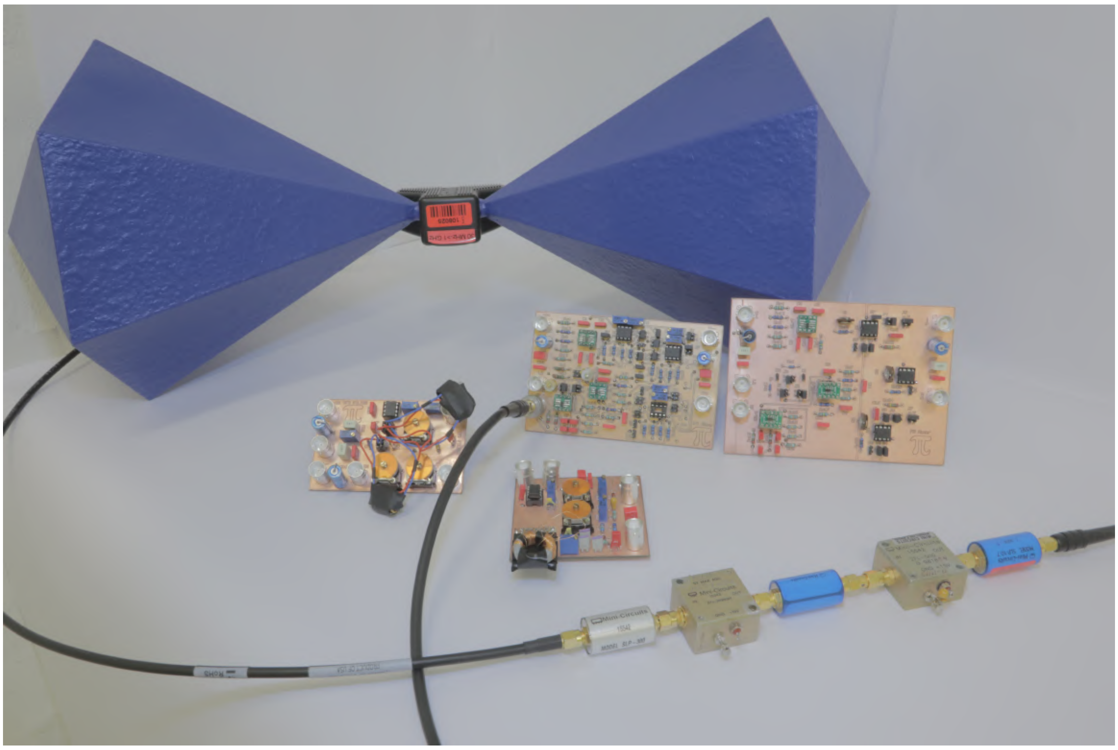
Frequency Modulation Radio

- P3 -



Project Report

Group 314

Aalborg University
Department of Electronic Systems
Fredrik Bajers Vej 7B
DK-9220 Aalborg

This report was written in LaTeX and has been shared online in-between the group members using ShareLatex licensed to all students at Aalborg University.

AALBORG UNIVERSITY
STUDENT REPORT

Department of Electronic Systems
Fredrik Bajers Vej 7
DK-9220 Aalborg Ø
http://es.aau.dk

Title:
Frequency Modulation Radio

Theme:
Analogue Circuit Design

Project Period:
Fall Semester 2017

Project Group:
Group 314

Participant(s):
Frederik Skyt Dæncker Rasmussen
Mads Jacob Riisager
Magnus Bøgh Borregaard Christensen
Max Væhrens
Peter Kjær Fisker
Peter Hammerich Graversen

Supervisor(s):
Jimmy Jessen Nielsen
René Brandborg Sørensen

Copies: 1

Page Numbers: 214

Date of Completion:
December 20, 2017

Abstract:

The purpose of this report is to document the design and construction of a superheterodyne radio receiver and a class A amplifier. The report will go through specific standards for radio receivers and HiFi audio amplifiers, which define the requirements for both.
The superheterodyne radio receiver was designed based on the theory in the preliminary study and the radio receiver requirements.
The radio receiver was split into modules, which were designed, constructed and tested individually. The tests of the individual modules of the radio receiver showed that the majority of the requirements were met. The tests of the class A amplifier showed that it met none of the requirements, and additional research later uncovered several flaws with the design. Therefore a suggestion for a better design was made, but did not get implemented due to lack of time.
Additionally due to time restraints the class A amplifier was not available for testing during the acceptance test. Thus the acceptance test was carried out with the implemented radio receiver alone.
The acceptance test showed that the different modules of the radio receiver were able to be integrated successfully into a complete functioning radio. It was therefore concluded that the design of the radio receiver was a success but the design of a class A amplifier was not.

Preface

This report is about a project which aims to create an FM radio and a high fidelity amplifier. This project takes place on the third semester of the Electronics and IT bachelor at Aalborg University, during the Fall semester of 2017. The main theme of the third semester is *Analogue Circuits and Systems*.
The author of this report is a project group made up of students, and the students are all listed on the title page. During the project two supervisors have guided the group.

The first part of this report is the preliminary studies, and they form the foundation for most of the theory in the report.
Next is the design and implementation part, which seeks to answer the problem statement of the project. This includes chapters such as requirements and limitations of the product, design, production, and test of each of the sub-systems. This part is concluded with the acceptance test of the radio receiver and amplifier.
Finally, the third part is the conclusion of the report. First is the conclusion on the whole project, and this is accompanied by a final chapter putting the work process during project into perspective.
Due to the time constraints of this project, not all appendices have been completed within the project period. However, the missing or lacking appendices will be updated via Github https://github.com/bifr05t/P3_radio_receiver.

First off, the group would like to thank Aalborg University and the Electronics Institute for setting a good study environment for Electronics and IT students, and further allowing ambitious projects to come to life. We would like to thank the following people in no particular order:
The two supervisors; René Brandborg Sørensen and Jimmy Jessen Nielsen for supervision and thorough feedback on the report.
Hans Ebert, for his willingness to spend several hours with technical supervision despite not being our supervisor.
Kenneth Knirke, for his help obtaining equipment and components for the project, and additionally lending support with measurement equipment in the electronics lab.
Ben Krøyer, for his help with designing and milling Printed Circuit Boards.
Sofus Birkedal Nielsen, for his feedback on amplifier designs.

Contents

Part I

Preliminary Study

Chapter 1

Introduction

This report aims to document the design and development of a HiFi sound system consisting of a radio receiver and a HiFi audio amplifier. The focus of the documentation and the efforts is the radio receiver, designed as a super heterodyne receiver. The superheterodyne radio receiver can be split into different subsystems, as shown on figure 1.1.

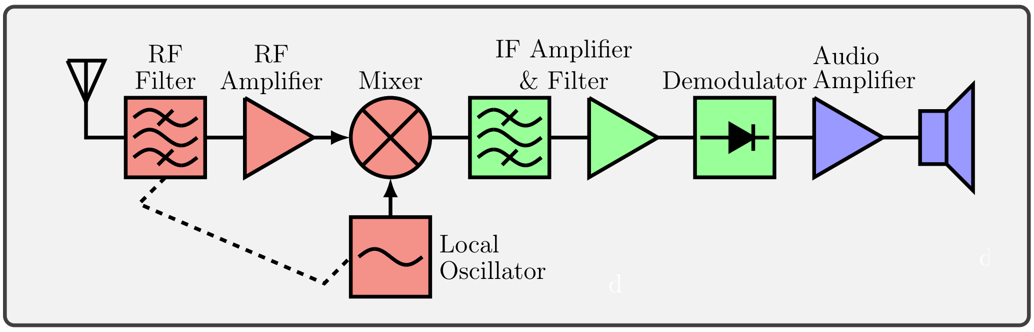


Figure 1.1: Block diagram of a typical superheterodyne receiver.

To be able to design and develop such a system, a preliminary study is conducted to answer some of the questions that arise when trying to understand the different subsystems. The questions sought answered in this part of the report are as follows:

- What are audio signals?
- What are radio frequency (RF)-signals and how do they work?
- How is it possible to extract audio signals from the transmitted RF-signal?
- What is an audio amplifier and how does it work?

To answer the first question, the general principle of sound and how humans perceive sound as audio is studied.

1.1 Sound

Sound is a form of mechanical wave, meaning that it is an oscillation that travels through a medium such as a gas, liquid or solid. These vibrations can then be sensed by the human auditory system. The waves are amplified with a directional information by the pinna, then modulated by the ossicles in the middle ear and then transported to the auditory vestibular nerve in the inner ear. Here the nerves transfer the information to the temporal lobe, where it is perceived as sound. The presence of two independent hearing sensors on humans add additional signal processing capability allowing e.g. to identify the spatial location of a sound source. This is outside the scope of the contents of this report.

More specifically, sound falls into the longitudinal wave category. In longitudinal waves the oscillations are parallel to the direction of the energy transfer. In other words, the propagation of a sound wave is parallel to the displacement of the medium.[1]

Regarding sound waves, this displacement of matter is observed as the vibration of particles. This effect is illustrated in figure 1.2.

Figure 1.2: Illustration of vibrating particles.

As seen on the illustration 1.2, there are regions where the particles are close together, these are called compressions. There are also regions where the particles are further apart, these are called rarefactions. The distance between two compressions or two rarefactions is called the wavelength (denoted by λ). The amplitude is the furthest point the wave vibrate from the center point. An illustration of this can be seen on figure 1.3.

A sound wave with a small amplitude is perceived as quiet, whereas a sound wave with a large amplitude will be perceived as being louder. The frequency is defined by the total number of waves per second. This determines the pitch of sound; thus, a high frequency wave is perceived as a high-pitched sound and vice versa for low frequency.

To see an illustration of the effects above see figure 1.3.

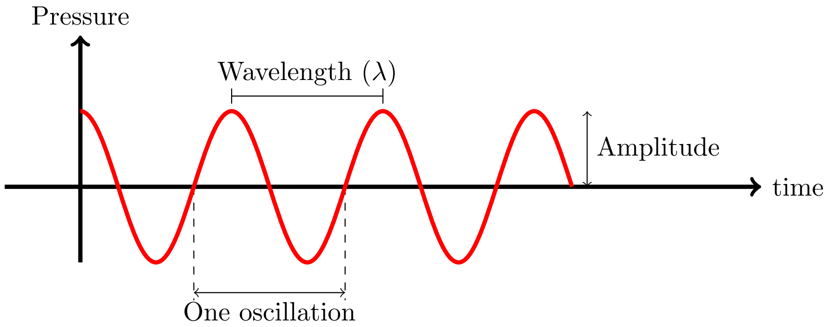


Figure 1.3: Sound wave properties.

1.1.1 Human hearing range

Having defined the properties of sound, it is now prudent to take a brief look at the human hearing range. The human ear is sensitive only to vibrations in what is known as the audible range. The audible range is sound waves with frequencies between 20 Hz and 20 kHz, however this may vary from person to person depending on factors such as age, sex etc. Sound below 20 Hz is classified as infrasound while frequencies above 20 kHz fall into the ultrasound category.[2] In addition to this, the intensity (loudness) of sound is measured in decibels (dB).

Chapter 2

Elementary Radio Theory

This chapter deals with the fundamental radio theory employed in radio communication, and more specifically Frequency Modulation (FM) communication. To successfully transmit and receive a radio signal it is required that a transmitting device includes a modulator to modulate the information bearing signal, and that the receiving device has a demodulator to recover the information bearing signal.
Furthermore, it is also necessary for the receiving device to have a receiver that can extract the desired radio frequencies and filter the remaining frequencies received from the antenna. In addition, both devices also need an antenna to act as an interface between radio waves (electromagnetic waves) and the electric circuitry. The way these systems interact to transmit and receive signals is illustrated in figure 2.1.

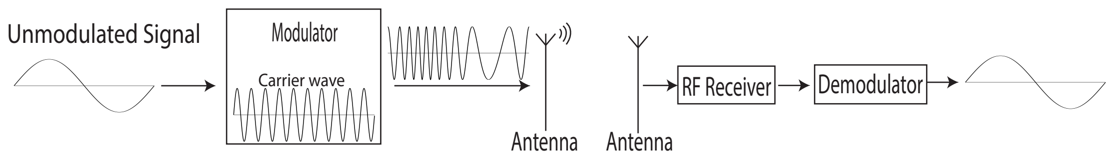


Figure 2.1: A simplified illustration of how a signal (from left to right) is frequency modulated, transmitted, and then finally received and demodulated back to the source signal.

This chapter will describe the above mentioned (and more) properties and techniques employed in radio communication. First the basics of modulation, noise, and transmission will be explored. The chapter will then go on to describe the specifics of FM theory, as well as some of the methods used for FM reception. It should also be noted that the following theory will primarily focus on the transmission of analogue baseband signals, as the transmissions of digital signals is outside the scope of this report. The reader should also be aware that unless otherwise specified, it should be assumed that all presented information is derived from the second edition of the book "Analog Communication" by Simon Haykin. [3]

2.1 A Brief History of Radio Communication

Throughout the twentieth century, a variety of wireless communication technologies has been invented.

Since its inception it has made communication between different regions and continents much more efficient and accessible. Seeing how wireless communication travels in the aether it does not require the same amount of cost or establish to setup as wired communication does.

Wireless communication was first demonstrated in 1895, when Guglielmo Marconi transmitted and received a signal over a distance of two kilometers. Two years later Marconi had achieved almost 20 kilometers. [4]
Four years later in 1901, Marconi received a radio signal in Newfoundland, eastern Canada, which were transmitted from Cornwall, England. This achievement was accomplished by transmitting the Morse code for the letter 's' from an antenna in Cornwall and elevating 150 meters of antenna wire into the air with a balloon in Newfoundland. [5]
Marconi's experiment proved that it was possible to transmit signals on a global scale. This achievement helped shape communication in the twentieth century, where the wireless telegraph helped pave the way in the early days of the century allowing for expansion of communications at a lower cost than the cable telegraph.

Later in 1906 Reginald Fessenden performed the first radio broadcast, using a system implementing amplitude modulation (AM). Following in 1918, Edwin H. Armstrong invents the superheterodyne radio receiver which would later become the basis for all modern radios. 15 years later Edwin would revolutionize the industry, again, by inventing Frequency Modulation (FM), thus allowing the transmission of high quality audio broadcasts. [6, p. 1-2]
From the late 1880s and onward into the twentieth century, land-line telephones had been in use along with the switchboard technology. In 1977, a cellular network was for the first time established in Chicago by the company AT&T. The cellular network, based on radio waves, successfully made telecommunication wireless and later made its way into a hand-held format. By 1984, Japan was the first country with a nationwide coverage of the first generation of cellular network, namely the NTT network. [7, p. 209-217]

2.2 Frequency Spectrum and Modulation

Radio communication refers to the use of radio waves[1] with the purpose of transmitting information. However, it is rarely practical to directly transmit the raw information in the form of an electrical signal. The primary reason being that multiple simultaneous radio transmissions, transmitting signals of approximately equal frequency, are not possible as it would be unfeasible to differentiate the transmissions from one another. For example, a common application of radio is the broadcasting of audio (i.e. frequencies ranging from 20 Hz to 20 kHz). If the source analogue signal were to be broadcasted,

[1] **Radio Waves:** Electromagnetic waves with a frequency ranging from 3 Hz to 3 THz

then, for a given area, it would only be possible to have one ongoing broadcast at time, since any additional audio broadcasts would cause interference. To overcome this restriction the source signal must first be modulated on to a carrier wave before it can be transmitted.

Modulating signals on to different carriers allows multiple signals, regardless of their frequencies, to be transmitted concurrently. Modulation can be defined as "The process by which some characteristic of a carrier wave is varied in accordance with an information-bearing signal" [3]. Where the information-bearing signal refers to the signal that is to be transmitted (such as audio), and the carrier wave is a sinusoidal waveform that "carries" the information-bearing signal. The carrier wave dictates the frequency of the transmission, and by changing the frequency of the wave it becomes possible to have multiple concurrent transmissions - even if the information-bearing signals are of similar frequencies. Figure 2.2 portrays how signals can be separated from one another by using different carrier frequencies:

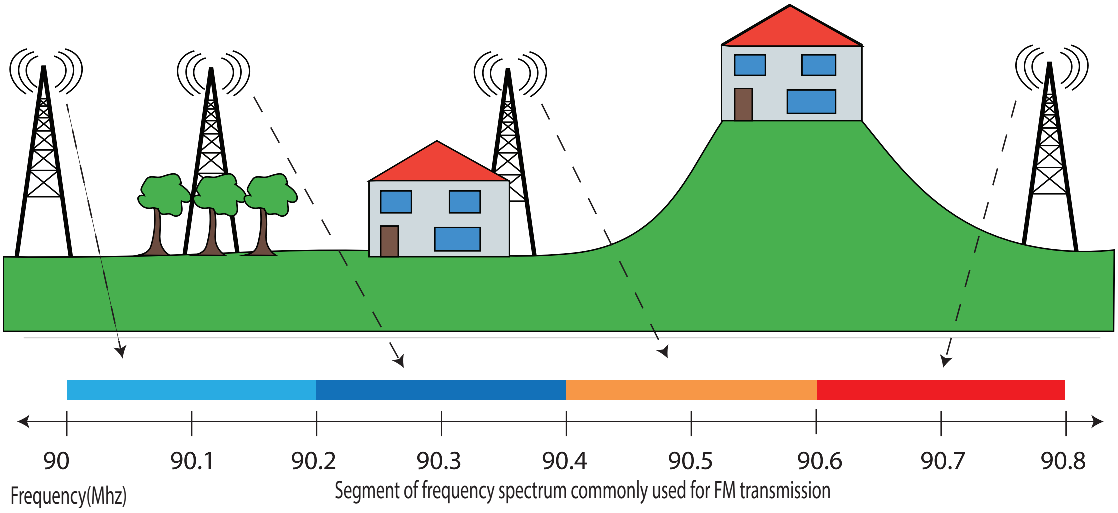


Figure 2.2: Depicts how subcarrier modulation allows simultaneous transmission, by varying the frequency of the transmission

For modulating analogue signals, the two most widely adopted methods are amplitude modulation (AM) and frequency modulation (FM). In AM the instantaneous amplitude of the carrier wave is varied with respect to the instantaneous amplitude of the information-bearing signal. The modulated wave can mathematically be described as:

$$s(t) = A_c[1 + k_a m(t)] \cos(2\pi f_c t) \qquad [\mathrm{V}] \qquad (2.1)$$

where:

$s(t)$ is the AM wave as a function of t	[V]
$m(t)$ is the information signal	[V]
t is time	[s]
A_c is carrier amplitude	[V]
f_c is the carrier frequency	[Hz]
k_a is the amplitude sensitivity	[V^{-1}]

AM is relatively simple to implement, however it suffers from very high noise sensitivity (i.e. it is easy to distort the signal), and thus it has largely been replaced by other modulation methods, such as FM.

FM works by changing the instantaneous frequency of the carrier wave with respect to the information bearing signal. FM can mathematically be described as:

$$s(t) = A_c \cos[2\pi f_c t + 2\pi k_f \int_0^t m(\tau) d\tau)] \qquad [\mathrm{V}] \qquad (2.2)$$

where:

$s(t)$ is the FM wave as a function of t	[V]
$m(\tau)$ is the information signal	[V]
t is time	[s]
A_c is carrier amplitude	[V]
f_c is the carrier frequency	[Hz]
k_f is the frequency sensitivity	[$\mathrm{Hz\,V}^{-1}$]
τ is an integration variable	[$\cdot$]

Figure 2.3 illustrates how the information-bearing signal is carried by the carrier wave in both amplitude and frequency modulation:

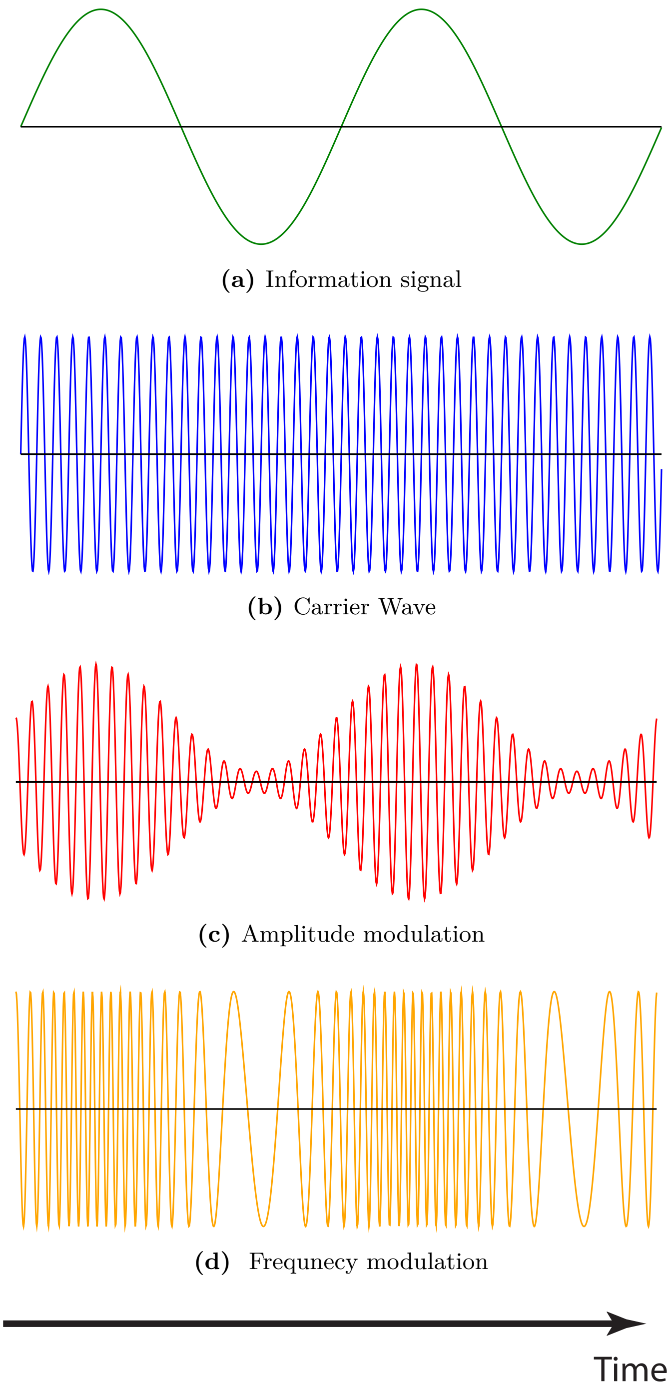
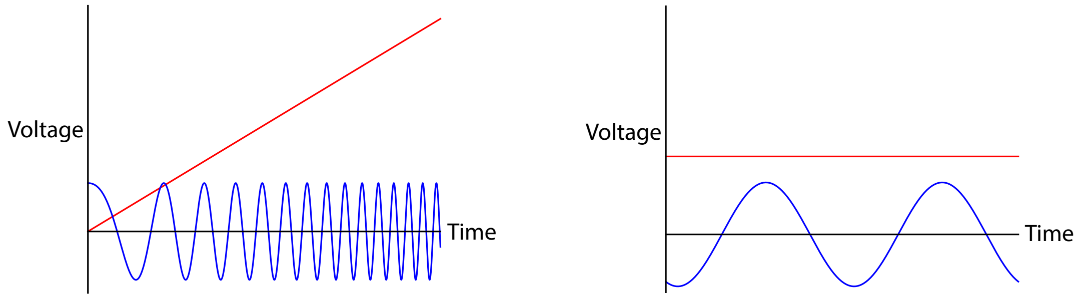


Figure 2.3: Depiction of how an unmodulated signal,(a), could be modulated onto a carrier wave,(b), to produce an AM,(c), or an FM,(d), waveform

Figure 2.4 further illustrates how two different information-bearing signals are frequency modulated onto carrier waves:



(a) **(b)**

Figure 2.4: The red signal is modulated onto a carrier wave to produce a modulated signal (shown in blue)

2.3 Noise

In communication systems, noise is defined as "any unwanted signal interfering with or distorting the signal being communicated". In electrical systems the unwanted signal manifests itself as electrical fluctuations. Noise originates from a variety of manmade and natural sources, such as from the atmosphere, distant stars (cosmic noise), or even sub-atomic interactions within circuits. With regards to communication systems, the biggest source of noise originates from other nearby communication devices transmitting at similar frequencies compared to the desired to transmission. However, radio designers are rarely able to affect the amount of external noise, and must therefore instead focus their attention on minimizing internal noise caused by the physical properties of electronic circuits.

Noise is by its very nature a randomly fluctuating quantity and can therefore not be predicted. Instead noise is modelled as a stochastic process, where the noise randomly changes with respect to time. The mathematical details regarding stochastic processes are well beyond the scope of this report, however to gain a basic understanding of noise, a few basic concepts of probability theory must be understood.

For most noise models, it can generally be assumed that the noise is white, i.e. it has a uniform power spectrum density. The power density spectrum of a signal describes the power of the signal as a function of frequency, it can be found using Fourier analysis.

The noise of interest in this report is the subset of white noise following a gaussian distribution with zero mean (in electrical systems this is equivalent to having no dc offset).

The gaussian distribution is a probability distribution commonly used to represent a variety of random variables.

Figure 2.5 illustrates the gaussian probability density function:

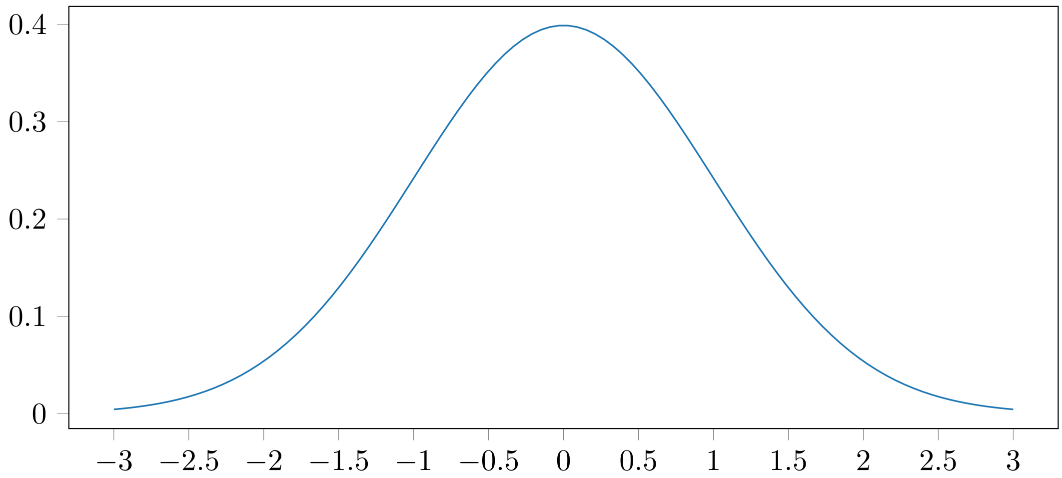
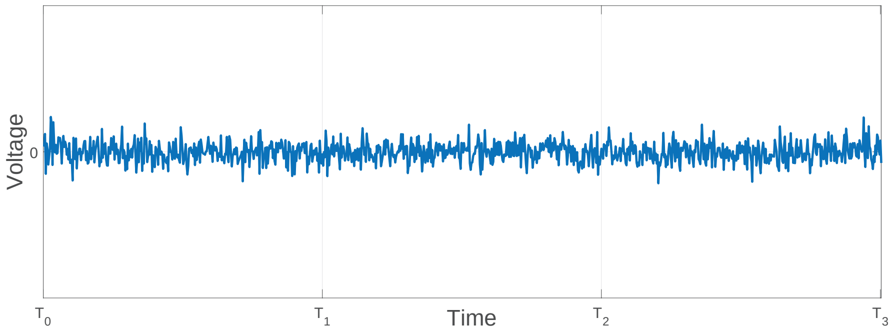
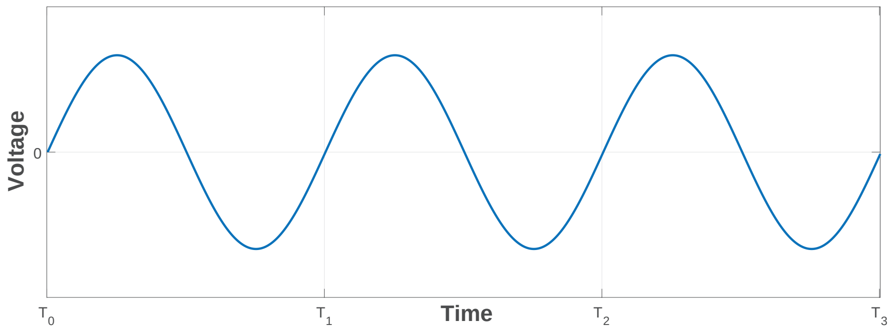


Figure 2.5: Depicts the standard gaussian distribution, with $\mu = 0$ and $\sigma = 1$

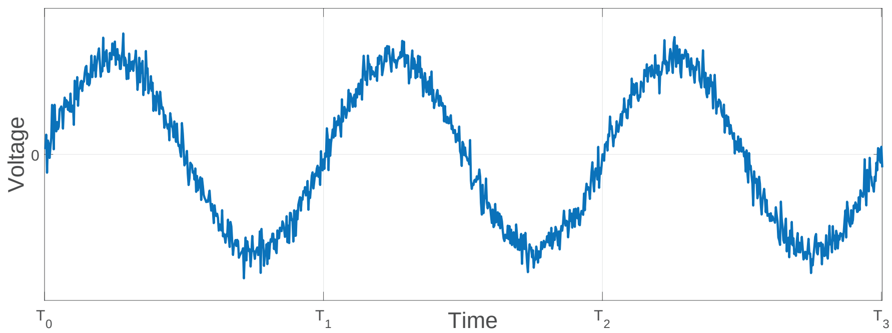
While the gaussian distribution has many unique properties, in noise theory its most prominent attribute is its connection to the central limit theorem. The central limit theorem suggests that a sampled mean of most probability distributions approach a gaussian distribution as the number of samples approach infinity. This theorem is essential as it establishes that most forms of white noise can be modelled using a gaussian distribution. This type of noise is referred to as **A**dditive **W**hite **G**aussian **N**oise (AWGN). Figure 2.6 illustrates how AWGN is combined with a source signal to produce a distorted signal:



(a)



(b)



(c)

Figure 2.6: (c) is the combination of AWGN,(a), and a source signal, (b)

Usually the amount of noise present in a system is by itself of little interest, as it provides little insight into its effects. Instead it is often compared to a signal using the signal-to-noise ratio (SNR). The signal to noise ratio is formally defined as:

$$\mathbf{SNR} = \frac{P_{signal}}{P_{noise}} \qquad [\cdot] \quad (2.3)$$

where:

SNR is the signal to noise ratio [·]

P_{signal} is the average power of a signal [W]

P_{noise} is the average power of the noise [W]

When the SNR is large the signal is large compared to the noise, and thus the noise will have little impact on the integrity of the information. The opposite is true for a low SNR.

As mentioned earlier in the section, electrical noise is usually the only type of noise that can be affected when designing communication systems. There exist various kinds of electrical noise, such as shot noise produced by semiconductor properties, and one-over-f noise affecting low frequency circuits (more than a few kHz). However, for high frequency circuits thermal noise is often the most significant type, and therefore it will be the main type discussed in this report. (source)

Thermal noise occurs because of random motion of electrons within circuit elements. The motion (and in extension the total thermal noise) of electrons increase as a function of temperature and frequency. The power spectral density of thermal noise in a resistor is given by:

$$S_{TN}(f) \approx 2kT \qquad [\mathrm{W\,Hz^{-1}}] \qquad (2.4)$$

where:

S_{TN} Power spectral density [$\mathrm{WHz^{-1}}$]

k is the boltzman constant [$\mathrm{J\,K^{-1}}$]

T is the temperature [K]

It should be noted that equation 2.4 approximates the complete thermal noise formula.

Other properties of thermal noise, such as its mean square voltage and current, can be derived using equation 2.4. However, for calculating the SNR of a communication system, the available noise power is of primary interest. The available noise power is the total noise present in a given bandwidth, and can be defined as the integral of the power spectral density over the bandwidth. The available noise power generated by thermal noise is represented as:

$$P_n = kTB_N \qquad [\mathrm{W}] \qquad (2.5)$$

where:

P_N is the noise power [W]

k is boltzman constant [$\mathrm{J\,K^{-1}}$]

T is the temperature [K]

B_N is the bandwidth measured across [Ω]

Usually noise calculations of a circuit are performed as a series of two-port devices. For a single two-port device the noise performance, commonly known as the noise figure, is usually expressed as a ratio between the input and output noise power (normalized for gain):

$$F = \frac{GN_i + N_d}{GN_i} \quad [\cdot] \qquad (2.6)$$

where:

F is the ratio between the input and output noise [$\cdot$]

G is the gain of the device [$\cdot$]

N_i is the input noise power [W]

N_d is the noise contributed by the device [W]

An example of a two-port network is illustrated below:

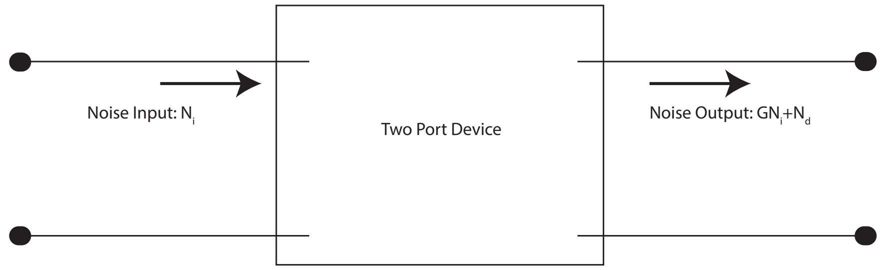


Figure 2.7: Two-port network and its input/output noise relationship.

When two-port networks are serially linked, the overall noise can be expressed as:

$$F = F_1 + \frac{F_2 - 1}{G_1} + \frac{F_3 - 1}{G_1 G_2} + ... + \frac{F_n - 1}{\prod_{i=1}^{n-1} G_i} \quad [\cdot] \qquad (2.7)$$

where:

F is the ratio between the input and output noise [$\cdot$]

F_n is the noise figure of each device n [$\cdot$]

G_n is the gain of the two-port devices [$\cdot$]

From eq. 2.7 it can be shown that if the gain of the first two-port network is large, then the noise power contributed at later stages of the two-port network barely affects the total noise power. Thus, when designing a radio, special care must be taken when creating the receiver (which will be the first two-port device in the circuit network), as it is will be a major source of noise.

2.4 Range and Link Budget

As described in section 2.3, to be able to successfully retrieve and extract information from a signal, the power of the information component must be significantly larger than that of additive noise. This minimum requirement for the reception SNR is also the limiting factor that determines the range of radio transmissions. The power of a received signal can be determined by accounting for the originally transmitted signal power, as well as any gains or losses that the signal may undergo when travelling between the transmitter and receiver. This summary of signal affecting factors is referred to as the link budget, and can in simple terms be expressed as:

$$P_R = P_{ET} + G - L \qquad \text{[dBm]} \qquad (2.8)$$

where:

P_R is the received power [dBm]

P_{ET} is the effective transmission power [dBm]

G represents any gain of signal power [dB]

L represents any loss of signal power [dB]

It is often a difficult task to fully quantify the link budget of a radio transmission. While a detailed explanation of the methods is outside the scope of this report, a brief introduction to the underlying principles involved in free-space link calculations is warranted.

In free-space transmission models, there must be a line of sight path between the transmitter and receiver, furthermore it is assumed that waves always propagate in a direct path.

In free space link calculations, the primary signal gain/loss contributors are the transmission power, transmitter antenna gain, receiver antenna gain, the transmission wavelength, and the distance between the transmitter and receiver.

In ideal free space propagation, the distance between the transmitter and receiver is the reason for signal loss. This loss is commonly referred to as *free space path loss* (FSPL) and is attributed to the decreasing power density, also known as intensity, of the signal as it propagates away from the transmission source. The intensity of the signal adheres to the inverse square law, meaning that the intensity of the signal is inversely proportional to the square of the travelled distance. This is usually presented as:

$$\text{Intensity} \propto \frac{1}{\text{distance}^2} \qquad [\cdot] \quad (2.9)$$

For an isotropic transmission antenna, the signal is transmitted uniformly in all directions, thus the transmitted energy is distributed in the shape of a sphere. As the signal then propagates away from the antenna, its power must be distributed equally across the increasing area of the sphere. This is represented in figure 2.8:

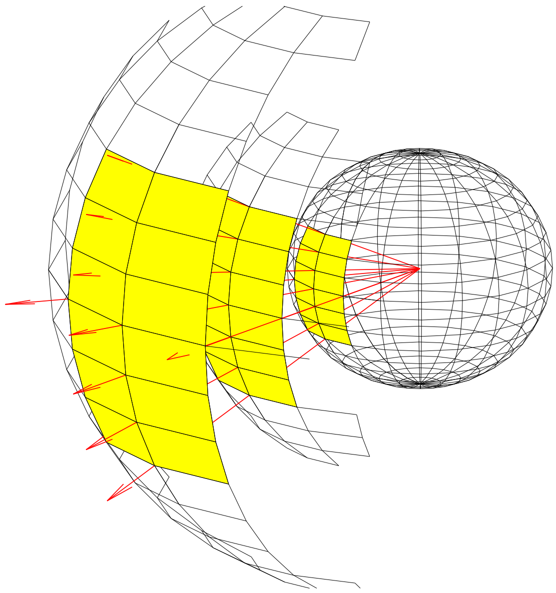


Figure 2.8: Depicts how the power density (signal strength) decreases, as a function of distance, when emitted from an isotropic transmitter. The lines represent a subset of the radiated waves.

Thus, for an isotropic transmission, the intensity can mathematically be described as:

$$\text{Intensity} \propto \frac{1}{4\pi r^2} \qquad [\cdot] \quad (2.10)$$

where

r is the distance from the antenna [dBm]

However, power loss caused by the inverse square law only accounts for part of the FSPL. The other part is determined by the receiving antenna's ability to absorb radio waves. This is referred to as the effective aperture area (A_{eff}). For an isotropic antenna the A_{eff} can be modelled as a function of the radio wavelength, and is described as:

$$A_{eff} = \frac{\lambda^2}{4\pi} \quad [\mathrm{m}^2] \quad (2.11)$$

where

A_{eff} is the effective aperture area [m^2]

λ is the wavelength of the transmission [m]

Finally, equation 2.10 and 2.4 can be combined to form the FSPL formula:

$$G_{FSPL} = (\frac{1}{4\pi r^2})(\frac{\lambda^2}{4\pi}) = (\frac{\lambda}{4\pi r})^2 \quad [\mathrm{dB}] \quad (2.12)$$

To make future calculations easier, equation 2.12 is often expressed as dB:

$$G_{FSPL} = 20\log_{10}(\frac{\lambda}{4\pi r}) \quad [\mathrm{dB}] \quad (2.13)$$

While the FSPL accounts for signal loss, the design of the transmitting and receiving antennas can be a major source of gain. The effective gain depends on the design of the antenna, for example a simple quarter-wave monopole (i.e. the antenna length equals one fourth of the wavelength) will under ideal conditions have a gain of ~5.2 dB [8]. Further details regarding antenna design is outside the scope of this report, and will not be explored further.

The final factor required for calculating the free space link budget (as well as any other link calculation) is the power radiated by the transmitter. This can in many cases be tweaked by the radio designer, and its value often depends on the application of the transmitter. For example, a radio mast used for FM radio will often have a power level of ~80 dBm [9], while a cell phone will at their maximum output ~33 dBm [10].

The above-mentioned effects can be combined using a variation of Friis' transmission formula, to create a model for the link budget for ideal free space propagation:

$$P_R = P_T + G_T + G_R + G_{FSPL} \quad [\mathrm{dBm}] \quad (2.14)$$

where:

P_R is the received power	[dBm]
P_T is the power radiated by the transmitter	[dBm]
G_T is the gain of the transmitting antenna	[dBi]
G_R is the gain of the receiving antenna	[dBi]
G_{FSPL} is the free space path loss	[dB]

It is crucial to note that equation 2.14 only applies for ideal free space transmission models, and does not account for wave propagation properties such as absorption, reflection, and refraction. Nor does it account for signal loss caused by the impedance of the receiving electrical components. Since even the most minor variations in the propagation environment will affect the behavior of the transmitted waves, it is difficult to create accurate link budget models for most forms of terrestrial communication. To produce these models, computer simulations are often combined with land surveys. Despite the complexity associated creating accurate link budget models, the free space link budget model is still extensively used to guide the creation for more complete models. Furthermore, certain situations can comprehensively be modelled by the free space model. An example of this is in space based communication, where the signal largely follows a direct path between the transmitter (e.g. an earth based station) and the receiver (e.g. satellite). Another case is FM radio broadcasting, where the free space model can adequately be used to approximate the link budget across flat rural land. Free space link budget calculations are relatively easy to perform.

For example, the Danish broadcasting corporation (DR) broadcasts 93.3 MHz FM radio from a 60 kW monopole antenna in Frejlev, located ~10 km from Aalborg university's institute for electronic systems [11]. By assuming free space, and that all antennas are quarter wave monopoles, the received power,P_R, at the institute can be calculated. To find the free space path loss, G_{FSPL}, both the distance between the transmission and the reception point is required, as well as the wavelength of the transmission. Since the transmission is at 93.3 MHz, the wavelength is found to be ~3.2 m, and the distance is given as 10 thousand meters. When these two values are used in equation 2.13, the G_{FSPL} is found to be ~-91.9 dB. Since the both the receiving and transmitting antenna are quarter wave monopoles, their gains, G_T and G_R, are ~5.2 dB. The power radiated by the antenna, P_T is 60 kW, or ~77.8 dBm. When these values are inserted into equation 2.14, the received power is calculated to be ~3.7 dBm, or 0.42 mW.

2.5 FM Theory

This section will, with regards to FM radio, expand upon the theory described in the previous section

2.5.1 Mathematical description of FM

The basics of frequency modulation were presented in section 2.2 in the form of a basic comparison between AM and FM. In the following, the previously presented theory will be greatly expanded upon. It should be noted that the following only accounts for a single tone modulating wave (e.g. a sine wave), as multitone waves are considerably more complicated to account for.

The single tone modulating wave is defined to be:

$$m(t) = A_m \cos(2\pi f_m t) \qquad [\mathrm{V}] \qquad (2.15)$$

where

m_t Is the unmodulated information signal [V]

A_m Is the amplitude of the waveform [V]

f_m Is the frequency of the waveform [Hz]

t Is time [s]

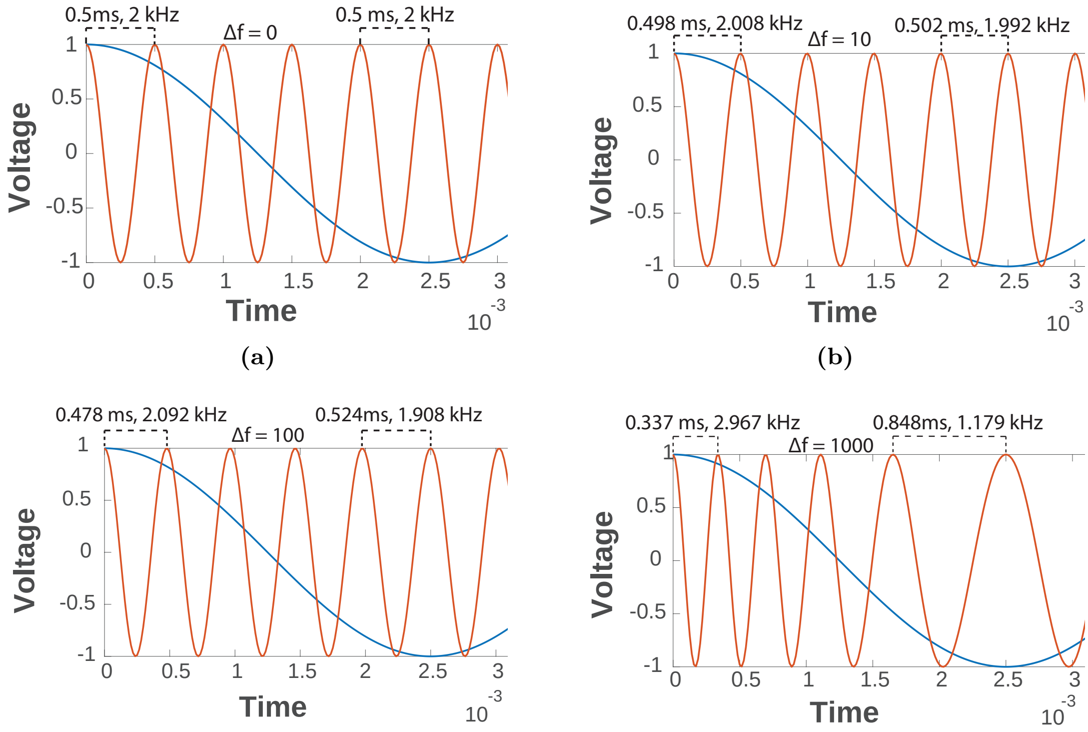
The FM wave of equation 2.15 can be found by applying equation 2.2:

$$s(t) = A_c \cos[2\pi f_c t + \frac{\Delta f}{f_m} \sin(2\pi f_m t)] \qquad [\mathrm{V}] \qquad (2.16)$$

where:

$$\Delta f = k_f \cdot A_m$$

The frequency deviation, Δf, is directly proportional with the frequency sensitivity, as it is the sensitivity multiplied with the amplitude of the information-signal, A_m. Δf determines the maximum frequency shift away from f_c in either direction. Figure 2.9 exemplifies this:



(a) (b) (c) (d)

Figure 2.9: The four figures shows how Δf affects the max/min frequency of the modulated signal for Δf = 0,10,100, and 1000. The blue line is an unmodulated 200 Hz cosine wave. The orange line is the modulated signal, where the carrier wave frequency is 2000 Hz.

The frequency deviation largely determines the significant bandwidth of an FM signal, this is shown in figure 2.10. The guard band at both ends of the bandwidth is used in practical applications to reduce the risk of signals of nearby frequencies interfering with the desired channel.

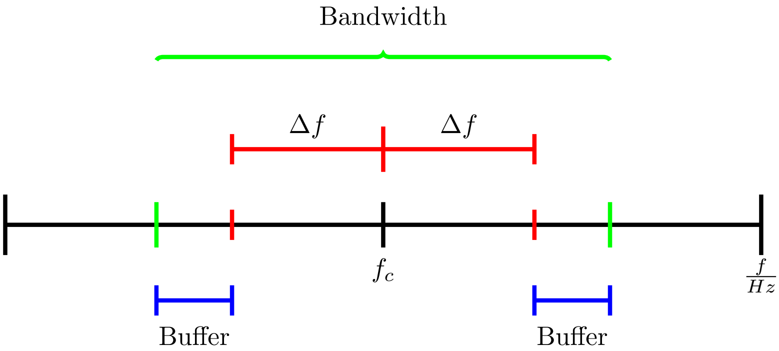


Figure 2.10: The frequency deviation of an FM signal.

When the bandwidth is increased the noise tolerance of the transmission will increase as well. In simple terms, this trait of FM allows operators to trade bandwidth with signal integrity, thus making it a more versatile modulation method compared to AM. The reason that operators don't use an arbitrarily large bandwidth to minimise the effects of noise, is that the radio spectrum is a limited resource, and therefore expensive to rent. For example, in 2016 the Danish Energy Agency auctioned 130 Hz in the 1800 MHz frequency band (used for mobile communication) for more than 1 billion DKK [12].

Equation 2.16 is usually presented as:

$$s(t) = A_c \cos[2\pi f_c t + \beta \sin(2\pi f_m t)] \quad [\mathrm{V}] \quad (2.17)$$

where:

$$\beta = \frac{\Delta f}{f_m} = \frac{k_f A_m}{f_m} \quad [\mathrm{rad}] \quad (2.18)$$

Beta, commonly known as the modulation index, is a critical variable in FM, and it represents the maximum phase deviation of the modulated carrier wave.

When beta is small (usually less than 0.3), the modulation is said to be narrow band, and equation 2.16 can be simplified (using trigonometric approximations) to:

$$s(t) \approx A_c \cos(2\pi f_c t) + \frac{1}{2}\beta A_c\{\cos[2\pi(f_c + f_m)t] - cos[2\pi(f_c - f_m)t]\} \quad [\mathrm{V}] \quad (2.19)$$

Equation 2.19, shows that narrow band FM modulation is composed of three distinct wave forms: the carrier wave, and one side band in the form of an upper ($f_c + f_m$) and lower frequency ($f_c - f_m$).

Their interaction is shown in figure 2.11.

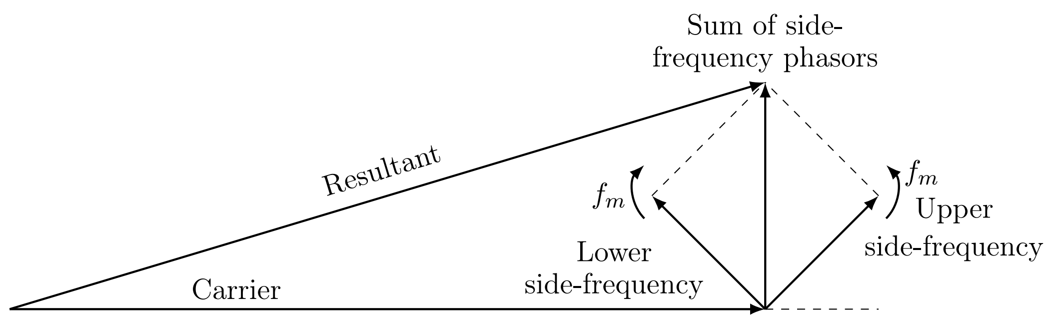


Figure 2.11: Phasor Diagram.

As can be seen in equation 2.19, the effective bandwidth of narrow band FM is $2f_m$.

Because of the approximation used to derive equation 2.19, it only accounts for one set of side bands. An FM signal has infinite side bands, however most of these are of insignificant power and thus their contribution to the final signal is negligible. When discussing side bands, the common practice is to only account for side bands if their relative power makes up more than 1 % of the total signal.

