pi k(N_1+1/2)}{N}\right]}{\sin\left(\frac{\pi/k}{N}\right)} & k \neq 0, \pm N, \pm 2N, \ldots \\ \frac{2N_1+1}{N} & k = 0, \pm N, \pm 2N, \ldots \end{cases}
\tag{6.29}
$$

The scaled Fourier series coefficients Na_k are plotted in Fig. 6.10 for $2N_1 + 1 = 5$ and $N = 40$.

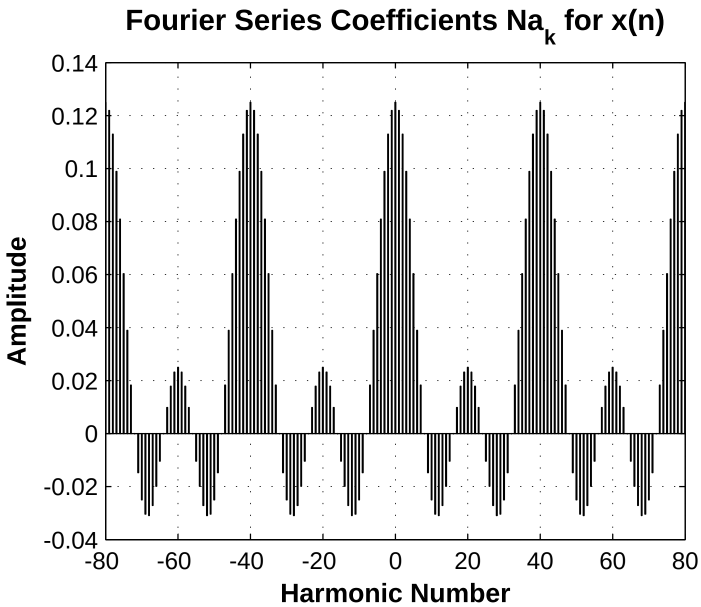


Figure 6.10: Fourier series coefficients of the discrete periodic square wave

NOTE

There are some important differences between Fourier series representation for continuous time periodic signals and Fourier series representation for discrete time periodic signals. The Fourier series representation for discrete time periodic signals is a finite series, while Fourier series representation for continuous time periodic signals is infinite series. Also, an important property that must be noted is that discrete Fourier series coefficients are periodic with period N, i.e.

$$
a_k = a_{k+N} \tag{6.30}
$$

Compare Fig. 6.4 ($T = 20T_1$) and Fig. 6.10 ($N = 20N_1$) and you can notice the two differences clearly.

B. Discrete time Fourier transform

Computers and other digital electronic based systems cannot handle continuous time signals. These systems can only process discrete data. Therefore, a discrete form of the Fourier Transform, i.e. a numerical computation of the Fourier transform, is needed to give us spectral analysis of discrete signals. This transform is called discrete Fourier transform.
The discrete Fourier transform converts a time domain sequence resulting from sampling a continuous time signal into an equivalent frequency domain sequence representing the frequency content of the given signal. Thus, the DFT is given by the following equation.

$$X[k] = \sum_{n=0}^{N-1} x[n]e^{-j2\pi kn/N} \qquad k = 0, 1, 2, ..., N-1 \tag{6.31}$$

The Inverse Discrete Fourier Transform (IDFT) performs the reverse operation and converts a frequency domain sequence into an equivalent time domain sequence

$$x[n] = \frac{1}{N}\sum_{k=0}^{N-1} X[k]e^{-j2\pi kn/N} \qquad n = 0, 1, 2, ..., N-1 \tag{6.32}$$

The DFT plays an important role in many applications of digital signal processing including linear filtering, correlation analysis and spectrum analysis.
The computation of the DFT is computationally expensive as the DFT computes the sequence $X[k]$ of N complex valued numbers given another sequence of data $x[n]$ of length N. From the equations for the DFT, it can be seen that to compute all N values N^2 complex multiplications and $N^2 - N$ complex additions are required. The direct computation using the DFT is inefficient because it does not exploit the symmetry and periodicity properties. Thus a number of algorithms exist that makes these computations more efficient. An important algorithm for computer the DFT is the Fast Fourier Transform (FFT).
The FFT algorithms exploit these two basic properties and make the computation more efficient.

NOTE
It should be clear that FFT is not an approximation of the DFT. It yield the same result as the DFT with fewer computations required.

In MATLAB, the FFT is performed using the function (fft). The output of the fft() function by itself is a vector of complex numbers. The following formula returns a vector of the magnitudes of each of the frequencie's contributions to the signal's amplitude.

$$y = 2 * \frac{abs(\text{fft\_data})}{length(data)} \tag{6.33}$$

To achieve the absolute value of the complex magnitude we need the abs() function. Since the fft() returns the complex amplitudes scaled by the overall length of the data, we need to divide by length of the data. Finally the equation has to be multiplied by 2 (because of Euler's Relation!?). Only the first half of the vector y contains relevant data, so

$$y = y\left(1 : \frac{length(data)}{2}\right) \tag{6.34}$$

The highest frequency that can be perceived in a signal is given by the Nyquist Frequency:

$$f_{nyqu} = \frac{F_s}{2} \quad \text{where } F_s \text{ is the sampling frequency} \tag{6.35}$$

The frequencies that correspond to the y vector range from $0\,Hz$ to the Nyquist Frequency can be generated by:

$$\text{f} = \text{linspace}\ (0,\ f_{nyqu},\ \text{length(y)}) \tag{6.36}$$

Finally the plot command 'plot(f, y)' shows the frequency components from the y vector. A detailed description of fft() function can be found along with examples in the Matlab documentation [5].

6.1.5 Definitions and practical hints

The sampling process is the transfer of a continuous time signal into a discrete time signal or the transfer from the world of analog signal processing to the world of digital signal processing. In practice, most signal processing is performed on discrete time signals and not on continuous time signals. This applies despite the fact that most of the signals encountered in science and engineering are analog in nature. In the next experiment, the sampling theory and sampling techniques will be discussed in more details.

Continuous Fourier transform

The Continuous Fourier transform is used to transform a continuous time signal into the frequency domain. It describes the continuous spectrum of a non-periodic time signal.

Discrete Fourier transform

Is used in the case where both the time and the frequency variables are discrete.

Fast Fourier transform

Is a special algorithm which implements the discrete Fourier transform with considerable savings in computational time. It must be clear that the FFT is not a different transform from the DFT, but rather just a means of computing the DFT with a considerable reduction in the number of calculations required.

FFT using the oscilloscope

As part of the experiment, the FFT has to be obtained using the oscilloscope. In order to obtain the FFT, the following steps should be carried out. First the time domain signal should be set properly by the following steps:

a. Press **AUTOSET** to display the time domain waveform.

b. Position the time domain waveform vertically at center of the screen to get the "true" DC value.

c. Position the time domain waveform horizontally so that the signal of interest is contained in the center eight divisions.

d. Set the YT waveform SEC/DIV timebase to provide the resolution you want in the FFT waveform. This decides how high the frequency content of the transform will be. The smaller the time scale used the higher the frequency content.

Then the FFT can be performed as follows:

a. Locate the **FFT** button on the front panel of the oscilloscope.

b. On pressing it the oscilloscope will change into FFT mode and you will enter the FFT menu.

c. In function choose fft.

d. The following screen will appear. (Clearly the waveform in your case will be different.)

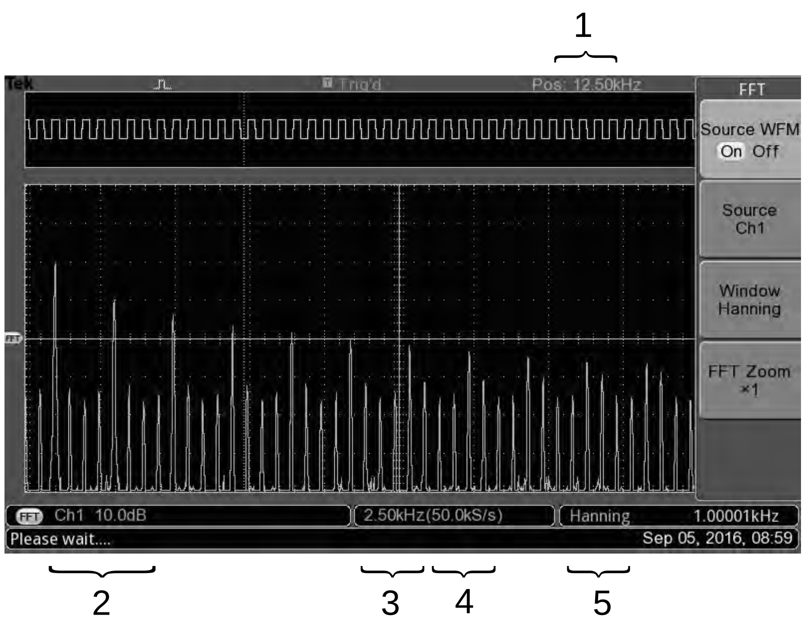


Figure 6.11: Hardcopy from oscilloscope screen

1. Frequency value at center position of the screen.

2. Vertical scale in dB/division.
Magnitude is referenced to $0dB$, where $0\,dB$ equals $1\,V_{RMS}$.

3. Horizontal scale in frequency/division

4. Sample rate in number of samples/sec

5. FFT window type.

The Fourier series represents a periodic waveform as an infinite series of harmonically related sinusoids. Since the Fourier series contains only discrete frequencies, each sinusoidal component of the waveform is represented by a vertical line on a plot of the signal magnitude versus frequency. The height of the line represents the magnitude of the contribution from that particular frequency. The location of the line along the horizontal axis identifies its frequency.
For all FFT's in this experiment always use the Hanning window, as it is best suited among the given options for a periodic signal. A windowing function is basically a function used to cut out a part of the signal in time domain so that an FFT can be carried out on it. Thus basically it is multiplication by some kind of rectangular pulse in time domain, which implies convolution with some kind of a sinc in frequency domain. Using a windowing function affects the transform but is the only practical method for obtaining it.
Once the time domain signal has been set up as discussed previously and you are in the FFT screen. The following steps must be carried out:

a. Bring the region of the frequency domain that you are interested in towards the middle of the screen.

b. Adjust the FFT zoom button in order to zoom into the transform sufficiently till you reach the magnitude of frequency that you desire. At this point you should normally make your hardcopy, as it is where you would be able to see the transform most clearly.

6.1.6 References

1. A. V. Oppenheim, A. S. Willsky, S. H. Nawab, "Signals and Systems", Prentice Hall, Second Edition (1997).

2. Raymond A. DeCarlo, Pen-Min Lin, "Linear Circuit Analysis", Oxford University Press, Second Edition (2001).

3. J. G. Proakis, D. G. Manolakis, "Digital Signal Processing", Prentice Hall, Third Edition (2002).

4. Sarma, M.S., "Introduction to Electrical Engineering", Oxford University Press, 2001.

5. MATLAB Documentation.

6. TDS220 manual.

7. TDS200-Series Extension Modules manual.

6.2 Prelab Fourier Series and fourier Transform

6.2.1 Problem 1 : Decibels

In the lab, you must be able to express the signal amplitude in V_{pp} and V_{rms}, also you have to know what dBV_{rms} corresponds to.

1. Given $x(t) = 2\cos(2\pi 1000t)$,

   a. What is the signal amplitude in V_{pp}?

   b. What is the root-mean-square value of the provided signal in V_{rms}?

   c. What is the amplitude of the spectral peak in dBV_{rms}?

2. For a square wave of $4\,\mathrm{V}_{pp}$ and the voltage level changes between $-2\,\mathrm{V}$ and $2\,\mathrm{V}$,

   a. What is the signal amplitude in V_{rms}?

   b. What is the amplitude in dBV_{rms}?

6.2.2 Problem 2 : Determination of Fourier series coefficients

1. Determine the Fourier series coefficients up to the 5th harmonic of the function

$$f(t) = \frac{2}{T}t \qquad\qquad -\frac{T}{2} < 0 < \frac{T}{2}$$

2. Use MatLab to plot the original function.

3. Use MatLab and plot function using the calculated coefficients.

6.2.3 Problem 3 : FFT of a Square/Rectangular Wave

Write a Matlab script:

1. Generate a square wave of 1 ms period, $2\,\mathrm{V}_{pp}$ amplitude, no offset, and duty cycle 50% (**hint** : use 'square' function). Use 200 kHz as the sampling frequency for the problem.

