Design/Simulation – Part 1

Jackman Lin, Seho Kim, Dariusz Kacprzak

2022 Semester One

Introduction

The goal of this assignment is to familiarize the class with using LTspice to design complex analogue circuitry. This document can be completed by attending the six laboratories between March 8, to March 21, 2022. The labs are going to be **online.** See CANVAS for the zoom link for these labs. TAs and lecturers will be present to help with the tasks presented in this assignment.

This document is part of the submission for the first part of the Design/Simulation, focused on the transmitter. It is due on **March 22, 2022**. Read each task carefully and ensure that all instructions are followed.

This assignment is worth **10 marks**, only one document needs 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 the mandatory tasks.
- Tasks are **bolded**.
- When discussing the work, be specific and refer to any images accurately
- When copy pasting schematics from LTSpice, use the snipping tool available in windows and ensure that all the relevant components are visible, with values clearly shown
- When copy pasting output graphs from LTSpice, include only **one or two periods** of any ac waveforms.
- Use the nearest E12 value when designing your circuitry to ensure that the simulation is as practical as possible.
- The tasks are not presented in order, they are instead split into functional blocks.

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 any screenshots presented
2. An LTspice simulation file of the entire transmitter side along with any related libraries required to run it.
   Marks will be given based on:
   - Clarity, correctness, and tidiness of the simulation.
   - Understanding of how to use LTSpice features
   - Use of techniques to simplify the simulation without compromising the 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_Group*XX*_Tx_2022.zip"**, where *XX* is your group number.

1. Chosen design Parameters

In this project, students must select various design parameters based on the information given in the design meetings. These parameters include:

- f_{PWM}: The frequency of the PWM Modulation signal
- $v_{tri,p-p}$: The peak-to-peak voltage of the triangle wave in the triangle wave generator
- V_{supply}: The supply voltage of the transmitter side (indicate whether you are using single or dual sided supplies)
- V_{ref}: The reference voltage for the Op-amps and Comparators
- i_f: The maximum forward current that the transmitting LED will be driven at
- $v_{con,p-p}$: The peak-to-peak voltage of the conditioned input ac sine wave
- v_{hyst}: The hysteresis band of the PWM generator

1.1. Design Parameters

Choose suitable values for each of the parameters discussed in task 1. Some parameters will make sense as you progress through the lectures

1.1.1. Fill in Table 1 with the standard design parameters chosen, including units.

Table 1: Chosen parameters for design

f_{PWM}	40kHz
$v_{tri,p-p}$	2V
V_{supply}	5V
V_{ref}	2.5V
$i_{LED,max}$	100mA
$v_{con,p-p}$	1.9V
v_{hyst}	2V

1.2. Design Justifications

1.2.1. Give a brief (one to two sentences) justification for the value of each parameter chosen in Table 1.

f_{PWM}:

40kHz would allow for 16 triangular waveforms for every 1 input signal waveform to have a high number of intersections between the conditioned input signal and the triangular waveform and a higher PWM resolution. A lower frequency than 40kHz could cause more distortion on the input signal of the receiver and a higher frequency than 40kHz might distort the waveform due to components with lower slew rates.

$v_{tri,p-p}$:

We would like to not favor a higher or lower PWM duty cycle, so we would need to make the $v_{tri,p-p}$ similar in value to the peak-to-peak voltage of the conditioned input signal. We would condition the peak-to-peak voltage of the input signal to be slightly lower than $v_{tri,p-p}$ so that we don't risk having consecutive intersections of the conditioned input and triangular waveforms occur to soon after one another and risk being a shorter time period than the response time of the comparator that generates the PWM signal.

V_{supply}:

We would want to regulate the variable input voltage (9V to 16V) so that we have a more stable voltage, but we would want to take the worst-case scenario (9V) and have a regulator integrated circuit that converts that 9V into 5V, which is a supply voltage commonly used by cheaper OpAmps and Comparators.

V_{ref}:

We know from our calculations that it would be best to choose a V_{ref} that is half the voltage of the voltage top value of our triangular waveform, to ensure that our triangular waveform is symmetrical, so we have decided on a V_{ref} of 2.5V (5V/2).

i_f:

Assuming that the LED to be used is the KA-3529 LED, the datasheet of the KA-3529 has an input DC current of 150mA. To ensure that we don't damage the LED, for protection we would drive it at 100mA instead of 150mA

$v_{con,p-p}$:

The input signal of 2Vpk-pk would need to be slightly conditioned to a lower value so that we don't run risk of missing several switching time periods between the maximum and the minimum value in the PWM signal if two consecutive intersections of the triangular and input waveforms are too soon after each other. 1.5Vpk-pk seems suitable.

v_{hyst}:

The hysteresis voltage band of the PWM generator would be what creates our desired peak-to-peak voltage of our triangular waveform of 2V. With that said, $v_{hyst} = 2V$

2. Triangle wave generator

In this project, a Triangle Wave Generator (TWG) is used to produce a high frequency modulation signal to encode the input sinusoid.

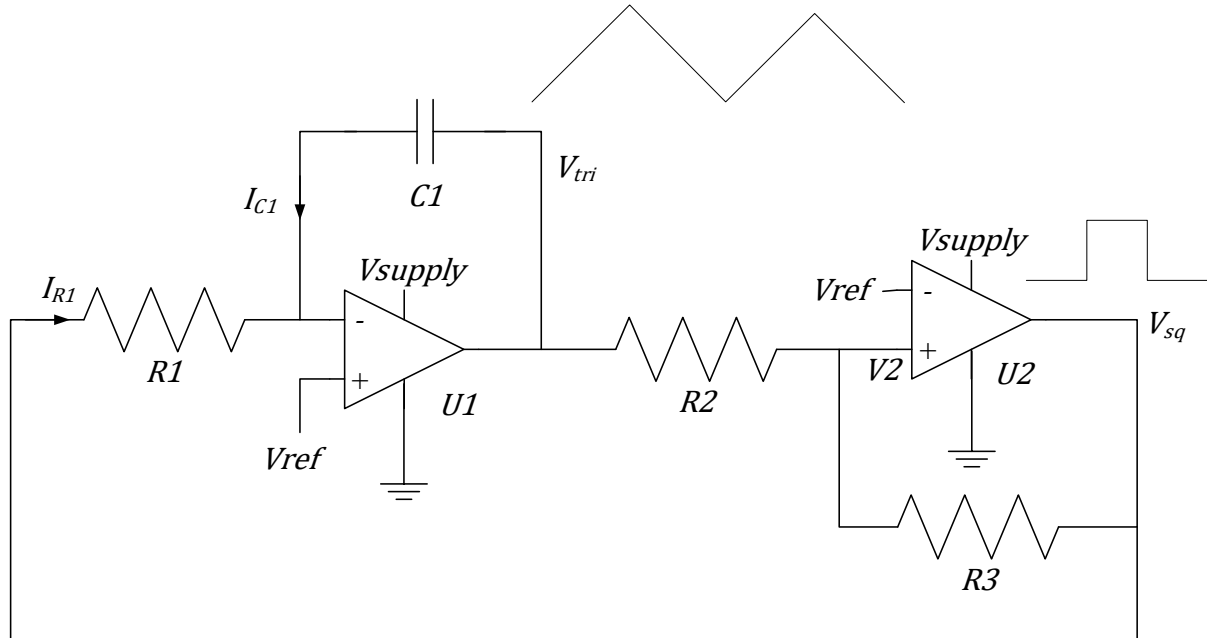A standard TWG with labelled components is given in Figure 1

Figure 1: A standard TWG

2.1. Calculating theoretical values

Using the theory learned in the design meetings, calculate the required values for the Triangle Wave Generator to achieve the f_{PWM} stated in Table 1.

2.1.1. Fill out Table 2 with the expected values, including units.
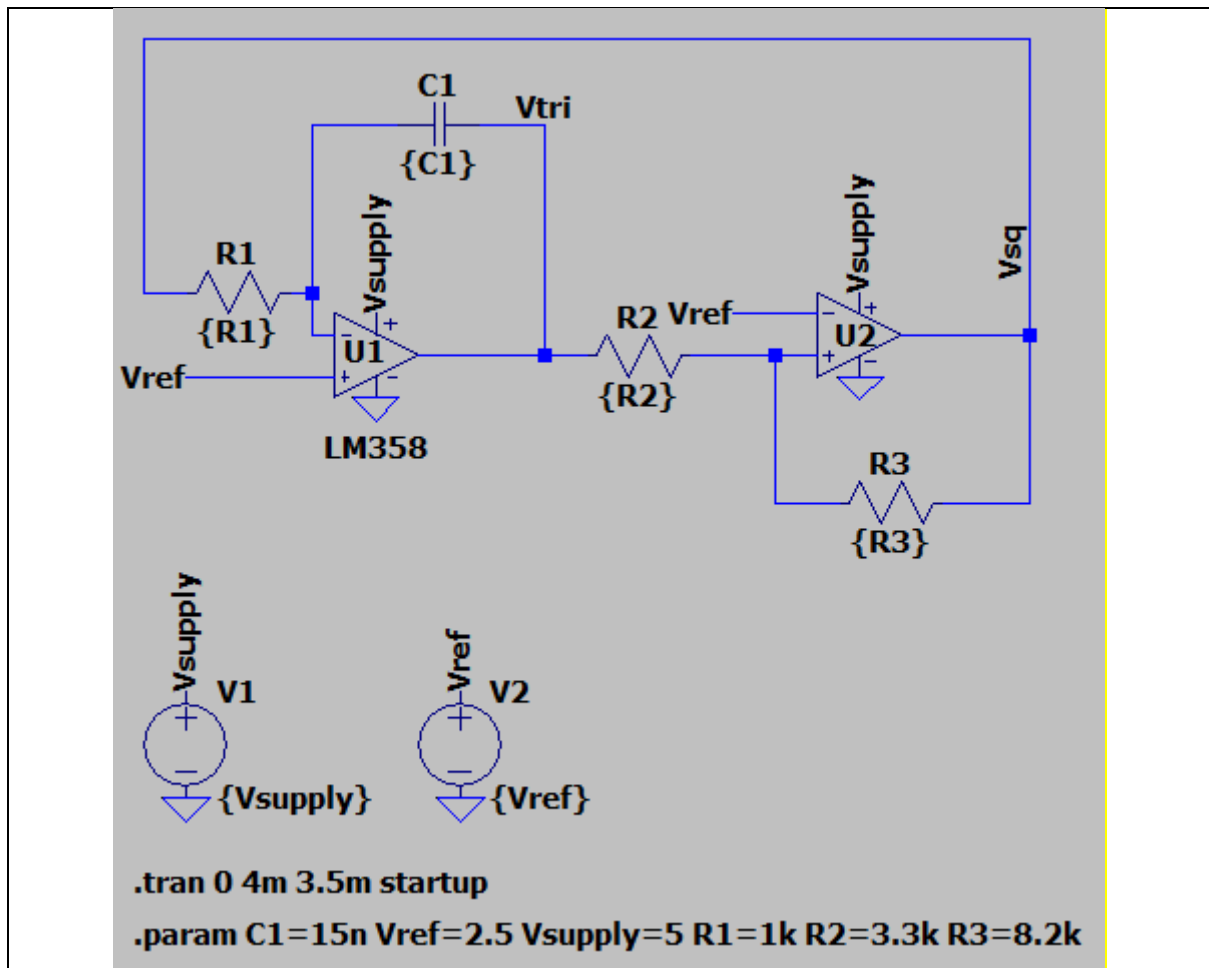
Table 2: Calculate theoretical values for the TWG