For large beta values (larger than narrow band), the frequency modulation is said to be wide band. In wide band frequency modulation, the number of significant side band's increase as beta increase. Deriving the equation for wide band FM signal is an extensive and complex procedure and has been omitted [2]. However, it is usually presented as:

$$s(t) = A_c \sum_{n=-\infty}^{\infty} J_n(\beta)\cos[2\pi(f_c + nf_m)t] \qquad [\mathrm{V}] \quad (2.20)$$

where

$J_n(\beta)$ Is the Bessel function of the first kind, with respect to β
$[\cdot]$

The Bessel function is defined as:

$$J_n(\beta) = \frac{1}{2\pi}\int_{-\pi}^{\pi} e^{j(\beta\sin(x) - nx)}dx \qquad [\cdot] \quad (2.21)$$

where:

$$x = 2\pi f_m t \qquad [\cdot] \quad (2.22)$$

The Bessel function with respect to β determines the relative amplitude of each side band. Each side band is represented by the integer order, n, of the Bessel function. A graphical representation of the Bessel function is shown below:

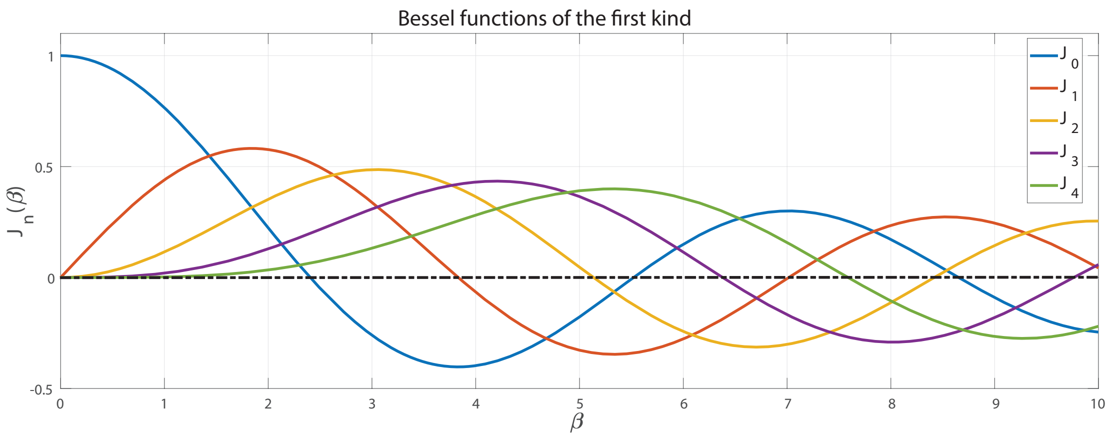


Figure 2.12: Depicts the Bessel function with respect to β, for integer orders: n =1,2,3,4.

[2]Left as an exercise for the reader.

Figure 2.13, further illustrates how the Bessel function is used to find the side bands of an FM signal. Each side band is f_m hertz apart, and their amplitude is found with the Bessel function when $\beta = 1$:

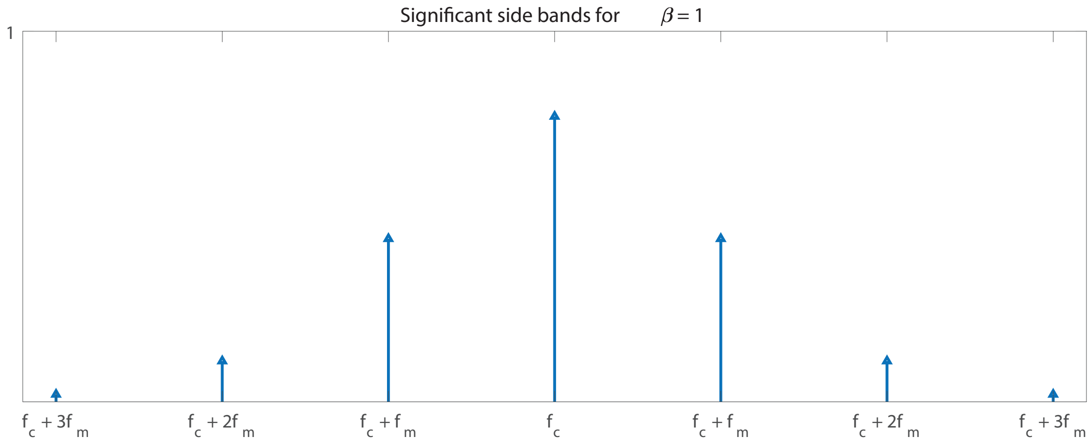


Figure 2.13: Depicts the amplitude of the significant side bands, where $\beta = 1$.

As an FM signal theoretically has infinite side bands, its theoretical bandwidth should also be infinite. However, as the majority of side bands are insignificant, most frequencies can be ignored. Thus, the effective bandwidth is the bandwidth that contains most of the signal power. One popular method used to estimate effective bandwidth is Carson's rule. Carson's rule is defined as:

$$\text{Bandwidth} = 2\Delta f(1 + \frac{1}{\beta}) \qquad [\text{Hz}] \qquad (2.23)$$

The Bandwidth found using Carson's rule account for $\approx 98\%$ of sideband energy.

2.5.2 Receivers

Throughout radio history a variety of radio receivers have been designed, such as the simple crystal radio or the more complicated tuned radio receiver. Today however, only the superheterodyne receiver design has survived the test of time to become the universal receiver used in demodulating analogue FM carrier signals.

The basic superheterodyne receiver is divided into the following stages: RF Filter and amplifier, mixer (connected to an oscillator), IF filter and amplifier, and finally the demodulator. A block diagram of this setup is shown in figure 2.14:

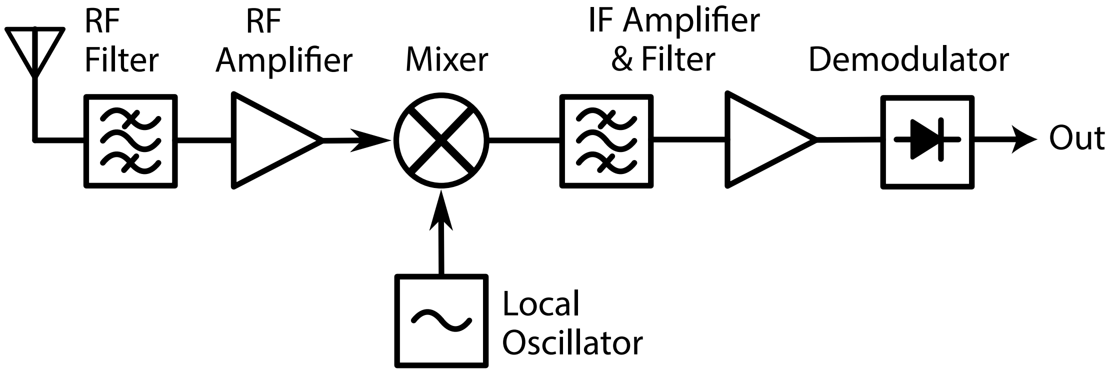


Figure 2.14: Simple block diagram illustration of a superheterodyne radio receiver.

The antenna receives the radio signals in the form of tiny voltages. These signals are then fed into the RF filter and amplifier. The purpose of the RF filter is to allow the desired frequency to pass through freely, while attenuating the image frequency produced by the mixer. This is usually implemented using a band-pass filter with a modifiable centre frequency. The signal then enters an amplifier used to amplify the signal to ensure that it is powerful enough to drive the following circuitry, as well to minimise the effect of future noise. It should be noted that special care must be taken to design a low noise RF filter and amplifier stage, as the signal will be at its maximum noise susceptibility during this stage due to its low voltage.

After the signals has been amplified, it is led into the multiplicative mixer. The mixer is a special type of electrical device which multiplies two time varying signals together. This is significant since multiplication in the time domain translates to convolution in the frequency domain, thus the mixer allows one to shift the frequency (f_{RF}) of a radio signal using an oscillator (with frequency f_{LO}). The output of the mixer will be a convolution of two signals with their frequency being the sum ($f_{RF} + f_{LO}$) and difference ($|f_{RF} - f_{LO}|$) of the frequencies of the original signals. This mixing, also referred to as heterodyning from which the receiver gets its name, is what characterises the superheterodyne receiver. Since all frequencies that enter the mixer will be mixed with the signal from the local oscillator, the intermediate frequency (IF) filter is placed after the mixer to filter all frequencies except for a single predetermined frequency called the intermediate frequency. For FM the IF is commonly placed at 10.7 MHz. By changing the frequency of the LO the RF frequency which is mixed to become IF frequency also changes. For example, if it is desired to tune an FM radio with an IF of 10.7 MHz to 100 MHz the LO would be set to 89.3 MHz since 100-89.3=10.7 . If the tuned radio frequency is then to be changed, then only the LO frequency must be modified. However, there is a slight error in the previous example since the mixer will mix two different RF frequencies to the desired IF frequency. Using the values from the above example, the second RF frequency mixed to the IF frequency would be 78.6 MHz since |78.6-89.3| also equals 10.7 MHz. This second undesired frequency is referred to as the image frequency and it will interfere with the desired signal. To combat the image frequency's effect on the desired signal the RF filter is used to attenuate it in the very first stage of the receiver. To remain effective throughout the entire FM radio spectrum, the RF filters modifiable band pass filter must be matched with LO frequency. In other words when the LO frequency is changed, the RF band pass centre frequency must be changed by the same magnitude.

To quantify a superheterodyne receiver's ability to suppress the image frequencies, the image rejection ratio (IMRR) is used. The IMRR **is the ratio of the intermediate frequency (IF) signal level produced by the desired input frequency to that produced by the image frequency.**

The final stage of the receiver is the demodulator which is used to acquire the originally transmitted information signal. The demodulator can be designed for any kind of modulation scheme (such as an AM) however in this report it will be an FM demodulator.

Figure 2.15 summarises how the key stages of the superheterodyne process radio signals:

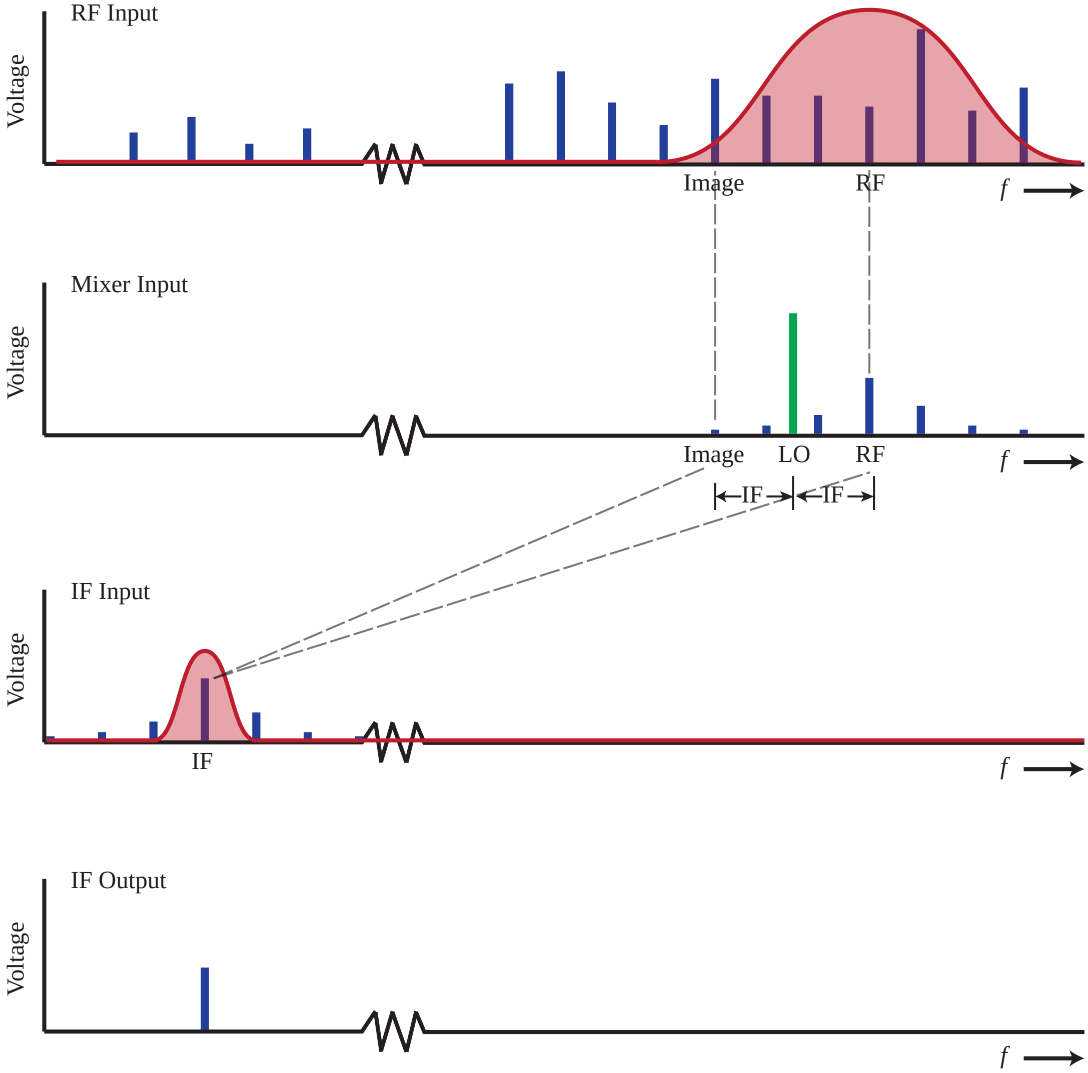


Figure 2.15: Illustration of the key stages of the superheterodyne processing of radio signals.

The basic superheterodyne design can be improved in many ways, the most common being the addition of an additional IF mixer stage to improve image rejection and channel selection. The block diagram of this design is shown in figure 2.16:

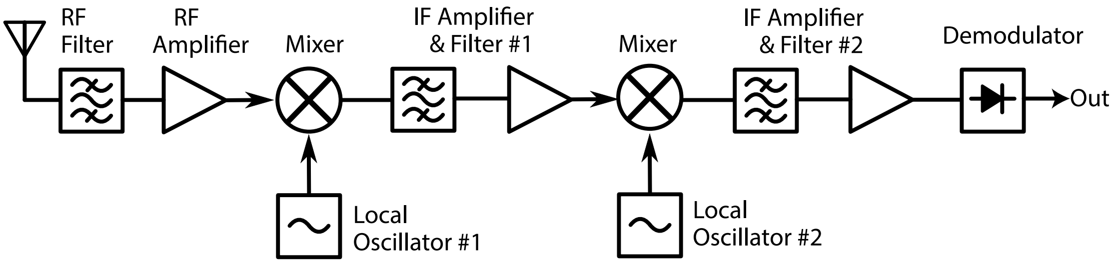


Figure 2.16: Illustration of modules implemented in a superheterodyne radio receiver.

The design shown in figure 2.16 includes an additional mixer and IF stage compared to the basic design. This design is advantageous since it allows designers to combine the advantages provided by using both low and high IF's. A high IF generally improves IMRR as it creates a larger frequency difference between the image frequency and the desired frequency, and thus the image frequency will undergo more attenuation from the RF filter. On the other hand, a low IF improves selectivity, i.e. the radios ability to only respond to its tuned frequency, as it is much easier to create narrow bandwidth bandpass filters for low frequencies. For an FM superheterodyne receiver using two mixers and IF stages, the IF are commonly set to 10.7 MHz, and 470 kHz.

2.5.3 FM Stereo Multiplexing

Before a signal can be modulated onto a carrier wave and transmitted, the necessary signals must be gathered at a summary point.
In Danish FM Broadcasting, both mono and stereo audio signals are transmitted. In the FM transmitter both the left and right channel of a stereo signal are the input of a multiplexing system. This system outputs a mono signal which is the sum of the left and right channel, which lies in the first 15 kHz. An illustration of the baseband for FM broadcasting can be seen on figure 2.17.

Furthermore, its worth mentioning the existence of Radio Data System (RDS). RDS is small embedded digital code within the FM band. The reason for using RDS is to increase the functionality of FM receivers and ease of use by, for example by implementing automatic tuning through the RDS for portable radios[13].

Often used applications for RDS includes

- radio station (PS)
- program information (PI)
- time (CT)

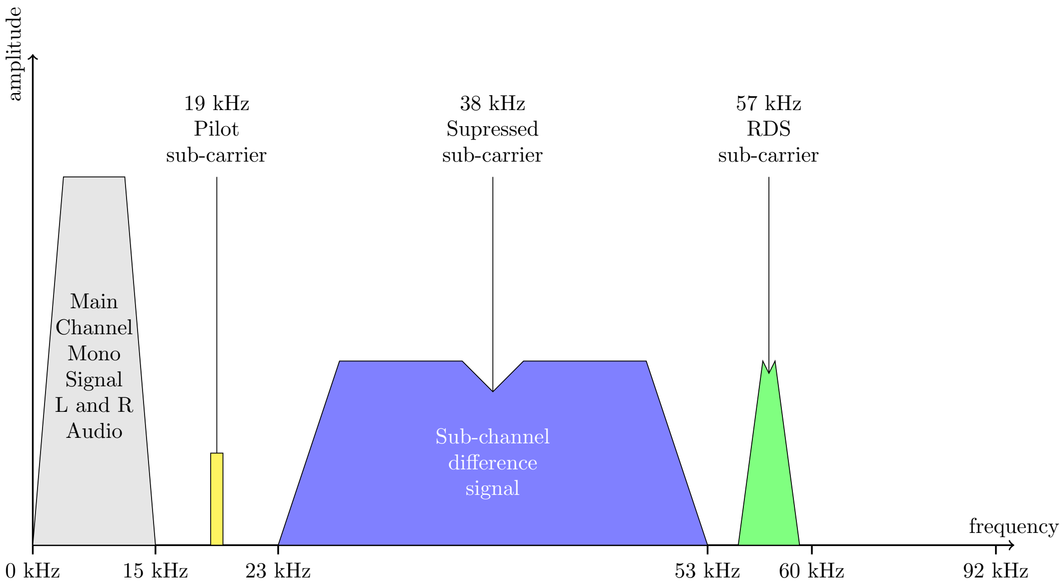


Figure 2.17: An overview of the baseband layout for FM broadcasting [13].

RDS is however out of scope for this project.

2.5.4 Pre- and De-emphasis

When a FM signal is transmitted the input frequency range most vulnerable to noise is enhanced. This process is known as "pre-emphasis". To accomplish this enhancement the modulated signal is passed through a pre-emphasis network designed to amplify these frequencies. The increased ampliutde makes the vulnerable frequencies more resistance to noise.

When the transmitted FM signal arrives at a receiver it needs to be returned to normal. This is process is known as "de-emphasis". To return the signal to its original state a de-emphasis circuit is implemented. This is a low-pass filter with a predetermined time constant of 75 µs [14].

2.6 Standards and recommendations applying to FM radio systems

When developing FM radio systems there are standards that apply to the minimum requirements of the system, and to the methods with which these shall be measured. To specify a radio system as having high fidelity sensitivity a specific set of performance criteria must be met. [15] These criteria are:

- The effective frequency range must be from at least 40 Hz to maximum 12,5 kHz.
- The total harmonic distortion must not exceed 1%.
- Channel separation must be at least 30 dB at 6,3 kHz and at least 20 dB at 12,5 kHz.
- The unweighted SNR must be at least 57 dB

When testing all measurement shall be made at a radio-frequency input level of 70 dB (fW), which is equivalent to an e.m.f. source of 1,7 mV in a 75 Ω system. [15]

2.6.1 Reference audio-frequency output level

According to IEC standard 61305-2, the 0 dB reference level for all a.f. output voltage measurements expressed in decibels shall (except if otherwise stated) be the a.f. output voltage produced by an r.f. input signal level of 70 dB (fW), modulated at 1 kHz with ± 67.5 kHz deviation. [15]

2.6.2 Recommendations

In addition to the standards, are some recommendations from The ITU Radiocommunication Assembly. They recommend that for FM sound broadcasting in band 8 the following standards should be used: The RF signal shall consist of a carrier which is frequency-modulated by the sound signal to be broadcast with a maximum frequency deviation of ±75 kHz. The intermediate frequency used when manipulating received FM radio signals before demodulation should be 10,7 MHz. Likewise, it is recommended that the AF signal-to-noise ratio should be at least 50 dB at the critical frequencies which are integer multiples of the intermediate frequency (e.g. 96.3 MHz, 107 MHz). [16]
Lastly it is worth noting that in Denmark, the radio band used for sound broadcasting ranges from 87,5 MHz to 108 MHz, which means that a radio receiver should be designed in a way that allows it to tune into frequencies in this range if the goal is to listen to sound broadcasts.

Chapter 3

High Fidelity Audio

Through the last century, audio technology has undergone a major transformation in a never-ending quest to perfect audio quality.

During the beginning of the 1930s, scientists at Bells Laboratories began experimenting with a wider range of recording techniques. The conducted experiments created the foundation for the high-fidelity audio.
At around this time an amateur violist by the name Avery Fisher began experimenting with audio and acoustics to create a record player that could reproduce the sensation of listening to an actual live orchestra.

In the late 1940s after the end of World War II an engineer at *RCA Victor* conducted a series of experiments where he exposed a group of people to sound from a live orchestra through a hidden acoustic filter. The results proved that people preferred high fidelity. In 1947, a game changer was introduced to the field. The Williamson amplifier, which is an audio power amplifier with less than 0.1 percent Total Harmonic Distortion (THD). This meant that the components in e.g. a speaker could achieve a more accurate reproduction of an audio recording.

Other inventions such as:

- The electrodynamic speaker
- Electrostatic speakers
- Radio tubes
- The microphones
- FM radio
- Long Play (LP) vinyl record
- Open-reel audio tape recording
- Better amplifier designs

all played their part in the quest for high-fidelity.[17]

3.1 Amplifier Theory

An amplifier is used for amplifying the power of an arbitrary signal. The amplifier is fed a signal, and proceeds to amplify either the current or the voltage (depending on the type). The magnitude of the amplification is measured in **amplifier gain**. An amplifier can therefore be described simply as a “circuit with a power gain greater than 1. [18, p. 15-16]

An amplifier has many practical applications in the modern electronic age. An example is when a phone is connected to an audio system. The phone sends a signal through an audio cable to the audio system, where an amplifier amplifies the signal before it reaches the loudspeaker. In this example the signal is amplified multiple times before the sound is leaving the speakers. The signal will have to increase its voltage and current. This can be done by any combination of amplifiers, but will at least need a signal amplifier and a current amplifier. These amplifiers and some of their common types and their characteristics will be explained further in this section. In short, the signal amplifier isus ed as a preamplifier, and the current amplifier is used as the output stage and can deliver high power.

The ideal amplifier would have an efficiency of 100 % conservation rate of the energy supplied to the amplifier into the amplification process of the signal, however in practice this is not possible. A lot of power is inevitably lost as heat during amplification due to the resistance of the components used. Still, a lot of effort and research is conducted to get as close as possible to the ideal amplifier. The efficiency can be determined by the following formula: 3.1.

$$\text{Efficiency} = \frac{\text{power delivered to the load}}{\text{DC power taken from supply}} = \frac{P_{\text{out}}}{P_{\text{in}}} \qquad [\cdot] \tag{3.1}$$

Other characteristics of an ideal amplifier would include: [19]

- The amplifiers gain (A) must remain constant for all input signals.
- Gain is not affected by frequency. Signals of all frequencies must be amplified by exactly the same amount.
- The amplifiers gain must not add additional noise to the output signal.
- The amplifiers gain must not be affected by changes in temperature.
- The gain of the amplifier must remain stable for an indefinite amount of time.
- The amplifier must be able to provide sufficient power for any load.

These characteristics are not possible to achieve it is however desired to get as close as possible. Thus, depending on the application, it will be attempted to get the right balance of these parameters. Different amplifiers have different strengths and by combining multiple amplifiers it is possible to take advantage of these strengths to achieve the best amplification for the given application.

3.1.1 Different kinds of amplifiers

There are different kinds of amplifiers, the most common of which are described below.

Signal Amplifier

The most common amplifier is the signal amplifier. Its purpose is to amplify the voltage of the signal. The signal amplifier is used for low power signals which are signals below 1 watt. Usually the most important property of a signal amplifier, aside from amplifying the voltage, is to ensure that the output signal is as true in frequency and consistency to the input signal as possible. Any variation from the original input signal, is considered distortion.

The purpose of a signal amplifier is to increase the magnitude of the signal for practical use within circuitry. An example of such an amplifier is the preamplifier. The purpose of the preamplifier in a stereo system is to amplify the audio signal, often from around 1 V to tens of volts for the loudspeaker output. However most signal amplifiers cannot supply sufficient current to drive the loudspeaker, in such cases a power amplifier is used.

Power amplifiers

Power amplifiers are used to amplify the power of the input signal, so that the output signal delivered to a load can drive e.g. a loudspeaker. Most commonly the power amplifier simply increases the available current for the load and has a voltage gain of around 1. This is because several watts are usually required to drive a loudspeaker, and most audio circuits can only supply a few milliwatts because they consist of signal amplifiers.

The power gain can be expressed as: [18, p. 18]

$$A_p = A_i \cdot A_v \qquad [\cdot] \quad (3.2)$$

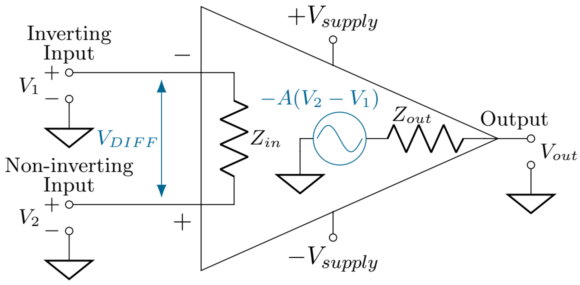
A_p is the power gain $[\cdot]$

A_i is the current gain $[\cdot]$

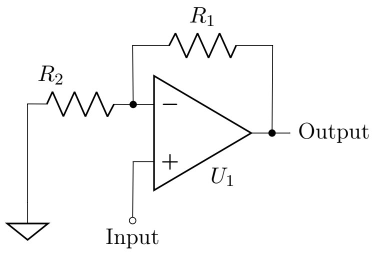
A_v is the voltage gain [·]

Operational amplifiers (op-amps)

Operational amplifiers, or op-amps, are linear devices which are close to having all the properties for ideal DC amplification. The op-amp is used for amplifying voltage and uses external feedback components such as resistors and capacitors, to determine its function. An op-amp usually has five terminals as seen in figure 3.1(a). There are two terminals used for supplying the op-amp with power for the amplification, one output terminal for the amplified signal, and two input-terminals called inverting input (-) and non-inverting input (+). As with other amplifiers the amplification is described as **gain**. [18, p. 95-]



(a) Op-amp with labelled pins



(b) Op-amp set up as amplifier with negative feedback

Figure 3.1: Illustration of op-amps.

3.1.2 Features of the amplifiers

Distortion

When amplifying a signal, it is desired to get as true an amplification as possible. In a perfect amplifier the output signal would be true to the input signal, with only a voltage gain. A perfect amplifier would have the following relationship between input and output signal.

$$v_o(t) = A \cdot v_i(t) \qquad [\mathrm{V}] \quad (3.3)$$

$v_o(t)$ is the input voltage [V]

$v_i(t)$ is the output voltage [V]

t is time [s]

A is the amplification of the input signal $[\cdot]$

Such an amplification is linear and contains no distortion nor noise, but only a voltage gain A. All electrical components are considered either linear or nonlinear, and linear components have a linear relationship between the current and the voltage across the component and follows Ohm's Law. These components have resistance, capacitance and/or inductance. The nonlinear components have nonlinear voltage/current relations. An example of such components are diodes and transistors. These components will distort the signal to some degree. It is however possible to approximate linear models for small variations in input for these otherwise nonlinear components. Amplifiers are made using nonlinear components, thus there will always be some distortion. Though it is possible to make a practically perfect amplification, there will always be some distortion, and of course also noise. Distortion can have multiple causes and multiple effects on the signal, though in audio distortion usually manifests itself as impurities in the sound. It should be mentioned that though noise and distortion is always present, it is often possible to reduce it to negligible amounts.

Amplifier saturation

Even the most linear amplifiers, are only linear within a certain range of inputs and outputs. If the signal exceeds the amplifiers output range the amplifier is said to be saturated and the output signal will be clipped. This can happen in most circuits if trying to amplify a signal with a magnitude greater than its supply voltage. The output is limited by the supply voltage, and when the amplifier tries to let more current trough than is supplied, the signal will clip and supply the maximum supply voltage until the signal is below the maximum supply voltage as seen on figure 3.2.

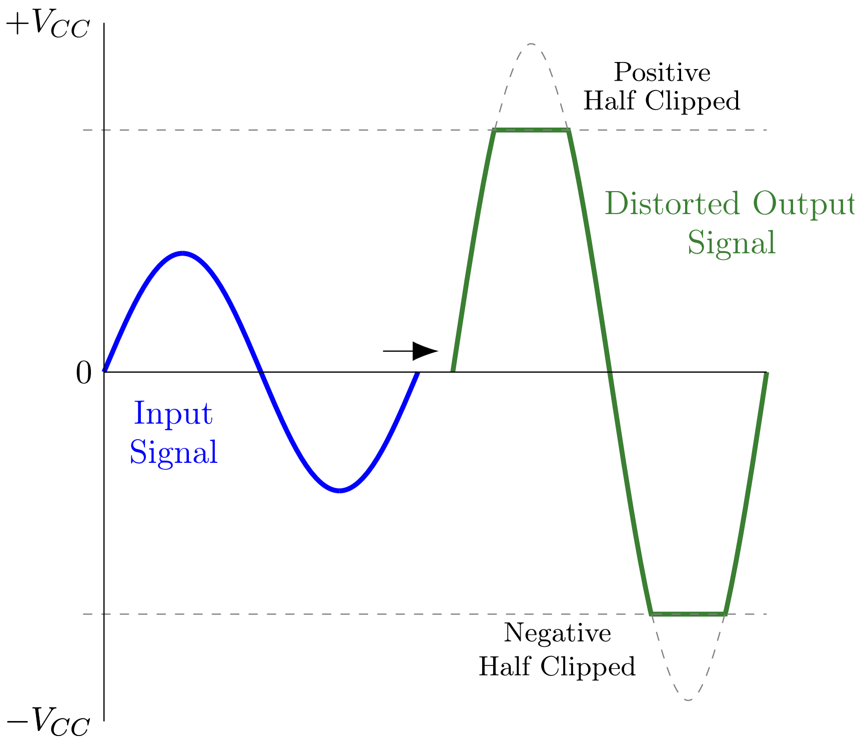


Figure 3.2: Illustration of clipping due to amplifier saturation.

When designing an amplifier circuit, it is thus important to consider which range of signals the circuit is supposed to amplify. If the supply voltage is the limiting factor either the supply voltage can be increased, while of course considering the limits of the components, or the gain can be reduced to allow a greater range of inputs without clipping the signal. It should be noted that amplifier saturation is a limit on the output signal.

Bandwidth

The bandwidth of an amplifier is based on its frequency response. The bandwidth is the interval with a constant gain within 3 dB. The bandwidth of an amplifier is like that of a bandpass filter, that is the frequencies in-between two frequencies ω_1 and ω_2 (f_1 and f_2 on the figure). Where the frequencies inside ω_1 and ω_2 have a gain within 3 dB. Frequencies outside the bandwidth will be distorted due to low gain.

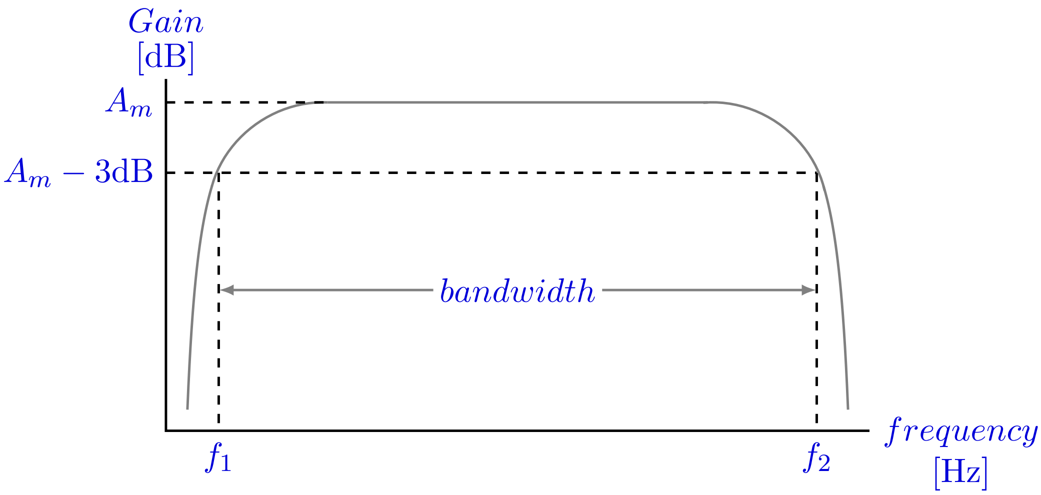


Figure 3.3: Illustration of bandwidth.

The bandwidth of an amplifier limits its output range. The lower cut off frequency is determined by the decoupling capacitors and can be adjusted by changing the size of the capacitors. The higher frequency cut off however is determined by the inner capacitance of the transistor. To adjust the upper cut off frequency either the transistor used should be changed, or the resistors can be selected based on the transfer function of the system. This however, can have other undesired effects on the circuit and sometimes is not an option depending on other requirements for the amplifier. This is often taken into consideration when designing audio amplifiers, where the entire human hearing range is amplified.

Crossover distortion

In some amplifiers the transfer characteristics is not linear around 0 volts (due to transistor properties), there will be crossover distortion in the form of a dead band. The class B amplifier's transfer characteristic has a dead band. This dead band is approximately from -0.5V to +0.5V. Any part of the input signal within this dead band will not be amplified.

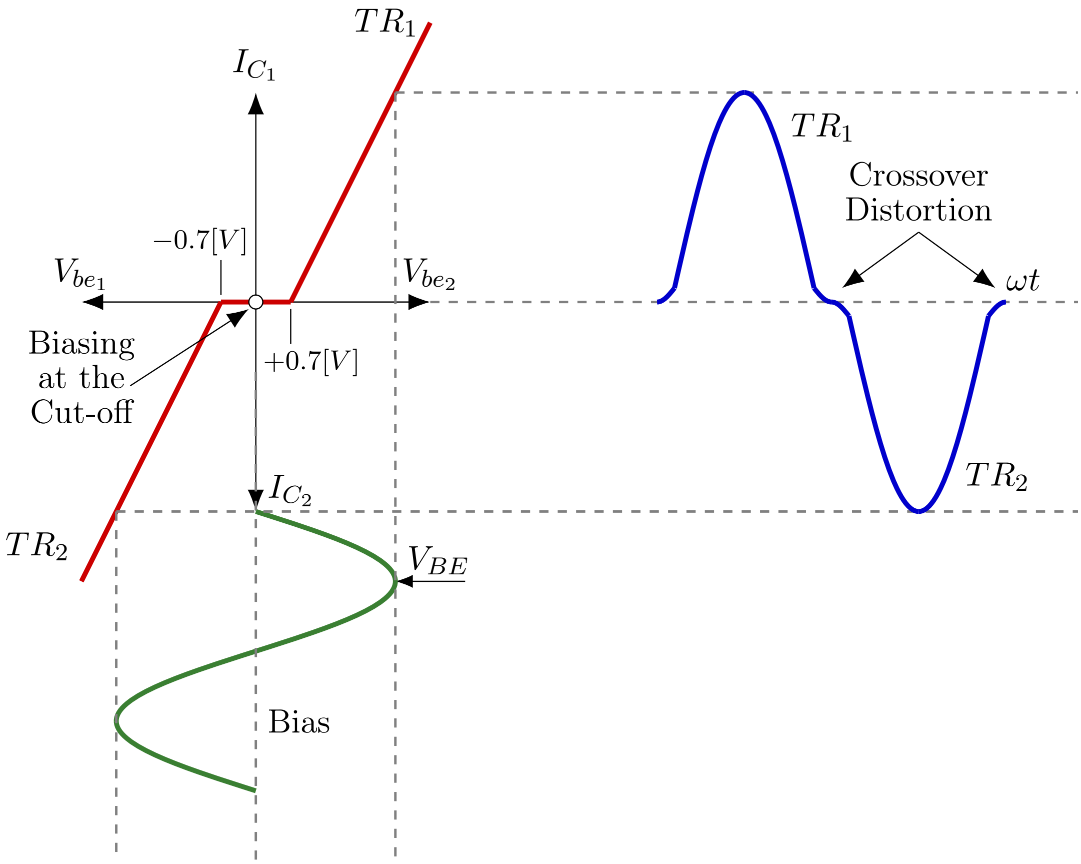


Figure 3.4: Illustration of crossover distortion due to nonlinear transfer characteristics.

Crossover distortion is undesirable, especially when amplifying audio. This has the most pronounced effect when using small voltage signals as the dead band is in a static voltage interval. A common example of crossover distortion is the class B amplifier(see section 3.1.4). The crossover distortion is due to the transfer function of an unbiased transistor used for the class B amplifier. However, this can be almost eliminated by biasing the transistors just within the linear range of the transistor, such an amplifier is called a class AB amplifier. Thus, with only a small biasing current it is possible to get almost completely rid of the crossover distortion. In the Class A amplifier, the transistor is biased enough to amplify the entire signal with no crossover distortion, but it is very inefficient.

3.1.3 Intermodulation & Harmonic Distortion

No electrical device adheres to a perfectly linear transfer function, and therefore harmonic and intermodulation distortion will occur. While this non-linearity, and in extension the distortion caused by it, is usually negligible for passive devices such as resistors, it must often be accounted for when dealing with highly nonlinear devices such as diodes or transistors.

To analyse the effects of harmonic and intermodulation distortion, the nonlinear transfer function is usually expanded into its power series. As an example, the transfer function e^x (a typical transistor transfer function) can be expanded, as shown in equation 3.4: [20]

$$y(x) = e^x = 1 + x + \frac{x^2}{2} + \frac{x^3}{6} + \frac{x^4}{24} + \frac{x^5}{120} + ... + \frac{x^n}{n!} \qquad [\cdot] \quad (3.4)$$

For small values of x, the linear term of the power series (1+x) provides a sufficient estimate of the transfer function. However as x increases an, increasing number of terms are needed to accurately model the transfer function. Figure 3.5 illustrates this idea[20] :

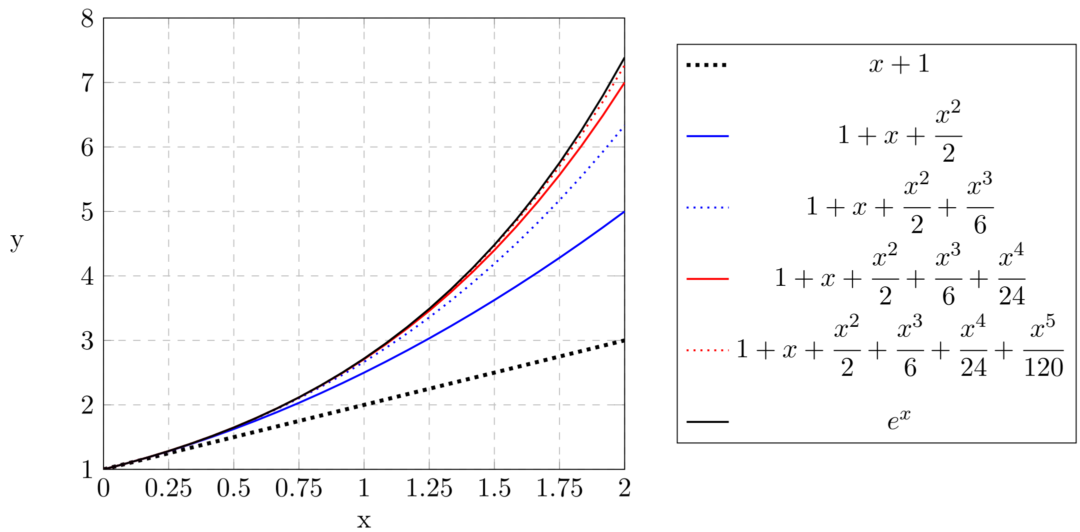


Figure 3.5: Graphical illustration for the general estimated transistor transfer function ($e^x \ldots$).

When variable x in equation 3.4 is a sinusoidal signal, such as $x = A \cdot \sin \omega t$, equation 3.4 becomes[20]:

$$\begin{aligned} y(x) = K_0 + K_1 A \cdot \sin \omega t + K_2 A^2 \cdot \sin^2 \omega t + K_3 A^3 \cdot \sin^3 \omega t + ... \\ + K_n A^n \cdot \sin^n (\omega t) \end{aligned} \qquad [\cdot] \quad (3.5)$$

Where K_n are the constant scaling factors (in this case: $K_0 = 1, K_1 = 1, K_2 = \frac{1}{2}$, etc)

By applying trigonometric identities, equation 3.5 can be rewritten as[20]:

$$y(x) = K_0 + K_1 A \cdot \sin \omega t + K_2 A^2 \frac{1 - \cos(2\omega t)}{2} + K_3 A^3 \frac{3\sin(\omega t) - \sin(3\omega t)}{4} + \ldots \quad [\cdot] \tag{3.6}$$

Equation 3.6 shows how the second and third order terms of the power series result in additional distortion products located at 2ω and 3ω (HD_2 and HD_3 respectfully) in the frequency domain. Higher order terms from the derived power series result in higher orders of Harmonic distortion.

The total harmonic distortion, THD, is a measurement of all the harmonic distortion present in a signal. The THD is given by equation 3.7

$$\mathrm{THD} = \frac{\sqrt{\sum_{n=2}^{\infty} V_n^2}}{V_1} \quad [\cdot] \tag{3.7}$$

where:

THD Total harmonic distortion [·]

V_n is the n harmonics contribution (in RMS amplitude) to the signal [V]

V_1 is the fundamentals contribution (in RMS amplitude) to the signal [V]

THD is usually either expressed as a percentage or in decibels.

[21, p. 2478-2491]

Figure 3.6 shows an example of harmonic distortion:

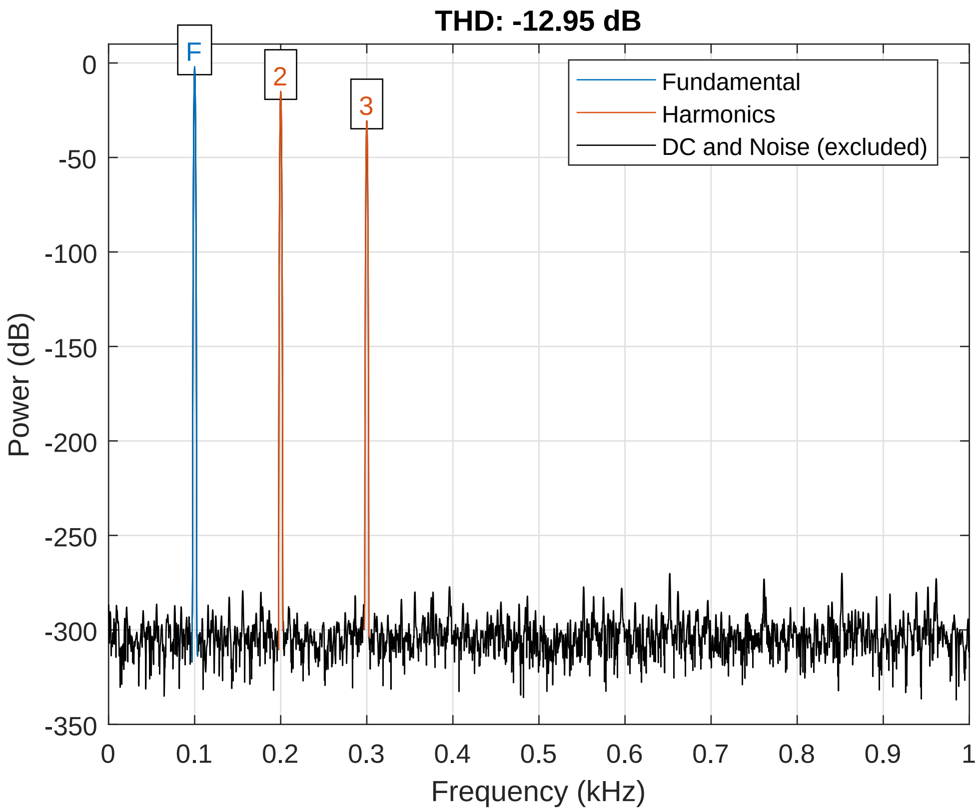


Figure 3.6: Illustration of harmonic distortion estimated to the third order, with a fundamental frequency of 100 Hz, based on equation 3.6

While a single frequency sinusoidal input signal explains how harmonic distortions can form, it is a rather simplistic model, as most audio and radio communications are comprised of signals containing multiple frequency components.

If instead of setting variable x from equation 3.4 equal to a single frequency sinusoidal signal, but rather a signal containing multiple frequencies, such as $x = A_1 \cdot \sin(\omega_1 t) + A_2 \cdot \sin(\omega_2 t)$, equation 3.4 can now be written as[20]:

$$\begin{aligned} y(x) = K_0 &+ K_1 \cdot [A_1 \cdot \sin(\omega_1 t) + A_2 \cdot \sin(\omega_2 t)] \\ &+ K_2 \cdot [A_1 \cdot \sin(\omega_1 t) + A_2 \cdot \sin(\omega_2 t)]^2 \\ &+ K_3 \cdot [A_1 \cdot \sin(\omega_1 t) + A_2 \cdot \sin(\omega_2 t)]^3 \\ &+ \ldots \\ &+ K_n \cdot [A_1 \cdot \sin(\omega_1 t) + A_2 \cdot \sin(\omega_2 t)]^n \end{aligned} \qquad [\cdot] \quad (3.8)$$

These terms can then be expanded and then simplified using trigonometric identities:

- The first and second term remain identical
- The third (quadratic) term becomes:

$$\begin{aligned} & K_2 \cdot \Big(A_1 \cdot \sin(\omega_1 t) + A_2 \cdot \sin(\omega_2 t)\Big)^2 \\ & = K_2 \Bigg(A_1^2 \frac{1-\cos(2\omega_1 t)}{2} + A_2^2 \frac{1 - cos(2\omega_2 t)}{2} \\ & + 2A_1A_2 \frac{\cos(\omega_1 t - \omega_2 t) - \cos(\omega_1 t + \omega_2 t)}{2} \Bigg) \end{aligned} \quad [\cdot] \quad (3.9)$$

- The fourth (cubic) term becomes:

$$\begin{aligned} & K_3 \cdot \Big(A_1 \cdot \sin(\omega_1 t) + A_2 \cdot \sin(\omega_2 t)\Big)^3 \\ & = K_3 \Bigg(A_1^3 \frac{3 \cdot \sin(\omega_1 t) - \sin(3\omega_1 t)}{4} \\ & + 3A_1A_2 \frac{2\sin(\omega_2 t) - \sin(2\omega_1 t + \omega_2 t) - \sin(2\omega_2 t + \omega_1 t)}{4} \\ & + 3A_1A_2 \frac{2\sin(\omega_1 t) - \sin(2\omega_2 t + \omega_1 t) - \sin(2\omega_1 t + \omega_2 t)}{4} \\ & + A_2^3 \frac{3 \cdot \sin(\omega_2 t) - \sin(3\omega_2 t)}{4} \Bigg) \end{aligned} \quad [\cdot] \quad (3.10)$$

Equation 3.9 and 3.10 shows that not only does a multi-frequency input still produce harmonic distortion. It also produces *intermodulation distortion* in the form of sum products of ω_1 and ω_2, such as $2\omega_1 + \omega_2$. All distortion frequencies caused by the quadratic and cubic term are listed in table 3.1[20]:

Table 3.1: Distortion frequencies produced up to the third order when applying two sinusoidal signals to equation 3.5.

2^{nd} and 3^{rd} order distortion frequencies				
2^{nd} Order:	$2\omega_1$	$2\omega_2$	$\omega_1 - \omega_2$	$\omega_1 + \omega_2$
3^{rd} Order:	$3\omega_1$	$3\omega_2$	$2\omega_1 - \omega_2$	$2\omega_2 - \omega_1$
	$2\omega_1 + \omega_2$	$\omega_1 + 2\omega_2$		

It should be noted that while an infinite order of intermodulation frequencies exists, only odd orders contribute to intermodulation. Additionally, higher orders of IMD are in many cases ignored in calculation due their negligible magnitude. Furthermore, as an input signal contains an increasing amount of frequency component, so does the number of resultant intermodulation frequencies.

While most intermodulation distortion is either of insignificant magnitude or can easily be filtered, certain frequencies of low order (primarily the third order) can be of a significantly large magnitude to cause distortion, and be located too close in frequency to the desired signal to be effectively filtered. For example, if ω_1 and ω_2 from equation 3.8 were equal to $2\pi 101$ Mrad s^{-1} and $2\pi 99$ Mrad s^{-1} respectfully. Then intermodulation distortion would occur at $2\pi 202 - 2\pi 99 = 2\pi 103$ and $2\pi 198 - 2\pi 101 = 2\pi 97$ Mrad s^{-1}, which is relatively close in frequency to the two input signals, and thus cannot easily be filtered. Figure 3.7 illustrates a couple of distortion frequencies produced by second and third order nonlinearities:

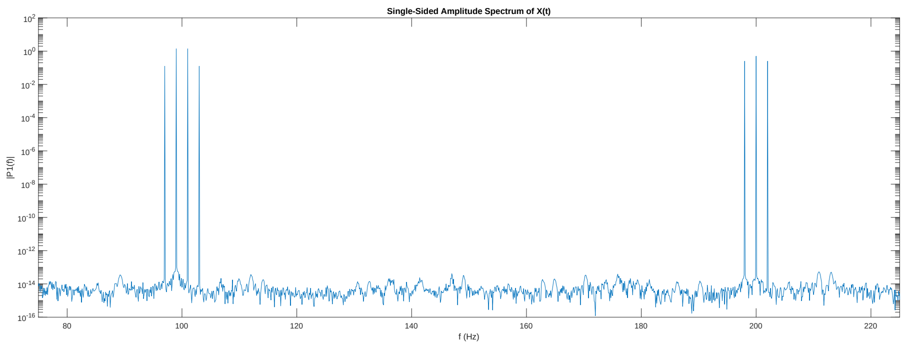


Figure 3.7: Illustration of harmonic distortion estimated to the third order, with a fundamental frequency of 100 Hz.

Noise

Noise is separate from the distortion of the signal, as it originates from other sources than the signals interaction with the circuit itself. As mentioned in section 2.3, Amplifiers, as any electronic component, adds noise to the signal. Furthermore, when the input signal is amplified the noise in the input signal is also amplified. Amplifiers however, are used to reduce the impact of the noise that is added to the signal after the amplification. This is because the noise added to the signal has a static magnitude, thus when increasing the magnitude of the signal the signal to noise ratio is reduced.

