Design and simulation – part 2

Receiver

2022 Semester One

# Introduction

This assignment focuses on the receiver part of Electeng 310 project 1. The due date for the assignment is 7th April 2022. This assignment is worth 10 marks. Only one copy of the document and simulation need to be submitted for each group by the group leader.

# Instructions

Please follow the guidelines outlined here:

• Read the assignment document carefully and complete all of the tasks.

• Tasks are **in bold**.

• Be specific and refer to any images accurately when discussing the work.

• Use the snipping tool available in windows and ensure that all the relevant components are visible, with values clearly shown when copying schematics from LTSpice on to this document.

• Include only one or two periods of any ac waveforms when copying output graphs from LTSpice on to this document.

• Use the closest E12 values for resistors and capacitors to ensure that the circuit is as practical as possible.

# Deliverables

The deliverables of this assignment are:

1. A completed version of this document.

Marks will be given based on:

• Completeness and correctness of the answers in this document.

• Clarity of the screenshots presented.

2. An LTspice simulation file of the entire transmitter and the receiver along with any related libraries required to run it.

Marks will be given based on:

• Clarity, correctness, and tidiness of the simulation.

• Understanding of LTSpice features.

• Ability to simplify the simulation without compromising on the accuracy and practicality of the simulation.

Submit a zipped file with a pdf version of this word document, the .asc file of the simulation, and any libraries of the practical components used. Name the zip file “EE310\_GroupXX\_Rx\_2022.zip”, where XX is your group number.

# 1. Light detector

A light detector circuit is a transimpedance amplifier that converts the photo-current from the photo-diode to an output voltage.

An example light detector circuit with labelled components is shown in Figure 1.



Figure 1: An example of the light detector circuit.

## 1.1 Calculation of theoretical values

Using the theory learned in the design meetings, calculate the values for the light detector circuit in Table 1. Remember to use practical E12 values for all resistors and capacitors.

### 1.1.1 Fill out Table 1 with the expected values, including units.

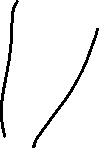


Table 1: Calculate theoretical values for the light detector

|  |  |
| --- | --- |
|  | 100k Ohms |
|  | 7.5pF (Two 15pF in series) |
|  | 212khz |

Here, is the cut-off frequency of the light detector circuit.

## 1.2 Design justifications

**Give a brief justification for the value of each parameter chosen in Table 1.**

|  |
| --- |
| : Considering our PWM signal could have noise that we assume to be approximately 1mV in amplitude, we would want to have a larger gain for our Light detection circuit. We decided on a peak-to-peak value of 5mV for the worst case Ip of approximately 50nA, which made us decide on a value for of 100k Ohms. |
| : While we are amplifying the signal, we would want to reduce the amount of noise that gets amplified. This could be done if we have a small enough capacitance that is able to remove high frequency noise, but still large enough to keep the shape of PWM signal that is detected by the photodiode, which is why we have found a suitable value to be 7.5pF (made with two 15pF capacitors in series). |
| : The cuttoff frequency we decided to go with is 200kHz as our fundamental frequency of the PWM waveform is 40kHz and since the PWM signal is made up of odd multiple integers of the fundamental harmonic, we saw that we only needed 3 harmonics above the fundamental that reaches 200kHz to filter out noise while retaining the shape of the PWM signal. By going slightly above that frequency(we decided on 212kHz) we can account for any inaccuracies and tolerances. |

## 1.3 Simulation

Open the simulation called “Assignment2.asc”. Copy paste the transmitter your group developed in assignment 1 to this project and finish the light detector circuit with the values that you have chosen. Assume the input signal frequency is 1kHz, offset at 3V, with an amplitude of 1V unless stated otherwise.

Make sure that all the values are clearly labelled, and the simulations include any auxiliary components such as voltage or current sources.

### 1.3.1 Paste a screenshot of the light detector circuit from the LTSpice simulation into the box below.

|  |
| --- |
|  |

### 1.3.2 Paste a screenshot of the simulated detector voltage () into the box below when the receiver is 100mm and 400mm away from the transmitter.