R_1	1k
R_2	3.3k
R_3	8.2k
C_1	15n

2.2. Simulation with ideal values

Open the simulation called "Assignment1_Task2_2.asc". This file contains a simulation of an ideal TWG. Follow the instructions in the simulation to complete task 2.2.

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

2.2.1. Paste a screenshot of the final schematic from the LTSpice simulation into the box below.

```
                    C1        Vtri
                    ||
                   {C1}
        R1              Vsupply              Vsupply      Vsq
        W                                    -
       {R1}           +  U1      R2 Vref        U2
   Vref              -                W          +
                        -            {R2}
                    LM358                      R3
                                               W
                                              {R3}

   Vsupply          Vref
      V1              V2
   {Vsupply}       {Vref}

.tran 0 4m 3.5m startup
.param C1=15n Vref=2.5 Vsupply=5 R1=1k R2=3.3k R3=8.2k
```

2.2.2. **Paste a screenshot of the simulated Triangle wave (V_{tri}) and Square wave (V_{sq}) into the box below.**
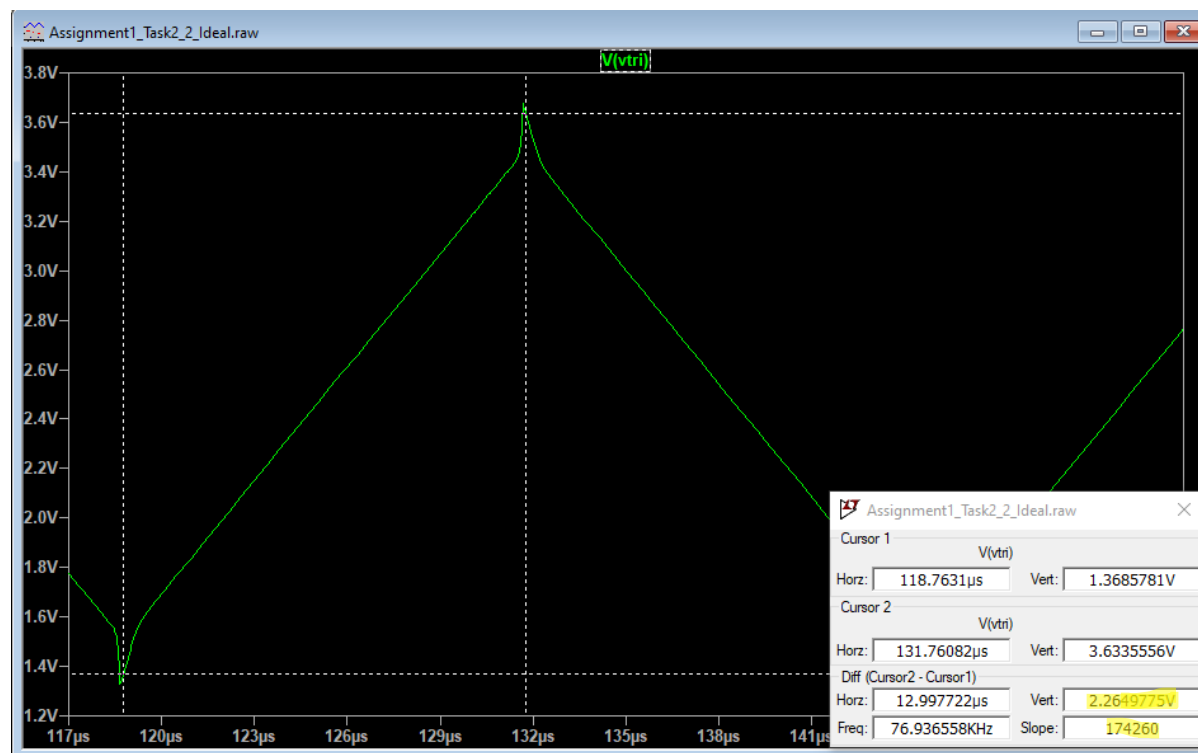
Using the cursor tool in LTSpice, measure

- f_{PWM} of the square wave
- $V_{p-p,tri}$ of the triangle wave
- The rising slope of the triangle wave ($\frac{dv_{tri}}{dt}$)
- The falling slope of the triangle wave ($\frac{dv_{tri}}{dt}$)
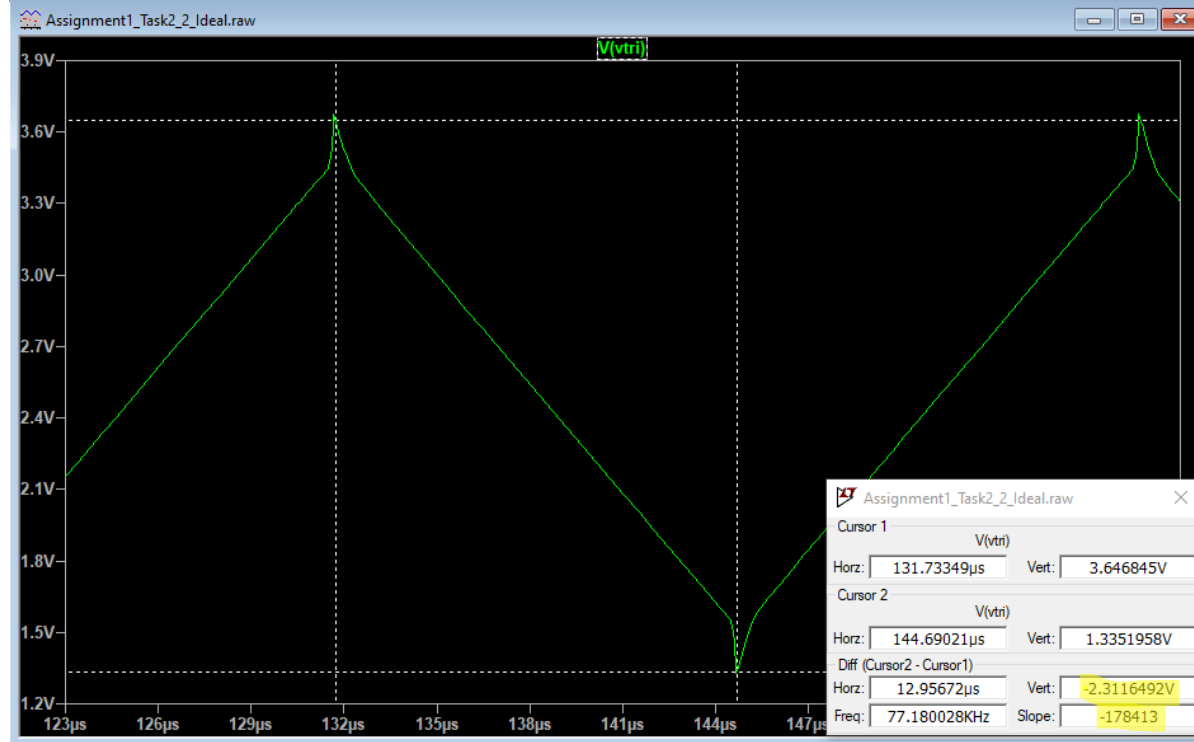
Screenshot of Square wave showing V_{sq} and f_{PWM}
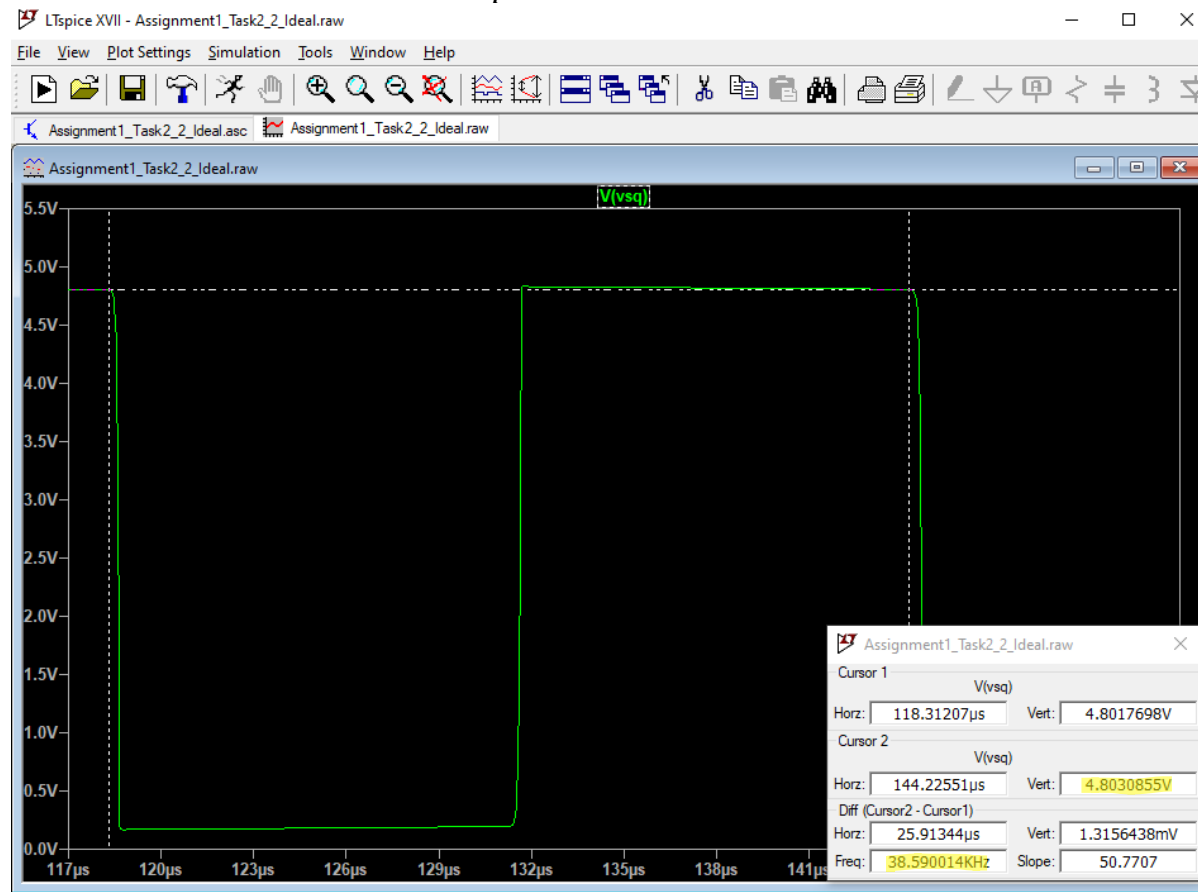
Screenshot of triangle wave showing $V_{p-p,tri}$ and the rising slope of the triangle wave $\frac{dv_{tri}}{dt}$ (measured using cursors)