3.1.4 Output stages

The amplifier that delivers the signal to the load is called the output stage. Depending on the requirements, the output stage can be made in several ways. The output stages are classified based on how they are operating. Some classes of amplifiers have been standardized and will be summarised below. More classes of operation have been produced, but have yet to be standardized.

Class A

The class A amplifier is the simplest of the amplifiers, and it has the advantage of producing minimum distortion and crossover. It achieves this by always drawing power from the DC power supply. [18, p. 916] However this constant current draw also makes the class A amplifier the most inefficient amplifier there is, since it is drawing power, even if there is no AC signal to amplify.

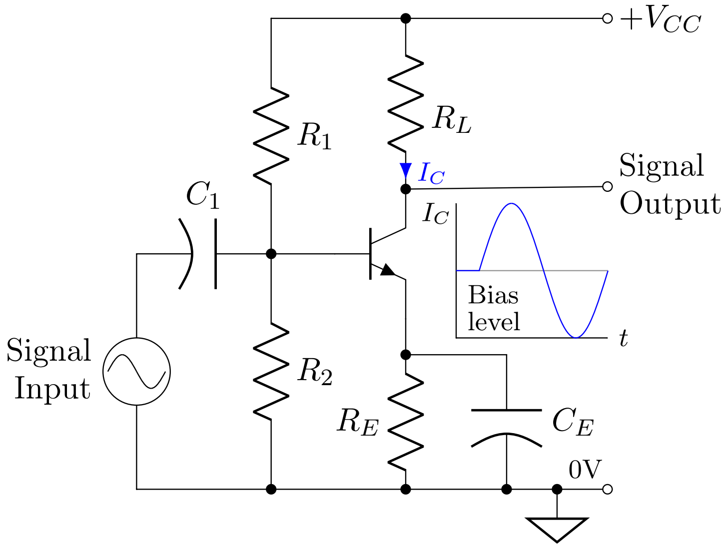
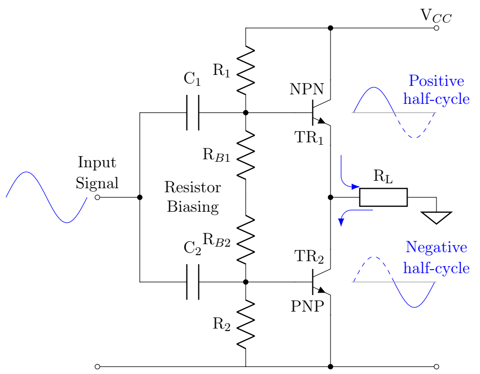


Figure 3.8: Illustration of a Class A amplifier.

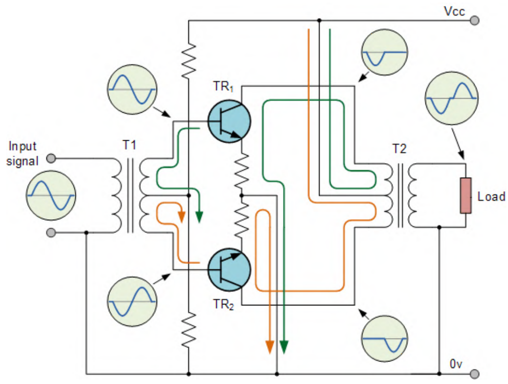
In practice the class A amplifier is used as low-powered applications, low-signal amplifier, a power amplifier, etc. The amplifier is ideal for these purposes, since it has no crossover distortion, but it is rarely used in high-powered applications, since the amplifier can only have a maximum efficiency of 25 %. [18, p. 920]

Class B

A class B amplifier contains at least two transistors, to create an overall more energy efficient amplifier compared to the class A amplifier. The transistors receive the same input signal with the same magnitude, but where each transistor only amplifies 180 degrees of the input AC signal. This is done by implementing a NPN and a PNP transistor in the class B circuit. The NPN transistor amplifies the positive half-cycle of the signal, and the PNP amplifies the negative half-cycle of the signal. This is illustrated in figure 3.9(a) and the type of circuit is called a push-pull setup. The name push-pull is based on the fact, that the NPN transistor *pushes* current into the load, when the input voltage is positive, and the PNP transistor *pulls* current from the load, when the input voltage is negative. [18, p. 921]



(a) Illustration of standard Class B amplifier.



(b) Illustration of Class B amplifier implementing transformers.

Figure 3.9: Illustration of Class B amplifiers.

Figure 3.9(b) shows another type of class B amplifier, where the signal is delivered to the transistors through a transformer. In this circuit both transistors are NPN-transistors, since the internal wiring will determine the phase of the AC signal each transistor will receive. After the signal has been amplified, it is then combined by another transformer. In this circuit both halves of the output signal will swing from zero to twice the quiescent current, which reduces the dissipation of energy as heat. [22]

The class B's ability to only draw power when needed, makes it more efficient than the class A amplifier. Specifically, 25 % more efficient, since the class B amplifier can have a maximum efficiency of 50 %. [18, p. 924]

However, the class B amplifier produces a higher level of distortion, than the class A amplifier, which is a disadvantage of the class B. This higher level of distortion is a combination of regular distortion and crossover distortion. Crossover distortion happens, when the signal magnitude is zero. This problem can in the meantime be minimised by combing the class A and B amplifier to a class AB amplifier. [18, p. 921-922; 923]

Class AB

The class AB amplifier is a combination of the class A and B amplifier. The crossover distortion in the class B design is overcome in the class AB amplifier by designing it so the two transistors only conduct between 180° and 360° of the input signal.
This is illustrated in figure 3.10. [23]

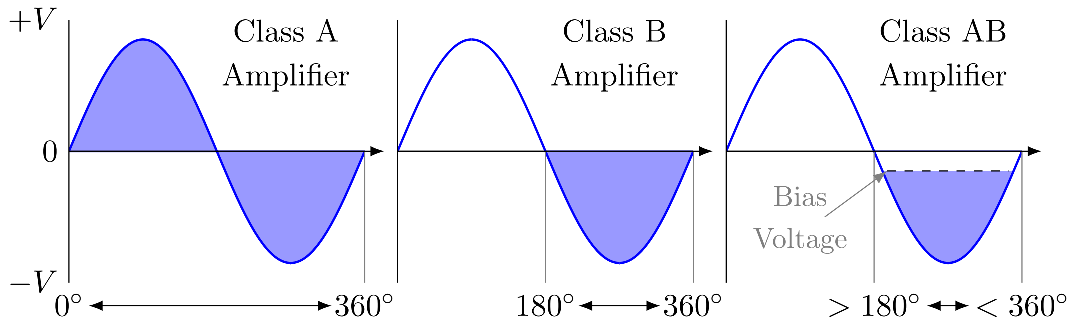
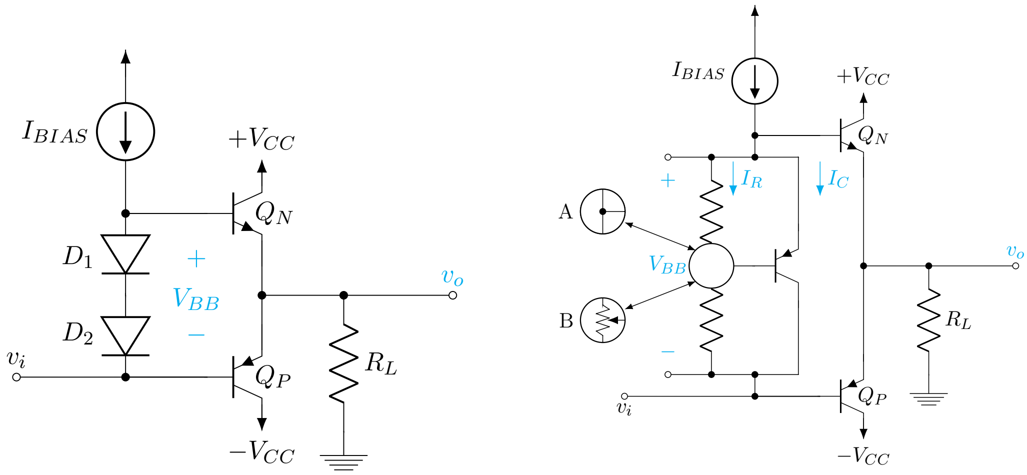


Figure 3.10: Illustration of which phase of a AC input signal a Class A, Class B and Class AB amplifier amplifies.

The amplifying process illustrated in figure 3.10 for the class AB amplifier, is also known as biasing. By restricting the class AB amplifier to only amplify between 180° and 360°, the crossover points, where the crossover distortion happens, are removed. To ensure that the amplifier still produces an output signal at the crossover points, a small voltage known as the biasing voltage [V_{BB}] is added. [18, p. 927-929] Thereby the class AB amplifier has minimal distortion, and an efficiency around 75 % can be achieved. [23]

There exist different ways of biasing a class AB amplifier. Two examples are using diodes and using the V_{BE} multiplier. Biasing using diodes works by connecting the transistors in the circuit to diodes and then passing a bias current, I_{BIAS}, through the diodes to produce a bias voltage V_{BB}. This in illustrated in figure 3.11(a). [18, p. 932] Alternatively, biasing using a multiplier works by adding a circuit of two resistors and a transistor to the class AB amplifier circuit. The multiplier is controlled by the resistors, which make it easy for manufacturers to tailor a AB amplifier to have a specific bias voltage. It is furthermore possible to add a potentiometer to the multiplier, so the AB amplifier can be calibrated, even after production.[18, p. 934-935]



(a) Illustration of biasing using diodes.

(b) Illustration of biasing using a V_{BE} multiplier without a potentiometer (A) and with a potentiometer (B).

Figure 3.11: Illustration of biasing in a Class AB amplifier.

3.2 Standards applying to amplifiers in high fidelity systems.

For a system to be classified as high fidelity, many standards and requirements must be met.

These standards have been determined by the International Electrotechnical Commission (IEC) and those cited in this report can be found in their entirety in the following publications:

- IEC 581 High fidelity audio equipment and systems; Minimum performance requirements. Part 1 - General, Part 6 - Amplifiers and Part 8 - Combination equipment. [24], [25], [26]
- IEC 1305 Household high-fidelity audio equipment and systems; Methods of measuring and specifying the performance. Part 1 - General and Part 3 - Amplifiers. [27], [28]
- IEC 60268-3:2013 Sound system equipment. Part 3 - Amplifiers. [29]
- IEC 61938:2013 Multimedia systems — Guide to the recommended characteristics of analogue interfaces to achieve interoperability . [30]

Due to the limited scope of this report, the limited possibility of testing the standards, and it being non-essential to explain all the standards, only a few of the requirements from the publications will be discussed in this section.

3.2.1 Rated output power

The amplifier must be able to deliver a power of at least 10 W RMS per channel. If the amplifier has multiple channels, it shall be rated when all channels are operating simultaneously at rated output power. Furthermore, the amplifier shall be able to deliver the rated output power for at least 10 minutes with all channels operating simultaneously at rated output power, and at ambient temperature between 15 °C and 35 °C.[25]

3.2.2 Effective frequency range

The effective frequency range or gain-limited effective frequency range is the frequency range within which the deviations from the required frequency response under standard conditions do not exceed stated limits.[29]
In IEC 581-6 it is stated that the minimum effective frequency range shall be from 40 Hz to 16 kHz, within a tolerance of $\pm$ 1,5 dB for non-equalized inputs related to 1 kHz and $\pm$ 2,0 dB for equalized inputs related to 1 kHz. [25]
Furthermore, it is stated that if the effective frequency range is broader than the minimum requirement, the tolerances given earlier still apply.
This means that for non-equalized inputs, the frequency response across the entire range must not deviate from the response at 1 kHz, with more than 1,5 dB.

3.2.3 Total harmonic distortion

The total harmonic distortion present when operated at the rated output power for amplifiers and at rated output voltage for preamplifiers, shall not exceed the following limits over the minimum effective frequency range:

- $\leq$ 0.5% for preamplifiers
- $\leq$ 0.5% for amplifiers
- $\leq$ 0.7% for integrated amplifiers

Furthermore, the specified distortion values shall not be exceeded at any output level above -26 dB, referred to rated output power. [25] When determining the value of the total harmonic distortion the most unfavourable value at any frequency between 63 Hz to 12,5 kHz, at any level from 1 dB below the rated (distortion-limited) output power or voltage, to −30 dB or 50 mW, whichever is lower, shall be given. [28]

3.2.4 Wideband signal-to-noise ratio

The signal-to-noise ratio or SNR describes the ratio between a desired signal and the background noise experienced by the system. The higher the ratio, the less noise is interfering and thus the desired signal is clearer. The minimum requirements for the signal-to-noise ratio is as follows:

- *Preamplifier*: $\geq 58\,\mathrm{dB}$
  This requirement shall be met at all settings of the volume control.
- *Power amplifier (without volume control)*: $\geq 81\,\mathrm{dB}$
- *Integrated amplifier*:
  a. $\geq 58\,\mathrm{dB}$
  This requirement shall be met at that setting of the volume control which gives the rated output power.
  b. $\geq 78\,\mathrm{dB}$
  This requirement shall be met at that setting of the volume control which gives an output power of 23 dB below the rated output power

3.2.5 Interoperability

To ensure compatibility between different electronics, IEC has defined some interoperability standards. In this report it is relevant to study the standards applying to in- and output impedances as well as in- and output voltages.

Between amplifier and loudspeaker

To ensure interoperability between an amplifier and a loudspeaker it is recommended that the output impedance of the amplifier is $< \frac{1}{10}$ of the input impedance of the loudspeaker. Input impedances of loudspeakers is typically $4\,\Omega$, $8\,\Omega$ or $16\,\Omega$, and the amplifier should be designed accordingly.

Between audio inputs and amplifier

To preserve flexibility and compatibility between audio inputs and amplifiers, a set of general purpose values for interfaces is stated. According to [30] the input impedance for audio inputs should be $\geq 22\,\mathrm{k\Omega}$ and the output impedance should be $\leq 2{,}2\,\mathrm{k\Omega}$. Similarly, a set of minimum and maximum voltages is defined for the audio signals. The voltage shall be between $0{,}2\,\mathrm{V_{RMS}}$ and $2\,\mathrm{V_{RMS}}$, though it is recommended that the voltage is at $0{,}5\,\mathrm{V_{RMS}}$.

Part II

Design and Implementation

Part II will delve into how the product is designed and implemented by the project group. In the first chapter the considerations and requirements for the product is detailed. Additionally, the limitations of the product is also covered.

Following the requirements, the product is divided into sub-systems. Each of these sub-systems are explored in individual chapters, where the theory, design and implementation of the systems will be covered in detail. Every sub-system will also undergo thorough testing which is covered in the appendix, with the conclusion of each test outlined and reported in the chapters with results in the appendices.

This leads to the following problem statement which the group has decided to solve in this project. Although the problem statement will be concluded in the part III of the report, it is part II which seeks to develop and document a solution for the following problem.

- How does one design and produce an FM receiver and an audio amplifier using analog circuits?
- Furthermore, how are these two systems designed and produced to work separately and connected together?

Chapter 4

Requirements

This chapter first covers the considerations for the product and then presents the requirements for the product. Some of the requirements are based on standards for both radio receivers and amplifiers, for example Hi-Fi standards. While other requirements are based on necessity or a desired feature. Furthermore, the limitations of the product is presented in both the considerations and standards.
Including and making proper requirements in a design phase is important because they have a direct effect on the quality of a product. The requirements and limitations in this chapter, will place the boundaries for the product that is to be designed and built in the following chapters.

4.1 Project considerations and boundaries

In this project the group has decided to create a double superheterodyne radio receiver as seen in figure 2.16 in section 2.5.2. This receiver was chosen because each subsidiary system within the receiver is very modular and can be designed for very specific functions. E.g. the subsidiary systems described in subsection 2.5.2. Finally, the receiver needs to be able to deliver a sound signal to an audio amplifier.
In addition, the group has decided to design a class A amplifier. This amplifier class was chosen because it delivers a signal to the loudspeaker with minimal distortion. When implementing this amplifier, there will be both a pre-amplifier and a power-amplifier in the same system. The group has also chosen to base the amplifier requirements off standards for Hi-Fi amplifiers, which is mentioned in subsection 3.2.

In Denmark the FM radio frequency band is between, 87.5 MHz to 108 MHz, which is the frequency band which the project's receiver will tune to. [31] Additionally, the radio can be tuned manually so that a user can select a desired broadcast channel. Therefor, the receiver will also need to tune accurately enough so that only a single broadcast channel is selected. Furthermore, to limit the complexity of the receiver and keep it within the scope of this report, the radio will only deliver a single sound channel (mono).

The group has also decided upon adding the following features to the overall system. An auxiliary input for the amplifier, which allows for a stereo signal to be sent through to the output. This feature will require that two identical amplifiers are built for each sound channel. This feature adds the need for the ability to switch between input signals. Although the auxiliary input is a stereo signal, the radio will only output a mono signal which is to be played

When working in the university's electronic laboratories the group faces the following environment, health and safety restrictions:

- Direct Current limit: 60 V
- Alternating Current: 25 V (RMS value)

However, with supervision the group is allowed to perform tests at higher voltages.

4.2 Amplifier requirements

The requirements for the amplifier is based upon the preliminary studies and design choices made by the group. The intention of these requirements is to have guidelines for how the product will be built and how it is tested. In addition, the functional requirements state the required features of the amplifier.
Several of the requirements for the amplifier are based on Hi-Fi standards, introduced in section 3.2.

Table 4.1: Amplifier requirements.

Technical requirements for the amplifier					
No.	*Description*	*Min.*	*Max.*	*Unit*	*Comments*
01	Total Harmonic Distortion	-	0.7	%	Specified value is for an integrated amplifier at rated output power. Cf. section 3.2.
02	Effective frequency range	20	$20 \cdot 10^3$	Hz	Cf. human hearing frequency range, see section 1.1
03	Signal-to-noise ratio	≥ 58	-	dB	Wide band ratio. Specified value is for an integrated amplifier at rated output power. Cf. section 3.2
04	Rated output power	10		W	Average value, cf. section 3.2
05	Effectiveness: Average	10	25	%	10% is the usual efficiency of a practical application and 25% is the maximum attainable efficiency of a class A amplifier, cf. [18, p. 920]
06	Input impedance (Line in)	≥ 22	-	kΩ	cf. section 3.2
07	Output impedance	-	≤ 0.8	Ω	With an 8 Ω load (speaker), cf. 3.2.
09	Typical input signal voltage between	0.2	2.0	V	cf. 3.2
Functional requirements					
No.	*Description*			*Comments*	
10	Stereo input (AUX input)			See section 4.1	
11	Stereo output			See section 4.1	
12	Mono radio-line input			See section 4.1	
13	Logarithmic volume control			See section 1.1	

4.3 Radio receiver requirements

The specific radio receiver requirements are based upon both international standards, national law and functional features. The intention of these requirements is to have guidelines for how the product will be built and how it is tested. In addition, the functional requirements state the required features of the receiver.

Table 4.2: Radio requirements.

Requirements for radio					
No.	*Description*	*Min.*	*Max.*	*Unit*	*Comments*
01	Output a signal within the audible frequency range	20	$20 \cdot 10^3$	Hz	Cf. human hearing frequency range, see section 1.1
02	Total harmonic distortion	-	≤ 1	%	Cf. 2.6
03	Channel separation	30	-	dB	At 6,3 kHz, cf. section 2.6
04	Unweighted signal-to-noise ratio	57	-	dB	Cf. section 2.6
05	Output impedance		≤ 2.2	Ω	Cf. section 3.2
06	Output voltage	0.2	2	V	Cf. section 3.2
07	Tuning range	87.5	108	MHz	Cf. The Danish FM band [31]
Functional requirements					
No.	*Description*			*Comments*	
08	Receives a transmitted FM signal			See section 4.1	
09	Output a sound signal			See section 4.1	
10	The radio can be tuned manually			See section 4.1	
11	The radio is precise enough to tune to a single radio broadcast channel			See section 4.1	

4.4 System Overview

In the following chapters the design of the modules, which as a whole will be a superheterodyne radio receiver, will be described in detail. The modules are the following:

- Atenna in chapter 8.
- Filters in chapter 6.
- Mixers in chapter 5.
- Demodulator in chapter 7.
- Audio Power Amplifier in chapter 9.

Figure 4.1 illustrates a overview of the superheterodyne radio receiver divided by modules. Each group of modules is divided in colours, for a better overview. The figure is based on figure 2.16 in section 2.5.2.

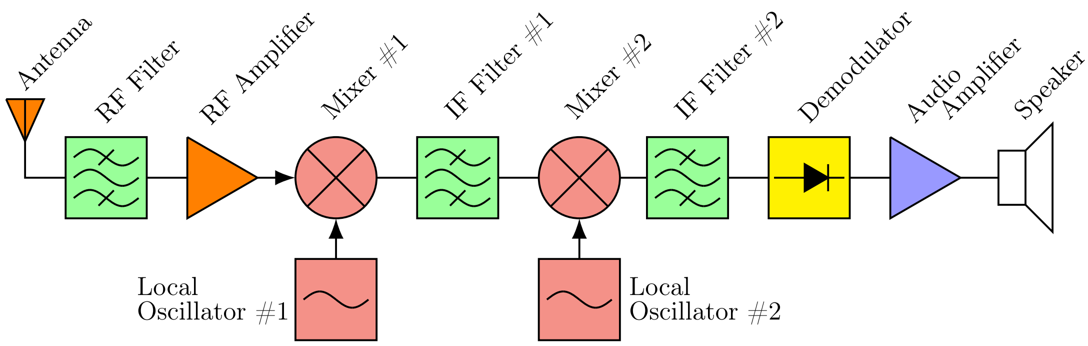


Figure 4.1: Illustration of the modules in the superheterodyne radio receiver

Chapter 5

Mixers

This chapter will focus on the design and theory of the frequency mixer(s). First basic analogue mixer theory will be introduced. The theory will then be used to choose the mixer topology, followed by a design phase, and finally test(s) of the mixer(s). The mixer modules in the superheterodyne radio receiver is highlighted in figure 5.1.

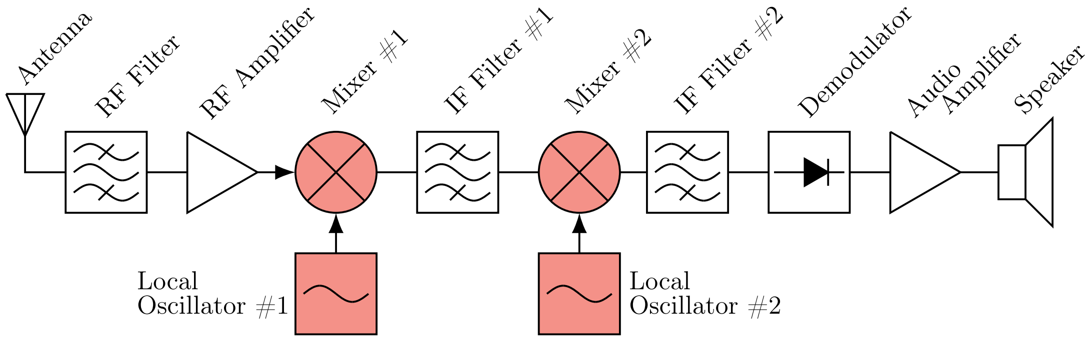


Figure 5.1: Mixer modules highlighted in the overall superheterodyne radio receiver design.

This section will not design the local oscillators, as it has instead been decided to use two external signal generators with an output impedance of $50\ \Omega$.

5.1 Analog Mixer Theory

In section 2.5, the use and operation of an ideal mixer was described. This section will expand upon this by characterising the attributes of practical analog mixers.

Balanced & Unbalanced Mixers

The most fundamental property of a mixer is its balance level. In the most basic mixer, referred to as an *unbalanced mixer* neither of the Local Oscillator (LO) nor the Radio Frequency (RF) signal will be filtered, and thus they will both be a part of the resulting output signal. For many applications this is unacceptable as it may be impractical to filter these undesired frequencies in subsequent circuits. In case it is sufficient to only suppress one of the input signals, a *single balanced mixer* is implemented. This configuration will suppress either the RF or the LO input. However in cases where neither of these configurations will be satisfactory, it is possible to use a *double balanced mixer* configuration, where both of the inputs are suppressed. While this configuration is generally considered the best (since it approaches the performance of the ideal mixer) it is also by far the most complicated to design.

Gain

In almost all types of mixers, there is an amplification of the input known as the gain G of the mixer. The gain is, as in any other electrical component, the difference between input and output in decibels [dB]. It can furthermore be described as the difference in signal amplitude or power. In this project it will be described as the power of the signal amplitude.
The equation for the power gain is seen in 5.1, where the output is denoted P_{OUT} and the input P_{RF}. The input is denoted with $_{RF}$, because the mixer will be used in a radio, and therefore the input will be radio frequencies.[18]

$$G = 10 \cdot \log_{10}\left(\frac{P_{OUT}}{P_{RF}}\right) \quad [\text{dB}] \tag{5.1}$$

The gain can be determined by testing the mixer and measure the input amplitude and the corresponding output amplitude of the signal, or by calculation. In the case of calculation the transfer function for the mixer is used. To find the transfer function for a mixer, the mixer type and design must be known, along with all the values of the internal components.
This is only done, when the mixer is designed in house. If the mixer is bought from a manufacturer, the gain can be found in the datasheet for the mixer.

5.2 Mixer Specification

The specifications for the design of the mixer are set up in tabel 5.1 and 5.2. These specifications define the scope limits of the test of the mixer(s). The tests can be found in appendix A, B and C.

Table 5.1: Specifications for the IF 1Mixer Circuit.

Mixer #1 Specifications				
Description	*Min.*	*Max.*	*Unit*	*Comments*
Conversion gain	20	-	dB	-
LO signal input	5	-	mV	-
RF input	1	-	mV	-
Frequency Range	≤ 87	≥ 107	MHz	-
Image Rejection Ratio	6	-	dB	-

Table 5.2: Specifications for the IF 2 Mixer Circuit.

Mixer #2 Specifications				
Description	*Min.*	*Max.*	*Unit*	*Comments*
Conversion gain	20	-	dB	-
LO signal input	5	20	mV	-
RF input	1	20	mV	-
Frequency Range	≤ 0.455	≥ 10.7	MHz	-
Image Rejection Ratio	6	-	dB	-

5.3 Mixer Topologies

This section will identify and describe a mixer circuit which can fulfil the presented specifications, and will be implemented in the following design phase.

The first mixer characteristic to be determined is, if it should be a passive or active mixer. The specification tables 5.1 and 5.2 both specify the mixers to have a gain of at least 20 dB. Therefore both mixers in this project will be active mixers, since only active mixers can have a gain above 0 dB.

There are many different active mixer configurations, but the three mentioned in this project will be the Gilbert Cell Mixer, the single transistor mixer and the differential amplifier mixer. Both the Gilbert Cell and differential mixer implements two differential pairs in the mixing and amplifications of the LO and RF signal. The differential pair is illustrated in figure 5.2.[18]

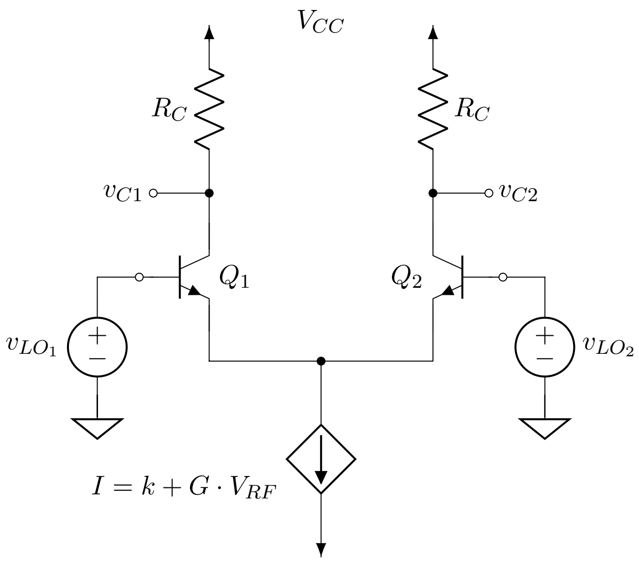


Figure 5.2: The basic Bipolar-Junction-Transistor (BJT) differential-pair configuration.

The Gilbert Cell, also known as a four-quadrant multiplier, is a double-balanced active mixer and the most widely used mixer in RF integrated circuits. The mixer works by having one of the input signals controlling the bias current of both differential pairs, while the other input signal drives the base electrodes of the transistors in the differential pair.

A basic Gilbert Cell Mixer configuration can be seen in figure 5.3, where the RF input controls the bias current, and the LO input drives the transistors.[32]

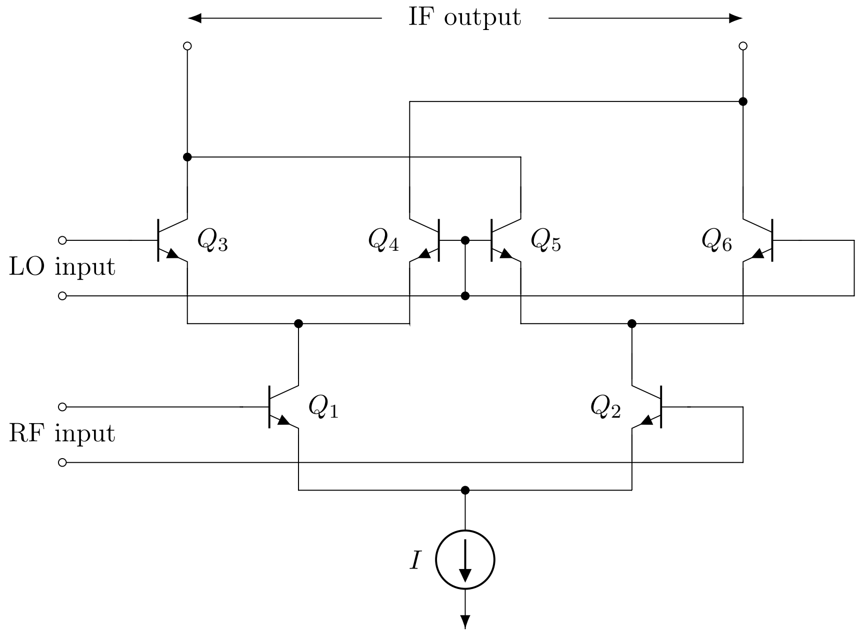


Figure 5.3: Illustration of a basic Gilbert Cell Mixer configuration.

The Gilbert Cell mixer has almost perfect performance, but has the disadvantage, that it is complicate to design. An example is, when the Gilbert Cell is used in a analogue circuit. Here the mixer requires, that a diode circuit ads distortion to one of the input signals. This added level of distortion is then equal and opposite to the distortion produced by the differential pair in the Gilbert Cell. Therefore the group has not opted for a Gilbert Cell mixer for this project.

The basic BJT transistor mixer, or single transistor mixer, is, in addition to the Gilbert Cell, a possible active mixer to use in a radio. The simple BJT mixer is often used in cheap and portable radios, where low cost is a priority. The mixer can have the required performance and conversion gain, but just barely, and with poor linearity. The most common configuration of a single transistor mixer is seen in figure 5.4.[33]

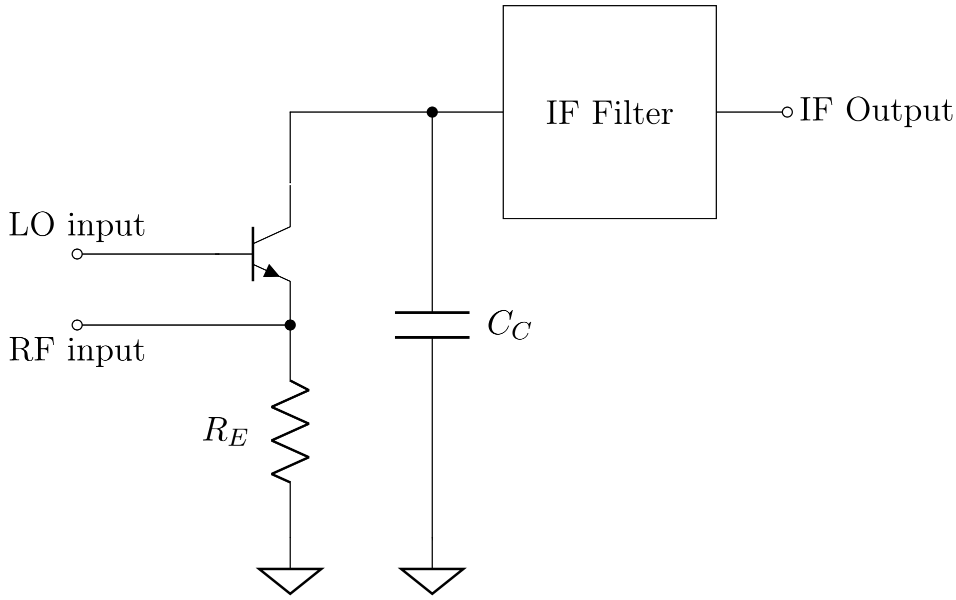


Figure 5.4: Illustration of the most common basic BJT transistor mixer or single transistor mixer.

The single transistor configuration in figure 5.4 uses the LO input to drive the base of the transistor, while the RF input is applied to the emitter of the transistor. The capacitor on the output is used to remove the RF and LO signals from the output. The configuration form the basis for almost all other transistor mixers.
Furthermore, an advantage to the single transistor mixer is, that it solves the Gilbert Cell problem of being complicated to design. Unfortunately it is not of a high enough quality for this project. The single transistor mixer has therefore, as the Gilbert Cell, not been opted for.

The last mixer configuration to be mentioned in this project is the differential amplifier mixer. Furthermore, it's the mixer the group has opted for to use in the radio. The differential amplifier mixer has some elements from the Gilbert Cell mixer, ex. both mixers implements two BJT differential pairs. The difference between the two are, that the differential amplifier mixer is single-balanced, whereas the Gilbert Cell is double-balanced. The former is therefore simpler to design and implement. In addition to being simpler, the differential amplifier mixer will also perform better, than the basic BJT transistor mixer and with better linearity. A more detailed description of the differential amplifier mixer is found in section 5.3.1.

5.3.1 The BJT differential amplifier mixer

The BJT differential amplifier, shown earlier in figure 5.2, amplifies the difference between V_{LO1} and V_{LO2} by the V_{RF} controlled current source, I. To design this circuit the V_{RF} controlled current source is implemented using a BJT common emitter configuration. Furthermore, since the local oscillator signal is not a balanced signal the differential pair will be fed in a single ended fashion. The complete differential mixing circuit (without biasing resistors) is shown in figure 5.5[18]:

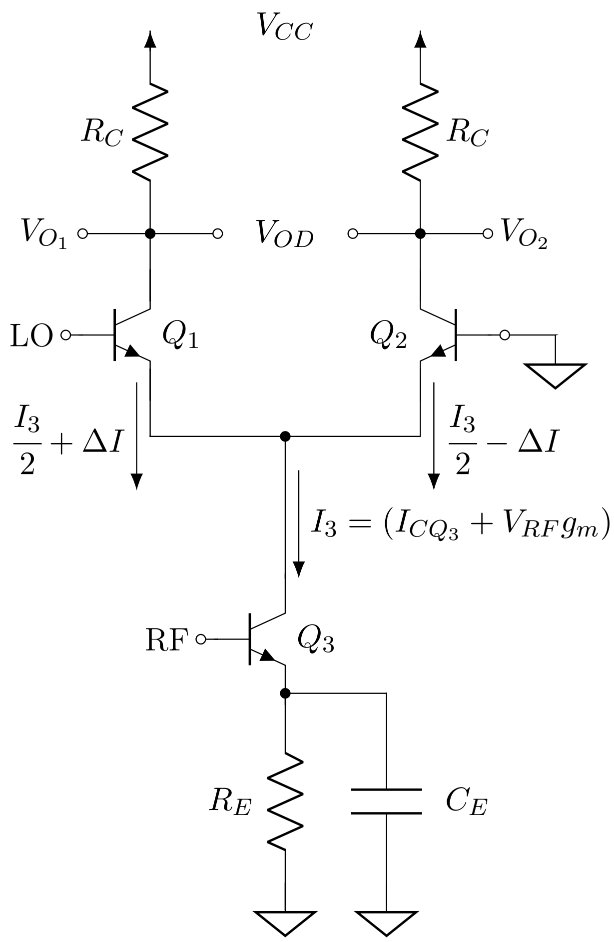


Figure 5.5: The basic BJT differential-pair mixer. Biasing networks are neglected.

In the calculations following, a subscript followed by a numbered suffix indicates the transistor number from figure 5.5. For example I_{C3} refers to the collector current in transistor Q3.

For BJT's the following apply:

$$I_C = \alpha I_E \qquad [A] \quad (5.2)$$

To simplify the following computations, it will be assumed that $\alpha = 1$.Thus equation 5.2 becomes:

$$I_C = I_E \qquad [A] \quad (5.3)$$

In figure 5.5 the following relations apply [18]:

$$V_{O1} = (V_{cc} - \frac{I_3}{2} R_c) - \Delta I R_C \qquad [V] \quad (5.4)$$

$$V_{O2} = (V_{cc} - \frac{I_3}{2} R_c) + \Delta I R_C \qquad [V] \quad (5.5)$$

and the total differential output can be expressed as[18]:

$$V_{OD} = V_{O2} - V_{O1} = 2\Delta I R_c \qquad [V] \quad (5.6)$$

Again assume that $\alpha = 1$, then equation 5.7 applies.

$$I_C = I_E \qquad [A] \quad (5.7)$$

Then for the differential amplifier shown in figure 5.5, the following two equations apply[18]:

$$I_{C1} = \frac{I_3}{1 + e^{-\frac{V_{LO}}{V_T}}} \qquad [A] \quad (5.8)$$

$$I_{C2} = \frac{I_3}{1 + e^{\frac{V_{LO}}{V_T}}} \qquad [A] \quad (5.9)$$

where:

V_T is the thermal voltage, it is approximately equal to 25 mV at 25 C°
[V]

ΔI is then found by taking the difference between equation 5.8 and equation 5.9 :

$$\begin{aligned}\Delta I &= I_{C1} - I_{C2} \\ &= \frac{I_3}{1+e^{-\frac{V_{LO}}{V_T}}} - \frac{I_3}{1+e^{\frac{V_{LO}}{V_T}}} \\ &= \frac{(I_3 + I_3 e^{\frac{V_{LO}}{V_T}}) - (I_3 + I_3 e^{-\frac{V_{LO}}{V_T}})}{(1+e^{-\frac{V_{LO}}{V_T}})(1+e^{\frac{V_{LO}}{V_T}})} \qquad [\mathrm{A}] \qquad (5.10)\end{aligned}$$

Further reducing equation 5.10 to equation 5.11.

$$\Delta I = I_3 \frac{e^{\frac{V_{LO}}{V_T}} - e^{-\frac{V_{LO}}{V_T}}}{e^{\frac{V_{LO}}{V_T}} + e^{-\frac{V_{LO}}{V_T}} + 2} \qquad [\mathrm{A}] \qquad (5.11)$$

Which can be simplified to:

$$\Delta I = I_3 \tanh\left(\frac{V_{LO}}{2V_T}\right) \qquad [\mathrm{A}] \qquad (5.12)$$

$$\tanh\left(\frac{V_{LO}}{2V_T}\right) \approx \frac{V_{LO}}{2V_T} \quad \text{when:} \quad \frac{V_{LO}}{2V_T} \ll 1 \qquad [\cdot] \qquad (5.13)$$

thus equation 5.12 becomes:

$$\Delta I \approx I_3 \frac{V_{LO}}{2V_T} \qquad [\mathrm{A}] \qquad (5.14)$$

As is evident from figure 5.5

$$I_3 = I_{CQ3} + V_{RF} \cdot g_m \qquad [\mathrm{A}] \qquad (5.15)$$

Which can be simplified to:

$$I_3 = I_{CQ3}(1 + \frac{V_{RF}}{V_T}) \qquad [\mathrm{A}] \qquad (5.16)$$

Where $V_{RF} \cdot g_m$ must be less than I_{CQ3}.

When equation 5.14 and 5.16 are combined:

$$\Delta I \approx I_{CQ3}(1 + \frac{V_{RF}}{V_T})\frac{V_{LO}}{2V_T}$$
$$\approx I_{CQ3}(\frac{V_{LO}}{2V_T} + \frac{V_{LO}V_{RF}}{2V_T^2}) \quad [A] \quad (5.17)$$

When equation 5.17 is used in conjunction with equation 5.6 the output voltage can be expressed as:

$$V_o = 2R_C I_{CQ3}(\frac{V_{LO}}{2V_T} + \frac{V_{LO}V_{RF}}{2V_T^2}) \quad [V] \quad (5.18)$$

5.3.2 Image Rejection Mixer

The image rejection mixer is a special type of mixer configuration which is able to suppress the image frequency. In its most basic form the image rejection mixer is made up of twF sub mixers (in this case the BJT differential mixer), two 90° phase shifters, and a summation/difference block. This configuration is shown in figure 5.6 [34]. The including mathematical example is provided to aid in the following explanation of the image rejection mixers mode of operation.

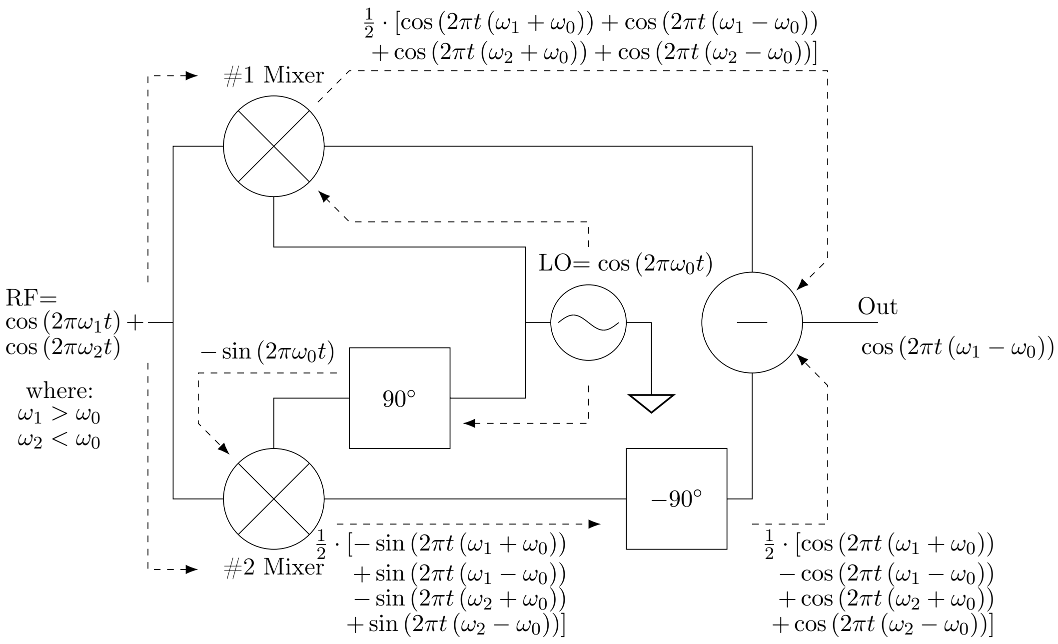


Figure 5.6: Image rejection mixer.

As described in section 2.5.2, the mixer converts two frequencies to the intermediate frequency: the signal of interest, as well as a complementary image frequency. It is decided that the system will utilise low side injection, therefore the frequency of the image and the desired frequency can be found with the following formula:

$$2\pi\omega_0 + IF = 2\pi\omega_1 \qquad [\mathrm{Hz}] \quad (5.19)$$

$$2\pi\omega_0 - IF = 2\pi\omega_1 \qquad [\mathrm{Hz}] \quad (5.20)$$

where:

IF is the intermediate frequency	[Hz]
ω_0 is the local oscillator frequency	[rad s^{-1}]
ω_1 is the frequency of the desired signal	[rad s^{-1}]
ω_2 is the image frequency	[rad s^{-1}]

Using figure 5.6 as a reference point, the desired signal and its complementary image frequency is used as the RF input:

$$RF = \cos(2\pi\omega_1 t) + \cos(2\pi\omega_2 t) \qquad [\mathrm{Hz}] \quad (5.21)$$

In a similar fashion, the local oscillator is a sinusoid of frequency ω_0:

$$LO = \cos(2\pi\omega_0 t) \qquad [\mathrm{Hz}] \quad (5.22)$$

Having defined the input variables of the mixer, figure 5.6 will be stepped through block by block.

In the first mixer the LO signal is mixed (i.e multiplied) with the RF signal. To simplify this the following trigonometric identity is used:

$$\cos(\theta)\cos(\phi) = \frac{1}{2}\left[\cos(\theta - \phi) + \cos(\theta + \phi)\right] \qquad [\cdot] \quad (5.23)$$

Thus the output of the first mixer can be expressed as:

$$\begin{aligned} M_1 &= \cos(2\pi\omega_0 t)\left[\cos(2\pi\omega_1 t) + \cos(2\pi\omega_2 t)\right] \\ &= \frac{1}{2}[\cos(2\pi t(\omega_1 - \omega_0)) + \cos(2\pi t(\omega_1 + \omega_0)) \\ &+ \cos(2\pi t(\omega_2 - \omega_0)) + \cos(2\pi t(\omega_2 + \omega_0))] \end{aligned} \quad [\mathrm{V}] \quad (5.24)$$

where:

M_1 is the output of the first mixer stage. [V]

Before the LO and RF signal can be mixed in the second mixer the LO is phase shifted by 90°. This phase shift convert convert the LO signal from a cosine function to a sine function, as shown in equation 5.25:

$$LO_{90^\circ} = \cos(2\pi\omega_0 t + \frac{\pi}{2}) = -\sin(2\pi\omega_0 t) \quad [\mathrm{V}] \quad (5.25)$$

where

LO_{90° is the 90°phase shifted LO signal. [V]

To find the simplified output of LO_{90° and RF being mixed together, another trigonometric identity is used:

$$\cos(\theta)\sin(\phi) = \frac{1}{2}\left[\sin(\theta + \phi) - \sin(\theta - \phi)\right] \quad [\cdot] \quad (5.26)$$

By applying equation 5.26, the output of the second mixer becomes:

$$\begin{aligned} M_2 &= -\sin(2\pi\omega_0 t)\left[\cos(2\pi\omega_1 t) + \cos(2\pi\omega_2 t)\right] \\ &= \frac{1}{2}[-\sin(2\pi t(\omega_1 + \omega_0)) + \sin(2\pi t(\omega_1 - \omega_0)) \\ &- \sin(2\pi t(\omega_2 + \omega_0)) + \sin(2\pi t(\omega_2 - \omega_0))] \end{aligned} \quad [\mathrm{V}] \quad (5.27)$$

where:

M_2 is the output of the first mixer stage [V]

Using known information, it is possible to further manipulate equation 5.27.

According to equation 5.19 and 5.20 the following relation must always hold true:

$$\omega_1 > \omega_0 > \omega_2 \qquad [\mathrm{rad\,s^{-1}}] \quad (5.28)$$

Therefore:

$$\omega_2 - \omega_0 < 0 \qquad [\mathrm{rad\,s^{-1}}] \quad (5.29)$$

Since the sine function is an odd function the following must apply:

$$\sin(\omega_2 - \omega_0) = -\sin|\omega_2 - \omega_0| \qquad [\mathrm{rad\,s^{-1}}] \quad (5.30)$$

When equation 5.30 is used in conjunction with equation 5.27 the very last term becomes negative:

$$\begin{aligned} M_2 = \frac{1}{2}[&-\sin(2\pi t(\omega_1+\omega_0)) + \sin(2\pi t(\omega_1-\omega_0)) \\ &-\sin(2\pi t(\omega_2+\omega_0)) - \sin(2\pi t|\omega_2-\omega_0|)] \end{aligned} \qquad [\mathrm{V}] \quad (5.31)$$

M_2 is then inserted into a -90° phase shifter, resulting in the following equation:

$$\begin{aligned} M_{2_{90^\circ}} &= \frac{1}{2}[-\sin(2\pi t(\omega_1+\omega_0) - \frac{\pi}{2}) + \sin(2\pi t(\omega_1-\omega_0) - \frac{\pi}{2}) \\ &\quad -\sin(2\pi t(\omega_2+\omega_0) - \frac{\pi}{2}) - \sin(2\pi t|\omega_2-\omega_0| - \frac{\pi}{2})] \\ &= \frac{1}{2}[\cos(2\pi t(\omega_1+\omega_0)) - \cos(2\pi t(\omega_1-\omega_0)) \\ &\quad + \cos(2\pi t(\omega_2+\omega_0)) + \cos(2\pi t|\omega_2-\omega_0|)] \end{aligned} \qquad [\mathrm{V}] \quad (5.32)$$

where

M_{90° is the -90° phase shifted Mixer #2 signal. [V]

Since:

$$\cos(-x) = \cos(x) \qquad [\cdot] \quad (5.33)$$

equation 5.32 can be simplified to:

$$\begin{aligned} M_{2_{90^\circ}} = \frac{1}{2}[&\cos(2\pi t(\omega_1 + \omega_0)) - \cos(2\pi t(\omega_1 - \omega_0)) \\ &+ \cos(2\pi t(\omega_2 + \omega_0)) + \cos(2\pi t(\omega_2 - \omega_0))] \end{aligned} \qquad [\mathrm{V}] \quad (5.34)$$

Finally the difference block computes the difference between M_1 and $M_{2_{90^\circ}}$:

$$\begin{aligned} Out = M_1 - M_{2_{90^\circ}} &= \frac{1}{2}[\cos(2\pi t(\omega_1 - \omega_0)) + \cos(2\pi t(\omega_1 + \omega_0)) \\ &+ \cos(2\pi t(\omega_2 - \omega_0)) + \cos(2\pi t(\omega_2 + \omega_0))] \\ &- \frac{1}{2}[\cos(2\pi t(\omega_1 + \omega_0)) - \cos(2\pi t(\omega_1 - \omega_0)) \\ &+ \cos(2\pi t(\omega_2 + \omega_0)) + \cos(2\pi t(\omega_2 - \omega_0))] \\ &= \cos(2\pi t(\omega_1 - \omega_0)) \end{aligned} \qquad [\mathrm{V}] \quad (5.35)$$

As is evident from equation 5.35, the image rejection mixer shown in figure 5.6 is able to suppress the image frequency.