2. Plot the square wave in time domain.

3. Obtain the FFT spectrum using Matlab FFT function. Make the FFT length to be the length of the square wave data vector.

4. Plot the single-sided amplitude spectrum in dBV_{rms}.

5. Plot the spectrum including only the first four harmonics in dBV_{rms}.
   Hint: Use the Matlab command 'xlim'.

6. Repeat the previous steps using 20% and 33% duty cycles, respectively. Keep period and amplitude constant.

7. Discuss the changes for smaller pulse width. Use Eq. (6.11) to prove your statement.
   Hint: Use the **subplot** command to plot the spectrum magnitude for the three cases 50%, 33% and 20% duty cycle to ease the comparison.

6.2.4 Problem 4 : FFT of a sound sample

The Matlab command '[y,Fs] = audioread(filename)' reads a wave file specified by the string 'filename', returning the sampled data in 'y' and the sample rate in 'Fs' in Hertz.

1. Download the sound file ('Sound_Sample.wav') from the Campusnet webpage.
2. Using Matlab, read the sound file and plot the first 10 ms of the signal in time domain.
3. Use the Matlab FFT function to compute the spectrum and plot the single-sided amplitude spectrum in dBV_{rms}..
4. What are the tones forming this signal?

6.3 Execution Fourier Series and fourier Transform

6.3.1 Problem 1 : FFT of Single Tone sinusoidal wave

1. Use the function generator to generate a sinusoidal wave having 500 Hz frequency, 2 V_{pp} amplitude and no offset. Use the measure function to verify all properties. Take a hard copy in time domain.

2. Obtain the FFT spectrum using the oscilloscope FFT function. Use the cursor to measure the properties. Take hard copies of the complete spectra and the zoomed spectra peak.

3. Generate a sinusoidal wave having 0 dB spectrum peak, 2 KHz frequency, without a dc offset. What is the amplitude value? Use the measure function and the cursors. Take hard copies of time and frequency domain.

6.3.2 Problem 2 : FFT of square wave

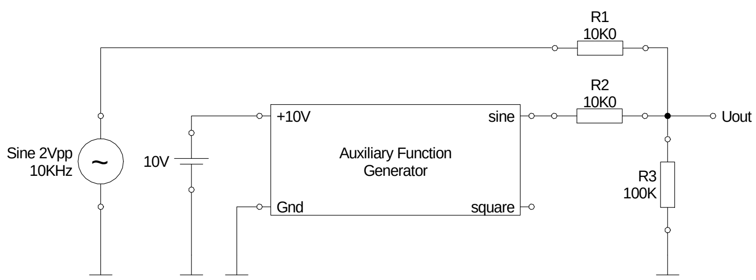
1. Use the function generator to generate a square wave having 1 ms period, 2 V_{pp} amplitude, and no offset. Check the properties with the measure function. Take a hard copy.

2. Obtain the FFT spectrum. Instead of using the time base (sec/div) control to accurately measure the frequency components, use the FFT zoom control that provides a zoom factor up to 10 and use the cursors to determine the amplitudes and the frequency of the fundamental and the first four harmonics. Take hard copies of the FFT signal.

3. Obtain the FFT spectrum for 20 % duty cycles. Determine the amplitudes of the fundamental frequency and the first four harmonics. Take hardcopies of the signal in time and frequency domains.

6.3.3 Problem 2 : FFT of Multiple-Tone sinusoidal wave

1. Combine the signal from the sine output of the auxiliary signal generator and a 2 V_{pp}, 10 KHz sinusoidal wave from the Agilent signal generator. Use the following circuit.



2. Take a hard copy of the signal in time domain.

3. Take a hard copy of the FFT spectrum of the signal.

6.4 Evaluation

6.4.1 Problem 1 : FFT of Single Tone sinusoidal wave

In the lab report:

1. What is the reference value of the oscilloscope for $0dB$.
2. Use Matlab to calculate the expected FFT spectra for the parameters given in part 6.3.1.1. Is the calculated spectra consistent with the measured spectra?
3. Use Matlab to calculate the expected FFT spectra for the parameters given in part 6.3.1.3. Is the calculated spectra consistent with the measured spectra?
4. Compare the results from Matlab with the measured values. Discuss the differences.

6.4.2 Problem 2 : FFT of square wave

In the lab report:

1. For frequency domain measurements, the frequency scale needs to be expanded in order to accurately measure the frequency components. This could be done with the time base (sec/div) control. What is the effect of doing this on the measured bandwidth? Information can be found in reference [6] and [7].
2. Use the hardcopies taken to discuss the effect of changing the duty cycle on the FFT results.

6.4.3 Problem 3 : FFT of square wave

In the lab report:
Use the hardcopy of the spectrum and discuss the linearity of the FFT.

7. Theory 4 : Sampling

7.1 Introduction to the experiment

7.1.1 Objectives of the experiment

The goal of this experiment is to introduce the basics of sampling and to implement different sampling circuits.
The foundations of sampling will be discussed on the signals and systems and on the component level. Different sampling schemes like impulse train sampling, rectangular pulses sampling and sample and hold will be introduced. In addition, the consequence of the different sampling schemes on the reconstructed signals will be described.
Furthermore, two different types of sampling circuits will be discussed. The first sampling circuit is the sampling bridge used for high-speed sampling applications, e.g. as part of a digital oscilloscope. The second sampling circuit is a sampling circuit based on Metal Oxide Semiconductor Field Effect Transistors (MOSFETs).

7.1.2 Introduction

Most of the signals in nature exist in an analog form. Sampling is the transfer of a continuous time signal into a discrete time signal. Sampling is the first and a very important step to provide signals, which can be digitally processed. Therefore, sampling is the connecting element between the world of analog and digital signal processing.
The handout is divided into three parts, where the first part of the handout introduces the sampling theory and discusses the difference between sampling and quantization of signals. The second part will explain different ways of sampling like impulse train sampling, rectangular pulses sampling and sample and hold schemes. Finally, different implementations of sampling circuits will be discussed.

7.1.3 Sampling Theory

Sampling step versus quantization step

Sampling is the transfer of a continuous time signal into a discrete time signal. However, sampling should be clearly distinguished from the quantization of a signal. Both steps are necessary to carry out an analog-to-digital (A/D) conversion.
Quantization is the transformation of a continuous signal into a discrete signal in terms of discrete amplitudes or discrete signal levels. The quantization leads to discrete values for the amplitude, whereas the sampling leads to values of discrete times. The difference between sampling and quantization is illustrated in Fig. 7.1.
Fig. 7.1 provides a schematic description of an A/D conversion process. In order to transfer continuous time signals into a stream of bits, the signal can be first

quantized and afterwards sampled or the signal can be first sampled and quantized afterwards.
In both cases, we get a signal that is discrete in terms of time and amplitude. After the quantization and the sampling the signal is coded, meaning the signal is transformed in a stream of bits, which can be processed or transmitted. After carrying

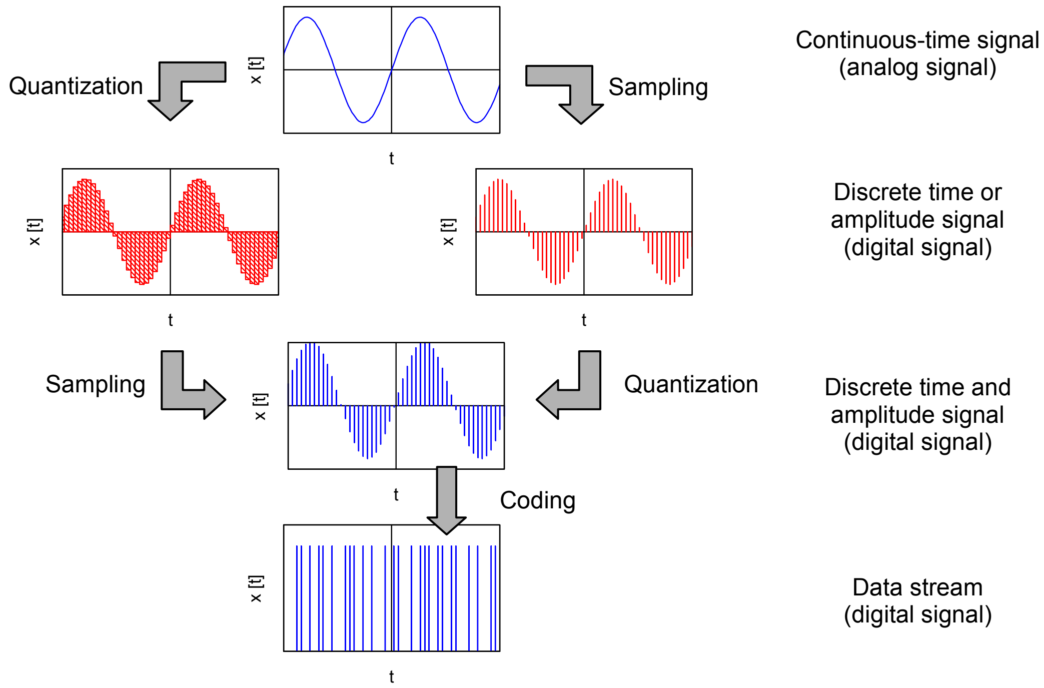


Figure 7.1: Schematic illustration of an A/D conversion process using a sampling, quantization and coding steps.

out certain processing steps the digital signal is very often converted back into an analog signal. Under certain circumstances, it is possible to completely recover the initial signal. This very important property follows the sampling theorem. This theorem is simple, but very important and useful, because based on the sampling theorem we can decide whether a signal can be completely recovered or not.

The sampling theorem

We assume that $x(t)$ is a band-pass limited signal which has a Fourier Transform $X(j\omega) = 0$ for frequencies larger than a maximum cut-off frequency ω_M. The initial signal can be reconstructed from the sampled signal if the sampling frequency ω_S is two times larger than the maximum cut-off frequency of the band pass filter.

$$\omega_S > 2\omega_M \tag{7.1}$$

The sampling frequency ω_S can be described by

$$\omega_S = \frac{2\pi}{T} \tag{7.2}$$

where T is the period of an impulse train sampling signal. If the sampling frequency is smaller than 2 times the maximum frequency of the band-pass limited signal, the initial signal cannot be completely reconstructed afterwards. In this case, the sample rate is not high enough and the term under-sampling or aliasing is used. If the sampling frequency is exactly equal to 2 times the maximum frequency of the band-pass limited signal, then we speak about the Nyquist frequency or the Nyquist rate. The sampling frequency has to be higher than the Nyquist rate. Otherwise, the signal cannot be reconstructed.

Remark: Sampling is not only important when we deal with voltages or currents. There are several other areas, where we have to deal with sampling. Take for example the area of digital photography or digital image processing. The transfer of an analog to a digital picture requires a sampling step. The original picture is sampled for example by a digital camera and the sampling signal is in this case defined by the size of the pixel of your camera in combination with the optics of your camera. So, if you complain about the resolution of your digital camera you already made your experience with the sampling theorem.

7.1.4 Sampling Methods

1. Impulse Train sampling (Ideal sampling)

We will first discuss the ideal sampling, where a periodic series of unit impulses is used as the sampling signal. The sampling scheme is therefore called Impulse Train sampling. The schematic sampling procedure is shown in Fig. 7.2.