Use the cursor tool in LTSpice measure the amplitude of the V\_det in both cases.

|  |  |
| --- | --- |
| **Screenshot of when range is 100mm.** | |
| **Screenshot of when range is 400mm.** | |
| Calculated amplitude of (V) at 100mm range |  |
| Simulated amplitude of (V) at 100mm range |  |
| Calculated amplitude of (V) at 100mm range |  |
| Simulated amplitude of (V) at 100mm range |  |

### 1.3.3 (Optional) Discuss the impact the cut-off frequency () has on the detector voltage () waveform.

Some factors to consider:

How does changing the affect the waveform of ?

What limits the from being set higher in this circuit?

Does gain-bandwidth product have an impact on your design of the light detector?

|  |
| --- |
|  |

# 2. High-pass filter and non-inverting amplifier

A high-pass filter sets a DC bias for and the non-inverting amplifiers are used to amplify lower ranges of .

An example high-pass filter and amplifier circuit with labelled components is shown in Figure 2.

A picture containing text, antenna

Description automatically generatedFigure 2: An example of a high-pass filter and non-inverting amplifiers.

## 2.1 Calculation of theoretical values

Using the theory learned in the design meetings, calculate the values for the light detector circuit in Table 1. Remember to use practical E12 values for all resistors and capacitors. Assume all op-amps here are ideal.

### 2.1.1. Fill out Table 2 with the expected values, including units.

Table 2: Calculate theoretical values for the high-pass filter and amplifiers.

|  |  |
| --- | --- |
|  | 100k Ohms |
|  | 10nF |
|  | 589Hz |
|  | 2k Ohms |
|  | 68k Ohms |
|  | Not Applied |
|  | Not Applied |

Here, is the cut-off frequency of the high-pass filter.

## 2.2 Design Justifications

**Give a brief justification for the values chosen in Table 2.**

|  |
| --- |
| :  We wanted to have a R combination that allowed for a suitable cut-off frequency. We can’t make C\_FH too large or else, we would have a long transient period and that would make it hard to simulate in LTspice. However too large a capacitance value for C\_FH, we would need a smaller value for R\_FH which could increase power losses of the circuit.  : If the cut-off frequency is too large, we start attenuating the input signal frequency from V\_HF. This results in a lot of oscillation in VH\_F. And if the oscillation causes the peaks of the V\_HF to go below V\_bias, that could cause the PWM generator circuit to skip a few peaks and troughs of V\_HF. But if the cut-off frequency is too low, then we wouldn’t remove the DC offset from V\_\_DET when it is filtered to V\_HF. That would cause the DC offset to be amplified by the non-inverting amplifier and would cause the the PWM generator to completely miss the input signal unless not having a really tiny (or unhelpful) hysteresis band. As a result, we have decided on a cut-off frequency (half a decade below the input signal frequency) that has a bit of oscillation, but properly centers V\_HF with a V\_bias DC offset. |
| and : For our non-inverting amplifier circuit, we wanted to have a worst-case amplitude of Vamp that is greater than the amplitude of noise. Therefore, we might large R2 to R1 ratio to increase the gain.  On the other hand, if we increase the gain too much, the OpAmp could become gainbandwidth limited or slew rate limited which could distort the signal.  We decided on a gain of 5mV for the worst case so that our Vamp is still larger than the amplitude of noise (assumed to be around 1mV) but is not so large that we have to use a more expensive OpAmp with a greater slew rate and/or gainbandwidth. |
| and :  This does not apply since we have decided to go with a single gain stage. Justifications for only having a single gain stage:   * The practical OpAmp that we have decided to use (LM324) is not slew rate or gain bandwidth limited for the amplification we desire from the non-inverting Amplifier. * Number of components is reduced, and the cost of the system is reduced for a negligible reduction in performance. * Having one gain stage would also likely increase the response time of the system. |

## 2.3 Simulation

In the LTSpice model “Assignment2.asc”, finish the high-pass filter and the non-inverting amplifier circuit with the values that you have chosen. Choose appropriate practical op-amps for this task.