5.4 Design

For the mixer, it has been decided to use BCM847DS transistors, a SMD variant of the BC547 with two matched transistors. While superior high frequency transistors exist, the BCM847DS was primarily chosen due to the authors limited access to high frequency transistors.

As described in section 5.3.1, the mixer is realised using a differential pair biased by a voltage controlled current source, in the form of a transistor in a common emitter configuration.
To simplify [18] the biasing phase, the differential pair is simplified to a differential half circuit as shown in figure 5.7:

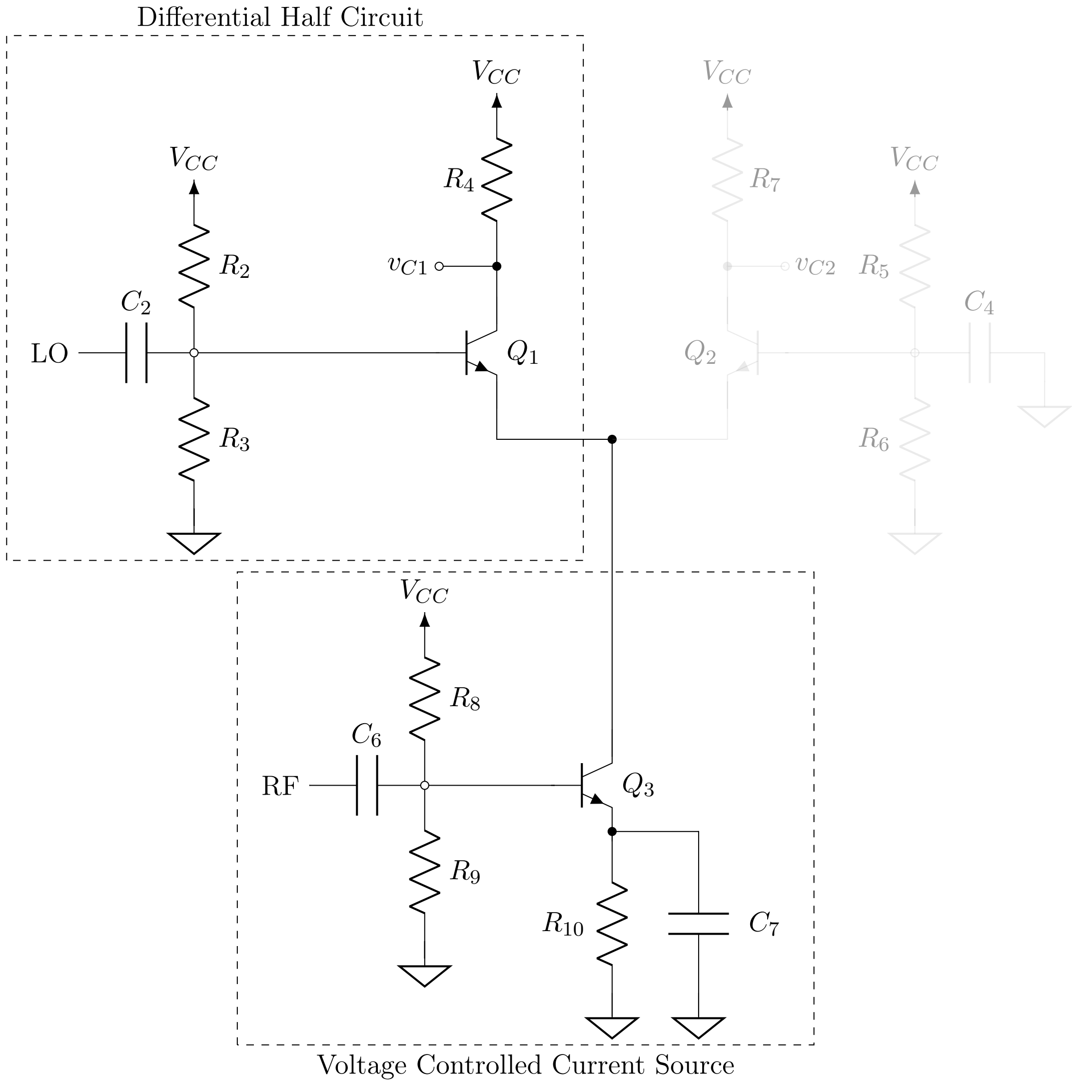


Figure 5.7: Illustration of a basic differential pair with expanded current source.

To bias the above circuit the following two conditions must be met:

- $I_{E1} = \frac{I_{CQ3}}{2}$ [A]
- $V_{E1} > V_{E3} + V_{BE3}$ [V]

Furthermore, to ensure the current source does not clip part of the negative signal swing, its I_{CQ3} must be sufficiently large. Similarly, to ensure the differential stage does not clip its output signal, its quiescent point should approximately be placed in the middle of its load line.

According to the BCM847DS's datasheet [35], its performance starts to deteriorate when I_C goes above ~20 mA. Due to this limitation it is decided to place the current sources I_{CQ} at 20 mA. Additionally, its V_{CEQ} is placed at 2 V and its V_E at 1 V. According to the transistor's datasheet, the chosen quiescent point results in a β of approximately 300, and a V_{BE} of 0.8 V. As a starting point for the biasing design, R_9 is set equal to 110 kΩ.

To find the value of R_8 the following formula is used [36]:

$$R_5 = \frac{V_{R_8}}{I_{R_8}} = 122.9 \cdot 10^3 \qquad [\Omega] \quad (5.36)$$

where:

$$V_{R_8} = V_{cc} - V_E - V_{BE} = 12 - 1 - 0.8 = 10.2 \qquad [\mathrm{V}] \quad (5.37)$$

and

$$I_{R_6} = \frac{I_C}{\beta} + \frac{V_E + V_{BE}}{R_9} = \frac{0.02}{300} + \frac{1.8}{110 \cdot 10^3} = 83 \cdot 10^{-6} \qquad [\mathrm{A}] \quad (5.38)$$

Next, to find R_{10} equation 5.39 is used[36]:

$$R_{10} = \frac{V_{10}}{I_{10}} = \frac{V_E}{I_C + \frac{I_C}{\beta}} = \frac{1}{0.02 + 6.7 \cdot 10^{-5}} = 49.8 \qquad [\Omega] \quad (5.39)$$

Next, to bias the differential half circuit, a similar approach is used. Since the I_{CQ3} divides equally into each of the emitters of the half circuits, I_{CQ1} can easily be deduced:

$$I_{CQ1} \approx I_{E1} = \frac{I_{CQ3}}{2} = 0.01 \qquad [\mathrm{A}] \quad (5.40)$$

Furthermore, from inspection it is evident that the following apply:

$$V_{E1} = V_{E2} = V_{CEQ3} + V_{E3} = 1 + 2 = 3 \qquad [\mathrm{V}] \quad (5.41)$$

Finally, a V_{CEQ1} point needs to be chosen. Since $V_{E1} = 3$ and $V_{CC} = 12$, a $V_{CEQ1} = 4$ results in Q point located approximately in the middle of the transistor load line. For this Q-point $\beta = 300$ and $V_{BE} = 0.75$V [35]. As when biasing the constant current source, R_3 is set equal to 110 kΩ as a starting point for the biasing design. To find the appropriate value for R_2 equation 5.36, 5.37, and 5.38 are again used to find that $R_4 = 122.4$ kΩ.

To find the value of R_4, equation 5.42 is used[36]:

$$R_4 = \frac{V_{CC} - V_{CE} - V_E}{I_C} = \frac{12 - 4 - 3}{0.01} = \frac{5}{0.01} = 500 \qquad [\Omega] \quad (5.42)$$

All of the calculated values for the bias resistors are then rounded to the nearest E96 resistor value, and subsequently inserted into LTSpice. In Spice the values of the resistors are then slightly tuned to accommodate for more detailed model parameters that the above equation's do no take into account. The final resistor values used in the two mixers are shown in figure 5.8. Figure 5.8 also shows the DC voltages and currents at various points in the circuit (as simulated by spice):

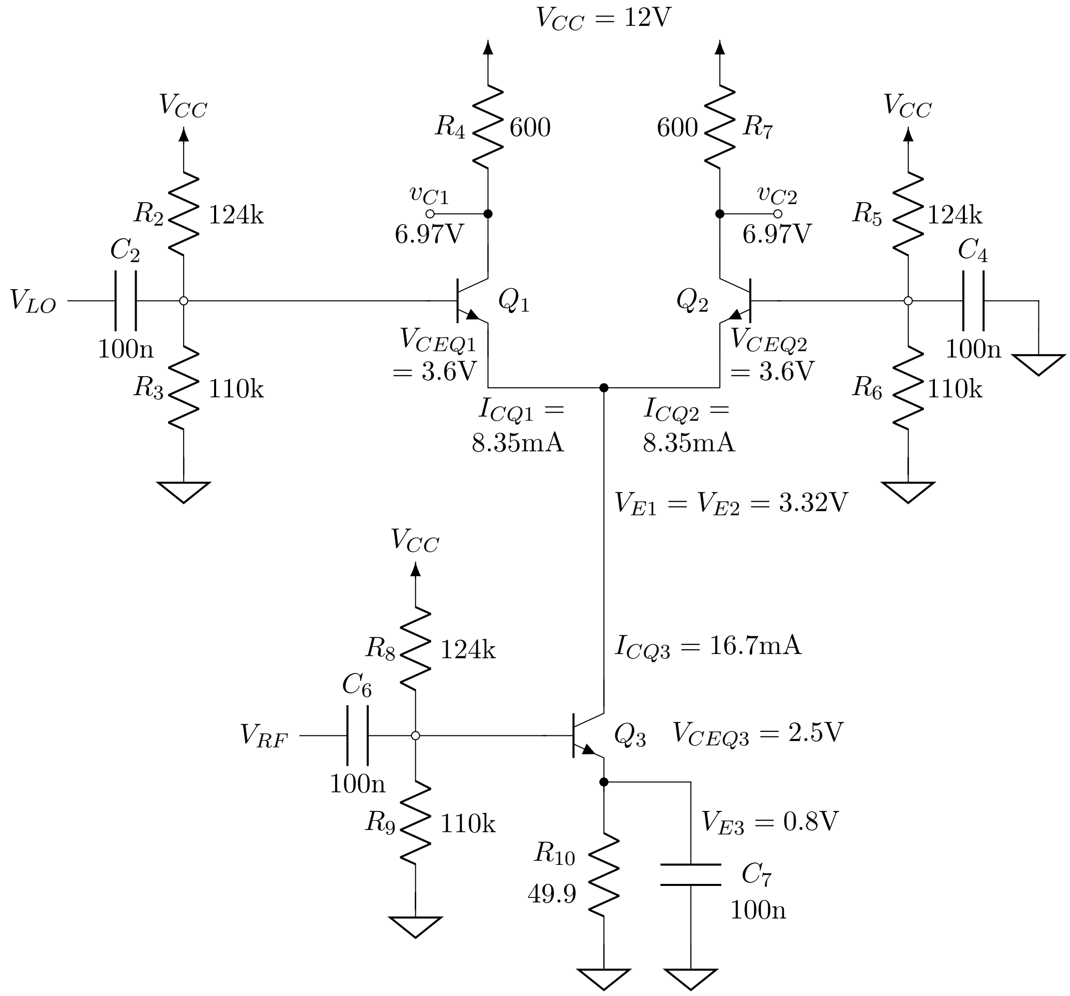
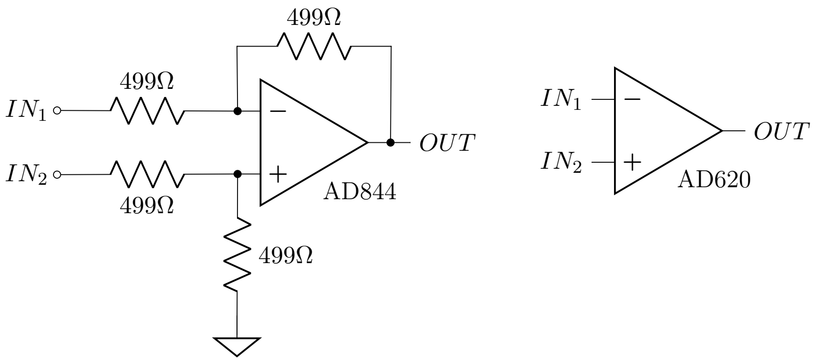


Figure 5.8: Mixer bias Network.

To convert the differential output signal produced by the mixer to a single ended signal, an operational amplifier is used. For the first mixer stage an AD844 high frequency op-amp is used in a differential configuration, where all the resistors have been set to 499 Ω. For the second mixer stage an AD620 instrumentation op-amp is used. Both configurations are shown in figure 5.9.



(a) Op-amp configuration used to combine the first mixer's differential output to a single ended signal.

(b) Instrumentation op-amp used to combine the second mixer's differential output to a single ended signal.

Figure 5.9: Op-amp configuration used to combine the differential signals.

Image rejection design

In section 5.3.2 it was described how it was possible to create an image rejection mixer using 90°phase shifters. However in reality it is very difficult to achieve a true 90° phase shift, and most literature seems to encourage the use of hybrid couplers to achieve this. Unfortunately it was not possible to acquire sub 1 GHz 90° hybrid couplers for this project. To overcome this limitation the authors came up with a way to use the phase shift of simple RC filters to achieve the same effect. At the 3 dB frequency of a simple low pass RC filter the phase shift is equal to -45°. Similarly a high pass filter has a +45° phase shift at its 3 dB frequency. By combining the low pass and high pass filters phase shifts, it is possible to achieve the intended relative signal phase shift of 90°. Figure 5.10 illustrates a conceptual block model of the setup:

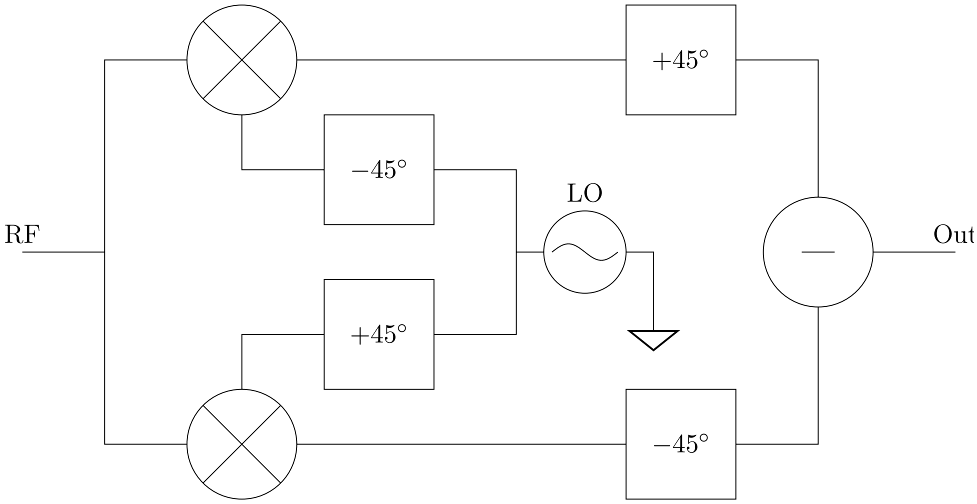


Figure 5.10: System diagram of the Double Balanced Image Rejection Mixer.

To realise the first 90° phase shift presented in the above implementation, the following circuit is used:

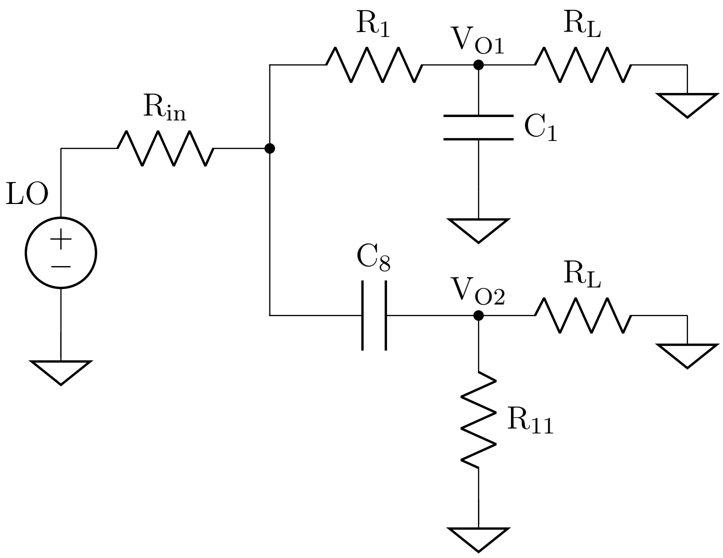


Figure 5.11: First 90° phase shifter.

To find the appropriate values for R_2, R_4, C_1, and C_2 the circuit will have to be analysed, and the transfer function must be found. To simplify this procedure the load resistance is assumed to be infinite.

First the impedance of each filter branch needs to be identified.

The input impedance of the RC high pass filter is:

$$Z_H = R4 + \frac{1}{sC_2} \qquad [\Omega] \quad (5.43)$$

where

Z_H is the input impedance of the high pass filter. [Ω]

Similarly, the impedance of the RC low pass filter is:

$$Z_L = R2 + \frac{1}{sC_1} \qquad [\Omega] \quad (5.44)$$

where

Z_L is the low pass transfer function. [Ω]

Using equation 5.43 and 5.44 a simplified equivalent diagram is drawn:

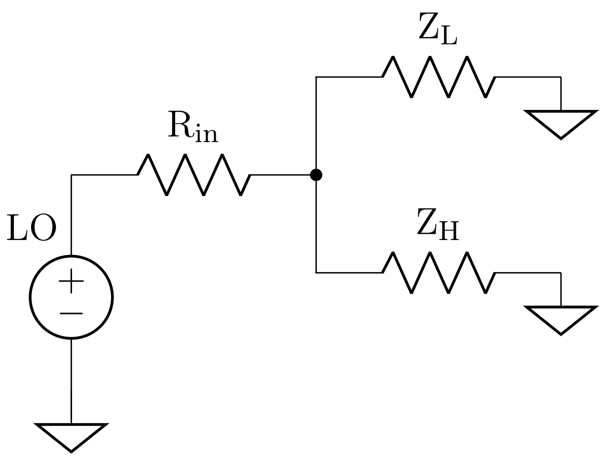


Figure 5.12: Two filters in parallel

The next step is to find the parallel impedance of the low and high pass filters:

$$
\begin{aligned}
Z_L||Z_H &= \frac{Z_L \cdot Z_H}{Z_L + Z_H} \\
&= \frac{(R2 + \frac{1}{sC_1})(R4 + \frac{1}{sC_2})}{R2 + \frac{1}{sC_1} + R4 + \frac{1}{sC_2}} \\
&= \frac{s^2 R_2 R_4 C_1 C_2 + s(R_2 C_1 + R_4 C_2) + 1}{s^2(C_1 C_2 (R_2 + R_4) + s(C_1 + C_2)} \qquad [\Omega] \qquad (5.45)
\end{aligned}
$$

Resulting in the following equivalent diagram:

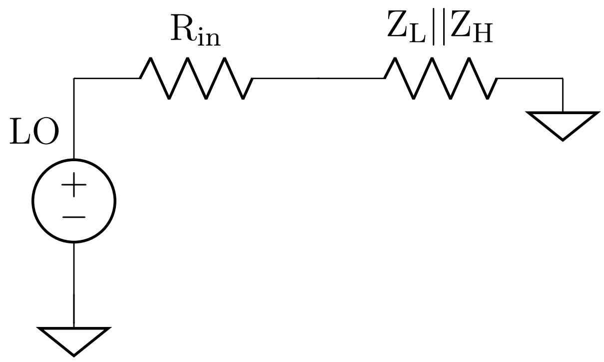


Figure 5.13: Two filters in parallel

Next the voltage across $Z_L||Z_H$ is found using the voltage divider formula:

$$
\begin{aligned}
V_{across} &= V_{IN} \frac{Z_L||Z_H}{R_{In} + Z_L||Z_H} \\
&= \frac{\dfrac{s^2 R_2 R_4 C_1 C_2 + s(R_2 C_1 + R_4 C_2) + 1}{s^2(C_1 C_2 (R_2 + R_4) + s(C_1 + C_2)}}{R_{In} + \dfrac{s^2 R_2 R_4 C_1 C_2 + s(R_2 C_1 + R_4 C_2) + 1}{s^2(C_1 C_2 (R_2 + R_4) + s(C_1 + C_2)}} \\
&= V_{IN} \frac{s^2 R_2 R_4 C_1 C_2 + s(R_2 C_1 + R_4 C_2) + 1}{s^2(C_1 C_2 (R_2 R_{in} + R_4 R_{in} + R_2 R_4)) + s(R_2 C_1 + R_4 C_2 + R_{in}(C_1 + C_2))} \qquad [\text{V}] \qquad (5.46)
\end{aligned}
$$

Finally, to find the value at V_{O1} and V_{O2} the standard RC lowpass and high pass transfer function needs to be multiplied by equation 5.46.

The standard RC lowpass filter transfer function is:

$$LP = \frac{1}{sR_2C_1 + 1} \quad [\cdot] \quad (5.47)$$

and the RC highpass transfer function is:

$$HP = \frac{sR_4C_2}{sR_2C_2 + 1} \quad [\cdot] \quad (5.48)$$

Thus V_{O1} is:

$$\begin{aligned}\frac{V_{O1}}{V_{IN}} &= \frac{LP \cdot V_{across}}{V_{IN}} \\ &= \frac{C_2\,R_4\,s + 1}{\begin{array}{r}C_1R_{in}s + C_1R_{in}s + C_2(R_{in}s + C_1\,R_2\,R_4\,s^2 \\ + R_4\,s + C_1\,R_2R_{in}s^2 + C_1\,R_4R_{in}s^2 + 1)\end{array}}\end{aligned} \quad [\cdot] \quad (5.49)$$

Similarly for V_{O2}:

$$\begin{aligned}\frac{V_{O2}}{V_{IN}} &= \frac{HP \cdot V_{across}}{V_{IN}} \\ &= \frac{C_2\,R_4\,s\,\,(C_1\,R_2\,s + 1)}{\begin{array}{r}C_1\,R_2\,s + C_2\,R_4\,s + C_1R_{in}s + C_2R_{in}s \\ + C_1\,C_2\,R_2\,R_4\,s^2 + C_1\,C_2(\,R_2R_{in}s^2 + \,R_4R_{in}s^2) + 1\end{array}}\end{aligned} \quad [\cdot] \quad (5.50)$$

If it is assumed that $C_1 = C_2$ and $R_2 = R_4$, then equation 5.49 and 5.50 can be simplified to:

$$\frac{V_{O1}}{V_{IN}} = \frac{1}{1 + C_1(R_1 + 2R_{in})s} \quad [\cdot] \quad (5.51)$$

and

$$\frac{V_{O2}}{V_{IN}} = \frac{C_1R_1s}{1 + C_1(R_1 + 2R_{in})s} \quad [\cdot] \quad (5.52)$$

Since the both transfer functions have a pole at $s = -\frac{1}{C_1(R_1+2R_{in})}$ the cutoff frequency can be expressed as:

$$f_c = \frac{1}{2\pi C_1(R_1 + 2R_{in})} \qquad [\text{Hz}] \quad (5.53)$$

It is also important to note, that while the low pass filter will have 0 dB gain in its pass-band and therefore in extension a cut off frequency gain of -3dB relative to the input, this is not the case for the high pass filter. For the high pass filter the gain in the pass band will be:

$$\text{gain}_{\text{pass}} = 20\log_{10}\left(\frac{C_1R_1}{C_1(R_1 + 2R_{in})}\right) \qquad [\cdot] \quad (5.54)$$

and therefore the cutoff frequency gain is:

$$\text{gain}_{\text{Cut Off}} = 20\log_{10}\left(\frac{C_1R_1}{C_1(R_1 + 2R_{in})}\right) - 3 \qquad [\cdot] \quad (5.55)$$

Since the gain of the two filter should under ideal conditions be equal, the R_1 resistor should ideally be much greater than R_{in} to ensure that the high pass's pass band gain approaches unity. Unfortunately this approach does not take the differential mixer's input impedance into account. The input impedance of the differential mixer can be found with equation 5.56 [18]:

$$R_{in} = (R_2||R_3)||R_\pi \approx R_\pi \qquad [\Omega] \quad (5.56)$$

Where R_π is a small signal parameter for the BJT transistor. R_π is defined as[18]:

$$R_\pi = \frac{\beta_{ac}}{g_m} \qquad [\Omega] \quad (5.57)$$

where gm is [18]:

$$g_m = \frac{I_{CQ}}{V_T} \qquad [\text{S}] \quad (5.58)$$

By using the following assumption

$$\beta_{AC} = \beta_{DC} \qquad [\cdot] \quad (5.59)$$

it is possible to calculate R_π using the known properties of the circuit.

Assuming that $V_T = 25$ mV g_m becomes:

$$g_m = \frac{7.5}{25} = 0.3 \qquad [\text{S}] \quad (5.60)$$

Since $\beta \approx 300$, R_π (and thus R_{in} becomes:

$$R_\pi = \frac{300}{0.3} = 1 \qquad [\text{k}\Omega] \quad (5.61)$$

Thus, R_1 should also ideally be much less than R_π to ensure that the maximum voltage is transferred to the mixer. As it is not possible for R_1 to be much greater than R_{in} (50 Ω) and much less than R_π (1 kΩ) compromises must be made. Through simulation experimentation, it was determined that an R_1 value of 150 Ω provided optimal results. Since the second mixers LO input is a constant 10.245 MHz, it is possible to solve for C_1 using equation 5.53:

$$C_1 = \frac{1}{2\pi(R_1 + 2R_{in})f_c}, C_1 = \frac{1}{2\pi(150 + 100)10.245 \cdot 10^6} \approx 62 \ [\text{pF}] \quad (5.62)$$

Using a similar approach for the first mixer (though with $f_C = 86.3$ MHz) $C_1 = 7.3 \approx 6.8$ pF.

Finding the values for the second 90° phase shift is much simpler as the standard equation for a RC low and high pass (equation 5.47 and 5.48).

In the first mixer the filter cut off frequency must be tuned to the first IF (10.7 MHz). The following resistor and capacitor value full fill this:

- $C_{15} = C_{16} = = 430$ pF
- $R_{25} = R_{30} = 35\ \Omega$

In the second mixer the filters cut off frequency must be tuned to the second IF (0.455 MHz). The following resistor and capacitor value full fill this:

- $C_{15} = C_{16} = = 10$ nF
- $R_{21} = C_{22} = 35\ \Omega$

5.5 Test & Conclusion

Initial testing of the first image rejection mixer showed that it was essentially non-functional when the 90° phase filters were connected to the local oscillator. It is theorised that this is due to the poor high frequency performance of the selected capacitors. However, this hypothesis was not tested further, and instead it was decided that going forward the first mixer would be used in a standalone configuration (i.e. without image rejection abilities). In the gain conversion test of the first mixer (documented in Appendix A) the mixer was shown to have a -25 and -10 dB conversion gain when converting 107 and 87 MHz respectively to the IF frequency. These results are very poor, and do not full fill the designated specification of minimum conversion gain of 20 dB. In fact, the results are on par with what could be expected from a passive mixer. One interesting aspect of the results is the large difference between the gain when converting from 87 MHz as opposed to 107 MHz. This discrepancy indicates that the chosen transistors are operating outside their pass band, and are therefore attenuating the signal.

In contrast to the first mixer, the second mixer (which operates at much lower frequencies) performs far better. The conversion gain was tested (documented in Appendix B to be 25 dB, which is above the target minimum. Similarly the image rejection test (documented in Appendix C of the second mixer was also a success. It proved that the design was able to suppress the image frequency by 20 dB, which is well above the specified minimum target.

In conclusion, the designed circuits operate as intended at lower frequencies (less than 10.7 MHz, however more care should have been taken when choosing components for high frequency operation.

Chapter 6

Filters

This chapter will describe how to design a filter by using the low-pass prototype then transforming and scaling to the desired type of filter and frequency. This method is then used to design three filters used at different locations in the receiver design. The filters designed for this report are henceforth referred to as RF-filter, IF_1-filter and IF_2-filter, and will be described in detail in their own section later on. The filter modules in the superheterodyne radio receiver is highlighted in figure 6.1.

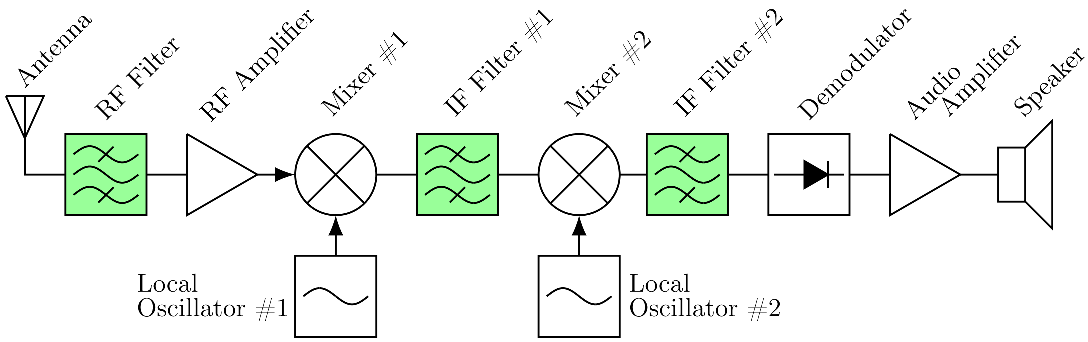


Figure 6.1: Filter modules highlighted in the overall superheterodyne radio receiver design.

6.1 Introduction

Often in electronics it is desired to work only at specific frequencies or as the introduction of unwanted frequencies components can cause the system performance to deteriorate. To remove these undesired signals a filter is used. When designing a radio receiver the undesired signal is located either at a frequency called the image frequency, which is described in section 2.5 or at the frequency of the neighbouring radio stations depending on where in the radio the signal is.

6.2 Basic filter design theory

The contents of this section is based upon the book "RF Circuit Design" by Chris Bowick [37], unless otherwise stated.

In order to simplify the process of designing filters, at one point it became necessary to standardise the way filters were presented so as to make it easier to compare different types of filters. This standardisation resulted in filters normalised for a cutoff frequency of 1 rad s^{-1} and source and load resistors of 1 Ω. The performance and characteristics of different filters then became easy compare, and when a specific type is chosen that fits ones needs it is then possible to reach realisable component values through a process called scaling.

The low-pass prototype is a low-pass filter normalised as described above, and the characteristic response of a filter like this is illustrated in figure 6.2.

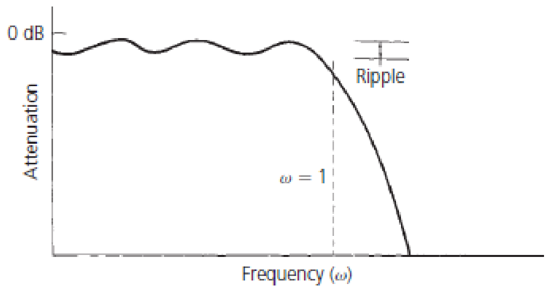


Figure 6.2: Illustration of a normalised low-pass response. [37]

This response changes with the type of filter and there exists many different types of filters. As it is outside the scope this report to describe and compare them all, instead only two different types of filters are chosen and their attenuation characteristics are compared. The two types filters discussed are the Butterworth and the Chebyshev types.

6.2.1 The Butterworth Filter

The Butterworth filter has the flattest response in the passband, compared to other types of filters, and doesn't contain any ripples i passband. An example of the response of a Butterworth filter can be seen in figure 6.3

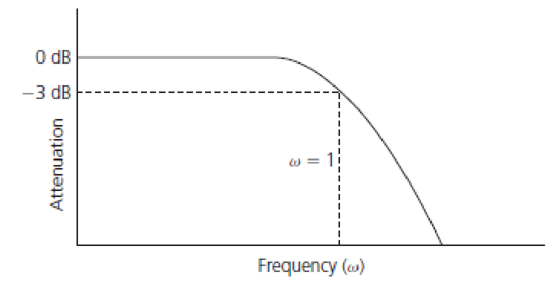


Figure 6.3: Illustration of a Butterworth low-pass response. [37]

It is clear that the Butterworth response is free of any ripple in the passband, but the steepness of the attenuation beyond the cutoff frequency is not very great. The attenuation of a Butterworth filter at a given frequency is given by

$$A_{\mathrm{dB}} = 10 \cdot \log_{10}\left(1 + f_{\mathrm{r}}^{2n}\right) \qquad \text{[dB]} \tag{6.1}$$

where

- f_r is the ratio between a given frequency and the cutoff frequency of the filter.
- n is the order of the filter.

When the attenuation is calculated for different frequency ratios at different orders and then plotted in the same graph, one gets a family of curves as the one seen in figure 6.4

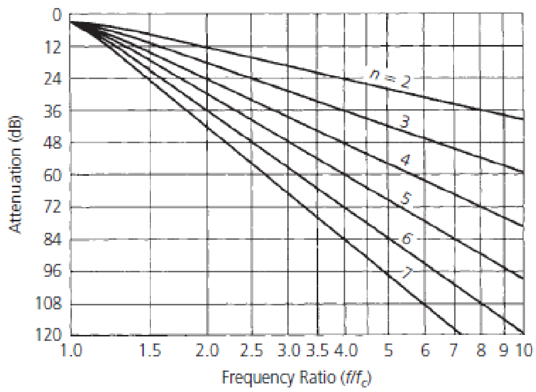
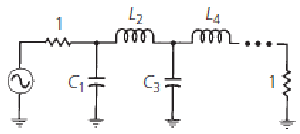


Figure 6.4: Illustration of the attenuation characteristics for Butterworth filters. [37]

This plot gives a great overview of the attenuation characteristics for a Butterworth filter of a given order and helps in the design of a filter. To give an example on how to use the plot, take the frequency ratio of 2 which corresponds to the frequency that is double that of the cutoff frequency. Then choose an order at which you wish to know the attenuation, for example 3, and find where the line corresponding to this order crosses the vertical line at the chosen frequency ratio. From this point, find the attenuation on the scale on the left of the plot, in this case 24 dB. From this example it is shown that for a 3rd order Butterworth filter, the attenuation at double the cutoff frequency is 24 dB.

When designing a filter, first the order must be chosen as this determines how many components there will be in the filter, and what value each component has. For a given frequency ratio and desired attenuation, figure 6.4 can be used to determine the necessary order of the filter. When the order has been chosen, a table of low-pass prototype element values is used to determine the component values. Each type of filter has its own table of element values, which depend on the ratio between the source and load resistors. The table for Butterworth filters with equal terminations and orders between two and seven is shown in table 6.5



n	C_1	L_2	C_3	L_4	C_5	L_6	C_7
2	1.414	1.414					
3	1.000	2.000	1.000				
4	0.765	1.848	1.848	0.765			
5	0.618	1.618	2.000	1.618	0.618		
6	0.518	1.414	1.932	1.932	1.414	0.518	
7	0.445	1.247	1.802	2.000	1.802	1.247	0.445
n	L_1	C_2	L_3	C_4	L_5	C_6	L_7

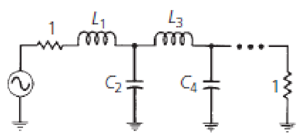


Figure 6.5: Table of low-pass element values for equal termination ($R_s = R_l$) Butterworth filters. [37]

At the top and bottom of the table is shown how each of the components in the filter is placed, depending on whether a shunt element or series element is chosen as the first element. At times it is necessary to design filters where the termination is not equal and a different table of values is used, with different values depending on the ratio between the source and load resistors. These tables and their calculation is deemed outside the scope of this report and can be studied by the reader in [37].

6.2.2 The Chebyshev Filter

Another type of filter is the Chebyshev filter. This filter has a steeper attenuation beyond the cutoff frequency than the Butterworth filter, but has ripple in the passband instead. As the allowed amount of ripple is increased the attenuation curve of the filter gets steeper as well. To demonstrate this, a 3rd order Butterworth filter and a 3rd order Chebyshev filter can be compared by overlaying the first over the other as seen in figure 6.6.

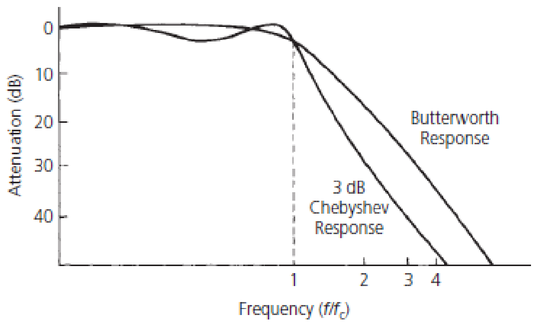


Figure 6.6: Illustration that shows the difference between equal order Chebyshev and Butterworth filters. The Chebyshev filter has a ripple of 3 dB in the passband. [37]

Comparing the two filters it is clear that the Chebyshev filter has better attenuation as the frequency increases. The Chebyshev response in fact quickly approaches an attenuation 10 dB larger than the Butterworth response as the frequency increases.

As with the Butterworth filter it is possible to calculate the attenuation of a Chebyshev filter as well. However the attenuation now depends on both the order and the ripple in the passband. The attenuation of the Chebyshev filter can be calculated using equation (6.2)

$$A_{\mathrm{dB}} = 10 \cdot \log_{10}\left(1 + \varepsilon^2 \cdot C_n^2 \cdot \left(\frac{\omega}{\omega_c}\right)'\right) \quad [\mathrm{dB}] \tag{6.2}$$

Where

- $C_n^2 \cdot \left(\frac{\omega}{\omega_c}\right)'$ is the nth order Chebyshev polynomial evaluated at $\left(\frac{\omega}{\omega_c}\right)'$. The first seven polynomial can be seen in the table in table 6.1
- ε is a parameter depending on the allowed ripple. It is given by $\varepsilon = \sqrt{10^{\frac{R_{\mathrm{dB}}}{10}} - 1}$ where R_{dB} is the allowed passband ripple

Beware that $\left(\frac{\omega}{\omega_c}\right)'$ does not equal $\left(\frac{\omega}{\omega_c}\right)$, and has to be calculated by using equation (6.3)

Table 6.1: The first seven orders of Chebyshev filters.

n	***Chebyshev Polynomial***
1	$\frac{\omega}{\omega_c}$
2	$2(\frac{\omega}{\omega_c})^2 - 1$
3	$4(\frac{\omega}{\omega_c})^3 - 3(\frac{\omega}{\omega_c})$
4	$8(\frac{\omega}{\omega_c})^4 - 8(\frac{\omega}{\omega_c})^2 + 1$
5	$16(\frac{\omega}{\omega_c})^5 - 20(\frac{\omega}{\omega_c})^3 + 5(\frac{\omega}{\omega_c})$
6	$32(\frac{\omega}{\omega_c})^6 - 48(\frac{\omega}{\omega_c})^4 + 18(\frac{\omega}{\omega_c})^2 - 1$
7	$64(\frac{\omega}{\omega_c})^7 - 112(\frac{\omega}{\omega_c})^5 + 58(\frac{\omega}{\omega_c})^3 - 7(\frac{\omega}{\omega_c})$

$$\left(\frac{\omega}{\omega_c}\right)' = \left(\frac{\omega}{\omega_c}\right) \cdot \cosh\left(\frac{1}{n} \cdot \operatorname{arccosh}\left(\frac{1}{\varepsilon}\right)\right) \quad [\cdot] \quad (6.3)$$

By varying the order and allowed ripple of the filter it is possible to get similar families of curves as for the Butterworth filter. The curves for filters with a ripple of 0.01 dB and 1.0 dB can be seen in figures 6.7 and 6.8.

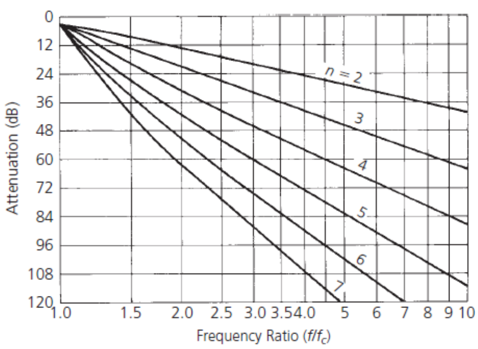


Figure 6.7: Illustration of the attenuation characteristics for Chebyshev filters with a passband ripple of 0.01 dB. [37]

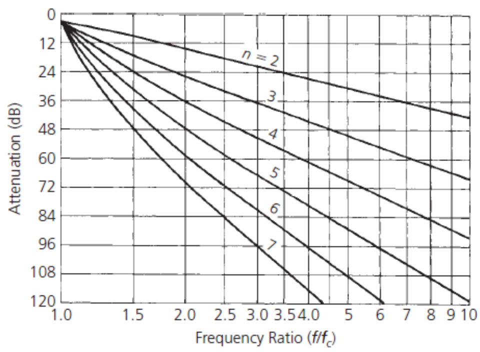


Figure 6.8: Illustration of the attenuation characteristics for Chebyshev filters with a passband ripple of 1.0 dB. [37]

Lastly the element values for a Chebyshev filter can be found using tables similar to those for the Butterworth filter. However, as Chebyshev filters of order n where n is an even number, does not function correctly if they have equal terminations, the tables have been expanded with different source-to-load resistance ratios for each order. [37] An example of this is shown in table 6.2.

By using the same technique described earlier with the Butterworth filter, it is possible to design a normalised Chebyshev filter low-pass filter using the attenuation curves and

Table 6.2: Table of low-pass element values for a second order Chebyshev filter with 0.1 dB ripple. Note the different values for the different ratios between source and load terminations. [37]

n	R_S/R_L	C_1	L_2	C_3	L_4
2	1.355	1.209	1.638		
	1.429	0.977	1.982		
	1.667	0.733	2.489		
	2.000	0.560	3.054		
	2.500	0.417	3.827		
	3.333	0.293	5.050		
	5.000	0.184	7.426		
	10.000	0.087	14.433		
	∞	1.391	0.819		

tables of element values.

6.2.3 Frequency and Impedance Scaling

We are now able to design low-pass filters with a cut-off frequency of $1\,\mathrm{rad\,s^{-1}}$, but this is rarely the desired response for a filter as $1\,\mathrm{rad\,s^{-1}}$ is only 0.159 Hz, and a normalised load of $1\,\Omega$. More often a filter with a different cut-off frequency and load is desired and the need for frequency and impedance scaling arises. By scaling the frequency and impedance of the low-pass prototype, a filter can be designed as needed.

A simple formula can be derived for calculating the new capacitor and inductor values for the scaled filter, by analysing the transfer function of the low-pass filter, $H(j\omega)$. Starting with the simplest filter, the second order filter, it follows that the calculations holds for any order of filter.

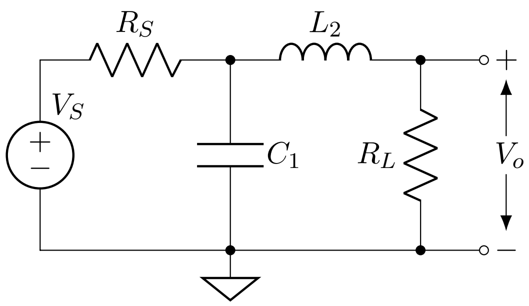


Figure 6.9: Example of a second order low-pass filter.

By analysing the low-pass filter shown in figure 6.9 its transfer function can be written as

$$H(s) = \frac{R_L}{R_S + R_L} \cdot \frac{1}{\frac{R_S L_2 C1}{R_S + R_L} s^2 + \frac{R_S R_L C_1 + L2}{R_S R_L} s + 1} \qquad [\cdot] \quad (6.4)$$

This transfer function has two poles which are the roots of the second order polynomial of the denominator. These roots are defined as $s_1 = -\frac{1}{A}$ and $s_2 = -\frac{1}{B}$, where the coefficients A and B can be calculated with the component values in the system. Writing the transfer function using A and B we get

$$H(s) = K \cdot \frac{1}{(As+1)+(Bs+1)} \qquad [\cdot] \quad (6.5)$$

Where $K = \frac{R_L}{R_S + R_L}$. If applying the transformation of $s = j\omega$ we can define a constant k_f such that

$$j\omega_{ref} = j(\frac{\omega_f}{k_f}) => k_f = \frac{\omega_f}{\omega_{ref}} \qquad [\cdot] \quad (6.6)$$

Where ω_{ref} is the reference frequency of 1 Rad/s and ω_f is the desired frequency. Transferring back to the Laplace domain and substituting into the transfer function we get

$$\begin{aligned} H(\frac{s_f}{k_f}) &= K \cdot \frac{1}{(A \cdot \frac{s_f}{k_f} + 1) + (B \cdot \frac{s_f}{k_f} + 1)} \\ &= K \cdot \frac{1}{(s_f \cdot \frac{A}{k_f} + 1) + (s_f \cdot \frac{B}{k_f} + 1)} \end{aligned} \qquad [\cdot] \quad (6.7)$$

This apllies to the impedances of components in the Laplace domain as well, and we get that

$$s_{ref} \cdot L \mapsto s_f \cdot \frac{L}{k_f} \quad \wedge \quad \frac{1}{s_{ref} \cdot C} \mapsto \frac{1}{s_f \cdot \frac{C}{k_f}} \qquad [\cdot] \quad (6.8)$$

And by that we can calculate new component values for different frequencies using

$$L_{new} = \frac{L_{old}}{k_f}[\text{H}] \quad \wedge \quad C_{new} = \frac{C_{old}}{k_f}[\text{F}] \qquad (6.9)$$

The same principle applies to scaling of the impedances, where a constant k_r is defined as

$$k_r = \frac{R_f}{R_{ref}} \qquad [\cdot] \quad (6.10)$$

Where R_{ref} is the reference load of 1 Ω and R_f is the desired load. As the impedances in the transfer function are scaled, the capacitor and inductor values have to be scaled as well for transfer function to be unchanged. Thus the component values after the impedance scaling can be calculated as before, this time by scaling with the factor k_r which gives

$$R_{new} = k_r \cdot R_{old}[\Omega] \quad \wedge \quad L_{new} = k_f \cdot L_{old}[\mathrm{H}] \quad \wedge \quad C_{new} = \frac{C_{old}}{k_r}[\mathrm{F}] \tag{6.11}$$

Combining both the frequency and impedance scaling, the new component values can be calculated as

$$R_{new} = k_r \cdot R_{old}[\Omega] \quad \wedge \quad L_{new} = \frac{L_{old} \cdot k_r}{k_f}[\mathrm{H}] \quad \wedge \quad C_{new} = \frac{C_{old}}{k_r \cdot k_f}[\mathrm{F}] \tag{6.12}$$

Keeping in mind that the reference values are 1 Rad/s and 1 ohm, the expressions for the capacitor and inductor values reduce to

$$L = \frac{L_n \cdot R_f}{\omega_f}[\mathrm{H}] \quad \wedge \quad C = \frac{C_n}{R_f \cdot \omega_f}[\mathrm{F}] \tag{6.13}$$

Where

- L_n is the low-pass prototype element value for the inductor.
- C_n is the low-pass prototype element value for the capacitor.
- L is resulting inductor value.
- C is the resulting capacitor value.
- R_f is the desired load impedance.
- ω_f is the desired cut-off frequency.

With these formulas it is now possible to design low-pass filters with any cut-off frequency and load impedance.

6.2.4 Filter Transformations

Sometimes a different type of filter than a low-pass filter is needed, such as a high-pass or a band-pass filter. Based on the low-pass prototype, it is possible to design the other types of filters by transforming the low-pass filter into the desired type of filter. As the desired function of the filters in this report is filter image frequencies as well as noise from the rest of the spectrum, band-pass filters are chosen for each filter stage. Thus only the lowpass-to-bandpass transformation is discussed here.

One method of transforming a low-pass filter to a band-pass filter is by looking at the bandwidth ratios between the 3-dB cut-off frequency and the frequency of a desired attenuation. From the low-pass prototype we saw that the ratio was expressed as

$$\text{Bandwidth ratio} = \frac{f}{f_c} \qquad [\cdot] \quad (6.14)$$

Where f is the frequency with the desired attenuation and f_c is the cut-off frequency. When transforming a low-pass filter to a band-pass filter, the bandwidth ratios stay constant, such that

$$\text{Bandwidth ratio} = \frac{f}{f_c} = \frac{BW}{BW_c} \qquad [\cdot] \quad (6.15)$$

where BW is the bandwidth at the desired attenuation and BW_c is the 3-db bandwidth.

This means that the attenuation curves from the low-pass prototype can be used for a band-pass filter as well. Sometimes the specifications for the filter does not contain both BW and BW_c, but only BW_c and a specific frequency which needs to be attenuated a certain amount. In this case it is necessary to first calculate the center frequency and then use that to calculate BW. As the frequency response of a band-pass filter is geometrically symmetrical, the center frequency is given by:

$$f_c = \sqrt{f_1 \cdot f_2} \qquad [\text{Hz}] \quad (6.16)$$

where f_1 and f_2 are any two frequencies on either side of the center frequency, that have the same attenuation.

When the bandwidth ratio has been found, and the necessary order chosen, the components of the actual filter can be determined. It follows instinctively that the band-pass-filter must have twice the reactive elements as the low-pass filter, and the way the band-pass filter is build is by resonating each component with a component of opposite type and same value. An example of this transformation is shown in figure 6.10.

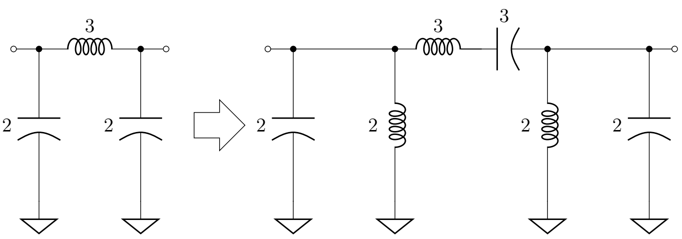


Figure 6.10: Transformation of components from a low-pass filter to a band-pass filter. [37]

Finally the normalised values needs to be frequency and impedance scaled as before using some slightly different formulas. For the parallel-resonant parts component values are scaled by using

$$C = \frac{C_n}{2RB\pi} \qquad [\text{F}] \quad (6.17)$$

$$L = \frac{RB}{2L_n {f_0}^2 \pi} \qquad [\text{H}] \quad (6.18)$$

and for the series-resonant parts,

$$C = \frac{B}{2\pi f_0^2 C_n R} \qquad [\text{F}] \quad (6.19)$$

$$L = \frac{RL_n}{2\pi B} \qquad [\text{H}] \quad (6.20)$$

where

- R is the desired load impedance of the filter.
- B is the desired 3-dB bandwidth of the filter.
- f_0 is the center frequency of the filter.
- L_n is the normalised inductive element value.
- C_n is the normalised capacitive element value.

