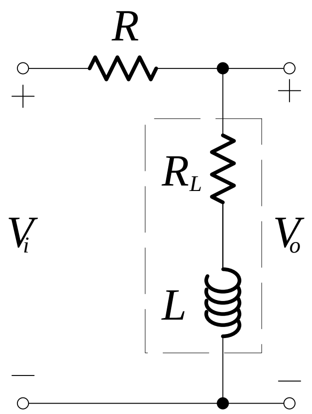
Family Name: ____________________

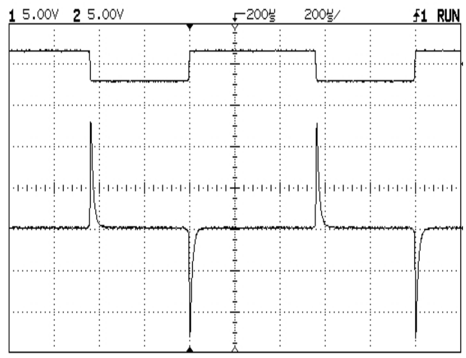
First Name: ____________________

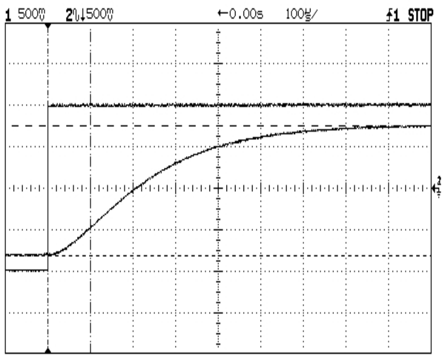
48520 Electronics and Circuits

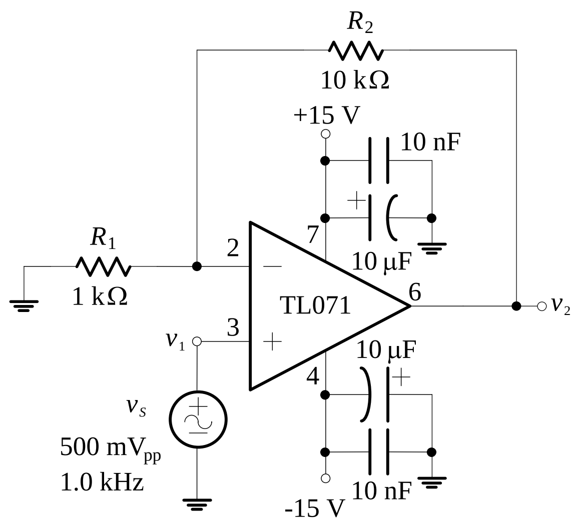
Lab Notes

2015









PMcL

Introduction

Circuit breadboarding. Layout plan. Circuit construction. Circuit testing. Connecting laboratory power supplies. Decoupling capacitors.

Introduction

This subject places a particular emphasis on the practical, hands-on aspects of Electronics and Circuits. In-depth understanding and mastery of Electronics and Circuits can be gained by:

- Finding out by measurements the *characteristics* and *limitations* of basic electronic devices

- Practicing the *analysis*, *design*, *building* and *testing* of some fundamental electronic circuits.

These laboratory experiments will help you acquire key testing, troubleshooting and measuring skills, vital for any electrical or computer engineer.

The laboratory experiments concentrate on characteristics and applications of the operational amplifier (op-amp). The topics selected for the experiments are relevant not only for future electrical engineers, but also for information and communication technology engineers and mechatronic engineers, because the experiments refer to fundamental signal responses, devices and circuits used in all electronic systems.

Computer simulation of electronic devices and circuits can produce meaningful results only if the user is aware of the physical characteristics, limitations and real-life interactions of the devices and circuits the user is attempting to simulate. The lab experiments should give you a better understanding and knowledge of these characteristics, limitations and interactions.

We hope that you will enjoy the laboratory experience, and benefit from it for the entire duration of your professional life!

Circuit Breadboarding

Once a circuit has been designed, it must be tested. To do this quickly and reliably, a good breadboarding system is needed. It should allow for the easy interconnection and removal of the analog ICs, discrete components, power supplies, and test equipment. It is *absolutely critical* that connections between the breadboard, the components, the power supplies and the test equipment be mechanically and electrically sound. Most beginners spend more time running down poor or wrong breadboarding connections than they spend actually evaluating the circuit they have built. In this section you will find breadboarding hints that will help you minimize problems and errors in building your circuit for testing.

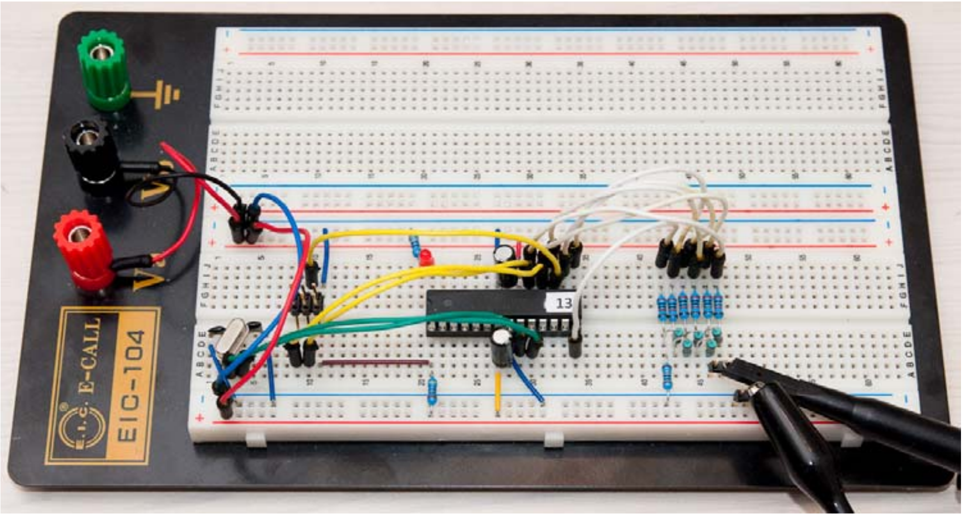


Figure 1 – Breadboard with ICs and other components inserted

The universal breadboard illustrated in Figure 1 provides a popular and convenient technique for circuit prototyping. Typically they give two to four busses (rails) for power supplies and ground, running along the edges. The body provides an array of solderless connections properly spaced and sized for most analog and digital ICs, transistors, diodes, small capacitors, 1/4 W resistors, and 22 AWG solid hook-up wire. Using it, you can construct circuits quickly, compactly, and reliably. These breadboards are available in a variety of styles and qualities from most electronic component suppliers.

The connection diagram of a typical breadboard is shown below:

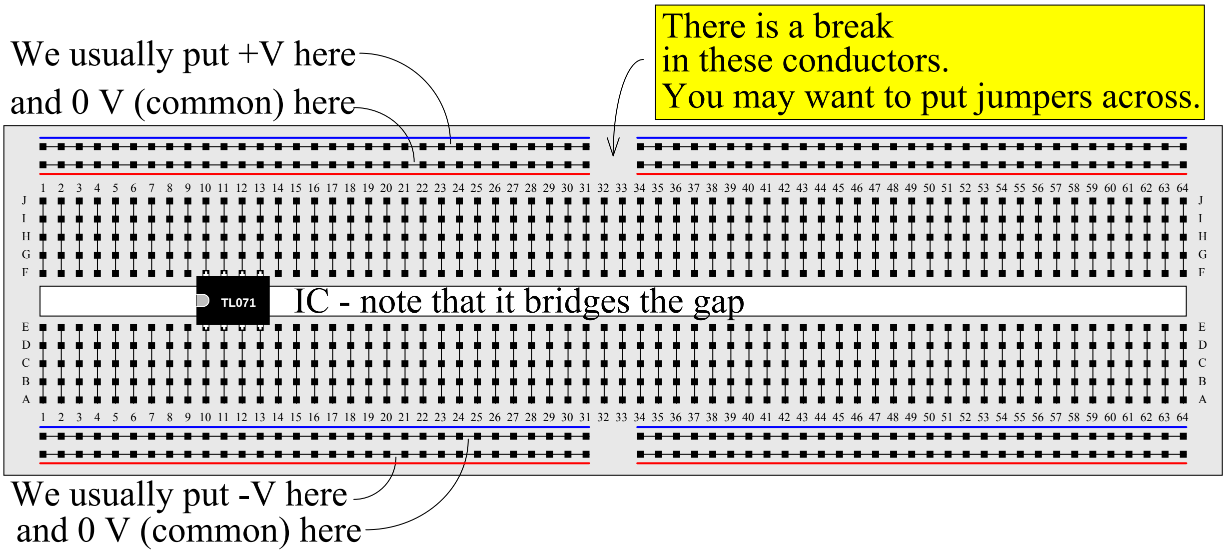


Figure 2 – Connection diagram of wire sockets (holes) on a breadboard

The breadboard consists of two regions – rows and columns:

- There are two sets of 64 columns each of 5 interconnected holes (A-E and F-J), to plug in components and connection wires.

- There are four sets of 2 rows each of 31 interconnected holes, called ‘rails’. The two rails on each side are for connecting the power supply(ies). Typically, the rails are for the positive supply +V, the negative supply -V and for 0 V (common).

The universal breadboard provides a good interface between the components of the circuit, but care must be taken when you connect it to power supplies and test equipment. The breadboard is usually mounted on some larger, sturdier base (an aluminium plate).

Just as a chain is only as good as its weakest link, test equipment can perform no better than the technique used to connect it to the circuit under test. Excellent standard leads supplied with banana plugs, BNC connectors, or probes are common. Use them.

Hours of careful design and breadboarding can literally go up in smoke because of a shorted or open wire to a power supply or from two alligator clips which accidentally touch, or jump off at just the wrong time. Alligator clips are a major source of trouble. They are often too large for use on a breadboard, short together, or fail to hold adequately.

Instead of connecting test equipment to the breadboard with alligator clips, we use binding posts (that have a socket for 4mm banana plugs) mounted on the side of the base plate. There are 5 binding posts: three for a dual power supply: +V, 0 (common) and –V, and the other two for the input and output signals.

Connect signals and supply sources from the test instrument to the breadboarding system, and from breadboard to instruments using standard leads with 4mm banana plugs. Then wire from the binding posts to the breadboard with 22 AWG wire, inserting the wire into the desired connector. This technique will provide an electrically and mechanically sound and professional way to build circuits, eliminating the cause of most breadboarding headaches, bad connections.

Use only standard connectors to connect test equipment to the breadboard. Never use alligator clips.

Find a suitable box to contain the breadboard with its base and the components you have plugged into it, to enable you to carry the breadboard around from home to the Lab without unplugging components and disturbing the assembled circuit.

An example of a properly assembled breadboard is shown below:

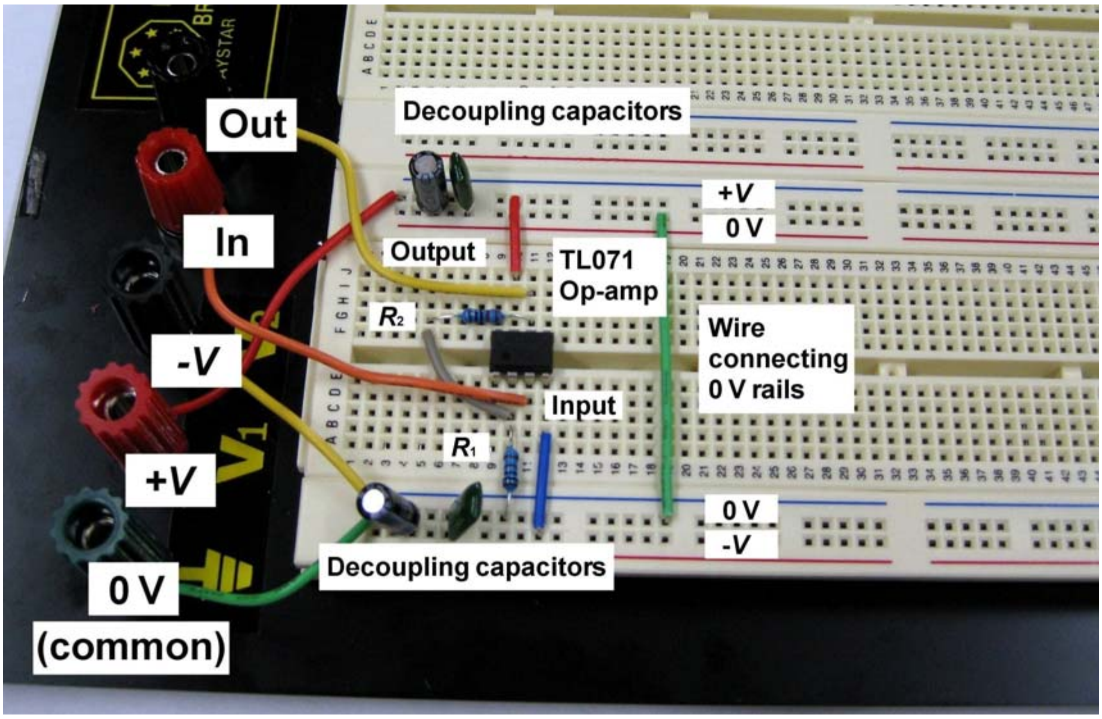


Figure 3 – Neatly and correctly assembled circuit on breadboard

Observe the two sets of decoupling capacitors (one electrolytic, one ceramic in each set) connected as explained below:

- One set of two capacitors connected between the +V rail and upper 0 V (common) rail.
- The other set of two capacitors between the –V rail and lower 0 V (common) rail.
- Of course, the upper and lower ground rails are interconnected with a wire strap.

Probes must also be used carefully. It is far too easy, when you are trying to touch a pin on an IC, for the probe to slip between two pins, shorting them together. This could damage the IC or supporting equipment. Instead of probing IC pins directly, you should connect a wire from the point you want to probe to a vacant part of the socket, where it can be secured and safely probed. ***Never probe IC pins directly***.

Layout Plan

a) Simplify the schematic and layout as much as possible for initial testing. Fine-tuning, zeroing, and additional stages can easily be added after you have the basic circuit working.

b) Be sure to include IC number, package type suffix, and pin numbers on each IC on the schematic diagram.

c) Make the layout look as much as possible like the schematic. Refer to the schematic whenever you debug your circuit.

d) Locate input and feedback resistors as physically close to the IC as possible. Long leads, connecting to remotely located resistors, pick up noise. This noise is then coupled to the highly sensitive input pin of the IC.

e) Keep the inputs well separated from the outputs to prevent oscillations.

Circuit Construction

a) Always clear the breadboard of any old circuits before beginning to build a new circuit.

b) Exercise care in inserting and removing ICs. Pins are easily bent and jabbed into your fingers.

c) Solder 22 A WG solid wire to the leads of components with large leads.

d) Devise and carefully follow a colour code scheme for +V, -V, 0 V (common) and signal wires. The usual colour code is:

- RED: +V
- BLACK: -V
- GREEN: 0 and/or EARTH

e) Avoid jungles. Make all components lie flat. Trim and bend leads and wires to fit the layout. Neat, flat layouts work better and are far easier to troubleshoot than a jungle of components and wires.

f) Do not forget to connect the power supplies to each IC. Although not always shown on a schematic, power is required by the ICs. This simple oversight is responsible for many lost hours of fruitless troubleshooting.

g) Select one connector as the common point. Tie the breadboard's 0 V rail, power supply common, and all test instruments' earths to that single point.

h) Insert suitable decoupling capacitors between the +V, -V supply rails and the 0 V (common) rail, preferably close to the power supply's connection points to the rails. See the layout in Figure 3 for an example.

Circuit Testing

a) Analyse the circuit before applying power to ensure that you know what to expect.

b) Double check all connections, especially power supply connections, before applying power.

c) Apply power to the IC before applying the signals.

d) Measure voltages with respect to circuit “common”. If you need the difference in potential between two points, measure each with respect to earth and then subtract. The common terminal of some instruments (particularly the oscilloscope) may be tied to earth and would short out some part of your circuit. Or it may inject noise into a sensitive portion of your circuit.

e) When using the oscilloscope to measure voltages, be aware that the accuracy of an oscilloscope, as a voltmeter, is of the order of 3%.

f) To measure voltages accurately (better than 0.5% accuracy) use the Digital Multimeter. When measuring AC voltages with the Digital Multimeter, make sure that the frequency of the signal you are measuring is within the limits specified for your Digital Multimeter.

g) Measure current by determining the voltage across a known resistor. Then calculate the current. Ammeters are rarely sensitive enough, tend to load the circuit, and often inject noise into sensitive nodes.

h) Remove the signal from the IC before removing the power.

i) Change components and connections with the power off.

Connecting Laboratory Power Supplies

Most regulated DC power supplies used in the laboratories usually contain *two separate, adjustable DC power supplies, isolated from one another* and *'floating'*, i.e., *not connected to earth*. This is shown below:

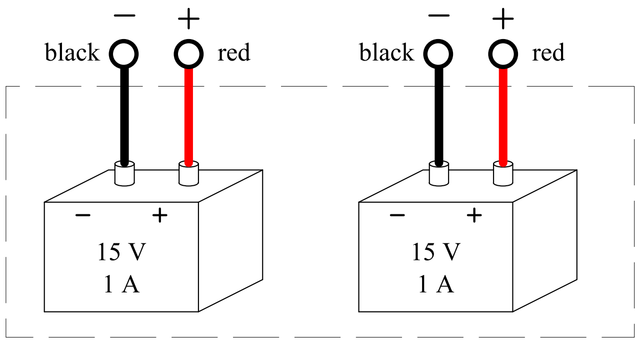


Figure 4 – Dual Independent Power Supplies

The BWD 604 Mini-Labs used in some laboratories do not have independent DC power supplies – they are connected in series and have one common terminal, as shown below:

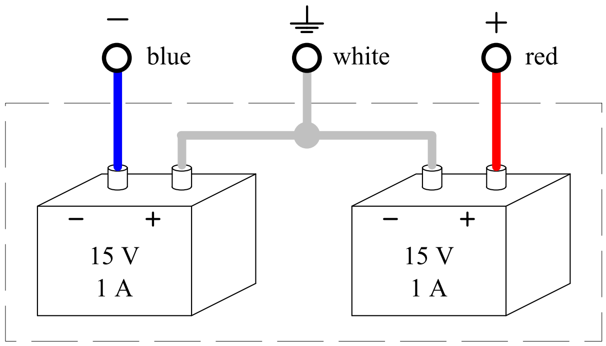


Figure 5 – Mini-Lab Dual Power Supply

There is also a third, fixed 5 V DC power supply, intended specifically for digital circuits. The Mini-Lab ties the 'negative' side of this 5 V DC power supply to earth (via the GPO). The Mini-Lab power supply therefore looks like:

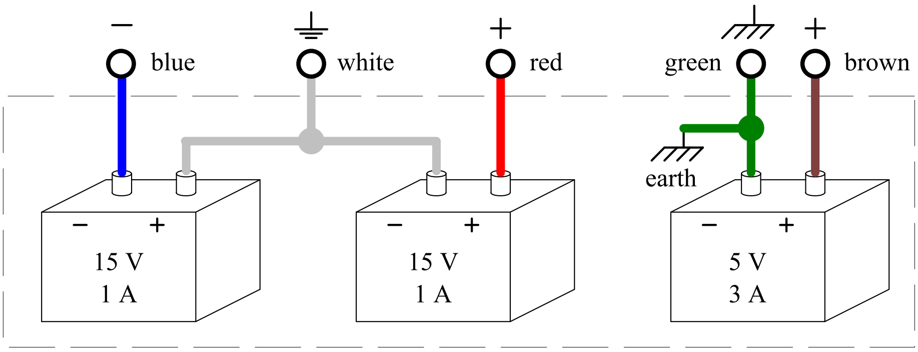


Figure 6 – Mini-Lab Triple Power Supply

For laboratory experiments, ***the 0 V middle connection point of the <u>dual power supply</u> must be connected to earth, as shown below***. Otherwise, the 'floating' supplies might pick up stray DC or AC voltages that could endanger the circuit you are studying, or yourself.

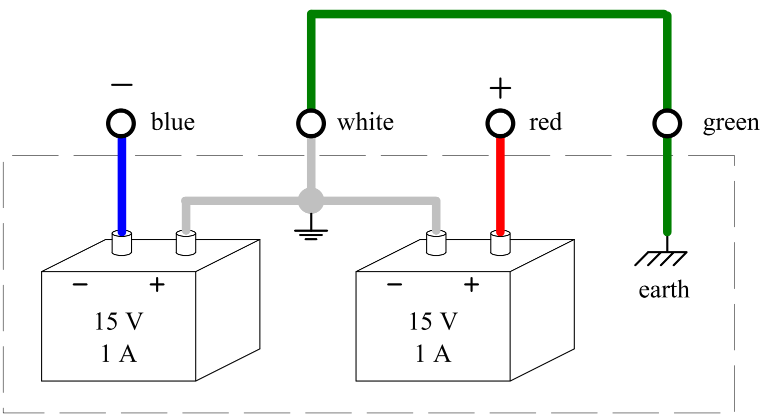


Figure 7 – Mini-Lab Dual Power Supply with Earth

The photographs below show an example of such a Mini-Lab earthing connection according to the wiring diagram of Figure 7.

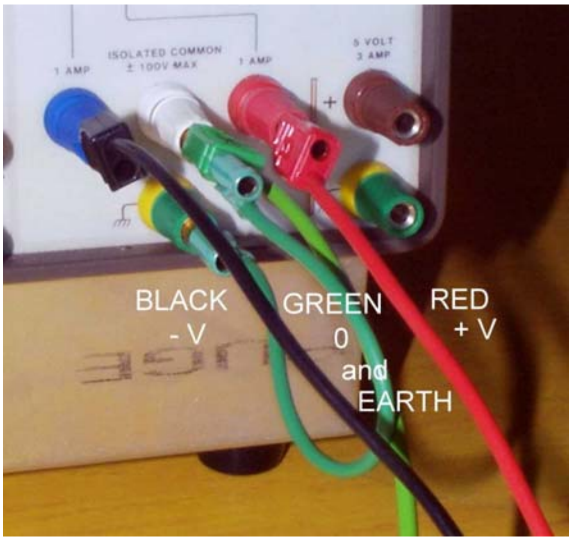
Blue Terminal: -V

Red Terminal: +V

White Terminal: 0

Green Terminal: Earth

Figure 8 – Mini-Lab Power Supply Earthing



Connection wires to the breadboard circuit:

Black: –V

Green: 0

Red: +V

Figure 9 – Details of "earthing" the common on the Mini-Lab Power Supply

Power supply connections to the breadboard and to the individual ICs can cause some other problems. For example, one sure way to damage an analog IC is to reverse the power supply connections. This can be easily prevented when you are breadboarding, by first labelling each bus in some highly visible way, for example, by colour coding. This should prevent you from connecting the IC to the wrong supply bus.

Decoupling Capacitors

When analysing the AC small-signal operation of an electronic circuit, one *assumes that the DC power supply of the circuit is a* ***short-circuit*** *for all the AC signals likely to occur in the circuit*. In real-life situations, this assumption might be only wishful thinking, unless you make sure with appropriate measures that it really happens.

The *laboratory power supply itself* usually complies with this requirement, i.e. its output impedance is typically only a few milliohms over a wide range of frequencies.

On the other hand, *the leads running from the power supply to the breadboard* have some *resistance* and some *inductance*; therefore, the power supply does not actually behave as a short circuit *when seen from the breadboard*. The stray impedance of the leads can cause stray coupling of signals from the output to the input of your circuit, producing unwanted feedback and unpredictable behaviour.

Also, high-frequency (often noise) signals can be picked-up by the leads. When coupled to or from one IC to another IC and amplified, these high frequency signals on the supply rails can cause the entire circuit to oscillate.

To avoid stray coupling via lead impedances, the connections to the power supply must be '***decoupled***' or '***bypassed***' with capacitors ***directly on the breadboard***. The decoupling capacitors must provide, ***between the power supply connection points to the breadboard, a negligibly small impedance for all likely AC signal frequencies***.

Therefore it is ***strongly recommended for all circuits***, to place a large capacitor, say an aluminium electrolytic $10\,\mu F$ or $100\,\mu F$, in parallel to a smaller capacitor, say a 10 nF or 100 nF polyester film capacitor across the +V to common and –V to common connection points at the power supply inputs on the breadboarding socket as shown below:

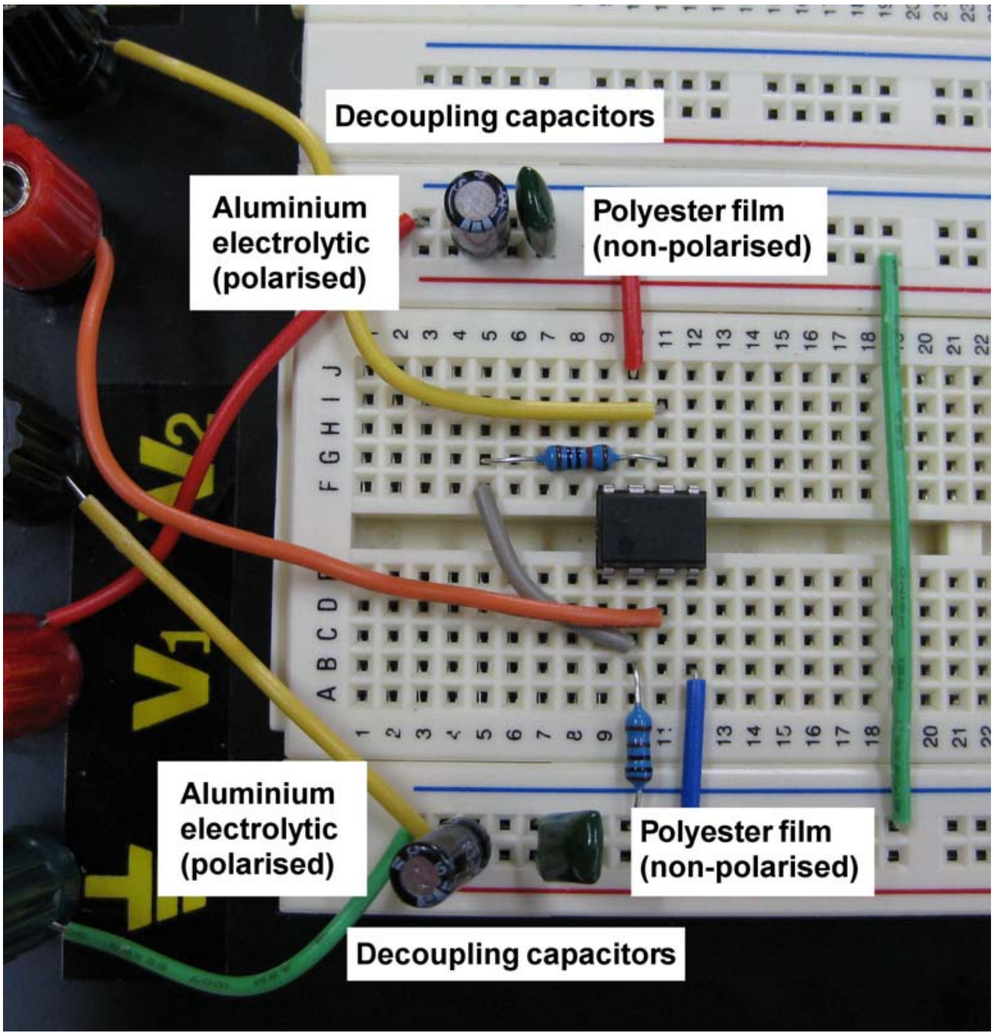


Figure 10 – Decoupling capacitors

Additionally, for decoupling the supply terminals of fast pulse ICs or high-frequency ICs, and to avoid stray signals being inadvertently transmitted from one IC to another one, it is strongly recommended to place 10 nF or 100 nF capacitors from each power supply pin of each IC to common, adjacent to the IC. The stray signals are then passed to common as they leave the IC, before they can contaminate the supply rails.

Lab 1 – Lab Equipment

DSO. Vertical setup. Horizontal setup. Trigger setup. Coupling of input signals. Automatic time measurements. Automatic voltage measurements. Cursor measurements. Reducing random noise on a signal. Dual power supply. Earthing the supply. Using triple supplies.

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. Like any tool though, it has its limitations, and careful operation is required to interpret results correctly.

For professional design and testing, a constant DC voltage is usually required where the voltage can be adjusted from the front panel – such devices are DC power supplies. A power supply may have one pair of terminals, or two (a 'dual' power supply) or three pair (a 'triple' power supply). Some can be operated in series or parallel. You need to become familiar with the laboratory power supplies so that in future when you need to use one you know how they operate.

Another useful device for testing is the "function generator". This device is capable of generating sinusoidal, triangular, and square waves of varying frequency and amplitude. It is generally used as the "input signal" to a circuit so that a circuit's time and frequency characteristics can be determined.

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 review the operation of a dual and triple power supply.

L1.2

Equipment

- 1 Digital Storage Oscilloscope (DSO) – Agilent 54621A
- 1 Mini-Lab – BWD 604
- Resistors – 1 x 4.7 kΩ , 1 x 10 kΩ
- Breadboard, Hook-up wire
- 4mm leads (assorted colours), 2 BNC to 4mm leads

Safety

Cat. A lab

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

Basic Setup

You will be asked to perform various and wide-ranging tasks with the DSO during the laboratories, so it is important that you become familiar with its capabilities and limitations.

Function Generator Setup

1. Turn the Mini-Lab on and set the function generator (FG) up for a sinusoidal wave of around 2 kHz. Set the amplitude to one quarter of the full range. Ensure the DC offset knob is set to ‘off’.

2. Turn the DSO on and ensure the DSO has been set to its default setup configuration, by pressing the `Save/Recall` key on the front panel, then press the `Default Setup` softkey under the display.

3. Observe the FG output using Channel 1 of the DSO.

Vertical Setup

1. Push the `1` button. In the Channel 1 Menu, select the `BW Limit` softkey to “bandwidth limit” the channel, i.e. to attenuate high frequencies, which is generally “noise”. Bandwidth limiting Channel 1 will help create a “stable trigger”. Note the illuminated “BW” next to the `Position` knob..

2. Turn the `Volts/Div` knob to 500 mV/div.

3. Set the FG so that the sinusoid is 3 V peak-to-peak.

4. Turn the `Position` knob and note the effect. Return the position to 0.0V.

5. In the Channel 1 Menu, press the `Coupling` softkey until `AC` is selected. Note the illuminated “AC” next to the `Position` knob. Use the `Coupling` softkey to reselect `DC`.

6. In the Channel 1 Menu, select the `Invert` softkey to “Invert” the channel. Note the status line shows that channel 1 is inverted (it has a bar over the 1). Turn the “Invert” off.

L1.4

Horizontal Setup

1. Turn the `Time/Div` knob and notice the change it makes to the status line.

2. Press `Main/Delayed`. Change the `Time Ref` softkey to see the effect (note the trigger point / time reference triangle beneath the status line moves to show the position of the time reference). Return the Time Ref to “Center”.

3. Use the softkeys to select different horizontal modes, and note the effect.

4. Restore the horizontal mode to `Main` and display two cycles of the sinusoid.

5. Turn the `Delay` knob to see the effect, and notice that its value is displayed in the status line. Reset the delay to 0.0s.

Trigger Setup

1. Turn the trigger Level knob and notice the changes it makes to the display. Note that when the trigger level is set to a value that exceeds the bounds of our input signal, we lose the ability to trigger because the input signal never reaches the trigger level. Use the value in the status line to return the trigger level to 0.0V.

2. Press `Edge`. Toggle each of the softkeys to see the effect and notice the change to the status line. Set the trigger to a positive edge on Channel 1.

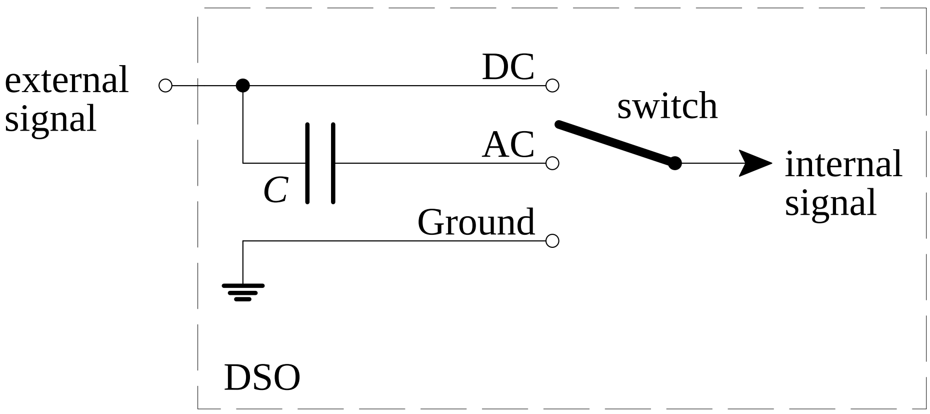
3. Press `Mode/Coupling`. Toggle between the Modes to see the effect on the status line. Set the Mode to `Auto Lvl`.

4. Change the FG frequency to 3 kHz, then push the 10 Hz range button to obtain 3 Hz. Adjust the `Time/Div` knob to display two cycles of the sinusoid. Press `Main/Delayed`. Press the `Roll` softkey. Change the FG wave shape to triangle, then square, then back to sinusoid. Press the `Single` key. Press the `Run-Stop` key to trigger the DSO again.

5. Set the DSO to `Main` Horizontal Mode and `Auto Lvl` Trigger Mode.

Coupling of Input Signals

The DSO has the ability to insert a capacitor between the external input and its internal analog acquisition circuitry. This can be represented by the circuit below, known as the input 'coupling' circuit:

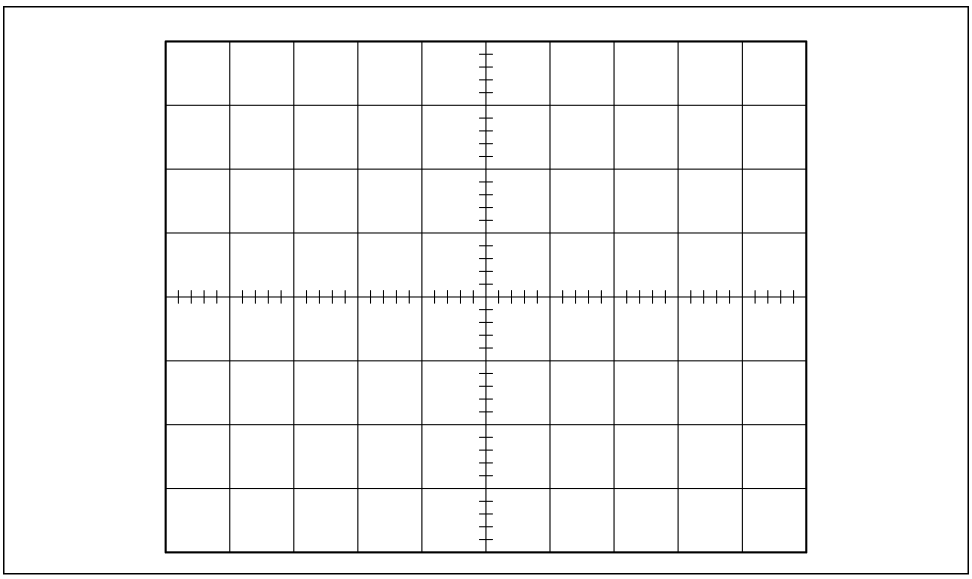


We will investigate the effect and use of the input coupling circuit.

1. Turn Channel 2 on by pressing the 2 key. Set 1 V/div on both channels.

2. Set the FG frequency to around 3 Hz and measure the FG output on DSO Channels 1 and 2 simultaneously. Adjust the `Time/Div` knob to display approximately 2 cycles of the sinusoid.

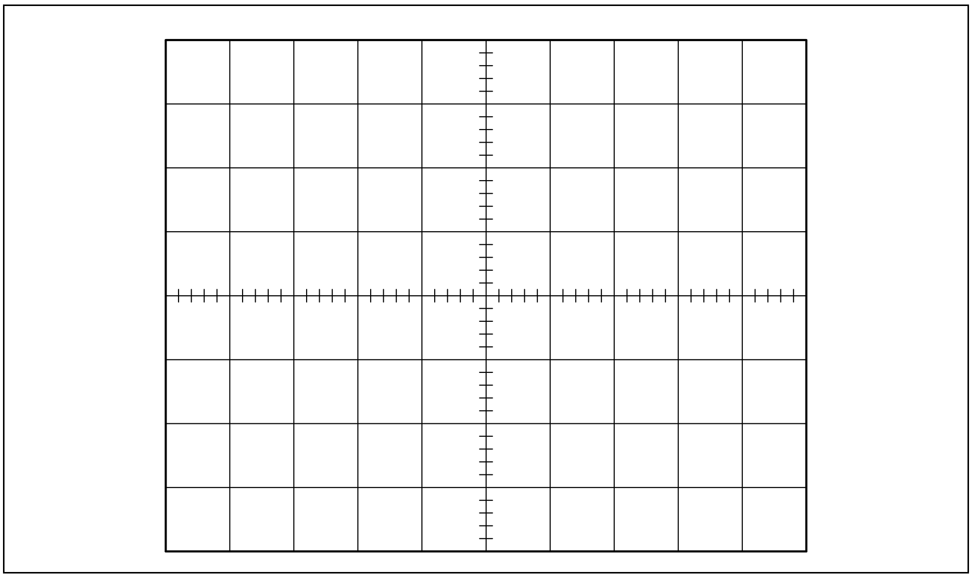
3. On the Channel 2 menu, set the Coupling to `AC`. You should see a "shifted" sinusoid on Channel 2. Sketch the observed waveforms in the correct time relationship below. Show the voltages and time on your plot.



Phase-shifted sinusoids

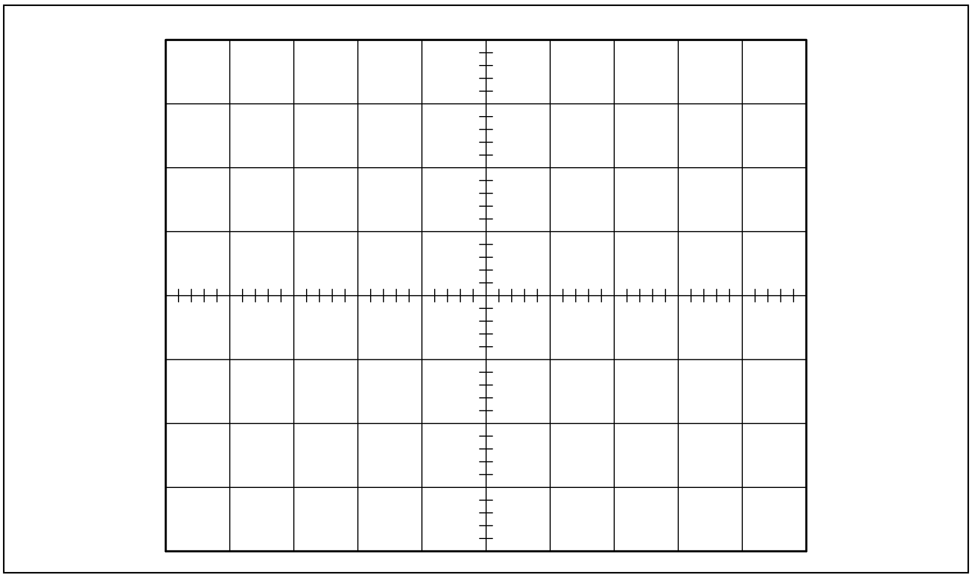
4. Change the FG to a square wave. Note the significant change in wave shape. Sketch the observed waveforms in the correct time relationship below. Show the voltages and time on your plot.

Square wave and AC-coupled square wave



5. Change the FG to a triangle wave. Note the significant change in wave shape. Sketch the observed waveforms in the correct time relationship below. Show the voltages and time on your plot.

Triangle wave and AC-coupled triangle wave



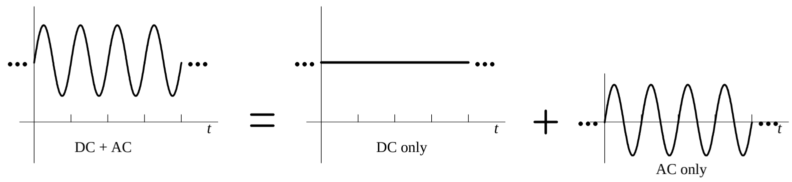
<u>Note</u>: AC coupling should be used with caution, because at low frequencies it can radically alter the observed waveform!

6. Change the FG to a 20 kHz sinusoid, and adjust the `Time/Div` knob to display 2 cycles of the sinusoid.

7. Set the trigger on the DSO to use Channel 2 with `HF Reject` selected.

8. On the FG, turn on the "DC offset" and apply approximately 3 V of DC to the sinusoid.

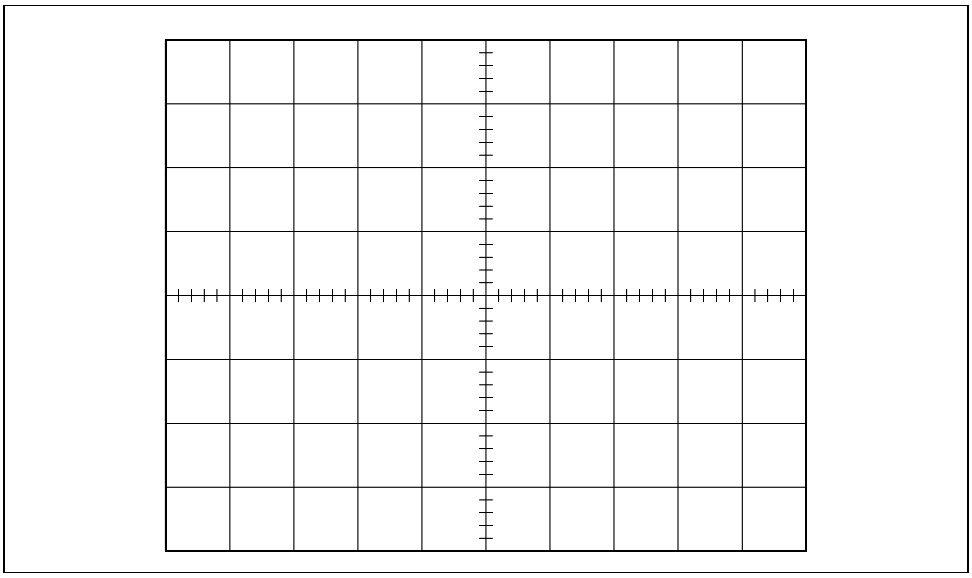
9. Now reduce the amplitude of the sinusoid to a minimum. Turn the `Volts/Div` knob on Channel 2 to 100 mV/div to display a fairly large sinusoid. We can now get "AC detail" from a waveform that has a large DC component:



AC coupling will remove the DC component of a waveform

<u>Note</u>: The AC input coupling capacitor has blocked the DC component of the waveform, allowing us to observe only the AC component.

10. Sketch the observed waveforms in the correct time relationship below. Show the voltages and time on your plot.



Shifted sinusoid and AC component

11. Set the trigger on the DSO to use Channel 1 with `HF Reject` off.

12. Set Channel 2 to 1.00 V/div and then turn it off.

13. On the FG, turn the DC offset to 'off'.

L1.8

Time-domain Measurement

Automatic Time Measurements

1. Set the FG to a 3 V p-p 20 kHz sinusoid. Set the time base to 200 µs/div.

2. Press `Quick Meas`. Note that the frequency is automatically displayed. Change the FG wave shape to square. On the FG, turn the Symmetry switch on (up) and turn the Symmetry knob fully clockwise.

3. Press the `Select:` softkey. Use the Entry knob to select `Duty Cycle`. Press the `Measure Duty` softkey. Note the duty cycle range of the FG by turning the FG's Symmetry knob. Change the FG wave shape back to sinusoid, and turn the Symmetry switch off.

4. Turn Channel 2 on and set the time base to 50.0 ms/div. Set the FG to about 5 kHz, then push the 10 Hz range button to get 5 Hz.

5. Press `Quick Meas`. Use the Entry knob to select `Phase`. Press the `Measure Phase` softkey. Measure the phase difference between the two waveforms. Determine which channel is used as the reference by the DSO for the phase measurement. You can set up the way the DSO measures phase by using the softkey `Settings`.

Automatic Voltage Measurements

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

1. Turn Channel 2 off.

2. Press `Quick Meas`. Measure the Peak-Peak voltage of Channel 1. Measure the Average of Channel 1. Measure the RMS of Channel 1. Change the FG waveform to triangle, then to square, and observe the change in the measurements.

3. Set the FG to a sinusoidal wave, and vary the DC offset. Note the effect on the Pk-Pk, Avg and RMS values. Turn the DC offset to 'off'.

Mini-Lab Amplifier Setup

The function generator has an output resistance of $50\,\Omega$ and so any load that draws considerable current will cause the output to experience a significant internal *Ri* voltage drop, resulting in a "droop" in the output voltage. The Mini-lab provides us with a "buffer amplifier" that is capable of delivering large currents with minimal voltage drop.

Mini-lab Amplifier

1. Identify the section under the power switch labelled "AMPLIFIER OR BI-POLAR POWER SUPPLY".

2. Ensure that the left-most pushbutton is out (F. GEN) so that the internal function generator is selected as the input.

3. Ensure that the middle pushbutton is out (NORM) so that the output is normal.

4. Ensure that the right-most pushbutton is out (AMP) so that the unit acts as an amplifier.

5. Ensure that the knob is fully rotated counter-clockwise to select a gain of "X 1".

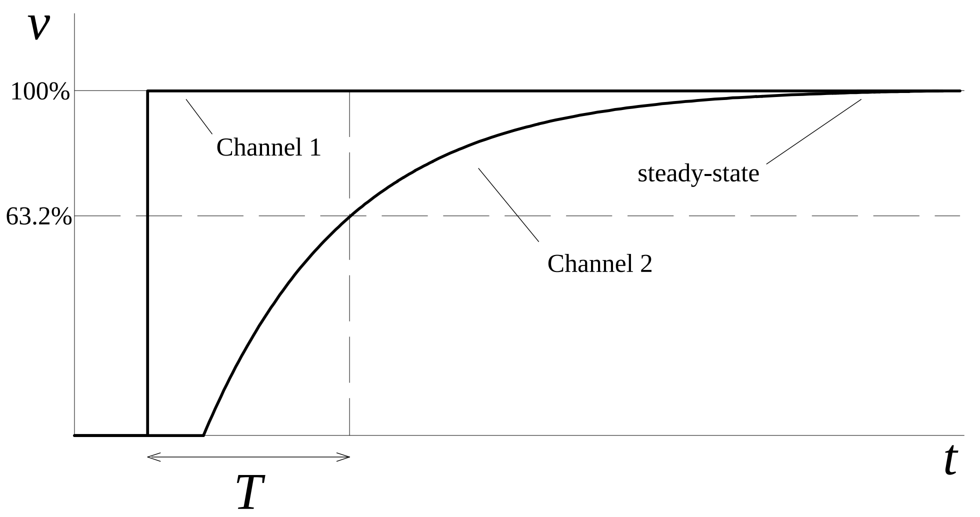
6. With these settings a "buffered" version of the function generator output is provided directly from the red output terminal.

Cursor Measurements

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

For example, we would like to measure the time it takes for a particular waveform to respond to a stimulus and reach 63.2% of its final “steady-state” value. We take a measurement of the time *T* as shown below:

Cursors are used for custom measurements



1. Set the FG to a 2 V p-p, 5 kHz square wave.

2. Turn Channel 2 on and set the coupling to `DC`. Measure the output of the Mini-Lab amplifier with Channel 2 of the DSO.

3. Press `Main/Delayed`. Set the `Time Ref` softkey to `Left`. Set the `Time/Div` to 500 ns.

4. Press `Cursors`.

5. `Source` selects a channel for the cursor measurements. Change the cursors’ source to Channel 2 by pressing the `Source` softkey.

6. Press the softkey labelled `X Y` to select the Y (voltage) cursors.

7. Press the softkey labelled `Y1`. Move the Y1 cursor to align with the bottom of the output response by rotating the Entry knob ↻.

8. Press the softkey labelled `Y2` to enable the second Y (voltage) cursor. Move the Y2 cursor to align with the top (steady-state value) of the output response. Check that the cursor measurement displays $\Delta\mathrm{Y}(2) \approx 2.000\ \mathrm{V}$.

9. Now calculate 63.2% of the steady-state value. e.g. $63.2\% \times 2.000\ \mathrm{V} = 1.264\ \mathrm{V}$.

10. Adjust the Y2 cursor so that $\Delta\mathrm{Y}(2)$ is close to the 63.2% value. You will not be able to set the exact value. Choose the closest value available.

11. Press the softkey labelled `X Y` to select the X (time) cursors.

12. Press the softkey labelled `X1`. Move the X1 cursor to align with the vertical edge of the input square wave.

13. Press the softkey labelled `X2`. Move the X2 cursor to align with the intersection of the Y2 cursor and the channel 2 waveform.

14. Record the following measurement, using the value for $\Delta\mathrm{X}$:

$\Delta\mathrm{X} = T =$

15. Turn the cursors off.

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

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

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.

1. Connect Channel 2 to the SYNC output of the Mini-Lab (it's on the far left). Press `Edge` and then `2` so that the DSO triggers off Channel 2. The SYNC output from the Mini-Lab is in frequency synchronism with the FG output, and will provide a stable trigger for the DSO. Turn Channel 2 off (we don't need to display the SYNC waveform).

2. Change the FG waveform to a sinusoid. Set 50.0 μs/div. Reduce the amplitude to a minimum. Press the FG's 20 dB ATTENUATOR button to apply 20 dB of attenuation (i.e. the output is reduced by a factor of 10).

3. Change the DSO vertical scale so that the peaks of the sinusoid are visible. It should be a noisy sinusoid.

4. Press `1`. Press the `BW Limit` softkey. The noise is increased because bandwidth limiting is off, and we are "letting through more noise".

5. Turn bandwidth limiting on 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 `Edge` and then `1` so that the DSO triggers off Channel 1.

2. Press `Mode/Coupling`.

3. 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). `Noise Rej` adds additional "hysteresis" to the trigger circuitry. `HF Reject` adds a 50 kHz lowpass filter in the trigger path to remove high frequency components from the trigger waveform.

4. Press `Acquire`, then press the `Averaging` softkey.

5. Turn the Entry knob to select the number of averages that best eliminates the noise from the displayed waveform. The higher the number of averages, the slower the displayed waveform responds to waveform changes. Set `# Avgs` to `64`.

6. Change the FG wave shape to triangle, then square, then back to sinusoid to see the effect of averaging.

7. Turn off the FG's 20 dB ATTENUATOR button.

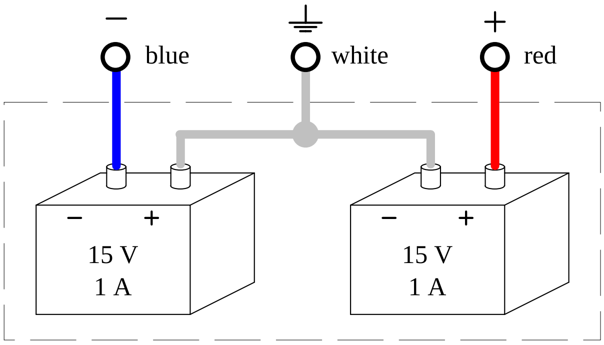
Dual Power Supply

Refer to the Lab Equipment Guide

A dual power supply is really just two independent power supplies, either with or without a ‘common’ connection. If the power supplies are truly independent, the output can be connected in series for additional voltage, or they can be connected in parallel for additional current capacity. This section will explore the operation and connections of a dual power supply.

Mini-Lab Dual Power Supply

Conceptually, the Mini-Lab dual power supply looks like:



Setting up the Supply

We are going to set up the power supply for a 5 volt output on one pair of output connectors.