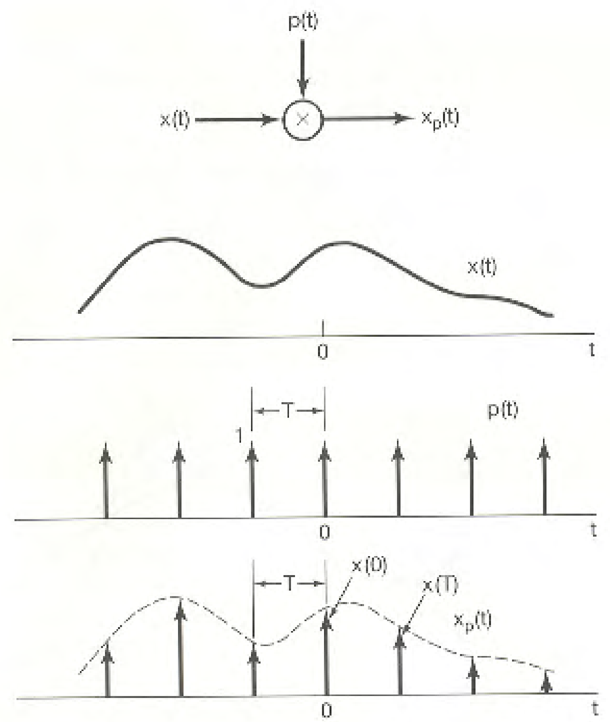


Figure 7.2: Schematic illustration of Impulse Train sampling [1]. Top: Continuous time signal, Middle: Impulse train, Bottom: Sampled signal

The continuous signal $x(t)$ is sampled by the signals $p(t)$, which represents the impulse train signal. The periodic impulse train $p(t)$ is described by

$$p(t) = \sum_{n=-\infty}^{\infty} \delta(t - nT) \tag{7.3}$$

The sampling signal is multiplied with the input signal $x(t)$. After multiplication, we get the signal $x_p(t)$. The index p indicates that an impulse samples the signal.

$$x_p(t) = x(t) \sum_{n=-\infty}^{\infty} \delta(t - nT) \tag{7.4}$$

We can rewrite Eq. (7.4), so that we get the following equation

$$x_p(t) = \sum_{n=-\infty}^{\infty} x(nT) \cdot \delta(t - nT) \tag{7.5}$$

The sampled signal $x_p(t)$ is illustrated in Fig. 7.2. Further information regarding Impulse Train sampling can be found in chapter 7 of reference [1].

Now, let us discuss how the impulse train sampled signal can be reconstructed as shown in Fig. 7.3. We still assume that the initial band-pass limited signal was sampled by an impulse train and that the sampling frequency was higher than 2 times the maximum frequency of the band-pass limited input signal. The output signal $x_p(t)$ in the frequency domain can be described by

$$X_P(f) = X(f) \sum_{n=-\infty}^{\infty} \delta\left(f - \frac{n}{T}\right) \tag{7.6}$$

The initial signal is convolved with an impulse train in the frequency domain. It is assumed that the input signal in the frequency domain corresponds to a triangle. It can be seen that the signal is reproduced at integer multiples of the sampling frequency. The output signal $X_p(j\omega)$ of the sampling circuit is shown in Fig. 7.3c. The input signal can be recovered if the signal $X_P(j\omega)$ is filtered by a low-pass filter with a gain of T and a cut-off frequency greater than ω_M and less than $\omega_s - \omega_M$ to cut-off the redundant part of the signal as indicated in Fig. 7.3d. The final output signal, which is a perfect reconstruction of the input signal is shown in Fig. 7.3e.

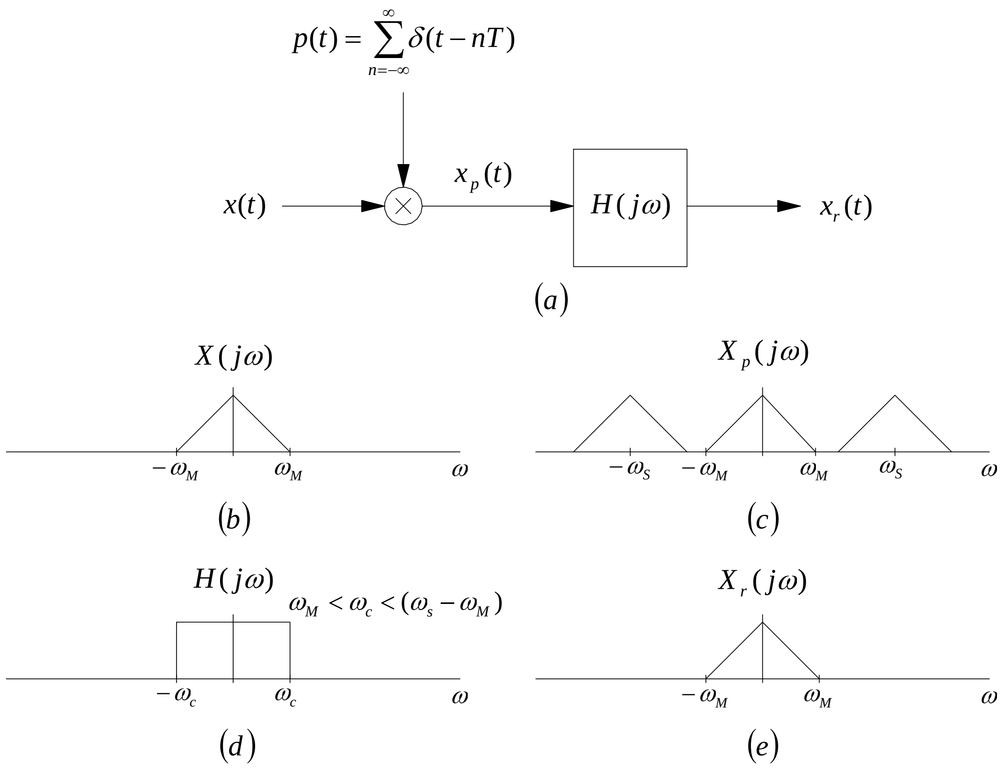


Figure 7.3: Recovery of a continuous-time signal from its samples using an ideal low pass filter [1].

2. Rectangular Pulses sampling (Real sampling)

So far, we discussed the sampling of a continuous signal by using an impulse train. However, in practice it is difficult to generate and transmit unit impulses. Therefore, it is very convenient to generate rectangular pulses. The sampling signal $p(t)$ in Fig. 7.2 can be therefore described by

$$p(t) = \sum_{n=-\infty}^{\infty} \text{rect}\left(\frac{t-nT}{T_0}\right) \tag{7.7}$$

where T_0 the width of the rectangular pulse which is used as the sampling signal. Consequently, the output signal is given by:

$$x_p(t) = \sum_{n=-\infty}^{\infty} x(nT) \cdot \text{rect}\left(\frac{t-nT}{T_0}\right) \tag{7.8}$$

In practice, it is common to use a first order sample and hold scheme rather than a Rectangular Pulses sampling scheme.

3. First order Sample and Hold

In such a case, the sampling signal is held constant for a certain period. For example, the sampling signal is held constant until the next sample is taken or the signal is held constant for a shorter period.

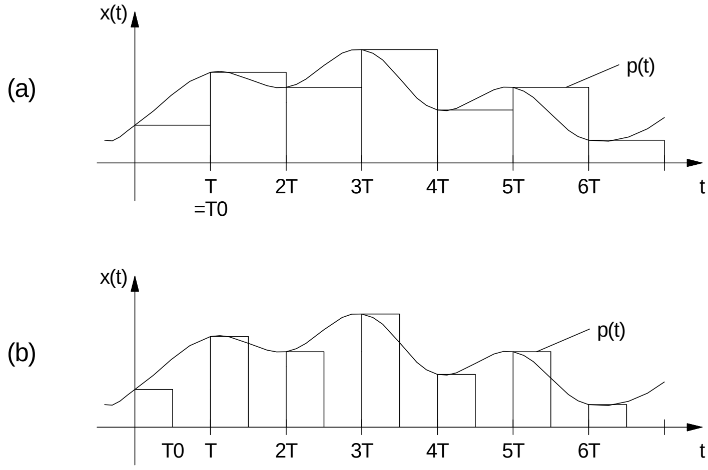


Figure 7.4: A hold function (a) the value is held until the next sample is taken. (b) held for half of the period of the sampling signal [3].

Is the value held for half the sampling period the sampling signal is given by

$$p(t) = \sum_{n=-\infty}^{\infty} \mathrm{rect}\left(\frac{t - T_0/2 - nT}{T_0}\right) \tag{7.9}$$

where $T_0 = T$. Consequently, the output signal is given by

$$x_{p,SH}(t) = \sum_{n=-\infty}^{\infty} x(nT) \cdot \mathrm{rect}\left(\frac{t - T_0/2 - nT}{T_0}\right) \tag{7.10}$$

The expression can be rewritten by

$$\begin{aligned} x_{p,SH}(t) &= \left[\sum_{n=-\infty}^{\infty} x(nT) \cdot \delta(t - nT)\right] \cdot \mathrm{rect}\left(\frac{t - T_0/2}{T_0}\right) \\ &= \left[x(t) \sum_{n=-\infty}^{\infty} \delta(t - nT)\right] \cdot \mathrm{rect}\left(\frac{t}{T_0}\right) \cdot \delta\left(t - \frac{T_0}{2}\right) \end{aligned} \tag{7.11}$$

The sample and hold procedure can be described by an impulse train sampling step, where the output signal of the impulse train sampling step is convolved with a rectangular pulse. A schematic implementation of a first order sample and hold procedure is shown in Fig. 7.5.
A simple low-pass filter with a constant gain cannot be used to reconstruct the initial signal $x(t)$ from $xp.SH(t)$. The Fourier transform of Eq. (7.10) explains why the initial signal $x(t)$ cannot be completely reconstructed after being sampled and held.

$$X_{p,SH} = X(f) \sum_{n=-\infty}^{\infty} \delta\left(f - \frac{n}{T}\right) \cdot \mathrm{sinc}(\pi f T_0) \cdot \exp\left(-j\pi f T_0/2\right) \tag{7.12}$$

The first two terms of Eq. (7.12) are identical with Eq. (7.6). However, the convolution of the impulse train sampled signal in the time domain corresponds to a

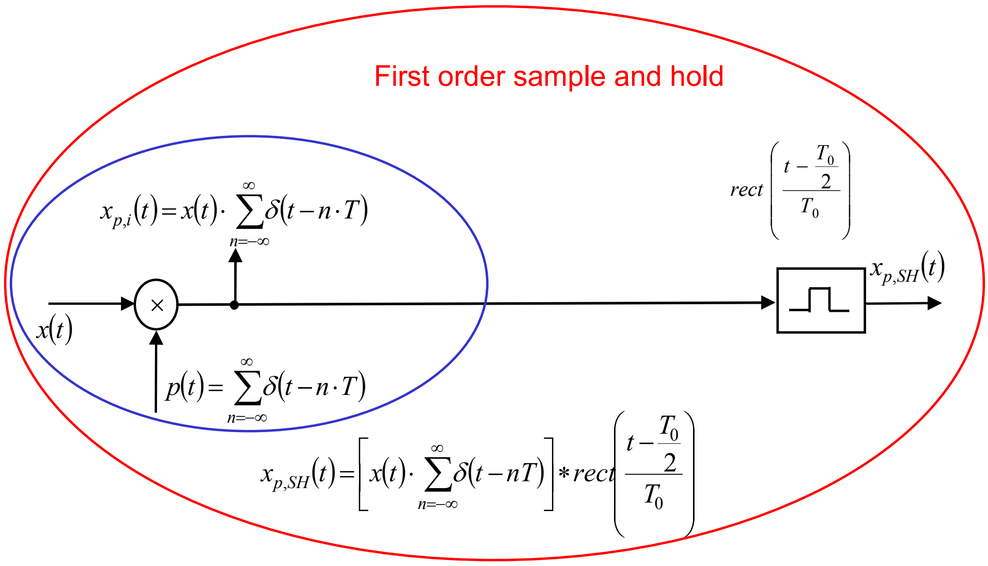
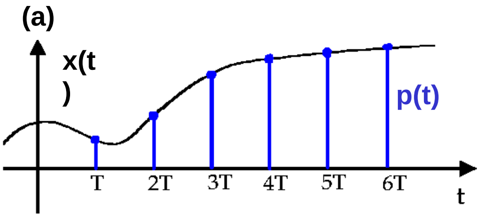


Figure 7.5: Description of a first order sample and hold implementation. The first order sample and hold implementation can be described by an ideal or Impulse-train sampled signal, which is convolved with a rectangular signal.