Now that the basic theories behind designing simple band-pass filters have been discussed, the requirements for the filters in the radio is listed and each filters design topology is chosen.

6.3 Filter Specification

As described earlier in this chapter, three filters are designed for the receiver. Each filter and its specifications are discussed further in the following subsections.

6.3.1 Radio Frequency-filter

The first filter in the radio is the Radio Frequency-filter (RF-filter). The purpose of this filter is to attenuate all the frequencies outside the danish FM spectrum, especially the image frequencies that arise as the radio tunes into different radio stations. As the mixers use lowside injection and the first intermediate frequency is at 10.7 MHz, the image frequencies will lie below the carrier wave starting at 66.1 MHz and spanning towards 86.6 MHz. In table 6.3 are listed the specifications for the RF-filter.

Table 6.3: Specifications for the RF-filter.

RF-filter Specifications				
Description	*Min.*	*Max.*	*Unit*	*Comments*
Lower bound frequency of passband	87.0	87.5	MHz	-
Upper bound frequency of passband	108	110	MHz	-
Attenuation of image frequency	20	-	dB	-

6.3.2 IF1-filter

The second filter is the first of the two intermediate frequency filters, located between the two mixers in the system. The purpose of this filter is the same as the RF-filter, to attenuate an image frequency. Where the RF-filter had to attenuate a band of image frequencies spanning 20 MHz, the IF1-filter only has to attenuate one image frequency. As the second intermediate frequency is at 455 kHz, the image frequency between the two mixers is located at 9.79 MHz. Due to the 75 kHz frequency deviation of the carrier wave, the filter must have a bandwidth of at least 150 kHz at a center frequency of 10.7 MHz, or a part of the signal modulated into the carrier will be lost. In table 6.4 the specifications for the IF1-filter are listed.

Table 6.4: Specifications for the IF1-filter.

IF1-filter Specifications				
Description	*Min.*	*Max.*	*Unit*	*Comments*
Lower bound frequency of passband	9.79	10.6	MHz	-
Upper bound frequency of passband	10.8	-	MHz	-
Attenuation of image frequency	20	-	dB	-

6.3.3 IF2-filter

The final filter is the second intermediate filter, located between the last mixer and the demodulator. The purpose of this filter is to attenuate all signals except the carrier wave. Before the signal reaches the demodulator, it is important that all other frequencies than the carrier wave has been filtered away. If this is not the case, signals from neighbouring radio stations may reach the demodulator together with the radio station the radio is supposed to be tuned into. This results in an effect called adjacent-channel interference, where both the intended station and the neighbouring station can be heard simultaneously. In Denmark radio stations has to be spaced with at least 200 kHz between the carrier frequencies, but a frequency deviation of 75 kHz and therefore the filter must have a bandwidth between 150 and 250 kHz. In table 6.5 the specifications for the IF1-filter are listed.

Table 6.5: Specifications for the IF2-filter.

IF2-filter Specifications				
Description	*Min.*	*Max.*	*Unit*	*Comments*
Lower bound frequency of passband	330	380	kHz	-
Upper bound frequency of passband	530	580	kHz	-
Attenuation neighbouring stations	20	-	dB	-

6.4 Filter Design

In this section the design progress of the filters continue. Following the previous section, the design will be based on each filter's specification. Due to the characteristics of these filters, the Chebyshev band-pass design is the method which these filters will be designed. This is because some ripple in the passband can be allowed to achieve the greater steepness which is needed in the filters. The desired frequency response for this filter is illustrated on figure 6.11. The parameters on this figure are to be determined for each individual filter.

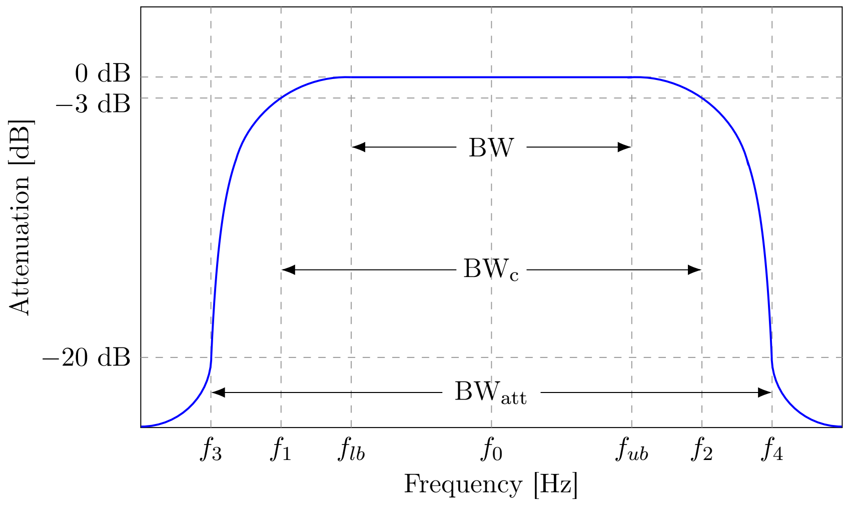


Figure 6.11: Frequency response of a Chebyshev band-pass filter with frequency parameters.

Note that the RF filter has been split into two separate filters (RF1 and RF2) because its bandwidth is too wide to realise the needed steepness in the skirts of the frequency response. To ensure that all channels are still available, there will be an overlap between responses of the two filters.
The first step is to setup the parameters for where the band-pass lies in each of the four filters' frequency range, this is done in subsection 6.4.1. The next step is to determine the component values for which the filter will be based upon, this is done in subsection 6.4.2. Thereafter, the filter circuits will be simulated in LTSpice before they can be tested.

6.4.1 Band-Pass Filter Parameters

In section 6.3, the lower and upper outer bounds (f_{lb} and f_{ub}) of the bandwidth of the three filters are defined. This also gives a bandwidth (BW) for the pass-band filter. Additionally, the centre frequency (f_0) of the IF1 and IF2 filters are given.
Note that these frequency parameters throughout the subsection are shown on the frequency response of the filters on figure 6.11.
For the RF1 and RF2 filters, the bandwidth and center frequency are roughly estimated to cover the entire RF filter's bandwidth. For the four filters the first parameters are the following values described in table 6.6.

Table 6.6: Band-pass parameters for the four filters based on section 6.3.

First Band-Pass Parameters				
Filter:	*BW [Hz]:*	f_0 *[Hz]:*	f_{lb} *[Hz]:*	f_{ub} *[Hz]:*
RF1	$11 \cdot 10^6$	$93 \cdot 10^6$	$87.5 \cdot 10^6$	$98.5 \cdot 10^6$
RF2	$11 \cdot 10^6$	$102.5 \cdot 10^6$	$97 \cdot 10^6$	$108 \cdot 10^6$
IF1	$\geq 150 \cdot 10^3$	$10.7 \cdot 10^6$	$\leq 10.625 \cdot 10^6$	$\geq 10.775 \cdot 10^6$
IF2	$\geq 150 \cdot 10^3$	$455 \cdot 10^3$	$\leq 380 \cdot 10^3$	$\geq 530 \cdot 10^3$

Once again conferring to the requirements of the filters, in section 6.3, there is a given -20dB-Attenuation frequency (f_3) for IF1 and IF2. For RF1 and RF2, the f_3 frequency is calculated by using formula 6.21

$$f_3 = f_{ub} - 2 \cdot f_{0_{IF1}} \quad \text{[Hz]} \qquad (6.21)$$

where

- $f_{0_{IF1}}$ is the center frequency of the IF1 band-pass filter

Using f_3 and f_0, the adjacent -20dB-Attenuation frequency (f_4) can be calculated. This calculation is done using the formula 6.22.

$$f_4 = \frac{f_0^2}{f_3} \quad \text{[Hz]} \qquad (6.22)$$

Now, the −20dB-Attenuation bandwidth (BW_{att}) can be calculated using formula 6.23.

$$BW_{att} = f_4 - f_3 \quad \text{[Hz]} \qquad (6.23)$$

Finally, the bandwidth ratio (BW_{ratio}) between BW and BW_{att} can be calculated using formula 6.24.

$$BW_{ratio} = \frac{BW}{BW_{att}} \quad [\cdot] \qquad (6.24)$$

The important parameters listed above are gathered in table 6.7

Table 6.7: Additional band-pass filter parameters based on above formulas and requirements from section 6.3.

Second Band-Pass Parameters				
Filter:	f_3 *[Hz]:*	f_4 *[Hz]:*	BW_{att} *[Hz]:*	BW_{ratio} *[·]:*
RF1	$77.1 \cdot 10^6$	$112.17 \cdot 10^6$	$35.1 \cdot 10^6$	3.19
RF2	$86.6 \cdot 10^6$	$121.3 \cdot 10^6$	$34.7 \cdot 10^6$	3.15
IF1	$9.79 \cdot 10^6$	$11.79 \cdot 10^6$	$2 \cdot 10^6$	≤ 13.33
IF2	$330 \cdot 10^3$	$580 \cdot 10^3$	$250 \cdot 10^3$	≤ 1.66

On table 6.7, the IF1 BW_{ratio} is set at its maximum value where the IF1 BW is set at its minimum value at 150 kHz. However, for this design, the IF1 BW is designed to 300 kHz which results in a BW_{ratio} of ≤ 6.67. This choice is made to ensure that the entire 150 kHz band is within the band-pass and no component tolerance or noise can cause interference.

Before moving onto the next step, the two cut-off frequencies (f_1 and f_2) for each filter must be calculated.

For IF1, the cut-off frequencies can be put at f_{lb} and f_{ub} which for 300 kHz is 10.55 MHz and 10.85 MHz respectively. However, for RF1, RF2 and IF2 the cut-off frequencies must be roughly estimated below f_{lb} and above f_{ub} to make sure that there is no attenuation within BW. These estimations for RF1, RF2 and IF2 can be seen on table 6.8

Table 6.8: Estimated band-pass parameters for the three filters.

Third Band-Pass Parameters			
Filter:	f_1 *[Hz]:*	f_2 *[Hz]:*	BW_c *[Hz]:*
RF1	$86 \cdot 10^6$	$100 \cdot 10^6$	$14 \cdot 10^6$
RF2	$95.5 \cdot 10^6$	$109.5 \cdot 10^6$	$19 \cdot 10^6$
IF1	$10.55 \cdot 10^6$	$10.85 \cdot 10^6$	$300 \cdot 10^3$
IF2	$370 \cdot 10^3$	$530 \cdot 10^3$	$160 \cdot 10^3$

6.4.2 Band-pass Filter Components

In this part of the design phase, the real world components of the filter are calculated. These calculations are based on the frequency parameters from subsection 6.4.1 and the formulas 6.17 through 6.20 in subsection 6.2.4.
Before applying the formulas from subsection 6.2.4, the order of the filter based on BW_{ratio} and desired attenuation must be determined. For the four filters the BW_{ratio} were determined in subsection 6.4.1. To determine the order of the filter, figure 6.8 is used by aligning the minimum attenuation with the BW_{ratio} on the graph and finding the closest function off an odd-numbered order. These characteristics of the filters are compiled in table 6.9.

Table 6.9: Determination of the order of the band-pass filters and the approximate attenuation of f_3 and f_4.

Final characteristics of the Band-Pass Filters		
Filter:	*Order*	*Approximate Attenuation [dB]:*
RF1	3	40
RF2	3	40
IF1	3	52
IF2	5	30

Now the next sequence is to find the ratio between input and load impedance (R_{ratio}, this ratio along with the order of the filter is used to find the element values which are used to calculate the exact component values of the given filter. For the four filters the value of R_{ratio} is 1 because the input and output impedance is 50 Ω.
Hereafter the specific element values can be found for each filter's order. When looking at element value tables, there are different tables for either low or high ripple in the pass-band. For the four filters, a 1.0 dB ripple is selected.
Finally, all the required values to calculate the parallel-resonant and series-resonant components can be made. The formulas for the parallel-resonant components are given by equation 6.17 and 6.18, and the series-resonant components are given by equation 6.19 and 6.20.
The four formulas and their input values are calculated and the results can be found in

table 6.10. These values together make up one parallel-resonant part (C_1 and L_{C_1}) and two series-resonant parts (L_1, C_{L_1}, L_2 and C_{L_2}).

Table 6.10: Component values for third order Chebyshev filters.

Component Values of Third Order Chebyshev Band-Pass Filters with 1.0 dB ripple						
Filter:	L_1	C_{L_1}	C_1	L_{C_1}	L_2	C_{L_2}
RF1	1.26 µH	2.33 pF	182.27 pF	13.23 nH	1.26 µH	2.33 pF
RF2	1.26 µH	1.91 pF	247.37 pF	9.75 nH	1.26 µH	1.91 pF
IF1	58.78 µH	3.76 pF	11.54 nF	19.17 nH	58.78 µH	3.76 pF

The circuit for RF1, RF2 and IF1 is illustrated on figure 6.12

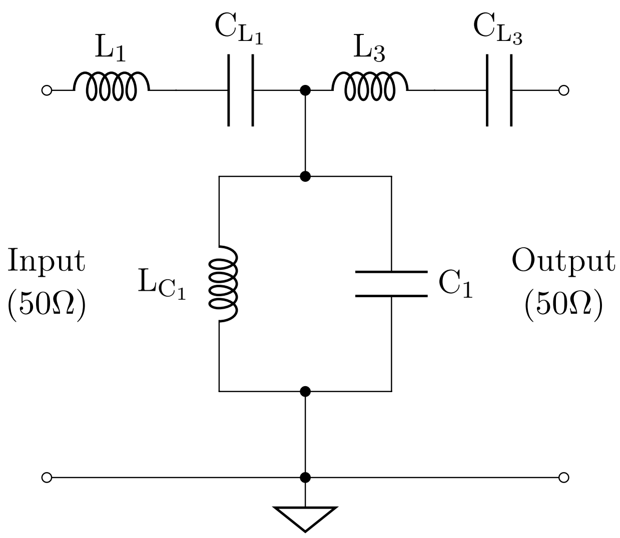


Figure 6.12: Circuit schematic of a third order Chebyshev filter.

The IF2 filter is a Chebyshev band-pass filter of fifth order, and therefor requires an additional parallel-resonant and series-resonant part. However, this does not add much additional work to the component calculation. Once again, the component values are calculated and compiled in table 6.11

Table 6.11: Component values for the IF2 band-pass filter.

IF2 Seventh Order Chebyshev Band-Pass Filter with 1.0 dB ripple					
L_1 *and* L_5	C_{L_1} *and* C_{L_5}	C_2 *and* C_4	L_{C_1} and L_{C_4}	L_3	C_{L_3}
109.77 µH	1.14 nF	22.44 nF	5.57 µH	154.33 µH	810.52 pF

The circuit for the IF2 filter is very much similar to that of the third order Chebyshev filters. However, the first series-resonant and parallel-resonant parts are mirrored on the other side of the second series-resonant part. These mirrored series and parallel parts also have the same values as the first two, this is because they have the same element value. This is similar to the third order filter, where the two series-resonant parts have the same values. The IF2 filter's circuit is illustrated on figure 6.13.

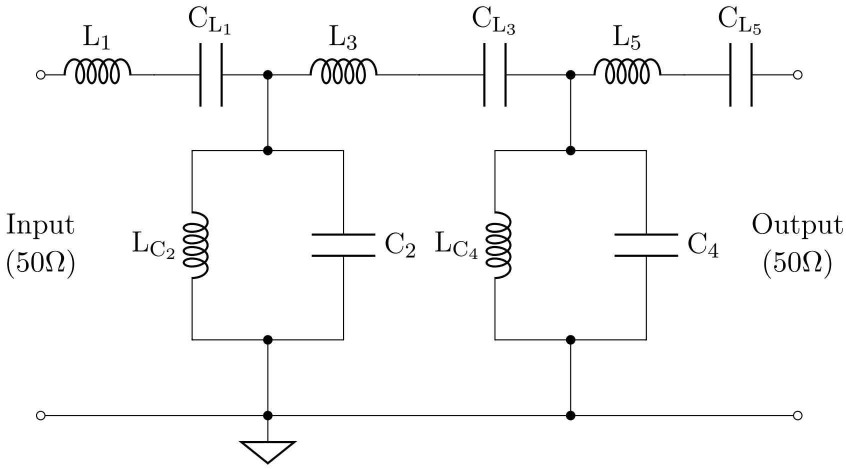


Figure 6.13: Circuit schematic of a fifth order Chebyshev filter.

Now that the components and their values for the filters have been realised, it is time to simulate the filters and confirm that their requirements have been met.

6.4.3 Chebyshev Filter Simulation

Simulating the filters is the final step before the filters can be implemented on a PCB and eventually tested. The simulations are done in LTSpice made by Linear Technology. The data and graphs which are simulated are all frequency responses of the attenuation (The signal's amplitude) in decibel and the phase.
First the RF1 and RF2 filters are simulated and the respective frequency response can be seen on figure 6.14 and 6.15.

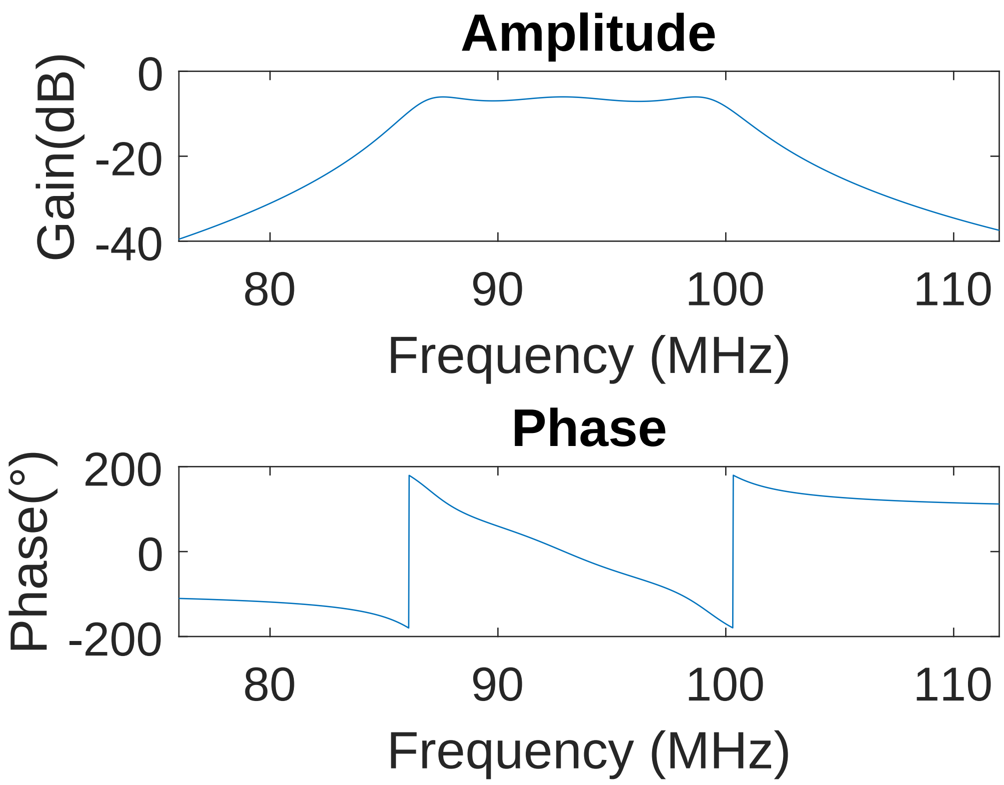


Figure 6.14: Simulation of the RF1 filter, the data points can be found in the attached files (RF1-data.csv).

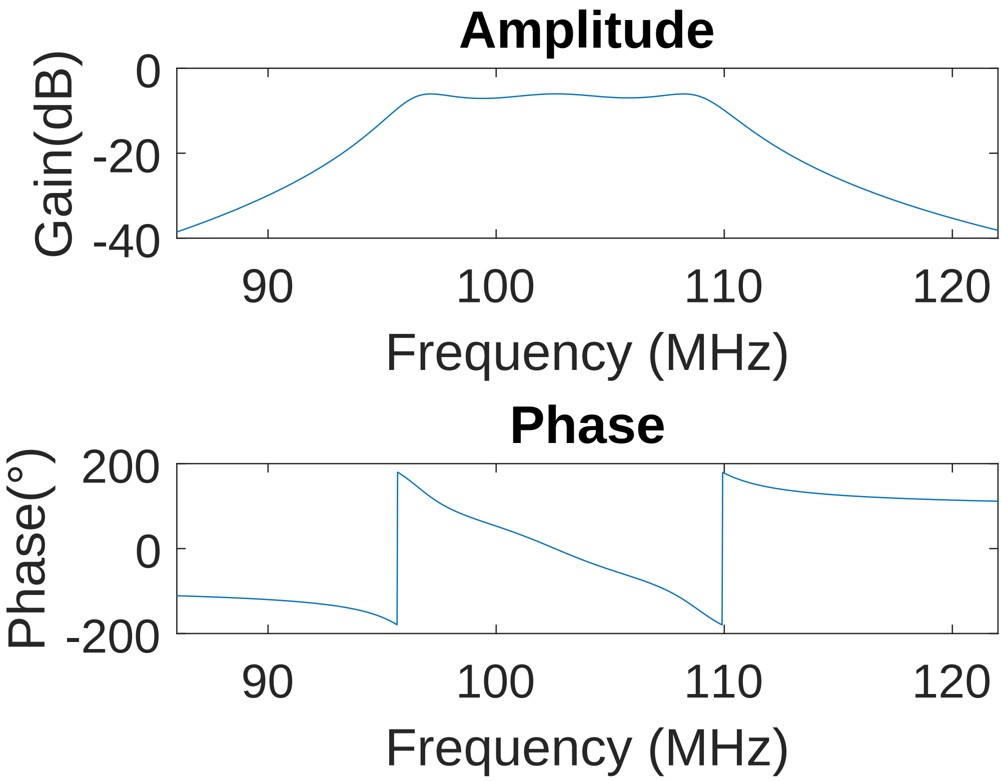


Figure 6.15: Simulation of the RF2 filter, the data points can be found in the attached files (RF2-data.csv).

The responses on the RF1 and RF2 simulation meet the requirement of the filters. For RF1 there is 37 dB attenuation at f_3 and f_4, while the same is the case for RF2 at it's respective f_3 and f_4.
The next two filters which are simulated are IF1 and IF2, these can be seen on figure 6.16 and 6.17 respectively.

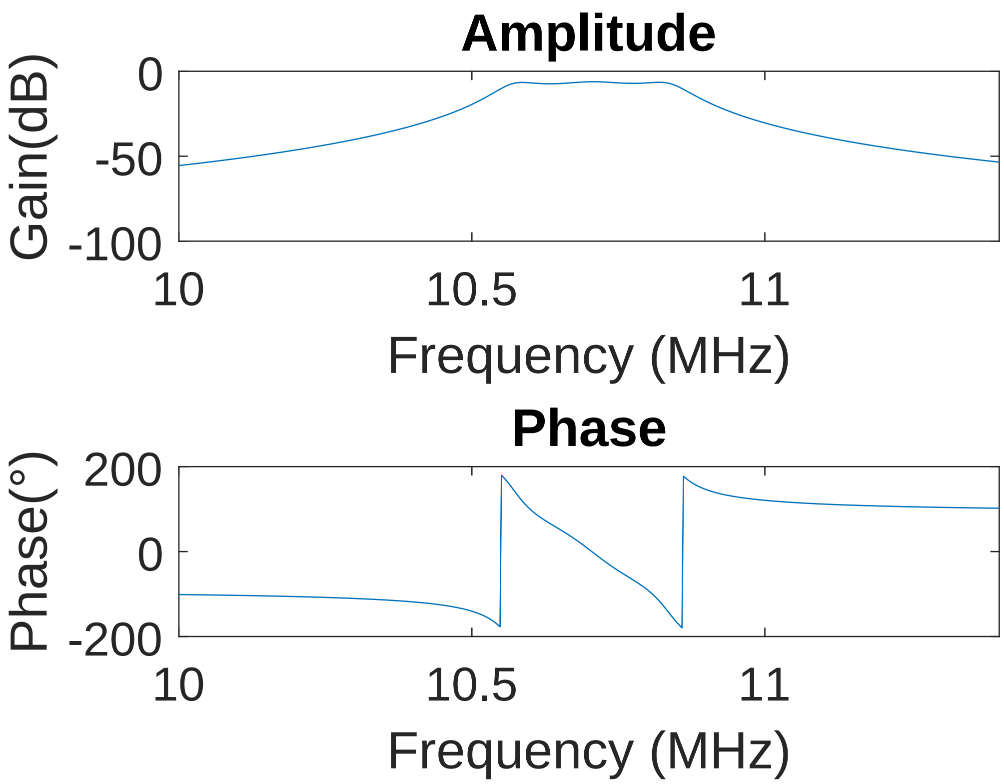


Figure 6.16: Simulation of the IF1 filter, the data points can be found in the attached files (IF1-data.csv).

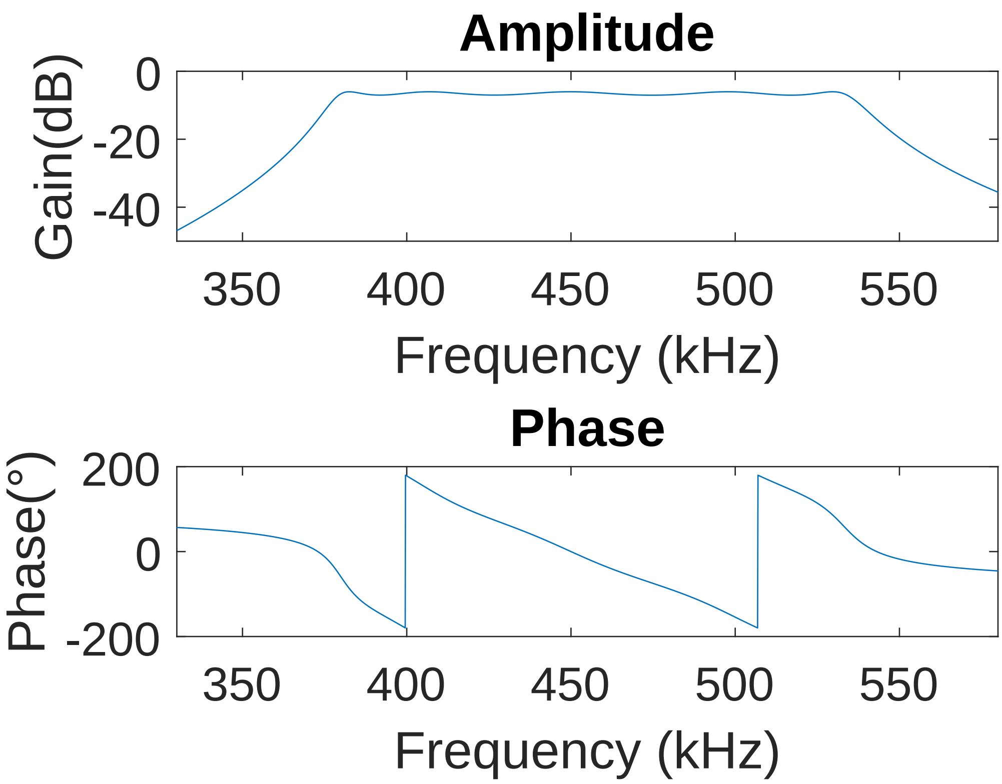


Figure 6.17: Simulation of the IF2 filter, the data points can be found in the attached files (IF2-data.csv).

For IF1 and IF2, the responses of the amplitudes also meet the stated requirements of the band-pass. The IF1 filter shows a 63 dB attenuation at both f_3 and f_4. The IF2 there is a 46 dB attenuation at f_3 and 35 dB attenuation at f_4.
Since these simulations show that the requirements can be met with third and fifth order Chebyshev band-pass filters the next step in the implementation is to build the filters and test them.

6.4.4 Conclusion of the Chebyshev band-pass filter implementation

The RF1, RF2 and IF1 filters were the first filters to be constructed. The three filters were constructed with values similar to that of the design in subsection 6.4.2. Several specific component values of both capacitors and inductors were achieved by combining two or more of said component in series and parallel. Furthermore, several of the component values were tested to make sure that component tolerances were no higher than five percentage. As a result these circuits were very closely built to that of the design,

with some component values being as close as a few decimal points of Farad and Henry off the design (for a given Engineering magnitude).
However, during the initial testing of these three filters their characteristics were nothing like the simulations shown in subsection 6.4.3. Attempts were made to shield the inductors from creating unwanted magnetic coupling, however this measure had no significant impact on the filters' characteristics.
During research into the problem it was uncovered that the utilised axial inductors have a lower self-resonance frequency (SRF) than what frequency they were utilised for. The SRF of an inductor is the frequency at which the inductor's impedance becomes a capacitor [38]. This characteristic of the inductors is due to the self-capacitance between turns of each winding. At frequencies lower than the SRF this capacitance serves as a reduction of the inductor's quality factor (Q-value) [39] A graph of the SRF for the utilised axial inductors (in the µH range) can be seen on figure 6.18.

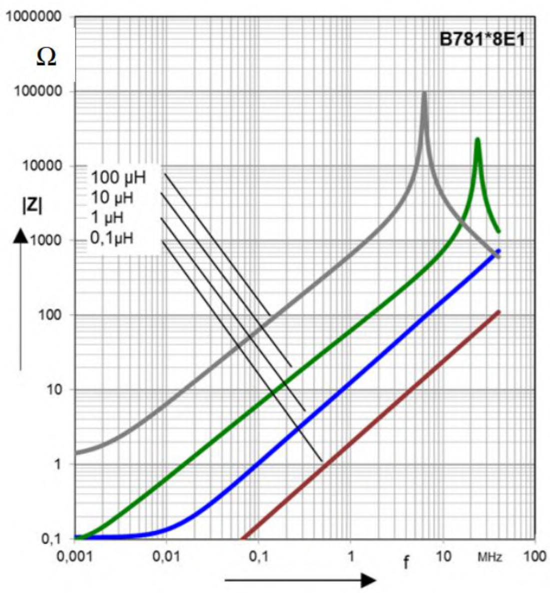


Figure 6.18: Axial inductor impedance versus frequency [40]

A solution to this problem, would be to utilise surface-mounted devices (SMD) Inductors which have a much higher Q-value and SRF. However, PCB production including SMD components is outside the scope of this project and was therefor not attempted for these filters. Additionally, reaching high enough µH-values using SMD inductors becomes rather difficult.
Similarly to inductors, ceramic capacitors also have an SRF, however this frequency is usually higher than inductors. For capacitors, at high frequencies there are traces of residual inductance and beyond the SRF the impedance becomes an inductance. [41]

Similarly to the other three filters, the IF2 filter was also built following the design.

Likewise for this filter, the initial testing also showed fault in the filter's characteristics. The reason for this, which was pointed out by a professor at the Electronic Institute at AAU, is that when a fifth order filter with 10 components (aproximate double that amount of components to match the design values) is constructed it is almost impossible to get it right due to component tolerances, non-linearity, stray-capacitance, etc.

The conclusion of the Chebyshev band-pass design is that when simulated, the designed filters have an accurate response compared to their design parameters and requirements. However, when assembling the filters with unsuitable and high tolerance components available in the electronic laboratory it is impossible to realise the simulated results. Finally, it is estimated that these filter designs would work if SMD Inductors with high Q-values and low tolerances in addition to capacitors with low tolerances were used in the circuits.
Seeing how there is still a need for filters in the complete system the IF1 and IF2 filter will be described in section 6.5. The RF filter will be discontinued, but for the RF input there will still be filters near the Low-noise amplifier to counter excess signals and oscillation which occours at the antenna and amplifier. More documentation of the RF front-end can be found in chapter 8.

6.5 New Design and Implementation of Filters

Instead of attempting to design and implement additional filters within a limited time-line, the group has decided on acquiring an IF1 filter at 10.7 MHz, this filter is described in subsection 6.5.1.
Furthermore, the IF2 filter will also be remade, this time as a tunable cascade-coupled parallel-resonance circuit with magnetic coupling filter, this filter is described in subsection 6.5.2.

6.5.1 Ceramic 10.7 MHz Filter

For the new IF1 filter, a ceramic filter is used, three ceramic filters were provided by lecturer Hans Ebert.
Ceramic filters have a piezoelectric effect similar to that of a crystal lattice filter, however with a lower Q-value and wider bandwidth. [41] The piezoelectric effect is when a mechanical force is converted into an electrical field and when an electrical signal is converted into a mechanical force. For filters this piezoelectric effects affects the input signal, and in turns gives a filtered output. For a ceramic filter, the piezolectric effect happens when there is a vibration, which can occur when atoms vibrate along with temperature change. [42] [43]
The ceramic filters which were provided are to be tested to verify which filter best

matches the requirements stated in 6.3.2. These tests were completed and the test journal is presented in appendix G. The conclusion of this test is that the

6.5.2 Optimized Cascade-coupled Parallel-resonance Circuit with Magnetic Coupling Intermediate Frequency Filter with Amplification for a Double Superheterodyne Radio Receiver

This filter is based on a theoretical design presented by lecturer Hans Ebert, thereafter it is further developed by the project group. The filter's topology is cascade-coupled parallel-resonance circuit with magnetic coupling between inductors. The design is based upon stagger tuning and multiple feedback band-pass active filter principles, however with attenuation instead of amplification. Additionally, the filters are made highly tunable, in turns this means that most of the resonant and magnetic coupled inductors, resonant capacitors and resistors are variable on the final PCB.
The principle of stagger tuning when designing a band-pass filter is to have separate stages (at least two stages) of narrow-bandwidth filters. To obtain the best bandwidth response of the band-pass filter, the individual stages must be designed closely to the center frequency of the band-pass. This principle is also known as the lowpass-to-bandpass transformation [44]. The principle of stagger tuning is shown on figure 6.19.

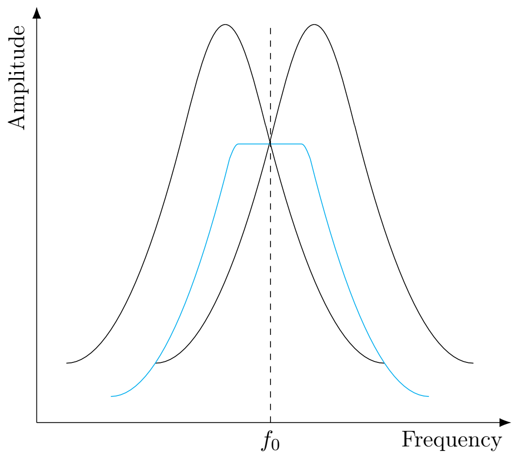


Figure 6.19: The frequency response of a stagger tuning band-pass filter (blue) and it's individual narrow-band stages (black) [44].

For this variation of stagger tuning, instead of separating the individual stages with (unit-gain) amplifiers, the individual stages are separated with magnetic coupled inductors. Each stage are made up of parallel resonance circuits with a variable potentiometer in series in-between the resonance circuit and ground. These potentiometers serve as a tunable component, by adjusting the resistance the overall flatness or ripple of the filter's attenuation can be tuned. This is possible because of the stagger principle, where

each stage can be tuned individually to improve the overall band-pass response.
In addition to the filter, there is an amplifier in the front-end of the filter. This amplifier serves as a limiter, and while amplifying the signal going into the filter it also creates harmonic distortion in the signal. Although added harmonic distortion is usually an unwanted aspect of any audio system, this is not a problem because the distortion lies at frequencies which are filtered away from the signal.
The circuit of this filter can be seen on figure J.4, it is illustrated with the maximum size of the tunable components.

Because of the highly tunable characteristics of this filter, it is difficult to simulate an exact response of the band-pass. However, on figure 6.20 a simulation of a working configuration has been made in LTSpice.

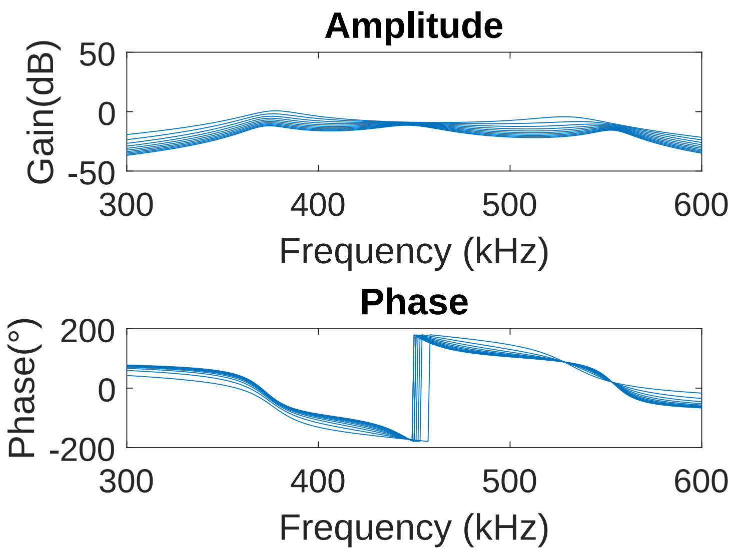


Figure 6.20: Simulation of the band-pass filter where R1 is a 10 kΩ potentiometer stepped at 1 kΩ for each function.

Based on the above simulation the filter is constructed and tuned with an Oscilloscope showing the frequency response in real time. The tuning is accomplished by varying the potentiometers, variable capacitors and magnetic coupling between the inductors and their ferrite cores.
Once the filter has been properly tuned, it is tested and the test journal can be found in appendix H.

6.6 Conclusion

The initial filter design, although properly designed does not meet the requirements erased for the filters and thus will not be included in the acceptance test. Instead, the RF filter replacement is documented in section, while replacements for the IF1 and IF2 filters have been documented in subsection 6.5.1 and 6.5.2 respectively. During testing of the IF1 and IF2 filters, it has been documented that both filters meet the requirements. Although the filters function, it does not mean that they are perfect by any means.

After the implementation and testing of the ceramic IF1 filter, it has been uncovered that the filter functions significantly better with impedance matching across the input and output terminal. [42] Specifically for this filter, the input and output should always be matched at 330 Ω and no reactance. This is easily implemented with potentiometers which can be tuned to the changing input and output of the filter [43].

Although the design works in both simulations and real-world implementation, the design is deemed as faulty. This is due to the functionality of the filter completely relying on being tuned to it's intended purpose. More so, this is the case because each of the separate narrow band filter within the band-pass is tuned by four different variables being: parallel capacitance and inductance (coupled-resonance), series resistances and magnetic coupling of two coupled inductors. It is true, that each stage of the filter can be tuned but unlike a classic multiple feedback active filter, this filter is connected throughout like a ladder circuit (where all components affect each-other).
Despite the above stated, the filter is still used because it's fine-tuned characteristics, as highlighted by the test journal, meet the set requirements of the filter.

Chapter 7

Demodulation

This chapter will describe the process and consideration made in designing the FM demodulator. The demodulator module in the superheterodyne radio receiver is highlighted in figure 7.1.

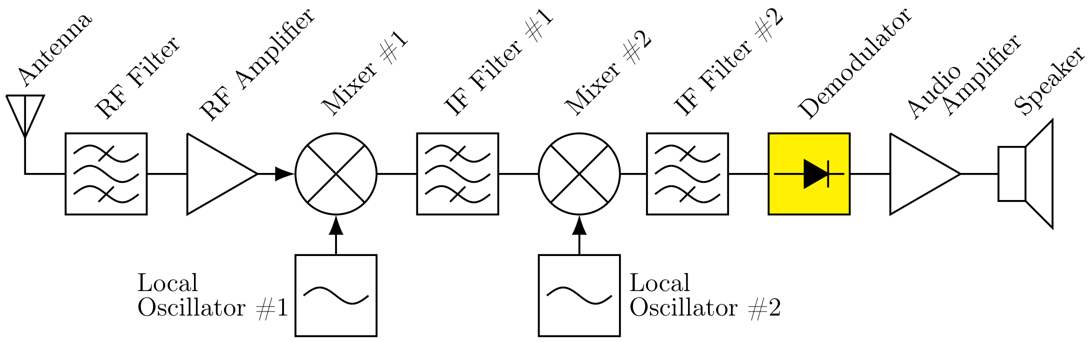
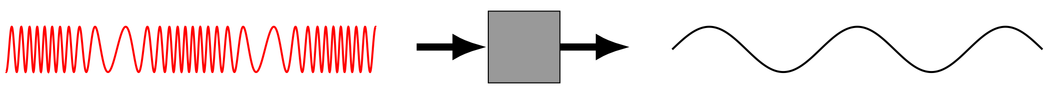


Figure 7.1: Demodulator module highlighted in the overall superheterodyne radio receiver design.

7.1 Theory

As previously touched upon in section 2.2 carrier waves are modulated with information and are in need of a demodulator to recover/extract the original signal information from a carrier wave, to produce sound.



(a) FM signal **(b)** Demodulator **(c)** Demodulated signal

Figure 7.2: An overview of a FM signal before and after demodulation.

The primary functionally of the FM demodulator is therefore to convert frequency variations into of the input signal into amplitude variations at the output, as illustrated on figure 7.2. For optimal demodulation, the amplitude output has to be proportional to the deviation of its frequency input, thus a linear relation between frequency and

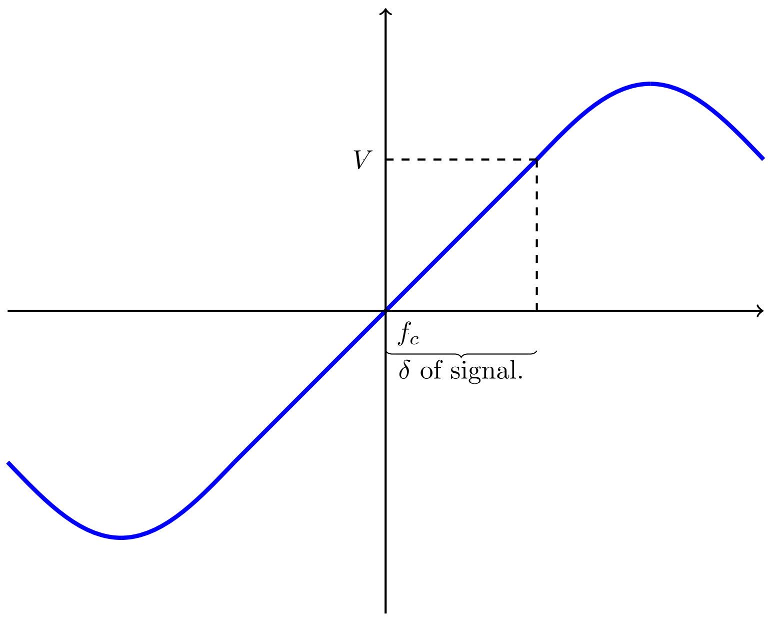


Figure 7.3: The s-curve characteristics.

voltage. This is described as:

$$V_o = k_d \cdot \delta \qquad [\mathrm{V}] \tag{7.1}$$

V_o is output voltage	[V]
k_d is the demodulator's sensitivity in volts per hertz.	[V Hz^{-1}]
δ is frequency deviation required for the output voltage.	[Hz]

However it is difficult to realise for a wide range of frequencies. Since the carrier wave has a certain bandwidth, it is only required to have a linear characteristic within that bandwidth, which is possible. To realise this, frequency dependent circuits are required. This often results in a linear characteristic with a nonlinear characteristic outside the carrier wave bandwidth, often in the shape of an the letter 'S'.

The sensitivity of a detector is the slope of the straight line portion of the S-curve of figure 7.3. [45, Chapter 12]

To accomplish this task of demodulation, several demodulators have been designed through the last century. Each of these demodulators have their own advantages and disadvantages.
For the sake of brevity only a handful of the most well known ones will be presented in this report. To quickly navigate between these, the most common ones will be listed

below together with a brief description of their respective characteristics and functionalities.

Slope Detector

One if the most elementary methods of demodulation is known as slope detection. At the core of its circuit the slope detector consists of a simple LC resonator tank tuned at f_0 and a AM detector diode in series. The carrier frequency of the FM wave f_{FM} is not equal to f_0. Due to this arrangement, the incoming FM signal falls on the tuned circuit side of gain peak of the tank circuit, which causes the amplitude vary in line with the frequency variations. Once the FM signal has passed through the LC tank, the diode and a lowpass filter, it has been demodulated[45, Chapter 12].

The advantages of this detector is that it is simple to use and allows demodulation without implementing additional circuits, however the output of the detector is not very linear.

Foster-Seeley Detector

Another demodulator is the modified version of the slope demodulator, the "Foster-Seeley" invented by Dudley E. Foster and Stuart William Seeley in 1936.
This demodulator has a RF transformer, a pair of diodes and a RF choke, which functions by differentiating between mistuned signals on the high frequency side of resonance and those on the low frequency side hence creating a phase difference between signals. This results in a signal which is ideally 90° out of phase. This signal is then passed through the two diodes which results in the signal being demodulated as a voltage difference between the two diodes.[45, Chapter 12].

The advantages of this detector is that it offers good performance and in general a good linearity, however a high cost of the transformer, and poor AM rejection without a limiter.

Ratio Detector

The Ratio Detector invented by Stuart William Seeley and Jack Avins is a variation of the before mentioned “Foster-Seeley” demodulator. The Ratio demodulator was widely used back in the days before the introduction of digital components, due to its great capability to reject AM, which meant it worked both as a demodualtor and a limiter.
It consists of a transformer at its core with three windings, which detects frequency changes in an incoming signal. It is done by adding three signal that are out of phase,

the two signals from the two parts of the centre tapped secondary are a part of a tank circuit and thus the phase vary with the frequency and the signals create a voltage difference that can be tapped for the demodulated signal. [46]

The advantages of this detector is that it is relatively easy to construct with discrete components while offering a good linearity, however as with the Foster-Seeley is the high cost of the transformer.

Phase Locked Loop

The Phase Locked Loop (PLL) is a closed loop feedback circuit, which consists of a three components: A phase detector, a low pass filter and a voltage-controlled oscillator (VCO) which oscillation frequency is determined by an external voltage input. A block diagram of the PLL can be seen on figure 7.4. In a PLL circuit the VCO is following the input frequency. This is done by a feedback loop which generates a voltage based on the deviation from the IF. this feed back can be tapped and is the demodulated signal. This

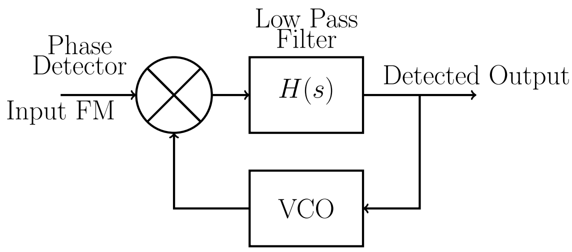


Figure 7.4: Principles behind the PLL.

method is today one of the most common methods of demodulation since it is easy to implement in an IC And offer excellent linearity. However it is complicated to construct with discrete components. [47, Chapter 2].

Quadrature Detector

The quadrature detector is quite similar to the slope detector, but instead of converting FM directly to AM, its instead converted to PM and then to AM. The FM signal is split into two separate paths. The first path goes directly to the mixer within in the system. The second path adds a 90° and then an additional amount PM to the signal due to its resonant network, which then is fed into the mixer as well. The mixer then produces an output proportional to the phase difference between the two signals, hence demodulating the signal [45, Chapter 12].

The advantages of this detector is its good linearity and performance, however it is difficult to construct because of an additional mixer.

7.2 Requirements

The requirements for the demodulator should be considered relative to the radio receiver, which requirements were previously specified in chapter 4.3. While the noise is added to the system by all components, the THD should only be introduced by the demodulator since the frequency should not be distorted by any components, thus the THD is only relevant requirement for the demodulator, and the THD specification can be transferred directly to the demodulator. The SNR has to be considered for every component. However, noise is suppressed by the demodulator and thus it is not just a simple addition of the noise in the system. The actual calculations are way out of scope for this project and thus the noise should just be as low as practically possible. Since the demodulator has to be built to retain certain characteristics, the output voltage has to be strong enough to be picked up and amplified and with a small output impedance. The input impedance should be as high as possible. The demodulator has certain unique requirements that directly effect the other characteristics. The demodulator should also have a linear frequency to output voltage characteristic within the bandwidth of the FM signal. Last but not least the output frequency should match the specifications for the receiver as the demodulator generates the audio output.

7.2.1 Choice of demodulator type

When choosing the type of demodulator several things has to be considered. Since the the purpose of this project was to build an FM radio of mainly discrete components several of the modulators can be excluded since they would be too complex to integrate using discrete components due to the time constraints of this project. The PLL is mostly fit for ICs. The quadrature detector would require another mixer, but could have been an alternate solution. The slope detector, while simple to implement, have poor linearity and no AM rejection without a limiter, it was decided that this would not be able to fulfil the requirements. Thus it was a choice between the Fooster-Seely detector or the ratio-detector (RD). The RD was the final choice due to the superior AM rejection and the seemingly simple design with no need for a limiter.

7.3 Design

This section will describe the design of the demodulator used in the final product. First an understanding of the principles behind the RD will be established, followed by the realisation of the design and the components. An overview of the ratio detector can be seen on figure 7.5. Unless otherwise specified the source material used for the design of the ratio detector is the technical journal RCA Review Volume VIII[46].

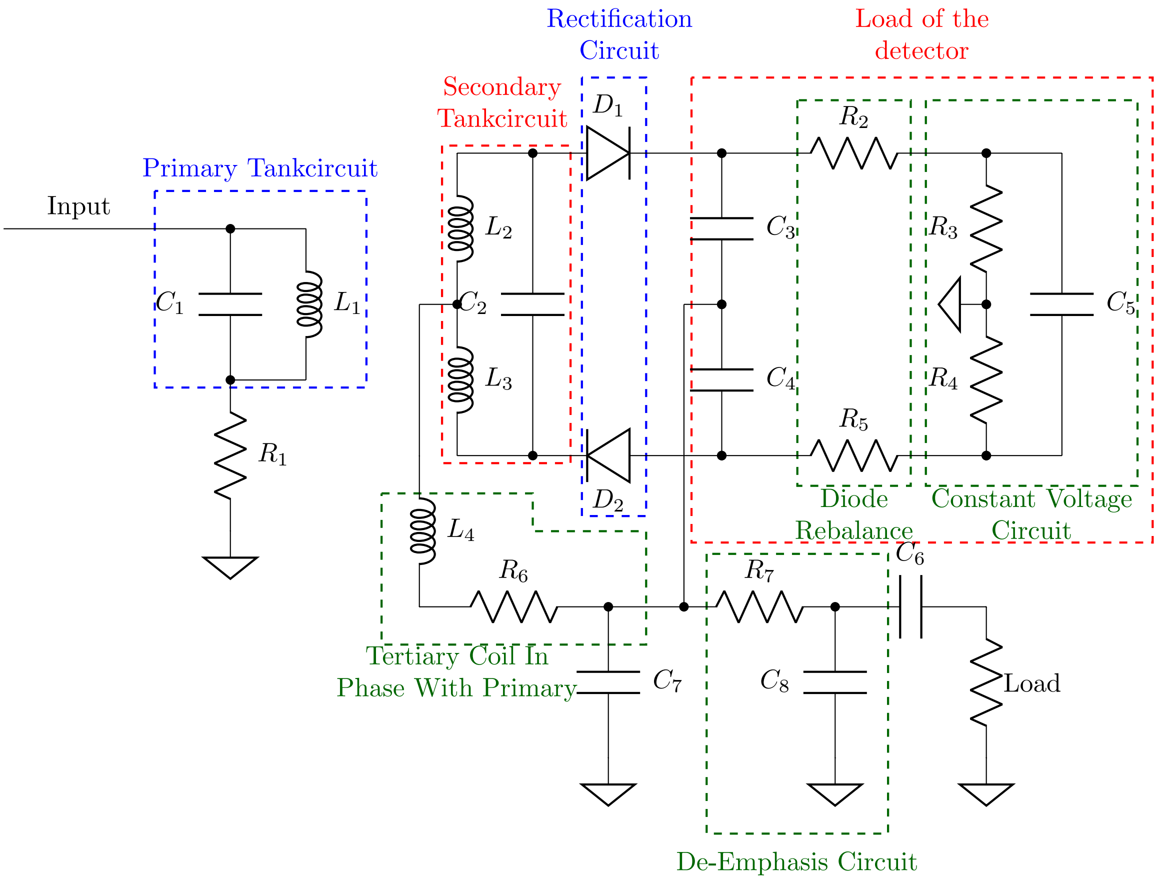


Figure 7.5: Overview of the basic components of the ratio detector.

This section will only touch upon the basics of the ratio detector. The design of the ratio detector can be optimised to great extend, however, due to the time frame it is out of the scope of this project. A functional ratio detector can instead be constructed with a basic understanding of the operation.

7.3.1 General mode of operation

As previously mentioned a frequency dependent circuit is required in order to demodulate an FM signal. The ratio detector is utilising the addition of phase shifted signals. The signals in the secondary winding of the transformer can be seen on figure 7.6.

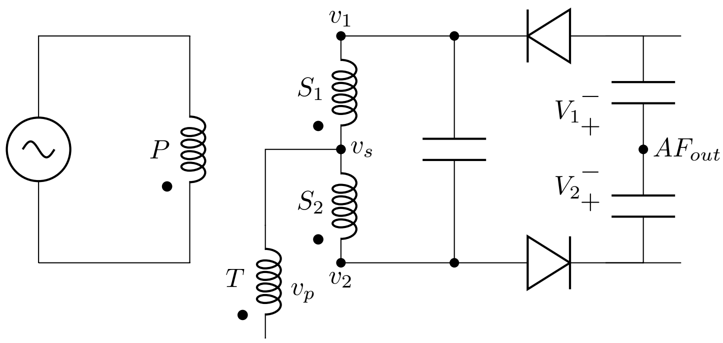


Figure 7.6: Overview of the relevant signals in the ratio detector. The dots are standard notation for relative direction of magnetic coupling.

The signal v_p from the tertiary winding is directly coupled with the primary and is therefore in phase with the primary. The phase of v_s is dependent on the the frequency since it is part of a tuned tank circuit. The phase of the secondary is as illustrated on figure 7.7.

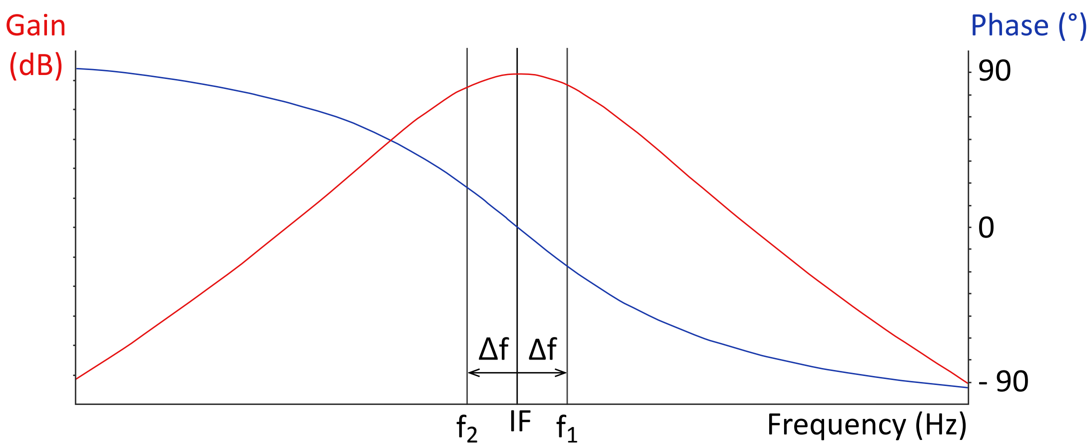


Figure 7.7: Phase and gain of tank circuit around the IF. Blue is phase. Red is gain.

The secondary tank circuit is 90° out of phase with the primary at IF, because it is centre tapped. The two parts of the secondary coil is 180° out of phase relative to each other. Again referring to figure 7.6 the signal at v_1 is equal to the sum of the primary and half that of the secondary. The signal at v_2 is equal to the sum of the signals from the primary, and half that of the secondary. But since the relative signal direction is reversed relative to the tertiary, the resulting signal is equal to the signal of the primary subtracted from half the signal at the secondary. Thus these signals can be expressed as:

$$v_1 = v_p + \frac{v_s}{2} \qquad [\mathrm{V}] \quad (7.2)$$

$$v_2 = v_p - \frac{v_s}{2} \qquad [\mathrm{V}] \quad (7.3)$$

The relation of the signals are illustrated with vectors in the complex plane in fig 7.8.

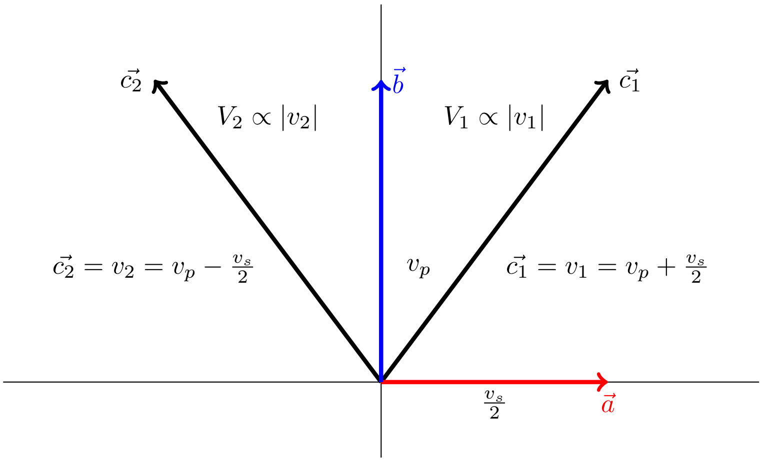
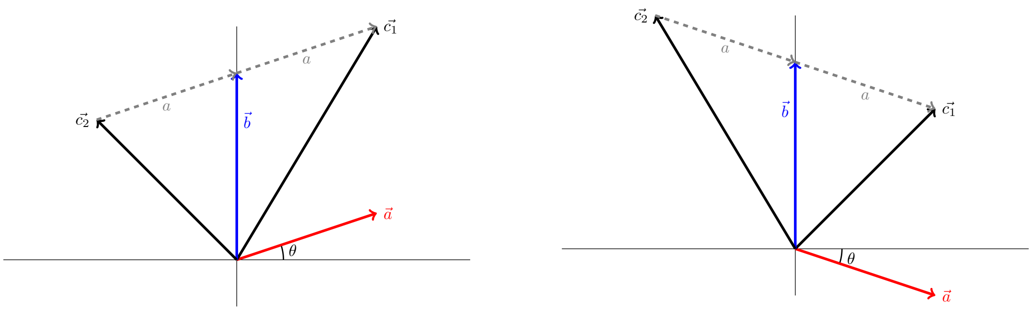


Figure 7.8: Vector representation of the signals in the secondary of the transformer.



(a) Vector representation of the signals in the ratio detector above IF.

(b) Vector representation of the signals in the ratio detector bellow IF.

Figure 7.9: Vector representation of the signals in secondary of the transformer in the ratio detector.

The signals will now be referred to by their respective vector names, and thus:

$\vec{a} = v_\mathrm{p}$ is the signal from the primary [V]

$\vec{b} = \frac{v_\mathrm{s}}{2}$ is half the signal from the secondary [V]

$\vec{c_1} = v_1$ is the voltage into D_1 [V]

$\vec{c_2} = v_2$ is the voltage into D_2 [V]

When the phase of $\vec{a}$ shifts, the magnitude of $\vec{c_1}$ and $\vec{c_2}$ change accordingly as seen on figure 7.9. The two vectors $\vec{c_1}$ and $\vec{c_2}$ are representing v_1 and v_2. The magnitude of v_1 and v_2 are proportional to the Voltage V_1 and V_2 seen on figure 7.6. The demodulated signal is extracted at AF_{out}, and is equal to:

$$v_\mathrm{AF} = V_1 - V_2 \qquad [\mathrm{V}] \tag{7.4}$$

This is roughly the same functionality that the Foster-Seeley detector utilise. The problem is when the amplitude of the input signal varies then the magnitude of both V_1 and V_2 varies. This can be expressed as k_s in equation where any variation of k_s would be multiplied with audio output and thus resulting in potentially audible variations. 7.5

$$k_s \cdot v_\mathrm{AF} = k_s \cdot V_1 - k_s \cdot V_2 = k_s(V_1 - V_2) \qquad [\mathrm{V}] \tag{7.5}$$

this can be taken care of by keeping the voltage $V_1 + V_2$ constant, this will cause the output to only be dependent on the ratio between V_1 and V_2, and thereby the name 'ratio detector'. This can be achieved by shunting a capacitor across the output of the diodes, with a time constant of less than 0.2 sec. The capacitor cannot change in an audible frequency. This will essentially remove the AM signal. A compensation for the nonlinear nature of a real diode can be achieved by adding different resistances between the constant voltage and the two diodes to offset the unbalance.

7.3.2 Circuit design

This section will focus on designing a circuit that operates as described in the previous section 7.3.1, and realise the components needed to accomplish the product specification.

Transformer

The transformer is the core of the ratio detector. The transformer was made by winding the primary, secondary and tertiary coil around the same toroid. The selected model was the T50-3 which was made for frequencies between 20 kHz and 1 MHz, and is fine for 455 kHz. The wire was wound as seen on figure 7.10.

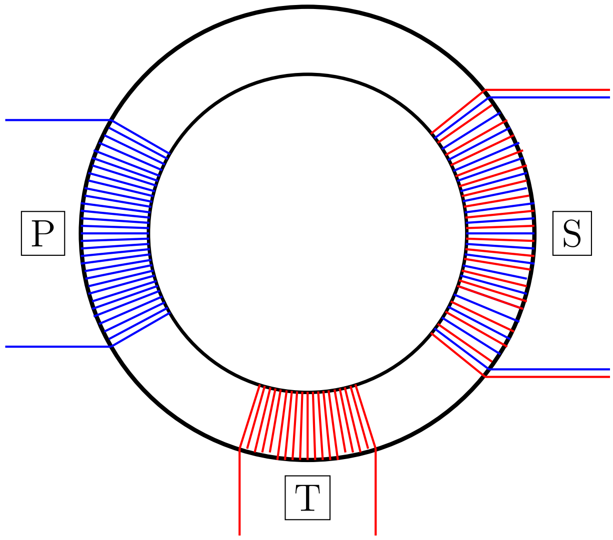


Figure 7.10: The windings on the torodial transformer

The primary winding is marked with a P on figure and was wound to have an impedance of approximately 200µH. In order to calculate the required windings, the simplified equation found in the data sheet[48] 7.6 was used.

$$N = \sqrt{\frac{L_{\text{desired}}}{A_L}} \quad [\cdot] \quad (7.6)$$

N is the required turns [·]

L_{desired} is the desired impedance [nH]

A_L is constant for toroid by the manufacturer [nH/N^2]

Thus the the primary windings is calculated in equation 7.7, and the A_L of the T80-3 toroid is 18.

$$N = \sqrt{\frac{200 \cdot 10^3}{18}} \approx 105 \quad [\cdot] \quad (7.7)$$

It was then wound 105 times, and the measured impedance matched the calculation. The secondary winding was based on a turn ratio found in the source [46] relative to the primary to have an inductance of 108 µH and was wound with bifilar wire to match the inductance of the two parts of the secondary. The tertiary was wound to have a inductance of 6.4 µH based on the same source as the secondary. It was decided not spend additional time to further investigate the turn ratio as it was not specifically explained in the source material. The factors considered when designing the coils was the time required to wind the coil and how easy it would be to tune the capacitance. Since the more windings the smaller the capacitor needed and the easier tuning which is explained in the next section, but also the more time spend winding the coil which is a time consuming task.

7.3.3 Capacitance for Primary and Secondary Tank Circuit

The ratio detector require two tank circuits tuned at the IF frequency of 455 kHz. The tuning is required since the performance require exact values, and because of factors such as stray capacitance the numbers can't be exactly calculated. The two tank circuits each consists of their own respective inductor that is wound on the transformer, and a capacitor. Equation 7.8 is used for calculating the resonant frequency of a tank circuit.

$$f_0 = \frac{1}{2 \cdot \pi \cdot C \cdot L} \qquad [\text{Hz}] \quad (7.8)$$

f_0 is the resonant frequency [Hz]

C is the capacitance [F]

L is the inductance [H]

As it is desired to be able to fine tune the tank circuits. And since the largest tuneable capacitor available can only be tuned to a capacitance between approximately 40pF-200pF, it is required to use a parallel capacitor or a large inductor. To be able to tune the circuit the parallel capacitor should not be much larger than the tune able one. Since capacitors size increase with approximately 10% between the available values, it is decided that the tuneable capacitor should be at least at 10% the capacitance in the tank circuit. Thus the maximum capacitance in the tank circuit is calculated by equation 7.9

$$C_{\text{totalMax}} = C_{\text{tuneable}} \cdot 10 = 200\text{pF} \cdot 10 = 2\text{nF} \qquad [\text{F}] \quad (7.9)$$

C_{totalMax} maximum desired capacitance [F]

C_{tuneable} tuneable capacitance [F]

When designing the tank circuit the maximum capacitance should not exceed 2 nF. A lower value is preferable since it will make it easier to tune. This was considered when designing the transformer, and the exact values can now be calculated. Equation 7.8 was used by isolating the capacitance C, resulting in equation 7.10 for the primary capacitance, and equation 7.11 for the secondary capacitance.

$$C_P = \frac{1}{2 \cdot \pi \cdot 455\text{kHz} \cdot 200\mu\text{H}} = 611\text{pF} \qquad [\text{F}] \quad (7.10)$$

$$C_S = \frac{1}{2 \cdot \pi \cdot 455\text{kHz} \cdot 108\text{µH}} = 1139\text{pF} \qquad [\text{F}] \quad (7.11)$$

The value of the static capacitor should be close to the value of the total capacitance subtracted from half the maximum variation of the variable capacitor, and thus the static capacitor is calculated in equation 7.12 for the primary tank, and in equation 7.13 for the secondary.

$$C_{\text{Pstatic}} = C_{\text{P}} - \frac{C_{\text{tuneable}}}{2} = 611\text{pF} - \frac{200\text{pF}}{2} = 511\text{pF} \qquad [\text{F}] \quad (7.12)$$

$$C_{\text{Sstatic}} = C_{\text{S}} - \frac{C_{\text{tuneable}}}{2} = 1139\text{pF} - \frac{200\text{pF}}{2} = 1039\text{pF} \qquad [\text{F}] \quad (7.13)$$

By the values from the calculations we can now decide which components are needed to realise the capacitance's for the primary and secondary tank circuit. For the primary a 470pF static capacitor in parallel with a 200 pF variable capacitor would be sufficient and for the secondary a 1000pF static capacitor in parallel with a 200 pF variable capacitor would suffice.

7.3.4 Tertiary winding on the transformer

The tertiary winding on the transformer will not be in a tuned tank circuit as its function is to tap the signal from the primary. This could be done in other ways, but by using a winding on the transformer impedance matching should not be needed. The resistor in parallel can be used to adjust the peak current of the diodes and can be adjusted to further improve the AM rejection by tuning, however this will not be explained further as it is out of the scope of this project.

7.3.5 Rectification circuit

The diodes in this circuit are an essential part of the ratio detector as they allows for the voltage differences V_1 and V_2 seen on figure 7.6. The diodes has to be high frequency diodes with as low internal capacitance, the diodes used is the 1N4748 as they were the best ones available.

7.3.6 Diode Rebalance Resistors

The resistors R_2 and R_5 seen on figure7.5 are implemented because of the unlinearity of the diodes, and are used to balance out these unlinearities. The basic function is to bias the diodes. Instead of calculating the value of the resistors, which is out of the scope of this project, they will instead be replaced by potentiometers which can be used for further tuning. These resistances can be tuned to virtually eliminate the effect of the AM signals, simply by tuning.

7.4 Constant voltage circuit

As described is section 7.3.1 it is required to keep the voltage V_1+V_2 constant to eliminate the effect of AM on the audio output. This could be done either by a voltage source such as a battery or a voltage supply. But same result can also be accomplished by shunting a capacitor across the circuit as seen on figure 7.5. To remove any audible amplitude variations a time constant below the human hearing range is required. A time constant of around 0,2 seconds is recommended, and a too large time constant can have undesirable effects. Because the exact calculations of the effect of this circuit is out of scope for this project, a recommended setup with a time constant of 0,1 second is designed. Thus the resistors are selected at 6.8kΩ and the capacitor at 8 µF. The time constant is then calculated in equation 7.14.

$$\tau = R \cdot C = (6.8\text{k}\Omega \cdot 2) \cdot 8\mu\text{F} = 0,109\text{s} \qquad [\text{s}] \qquad (7.14)$$

7.5 De-Emphasis Circuit

As described in section 2.5.4 the FM signals are pre-emphasised before they a modulated. Therefore a de-emphasis circuit is needed. The de-emphasis circuit is a standard circuit. It is a low-pass filter with a time constant of approximately 75 µs. A standard high-pass filter with a capacitor at 2 nF and a resistor with a resistance of 39 kΩ. The time constant is calculated in equation 7.15.

$$\tau = R \cdot C = 39\text{k}\Omega \cdot 8\mu\text{F} = 78\mu\text{s} \qquad [\text{s}] \qquad (7.15)$$

7.6 Other circuit considerations

The majority of the ratio detector circuit has now been accounted for. The remaining components on figure 7.5 will now be addressed. The Resistor R_1 is for tuning the bandwidth of the primary tank circuit, and will be implemented as a potentiometer.

The capacitor C_7 is for decoupling the IF signal from the tertiary and the secondary coils.

The capacitor C_6 is to avoid any DC leakage into the load.

The capacitors C_3 and C_4 is to only extract the AC signal across the diodes.

7.7 Tests

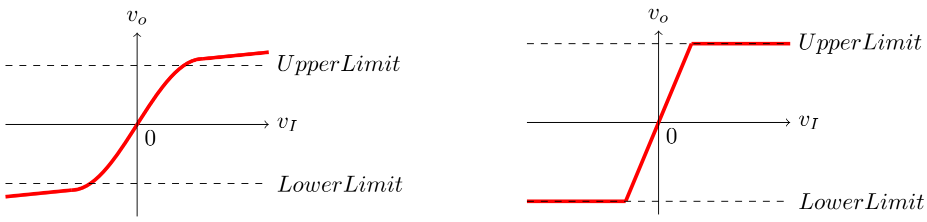
After designing the demodulator the first implementation was made on a breadboard. The initial tuning was done by using an oscilloscope and doing an FFT and adjusting the potentiometers and variable capacitors to get the lowest harmonic distortion. It was however at this point that a mistake was made. While tuning the detector the results seemed to get better by reducing the capacitance, and thus a smaller static capacitor was used. This improved the performance in an unintended way, it was assumed that some calculations where wrong, since the results got better. The problem was however that the tuning was done on very little information. It was purely based on the numbers. This will be further explained in the final test section. When the sound seemed convincing it was decided to get the design on some strip board. After getting the design on some strip board it was once again tuned, this time it was used by repeatedly tuning it for both THD and AM rejection. And while results where good when measuring the THD, the AM rejection was unsatisfactory . The poor AM rejection could partly be because of the aforementioned mistake. However at the time it was decided that a limiter was needed, which ended up solving multiple problems anyways.

7.8 Test review and the need for a limiter

Due to the unsatisfactory AM rejection of the demodulator, it has been decided to construct a limiter which should improve the AM rejection.

7.9 Limiter

To accommodate the Ratio Detector by removing unwanted AM signals or noise, a limiter will be designed and implemented.
The main advantages of implementing a limiter circuit prior to the FM Demodulator, is that the quality/precision of the demodulation would benefit from receiving a prescribed standard waveform.



(a) Soft limiting. **(b)** Hard limiting.

Figure 7.11: Illustration of soft and hard limiting.

7.10 Theory

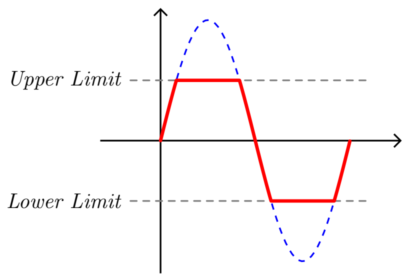
The theory in this section is based on [18, p. 217-220].

A limiter is an electrical circuit which purpose is to produce saturation on the output by preventing its output from exceeding a predetermined value.
There are two different types of limiting, either hard limiting or soft limiting, the difference between these can be seen in figure 7.11.
Hard limiting occurs when the input waveform reaches either upper or lower limit in the limiter. At either of these points the slope of the input becomes zero, thus clipping the output waveform. This is the ideal limiter characteristic. Soft limiting is a bit different. Instead of the slope instantly becoming zero, it decreases as it reach either the upper or lower limit, thus creating a smooth transition.

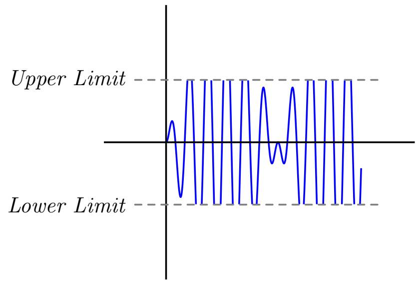
Furthermore there are both single limiter and double limiter circuits. Two illustrations of limiting a sine wave and an AM can be seen on figure 7.12. A limiter is called a single limiter when it only limits either the upper or the lower limit and a double limiter is when both limits are present.

7.10.1 Requirements

To best aid the demodulators am rejection the limiter should be clipping as hard as possible. It is also desired to have a gain to also increase the performance of the demodulator itself, as a higher input gives a better AM rejection of the demodulator.



(a) Soft limiting.



(b) Hard limiting.

Figure 7.12: Illustration of limiting of a sine wave and an AM wave, where the x axis is v_I and the y axis is v_o with respect to time.

7.10.2 Design

A double hard limiter design has been chosen. This is done due to the fact that the demodulator benefits from getting a prescribed standard waveform, hence the need for the double hard limiter.
An op-amp by the model AD844 will be used.The limminting is a result of the op-amp amplifying the signal beyound the suplyvoltage. This has numerous advantages. First of the signal is amplified to the maximum violtage available in the system, resulting in a 24 V peak-to-peak signal. And at the same time the signal will be clipped. The reason for choosing this particular op-amp is that its slew rate of 2000V μs^{-1} enables it to shift between a positive and negative voltage more than 530 times a second without too much of a performance drop as is required at the maximum FM signal.

The final design can be seen on figure 7.13.

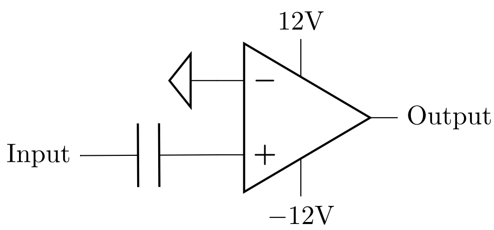


Figure 7.13: AD844 limiter design.

The capacitor on the input of the op-amp is simply to remove any DC bias from the signal that else would have passed through. The +- 12 V power supplies are equal to the power used in the rest of the radio receiver system. Furthermore no feedback has been chosen, because it will result in more gain, which in this case is better.

7.10.3 Tests

In this section there will be a brief description of how the tests were conducted and an evaluation of the results.

Undocumented tests

Once the limiter was constructed a test with the limiter and demodulator was made to make sure they behaved as expected. These tests where not documented as they where simply to check that they worked as intended. It was also not documented as they were supposed to be implemented on a PCB before the tests. The tests at were however great and with a THD measured at -42 dB and when an AM tune at 80% was send into the system and was almost inaudible and could not be seen on the FFT, but These data was not recorded, as it was assumed a better result would be measured once implemented on the PCB. However, these test will not be compared with the product specifications as they are not documented and the circuits has been disassembled when the PCB was made. But they are mentioned to give a better understanding of the problems with the final implementation.

The final tests of the demodulator

The final tests of the demodulator where made on a PCB implementation. The demodulator were tested for THD, frequency response, SNR and AM rejection.

The first test described in test journal E was done to measure the frequency response. The response can be seen with the relevant frequencies marked on figure 7.14. Just like the expected the characteristic that is shown on figure 7.3 the demodulation is happening on a linear part of the characteristic, because of the empirical nature of the way it was tuned, th circuit was changed and thus the wrong part of the characteristic is used for demodulation. The tuning was done like this because it was first realised how the characteristic could be measured in time for the final test. And there was no time to re solder the circuit and tune the demodulator again to make a new test. It is likely that the reason some of the early tests had better results. The characteristic also suggest that the other tests will have unsatisfactory.

The next test was to test both the THD and AM rejection of the final demodulator. The

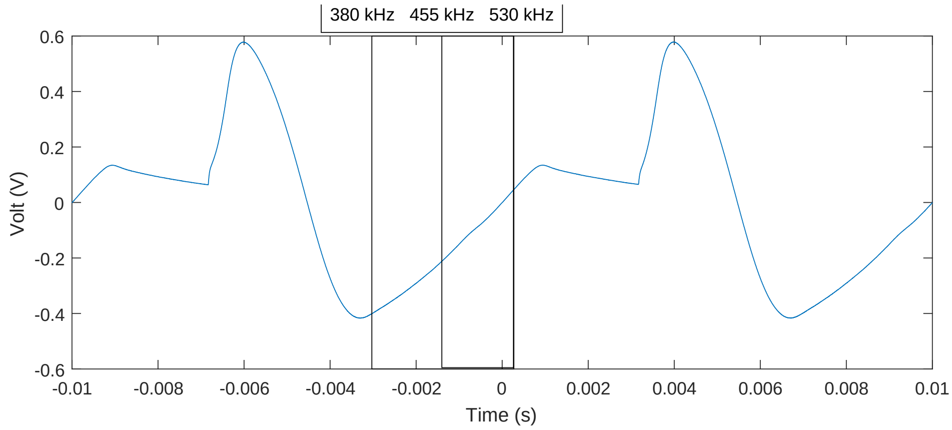


Figure 7.14: The frequency response of the demodulator. The IF and the maximum deviation in frequency is marked on the characteristic.

results can be seen in the test journal D. The results show that with no AM the THD is at -24dBc. And the SNR is at 32 dBc. As the AM modulation is increased the THD starts falling towards -33dBc at 80 % AM. And the SNR falls to a mere 3 dBc at 80 % AM.

The final test was made to test the gain of the demodulator with and without the limiter, see test journal F. The results where noted and thus the conclusion on that test is that the demodulator with the limiter will have a output of around 316 mV if the input is grater than 12 mV.

7.10.4 Review of tests

It is clear that the requirements 4.3 are not met. The THD should be below -40 dBc to met the requirements. The THD was measured to be -24 dBc for the final implementation which is 16 dBc off. The SNR was at best at 32 dBc, while the requirements was 56 dBc this is 24 dBc off. It is however likely that the performance could have been better if not for certain mistakes that were previously discussed in section 7.7.

Chapter 8

Antenna & Low Noise Radio Amplifier

The Antenna and low-noise amplfier (LNA) modules in the superheterodyne radio receiver is highlighted in figure 8.1.

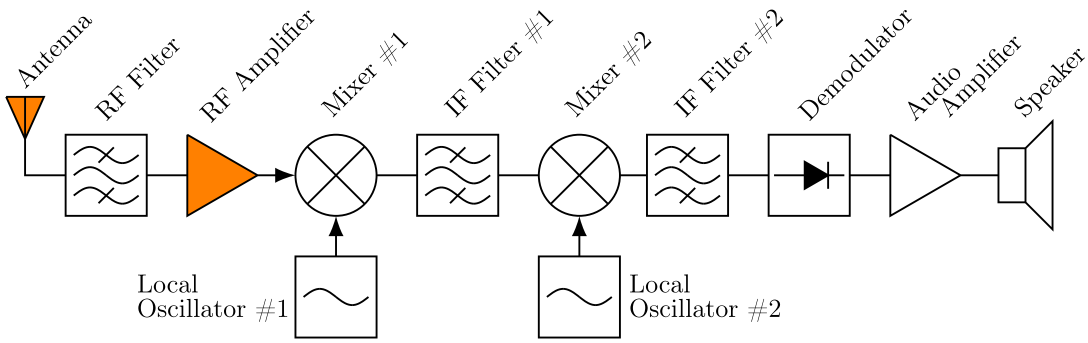


Figure 8.1: Antenna and LNA modules highlighted in the overall superheterodyne radio receiver design.

8.1 Theory

A lot of theory covering the antenna has already been highlighted in section 2.3 and 2.4. In addition to this, amplifiers have also been covered in chapter 3.

8.2 Requirements

The requirements for the antenna is that it should be able to receive broadcasted signals between $87.5 - 108$ MHz while maintaining a dBi above -20.

8.3 Design

For the actual design of the antenna prefabricated equipment were used. For the antenna a BicoLOG 30100E[49] was used. According to its datasheet the antenna has a gain of around -5dBi to -3.5dBi in the frequency domain between $87.5 - 108$ MHz. This is satisfactory to the requirements.

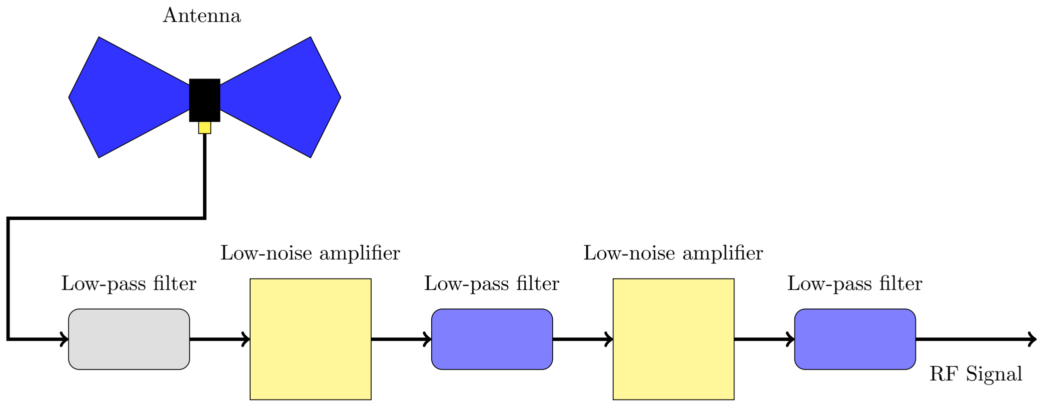


Figure 8.2: Antenna design.

Because of the antennas low gain, it was necessary to implement two low-noise amplifiers of the model ZFL-500HLN[50]. To avoid unwanted oscillations from the amplifiers additional three low-pass filters were added. The first one a SLP-300+[51] was connected to the output of the antenna and then the remaining two SLP-150+[52] were connected in continuation. These can be viewed as a black box. An illustration of the design setup can be on 8.2.

8.4 Tests

Since the equipment used in this chapter was prefabricated high-end equipment used for research projects it was deemed unnecessary to conduct test any tests other than observing that they are not broken.

8.5 Conclusion

There was not time to design and implement an antenna and LNA. Therefore the prefabricated solution was decided upon, and worked without problems after counselling about the setup and use of filters.

Chapter 9

Audio Power Amplifier

This chapter will go through the design of a class A audio power amplifier, and the following tests of the amplifier. The audio power amplifier module in the superheterodyne radio receiver is highlighted in figure 9.1.

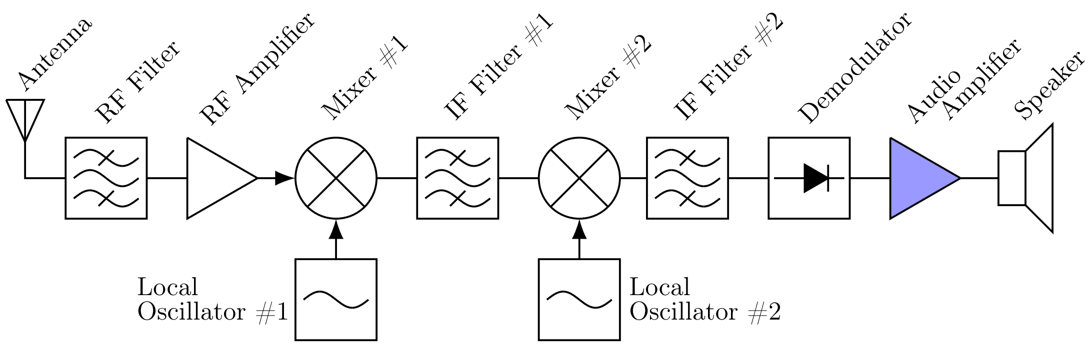


Figure 9.1: Audio power amplifier module highlighted in the overall superheterodyne radio receiver design.

9.1 Introduction

When the music from a radio station has been received and demodulated by the radio stage, it is necessary to have an audio power amplification stage to be able to play the music through a loudspeaker. As the main focus of this report was to design and build an FM radio receiver, the Class A type of amplifier was chosen because of the low distortion associated with this class. As a result of this, the amplifier should not be the limiting factor in the quality of the sound produced by the loudspeaker.

9.2 Amplifier Specifications

As mentioned earlier in section 4.2, the amplifier has to meet some specifications based on international standards. These specifications can be seen in table 4.1 in section 4.2.

9.3 Design

When designing a class A amplifier, different design topologies may be chosen based on the wanted complexity and efficiency of the amplifier. In this report the most simple of the topologies, the common emitter configuration (CE), was chosen due to time restrictions and no requirements for the efficiency of the amplifier. As the power source used for the amplifier was able to deliver 2 A and 30 V, the design of the amplifier has been made to accommodate this restriction.

9.3.1 Power amplifying stage

When designing an audio power amplifier, it is custom to start at the power amplifying stage and work inwards towards the audio signal source. The power amplifying stage is chosen to be of the CE configuration, and an example of this configuration is seen in figure 9.2

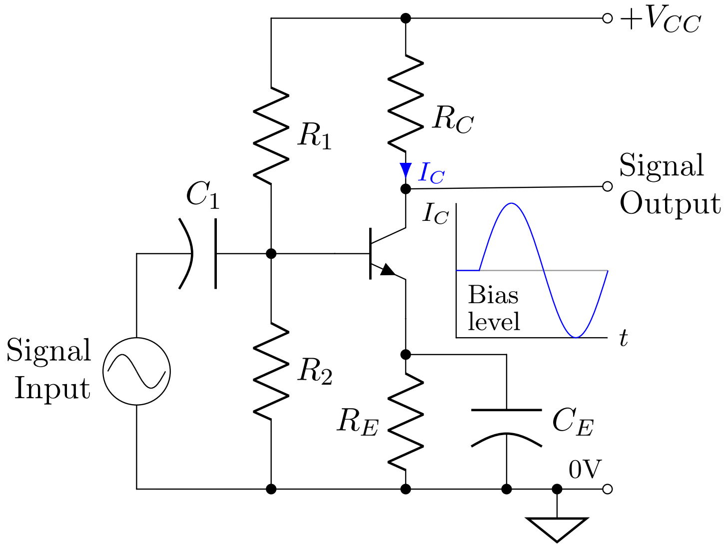


Figure 9.2: Illustration of a CE configuration class A amplifier.

As seen on the figure, the output AC current lies around a bias level, which has to be at least the size of the negative amplitude of the signal to avoid clipping of the signal. According to the HIFI standards, the power in the loudspeaker has to be at least 10 W per channel. As the chosen loudspeaker has an impedance of 8 Ω, the needed current through it is calculated as

$$\begin{aligned} I_{RMS} &= \sqrt{\frac{P}{R_L}} \\ &= \sqrt{\frac{10W}{8\Omega}} \\ &= 1.1A \end{aligned} \tag{9.1}$$

This is an RMS-value, and to find the peak-peak value the RMS-value has to be multiplied by two times the square root of two.

$$\begin{aligned} I_pp &= I_{RMS} \cdot 2\sqrt{2} \\ &= 1.1A \cdot 2\sqrt{2} \\ &= 3.16A \end{aligned} \tag{9.2}$$

This is beyond the 2 A available from the power source, which means that the signal would clip at the top if the output power should become 10 W. To ensure that the signal does not clip, the peak current can not go beyond 2 A, which results in an RMS-value of

$$\begin{aligned} I_{RMS} &= \frac{I_p}{2\sqrt{2}} \\ &= 0.707A, \end{aligned} \tag{9.3}$$

which gives a power through the loudspeaker of

$$\begin{aligned} P &= I_{RMS}^2 \cdot R_L \\ &= 0.707^2 A \cdot 8\Omega \\ &= 4.0W \end{aligned} \tag{9.4}$$

Thus the maximum power delivered to the loudspeaker is 4 W, and henceforth this will be the design parameter for the amplifier.

This results in a requirement for the transistor to be able to withstand a current of 2 A and a V_{CE} of 30 V if the rest of the circuit shorts out. The transistor chosen is the MJE3055 NPN power transistor which according to its datasheet can withstand a collector current of 10 A and a collector-emitter voltage of 60 V.

Operation Point

The operation point has to placed sufficiently high, to ensure that the voltage signal over the load does not clip at the saturation voltage of the transistor, but not so high that it clips at the voltage source. A current with an RMS-value of 0.707 A through a load of 8 Ohms results in an RMS voltage of

$$\begin{aligned} V_{RMS} &= I_{RMS} \cdot R_L \\ &= 0.707A \cdot 8\Omega \\ &= 5.65V \end{aligned} \tag{9.5}$$

Which corresponds to a peak voltage of, $V_p = 8V$, and thus the operating point should be at least 8 V above the collector-emitter saturation voltage $V_{CE,sat}$. The saturation voltage for the MJE3055T transistor at 1 A, is rated at 0.1 V in its datasheet. [53] The operating point is chosen to be at $Q(1A, 11V)$, giving the AC-signal a buffer of 3 V before getting into the saturation part. This ensures that the negative part of the signal is not clipping, when at its peak. To ensure the same for the positive part of the signal, the collector resistor has to be dimensioned to allow for the same peak voltage. This is because the maximum positive signal peak is found as

$$V_{ac+,peak} = I_C \cdot R_C$$

To allow for the same maximum peak as the minimum peak, the resistor is chosen to be 11 Ohm.

This leaves the rest of the voltage from the voltage source at the emitter resistor, R_E, and thus the resistor value can be calculated using Ohms Law, when assuming the emitter current is equal to the collector current.

$$\begin{aligned} R_E &= \frac{V_{CC} - V_{RC} - V_{CE}}{I_C} \\ &= \frac{30 - 11 - 11V}{1A} \\ &= 8\Omega \end{aligned} \tag{9.6}$$

The emitter resistor is placed in the circuit ensure stability of the transistor. This is because when the transistor heats up and the current through it increases, the increased current through the resistor results in a higher voltage across it. This in turn forces a reduction in the base-emitter voltage which in turn reduces the current through the transistor.

Biasing

According to the datasheet for the MJE3055T transistor, the DC-current gain at a collector current of 1 A is approximately 80. Using this, the base current, I_B, is determined by

$$\begin{aligned} I_B &= \frac{I_C}{h_F E} \\ &= \frac{1A}{80} \\ &= 12.5mA \end{aligned}$$

which has to be delivered by the bias resistors R_1 and R_2 to the base of the transistor. Using Kirchhoffs current law, an equation for the current at the point between the bias resistors and the base is formed, which can be seen in equation 9.7.

$$I_{R1} - I_B - I_{R2} = 0 \qquad [\mathrm{A}] \quad (9.7)$$

The value of resistor R_2 is arbitrarily chosen to be 1000 Ω as a starting point for the design. As the voltage across the resistor must be the same as the total voltage across the base-emitter part of the transistor, V_{BE} and the emitter voltage V_E, the current through R_2 is calculated using Ohms law as shown in equation 9.8

$$I_{R2} = \frac{(V_{BE} + V_E)}{R2} \qquad [\mathrm{A}] \quad (9.8)$$

In the datasheet is found a value for V_{BE} of 1.8 V at $V_{CE} = 4V$ which results in a current of 9,8 mA. Using equation 9.7, the current through R_1 is found to be 22,3 mA, and considering the voltage division between R_1 and R_2 the voltage is found to be 20,9 V. Again using Ohms law, the resistor value of R_1 is calculated to be 905,8 Ω. Lastly the capacitor C_E, which shorts for AC-signals and thereby connects the emitter to ground for AC-signals, is chosen to have a large capacitance to ensure its impedance acts as short circuit. A capacitance of 220 µF is deemed sufficient for AC grounding the signal. The power amplifying stage, with calculated component values is illustrated in figure 9.3

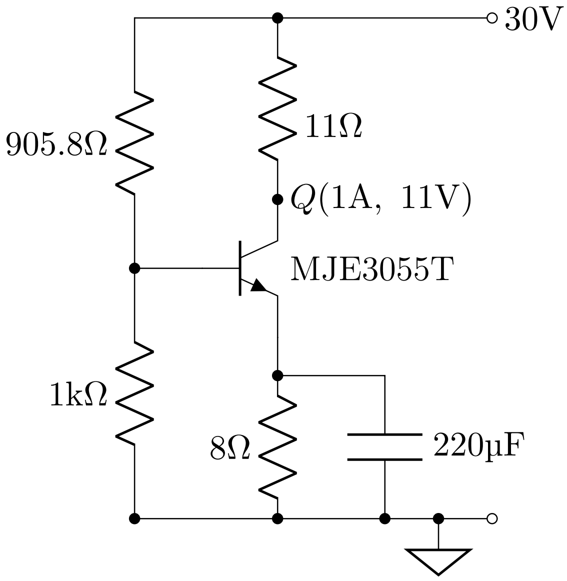


Figure 9.3: Illustration of the power amplifying stage with calculated component values.

Analysing the voltage gain of the power amplifying stage to determine if further voltage amplifying stages are needed before the power amplifying stage, equation 9.9 from [18] is used

$$A_v = -g_m \cdot R_L \qquad [\cdot] \quad (9.9)$$

where g_m is the transconductance determined by

$$g_m = \frac{I_C}{V_T}$$

where V_T is the thermal voltage, which at 25 degrees Celsius is 25,7 mV. This results in a voltage gain of

$$A_v = -\frac{1A}{25.7mV} \cdot 11\Omega = 428 \qquad (9.10)$$

This is a very large voltage gain compared to the voltages between 0.2 V og 2 V that is expected according to the standards. Thus some attenuation of the signal is necessary before the power amplifying stage. This attenuation is produced by a simple voltage divider right after the input signal. The desired voltage at the output is 8 V, which means that the needed voltage gain is

$$A_v = \frac{8V}{0.2V} = 40 \qquad (9.11)$$

and thus the signal has to be attenuated by a factor of $\frac{428}{40} = 10.7$. This is accomplished with a voltage divider with resistor values of 1 kΩ and 9,7 kΩ.

The final design is shown in figure 9.4.

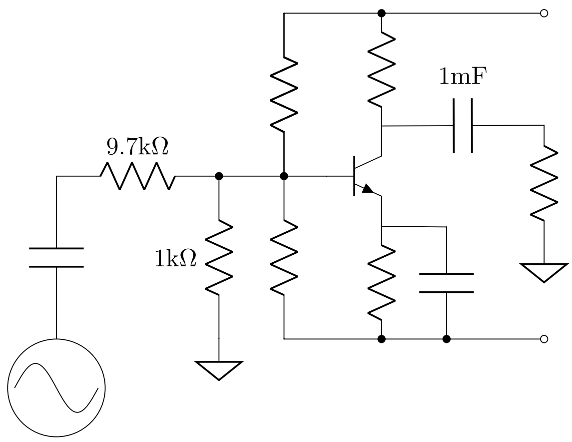


Figure 9.4: Illustration of the final audio power amplifier circuit.

Between the load and the amplifier is placed a decoupling capacitor of 1 mF, to remove the DC-offset at the operating point of the transistor so only the AC signal is sent to

Table 9.1: Table of measured values in LTSpice for the DC operating point.

Measurement	Expected value	Measured value	Deviation
Operating point voltage DC	11 V	7.6 V	−31\%
Operating point current DC	1 A	1.17 A	17\%
Base current DC	12.5 mA	11.4 mA	8.8\%

the loudspeaker. The capacitor is chosen large as it acts as a filter together with the loudspeaker, and thus should let all audio frequencies pass.

9.4 Simulations

To test the design before constructing it in the laboratory, a simulation was created in LTSpice. The circuit is first simulated for DC to determine if the calculated DC operating point was achieved. The results is shown in table 9.1.

It is clearly seen that the measured values deviate from the expected values. The operating point seems to have shifted from 11 V and 1 A, to 7.6 V and 1.17. This will result in clipping of the negative part of the signal. Next, the circuit is simulated with an input signal of 0.2 V and frequency of 1 kHz. This simulation showed much more attenuation at the load than expected. An additional simulation was then made, without the voltage divider, and this showed that the voltage gain of the amplifier was much lower than anticipated, and thus it was decided to continue without the voltage divider during the tests of the amplifier.

9.5 Tests

Following the simulations from LTSpice, it was chosen to conduct the tests of the amplifier without the designed voltage divider between the input signal and the power amplifying stage as it proved unnecessary. Due to time restraints most of the requirements were not tested for, and as a result of this can not be concluded upon. The requirements tested for was the effective frequency range and the total harmonic distortion. These tests have each been documented in a test journal which are found as appendices to the report.

9.5.1 Effective frequency range

In accordance with the standards the effective frequency range was determined based on a frequency response analysis of the amplifier. The test journal documenting this test is found in appendix H.4. The result of the test was that the amplifier had an effective frequency range from approximately 780 Hz to 1330 Hz equal to a range of 550 Hz. Compared to the requirement in section 4.2 that requires a range from 20 Hz to 20 kHz equal to approximately 20 kHz, there is a deviation of -97.25 % between the measured results and the required specifications.

9.5.2 Total harmonic distortion

The purpose with this test was to determine the maximum total harmonic distortion (THD) of the amplifier. The test journal documenting this test is found in appendix H.8. The result of this test is that the maximum THD of the amplifier is 49.5 %. Compared to the requirement in section 4.2 that requires a THD of maximum 0.7 %, there is a deviation of 6971 % between the measured results and the required specifications.

9.6 Discussion

The design and implementation of the audio power amplifier did not go very well. The design was started too late, and thus not enough research into the theories behind amplifiers was achieved in time to implement in the designing phase. After the CE configuration had been designed and simulated, showing some errors in the design, it was chosen to continue with the design despite the deviations from the design calculations as there was no time left for an entirely new design. The circuit was constructed, though based on the simulations from LTSpice the voltage divider at the input was omitted, and the tests were conducted with this amplifier. The results of these tests will be discussed in detail later in this section.

Simultaneously with testing the already constructed amplifier, some time went into research for a better design. This research did not get implemented into the final design, but it explained some of the difficulties and errors we encountered during the test of the amplifier. In the following paragraphs the results of the research will explained, and an idea for what seems like a better design is described, though it has not been constructed or tested.

9.6.1 Results of additional research

First of all did the research reveal some fundamental errors in the way the CE configuration was designed, and these errors will be described first. When designing a power amplifying stage for a loudspeaker which has a very low resistance, it is necessary to consider AC load line when designing the circuit with regards to the DC operating point.

The AC load line will always be steeper than the DC load line, because the maximum AC voltage is dependent of the load resistor in the loudspeaker in parallel with the collector resistor. During the design in section 9.3, this aspect was not taken into consideration as it was assumed that the AC-signal was able to go from the transistor saturation voltage to the source voltage, i.e that it only depended on the collector resistor. This would result in a maximum positive AC voltage swing of

$$\begin{aligned} V_{AC} &= I_C \cdot R_L||R_C \\ &= 1A \cdot 8\Omega||11\Omega \\ &= 1A \cdot 4.6\Omega \\ &= 4.6V \end{aligned}$$

which is close to half of the needed 8 V. To compensate for this, the circuit should be designed for less power dissipated in the loudspeaker.

Additionally when choosing the resistors for the bias of the transistor, a rule of thumb from the course "Analog Circuit Design" was forgotten. This rule of thumb regarded the current through the resistors, which should be about 10 times the base current. [54] According to [55] this is to stabilise the biasing of the transistor. Instead of this, a current twice the base current was chosen for R_1 and a current equal to I_B for R_2. This might have resulted in the bias of the transistor getting shifted unpredictably.
Lastly the choice of the emitter resistor, R_E, did not take into consideration another rule of thumb that recommends that the voltage across R_E should be between $10-15\%$ of V_CC.[55] Instead approximately a fourth of V_CC was placed across R_E. This resistor causes stability in the circuit, and should still do this even though it is larger than necessary, but the larger resistor value results in a larger power dissipated and the resistor gets very hot.

If these design considerations were to be implemented in the design, one would expect the resulting amplifier to produce better results when tested. But even then, the requirement of a power of 10 W in the speaker would still not be realisable, due to the way the CE configuration works. Instead a different configuration should be chosen, that allows for a larger power in the load. Another used configuration is the Common-Collector (CC) configuration together with an active current source.

The CE configuration has a voltage gain of approximately 1, which is why it is also called the emitter follower as the voltage at the emitter follows that at the base. This configuration, together with an active current source to hold the operating point, eliminates the need for a resistor to set operating current. The current gain of the CC configuration is defined as

$$A_i = \beta + 1 \tag{9.12}$$

so when used with the MJE3055T transistor the current gain would be approximately 81 times. As there is no voltage gain, it is necessary to have a voltage amplifying stage

(VAS) before the power amplifying stage. This stage can be designed using various design topologies which will not be described further in this report.
The VAS has multiple purposes, one is adding a large voltage gain and another is ensuring stability of the entire audio power amplifier when used together with negative feedback in the circuit. Adding negative feedback to the circuit is another way to improve the characteristics of the audio amplifier, as it helps decrease harmonic distortion and stabilises the gain of the amplifier. If negative feedback is to be implemented another stage, called a differential stage, has to added to the system between the VAS and the audio signal input. This stage consist of a differential amplifier where one input is the audio signal and the other is the negative feedback.

Thus a design of a better class A audio amplifier could be made with a CC configuration together with a VAS, differential amplifier and negative feedback. A block diagram of such a system is shown in figure 9.5

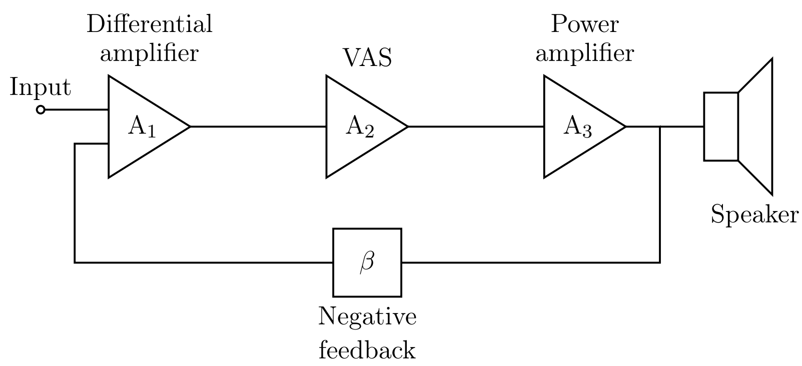


Figure 9.5: Block diagram showing the different blocks in the design suggested by the additional research.

9.6.2 Results from tests

As discussed earlier, due to time restraints only a few of the requirements from the specifications in section 4.2 was actually tested. Thus it is unknown if most of the requirements were met or not, and it must be assumed that they were not. The two requirements tested for was the effective frequency range and the total harmonic distortion.
Starting with the test of the effective frequency range, it showed that the amplifier had a much smaller effective range than required, approximately 97.25 % smaller. The most likely explanation for this deviation is the faulty design of the amplifier as discussed above. As the design has been deemed not functional, the no further explanation is offered to as why the results where as far from desired as they were.

Continuing to the test of the total harmonic distortion, this too showed a large deviation from the requirements, as it was 6971 % higher than the required 0.7 % maximum total harmonic distortion. As before, this deviation is most likely due to the faulty design of the amplifier.

9.7 Conclusion

This chapter has documented the process through which the audio power amplifier has been designed, simulated, and tested, and the results of these tests. Based on the simulations it was decided to do some additional research into amplifier design, though the results from this research did not get implemented due to time restraints. Thus the tests were conducted using the amplifier constructed based on the design section above. Based on these tests it can be concluded that the designed and constructed amplifier did not meet any of the requirements stated in the specifications in section 4.2.

Chapter 10

Acceptance Test

This chapter will focus on the implementation of the entire system and the final acceptance test. The system will be tested for the system specifications. This will be done according to the requirement analysis established earlier in section 4.2 & 4.3.

10.1 Execution of tests

Tests where performed to measure the degree of fulfilment for most of the requirements. But because of the time restraints towards the end of the project it was not possible fully analyse the data make supporting the test journals. It has been decided to briefly compare the requirements with the test results in a table and then comment on the most relevant data. All the raw data has been collected but there was no time to process it all. All the raw data will be available as an attachment.

10.2 Fulfilment of the requirements

The table 10.1 shows all the requirements set for the system and any relevant results from the tests.

10.3 Printed Circuit Board Implementation

Throughout the project, circuit prototypes have been constructed on breadboards, veroboards and stripboard. However, for the final design which has been used for both unit testing as well as acceptance testing Printed Circuit Boards (PCB). Each system have been designed in Altium Designer on double sided PCB and milled in a PCB milling machine.
The PCBs designed in Altium Designer are included in appendix K. While real world realisation of the PCBs can be found in L.

Table 10.1: Receiver requirements and relevant test results

<table>
<tr><th colspan="7">Requirements for receiver and test results</th></tr>
<tr><th>No.</th><th>Description</th><th>Min.</th><th>Max.</th><th>Unit</th><th>Test result</th><th>Deviation and comments</th></tr>
<tr><td>01</td><td>Output a signal within the audible frequency range</td><td>20</td><td>$20 \cdot 10^3$</td><td>Hz</td><td>The receiver could output sounds from the entire frequency range of the radio.</td><td>Fulfilled</td></tr>
<tr><td>02</td><td>Total harmonic distortion</td><td>-</td><td>≤ 1</td><td>%</td><td>The THD within the audible frequencies was approximately 2 %</td><td>not fulfilled: 1% THD off target</td></tr>
<tr><td>03</td><td>Channel separation</td><td>30</td><td>-</td><td>dB</td><td>Channel separation for channels under the channel of interest was at least 30 dB and for channels above it was 10 dB</td><td>Partially fulfilled: The upper channel separation was 20 dB off target</td></tr>
<tr><td>04</td><td>Unweighted signal-to-noise ratio</td><td>57</td><td>-</td><td>dB</td><td>The SNR of the demodulator alone was at approximately 30 dB</td><td>Not fulfilled: at least 27 dB off target</td></tr>
<tr><td>05</td><td>Output impedance</td><td></td><td>≤ 2.2</td><td>Ω</td><td>No tests were made</td><td>TBD</td></tr>
<tr><td>06</td><td>Output voltage</td><td>0.2</td><td>2</td><td>V</td><td>The output of the demodulator was measured to be about 300 mV.</td><td>Fulfilled</td></tr>
<tr><td>07</td><td>Tuning range</td><td>87.5</td><td>108</td><td>MHz</td><td>The radio was able to tune into all channels to and audible degree</td><td>Fulfilled</td></tr>
<tr><th colspan="7">Functional requirements</th></tr>
<tr><td>No.</td><td colspan="3">Description</td><td colspan="3">Result</td></tr>
<tr><td>08</td><td colspan="3">Receives a transmitted FM signal</td><td colspan="3">Fulfilled</td></tr>
<tr><td>09</td><td colspan="3">Output a sound signal</td><td colspan="3">Fulfilled</td></tr>
<tr><td>10</td><td colspan="3">The radio can be tuned manually</td><td colspan="3">Fulfilled</td></tr>
<tr><td>11</td><td colspan="3">The radio is precise enough to tune to a single radio broadcast channel</td><td colspan="3">Fulfilled</td></tr>
</table>

Part III

Discussion & Conclusion

This part will include the discussion and the conclusion for the project.

Chapter 11

Conclusion

In this chapter is the collective conclusion on the report and the project as a whole. The conducted tests and results are compared with the specifications from section 4.2, and an overall assessment are made based on this.
The purpose of this project was primarily to design and implement a radio receiver. It was decided to build a radio receiver together with a class A audio amplifier. For the design of the radio receiver, the double superheterodyne topology was selected, with priority being put on building the mixers, filters and the demodulator. For the rest of parts of the radio receiver, namely the front end antenna, LNA's and local oscillators it was chosen to use prefabricated units.

During the implementation of the designs, some unforeseen challenges with high frequencies arose. These challenges were primarily due to a lack of prior experience and knowledge of designing circuits for high frequencies. This led to a number of delays during the implementation, and the time table got pushed back, resulting in otherwise important aspects of the system not being implemented in an effective manner. This was primarily observed in the design and construction of the audio power amplifier. Due to the delays, the design of the amplifier ended being unsatisfactory and in addition did not meet any requirements. In lieu of designing a completely new amplifier, additional research into amplifiers was done and a suggestion for a better design was put forward, explaining some of the errors made in the first design and expanding on the topology to correct some of the discovered discrepancies between the expected results and the ones measured during simulations and tests. Additionally other parts such as the processing of the data from tests where not finished as well.

However, the final implementation of the receiver was completed and the results from the unit tests of each part as well as the accept test for all the radio receiver parts together, showed that most of the requirements were met. Thus the group is ultimately very satisfied with the final implementation of the radio receiver. All things considered the group is still overall very happy with the results of the project.

Chapter 12

Perspectivation

As mentioned in the conclusion the prioritisation of manpower and time management during this project was not done in an effective way. In the beginning of the design process the project was divided into three primary parts with two people assigned with the design and construction of each part. The parts were the mixers, the demodulator and the filters in combination with power amplifier design. It was expected that the filter design process would be the least challenging and time consuming part of the project and therefore could be completed within a few weeks. Thus the team designated to this would quickly be able to turn their attention to the design of the power amplifier. However, this turned out not to be the case, in fact filter design turned out to be the most challenging and problematic part of the project. Furthermore, when we in the last few days of the project period finally assigned people to the amplifier design it was again falsely believed that this phase would prove to be trivial. Once again this turned out not to be the case.

Due to this naivety and lack of insight, time was managed poorly which ultimately resulted in the power amplifier not being able to meet any of the specified requirements or even our own expectations.

In retrospect, one of the biggest mistakes performed was unwillingness of the group to discontinue the pursuit of the filter design despite it proving tedious and being out of scope for the semesters curriculum. Furthermore, the failure to thoroughly study amplifier design before creating the project time schedule, and instead partially basing it on our own undocumented expectations.

Retrospectively more research into pitfalls should ideally be conducted before starting the planning phase.

Bibliography

[1] C. R. Nave, "Hyperphysics," *http://hyperphysics.phy-astr.gsu.edu/hbase/Sound/tralon.html*, 2016, read: September 14th 2017.

[2] ——, "Hyperphysics," *http://hyperphysics.phy-astr.gsu.edu/hbase/Sound/earsens.html*, 2016, read: September 14th 2017.

[3] S. Haykin and M. Moher, *Introduction to Analog & Digital Communications*, 2nd ed. John Wiley & Sons, inc., 2007.

[4] Elsevier , *Nobel Lectures, Physics 1901-1921*, ser. Nobel Lectures. Elsevier Publishing Company, 1967.

[5] R. Alfred, "Dec. 12, 1896: Marconi demos radio — dec. 12, 1901: Marconi transmits across atlantic," *https://www.wired.com*, 2011, september 24Th 2017.

[6] S. Haykin and M. Moher, *Introduction to Analog & Digital Communications*, 2nd ed. John Wiley & Sons, inc., 2007.

[7] P. Simonson, J. Peck, R. T. Craig, and J. P. J. Jr., *The Handbook of Communication History*, 1st ed. Routledge, 2013.

[8] S. J. Orfanidis, *Electromagnetic Waves and Antennas.* Rutgers University, 2016.

[9] "Fm query broadcast station search," Dec 2015. [Online]. Available: https://www.fcc.gov

[10] D. Hardy, G. Malleus, and J.-N. Mereur, *Networks: internet - telephony - multimedia*, M. Byrne and M. Horne, Eds. Springer, 2014.

[11] "Fm-sendenettet."

[12] "Auktioner og udbud for frekvenser," Sep 2016.

[13] "Fm stereo/rds theory," *http://rfmw.em.keysight.com/wireless/helpfiles/n7611b/Content/Main/FM_Broadcasting.htm*, last visited: 2017-19-12.

[14] daenotes, "Pre-emphasis and de-emphasis," http://www.daenotes.com/electronics/communication-system/pre-emphasis-and-de-emphasis, 2017-20-12.

[15] International Electrotechnical Commission (IEC), *Household high-fidelity audio equipment and systems; Methods of measuring and specifying the performance. Part 2 - FM radio tuners*, 1st ed. IEC, 1998.

[16] International Telecommunication Union (ITU), *Characteristics of FM sound broadcasting reference receivers for planning purposes*, 1st ed. ITU, 1990.

[17] G. Milner, *Perfecting Sound Forever: The Story of Recorded Music*, 1st ed. Granta Books, 2009.

[18] A. S. Sedra and K. C. Smith, *Microelectronic Circuits*, 7th ed. Oxford University Press, 2016, international Edition.

[19] Electronics Tutorials, "Introduction to the amplifier," http://www.electronics-tutorials.ws/amplifier/amp_1.html, last viewed: 2017.10.23.

[20] J. Karki, "Calculating noise figure and third-order intercept in adcs."

[21] I. Blagouchine and E. Moreau, *IEEE Transactions on Communications.* IEEE Communications Society, Sep 2011, vol. 59.

[22] Electronics Tutorials, "Class b amplifier," http://www.electronics-tutorials.ws/amplifier/amp_6.html, last viewed: 2017.10.23.

[23] ——, "Class ab amplifier," http://www.electronics-tutorials.ws/amplifier/class-ab-amplifier.html, last viewed: 2017.10.23.

[24] International Electrotechnical Commission (IEC), *High fidelity audio equipment and systems; Minimum performance requirements. Part 1: General*, 1st ed. IEC, 1977.

[25] ——, *High fidelity audio equipment and systems; Minimum performance requirements. Part 6: Amplifiers*, 1st ed. IEC, 1979.

[26] ——, *High fidelity audio equipment and systems; Minimum performance requirements. Part 8: Combination equipment*, 1st ed. IEC, 1986.

[27] ——, *Household high-fidelity audio equipment and systems; Methods of measuring and specifying the performance. Part 1 - General*, 1st ed. IEC, 1996.

[28] ——, *Household high-fidelity audio equipment and systems; Methods of measuring and specifying the performance. Part 3 - Amplifiers*, 1st ed. IEC, 1996.

[29] ——, *Sound system equipment. Part 3 - Amplifiers*, 2nd ed. IEC, 2013.

[30] ——, *Multimedia systems - Guide to the recommended characteristics of analogue interfaces to achieve interoperability*, 2nd ed. IEC, 2013.

[31] RadioPlay, "Radio frekvenser," https://radioplay.dk/frekvenser/, 2017.

[32] Radio-Electronics, "Gilbert cell mixer / multiplier," http://www.radio-electronics.com/info/rf-technology-design/mixers/gilbert-cell-mixer-multiplier.php.

[33] ——, "Bjt transistor mixer / multiplier," http://www.radio-electronics.com/info/rf-technology-design/mixers/bjt-transistor-mixer-multiplier.php.

[34]

[35] "Bcm847bv; bcm847bs; bcm847ds, datasheet," 2017.

[36] J. H. Mikkelsen, "Acd5.slides," 2017.

[37] C. Bowick, *RF Circuit Design*, 2nd ed. Elsevier, 2008.

[38] H. K. Krauss, C. W. Bostian, and F. H. Raab, *Solid State Radio Engineering*, 1st ed. John Wiley & Sons, 1980, p. 126.

[39] F. Langford-SMith, *Radio Designer's Handbook*, 4th ed. Reed Educational and Professional Publishing Ltd., 1967, p. 126.

[40] TDK, "B78108e, b78148e datasheet," http://www.mouser.com/ds/2/400/b78108_148e-961548.pdf, 2015.

[41] R. Blake, *Basic Electronic Communication*, 1st ed. West Publishing Company, 1993, p. 56.

[42] [Murata], "Ceramic filters application manual," 2011.

[43] ——, "Ceramic filter (cerafil) for fm receivers," 2000.

[44] A. S. Sedra and K. C. Smith, *Microelectronic Circuits Appendix H*, 7th ed. Oxford University Press, 2015, international Edition.

[45] R. Sobot, *Wireless Communication Electronics, Introduction to RF Circuits and Design Techniques*, 1st ed. Springer, 2012.

[46] S. W. Seeley and J. Avins, *RCA REVIEW a technical journal, Volume VII*, 1st ed. RADIO CORPORATION OF AMERICA RCA LABORATORIES DIVISION, 1947.

[47] S. Haykin, *Communication Systems*, 4th ed. John Wiley & Sons, 2001.

[48] Micrometals, "Toroid datasheet," *RF Applications*, no. H, Dec 2005, last viewed: 2017.12.18. [Online]. Available: http://c1170156.r56.cf3.rackcdn.com/UK_MIS_RF_Series_Issue_H_DS.pdf

[49] A. AG, "Biconical emc broadband antennas - bicolog series," http://www.aaronia.com/Datasheets/Antennas/EMC-Measurement_Antenna_BicoLOG-Series.pdf.

[50] Mini-Circuits, "Coaxial low noise amplifier, zfl-500hln+ & zfl-500hln," https://ww2.minicircuits.com/pdfs/ZFL-500HLN.pdf.

[51] ——, "Coaxial low noise amplifier, slp-300+," https://ww2.minicircuits.com/pdfs/SLP-300+.pdf.

[52] ——, "Coaxial low noise amplifier, slp-150+," https://ww2.minicircuits.com/pdfs/SLP-150+.pdf.

[53] Fairchild, “Mje3055t npn silicon transistor datasheet,” http://www.mouser.com/ds/2/149/MJE3055T-889641.pdf.

[54] J. Mikkelsen, *Analog Kredsløbsdesign slides kursusgang 5*, 1st ed., 2017, last viewed: 2017.12.19.

[55] E. Coates, “Class a amplifier design,” http://www.learnabout-electronics.org, 2017.

Appendix A

1^{st} Mixer Gain Test

A.1 Test purpose

The purpose of this test, is to identify the gain of the first mixer when down converting an FM signal to the first IF, 10.7 MHz mixer.

A.2 Test frame

Theoretical background

The theory for the mixer was previously discussed in section 5.

Test setup, equipment and test procedure

This test was conducted on December the 16th 2017, at 05:50 UTC+01:00 in room B1-101 on Fredrik Bajers Vej 7, 9000 Aalborg, Denmark. The temperature in the room during the test was 23 degrees Celcius.

Table A.1 lists all components and equipment used in the gain test of the first mixer. The test setup with the equipment included is furthermore seen in figure A.1.

Table A.1: List of equipment used in gain test of the 10.7 MHz mixer.

Equipment list - Gain Test - #1 Mixer		
Name / Description	*Type*	*AAU-ID*
Marconi Instruments	10kHz - 1Ghz signal generator 2022D	33337
Marconi Instruments	10kHz-1000MHz signal generator 2022	-
Hameg Power Supply	HM7042	33885
Keysight Digital Oscilloscope	DSOX1102G	AAU-2180-08

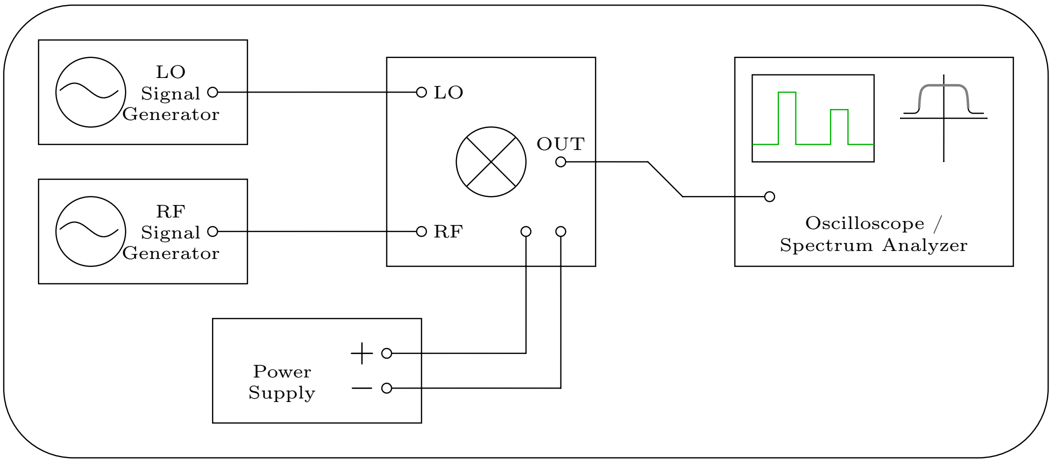


Figure A.1: Illustration of test setup for gain test of mixer #1.

The two Marconi Signal generators output were each set to -37dBm. Two measurements were made:

- One at the very maxima of the FM band, with the RF frequency being 107 MHz and the LO being 96.3 MHz
- One at the minimum frequency of the FM band, with the RF frequency being 87 MHz and the LO being 85.6 MHz

To capture the input and output amplitude spectrum, the DSOX1102G internal FFT function was used. To calculate the gain, the difference between the output amplitude and the RF input amplitude was calculated

A.3 Test results and data processing

The following results where measured:

Table A.2: Test of first mixer gain

Tests data summary					
LO Frequency	*RF Frequency*	*LF Input Amplitude*	*RF Input Amplitude*	*Output amplitude*	*Gain*
96.3 MHz	107 MHz	-53.5 dB	-51 dB	-76dB	-25 dB
76.3 MHz	87 MHz	-50 dB	-60 dB	-70dB	-10dB

A.4 Sources of error and other uncertainties

The primary source of error was the oscilloscope. It is a relatively low end oscilloscope which isn't designed to analyse RF. Furthermore, it would have been preferable to use a real spectrum analyser, as opposed to the simple integrated software FFT.

Appendix B

2^{nd} Mixer Gain Test

B.1 Test purpose

The purpose of this test, is to identify the gain of the second mixer when down converting the first IF from 10.7 MHz to 455 kHz (second IF) .

B.2 Test frame

Theoretical background

The theory for the mixer was previously discussed in section 5.

Test setup, equipment and test procedure

This test was conducted on December the 16th 2017, at 05:40 UTC+01:00 in room B1-101 on Fredrik Bajers Vej 7, 9000 Aalborg, Denmark. The temperature in the room during the test was 23 degrees Celcius.

Table B.1 lists of all components and equipment used in the gain test of the 2^{nd} mixer. The test setup with the equipment included is also seen in figure B.1.

Table B.1: List of equipment used in gain test of the second mixer.

Equipment list - Gain Test - #2 Mixer		
Name / Description	*Type*	*AAU-ID*
Marconi Instruments	10kHz - 1Ghz signal generator 2022D	33337
Marconi Instruments	10kHz-1000MHz signal generator 2022	-
Hameg Power Supply	HM7042	33885
Keysight Digital Oscilloscope	DSOX1102G	AAU-2180-08

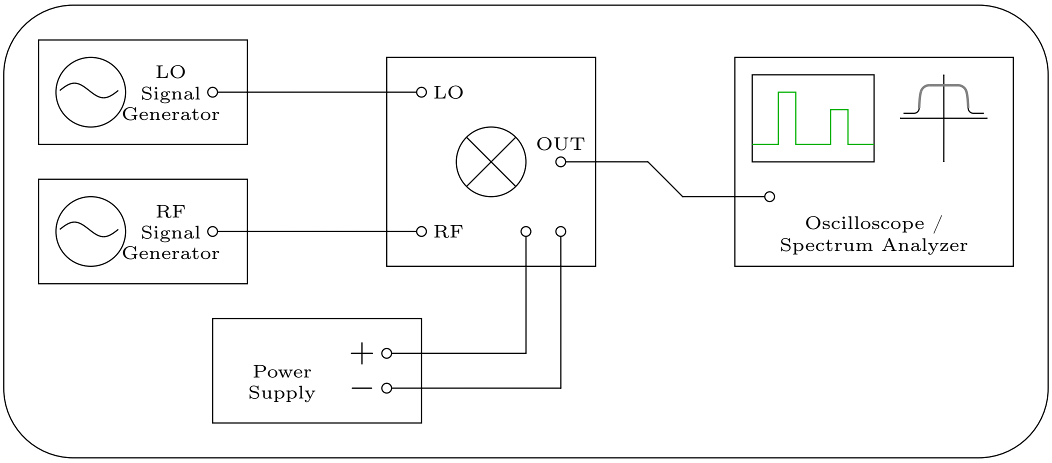


Figure B.1: Illustration of test setup for gain test of mixer #2.

The two Marconi Signal generators output were each set to -37dBm. One measurements was made:

- One where the RF is at 10.7 MHz and the LO is at 10.245 MHz

To capture the input and output amplitude spectrum, the DSOX1102G internal FFT function was used. To calculate the gain, the difference between the output amplitude and the RF input amplitude was calculated

B.3 Test results and data processing

The following results where measured:

Table B.2: Test of second mixer gain

Tests data summary					
LO Frequency	*RF Frequency*	*LF Input Amplitude*	*RF Input Amplitude*	*Output amplitude*	*Gain*
10.245 MHz	10.7 MHz	-50 dB	-50 dB	-25dB	25 dB

B.4 Sources of error and other uncertainties

The primary source of error was the oscilloscope. It is a relatively low end oscilloscope which isn't designed to analyse RF. Furthermore, it would have been preferable to use a real spectrum analyser, as opposed to the simple integrated software FFT.

Appendix C

2^{nd} Mixer Image Rejection Ratio test

C.1 Test purpose

The purpose of this test, is to find the Image Rejection Ratio of mixer #2.

C.2 Test frame

Theoretical background

The theory for the mixer were previously discussed in section 5.

Test setup, equipment and test procedure

This test was conducted on December the 16th 2017, at 05:30 UTC+01:00 in room B1-101 on Fredrik Bajers Vej 7, 9000 Aalborg, Denmark. The temperature in the room during the test was 23 degrees Celcius.

Table C.1 lists all components and equipment used in the image rejection ratio test of the second mixer. The test setup with the equipment included is also seen in figure C.1.

Table C.1: List of equipment used in image rejection ratio test of the 455 kHz mixer.

Equipment list - Gain Test - 10.7 MHz mixer		
Name / Description	*Type*	*AAU-ID*
Agilent Waveform Generator	33220A	43686
Agilent Waveform Generator	33220A	43686
Analog discovery osciloscope	-	-
Hameg Power Supply	HM7042	33885

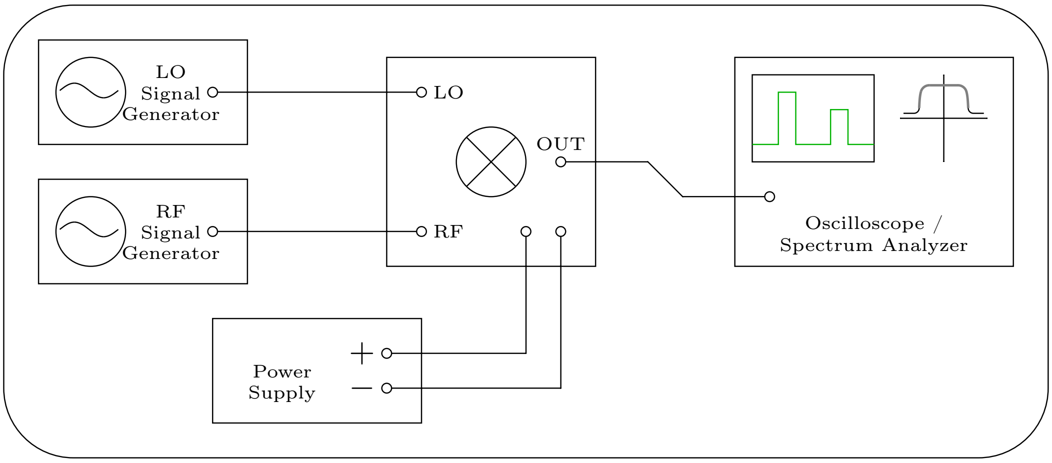


Figure C.1: Illustration of test setup for test of the 455 kHz mixer.

To measure the IMRR, each of the waveform generators were configured to output a 20 mV peak peak sine wave. The first the RF input was set to 10.7 MHz and the LO was to set to 10.245 MHz and the resulting output amplitude was measured at .455 MHz. Next the output amplitude at .455 MHz was again measured when using the corresponding image frequency (9.79 MHz) as an RF input. To find the image suppression ratio, the difference was taken between each of the output measurements.

C.3 Test results and data processing

The following results where measured:

Table C.2: Test of second mixer IMRR

Tests data summary		
Image Amplitude	*Desired amplitude*	*Image Suppression ratio*
-34 dB	-14 dB	-20dB

C.4 Sources of error and other uncertainties

The only significant source of error was the oscilloscope. It is a hobbyist oscilloscope, and as such it can not possibly adhere to professional lab equipment tolerances.

Appendix D

FM Demodulator THD and AM Rejection Test

D.1 Test purpose

The purpose of this test is to test the AM rejection of the constructed Ratio Detector.

D.2 Test frame

Theoretical background

The theory for the demodulator was previously discussed in section 7.

Test setup, equipment and test procedure

This test was conducted on December the 16th 2017, at 05:10 UTC+01:00 in room B1-101 on Fredrik Bajers Vej 7, 9000 Aalborg, Denmark. The temperature in the room during the test was 23 degrees Celcius.

The setup was as seen on figure D.1, and then a FFT analysis was conducted using the software *"WaveForms"* through the Analog Discovery 2. The wave generator was used to generate and send a signal that was both AM and FM. The carrier frequency was set to 455 kHz. The FM modulation had a bandwidth of 150 kHz. The AM signal was then varied to see the effect on the output into the demodulator and the output signal was measured with the integrated oscilloscope. The software WaveForms then performed an FFT which was saved as the output. The following AM percentages was measured: 0%, 10%, 20%, 50% and 80%. The measurement with no AM was used to measure THD of the demodulator.

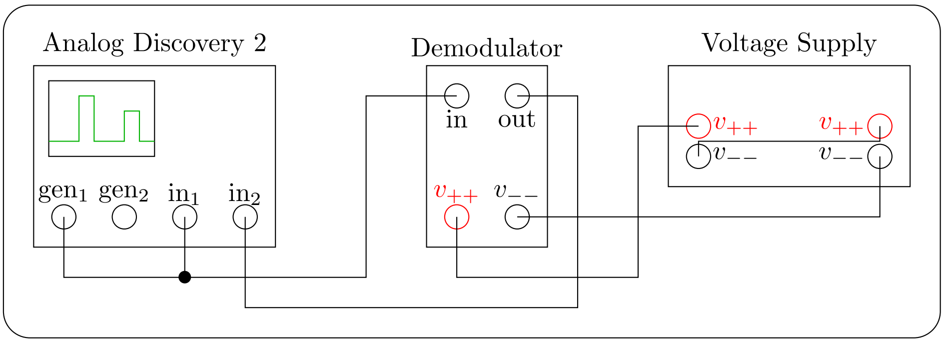


Figure D.1: Illustration of the setup.

Table D.1: Test equipment for the AM rejection test.

Equipment		
Name / Description	*Type*	*AAU-ID*
Analog Discovery 2	-	2179-06
Hameg Power Supply	HM7042	33905
FM Demodulator	Ratio Detector	-
Oscilloscope software	WaveForms Version: 2015(3.6.8 64-bit)	-

D.3 Test results and data processing

The following results where measured:

Table D.2: Test at different percent of AM, with a carrier frequency of 455 kHz.

Tests data summary				
FM Frequency	*AM Frequency*	*AM %*	*THD*	*SNR*
1 kHz	5,3 kHz	0	-24.00 dBc	31.89 dBc
1 kHz	5,3 kHz	10	-24.04 dBc	28.90 dBc
1 kHz	5,3 kHz	20	-24.04 dBc	25.09 dBc
1 kHz	5,3 kHz	50	-24.26 dBc	16.17 dBc
1 kHz	5,3 kHz	20	-33.74 dBc	3.73 dBc

D.4 Sources of error and other uncertainties

The only significant source of error was the oscilloscope. It is a hobbyist oscilloscope and signal generator. The oscilloscope and the signal generator are not as precise compared to the rest of the equipment used, which was all lab equipment.

Appendix E

FM Demodulator frequency response

E.1 Test purpose

The purpose of this test is to measure the ratio detectors frequency response.

E.2 Test frame

Theoretical background

The theory for the demodulator was previously discussed in section 7.

Test setup, equipment and test procedure

This test was conducted on December the 16th 2017, at 05:30 UTC+01:00 in room B1-101 on Fredrik Bajers Vej 7, 9000 Aalborg, Denmark. The temperature in the room during the test was 23 degrees Celcius.

The setup was as seen on figure E.1. The measurements were made by repeatedly making a frequency sweep from 200 kHz to 700 kHz every 10 ms. Then the scope was used to record the output voltage and thus draw the output V/Hz response for the demodulator.

Table E.1: Test equipment for the Frequency response test.

Equipment		
Name / Description	*Type*	*AAU-ID*
Analog Discovery 2	-	2179-06
Hameg Power Supply	HM7042	33905
FM Demodulator	Ratio Detector	-
Oscilloscope software	WaveForms Version: 2015(3.6.8 64-bit)	-

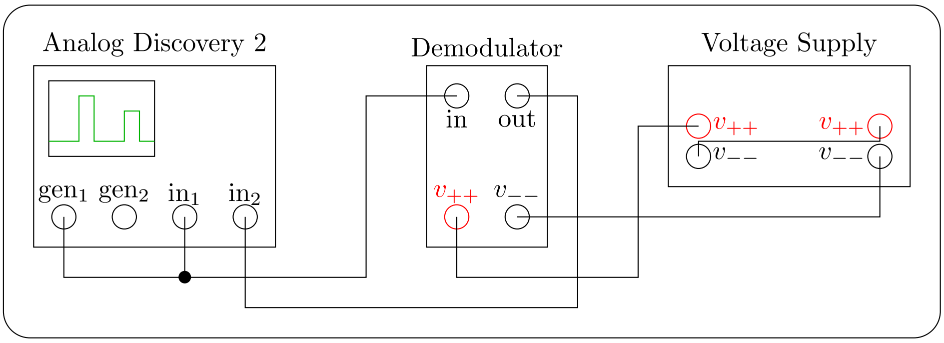


Figure E.1: Illustration of the setup.

E.3 Test results and data processing

The following results where measured:

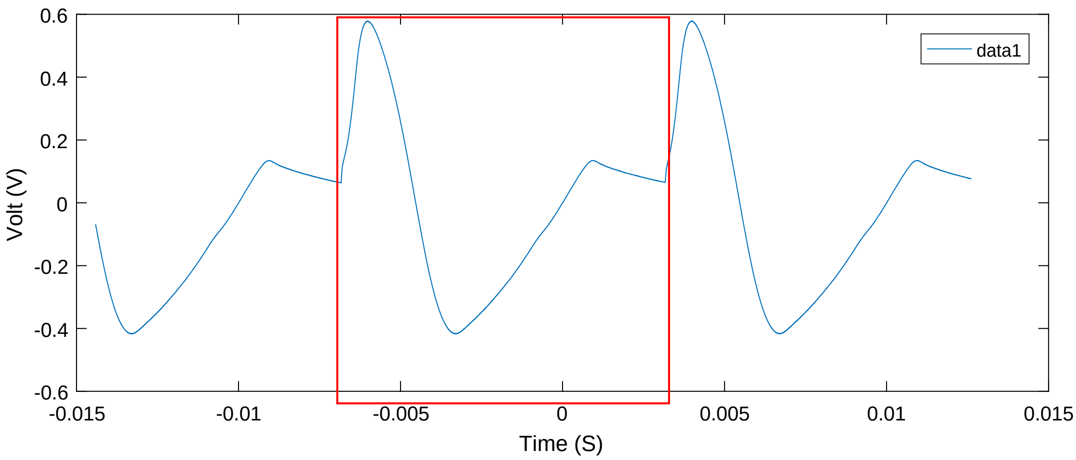


Figure E.2: Results form the frequency response test., The data in the red box is the frequency response from 200 to 700kHz.

E.4 Sources of error and other uncertainties

The only significant source of error was the oscilloscope. It is a hobbyist oscilloscope and signal generator. The oscilloscope and the signal generator are not as precise compared to the rest of the equipment used, which was all lab equipment.

Appendix F

FM Demodulator and limiter Gain Test

F.1 Test purpose

The purpose of this test is to measure the gain/attenuation of the demodulator with and without the limiter.

F.2 Test frame

Theoretical background

The theory for the demodulator was previously discussed in section 7. And the theory for the limiter was discussed in section 7.9.

Test setup, equipment and test procedure

This test was conducted on December the 16th 2017, at 05:10 UTC+01:00 in room B1-101 on Fredrik Bajers Vej 7, 9000 Aalborg, Denmark. The temperature in the room during the test was 23 degrees Celcius.

First the gain of the limiter was measured. The setup was as seen on figure F.1. The WaveForms program was used to determine the gain by comparing the input and the output of the limiter.

Second the attenuation of the demodulator was measured. The setup was as seen on figure F.2. The input signal and the output signal where noted to calculate the attenuation.

Finally the total gain of the demodulator and limiter was measured. The setup was as seen on figure F.3. The input signal and the output signal where noted to calculate the total gain.

Table F.1: Test equipment for the gain test.

Equipment		
Name / Description	*Type*	*AAU-ID*
Analog Discovery 2	-	2179-06
Hameg Power Supply	HM7042	33905
FM Demodulator	Ratio Detector	-
Oscilloscope software	WaveForms Version: 2015(3.6.8 64-bit)	-

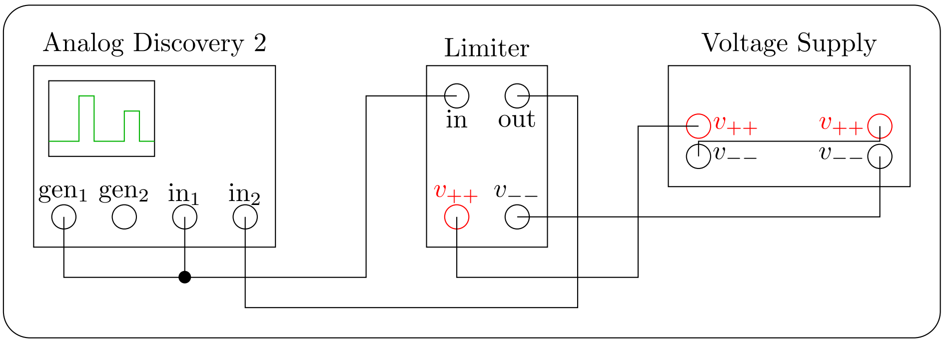


Figure F.1: Illustration of the setup.

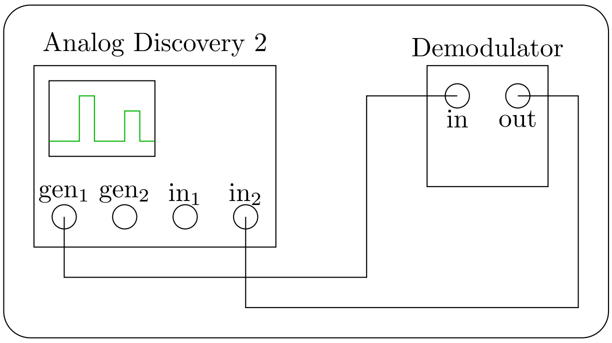


Figure F.2: Illustration of the setup.

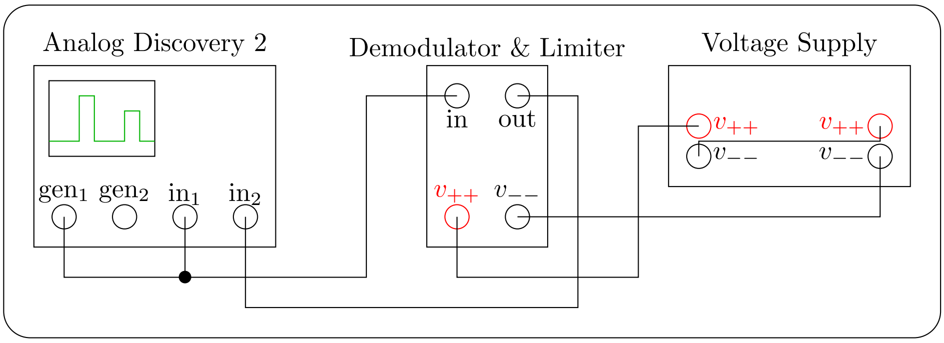


Figure F.3: Illustration of the setup.

F.3 Test results

The following results from the three tests where noted:

As expected, since the op-amp have no negative feedback, the gain was purely that of the op-amp which was measured at 60 dB with 1 mV peak to peak (p2p) input and 1 V p2p output at the IF, as was specified in the data sheet. However the op-amp was limited by the supply voltage of +/- 12 V, thus any signal larger than 12 mV p2p was clipped.

Secondly the gain of the demodulator was measured by comparing the input signal with the output of the demodulator. The input was a 24V p2p signal and the output was 316 mV p2p. Thus the attenuation was:

$$20 \cdot log(\frac{316m\text{V}}{24\text{V}}) = -37dB \tag{F.1}$$

Finally the limiter was tested with the demodulator. The total gain of the filter and the demodulator was up to $23dB$ at inputs lower than 12 mV. This was as expected since:

$$A_{\text{total}} = A_{\text{limiter}} + A_{\text{demodulator}} = 60dB - 37dB = 23dB \tag{F.2}$$

Above 12 mV the output was fixed at 316 mV because of the limiter.

F.4 Sources of error and other uncertainties

The only significant source of error was the oscilloscope. It is a hobbyist oscilloscope and signal generator. The oscilloscope and the signal generator are not as precise compared to the rest of the equipment used, which was all lab equipment.

F.5 Additional comments

This test was done with very little documentation, but because the time restrains of the project, it was not prioritised to do the test again.

Appendix G

Test Journal: Intermediate Filter 1

This test journal documents the tests of different ceramic filters intended to be used as intermediate filter 1. The tests were performed on December 10th 2017.

G.1 Test purpose

The purpose of these tests are to determine the frequency response of the ceramic filters, with the intent to compare this to the specifications put forth for the filter and determine if the specifications are met. If these specifications are met, the appropriate filter is to be used in the complete system.

G.2 Test frame

During this test, three different types of ceramic filters were tested. The filters were supplied by lecturer Hans Ebert, without data sheets or other means of telling them apart than their different colours and few letters that did not seem to correspond to any known manufacturers. In addition, Hans Ebert informed that for all the filters, the middle pin is for ground, and the outer pins are input and output.

In table G.1 the different filters have been given names based on the letters on them, and these will be used henceforth.

Table G.1: Table with names and description of the tested filters.

Filter name:	*Filter Description:*
J4	Brown ceramic with the letters 107 J4
CFS	Red ceramic with the letters CFS
RWBS	Red ceramic with black stripe

Theoretical background

Test setup, equipment and test procedure

In this test a Vector Network Analyzer was used to determine the frequency response of the tested filters. This was done by using the machine to measure the S21 parameter of the filters, corresponding to the forward voltage gain. A sketch of the test configuration can be seen on figure G.1

The test procedure was as follows:
First the machine was calibrated to frequency spectrum from 1 MHz to 500 MHz in which the test was to be performed. The calibration was done by a lab-technician. Then the filters were connected to the machine and tested one at a time. The frequency response was measured for three different ranges. This was done to be able to compare the characteristics of each filter both close to the pass-band and further up the frequency spectrum. The three ranges are shown in table G.2

Table G.2: Table of frequency ranges used in test of IF1-filter.

Frequency ranges for test IF1 filters		
Range 1	Range 2	Range 3
10 - 11.4 MHz	1 - 20 MHz	1 - 100 MHz

After each test, the measured data points were saved to a floppy disk, this data are included in the csv-files.

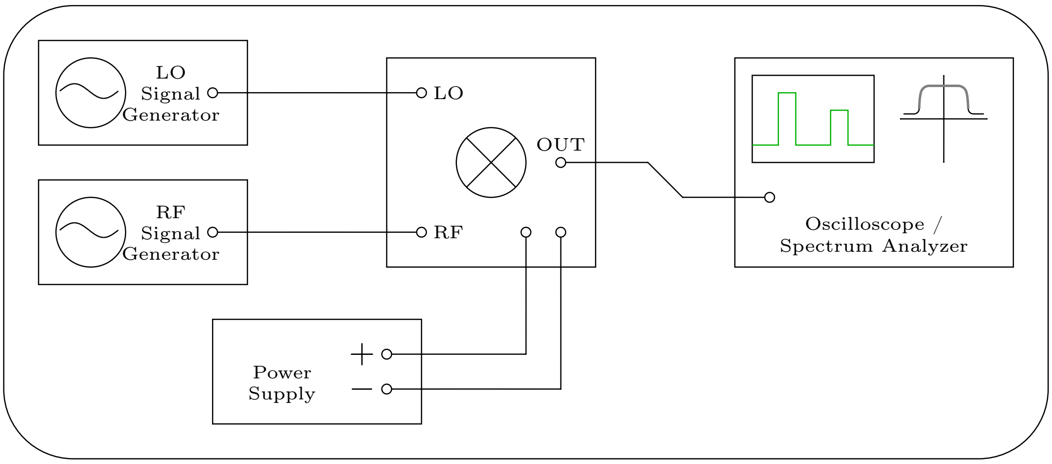


Figure G.1: Illustration of test setup for test of the IF1 filters.

Test equipment used:

Table G.3: Table of used equipment

Test Equipment:	*AAU-Number:*	*Name of Device:*
Vector Network Analyzer	56979	Rhode&Schwarz Vector Network Analyzer (9kHz - 4 GHz)
RG-58 cable	-	-
Coax to Bananna splitters (2x)	-	-
Test probes (3x)	-	-

G.3 Test results and data processing

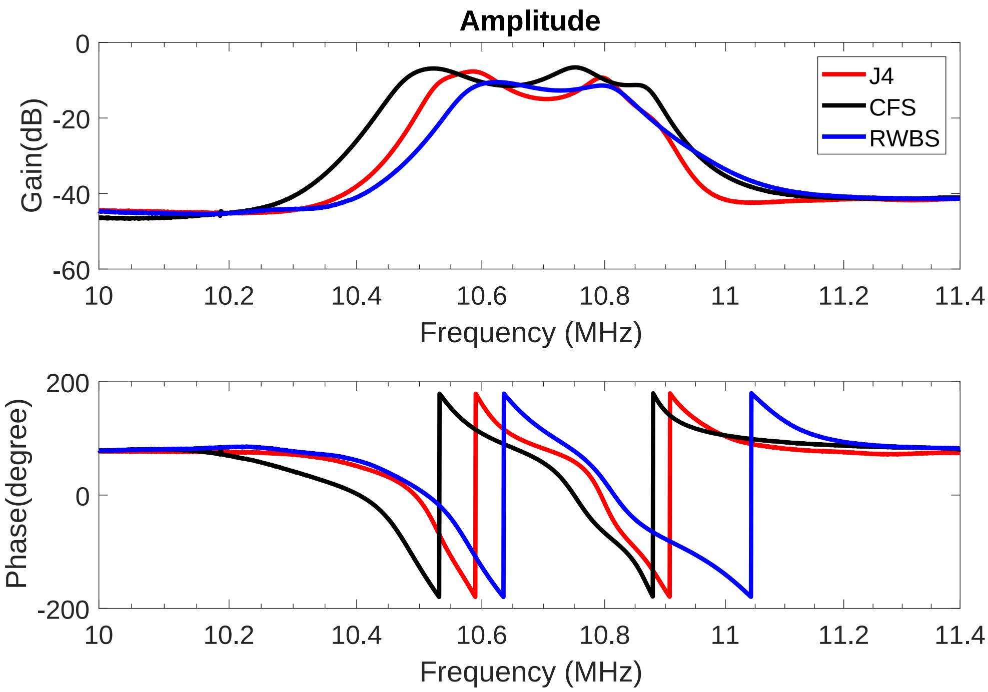


Figure G.2: Test results of the three filters, the test sweep is between 10 and 11.4 MHz

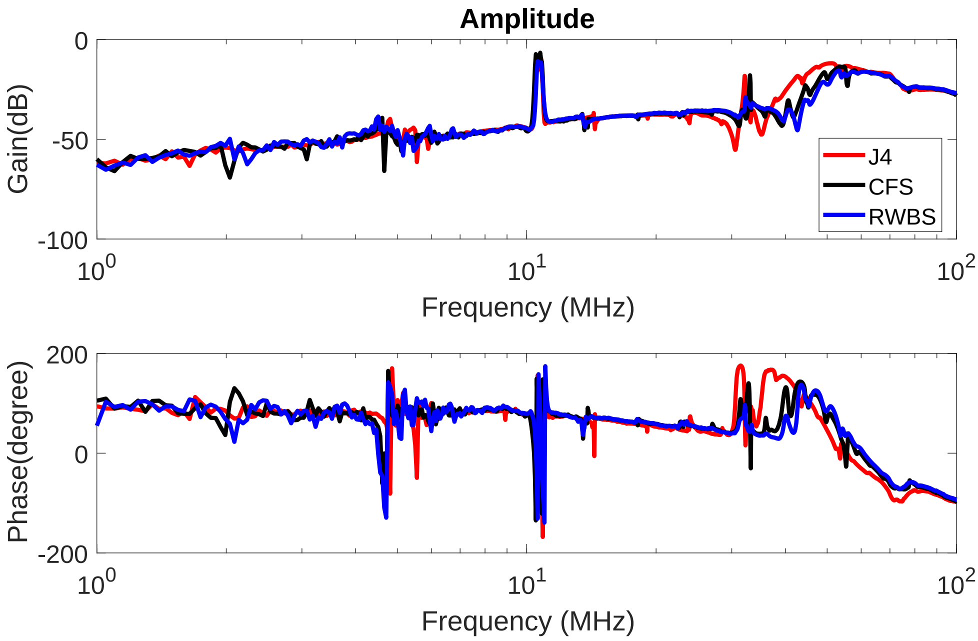


Figure G.3: Test results of the three filters, the test sweep is between 1 and 100 MHz, the filter's band-pass response is seen in the middle of the figure

Appendix H

Test Journal: Intermediate Filter 2

This test journal documents the tests of the filter described in section 6.5.2 intended to be used as intermediate frequency 2 (IF2) filter. The tests were performed on December 16th 2017.

H.1 Test purpose

The purpose of this test is to determine the frequency response of the IF2 filter, with the intent to compare this to the specifications put forth for the filter and determine if the specifications are met. If these specifications are met, the appropriate filter is to be used in the complete system.

H.2 Test frame

The filter was tested using the combined oscilloscope and signal generator Analog Discovery 2 and the WaveForms software.

Theoretical background

Test setup, equipment and test procedure

Test equipment used:

Table H.1: Test equipment for the AM rejection test.

Equipment		
Name / Description	*Type*	*AAU-ID*
Analog Discovery 2	-	2179-06
Hameg Power Supply	HM7042	33905
IF 2 Filter	Filter	-
Oscilloscope software	WaveForms Version: 2015(3.6.8 64-bit)	-

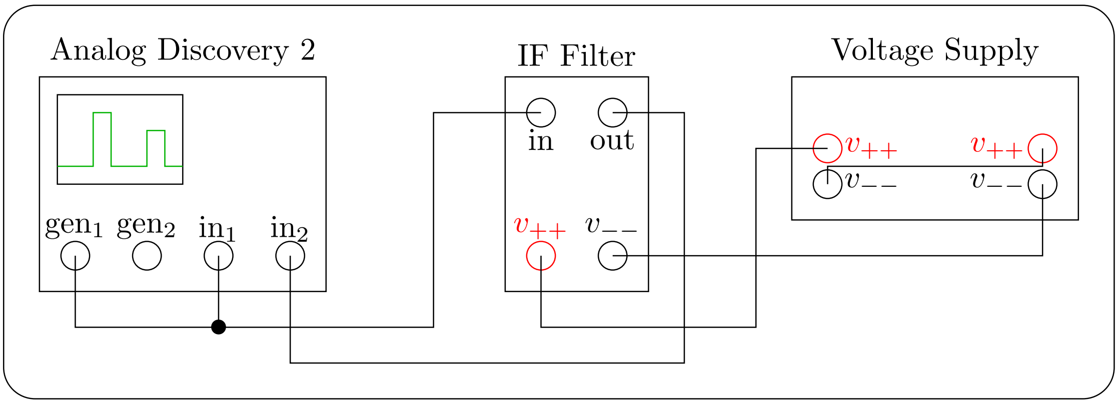


Figure H.1: Illustration of test setup for test of the IF2 filter.

H.3 Test results and data processing

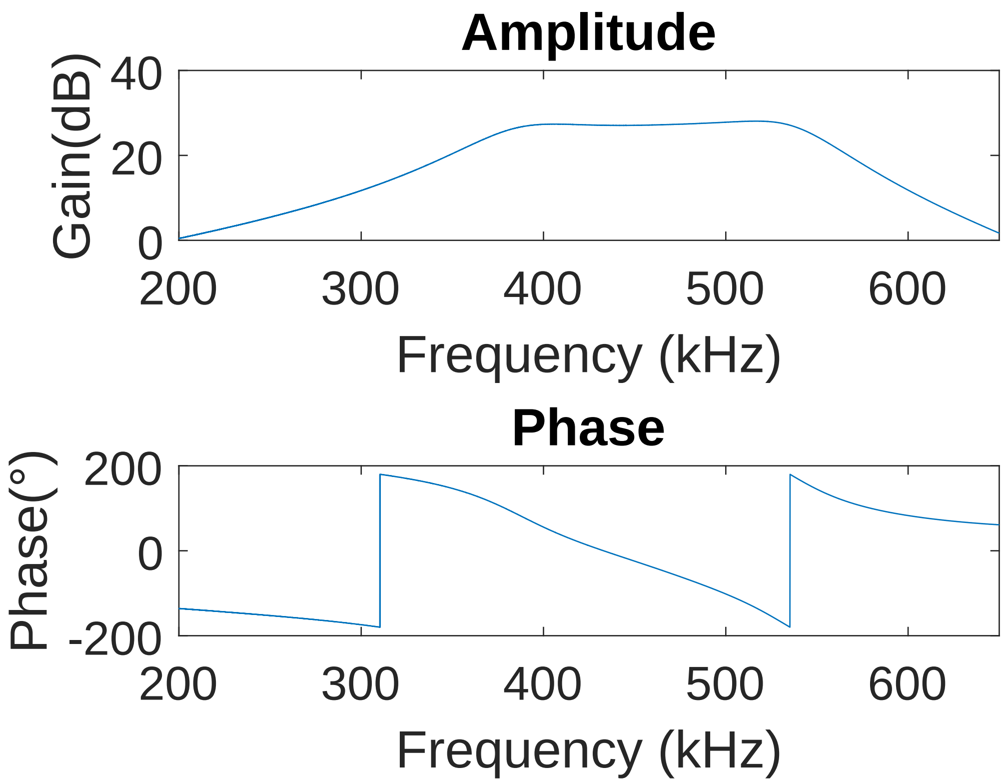


Figure H.2: IF2

H.4 Sources of error and other uncertainties

The only significant source of error was the oscilloscope. It is a hobbyist oscilloscope and signal generator. The oscilloscope and the signal generator are not as precise compared to the rest of the equipment used, which was all lab equipment.

Test journal: Frequency response of audio power amplifier

This test journal documents the test of the audio power amplifier with the

H.5 Test purpose

The purpose of this test is to determine the effective frequency range of the constructed audio power amplifier.

H.6 Test frame

Theoretical background

The effective frequency range was described further in section 3.2. According to the standards the effective frequency response is obtained from a graph showing the frequency response of the amplifier.

Test setup, equipment and test procedure

This test was conducted on December the 20th 2017, at 01:05 UTC+01:00 in room B1-101 on Fredrik Bajers Vej 7, 9000 Aalborg, Denmark. The temperature in the room during the test was 22 degrees Celcius.

The test was conducted by measuring the ratio between the input and output voltage while stepping the frequency of the input signal from 20 Hz to 20 kHz. This was done using the computer in the laboratory at Frederik Bajers Vej 7. The results were then plotted on a graph as a function of frequency.

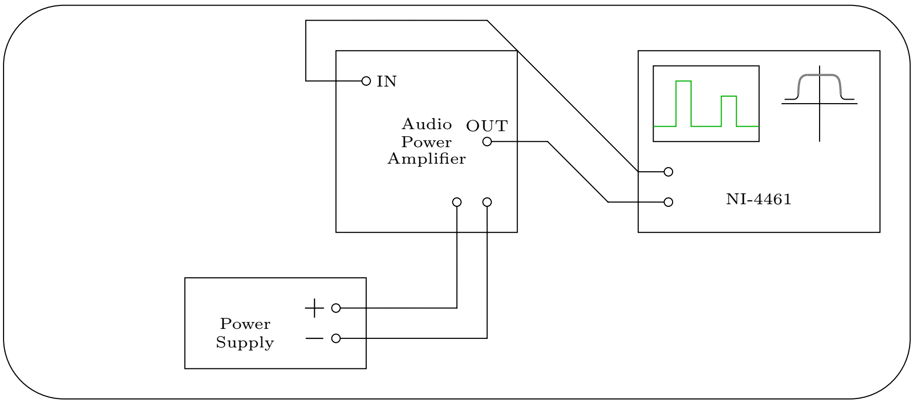


Figure H.3: Illustration of the audio power amplifier test setup.

Equipment list		
Name / Description	*Type*	*AAU-ID*
Laboratory computer with audio analyzer	NI-4461	64640
Hameg Power Supply	HM7042	33905

Table H.2: Table with the list of equipment used during the test.

H.7 Test results and data processing

The results from the test can be seen in figure H.4.

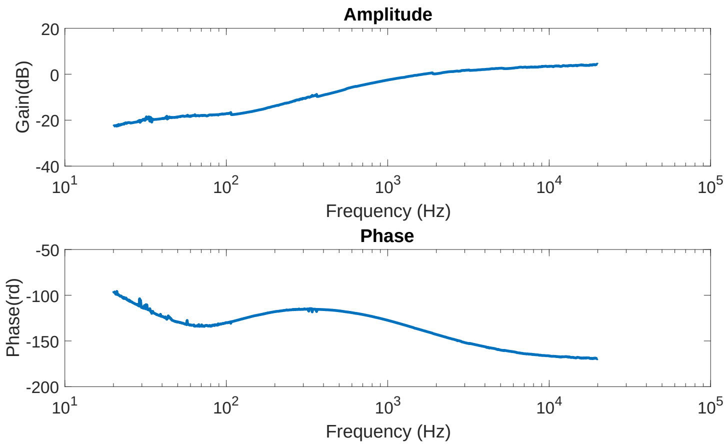


Figure H.4: Graph showing the voltage gain (upper) and the phase response (lower) as a function of frequency.

By inspecting the measurements from the test, the effective frequency range of the amplifier is determined to be approximately from 780 Hz to 1330 Hz.

H.8 Sources of error and other uncertainties

This test was conducted using old equipment that may have suffered some decay during the years it has been used.

Test journal: Total harmonic distortion of audio power amplifier

This test journal documents the test of the audio power amplifier with the

H.9 Test purpose

The purpose of this test is to determine the maximum total harmonic distortion of the constructed audio power amplifier.

H.10 Test frame

Theoretical background

Harmonic distortion was discussed in section 3.1.

Test setup, equipment and test procedure

This test was conducted on December the 20th 2017, at 01:25 UTC+01:00 in room B1-101 on Fredrik Bajers Vej 7, 9000 Aalborg, Denmark. The temperature in the room during the test was 22 degrees Celcius.

The test was conducted using the audio analyser integrated in the computer in the laboratory at Frederik Bajers Vej 7. The measurements were then plotted on a graph.

Equipment list		
Name / Description	*Type*	*AAU-ID*
Laboratory computer with audio analyzer	NI-4461	64640
Hameg Power Supply	HM7042	33905

Table H.3: Table with the list of equipment used during the test.

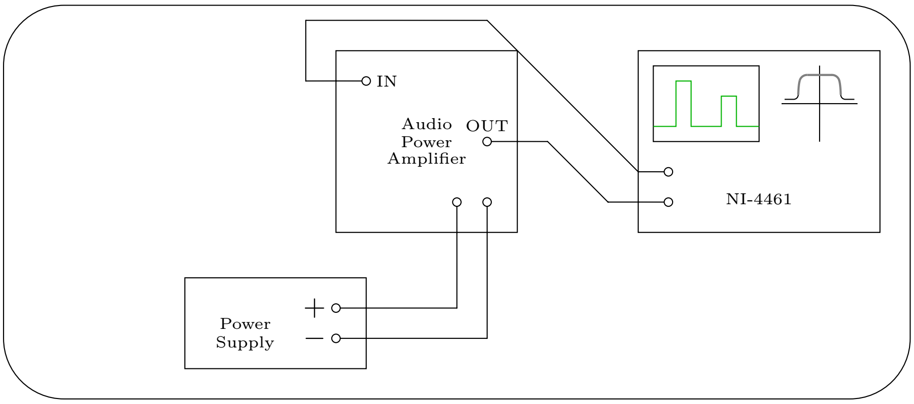


Figure H.5: Illustration of the audio power amplifier test setup.

H.11 Test results and data processing

The results from the test can be seen in figures H.6.

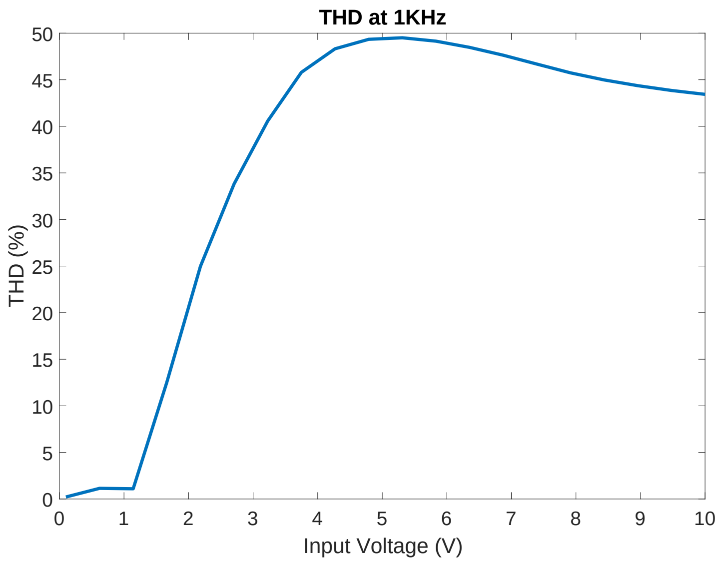


Figure H.6: Graph showing the total harmonic distortion as a function of amplitude of the input signal.

From figure H.6 the maximum total harmonic distortion for the amplifier can be observed as approximately 50 % which occurs at approximately 5 V. The precise value has been determined from the raw measurements to be 49.5% at 5.3 V.

H.12 Sources of error and other uncertainties

This test was conducted using old equipment that may have suffered some decay during the years it has been used.

Appendix I

Test journal: Acceptance Test

Acceptance test among others can be found https://github.com/bifr05t/P3_radio_receiver

Appendix J

Schematics

Table J.1: Component reference list for figure J.1

Reference:	Value:	Reference:	Value:	Reference:	Value:
V_1	12 V	R_{21}	499Ω	C_9	100 nF
V_2	-12 V	R_{22}	499 Ω	C_{10}	100 nF
R_1	150 Ω	R_{23}	499 Ω	C_{11}	100 nF
R_2	124 kΩ	R_{24}	499 Ω	C_{12}	100 nF
R_3	110 kΩ	R_{25}	35 Ω	C_{13}	100 nF
R_4	600 Ω	R_{26}	499 Ω	C_{14}	100 nF
R_5	124 kΩ	R_{27}	499 Ω	C_{15}	10 nF
R_6	110 kΩ	R_{28}	499 Ω	C_{17}	47 µF
R_7	600 Ω	R_{29}	499 Ω	C_{18}	1 µF
R_8	124 kΩ	R_{30}	35 Ω	C_{19}	100 nF
R_9	110 kΩ	R_{31}	499 Ω	C_{20}	47 µF
R_{10}	49.9 Ω	R_{32}	499 Ω	C_{21}	1 µF
R_{11}	150 Ω	R_{33}	499 Ω	C_{22}	100 nF
R_{12}	124 kΩ	R_{34}	499 Ω	Q_1	BCM847DS
R_{13}	110 kΩ	C_1	6.8 pF	Q_2	BCM847DS
R_{14}	600 Ω	C_2	100 nF	Q_3	BCM847DS
R_{15}	124 kΩ	C_3	100 nF	Q_4	BCM847DS
R_{16}	110 kΩ	C_4	100 nF	Q_5	BCM847DS
R_{17}	600 Ω	C_5	100 nF	Q_6	BCM847DS
R_{18}	124 kΩ	C_6	100 nF	U_1	AD820
R_{19}	110 kΩ	C_7	100 nF	U_2	AD820
R_{20}	49.9 Ω	C_8	6.8 pF	U_3	AD820

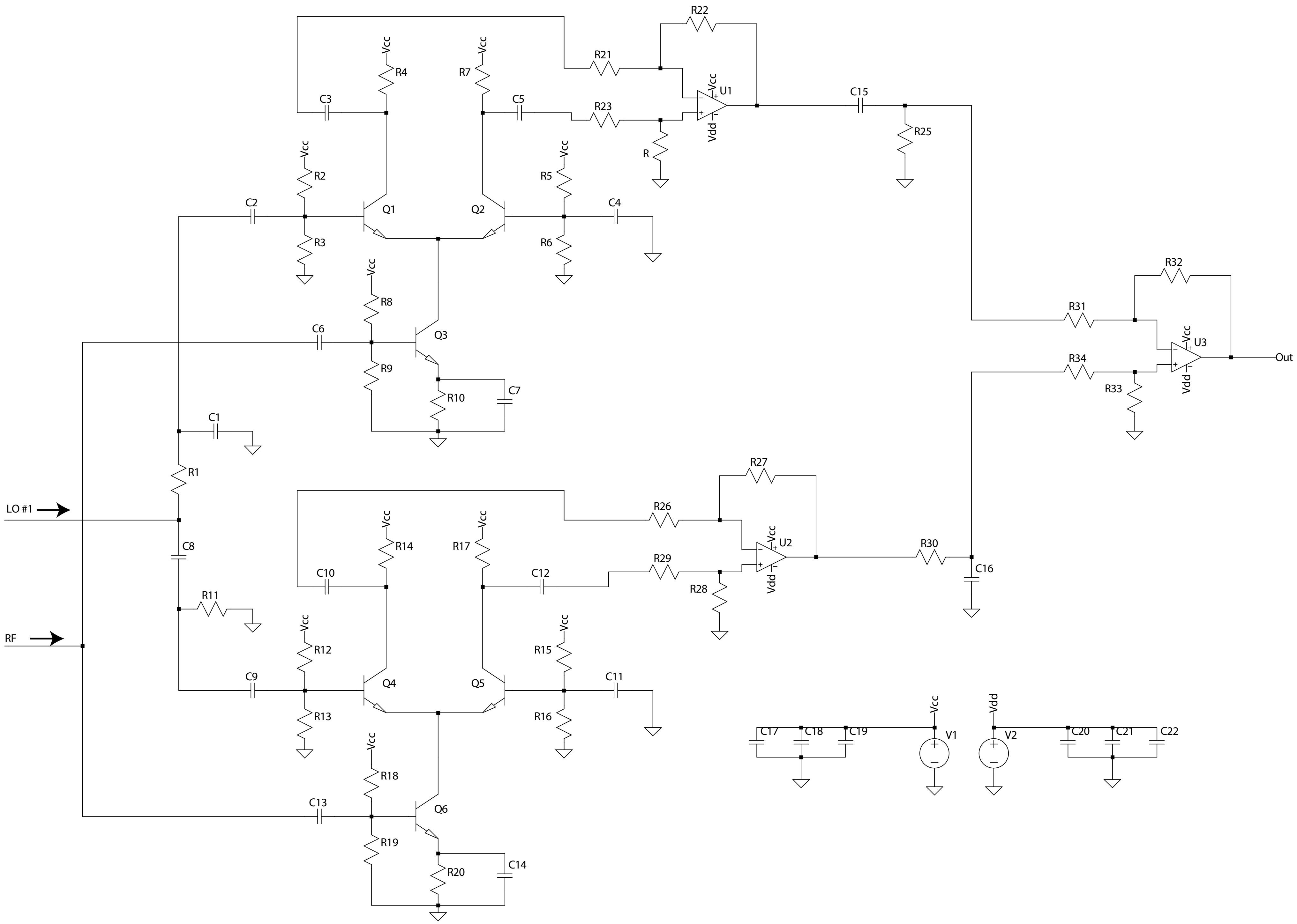


Figure J.1: IF1 Mixer Schematics

Table J.2: Component reference list for figure J.2

Reference:	Value:	Reference:	Value:	Reference:	Value:
V_1	12 V	R_{20}	49.9 Ω	C_{20}	47 µF
V_2	-12 V	R_{21}	34.9 Ω	C_{21}	1 µF
R_1	150 Ω	R_{22}	34.9 Ω	C_{22}	100 nF
R_2	124 kΩ	C_1	62 pF	Q_1	BCM847DS
R_3	110 kΩ	C_2	100 nF	Q_2	BCM847DS
R_4	600 Ω	C_3	100 nF	Q_3	BCM847DS
R_5	124 kΩ	C_4	100 nF	Q_4	BCM847DS
R_6	110 kΩ	C_5	100 nF	Q_5	BCM847DS
R_7	600 Ω	C_6	100 nF	Q_6	BCM847DS
R_8	124 kΩ	C_7	100 nF	U_1	AD620
R_9	110 kΩ	C_8	62 pF	U_2	AD620
R_{10}	49.9 Ω	C_9	100 nF	U_3	AD620
R_{11}	150 Ω	C_{10}	100 nF		
R_{12}	124 kΩ	C_{11}	100 nF		
R_{13}	110 kΩ	C_{12}	100 nF		
R_{14}	600 Ω	C_{13}	100 nF		
R_{15}	124 kΩ	C_{14}	100 nF		
R_{16}	110 kΩ	C_{15}	10 nF		
R_{17}	600 Ω	C_{17}	47 µF		
R_{18}	124 kΩ	C_{18}	1 µF		
R_{19}	110 kΩ	C_{19}	100 nF		

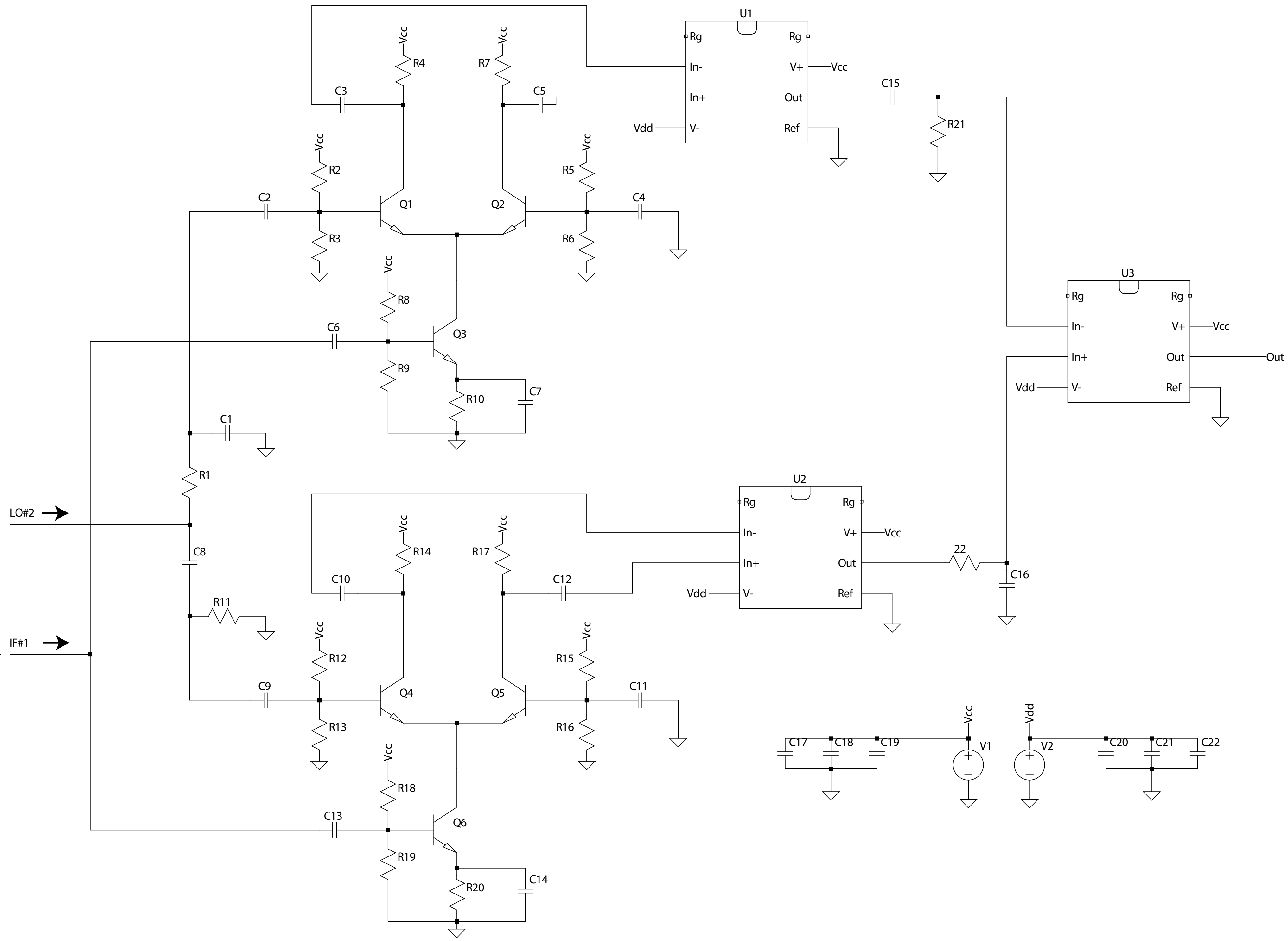


Figure J.2: IF2 Mixer Schematics

Table J.3: Component reference list for figure J.3

Reference:	Value:
V_1	12 V
V_2	-12 V
R_1	0-10 kΩ
R_2	5 kΩ
R_2V	0-10 kΩ
R_3	6.8 kΩ
R_4	6.8 kΩ
R_5	124 kΩ
R_6	0-1 kΩ
R_7	39 kΩ
C_1	470 pF
C_1V	40-200 pF
C_2	1 nF
C_2V	40-200 pF
C_3	330 pF
C_4	330 pF
C_5	8 µF
C_6	100 nF
C_7	330 pF
C_8	2 nF
C_9	100 nF
C_10	100 nF
C_11	1 µF
C_12	100 nF
C_13	1 µF
L_1	200 µH
L_2	27 µH
L_3	27 µH
L_4	6.4 µH
U_2	AD844
T_1	T80-3

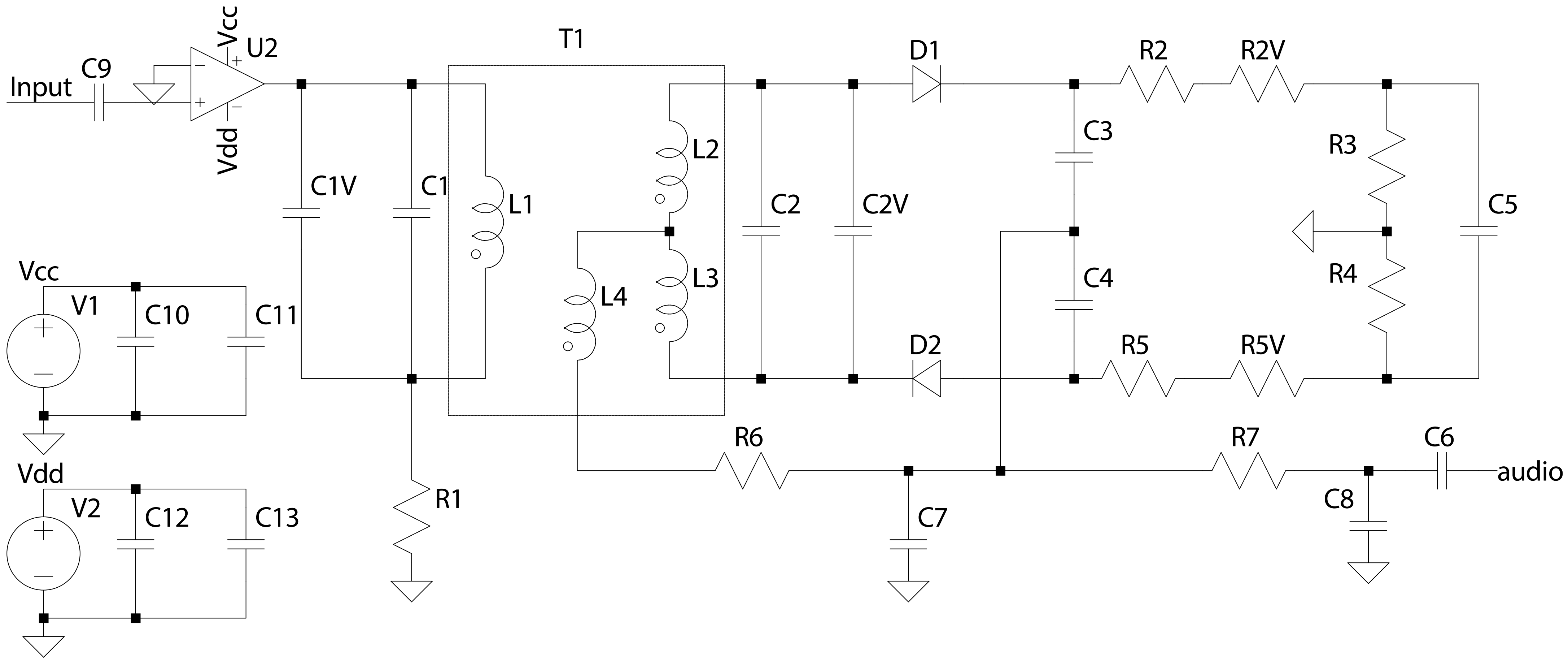


Figure J.3: Ratio Detector

Table J.4: Component reference list for figure J.3

Reference:	Value:
V_1	12 V
V_2	-12 V
R_1	0-10 kΩ
R_2	0-500 Ω
R_3	0-10 kΩ
C_1	440 pF
C_1V	40-200 pF
C_2	0.8 nF
C_2V	40-200 pF
C_3	550 pF
C_3V	40-200 pF
C_4	100 pF
C_5	100 nF
C_6	1 µF
C_7	100 pF
C_8	1 µF
U_2	AD844
L_1	260 µH
L_2	260 µH
L_3	260 µH
L_4	260 µH
T_1	custom
T_2	custom

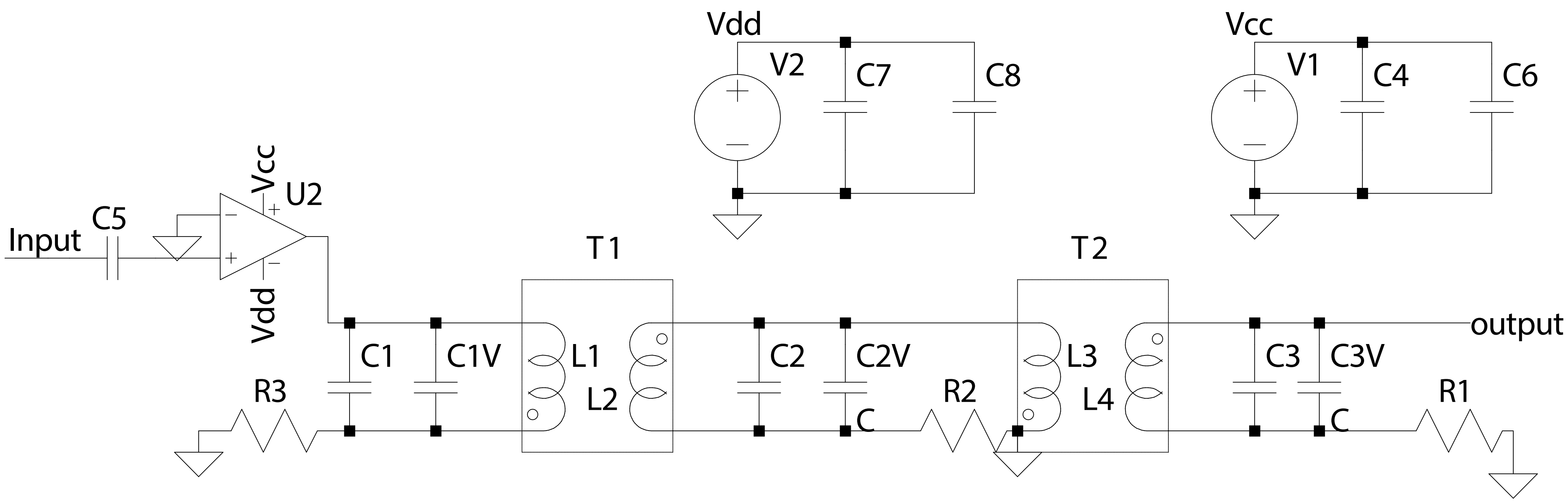


Figure J.4: IF filter 2

Appendix K

PCB Layout

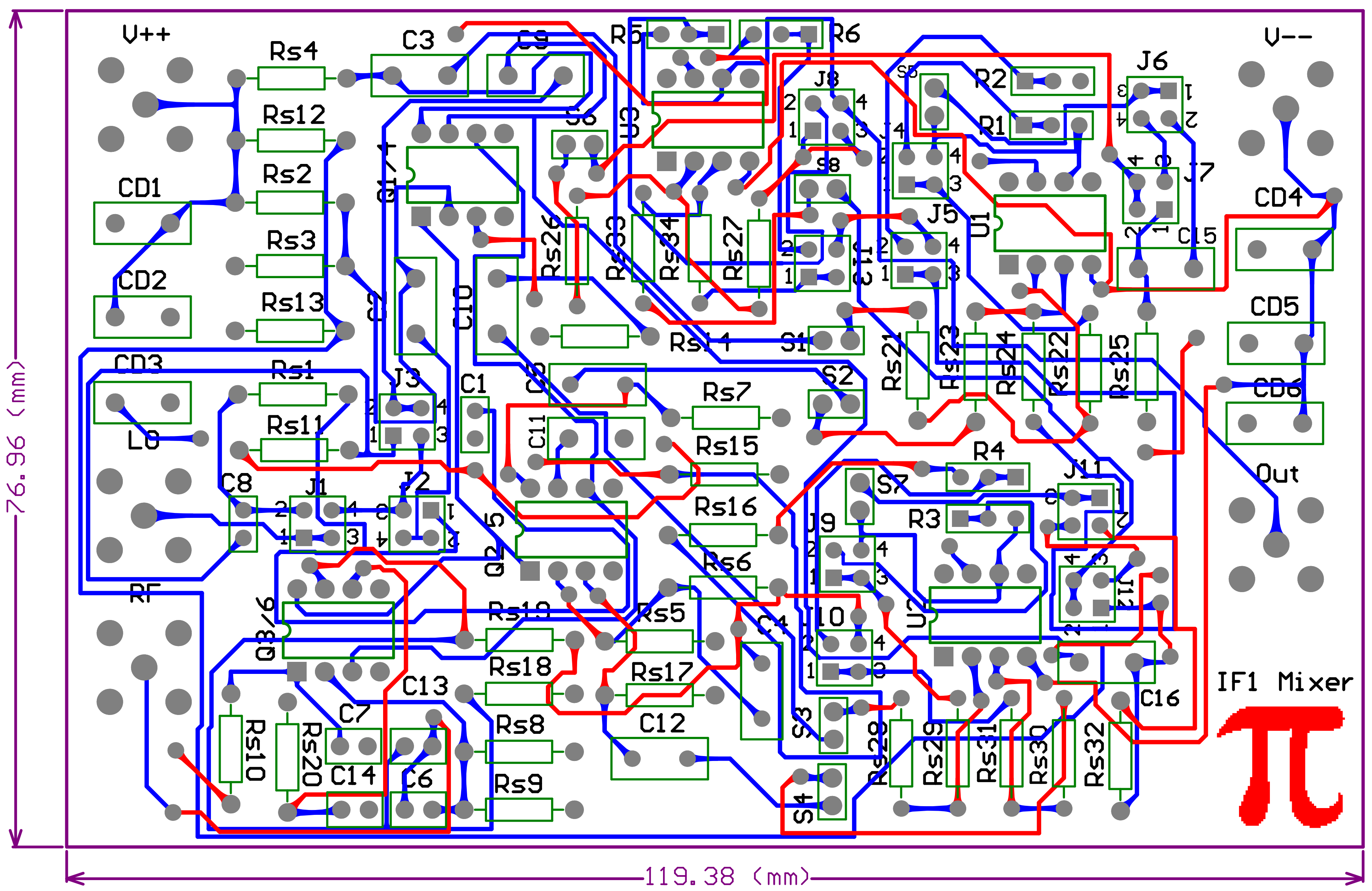


Figure K.1: Two layer PCB of the IF1 Mixer

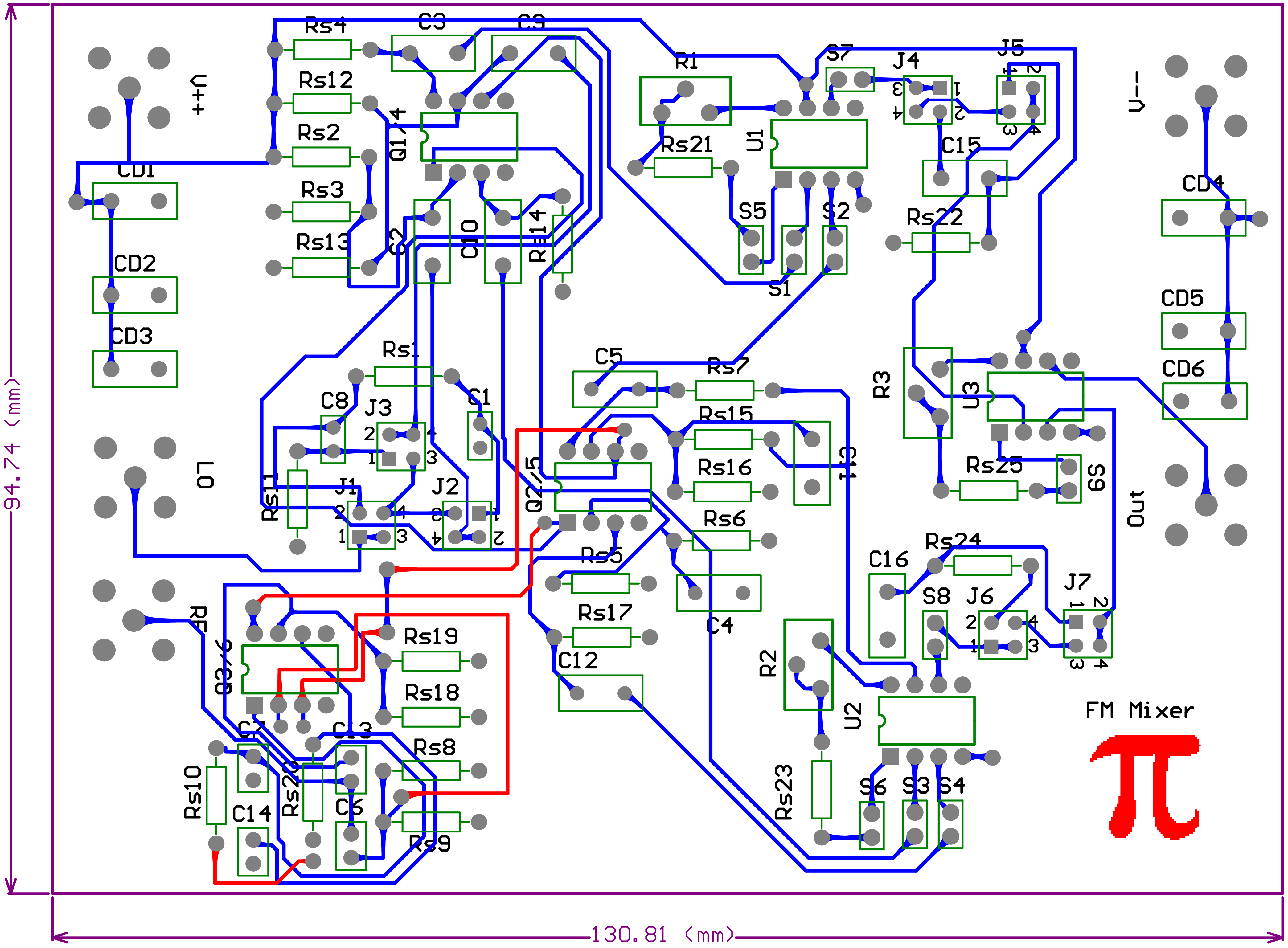


Figure K.2: Two layer PCB of the IF2 Mixer

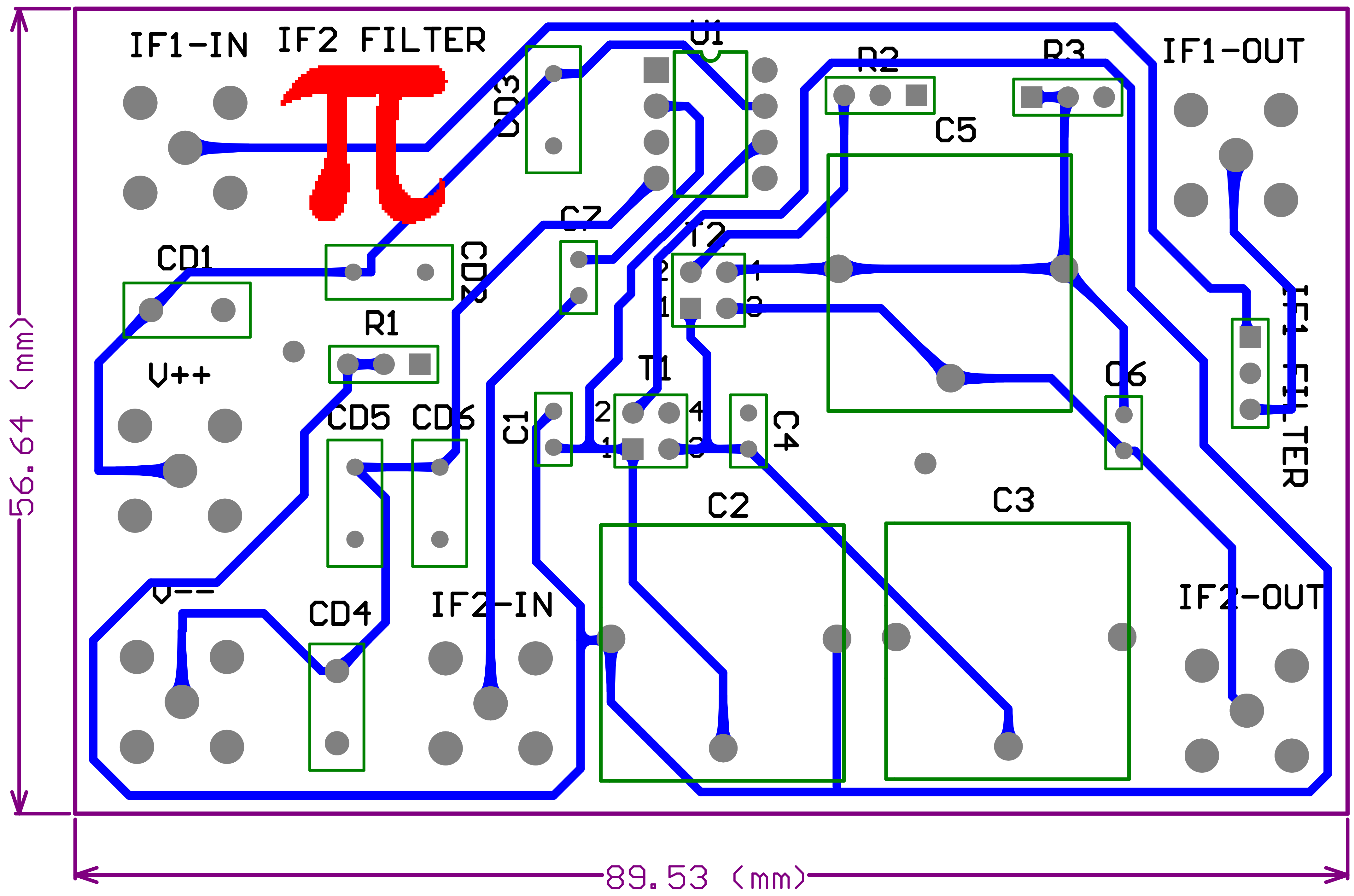


Figure K.3: Two layer PCB of the IF1 and IF2 filter

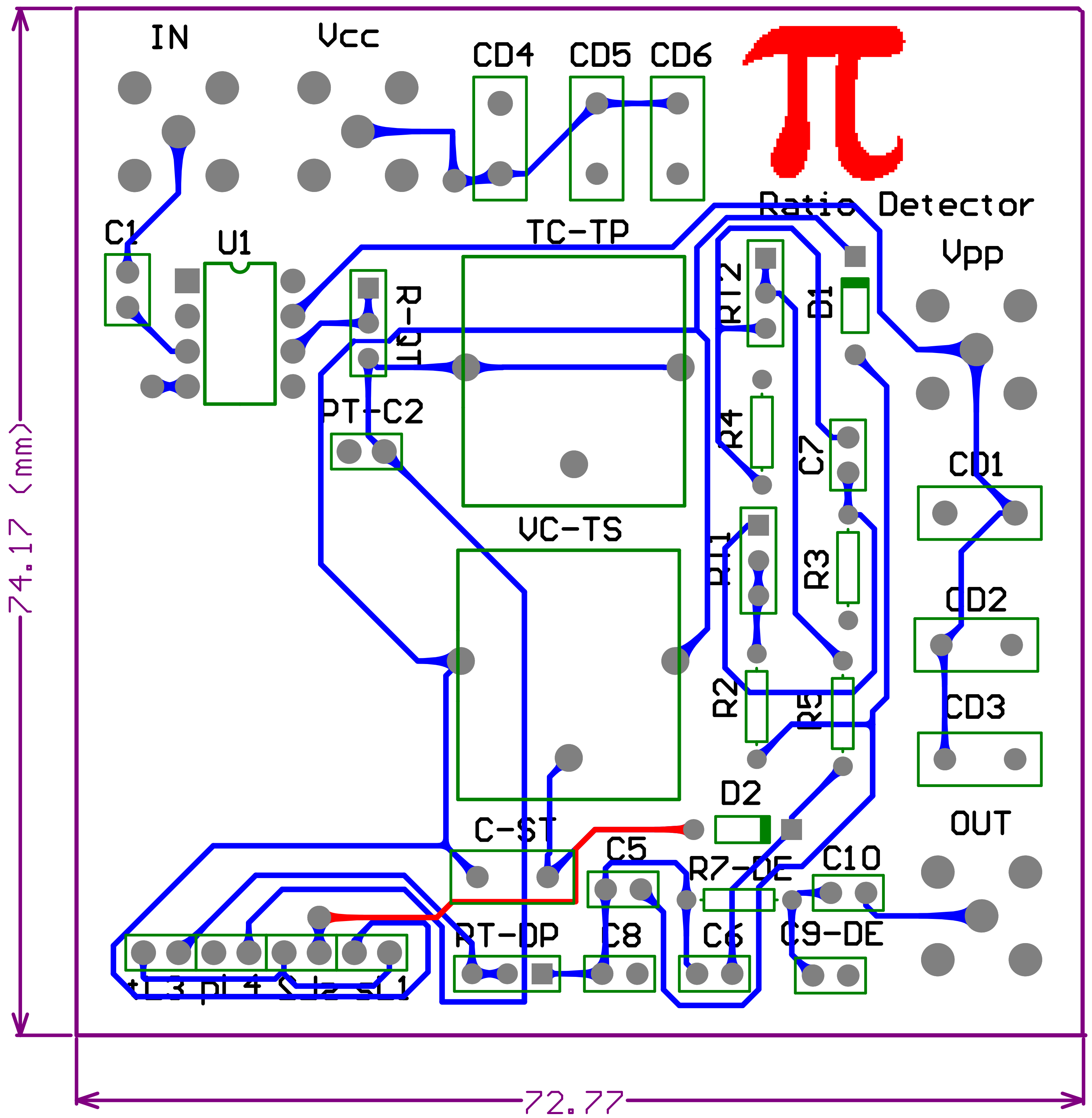


Figure K.4: Two layer PCB of the Ratio Detector

Appendix L

Printed Circuit Boards

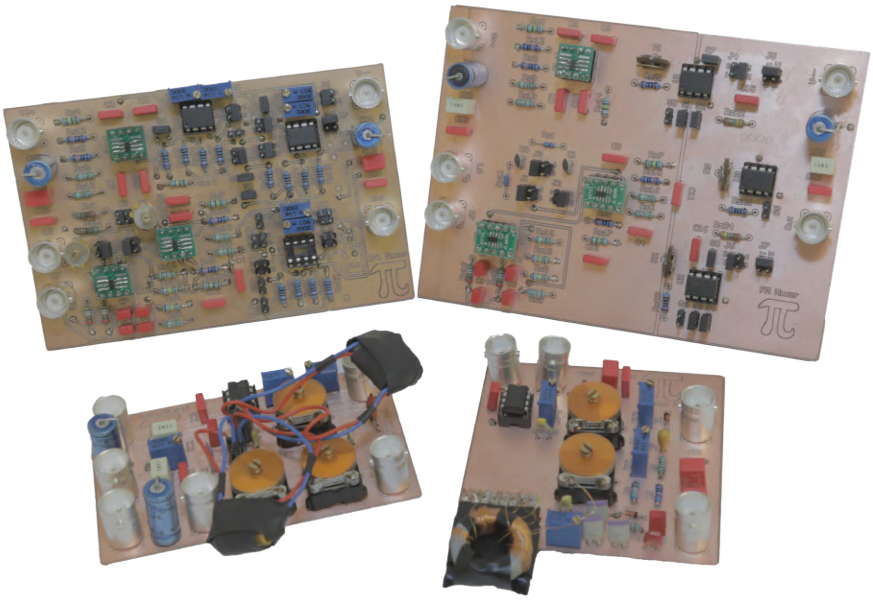


Figure L.1: Two layer PCB of IF1 and IF2 mixer, IF1 and IF2 filters and ratio detector.

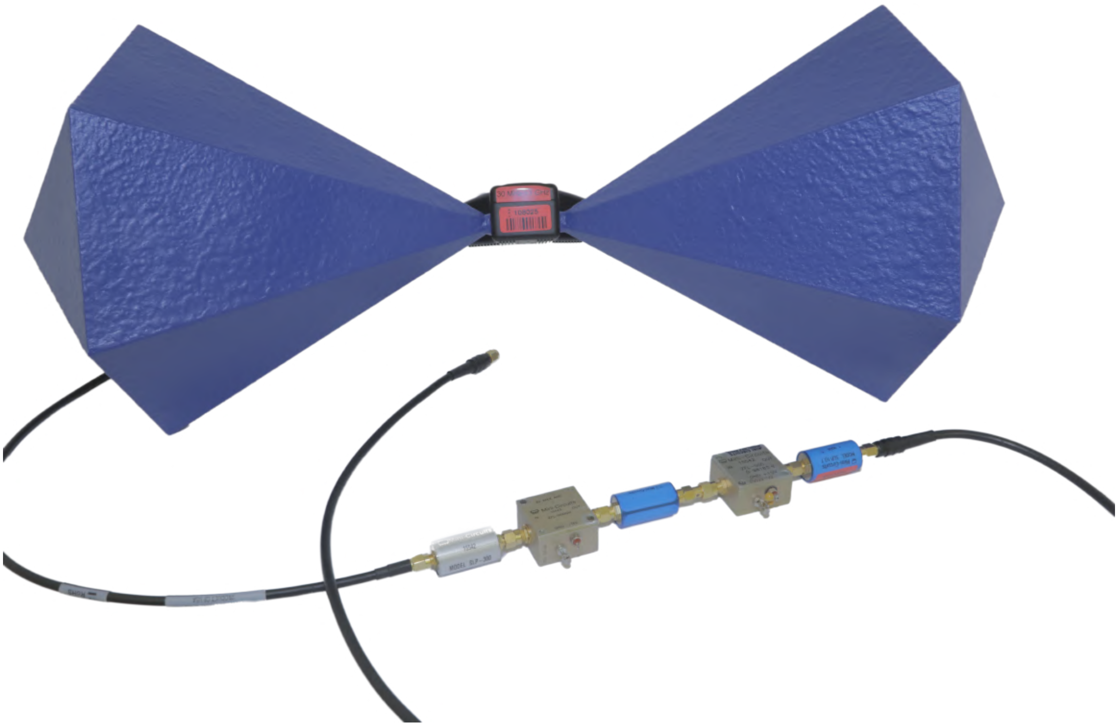


Figure L.2: Antenna, Low Noise Amplifier and noise filtering