Screenshot of triangle wave showing $V_{p-p,tri}$ **and the falling slope of the triangle wave** $\frac{dv_{tri}}{dt}$ (measured using cursors)

Calculated f_{PWM} (kHz)	40kHz
Simulated f_{PWM} (kHz)	38.59kHz
Calculated $V_{p-p,tri}$ (V)	2.00
Simulated $V_{p-p,tri}$ (V)	2.26
Simulated rising slope of $\frac{dv_{tri}}{dt}$ $(v/\mu s)$	0.17
Calculated rising slope of $\frac{dv_{tri}}{dt}$ $(v/\mu s)$	0.16
Simulated falling slope of $\frac{dv_{tri}}{dt}$ $(v/\mu s)$	-0.18
Calculated falling slope of $\frac{dv_{tri}}{dt}$ $(v/\mu s)$	-0.16

Save a copy of this simulation as "EE310_Assignment1_TX_Group_X.asc ", where X is your group number. This new schematic will be the file that needs to be submitted for the assessment.

      2.2.3. Optional task: Make some observations on the accuracy of the theory and the ideal simulation.

2.3. Practical components

Using the new saved file developed in task 2.2.2, each group will develop their model using practical models for each component. In practice, op-amps have limited GBWs and slew rates, and comparators have different output configurations and response times.

Libraries for the LM358, and TLC082 op-amps have been included. The LM393 comparator is also included. Please read the included pdf on how to include various components. Further support is offered in the labs.

2.3.1.**Fill in** Table 3 **with the general detail of the op-amp and comparator selected.**

Table 3: Op-amp selection table

Op-amp	LM358
Op-amp GBW	100kHz
Op-amp slew rate (V/us)	0.2V/us
Comparator	LM393
Output response time (us)	500ns
Output type (TTL, Open Collector)	Open Collector

In the new LTSpice schematic, replace the op-amp and the comparator with your chosen models.

2.3.2.**Paste a screenshot of the schematic into the box below.**
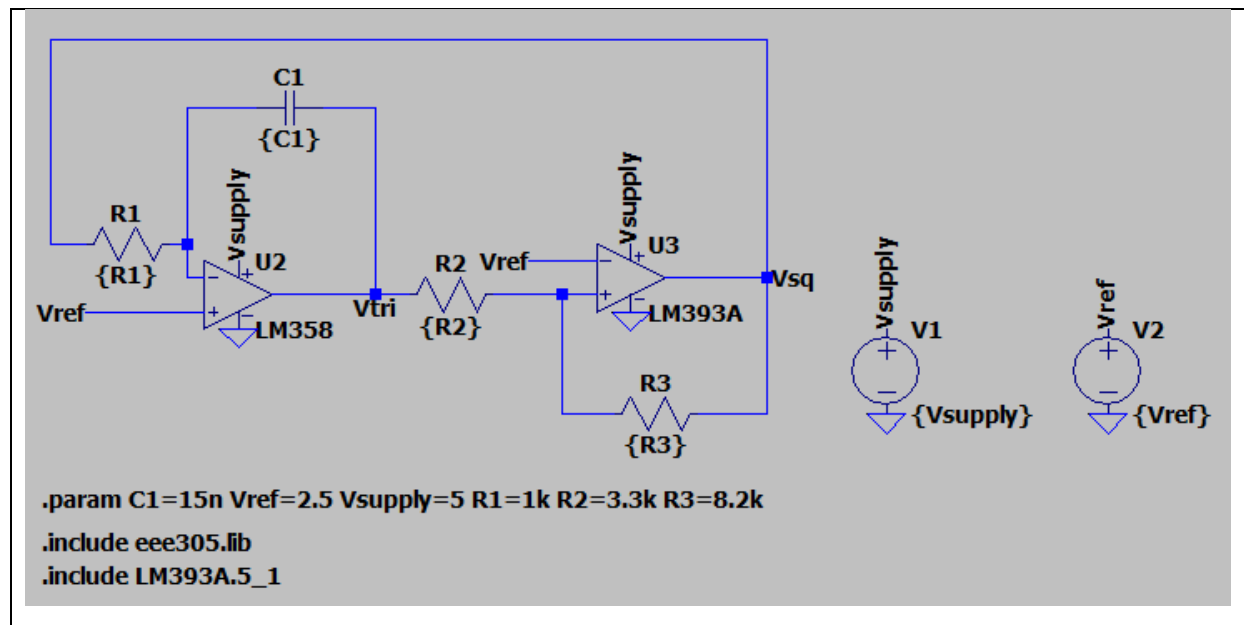
Paste a screenshot of the simulated Triangle wave (V_{tri}) and Square wave (V_{sq}) into the box below.

Using the cursor tool in LTSpice, measure

- f_{PWM} of the square wave,

- $V_{p-p,tri}$ of the triangle wave
- The rising slope of the triangle wave ($\frac{dv_{tri}}{dt}$)
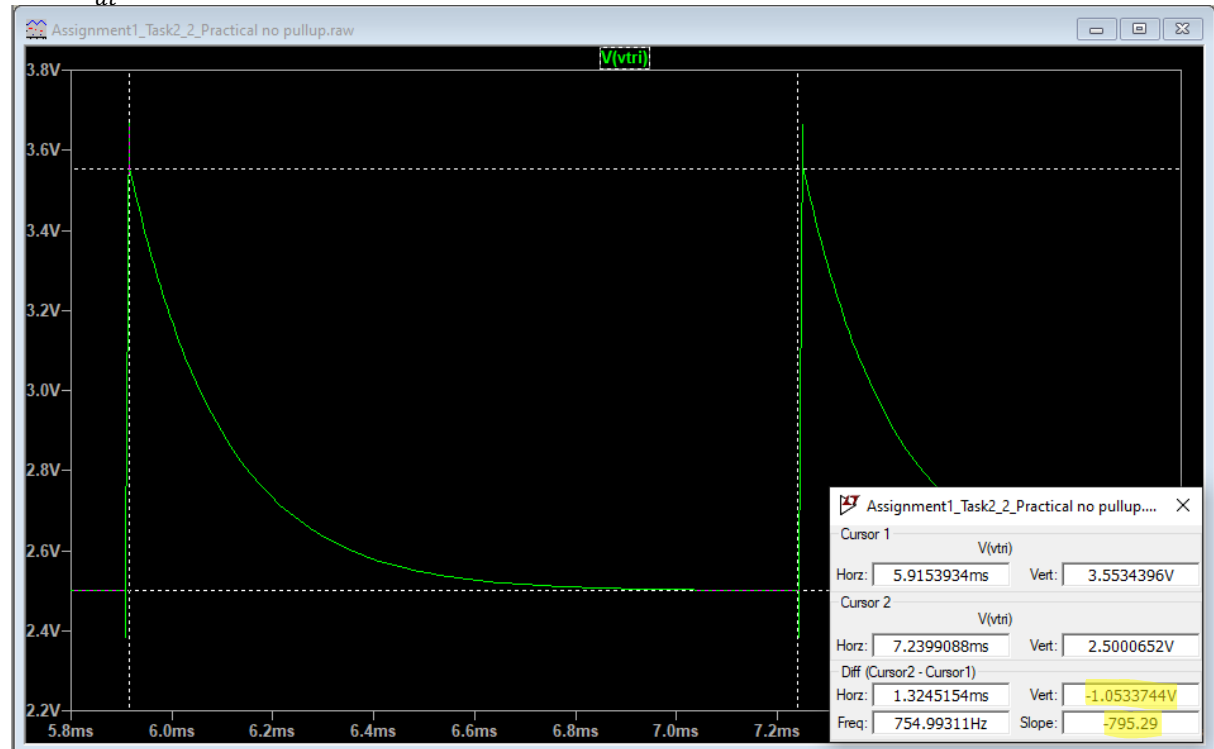- The falling slope of the triangle wave ($\frac{dv_{tri}}{dt}$)

Screenshot of the practical square wave showing V_{sq} and f_{PWM}
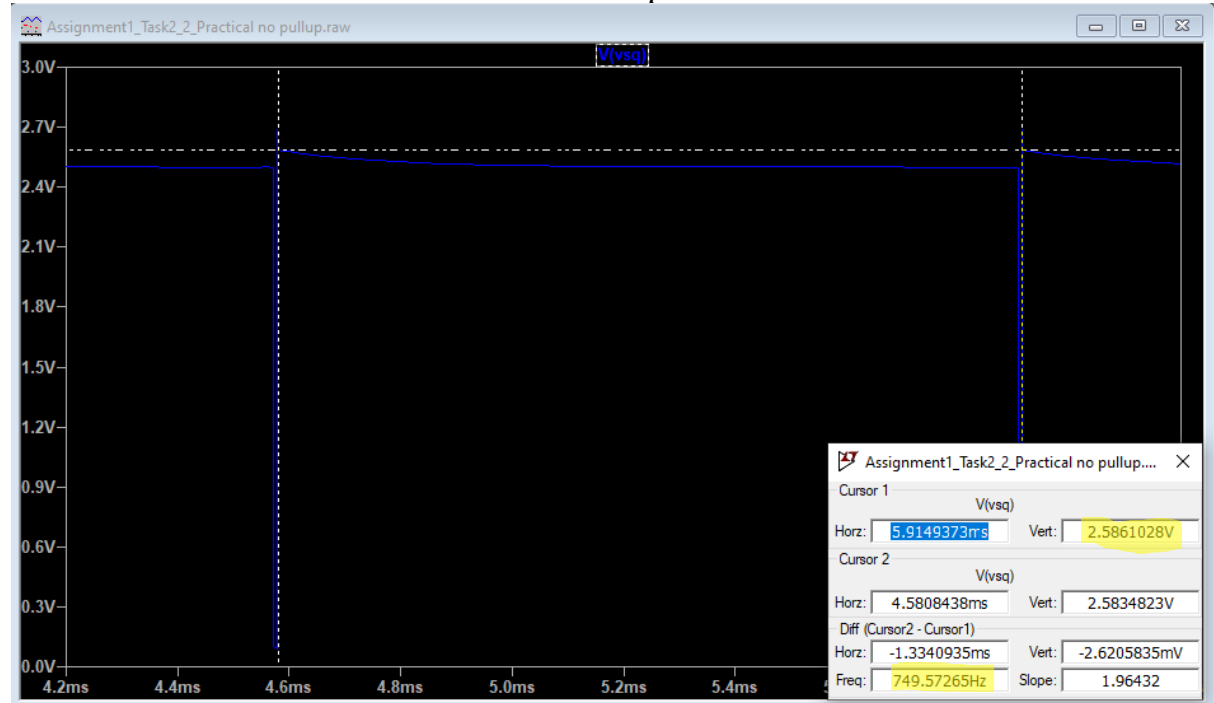
Screenshot of practical triangle wave showing $V_{p-p,tri}$ and the measured rising slope of the triangle wave $\frac{dv_{tri}}{dt}$
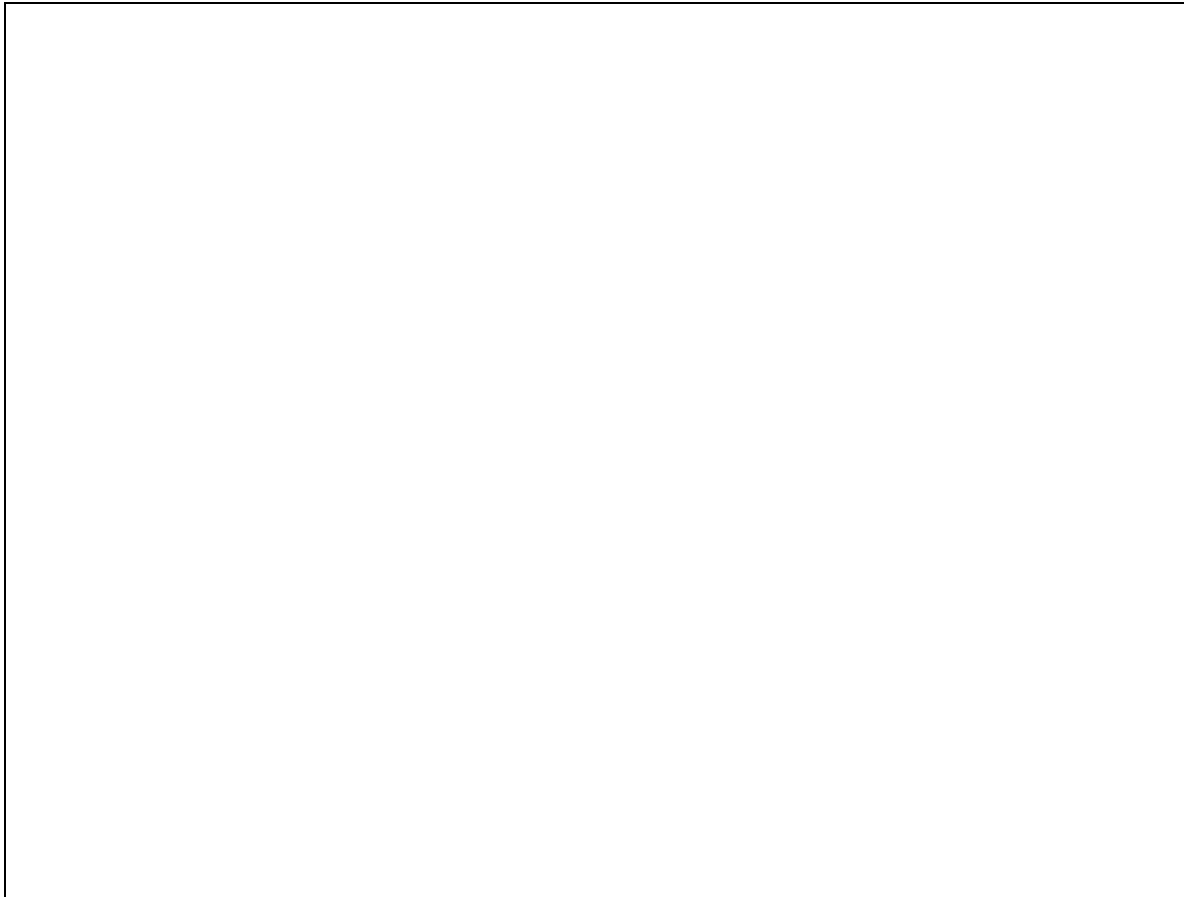
Screenshot of practical triangle wave showing $V_{p-p,tri}$ and the measured falling slope of the triangle wave $\frac{dv_{tri}}{dt}$

Ideal f_{PWM} (kHz)	40k
Simulated f_{PWM} (kHz)	0.75
Ideal $V_{p-p,tri}$ (V)	2.00
Simulated $V_{p-p,tri}$ (V)	1.28
Simulated rising slope of $\frac{dv_{tri}}{dt}$ $(v/\mu s)$	0.19
Calculated rising slope of $\frac{dv_{tri}}{dt}$ $(v/\mu s)$	0.16
Simulated falling slope of $\frac{dv_{tri}}{dt}$ $(v/\mu s)$	-0.000795
Calculated falling slope of $\frac{dv_{tri}}{dt}$ $(v/\mu s)$	-0.16

2.3.3. Optional task: Discuss any differences observed between the practical simulation and the ideal simulation performed in Task 2.2.
- Think about the cause of any differences in f_{PWM} and V_{tri} between the ideal model and the practical model
- Think about what needs to be done to set the frequency to the desired value as stated in Table 1
- Think about what needs to be done to set the $v_{p-p,tri}$ to the desired value as stated in Table 1
- Think about any additional components required for the practical simulation and why they are needed.
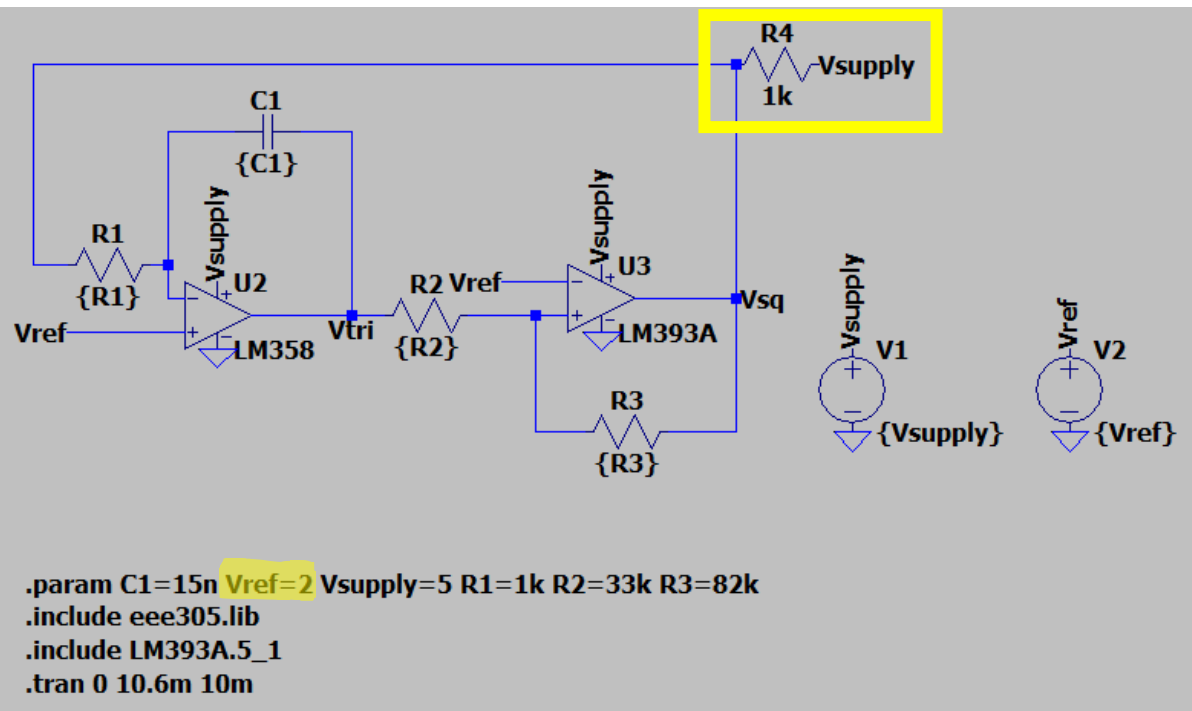
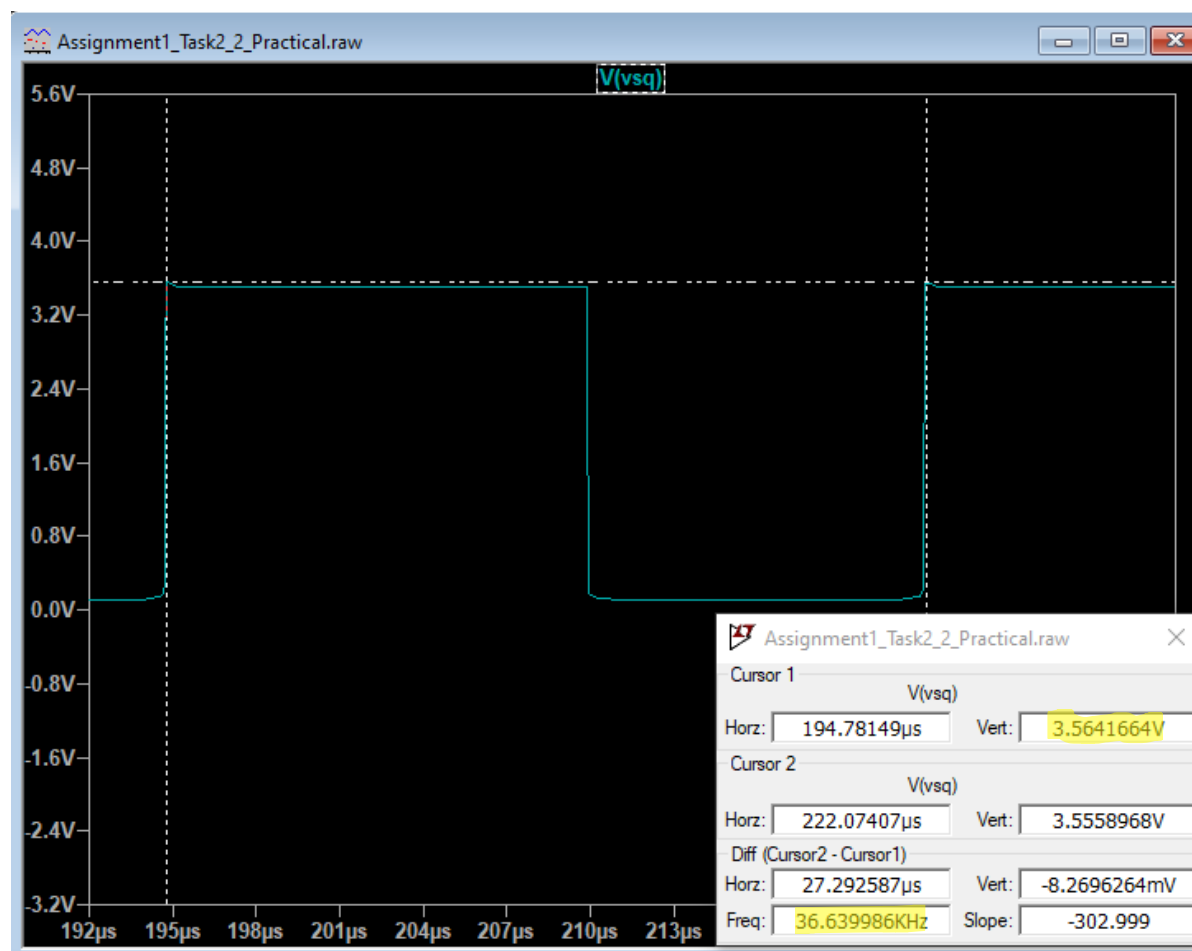2.3.4. Implement any changes required so the parameters set in Table 1 are matched

Paste a screenshot of the simulated Triangle wave (V_{tri}) and square wave (V_{sq}) into the box below.

Using the cursor tool in LTSpice, measure

- f_{PWM} of the square wave,
- $V_{p-p,tri}$ of the triangle wave
- The rising slope of the triangle wave ($\frac{dv_{tri}}{dt}$)
- The falling slope of the triangle wave ($\frac{dv_{tri}}{dt}$)

R4 —Vsupply
1k

C1
{C1}

R1
{R1}

Vref

U2
LM358
Vtri

R2 Vref
{R2}

U3
LM393A
Vsq

R3
{R3}

V1
{Vsupply}

V2
{Vref}

.param C1=15n Vref=2 Vsupply=5 R1=1k R2=33k R3=82k
.include eee305.lib
.include LM393A.5_1
.tran 0 10.6m 10m

Screenshot of the practical square wave showing V_{sq} and f_{PWM}

Assignment1_Task2_2_Practical.raw

V(vsq)

Cursor 1	V(vsq)	
Horz: 194.78149µs	Vert:	3.5641664V
Cursor 2	V(vsq)	
Horz: 222.07407µs	Vert:	3.5558968V
Diff (Cursor2 - Cursor1)		
Horz: 27.292587µs	Vert:	-8.2696264mV
Freq: 36.639986KHz	Slope:	-302.999

Screenshot of practical triangle wave showing $V_{p-p,tri}$ and the measured rising slope of the triangle wave $\frac{dv_{tri}}{dt}$

Screenshot of practical triangle wave showing $V_{p-p,tri}$ and the measured falling slope of the triangle wave $\frac{dv_{tri}}{dt}$

Ideal f_{PWM} (kHz)	**40k**
Simulated f_{PWM} (kHz)	**36.64**
Ideal $V_{p-p,tri}$ (V)	**2.00**
Simulated $V_{p-p,tri}$ (V)	**1.74**
Ideal rising slope of $\frac{dv_{tri}}{dt}$ ($v/\mu s$)	**0.17**
Simulated rising slope of $\frac{dv_{tri}}{dt}$ ($v/\mu s$)	**0.14**
Ideal falling slope of $\frac{dv_{tri}}{dt}$ ($v/\mu s$)	**-0.18**
Simulated falling slope of $\frac{dv_{tri}}{dt}$ ($v/\mu s$)	**-0.11**

3. Input Signal Conditioner

The purpose of the input signal conditioner (ISC) is to process your input sinusoid, which represents the speed of the car, into a waveform which is compatible with the TGW. The input waveform is a sinusoid with a fixed amplitude of 1 V, a variable offset between 1-3V, and a variable frequency between 1 – 2.5 kHz. The frequency of the waveform relates to the speed of the front vehicle.

3.1. Inputs and Outputs

3.1.1. **Write a brief (one paragraph) description of what factors governs what the output of the ISC needs to be. Consider the design of the TWG in the previous section.**

Our input signal has a variable DC offset. To make out input signal more stable, we would first remove the DC offset using a high-pass filter and then shift the output of the high-pass filter upwards by 2V DC offset. The 2V DC shift is done so that the input signal has the same DC offset as the triangular waveform created in the previous stage. While changing the DC offset of the input signal to 2V, we would also reduce the signal to a 1V pk-pk voltage signal to ensure that the triangular waveform has a larger pk-pk voltage than the input signal so that later on when our PWM signal is generated, we do not skip over any intersections that occur between our input signal and our triangular waveform.

3.2. ISC Circuit simulation

Your group needs to design an ISC that can meet the requirements outlined in task 3.1.1. Design a practical ISC circuit using the same schematic file as the TWG.

### 3.1.1. **Paste a screenshot of the proposed LTSpice schematic of the ISC**
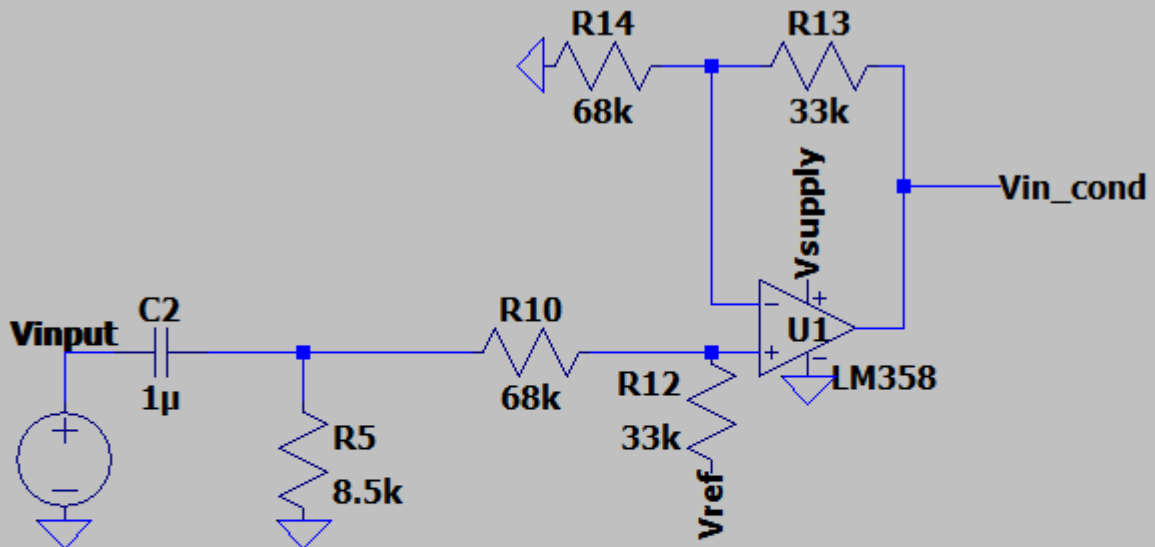
.param Vinput_frequency=2.5k Vinput_offset=1
.param C1=15n Vref=2 Vsupply=5 R1=1k R2=33k R3=82k
.include eee305.lib
.include LM393A.5_1

3.1.2. For this task, set the input waveform to 2V offset with 1 V amplitude. Set the frequency to 1 kHz

Paste the input and output sinusoids of the ISC below.

Use the cursor tools to confirm that the input waveform is correct, and that the output waveform matches the description written in task 3.1.1
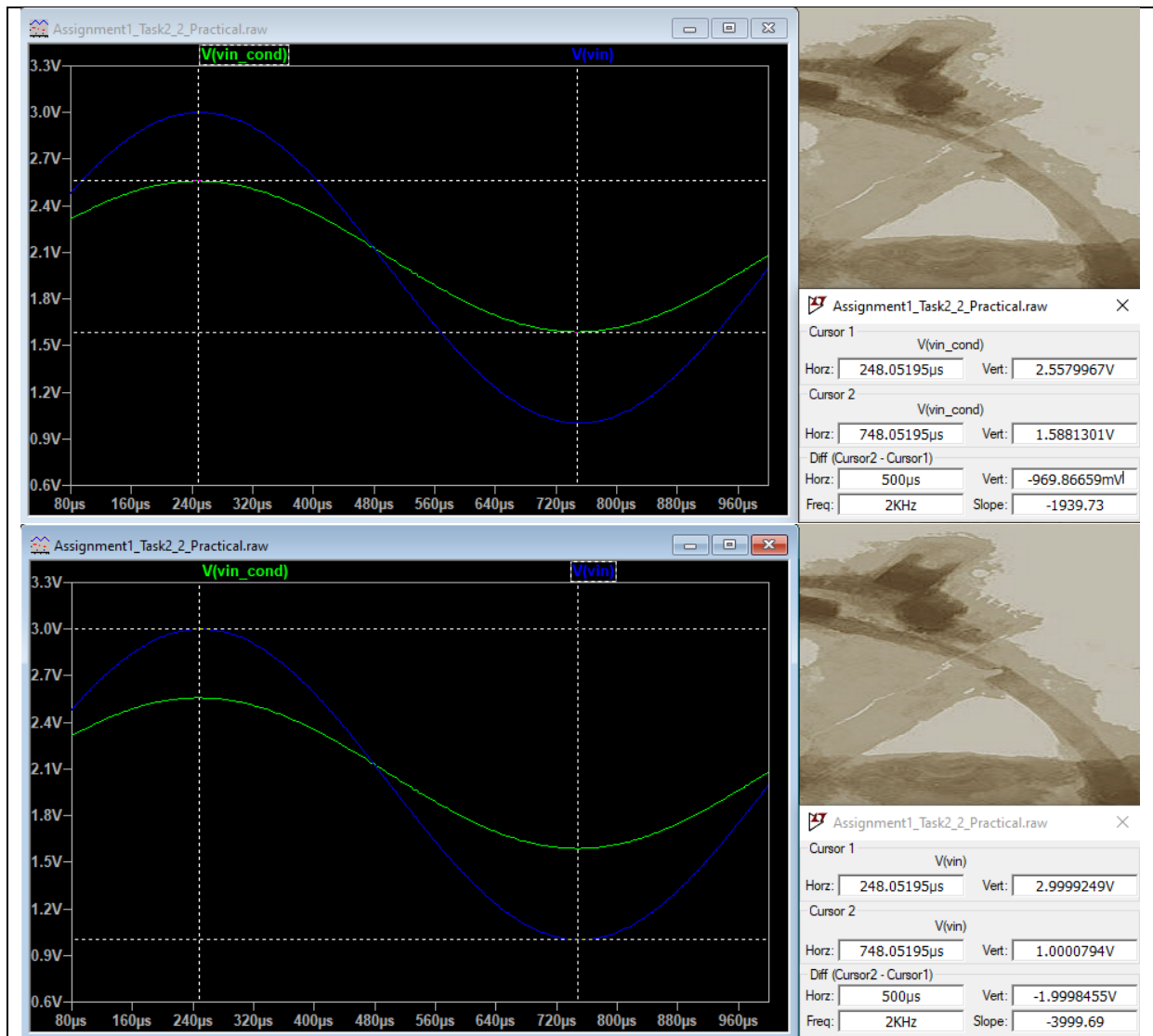
3.1.3. For this task, set the input waveform to 3V offset with 1 V amplitude. Set the frequency to 1 kHz

Paste the input and output sinusoids of the ISC below.

Use the cursor tools to confirm that the input waveform is correct, and that the output waveform matches the description written in task 3.1.1

3.1.4. **For this task, set the input waveform to 2V offset with 1 V amplitude. Set the frequency to 2.5 kHz**
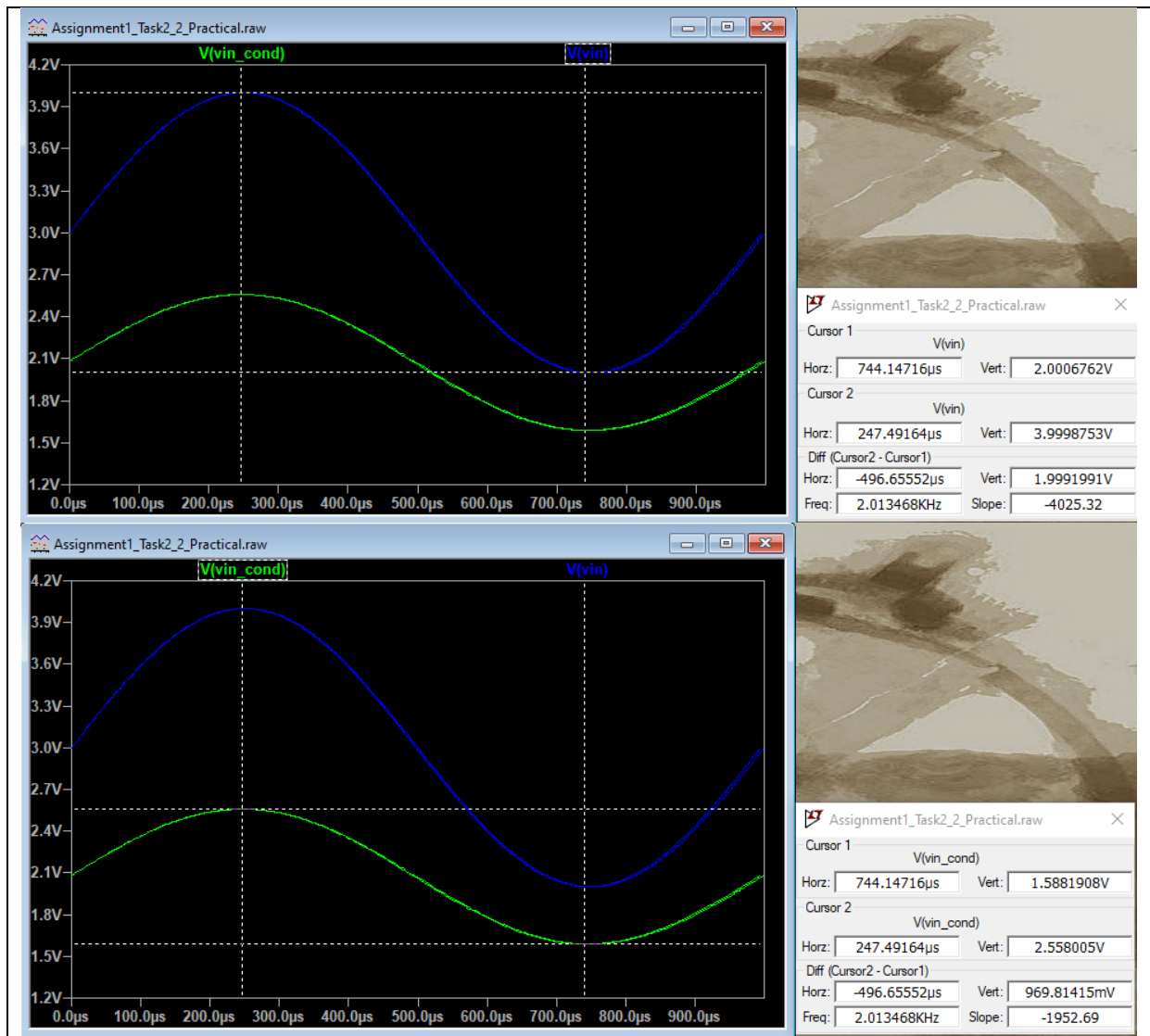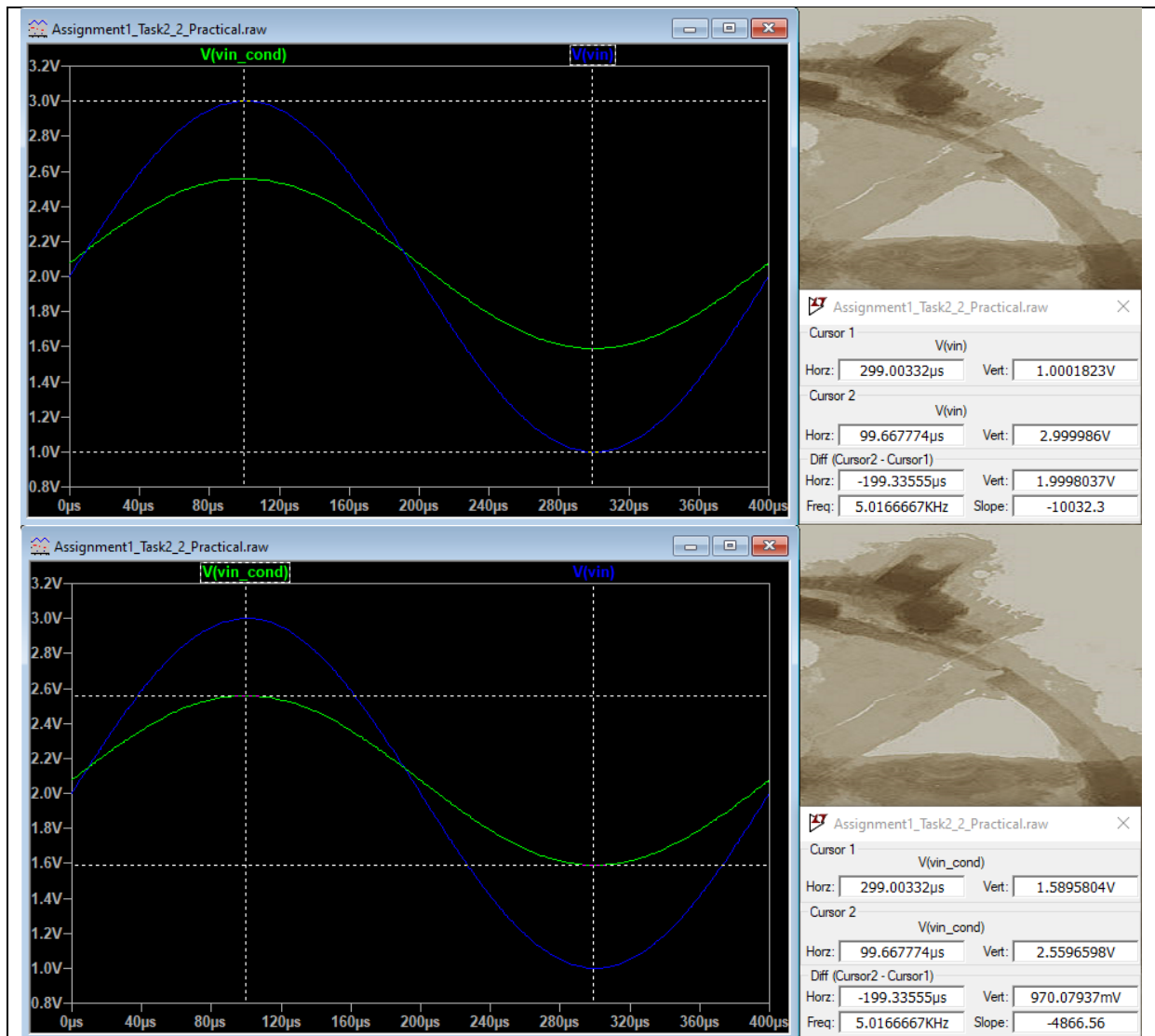
Paste the input and output sinusoids of the ISC below.