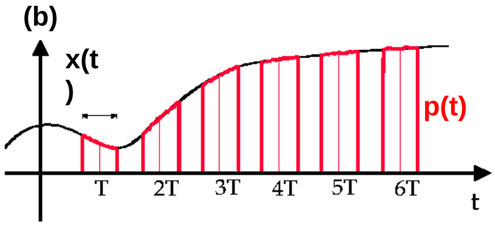
multiplication with a *sinc* function in the frequency domain and the additional shift in the time domain leads to a phase shift in the frequency domain.
The problem can be solved by the implementation of a filter, which compensates for the introduced nonlinearities. Therefore, a filter with the following properties can be used.

$$H_r(f) = \frac{\exp\left(j\pi f T_0/2\right)}{\mathrm{sinc}(\pi f T_0/2)} \tag{7.13}$$

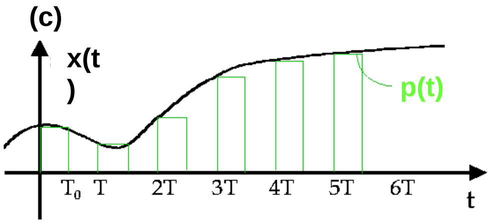
A summary of the different sampling schemes is given in table (7.1).



$$x_i(t) = \sum_{n=-\infty}^{\infty} x(nT)\cdot\delta(t-n\cdot T)$$



$$x_p(t) = x(t)\cdot \sum_{n=-\infty}^{\infty} rect\left(\frac{t-n\cdot T}{T_0}\right)$$



$$x_p(t) = \left[x(t)\cdot\sum_{n=-\infty}^{\infty}\delta(t-nT)\right] * rect\left(\frac{t}{T_0}\right) * \delta\left(t-\frac{T_0}{2}\right)$$

Table 7.1: Overview of the different sampling schemes (a) impulse train sampling, (b) real or rectangular sampling and (c) sample and hold [3]

7.1.5 Sampling Circuit Applications

Several different sampling circuits are known. Very often, the sampling circuit is the first stage of an A/D conversion circuits. In the following, we will discuss two implementations of sampling circuits. Depending on the application of the A/D conversion circuit, the requirements are different. The most important parameters are the sampling rate and the resolution.
For the A/D conversion of speech signals the sampling rate can be relatively low, however, the resolution has to be relatively high. In the case of a digital oscilloscope, the sampling rate has to be high, but the resolution can be low in comparison to other A/D converters. The following specifications are important for sampling circuits:

1. The transfer function of the sampling circuit should be close to being ideal. If the sampling circuit is switched on the attenuation of the signal should be as low as possible, whereas in the switched off state the attenuation should be as high as possible.

2. The sampling rate should be as high as possible.

3. The propagation delay between the input and the output side should be as low as possible.

4. The input and the output side should be decoupled.

The sampling bridge

The sampling bridge is used for high-speed sampling. An implementation of a sampling bridge is shown in Fig. 7.6. Generally, two reasons exist why sampling bridges are applied for high-speed sampling. The diodes are typically realized by Gallium Arsenide rather than silicon. Electrons "travel" faster inside the Gallium Arsenide crystal than inside a silicon crystal so that the switching speed of a Gallium arsenide diode is higher. Furthermore, the Gallium Arsenide diodes used for sampling circuits are Schottky diodes rather then pn-diodes. Schottky diodes usually have a very small equivalent capacitance, which allows fast switching.
Remark: As part of the lab we will use regular silicon pn-diodes to demonstrate the working principle of a sampling bridge. The general working principle of the circuits is not affected by this modification.

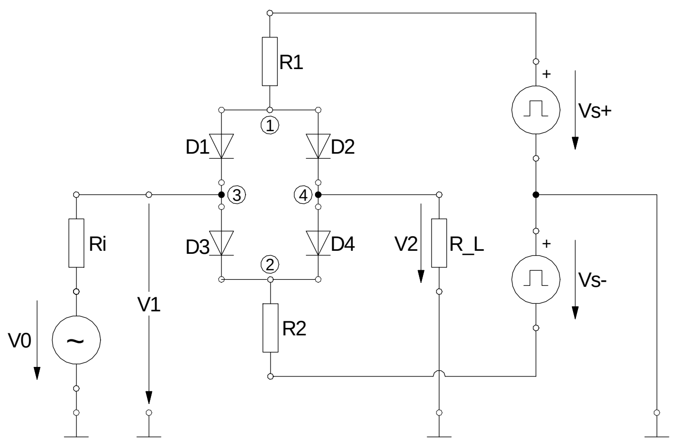


Figure 7.6: High speed Sampling circuit based on a four diodes.

The operating principle of the circuit can be described as follows. The input voltage is given by the voltage V_0. The resistor R_i is the internal resistance of the input source. The sampling circuit is formed by the four diodes D_1 to D_4, the two resistors R_1 and R_2 and the sampling signal V_s+ and V_s-. At the same time, positive and negative sampling pulses are applied to the node (1) and (2). The voltage V_s is high enough so that the diodes are forward biased. Consequently, the current path between the node (3) and (4) gets conductive and the signal from the input side is applied to the load resistor on the output side. In the ideal case, the voltage V_2 is equal to V_1, where V_1 is the voltage V_0 plus the voltage drop across the resistor R_i. The following aspects have to be considered while designing a sampling bridge. The amplitude of the sampling signal V_s has to be higher than the voltage V_1. Otherwise, not all diodes of the bridge are forward biased. Furthermore, the resistors R_1 and R_2 have to be in a certain range. If the resistance for R_1 and R_2 are too high, the voltage drop across the diodes is not high enough and the diodes are not under forward bias conditions. If the resistance is too low, the input signal will not be applied to the load, because the resistance of the sampling circuit is smaller than the resistance of the load, so that the current will pass through the sampling circuits.

Furthermore, it is obviously clear that the diodes are non-ideal devices. Therefore, the amplitude of the sampling voltage has to be at least equal to the diffusion voltage of the diodes.

CMOS sampling circuits

For most of the applications, the sampling rate does not have to be extremely high. For example, in the area of speech processing or real time processing the sampling rate is in the range of several 1000 samples per second (Ksamples/s). In such cases, CMOS circuits are typically used to realize sampling circuits. The major advantage of CMOS circuits is that the sampling circuit can be directly combined with other electronic components on the same chip.
A CMOS sampling circuit is shown in Fig. 7.7. The circuit consist of two major parts, which are the inverter, and two pass transistors. The circuit in Fig. 7.7 can be used for bidirectional data transfer as well as for the implementation of a sampling circuit. An implementation of the CMOS switch based on the transistor level is shown in Fig. 7.8. The inverter can be realized by two MOS field effect transistors.

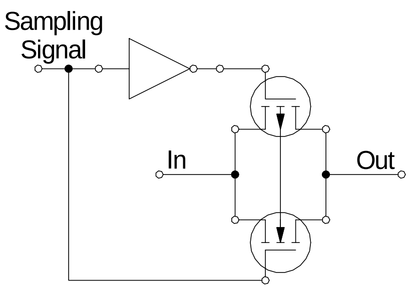


Figure 7.7: The sampling circuit based on two pass transistors in combination with an inverter.

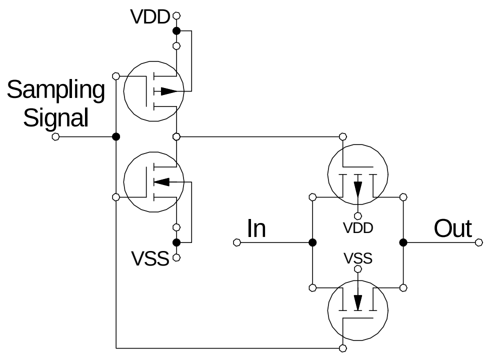


Figure 7.8: Implementation of the sampling circuit on the transistor level.

Bidirectional data transfer means that the circuit can be used for a data transfer from the input to the output side or a data transfer from the output to the input side. The data transfer is realized by two pass transistors. In order to transfer the data from the input to the output side or vice versa the gate of the NMOS transistor (arrow towards the gate) is applied to V_{DD} and the gate voltage of the PMOS transistor (arrow away from the gate) is applied to V_{ss}. A CMOS inverter provides the inverted sampling signal. The sampling signal is directly connected to the gate of the NMOS and the inverted sampling signal is applied to the gate of the PMOS transistor. Consequently, the voltages applied to both of the gates of the pass transistors are always inverted to each other. In the off-state the gate of the NMOS transistor is applied to V_{ss} and the gate of the PMOS transistor is on V_{DD}. It is important to mention that the sampling circuit in Fig. 7.8 has limitations in terms of the range of operation. Only signals can be sampled or transferred from the input to the output side or vice versa, which have voltage levels between V_{DD} and V_{SS}.

7.1.6 Reference

1. A.V. Oppenheim, A.S. Willsky, S.H. Nawab, Signals and Systems, 3rd edition, Prentice Hall Signal Processing Series (1997).

2. Fairchild Semiconductors, CD4016BC data sheet

3. O. Loffeld, Allgemeine Nachrichtentechnik, University Siegen.

7.2 Prelab Sampling

7.2.1 Problem 1: The Sampling Theorem

1. Analog signals are usually passed through a low-pass filter prior to sampling. Why is this necessary?

2. What is the minimum sampling frequency for a pure sine wave input at $3KHz$? Assume that the signal can be completely reconstructed.

3. What is the Nyquist frequency?

4. What are the resulting frequencies for the following input sinusoids $500Hz$, $2.5KHz$, $5KHz$ and $5.5KHz$ if the signals are sampled by a sampling frequency of $5KHz$?

5. Mention three frequencies of signal that alias to a $7Hz$ signal. The signal is sampled by a constant 30 Hz sampling frequency.

7.2.2 Problem 2: Impulse Train Sampling and Real Sampling

Consider the circuit shown in figure (7.2). The input signal x(t) is given by a sine function, with an amplitude of 5 V peak and a frequency of 50 Hz. The sampling signal p(t) is represented by a unity impulse train. Use an overall sampling rate of 100 k samples/s) for the whole problem.

1. Carry out simulations for the following cases:

   (a) Under Sampling (use 48 Hz)

   (b) Nyquist Sampling

   (c) Over Sampling (use 1000 Hz

   Use the command **subplot** to visualize the continuous signal x(t), the sampling signal p(t) and the result for each of these cases.

2. The signal x(t) should be sampled by a rectangular pulse train. Modify the sampling function p(t), so that the width of the sampling pulse is 50% of the sampling period. Carry out simulations for the following cases:

   (a) Under Sampling

   (b) Nyquist Sampling

   (c) Over Sampling

   Use the same sampling rates and the same plot setup as before.

7.2.3 Problem 3: Sampling using a Sampling bridge

Modify the circuit in figure (7.6) in such a way that a single sampling source can be used to sample the input signal.

1. Sketch the modified circuit.
2. Explain the operation of the modified circuit.

8. Experiment 5 : AM Modulation

8.1 Introduction to AM and FM experiments

8.1.1 Objectives of the experiments

The goal of this experiment of the Signals and Systems Lab is to study different analog modulation techniques.
In the first part of the experiment amplitude modulation will be investigated. We will examine the properties of double-sideband (DSB) modulation, double-sideband suppressed carrier (DSB-SC) modulation, and single-sideband amplitude (SSB) modulation and their frequency spectra. The techniques used for demodulation will be explained. Practically, the oscilloscope will be used to demonstrate the impact of the amplitude modulation parameters on the modulated signal in time and frequency domain. Furthermore, you will build a complete amplitude modulation based system using the function generator as a modulator and the envelope detector circuit as a demodulator.
In the second part of the experiment frequency modulation will be investigated. The influence of frequency modulation parameters on the bandwidth will be explained. Practically, the oscilloscope will be used as a spectrum analyzer to demonstrate the impact of the frequency modulation parameters on the frequency domain. Furthermore, you will build a simple demodulation circuit consisting of a slope detector.

8.1.2 Introduction

Communication systems play a key role in the modern world in transmitting information. A Modulator is a part of all modern day electronic communication systems such as radio, television, and telephony.
One of the final steps before the transmission of the signal is modulation and one of the first steps on receiving the signal is demodulation. Modulation is the process of embedding an information-bearing signal into a second carrier signal while extracting the information-bearing signal is known as the demodulation process.
One large class of modulation methods relies on the concept of amplitude modulation (AM) in which the signal we wish to transmit is used to modulate the amplitude of another signal. A very common form of amplitude modulation is sinusoidal amplitude modulation in which the information signal is used to vary the amplitude of a sinusoidal signal. Another important form of AM systems involves the modulation of the amplitude of a pulsed signal, which is called pulse amplitude modulation (PAM). A wide variety of modulation methods are used in practice. In the handout we will examine some of the most important of these amplitude modulation techniques.
In frequency modulation, the information-bearing signal or the message signal we wish to transmit is used to modulate the frequency of another signal that is the carrier signal rather than amplitude variations in the carrier signal as in case of

amplitude modulation. Frequency Modulation is part of a more general class of modulation schemes known as angle modulation. Angle modulation includes both phase modulation and frequency modulation. Theories and concepts are similar for phase modulation and frequency modulation, but we will only refer to frequency modulation in this lab.