Make sure that all the values are clearly labelled, and the simulations include any auxiliary components such as sources.

### 2.3.1 Paste a screenshot of the high-pass filter and the non-inverting amplifier circuit from the LTSpice simulation into the box below.

|  |
| --- |
|  |

### 2.3.2 Paste a screenshot of the simulated voltage waveform after the high-pass filter () and voltage waveform after the amplifier () into the box below when the receiver is 100mm away from the transmitter.

Use the cursor tool in LTSpice measure the amplitude of the and .

Note: Using the cursor tools of LTSpice, we would not be able to accurately measure the amplitudes of the waveforms, so we have provided the peak-to-peak measurements instead and the corresponding amplitudes can be found by dividing the peak-to-peak value by 2.

|  |  |
| --- | --- |
| **Screenshot of when range is 100mm.** | |
| **Screenshot of when range is 100mm.** | |
| Simulated amplitude of (V) at 100mm range | (5.30mV/2) = 2.65 |
| Calculated amplitude of (V) at 100mm range | (5mV/2) = 2.5V |
| Simulated amplitude of (V) at 100mm range | (152.26mV/2) = 76.13mV |
| Calculated amplitude of (V) at 100mm range | (175mV/2) = 87.5mV |

### 2.3.3 (Optional) Discuss the impact the high-pass filter cut-off frequency () and the amplifier gain have on the amplifier voltage () waveform.

Some factors to consider:

How does changing the affect the waveform of ?

Does gain-bandwidth product have an impact on your design of the amplifier?

Do you need both stages of the amplifier?

|  |
| --- |
| If our cut-off frequency is too low, then then VHF would still have a DC offset. When this is compared to V\_bias in the amplification stages, then the offset is amplified when V\_amp is generated. If the cut-off frequency is too high, then we will filter out some of the fundamental harmonic from from VH\_F which would causeV\_HF to oscillate. This oscillation gets amplified when passing though the amplification stage(s).  If we only had a single amplification stage and desired a greater amplitude of V\_amp, we would need to increase the gain of the non-inverting amplifier circuit. The problem with this is that the the OpAmp chosen would need a higher gainbandwidth. So an alternative is to have 2 gain stages so that we don’t gainbandwidth limit the OpAmp that we have chosen or need a more expensive Operational Amplifier. |

# 3. PWM regenerator

A PWM regenerator receives and re-creates the PWM signal to remove noise and distortions.

An example PWM regenerator circuit with labelled components is shown in Figure 3.



Figure 3: An example of a PWM regenerator.

## 3.1 Calculation of theoretical values

Using the theory learned in the design meetings, calculate the values for the PWM regenerator circuit in Figure 3. Remember to use practical E12 values for all resistors and capacitors.

### 3.1.1. Fill out Table 3 with the expected values, including units.

Table 3: Calculate theoretical values for the PWM regenerator

|  |  |
| --- | --- |
|  |  |
|  |  |
|  |  |

## 3.2 Design Justification

### 3.2.1 Give a brief justification for the values chosen in Table 3.

|  |
| --- |
| and : |
| : |

## 3.3 Simulation

In the LTSpice model “Assignment2.asc”,finish the PWM regenerator with the values that you have chosen with a practical comparator.

Make sure that all the values are clearly labelled, and the simulations include any auxiliary components such as sources.

### 3.3.1 Paste a screenshot of the PWM regenerator circuit from the LTSpice simulation into the box below.

|  |
| --- |
|  |

### 3.3.2 Paste a screenshot of the simulated regenerated PWM waveform () into the box below when the receiver is 400mm away from the transmitter.

