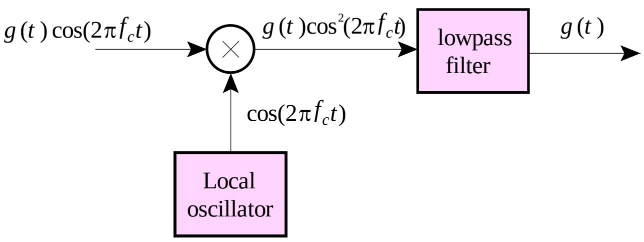
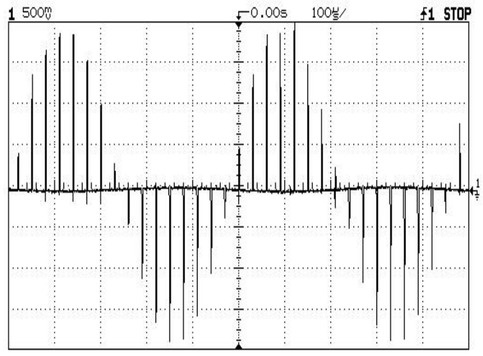
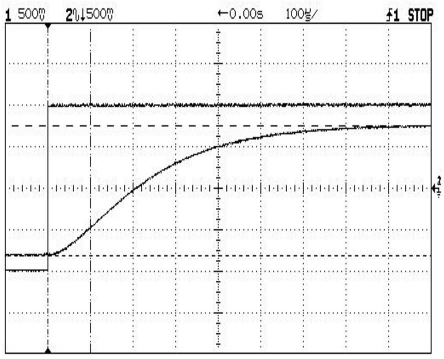
48540
Signals and Systems

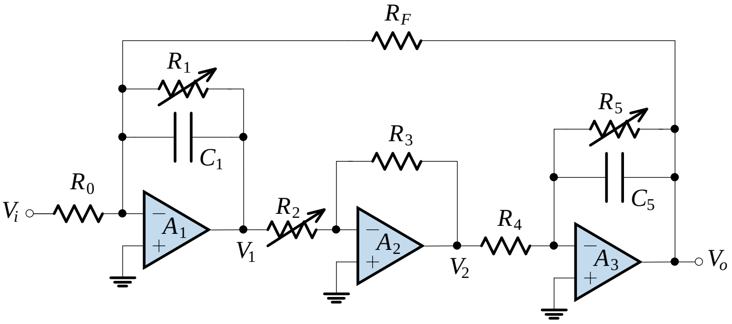
Lab Notes

Spring 2015









PMcL

University of Technology, Sydney
Faculty of Engineering and Information Technology

Subject:	**48540 Signals and Systems**
Assessment Number:	**1**
Assessment Title:	**Lab 1 – DSO Measurements**
Tutorial Group:	

Students Name(s) and Number(s)

Student Number	Family Name	First Name

Declaration of Originality:

The work contained in this assignment, other than that specifically attributed to another source, is that of the author(s). It is recognised that, should this declaration be found to be false, disciplinary action could be taken and the assignments of all students involved will be given zero marks. In the statement below, I have indicated the extent to which I have collaborated with other students, whom I have named.

Statement of Collaboration:

Signature(s)

Marks

Practical Exam:	/2
Questions:	/2
TOTAL	/4

Office use only ☺

key

Assessment Submission Receipt

Assessment Title:	**Lab 1 – DSO Measurements**
Student's Name:	
Date Submitted:	
Tutor Signature:	

Lab 1 – DSO Measurements

Vertical setup. Horizontal setup. Trigger setup. Run control. Automatic time measurements. Automatic voltage measurements. Cursor measurements. Reducing random noise on a signal. Frequency-domain measurement. Aliasing. Windowing.

Introduction

The digital storage oscilloscope (DSO) is a versatile tool for the engineer. It has the ability to sample and store voltage waveforms, giving it the ability to "capture" transient waveforms and also the ability to perform mathematical operations on the sample values. One very important operation is known as the Fast Fourier Transform (FFT) which gives the DSO the ability to display the spectral content of a waveform. Like any tool though, it has its limitations, and careful operation is required to interpret results correctly.

Objectives

1. To become familiar with setting up a DSO.
2. To become familiar with basic time and voltage measurement techniques using a DSO.
3. To become familiar with the FFT and aliasing when using a DSO.

Equipment

- 1 Digital Storage Oscilloscope (DSO) – Agilent DSO-X 2004A with Wave Gen
- 1 Arbitrary Waveform Generator (AWG) – Agilent 33210A with Option 002
- 1 TIMS trainer with 1 adder, 1 headphone amplifier or tunable LPF
- 4mm leads (assorted colours), 2 BNC to 4mm leads, 1 BNC to 4mm adaptor

Safety

This is a Category A laboratory experiment. Please adhere to the Category A safety guidelines (issued separately). Cat. A lab

L1.2

Basic Setup

Refer to the Lab Equipment Guide

You will be asked to perform various and wide-ranging tasks with the DSO, so it is important that you have the Lab Equipment Guide (LEG) as a reference.

Function Generator Setup

1. Refer to the LEG and set the arbitrary waveform generator (AWG) up for a sinusoid of 2 kHz, with an amplitude to 5 Vp-p and 0 V DC offset.

2. Ensure the DSO has been set to its default setup configuration.

3. Connect the AWG output to Channel 1 of the DSO.

Vertical Setup

1. If necessary, set the Channel 1 position to 0.0 V with the vertical `Position` knob.

2. Press `1` and select each softkey option within the vertical setup menu and notice that each change affects the status line and information area differently. Adjust the vertical scale (volts/div) to 1 V/div.

Horizontal Setup

1. Turn the Horizontal knob and notice the change it makes to the status line.

2. Press `Horiz`. Toggle the `Time Ref` softkey to see the effect of moving the trigger point ▼. Return the time reference to the centre.

3. Change the `Time Mode` to see the effect. Restore the `Time Mode` to `Normal` and set the horizontal scale to 100 μs/div.

4. Turn the horizontal delay knob to see the effect. Push the horizontal delay knob to reset the delay to 0.00 s.

Trigger Setup

1. Turn the trigger Level knob and notice the changes it makes to the display.
2. Press `Trigger` then use the `Source` softkey to scroll through each of the options and notice that each change is shown in the status line.
3. Press `Mode/Coupling`. Change the `Mode` to see the effect on the status line.
4. Leave the mode on `Normal`. Adjust the trigger level so that it is above the positive peak of the displayed sine wave and notice the change in the status line. Return the trigger mode to `Auto` and notice the effect. Move the trigger level back to 0.0 V.
5. Change the `Coupling` to `LF Reject` and observe the display and status line. Return the `Coupling` to `DC`.
6. Change the AWG frequency to 2 Hz. Adjust the time base to 100 ms/div. Press `Horiz` and change the `Time Mode` to `Roll`.
7. Change the AWG wave shape to square, then ramp, then back to sinusoid.
8. Press `Run/Stop` to freeze the display. Press `Run/Stop` again to resume acquisition and refresh the display.
9. Set the AWG to a 20 kHz sinusoid. Try to adjust the timebase to display two cycles – note that the timebase setting is restricted in roll mode.
10. Set the `Time Mode` to `Normal` and adjust the timebase to 10 μs/div.

Run Control

1. Press `Single`. Note that the `Run/Stop` button is illuminated in red and the trigger indicator in the status line shows “Stop”.
2. Set the AWG for ramp. Does the displayed waveform change?
3. Disconnect the input from DSO channel 1. Does the displayed waveform change?
4. Reconnect the input to DSO channel 1. Press `Single`. Does the displayed waveform change?
5. Press `Run/Stop` to resume continuous acquisition.

Time-domain Measurement

Automatic Time Measurements

1. Press `Meas`. Notice that measurements for frequency and peak to peak are automatically displayed for channel 1.
2. Change the AWG wave shape to square. Adjust the duty cycle to the minimum available.
3. On the DSO, press the `Type:` softkey. Select `Duty Cycle` then press `Add Measurement`. Note the *measured* duty cycle.
4. Adjust the duty cycle in steps across its full range using the AWG knob. Note the maximum duty cycle available.
5. Use the AWG keypad to return the duty cycle to 50%. Set the wave shape to sinusoid. Set the frequency to 2 kHz.
6. Set the DSO timebase to 100 μs/div.
7. Remove the Phase Shifter module from the TIMS trainer and set the “Frequency Range” switch to “LO”. Carefully replace the Phase Shifter module into the TIMS unit.

8. Turn the TIMS trainer on. Connect the earth of the AWG lead to the green GND terminal on the TIMS trainer. You may leave the earth of the DSO lead floating (since it is connected to earth anyway).

The TIMS is turned on at the back – when on, you should see the LEDs in the counter section light up

9. Connect the AWG output to the Phase Shifter unit's "IN" terminal. Measure the Phase Shifter input and output on DSO Channels 1 and 2 respectively. On the Phase Shifter unit, turn the coarse and fine knobs fully anticlockwise, and turn the ±180° switch on.

10. Press `Meas`. Press `Type` and then select `Phase` measurement. Press `Settings` and then set `Source 1` to channel 2 and `Source 2` to channel 1. Press (Back) then `Add Measurement`. Measure the phase difference between the two waveforms. Determine which channel (Source) is used as the reference by the DSO for the phase measurement. Vary the Phase Shifter Coarse knob to see the effect. Turn off Channel 2.

Automatic Voltage Measurements

1. Press `Meas`. Press `Source` and select "Channel 1".

Be careful when using the automatic voltage measurements – the DSO can't differentiate between a noise peak and a signal peak

2. Press `Type` and select "Peak-Peak". Press `Add Measurement`. Change the measurement `Type` to "Average N cycles" and add this measurement for channel 1. Change the measurement `Type` to "DC RMS N cycles" and add this measurement for channel 1. You should now have three measurements for channel 1 voltage.

3. Note each of the three measurements for a sinusoidal signal then change the AWG waveform to square, then to ramp, and observe the change in the measurements.

4. Set the AWG to a sinusoidal wave, and vary the DC offset up and down in 1 V steps using the AWG knob. Note the effect on the Vp-p, Vavg and Vrms values. Return the DC offset to 0.00 V.

Cursor Measurements

The cursor keys are useful for making custom time or voltage measurements on a signal.

The RC LPF is located in the Utilities module

1. Connect a 2 V p-p, 200 Hz square wave from the AWG to the TIMS trainer RC LPF input in the Utilities module. The RC LPF is a simple first-order *RC* circuit that acts as a “lowpass filter”. Measure the AWG output with Channel 1. Measure the RC LPF output with Channel 2.

2. Press `Horiz`. Set the `Time Ref` softkey to `Left`. Set the timebase to 50 μs/div. Set the vertical scale to 500 mV/div for each channel.

3. Press `Cursors`. `Source` selects a channel for the voltage (vertical) cursor measurements. `Cursors` selects the cursor(s) to be positioned by turning the cursors knob. To erase the cursor readings and remove the cursors from the display press the `Cursors` button again.

4. Press `Cursors`. Change the cursors’ source to Channel 2 by pressing the `Source` softkey.

5. Press the `Cursors` softkey and select `Y1` as the active cursor. Move the Y1 cursor to align with the bottom of the output response by rotating the cursors knob.

6. Press the `Cursors` softkey and select `Y2` as the active cursor. Move the Y2 cursor to align with the top (steady-state value) of the output response. Check that the cursor measurement displays $\Delta Y(2) \approx 2.000\,\mathrm{V}$.

7. Press the `Units` softkey. Press `Y Units` and select `Ratio (%)`. The cursors will now measure in percent but this measurement needs a reference. Press `Use Y Cursors As 100%` to set the current Y cursor positions as the reference values. Y1 will now show 0%, Y2 100% and $\Delta Y(2)$ 100%.

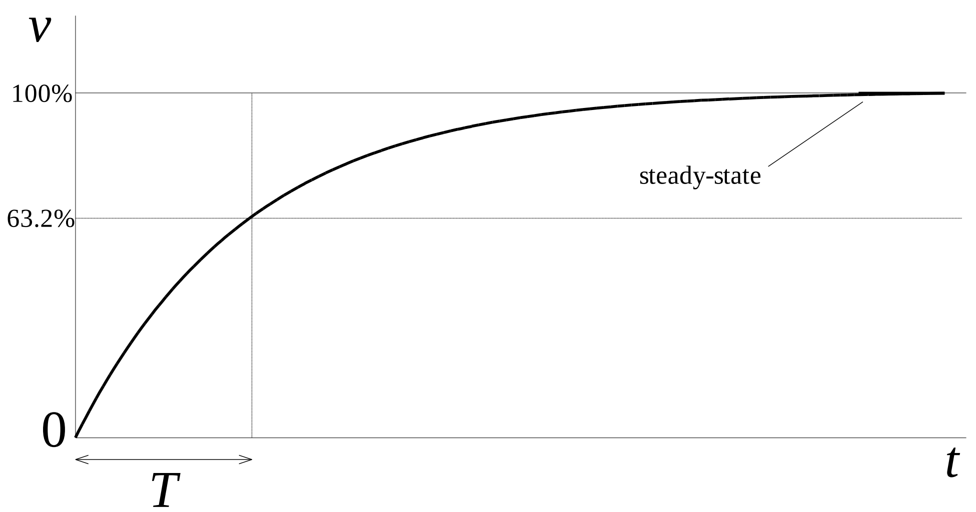
8. Move Y2 to read as close as possible to $\Delta Y(2)$ = 63.2%.

9. Press 

. Press the `Cursors` softkey then select X1 as the active cursor. Position X1 to align with the vertical edge of the input square wave on channel 1. (**The timescale is the same for all channels!**)

10. Select X2 as the active cursor and position it to align with the intersection of the Y2 cursor and the channel 2 waveform.

11. Record the following measurement:



A step-response for a first-order system

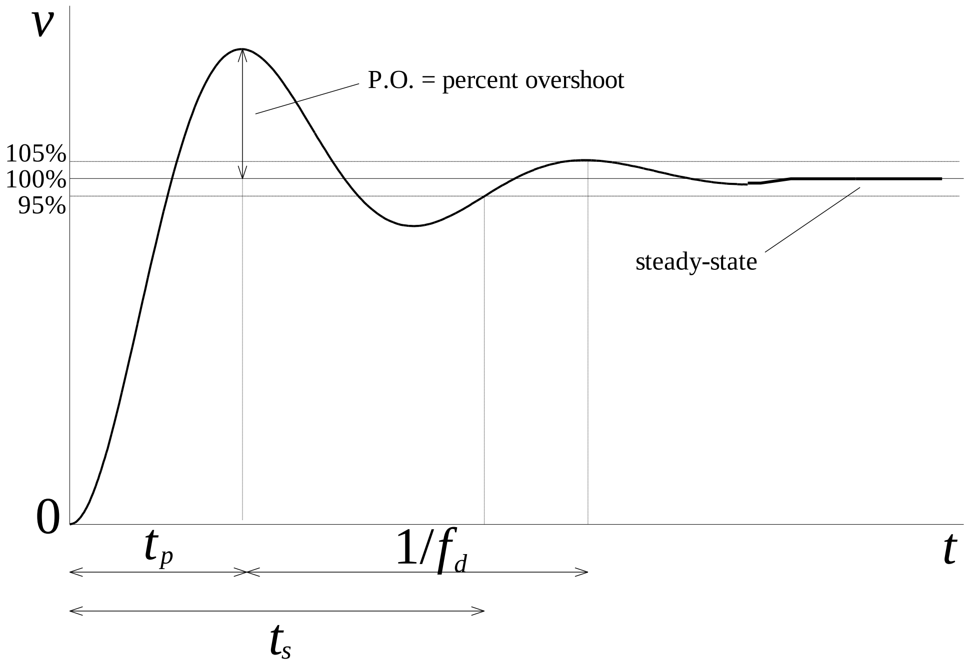
$T =$

12. Connect the 200 Hz square wave from the AWG to the TIMS trainer Headphone Amplifier input “A”. Measure the AWG output with Channel 1. Measure the Headphone Amplifier “LPF” output with Channel 2.

13. Set the timebase to 200 μs/div.

14. Use the cursors to record the following quantities:

A typical step-response for an underdamped second-order system



$t_p =$	P.O.=
$t_s =$	$f_d =$

15. Turn off the cursors. Turn Channel 2 off.

16. Disconnect the DSO and AWG leads from the TIMS unit.

Reducing Random Noise on a Signal

If the signal you are applying to the DSO is noisy, you can set up the DSO to reduce the noise on the waveform. There are two methods to reduce noise – bandwidth limiting and averaging.

Bandwidth Limiting

This method applies the incoming signal to a lowpass filter before it is sampled by the DSO. This method works only when the noise has very high frequency content. The bandwidth limiter "cuts off" frequencies above 20 MHz.

Bandwidth limiting will only help if the signal frequency is less than about 1 MHz.

1. Change the AWG waveform to a sinusoid. Set the **amplitude** to 20 mVp-p and the frequency to 20 Hz.

2. Connect DSO channel 1 to the AWG output.

3. Set the vertical scale to 20 mV/div and the horizontal scale to 10 ms/div. If necessary, press `Mode/Coupling` and select `HF Reject` for trigger coupling. You should see a noisy sinusoid.

4. Press `1`. Press the `BW Limit` softkey. The noise should be reduced.

5. Turn bandwidth limiting off by pressing the `BW Limit` softkey again.

Averaging

The second method of reducing noise works when noise is present below the cutoff frequency of the bandwidth limit filter. First, you stabilize the displayed waveform by removing the noise from the trigger path. Second, you reduce the noise on the displayed waveform by averaging the samples.

Averaging can only be used to clean up a signal if the noise is "uncorrelated"

1. Press `Mode/Coupling`. Remove the noise from the trigger path by turning on either `Noise Rej` or `HF Reject` (choose the one that results in a stable trigger). The displayed sinusoid should be noisy but stable.

2. Press `Acquire`, then press the `Acq Mode` softkey to select "Averaging".

3. Vary the setting for `# Avgs` to select the number of averages that best eliminates the noise from the displayed waveform. In this mode the DSO calculates the average value of each point over the previous # frames of acquired data. The higher the number of averages, the slower the displayed waveform responds to waveform changes. Set `# Avgs` to `8`.

4. Change the AWG wave shape to square, then ramp, then back to sinusoid to see the effect of averaging.

5. Press the `Acq Mode` softkey to select "High Resolution". This mode averages sequential samples within the same acquisition rather than corresponding samples from previous acquisitions. At low sweep speeds the DSO oversamples the signal and the effective resolution is increased by averaging samples which would otherwise be ignored. The update rate of the display is not affected.

6. Change the AWG waveform to a sinusoid of 2 Vp-p.

7. **Disconnect** the lead from the DSO channel 2 BNC input (i.e. physically remove the lead from the DSO input).

The DSO autodetects the probes attached to the inputs, so it is important to remove any unwanted signals before hitting the Auto Scale button

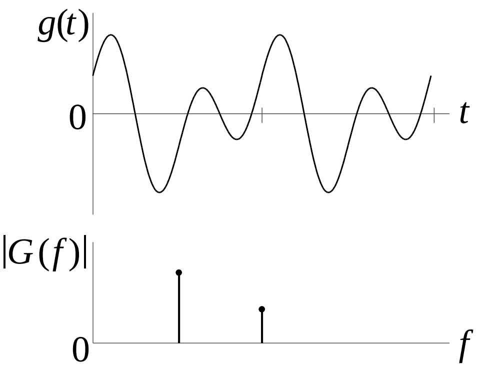
8. Press the DSO's `Auto Scale` button. If you did not disconnect the lead from channel 2 (which has no signal, apart from noise) the DSO will try to set the horizontal and vertical scales to view this "interesting" noisy signal, rather than setting the scales for the signal attached to channel 1 (if required, you can undo the effect of the Auto Scale button by choosing the `Undo Autoscale` softkey).

9. Set the Channel 1 position to 0.0 V and the trigger level to 0.0 V. These may have changed due to the Auto Scale feature.

10. Reconnect the DSO channel 2 lead to the BNC input.

Frequency-domain Measurement

Normally, when a signal is viewed on an oscilloscope, it is viewed in the time-domain. That is, the vertical axis is voltage and the horizontal axis is time. For many signals, this is the most logical and intuitive way to view them. But when the frequency content of the signal is of interest, it makes sense to view the signal in the frequency-domain. In the frequency-domain the horizontal axis is frequency and the vertical axis is voltage but is usually scaled in dBV (decibels relative to 1 V RMS).



The frequency-domain representation, or spectrum, is a graph of the sinusoids present in a signal

The Fast Fourier Transform (FFT) is an algorithm that efficiently converts a time-domain signal into its frequency-domain representation.

Sample Rate and Frequency Resolution

1. Set up a 2.2 kHz, 4 Vp-p sinusoid on the AWG. Set the DSO vertical scale to 1 V/div and horizontal scale to 100 μs/div.

2. Press the `Math` button and set `Function` to "g(t): Internal". Ensure `Operator` is "+", `Source 1` is "1" and `Source 2` is "2".

3. Set `Function` to "f(t): Displayed". Set `Operator` to "FFT". Set `Source 1` to "g(t)". Check the function information displayed just above the softkey descriptions. It should show "f(t)=FFT(Ch1+Ch2)".

The DSO's FFT function displays the frequency content of the signal

4. Use the shared position knob adjacent to the `Math` key to adjust the offset to approximately -30 dBV.

5. Use the shared scale knob to adjust the vertical scale to 10 dB/div.

6. Turn the Horizontal knob and observe the sample rate and FFT Resolution shown on the display. Set the horizontal scale to 1.000 ms/div.

   What is the sample rate at this setting? What is the FFT Resolution?

$f_s =$	$f_s/N =$

An FFT only displays frequencies from 0 to half the sample rate

   The FFT will only display frequencies from 0 to *half* the sample rate, $f_s/2$.

7. Set the frequency `Span` to 5 kHz and the `Center` frequency to 2.5 kHz. The `Span` and `Center` controls can be used to zoom in on a part of the FFT display.

8. Connect channel 2 to the TIMS 2 kHz Message signal in the “Master Signals” section. Turn on channel 2 and adjust the vertical scale and position to accommodate the signal in the display without clipping (it will not be a stable sinusoid, but you can see the peaks).

9. Turn off channel 1 and channel 2 so that only the FFT is displayed. Note that although the display of the channels is turned off the data acquisition still takes place and the data is available to calculate the FFT.

10. Note that even though two sinusoids are present in the signal $g(t)$, the DSO only displays a single “peak”, with a fairly wide side lobe.

The timebase and sample rate are inversely proportional to one another

11. Change the DSO horizontal scale to 10.00 ms/div, so that the sample rate changes to 250 kSa/s. The DSO now displays two peaks corresponding to the two sinusoids that make up “g(t)”.

12. Note that the side lobes of the “peaks” are reduced and that the difference between frequencies which can be distinguished on the display is also reduced. This is a consequence of increasing the frequency resolution by decreasing the sample rate. Be aware though, that although the frequency resolution of the FFT is determined by the sample rate and number of points calculated, the selected display settings may reduce the effective resolution visible on the display.

13. Disconnect Channel 2 from the TIMS.

Aliasing

The frequency $f_s/2$ is also known as the folding frequency. Frequencies that would normally appear above $f_s/2$ (and therefore outside the range of the FFT) are folded back into the normal range of the FFT. These unwanted frequency components are called aliases, since they erroneously appear under the alias of another frequency. To prevent aliasing, the DSO has to sample at greater than twice the highest frequency in the signal being measured (twice the bandwidth). It is therefore necessary to have some idea of the frequency content of the signal being measured to interpret the DSO's FFT results correctly.

To prevent aliasing, we have to sample at greater than twice the bandwidth of the signal

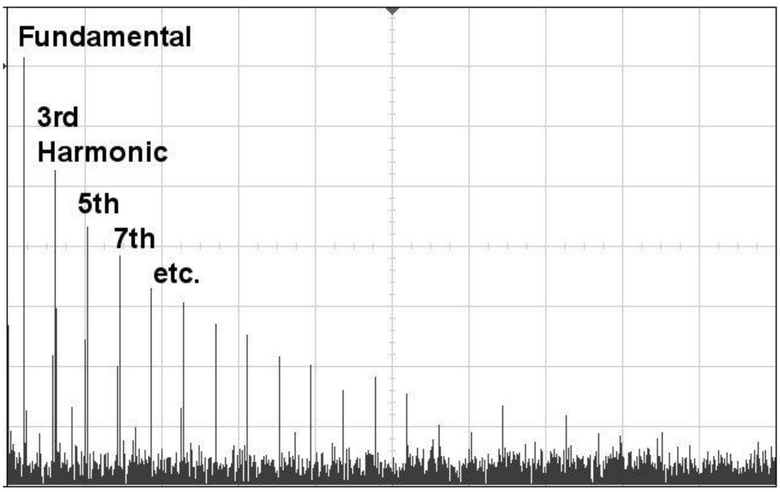
Fixed Signal with Varying Sample Rate

1. Set the AWG frequency to a **triangle** wave of approximately 2.6 kHz (you will need to change the ramp symmetry to 50%). Check the waveform on Channel 1 at 100 μs/div, then turn Channel 1 off.

2. Press `Math`. Press `More FFT`. Press `Auto Setup`. This will return the FFT frequency span and centre frequency to the default settings.

3. Set the FFT display offset to approximately -30 dBV and set the FFT vertical scale to 10 dB/div.

4. Observe the spectrum at a sample rate of 250 kSa/s. You should observe a spectrum similar to the following.



A triangle wave's spectrum with no aliasing

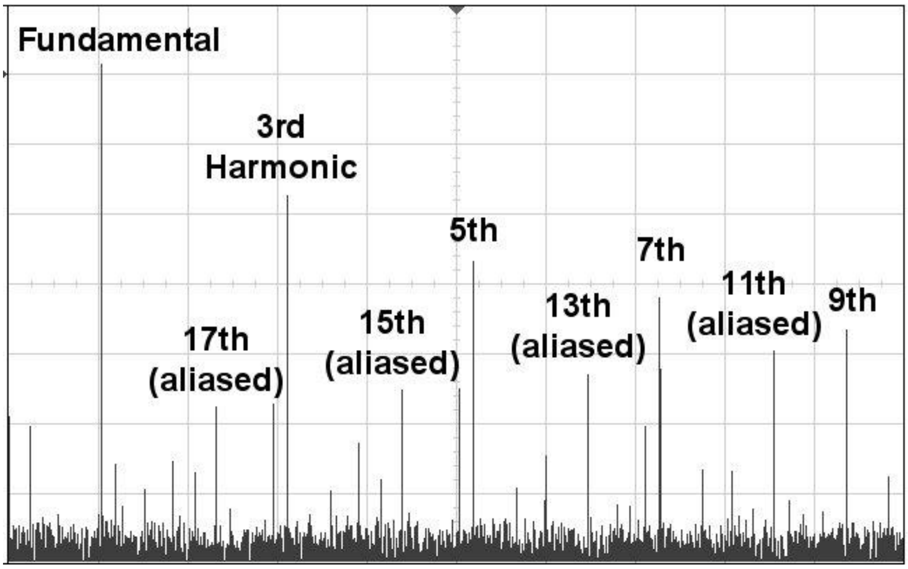
The leftmost spectral line is the fundamental. The next line is the 3rd harmonic. The next is the 5th harmonic and so forth. The higher harmonics are small in amplitude with the 35th harmonic just visible above the FFT noise floor. The frequency of the 35th harmonic is 35 x 2.6 kHz = 91 kHz, which is within the folding frequency of $f_s/2$ (125 kSa/s). Therefore, no significant aliasing is occurring.

5. Change to the following settings:

Sample rate	Freq Span	Center Freq
50 kSa/s	25 kHz	12.5 kHz

Now the upper harmonics of the triangle wave exceed the folding frequency and appear as aliases in the display.

A triangle wave's spectrum with aliasing



6. Change the sample rate to 12.5 kSa/s, then 5 kSa/s. The frequency plot is severely aliased.

Often the effects of aliasing are obvious, especially if you have some idea as to the frequency content of the signal. Spectral lines may appear where no frequency components exist.

It is important to recognise aliasing and take steps to prevent it

Signals that are bandlimited (that is, have no frequency components above a certain frequency) can be viewed alias-free by making sure that the effective sample rate is high enough.

Varying Signal with Fixed Sample Rate

1. Set the AWG waveform to a sinusoid with a frequency of 6 kHz. Push `Auto Scale` on the DSO. Turn all the channels off and observe just the spectrum of channel 1 with the following settings.

Sample rate	Freq Span	Center Freq
25.0 kSa/s	12.5 kHz	6.25 kHz

2. Press `Cursors` and measure the frequency of the spectral peak. Confirm that the DSO is detecting a large spectral peak at 6 kHz.

3. Increase the AWG frequency in steps of 1 kHz using the cursors and knob. The spectral peak (representing the AWG sinusoid) should move to the right as you increase the frequency – this is what we expect. Slowly increase the frequency to 12 kHz.

4. Continue increasing the AWG frequency slowly. Aliasing occurs as the frequency exceeds 12.5 kHz. Slowly increase the frequency from 14 kHz to 20 kHz. The spectral peak moves to the **left** on the display.

We're setting the measured signal's bandwidth above the DSO's folding frequency

5. Slowly decrease the AWG frequency in steps of 100 Hz so that the spectral peak returns **<u>for the first time</u>** to the vertical cursor positioned at 6 kHz. (The AWG's frequency should still be greater than 12.5 kHz). The DSO is now telling us that a frequency component exists at 6 kHz!

6. Read the frequency off the AWG display:

$f_1 =$

Record the frequency of the source

State the relation between the DSO's observed frequency, the sample rate, and the actual frequency of the sinusoid:

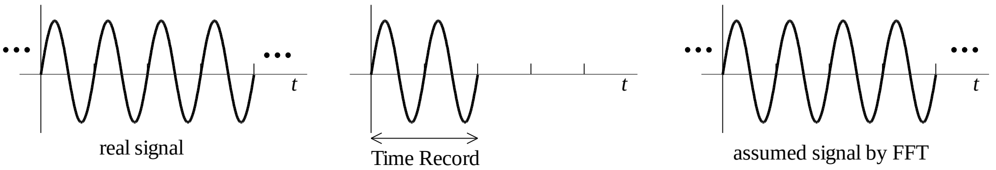
$f_{DSO} =$

7. Turn the `Cursors` off.

Windowing

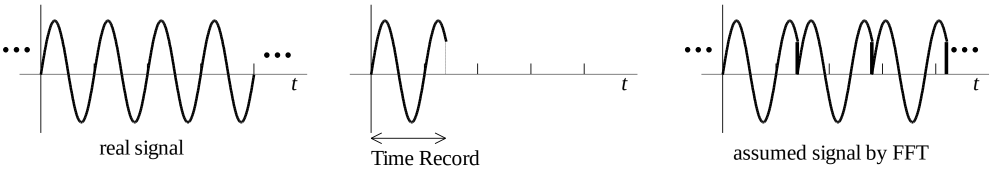
The FFT operates on a finite length time record, but assumes that this time record is exactly one period of an infinitely long periodic signal. With the waveform shown below, where an integral number of periods fits **exactly** within the time record, the infinitely long signal assumed by the FFT is correct.

FFT replicas producing the desired waveform



However, we do not normally have control over how the waveform fits into the time record of the DSO, with the result that discontinuities are introduced by the replication of the time record by the FFT over all time:

FFT replicas producing discontinuities



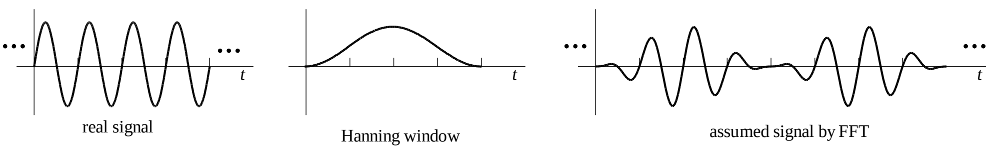
This effect is known as *leakage*, and the effect in the frequency-domain is very apparent. For the case of a single sinusoid as shown, the normally thin spectral line will spread out in a peculiar pattern.

The solution to the problem of leakage is to force the waveform to zero at the ends of the time record so that no discontinuity will exist when the time record is replicated. This is accomplished by multiplying the time record by a *window* function.

The window function modifies the time record and will produce its own effect in the frequency domain, but for a properly designed window, the effect is a vast improvement over no window at all.

The Hanning window, and it's effect in the time-domain, is shown below:

Windowing reduces spectral leakage



Even though the overall shape of a time-domain signal is changed by a window, the frequency content remains basically the same. There are many windows, all suited to different purposes. The Agilent DSO-X 2004A DSO has four, and are used for the following measurements:

Window	Useful for:
Hanning	making accurate frequency measurements or for resolving two frequencies that are close together
Flat Top	making accurate amplitude measurements of frequency peaks
Rectangular	good frequency resolution and amplitude accuracy, but only where there will be no "leakage" effects, e.g. self-windowing or synchronized waveforms
Blackman-Harris	greatly reducing "leakage" into adjacent FFT "bins" compared to the rectangular window, but at a cost of reduced frequency resolution

The Hanning and Flat Top windows should be used most of the time

We normally use the Hanning or Flat Top window. The Rectangular and Blackman-Harris windows should be considered windows for special situations.

L1.18

Windows

1. Press `Math`. Press the `More FFT` softkey and ensure `Window` is set to "Hanning".

2. Set the AWG frequency to 2 kHz. Observe just the spectrum on the following settings.

Sample rate	Freq Span	Center Freq	Window
25.0 kSa/s	100 Hz	2.00 kHz	Hanning

Sketch the spectrum:

3. Set the following.

Sample rate	Freq Span	Center Freq	Window
25.0 kSa/s	100 Hz	2.00 kHz	Flat Top

Sketch the spectrum:

4. Set the following.

Sample rate	Freq Span	Center Freq	Window
25.0 kSa/s	100 Hz	2.00 kHz	Rectangular

Sketch the spectrum:

5. Set the following.

Sample rate	Freq Span	Center Freq	Window
25.0 kSa/s	100 Hz	2.00 kHz	Blackman-Harris

Sketch the spectrum:

Practical Exam [2 marks]

When **all** lab work is completed, you will be asked by a tutor to:

1. Set up a 3 Vp-p sinusoid at 3 kHz, with 3 V DC offset. Display the entire waveform on the DSO with the 0 V reference set to the middle of the display.

2. Set the amplitude to 30 mVp-p and the DC offset to 30 mV. Set up the DSO to get a stable, noise-free display. Use averaging if necessary.

3. Set the amplitude to 3 Vp-p and the DC offset to 0 V. Apply the AWG signal to the IN input of the TIMS phase shifter. Set the coarse knob halfway and the ±180° switch to on. Measure the phase difference between IN and OUT.

4. Apply the AWG to the A input of a TIMS adder. Apply the 2 kHz TIMS MESSAGE signal to the B input of the TIMS adder. Set the gain controls on the adder module to halfway. Observe the output of the adder on channel 2 of the DSO. Turn Channel 1 off to double the acquisition rate – Ch 1 & 2 share the same acquisition memory. Display the spectrum using a sample rate of 10 kSa/s. Use cursors to measure the two dominant frequencies in the signal.

- **Turn off and disconnect all equipment.**

Marking

Assessment item	Mark	Tutor Signature
1	/0.5	
2	/0.5	
3	/0.5	
4	/0.5	
TOTAL	/2	

Questions [2 marks]

Encircle the correct answer, cross out the wrong answers. [one or none correct]

All questions are worth 0.2 marks each.

1. DSO Basics

(i)

The "output termination" setting of an AWG is set to 50 Ω, but the output is connected to a DSO high impedance input. The AWG amplitude is set to 2 V. The displayed waveform on the DSO has an amplitude of:

(a) 2V (b) 4 V (c) 1 V

(ii)

A DSO always takes 65536 samples before performing an FFT. By decreasing the sample rate (e.g. from 50 kSa/s to 10 kSa/s), the resolution of the FFT will:

(a) increase (b) decrease (c) remain the same

(iii)

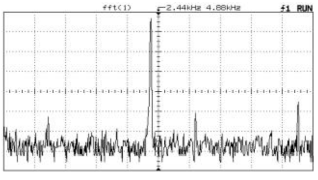
The spectral leakage of the Hanning window, compared to the rectangular window, is:

(a) less (b) more (c) the same

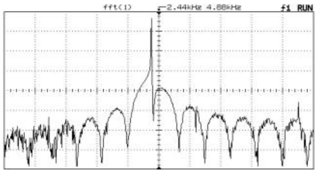
(iv)

The spectrum of a single sinusoid on a DSO that uses a rectangular window will in general look like:

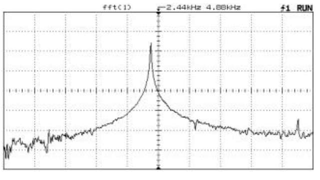
(a)



(b)



(c)



(v)

A DSO's sample rate is set to 100 kSa/s. The DSO display will show frequencies in the range:

(a) –50 kHz to 50 kHz (b) 0 to 100 kHz (c) 0 to 50 kHz

2. Aliasing

(i)

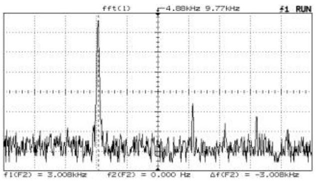
A signal has bandwidth B and is ideally sampled at a rate of f_S. Aliasing will NOT occur when:

(a) $f_S < B$ (b) $f_S > B/2$ (c) $f_S > 2B$

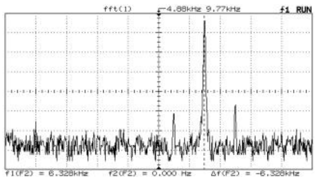
(ii)

A sinusoid of frequency 13 kHz is sampled at 20 kHz. The DSO display has a span of 0 Hz to 9.77 kHz. The display will look like:

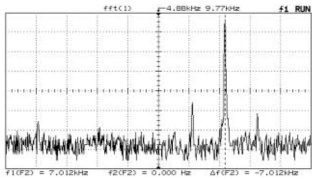
(a)



(b)



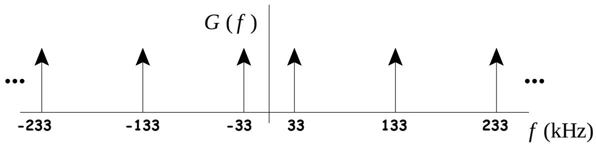
(c)



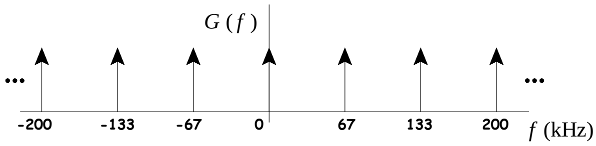
(iii)

A 67 kHz sinusoid is ideally sampled at 100 kHz. The ideal spectrum is:

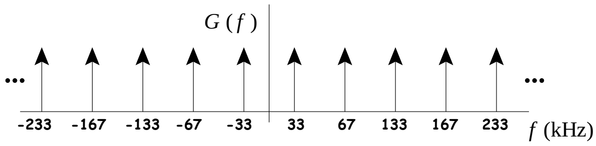
(a)



(b)



(c)



(iv)

A signal has a known bandwidth of 23 kHz. For maximum frequency resolution, the DSO sample rate should be set to:

(a) 20 kSa/s (b) 50 kSa/s (c) 100 kSa/s

(v)

Only two spectral peaks are present on a DSO display, and they are known to be parts of a single signal $g(t) = \cos(2\pi f_1 t)\cos(2\pi f_0 t)$, where $f_1 < 5\,\text{kHz}$ and $f_0 = 20\,\text{kHz}$. It is desired to measure the frequency f_1. For maximum frequency resolution, the DSO sample rate should be set to:

(a) 20 kSa/s (b) 50 kSa/s (c) 100 kSa/s

<u>**Complete the questions as part of your lab report.**</u>

Report

Only submit ***ONE*** report per lab group.

Complete the assignment cover sheet.

Ensure you have completed:

1. ***Lab Work*** – measurements and sketches.
2. ***Post-Lab Work*** – complete the multiple choice questions.

The lab report is due on the date specified in the Learning Guide.

You should hand the report directly to your tutor.

University of Technology, Sydney

Faculty of Engineering and Information Technology

Subject:	**48540 Signals and Systems**
Assessment Number:	**2**
Assessment Title:	**Lab 2 – Sampling and Reconstruction**
Tutorial Group:	

Students Name(s) and Number(s)

Student Number	Family Name	First Name

Declaration of Originality:

The work contained in this assignment, other than that specifically attributed to another source, is that of the author(s). It is recognised that, should this declaration be found to be false, disciplinary action could be taken and the assignments of all students involved will be given zero marks. In the statement below, I have indicated the extent to which I have collaborated with other students, whom I have named.

Statement of Collaboration:

Signature(s)

Marks

Hand Analysis	/0.5
MATLAB® Simulation	/1
TIMS Wiring Diagram	/0.5
Lab Sketches	/1
Questions	/1
TOTAL	/4

Office use only ☺

key

Assessment Submission Receipt

Assessment Title:	**Lab 2 – Sampling and Reconstruction**
Student's Name:	
Date Submitted:	
Tutor Signature:	

Lab 2 – Sampling and Reconstruction

Sampling. Reconstruction.

Introduction

Sampling is a simple yet extremely important process we perform on signals. Once a signal is sampled, we can "process" it with digital signal processors. We then convert the processed signal back to a continuous-time signal by a process known as *reconstruction*.

Objectives

1. To become familiar with one of the most important signal processing principles: sampling and reconstruction.

Equipment

- 1 Digital Storage Oscilloscope (DSO) – Agilent DSO-X 2004A with Wave Gen
- 1 Arbitrary Waveform Generator (AWG) – Agilent 33210A with Option 002
- 1 TIMS trainer with:
  - 1 multiplier
  - 1 headphone amplifier
  - 1 buffer amplifier
  - 1 master signals module

Safety

This is a Category A laboratory experiment. Please adhere to the Category A safety guidelines (issued separately).

Cat. A lab

L2.2

Sampling

Sampling is one of the most important operations we can perform on a signal. Samples can be quantized and then operated upon digitally (digital signal processing). Once processed, the samples are turned back into a continuous-time waveform. (eg. CD, mobile phone!) Here we demonstrate how, if certain parameters are right, a sampled signal can be reconstructed from its samples almost perfectly.

Ideal sampling involves multiplying a waveform by a train of impulses. The weights of the impulses are the sample values used by a digital signal processor (computer). The sampling waveform can be made up of any repeated pulse shape, such as a rect, triangle or even a sinc function:

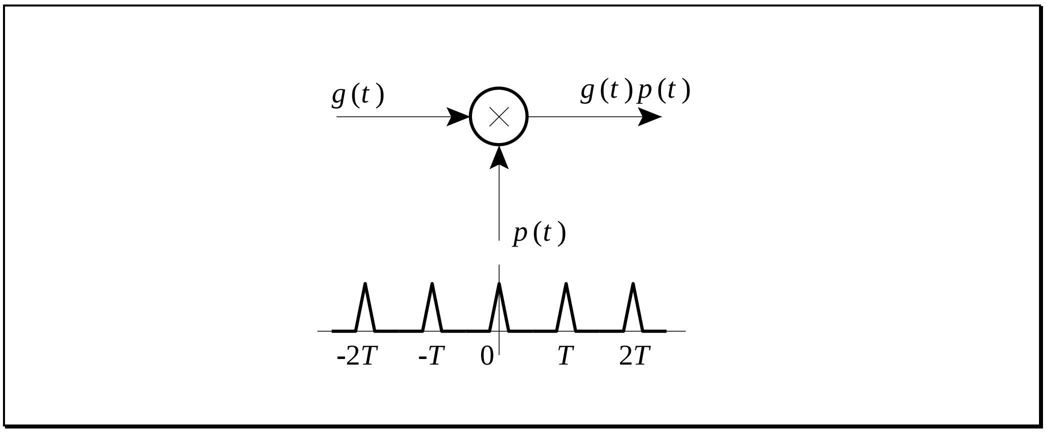


Figure L2.1

Even though the sampling waveform is non-ideal, we can still perfectly reconstruct the original waveform with an appropriate filter.

Reconstruction

Reconstruction of a waveform from its samples is accomplished by applying the sampled signal to a lowpass filter.

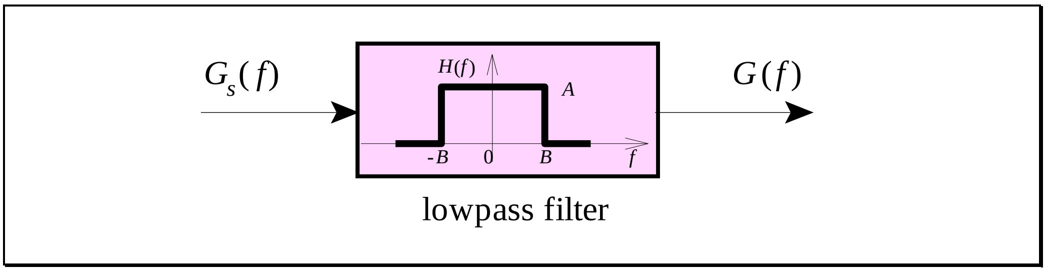


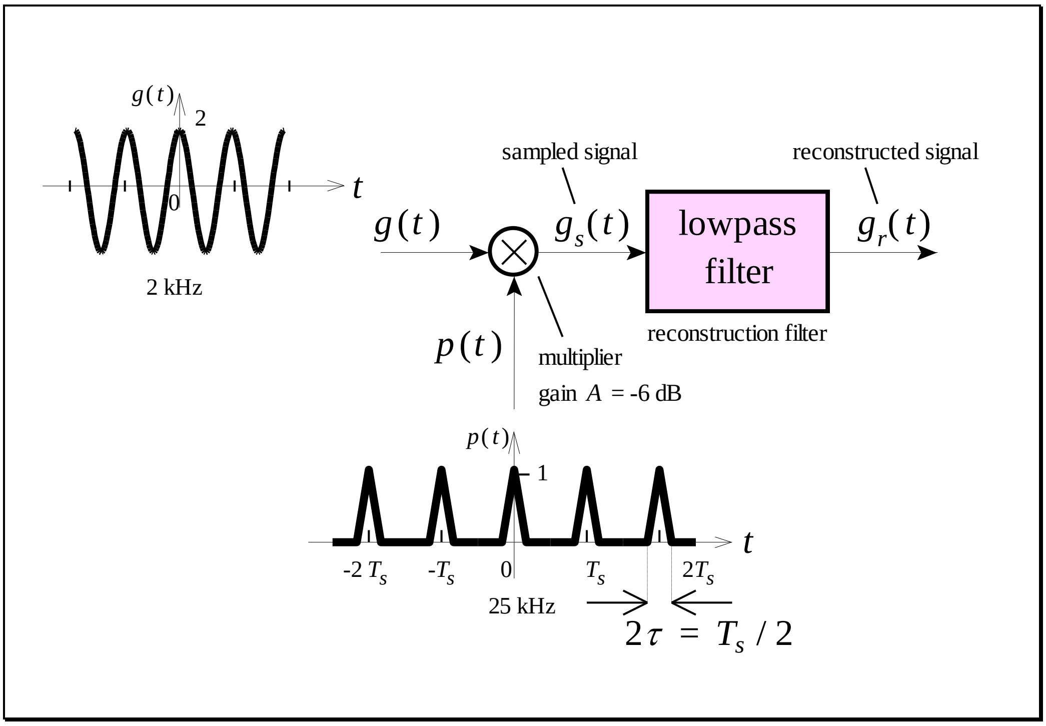
Figure L2.2

Pre-Lab Work – Hand Analysis [0.5 mark]

1. Perform a theoretical analysis by hand on the sampling / reconstruction scheme shown below. Note that the multiplier has an associated gain of 0.5 (-6 dB). Also, the triangle pulse train has been defined so that the base of each triangle has a width of 2τ.

    - Sketch **time-domain waveforms** and **magnitude spectra** at each point in the system.

    - Determine an ideal magnitude response of the reconstruction filter (specify gain, *A*, and cutoff frequency, *B*) to achieve reconstruction of the original signal.



Sampling and reconstruction scheme

Figure L2.3

Attach your hand analysis as part of your pre-lab work .

Pre-Lab Work – MATLAB® Simulation [1 mark]

2. Perform a MATLAB® simulation of the sampling / reconstruction scheme shown in Figure L2.4, showing ***ALL*** labelled signals as both a **time-domain waveform** and a **magnitude spectrum**:

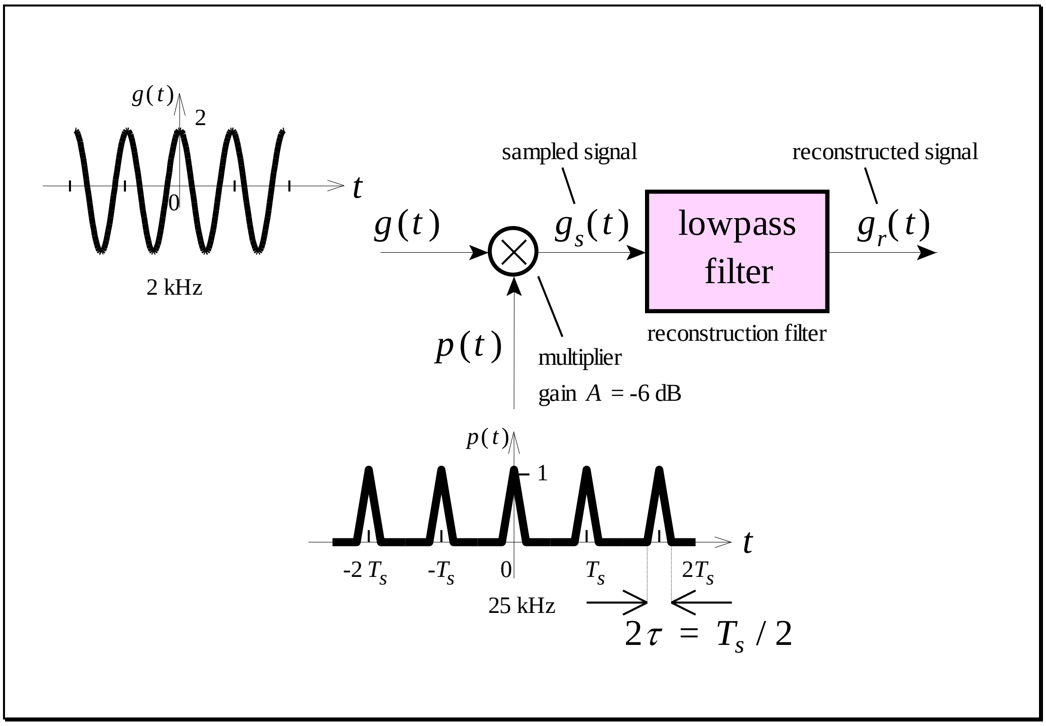


Figure L2.4

- Choose a MATLAB® "sample rate" of `fs=500e3` and choose `N=16384` samples. Do not confuse the MATLAB® "sample rate" with the sample rate of the sampling / reconstruction scheme we're simulating.
- The signal $g(t)$ is a 2 kHz sinusoid with an amplitude of 2.
- The signal $p(t)$ is obtained by using the MATLAB® *pulstran* and *tripuls* functions:

```
fc=25e3;
Tc=1/fc;
D=0:Tc:To-Ts;
p=pulstran(t,D,'tripuls',Tc/2);
```

- The multiplier has a gain of –6 dB.
- The reconstruction filter is an Elliptic lowpass filter with a cutoff frequency of $f_0 = 3\,\text{kHz}$. Use the MATLAB® *ellip* function:

```
[b,a]=ellip(5,0.2,50,3000/(fs/2));
```

  The vectors `b` & `a` can be used with the MATLAB® *filter* function:

```
output=A*filter(b,a,input);
```

  You will need to adjust the filter's passband gain, `A`, to achieve the correct amplitude of the reconstructed signal.
- All time-domain waveforms should be graphed from 0 to 2 milliseconds, with a range of –2 to 2.
- All magnitude spectra should be graphed from 0 to 250 kHz, with a range of –80 dB to 0 dB.

Attach a hardcopy of the MATLAB® simulation and code as part of your pre-lab work.

L2.6

Pre-Lab Work - TIMS Wiring Diagram [0.5 mark]

3. Construct a wiring diagram for the TIMS trainer that implements the scheme shown in Figure L2.4.

    - Use the 2 kHz MESSAGE output, located in the MASTER SIGNALS module, for the sinusoid $g(t)$.

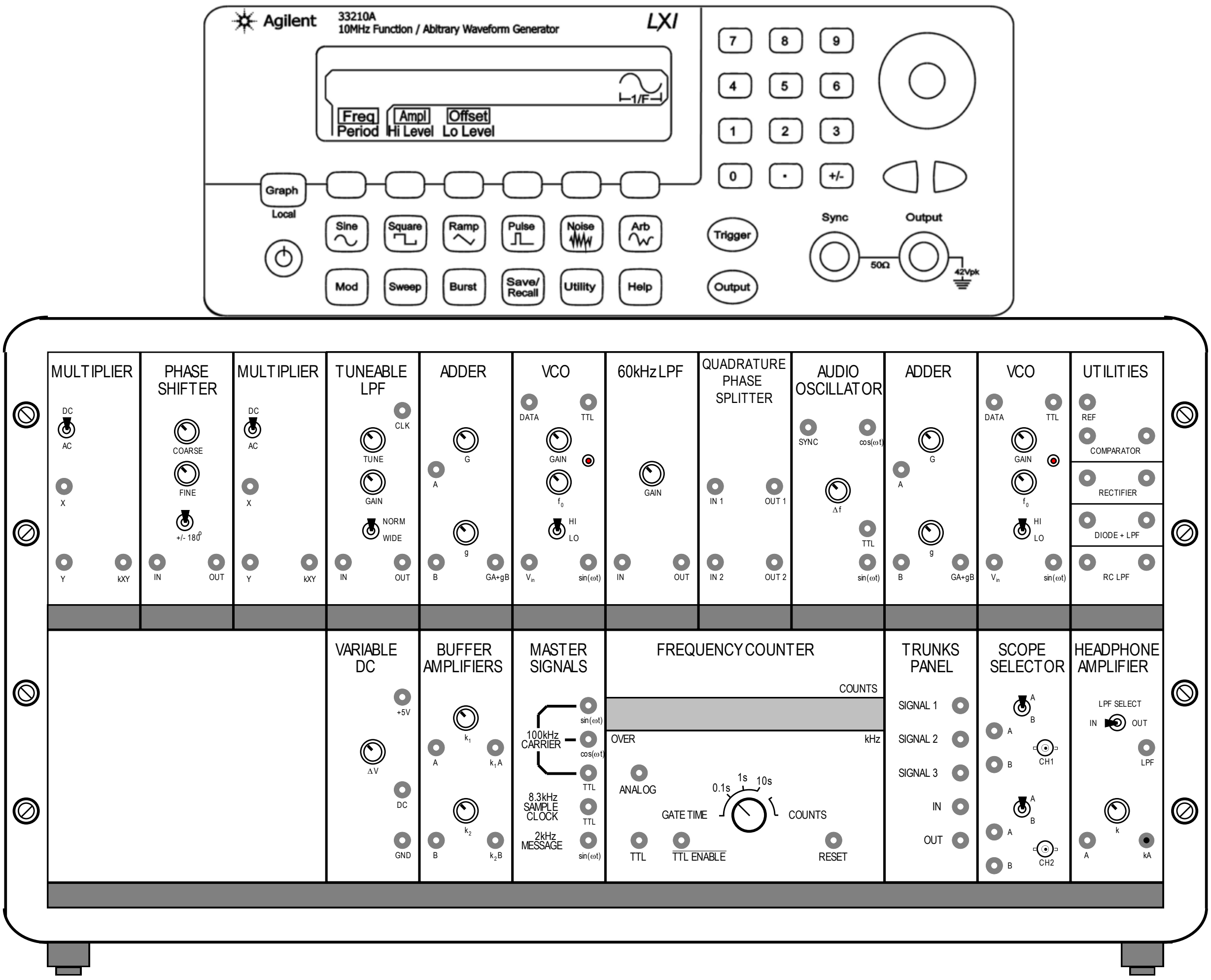
    - Use the AWG for the triangle pulse train $p(t)$.

    - Use the headphone lowpass filter for the reconstruction filter, followed by a buffer amplifier to provide the correct passband gain (to correct for the multiplier which has a gain of 0.5 and to provide any additional gain required by the design of your ideal reconstruction filter).

    - **Indicate the direction of signal flow on the tims wiring diagram.**

The headphone amplifier has a gain of 0 dB in the passband

We need gain to correct the multiplier's output – it's attenuated

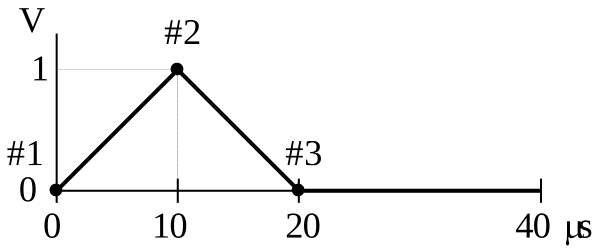


<u>Construct a tims wiring diagram on the above picture as part of your pre-lab work.</u>

Lab Work [1 mark]

1. Wire up the sampling and reconstruction scheme, following your TIMS wiring diagram from the pre-lab work.

2. To set up the triangle pulse train, use the AWG to create a new waveform (using the waveform editor) with 3 interpolated points that looks like:



Refer to the Lab Equipment Guide (LEG) for further information on setting up the AWG.

3. Don’t forget to add a buffer amplifier after the reconstruction filter (headphone LPF) to achieve the desired passband gain of your design.

4. To minimise noise, connect the AWG black lead to the TIMS GND terminal.

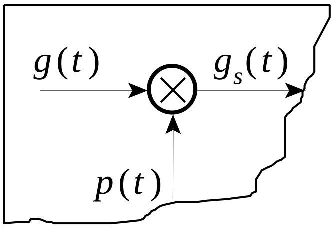
5. Ensure the TIMS MULTIPLIER switch is set to DC.

6. On the following graphs, sketch the time-domain waveform and the corresponding magnitude spectrum.
Choose scales that correspond to your MATLAB® simulations.
Use the following settings for the FFT:

Sample rate	Freq Span	Center Freq	Window
500 kSa/s	250 kHz	125 kHz	Hanning

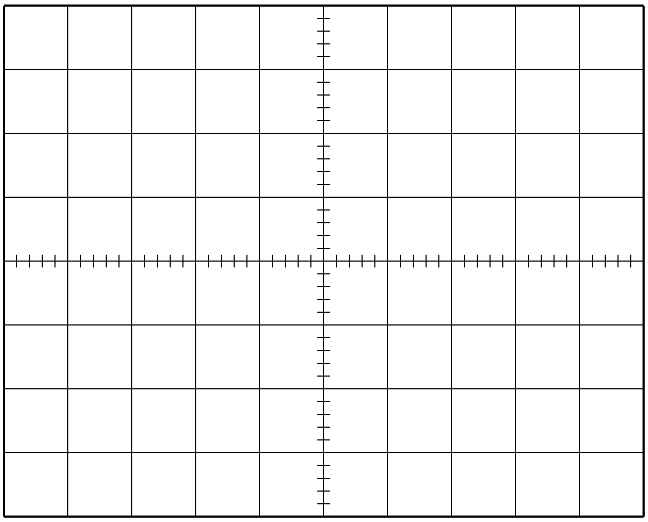
Set the Offset of the FFT to ≈ -40 dB, and the Scale to 10 dB/div.

L2.8



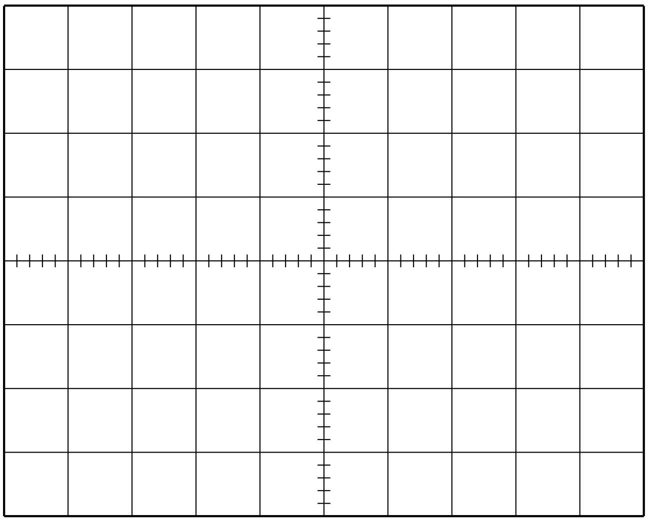
$g(t)$

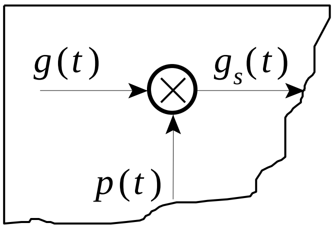
Time-domain view
of original sinusoid



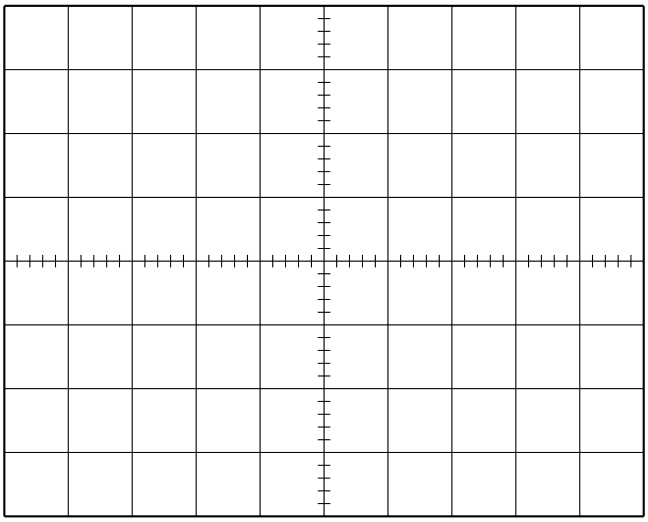
$|G(f)|$

Magnitude spectrum
of original sinusoid



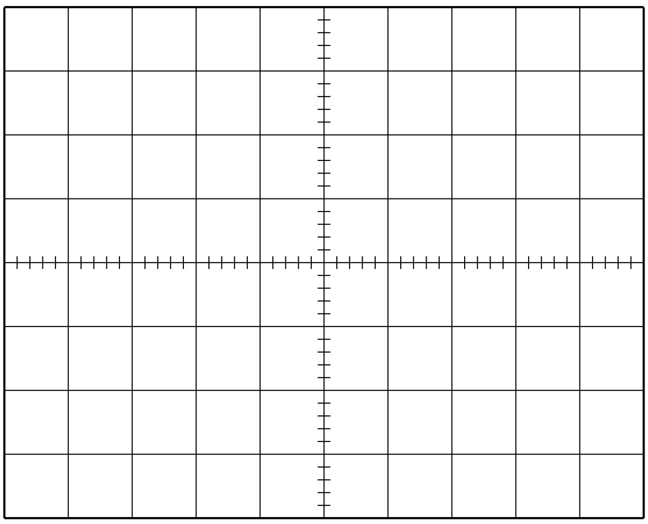


$p(t)$



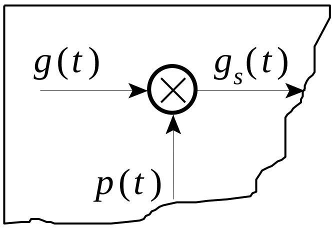
Time-domain view of uniform triangle pulse train

$|P(f)|$



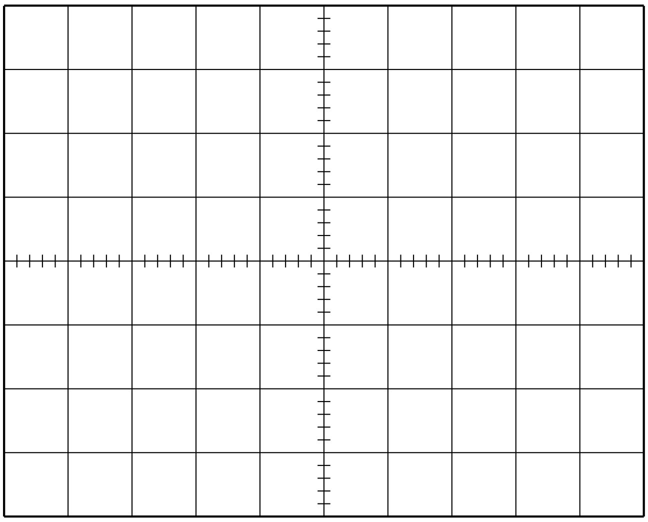
Magnitude spectrum of uniform triangle pulse train

L2.10



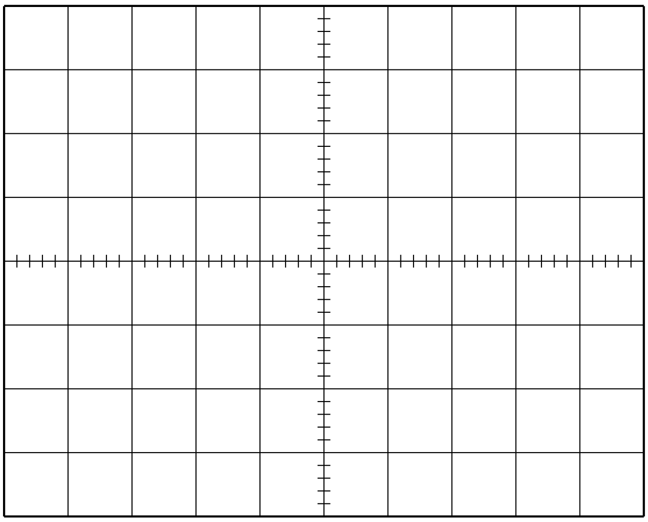
$g_s(t)$

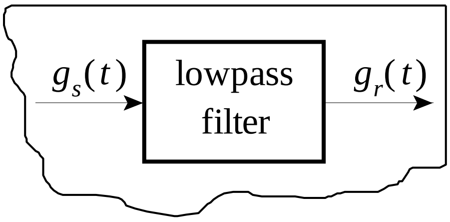
Time-domain view
of sampled sinusoid



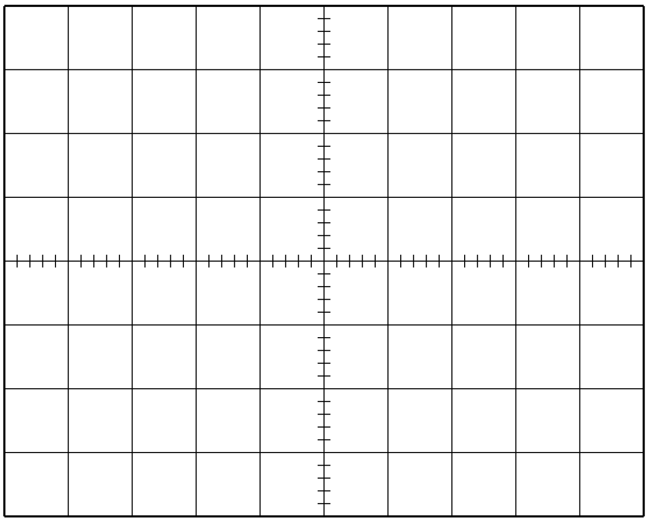
$|G_s(f)|$

Magnitude spectrum
of sampled sinusoid



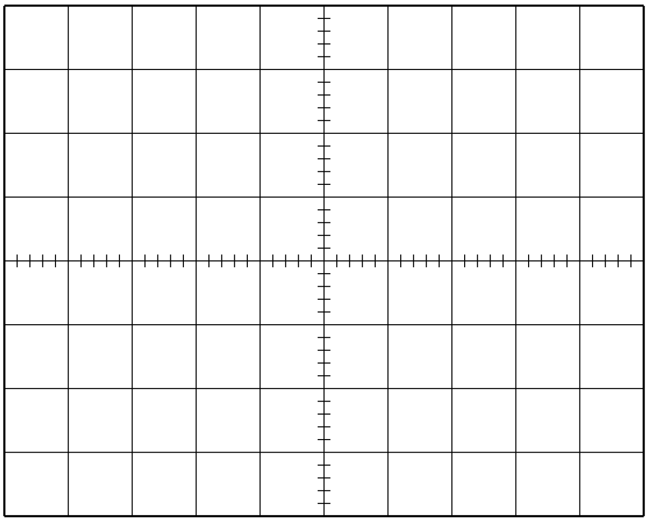


$g_r(t)$



Time-domain view of reconstructed sinusoid

$|G_r(f)|$



Magnitude spectrum of reconstructed sinusoid

L2.12

Questions [1 mark]

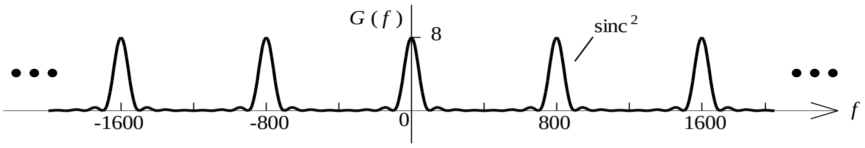
Encircle the correct answer, cross out the wrong answers. [one or none correct]

All questions are worth 0.1 marks each.

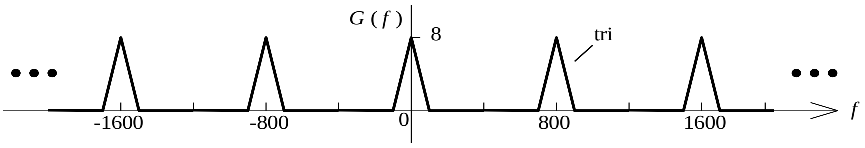
1. Sampling

(i)

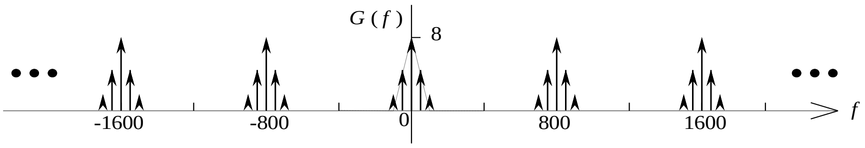
The signal $g(t) = \text{tri}(100t)$ is ideally sampled at a rate of $f_s = 800$ Sa/s. The resulting spectrum is:



(a)



(b)



(c)

(ii)

The number of sample values, N, required to yield enough information to exactly describe the signal $x(t) = 8 + 3\cos(8\pi t) + 9\sin(4\pi t)$ is:

(a) $N > 3$ (b) $N > 4$ (c) infinite

(iii)

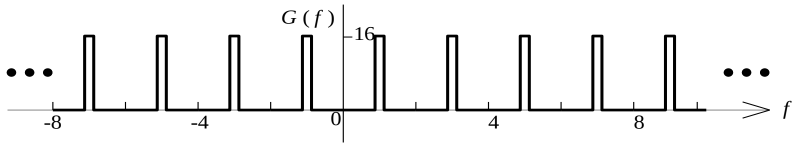
The Nyquist rate for the signal $x(t) = \text{rect}(300t)$ is:

(a) 150 Sa/s (b) 300 Sa/s (c) infinite

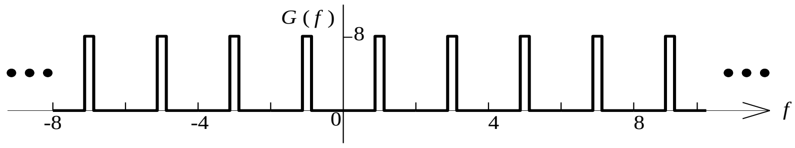
(iv)

The band-limited signal $g(t) = \text{sinc}(t/4)\cos(2\pi t)$ is ideally sampled at a rate $f_s = 4\,\text{Sa/s}$. The resulting spectrum is:

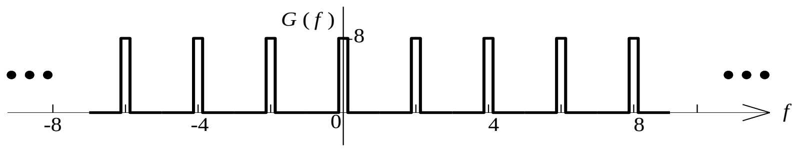
(a)



(b)



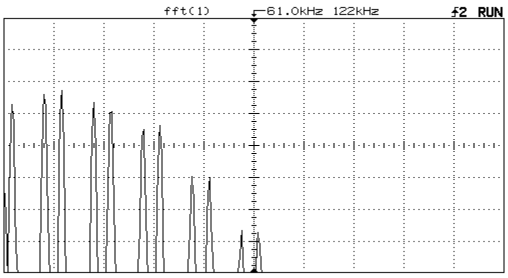
(c)



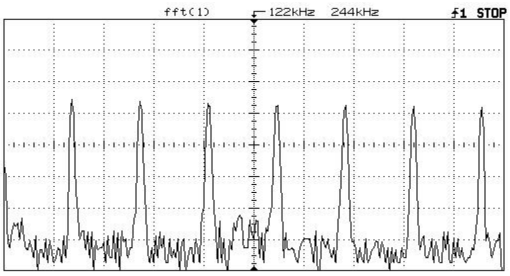
(v)

A 2 kHz sinusoid is multiplied by a 5% duty cycle 33 kHz square wave. The spectrum looks like:

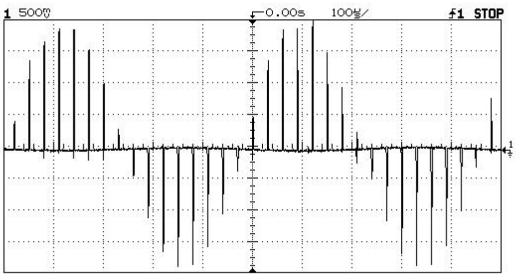
(a)



(b)



(c)



L2.14

2. Reconstruction

(i)

A 50% duty cycle 20 kHz square wave is one input to a multiplier. The other input is a sinusoid of 2 kHz. The finite pulse width of the square wave (compared to an impulse) will cause the multiplier output spectrum to:

(a) be weighted continuously by a sinc function.

(b) have repeats of the original spectrum spaced by 20 kHz.

(c) have repeats of the original spectrum spaced by 40 kHz.

(ii)

A signal $x(t)$ with bandwidth B is bandlimited to $f_s/2$ Hz before sampling at a rate of f_s Sa/s. Perfect reconstruction of $x(t)$ from its samples, using an ideal LPF with cutoff frequency f_0 Hz will only occur when:

(a) $B < f_0 < f_s/2$ (b) $B < f_0 < f_s$ (c) $f_s/2 < B < f_0$

(iii)

In the time-domain, ideal reconstruction of a waveform from its samples is achieved by:

(a) using the samples to weight a train of impulses.

(b) linearly interpolating between samples.

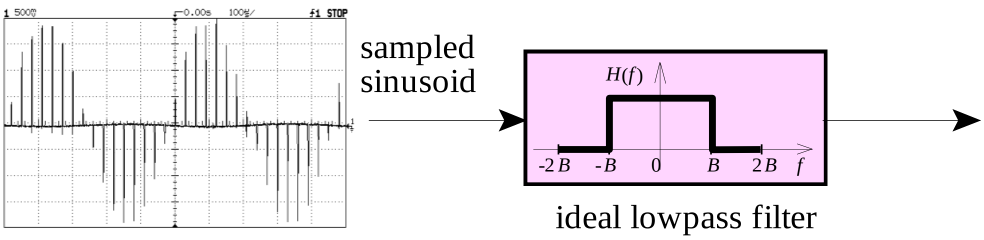
(c) holding the samples for a sample period.

(iv)

A signal has a bandwidth B and is ideally sampled at a frequency of $f_s = 3B$. The original signal can be recovered by an ideal LPF with cutoff frequency:

(a) $2B$ (b) $3B$ (c) $4B$

(v)



(a) The input spectrum is convolved with $H(f)$.

(b) The output is a stair-case waveform with a similar shape to a sinusoid.

(c) If B is equal to half the sample rate then the output is a sinusoid.

<u>Complete the questions as part of your lab report.</u>

Report

Only submit ***ONE*** report per lab group.

Complete the assignment cover sheet and attach your pre-lab work.

Ensure you have completed:

1. ***Pre-Lab Work*** – hand analysis, MATLAB® simulation and TIMS wiring diagram.

2. ***Lab Work*** – sketches of the time-domain waveforms and magnitude spectra.

3. ***Post-Lab Work*** – complete the multiple choice questions.

The lab report is due on the date specified in the Learning Guide.

You should hand the report directly to your tutor.

University of Technology, Sydney
Faculty of Engineering and Information Technology

Subject:	**48540 Signals and Systems**
Assessment Number:	**3**
Assessment Title:	**Lab 3 – First-Order Systems**
Tutorial Group:	

Students Name(s) and Number(s)

Student Number	Family Name	First Name

Declaration of Originality:

The work contained in this assignment, other than that specifically attributed to another source, is that of the author(s). It is recognised that, should this declaration be found to be false, disciplinary action could be taken and the assignments of all students involved will be given zero marks. In the statement below, I have indicated the extent to which I have collaborated with other students, whom I have named.

Statement of Collaboration:

Signature(s)

Marks

Hand Analysis	/1
MATLAB® Simulation	/1
Lab Work	/0.5
MATLAB® Verification	/0.5
Questions	/1
TOTAL	/4

Office use only ☺

key

Assessment Submission Receipt

Assessment Title:	**Lab 3 – First-Order Systems**
Student's Name:	
Date Submitted:	
Tutor Signature:	

Lab 3 – First-Order Systems

Frequency response. Step response.

Introduction

A first-order continuous-time system is one characterised by a first-order differential equation. Many real systems are modelled as first-order systems due to the relatively simple mathematics associated with them.

First-order systems can be characterised with two parameters – a gain K and a time constant T. We usually seek out these two numbers by performing two types of test on the system (but only if we can *model* the system as a first-order system!).

The frequency response of a system is a powerful description of a system. We have already seen that it completely specifies the system behaviour for any input (frequency response is the FT of the impulse response!) It tells us the order of the system, and whether or not a first-order model of the system is valid. We can get the constants K and T directly from the frequency response.

The step response is a familiar and important response for a system when viewed in the time-domain. By taking various measurements in the time-domain, we can find K and T. We normally do this to verify those values obtained from the frequency response.

The system we will be characterising will be an electronic system (a circuit with op-amps), but the techniques will be applicable for any system.

Objectives

1. To measure the frequency response of a system.

2. To determine a system's order, and to measure parameters that will completely characterise the system.

3. To verify a system's model by measuring parameters in the time-domain.

L3.2

Equipment

- 1 Digital Storage Oscilloscope (DSO) – Agilent DSO-X 2004A with Wave Gen
- 1 Arbitrary Waveform Generator (AWG) – Agilent 33210A with Option 002
- 1 state variable filter
- 4mm leads (assorted colours), BNC to 4mm adaptors

Note

Quality!!!

In this lab, "draw" means to make an accurate recording – one showing <u>times</u> and <u>amplitudes</u> as accurately as possible – this is the only way to interpret results after leaving the lab. Quick sketches are not acceptable – and are almost certainly useless when it comes to tying up theory with practice.

"Sketch" means to quickly give an overview, but showing important features.

Safety

Cat. A lab

This is a Category A laboratory experiment. Please adhere to the Category A safety guidelines (issued separately).

Pre-Lab Work – Hand Analysis [1 mark]

Single First-Order System

1. Show that the circuit:

First-order circuit

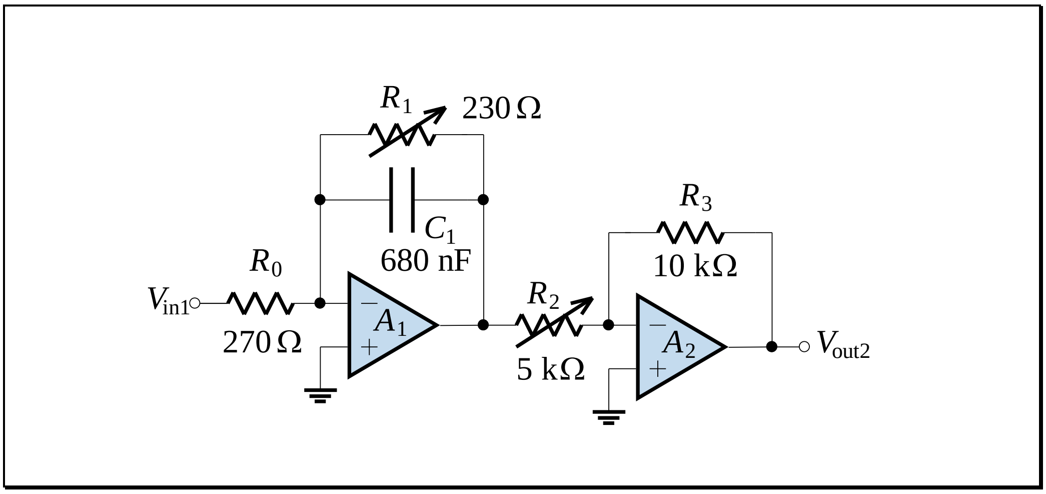


Figure L3.1

can be represented by the block diagram:

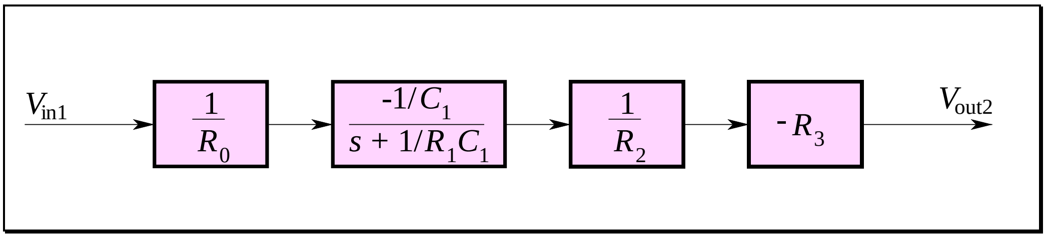


Figure L3.2

2. By comparing the transfer function of the system in Figure L3.2 with the standard form of a first-order lowpass transfer function,

$$T(s) = \frac{K_1}{1 + sT_1}, \qquad \text{(L3.1)}$$

write expressions for K_1 and T_1 in terms of R's and C's.

3. Which circuit element(s) may be used to set the location of the pole?

4. For the element values shown in Figure L3.1, calculate K_1 and T_1.

L3.4

5. Draw a pole-zero plot for the circuit.

6. Determine theoretical expressions for the magnitude response and phase response of the circuit.

7. Sketch a magnitude and phase Bode plot for the circuit.

8. Determine the theoretical expression for the unit-step response of the circuit, in terms of K_1 and T_1.

9. If the settling time is defined as $t_s = 4T_1$, what is the unit-step response *value* after a time t_s, as a percentage of the *steady-state value*? How can this be used to *experimentally* measure T_1?

10. What is the unit-step response *value* after a time T_1, as a percentage of the *steady-state value*? How can this be used to *experimentally* measure T_1?

11. What is the *initial* slope of the unit-step response? How can this be used to *experimentally* measure T_1?

Cascaded First-Order Systems

12. Determine the transfer function of the following second-order circuit:

Second-order circuit

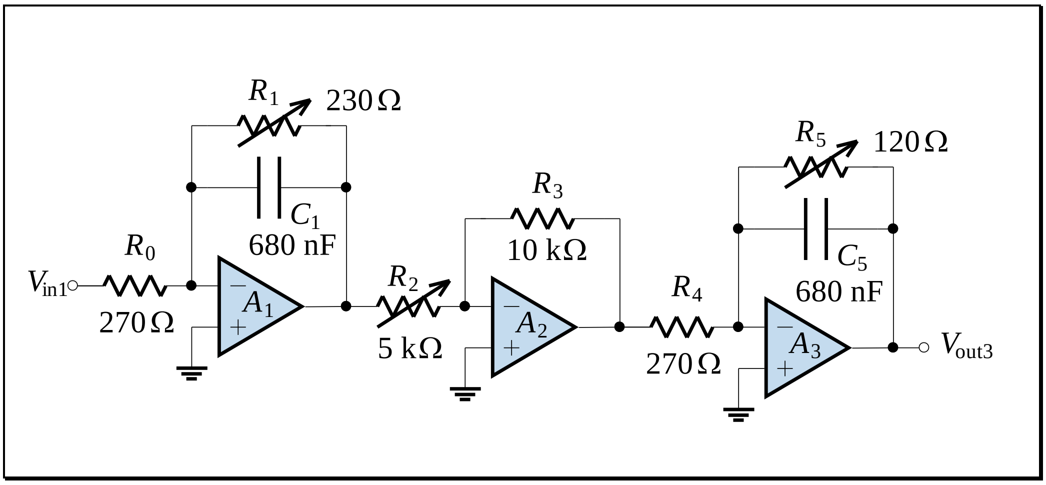


Figure L3.3

13. Draw a pole-zero plot for the circuit.
14. Sketch a magnitude and phase Bode plot for the circuit.

<u>**Attach your hand analysis as part of your pre-lab work.**</u>

Pre-Lab Work – MATLAB® Simulation [1 mark]

1. Use MATLAB® to plot the theoretical frequency response and step response of the first-order circuit shown in Figure L3.1 and the second-order circuit shown in Figure L3.3.

- You can create a log-spaced frequency vector using the MATLAB® *logspace* function:

```
f=logspace(log10(f1),log10(f2),100); w=2*pi*f;
```

- The theoretical frequency response can be obtained with the MATLAB® *freqs* function:

```
H=freqs(b,a,w);
```

  The vectors *b* and *a* are the numerator and denominator coefficient vectors of the transfer function, respectively.

- Use the following tables for the frequency response scales:

Frequency response scales

First-order Circuit	Horizontal axis	Vertical axis
Magnitude Response	$46\text{ Hz} \le f \le 46\text{ kHz}$	$-50\text{ dB} \le 20\log\lvert\mathbf{T}\rvert \le 10\text{ dB}$
Phase Response	$46\text{ Hz} \le f \le 46\text{ kHz}$	$-90° \le \angle\mathbf{T} \le 0°$

Second-order Circuit	Horizontal axis	Vertical axis
Magnitude Response	$46\text{ Hz} \le f \le 46\text{ kHz}$	$-50\text{ dB} \le 20\log\lvert\mathbf{T}\rvert \le 10\text{ dB}$
Phase Response	$46\text{ Hz} \le f \le 46\text{ kHz}$	$0° \le \angle\mathbf{T} \le 180°$

- The theoretical step response can be obtained with the MATLAB® *step* function.

- Use the following tables for the step response scales:

Step response scales

First-order Circuit	Horizontal axis	Vertical axis
Step Response	$0\text{ ms} \le t \le 1\text{ ms}$	$0\text{ V} \le v_{\text{out2}}(t) \le 2\text{ V}$

Second-order Circuit	Horizontal axis	Vertical axis
Step Response	$0\text{ ms} \le t \le 1\text{ ms}$	$-1\text{ V} \le v_{\text{out3}}(t) \le 0\text{ V}$

Bring print-outs of your frequency response, step response and MATLAB® code to the lab as part of your pre-lab work.

L3.6

Lab Work [0.5 mark]

Frequency Response

Refer to the Lab Equipment Guide

1. Set up the SVF to give the circuit in Figure L3.3. Don't forget to remove the link joining Vout3 to Vin1'.

2. Examine the circuit in Figure L3.1 to determine its input impedance. Now set the AWG output Load setting to match the input impedance of the circuit. This will ensure that the amplitude settings on the AWG display will match the actual physical voltage appearing at the AWG terminals.

First-Order Frequency Response

3. Measure the first-order frequency response $\mathbf{V}_{\text{out2}}/\mathbf{V}_{\text{in1}}$.

The desired frequencies are linearly spaced on a log-scale

f (kHz)	$\lvert\mathbf{V}_{\text{in1}}\rvert$ (V)	$\lvert\mathbf{V}_{\text{out2}}\rvert$ (V)	$\lvert\mathbf{V}_{\text{out2}}\rvert/\lvert\mathbf{V}_{\text{in1}}\rvert$ (dB)	$\angle\mathbf{V}_{\text{out2}} - \angle\mathbf{V}_{\text{in1}}$ (°)
0.010				
0.022				
0.046				
0.100				
0.215				
0.464				
1.00				
2.15				
4.64				
10.0				
21.5				
46.4				
100.0				

Second-Order Frequency Response

4. Measure the second-order frequency response $\mathbf{V}_{out3}/\mathbf{V}_{in1}$.

f (kHz)	$\lvert\mathbf{V}_{in1}\rvert$ (V)	$\lvert\mathbf{V}_{out3}\rvert$ (V)	$\lvert\mathbf{V}_{out3}\rvert/\lvert\mathbf{V}_{in1}\rvert$ (dB)	$\angle\mathbf{V}_{out3}-\angle\mathbf{V}_{in1}$ (°)
0.010				
0.022				
0.046				
0.100				
0.215				
0.464				
1.00				
2.15				
4.64				
10.0				
21.5				

Step Response

<u>The step response is the response of a system to a step input. Use a low frequency square wave for step response!</u>

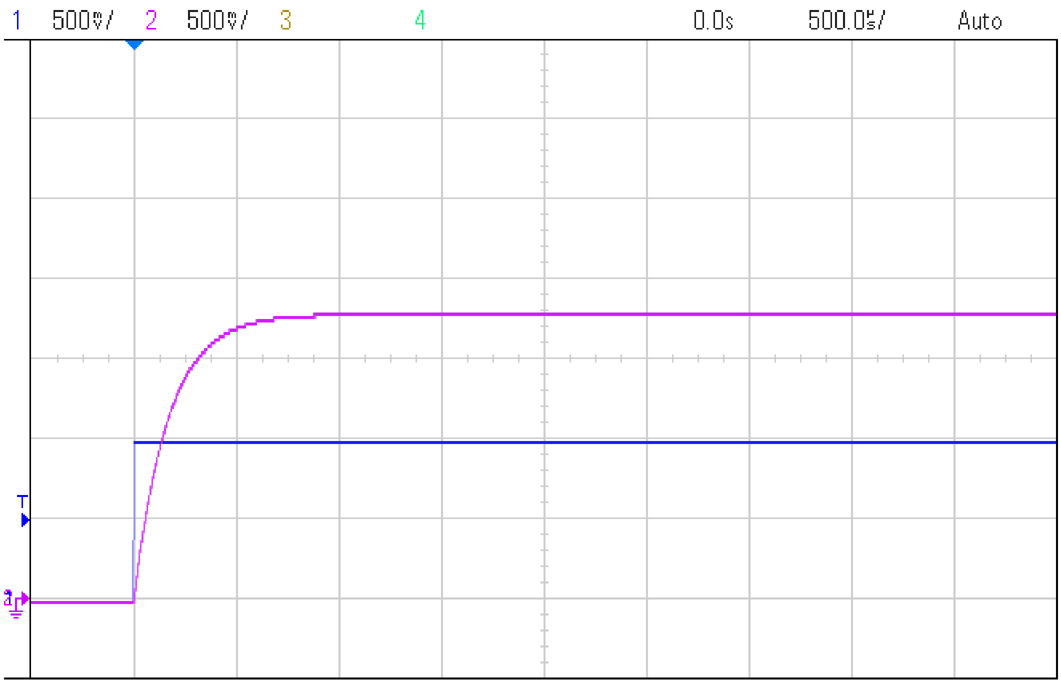
We're just making a "step" using a square wave

Since we're going to be looking at the step response using a repetitive waveform, we need to set up a square wave with a period long enough to *look like* a step – in other words the output should be steady within half a period.

1. Set the AWG to generate a 0 V to 1 V, 100 Hz square wave.

2. Press `Horiz`. Set the `Time Ref` softkey to `Left`. This positions the trigger point on the oscilloscope one division from the left edge of the screen (which will maximise the view of the transient response).

3. Set the reference level (⏚) for the DSO channels one division from the bottom of the display. This will maximise the viewing area of the step response:



First-Order Step Response

4. Measure the time constant T_1 of the first-order circuit using each of the three methods below:

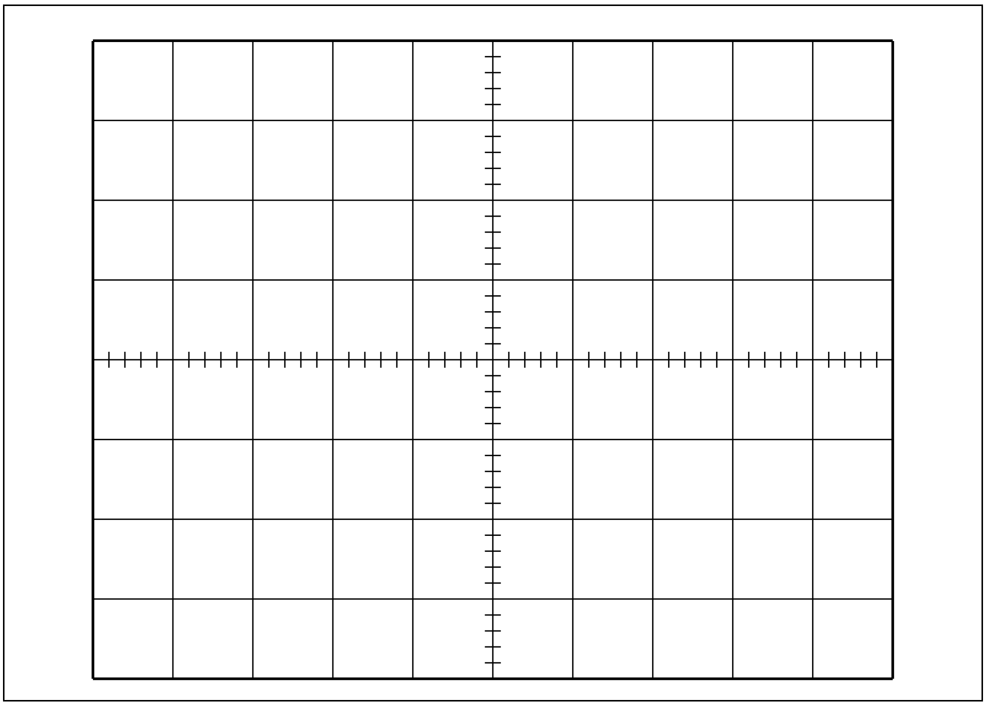
Settling time	One time constant	Initial slope
$v(t_s) = 0.98 v_{ss}$ $t_s \approx 4T_1$	$v(T_1) \approx 0.63 v_{ss}$	$\left.\frac{dv}{dt}\right\|_{t=0} = \frac{v_{ss}}{T_1}$
Answer: $T_1 =$	Answer: $T_1 =$	Answer: $T_1 =$

Methods to measure the time constant of a first-order circuit

5. Measure the constant K_1 used in the transfer function of Eq. (L3.1).

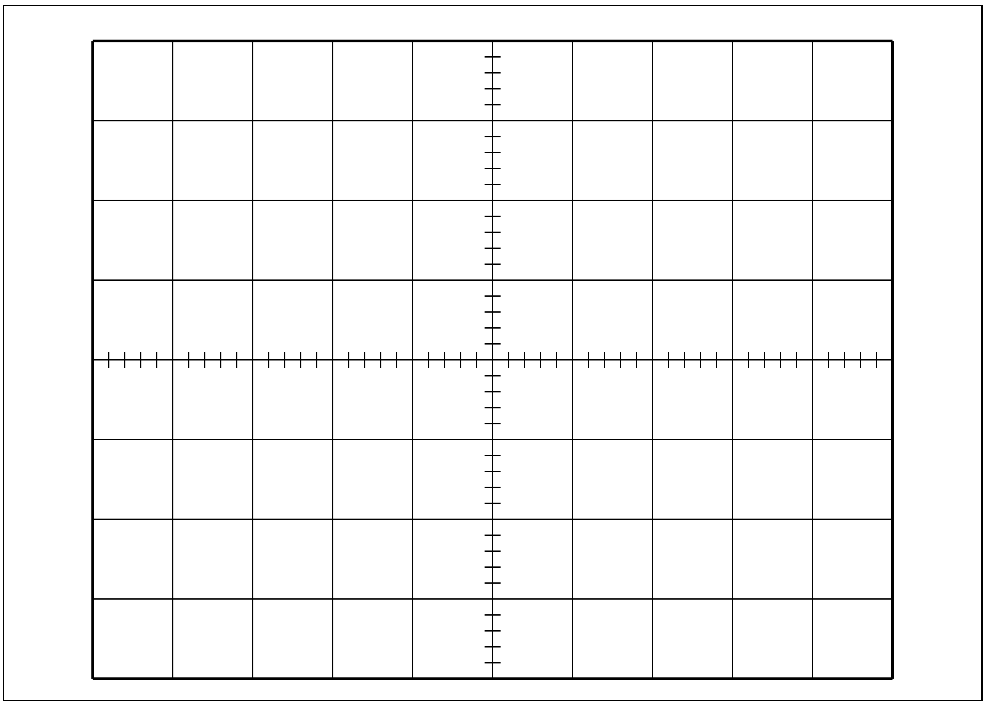
Answer: $K_1 =$

6. Sketch the step response of the fist-order circuit:



Second-Order Step Response

7. Invert the output of the second-order circuit using the DSO. Sketch the step response, showing clearly the difference between this and the first-order step response (which occurs near time $t = 0$):



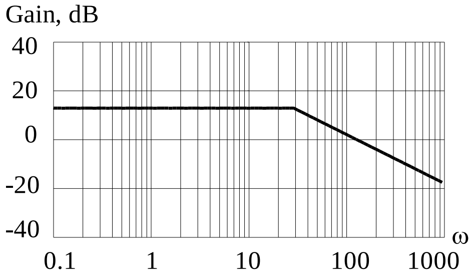
Questions [1 mark]

Encircle the correct answer, cross out the wrong answers. [one or none correct]

All questions are worth 0.1 marks each.

1. Frequency Responses

(i)



Asymptotic straight line approximation of a first-order magnitude response. The transfer function is:

(a) $T(s)=\dfrac{4.5}{s+30}$ (b) $T(s)=\dfrac{13}{1+s/60\pi}$ (c) $T(s)=\dfrac{135}{s+30}$

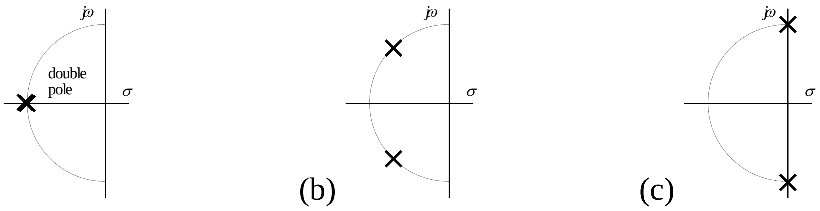
(ii)

The phase response of an all-pole 2nd-order lowpass system may vary between:

(a) 0 and -90° (b) -90° and +90° (c) -180° and -360°

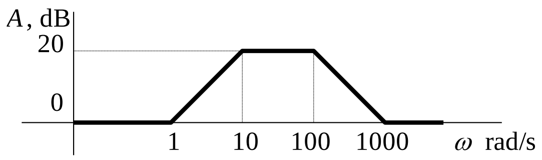
(iii)

The pole-zero plot corresponding to two identical unbuffered first-order systems in cascade is:



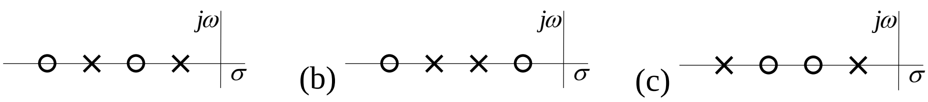
(a) (b) (c)

(iv)



The pole-zero plot corresponding to the asymptotic Bode plot is:

(a) (b) (c)



(v)

The phase response of an *RC* circuit varies from 0° to 90°. The circuit is:

(a) highpass (b) lowpass (c) bandpass

L3.12

2. Step Responses

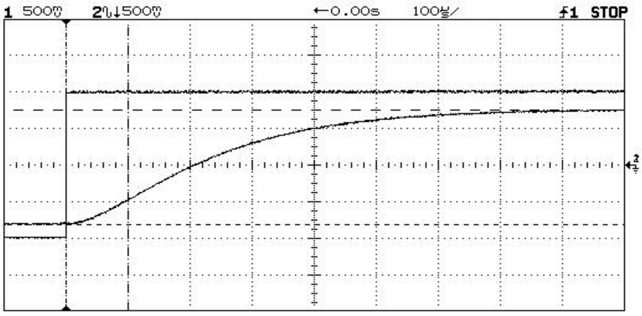
(i)

$$T(s) = \frac{K}{1+sT}$$

The DC gain is:

(a) KT (b) K/T (c) K

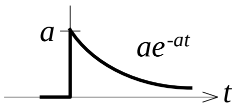
(ii)



Step response of a system. Assuming a first-order system, the transfer function is approximately:

(a) $T(s) = \frac{1560}{s+4000}$ (b) $T(s) = \frac{3333}{s+6667}$ (c) $T(s) = \frac{3120}{s+4000}$

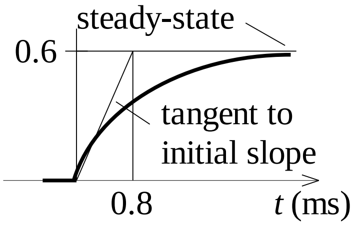
(iii)



Unit-step response of a system. The system is:

(a) lowpass (b) highpass (c) unstable

(iv)



Step response of a first-order system. The time constant is:

(a) 0.75 ms (b) 1.333 ms (c) 0.6 ms

(v)

The unit-step response of a first-order system was found to have a steady-state value of 0.8. The time it took for the output to rise to 80% of the steady-state value was 3.219 s. The impulse response of the system is:

(a) $1.6e^{-2t}$ (b) $0.8e^{-t/3.219}$ (c) $0.4e^{-t/2}$

<u>**Complete the questions as part of your lab report.**</u>

Report

Only submit ***ONE*** report per lab group.

Complete the assignment cover sheet and attach your pre-lab work.

Ensure you have completed:

1. ***Pre-Lab Work*** – hand analysis and MATLAB® simulations.

2. ***Lab Work*** – frequency response tables and step response calculations.

3. ***Post-Lab Work*** – complete the multiple choice questions. Modify your MATLAB® code to plot your experimental frequency response results on top of your theoretical pre-lab work.

The lab report is due on the date specified in the Learning Guide.

You should hand the report directly to your tutor.

University of Technology, Sydney

Faculty of Engineering and Information Technology

Subject: **48540 Signals and Systems**

Assessment Number: **4**

Assessment Title: **Lab 4 – Modulation and Demodulation**

Tutorial Group:

Students Name(s) and Number(s)

Student Number	Family Name	First Name

Declaration of Originality:

The work contained in this assignment, other than that specifically attributed to another source, is that of the author(s). it is recognised that, should this declaration be found to be false, disciplinary action could be taken and the assignments of all students involved will be given zero marks. In the statement below, I have indicated the extent to which I have collaborated with other students, whom I have named.

Statement of Collaboration:

Signature(s)

Marks

Hand Analysis	/1
MATLAB® Simulation	/1
TIMS Wiring diagram	/0.5
Lab Sketches	/0.5
Questions	/1
TOTAL	/4

Office use only ☺

key

Assessment Submission Receipt

Assessment Title:	**Lab 4 – Modulation and Demodulation**
Student's Name:	
Date Submitted:	
Tutor Signature:	

Lab 4 – Modulation and Demodulation

QAM modulation. QAM demodulation.

Introduction

Modulation is the process by which some characteristic (amplitude, frequency or phase) of a *carrier* sinusoid is varied in accordance with a message signal. We modulate high frequency carriers to achieve:

- sharing of the limited electromagnetic spectrum
- efficient transmission and reception with small antennas

Demodulation is the process of recovering the message signal from a part of the total spectrum. There is usually more than one way to demodulate a signal. We choose the method depending on circuit complexity and performance criteria.

Objectives

1. To become familiar with one of the most important signal processing principles: modulation and demodulation

Equipment

- 1 Digital Storage Oscilloscope (DSO) – Agilent DSO-X 2004A with Wave Gen
- 1 Arbitrary Waveform Generator (AWG) – Agilent 33210A with Option 002
- 1 TIMS trainer with:
    - 2 multipliers
    - 1 adder
    - 1 60 kHz lowpass filter (or 1 twin)
    - 1 phase shifter (HI frequency range)

Safety

This is a Category A laboratory experiment. Please adhere to the Category A safety guidelines (issued separately). Cat. A lab

L4.2

Modulation

Let $g(t)$ be a message signal such as an audio signal that is to be transmitted through a cable or the atmosphere. In amplitude modulation (AM), the message signal modifies (or modulates) the amplitude of a *carrier* sinusoid $\cos(2\pi f_c t)$. In one form of AM transmission, two messages that occupy the same part of the spectrum can be sent by combining their spectra in *quadrature*. If the first message signal $g_1(t)$ is multiplied by a carrier $\cos(2\pi f_c t)$, then the second message signal $g_2(t)$ is multiplied by $\sin(2\pi f_c t)$. The process is illustrated below:

QAM modulation

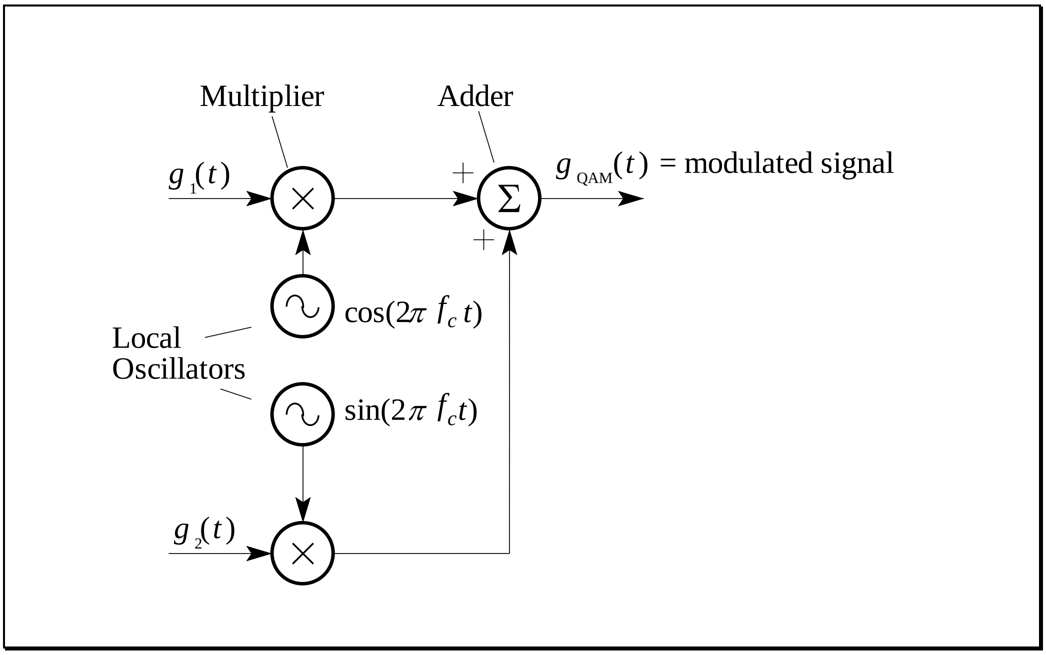


Figure L4.1

The *local oscillators* in Figure L4.1 are devices that produce sinusoidal signals. One oscillator is $\cos(2\pi f_c t)$. The other oscillator has a phase which is said to be in *quadrature*, or a phase of $-\pi/2$ with respect to the first oscillator, to give $\sin(2\pi f_c t)$. The multiplier is implemented with a non-linear device and the adder is a simple op-amp circuit (for input signals with a bandwidth less than 1 MHz).

The spectrum of the modulated signal has two parts in quadrature. Each part is a replica of the original message spectrum but "shifted up" in frequency. The parts do not "interfere" since the first message forms the real (or in-phase, *I*) part of the modulated spectrum and the second message forms the imaginary (or quadrature, *Q*) part. An abstract view of the spectrum showing its real and imaginary parts is shown below:

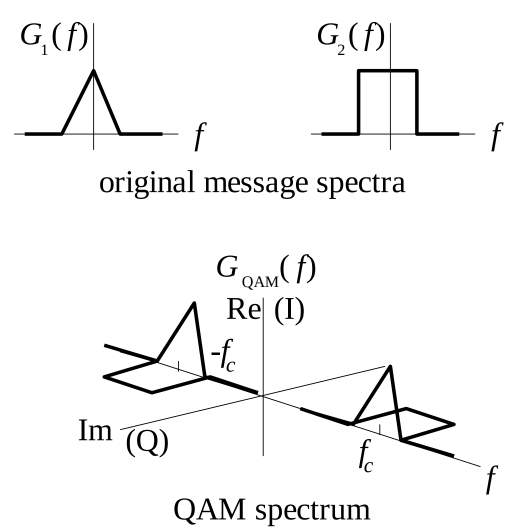


Figure L4. 2

Normally, we represent a spectrum by its magnitude and phase, and not its real and imaginary parts, but in this case it is easier to picture the spectrum in "rectangular coordinates" rather than "polar coordinates".

If the spectrum of both message signals is bandlimited to B Hz, then the modulated signal spectrum has an upper sideband from f_c to $f_c + B$ and a lower sideband from $f_c - B$ to f_c.

The appearance of the modulated signal in the time domain is that of a sinusoid with a time-varying amplitude and phase, but since the amplitude of the quadrature components (cos and sin) of the carrier vary in proportion to the message signals, this modulation technique is called *quadrature amplitude modulation*, or *QAM* for short.

Demodulation

The reconstruction of $g_1(t)$ and $g_2(t)$ from $g_{\text{QAM}}(t)$ is called *demodulation*. There are several ways to demodulate the QAM signal - we will consider a simple analog method called *coherent* (or *synchronous*) *demodulation*.

Coherent Demodulation

Coherent QAM demodulation

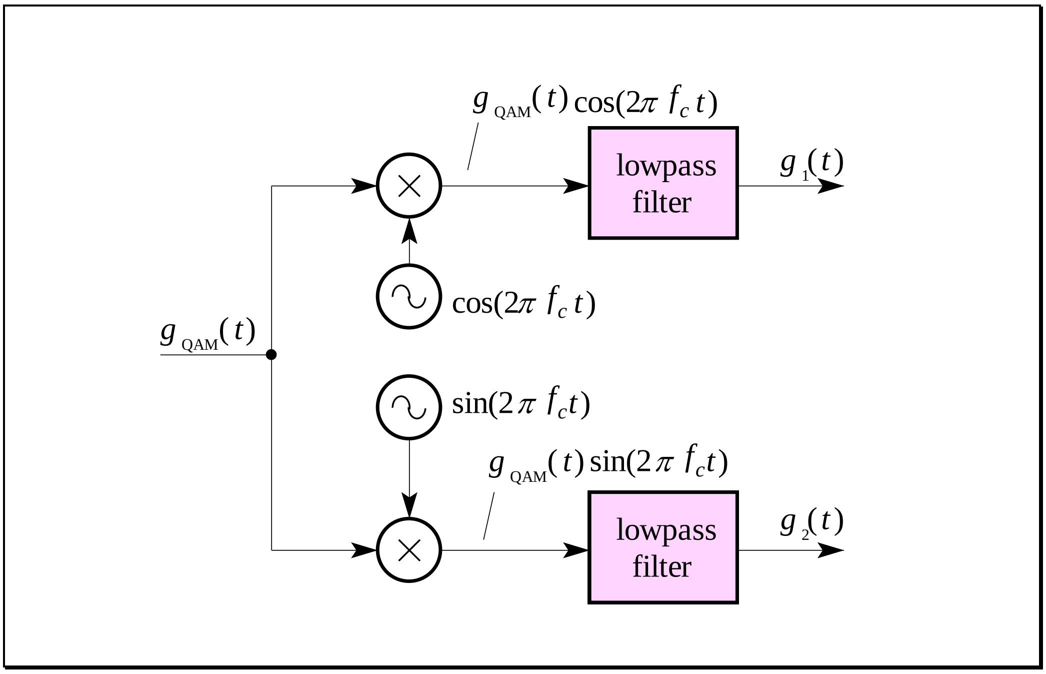


Figure L4.3

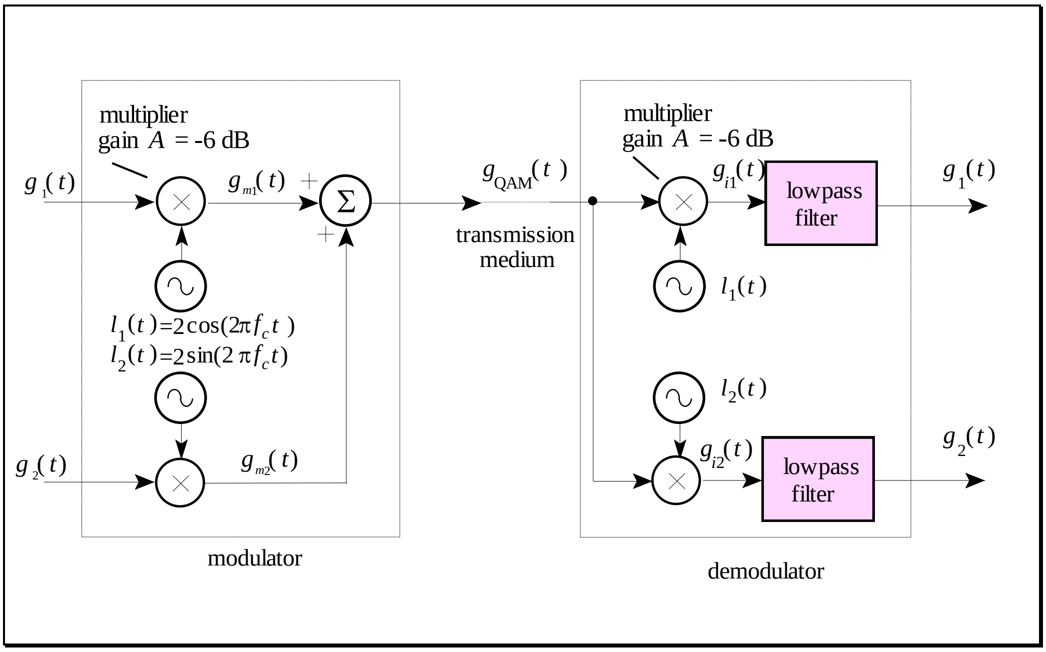
The first stage of the demodulation process involves applying the modulated waveform $g_{\text{QAM}}(t)$ to two separate multipliers. The other signals applied to each multiplier are local oscillators (in quadrature again) which are assumed to be synchronized with the modulator, i.e. the frequency and phase of the demodulator's local oscillators are exactly the same as the frequency and phase of the modulator's local oscillators. The signals $g_1(t)$ and $g_2(t)$ can be "extracted" from the output of each multiplier by lowpass filtering.

Pre-Lab Work – Hand Analysis [1 mark]

1. Perform a theoretical analysis by hand on the modulation / demodulation scheme shown below. Sketch time-domain waveforms and spectra (magnitude and phase) at each point in the system. The signal $g_1(t)$ is a $2\,\text{kHz}$ sinusoid with an amplitude of $2\,\text{V}$, and the signal $g_2(t)$ is a $6\,\text{kHz}$ sinusoid with an amplitude of $2\,\text{V}$. The modulator's local oscillator $l_1(t)$ is a $100\,\text{kHz}$ sinusoid, with an amplitude of $2\,\text{V}$.

   Note that the multipliers have a gain of –6 dB.

   Determine the magnitude responses of the lowpass filters to achieve ideal demodulation of the original signal.



Modulation and coherent demodulation scheme

Figure L4.4

<u>Attach your hand analysis as part of your pre-lab work .</u>

Pre-Lab Work – MATLAB® Simulation [1 mark]

2. Perform a MATLAB® simulation of the modulation / demodulation scheme, showing ***ALL*** signals as both a **time-domain waveform** and a **magnitude spectrum**:

   - Choose a MATLAB® "sample rate" of `fs=500e3` and choose `N=16384` samples.
   - The signal $g_1(t)$ is a $2\,\text{kHz}$ sinusoid with an amplitude of 2.
   - The signal $g_2(t)$ is a $6\,\text{kHz}$ sinusoid with an amplitude of 2.
   - The local oscillator $l_1(t)$ is a sinusoid with frequency $f_c = 100$ kHz and amplitude of $2\,\text{V}$.
   - The lowpass filter is an Elliptic filter with a cutoff frequency of 60 kHz. It can be created with the following MATLAB® *ellip* function:

     ```
     [b,a]=ellip(5,0.1,50,60000/(fs/2));
     ```

     The vectors *b* and *a* can be used with the MATLAB® *filter* function. You may need to adjust the magnitude of the filter response to achieve the correct amplitude in the reconstruction of the signal.
   - All time-domain waveforms should extend from 0 to 0.5 milliseconds, with a range of –4 to 4.
   - All magnitude spectra should extend from 0 to 250 kHz, with a range of –70 dB to 10 dB.

Attach a hardcopy of the MATLAB® simulation and code as part of your pre-lab work.

Pre-Lab Work - TIMS Wiring Diagram [0.5 marks]

In the lab we will demodulate each message individually rather than simultaneously (this is due to the limitations of the TIMS multipliers – specifically, they introduce an additional phase shift that needs to be compensated for in the demodulator). We will therefore implement the demodulator as:

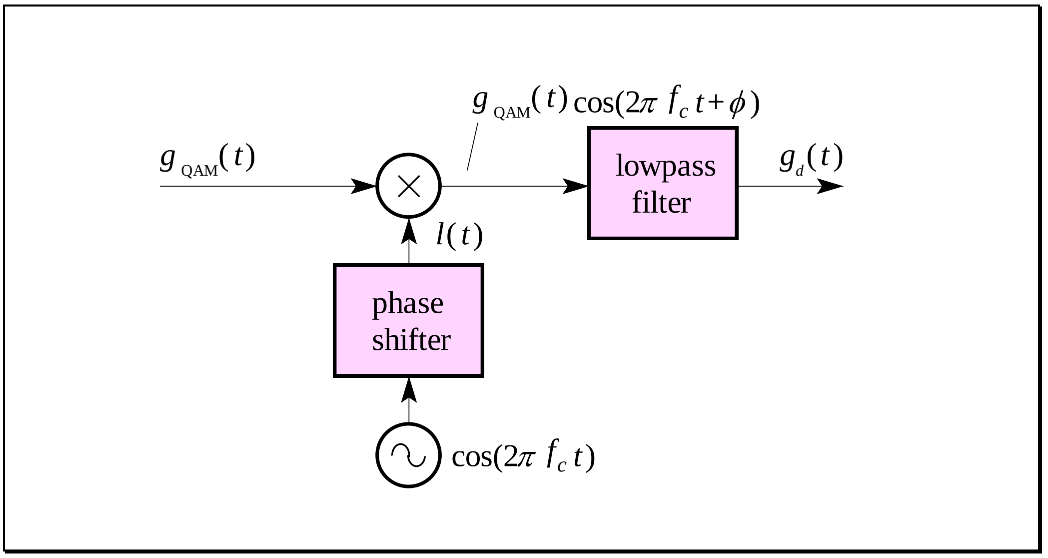


Figure L4.5

The theoretical output of the demodulator will be:

$$g_d(t) = g_1(t)\cos\phi + g_2(t)\sin\phi$$

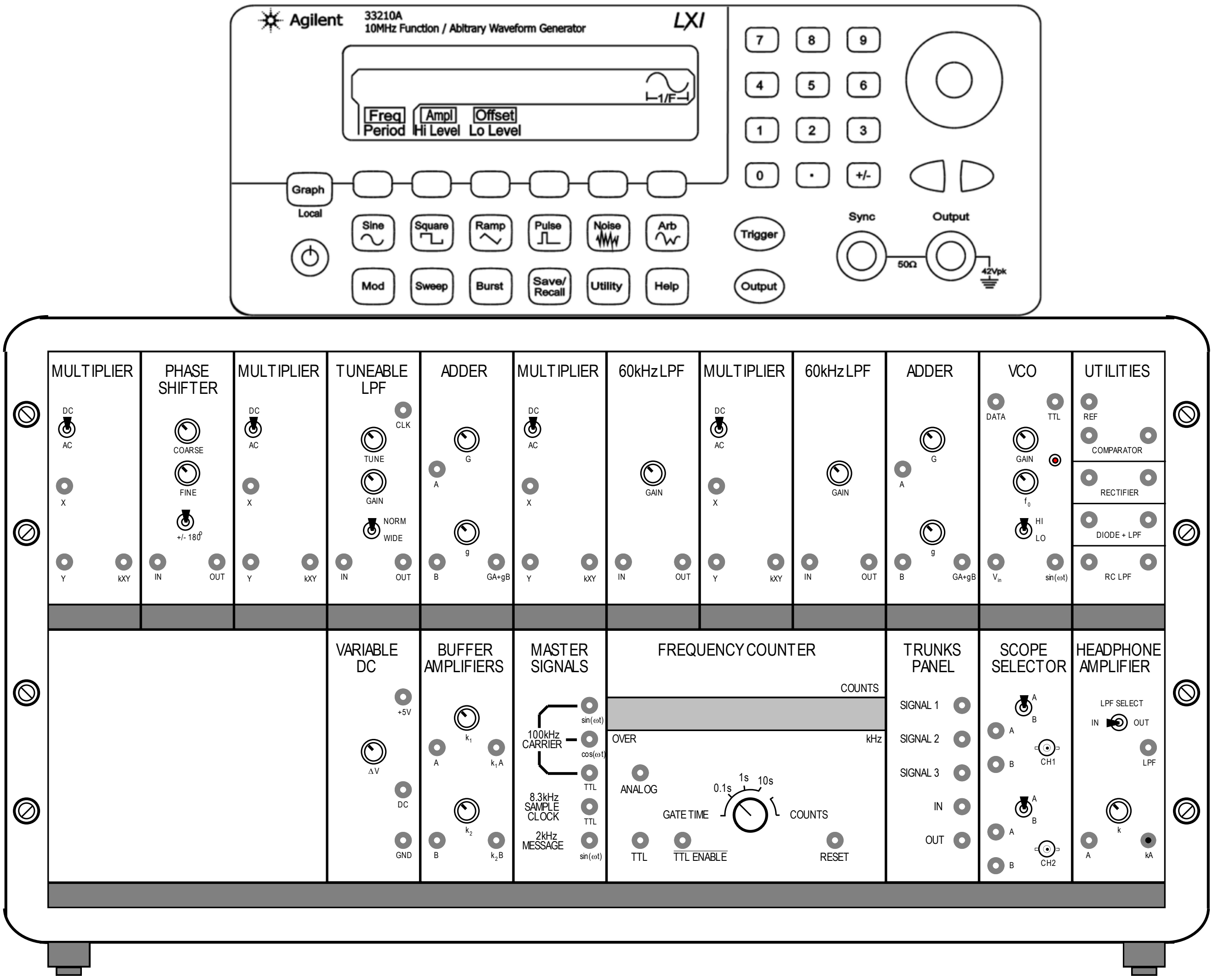
Thus, if we set the phase shifter to 0°, then we will demodulate $g_1(t)$, and if we set it to -90°, we will demodulate $g_2(t)$. In practice, the phase needs to be set to compensate for the TIMS multipliers as well.

If the phase is not set to obtain either of the message signals alone (e.g. -45°) then the output of the demodulator will be a “mixture” of the two message signals.

3. Construct a wiring diagram for the TIMS trainer that implements the modulation scheme shown in Figure L4.4 and the demodulation scheme shown in Figure L4.5.

    - Use the 100 kHz CARRIER outputs, located in the MASTER SIGNALS module, for the local oscillators $l_1(t)$ and $l_2(t)$.
    - Use the 2 kHz MESSAGE output for $g_1(t)$, and the AWG for $g_2(t)$.
    - Use the 60 kHz lowpass filter for the demodulator filter, and set the gain manually to restore the demodulated waveform to its correct amplitude.

We need gain to correct for attenuations throughout the system.



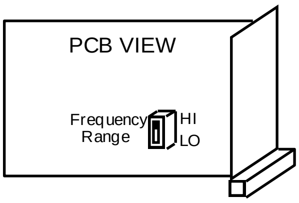
<u>Construct a TIMS wiring diagram on the above picture as part of your pre-lab work.</u>

Lab Work [0.5 mark]

1. Wire up the modulation and demodulation scheme, following your TIMS wiring diagram from the pre-lab work.

2. To minimise noise, connect the AWG black lead to the TIMS GND terminal.

3. Set the Phase Shifter to the “HI” range by pulling the module out of the TIMS and setting the Frequency Range to HI. Reinsert the module.



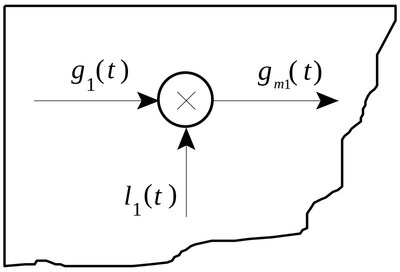
4. Adjust the gain of the 60 kHz lowpass filter to achieve the correct amplitude of the demodulated sinusoid.

5. On the following graphs, sketch the time-domain waveform and the corresponding magnitude spectrum.
   Choose scales that correspond to your MATLAB® simulations.
   Use the following settings for the FFT:

Sample rate	Freq Span	Center Freq	Window
500 kSa/s	250 kHz	125 kHz	Hanning

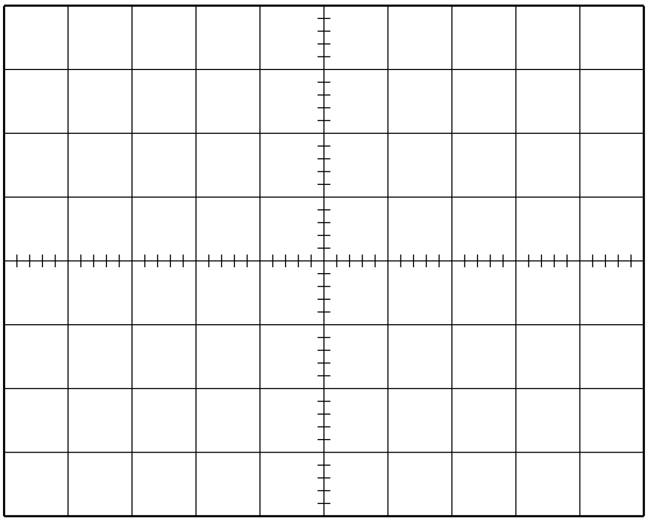
Set the Offset of the FFT to ≈ -30 dB, and the Scale to 10 dB/div.

L4.10



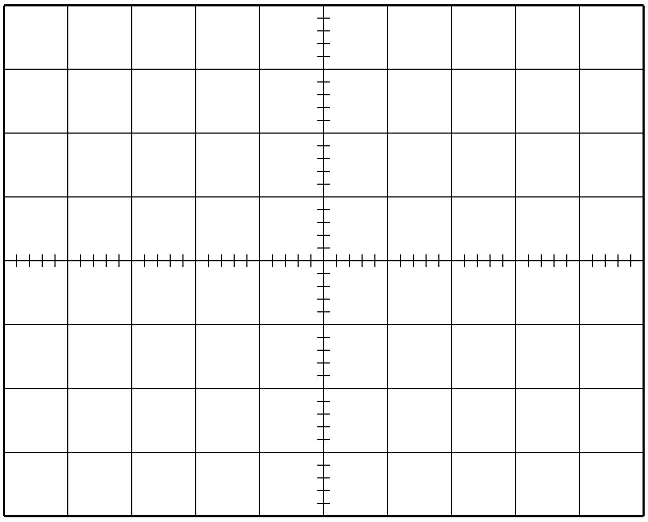
$g_1(t)$

Time-domain view of sinusoid

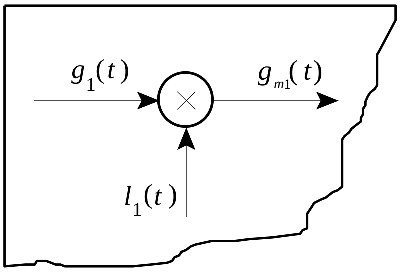


$|G_1(f)|$

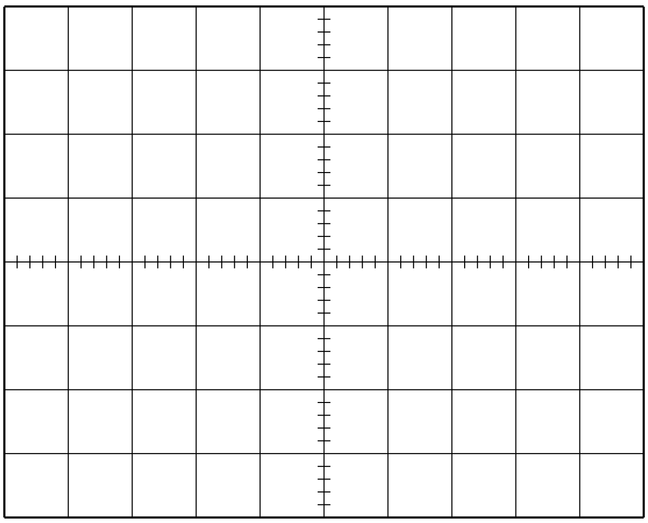
Magnitude spectrum of sinusoid



L4.11

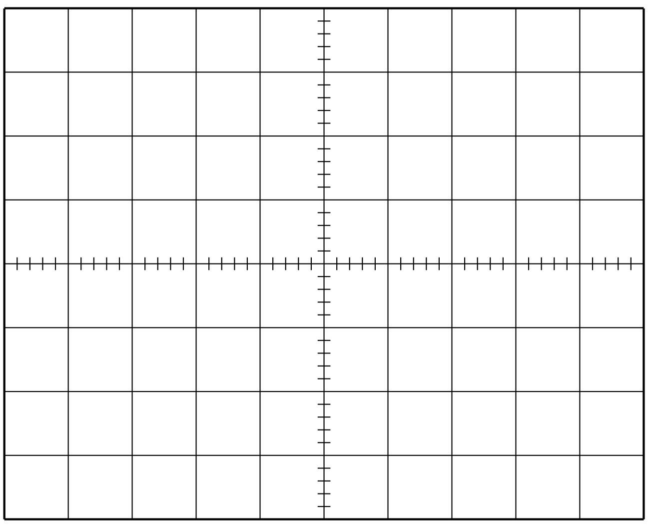


$g_{m1}(t)$



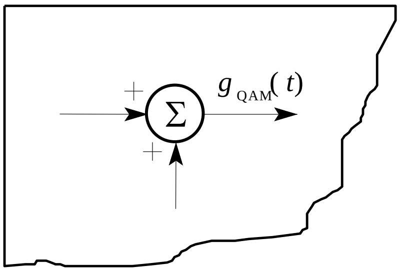
Time-domain view of the modulator's multiplier output

$|G_{m1}(f)|$



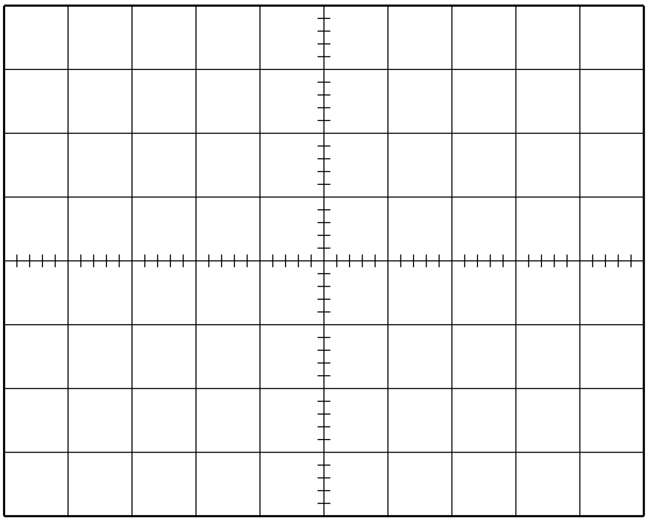
Magnitude spectrum of the modulator's multiplier output

L4.12



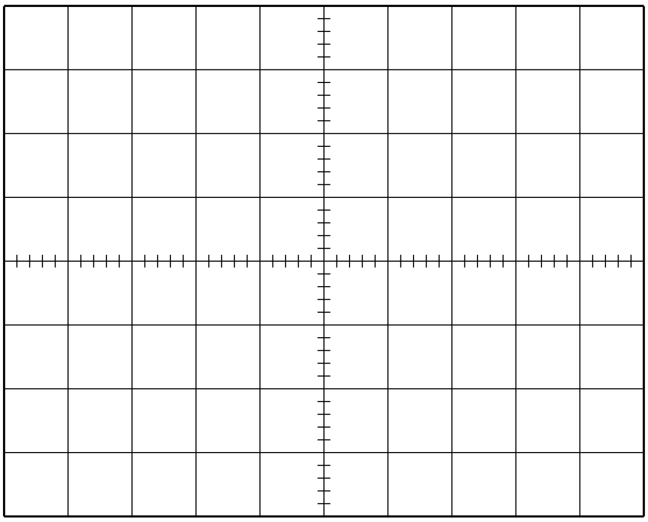
$g_{\text{QAM}}(t)$

Time-domain view
of QAM signal

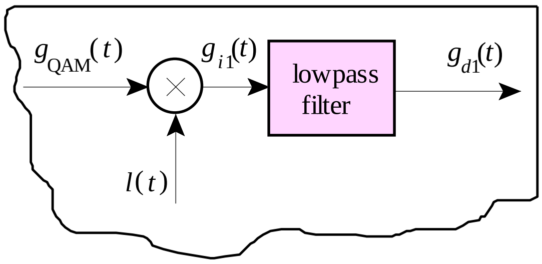


$\left|G_{\text{QAM}}(f)\right|$

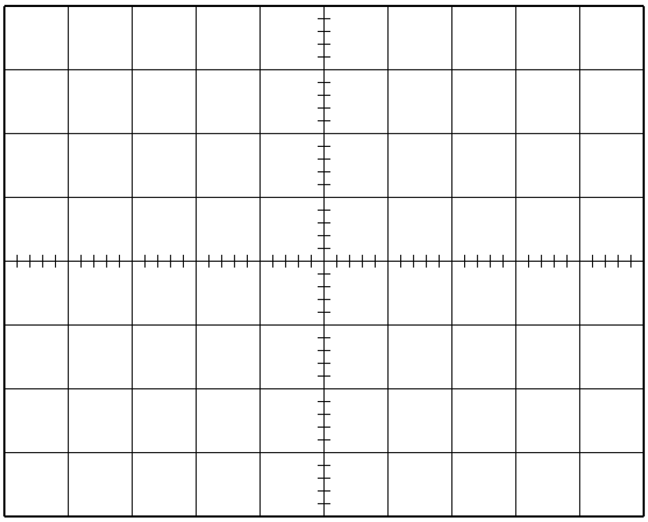
Magnitude spectrum
of QAM signal



6. Observe the output of the demodulator in the time-domain. Vary the phase of the local oscillator with the Phase Shifter to demodulate $g_1(t)$. Observe the signal at the **INPUT** of the lowpass filter.

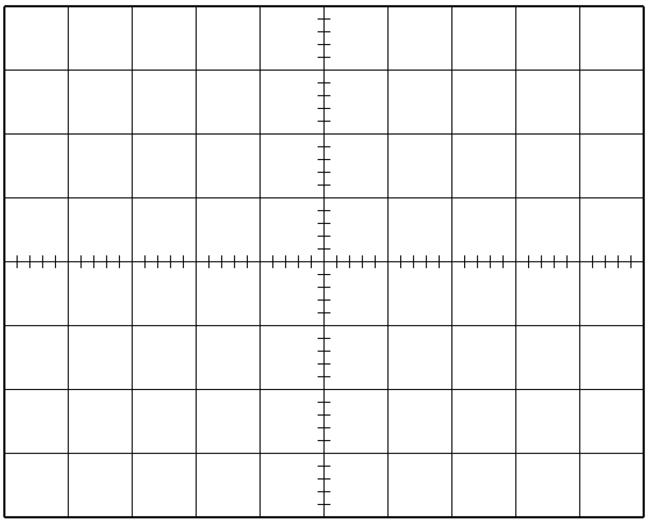


$g_{i1}(t)$



Time-domain view of QAM signal after multiplication with a local oscillator

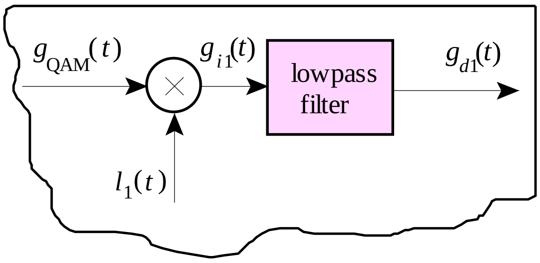
$|G_{i1}(f)|$



Magnitude spectrum of QAM signal after multiplication with a local oscillator

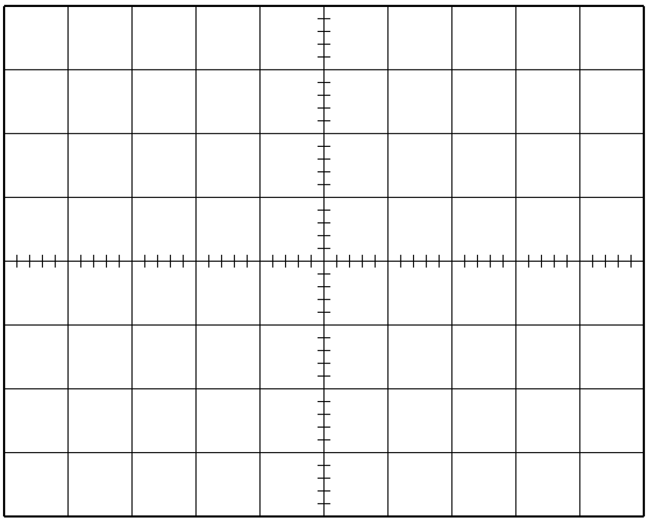
L4.14

7. Observe the signal at the **OUTPUT** of the lowpass filter.



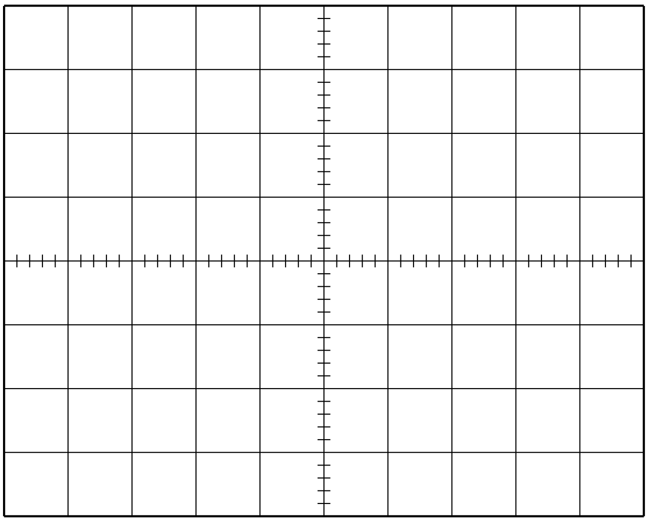
Time-domain view of demodulated signal

$g_{d1}(t)$

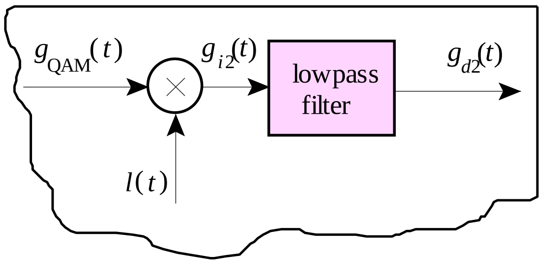


Magnitude spectrum of demodulated signal

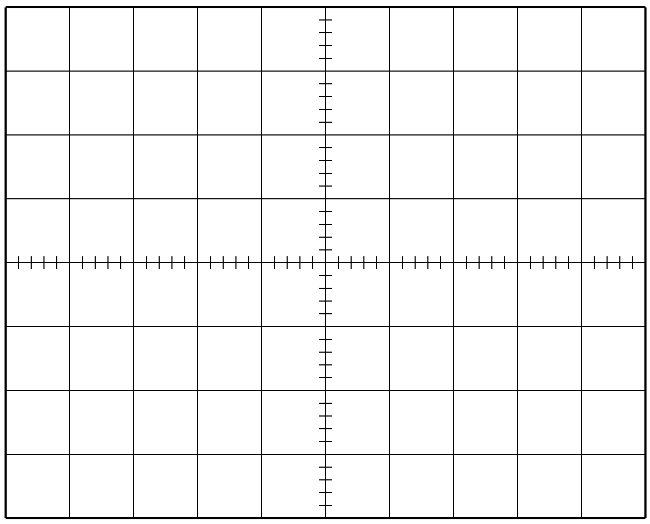
$|G_{d1}(f)|$



8. Observe the output of the demodulator in the time-domain. Vary the phase of the local oscillator with the Phase Shifter to demodulate $g_2(t)$. Observe the signal at the **INPUT** of the lowpass filter.

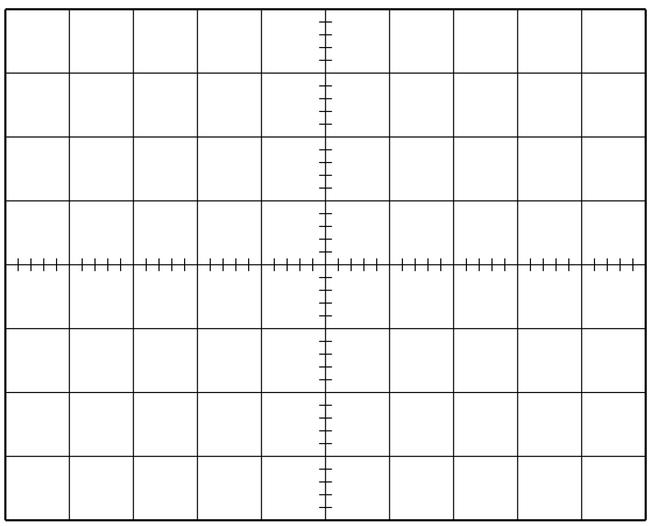


$g_{i2}(t)$



Time-domain view of QAM signal after multiplication with a local oscillator

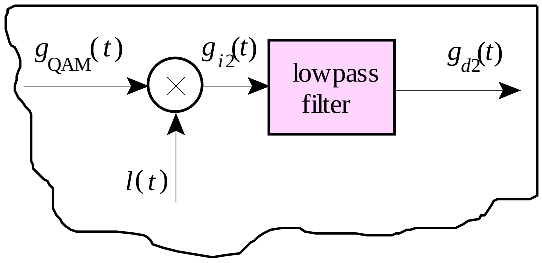
$|G_{i2}(f)|$



Magnitude spectrum of QAM signal after multiplication with a local oscillator

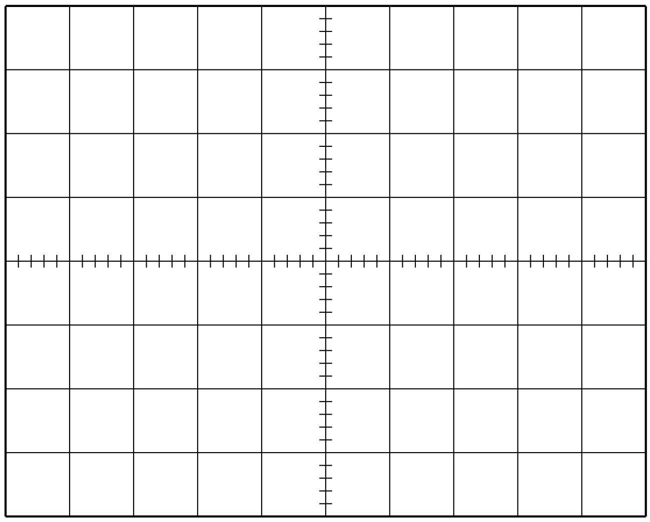
L4.16

9. Observe the signal at the **OUTPUT** of the lowpass filter.



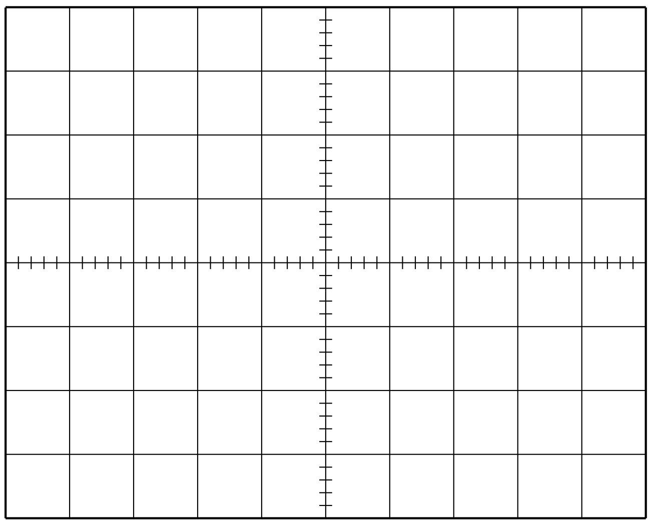
$g_{d2}(t)$

Time-domain view of demodulated signal



$|G_{d2}(f)|$

Magnitude spectrum of demodulated signal

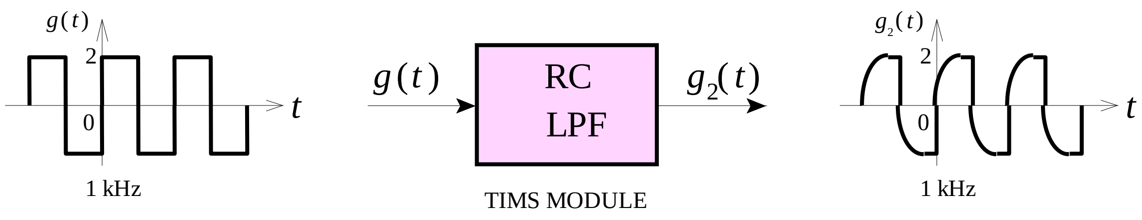


Sharing the Spectrum

1. Set the horizontal scale to 200 μs/div.

2. Practice using the Coarse knob of the phase shifter to extract either the 2 kHz message signal $g_1(t)$ or the 6 kHz message signal $g_2(t)$.

3. Now create a new $g_2(t)$ message signal by implementing the following:



   The new $g_2(t)$ message is therefore a square wave that has been "bandlimited" – i.e. it's high frequency components have been attenuated to such a degree that the resulting signal can be said to have finite bandwidth. Note that $g_2(t)$ has a fundamental frequency of 1 kHz and higher harmonics. It's spectrum therefore "overlaps" the spectrum of $g_1(t)$.

4. Check that the QAM scheme still works, even when the two message signals have similar spectra, i.e. use the Coarse knob of the phase shifter to extract either the message signal $g_1(t)$ or the message signal $g_2(t)$ (you will observe a strange wave shape when the demodulator is mixing the two signals).

L4.18

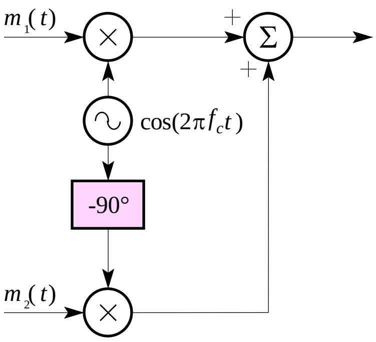
Questions [1 mark]

Encircle the correct answer, cross out the wrong answers. [one or none correct]

All questions are worth 0.1 marks each.

1. Modulation

(i)



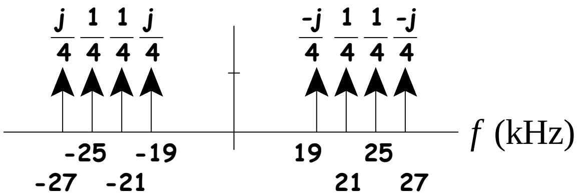
$f_c = 23$ kHz

$m_1(t) = \cos(2000\pi t)$

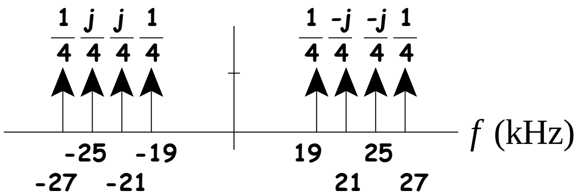
$m_2(t) = \cos(4000\pi t)$

The output spectrum is:

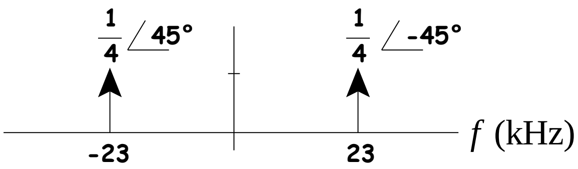
(a)



(b)



(c)



(ii)

In QAM modulation, message signal variations are evident in the carrier's:

(a) amplitude only (b) phase only (c) amplitude and phase

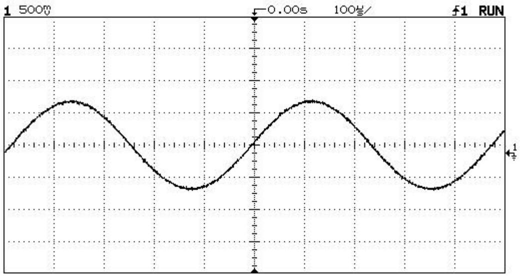
(iii)

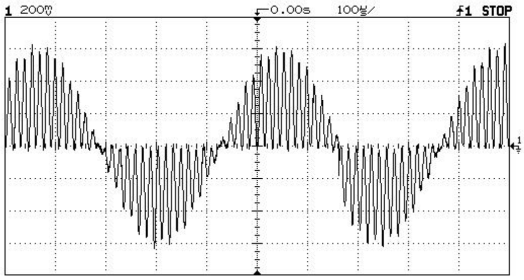
In QAM modulation, if the phases of the demodulator local oscillators are -90° out of phase with respect to the ideal, then the "message 1" output will be:

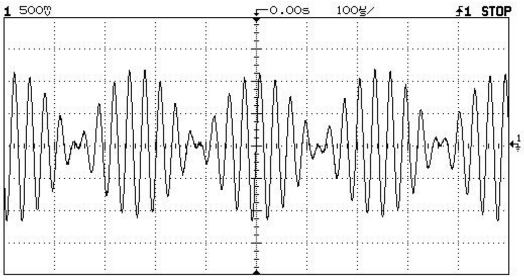
(a) zero (b) message 2 (c) inverted message 2

(iv)

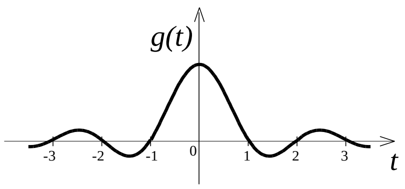
A 2 kHz sinusoid is applied as one message signal to a QAM modulator with a 33 kHz local oscillator. The other message signal is zero. The resulting time-domain output looks like:

(a)


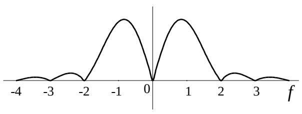
(b)


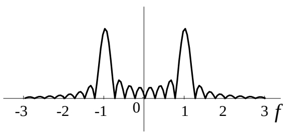
(c)


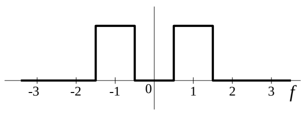
(v)



The magnitude spectrum after multiplying the signal $g(t)$ by $\sin(2\pi t)$ is:

(a)


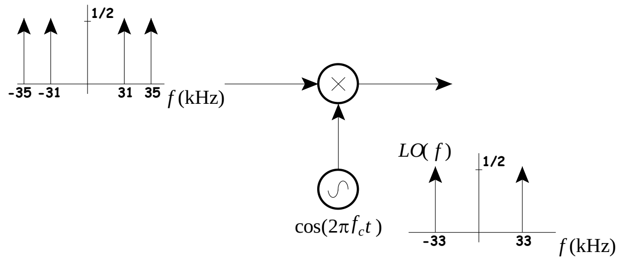
(b)


(c)


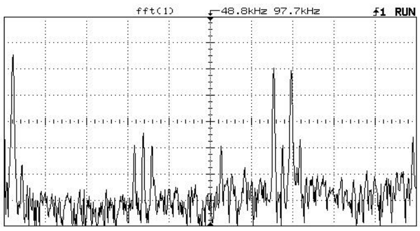
L4.20

2. Demodulation

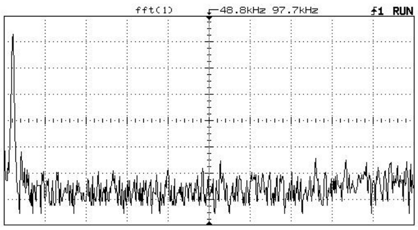
(i)



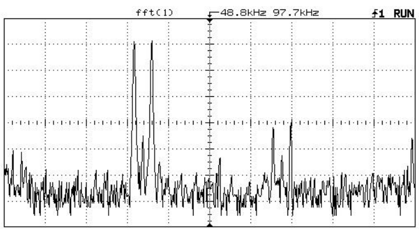
The output magnitude spectrum resembles (100 kHz frequency span):



(a)



(b)



(c)

(ii)

A local oscillator in a QAM coherent demodulation scheme is out of phase with the modulator's local oscillator by -45° (the frequency is the same). The two modulator input messages are a sinusoid with a power of 0 dB, and zero. One of the demodulator's outputs will be a sinusoid with a power of:

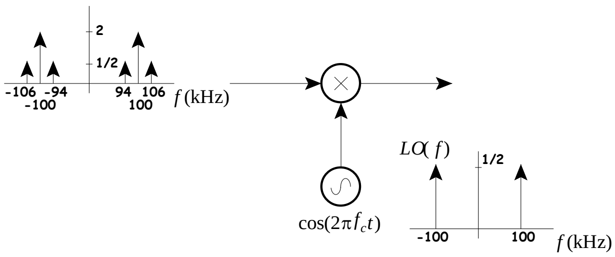
(a) 0 dB (b) –20 dB (c) –3 dB

(iii)

A 6 kHz sinusoid has been QAM modulated with a 23 kHz carrier. A coherent demodulator filter should span:

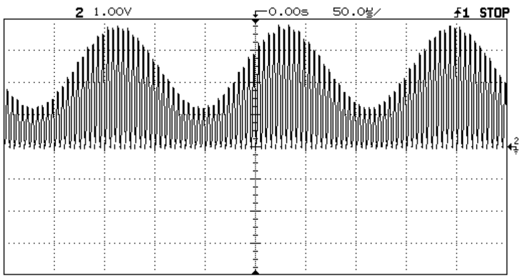
(a) 0 kHz to 23 kHz (b) 6 kHz to 44 kHz (c) 0 kHz to 46 kHz

(iv)

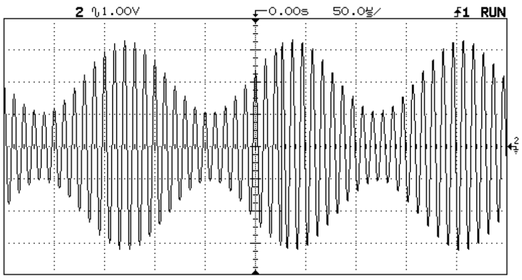


The output in the time-domain looks like:

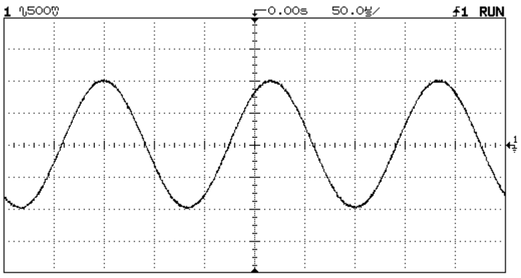
(a)



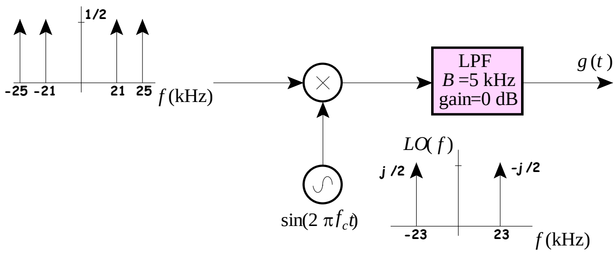
(b)



(c)



(v)



The magnitude of the resulting sinusoid is:

(a) 1/4 (b) 1/2 (c) zero

<u>Complete the questions as part of your lab report.</u>

Report

Only submit ***ONE*** report per lab group.

Complete the assignment cover sheet and attach your pre-lab work.

Ensure you have completed:

1. ***Pre-Lab Work*** – hand analysis, MATLAB® simulation and TIMS wiring diagram.
2. ***Lab Work*** – sketches of the time-domain waveforms and magnitude spectra at each point in the system.
3. ***Post-Lab Work*** – multiple choice questions.

The lab report is due on the date specified in the Learning Guide.

You should hand the report directly to your tutor.

University of Technology, Sydney
Faculty of Engineering and Information Technology

Subject:	**48540 Signals and Systems**
Assessment Number:	**5**
Assessment Title:	**Lab 5 – Second-Order Systems**
Tutorial Group:	

Students Name(s) and Number(s)

Student Number	Family Name	First Name

Declaration of Originality:

The work contained in this assignment, other than that specifically attributed to another source, is that of the author(s). it is recognised that, should this declaration be found to be false, disciplinary action could be taken and the assignments of all students involved will be given zero marks. In the statement below, I have indicated the extent to which I have collaborated with other students, whom I have named.

Statement of Collaboration:

Signature(s)

Marks

Hand Analysis	/1
MATLAB® Simulation	/1
Lab Work	/0.5
MATLAB® Verification	/0.5
Questions	/1
TOTAL	/4

Office use only ☺

key

Assessment Submission Receipt

Assessment Title:	**Lab 5 – Second-Order Systems**
Student's Name:	
Date Submitted:	
Tutor Signature:	

Lab 5 – Second-Order Systems

Effect of pole positions. Non-linear effects. Frequency responses and step responses of second-order circuits.

Introduction

Control systems are mostly designed to achieve time-domain specifications. Time-domain specifications are important because we usually want the output to track some reference input. We are interested in the transient response (How quick is it?) and the steady-state response (How accurate is it?).

We usually set certain system parameters to achieve a desired time-domain response – this is control system design. Second-order system specifications are often given, because many control systems can be approximated by second-order systems.

Communication systems are mostly designed to meet frequency-domain specifications. Frequency-domain specifications are important because we usually think of communication systems *processing* the information in signals, which are made up of sinusoids of different frequency, amplitude and phase. For example, when designing filters we want to pass certain frequencies and attenuate others. It is natural to think of these systems in the frequency-domain.

The time-domain and frequency-domain are related – they are just two ways to look at the same system. Insight in one domain will lead to a better understanding in the other.

Objectives

1. To realise that *s*-domain block diagrams model the *real* world – an electronic circuit is just one convenient example.

2. To become familiar with the time-domain and frequency-domain responses of a variety of second-order transfer functions.

L5.2

Equipment

- 1 Digital Storage Oscilloscope (DSO) – Agilent DSO-X 2004A with Wave Gen
- 1 Arbitrary Waveform Generator (AWG) – Agilent 33210A with Option 002
- 1 state variable filter
- 4mm leads (assorted colours), BNC to 4mm adaptors

Note

Quality!!!

In this lab, "draw" means to make an accurate recording – one showing times, and amplitudes as accurately as possible – this is the only way to interpret results after leaving the lab. Quick sketches are not acceptable – and are almost certainly useless when it comes to tying up theory with practice.

"Sketch" means to quickly give an overview, but showing important features.

Safety

Cat. A lab

This is a Category A laboratory experiment. Please adhere to the Category A safety guidelines (issued separately).

Pre-Lab Work – Hand Analysis [1 mark]

Op-Amp Circuit Analysis using Block Diagrams

1. Show that the circuit:

First-order circuit

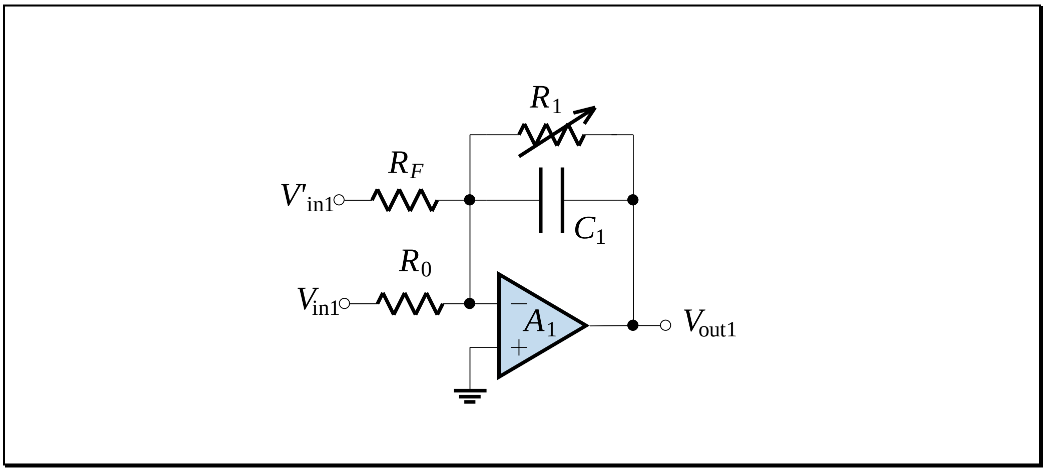


Figure L5.1

can be represented by the block diagram:

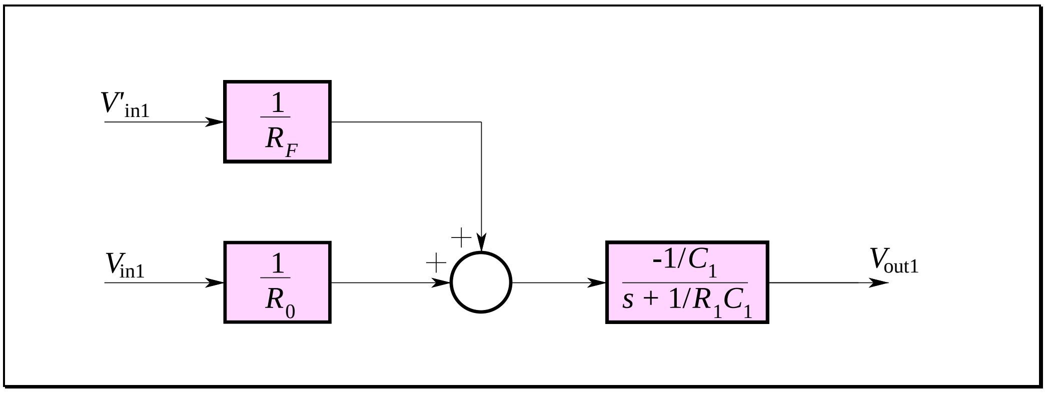


Figure L5.2

L5.4

Second-Order Universal Circuit

Second-order circuit with almost arbitrary pole locations

2. Construct a block diagram of the following circuit:

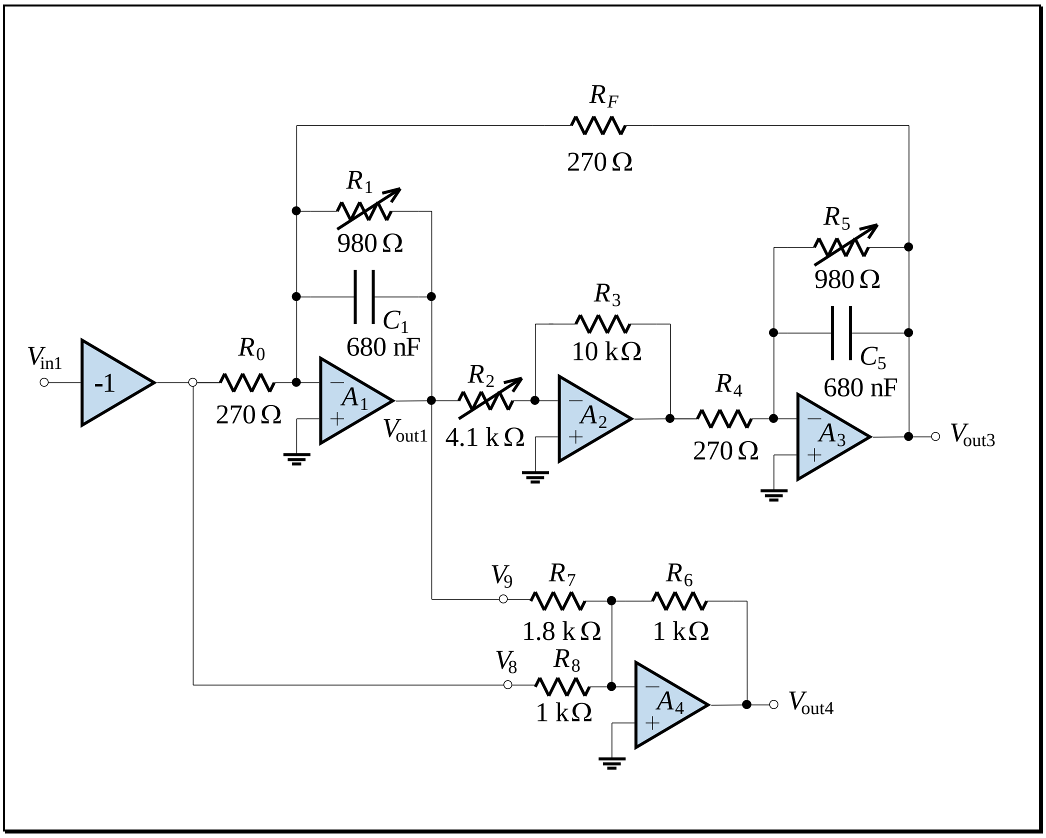


Figure L5.3

Hence show that the transfer function $T_1(s) = V_{out3}(s)/V_{in1}(s)$ is:

$$T_1(s) = \frac{R_3/R_0R_2R_4C_1C_5}{s^2 + \left(\frac{1}{R_1C_1} + \frac{1}{R_5C_5}\right)s + \left(\frac{R_3}{R_2R_4R_FC_1C_5} + \frac{1}{R_1R_5C_1C_5}\right)} \qquad \text{(L5.1)}$$

3. By comparing Eq. (L5.1) with the standard form of a second-order all-pole transfer function,

$$T_1(s) = \frac{K_1\omega_n^2}{s^2 + 2\alpha s + \omega_n^2}\,, \tag{L5.2}$$

write expressions for K_1, α and ω_n in terms of *R*'s and *C*'s.

4. By completing the square in the denominator, and defining $\omega_d = \sqrt{\omega_n^2 - \alpha^2}$, show that the poles of the transfer function are:

$$p_{1,2} = -\alpha \pm j\omega_d \tag{L5.3}$$

5. For the *special* case of when $R_1 = R_5$, $C_1 = C_5$ and $R_F = R_4$, show that:

$$\alpha = \frac{1}{R_1 C_1} \qquad \omega_d = \sqrt{\frac{R_3}{R_2}\frac{1}{R_4 C_1}} \tag{L5.4}$$

6. For the *special* case of when $R_1 = R_5$, $C_1 = C_5$ and $R_F = R_4$, which circuit element(s) may be used to set the real part of the poles (ie. α) without affecting the imaginary part (ie. ω_d)?

7. For the *special* case of when $R_1 = R_5$, $C_1 = C_5$ and $R_F = R_4$, which circuit element(s) may be used to set the imaginary part of the poles (ie. ω_d) without affecting the real part (ie. α)?

L5.6

Second-Order Circuit with a Real Zero

8. From your block diagram of Figure L5.3, and for the *special* case of when $R_1 = R_5$, $C_1 = C_5$ and $R_F = R_4$, show that the transfer function $T_2(s) = V_{\text{out1}}(s)/V_{\text{in1}}(s)$ can be put in the form:

$$T_2(s) = \frac{K_2(s+\alpha)}{s^2 + 2\alpha s + \omega_n^2} \tag{L5.5}$$

Second-Order Circuit with an Imaginary Zero Pair

9. The SVF's summing amplifier is used to add (with appropriate gain) the voltages V_{in1} and V_{out1}.

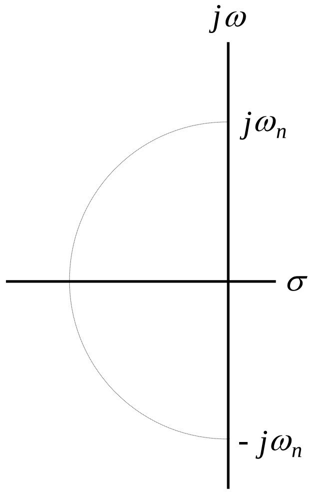
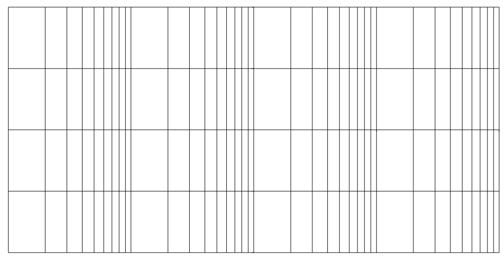
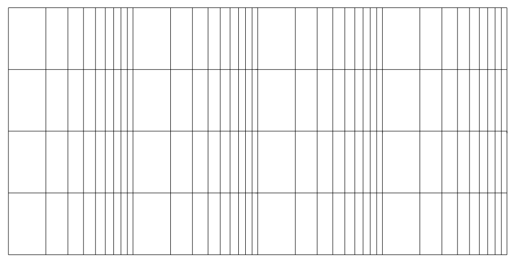
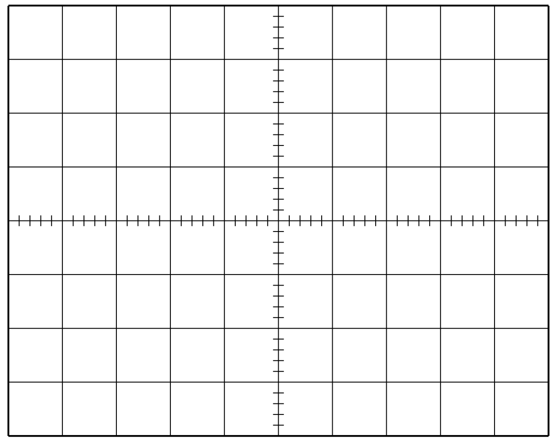
   Show that, for the *special* case of when $R_1 = R_5$, $C_1 = C_5$, $R_F = R_4$, $R_6 = R_8$ and $\frac{2R_0}{R_1} = \frac{R_8}{R_7}$, the transfer function $T_3(s) = V_{\text{out4}}(s)/V_{\text{in1}}(s)$ can be written as:

$$T_3(s) = \frac{K_3\left(s^2 + \omega_n^2 - 2\alpha^2\right)}{s^2 + 2\alpha s + \omega_n^2} \tag{L5.6}$$

Second-Order Circuit Summaries

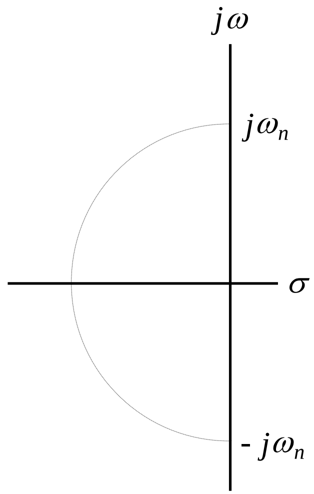
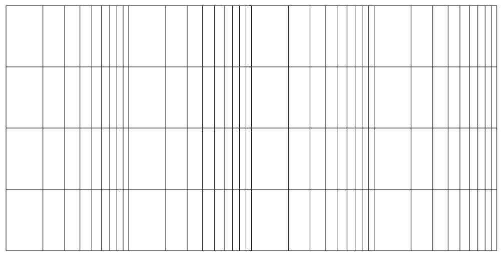
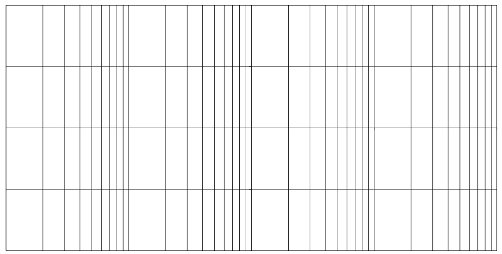
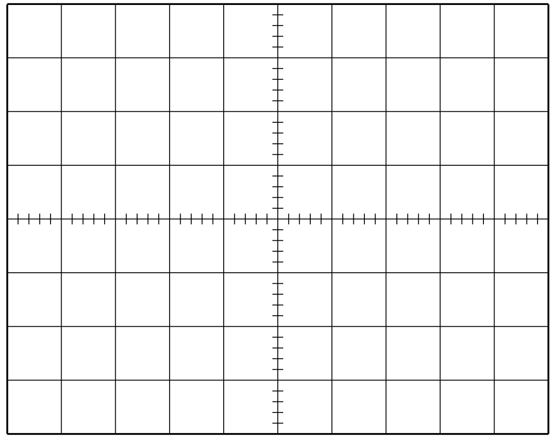
10. For the element values shown in Figure L5.3, complete the following tables (provide numerical values and sketch appropriate graphs).

Second-Order All-Pole Circuit

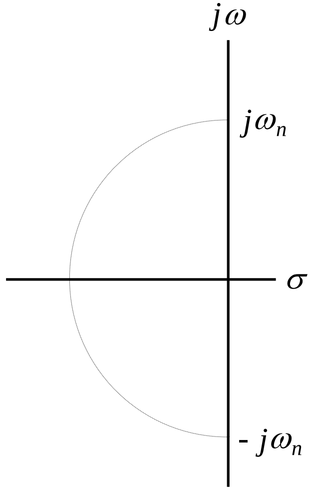
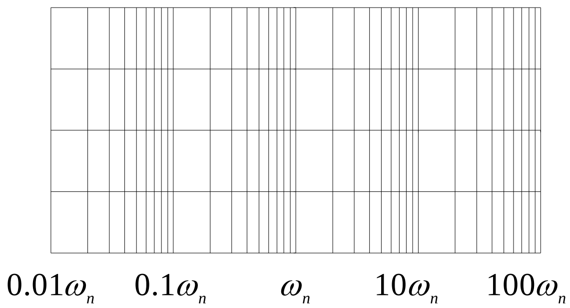
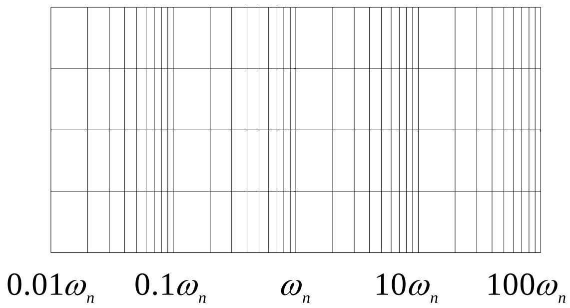
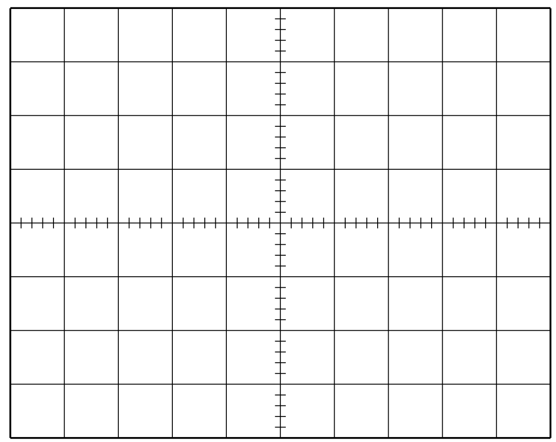
Transfer Function	Pole-zero Plot	Asymptotic Bode Plots	Step Response
$T_1(s)=\dfrac{K_1\omega_n^2}{s^2+2\alpha s+\omega_n^2}$ <u>DC Gain</u> $K_1 =$ <u>Pole locations</u> $-\alpha + j\omega_d =$ $-\alpha - j\omega_d =$	 	Magnitude Response (dB)  Phase response (°)  	

L5.8

Second-Order Circuit with a Real Zero

Transfer Function	Pole-zero Plot	Asymptotic Bode Plots	Step Response
$T_2(s) = \frac{K_2(s+\alpha)}{s^2 + 2\alpha s + \omega_n^2}$ <u>DC Gain</u> $\frac{K_2 \alpha}{\omega_n^2} =$ <u>Pole locations</u> $-\alpha + j\omega_d =$ $-\alpha - j\omega_d =$ <u>Zero location</u> $-\alpha =$		Magnitude Response (dB)  $0.01\omega_n$ $0.1\omega_n$ ω_n $10\omega_n$ $100\omega_n$ Phase response (°)  $0.01\omega_n$ $0.1\omega_n$ ω_n $10\omega_n$ $100\omega_n$	

Second-Order Circuit with an Imaginary Zero Pair

Transfer Function	Pole-zero Plot	Asymptotic Bode Plots	Step Response
$T_3(s) = \dfrac{K_3\left(s^2 + \omega_n^2 - 2\alpha^2\right)}{s^2 + 2\alpha s + \omega_n^2}$ DC Gain $\dfrac{K_3\left(\omega_n^2 - 2\alpha^2\right)}{\omega_n^2} =$ Pole locations $-\alpha + j\omega_d =$ $-\alpha - j\omega_d =$ Zero locations $+j\sqrt{\omega_n^2 - 2\alpha^2} =$ $-j\sqrt{\omega_n^2 - 2\alpha^2} =$		Magnitude Response (dB)  Phase response (°) 	

Pre-Lab Work – MATLAB® Simulation [1 mark]

11. Perform **frequency responses** and **unit-step responses** of the state variable filter at the labelled voltage outputs shown in Figure L5.3.

- Use the following tables for the response scales:

$T_1(s)=\dfrac{V_{\text{out3}}(s)}{V_{\text{in1}}(s)}$	Horizontal axis	Vertical axis
Magnitude Response	$10\text{ Hz} \le f \le 100\text{ kHz}$	$-50\,\text{dB} \le 20\log\lvert T_1\rvert \le 10\,\text{dB}$
Phase Response	$10\text{ Hz} \le f \le 100\text{ kHz}$	$-180° \le \angle T_1 \le 0°$
Step Response	$-0.5\,\text{ms} \le t \le 4.5\,\text{ms}$	$0\,\text{V} \le v_{\text{out3}} \le 1.6\,\text{V}$

$T_2(s)=\dfrac{V_{\text{out1}}(s)}{V_{\text{in1}}(s)}$	Horizontal axis	Vertical axis
Magnitude Response	$10\text{ Hz} \le f \le 100\text{ kHz}$	$-30\,\text{dB} \le 20\log\lvert T_2\rvert \le 10\,\text{dB}$
Phase Response	$10\text{ Hz} \le f \le 100\text{ kHz}$	$-100° \le \angle T_2 \le 100°$
Step Response	$-0.5\,\text{ms} \le t \le 4.5\,\text{ms}$	$-0.2\,\text{V} \le v_{\text{out1}} \le 0.8\,\text{V}$

$T_3(s)=\dfrac{V_{\text{out4}}(s)}{V_{\text{in1}}(s)}$	Horizontal axis	Vertical axis
Magnitude Response	$10\text{ Hz} \le f \le 100\text{ kHz}$	$-30\,\text{dB} \le 20\log\lvert T_3\rvert \le 10\,\text{dB}$
Phase Response	$10\text{ Hz} \le f \le 100\text{ kHz}$	$-100° \le \angle T_3 \le 100°$
Step Response	$-0.5\,\text{ms} \le t \le 4.5\,\text{ms}$	$0\,\text{V} \le v_{\text{out4}} \le 1.2\,\text{V}$

Bring print-outs of your frequency responses and step responses and MATLAB® code to the lab as part of your pre-lab work.

Lab Work [0.5 mark]

Second-order All-Pole Step Response – Effect of Pole Positions

We're just making a "step" using a square wave

1. Set the AWG to generate a 0 V to 1 V, 50 Hz square wave.

2. Examine the circuit in Figure L5.3 to determine its input impedance. Now set the AWG output Load setting to match the input impedance of the circuit. This will ensure that the amplitude settings on the AWG display will match the actual physical voltage appearing at the AWG terminals.

3. Set up the SVF to give the circuit of Figure L5.3. To implement the inverter shown in Figure L5.3, use the DSO's "invert channel 2" function.

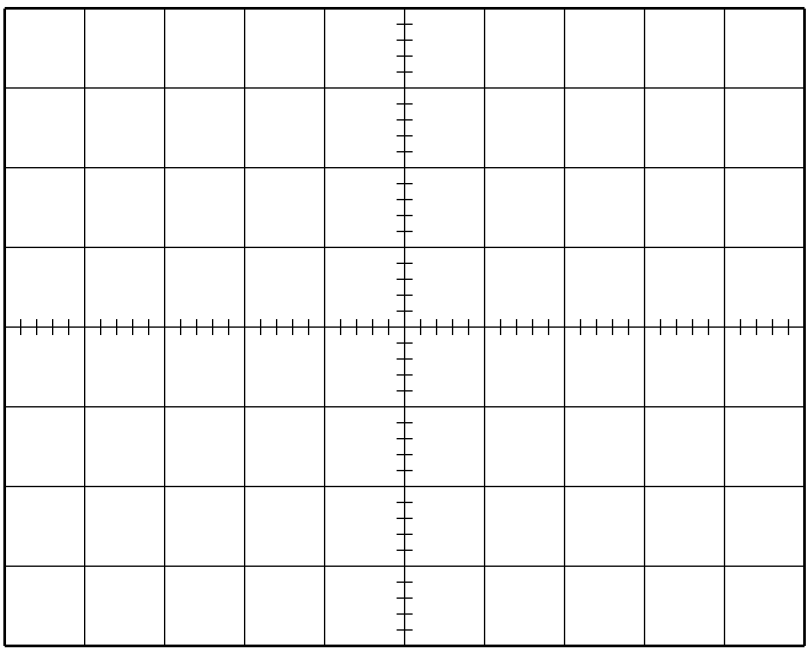
4. On the DSO, press `Horiz`. Set the `Time Ref` softkey to `Left`. This positions the trigger point on the oscilloscope one division from the left edge of the screen (which will maximise the view of the response).

5. Set the reference level (⏚) for the DSO channels near the bottom of the display. This will maximise the viewing area of the step response.

Second-Order All-Pole Step Response – Reference

6. Observe Vout3 on DSO channel 2. Use the following settings:

R_1	R_2	R_5	V/div	Time/div
097	409	097	250 m V	500 μs



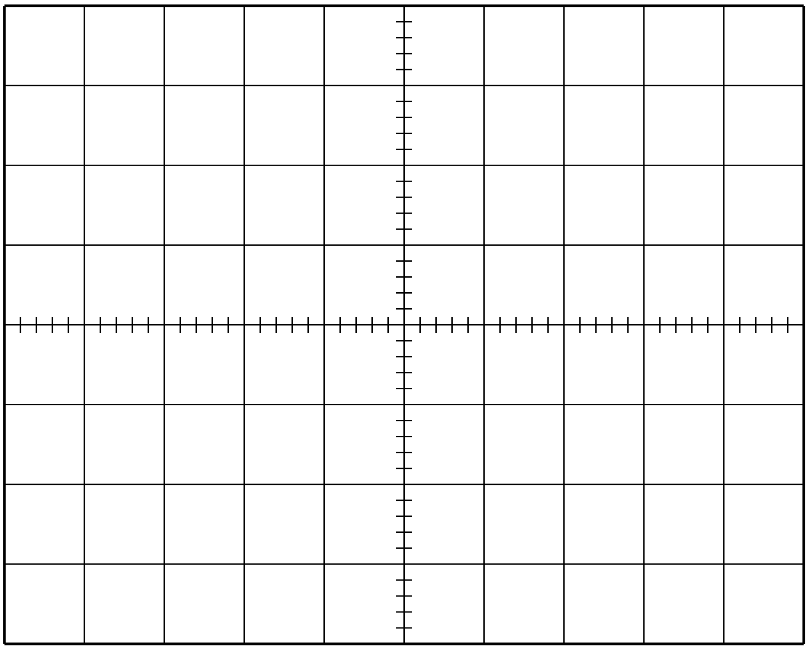
A "reference" step response for comparison purposes

Second-Order All-Pole Step Response – Varying α

Changing the real part of the complex poles

7. Since the system has complex conjugate poles, we're going to see the effect that changing the real part of the poles has on the response. Set:

R_1	R_2	R_5	V/div	Time/div
297	409	297	250 m V	500 μs

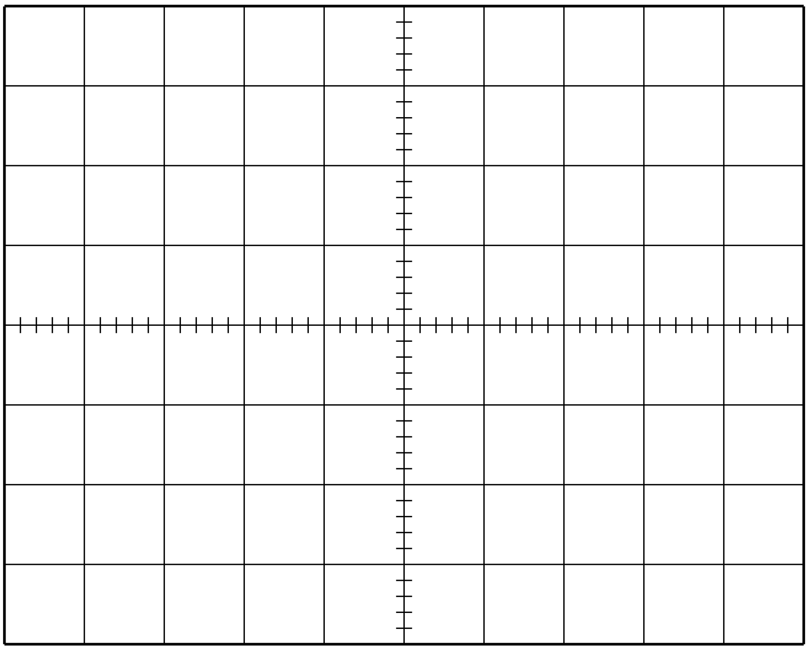


Second-Order All-Pole Step Response – Varying ω_d

Changing the imaginary part of the complex poles

8. Now we're going to change the imaginary part of the pole locations. Set the following:

R_1	R_2	R_5	V/div	Time/div
297	909	297	250 m V	500 μs



Non-linear Effects

The calculations in the pre-lab work were based upon a linear model of the op-amp. This model is deficient in *at least* one important aspect – real op-amps are linear only over certain voltage and current ranges. The effect of one or more of the op-amps connected in a feedback configuration and entering into saturation or current-limiting will be investigated.

Op-Amp Current-limiting Due to Small Resistance

1. Set:

R_1	R_2	R_5	V/div	Time/div
097	199	097	250 mV	500 μs

2. Set R_2 to 099, then 089, 079, 069… keep decreasing R_2 until a non-linear waveform results!!!
3. Remove the AWG from Vin1. There should still be an output, even though no input is being applied! Positive feedback is occurring, and the circuit is behaving as an oscillator.

Op-Amp Current-limiting Due to Large Input Signal

4. Turn off the SVF and set:

R_1	R_2	R_5	V/div	Time/div
297	409	297	2.0 V	500 μs

5. Set the AWG to a 1.25 kHz sinusoid with 0 V DC offset. Set the amplitude to 100 mVp-p and set 100 mV steps using the knob and cursor keys.
6. Set up the DSO to achieve a stable trigger with a reference level (⏚) at 0 V.
7. Turn the SVF on. Connect the AWG to Vin1. Gradually increase the amplitude until a non-linear waveform results! Decrease the amplitude.
8. Turn the AWG output off. The output may still be oscillating…
9. To stop the oscillation, return R_1 and R_5 to 097, or turn the SVF off.
10. Turn the amplitude of the AWG down. Turn the AWG output on.
    In future, keep $\hat{v} < 1\,\text{V}$!!!

In all subsequent use of the SVF, ensure that the input amplitude is chosen low enough to avoid oscillation.

L5.14

Second-Order All-Pole Circuit – V_{out3}

1. Set up the SVF to give the circuit of Figure L5.3.

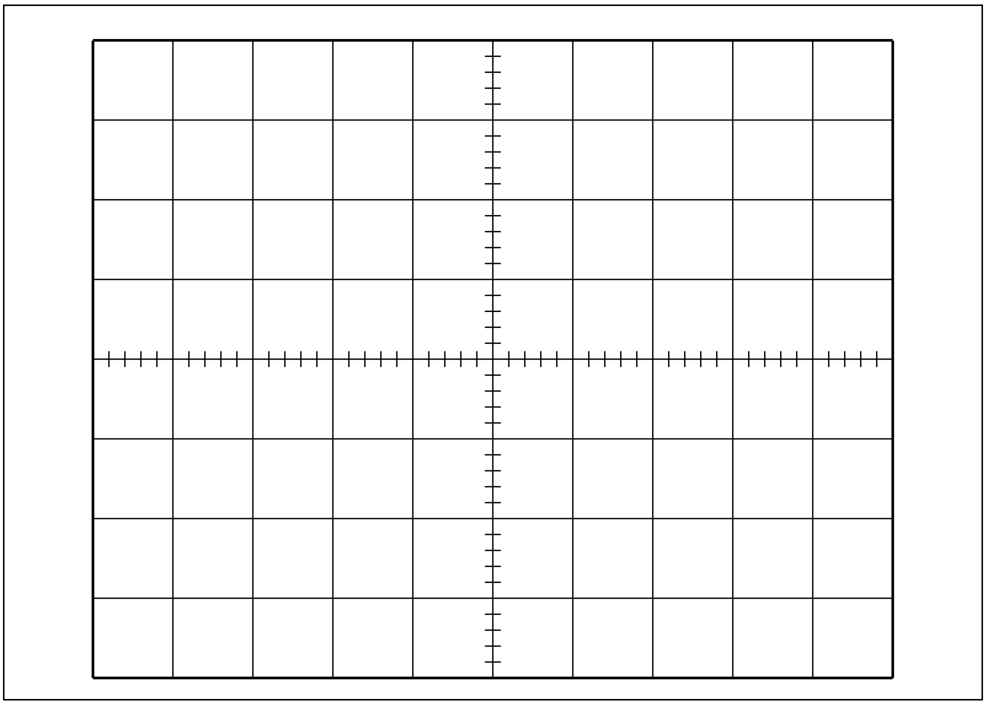
Frequency Response

2. Measure the frequency response using the following table. The frequency f_p is the frequency at which the output is a **maximum**.

f (Hz)	$\lvert\mathbf{V}_{in1}\rvert$ (V)	$\lvert\mathbf{V}_{out3}\rvert$ (V)	$\lvert\mathbf{V}_{out3}\rvert / \lvert\mathbf{V}_{in1}\rvert$ (dB)	$\angle\mathbf{V}_{out3} - \angle\mathbf{V}_{in1}$ (°)
20				
43				
93				
200				
430				
930				
2 000				
4 300				
9 300				
20 000				
f_p =				

Step Response

3. Draw or "capture" the step response:



Second-Order Circuit with a Real Zero – V_{out1}

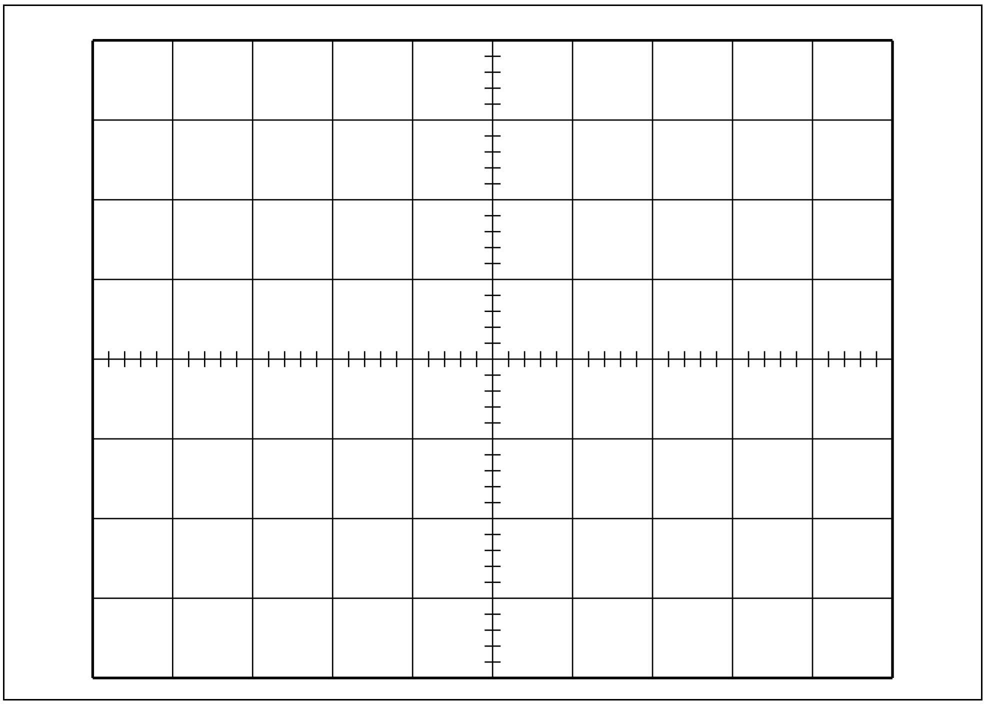
Frequency Response

1. Measure the frequency response using the following table. The frequency f_p is the frequency at which the output is a **maximum**.

f (Hz)	$\lvert \mathbf{V}_{in1} \rvert$ (V)	$\lvert \mathbf{V}_{out1} \rvert$ (V)	$\lvert \mathbf{V}_{out1} \rvert / \lvert \mathbf{V}_{in1} \rvert$ (dB)	$\angle \mathbf{V}_{out1} - \angle \mathbf{V}_{in1}$ (°)
20				
43				
93				
200				
430				
930				
2 000				
4 300				
9 300				
20 000				
$f_p =$				

Step Response

2. Draw or "capture" the step response:



L5.16

Second-Order Circuit with an Imaginary Zero Pair – V_{out4}

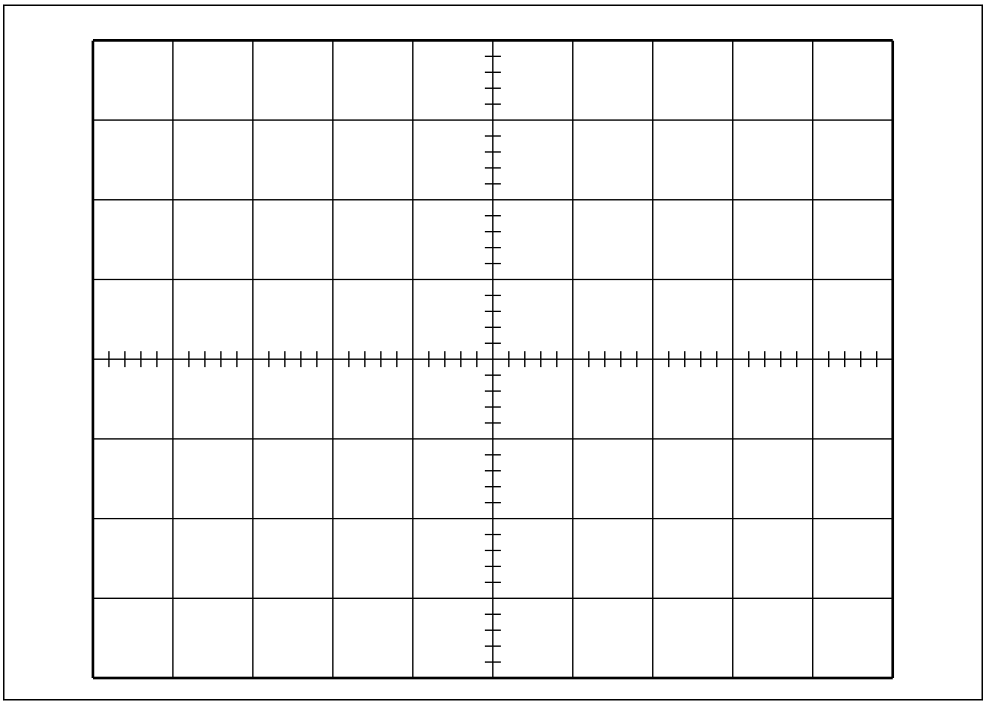
Frequency Response

1. Measure the magnitude response using the following table. The frequency f_n is the frequency at which the output is a **minimum**.

f (Hz)	$\lvert \mathbf{V}_{in1} \rvert$ (V)	$\lvert \mathbf{V}_{out4} \rvert$ (V)	$\lvert \mathbf{V}_{out4} \rvert / \lvert \mathbf{V}_{in1} \rvert$ (dB)	$\angle \mathbf{V}_{out4} - \angle \mathbf{V}_{in1}$ (°)
20				
43				
93				
200				
430				
930				
2 000				
4 300				
9 300				
20 000				
$f_n =$				

Step Response

2. Draw or "capture" the step response.



Questions [1 mark]

Encircle the correct answer, cross out the wrong answers. [one or none correct]

All questions are worth 0.1 marks each.

1. Step Responses

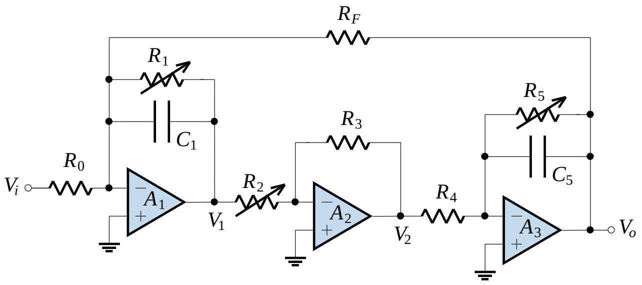
(i)

$$T(s)=\frac{K\omega_n^2}{s^2+2\alpha s+\omega_n^2}$$

When α is increased, the envelope of the step response:

(a) decays slower (b) is unchanged (c) decays quicker

(ii)



When R_1 and R_5 are decreased in unison:

(a) the imaginary part of the pole locations increases in magnitude

(b) the real part of the pole locations decreases in magnitude

(c) the frequency of oscillations in the step response decreases

(iii)

$$T(s)=\frac{K\omega_n^2}{s^2+2\alpha s+\omega_n^2}$$

The underdamped step response is:

(a) $y(t)=K\left[1-\frac{e^{-\zeta\omega_n t}}{\sqrt{1-\zeta^2}}\cos\left(\omega_d t+\cos^{-1}\zeta\right)\right]u(t)$

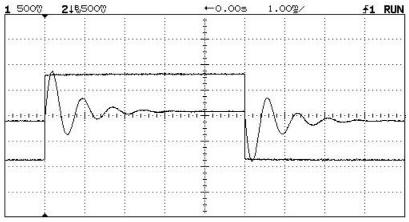
(b) $y(t)=K\left[1-\frac{e^{-\zeta\omega_n t}}{\sqrt{1-\zeta^2}}\sin\left(\omega_n\sqrt{1-\zeta^2}\,t+\cos^{-1}\zeta\right)\right]u(t)$

(c) $y(t)=K\left[1-\frac{e^{-\zeta\omega_n t}}{\sqrt{1-\zeta^2}}\sin\left(\omega_n t+\cos^{-1}\zeta\right)\right]u(t)$

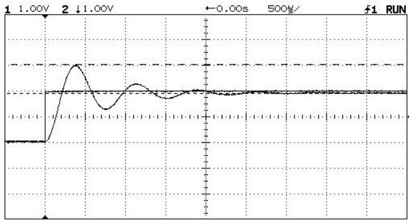
L5.18

(iv)

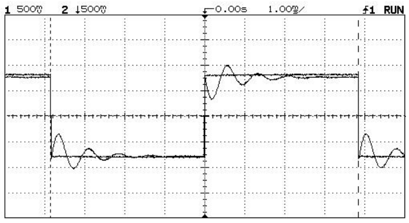
A second-order all-pole step response looks like:



(a)

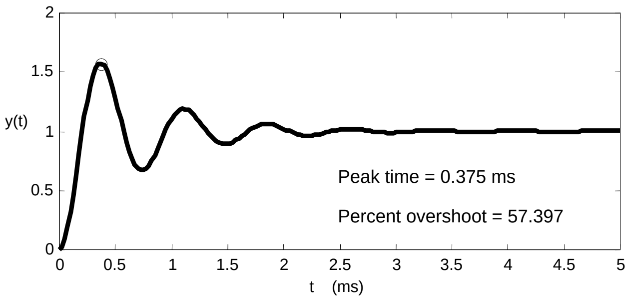


(b)



(c)

(v)

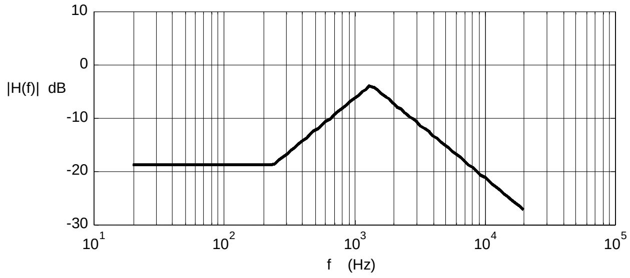


Step response of an all-pole second-order system:

(a) $\omega_d = 8369 \text{ rads}^{-1}$ (b) $\zeta = 0.5$ (c) $\zeta = 0.0174$

2. Frequency Response

(i)



Asymptotic magnitude response. The system has:

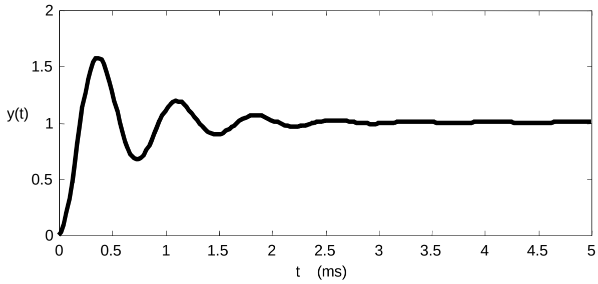
(a) one pole and one zero (b) two poles and one zero (c) two poles only

(ii)

A second-order non-inverting system has a real zero with a negative value. The asymptotic limit of the phase response, as $f \to 0$, is:

(a) -90° (b) 0° (c) 90°

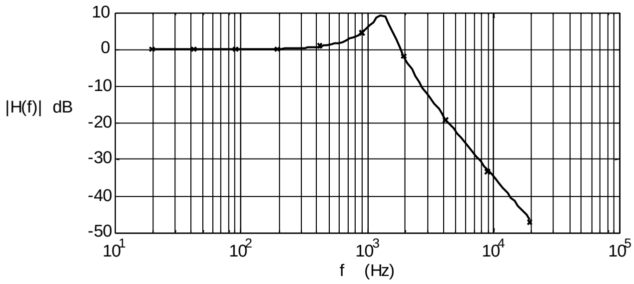
(iii)



The step response settles to 5 % of its final value after 2.5 ms. The peak occurs at 0.375 ms.

(a) $\alpha = 1200$ (b) $\alpha = 1500$ (c) $\omega_d = 8639$ rads^{-1}

(iv)



Magnitude response of a second-order system. The step response is:

(a) underdamped (b) critically damped (c) overdamped

(v)

A second-order transfer function has an imaginary zero pair. The magnitude response will, in general:

(a) be entirely flat (b) be zero as $f \to \infty$ (c) have a resonant peak

Attach the completed questions to your lab report.

Report

Only submit ***ONE*** report per lab group.

Complete the assignment cover sheet and attach your pre-lab work.

Ensure you have completed:

1. ***Pre-Lab Work*** – hand analysis and MATLAB® simulations.

2. ***Lab Work*** – frequency response tables and step response plots.

3. ***Post-Lab Work*** – complete the multiple choice questions. Modify your MATLAB® code to plot your experimental frequency response results on top of your theoretical pre-lab work.

The lab report is due on the date specified in the Learning Guide.

You should hand the report directly to your tutor.