8.1.3 Why to Modulate?

Why do we have to modulate a signal for transmission? Or, why can't the signal be sent as it is? Modulation is required for a variety of reasons. There are three reasons that force us to use modulation.
The first reason is the attenuation of the channel, i.e. air attenuation is high for low frequency voice signals, but the attenuation is significantly lower for higher frequencies.
The second reason has to do with the laws of electromagnetic propagation, which dictate that the size of the radiating element, the antenna, is a significant fraction of the wavelength of the signal being transmitted. For example, if we want to transmit a $1kHz$ signal by a quarter wave antenna, the size of the antenna would need to be $75km$. On the other hand, if the signal is being transmitted on a high frequency carrier, say $630kHz$, the corresponding size of the radiating antenna needs to be only 119 m.
The third reason is the possibility of simultaneous transmission of different signals. As audio signals relevant to humans lie from a few hertz to a few thousand hertz, we could broadcast only one base-band signal at a time. Simultaneous transmission would cause the overlap of signals and we would not be able to separate them.

8.2 Amplitude Modulation

8.2.1 Band-limited signal AM modulation

The concept of amplitude modulation is a complex or sinusoidal signal (carrier signal) has its amplitude modulated by the information-bearing signal (message signal or modulating signal). In the handout, we will explain the amplitude modulation on the form where the carrier signal is sinusoidal (i.e. sinusoidal amplitude modulation)

$$c(t) = A_c \cos(2\pi f_c t + \theta_c) \tag{8.1}$$

Where, A_c is the amplitude and f_c is the frequency of the carrier. For convenience, we choose $\theta_c = 0$,so that the carrier signal is

$$c(t) = A_c \cos(2\pi f_c t) \tag{8.2}$$

Fig. 8.1 and Fig. 8.2 shows the carrier signal in time and frequency domain respectively. The modulating signal $x(t)$, could be music, video, or any other bandlimited signal. The amplitude of the carrier wave is varied about a mean value, linearly with the modulating signal thus forming the envelope of the modulated signal that has essentially the same shape as the modulating signal.

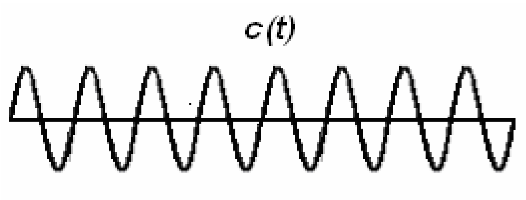


Figure 8.1: Time domain

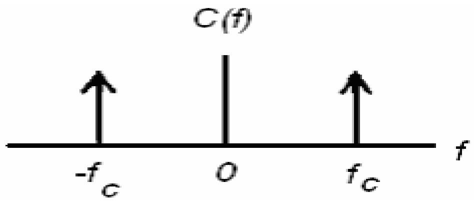


Figure 8.2: Frequency Domain

Mathematically this can be represented as:

$$y(t) = A_c[1 + kx(t)]\cos(2\pi f_c t) \tag{8.3}$$

where 'k' is the transmitter sensitivity. Two requirement must be satisfied in order to have a correct amplitude modulated signal that can be modulated properly:

1. The amplitude of $(kx(t))$ is always less than unity ,that is, $|km(t)| < 1$ for all t , otherwise the carrier wave becomes overmodulated and the modulated signal then exhibits envelope distortion.

2. The carrier signal fc must be greater than the highest frequency component W of the message signal $x(t)$, that is $f_c >> W$, otherwise an envelope cannot be visualized.

A band-limited signal x(t) in time domain is shown in Fig. 8.3.

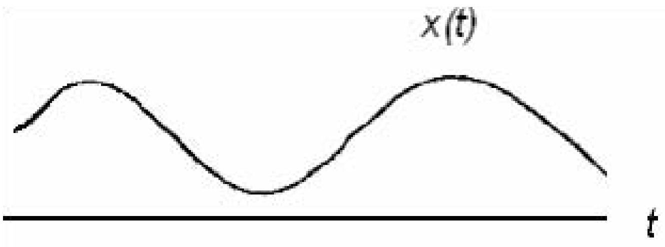


Figure 8.3: A band-limited signal in time domain

A band-limited signal is a signal whose Fourier transform is zero outside a given range of frequency, i.e. $X(j\omega) = 0$ for $|f| > W$, as shown in Fig. 8.4.

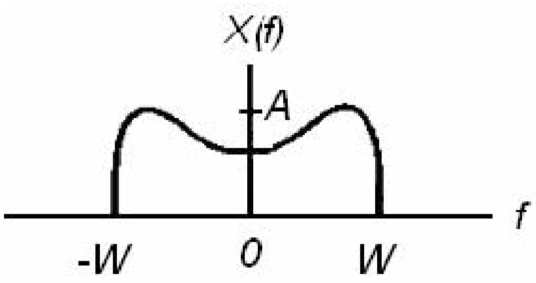


Figure 8.4: A band-limited signal in frequency domain

The carrier signal modulated by the band-limited modulating signal is shown in Fig. 8.5.

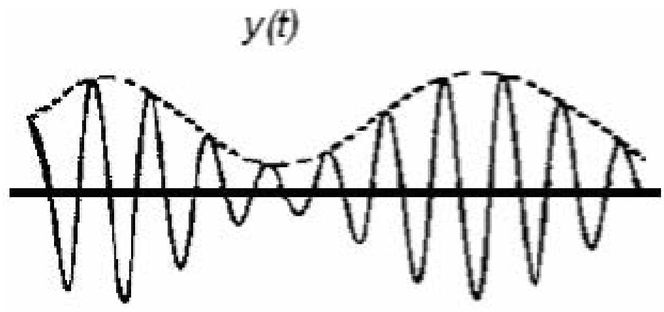


Figure 8.5: Modulated signal in time domain

The Fourier transform of the modulated signal calculated from Eq. (8.3) is given by,

$$Y(f) = \frac{A_C}{2}[\delta(f - f_C) + \delta(f + f_C)] + \frac{kA_C}{2}[X(f - f_C) + X(f + f_C)] \tag{8.4}$$

Fig. 8.6 illustrates the modulated signal spectrum.

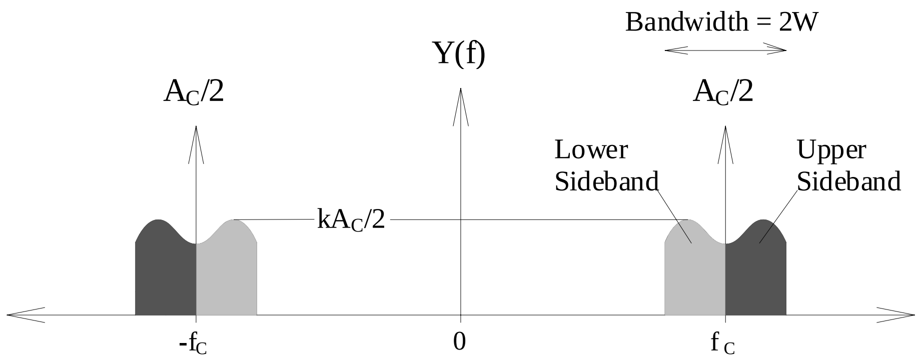


Figure 8.6: Modulated signal in frequency domain

The spectrum consists of two delta functions weighted by the factor $Ac/2$ at $\pm f_C$ and two copies of the base-band spectrum scaled by the factor $kAc/2$ and shifted at $\pm f_C$. The sidebands above and below the carrier frequency are called the upper and lower sidebands.

8.2.2 Single frequency AM modulation

Consider a modulating signal x(t) that consists of a single frequency component

$$x(t) = A_m \cos(2\pi f_m t) \tag{8.5}$$

Fig. 8.8 and Fig. 8.8 shows the modulating signal in time and frequency domain respectively.
The carrier signal modulated by the band-limited modulating signal is shown in Fig. 8.9.
Mathematically this can be represented as:

$$y(t) = A_c[1 + kA_m \cos(2\pi f_m t)] \cos(2\pi f_c t) \tag{8.6}$$

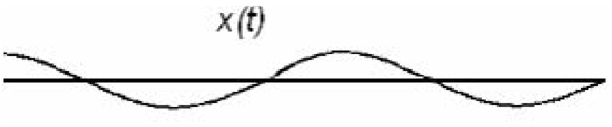


Figure 8.7: Time domain

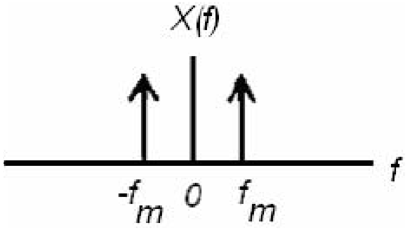


Figure 8.8: Frequency Domain

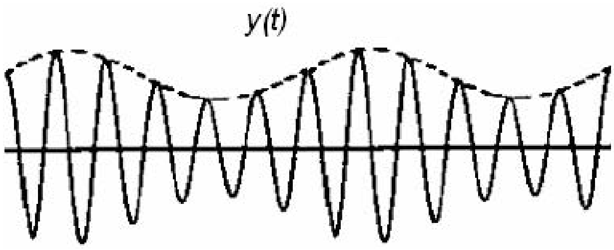


Figure 8.9: Modulated signal in time domain

where,

$$m = kA_m \tag{8.7}$$

'm' is called the modulation factor/index. To avoid overmodulation 'm' must be kept below unity. Using the trigonometric identities on Eq. (8.6), the Fourier transform of the modulated signal is given by

$$\begin{aligned} Y(f) =& \frac{A_C}{2}[\delta(f - f_C) + \delta(f + f_C)] + \\ & \frac{mA_C}{4}[\delta(f - f_C - f_m) + \delta(f + f_C + f_m)] + \\ & \frac{mA_C}{4}[\delta(f - f_C + f_m) + \delta(f + f_C - f_m)] \end{aligned} \tag{8.8}$$

Thus the spectrum of an AM signal consists of delta functions as shown in Fig. 8.10.

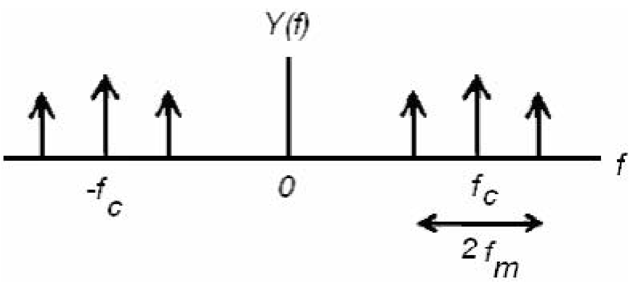


Figure 8.10: Modulated signal in frequency domain

8.2.3 AM signal power and bandwidth

The transmitted power and the channel bandwidth are two primary communication resources and should be used efficiently. The AM signal is a voltage function. The average power delivered to a resistor by the AM signal is compromised of three components, carrier power, upper side frequency power and lower side frequency power. The transmission bandwidth of the AM signal is equal to the difference between the highest frequency component (f_c+W) and the lowest frequency component (f_c-W) which is exactly twice the message bandwidth W, that is,

$$B_T = 2W \tag{8.9}$$

Where, W is the maximum frequency contained in the modulating message signal.

8.2.4 How to improve the efficiency of the DSB AM system

A. Double Sideband Suppressed Carrier (DSB-SC) modulation In double side band suppressed carrier (DSB-SC) modulation the transmitted signal consists only the upper and lower sidebands as shown in Fig. 8.11.

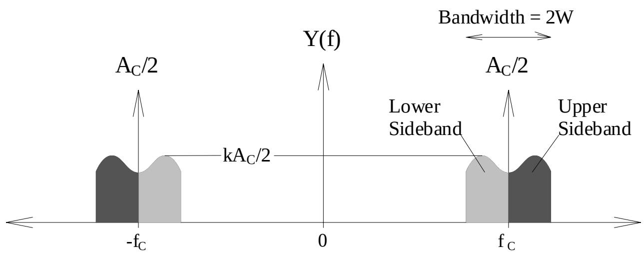


Figure 8.11: DSB-SC modulation in frequency domain

Transmitted power is saved through the suppression of the carrier signal, but the transmission bandwidth is the same as in DSB modulation.