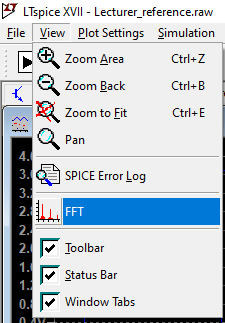
Use the cursor tool in LTSpice measure the amplitude of the . For the FFT waveform, use the cursor to measure the gain at the input signal frequency and carrier frequency.

|  |  |
| --- | --- |
| **Screenshot of when range is 400mm.** | |
| **Screenshot of the FFT of when range is 400mm.** | |
| Gain of at input frequency (1kHz) |  |
| Gain at the carrier frequency |  |

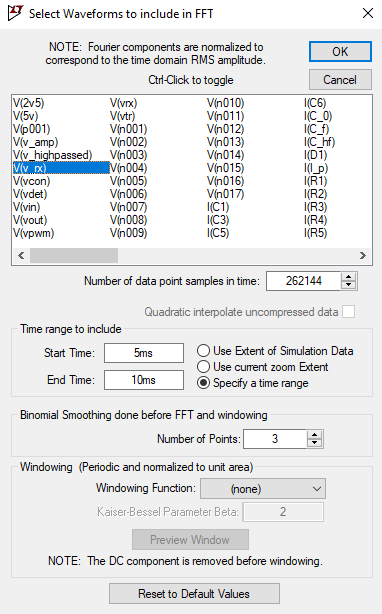
## Obtaining FFT in LTSpice

To obtain the FFT of the PWM signal, click on the LTSpice window with your plots.

Go to View -> FFT



Click on the waveform that you are interested in. In this case, we are interested in Vrx.



A time range can also be specified for the FFT so the transient waveforms that occur as the circuit starts up is not considered in the FFT. In this example, the time range is set to be between 5ms to 10ms to avoid the transients and only consider the steady-state waveform.

# 4. Low-pass filter

A low-pass filter is used to attenuate the higher frequency components of to make a sinusoidal waveform with minimal distortions.

An example low-pass filter circuit with labelled components is shown in Figure 4.



Figure 4: An example low-pass filter.

## 4.1 Calculation of theoretical values

Using the theory learned in the design meetings, calculate the values for the light detector circuit in Table 4. Remember to use practical E12 values for all resistors and capacitors.

### 4.1.1. Fill out Table 4 with the expected values, including units.

Table 4: Calculate theoretical values for the low-pass filter.

|  |  |
| --- | --- |
|  | 100k |
|  | 10k |
|  | 10n |
|  | 820p |
|  | 1.75kHz |

Here, is the passband edge frequency of the low-pass filter.

## 4.2 Design justifications

Give a brief justification for the values chosen in Table 4.

|  |
| --- |
| , , , and :  If we have too high a cut-off frequency, then we would allow the higher frequency of the carrier waveform to be present in Vout which would cause harmonic distortion to increase. If our cut-off frequency is too low, then we have the risk of over attenuating the fundamental frequency of the input signal.  Once we decide on a suitable f\_C, we decide on suitable values for R8 and R9 and find values for C1 and C2 so that the amount of ripple in the pass-band of the second-order type-1 Chebychev filter is below 1db, and a gain of 1 for the signal at our cut-off frequency.  When deciding on a cut-off frequency we also considered that the attenuation of the frequencies greater than our cut-off frequency gradually decrease in the gain for increasing frequencies. If we wanted a steeper attenuation that is closer to the ideal, we could make the filter a higher-order filter. This would increase the number of components, which would increase the cost of the implementation. My group has decided to continue to investigate the benefit of this for our implementation, but for this assignment we have decided to stay with a second order type-I chebychev filter as this provides sufficient attenuation in the stop band. |
|  |

## 4.3 Simulation

In the LTSpice model “Assignment2.asc”,finish the low-pass filter and the non-inverting amplifier circuit with the values that you have chosen. Use practical op-amps for this task

Make sure that all the values are clearly labelled, and the simulations include any auxiliary components such as sources.