1. Set the digital multimeter to read ‘V’ and ‘DC’ and select the 20 V range.

2. Connect 4mm leads from the right-hand power supply (white and red terminals) to the multimeter and adjust the output voltage until the meter reads as close to 5 V as possible. Record the multimeter reading:

$V =$

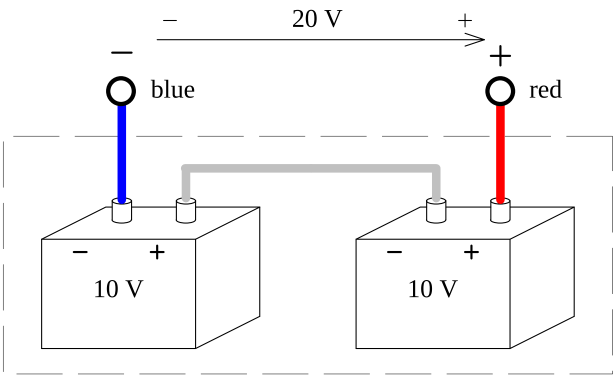
If you need an accurate output voltage, always use a digital multimeter connected to the output voltage at the load, not at the supply, since there may be a voltage drop in the leads due to the lead resistance, if the current is large.

Using Dual Supplies

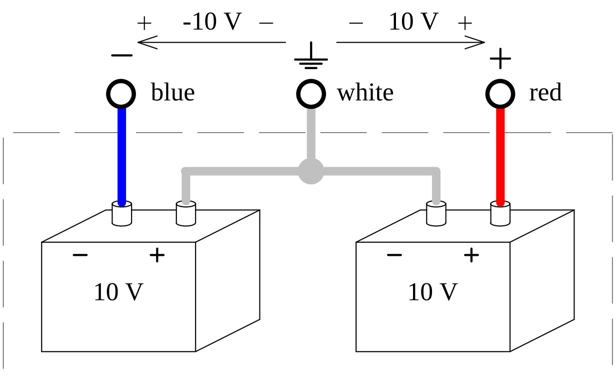
In this section we are going to demonstrate the various methods of taking 'an output' from the dual power supply. Each output of the power supply is 'floating' with respect to earth at the general power outlet (GPO), and thus is similar to a battery.

1. Adjust both variable outputs to 10 V using the multimeter.

2. Now measure the voltage between the '-' terminal on the left-hand side and the '+' terminal on the right-hand side of the power supply. *You should get 20 V, because you have connected the supplies in series, as shown below*:



3. Considering the white power supply terminal as a 'common' voltage reference, measure the voltage between this common and each of the other two terminals. *You should get +10 V and -10 V.*



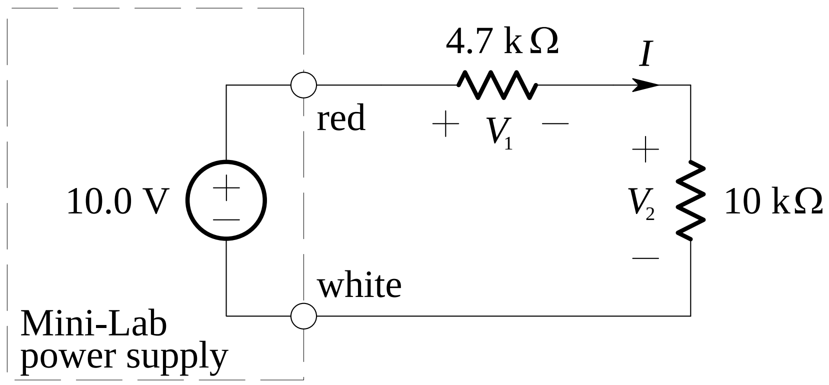
Voltage between 'plus' (red) and common =
Voltage between 'minus' (blue) and common =

<u>This is the way we get a 'plus and minus supply' for analog circuits.</u>

Earthing the Supply

Earthing lab power supplies is very important. The output of the supply is electrically 'floating', even if you connect two sides in series and use the interconnection point as a common reference. To make this common reference equal to the earth voltage, you must connect this point to the earth (green) terminal. This makes the circuit safe and allows you to use other test and measurement equipment on your circuit (e.g. a DSO), reducing the risk of damage. Let's see an example of this concept.

1. Set the output voltage of the right-hand supply to 10.0 volts. With the power off, construct the circuit shown below using the resistors and breadboard from your lab kit. Do ***<u>NOT</u>*** connect an earth lead to the circuit just yet.



2. Turn the power on. With the multimeter, measure the voltage across the $4.7\ \text{k}\Omega$ resistor, and then across the $10\ \text{k}\Omega$ resistor. Calculate the current in the circuit. Note the **<u>polarity</u>** of the voltages and current in the circuit above. Record your values for later reference:

Voltage across the $4.7\ \text{k}\Omega$ resistor , V_1 =
Voltage across the $10\ \text{k}\Omega$ resistor , V_2 =
Current, I =

3. Now using the DSO, measure the same voltages in the circuit:

Voltage across the $4.7\ \text{k}\Omega$ resistor, V_1 =
Voltage across the $10\ \text{k}\Omega$ resistor, V_2 =

4. Disconnect the DSO from the circuit.

5. Now earth the supply by connecting the common terminal (white) to earth (green). Measure the voltages in the circuit using the DSO:

Voltage across the $4.7\ \text{k}\Omega$ resistor, V_1 = Voltage across the $10\ \text{k}\Omega$ resistor, V_2 = Explain the results by drawing circuit diagrams of the measurements, showing the earthing connections.

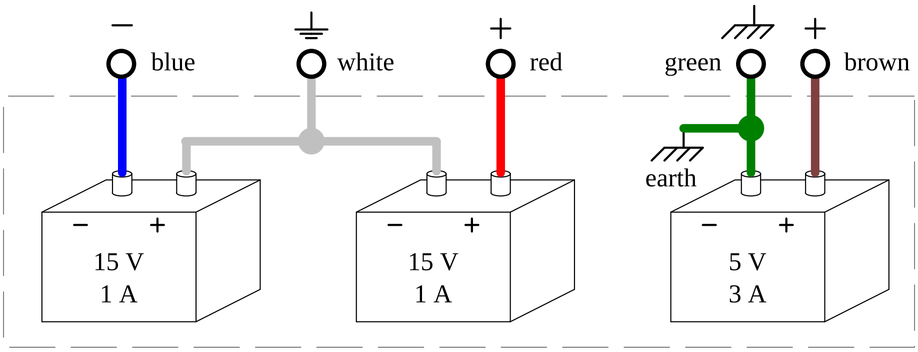
The reason is that when you connect the DSO earth to the point in between the $4.7\ \text{k}\Omega$ *and* $10\ \text{k}\Omega$ *resistors, you are earthing that point, and hence, shorting out the* $10\ \text{k}\Omega$ *resistor. You must always earth your circuit for your own safety and to avoid damage to the lab equipment and / or your circuit. When you do so, be careful when using a DSO or other earthed equipment. A DSO always measures voltages with respect to EARTH.*

To measure voltages across components in a circuit using a DSO, do one of the following:

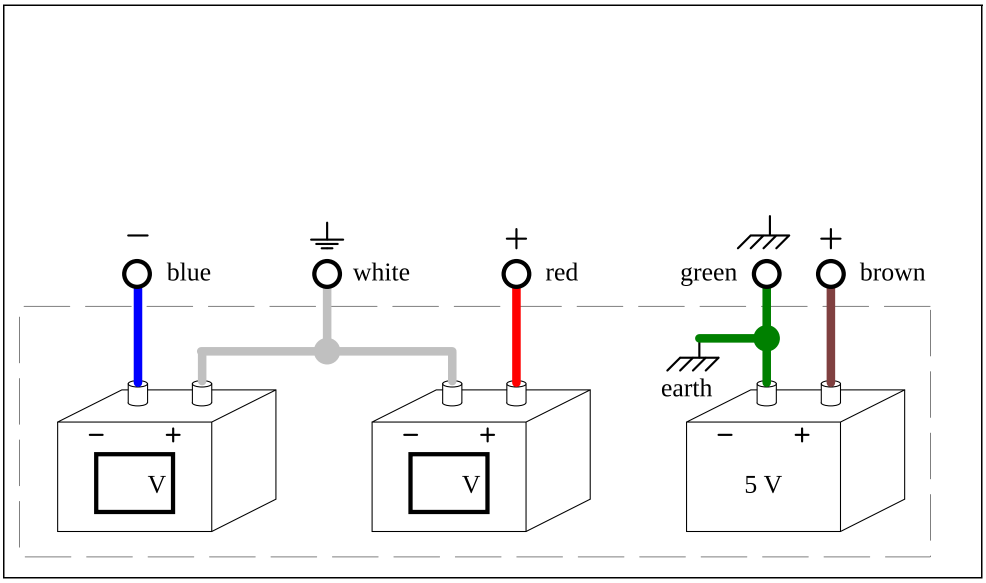
- *Use a DSO with a maths function to subtract the two channels.*
- *Measure one voltage, then the other, and subtract them.*

Using Triple Supplies

The Mini-Lab also has a fixed 5 V supply to facilitate the powering of digital integrated circuits (ICs). This fixed 5 V supply will occasionally be used, so it is important to note that its output is with respect to earth, and not the common of the dual power supply, as shown below:



1. Draw the connections you would use to create a triple power supply that provided +5 V, +10 V, and – 10V, with all voltages measured with respect to earth. Label the outputs, and show the various voltages.



2. Now confirm your connections by wiring the Mini-Lab and measuring the voltages with the digital multimeter.

Lab Assessment [2 marks]

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

1. Set up a 3 V p-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 on Channel 1. Show only the AC component of the waveform on Channel 2. Use the DSO `Quick Meas` feature to measure the average, peak-to-peak and RMS values of the waveform on Channel 1.

2. Use the FG attenuation pushbuttons to apply 30 dB attenuation to the signal. Set up the DSO to get a stable, noise-free (averaged) display.

3. Remove the attenuation and the DC offset and set the FG to 3 Hz. Apply the FG signal to Channel 1 with DC coupling, and to Channel 2 with AC coupling. Measure the phase difference.

4. Set up a triple power supply to provide +5 V, +10 V and +15 V with respect to earth. Use the DSO to observe and measure each voltage.

Marking

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

Lab 2 – Noninverting and Inverting Amplifiers

Noninverting amplifier. Inverting amplifier.

Introduction

The op-amp is the most versatile electronic building block. Circuits based on the op-amp nearly always use a *feedback* configuration. Feedback has many desirable properties, as we will see.

A noninverting amplifier uses a resistive negative feedback circuit around an op-amp to achieve a gain with a precision determined by the resistors (independent of the op-amp).

An inverting amplifier's gain is also determined by external resistors, except the output is inverted compared to the input.

Objectives

1. To build and test a non-inverting amplifier.

2. To build and test an inverting amplifier.

Equipment

- 1 Digital Storage Oscilloscope (DSO) – Agilent 54621A
- 1 Mini-Lab – BWD 604
- Op-amp – 1 x TL071
  Resistors – 1 x 1 kΩ, 1 x 10 kΩ
  Capacitors – 2 x 10 μF, 2 x 10 nF
  Breadboard, Hook-up wire, 2 x 4mm leads.

L2.2

Safety

Cat. A lab

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

Warning!

Remember:

1. **When wiring the circuits, ensure that the power supply is switched off.**

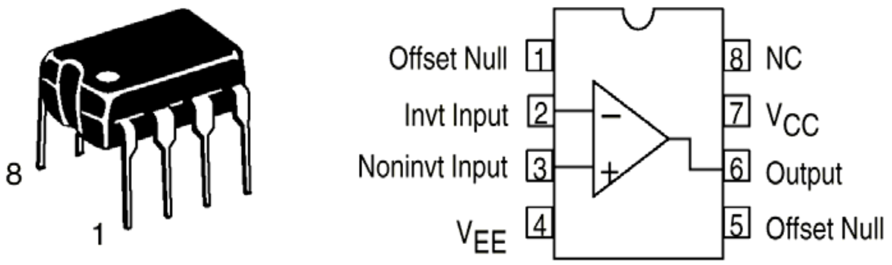
2. **<u>It is very important to place the polarised electrolytic capacitors into the circuit with the correct polarity.</u>**

   <u>Failure to do so will result in the capacitor failing catastrophically which may cause personal injury. If this happens, you will be awarded 0 marks for the lab and asked to leave!</u>

Laboratory Preparation

- We are going to be using several integrated circuits (ICs) in this and the following labs. It is important to be able to recognise the standard pin-outs of an IC. All ICs conform to a standard pin numbering scheme. There is usually a notch or mark on one end of the chip. With the notch oriented to the left, pin 1 is the first pin on the bottom of the package. The pins are then numbered in a counter-clockwise direction. An example is shown below for the TL071 op-amp used in this lab.

IC package details



TL071 package details

Figure L2.1

- Precautions should be taken to ensure that the power supply for the IC never becomes reversed in polarity or that the IC is not inadvertently installed backwards as an unlimited current surge through internal *p-n* junctions could cause fusing of the internal conductors and result in a destroyed IC.

- It would be a good idea to plan the layout of all the circuits as they will appear on your breadboard before you begin. This will minimise construction time in the lab, and assist in debugging circuits that do not appear to be working.

- A pair of pliers, a pair of wire cutters and a pair of wire strippers would be handy to wire a neat circuit; straighten bent leads; insert components into the breadboard etc. If you have any of these tools, bring them to the lab!

L2.4

Noninverting Amplifier

Noninverting amplifiers have an extremely high input resistance, and a very precise gain. The only disadvantage is that they can only produce a gain greater than or equal to 1.

A noninverting amplifier is illustrated in the figure below:

Noninverting amplifier

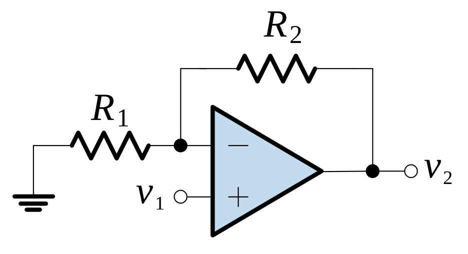


Figure L2.2

The closed-loop voltage gain is:

$$A_v = \frac{v_2}{v_1} = 1 + \frac{R_2}{R_1}$$

In the Lab – Noninverting Amplifier

1. Measure a 1 kΩ resistor for R_1 and a 10 kΩ resistor for R_2. Record the measured value of resistance in Table L2.1.

R_1 Measured	R_2 Measured	v_1 Measured	A_v Computed	v_2 Computed	v_2 Measured	$v_{(-)}$ Measured
		500 mV_{pp}				

Table L2.1

2. Using the measured resistances, compute the closed-loop gain of the noninverting amplifier. The closed-loop gain equation is given above.

3. Calculate v_2 using the computed closed-loop gain, and record the value in Table L2.1.

4. Connect the circuit shown in Figure L2.3. **Note the polarity of the DC supply's decoupling capacitors.** Set the function generator for a 500 mV_{pp} sinusoidal wave at 1 kHz. The sinusoid should have no DC offset.

Noninverting amplifier

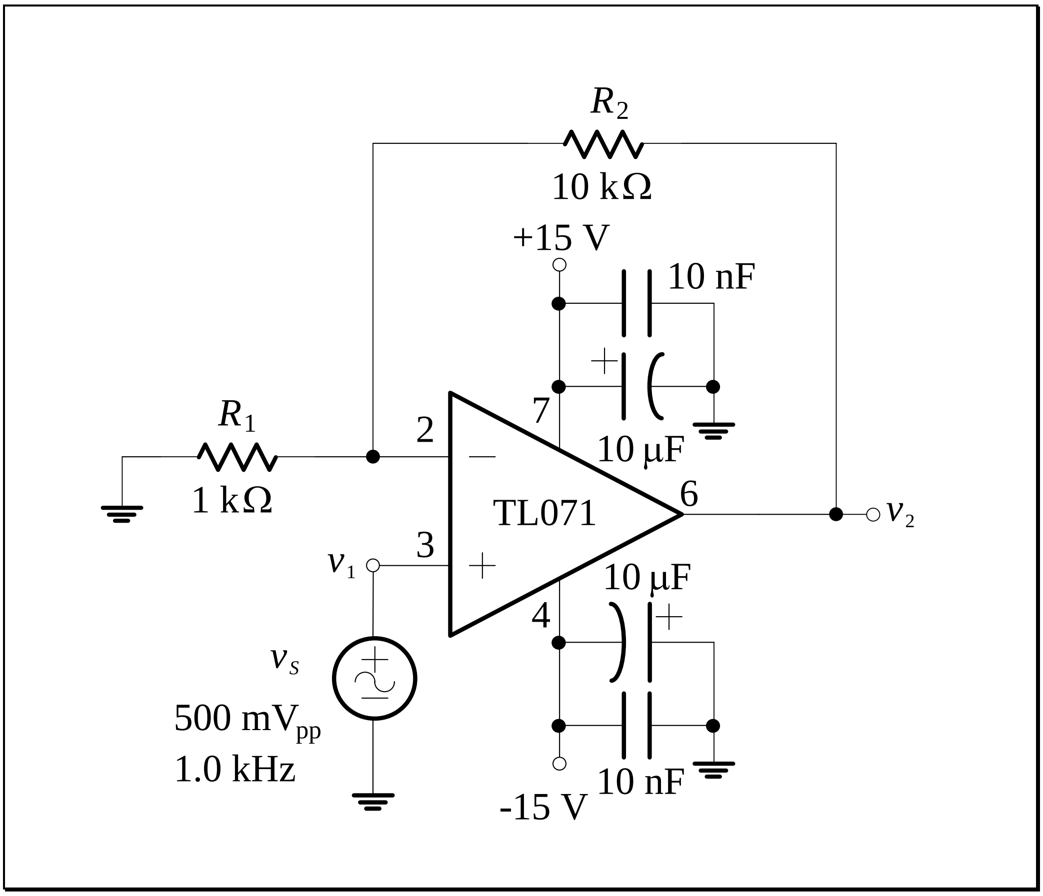


Figure L2.3

5. Observe the input, v_1 on channel 1 of the DSO and v_2 on channel 2.

In all subsequent parts of the lab, observe the input on channel 1 of the DSO and the output on channel 2. You may need to adjust the vertical attenuation settings on the DSO to obtain accurate readings.

6. Measure the output voltage, v_2. Record the measured value in Table L2.1.

7. Measure the feedback voltage at pin 2 using Channel 2. Record the measured value in Table L2.1.

L2.6

Questions – Noninverting Amplifier

1. Express the ***measured*** gain of the amplifier in dB.

   Answer:

2. If $R_2 = 0$ and $R_1 = \infty$, what is the gain?

   Answer:

   What is this amplifier called?

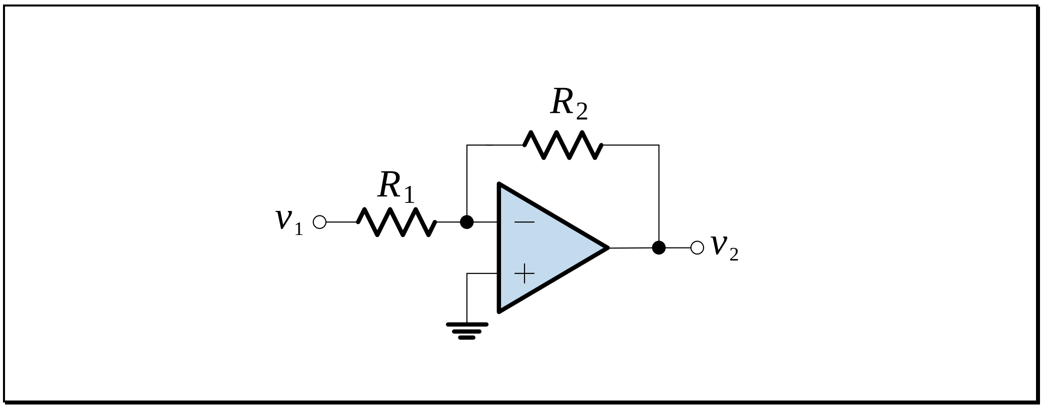
3. Explain the voltage measured at pin 2.

   Answer:

Inverting Amplifier

Inverting amplifiers can produce any value of gain, but they invert the output signal.

An inverting amplifier is illustrated in the figure below.



Inverting amplifier

Figure L2.4

The closed-loop voltage gain is:

$$A_v = \frac{v_2}{v_1} = -\frac{R_2}{R_1}$$

In the Lab – Inverting Amplifier

1. Use the same resistors for R_1 and R_2 as for the noninverting amplifier. Record the measured values in Table L2.2.

R_1 Measured	R_2 Measured	v_1 Measured	A_v Computed	v_2 Computed	v_2 Measured	$v_{(-)}$ Measured
		500 mV_{pp}				

Table L2.2

2. Using the measured resistances, compute and record the closed-loop gain of the inverting amplifier.

3. Calculate v_2 using the computed closed-loop gain, and record the value in Table L2.2.

4. Connect the circuit shown in Figure L2.5. Set the function generator for a 500 mV$_{pp}$ sinusoidal wave at 1 kHz, with no DC offset.

Inverting amplifier

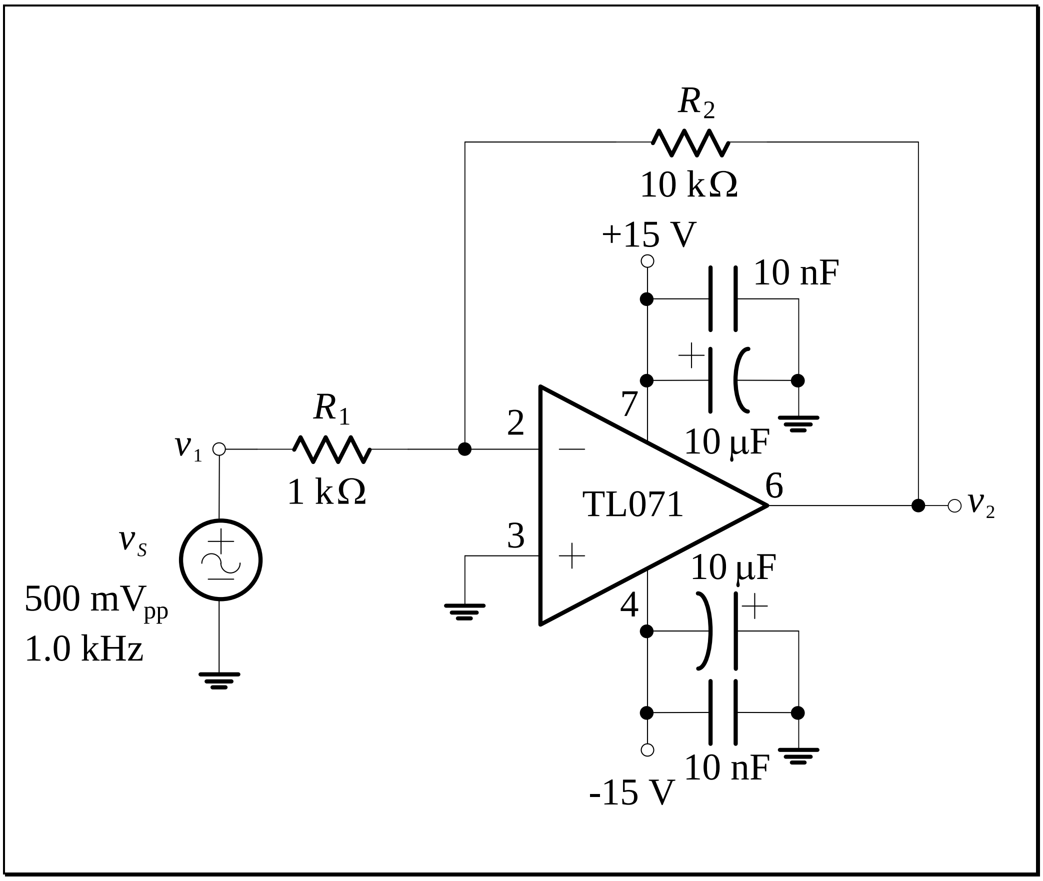


Figure L2.5

5. Measure the output voltage, v_2 (note the 180° phase compared to v_1). Record the measured value in Table L2.2.

6. Measure the voltage at pin 2. This point should be at a *virtual common* because of the effect of negative feedback. Record the measured value in Table L2.2.

Questions – Inverting Amplifier

1. Express the ***measured*** gain of the amplifier in dB.

   Answer:

2. What output would you expect if R_2 were open?

   Answer:

3. Explain the voltage measured at pin 2.

   Answer:

L2.10

Lab Assessment [2 marks]

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

1. Demonstrate the measurement of the gain, in dB, for the inverting amplifier shown in Figure L2.5.
2. Draw a schematic diagram of a noninverting amplifier with a gain of +6.021 dB, using only the components from the lab kit.
3. Explain the voltage measured at pin 2 of Figure L2.3.
4. In Figure L2.2, if $R_2 = 10\ \text{k}\Omega$ and $R_1 = \infty$, what is the gain?

Marking

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

Lab 3 – Comparator, Integrator, Differentiator

Comparator. Integrator. Differentiator.

Introduction

A comparator uses the op-amp in an *open-loop* mode. For a very small input voltage, the output will saturate close to one of the power supply voltages due to the very large gain of the op-amp.

With a capacitor placed in the feedback path of an inverting amplifier, we can make an integrator. A perfect integrator is hard to make due to limitations of real op-amps, but we can make an integrator very close to the ideal.

By putting a capacitor on the input instead of in the feedback path, we can make a differentiator. Both the integrator and differentiator have applications in waveform generation and signal processing.

Objectives

1. To build and test several op-amp circuits, and to determine their responses to several input signals.

Equipment

- 1 Digital Storage Oscilloscope (DSO) – Agilent 54621A
- 1 Mini-Lab – BWD 604
- Op-amp – 2 x TL071
  Resistors – 1 x 1 kΩ, 1 x 5.1 kΩ, 1 x 20 kΩ, 2 x 100 kΩ, 1 x 270 kΩ
  Potentiometer – 1 x 10 kΩ
  Capacitors – 2 x 10 μF, 3 x 10 nF
  Diodes –1 x green LED, 1 x red LED
  Breadboard, Hook-up wire, 2 x 4mm leads.

L3.2

Safety

Cat. A lab

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

Warning!

Remember:

1. **When wiring the circuits, ensure that the power supply is switched off.**

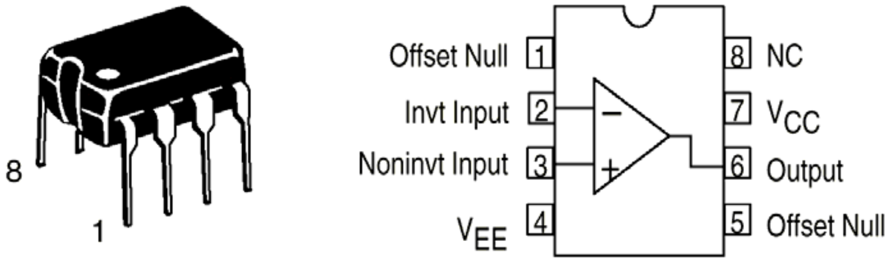
2. **<u>It is very important to place the polarised electrolytic capacitors into the circuit with the correct polarity.</u>**

   <u>Failure to do so will result in the capacitor failing catastrophically which may cause personal injury. If this happens, you will be awarded 0 marks for the lab and asked to leave!</u>

Laboratory Preparation

- The pin-out for the TL071 op-amp is given below:

Op-amp package details



TL071 package details

Figure L3.1

- ***For the TL071, pin 7 is connected to the positive supply and pin 4 is connected to the negative supply.***

- The pin-out for an LED is given below. The cathode is marked by a flat edge on the lens. New LEDs also have a shorter lead on the cathode.

LED package details

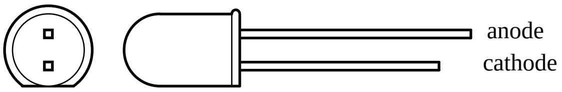


Figure L3.2

- It would be a good idea to plan the layout of all the circuits as they will appear on your breadboard before you begin. This will minimise construction time in the lab, and assist in debugging circuits that do not appear to be working.

- A pair of pliers, a pair of wire cutters and a pair of wire strippers would be handy to wire a neat circuit; straighten bent leads; insert components into the breadboard etc. If you have any of these tools, bring them to the lab!

Comparator

A comparator is an example of a non-linear op-amp circuit. It is a switching device that produces a high or low output, depending on which of the two inputs is larger. A comparator is made from an op-amp with no feedback connection (*open-loop*) as shown in Figure L3.3.

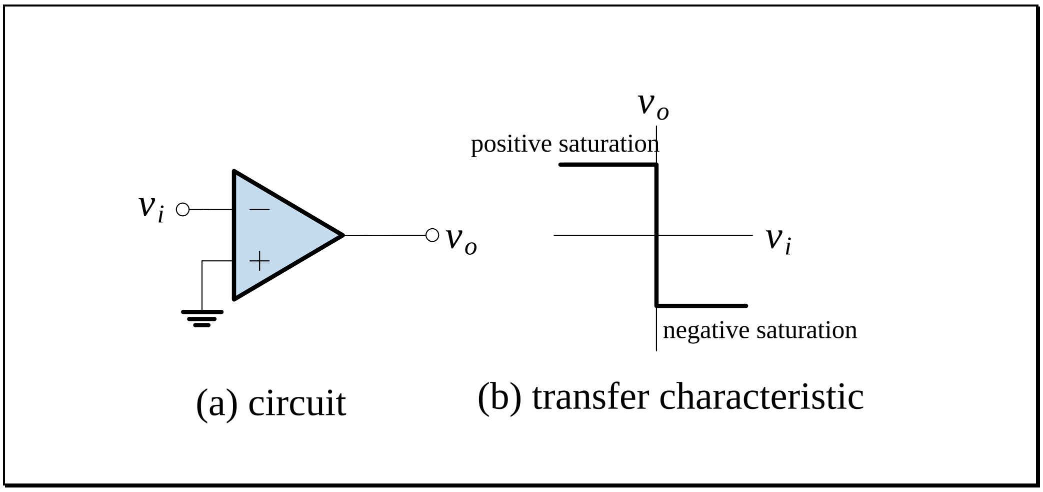


Figure L3.3

When the non-inverting input is only slightly larger than the inverting input, the output goes to positive saturation; otherwise it goes to negative saturation. Although general purpose op-amps (like the TL071) can be used as comparators, specially designed op-amps (like the LM311) can switch faster and have additional features not found on general-purpose op-amps. For non-critical applications, general purpose op-amps are satisfactory and will be used in this lab.

In the Lab – Comparator

1. Construct the comparator circuit shown in Figure L3.4. ***Note that the power connections on this and remaining circuits in this lab are not shown explicitly – connect the TL071's power supply according to the pin-out given in Figure L3.1. Use a ±15 V supply. Make sure you add 10 μF and 10 nF bypass capacitor from each DC supply to the common.***

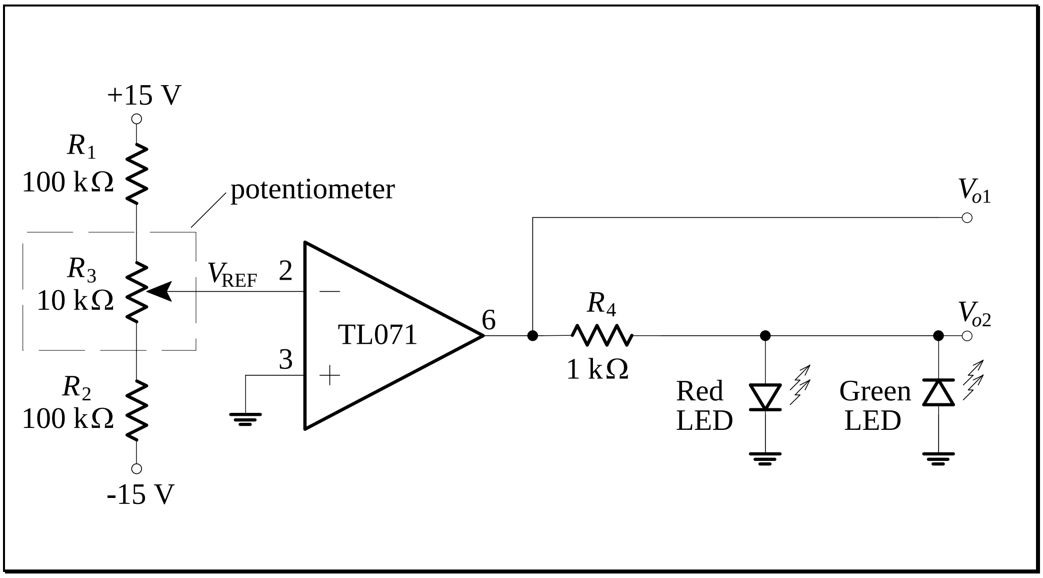


Figure L3.4

2. Vary the potentiometer. Measure the output voltages when the red LED is on and then when the green LED is on. Record the output voltages, V_{o1} and V_{o2}, in Table L3.1.

Red ON		Green ON		V_{REF} Threshold
V_{o1}	V_{o2}	V_{o1}	V_{o2}	

Table L3.1

3. Set the potentiometer to the threshold point (where one diode turns off and the other turns on). Measure and record V_{REF} at the threshold. It should be very close to 0 V.

L3.6

Integrator and Differentiator

Two circuits which have applications in waveform generation and signal processing are the integrator and differentiator.

An integrator produces an output voltage that is proportional to the *integral* (sum) of the input voltage waveform over time.

A differentiator circuit produces an output that is proportional to the *derivative* or rate of change of the input voltage over time.

Basic op-amp integrator and differentiator circuits are illustrated below.

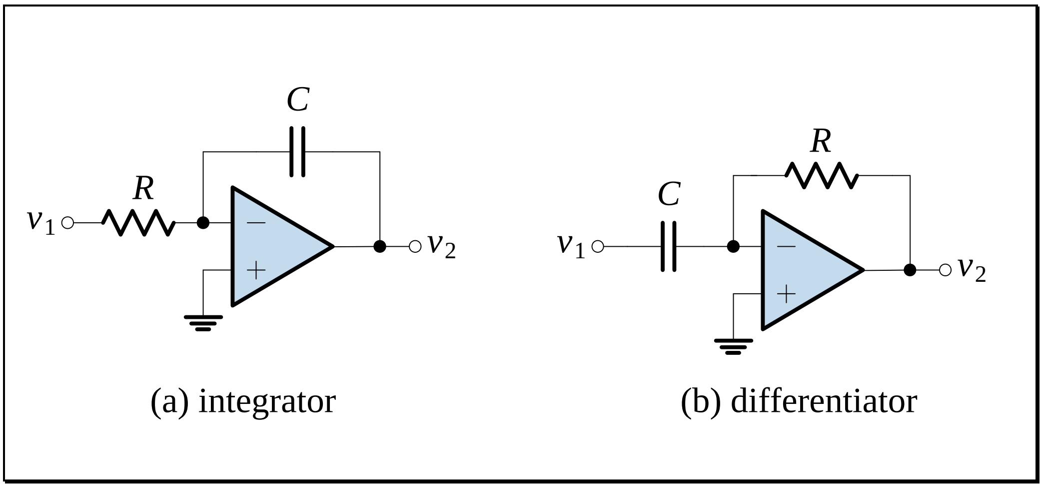


Figure L3.5

The output voltage of the integrator is given by:

$$v_2 = -\frac{1}{RC}\int_{-\infty}^{t} v_1 dt$$

The output voltage of the differentiator is given by:

$$v_2 = -RC\frac{dv_1}{dt}$$

In the Lab – Integrator

1. We will test the effects of the comparator on a sinusoidal wave input and add an integrating circuit to the output of the comparator. Connect the circuit shown in Figure L3.6 with a 1 V_{pp} sine wave input at 1 kHz as illustrated. Ensure that there is no DC offset in the FG's output.

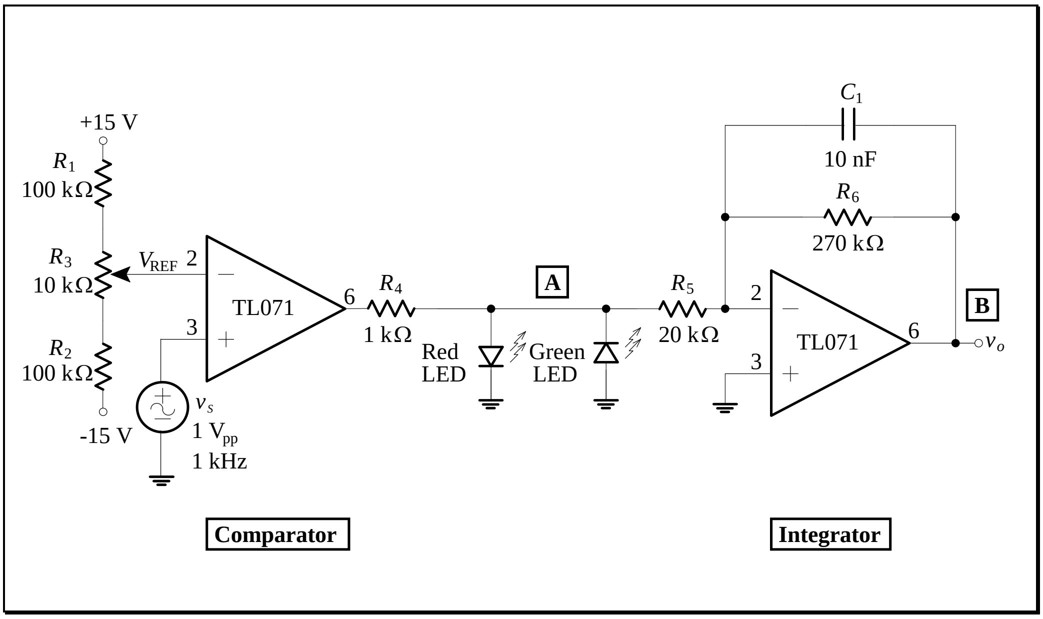
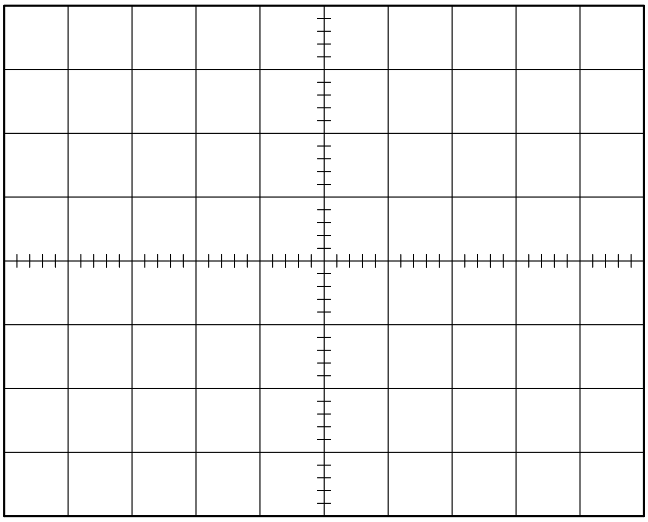


Figure L3.6

2. Observe the waveforms from the comparator (point **A**) and from the integrator (point **B**). Adjust R_3 so that the waveform at **B** is centred about 0 V. Sketch the observed waveforms in the correct time relationship below. Show the voltages and time on your plot.

Integrator input and output waveforms



3. Vary R_3 while observing the output of the comparator and the integrator.

4. For each of the faults listed in Table L3.2, see if you can predict the effect on the circuit. Then apply the fault and check your prediction. At the end of this step, restore the circuit to normal operation.

Fault	Symptoms
No negative power supply	
Red LED open	
C_1 open	
R_6 open	

Table L3.2

In the Lab – Differentiator

1. Replace the integrator part of the previous circuit with the differentiator shown below.

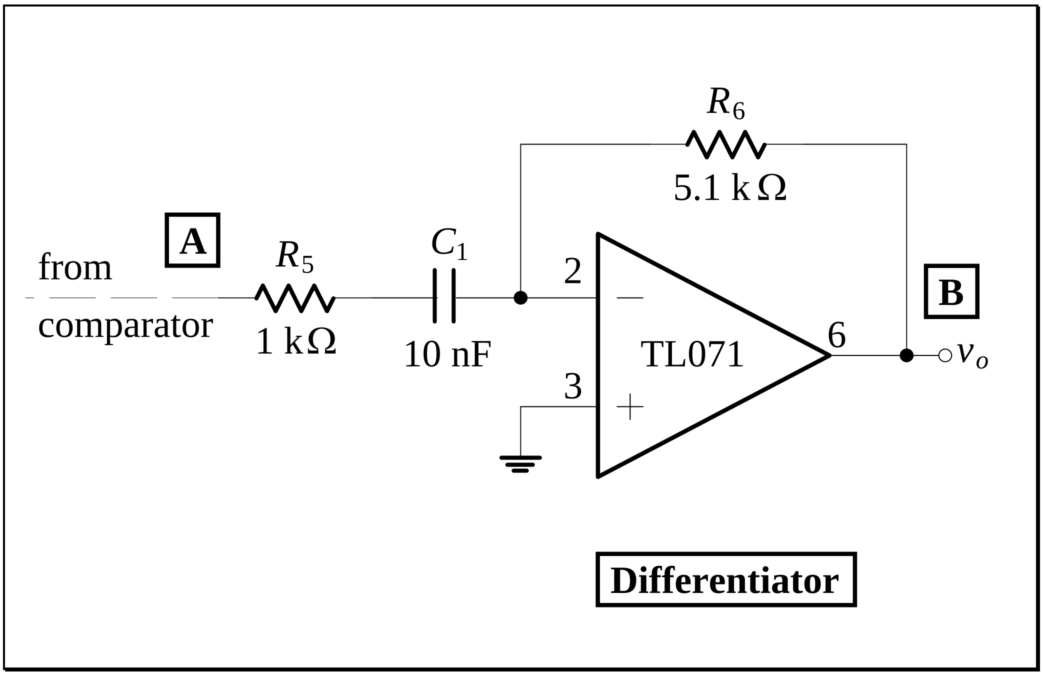
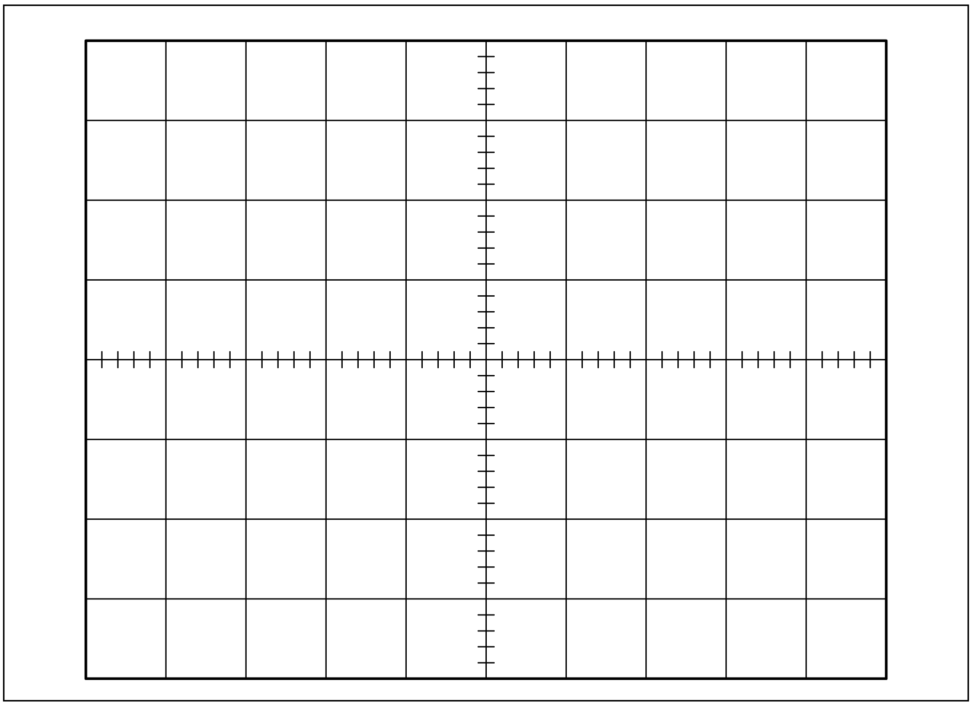


Figure L3.7

2. Observe the input and output waveforms of the differentiator. Sketch the observed waveforms below, showing the voltages and time.



Differentiator waveforms

L3.10

Questions – Integrator

1. For the integrator circuit in Figure L3.6, what is the purpose of R_6?

Answer:
(You may like to remove R_6 momentarily and observe the effect.)

Questions – Differentiator

1. What type of circuit will produce leading-edge and trailing-edge pulses from a square wave input?

Answer:

Lab Assessment [2 marks]

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

1. Show the integrator input and output waveforms.

2. Show the differentiator input and output waveforms.

3. Write down (do not solve) the differential equation governing the real integrator shown in Figure L3.6 (using symbols, not numerical values).

4. If the output of the comparator of Figure L3.6 has a DC component, what effect will be observed at the output of the integrator?

Marking

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

Lab 4 – Summing Amplifier, Precision FWR

Summing amplifier. Precision full-wave rectifier.

Introduction

The op-amp is the most versatile electronic building block. Circuits based on the op-amp nearly always use a *feedback* configuration. Feedback has many desirable properties, as we will see.

One advantage of the inverting amplifier configuration is that it can readily be converted to a summing amplifier. A summing amplifier can add multiple signals together.

A precision half-wave rectifier removes the forward-drop of a diode through the use of feedback, so we can rectify signals in the millivolt range. If we also use a summing amplifier, then we can make a precision full-wave rectifier.

Objectives

1. To build and test a summing amplifier in the configuration of a 3-bit digital-to-analog converter.

2. To build a precision full-wave rectifier.

Equipment

- 1 Digital Storage Oscilloscope (DSO) – Agilent 54621A
- 1 Mini-Lab – BWD 604
- Op-amp – 2 x TL071
  Digital IC – 1 x 74HC193 synchronous 4-bit up/down binary counter
  Diodes – 2 x 1N4148
  Resistors –1 x 3.9 kΩ, 1 x 5.1 kΩ, 4 x 10 kΩ, 1 x 20 kΩ
  Capacitors – 2 x 10 μF, 2 x 10 nF
  Breadboard, Hook-up wire, 2 x 4mm leads.

L4.2

Safety

Cat. A lab

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

Warning!

Remember:

1. **When wiring the circuits, ensure that the power supply is switched off.**

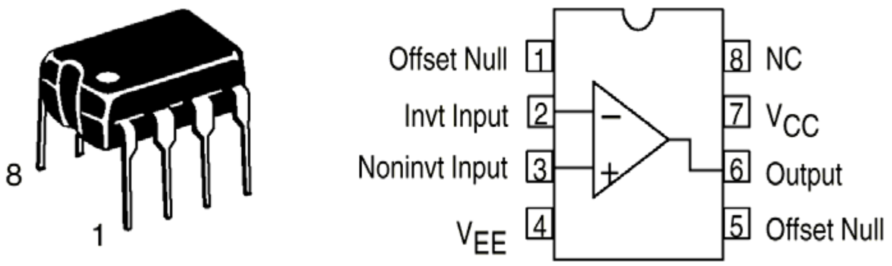
2. **<u>It is very important to place the polarised electrolytic capacitors into the circuit with the correct polarity.</u>**

   <u>Failure to do so will result in the capacitor failing catastrophically which may cause personal injury. If this happens, you will be awarded 0 marks for the lab and asked to leave!</u>

Laboratory Preparation

- The pin-out for the TL071 op-amp is given below:

IC package details



TL071 package details

Figure L4.1

- ***For the TL071, pin 7 is connected to the positive supply and pin 4 is connected to the negative supply.***

- Precautions should be taken to ensure that the power supply for the IC never becomes reversed in polarity or that the IC is not inadvertently installed backwards as an unlimited current surge through internal *p-n* junctions could cause fusing of the internal conductors and result in a destroyed IC.

- It would be a good idea to plan the layout of all the circuits as they will appear on your breadboard before you begin. This will minimise construction time in the lab, and assist in debugging circuits that do not appear to be working.

- A pair of pliers, a pair of wire cutters and a pair of wire strippers would be handy to wire a neat circuit; straighten bent leads; insert components into the breadboard etc. If you have any of these tools, bring them to the lab!

L4.4

Summing Amplifier

The summing amplifier shown in Figure L4.2 is just a multiple input version of an inverting amplifier. The current into the feedback resistor, R_f , is the sum of the currents in each input resistor. Since the inverting input is a *virtual common*, the total input current is $v_1/R_1 + v_2/R_2 + v_3/R_3$. The virtual common has the advantage of isolating the various inputs from each other. Also, the gain of each input can be set differently.

Summing amplifier

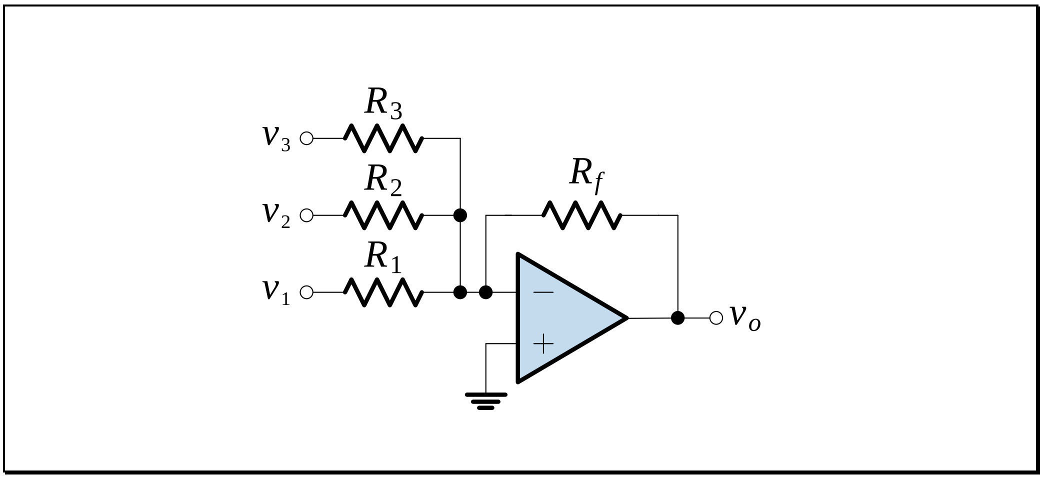


Figure L4.2

The output voltage is given by:

$$v_o = -R_f\left(\frac{v_1}{R_1} + \frac{v_2}{R_2} + \frac{v_3}{R_3}\right)$$

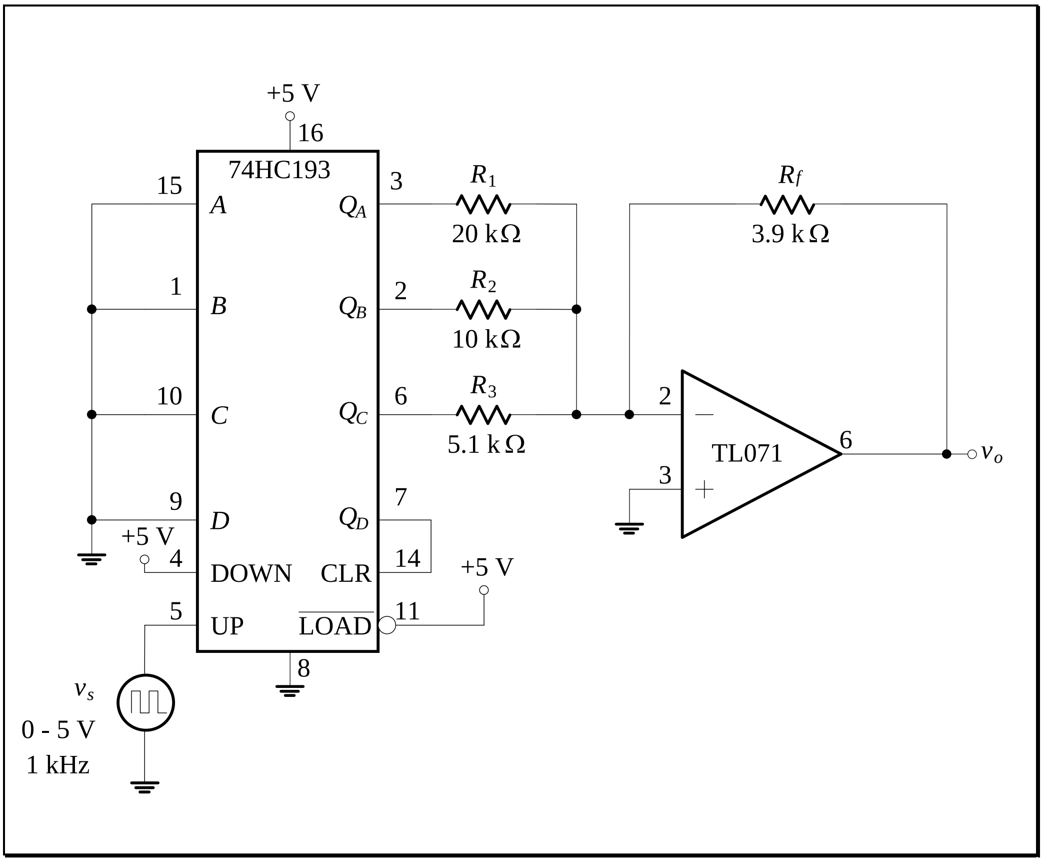
In the Lab – Summing Amplifier

1. Measure and record the values of the resistors listed in Table L4.1.

Resistor	Listed Value	Measured Value
R_1	20 kΩ	
R_2	10 kΩ	
R_3	5.1 kΩ	
R_f	3.9 kΩ	

Table L4.1

2. The circuit shown in Figure L4.3 is a summing amplifier connected to the outputs of a binary counter. The counter outputs are weighted differently by resistors R_1 through R_3, and added by the summing amplifier. The resistors and summing amplifier form a basic D/A converter.



Step generator (3-bit D/A)

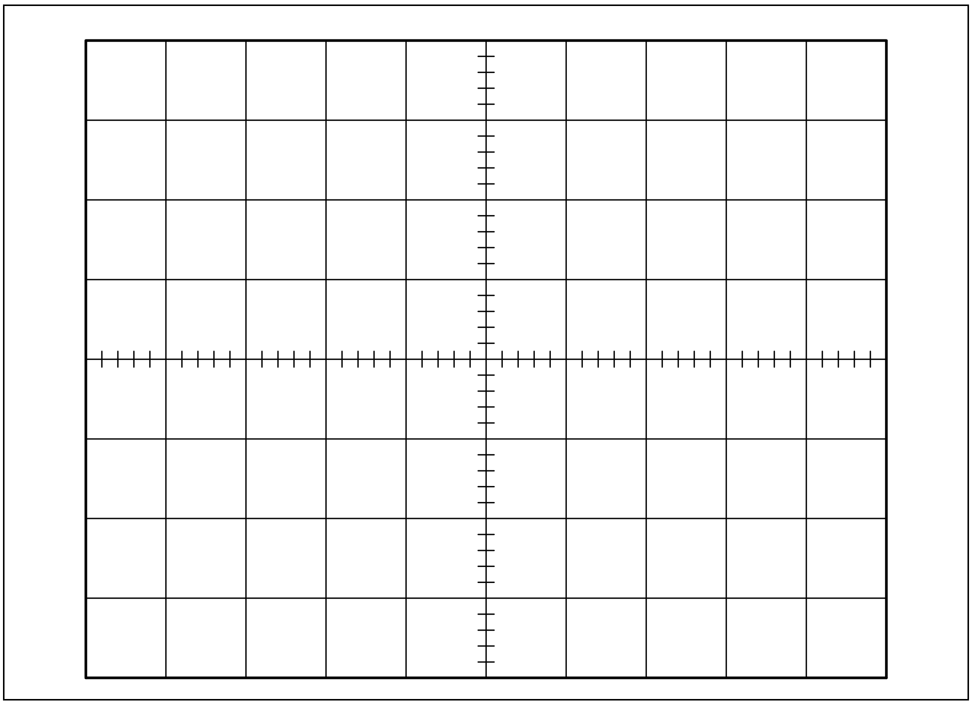
Figure L4.3

The input to the 74HC193 is a digital logic clock (approximately 0 to 5V) at 1 kHz from the function generator. ***Set up this waveform carefully using the DSO before connecting it to the circuit.***

3. Construct the circuit, using a ±15 V supply for the op-amp. ***Note that the 74HC193 counter is powered from a +5 V supply. The common of the +5 V supply must be connected to the common of the ±15 V op-amp supply.***

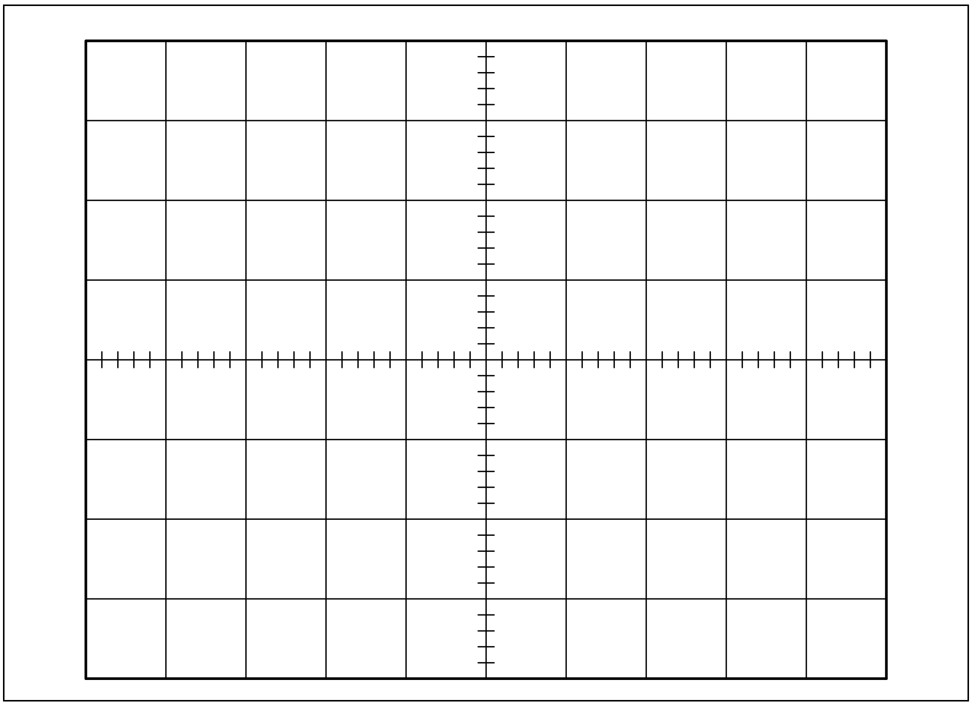
4. Observe v_o from the TL071 (trigger the DSO from channel 2, and set the mode to `Auto Level`). You should observe a series of steps. Sketch the output below. Label the voltage and time on your plot.

Step generator (D/A) waveforms



5. To see how the steps are formed, observe the Q_A, Q_B, and Q_C outputs from the 74HC193. To see the correct time relationship between the signals, keep channel 2 in place while moving the channel 1 probe. Sketch the waveforms in the correct relation below.

Summing amplifier input waveforms



Questions – Summing Amplifier

1. The step generator in Figure L4.3 forms negative falling steps starting at zero volts and going to a negative voltage (approximately –6.64 V). Explain why.

Answer: How could you modify the circuit to produce positive, rising steps at the output?

L4.8

2. Assume you have a function generator that does not have a DC offset control. Show how you could use a summing amplifier to add or subtract a DC offset from the output.

Answer:

Precision Full-Wave Rectifier

A precision inverting half-wave rectifier is shown in Figure L4.4. The circuit can be recognised as an inverting amplifier with a diode, D_2, added to the feedback path. When this diode is forward-biased, it closes the feedback loop, and the output is given by:

$$v_2 = -\frac{R_2}{R_1} v_1, \quad v_1 > 0$$

When D_1 is forward-biased ($v_1 < 0$), it closes the feedback loop and the output is 0V.

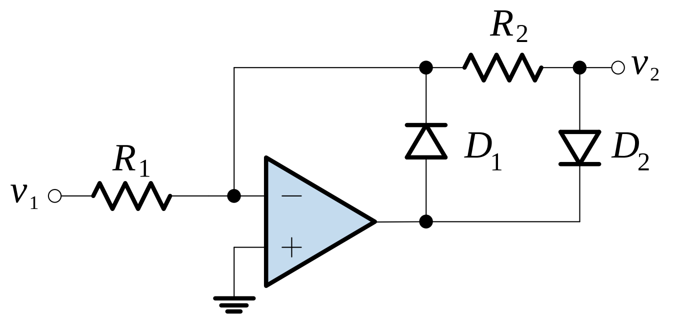


Figure L4.4

By combining the inverting half-wave rectifier with a summing amplifier, a precision full-wave rectifier can be constructed, as shown below.

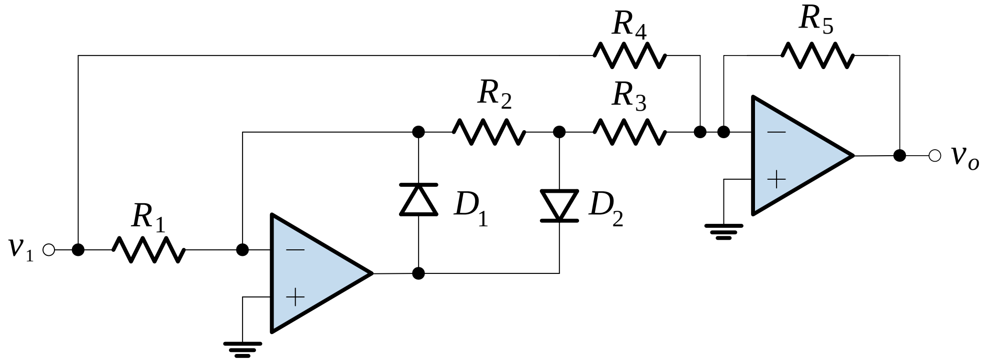


Figure L4.5

L4.10

In the Lab – Precision Full-Wave Rectifier

1. Construct the following circuit.

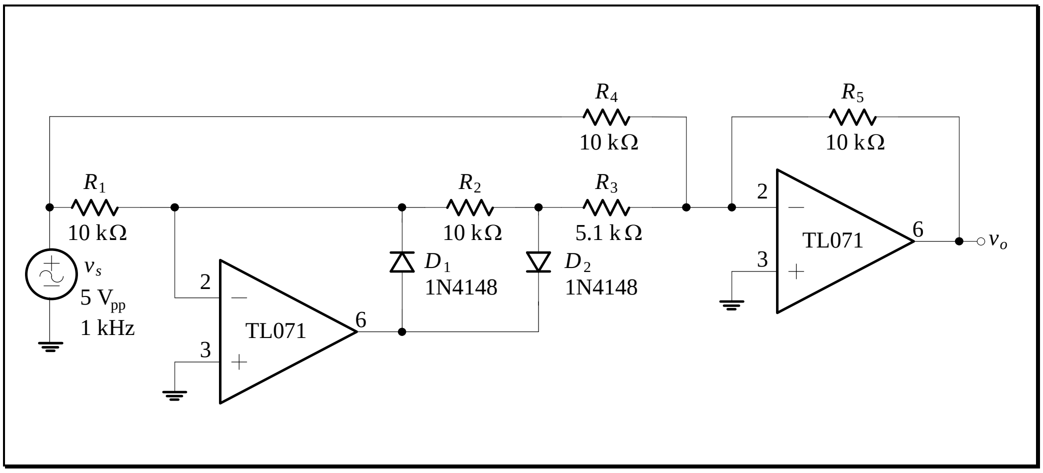
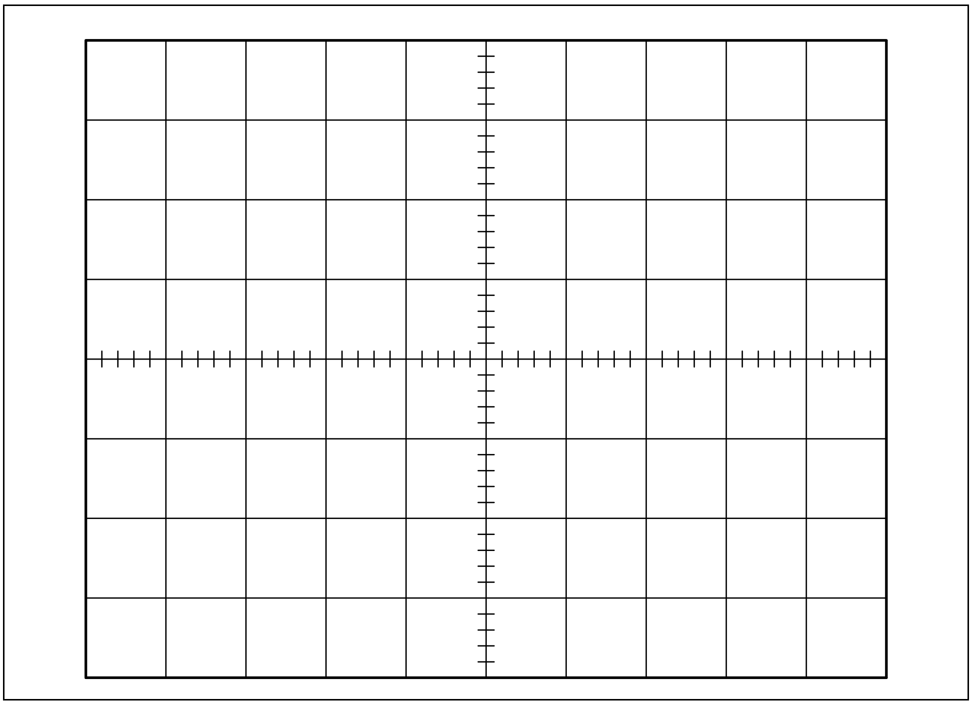


Figure L4.6

1. Sketch the waveforms at the left side of R_3 and R_4 (inputs to the summing amplifier) and v_o.

Summing amplifier input and output waveforms for a precision full-wave rectifier



Questions – Precision Full-Wave Rectifier

1. The gain for the summing amplifier in Figure L4.6 is not the same for both inputs. Explain why.

Answer:

2. What would be the output of the circuit if D_1 were removed? Explain why.

Answer:

(You may like to remove D_1 momentarily and observe the effect.)

L4.12

Lab Assessment [2 marks]

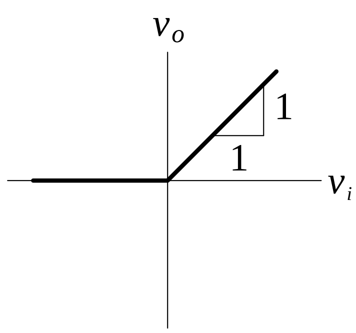
When **all** lab work is completed, you will be asked by a tutor to:

1. Show the step generator (D/A) waveforms.

2. Draw the schematic of a circuit that adds or subtracts a DC offset to a signal.

3. Sketch the output of the circuit shown in Figure L4.6 if D_1 were removed.

4. Draw the schematic of a precision non-inverting half-wave rectifier with transfer characteristic:



Marking

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

Lab 5 – Op-Amp Limitations

Op-amp limitations. Output voltage swing. Output current limiting. Input offset voltage. Input bias and offset currents. Slew rate limiting. Gain-bandwidth product.

Introduction

Real operational amplifiers do not exhibit the ideal characteristics assumed in the first-order analysis of op-amp circuits: infinite gain, infinite input impedance, zero output impedance, infinite bandwidth, zero output signal for zero input signal, etc. Some of the basic imperfections and limitations of real op-amps are expressed as:

Output Voltage Swing: The output voltage swing, $V_o^+ - V_o^-$, is defined as the maximum voltage available at the device output with a given load (usually it is 2-4 V less than $V_{CC} - V_{EE}$, i.e. the total supply voltage of the op-amp).

Output Current Limiting: The output current of an op-amp is normally limited by design to prevent excessive power dissipation within the device which would destroy it.

Input Offset Voltage: The input offset voltage, V_{OS}, is defined as the voltage required in series with the input to drive the output to zero.

Input Bias and Offset Currents: The input bias current, I_B, is defined as the average value of the DC bias current required at either input of the op-amp. The input offset current, I_{OS}, is defined as the difference between the two bias currents at the inputs of the op-amp.

Slew Rate Limiting: The slew rate, *SR*, may be defined as the limiting rate of change of output voltage in response to a large input step change.

Gain-Bandwidth Product: The gain-bandwidth product, *GB*, is defined as the frequency at which the open-loop gain would become unity (0 dB), if the amplifier had a single pole rolloff (i.e. -20 dB/decade gain slope, like a Single Time Constant – STC – lowpass network).

L5.2

Although no op-amp is ideal, modern processing techniques yield devices that come close, at least in some parameters. This is by design. In fact, different op-amps are optimized to be close to ideal for some parameters, while other parameters for the same op-amp may be quite ordinary (some parameters can be improved, but only at the expense of others). It is the designer's function to select the op-amp that is closest to ideal in ways that matter to the application, and to know which parameters can be discounted or ignored.

For this reason, it is very important to understand the specifications and to compare the limitations of the different commercially available op-amps, in order to select the right op-amp for a specific application.

Objectives

1. To examine some of the limitations of real operational amplifiers.

Equipment

- 1 Digital Storage Oscilloscope (DSO) – Agilent 54621A
- 1 Mini-Lab – BWD 604
- Op-amp – 1 x TL071

  Resistors – 1 x 390 Ω, 2 x 1 kΩ, 1 x 2.2 kΩ, 1 x 4.7 kΩ, 3 x 10 kΩ
  1 x 100 kΩ, 1 x 1 MΩ

  Capacitor –2 x 10 nF, 2 x 10 μF

  Breadboard, Hook-up wire, 2 x 4mm leads.

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

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

Remember:

Warning!

1. **When wiring the circuits, ensure that the power supply is switched off.**

2. **It is very important to place the polarised electrolytic capacitors into the circuit with the correct polarity.**

   Failure to do so will result in the capacitor failing catastrophically which may cause personal injury. If this happens, you will be awarded 0 marks for the lab and asked to leave!

L5.4

Lab Work

You will operate the op-amp at the extremes of its performance to witness some of the limitations of a real op-amp.

Output Voltage Swing

1. Connect the circuit shown in Figure L5.1, with $V_{CC} = +10\text{ V}$ and $V_{EE} = -10\text{ V}$, and no load resistor. Note that decoupling capacitors are not shown in the circuit, but they should be present as usual. Set the function generator for a 500 mV_{pp} sinusoidal wave at 500 Hz.

Noninverting amplifier for output testing

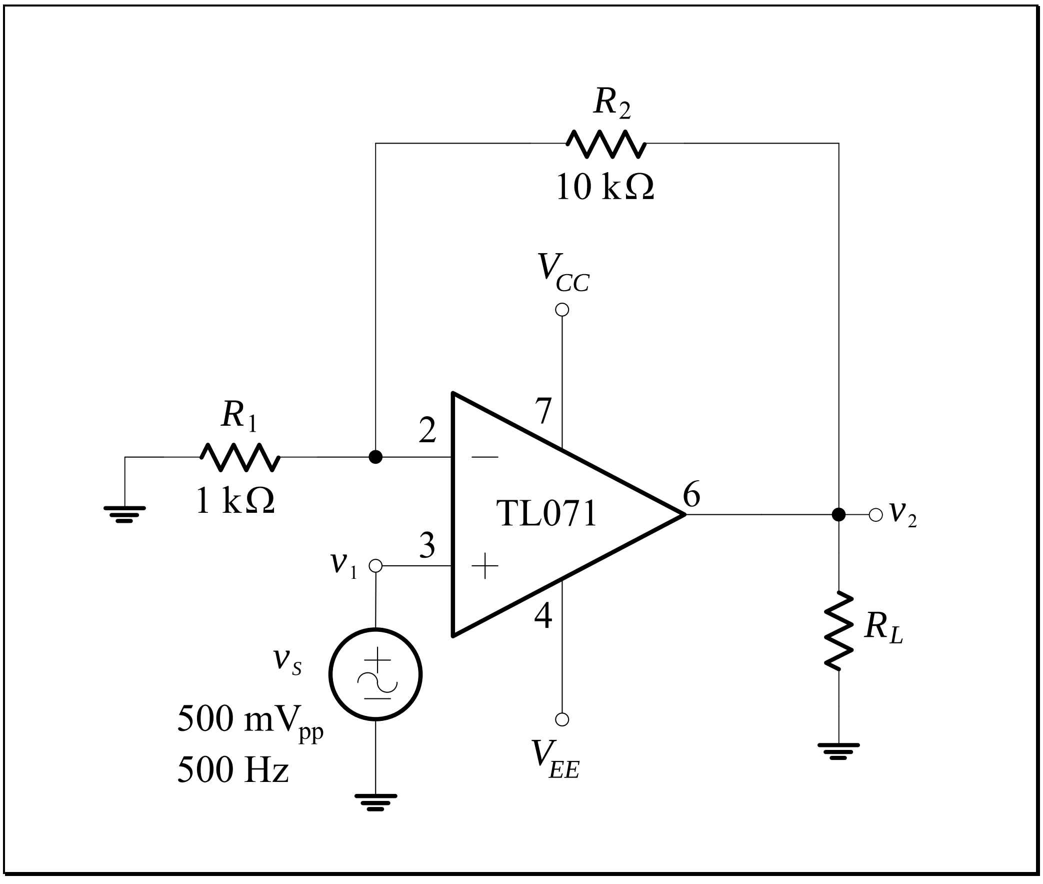


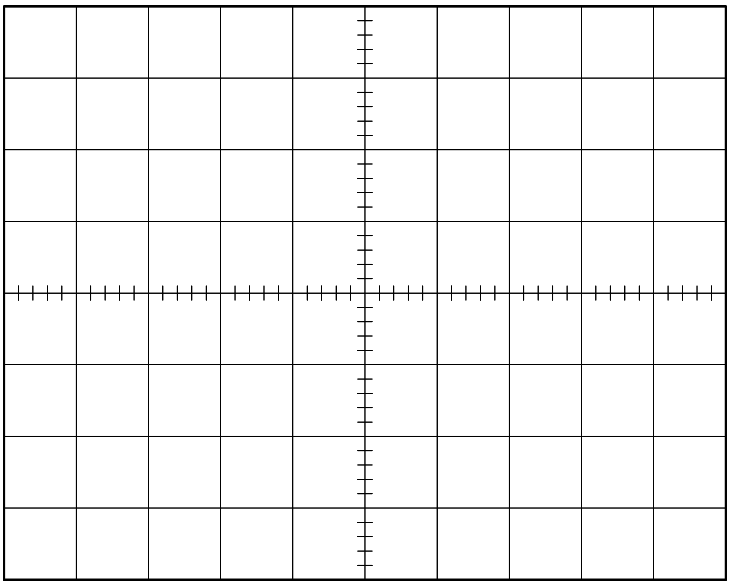
Figure L5.1 – Noninverting Amplifier for Output Testing

2. Observe the input, v_1 on Channel 1 of the DSO and the output v_2 on Channel 2 to confirm that there is a sinusoidal output of the correct magnitude.

3. Starting from an input amplitude of $500\,\mathrm{mV_{pp}}$, increase the function generator amplitude slowly until “clipping” occurs. Use the DSO cursors to record the maximum and minimum output voltage in the table below. Repeat with a load resistor of $R_L = 2.2\,\mathrm{k\Omega}$.

Power Supply	**Maximum Output Voltage**	
	No load resistor	$R_L = 2.2\,\mathrm{k\Omega}$
$V_{CC} = +10\,\mathrm{V}$ $V_{EE} = -10\,\mathrm{V}$	$v_{2\max} =$ $v_{2\min} =$	$v_{2\max} =$ $v_{2\min} =$
$V_{CC} = +15\,\mathrm{V}$ $V_{EE} = -15\,\mathrm{V}$	$v_{2\max} =$ $v_{2\min} =$	$v_{2\max} =$ $v_{2\min} =$

4. With a load resistor of $R_L = 2.2\,\mathrm{k\Omega}$ and the source set to $3\,\mathrm{V_{pp}}$, draw the output (Ch 2) waveform, ensuring that the sketch is labelled with voltage and time scales:



5. Change the power supplies to $V_{CC} = +15\,\mathrm{V}$ and $V_{EE} = -15\,\mathrm{V}$ and repeat steps 1 to 4.

L5.6

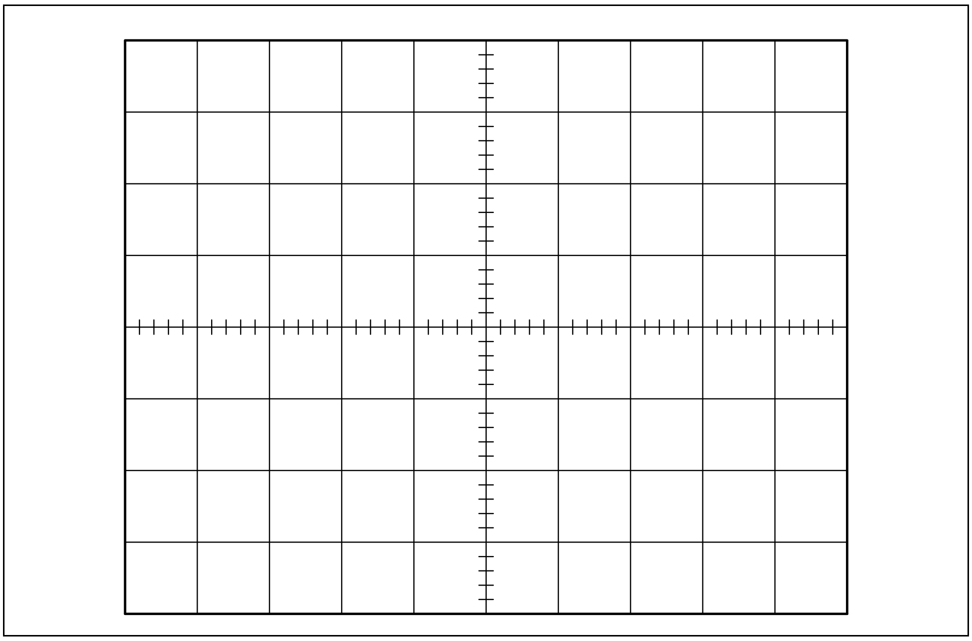
Output Current Limiting

1. Using the same circuit, with the DC supplies set to $V_{CC} = +15\,\text{V}$ and $V_{EE} = -15\,\text{V}$, change the load resistor to $R_L = 390\,\Omega$. Start with a small input signal, and increase the amplitude until distortion is observed on the output waveform.

2. Use the DSO cursors to record the measured values in the table below:

Maximum Output	
Voltage	**Current**
$\hat{v}_{2\,\max} =$	$\hat{i}_{2\,\max} = \hat{v}_{2\,\max} / R_L =$

3. With the source set to $3\,\text{V}_{\text{pp}}$, draw the input (Ch1) and output (Ch 2) waveforms, ensuring that the sketch is labelled with voltage and time scales:



4. Remove the load resistor.

Slew Rate Limiting

The slew-rate limit of an op-amp is caused by a current source within the amplifier that limits the amount of current that can be supplied by the first stage of the amplifier. When the amplifier is pushed to the point where this limit is reached, it can no longer function properly. The slew-rate limit manifests itself as a maximum value of dv_{out}/dt for the amplifier because there is an internal amplifier capacitance that must be charged by the first-stage output current and a first-stage current limit thus corresponds to a maximum dv/dt for this capacitor.

1. Build the following circuit:

Noninverting amplifier for slew rate testing

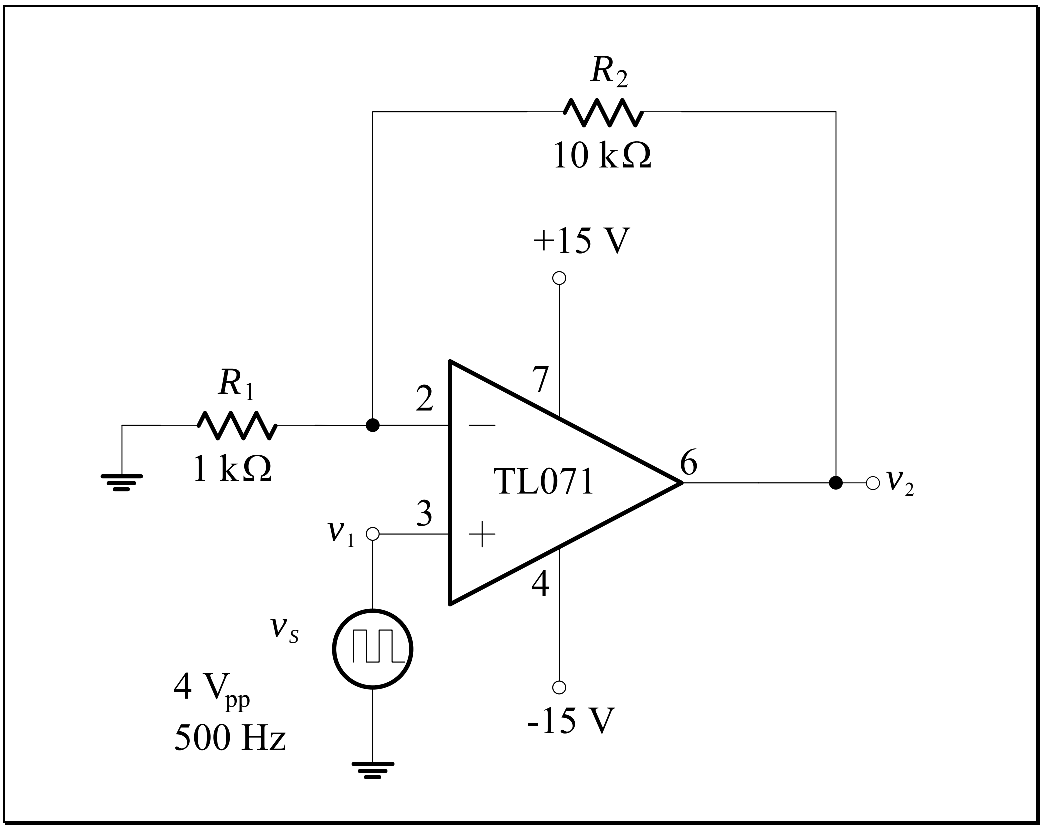


Figure L5.2 – Noninverting Amplifier for Slew Rate Testing

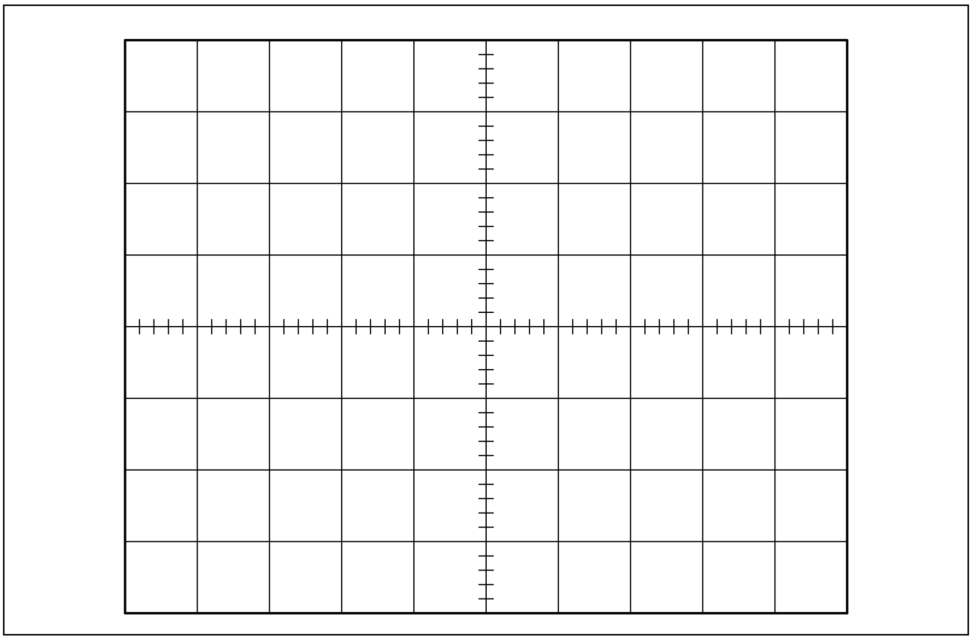
2. Apply a 500 Hz square wave input with an amplitude of $4\ \mathrm{V_{pp}}$.

3. Press `Main/Delayed`. Change the `Time Ref` softkey to Left. This will facilitate sketching the response.

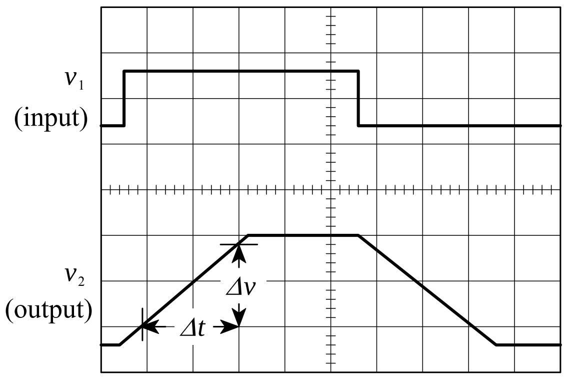
4. Due to the "high frequency" of the rising edge of the square wave, we need to ensure that the signal "gets through" the DSO's trigger path. Press `Main/Delayed`, and ensure that `HF Reject` is **off**.

5. Set the DSO horizontal time base to 500 ns / div.

6. Draw the input (Ch 1) and output (Ch 2) waveforms, ensuring that the sketch is labelled with voltage and time scales:



7. Measure the slope of the leading edge of the output waveform:



$$Slew\ rate = \frac{\Delta v}{\Delta t}$$

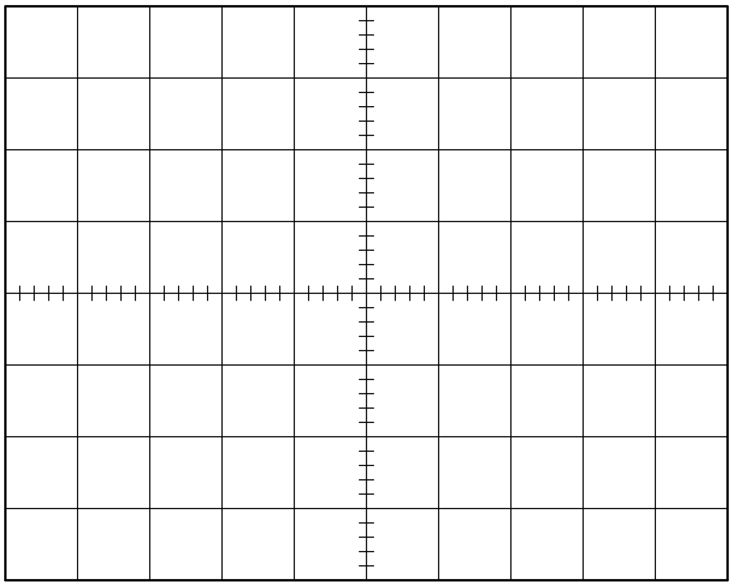
8. Record the measured value of the slew rate, expressed in units of V / μs :

$$SR = \frac{\Delta v}{\Delta t} =$$

9. Set $R_2 = 0\,\Omega$ to make a unity-gain buffer.

10. Change the input to a sinusoid with $10\,\mathrm{V_{pp}}$, and increase the frequency until a visually noticeable distortion is observed on the output waveform (i.e. the output is starting to deviate from an ideal sinusoid – it eventually turns into a triangle if the frequency is high enough).

11. Draw the input and output (Ch 1 and Ch 2) waveforms, ensuring that the sketch is labelled with voltage and time scales:



12. Record the frequency at which distortion occurs, and the output amplitude:

Frequency	**Voltage Magnitude**
$f =$	$\hat{v}_{2\max} =$

13. Calculate the slew rate from the above measurements, in units of $\mathrm{V/\mu s}$.

$$SR = 2\pi\ f\ \hat{v}_{2\max}$$
$$=$$

L5.10

Input Offset Voltage

The input offset voltage, V_{OS}, is a DC voltage which must be applied to the op-amp's noninverting *input* terminal to drive the *output* voltage to 0 V. The input offset voltage arises as a result of the unavoidable mismatches present in the input transistors (in the input differential stage) inside the op-amp.

1. Measure with the digital multimeter and write down the exact values of the resistors to be used in the circuit:

Resistor	Listed Value	Measured Value
R_1	10 kΩ	
R_2	1.0 MΩ	

Table L5.1

2. Build the following circuit:

Noninverting amplifier for input offset voltage testing.

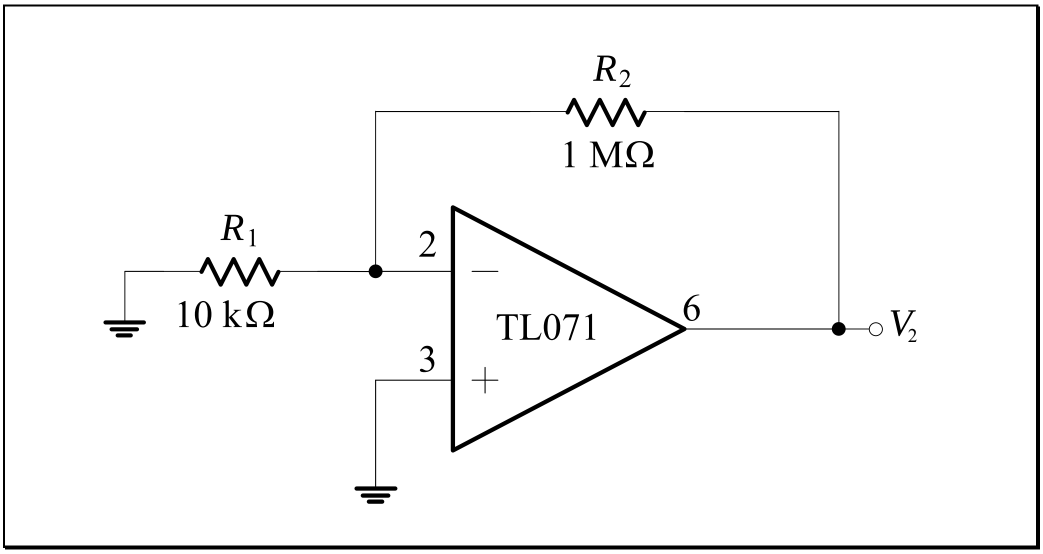


Figure L5.3 – Noninverting Amplifier for Input Offset Voltage Testing

3. Measure the output voltage on the DSO:

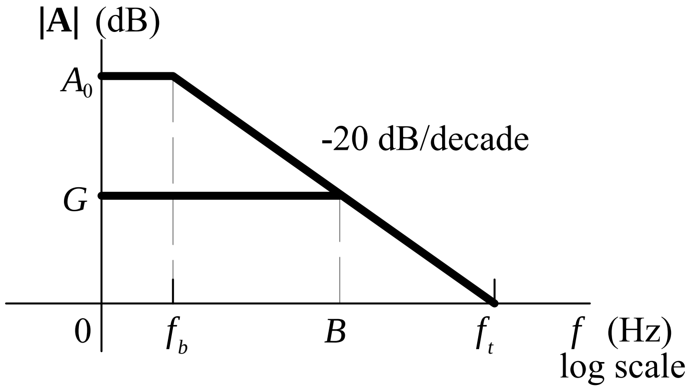
$$V_2 =$$

4. Calculate the input offset voltage:

$$V_{OS} = V_2 / (1 + R_2 / R_1) =$$

Gain-Bandwidth Product

The open-loop gain of an op-amp is finite and decreases with frequency. The gain is quite high at DC and low frequencies, but it starts to fall off at a rather low frequency (10's of Hz). Most op-amps have a capacitor included within the IC whose function is to cause the op-amp to have a single-time-constant (STC) lowpass response shown:



This process of modifying the open-loop gain is termed frequency compensation, and its purpose is to ensure that op-amp circuits will be stable (as opposed to oscillating).

For frequencies $f >> f_b$ (about 10 times and higher), the magnitude of the open-loop gain **A** can be approximated as:

$$|\mathbf{A}| \approx \frac{f_t}{f}$$

The frequency f_t where the op-amp has a gain of 1 (or 0 dB) is known as the *unity-gain bandwidth*. Datasheets of internally compensated op-amps normally call f_t the *gain-bandwidth product*, since:

$$f_t = A_0 f_b$$

The noninverting amplifier configuration exhibits a constant gain-bandwidth product equal to f_t of the op-amp. Thus, you can easily determine the "bandwidth", *B*, of a non-inverting amplifier with a gain, *G*, since the gain-bandwidth product is a constant:

$$GB = f_t = \text{constant}$$

1. Connect the circuit shown in Figure L5.4, with the feedback resistor set to $R_2 = 10\,\text{k}\Omega$ to make a non-inverting amplifier with a nominal gain of 11. The purpose of R_3 and R_4 is to attenuate the function generator voltage so that we can apply a very small sine wave to the input of the circuit; whilst maintaining a reasonably large and noise-free signal at the FG output for triggering purposes.

Noninverting amplifier for gain-bandwidth testing

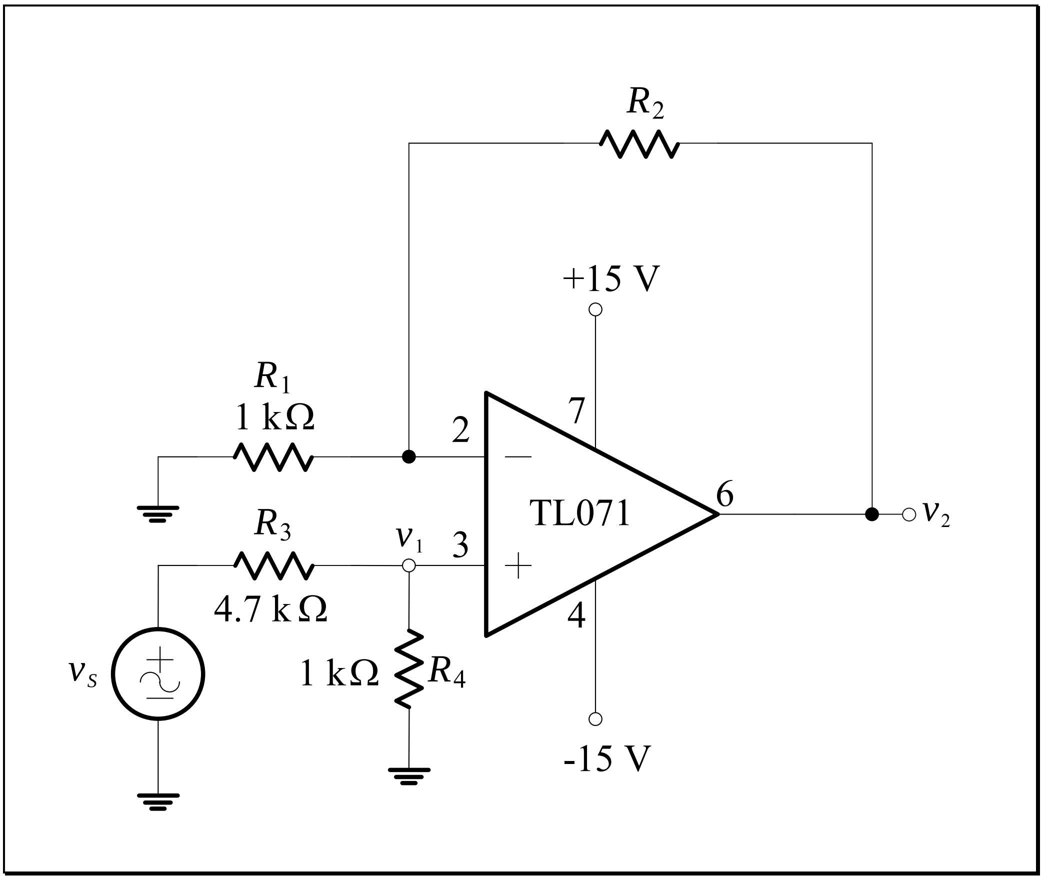


Figure L5.4 – Noninverting Amplifier for Gain-Bandwidth Testing

2. Measure the output of the FG on Channel 1 and set up the DSO trigger for `Noise Rej` to ensure a stable trigger.

3. Use waveform averaging with `# Avgs` set to `64` to achieve relatively noise-free waveforms.

4. To measure the gain-bandwidth product, a very small signal must be used to avoid slew rate limitations. Adjust the function generator for a $100\,\text{mV}_{\text{pp}}$ sine wave at the noninverting terminal (v_1) of the op-amp at a frequency of $1\,\text{kHz}$. Measure the output voltage and record the gain in Table L5.2.

5. Increase the frequency until the output amplitude falls to 70.7% of the output amplitude observed in Step 4. Adjust the function generator as necessary to maintain the input signal at $100\,\text{mV}_{\text{pp}}$. Measure and record the frequency at which this occurs, called the "closed-loop bandwidth", in Table L5.2. Calculate the corresponding gain-bandwidth (*GB*) product.

Step	**Computed Gain** (V/V)	**Measured Gain** *G* (V/V)	**Closed-loop Bandwidth** *B* (Hz)	**Gain-Bandwidth product** *GB* (Hz)
4, 5	11			
6	31			
7	101			

Table L5.2

6. Change the circuit to a noninverting amplifier with a gain of 31. Repeat Steps 4 and 5.

7. Change the circuit to a noninverting amplifier with a gain of 101. Repeat Steps 4 and 5.

Op-Amp Limitations Summary

In the following table, record the parameter values as specified by the TL071 datasheet, as well as the parameter values that your particular op-amp possesses based on your experimental results.

Parameter	**Datasheet (typical value)**	**Experimental**
Output Voltage Swing ($V_S = \pm 15\ \text{V}$, $R_L = 2.2\ \text{k}\Omega$)		
Output Current Limiting		
Slew Rate		
Input Offset Voltage		
Gain-Bandwidth Product		

Table L5.3

Lab Assessment [2 marks]

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

1. Show the result of the peak output voltage of the circuit in Figure L5.1 with a load resistance of $R_L = 2.2\ \text{k}\Omega$ and a supply of ±15 V, and compare with the datasheet. Determine the maximum output current of the op-amp you used before distortion occurred, and compare with the datasheet.

2. Determine the slew rate of the op-amp you used, and compare with the datasheet.

3. Determine the input offset voltage of the op-amp you used, and compare with the datasheet.

4. Determine the gain-bandwidth product of the op-amp you used, and compare with the datasheet.

Marking

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

TL071

Low noise JFET single operational amplifier

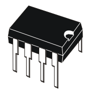
Features

- Wide common-mode (up to V_{CC}^+) and differential voltage range
- Low input bias and offset currenT
- Low noise e_n = 15 nV/ $\sqrt{Hz}$ (typ)
- Output short-circuit protection
- High input impedance JFET input stage
- Low harmonic distortion: 0.01 % (typ)
- Internal frequency compensation
- Latch-up free operation
- High slew rate: 16 V /µs (typ)

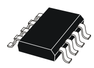
Description

The TL071 is a high-speed JFET input single operational amplifier. This JFET input operational amplifier incorporates well matched, high-voltage JFET and bipolar transistors in a monolithic integrated circuit.

The device features high slew rates, low input bias and offset currents, and low offset voltage temperature coefficient.

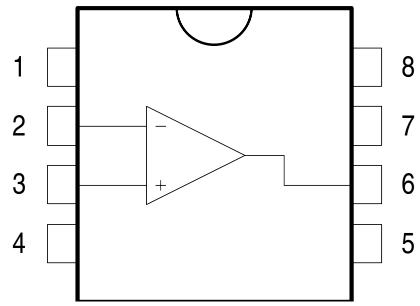


N
DIP8
(Plastic package)



D
SO-8
(Plastic micropackage)

Pin connections
(Top view)



1 - Offset null 1
2 - Inverting input
3 - Non-inverting input
4 - V_{CC}^-
5 - Offset null 2
6 - Output
7 - V_{CC}^+
8 - N.C.

1 Schematic diagram

Figure 1. Circuit schematics

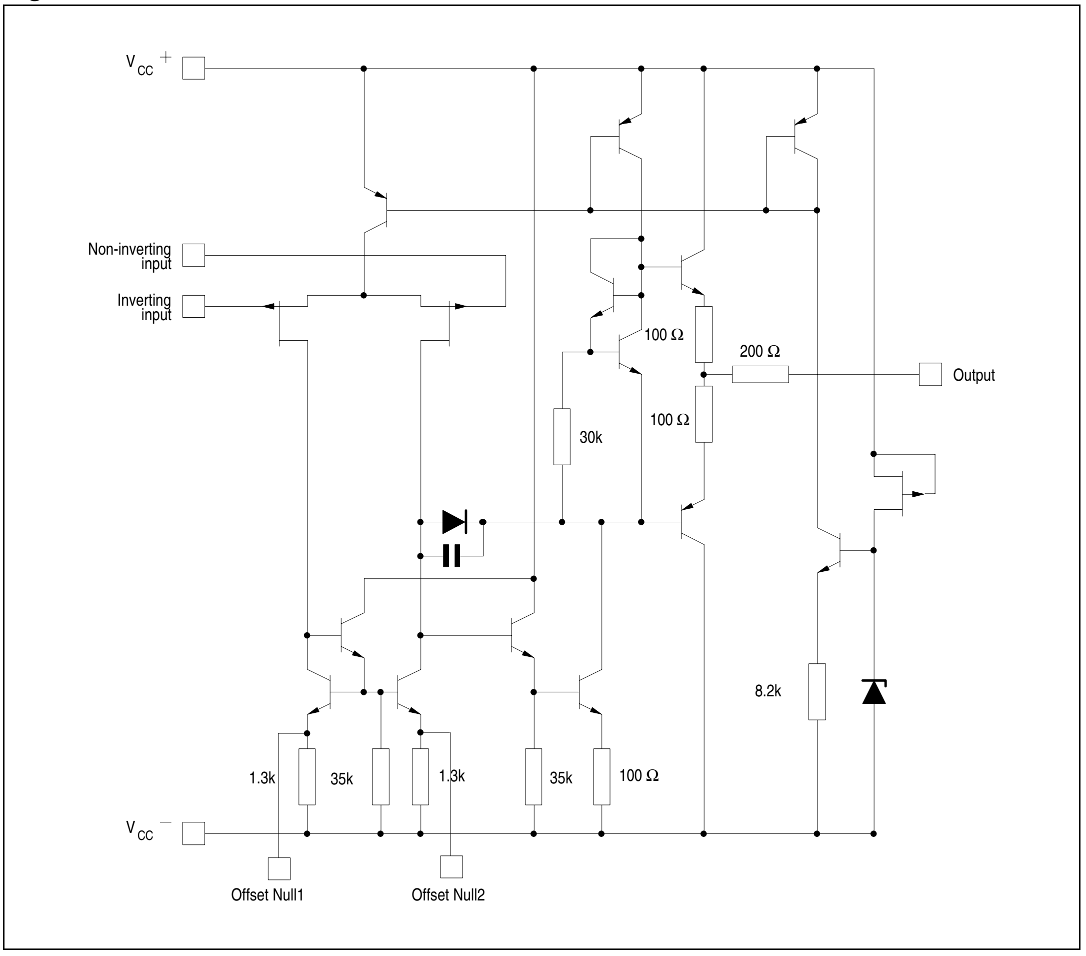
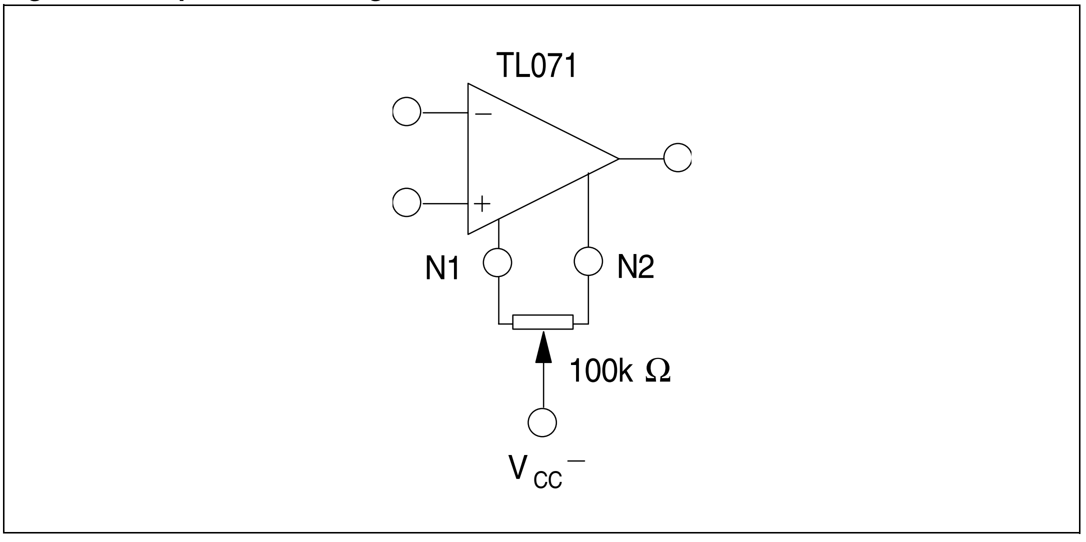


Figure 2. Input offset voltage null circuit



3 Electrical characteristics

Table 3. $V_{CC} = \pm15V$, $T_{amb} = +25°C$ (unless otherwise specified)

Symbol	Parameter	TL071I,M,AC,AI,AM, BC,BI,BM			TL071C			Unit
		Min.	Typ.	Max.	Min.	Typ.	Max.	
V_{io}	Input offset voltage ($R_s = 50\Omega$) $T_{amb} = +25°C$ TL071 TL071A TL071B $T_{min} \le T_{amb} \le T_{max}$ TL071 TL071A TL071B		 3 3 1	 10 6 3 13 7 5		 3	 10 13	mV
DV_{io}	Input offset voltage drift		10			10		µV/°C
I_{io}	Input offset current $T_{amb} = +25°C$ $T_{min} \le T_{amb} \le T_{max}$		 5	 100 4		 5	 100 10	 pA nA
I_{ib}	Input bias current (1) $T_{amb} = +25°C$ $T_{min} \le T_{amb} \le T_{max}$		 20	 200 20		 20	 200 20	 pA nA
A_{vd}	Large signal voltage gain ($R_L = 2k\Omega$, $V_o = \pm10V$) $T_{amb} = +25°C$ $T_{min} \le T_{amb} \le T_{max}$	 50 25	 200		 25 15	 200		V/mV
SVR	Supply voltage rejection ratio ($R_S = 50\Omega$) $T_{amb} = +25°C$ $T_{min} \le T_{amb} \le T_{max}$	 80 80	 86		 70 70	 86		dB
I_{CC}	Supply current, no load $T_{amb} = +25°C$ $T_{min} \le T_{amb} \le T_{max}$		 1.4	 2.5 2.5		 1.4	 2.5 2.5	mA
V_{icm}	Input common mode voltage range	±11	+15 -12		±11	+15 -12		V
CMR	Common mode rejection ratio ($R_S = 50\Omega$) $T_{amb} = +25°C$ $T_{min} \le T_{amb} \le T_{max}$	 80 80	 86		 70 70	 86		dB
I_{os}	Output short-circuit current $T_{amb} = +25°C$ $T_{min} \le T_{amb} \le T_{max}$	 10 10	 40	 60 60	 10 10	 40	 60 60	mA
$\pm V_{opp}$	Output voltage swing $T_{amb} = +25°C$ $R_L = 2k\Omega$ $R_L = 10k\Omega$ $T_{min} \le T_{amb} \le T_{max}$ $R_L = 2k\Omega$ $R_L = 10k\Omega$	 10 12 10 12	 12 13.5		 10 12 10 12	 12 13.5		V
SR	Slew rate $V_{in} = 10V$, $R_L = 2k\Omega$, $C_L = 100pF$, unity gain	8	16		8	16		V/µs

Table 3. V_{CC} = ±15V, T_{amb} = +25°C (unless otherwise specified) (continued)

Symbol	Parameter	TL071I,M,AC,AI,AM, BC,BI,BM			TL071C			Unit
		Min.	Typ.	Max.	Min.	Typ.	Max.	
t_r	Rise time V_{in} = 20mV, R_L = 2kΩ, C_L = 100pF, unity gain		0.1			0.1		µs
K_{ov}	Overshoot V_{in} = 20mV, R_L = 2kΩ, C_L = 100pF, unity gain		10			10		%
GBP	Gain bandwidth product V_{in} = 10mV, R_L = 2kΩ, C_L = 100pF, f= 100kHz	2.5	4		2.5	4		MHz
R_i	Input resistance		10^{12}			10^{12}		W
THD	Total harmonic distortion, f= 1kHz, R_L= 2kΩ,C_L = 100pF, A_v = 20dB, V_o = $2V_{pp}$)		0.01			0.01		%
e_n	Equivalent input noise voltage R_S = 100Ω, f = 1KHz		15			15		$\frac{nV}{\sqrt{Hz}}$
∅m	Phase margin		45			45		degrees

1. The input bias currents are junction leakage currents which approximately double for every 10°C increase in the junction temperature.

Lab 6 – First-Order *RC* Circuits

First-order RC *circuits. Step response. Frequency response.*

Introduction

A first-order network, also known as a single-time-constant (STC) network, is one that is composed of, or can be *reduced* to, one reactive component (capacitance or inductance) and one resistance. Some examples are shown below:

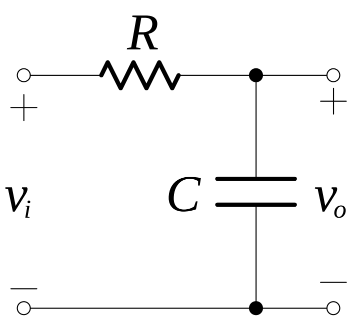
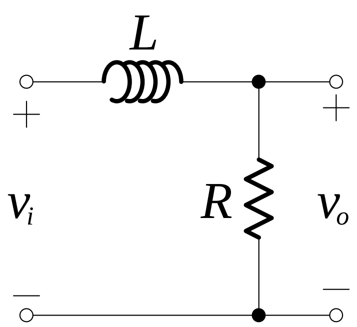
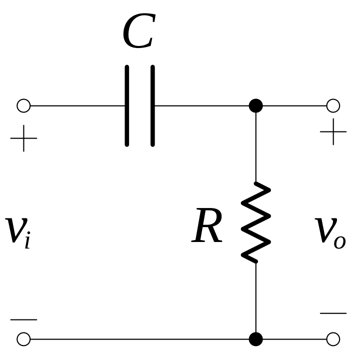
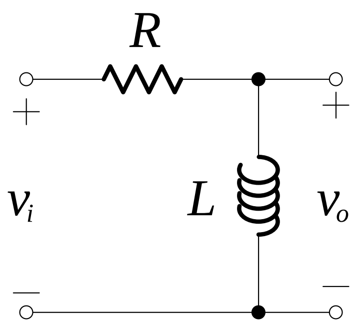
	RC Circuit	*RL* Circuit
Lowpass	 (a)	 (b)
Highpass	 (c)	 (d)

Table L6.1

Most first-order circuits can be classified into two categories, lowpass (LP) and highpass (HP), with each of the two categories displaying distinctly different signal responses (there is a third category called *allpass*).

As an example, the first-order circuit shown in Table L6.1 (a) is of the *lowpass* type and that in Table L6.1 (b) is of the *highpass* type. To see the reasoning behind this classification, observe that the frequency response of each of these two circuits can be expressed as a voltage-divider ratio, with the divider composed of a resistor and a capacitor. Now, recalling how the impedance of a capacitor varies with frequency ($\mathbf{Z} = 1/j\omega C$) it is easy to see that the voltage output of the circuit in Table L6.1 (a) will decrease with frequency and approach zero as ω approaches ∞. Thus the circuit of Table L6.1 (a) acts as a **lowpass filter**; it passes low-frequency sine-wave inputs with little or no attenuation and attenuates high-frequency input sinusoids. The circuit of Table L6.1 (b) does the opposite; the voltage output is unity at $\omega = \infty$ and decreases as ω is reduced, approaching 0 for $\omega = 0$. The latter circuit, therefore, performs as a **highpass filter**.

RC circuits such as those in Table L6.1 (a) and (c) are commonly used in electronics to provide timing functions. **In these applications the circuit's step response is of interest.**

RC networks are also used as simple filters . The circuit in Table L6.1 (a) is a lowpass filter which may be used to extract an audio signal (20 Hz to 20 kHz band) from a higher frequency carrier signal (1 MHz band used in AM broadcasting). The circuit of Table L6.1 (c) is a highpass filter. One application of such a filter is on the inputs to an oscilloscope (the filter is in place when the input is AC coupled). **When used as a filter, a circuit's frequency response provides the necessary characterization for the circuit.**

RL circuits are less commonly used in electronics because the inductors are more bulky than capacitors and more expensive since they are wound coils. *RL* circuits do have applications in power circuits such as power line filters and switch-mode power supplies but are most frequently found in electro-mechanical applications such as relays and electric motors. As with *RC* circuits, both the step and frequency responses are required depending on the application.

Objectives

1. To investigate the step response of a lowpass *RC* first-order circuit.

2. To investigate the frequency response of a lowpass *RC* first-order circuit.

Equipment

- 1 Digital Storage Oscilloscope (DSO) – Agilent 54621A
- 1 Mini-Lab – BWD 604
- Resistors – 1 x 390 Ω, 1 x 1.8 kΩ
  Capacitor – 1 x 10 nF
  Breadboard, Hook-up wire, 2 x 4mm leads.

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

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

L6.4

Pre-Lab Work

Lowpass *RC* First-Order Circuit

Step Response

Lowpass *RC* first-order circuit

1. The circuit shown below is an a "zero-state", $v_o(0^-) = 0\text{ V}$:

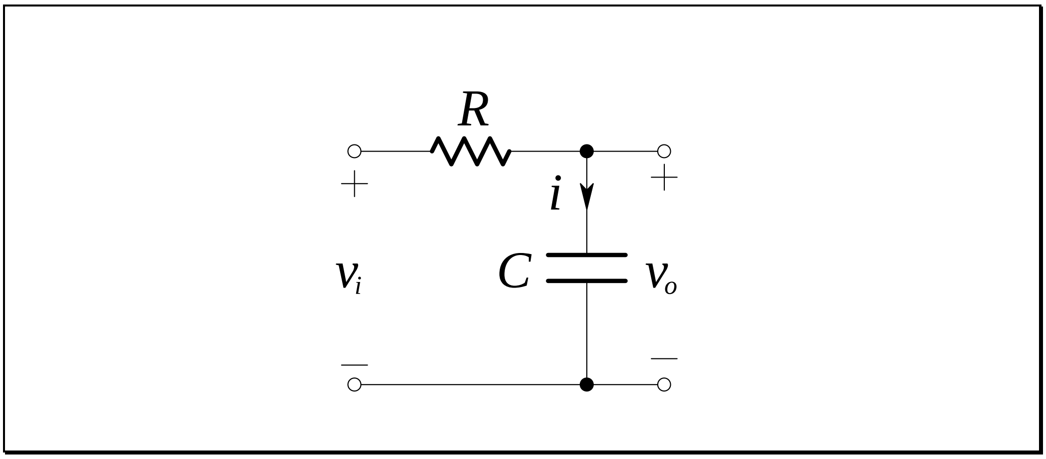


Figure L6.1

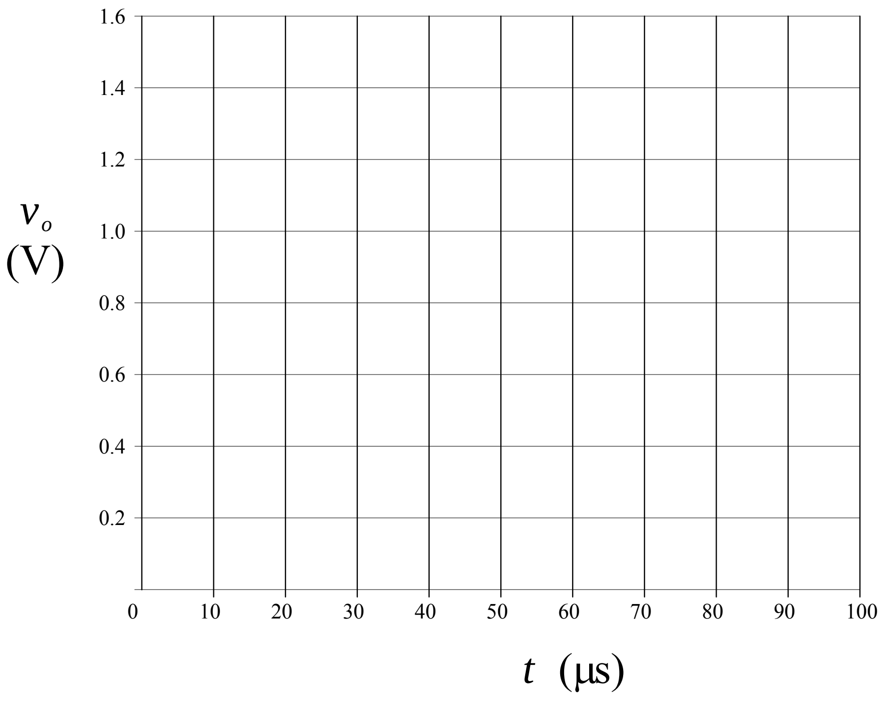
Derive an expression for the unit-step response of the circuit, $v_0(t)$.

2. What is the *time constant* for this circuit in terms of *R* and *C*?

$T =$

3. From the expression for the unit-step response, derive an expression for the current through the capacitor, $i(t)$.

4. Let $C = 10\,\text{nF}$ and $R = 390\,\Omega$ then $R = 1.8\,\text{k}\Omega$. Sketch, for both values of resistance, the unit-step response over the period $0 < t < 100\,\mu\text{s}$.



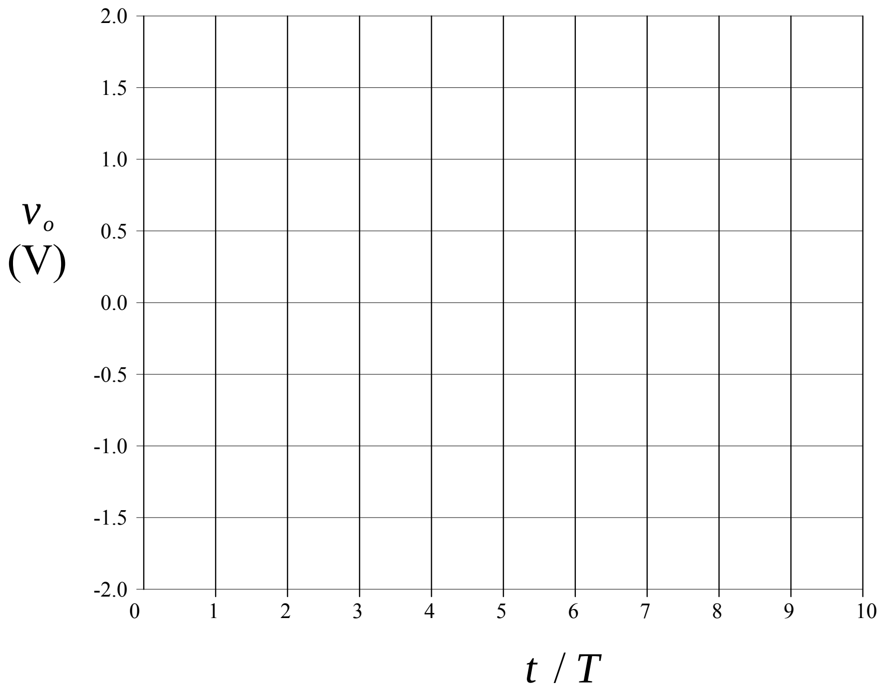
L6.6

5. The step response derived in 4 assumes zero initial conditions. If the input voltage is a square wave (-1 V to 1 V), the response to each half cycle of the input will be influenced by the response to the previous half cycle. The initial conditions will **NOT** be zero. If the period of the input is $T_0 = 10T$, derive an expression for the response in the first half-cycle under steady-state conditions.

   Hint: What will the initial condition be for each positive half cycle?

6. How do the above initial conditions affect the response of the circuit?

7. Sketch the circuit's response to a 2 Vp-p square wave, of period $10T$.



Frequency Response

1. For the circuit:

Lowpass *RC* first-order circuit

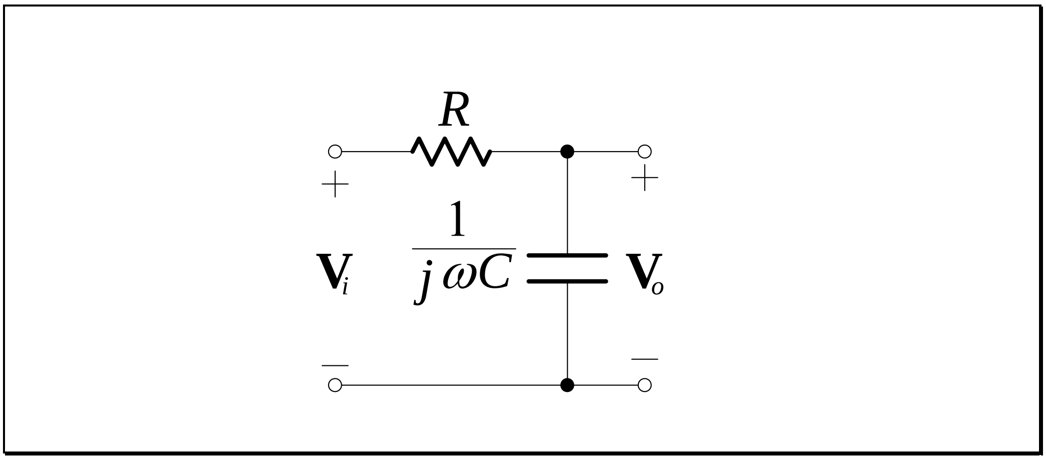


Figure L6.2

derive the frequency response:

$$\mathbf{H}(j\omega) = \frac{\mathbf{V}_o}{\mathbf{V}_i}, \tag{L6.1}$$

2. By comparing the frequency response of the circuit in Figure L6.2 with the standard form of a first-order lowpass frequency response:

$$\mathbf{H}(j\omega)=\frac{K}{1+(j\omega/\omega_0)}, \qquad \text{(L6.2)}$$

write expressions for K and ω_0 in terms of R and C.

$K =$
$\omega_0 =$

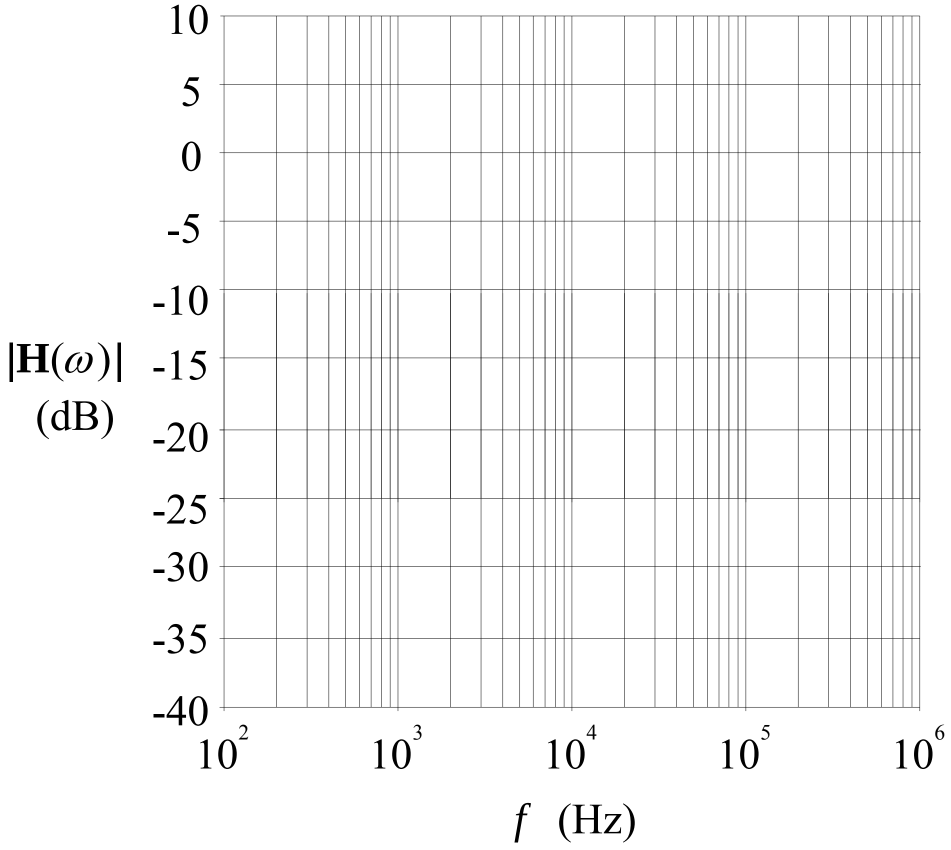
3. Let $C=10\,\text{nF}$ and $R=390\,\Omega$ then $R=1.8\,\text{k}\Omega$, and complete the following table:

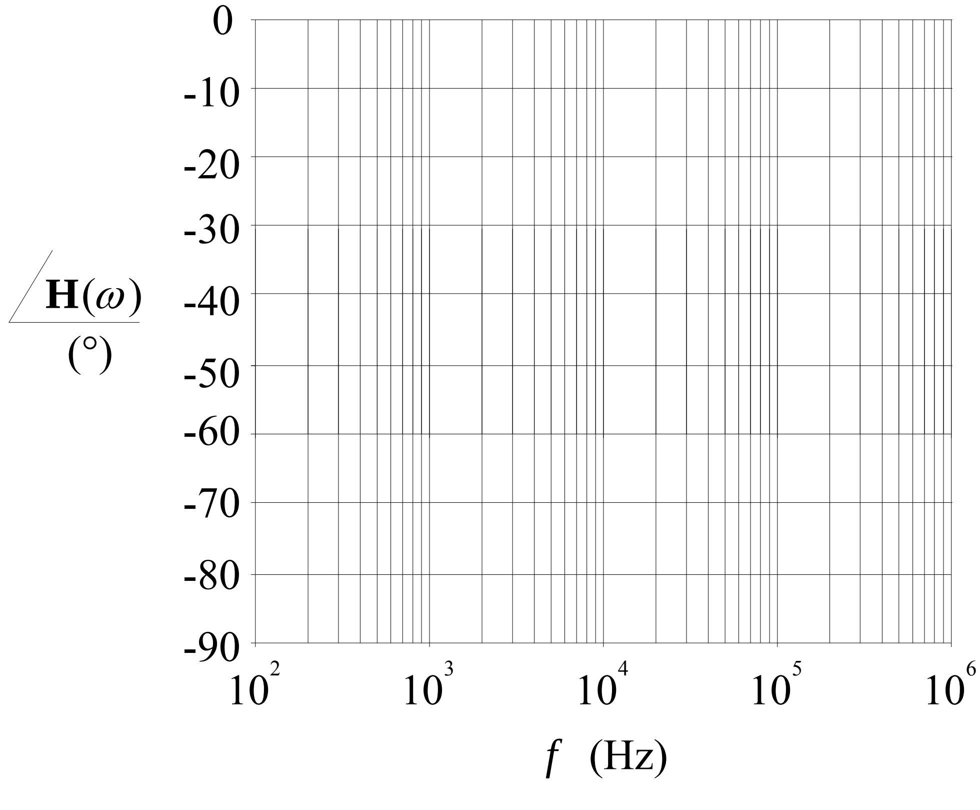
Frequency f (kHz)	Frequency ω (krads^{-1})	Gain $\frac{\lvert\mathbf{V}_o\rvert}{\lvert\mathbf{V}_i\rvert}$ (V/V)		Gain $20\log_{10}\frac{\lvert\mathbf{V}_o\rvert}{\lvert\mathbf{V}_i\rvert}$ (dB)		Phase $\angle\frac{\mathbf{V}_o}{\mathbf{V}_i}$ (°)	
		$R=390\,\Omega$	$R=1.8\,\text{k}\Omega$	$R=390\,\Omega$	$R=1.8\,\text{k}\Omega$	$R=390\,\Omega$	$R=1.8\,\text{k}\Omega$
0.1	0.6283	1.000	1.000	0.0000	-0.0006	-0.1	-0.6
0.2	1.257	1.000	0.9997	-0.0001	-0.0022	-0.3	-1.3
0.5	3.142	1.000	0.9984	-0.0007	-0.0139	-0.7	-3.2
1	6.283	0.9997	0.9937	-0.0026	-0.0552	-1.4	-6.5
2	12.57	0.9988	0.9754	-0.0104	-0.2167	-2.8	-12.7
5	31.42						
10	62.83	0.9713	0.6624	-0.2533	-3.578	-13.8	-48.5
20	125.7	0.8980	0.4043	-0.9349	-7.865	-26.1	-66.1
50	314.2						
100	628.3	0.3778	0.08808	-8.454	-21.10	-67.8	-84.9
200	1257	0.1999	0.04417	-13.98	-27.10	-78.5	-87.5
500	3142	0.08135	0.01768	-21.79	-35.05	-85.3	-89.0
1000	6283	0.04078	0.008842	-27.79	-41.07	-87.7	-89.5

Table L6.2

L6.10

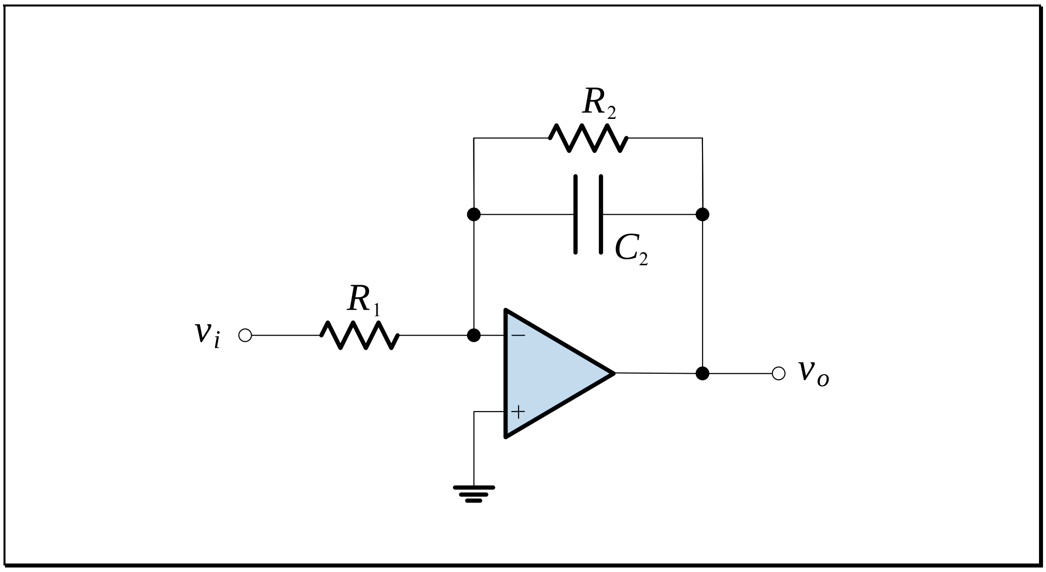
4. Plot the gain and phase values from Table L6.2 on the Bode plot below:





Lowpass Op-Amp Filter

1. For the circuit:



Lowpass op-amp filter

Figure L6.3

derive the frequency response:

$$\mathbf{H}(j\omega) = \frac{\mathbf{V}_o}{\mathbf{V}_i}, \tag{L6.3}$$

2. Design a first-order lowpass op-amp filter that has a DC gain magnitude of 5 V/V and a cutoff frequency of 3.183 kHz. Make sure your design uses components with practical sizes.

3. Verify your design by conducting a simulation of the frequency response using PSpice (e.g. OrCAD Demo).

   Use an LF411 for the op-amp in the simulation (in OrCAD Demo, do **Place ▶ Part…** then **Part Search…** to find it).

4. Print a hardcopy of the frequency response to bring to the lab.

   Use the following axes for graphing:

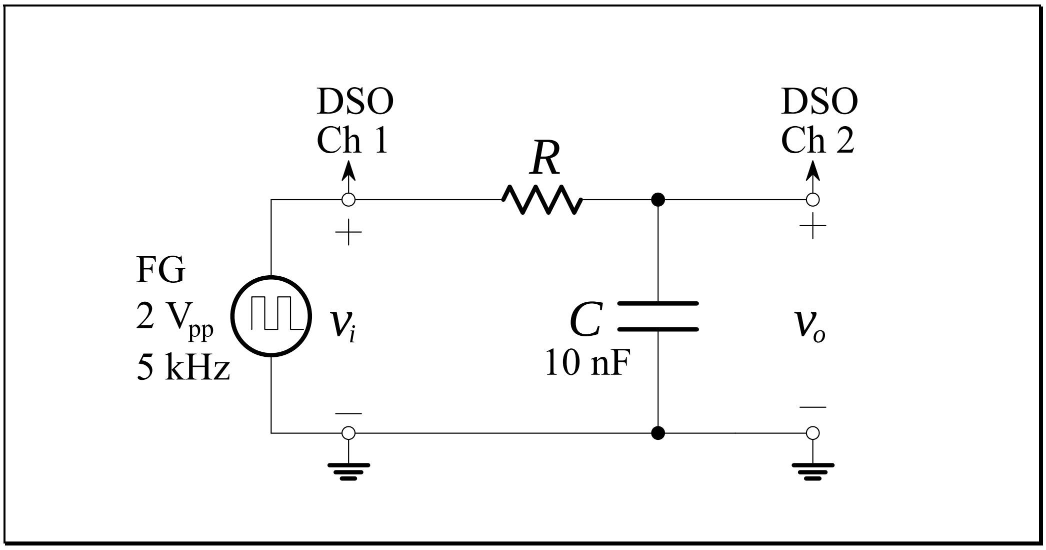
Response	**X**	**Y**
Magnitude	10 Hz to 100 kHz	-20 dB to 20 dB
Phase	10 Hz to 100 kHz	90° to 180°

Lab Work

Lowpass *RC* First-Order Circuit

Step Response

1. Construct the following circuit, using $R = 390\,\Omega$:



Lowpass *RC* first-order circuit

Figure L6.4

The input is a 2 V peak-to-peak **square wave** (i.e. -1 V to +1 V) with a frequency of 5 kHz.

Measurement of the step response of a circuit is conducted with a square wave!

2. Connect the input to Channel 1 of the DSO and the output to Channel 2 of the DSO.

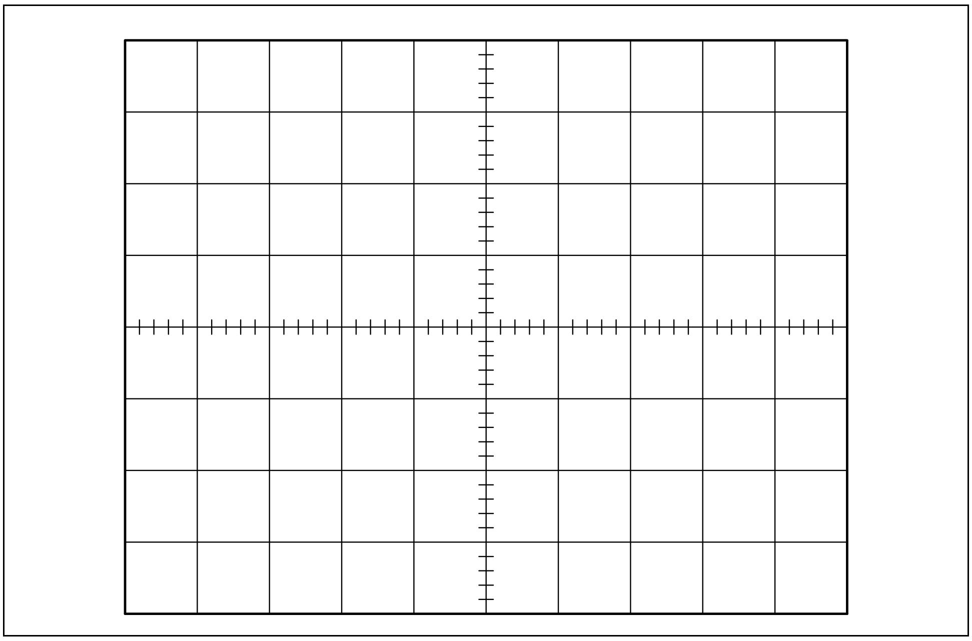
3. Press `Main/Delayed`. Change the `Time Ref` softkey to Left. This will facilitate sketching the step-response.

4. Set the DSO horizontal time base to 10 μs / div.

5. Set Channel 1 and Channel 2 to 500 mV/div.

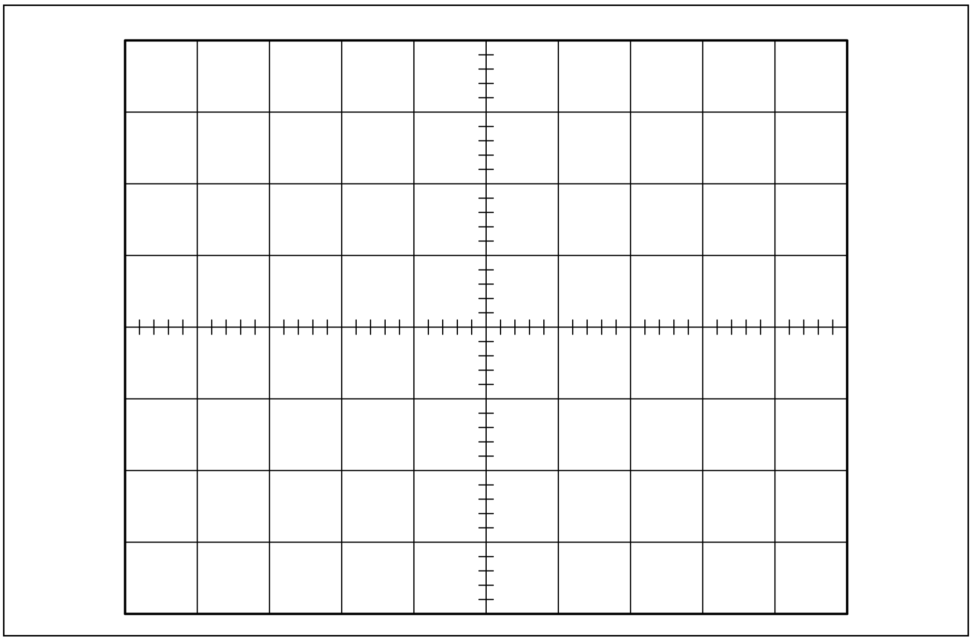
6. Ensure that "bandwidth limiting" is used for both Channels 1 and 2.

7. Draw the input and output (Ch 1 and Ch 2) waveforms, ensuring that the sketch is labelled with voltage and time scales.



8. On the 100 kHz range, sweep the frequency slowly from 5 kHz to 25 kHz, and observe how the output changes.

9. Draw the input and output waveforms at 25 kHz, ensuring that the sketch is labelled with voltage and time scales.

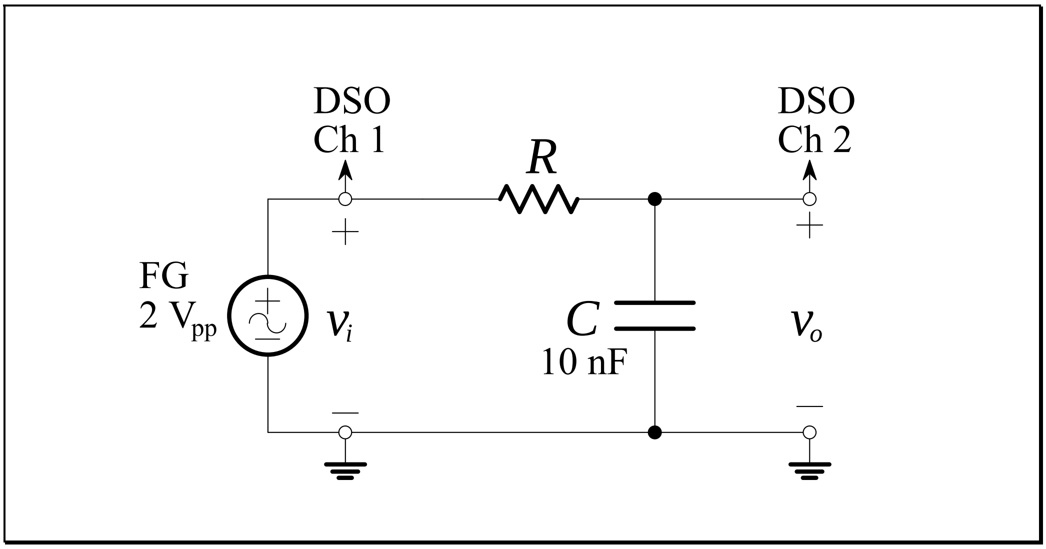


10. Change the resistor to $R = 1.8\ \mathrm{k\Omega}$, and reduce the frequency to 5 kHz.

11. Repeat steps 7 to 9.

Frequency Response

1. Construct the following circuit, using $R = 390\,\Omega$:



Lowpass *RC* first-order circuit

Figure L6.5

The input is a 2 V peak-to-peak **sine wave** (i.e. -1 V to +1 V) with a variable frequency.

Measurement of the frequency response of a circuit is conducted with a sine wave!

2. Connect the input to Channel 1 of the DSO and the output to Channel 2 of the DSO.

3. In measuring the frequency response, make sure to measure the Channel 1 amplitude, as it will decrease with increasing frequency due to the "bandwidth" of the function generator.

   In taking a frequency response of a circuit, the fastest measuring technique is to set the frequency vernier to a desired frequency, such as 100 Hz, then simply change the FG frequency range to get the 1 kHz reading, then the 10 kHz reading, etc.

4. Complete the following table:

Desired Frequency f_{desired} (kHz)	Actual Frequency f_{actual} (kHz)	Gain $\frac{\lvert \mathbf{V}_o \rvert}{\lvert \mathbf{V}_i \rvert}$ (V/V)		Gain $20\log_{10}\frac{\lvert \mathbf{V}_o \rvert}{\lvert \mathbf{V}_i \rvert}$ (dB)		Phase $\angle\frac{\mathbf{V}_o}{\mathbf{V}_i}$ (°)	
		$R = 390\,\Omega$	$R = 1.8\,\text{k}\Omega$	$R = 390\,\Omega$	$R = 1.8\,\text{k}\Omega$	$R = 390\,\Omega$	$R = 1.8\,\text{k}\Omega$
0.1							
0.2							
0.5							
1							
2							
5							
10							
20							
50							
100							
200							
500							
1000							

Table L6.3

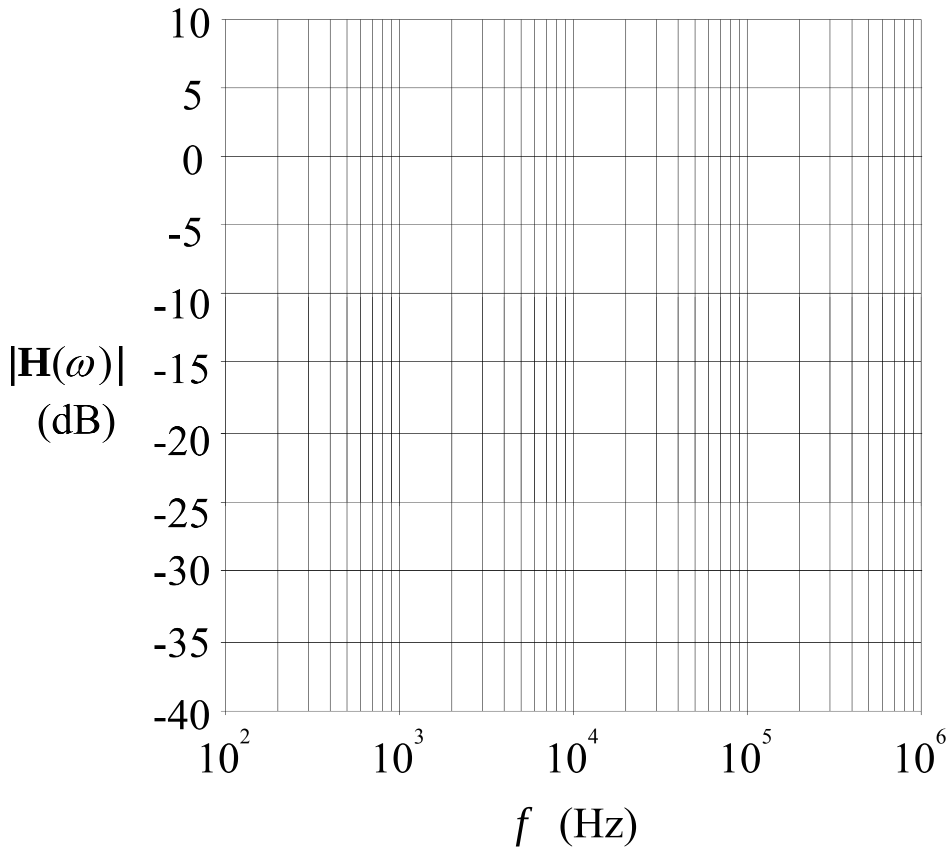
5. Find the cutoff frequency (also named the -3 dB frequency, or corner frequency, or break frequency), f_0, and record the magnitude of the gain and phase shift in the table below.

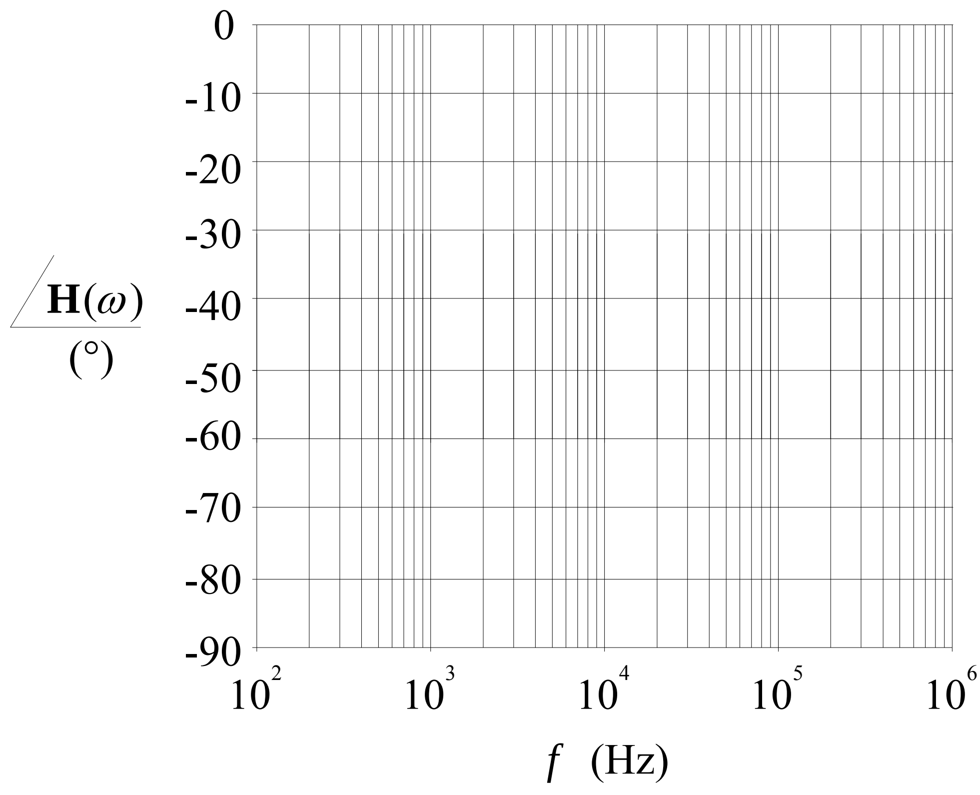
	Cutoff Frequency f_0 (kHz)	Gain $\frac{\lvert \mathbf{V}_o \rvert}{\lvert \mathbf{V}_i \rvert}$ (V/V)	Gain $20\log_{10}\frac{\lvert \mathbf{V}_o \rvert}{\lvert \mathbf{V}_i \rvert}$ (dB)	Phase $\angle\frac{\mathbf{V}_o}{\mathbf{V}_i}$ (°)
$R = 390\,\Omega$				
$R = 1.8\,\text{k}\Omega$				

Table L6.4

6. Repeat steps 4 and 5 with $R = 1.8\,\text{k}\Omega$.

7. Plot the gain and phase values from Table L6.3 on the Bode plot below:





L6.18

Questions – Lowpass Frequency Response

1. What is the relationship between the cutoff frequency and the time constant?

Answer:

2. At what rate does the response “fall off” at high frequencies? (Draw an asymptote on your graph and measure its slope).

Answer:

3. How do you experimentally determine the cutoff frequency?

Answer:

Lowpass Op-Amp Filter

1. In the pre-lab work, you designed a first-order lowpass op-amp filter with a DC gain magnitude of 5 and a cutoff frequency of 3.183 kHz. Build it.

2. Complete the following table:

Desired Frequency f_{desired} (kHz)	Actual Frequency f_{actual} (kHz)	Gain $\frac{\lvert\mathbf{V}_o\rvert}{\lvert\mathbf{V}_i\rvert}$ (V/V)	Gain $20\log_{10}\frac{\lvert\mathbf{V}_o\rvert}{\lvert\mathbf{V}_i\rvert}$ (dB)	Phase $\angle\frac{\mathbf{V}_o}{\mathbf{V}_i}$ (°)
0.1				
0.2				
0.5				
1				
2				
5				
10				
20				
50				
100				

Table L6.5

3. Find the cutoff frequency (also named the -3 dB frequency, or corner frequency, or break frequency), f_0, and record the magnitude of the gain and phase shift in the table below.

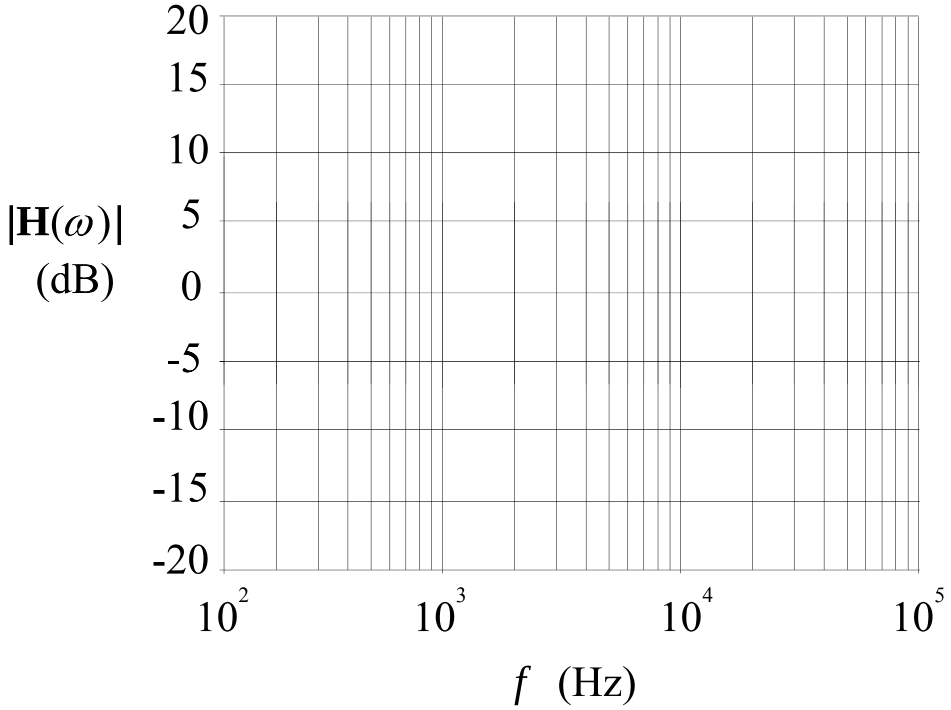
Cutoff Frequency f_0 (kHz)	Gain $\frac{\lvert\mathbf{V}_o\rvert}{\lvert\mathbf{V}_i\rvert}$ (V/V)	Gain $20\log_{10}\frac{\lvert\mathbf{V}_o\rvert}{\lvert\mathbf{V}_i\rvert}$ (dB)	Phase $\angle\frac{\mathbf{V}_o}{\mathbf{V}_i}$ (°)

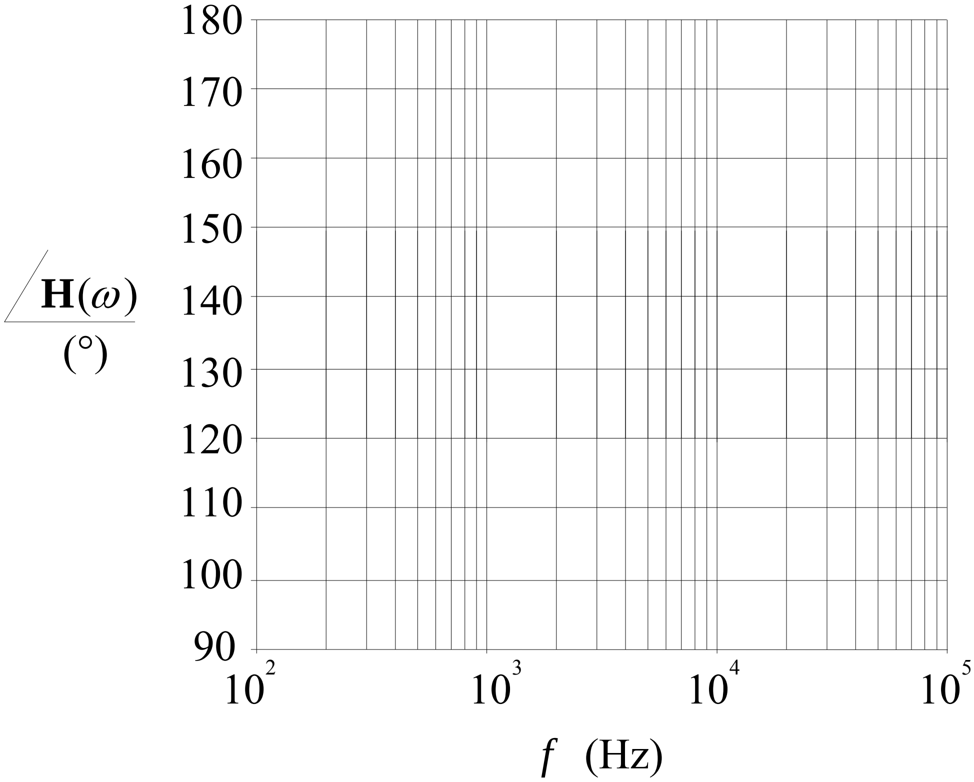
Table L6.6

4. Connect a load resistor $R_L = 1\,\text{k}\Omega$ to the output of your circuit. Does the frequency response change?

Answer:

5. Plot the gain and phase values from Table L6.5 on the Bode plot below:





Lab Assessment [2 marks]

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

1. Show the result of the pre-lab work, **Step Response** section, step 3.

2. Show the result of the pre-lab work, **Lowpass Op-Amp Filter** section, step 4.

3. Demonstrate the measurement of the time constant of a lowpass STC *RC* circuit using the step response (use $R = 1.8\,\mathrm{k\Omega}$, $C = 10\,\mathrm{nF}$).

4. Demonstrate the measurement of the cutoff frequency of a first-order lowpass op-amp filter using the frequency response.

Marking

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

Lab 7 – First-Order *RL* Circuits

First-order RL *circuits. Step response. Frequency response. Real inductors.*

Introduction

A first-order network, also known as a single-time-constant (STC) network, is one that is composed of, or can be *reduced* to, one reactive component (capacitance or inductance) and one resistance. Some examples are shown below:

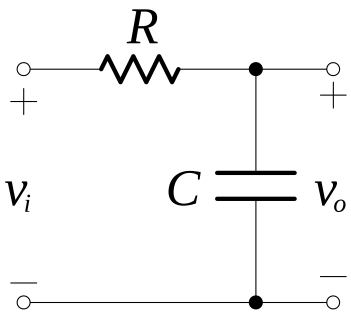
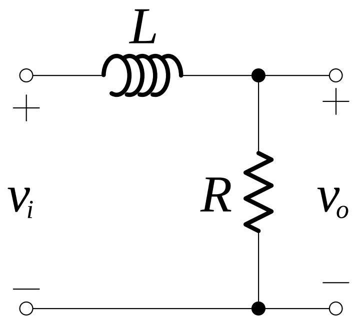
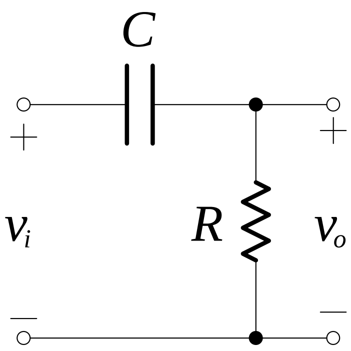
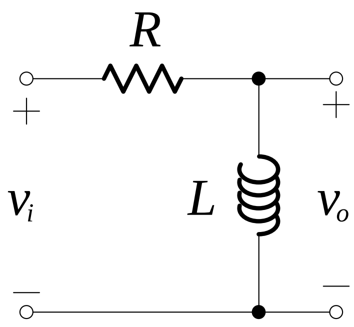
	RC Circuit	*RL* Circuit
Lowpass	 (a)	 (b)
Highpass	 (c)	 (d)

Table L7.1

Most first-order circuits can be classified into two categories, lowpass (LP) and highpass (HP), with each of the two categories displaying distinctly different signal responses (there is a third category called *allpass*).

As an example, the first-order circuit shown in Table L7.1 (a) is of the *lowpass* type and that in Table L7.1 (b) is of the *highpass* type. To see the reasoning behind this classification, observe that the frequency response of each of these two circuits can be expressed as a voltage-divider ratio, with the divider composed of a resistor and a capacitor. Now, recalling how the impedance of a capacitor varies with frequency ($\mathbf{Z} = 1/j\omega C$) it is easy to see that the voltage output of the circuit in Table L7.1 (a) will decrease with frequency and approach zero as ω approaches ∞. Thus the circuit of Table L7.1 (a) acts as a **lowpass filter**; it passes low-frequency sine-wave inputs with little or no attenuation and attenuates high-frequency input sinusoids. The circuit of Table L7.1 (b) does the opposite; the voltage output is unity at $\omega = \infty$ and decreases as ω is reduced, approaching 0 for $\omega = 0$. The latter circuit, therefore, performs as a **highpass filter**.

RC circuits such as those in Table L7.1 (a) and (c) are commonly used in electronics to provide timing functions. **In these applications the circuit's step response is of interest.**

RC networks are also used as simple filters . The circuit in Table L7.1 (a) is a lowpass filter which may be used to extract an audio signal (20 Hz to 20kHz band) from a higher frequency carrier signal (1 MHz band used in AM broadcasting). The circuit of Table L7.1 (c) is a highpass filter. One application of such a filter is on the inputs to an oscilloscope (the filter is in place when the input is AC coupled). **When used as a filter, a circuit's frequency response provides the necessary characterization for the circuit.**

RL circuits are less commonly used in electronics because the inductors are more bulky than capacitors and more expensive since they are wound coils. *RL* circuits do have applications in power circuits such as power line filters and switch-mode power supplies but are most frequently found in electro-mechanical applications such as relays and electric motors. As with *RC* circuits, both the step and frequency responses are required depending on the application.

Objectives

1. To investigate the step response of a highpass *RL* first-order circuit.

2. To investigate the frequency response of a highpass *RL* first-order circuit.

Equipment

- 1 Digital Storage Oscilloscope (DSO) – Agilent 54621A
- 1 Mini-Lab – BWD 604
- Resistors – 1 x 130 Ω, 3 x 100 Ω
  Inductor – 1 x 680 μH
  Breadboard, Hook-up wire, 2 x 4mm leads.

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).

L7.4

Pre-Lab Work

Highpass *RL* First-Order Circuit – Ideal Inductor

Step Response

1. The circuit shown below is an a “zero-state”, $i\left(0^-\right) = 0\text{ A}$:

Highpass *RL* first-order circuit

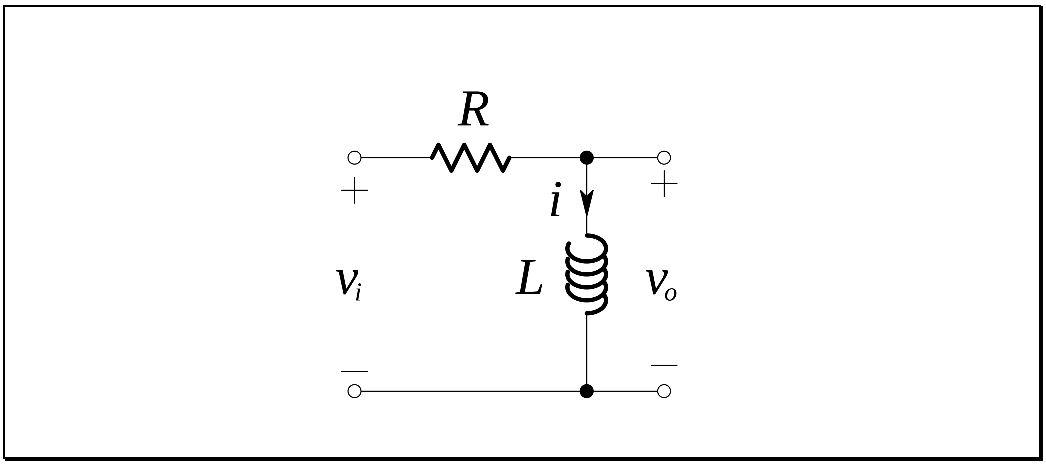


Figure L7.1

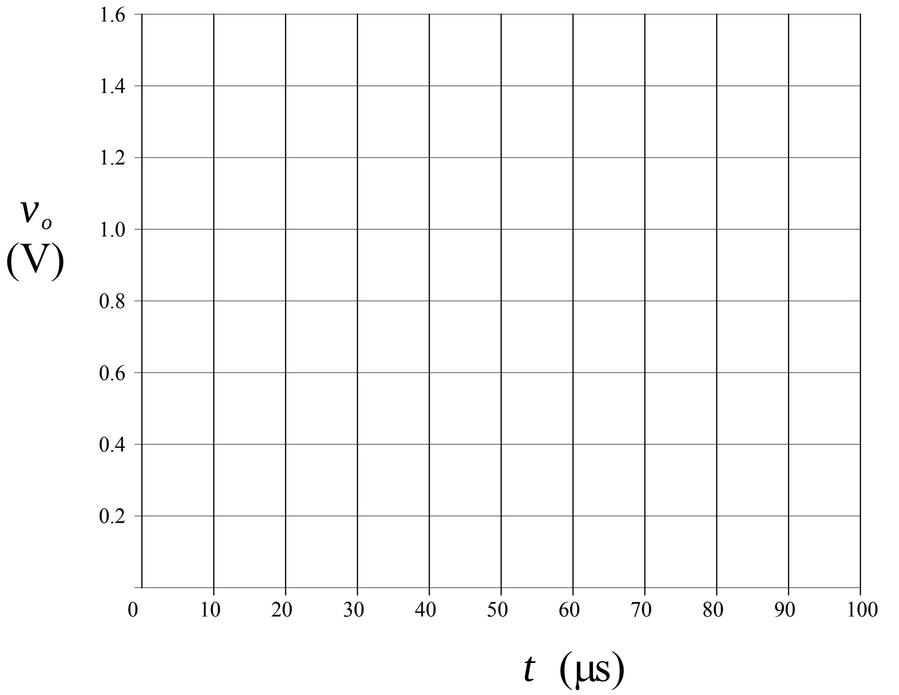
Derive an expression for the unit-step response of the circuit, $v_0(t)$.

2. What is the *time constant* for this circuit in terms of R and L?

$T =$

3. From the expression for the unit-step response, derive an expression for the current through the inductor, $i(t)$.

4. Let $L = 680\,\mu\text{H}$ and $R = 130\,\Omega$ then $R = 33\,\Omega$. Sketch, for both values of resistance, the unit-step response over the period $0 < t < 100\,\mu\text{s}$.

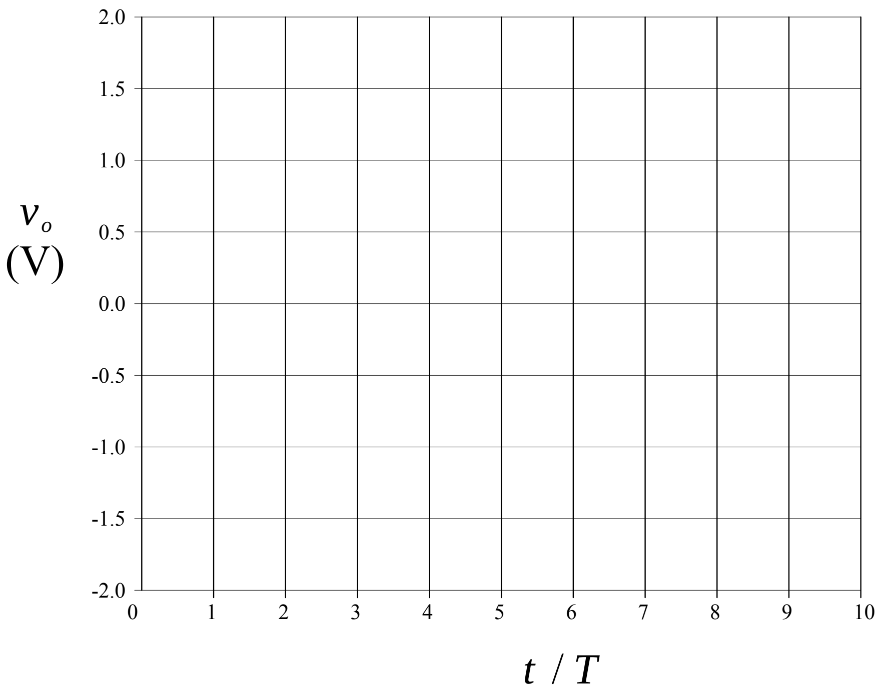


5. The step response derived in 4 assumes zero initial conditions. If the input voltage is a square wave (-1 V to 1 V), the response to each half cycle of the input will be influenced by the response to the previous half cycle. The initial conditions will **NOT** be zero. If the period of the input is $T_0 = 10T$, derive an expression for the response in the first half-cycle under steady-state conditions.

   Hint: What will the initial condition be for each positive half cycle?

6. How do the above initial conditions affect the response of the circuit?

7. Sketch the circuit's response to a 2 Vp-p square wave, of period $10T$.



Frequency Response

Highpass RL first-order circuit

1. For the circuit:

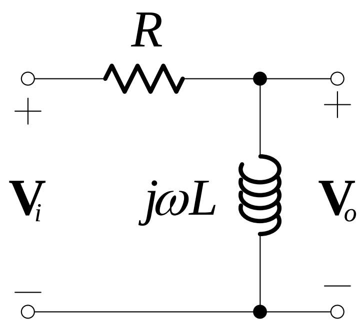


Figure L7.2

derive the frequency response:

$$\mathbf{H}(j\omega) = \frac{\mathbf{V}_o}{\mathbf{V}_i}, \tag{L7.1}$$

2. By comparing the frequency response of the circuit in Figure L7.2 with the standard form of a first-order highpass frequency response:

$$\mathbf{H}(j\omega) = K\frac{(j\omega/\omega_0)}{1+(j\omega/\omega_0)}, \tag{L7.2}$$

write expressions for K and ω_0 in terms of R and L.

$K =$

$\omega_0 =$

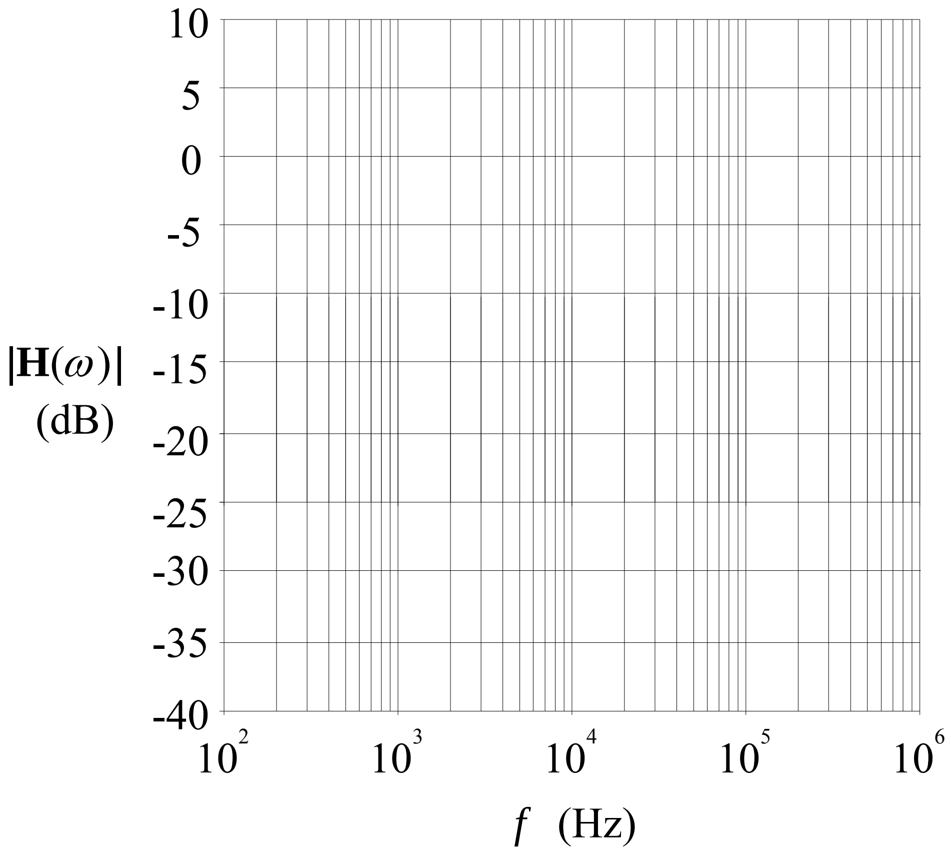
3. Let $L = 680\,\mu\text{H}$ and $R = 130\,\Omega$ then $R = 33\,\Omega$, and complete the following table:

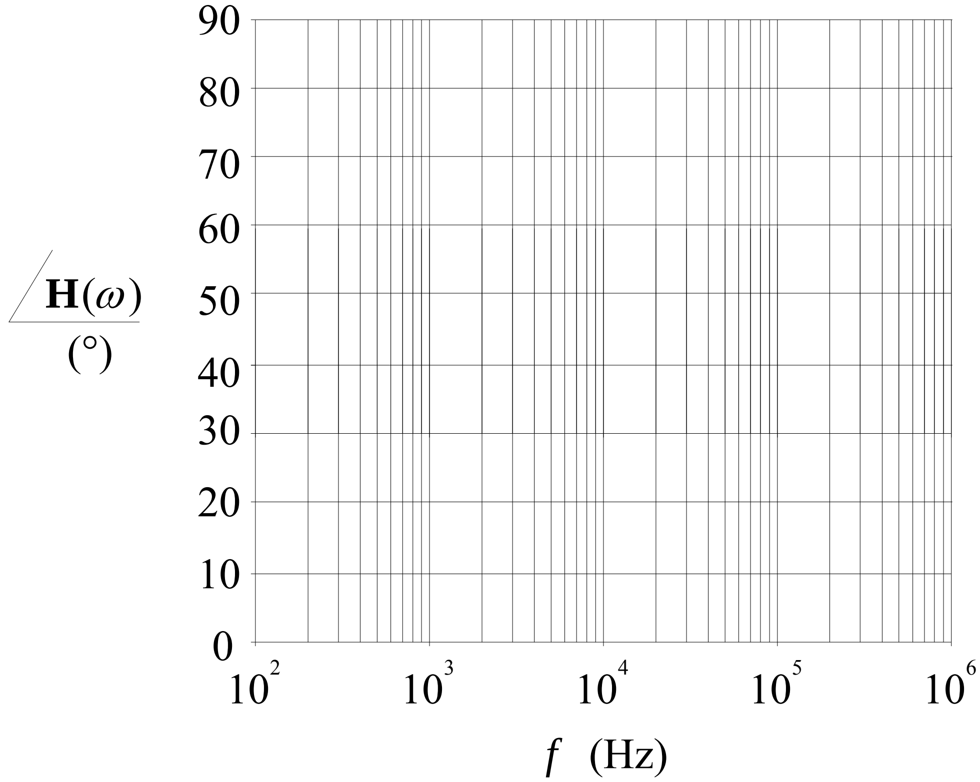
Frequency f (kHz)	Frequency ω (krads^{-1})	Gain $\frac{\lvert\mathbf{V}_o\rvert}{\lvert\mathbf{V}_i\rvert}$ (V/V)		Gain $20\log_{10}\frac{\lvert\mathbf{V}_o\rvert}{\lvert\mathbf{V}_i\rvert}$ (dB)		Phase $\angle\frac{\mathbf{V}_o}{\mathbf{V}_i}$ (°)	
		$R = 130\,\Omega$	$R = 33\,\Omega$	$R = 130\,\Omega$	$R = 33\,\Omega$	$R = 130\,\Omega$	$R = 33\,\Omega$
0.1	0.6283	0.0033	0.0129	-49.67	-37.76	89.8	89.3
0.2	1.257	0.0066	0.0259	-43.65	-31.74	89.6	88.5
0.5	3.142	0.0164	0.0646	-35.69	-23.80	89.1	86.3
1	6.283	0.0328	0.1284	-29.67	-17.83	88.1	82.6
2	12.57	0.0656	0.2507	-23.66	-12.02	86.2	75.5
5	31.42	0.1622	0.5434	-15.80	-5.297	80.7	57.1
10	62.83						
20	125.7	0.5493	0.9329	-5.204	-0.6037	56.7	21.1
50	314.2						
100	628.3	0.9567	0.9970	-0.3845	-0.0258	16.9	4.4
200	1257	0.9886	0.9993	-0.0994	-0.0065	8.7	2.2
500	3142	0.9982	0.9999	-0.0161	-0.0010	3.5	0.9
1000	6283	0.9995	1.0000	-0.0040	-0.0003	1.7	0.4

Table L7.2

L7.10

4. Plot the gain and phase values from Table L7.2 on the Bode plot below:





Real Inductors

An inductor is made by winding a coil of wire on a former:

A real inductor

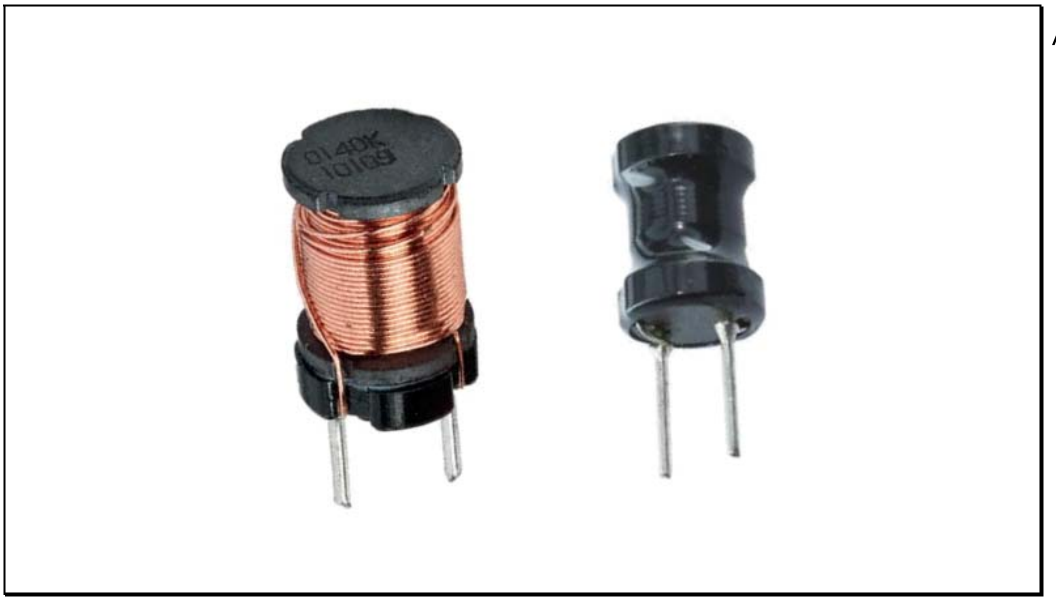


Figure L7.3

The wire used to wind the coil may have a considerable resistance if there are many turns of small diameter wire. In this case the inductor model used in the previous circuit is inadequate. A better model is shown below:

Real inductors

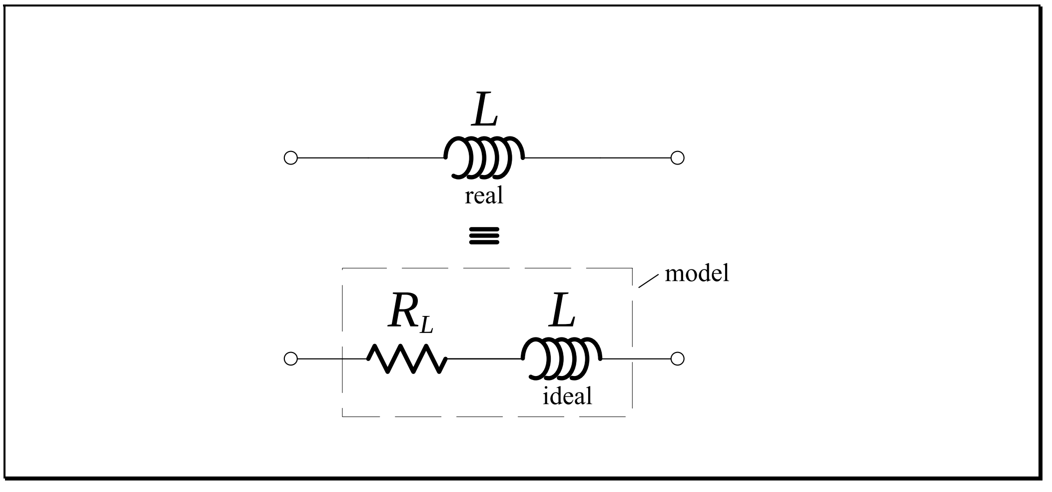


Figure L7.4

Note that the resistance and inductance cannot be separated – they are just the ideal model components of a real inductor. The model resistance of the inductor can be measured with a multimeter since the resistance of the model inductance component is zero.

L7.12

Highpass *RL* First-Order Circuit – Real Inductor

Step Response

1. The circuit shown below is an a “zero-state”, $i(0^-) = 0\text{ A}$:

Highpass *RL* first-order circuit with a real inductor

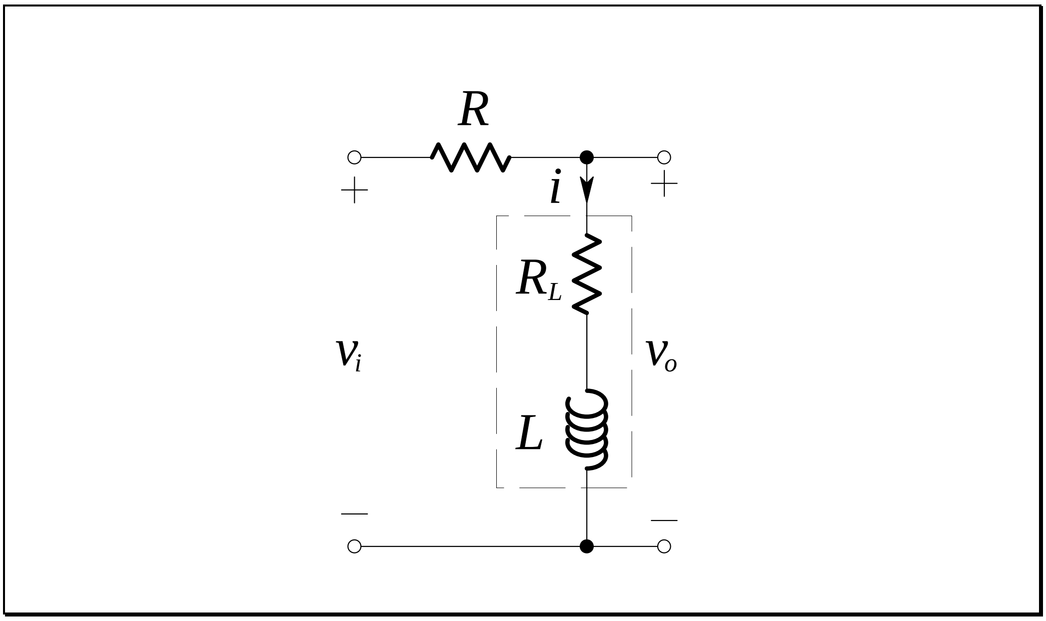


Figure L7.5

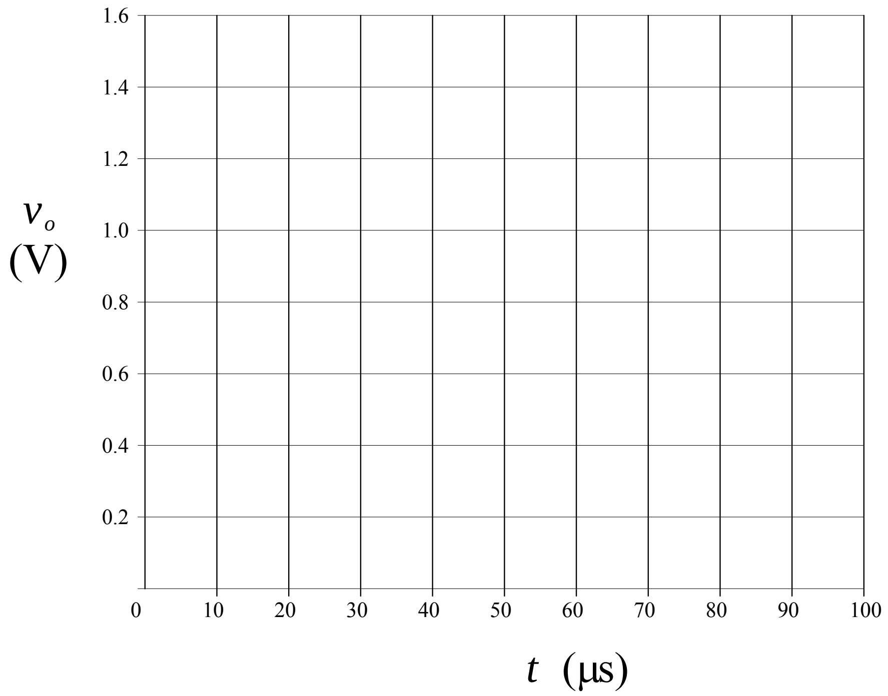
Derive an expression for the unit-step response of the circuit, $v_0(t)$.

2. What is the *time constant* for this circuit in terms of R, R_L and L?

$T =$

3. From the expression for the unit-step response, derive an expression for the current through the inductor, $i(t)$.

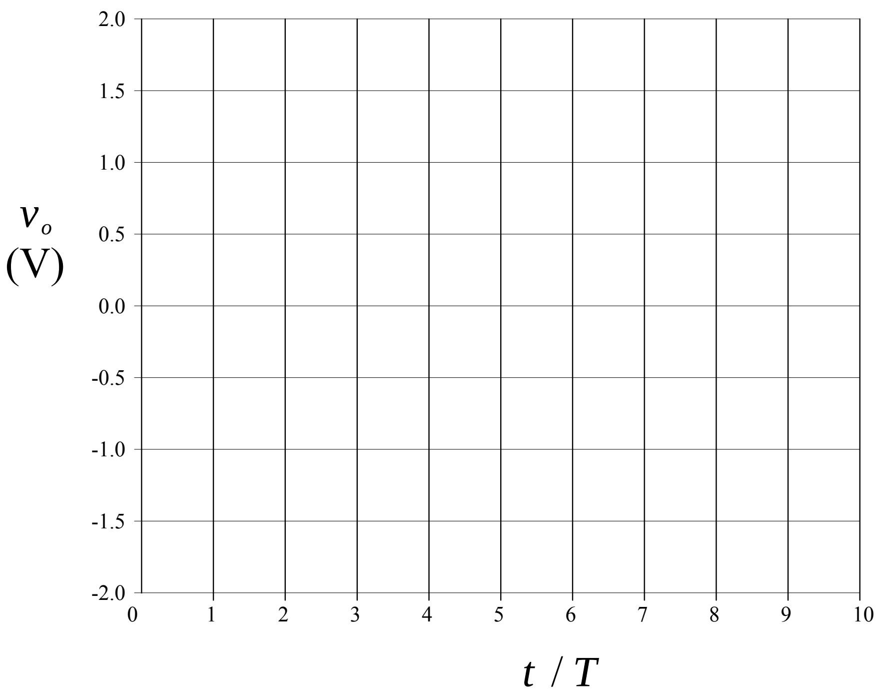
4. Let $L = 680\ \mu\text{H}$, $R_L = 2.6\ \Omega$ and $R = 130\ \Omega$ then $R = 33\ \Omega$. Sketch, for both values of resistance, the unit-step response over the period $0 < t < 100\ \mu\text{s}$.



5. The step response derived in 4 assumes zero initial conditions. If the input voltage is a square wave (-1 V to 1 V), the response to each half cycle of the input will be influenced by the response to the previous half cycle. The initial conditions will **NOT** be zero. If the period of the input is $T_0 = 10T$, derive an expression for the response in the first half-cycle under steady-state conditions.

   Hint: What will the initial condition be for each positive half cycle?

6. If $R_L = R/10$, sketch the circuit's response to a 2 Vp-p square wave, of period $10T$.



7. How is this different to the ideal inductor step response?

L7.16

Frequency Response

Highpass *RL* first-order circuit with a real inductor

1. For the circuit:

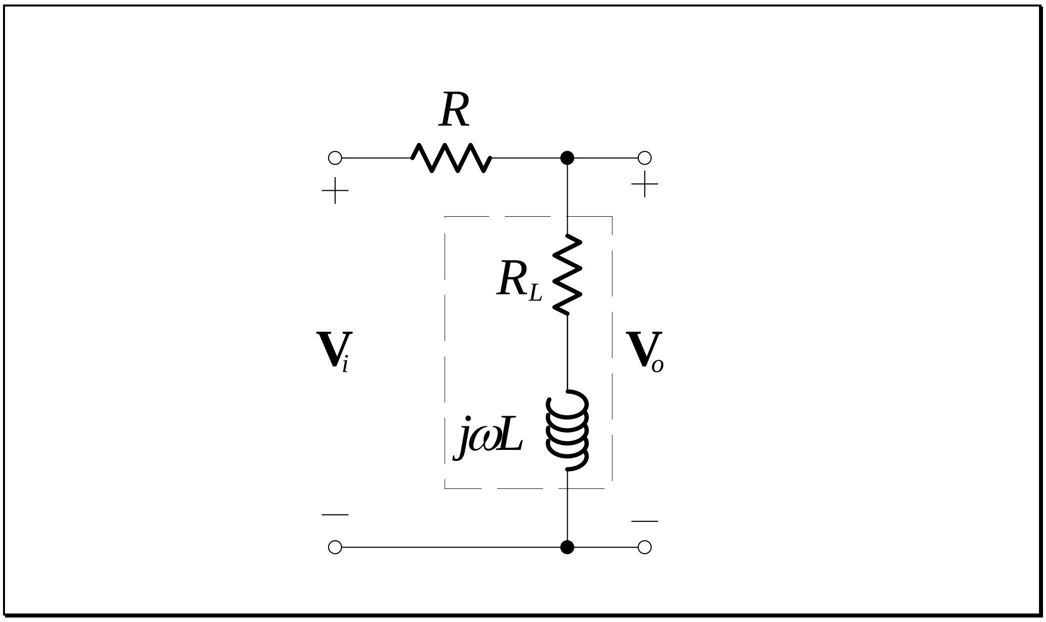


Figure L7.6

derive the frequency response:

$$\mathbf{H}(j\omega) = \frac{\mathbf{V}_o}{\mathbf{V}_i}, \tag{L7.3}$$

2. How is this different to the ideal inductor frequency response?

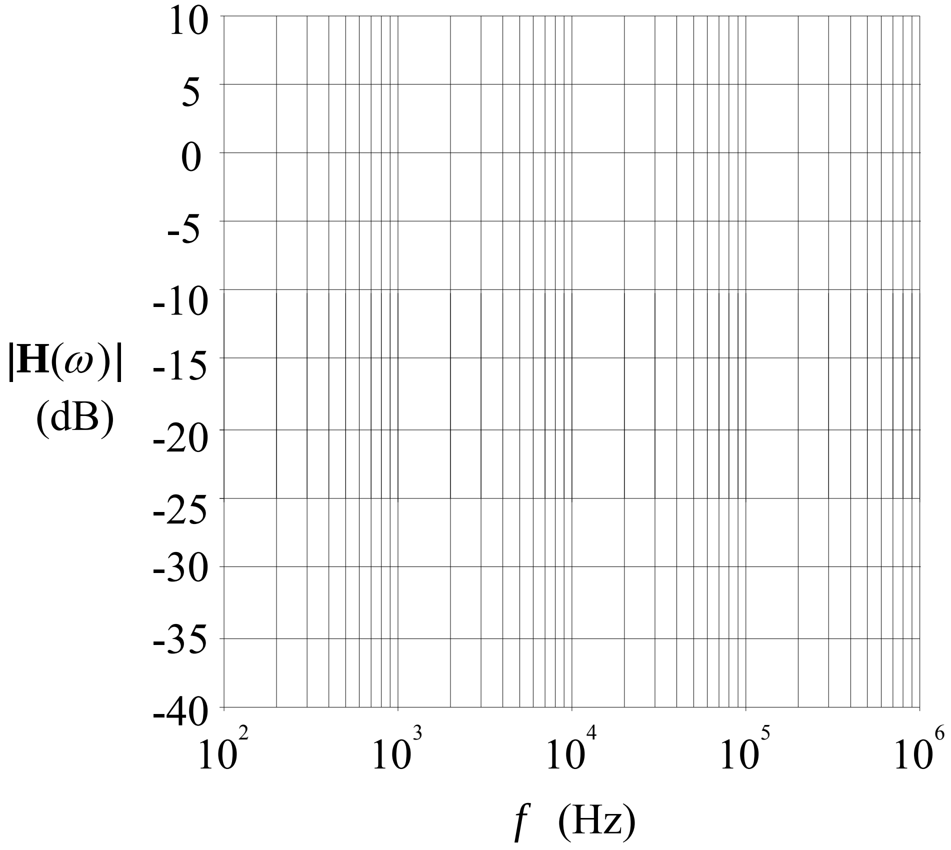
3. Let $L = 680\,\mu\text{H}$, $R_L = 2.6\,\Omega$ and $R = 130\,\Omega$ then $R = 33\,\Omega$, and complete the following table:

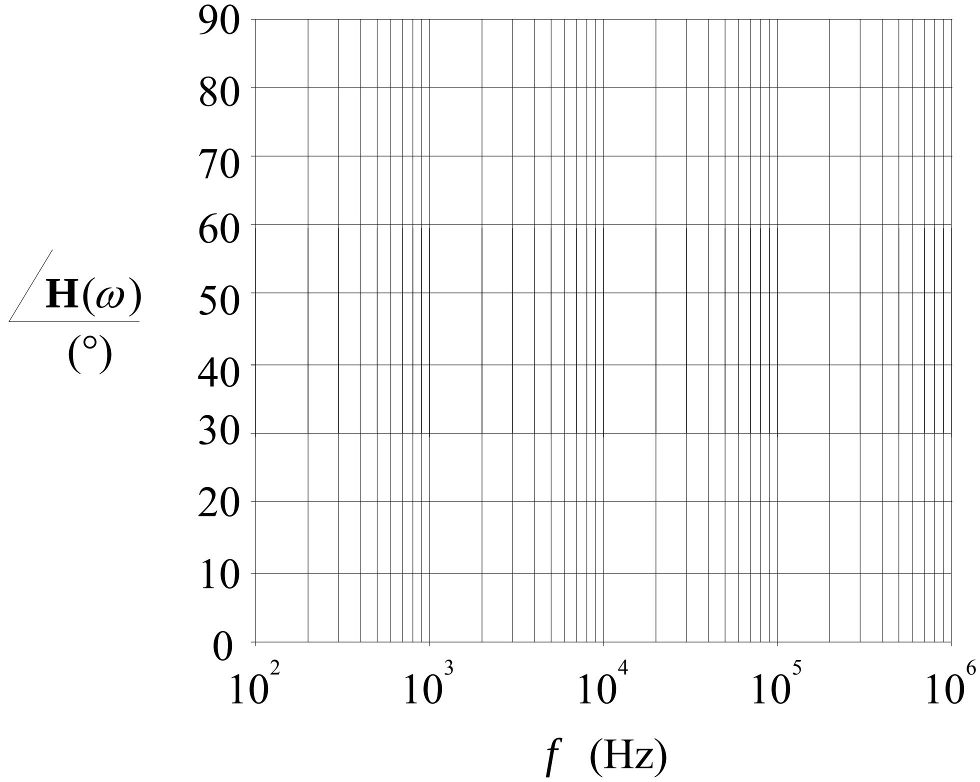
Frequency f (kHz)	Frequency ω (krads^{-1})	Gain $\frac{\lvert\mathbf{V}_o\rvert}{\lvert\mathbf{V}_i\rvert}$ (V/V)		Gain $20\log_{10}\frac{\lvert\mathbf{V}_o\rvert}{\lvert\mathbf{V}_i\rvert}$ (dB)		Phase $\angle\frac{\mathbf{V}_o}{\mathbf{V}_i}$ (°)	
		$R=130\,\Omega$	$R=33\,\Omega$	$R=130\,\Omega$	$R=33\,\Omega$	$R=130\,\Omega$	$R=33\,\Omega$
0.1	0.6283	0.0199	0.0740	-34.04	-22.61	9.1	8.6
0.2	1.257	0.0206	0.0769	-33.71	-22.29	17.8	16.8
0.5	3.141	0.0254	0.0944	-31.91	-20.51	38.5	36.0
1	6.283	0.0377	0.1395	-28.47	-17.11	56.8	51.8
2	12.57	0.0672	0.2440	-23.45	-12.25	69.4	59.6
5	31.41	0.1602	0.5183	-15.91	-5.708	73.9	52.1
10	62.83						
20	125.7	0.5419	0.9235	-5.321	-0.6911	55.5	20.9
50	314.1						
100	628.3	0.9551	0.9966	-0.3992	-0.0299	16.9	4.4
200	1257	0.9882	0.9991	-0.1033	-0.0075	8.6	2.2
500	3141	0.9981	0.9999	-0.0167	-0.0001	3.5	0.9
1000	6283	0.9995	1.0000	-0.0042	0.0000	1.7	0.4

Table L7.3

L7.18

4. Plot the gain and phase values from Table L7.3 on the Bode plot below:





Circuit Simulation

1. For the circuit:

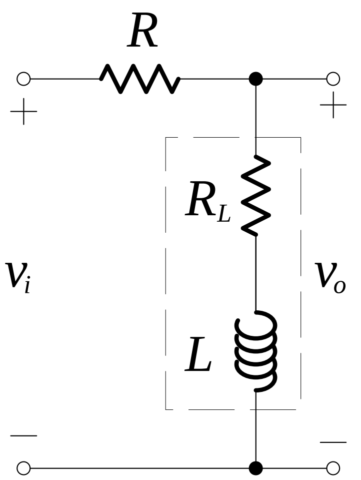


Figure L7.7

with $L = 680\ \mu\text{H}$, $R_L = 2.6\ \Omega$ and $R = 130\ \Omega$, conduct a simulation of the step response and frequency response using PSpice (e.g. OrCAD Demo).

2. Repeat with $L = 680\ \mu\text{H}$, $R_L = 2.6\ \Omega$ and $R = 33\ \Omega$.

3. Print hardcopies of the step and frequency responses to bring to the lab.

   Use the following axes for graphing:

Response	**X**	**Y**
Step	0 s to 100 μs	0 V to 2 V
Magnitude	100 Hz to 1 MHz	-40 dB to 10 dB
Phase	100 Hz to 1 MHz	0° to 90°

Lab Work

Highpass *RL* First-Order Circuit – Real Inductor

Mini-Lab Amplifier Setup

The *RL* circuit, when subjected to a step input, requires a fairly large current to be delivered from the input in a short span of time. The function generator has an output resistance of $50\,\Omega$ and so the output will therefore experience a significant internal Ri voltage drop, resulting in a "droop" in the applied voltage when delivering current. We therefore need to "buffer" the output of the function generator. The Mini-lab provides us with a way to do this.


Mini-lab Amplifier

1. Identify the section under the power switch labelled "AMPLIFIER OR BI-POLAR POWER SUPPLY".

2. Ensure that the left-most pushbutton is out (F. GEN) so that the internal function generator is selected as the input.

3. Ensure that the middle pushbutton is out (NORM) so that the output is normal.

4. Ensure that the right-most pushbutton is out (AMP) so that the unit acts as an amplifier.

5. Ensure that the knob is fully rotated counter-clockwise to select a gain of "x 1".

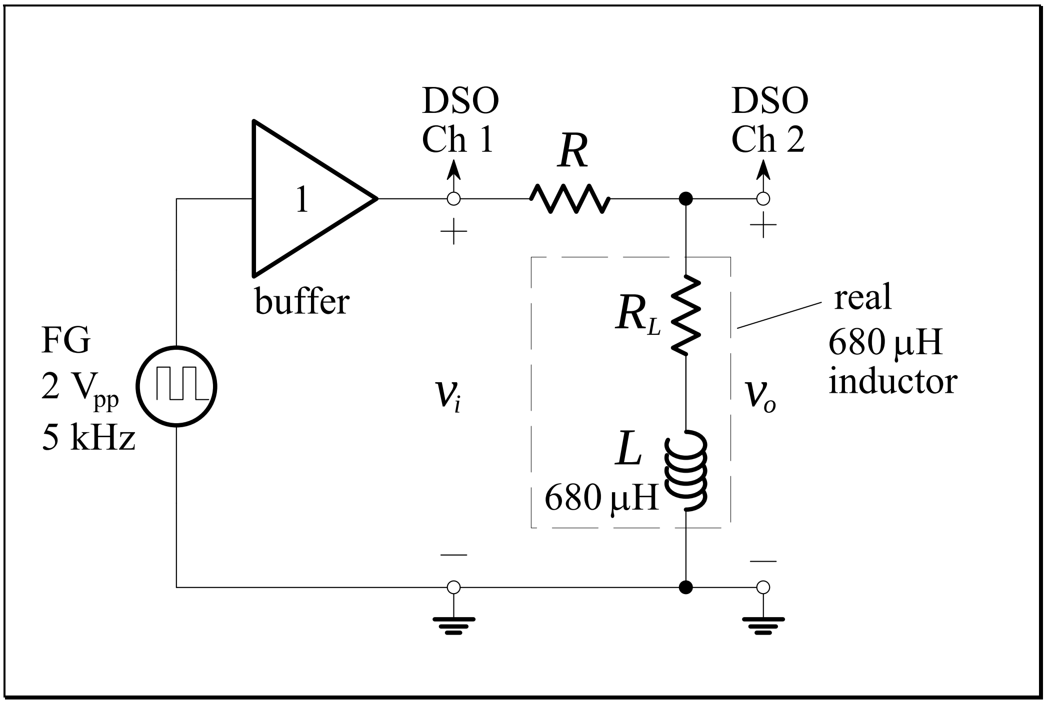
6. With these settings the buffered output of the function generator can be taken directly from the red output terminal.

Step Response

1. Using the DMM, measure and record the DC series equivalent resistance of your inductor:

$R_L =$

2. Construct the following circuit, using $R = 130\,\Omega$:



Highpass *RL* first-order circuit with a real inductor

Figure L7.8

The input is a 2 V peak-to-peak **square wave** (i.e. -1 V to +1 V) with a frequency of 5 kHz. Note that the output of the Mini-lab amplifier is the "buffered" function generator.

Measurement of the step response of a circuit is conducted with a square wave!

3. Connect the input to Channel 1 of the DSO and the output to Channel 2 of the DSO.

4. Press `Main/Delayed`. Change the `Time Ref` softkey to "Left". This will facilitate sketching the step-response.

5. Set the DSO horizontal time base to 10 µs / div.

6. Set Channel 1 to 2 V/div with a position of -5.000 V. This will put the input waveform at the top of the DSO display.

7. Set Channel 2 to 1 V/div with a position of 0.000 V. This will put the output waveform in the middle of the DSO display.

8. Ensure that "bandwidth limiting" is used for both Channels 1 and 2.

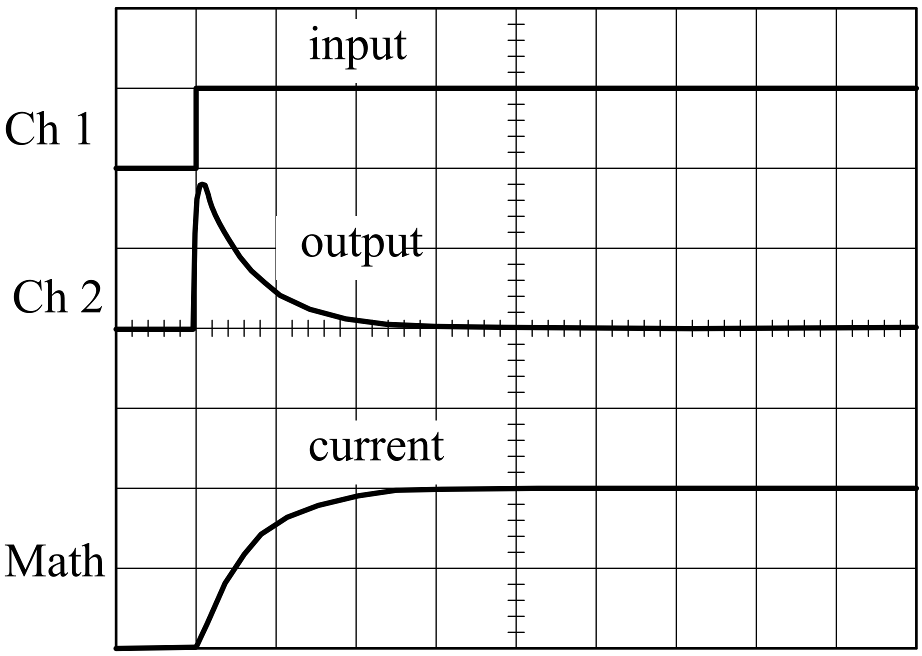
9. Use the DSO `Math` function to subtract Channel 2 from Channel 1 so that the DSO displays the voltage across the resistor (you cannot measure this voltage with the DSO leads because the black lead is connected to earth and will short out the inductor). This voltage is proportional to the current, since the voltage is across a resistor.

10. Under the `Math` function, press the `Settings` softkey. Set the `Scale` to 1.00 V/ and the `Offset` to 3.00 V/. This will position the voltage across the resistor, i.e. the current waveform, at the bottom of the DSO display.

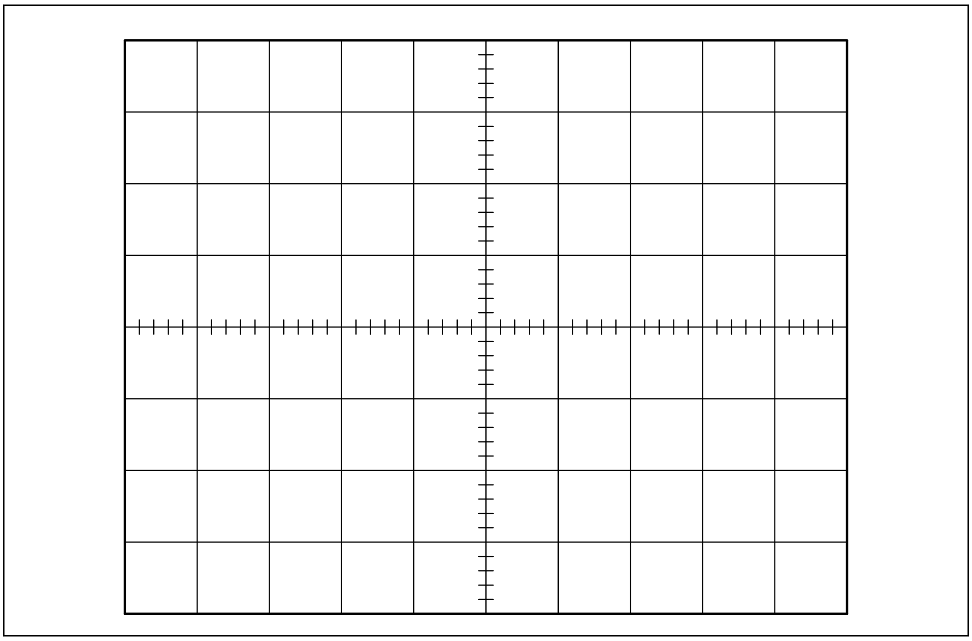
11. Ensure that the DSO has a stable trigger signal – you may need to choose "Noise Reject" in the trigger options.

12. Set up waveform averaging for 64 averages. This will ensure that almost noise-free waveforms are displayed on the DSO.

We have now set up the display to look like:

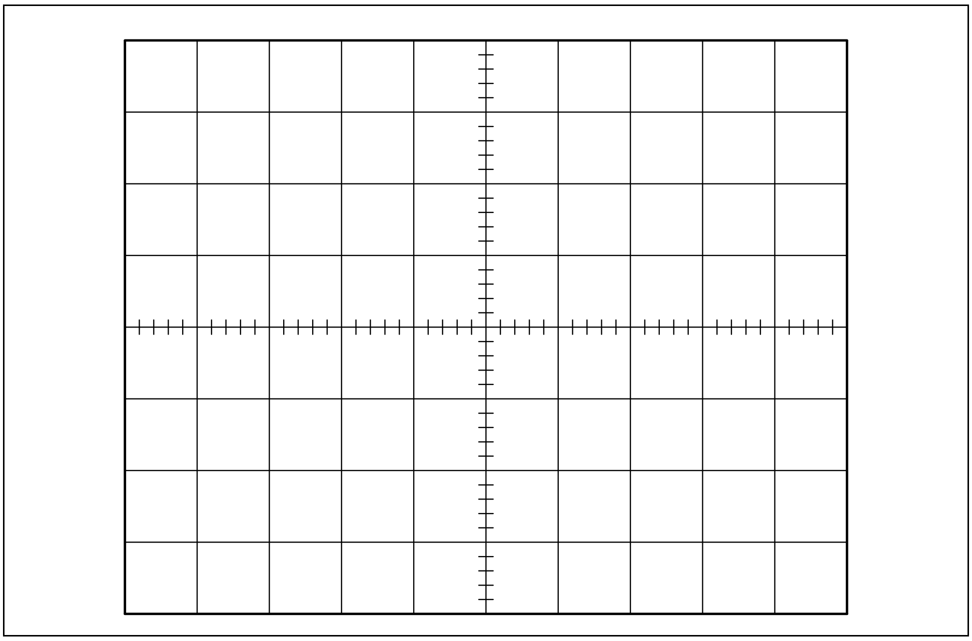


13. Draw the input (Ch 1), output (Ch 2) and current (Math) waveforms, ensuring that the sketch is labelled with both voltage and time scales.



14. Increase the frequency of the input to 25 kHz.

15. Draw the input (Ch 1), output (Ch 2) and current (Math) waveforms, ensuring that the sketch is labelled with both voltage and time scales.



16. Change the resistor to $R = 33\,\Omega$, and reduce the frequency to 5 kHz.

17. Repeat steps 13 to 15.

Frequency Response

Highpass *RL* first-order circuit with a real inductor

1. Construct the following circuit, using $R = 130\,\Omega$:

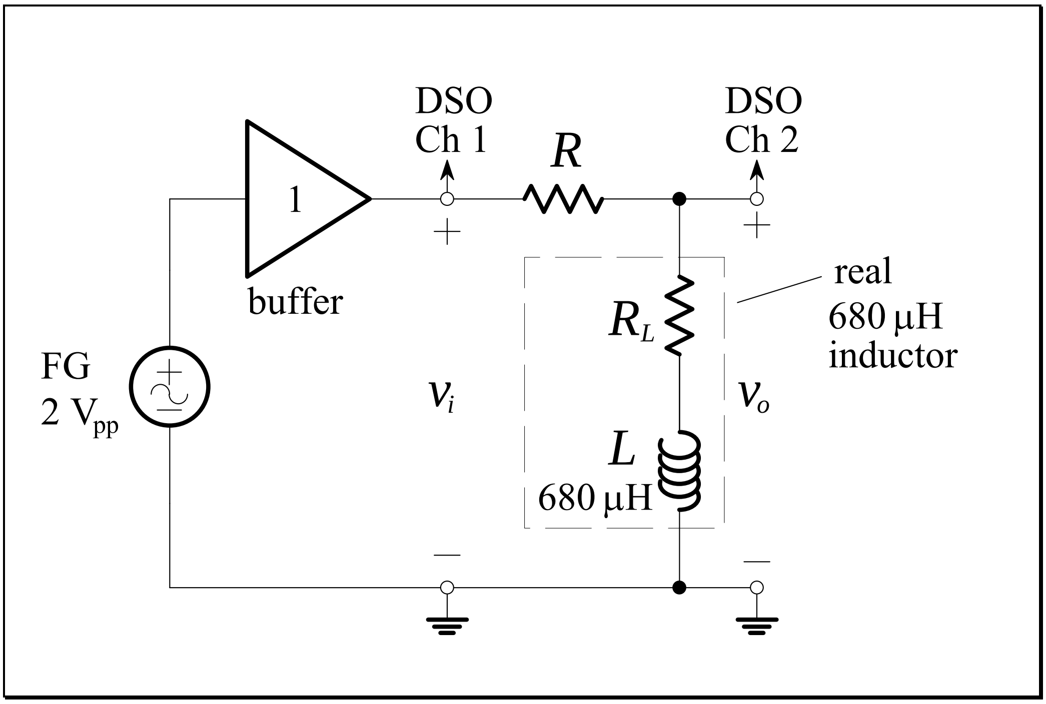


Figure L7.9

Measurement of the frequency response of a circuit is conducted with a sine wave!

The input is a 2 V peak-to-peak **sine wave** (i.e. -1 V to +1 V) with a variable frequency. Note that the output of the Mini-lab amplifier is the "buffered" function generator.

2. Connect the input to Channel 1 of the DSO and the output to Channel 2 of the DSO. Turn the `Math` function off.

3. Set Channel 1 to 500 mV/div with a position of 0.000 V. This will put the input sinusoid back into the middle of the DSO display.

4. In measuring the frequency response, make sure to measure the Channel 1 amplitude, as it will decrease with increasing frequency due to the "bandwidth" of the Mini-lab buffer.

   In taking a frequency response of a circuit, the fastest measuring technique is to set the frequency vernier to a desired frequency, such as 100 Hz, then simply change the FG frequency range to get the 1 kHz reading, then the 10 kHz reading etc.

5. Complete the following table:

Desired Frequency f_{desired} (kHz)	Actual Frequency f_{actual} (kHz)	Gain $\frac{\lvert\mathbf{V}_o\rvert}{\lvert\mathbf{V}_i\rvert}$ (V/V)		Gain $20\log_{10}\frac{\lvert\mathbf{V}_o\rvert}{\lvert\mathbf{V}_i\rvert}$ (dB)		Phase $\angle\frac{\mathbf{V}_o}{\mathbf{V}_i}$ (°)	
		$R = 130\,\Omega$	$R = 33\,\Omega$	$R = 130\,\Omega$	$R = 33\,\Omega$	$R = 130\,\Omega$	$R = 33\,\Omega$
0.1							
0.2							
0.5							
1							
2							
5							
10							
20							
50							
100							
200							
500							
1000							

Table L7.4

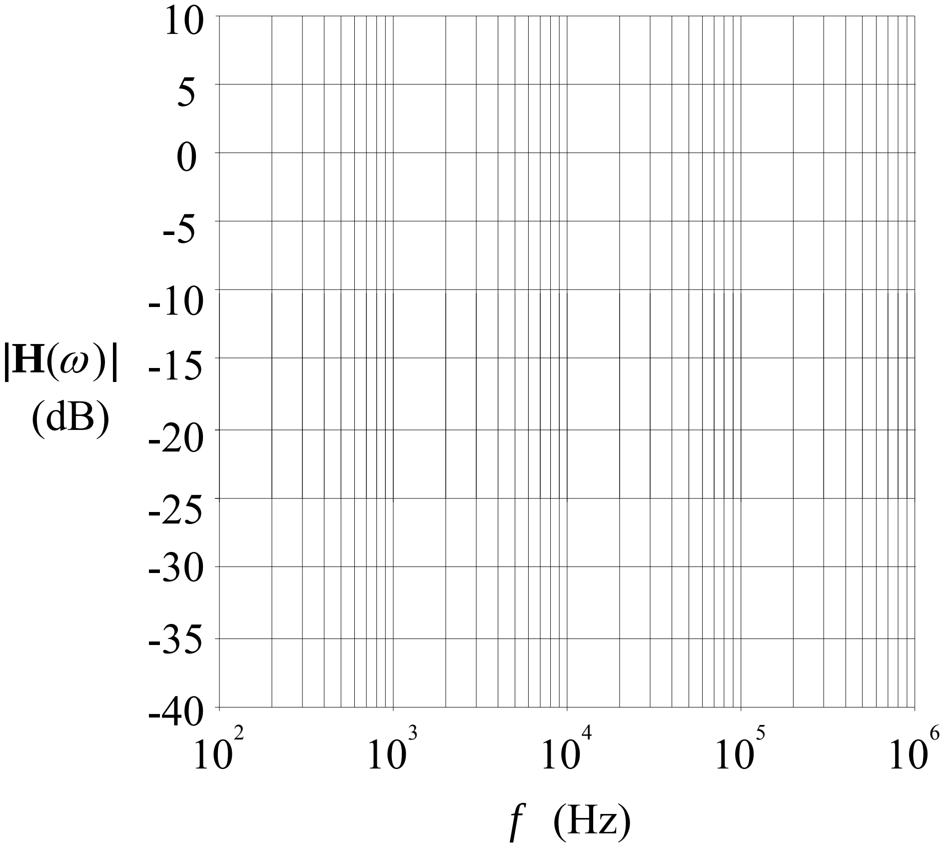
6. Find the break frequency (also named the -3 dB frequency, or corner frequency, or cutoff frequency), f_0, and record the magnitude of the gain and phase shift in the table below.

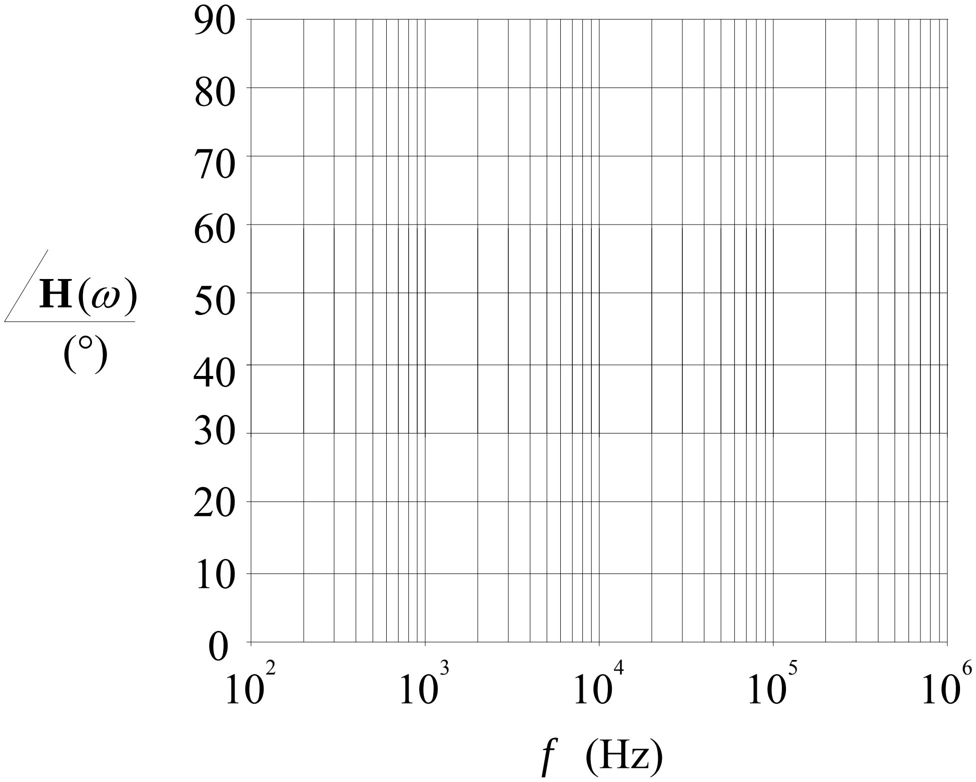
	Cutoff Frequency f_0 (kHz)	Gain $\frac{\lvert\mathbf{V}_o\rvert}{\lvert\mathbf{V}_i\rvert}$ (V/V)	Gain $20\log_{10}\frac{\lvert\mathbf{V}_o\rvert}{\lvert\mathbf{V}_i\rvert}$ (dB)	Phase $\angle\frac{\mathbf{V}_o}{\mathbf{V}_i}$ (°)
$R = 130\,\Omega$				
$R = 33\,\Omega$				

Table L7.5

7. Repeat steps 5 and 6 with $R = 33\,\Omega$.

8. Plot the gain and phase values from Table L7.4 on the Bode plot below:





Questions – Highpass Frequency Response

1. What is the relationship between the break frequency and the time constant?

Answer:

2. At what rate does the magnitude response rise towards the break frequency? (Draw an asymptote on your graph and measure its slope).

Answer:

3. How do you experimentally determine the break frequency?

Answer:

L7.28

Lab Assessment [2 marks]

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

1. Show the result of the pre-lab work, **real inductor Step Response** section, step 4.

2. Show the result of the pre-lab work, **real inductor Circuit Simulation** section, step 3.

3. Demonstrate the measurement of the time constant of a highpass STC *RL* circuit that uses a real inductor using the step response (use $R = 130\,\Omega$, $L = 680\ \mu\text{H}$).

4. Show the frequency response (magnitude and phase) of a highpass STC *RL* circuit that uses a real inductor (use $R = 130\,\Omega$, $L = 680\ \mu\text{H}$).

Marking

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

Lab 8 – Waveform Generation

Open-loop comparator. Comparator with hysteresis. Astable multivibrator. Waveform generator.

Introduction

A comparator uses the op-amp in an *open-loop* mode. For a very small input voltage, the output will saturate close to one of the power supply voltages due to the very large gain of the op-amp.

Positive feedback can be applied to a comparator to create hysteresis. This can be used to “clean-up” noisy digital waveforms, amongst other applications, and is an example of a *bistable* circuit (it has two stable states). It can also be used to make an *astable multivibrator*. The output will oscillate at a rate which can be set by a few passive components.

A comparator with hysteresis can also be used to generate simple waveforms such as square waves and triangle waves. With proper filtering, sinusoids can also be generated.

Objectives

1. To examine comparator circuits in more detail, including hysteresis, and to design and build a simple waveform generator.

Equipment

- 1 Digital Storage Oscilloscope (DSO) – Agilent 54621A
- 1 Mini-Lab – BWD 604
- Op-amp – 2 x TL071

  Resistors –1 x 1 kΩ, 1 x 3.9 kΩ, 1 x 4.7 kΩ, 1 x 10 kΩ, 1 x 22 kΩ,
  1 x 47 kΩ, 1 x 100 kΩ, 1 x 220 kΩ

  Potentiometer – 1 x 10 kΩ

  Capacitors – 3 x 10 nF, 1 x 68 nF, 1 x 470 nF, 2 x 10 μF

  Diodes – 1 x red LED

  Breadboard, Hook-up wire, 2 x 4mm leads.

L8.2

Safety

Cat. A lab

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

Warning!

Remember:

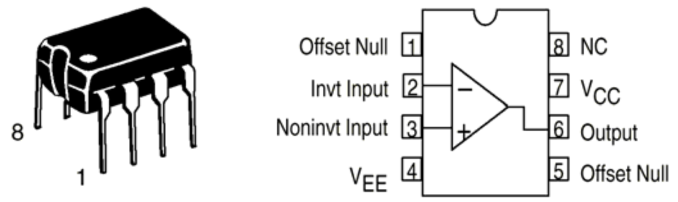
1. **When wiring the circuits, ensure that the power supply is switched off.**

2. **It is very important to place the polarised electrolytic capacitors into the circuit with the correct polarity.**

   Failure to do so will result in the capacitor failing catastrophically which may cause personal injury. If this happens, you will be awarded 0 marks for the lab and asked to leave!

Laboratory Preparation

- The pin-out for the TL071 op-amp is given below:

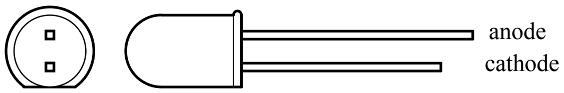


Op-amp package details

Figure L8.1

- ***For the TL071, pin 7 is connected to the positive supply and pin 4 is connected to the negative supply.***

- The pin-out for an LED is given below. The cathode is marked by a flat edge on the lens. New LEDs also have a shorter lead on the cathode.



LED package details

Figure L8.2

- It would be a good idea to plan the layout of all the circuits as they will appear on your breadboard before you begin. This will minimise construction time in the lab, and assist in debugging circuits that do not appear to be working.

- A pair of pliers, a pair of wire cutters and a pair of wire strippers would be handy to wire a neat circuit; straighten bent leads; insert components into the breadboard etc. If you have any of these tools, bring them to the lab!

L8.4

Open-Loop Comparator

A comparator is an example of a non-linear op-amp circuit. It is a switching device that produces a high or low output, depending on which of the two inputs is larger. A simple comparator can be made from an op-amp with no feedback connection (*open-loop*) as shown in the figure below:

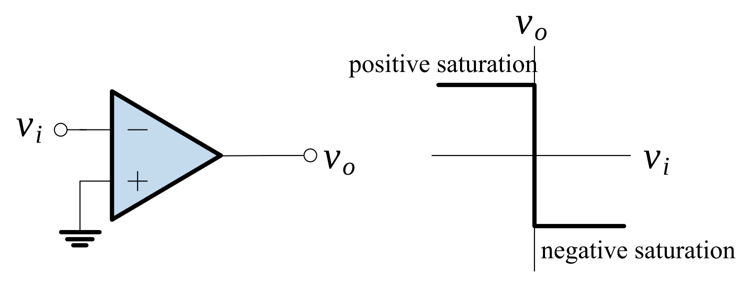


Figure L8.3

Since the open-loop voltage gain of an op-amp is very large, when there is no feedback an input voltage difference of only a few microvolts is sufficient to drive the output voltage to either its maximum (V_{OH}) or to its minimum value (V_{OL}). These values are determined by the op-amp supply voltages and its internal structure; their magnitudes are always slightly lower than that of their respective supply values ($V_{OH} < V_{CC}, |V_{OL}| < |V_{EE}|$).

This feature is used in comparator circuits, when one wishes to know whether a given input is larger or smaller than a reference value. It is especially useful in digital applications, such as in analog to digital converters (ADCs).

<u>Note</u>: In practical applications that require a comparator, an op-amp should **not** be used. This lab uses the op-amp as a comparator to demonstrate the basic principles. Semiconductor manufacturers produce specific integrated circuit comparators that have a different output stage to op-amps and are specifically designed to optimise operation in "saturation".

In the Lab – Open-Loop Comparator

1. Construct the comparator circuit shown below:

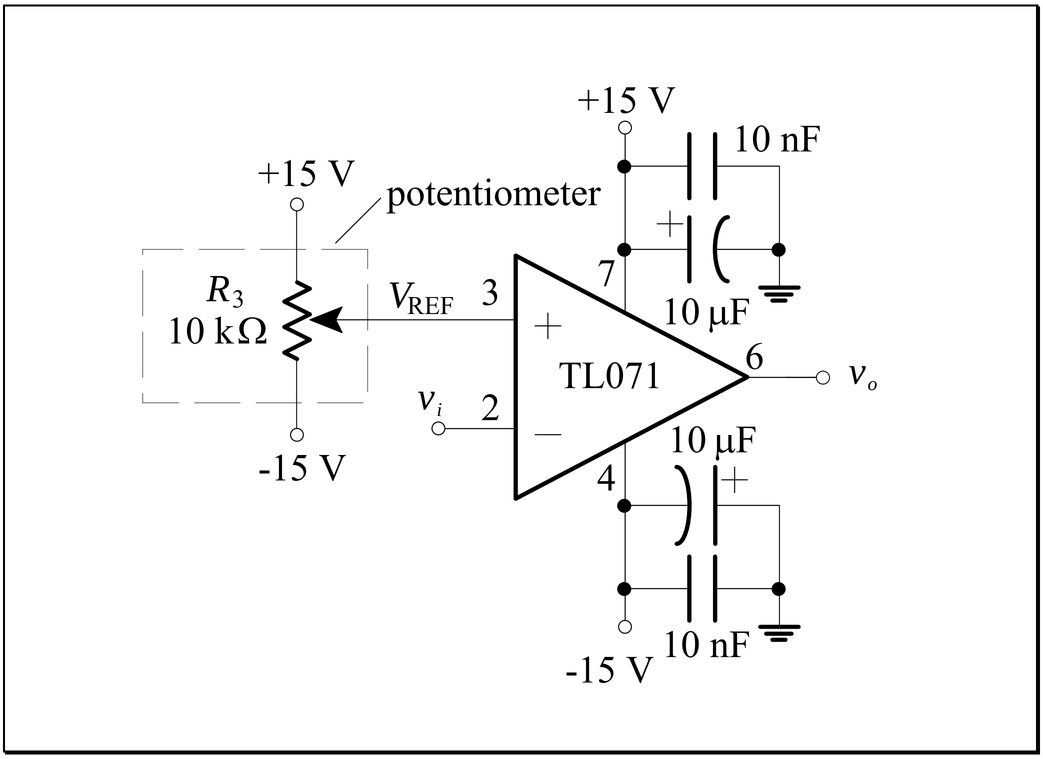


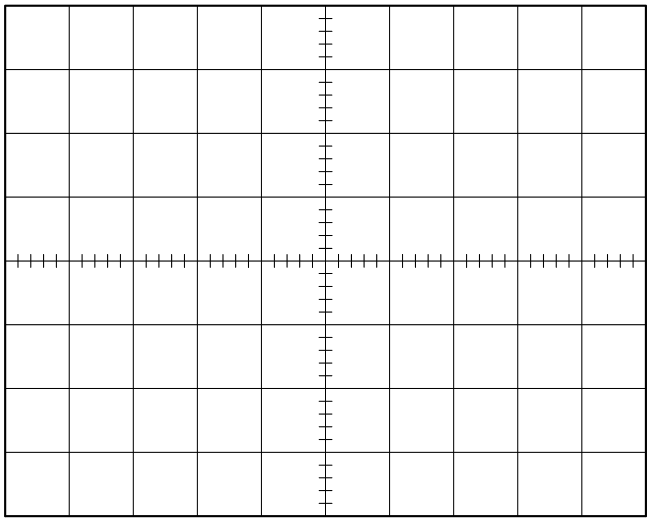
Figure L8.4

2. Set v_i to zero (by connecting the inverting terminal to common). Adjust the potentiometer to set V_{REF} above zero (say, $V_{REF} \approx 500$ mV). Measure and record $v_o = V_{OH}$. Then, adjust the potentiometer to set V_{REF} below zero (say, $V_{REF} \approx -500$ mV). Measure and record $v_o = V_{OL}$.

$V_{OH} =$
$V_{OL} =$

3. Set the function generator to a 1 kHz sinusoidal signal with an amplitude of approximately 2 V_{pp}. Apply this signal to the input of your circuit. For several values of V_{REF} (say, $V_{REF} \approx 500$ mV, 0 V, −500 mV), sketch the observed v_o vs. t waveforms on the oscilloscope on the plot below:

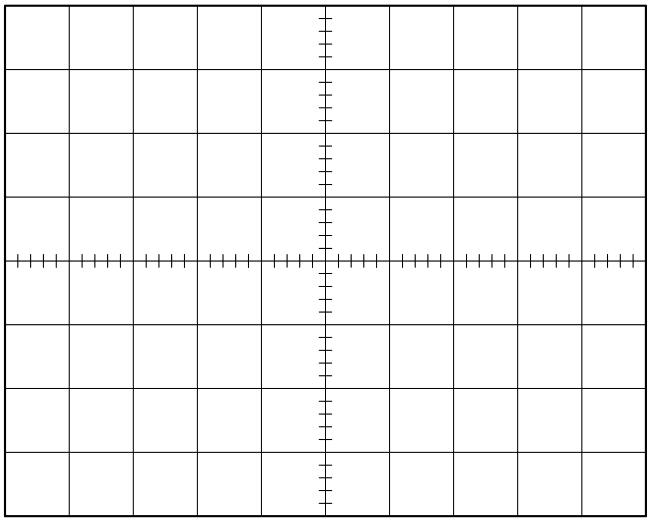
Comparator output waveforms



Explain the waveforms:

4. Display v_o vs. v_i using the X-Y mode of the oscilloscope. The image will be the *voltage transfer characteristic* of the comparator. Record its shape for several different values of V_{REF} (say, $V_{\text{REF}} \approx 500$ mV, 0 V, −500 mV), on the plot below:

Comparator transfer characteristics



Explain the characteristics:

Comparator with Hysteresis (Schmitt Trigger)

The Schmitt trigger shown in Figure L8.5 is an extension of the comparator. The positive feedback and absence of negative feedback ensures that the output will always be at either its highest (V_{OH}) or its lowest (V_{OL}) possible value. The voltage divider formed by R_1 and R_2 sets V_+ at a fraction of the output.

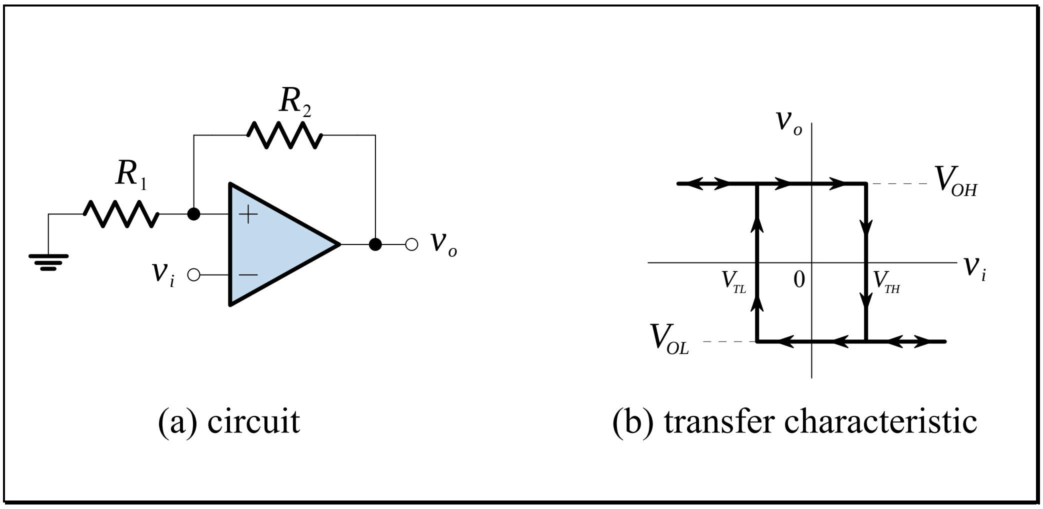


Figure L8.5

If $v_i > V_+$, the output is negative, if $v_i < V_+$ the output is positive. Each time the difference $v_i - V_+$ changes sign, the polarity of the output, and consequently of V_+, changes. No further change is possible until v_i reaches the new reference value V_+. The result is that the output may be at either extreme value (V_{OH} or V_{OL}) for the same value of the input; whether the output is positive or negative is determined by its previous state. The circuit therefore possesses memory. The consequence of this is that the voltage transfer characteristic of a Schmitt trigger follows a different curve, depending on whether the independent variable is increasing or decreasing. This property is called ***hysteresis*** and is depicted in Figure L8.5. Since the circuit has two stable states, it is also called a ***bistable*** circuit.

The thresholds for a change of an output state can be calculated as:

$$V_{TL} = V_{OL} \frac{R_1}{R_1 + R_2}, \qquad V_{TH} = V_{OH} \frac{R_1}{R_1 + R_2}$$

It is important to note that in order for the output to change state all that is needed is a short departure of the input voltage above or below the respective threshold. This initiates the regenerative process that results in changing the state.

The figure below shows a *noninverting* Schmitt trigger with an adjustable reference voltage.

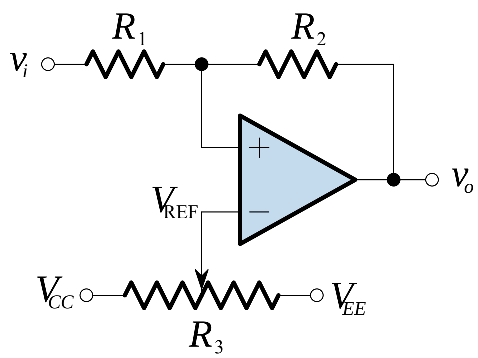


Figure L8.6

Using superposition, we can write the expression for v_+ :

$$v_+ = v_i \frac{R_2}{R_1 + R_2} + v_o \frac{R_1}{R_1 + R_2}$$

Let's assume that the circuit is in the positive stable state with $v_o = V_{OH}$. Then, in order to change this state to negative output, we must make $v_+ < V_- = V_{\mathrm{REF}}$. This means we need to apply:

$$v_i < V_{TL} = V_{\mathrm{REF}}\left(1 + \frac{R_1}{R_2}\right) - V_{OH}\frac{R_1}{R_2}$$

Similarly, to change the state from low to high, the input voltage must satisfy (even for a brief moment) the following inequality:

$$v_i > V_{TH} = V_{\mathrm{REF}}\left(1 + \frac{R_1}{R_2}\right) - V_{OL}\frac{R_1}{R_2}$$

In the Lab – Comparator with Hysteresis

1. Build the noninverting Schmitt trigger shown in Figure L8.7. Use a ±15 V supply. **Note**: Decoupling capacitors should be used but are not shown in the figure.

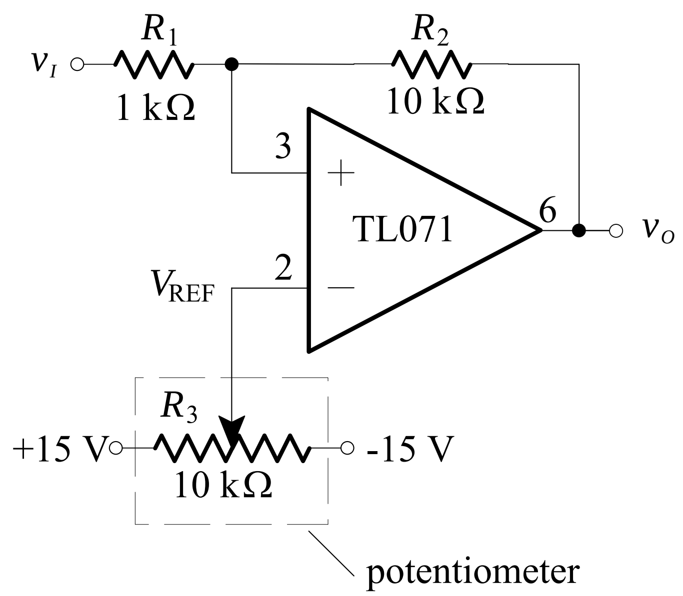
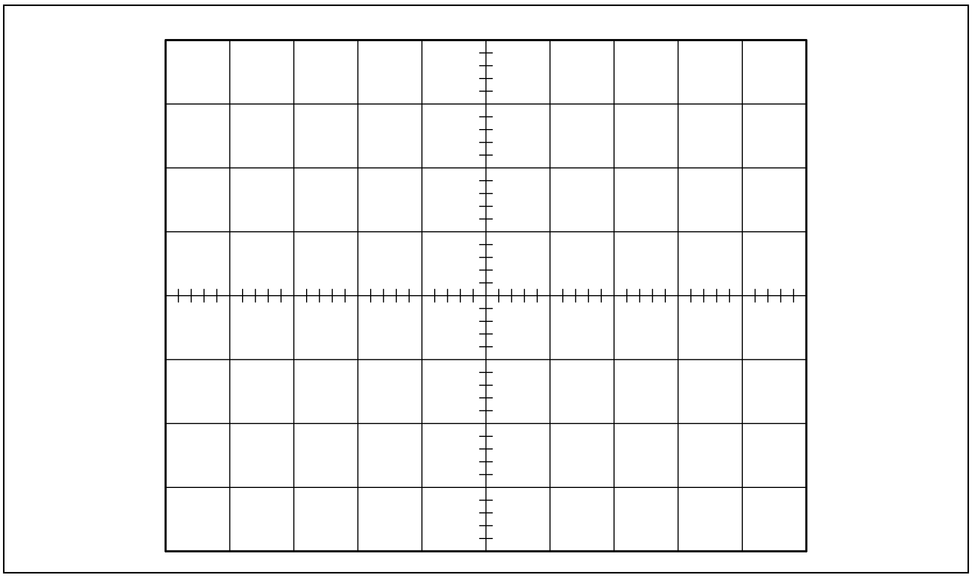


Figure L8.7

2. Calculate the low and high thresholds (V_{TL} and V_{TH}) for $V_{\text{REF}} = -2$ V, 0 V, +2 V. Use the values of V_{OL} and V_{OH} measured previously.

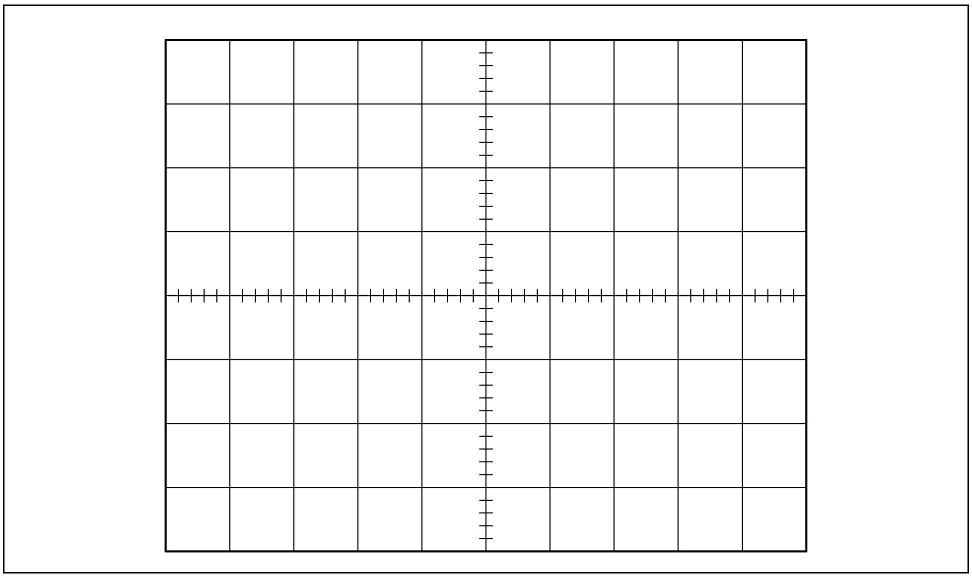
	$V_{\text{REF}} = -2$ V	$V_{\text{REF}} = 0$ V	$V_{\text{REF}} = 2$ V
V_{TL}			
V_{TH}			

3. Set the function generator to a 1 kHz ***triangular*** signal with an amplitude of approximately 5 V_{pp}. Apply this signal to the input of your Schmitt trigger. For several values of V_{REF} (say, $V_{REF} \approx -500$ mV, 0 V, +500 mV), sketch the observed v_o vs. t waveforms on the oscilloscope on the plot below:



Schmitt trigger output waveforms

4. Display v_o vs. v_i using the X-Y mode of the oscilloscope. The image will be the *voltage transfer characteristic* of the Schmitt trigger. You should observe hysteresis. Record its shape for several different values of V_{REF} (say, $V_{REF} \approx 500$ mV, 0 V, −500 mV), on the plot below:



Schmitt trigger transfer characteristics

Astable Multivibrator (Schmitt Trigger Clock)

When a negative feedback path consisting of a resistor R and a capacitor C is added to the Schmitt trigger in Figure L8.5, the new circuit has no stable state. The output will continuously switch between its two extremes at a rate determined by the time constant $T = RC$. The circuit is shown below:

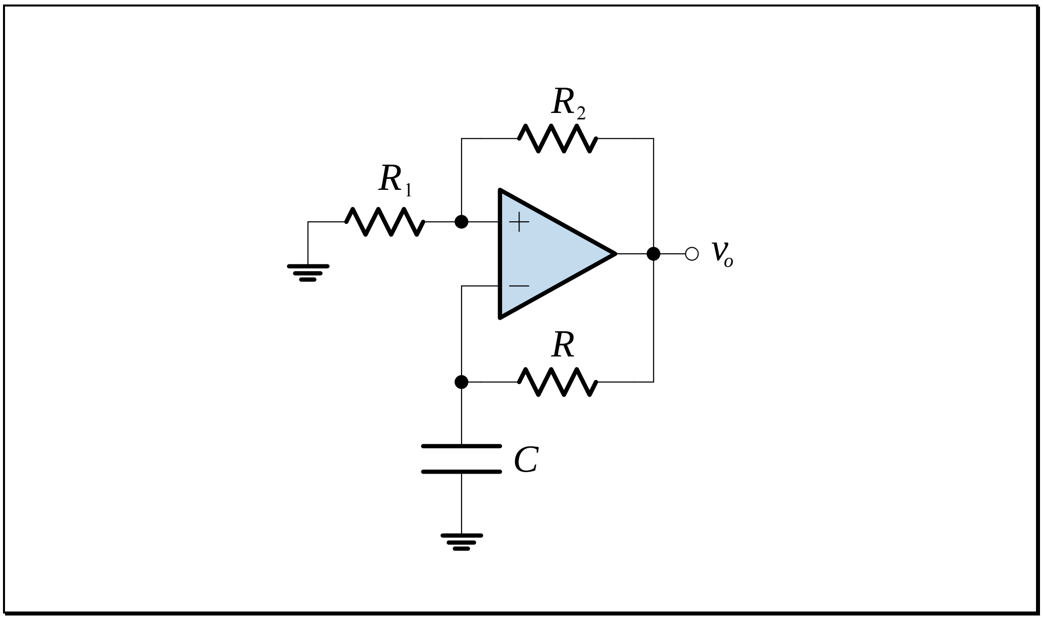
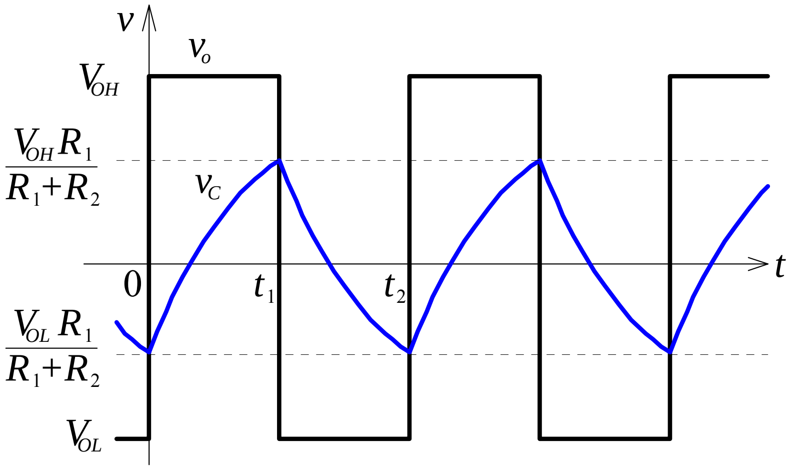


Figure L8.8

Immediately after a transition of the output to either its positive extreme (V_{OH}) or its negative extreme (V_{OL}), the RC network will begin an exponential transition; the capacitor will begin to charge or discharge, depending on its previous state, with its voltage approaching the new value of v_o. When the capacitor voltage v_- passes the value of v_+, which is determined by R_1 and R_2, the op-amp output will suddenly switch to its opposite extreme. The capacitor voltage will then begin to charge in the opposite direction until switching occurs again. The process will be repeated indefinitely, giving a square-wave output without the need for an input voltage source.



Suppose that at $t=0$ the output voltage is V_{OL}, and the capacitor voltage v_- has just fallen below $v_+ = V_{OL}R_1/(R_1+R_2)$. The output will switch from V_{OL} to V_{OH} because $v_+ - v_-$ has just become positive. The capacitor voltage begins to increase, and is given by:

$$v_C(t) = V_{OH} + \left(V_{OL}\frac{R_1}{R_1+R_2} - V_{OH}\right)e^{-\frac{t}{RC}} \qquad t \geq 0$$

Substitution of $t=0$ shows that the above equation indeed satisfies the initial condition $v_C(0) = V_{OL}R_1/(R_1+R_2)$. When $t \to \infty$, we obtain $\lim_{t\to\infty} v_C(t) = V_{OH}$. So, the capacitor voltage begins to increase toward V_{OH}, reaching $v_+ = V_{OH}R_1/(R_1+R_2)$ at time t_1. Solving the above equation for this condition, one gets:

$$t_1 = -RC\ln\left[\frac{V_{OH}R_2}{V_{OH}(R_1+R_2) - V_{OL}R_1}\right]$$

L8.14

At this point $v_+ - v_-$ changes sign and v_C begins to decrease, now governed by the equation:

$$v_C(t) = V_{OL} + \left(V_{OH} \frac{R_1}{R_1 + R_2} - V_{OL} \right) e^{-\frac{t-t_1}{RC}} \qquad t \ge t_1$$

At time t_2, v_C reaches $v_C(t_2) = V_{OL} R_1 / (R_1 + R_2)$. Solving the above equation for this condition, one gets:

$$t_2 - t_1 = -RC \ln \left[\frac{V_{OL} R_2}{V_{OL}(R_1 + R_2) - V_{OH} R_1} \right]$$

The period of the output waveform is just $T_0 = t_2$. Therefore we have:

$$\begin{aligned} T_0 &= (t_2 - t_1) + t_1 \\ &= -RC \ln \left[\frac{V_{OL} R_2}{V_{OL}(R_1 + R_2) - V_{OH} R_1} \right] - RC \ln \left[\frac{V_{OH} R_2}{V_{OH}(R_1 + R_2) - V_{OL} R_1} \right] \\ &= -RC \ln \left[\frac{V_{OL} R_2}{V_{OL}(R_1 + R_2) - V_{OH} R_1} \cdot \frac{V_{OH} R_2}{V_{OH}(R_1 + R_2) - V_{OL} R_1} \right] \end{aligned}$$

In the *special* case of $R_1 = R_2$ and $V_{OL} = -V_{OH}$, the above equation simplifies to a function of only R and C:

$$T_0 = RC \ln 9 \approx 2.2RC$$

In the Lab – Astable Multivibrator

1. Build the astable multivibrator shown below. Use a ±15 V supply.

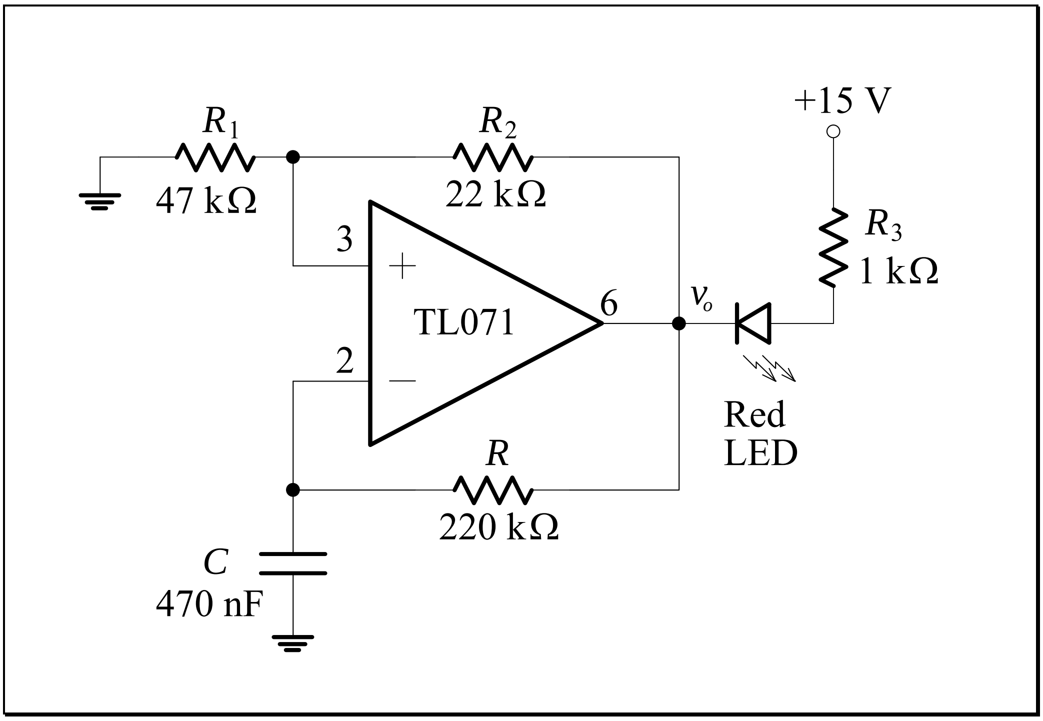


Figure L8.9

2. Calculate the oscillation period using the values of V_{OL} and V_{OH} measured previously.

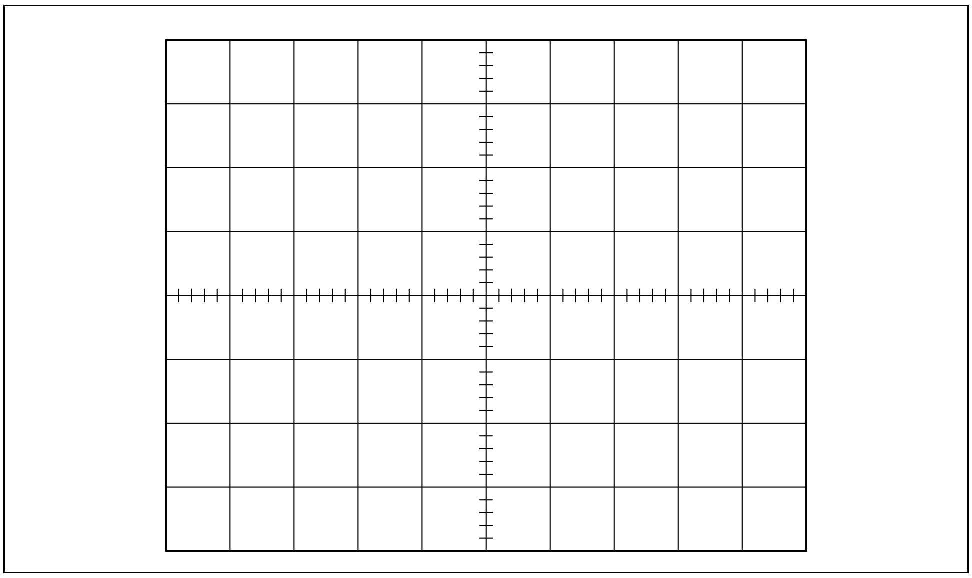
$T_0 =$

3. Measure the oscillation period using the DSO.

$T_0 =$
Compare with the calculated estimate:

Astable multivibrator waveforms

4. Display the output voltage, v_o, and the capacitor voltage, v_C, simultaneously on the DSO. Sketch the waveforms on the plot below:



Note: Measuring the capacitor voltage, v_C, with a DSO will cause the frequency of the output waveform to change. This is because the DSO input (and the lead) have a finite impedance – you can read the front of the DSO to see that each channel has a $1\,\mathrm{M\Omega} \,\|\, 14\,\mathrm{pF}$ input impedance. When you place the DSO lead in parallel with the circuit's capacitor, *C*, you are changing the impedance between pin 2 and ground. This is an occasional problem in electronics (especially so at high frequencies), where the measuring equipment can affect the circuit behaviour. In such cases, use of an "active probe" is required – an active probe has an extremely high input impedance amplifier built inside the probe tip.

You may like to see the effect of the DSO measurement by disconnecting the lead taking the v_C measurement from your circuit, and observing the change in the frequency of the output square wave.

5. Replace R and C with new components: $R = 100\,\text{k}\Omega$ and $C = 10\,\text{nF}$.

6. Calculate the oscillation period using the values of V_{OL} and V_{OH} measured previously.

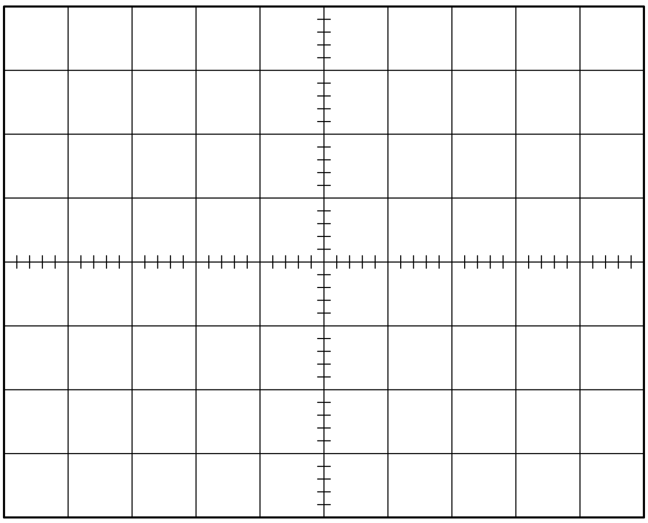
$T_0 =$

7. Measure the oscillation period using the DSO.

$T_0 =$

Compare with the calculated estimate:

8. Display the output voltage, v_o, and the capacitor voltage, v_C, simultaneously on the DSO. Sketch the waveforms on the plot below:

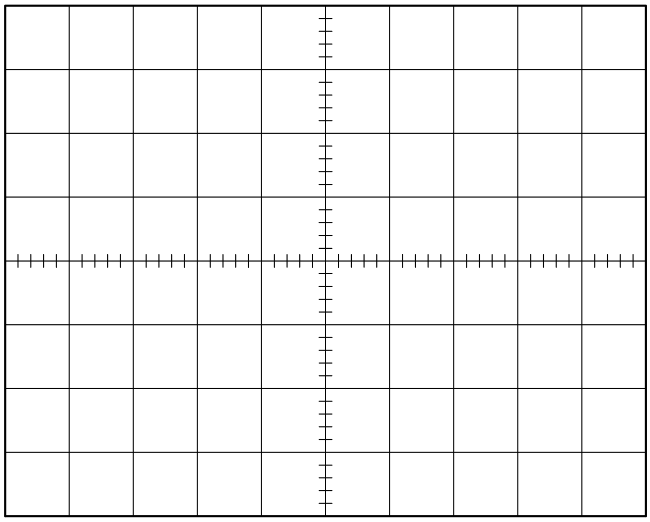


Astable multivibrator waveforms

Why does the LED always appear to be on?

Astable multivibrator output waveform

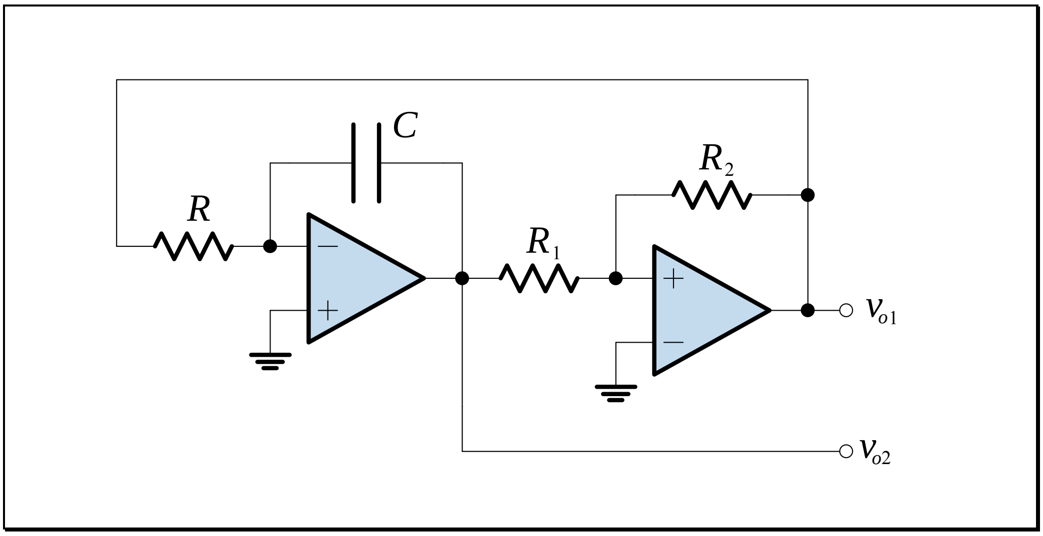
9. Observe the output waveform, v_o, on the DSO with a $1\,\mu s$ time/div setting, so you can examine the transition from a negative to a positive voltage. Sketch the waveform on the plot below:



Is it a true square wave? If not, how do you explain its shape?

Waveform Generator

The exponential waveform (across the capacitor) generated in the astable circuit of Figure L8.8 can be changed to triangular by replacing the lowpass *RC* circuit with an integrator (the integrator is, after all, a lowpass circuit with a corner frequency at DC). The integrator causes linear charging and discharging of the capacitor, thus producing a triangular waveform. The resulting circuit is shown below:



Triangular and rectangular waveform generator

Figure L8.10

This circuit oscillates and generates a square waveform at the output of the noninverting Schmitt trigger, v_{o1}, and a triangular waveform at the output of the inverting integrator, v_{o2}.

Let the output of the bistable circuit be at V_{OH}. A current equal to V_{OH}/R will go into the resistor *R* and then on to the capacitor *C*, causing the output of the integrator to *linearly* decrease with the slope $-V_{OH}/RC$, as shown in Figure L8.11. This will continue until the integrator output reaches the lower threshold, V_{TL}, of the bistable circuit.

L8.20

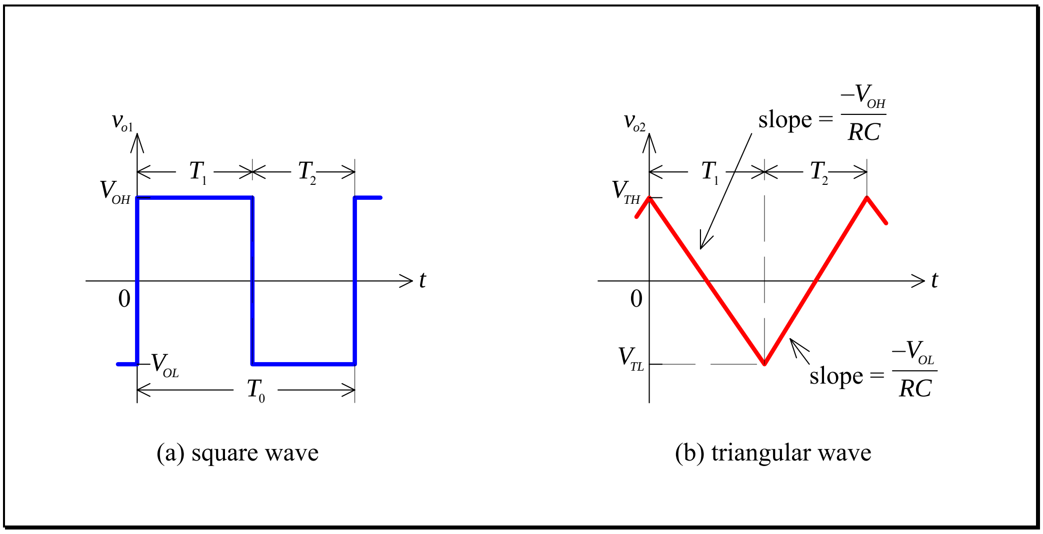


Figure L8.11

At this point the bistable circuit will switch states, its output becoming negative and equal to V_{OL}. At this moment the current through R will reverse direction and its value will become equal to $|V_{OL}|/R$. The output of the integrator will therefore linearly increase with time. This will continue until the integrator output voltage reaches the positive threshold of the Schmitt trigger, V_{TH}. The Schmitt trigger switches states again, starting the new cycle.

From Figure L8.11 it is relatively easy to derive an expression for the period T_0 of the square and triangular waveforms. During the interval T_1 we have:

$$\frac{V_{TH} - V_{TL}}{T_1} = \frac{V_{OH}}{RC} \quad \Rightarrow \quad T_1 = RC\frac{V_{TH} - V_{TL}}{V_{OH}}$$

Similarly, during T_2 we have:

$$\frac{V_{TH} - V_{TL}}{T_2} = \frac{-V_{OL}}{RC} \quad \Rightarrow \quad T_2 = RC\frac{V_{TH} - V_{TL}}{-V_{OL}}$$

Thus, to obtain symmetrical waveforms we need a bistable circuit with $V_{OL} = -V_{OH}$. The oscillation frequency is equal to:

$$f_0 = \frac{1}{T_0} = \frac{1}{T_1 + T_2} = \frac{1}{RC}\frac{V_{OL}V_{OH}}{(V_{TH} - V_{TL})(V_{OL} - V_{OH})}$$

In the Lab – Waveform Generator

1. Build the waveform generator shown below. Note that the integrator has **negative** feedback, whilst the Schmitt trigger has **positive** feedback. Use a ±15 V supply.

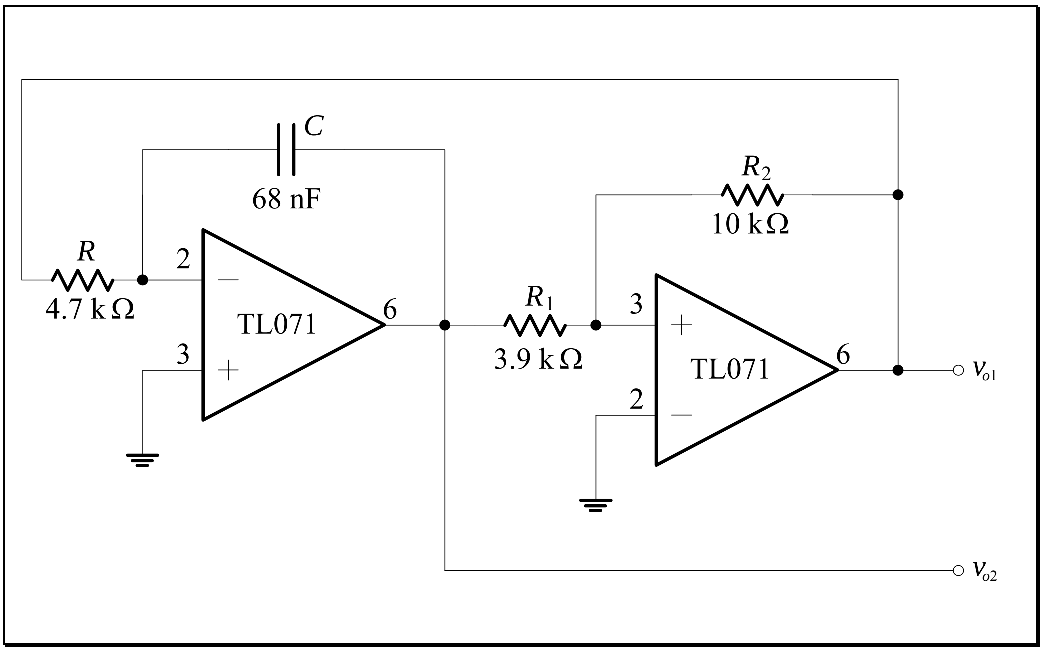


Figure L8.12

2. Calculate the oscillation frequency using the values of V_{OL} and V_{OH} measured previously (note that V_{TL} and V_{TH} are different for this circuit).

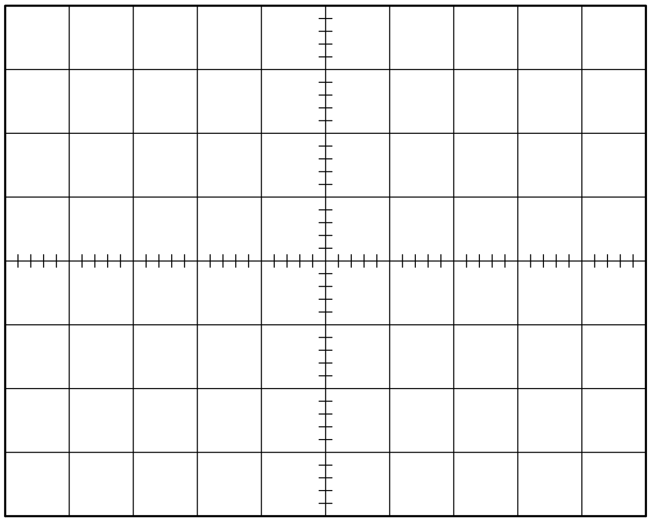
$f_0 =$

3. Measure the oscillation frequency using the DSO.

$f_0 =$
Compare with the calculated estimate:

4. Display the two output voltages, v_{o1} and v_{o2} on the DSO. Sketch the waveforms on the plot below:

Waveform generator waveforms



Explain the waveforms:

Lab Assessment [2 marks]

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

1. Show the result of the voltage transfer characteristic of the open-loop comparator.
2. Show the result of the voltage transfer characteristic of the Schmitt trigger.
3. Show the result of the measurement of the period of the astable multivibrator, and its comparison to the theoretical value (for $R = 220\ \text{k}\Omega$, $C = 470\ \text{nF}$).
4. Display both of the outputs of the waveform generator on the DSO, measure their frequency, and compare with the theoretical value.

Marking

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

Lab 9 – *RLC* Circuits

Lowpass series RLC *circuits. Step response. Frequency response.*

Introduction

A lowpass series *RLC* circuit is shown below:

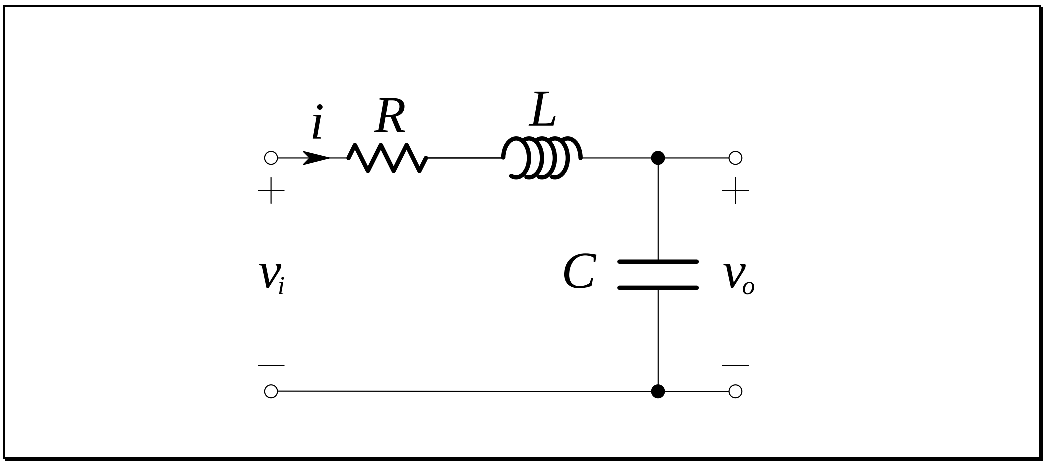


Figure L9.1

The describing differential equation is obtained by performing KVL around the circuit:

$$Ri + L\frac{di}{dt} + v_o = v_i \tag{L9.1}$$

Substituting $i = C\frac{dv_o}{dt}$ we get:

$$LC\frac{d^2 v_o}{dt^2} + RC\frac{dv_o}{dt} + v_o = v_i \tag{L9.2}$$

Dividing through by *LC*, we have the second-order differential equation that describes the circuit:

$$\frac{d^2 v_o}{dt^2} + \frac{R}{L}\frac{dv_o}{dt} + \frac{1}{LC}v_o = \frac{v_i}{LC} \tag{L9.3}$$

L9.2

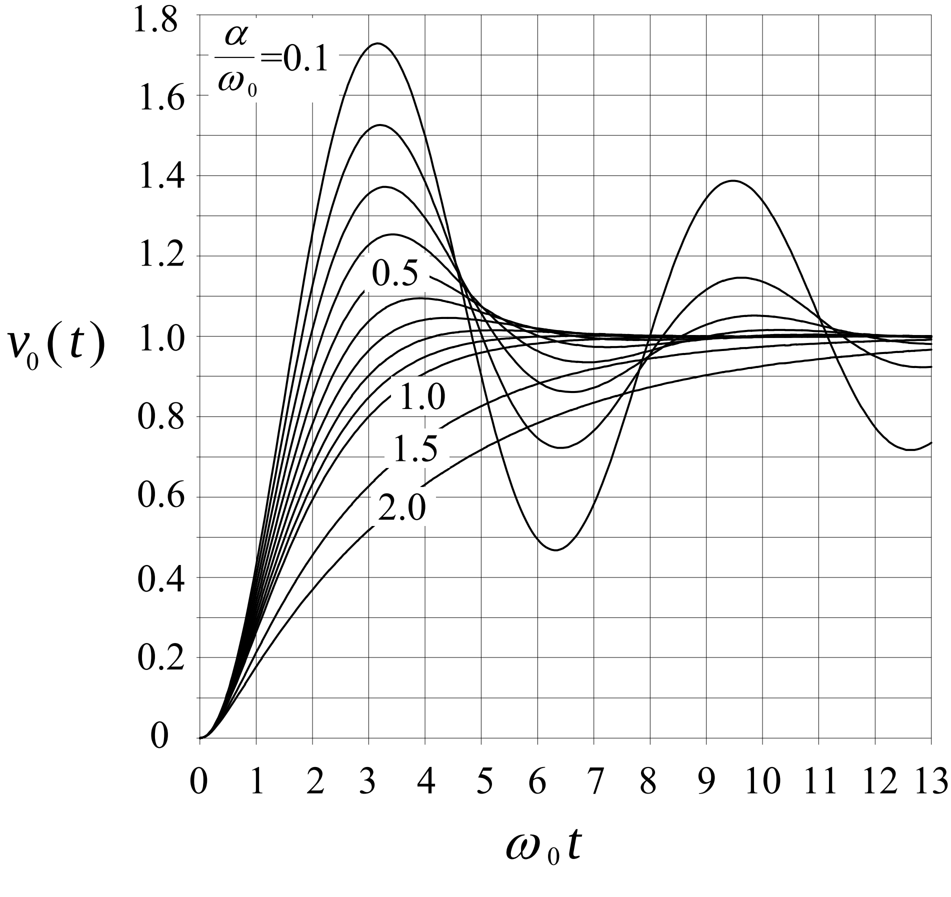
We normally let:

$$\omega_0 = \frac{1}{\sqrt{LC}} \qquad \text{and} \qquad \alpha = \frac{R}{2L} \tag{L9.4}$$

so that we can write it in a "standard" form:

$$\frac{d^2 v_o}{dt^2} + 2\alpha \frac{dv_o}{dt} + \omega_0^2 v_o = \frac{v_i}{LC} \tag{L9.5}$$

The solution of this equation for a step-input gives the *step response*. It is a very important response because many practical systems can be modelled by a second-order system (or made to be approximately a second-order system). The step-response has three different forms, depending on whether the system is overdamped, critically damped, or underdamped. Example step-responses are shown below:



This particular series second-order circuit is of the *lowpass* type. To see the reasoning behind this classification, observe that the frequency response of the circuit can be expressed as a voltage-divider ratio, with the divider composed of a capacitor and a series combination of a resistor and an inductor. Now, recalling how the impedance of a capacitor varies with frequency ($\mathbf{Z} = 1/j\omega C$) it is easy to see that the voltage output of the circuit will decrease with frequency and approach zero as ω approaches ∞. Thus the circuit acts as a **lowpass filter**; it passes low-frequency sine-wave inputs with little or no attenuation and attenuates high-frequency input sinusoids.

Objectives

1. To investigate the step response of a lowpass *RLC* circuit.

2. To investigate the frequency response of a lowpass *RLC* circuit.

Equipment

- 1 Digital Storage Oscilloscope (DSO) – Agilent 54621A
- 1 Mini-Lab – BWD 604
- Resistors – 1 x 20 Ω, 1 x 56 Ω, 1 x 75 Ω, 1 x 130 Ω
  Capacitor – 1 x 470 nF
  Inductor – 1 x 680 μH
  Breadboard, Hook-up wire, 2 x 4 mm leads.

Note

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.

Quality!!!

Safety

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

Cat. A lab

L9.4

Pre-Lab Work

Lowpass Series *RLC* Circuit

Step Response

Lowpass series *RLC* circuit

For the circuit:

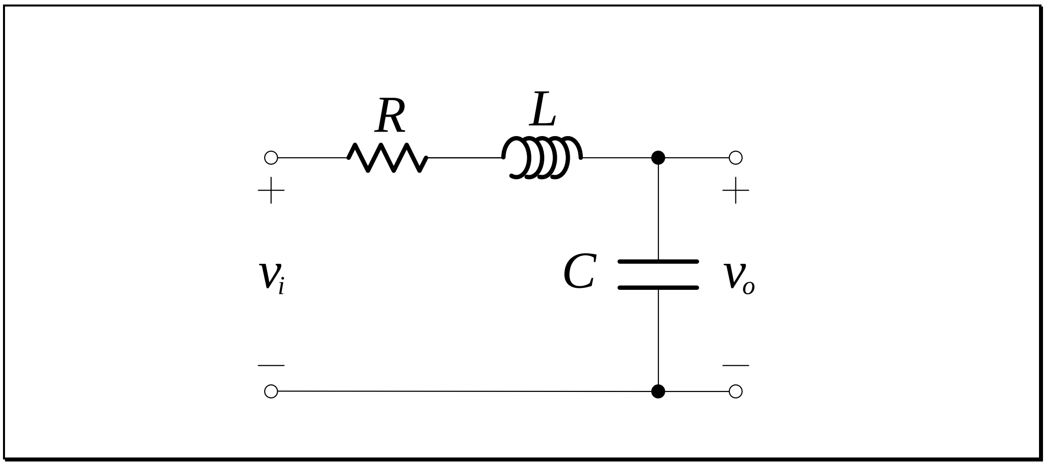


Figure L9.2

let:

$$\omega_0 = \frac{1}{\sqrt{LC}}, \quad \alpha = \frac{R}{2L} \quad \text{and} \quad \omega_d = \sqrt{\omega_0^2 - \alpha^2}$$

1. Determine the characteristic equation in terms of α and ω_0:

2. Determine the forced response of the system for a unit-step input:

3. Write down the *form* of the natural response of $v_0(t)$, for the following cases (do **not** evaluate arbitrary constants):

Overdamped $(\alpha > \omega_0)$:

Critically damped $(\alpha = \omega_0)$:

Underdamped $(\alpha < \omega_0)$:

Frequency Response

Lowpass series *RLC* circuit

For the circuit:

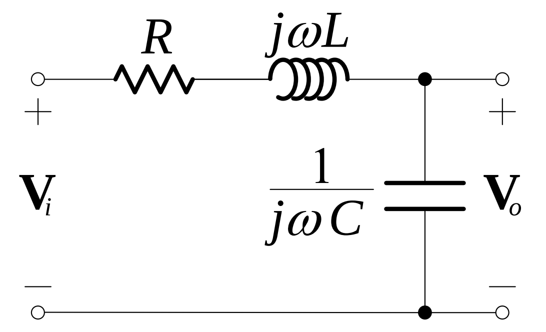


Figure L9.3

let:

$$\omega_0 = \frac{1}{\sqrt{LC}}, \quad Q_0 = \frac{\omega_0 L}{R} \quad \text{and} \quad \alpha = \frac{\omega_0}{2Q_0}$$

1. Determine the characteristic equation in terms of ω_0 and Q_0:

2. Derive an expression for the frequency response in terms of ω_0 and Q_0:

$$\mathbf{H}(j\omega) = \frac{\mathbf{V}_o}{\mathbf{V}_i} \tag{L9.6}$$

$\mathbf{H}(j\omega) =$ $\lvert\mathbf{H}(j\omega)\rvert =$ $\angle\mathbf{H}(j\omega) =$

3. Let $L = 680\,\mu\text{H}$ and $C = 470\,\text{nF}$ and complete the following tables:

$R = 130\,\Omega$ $\boxed{Q_0 = \quad}$

Desired Frequency f_{desired} (kHz)	Frequency ω (krads^{-1})	Gain $\dfrac{\lvert\mathbf{V}_o\rvert}{\lvert\mathbf{V}_i\rvert}$ (V/V)	Gain $20\log_{10}\dfrac{\lvert\mathbf{V}_o\rvert}{\lvert\mathbf{V}_i\rvert}$ (dB)	Phase $\angle\dfrac{\mathbf{V}_o}{\mathbf{V}_i}$ (°)
0.1	0.628	0.9994	-0.01	-2.2
0.2	1.257	0.9976	-0.02	-4.4
0.5	3.142			
1	6.283	0.9439	-0.50	-21.2
2	12.57	0.8189	-1.74	-39.0
5	31.42	0.4907	-6.18	-70.4
10	62.83			
20	125.7	0.1152	-18.77	-117.8
50	314.2	0.02772	-31.14	-147.9
100	628.3	0.007638	-42.34	-162.9
200	1257	0.001963	-54.14	-171.3
500	3142	0.0003165	-69.99	-176.5

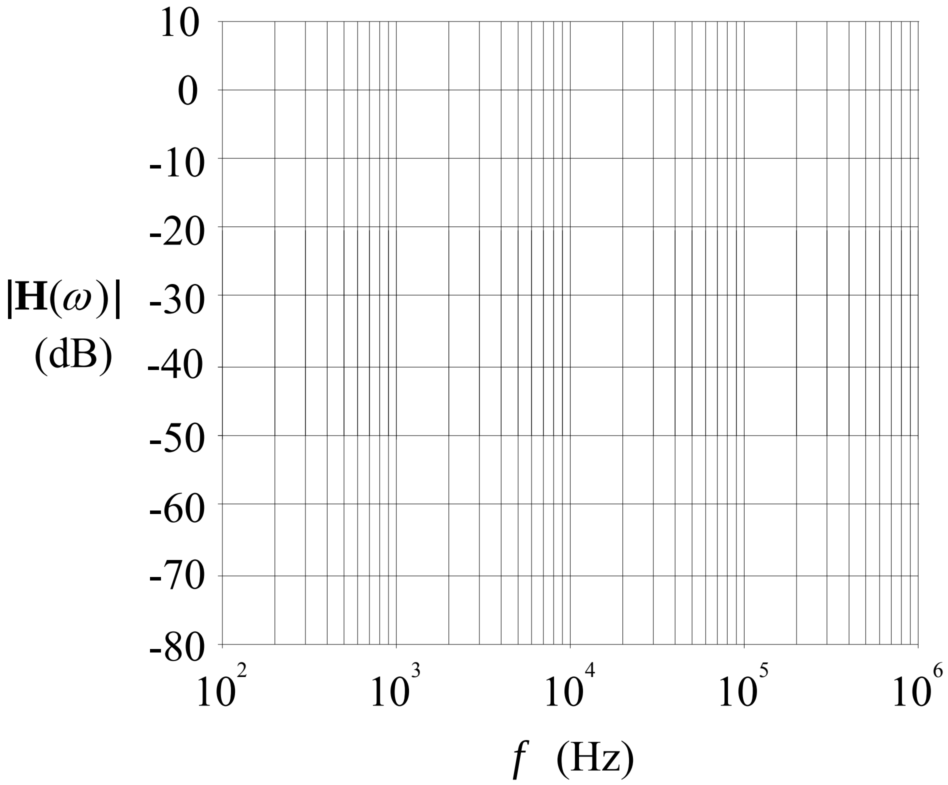
Table L9.1

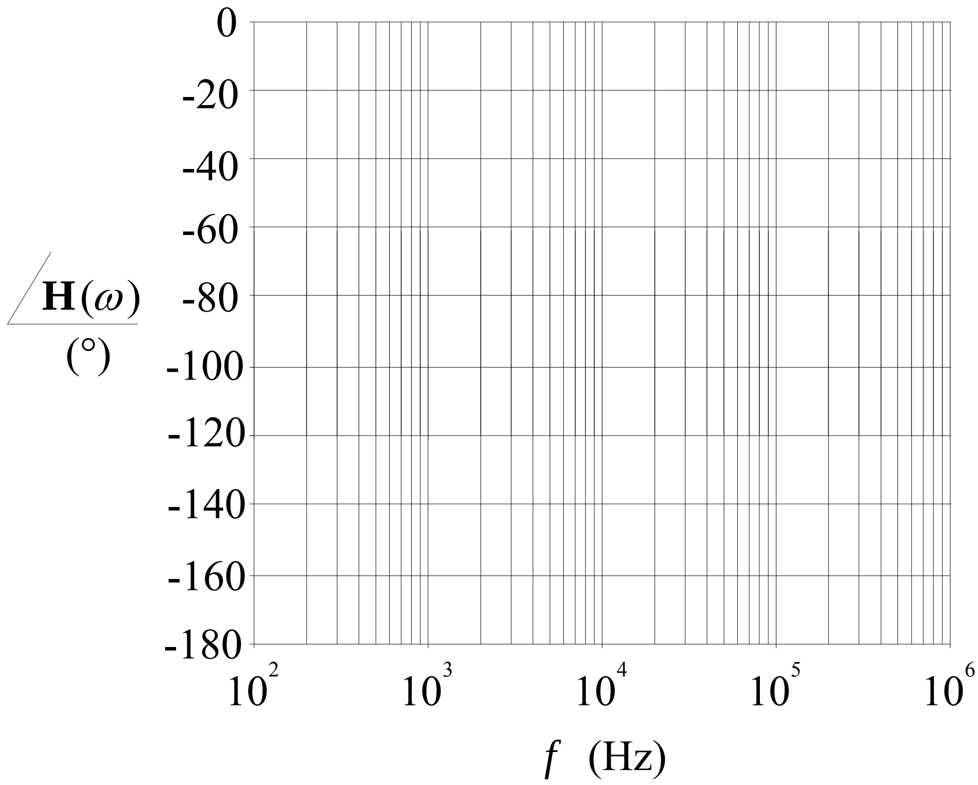
$R = 20\,\Omega$ $\boxed{Q_0 = \quad}$

Desired Frequency f_{desired} (kHz)	Frequency ω (krads^{-1})	Gain $\dfrac{\lvert\mathbf{V}_o\rvert}{\lvert\mathbf{V}_i\rvert}$ (V/V)	Gain $20\log_{10}\dfrac{\lvert\mathbf{V}_o\rvert}{\lvert\mathbf{V}_i\rvert}$ (dB)	Phase $\angle\dfrac{\mathbf{V}_o}{\mathbf{V}_i}$ (°)
0.1	0.628	1.000	0.001	-0.3
0.2	1.257	1.000	0.004	-0.7
0.5	3.142	1.003	0.024	-1.7
1	6.283	1.011	0.095	-3.4
2	12.57	1.045	0.383	-7.1
5	31.42	1.341	2.55	-23.3
10	62.83			
20	125.7	0.2372	-12.5	-163.7
50	314.2	0.03259	-29.7	-174.5
100	628.3	0.007980	-42.0	-177.3
200	1257			
500	3142	0.0003171	-70.0	-179.5

Table L9.2

4. Plot the gain and phase values from the tables on the Bode plots below:





L9.10

Circuit Simulation

1. For the circuit:

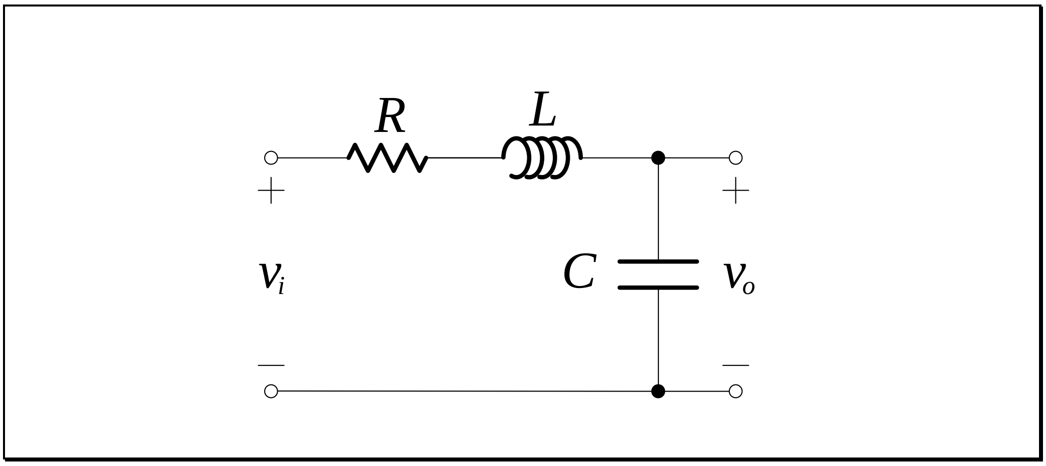


Figure L9.4

with $L = 680\,\mu\text{H}$ and $C = 470\,\text{nF}$, conduct a simulation of the step response and frequency response using PSpice (e.g. OrCAD Demo) for the following values:

(a) $R = 130\,\Omega$

(b) $R = 75\,\Omega$

(c) $R = 56\,\Omega$

(d) $R = 20\,\Omega$

The best approach is to make one series RLC circuit, then copy it 3 more times and change the resistor values and net labels so that all results can be graphed simultaneously.

2. Bring print-outs of the step responses and frequency responses to the lab.

Use the following axes for graphing:

Response	**X**	**Y**
Step	0 s to 500 μs	0 V to 1.5 V
Magnitude	100 Hz to 1 MHz	-80 dB to 10 dB
Phase	100 Hz to 1 MHz	-180° to 0°

Lab Work

Lowpass Series *RLC* Circuit

Mini-Lab Amplifier Setup

The *RLC* circuit, when subjected to a step input, requires a fairly large current to be delivered from the input in a short span of time. The function generator has an output resistance of $50\,\Omega$ and so the output will therefore experience a significant internal *Ri* voltage drop, resulting in a "droop" in the applied voltage when delivering current. We therefore need to "buffer" the output of the function generator. The Mini-lab provides us with a way to do this.

1. Identify the section under the power switch labelled "AMPLIFIER OR BI-POLAR POWER SUPPLY".


Mini-lab Amplifier

2. Ensure that the left-most pushbutton is out (F. GEN) so that the internal function generator is selected as the input.
3. Ensure that the middle pushbutton is out (NORM) so that the output is normal.
4. Ensure that the right-most pushbutton is out (AMP) so that the unit acts as an amplifier.
5. Ensure that the knob is fully rotated counter-clockwise to select a gain of "X 1".
6. With these settings the buffered output of the function generator can be taken directly from the red output terminal.

L9.12

Step Response

1. Using the DMM, measure and record the DC series equivalent resistance of your inductor:

$R_L =$

2. Construct the following circuit, using $R = 130\,\Omega$:

Series *RLC* circuit with a real inductor

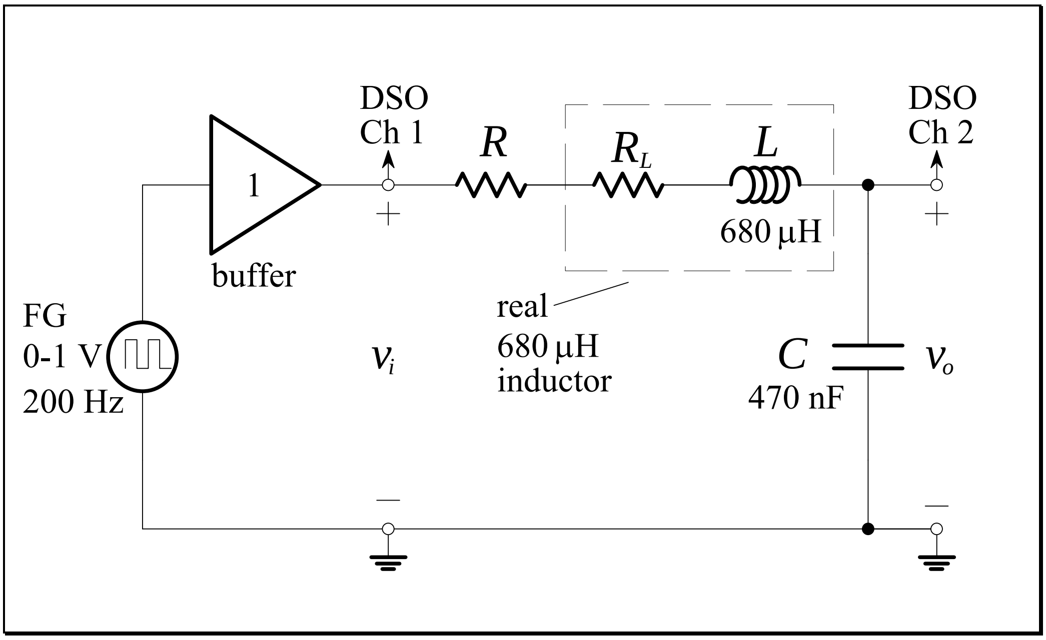


Figure L9.5

Measurement of the step response of a circuit is conducted with a square wave!

The input is a 0-1 V **square wave** (i.e. 0 V to +1 V) with a frequency of 200 Hz. Note that the output of the Mini-lab amplifier is the "buffered" function generator.

3. Connect the input to Channel 1 of the DSO and the output to Channel 2 of the DSO.

4. Press `Main/Delayed`. Change the `Time Ref` softkey to Left. This will facilitate sketching the step-response.

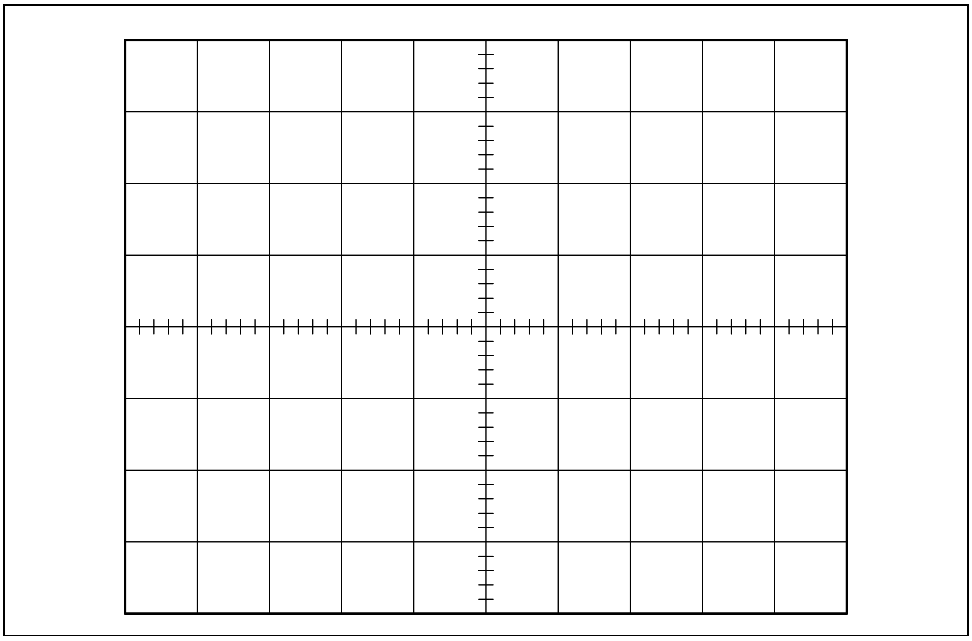
5. Set the DSO horizontal time base to display the positive-going step input of the square wave with a time scale of 50 µs / div.

6. Set the vertical scale of each DSO channel to 200 mV / div, and adjust the position of the channels so that 0 volts lies one division from the bottom.

7. Ensure that each input to the DSO is *bandwidth limited* and use *waveform averaging* to obtain a display with the least amount of noise.

8. Draw the input (Ch 1) waveform on the following graph, ensuring that the sketch is labelled with voltage and time scales.

9. Draw the output (Ch 2) waveform, and label it clearly.



10. We will now save this waveform on the DSO display:

    (a) Press the `Save/Recall` key in the "File" section.

    (b) Press the `Save` softkey.

    (c) Adjust the `To:` parameter in the leftmost softkey to `INTERN_n`. (where `n` will increment from 0 to 2). This is an internal memory location.

    (d) Press the `Press to Save` softkey.

    (e) Press the `Save/Recall` key.

    (f) Press the `Recall` softkey.

    (g) Change the `Recall` parameter in the leftmost softkey to `Trace`.

    (h) Change the `From` parameter to `INTERN_n` (the most recently used memory location).

    (i) Press the `Press to Recall` softkey. The step response has been saved and is displayed at low intensity in the background.

11. Repeat Steps 9 and 10 for the following values of R:

    $R = 75\,\Omega$

    $R = 56\,\Omega$

12. Repeat Step 9 for the following value of R:

    $R = 20\,\Omega$

13. We will now clear all the displayed waveforms on the DSO display:

    (a) Press the `Save/Recall` key in the “File” section.

    (b) Press the `Recall` softkey.

    (c) Press the `Clear Display` softkey.

14. For the last response with $R = 20\,\Omega$, use the DSO to measure:

Damped natural frequency:

$\omega_d =$

Peak time:

$t_p =$

Peak voltage:

$v_p =$

Questions – Lowpass *RLC* Step Response

With reference to the circuit of Figure L9.5:

1. Determine the theoretical undamped natural frequency:

$\omega_0 =$

2. For each resistor value, determine whether the circuit is overdamped, critically damped or underdamped.

Resistor Value	α	**Damping Type**
$R = 130\,\Omega$		
$R = 75\,\Omega$		
$R = 56\,\Omega$		
$R = 20\,\Omega$		

3. For the case $R = 20\,\Omega$, determine the theoretical values:

Damped natural frequency:

$$\omega_d = \sqrt{\omega_0^2 - \alpha^2} =$$

Peak time:

$$t_p = \frac{\pi}{\omega_d} =$$

Peak voltage:

$$v_p = 1 + e^{-\alpha t_p} =$$

Comment on the agreement (or otherwise) with the experimental results:

Frequency Response

1. Construct the following circuit, using $R = 130\,\Omega$:

Series *RLC* circuit with a real inductor

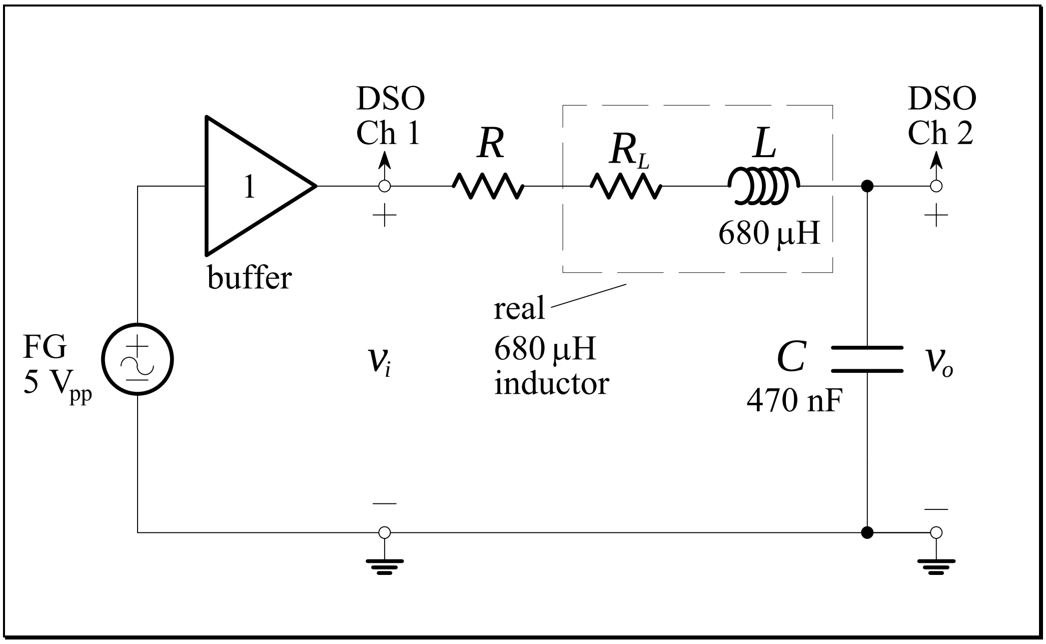


Figure L9.6

Measurement of the frequency response of a circuit is conducted with a sine wave!

The input is a 5 V peak-to-peak **sine wave** (i.e. -2.5 V to +2.5 V) with a variable frequency. Note that the output of the Mini-lab amplifier is the “buffered” function generator.

2. Connect the input to Channel 1 of the DSO and the output to Channel 2 of the DSO.

3. Reset the positions of the DSO channels on the display so they are centred at 0 V.

   In taking a frequency response of a circuit, the fastest measuring technique is to set the frequency vernier to a desired frequency, such as 100 Hz, then simply change the FG frequency range to get the 1 kHz reading, then the 10 kHz reading etc.

4. Complete the following table:

$R = 130\,\Omega$ $\boxed{Q_0 =\qquad}$

Desired Frequency f_{desired} (kHz)	Actual Frequency f_{actual} (kHz)	Gain $\dfrac{\lvert\mathbf{V}_o\rvert}{\lvert\mathbf{V}_i\rvert}$ (V/V)	Gain $20\log_{10}\dfrac{\lvert\mathbf{V}_o\rvert}{\lvert\mathbf{V}_i\rvert}$ (dB)	Phase $\angle\dfrac{\mathbf{V}_o}{\mathbf{V}_i}$ (°)
0.1				
0.2				
0.5				
1				
2				
5				
10				
20				
50				
100				
200				

Table L9.3

5. Set $R = 75\,\Omega$ and complete the following table:

$R = 75\,\Omega$ $\boxed{Q_0 =\qquad}$

Desired Frequency f_{desired} (kHz)	Actual Frequency f_{actual} (kHz)	Gain $\dfrac{\lvert\mathbf{V}_o\rvert}{\lvert\mathbf{V}_i\rvert}$ (V/V)	Gain $20\log_{10}\dfrac{\lvert\mathbf{V}_o\rvert}{\lvert\mathbf{V}_i\rvert}$ (dB)	Phase $\angle\dfrac{\mathbf{V}_o}{\mathbf{V}_i}$ (°)
0.1				
0.2				
0.5				
1				
2				
5				
10				
20				
50				
100				
200				

Table L9.4

6. Set $R = 56\,\Omega$ and complete the following table:

$R = 56\,\Omega$ $\boxed{Q_0 = \quad}$

Desired Frequency f_{desired} (kHz)	Actual Frequency f_{actual} (kHz)	Gain $\frac{\lvert\mathbf{V}_o\rvert}{\lvert\mathbf{V}_i\rvert}$ (V/V)	Gain $20\log_{10}\frac{\lvert\mathbf{V}_o\rvert}{\lvert\mathbf{V}_i\rvert}$ (dB)	Phase $\angle\frac{\mathbf{V}_o}{\mathbf{V}_i}$ (°)
0.1				
0.2				
0.5				
1				
2				
5				
10				
20				
50				
100				
200				

Table L9.5

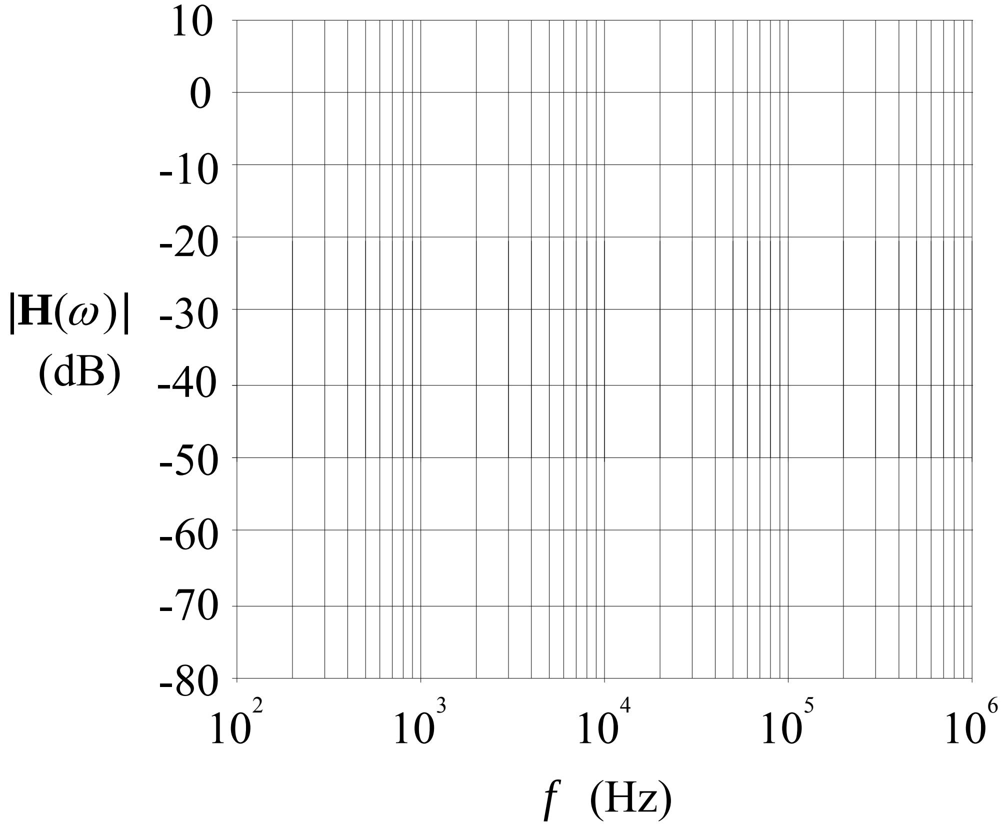
7. Set $R = 20\,\Omega$ and complete the following table. The frequency f_p is the frequency, obtained experimentally, at which the output is a **maximum**.

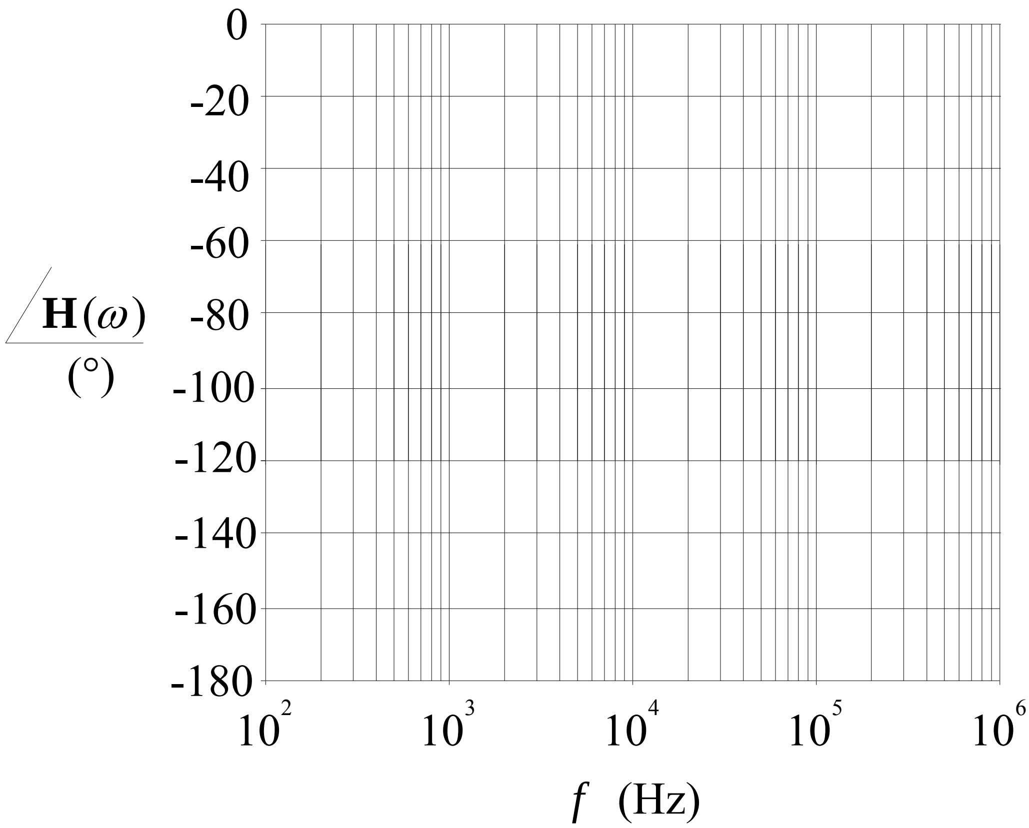
$R = 20\,\Omega$ $\boxed{Q_0 = \quad}$

Desired Frequency f_{desired} (kHz)	Actual Frequency f_{actual} (kHz)	Gain $\frac{\lvert\mathbf{V}_o\rvert}{\lvert\mathbf{V}_i\rvert}$ (V/V)	Gain $20\log_{10}\frac{\lvert\mathbf{V}_o\rvert}{\lvert\mathbf{V}_i\rvert}$ (dB)	Phase $\angle\frac{\mathbf{V}_o}{\mathbf{V}_i}$ (°)
0.1				
0.2				
0.5				
1				
2				
5				
10				
20				
50				
100				
200				
$f_p =$				

Table L9.6

8. Plot the gain and phase values from the four tables on the Bode plots:





L9.20

Questions – Lowpass *RLC* Frequency Response

1. For $R = 20\,\Omega$, use the total resistance of the circuit and the nominal values of L and C to calculate the undamped natural frequency, f_0, and quality factor, Q_0, of the circuit:

$$f_0 = \frac{1}{2\pi\sqrt{LC}} = \qquad\qquad Q_0 = \frac{1}{R_T}\sqrt{\frac{L}{C}} =$$

2. For $R = 20\,\Omega$, compute the theoretical frequency for which the output is a maximum, and compare it with the frequency obtained experimentally:

Theoretical:

$$f_p = f_0\sqrt{1-\frac{1}{2Q_0^2}} =$$

Experimental:

$$f_p =$$

Comment:

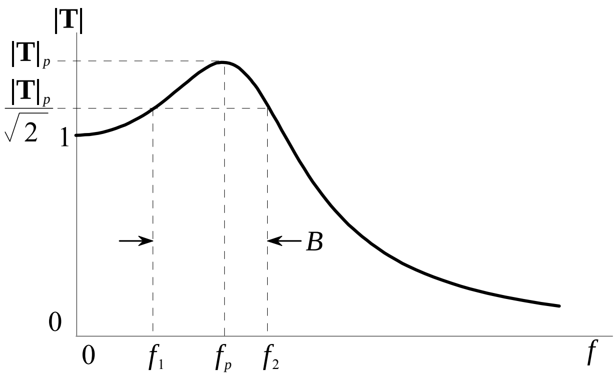
3. For each resistor value, estimate the -3 dB bandwidth of the circuit from the experimental results:

Estimate by using linear interpolation between the experimental measurements around the -3 dB point

Resistor Value	**-3 dB Bandwidth**
$R = 130\,\Omega$	
$R = 75\,\Omega$	
$R = 56\,\Omega$	

4. For the case $R = 20\,\Omega$, use the total resistance of the circuit and the nominal values of L and C to compute the bandwidth of the circuit and compare it to that obtained experimentally:

Theoretical:



$$u_{1,2} = 1 - \frac{1}{2Q_0^2} \mp \frac{1}{Q_0}\sqrt{1-\left(\frac{1}{2Q_0}\right)^2} =$$

$f_1 = \sqrt{u_1}\, f_0 =$ $\qquad$ $f_2 = \sqrt{u_2}\, f_0 =$

$B = f_2 - f_1 =$

Experimental:

$v_{op} =$ $\qquad$ $\dfrac{v_{op}}{\sqrt{2}} =$

$f_1 =$

$f_2 =$

$B = f_2 - f_1 =$

Comment:

Lab Assessment [2 marks]

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

1. Show the result of the pre-lab work, **Step Response** section, step 3.

2. Show the result of the pre-lab work, **Circuit Simulation** section, step 2.

3. Demonstrate the measurement of the damped natural frequency of the underdamped lowpass *RLC* circuit step response (use $R = 20\,\Omega$).

4. Show the Bode plots (magnitude and phase) of the lowpass *RLC* circuit for the four resistor values, and demonstrate the measurement of the bandwidth for the case $R = 20\,\Omega$.

Marking

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

Lab 10 – The Universal Filter

The Tow-Thomas biquad. Lowpass filter. Bandpass filter. Highpass filter. Notch filter. The spectrum. Audio filtering application.

Introduction

With the advent of op-amps and circuit miniaturization, engineers developed what is known as a *universal filter*. It’s frequency response takes the form of a biquadratic equation, and so it is also known as a *biquad*. Depending on the connections made and the point at which the output is taken, the universal filter can deliver lowpass, highpass, bandpass, bandstop (notch) and allpass responses. It is one of the most useful circuits to the electrical engineer and is widely available.

Objectives

1. To investigate the frequency response of a biquad circuit acting either as a lowpass filter, a bandpass filter, a highpass filter or as a notch filter.

2. To filter audio signals with a lowpass filter, a bandpass filter, a highpass filter and a notch filter, so as to gain an appreciation of circuit behaviour in the frequency-domain.

3. To observe the spectrum of a signal on a DSO.

Equipment

- 1 Digital Storage Oscilloscope (DSO) – Agilent 54621A
- 1 Mini-Lab – BWD 604
- 1 MP3 player with ear pieces – Dick Smith A8696
- 1 3.5 mm stereo plug – UTS
- 1 3.5 mm stereo socket – UTS
- Op-amp – 3 x TL071
  Resistors – 2 x 8.2 kΩ, 5 x 10 kΩ, 2 x 51 kΩ
  Capacitors – 5 x 10 nF
  Breadboard, Hook-up wire, 4mm leads.

L10.2

Safety

Cat. A lab

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

Warning!

Remember:

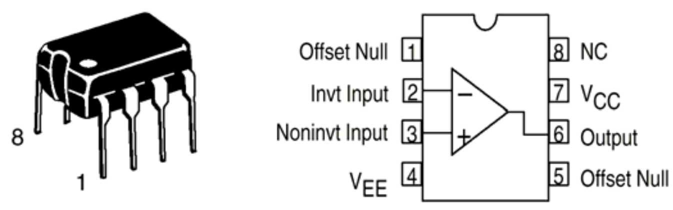
1. **When wiring the circuits, ensure that the power supply is switched off.**

2. **<u>It is very important to place the polarised electrolytic capacitors into the circuit with the correct polarity.</u>**

   <u>Failure to do so will result in the capacitor failing catastrophically which may cause personal injury. If this happens, you will be awarded 0 marks for the lab and asked to leave!</u>

Laboratory Preparation

- The pin-out for the TL071 op-amp is given below:



Op-amp package details

Figure L10.1

- ***For the TL071, pin 7 is connected to the positive supply and pin 4 is connected to the negative supply.***

- It would be a good idea to plan the layout of all the circuits as they will appear on your breadboard before you begin. This will minimise construction time in the lab, and assist in debugging circuits that do not appear to be working.

- A pair of pliers, a pair of wire cutters and a pair of wire strippers would be handy to wire a neat circuit; straighten bent leads; insert components into the breadboard etc. If you have any of these tools, bring them to the lab!

L10.4

The Tow-Thomas Biquad

The normalised Tow-Thomas biquad circuit is:

The normalised Tow-Thomas universal filter

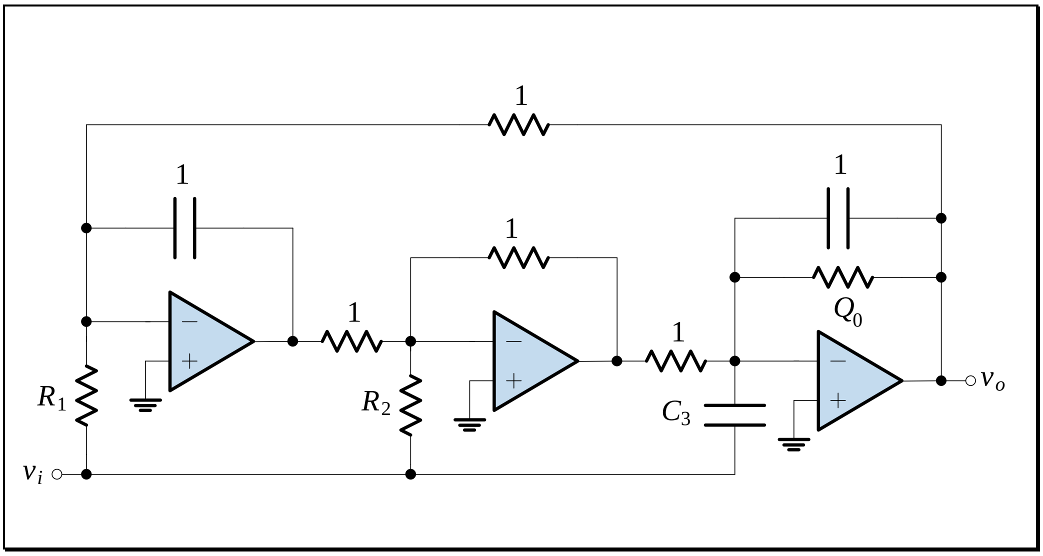


Figure L10.2

The normalised design values for various responses are given in the table below, where H is the passband gain.

Table of design values for a universal filter

Filter Type	Design Values		
	R_1	R_2	C_3
Lowpass	$1/H$	∞	0
Bandpass	∞	Q_0/H	0
Highpass	∞	∞	H
Notch	$\left(\omega_0/\omega_n\right)^2/H$	∞	H

Table L10.1 Design Values for the Tow-Thomas Universal Filter

Pre-Lab Work

The Tow-Thomas Biquad

1. Construct the biquad circuit shown in Figure L10.3. Calculate the correct value for the resistors labelled R to achieve a 2nd-order Butterworth response with a passband gain $H = 1$.

   Choose suitable resistors for their implementation.

   $R =$ kΩ

   Note: It will be beneficial to organise for the input signal to run along a breadboard *rail* (row) to enable the various inputs to be connected and disconnected easily.

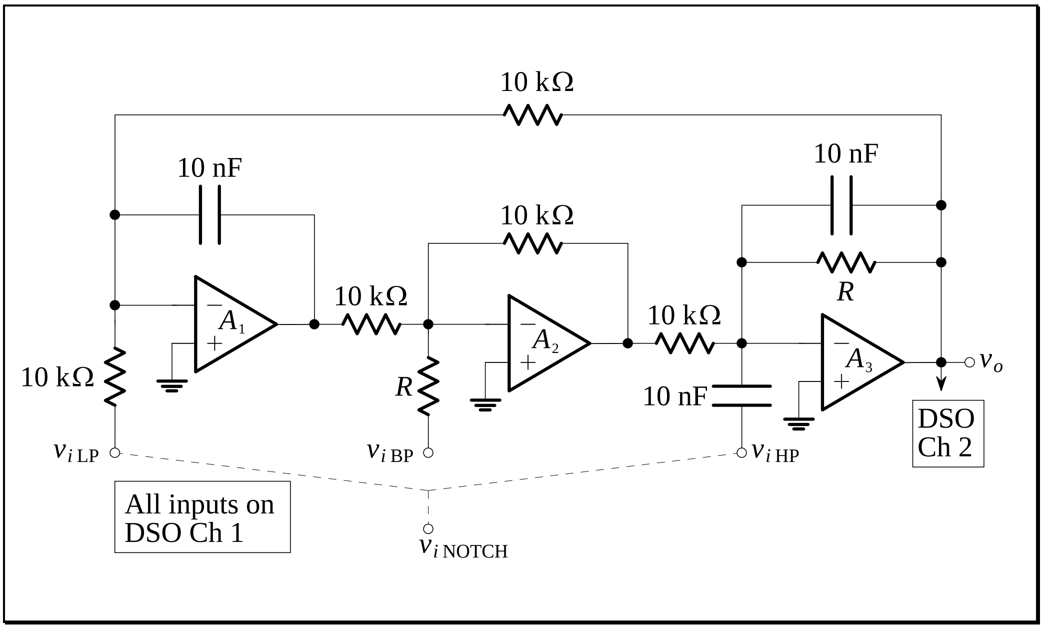


Figure L10.3

Note that the power connections on this circuit are not shown explicitly – connect the TL071's power supply according to the pin-out given in Figure L10.1. Use a ±15 V supply. Make sure you add 10 μF and 10 nF bypass capacitors from each DC supply to the common.

2. Determine the filter's ω_0 and f_0:

   $\omega_0 =$ $f_0 =$

L10.6

Lab Work

In this lab it is a good idea to **test the overall functionality of the filter** before taking precise measurements.

Observe correct circuit behaviour before taking measurements

When required to "check for correct filter operation", do the following:

(a) Set the DSO horizontal time base to 500 μs / div.

(b) Set the vertical scale of each DSO channel to 500 mV / div.

(c) **Invert Channel 2 on the DSO for better observation.**

(d) Set up the function generator to generate a 1V amplitude (2 Vpp) sinusoid using the Mini-Lab's 10 kHz range.

(e) Ensure that waveform averaging is off.

(f) Use the frequency vernier knob to manually sweep the frequency from 0.1 kHz to 10 kHz while visually observing the response of the filter on the DSO.

Once correct circuit operation is achieved, you will be able to take precise measurements.

1. Connect a suitable sinusoid to the input labelled $v_{i\mathrm{LP}}$.

   Check for correct filter operation.

   Measure the frequency response of the lowpass filter:

LOWPASS FILTER

Desired Frequency f_{desired} (kHz)	Actual Frequency f_{actual} (kHz)	Gain $\frac{\lvert\mathbf{V}_o\rvert}{\lvert\mathbf{V}_i\rvert}$ (V/V)	Gain $20\log_{10}\frac{\lvert\mathbf{V}_o\rvert}{\lvert\mathbf{V}_i\rvert}$ (dB)	Phase $\angle\frac{\mathbf{V}_o}{\mathbf{V}_i}$ (°)
0.1				
0.2				
0.5				
1				
2				
5				
10				
20				
50				

Table L10.2

2. Connect a suitable sinusoid to the input labelled $v_{i\mathrm{BP}}$.

   Check for correct filter operation.

   Measure the frequency response of the bandpass filter:

BANDPASS FILTER

Desired Frequency f_{desired} (kHz)	Actual Frequency f_{actual} (kHz)	Gain $\frac{\lvert\mathbf{V}_o\rvert}{\lvert\mathbf{V}_i\rvert}$ (V/V)	Gain $20\log_{10}\frac{\lvert\mathbf{V}_o\rvert}{\lvert\mathbf{V}_i\rvert}$ (dB)	Phase $\angle\frac{\mathbf{V}_o}{\mathbf{V}_i}$ (°)
0.1				
0.2				
0.5				
1				
2				
5				
10				
20				
50				

Table L10.3

3. Connect a suitable sinusoid to the input labelled $v_{i\mathrm{HP}}$.
   Check for correct filter operation.
   Measure the frequency response of the highpass filter:

HIGHPASS FILTER

Desired Frequency f_{desired} (kHz)	Actual Frequency f_{actual} (kHz)	Gain $\frac{\lvert\mathbf{V}_o\rvert}{\lvert\mathbf{V}_i\rvert}$ (V/V)	Gain $20\log_{10}\frac{\lvert\mathbf{V}_o\rvert}{\lvert\mathbf{V}_i\rvert}$ (dB)	Phase $\angle\frac{\mathbf{V}_o}{\mathbf{V}_i}$ (°)
0.1				
0.2				
0.5				
1				
2				
5				
10				
20				
50				

Table L10.4

4. Connect a suitable sinusoid to **both** inputs labelled $v_{i\mathrm{LP}}$ and $v_{i\mathrm{HP}}$.

   Check for correct filter operation.

   Measure the frequency response of the notch filter.

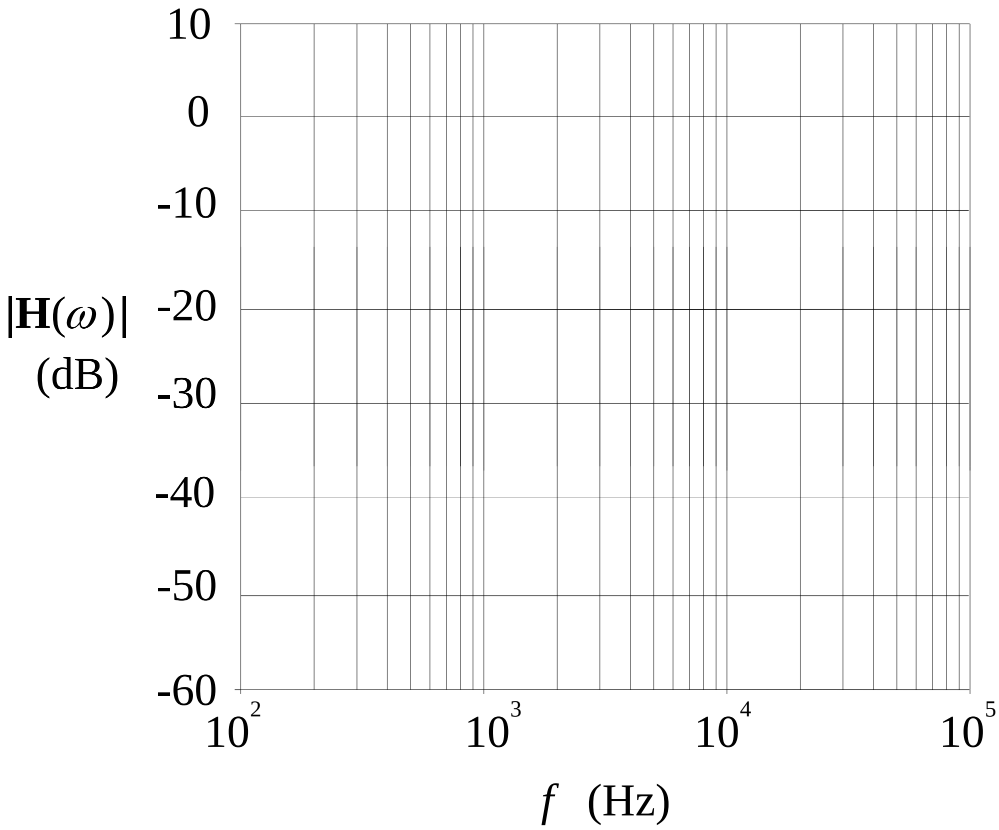
Note: f_n is the frequency, obtained experimentally, for which the output is a **minimum**.

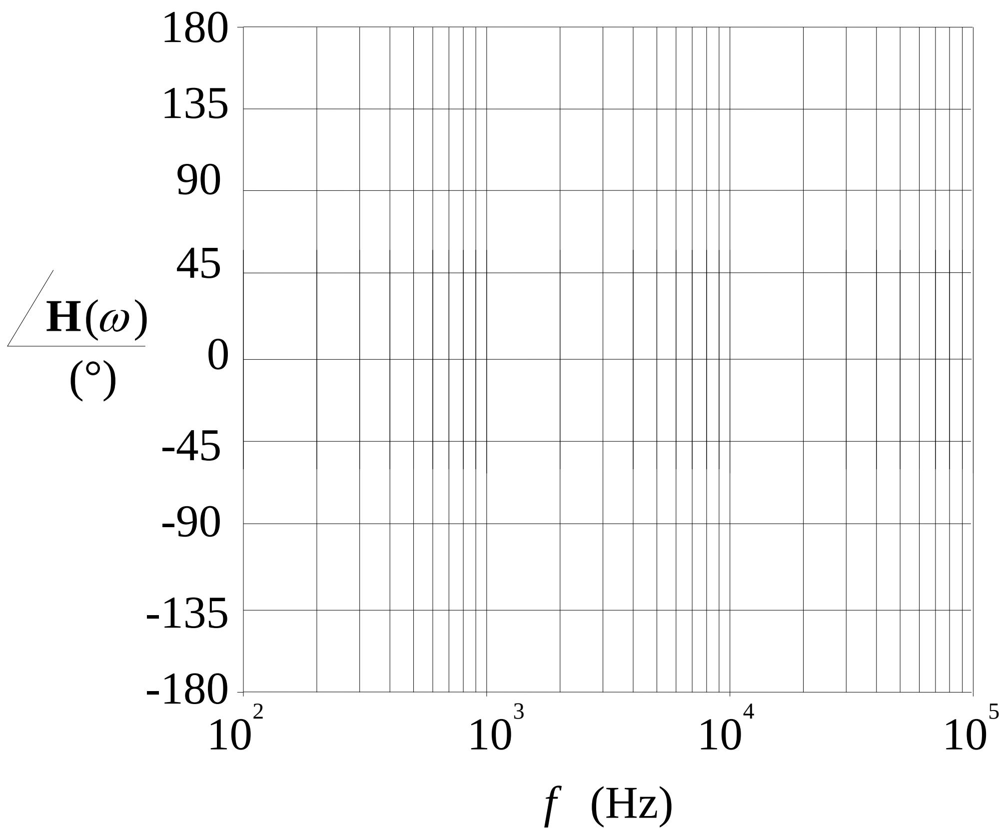
NOTCH FILTER

Desired Frequency f_{desired} (kHz)	Actual Frequency f_{actual} (kHz)	Gain $\frac{\lvert\mathbf{V}_o\rvert}{\lvert\mathbf{V}_i\rvert}$ (V/V)	Gain $20\log_{10}\frac{\lvert\mathbf{V}_o\rvert}{\lvert\mathbf{V}_i\rvert}$ (dB)	Phase $\angle\frac{\mathbf{V}_o}{\mathbf{V}_i}$ (°)
0.1				
0.2				
0.5				
1				
2				
5				
10				
20				
50				
$f_n =$				

Table L10.5

5. Plot the gain and phase values from the four tables on the Bode plots below, and label the responses clearly (LP, BP, HP, notch):





The Spectrum

We are familiar with the fact that light is composed of many different colours, each with a different wavelength. We observe the spectrum of white light when we look at a rainbow or pass light through a prism.

All sounds, including music and voice, are composed of many different sine waves. Normally, when a signal (such as music) is viewed on an oscilloscope, it is viewed such that the vertical axis is voltage and the horizontal axis is time. However, there is another way to observe the same signal. We can observe the "magnitude spectrum" of a signal on the DSO by observing the amplitude of its constituent sine waves (each with a different frequency, amplitude and phase).

With a spectrum, the vertical axis is still voltage but is usually expressed as a relative measurement in dB (e.g. dBV means the signal is expressed as a ratio with respect to 1 V rms). The horizontal axis is frequency, in Hz.

The magnitude spectrum is a graph of the sine wave magnitudes present in a signal, versus frequency

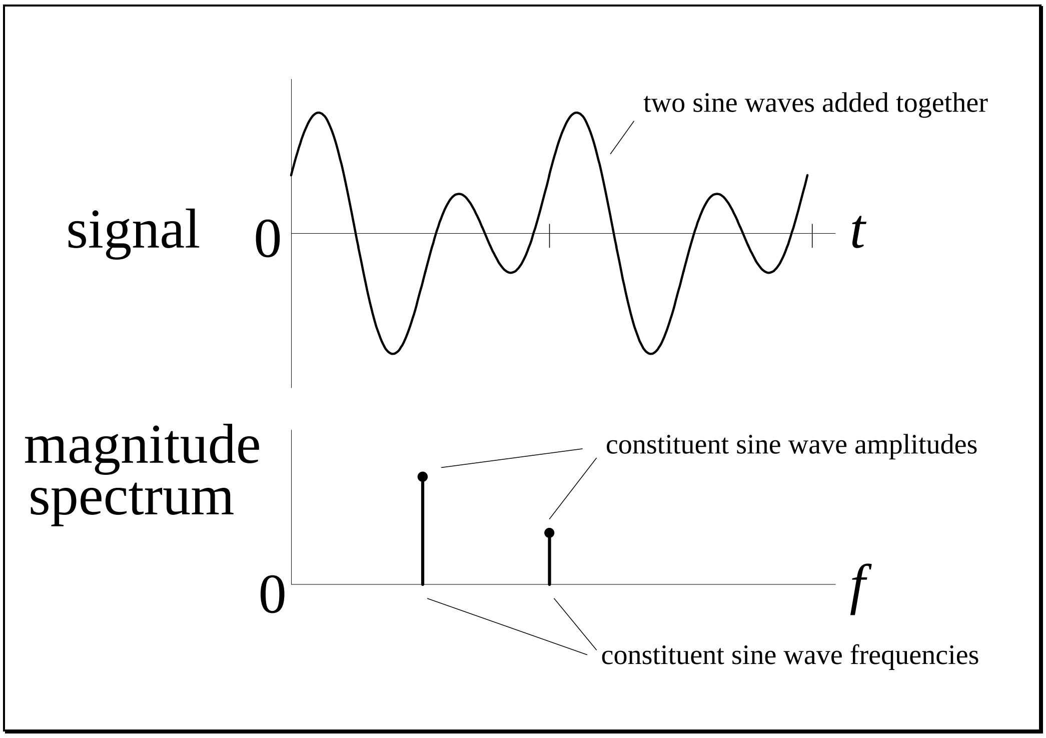


Figure L10.4 – An example spectrum

The Fast Fourier Transform (FFT) is an algorithm that efficiently converts a signal into its spectrum.

Observing a Magnitude Spectrum

1. Set up a 2 V peak-to-peak sine wave at a frequency of 1 kHz (on the Mini-Lab's 10 kHz range) and observe on Channel 1 of the DSO.

2. Push the `1` button so that it is no longer illuminated and the sine wave display turns off.

3. In the "Vertical" section of the DSO, press the `Math` button, press the `FFT` softkey, then press the `Settings` softkey to display the FFT menu.

4. In the "Horizontal" section of the DSO, turn the large knob and watch the display so that "`FFT Sample Rate = 40.0kSa/s`".

5. Press the `Center` softkey, then turn the Entry knob to set a centre frequency of `1.00 kHz`.

6. Press the `Span` softkey, then turn the Entry knob to set a frequency span of `2.00 kHz`.

7. Press the `More FFT` softkey to display additional FFT settings.

8. Press the `Offset` softkey, then turn the Entry knob to set an offset of `-40.0 dBV` (this adjusts the vertical scale of the display).

You should now see a magnitude spectrum similar to the following:

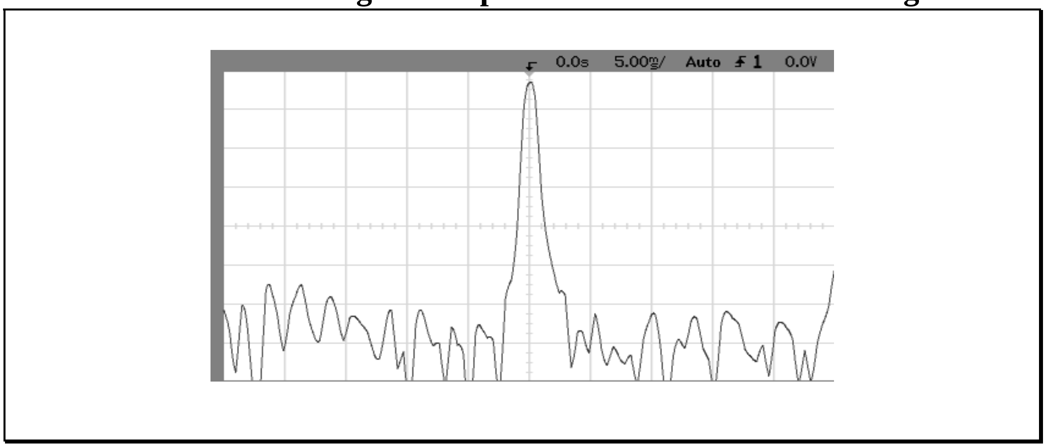


Figure L10.5 – An example magnitude spectrum

Measuring the Magnitude Spectrum

We can measure frequencies in the magnitude spectrum using the cursors.

1. To make cursor measurements on the magnitude spectrum, press the `Cursors` button and set the `Source` softkey to `Math`.

2. The magnitude spectrum looks like a whole series of mountains and valleys, or peaks and troughs, that move up and down. If there is a single and persistent sine wave in the signal, then there should be a dominant and consistent peak in the magnitude spectrum. The rest of the magnitude spectrum is referred to as "noise". Identify the dominant peak of the magnitude spectrum and align the X1 cursor with it.

3. Record the following measurement for the **frequency** of the sine wave, using the value for X1:

$\mathrm{X1} = f =$

4. Press the `Cursors` button to turn off the cursors.

5. Press the `Math` button.

6. Press the `Settings` softkey to display the FFT menu.

7. Press the `Preset` softkey to return the display to a 20 kHz span centred on 10 kHz.

8. Vary the frequency of the FG sinusoid and observe the behaviour of the magnitude spectrum on the DSO. Return the frequency to 1 kHz.

9. Observe the spectrum of a triangle wave. Note that a triangle wave is composed of many discrete sinusoids.

10. Observe the spectrum of a square wave. Note that a square wave is composed of many discrete sinusoids.

Magnitude Spectrum of Audio Signals

We will listen to an audio signal whilst simultaneously observing its spectrum.

1. The 3.5 mm stereo plug and socket:

Color	Use
red	signal
black	common
white	unused

will be used to connect the MP3 player to the breadboard. Construct the following system:

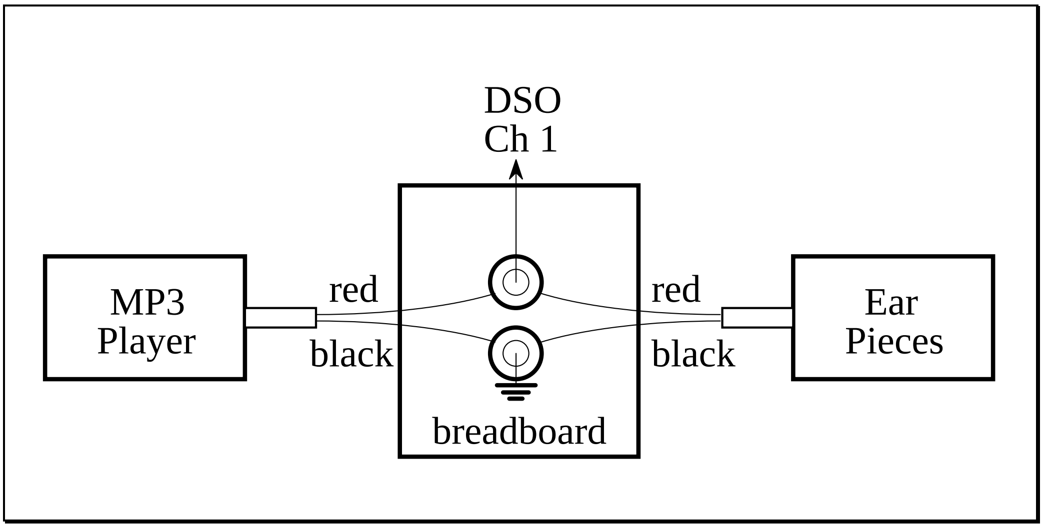


Figure L10.6

This will enable you to both listen to the audio signals and observe them on the DSO.

2. If you have been using “waveform averaging” to measure the frequency response of the universal filter (and you should have been for low amplitude responses), turn it **OFF**. The signals we will be looking at are difficult to trigger from - and waveform averaging is not correct unless we have a stable trigger!

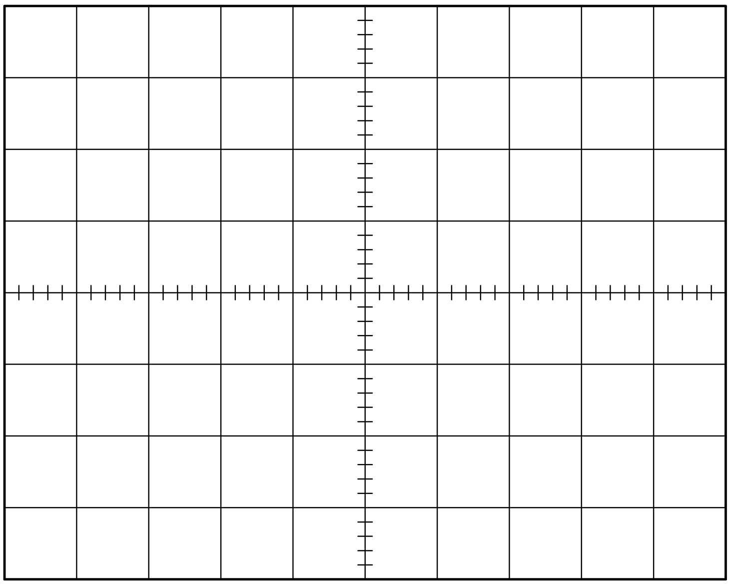
3. Turn the MP3 player on.

4. Play the track “Lab10 – 01 – Three Tones.mp3”.

5. Increase the volume to the maximum level (32).

6. **Do not put the ear pieces into your ears!** Listen closely to the right ear piece to hear the audio signal.

7. Sketch the spectrum of the audio signal.



8. Measure the **frequency** of the three dominant sinusoids present in the signal:

$f_1 =$

$f_2 =$

$f_3 =$

9. Play the track “Lab10 – 02 – Music.mp3” to hear music.

10. Observe the spectrum (do not sketch!).

11. When you have finished listening, **turn the MP3 player off**.

Filtering Audio Signals

1. Construct the following system:

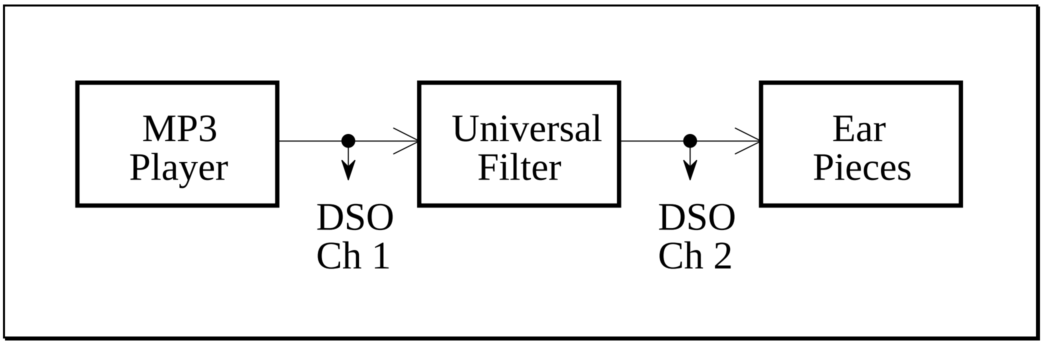


Figure L10.7

2. Turn the MP3 player on.

3. Play the track “Lab10 – 01 – Three Tones.mp3”.

4. Increase the volume to the maximum level (32).

5. In the “Vertical” section of the DSO, press the `Math` button, press the `FFT` softkey, then press the `Settings` softkey to display the FFT menu.

6. Set the Source of the FFT to `2` (Channel 2 is the filter output).

7. Observe the spectrum and listen to the output of the filter for the universal filter configured as LP, BP, HP and notch. For each filter type check for the presence of f_1, f_2 and f_3:

Filter Type	f_1	f_2	f_3
Lowpass	☐	☐	☐
Bandpass	☐	☐	☐
Highpass	☐	☐	☐
Notch	☐	☐	☐

8. Play the track “Lab10 – 02 – Music.mp3” to hear music.

9. Observe the spectrum and listen to the output of the filter for the universal filter configured as LP, BP, HP and notch. For each filter type, describe the effect on the audio signal:

Filter Type	**Effect (e.g. decreased bass / treble sounds)**
Lowpass	
Bandpass	
Highpass	
Notch	

10. You may have noticed and heard an annoying “tone” overlaying the music track. You can observe this tone by looking at the spectrum of Channel 1. Which type of filter is best to remove it, and why?

Best filter to remove unwanted “tone”:   Why?

12. When you have finished, **turn the MP3 player off**.

Lab Assessment [2 marks]

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

1. Show the magnitude and phase responses of the lowpass filter and bandpass filter.

2. Show the magnitude and phase responses of the highpass filter and notch filter.

3. Show the result of measuring the frequencies present in the first MP3 track (3 tones).

4. Explain the choice of the filter type to remove the unwanted "tone" from the second MP3 track (music).

Marking

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

Lab Equipment Guide

Mini-Lab. MP3 Player.

Introduction

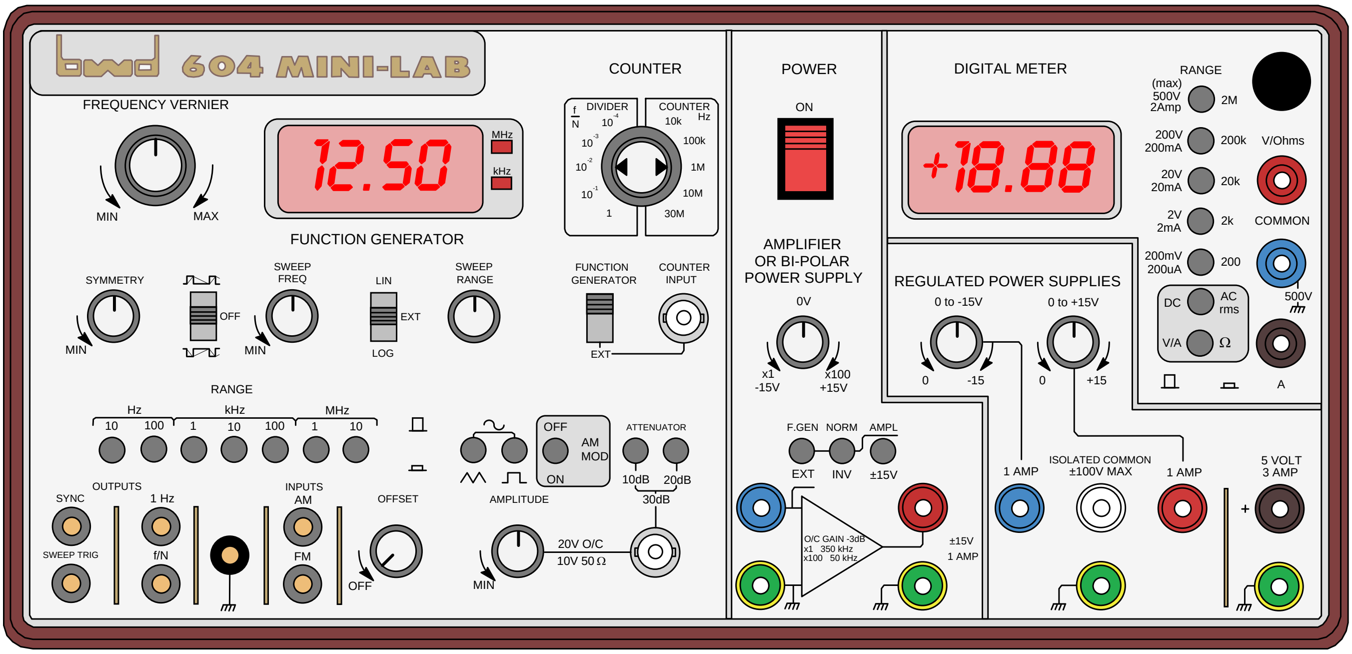
This guide is a reference for the following equipment:

Equipment

- Mini-Lab – BWD 604
- MP3 Player – Dick Smith A8696

Mini-Lab

The Mini-Lab front panel has the following layout:



It is seven instruments in one:

- 20 MHz function generator with AM and FM capabilities
- 30 MHz counter
- Power Amplifier
- ±15 V, 1 A Adjustable Bi-Polar Power Supply
- +15 V and -15 V, 1 A Adjustable Isolated Dual Power Supply
- 5V, 3A Power Supply
- 3 ½ digit Volt, Amp and Ohm Meter with true RMS AC readings

In addition, all inputs and outputs are short-circuit proof and protected.

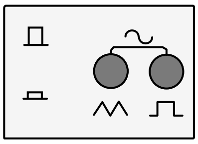
However, it is susceptible to damage if two outputs are short-circuited together, *such as connecting the power amplifier output to the function generator output* – never do this!

Function Generator and Counter

The function generator is capable of generating 20 Vp-p sine, triangular, square and ramp waveforms from 0.1 Hz to 20 MHz. It also provides for a "DC offset" with a range of ±10V. It has AM and FM modulation capabilities. The 4 digit counter has a range from 5 Hz to 30 MHz and updates every second.

LEG1.4

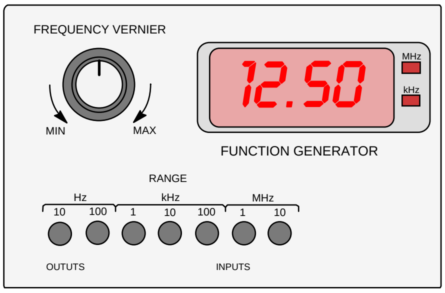
Function Generator



1. The type of waveform generated depends upon the two pushbuttons as shown in the table below:

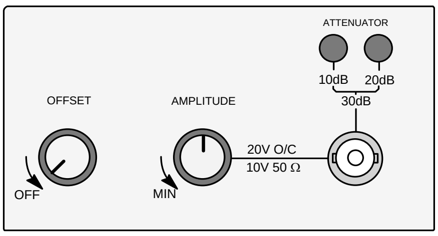
Pushbuttons Left	Pushbuttons Right	Waveform
Out	Out	Sinusoid
Out	In	Square
In	Out	Triangle
In	In	Undefined

Both pushbuttons "in" is undefined and should be avoided.



Frequency selection

2. The frequency of the waveform is selected by first changing the range using the pushbuttons, and then turning the frequency vernier knob that continuously varies the frequency within the set range. If the counter is set to read the internal function generator, then the frequency is displayed on the 4 digit LED display.



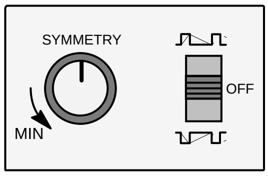
Amplitude selection

3. The amplitude of the waveform can be continuously varied in a 20:1 ratio using the amplitude knob. Additionally, the two attenuator pushbuttons can be used, either singly or together, to achieve attenuation of 10 dB, 20 dB or 30 dB. The waveform can have a DC offset from -10 V to +10 V by turning the DC offset knob – 0 volts is achieved around the vertical position.

If no DC offset is required, ensure the OFFSET knob is in the OFF position.

Also note that the function generator has an output resistance of 50 Ω.

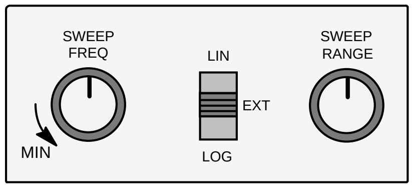
4. The symmetry of the waveform can be continuously varied between 30% and 70% by firstly selecting the direction of the asymmetry, and then continuously varying it using the knob.



Symmetry selection

If no asymmetry is required, ensure the switch is set to OFF.

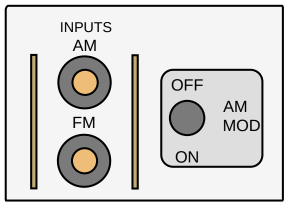
5. The frequency of the waveform can be "swept" – i.e. continuously varied from a minimum frequency to a maximum frequency in a repeated cycle – in either a linear or logarithmic fashion. The frequency can also be varied using an external signal that you provide. The rate of the sweep is continuously variable, as is the range.



Frequency sweep

If no frequency sweep is required, ensure the switch is set to EXT.

6. The generated waveform can be "modulated" using either amplitude modulation (AM) or frequency modulation (FM). In AM, the external signal will change the *amplitude* of the "carrier" sinusoid. In FM, the external signal will change the *frequency* of the "carrier" sinusoid.

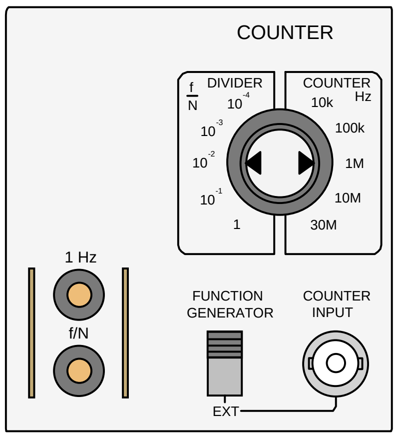


Modulation

If no modulation is required, ensure the AM MOD pushbutton is out (OFF).

Counter

1. The frequency counter also doubles as a decade frequency divider. With the switch set to FUNCTION GENERATOR, the counter will automatically display the frequency of the internally generated waveform. When the switch is set to EXT, the counter will display the frequency of the signal applied to the COUNTER INPUT, in the range set by the COUNTER knob. In addition, the DIVIDER setting will divide an internal 1 Hz square wave and make both signals available at the outputs labelled 1 Hz and f/N.



Counter / divider and clock output

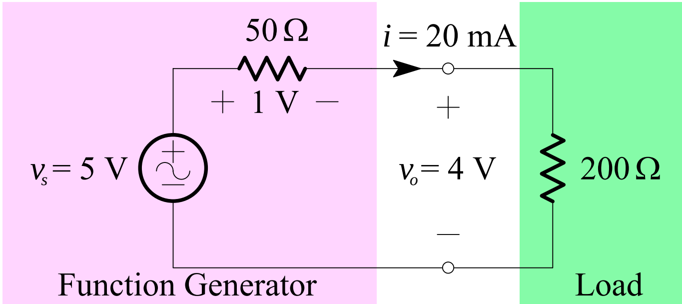
To read the frequency of the internal function generator waveform, ensure the switch is set to FUNCTION GENERATOR.

LEG1.6

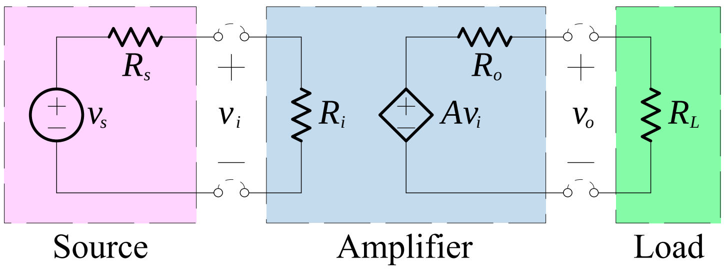
Amplifier or Bi-polar Power Supply

There are many types of voltage sources, e.g. power supplies, function generators, batteries, antennae, etc. When modelling these sources, it may turn out that they have large internal resistances (in comparison to an attached load).

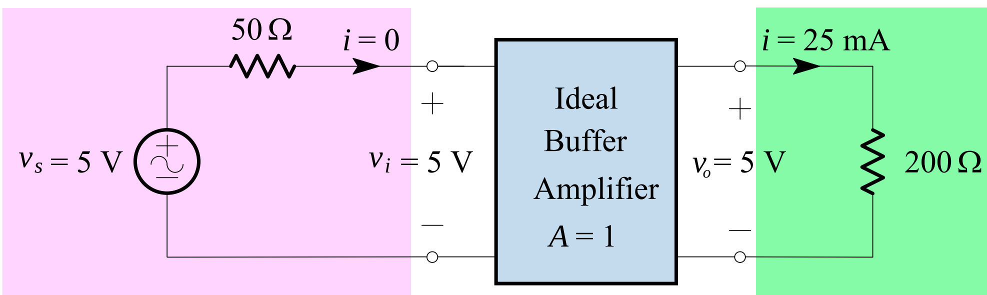
For example, the function generator has an output resistance of $50\,\Omega$ and the output will experience a significant internal *Ri* voltage drop when drawing "large" currents (> 10 mA), resulting in a drop in the output terminal voltage:



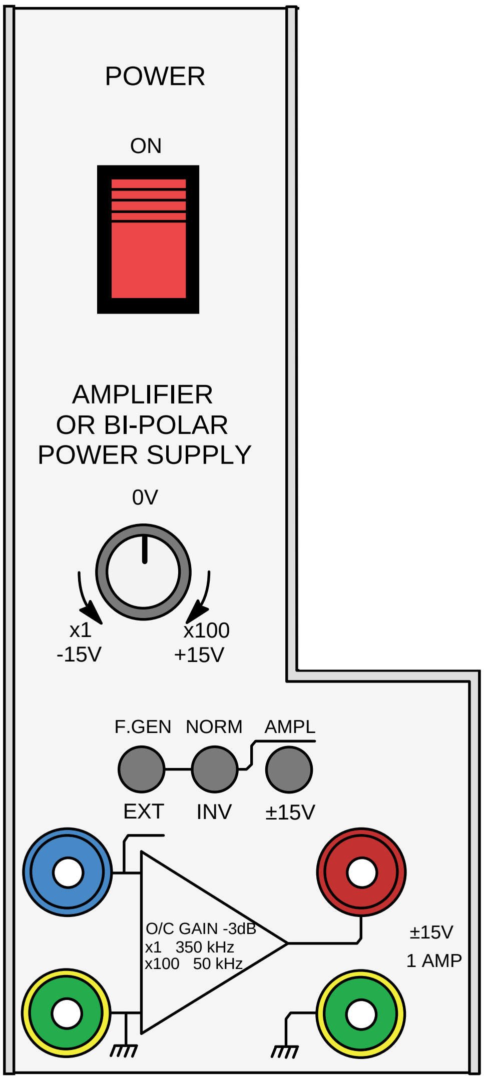
Therefore, we sometimes need to "buffer" a voltage source with an amplifier which presents a high input resistance to the source and which also provides a low output resistance to the load:



An ideal buffer amplifier with a gain of 1, when placed in between a function generator and a load, delivers the full source voltage to the load:



The Amplifier or Bi-Polar Power Supply section of the Mini-Lab provides us with a way to buffer a voltage source. The Amplifier presents a very high input resistance (100 kΩ) at its input terminals, whilst providing a very low output resistance (50 mΩ) at its output terminals. In addition, the gain (the amount by which the input signal is amplified) can be varied from 1 up to 100.



1. The left-most pushbutton selects the source of the amplifier – with the pushbutton out (F. GEN), the internal function generator is selected and no external connection is necessary. With the pushbutton in (EXT), you can apply an input signal to the blue terminal – with respect to earth, the green terminal.

2. The middle pushbutton selects whether the output of the buffer is normal (NORM) or inverted (INV).

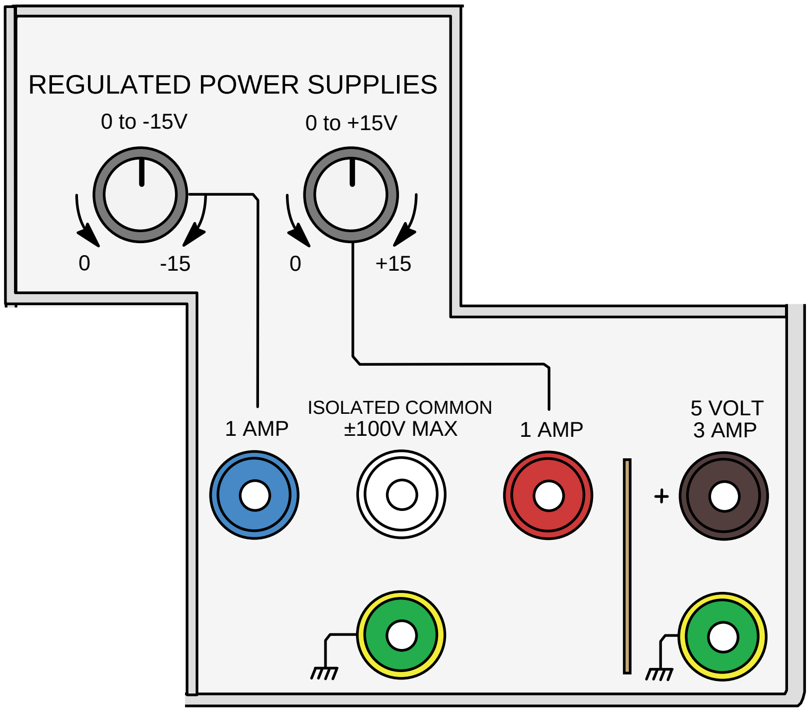
3. The right-most pushbutton selects whether the unit operates as an amplifier (AMP) or as a bi-polar power supply (± 15V).

4. When the unit is an amplifier, the knob varies the gain from 1 to 100. When the unit is a bi-polar power supply, the knob varies the output DC voltage from -15 V to +15 V.

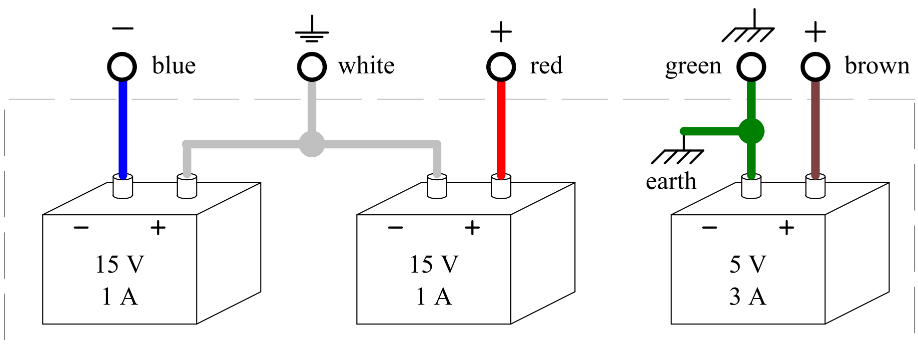
5. The output of the amplifier is taken from the red output terminal – with respect to earth, the green terminal.

Regulated Power Supplies

The Mini-Lab provides us with a ±15 V, 1 A adjustable isolated dual power supply and a 5V, 3A fixed power supply.



The power supplies are connected as shown below:

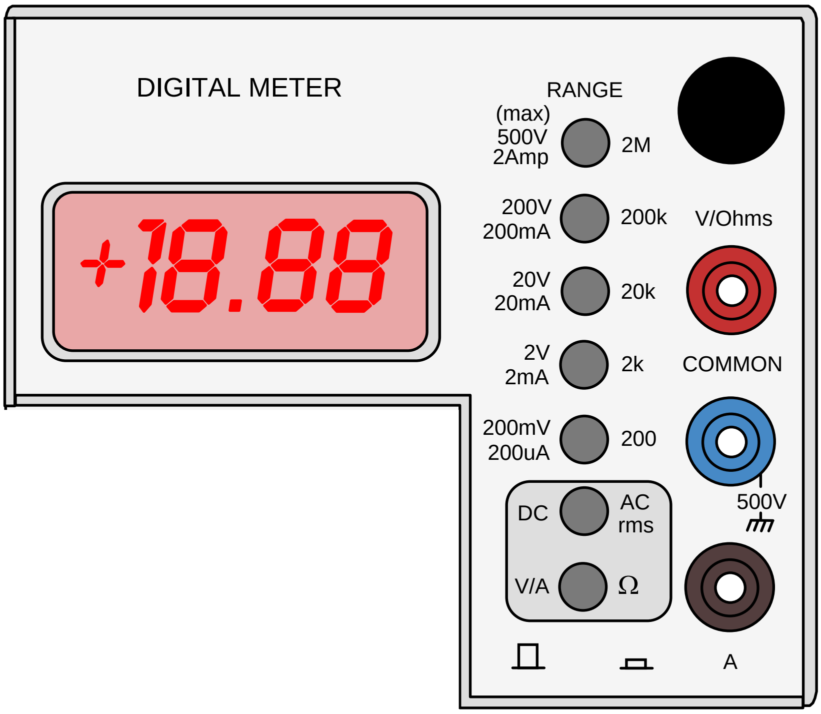


The outputs of the dual power supply are connected in series – this cannot be changed. Also, each output of the dual power supply is 'floating' with respect to earth at the general power outlet (GPO), and thus is similar to a battery. In contrast, the fixed 5 V supply has an output terminal that is taken with respect to earth, and is independent of the common of the dual power supply.

It is important to note the internal connections of the power supplies.

Digital Meter

The digital meter built into the Mini-Lab provides us with a 3½ digit volt, amp and ohm meter with true RMS AC readings, at an accuracy better than 3%.



The range buttons specify the maximum value that is displayed on that range. There is a "common" connection, which is isolated from earth, that must be used for all measurements. Separate physical inputs are provided for volts/ohms and amps measurements.

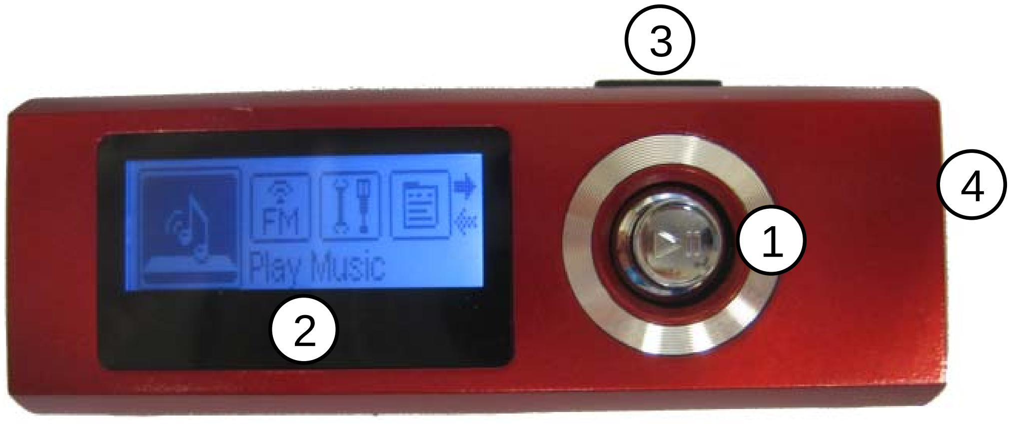
1. The bottom pushbutton selects the type of measurement. With the pushbutton out (V/A), the measurement will be volts or amps, depending on the physical connection. With the pushbutton in (Ω), the measurement will be ohms.

2. The second-from-bottom pushbutton selects whether the meter is a DC or AC meter for volt/amp measurements. With the pushbutton out (DC), the measurement will be the DC, or average value, of the voltage or current. With the pushbutton in (AC rms), the measurement will be a true RMS AC reading of the voltage or current.

MP3 Player

The MP3 player has the following layout:

MP3 player layout

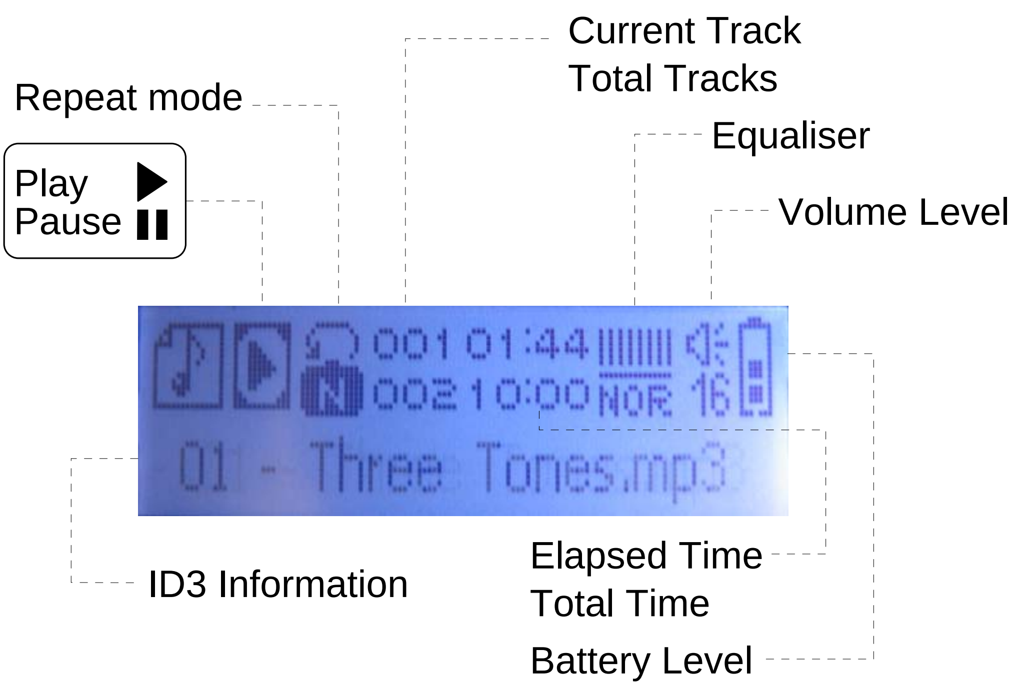


Joystick

1. The joystick can be moved in the usual four directions. It can also be pressed.

LCD display

2. The LCD display looks like:



Menu button

3. The MENU button is on the top of the device.

Headphone

4. The Headphone Out jack is on the side of the device.

Power On / Off

1. To turn the player **on**, press and hold ