

ECE 438 - Laboratory 3

Frequency Analysis

Last updated on January 23, 2022

```
In [1]: import numpy as np
import matplotlib.pyplot as plt

In [2]: # make sure the plot is displayed in this notebook
%matplotlib inline
# specify the size of the plot
plt.rcParams['figure.figsize'] = (16, 6)

# for auto-reloading extenrnal modules
%load_ext autoreload
%autoreload 2
```

1. Introduction

In this experiment, we will use Fourier series and Fourier transforms to analyze continuous-time and descrete-time signals and systems. The Fourier representations of signals involve the decomposition of the signal in terms of complex exponential functions. These decompositions are very important in the analysis of linear time-invariant (LTI) systems, due to the property that the response of an LTI system to a complex exponential input is a complex exponential of the same frequency! Only the amplitude and phase of the input signal are changed. Therefore, studying the frequency response of an LTI system gives complete insight into its behavior.

In this experiment and others to follow, we will use the Simulink extension to Matlab. Simulink is an icon-driven dynamic simulation package that allows the user to represent a system or a process by a block diagram. Once the representation is completed, Simulink may be used to digitally simulate the behavior of the continuous or discrete-time system. Simulink inputs can be Matlab variables from the workspace, or waveforms or sequences generated by Simulink itself. These Simulink-generated inputs can represent continuous- time or discrete-time sources. The behavior of the simulated system can be monitored using Simulink’s version of common lab instruments, such as scopes, spectrum analyzers and network analyzers.

2. Background Exercises

2.1 Synthesis of Periodic Signals

Each signal given below represents one periodic signal with period T_0 .

- Period $T_0 = 2$. For $t \in [0, 2]$:

$$s(t) = \text{rect}\left(t - \frac{1}{2}\right) \tag{1}$$

- Period $T_0 = 1$. For $t \in \left[-\frac{1}{2}, \frac{1}{2}\right]$:

$$s(t) = \text{rect}(2t) - \frac{1}{2} \tag{2}$$

Exercise 2.1

1. For each of these two signals, do the following on a blank sheet of paper (or type the equations in the Markdown cell if you are familiar with LaTeX):

- Compute the Fourier series expansion in the form

$$s(t) = a_0 + \sum_{k=1}^{\infty} A_k \sin(2\pi k f_0 t + \theta_k)$$

where $f_0 = \frac{1}{T_0}$.

Hint :You may want to use one of the following references:

Sec. 4.1 of “Digital Signal Processing”, by Proakis and Manolakis, 1996;

Sec. 4.2 of “Signals and Systems”, by A. Oppenheim and A. Willsky, 1983;

Sec. 3.3 of “Signals and Systems”, A. Oppenheim and A. Willsky, 1997.

Note that in the expression above, the function in the summation is $\sin(2\pi k f_0 t + \theta_k)$, rather than a complex sinusoid. The formulas in the above references must be modified to accommodate this. You can compute the cos/sin version of the Fourier series, then convert the coefficients.

write your answer here

2. Write code to approximate the two signals using the Fourier series expansion above. Use 200 (instead of infinite number of) Sine waves. Then, plot these two signals.

```
In [3]: # write your code here
```

3. Getting Started with Simulink

In this section, we will learn the basics of Simulink and build a simple system.