### 4.3.1 Paste a screenshot of the low-pass filter circuit from the LTSpice simulation into the box below.

|  |
| --- |
|  |

### 4.3.2 Paste a screenshot of the simulated voltage waveform after the low-pass filter (V\_out) into the box below when the receiver is 100mm away from the transmitter.

Use the cursor tool in LTSpice measure the amplitude and frequency of the .

|  |  |
| --- | --- |
| **Screenshot of when range is 100mm when is a 1kHz sinusoid with an offset of 3V and an amplitude of 1V.** | |
| **Screenshot of when range is 100mm when is a 2.5kHz sinusoid with an offset of 3V and an amplitude of 1V.** | |
| Simulated amplitude of (V) at = 1kHz |  |
| Simulated frequency of (V) at = 1kHz |  |
| Simulated amplitude of (V) at = 2.5kHz |  |
| Simulated frequency of (V) at = 2.5kHz |  |

|  |
| --- |
| **Screenshot of the FFT for when range is 100mm when = 1kHz.** |

### 4.3.3 Find the total harmonic distortion (THD) of and when input frequency is 1kHz.

|  |  |
| --- | --- |
| THD of (V) at = 1kHz |  |
| THD of (V) at = 1kHz |  |

### 4.3.4 (Optional) Discuss the impact the low-pass filter passband edge frequency () and the carrier frequency have on the output voltage () waveform.

Some factors to consider:

Compare the THD and FFT plots from and .

How does changing the affect the waveform of ?

How does changing the carrier frequency affect the waveform of ?

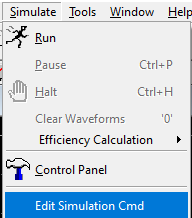
|  |
| --- |
|  |
|  |

## Obtaining THD in LTSpice

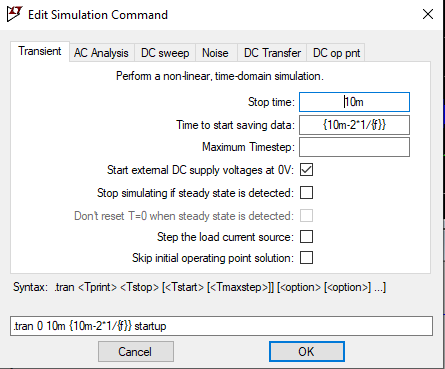
To obtain the THD of a signal in LTSpice, first run the simulation and save results only when the waveform has reached steady-state to disregard the transient waveforms.

An example of this could be to enable ‘Time to start saving data’ in the LTSpice.

Simulate -> Edit Simulation Cmd

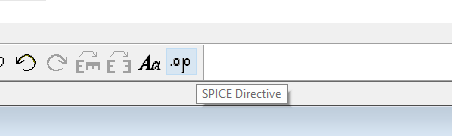
****

In this example, the time to start saving data is set to the last two periods of the simulation. The transients dies down at this point so only the steady-state is considered for THD.

****

**Time to start saving data: {10m-2\*1/{f}}**

A SPICE Directive needs to be set up in LTSpice to measure the frequency components starting from a specific frequency. Below is where SPICE Directives can be set.

****

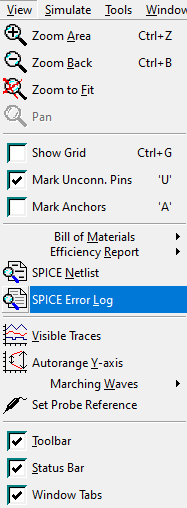
The LTSpice directive will be in the following format: .fourier frequency harmonics signal.  
An example is shown below to get the 100 harmonic components starting from the input frequency for .

**P460#yIS1**

**LTSpice directive: .fourier {f} 100 V(vout)**

*This is assuming the THD of the voltage waveform called vout is being measured. To measure THD of another voltage waveform, please change the V(vout) to the name of that waveform.*

To check the THD, View -> SPICE Error Log

****

In the error log, the harmonic components of the waveform analysed is shown. The THD measured by LTSpice is listed at the end of the harmonic components.