B. Single Sideband (SSB) modulation In single side band (SSB) modulation the transmitted signal consists only the upper sideband or the lower sideband. It is an optimum form of modulation, as it requires the minimum transmitted power and the minimum transmission bandwidth.

8.2.5 Amplitude demodulation

The information signal is recovered through demodulation. There are two commonly used methods for demodulation, each with its own advantages and disadvantages. The process referred to as synchronous demodulation, in which the transmitter and receiver are synchronized in phase, an alternative method referred to as asynchronous demodulation.

a. Synchronous Demodulation For the DSB-SC modulation, the modulating signal x(t) is multiplied by the carrier signal to produce the modulated output signal.

$$y(t) = x(t)\cos(2\pi f_C t) \tag{8.10}$$

The demodulation process involves multiplication of the modulated signal by the same carrier signal that was used in the modulator- hence the name synchronous. We multiply the modulated signal with the carrier signal to obtain the following:

$$w(t) = y(t)\cos(2\pi f_C t) \tag{8.11}$$

Thus:

$$w(t) = x(t)\cos^2(2\pi f_C t) \tag{8.12}$$

Using the trigonometric identity:

$$\cos^2 A = \frac{1}{2}(1 + \cos(2A)) \tag{8.13}$$

we get:

$$x(t) = \frac{1}{2}x(t) + \frac{1}{2}x(t)\cos\left(4\pi f_C t\right) \tag{8.14}$$

Thus, $w(t)$ consists of the sum of two terms, namely one-half the original signal and one-half the original signal modulated with a sinusoidal carrier at twice the original carrier frequency f_c. As discussed previously the band-limited signal has significantly lower frequencies as compared to the carrier signal. Thus, the original signal and the signal that has double the frequency of the carrier signal are sufficiently separated in frequency domain to apply a low pass filter to $w(t)$. On applying a low pass filter the first term, i.e., $1/2x(t)$ will remain and the second term will be eliminated. This is graphically represented in Fig. 8.12.

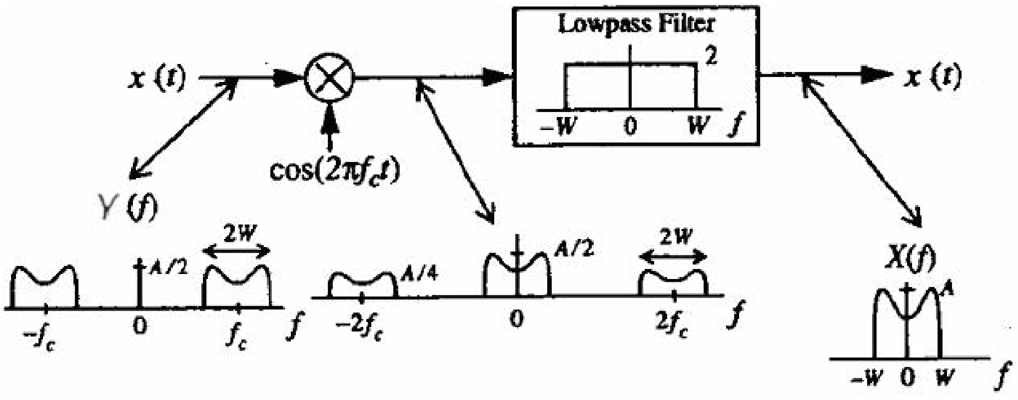


Figure 8.12: Synchronous Demodulation

This process is quite simple,but practically it is difficult to implement due to a the following problem.
For the sinusoidal carrier, let θ_C and φ_C denote the phase of the modulating and the demodulating carriers respectively. The input to the low pass filter is now:

$$w(t) = x(t)\cos\left(2\pi f_C t + \theta_C\right)\cos\left(2\pi f_C t + \varphi_C\right) \tag{8.15}$$

On using the trigonometric identities, we get the following equation

$$w(t) = \frac{1}{2}x(t)\cos\left(\theta_C - \varphi_C\right) + \frac{1}{2}\cos\left(4\pi f_C t + \theta_C + \varphi_C\right) \tag{8.16}$$

Thus, the message signal is multiplied by an amplitude factor of $1/2\cos\left(\theta_C - \varphi_C\right)$. From this equation, we can clearly see that the possibility of recovering the original information signal strongly depends on the phase difference, i.e. on the difference between the θ_C and φ_C. Thus, if $(\theta_C - \varphi_C)$ is zero, or a multiple of π, then the original signal is recovered perfectly. If $(\theta_C - \varphi_C)$ is a multiple of $\pi/2$ then the information is lost completely. More greater importance, the phase relation between the two oscillators must be maintained over time, so that the amplitude factor $\cos\left(\theta_C - \varphi_C\right)$ does not vary.
This requires careful synchronization between the modulator and the demodulator, which is often difficult, particularly when they are geographically separated, as

in a typical communications system. These effects of lack of synchronization are also between the frequencies of the carrier signals used in both the modulator and demodulator.
An alternative method can be used referred to as asynchronous demodulation, where a slight change to the modulation process makes a much simpler detector possible.

b. Asynchronous Demodulation Asynchronous Demodulation avoids the need for synchronization between the modulator and demodulator. Suppose that the modulating signal is always positive by adding a constant (dc offset) to the original signal. The modulating signal thus becomes

$$x(t) = x(t) + C \tag{8.17}$$

This basically means that the signal will be shifted upwards along the y-axis. This can be seen in Fig. 8.13.

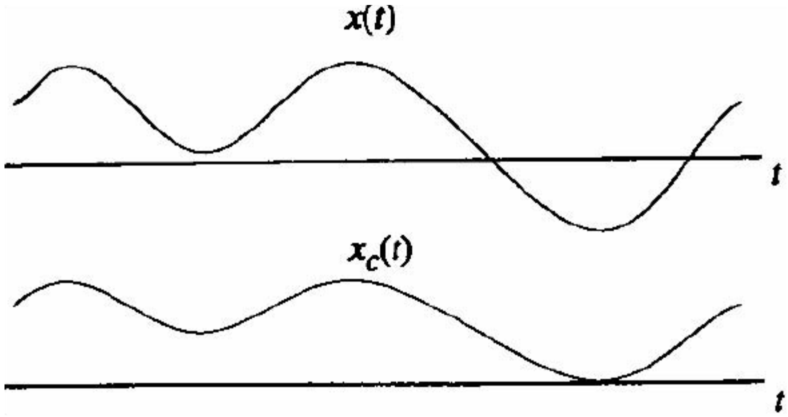


Figure 8.13: Signal shifting

The dc value C must chosen in an appropriate manner to shift the entire signal above the time axis. If we look to the frequency domain, the only effect is the addition of a delta function at zero frequency with the corresponding magnitude. The signal in frequency domain is shown in Fig. 8.14.

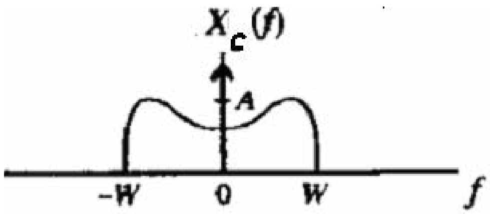


Figure 8.14: Signal shifting effect in frequency domain

The modulated signal is shown in Fig. 8.15.

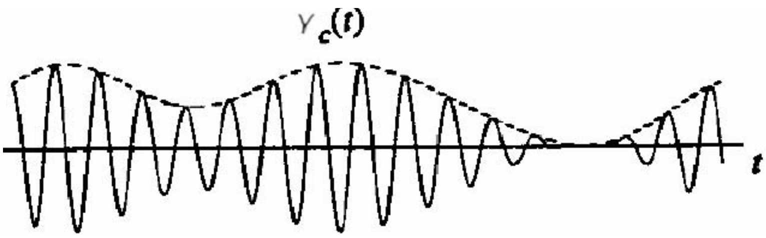


Figure 8.15: Modulated signal in time domain

Thus, the envelope of $x_c(t)$, can be approximately recovered through the use of a circuit that tracks these peaks to extract the envelope. Such a circuit is referred to as an envelope detector or peak detector. A very simple envelope detector can be made by low-pass filtering a full-wave rectified modulated signal as shown in Fig. 8.16.

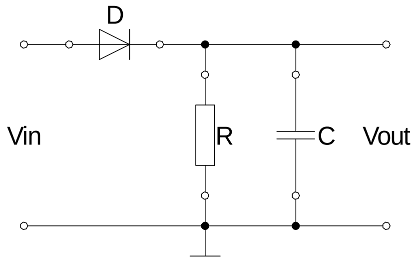


Figure 8.16: Envelope detector circuit

The diode in the circuit allows only the positive part of the cycle to pass and then a capacitor/resistor combination extracts the shape or envelope of the signal. Suitable values of R and C should be used for various carrier and modulation frequencies.
To use the envelope detector for demodulation, we require that C be sufficiently large, so that $x_c(t) = x(t) + C$ is positive. Assuming that A_m denotes the maximum amplitude of $x(t)$. For $x_c(t) = x(t) + C$ is positive, we require that $C > A_m$.

8.2.6 Asynchronous vs. synchronous modulation

The output from the asynchronous modulator has an additional delta component at f_c in the spectrum that is not present by using synchronous modulator. This carrier component in the output represents inefficiency in the amount of power required to transmit the modulated signal.
An advantage for the asynchronous modulation is the ability of a simple envelope detector to follow the input and extract the message signal.
For synchronous modulation complicated demodulator is needed because the oscillator in the demodulator must be synchronized with the oscillator in the modulator, in phase and frequency.

8.2.7 References

1. A. V. Oppenheim, A. S. Willsky, S. H. Nawab, "Signals and Systems," Prentice Hall, Second Edition 1997.

2. Theodore S. Rappaport, Wireless Communications: Principles and Practice (2nd Edition)

8.3 Prelab AM Modulation

8.3.1 Problem 1: Single frequency Amplitude Modulation

1. Derive an expression describing the modulation index m as a function of the modulation envelope, (use A_{min} and A_{max}!).

2. Derive an expression describing the ratio of the total sideband power to the total power $r_P = P_s/P_{tot}$ in the modulated wave delivered to a load resistor. Express the ratio in terms of the modulation index.

3. Calculate the ratio r_P assuming a modulation index of 100%.

4. A Carrier

   $$V_C(t) = 5\cos(20000\pi t)$$

   is AM modulated by a signal

   $$V_m(t) = 2 + \cos(2000\pi t)$$

   Calculate the ratio r_P. How would you change the input signals to maximize the sideband to total power ratio?

8.3.2 Problem 2: Amplitude Demodulation

Use a Matlab script to simulate the demodulation of an AM signal. The sinusoidal carrier frequency exhibits a frequency of 20 KHz and amplitude of 5 V. The sinusoidal modulation signal has a frequency of 500 Hz, and a modulation index of 50%.

1. Plot the modulated signal in time and frequency domain.

2. Design a first and a third order low pass filter (butterworth filter) to demodulate the signal. The cut-off frequencies of the filters should be 1 KHz. Plot the Bode diagram of these filters for a frequency range from 100 Hz to 100 KHz to verify the function.

3. Rectify the AM modulated signal and apply the 1. order low pass filter to the rectified signal. Plot the rectified and the demodulated signal.

4. Change the order of the filter from 1. to 3. Plot the demodulated signal.

5. Why is it better to use a higher order filter for the demodulation of the signal?

6. Attache the the full Matlab script to the prelab.

8.4 Execution AM Modulation

8.4.1 Problem 1: AM modulated Signals in Time Domain

The AM signal is generated by the function generator. Settings of the function generator:

Signal Shape = Sine
Modulation = AM
Carrier frequency = 20 kHz
Carrier Amplitude = 10 V_{PP}
Modulation Frequency = 500 Hz
Modulation index = 50%

1. Connect the function generator to the oscilloscope. Measure the frequency and the amplitude properties of the modulated signal and obtain the modulation index. Take hardcopies.

2. Repeat step 1 by setting the modulation index to 70%. Take hardcopies.

3. Adjust the modulation index to be 120% and observe the effect on the AM signal. Take a hardcopy.

8.4.2 Problem 2: AM Modulated Signals in Frequency Domain

Use the same setup as before. Set the modulation index at the function generator to 70%. At the oscilloscope display the amplitude modulated signal in frequency domain. (FFT!). Use the cursors to measure the magnitudes and the frequencies. Take hardcopies!

8.4.3 Problem 3: Demodulation of a message signal

Use the following circuit:

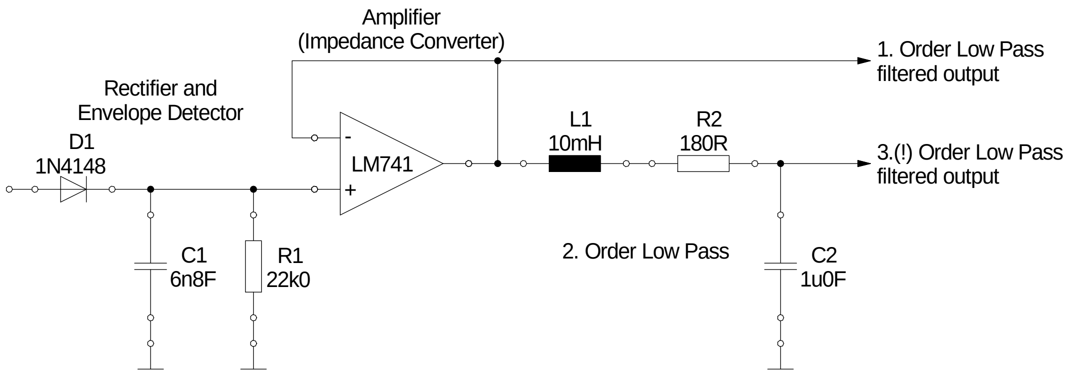


Fig. 1: Demodulation circuit

Do not forget that an OP-Amp needs a power supply. Below is the pinout and the necessary circuit.

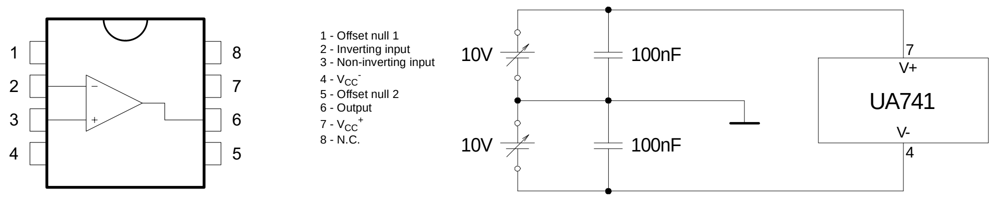


Fig. 2: LM741 pinout and supply circuit

Connect the function generator to the input of the demodulating circuit. Use the following settings:

Signal Shape = Sine
Modulation = AM
Carrier frequency = 20 KHz
Carrier Amplitude = 10 V_{PP}
Modulation Frequency = 500 Hz
Modulation index = 50%

1. Display the AM modulated signal together with the 1. order filter output. Take a hardcopy.

2. Display the AM modulated signal together with the 3. order filter output. Take a hardcopy.

3. Measure the amplitude of the demodulated signal at the 3. order output.

4. Take a FFT of the signal at the 3. order filer output. Check if there is still a 20kHz component. Take hardcopies.

8.5 Evaluation AM modulation

8.5.1 Problem 1: AM modulated Signals in Time Domain

1. What is the relation between the modulation index and the relative magnitudes of the frequency components?

2. Calculated the modulation index using the measurements! Compare to the index you have used to generate the AM signal.

3. Discuss the effect and the disadvantages of using a modulation index greater than 100

8.5.2 Problem 2: AM Modulated Signals in Frequency Domain

1. How does the spectrum look like in theory? Compare to the experiment!

2. Does the function generator generate a DSB or DSB-SC AM signal?

3. How does changing the carrier frequency affect the AM spectrum?

4. How does changing the message frequency affect the AM spectrum?

5. Determine the modulation index m using the measured values.

8.5.3 Problem 3: Demodulation of a message signal

1. Compare the 1. and 3. order filter output signal with the message signal.
2. Compare the measured signals with the MatLab results. What are the differences between simulation and measurement?

9. Theory 6 : FM Modulation

9.1 Frequency Modulation

9.1.1 Objective

This is the second part of the modulation experiment. The influence of frequency modulation parameters on the bandwidth will be explained. Practically, the oscilloscope will be used as a spectrum analyzer to demonstrate the impact of the frequency modulation parameters on the frequency domain. Furthermore, you will build a simple demodulation circuit consisting of a slope detector.

9.1.2 Single frequency FM modulation

In frequency modulation, the amplitude of the modulated carrier signal is kept constant while its frequency is varied by the modulating message signal. The basic idea of frequency modulation is shown in Fig. 9.1. The carrier frequency is controlled at each instant by the voltage of the modulating signal. The frequency of the modulated signal is increased if the input signal is positive, whereas the frequency is reduced if the input signal is negative.
The general definition of frequency modulated signal $S_{FM}(t)$ is given by the formula:

$$S_{FM}(t) = A_C \cos\left(2\pi f_C t + \theta(t)\right) = A_C \cos\left(2\pi f_C t + 2\pi K_f \int_0^t m(\tau)d\tau\right) \tag{9.1}$$

where,

$m(\tau)$	is the modulating signal.
A_C	is the amplitude of the carrier.
f_C	is the carrier frequency.
K_f	is the frequency deviation constant measured in Hz/V.

For the case of a single frequency sinusoidal modulating signal, $m(\tau) = A_m \cos(2\pi f_m \tau)$ the frequency modulated signal $S_{FM}(t)$ will be expressed as:

$$S_{FM}(t) = A_C \cos\left(2\pi f_c t + \frac{K_f A_m}{f_m} \sin(2\pi f_m t)\right) \tag{9.2}$$

The factor $K_f A_m$ is called the frequency deviation. It is defined as the maximum frequency shift away from f_c. The frequency modulation index β_f, is expressed as:

$$\beta_f = \frac{K_f A_m}{f_m} = \frac{\Delta f}{f_m} \tag{9.3}$$

So, the $S_{FM}(t)$ signal can be represented as:

$$S_{FM}(t) = A_C \cos\left(2\pi f_C t + \beta_f \sin(2\pi f_m t)\right) \tag{9.4}$$

Frequency modulated signals are classified into two categories based on the value of β_f.

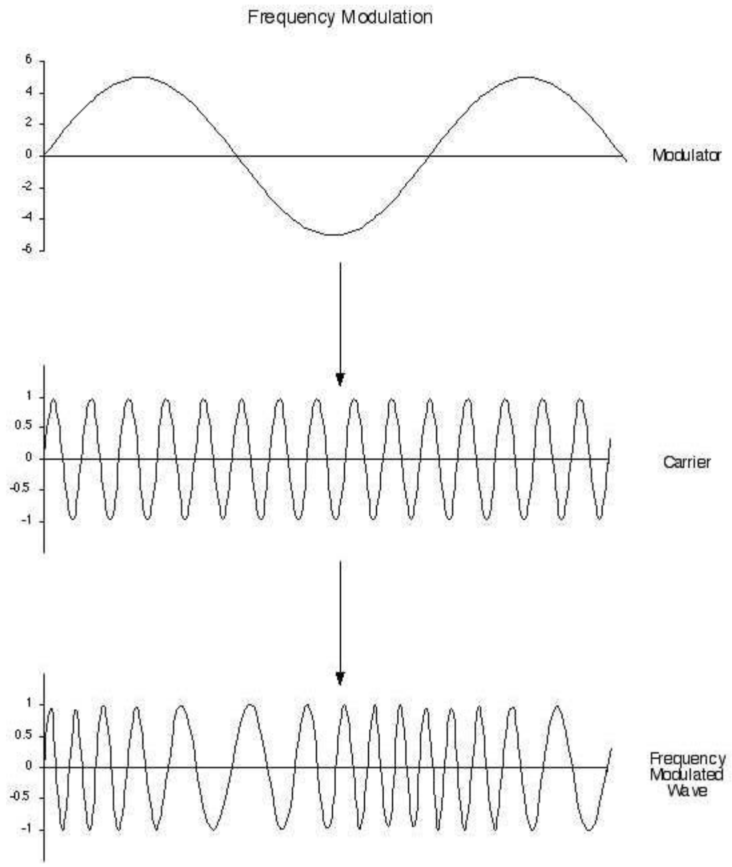


Figure 9.1: frequency modulation process

a. Narrow Band Frequency Modulation (NBFM) For small values of the frequency modulation index ($\beta_f << 1$), we have Narrow Band Frequency Modulation (NBFM). In this case, the frequency modulated signal $S_{FM}(t)$ becomes:

$$S_{FM}(t) = A_C \cos(2\pi f_C t) - A_C \beta_f \sin(2\pi f_C t) \sin(2\pi f_m t) \tag{9.5}$$

The derivation of Eq. (9.5) can be found in reference [2].

b. Wide Band Frequency Modulation (WBFM) As the modulation index increases, the signal occupies more bandwidth. In this case the modulation scheme is called Wide Band Frequency Modulation (WBFM). Therefore, for a single frequency sinusoidal modulating signal the frequency modulated signal could be written in the form:

$$S_{FM}(t) = A_C \sum_{n=-\infty}^{\infty} J_n(\beta) \cos(2\pi(f_C + n f_m)t) \tag{9.6}$$

The derivation of Eq. (9.6) can be found in reference [2].

9.1.3 FM spectrum

As with amplitude modulation, the modulation process causes sidebands to be produced at frequencies above and below the carrier. However, for a frequency modulation based system, there are a lot more, all spaced at multiples of fm from the carrier frequency f_C. As a result, the bandwidth needed to accommodate a frequency modulated signal is considerably larger than that for amplitude modulated signal having the same modulating frequency.

Fig. 9.2 shows the spectrum of a frequency modulated signal for various values of the modulation index β_f. The modulating signal in these examples is a single frequency sinusoidal signal.

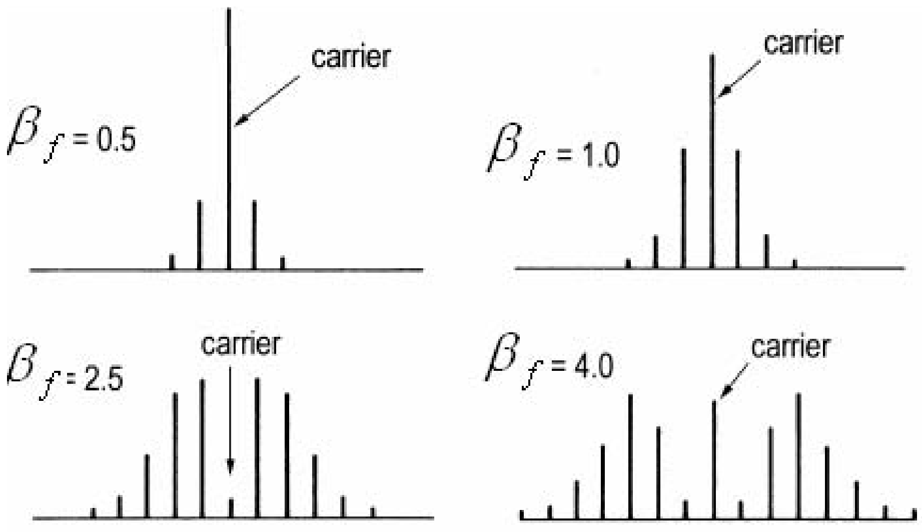


Figure 9.2: Frequency modulation index effect

When a sinusoidal signal such as $m(t) = A_m \cos(2\pi f_m t)$ is used, the spectrum contains a carrier component and many number of sidebands located on either side of the carrier frequency, spread at integer multiples of the modulating frequency f_m ($f_c \pm n f_m$), for all positive n ($n = 0$ is the carrier frequency component).

The only exception is at a very low frequency modulation index, most of the information is contained within the range of the first upper and lower sidebands, which makes the total bandwidth sufficient for transmission about the same as for amplitude modulation based system, that is $2f_m$. With larger frequency modulation indexes, the number of sidebands increases and we obtain larger bandwidth.

Fortunately, something else is happening which keeps the total bandwidth reasonable. To get a large frequency modulation index, we need a large frequency deviation but a small modulation frequency, according to the modulation index definition. The modulation frequency; however, determines the spacing between sidebands. So, at high modulation index, we may have many sidebands, but they will be closely spaced, so the total occupied bandwidth will not be much larger. This is demonstrated in Fig. 9.3.

Theoretically, the bandwidth of a frequency modulated carrier is infinite. In practice, however, we find that the frequency modulated signal is effectively limited to a finite number of significant sideband frequencies within an approximate bandwidth, B_T, given by Carson's rule.

Carson bandwidth rule is a rule defining the approximate bandwidth requirements of communications system components for a carrier signal that is frequency modulated.

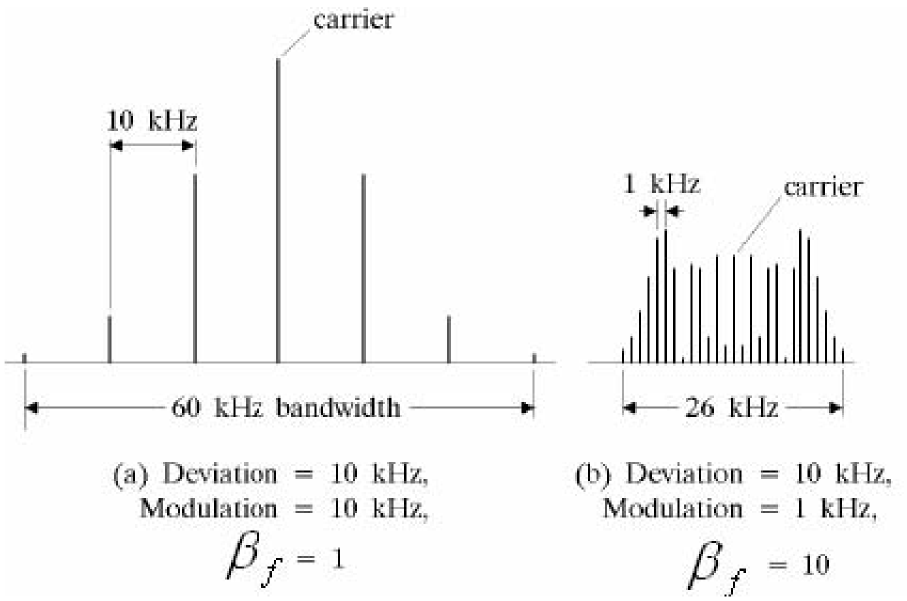


Figure 9.3: Same deviation, but different modulation index

In case of a single frequency modulation, the empirical Carson's rule is given by

$$B_T \cong 2f_m(\beta_f + 1) \tag{9.7}$$

For more practical case, an arbitrary modulating signal $m(t)$ is considered and its highest frequency component is denoted by W. Then, replacing β by D and replacing f_m with W in Eq. (9.7) we get

$$B_T \cong 2W(D + 1) \tag{9.8}$$

D is called the deviation ratio and it is defines as the ratio of the frequency deviation Δf, which corresponds to the maximum possible amplitude of the modulating signal $m(t)$, to the highest modulation frequency W.
The maximum frequency deviation depends on the maximum amplitude of the modulating signal and the sensitivity of the modulator. The sensitivity of the modulator is called the frequency-deviation constant, K_f. Thus, D is given by the following formula:

$$D = \frac{K_f * \max|m(t)|}{W} \tag{9.9}$$

where, W is the highest modulation frequency, $m(t)$ is the message signal and K_f is the frequency-deviation constant.
The deviation ratio D plays the same role for arbitrary modulation that the modulation index β_f plays for the case of a single sinusoidal modulation. From a practical viewpoint, Carson's rule somewhat underestimated the bandwidth.

9.1.4 Single frequency FM demodulation

There are many ways to recover the original information from a frequency modulated signal. The frequency demodulator should produce an output voltage with

instantaneous amplitude that is directly proportional to the instantaneous frequency of the input frequency modulated signal. Thus, a frequency-to-amplitude converter circuit is a frequency demodulator.
Various techniques such as slope detection, zero-crossing detection, phase locked discrimination and quadrature detection are used to demodulate the frequency modulated signal.
Universally, demodulators use a phase-locked loop (PLL), an extremely useful circuit that finds its way into all sorts of electronic systems. Briefly, it consists of an oscillator whose frequency can be varied by means of a voltage (that is, a voltage controlled oscillator or VCO), and a feedback loop, which results in the frequency of the oscillator being locked to the frequency of the incoming signal. In the process the circuit produces a voltage, which is proportional to the variation in the signal frequency.
In this experiment we will use a simple slope detector do demodulate an FM signal. A slope detector is essentially a resonator (tank) circuit which is tuned to a frequency either slightly above or below the fm carrier frequency.

9.1.5 Frequency Modulation vs. Amplitude Modulation

The main advantages of using frequency modulation over amplitude modulation are:

1. Frequency modulated signals have better noise immunity than amplitude modulated signals since signals are represented as frequency variations rather than amplitude variations. Frequency modulated signals are less susceptible to atmospheric and impulse noise, which tend to cause rapid fluctuations in the amplitude of the received radio signal.

2. In amplitude modulation, the peak amplitude of the envelope of the carrier is directly dependent on the amplitude of the modulating signal, which requires a large dynamic range of the transmitter and the receiver. For frequency modulation, the envelope carrier is constant. Consequently, the FM transmitter can always operate at peak power.

There are also some serious disadvantages in frequency modulation.

1. Frequency modulated signal typically requires larger bandwidth.

2. Frequency modulation based systems are much more complicated to analyze and build than amplitude modulation based systems.

9.2 References

1. A. V. Oppenheim, A. S. Willsky, S. H. Nawab, "Signals and Systems," Prentice Hall, Second Edition 1997.

2. Theodore S. Rappaport, Wireless Communications: Principles and Practice (2nd Edition) Signals and Systems Lab, Fall 2010, Jacobs University 19

9.3 Prelab FM Modulation

9.3.1 Problem 1: Frequency Modulator

A sinusoidal modulation signal,

$$m(t) = 4\cos(8000\pi t)$$

is applied to an FM modulator that has a frequency deviation constant $K_f = 10\,\text{kHz/V}$. Compute the frequency deviation and the frequency modulation index.

9.3.2 Problem 2: FM signal in the frequency domain

Plot a frequency modulated signal in the frequency domain. The signal exhibits 2.5 V peak carrier amplitude, kHz carrier frequency, 5 kHz modulation frequency. Vary β_f between 0.2, 1, and 2. Display the magnitudes in dB_{rms}! Calculate the bandwidth using Carlsons rule. Tabulate the peak magnitudes inside the bandwidth from the three plots.

9.3.3 Problem 3: Frequency demodulation

In Fig. 9.4 a simple slope detector circuit is shown . The circuit can be used as a FM demodulation circuit. Provide a bode diagram and explain the operation principle of the circuit.
Hint: The resonance frequency of the circuit is set higher than the carrier frequency fc of the FM signal.

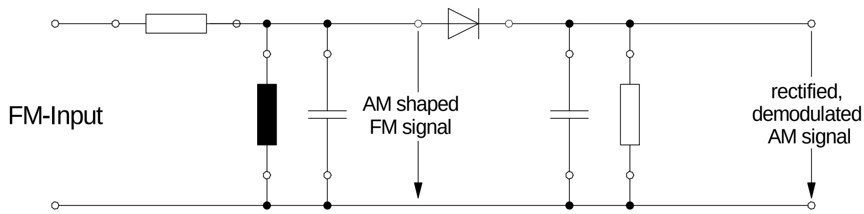


Figure 9.4: Schematic circuit of a slope detector.

Part III

Additional Information

A. Appendix

A.1 Hardcopy from oscilloscope screen

For the documentation and evaluation of an experiment it is useful to grab pictures from the oscilloscope screen. This is possible by using a printer, a computer, or an USB stick (dependant on the oscilloscope).

A.1.1 Tektronix TBS 1072B-EDU

The TBS series oscilloscope has an USB interface. Insert a USB stick into the plug on the front panel. For getting a Hardcopy press the button with the disk symbol. After you took all needed pictures do not forget to save the images from the stick to your computer.

A.1.2 Tektronix TDS Series

The TDS series oscilloscope have no USB interface. Getting hardcopies is done by an interface which connects the serial port of the oscilloscope to the network. A Java applet controls the oscilloscope and downloads the bitmap. The necessary prerequisites and the procedure how to do this is described in the following paragraphs.

Oscilloscope Settings
The oscilloscope is configured by the interface before requesting a hardcopy. The users only have to take care about the RS232 settings. In general we use the defaults. Only the baudrate is set to 19200.

Use following sequence to configure the RS232 interface of the oscilloscope:

Button	$\boxed{\text{UTILITY}}$							
	Button	$\boxed{\text{Options}}$						
		Button	$\boxed{\text{RS232 Set-Up}}$					
			Button	$\boxed{\text{Set to defaults}}$				
			Menu	$\boxed{\text{Baud}}$	set to	$\boxed{\text{19200}}$		

Computer Preparations
You need a Web-Browser which is enabled to execute Java applets. So you have to assure that executing Java (!! not Java Script !!) is allowed to run in your preferred browser. Second you need the Java Runtime Environment from Oracle. Dependent on your browser you have to download and install it manually (Windows IE) or the browser will find the necessary plugin and guide you through the installation when you use the convertor the first time (Firefox). In both cases the installation has to be done as 'Administrator'.

Ethernet to Serial Convertor Settings

The convertor is switched on when the workbench is powered with the key switch. There is nothing to configure! In case there is a problem using the convertor try resetting it with the reset switch in the front panel!

Make a Hardcopy

1. Connect the Oscilloscope to the Ethernet to serial convertor. Use the provided RS232 cables in the lab.

2. The main power switch of the workbench and the oscilloscope needs to be switched on.

3. Open the web browser. Start the applet in the convertor by using the following link:

   http://xxx.xxx.xxx.xxx/osci.html

   Instead of **xxx.xxx.xxx.xxx** insert the IP number of your workbench. (That is the number on the label you find at the convertor insert of the bench).

4. The applet will start. If you work the first time at this bench a warning about the certificate will pop up. Confirm that you trust the supplier of this certificate. (If you check you will find that 'Uwe Pagel' is the supplier!)

5. The applet shows up. You will see a status message in the upper part of the screen. It includes the status of the TCP/IP connection and the ID string of the oscilloscope.

6. The whole setup is now ready for usage. To get an image of the oscilloscope screen push the **'Start Hardcopy'** button in the applet window. In the status screen you will see the progress of the operation. If all data is transferred the image becomes visible. To store the bitmap use the button **'Save Image'**.

7. If it is necessary to get the raw data from the screen, then use the **'Capture Data from ->'** button. But first select the data source in the window beside the knob. Also here you will see the progress of the operation. If the data is transferred it is shown in the data window. To save the data use the **'Save Data'** button.

In case of problems!

Here it is assumed that everything is connected and powered and that the right link is used!!!

- The applet didn't show up. There is only a curious message somewhere on the screen!
  - The browser isn't able to execute an applet. Get and install the 'Java Runtime Environment' from Oracle (see above).
- The applet window is visible but you get an TCP/IP error.

- Somebody else is already talking to the interface box! It is **not** possible to call the applet several times.
- There is a problem with the interface. Use the reset button at the front, wait about 10 seconds and use the reload button of the browser.

- The applet window is visible, also the TCP/IP status is o.k. but instead of the oscilloscopes ID there is an error message that it is not on or connected.
  - Check if the oscilloscope is on and connected!
  - If it is on and connected check if the baudrate of the oscilloscope is set to 19200. If the baudrate is wrong correct it to 19200 and reload the applet.
  - Sometimes the controller in the oscilloscope fails! So switch the oscilloscope off and on and try to connect again.
- You pushed the **'Start Hardcopy'** button, the applet started but getting data is slow (even stops after a while).
  - The serial port of the oscilloscope **-is-** slow if the 'Measure' function is in use! Press the 'RUN/STOP' button at the oscilloscope and the download becomes faster.
  - If the download is slow and even interrupted after a while you might be connected to the network via a wireless link. If too much people are connected the bandwidth is to small. Use the network plug in the workbench instead.
- Failures during operation.
  - Use the reset button right at the front , wait about 10 seconds and use the reload button of the browser.

A.2 Books and other Tools

A.2.1 Book

- Sarma
- Floyd

A.2.2 Programs

- LTSpice
- Matlab
- Octave
- KiCad