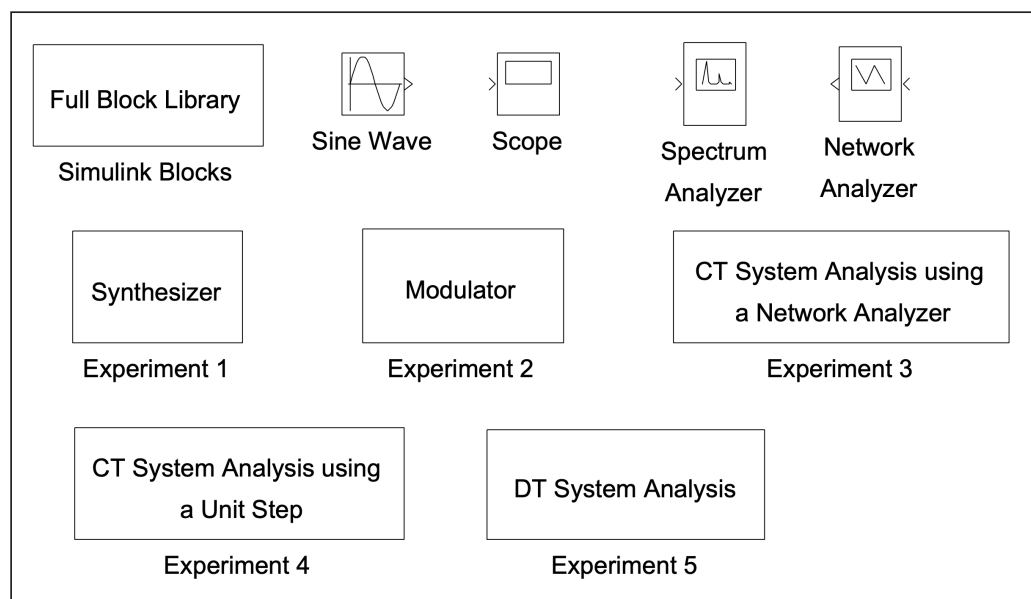


Figure 1: Simulink utilities for lab 3.

The library of Simulink functions for this laboratory is included in the folder `Lab3Utilities`. To get help on Simulink, open the file `simulink_help.pdf`. Once Matlab is started, change current directory to `Lab3Utilities` and then type `Lab3` to bring up the library of Simulink components shown in Fig. 1. This library contains a full library of Simulink blocks, a spectrum analyzer and network analyzer designed for this laboratory, a sine wave generator, a scope, and pre-design systems for each of the experiments that you will be running.

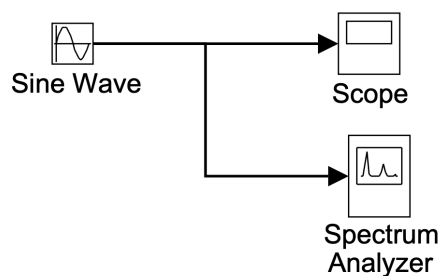


Figure 2: Simulink model for the introductory example.

In order to familiarize yourself with Simulink, you will first build the system shown in Fig. 2. This system consists of a sine wave generator that feeds a scope and a spectrum analyzer.

1. Open a window for a new system by using the **New** option from the **File** pull-down menu, and select **Model**.
2. Drag the **Sine Wave**, **Scope**, and **Spectrum Analyzer** blocks from the **Lab3** window into the new window you created.
3. Now you need to connect these three blocks. With the left mouse button, click on the output of the **Sine Wave** and drag it to the input of the **Scope**. Now use the right button to click on the line you just created, and drag to the input of the **Spectrum Analyzer** block. Your system should now look like Fig. 2.
4. Double click on the **Scope** block to make the plotting window for the scope appear.
5. Set the simulation parameters by selecting **Configuration Parameters** from the **Simulation** pull-down menu. Under the **Solver** tab, set the **Stop time** to **50**, and the **Max step size** to **0.02**. Then select **OK**. This will allow the **Spectrum Analyzer** to make a more accurate calculation.
6. Start the simulation by using the **Start** option from the **Simulation** pull-down menu. A standard Matlab figure window will pop up showing the output of the **Spectrum Analyzer**.
7. Change the frequency of the sine wave to 7π rad/sec by double clicking on the **Sine Wave** icon and changing the number in the **Frequency** field. Restart the simulation. Observe the change in the waveform and its spectral density. If you want to change the time scaling in the plot generated by the spectrum analyzer, from the Matlab prompt use the `subplot(2,1,1)` and `axis()` commands.
8. When you are done, close the system window you created by using the **Close** option from the **File** pull-down menu.

4. Continuous-Time Frequency Analysis

In this section, we will study the use and properties of the continuous-time Fourier transform with Simulink. The Simulink package is especially useful for continuous-time systems because it allows the simulation of their behavior on a digital computer.

4.1 Synthesis of Periodic Signals

Double click the icon labeled **Synthesizer** to bring up a model as shown in Fig. 3. This system may be used to synthesize periodic signals by adding together the harmonic components of a Fourier series expansion. Each **Sin Wave** block can be set to a specific frequency, amplitude and phase. The initial settings of the **Sin Wave** blocks are set to generate the Fourier series expansion

$$x(t) = 0 + \sum_{\substack{k=1 \\ k \text{ odd}}}^{13} \frac{4}{k\pi} \sin(2\pi kt)$$

These are the first 8 terms in the Fourier series of the periodic square wave shown in Fig. 4.

Run the model by selecting **Start** under the **Simulation** menu. A graph will pop up that shows the synthesized square wave signal and its spectrum. This is the output of the **Spectrum Analyzer**.

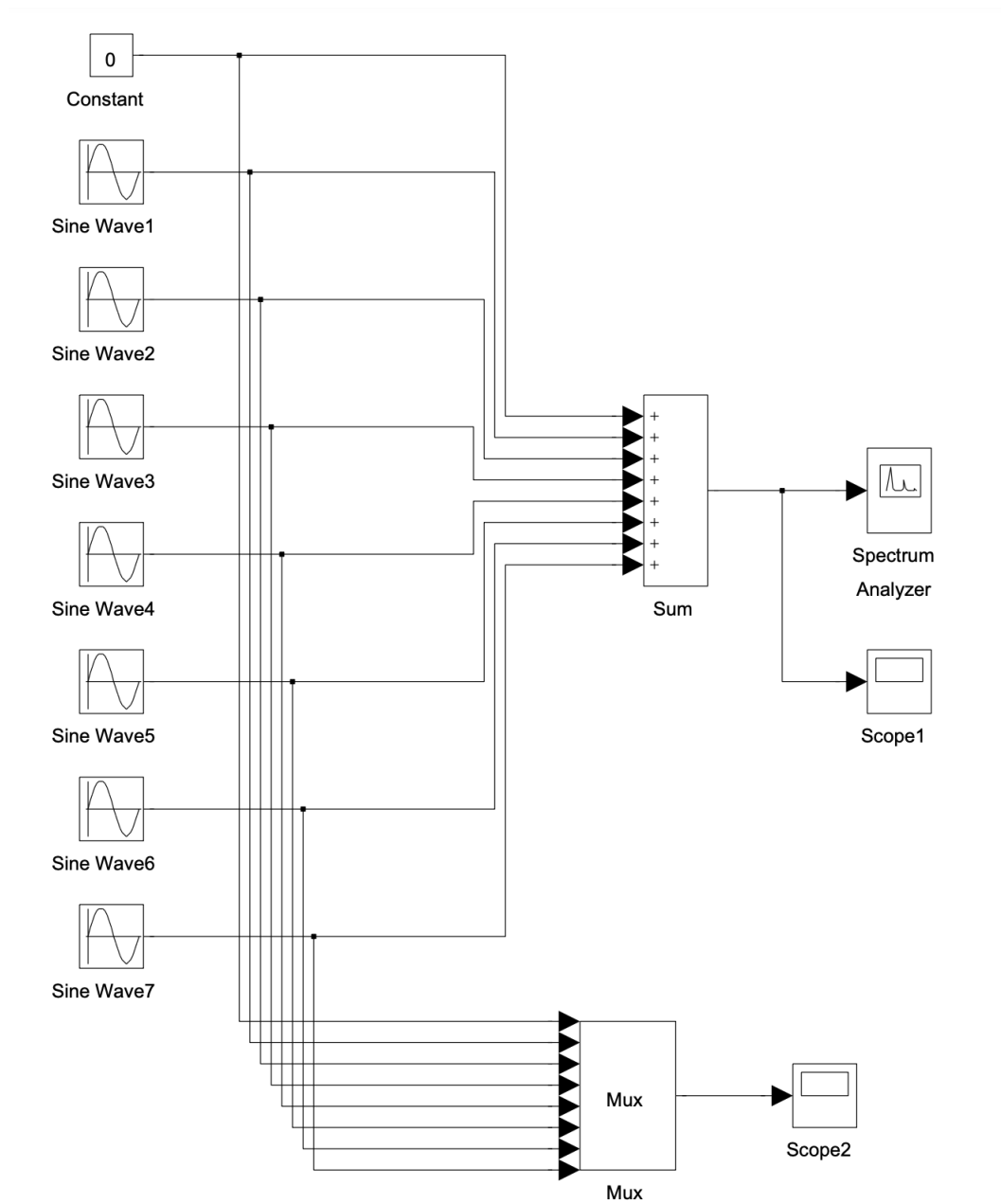


Figure 3: Simulink model for the synthesizer experiment.

After the simulation runs for a while, the **Spectrum Analyzer** element will update the plot of the spectral energy and the incoming waveform. Notice that the energy is concentrated in peaks corresponding to the individual sine waves. Print the output of the **Spectrum Analyzer**. You may have a closer look at the synthesized signal by double clicking on the **Scope1** icon. You can also see a plot of all the individual sine waves by double clicking on the **Scope2** icon. Synthesize the two periodic waveforms listed in [Section 2.1](#) of the background exercises.

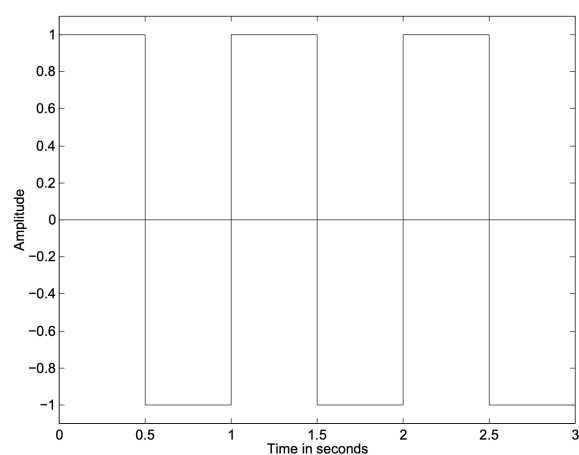


Figure 4: The desired waveform for the synthesizer experiment.

Do this by setting the frequency, amplitude, and phase of each sinewave generator to the proper values. For each case, print the output of the **Spectrum Analyzer**.

Exercise 4.1

1. Hand in plots of the Spectrum Analyzer output for each of the three synthesized waveforms.

insert your plots here

2. For each case in Q1, comment on how the synthesized waveform differs from the desired signal, and on the structure of the spectral density.

insert your answer here

4.2 Modulation Property

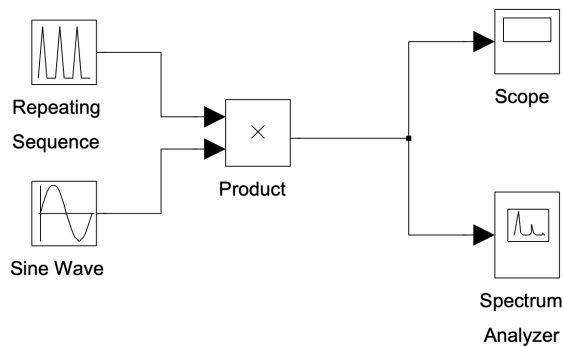


Figure 5: Simulink model for the modulation experiment.

Double click the icon labeled **Modulator** to bring up a system as shown in Fig. 5. This system modulates a triangular pulse signal with a sine wave. You can control the duration and duty cycle of the triangular envelope and the frequency of the modulating sine wave. The system also contains a spectrum analyzer which plots the modulated signal and its spectrum.

Generate the following signals by adjusting the **Time values** and **Output values** of the **Repeating Sequence** block and the **Frequency** of the **Sine Wave**. The **Time values** vector contains entries spanning one period of the repeating signal. The **Output values** vector contains the values of the repeating signal at the times specified in the **Time values** vector. Note that the **Repeating Sequence** block does **NOT** create a discrete time signal. It creates a continuous time signal by connecting the output values with line segments. Print the output of the **Spectrum Analyzer** for each signal.

1. Triangular pulse duration of 1 sec; period of 2 sec; modulating frequency of 10 Hz (initial settings of the experiment).
2. Triangular pulse duration of 1 sec; period of 2 sec; modulating frequency of 15 Hz.
3. Triangular pulse duration of 1 sec; period of 3 sec; modulating frequency of 10 Hz.
4. Triangular pulse duration of 1 sec; period of 6 sec; modulating frequency of 10 Hz.

Notice that the spectrum of the modulated signal consists of a comb of impulses in the frequency domain, arranged around a center frequency.

Exercise 4.2

1. Hand in plots of the output of the **Spectrum Analyzer** for each signal.

insert your plots here

2. What effect does changing the modulating frequency have on the spectral density?

write your answer here

3. Why does the spectrum have a comb structure and what is the spectral distance between impulses? Why?

write your answer here

4. What would happen to the spectral density if the period of the triangle pulse were to increase toward infinity? (in the limit)

write your answer here

4.3 System Analysis

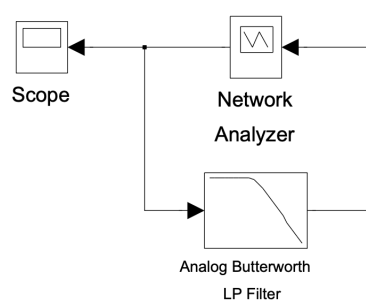


Figure 6: Simulink model for the continuous-time system analysis experiment using a network analyzer.

Double click the icon labeled **CT System Analysis** using a **Network Analyzer** to bring up a system as shown in Fig. 6. This system includes a **Network Analyzer** model for measuring the frequency response of a system. The **Network Analyzer** works by generating a weighted chirp signal (shown on the **Scope**) as an input to the system-under-test. The analyzer measures the frequency response of the input and output of the system and computes the transfer function. By computing the inverse Fourier transform, it then computes the impulse response of the system. Use this setup to compute the frequency and impulse response of the given fourth-order Butterworth filter with a cut-off frequency of 1Hz. Print the figure showing the magnitude response, the phase response and the impulse response of the system. To use the tall mode to obtain a larger printout, type `orient('tall')` directly before your print.

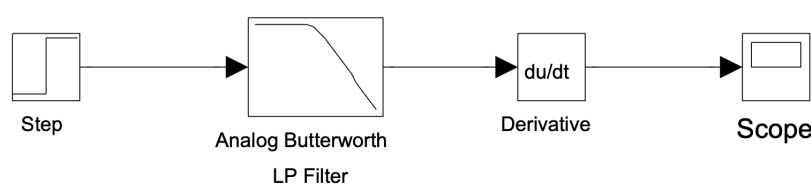


Figure 7: Simulink model for the continuous-time system analysis experiment using a unit step.

An alternative method for computing the impulse response is to input a step function into the system and then to compute the derivative of the output. The model for doing this is given in the **CT System Analysis** using a **Unit Step** block. Double click on this icon and compute the impulse response of the filter using this setup (Fig. 7). Make sure that the characteristics of the filter are the same as in the previous setup. After running the simulation, print the graph of the impulse response from the scope.

Exercise 4.3

1. Hand in the printout of the output of the Network Analyzer (magnitude and phase of the frequency response, and the impulse response).

insert your printout here

2. Hand in the plot of the impulse response obtained using a unit step.

insert your plot here

3. What are the advantages and disadvantages of each method?

write your answer here

5. Discrete-Time Frequency Analysis

In this section of the laboratory, we will study the use of the discrete-time Fourier transform.

5.1 Discrete-Time Fourier Transform

The DTFT (Discrete-Time Fourier Transform) is the Fourier representation used for finite energy discrete-time signals. For a discrete-time signal, $x(n)$, we denote the DTFT as the function $X(\omega)$ given by the expression

$$X(\omega) = \sum_{n=-\infty}^{\infty} x[n]e^{-j\omega n}$$

Since $X(\omega)$ is a periodic function of ω with a period of 2π , we need only to compute $X(\omega)$ for $-\pi < \omega < \pi$.

Exercise 5.1

1. Complete the follwing function that computes the DTFT of a discrete-time signal.

```
def DTFT(x,n0,w):
    """
    This function computes the DTFT of a discrete-time signal.

    Parameters
    ---
    x: the discrete-time signal
    n0: time index corresponding to the 1st element of the x vector
    w: frequencies

    Returns
    ---
    X: the computed DTFT
    """
    pass
```

Note that if \mathbf{x} is a vector of length N , then its DTFT is computed by

$$X(\omega) = \sum_{n=0}^{N-1} x[n]e^{-j\omega(n+n_0)}$$

where w is a vector that contains the frequencies from $-\pi$ to π .

Hint: In Python, $1j$ is defined as $\sqrt{-1}$. Use `np.exp(x)` to calculate e^x .

In [4]: # write your code here

2. For the following signals

- $x[n] = \delta[n]$
- $x[n] = \delta[n - 5]$
- $x[n] = (0.5)^n u[n]$

use your DTFT function to compute $X(\omega)$, and plots its magnitude and phase.

Hint: Use `np.power(a,b)` to calculate a^b . Use `np.abs()` and `np.angle()` to compute the magnitude and phase.

In [5]: # write your code here

Exercise 5.2: Magnitude and Phase of the Frequency Response of a Discrete-Time Systems

Consider the discrete-time system described by the following difference equation:

$$y[n] = 0.9y[n - 1] + 0.3x[n] + 0.24x[n - 1]$$

Assume that the system is **causal**.

1. Draw a system diagram.

insert your system diagram here

2. Obtain the impulse response of the system by replacing $x[n]$ with $\delta[n]$ in the above equation. (Use causality to set up the initial conditions.)

insert your answer here

3. Use your answer in Q2 to obtain the frequency response of the system.

insert your answer here

4. Find the frequency response of the system using another method. Specifically, take the DTFT of the left-hand-side and right-hand-side of the difference equation, and then use linearity and the time-shifting property of the DTFT along with the fact that $H(\omega) = \frac{Y(\omega)}{X(\omega)}$

insert your answer here

5. Write Python code to compute and plot the magnitude and phase responses, $|H(\omega)|$ and $\angle H(\omega)$, for $-\pi < \omega < \pi$.

```
In [6]: # write your code here
```

5.3 System Analysis

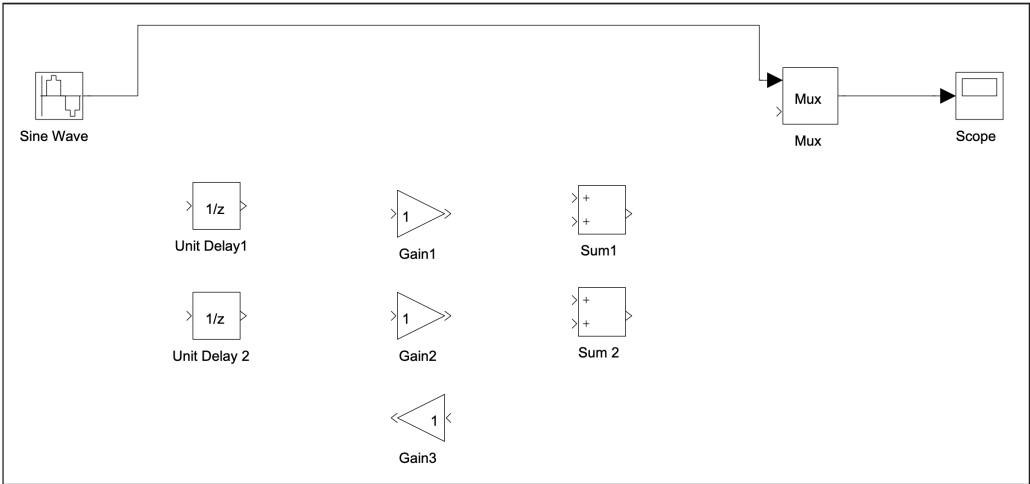


Figure 8: Incomplete Simulink setup for the discrete-time system analysis experiment.

Double click the icon labeled **DT System Analysis** to bring up an incomplete block diagram as shown in Fig. 8. It is for a model that takes a discrete-time sine signal, processes it according to a difference equation and plots the multiplexed input and output signals in a graph window. Complete this block diagram such that it implements the following difference equation given in [Section 5.2](#).

$$y(n) = 0.9y[n - 1] + 0.3x[n] + 0.24x[n - 1]$$

You are provided with the framework of the setup and the building blocks that you will need. You can change the values of the **Gain** blocks by double clicking on them. After you complete the setup, adjust the frequency of **Sine Wave** to the following frequencies: $\omega = \pi/16$, $\omega = \pi/8$, $\omega = \pi/4$. For each frequency, make magnitude response measurements using the input and output sequences shown in the graph window. Compare your measurements with the values of the magnitude response $|H(e^{j\omega})|$ which you computed in the background exercises at these frequencies.

An alternative way of finding the frequency response is taking the DTFT of the impulse response. Use your DTFT function to find the frequency response of this system from its impulse response. The impulse response was calculated in [Section 5.2](#) of the background exercises. Plot the impulse response, and the magnitude and phase of the frequency response.

Exercise 5.3

1. Insert the printout of your completed block diagram.

insert your diagram here

2. Enter both the amplitude measurements you made and their theoretical values.

ω	Measurements	Theoretical Values
$\pi/16$		
$\pi/8$		
$\pi/4$		

3. Plot the impulse response, and the magnitude and phase of the frequency response by using your DTFT function.

```
In [7]: # write your code here
```