Use the cursor tools to confirm that the input waveform is correct, and that the output waveform matches the description written in task 3.1.1

Window 1: Assignment1_Task2_2.raw — V(vin_cond) (green), V(vin) (blue)

Cursor panel (V(vin)):

Cursor 1	V(vin)		
Horz:	299.00332µs	Vert:	1.0001823V
Cursor 2	V(vin)		
Horz:	99.667774µs	Vert:	2.999986V
Diff (Cursor2 - Cursor1)			
Horz:	-199.33555µs	Vert:	1.9998037V
Freq:	5.0166667KHz	Slope:	-10032.3

Window 2: Assignment1_Task2_2.raw — V(vin_cond) (green), V(vin) (blue)

Cursor panel (V(vin_cond)):

Cursor 1	V(vin_cond)		
Horz:	299.00332µs	Vert:	1.5895804V
Cursor 2	V(vin_cond)		
Horz:	99.667774µs	Vert:	2.5596598V
Diff (Cursor2 - Cursor1)			
Horz:	-199.33555µs	Vert:	970.07937mV
Freq:	5.0166667KHz	Slope:	-4866.56

3.1.5. For this task, set the input waveform to 3V offset with 1 V amplitude. Set the frequency to 2.5 kHz

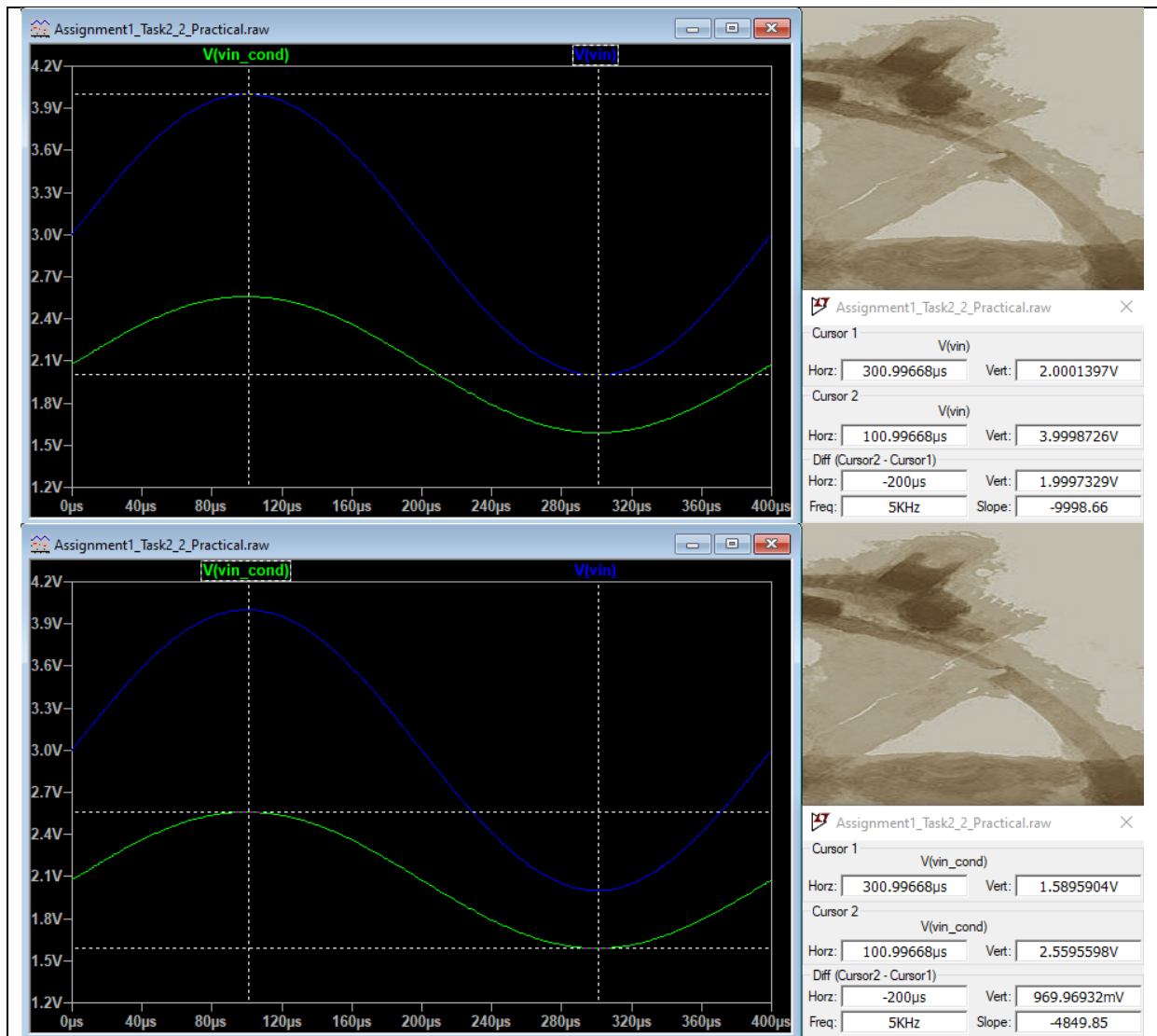**Paste the input and output sinusoids of the ISC below.**

Use the cursor tools to confirm that the input waveform is correct, and that the output waveform matches the description written in task 3.1.1
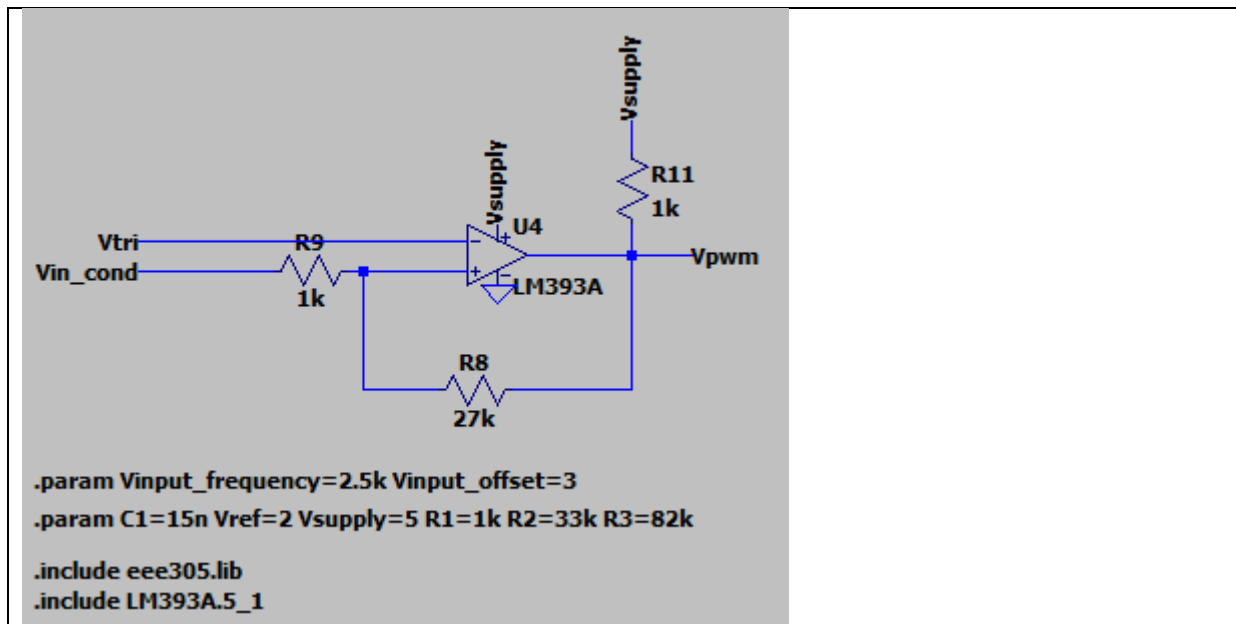
PWM Generator

The PWM generator takes the input from the ISC and the TWG and modulates the two signals together. For this task, design a PWM comparator circuit

3.1. PWM generator schematic

3.1.1. Design a PWM generator circuit in LTSpice and Paste it here

.param Vinput_frequency=2.5k Vinput_offset=3
.param C1=15n Vref=2 Vsupply=5 R1=1k R2=33k R3=82k

.include eee305.lib
.include LM393A.5_1

## 3.2. PWM generator simulation

In this task, the TWG, the ISC, and the PWM generator will be connected to modulate the input sinusoid into a PWM signal that represents it.

Connect the practical TWG, the practical ISC, and the PWM generator together in LTSpice

3.2.1. **Set the input sinusoid into the system with an offset of 3V, an 1 V amplitude and a frequency of 2.5 kHz. Run the full simulation and Paste the following waveforms into the box below.**

- **Input Triangle Wave from the TWG**
- **Input sinusoid from the ISC**
- **Output of the PWM**

Set up the time-scale of the simulation to show one cycle of the input 2.5 kHz signal
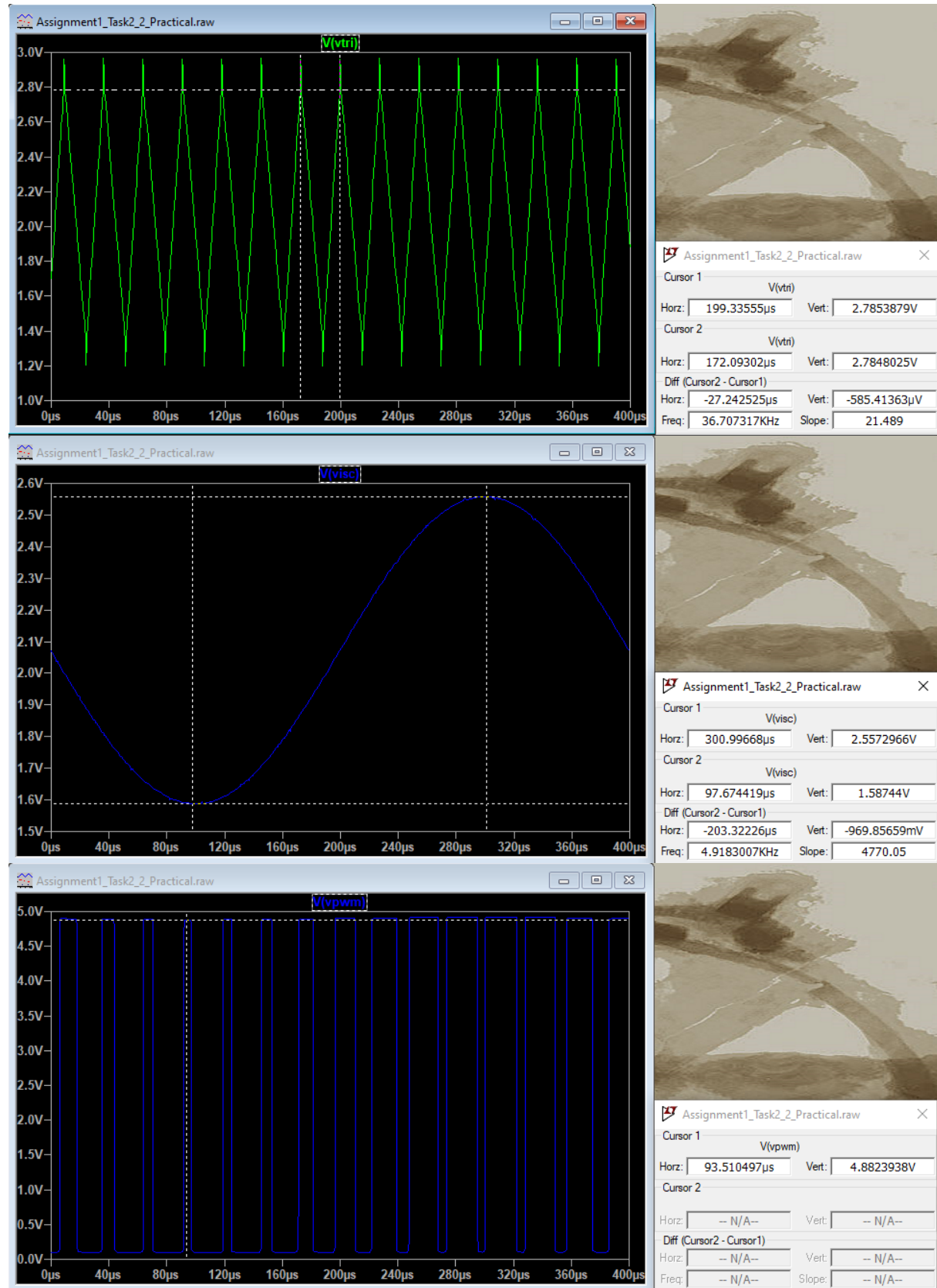
3.2.2. Optional task: Observe the salient features of the PWM signal. Discuss how the duty cycle of the PWM match relate to the triangle wave and sinusoidal speed inputs.

4. LED Amplifier

An amplifier is required to drive the LED which is used to send the encoded message to the receiver. For this project, a Darlington pair amplifier is suggested. A basic structure of this amplifier is shown in Figure 2.
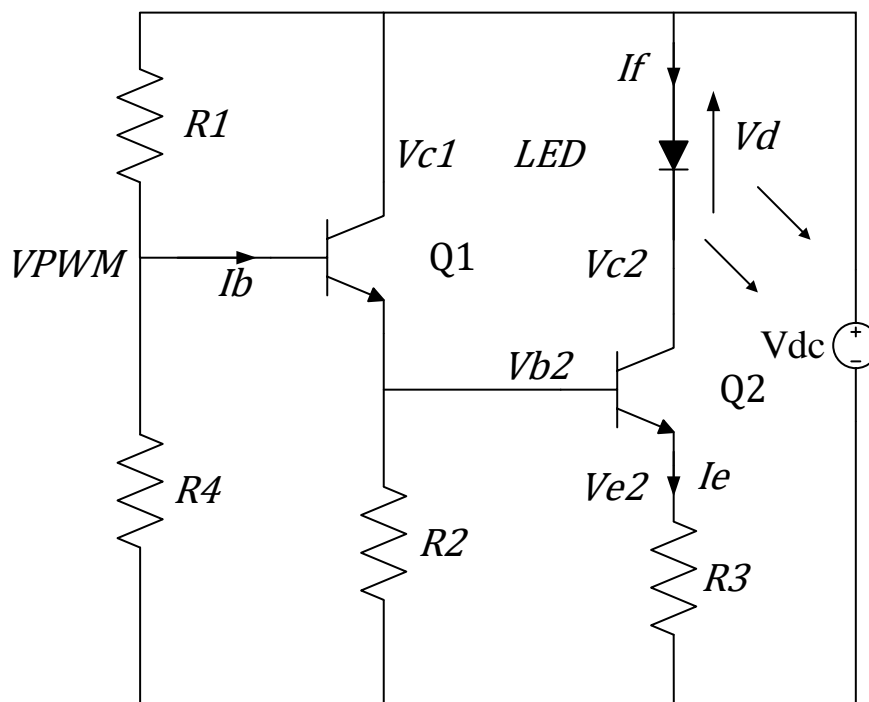

Figure 2: A standard structure of an ideal Darlington pair amplifier

4.1. Component calculations

Each resistor must be calculated to set i_f to the desired value stated in Table 1. Make sure that these resistors are E12 values.

4.1.1.   For each resistor R1 to R4, fill in the values chosen.

R1:
R2:
R3:

| R4: | |

4.1.2. Using the resistor values chosen in task 4.1.1, calculate the values of V_{PWM} and i_f when the signals are high

Calculated $V_{PWM,high}$	
Calculated $i_{f,high}$	

4.2. Amplifier simulation

Using LTspice, design the Darlington pair amplifier on the same simulation as the TWG, the ISC, and the PWM generator. Run the full transmitter simulation using the values decided on in Task

4.2.1. Set the input sinusoid to a 3V offset waveform with 0.5V amplitude. Set the frequency to be 1 kHz.

Run the simulation, Paste a single cycle (of the 1 kHz input) of the simulated i_f into the box below. Measure the required parameters using the LTspice measurement tools

Simulated $i_{f,high}$	
Simulated $V_{PWM,high}$	

4.2.2. Optional task: Discuss any differences (or lack of differences) between the calculated and simulated values for $i_{f,high}$ and $V_{PWM,high}$.

4.3. Practical Considerations

When designing an amplifier, the power loss in certain components need to be considered. This ensures that the system can run for a long time reliably.

4.3.1. **Using the measurement tools in LTSpice, measure the power dissipation of the transistors and resistors of the Darlington pair amplifier shown in Figure 2**

Part	Power Dissipation (mW)
Q1	
Q2	
R1	
R2	
R3	
R4	

4.3.2. **Briefly comment on whether any of the components in task 4.3.1 needs special attention when built in practice. Include details on components need to be modified and how the changes will be implemented.**

5. Voltage Regulation

Each group must decide on a practical means of meeting client requirements of being able to operate between 9 – 16V.

5.1. Vsupply setting

In this task, each group must decide on a practical method of generating the V_{supply} rail as chosen in Table 1

5.1.1. **Choose an appropriate voltage regulator for you application using the ECSE store stock list**

Chosen Regulator	
Vin range of chosen regulator	
Vout of chosen regulator	
Output current limits of chosen regulator	

5.1.2. **Give a brief description (1 to 2 sentences) of why this regulator was chosen over other parts**

5.1.3. Optional task: The voltage regulator is usually not simulated in LTSpice as an ideal voltage source can emulate its performance. However, in practice a voltage regulator must be physically populated.

Read the datasheet for the voltage regulator chosen and drawn a schematic showing the circuitry needed to implement a practical voltage regulator on a breadboard or PCB.

5.2. Vref setting

In this task, each group must decide on a practical method of generating V_{ref} as decided on in Table 1.

5.2.1. Paste a practical circuit to generate your V_{ref} voltage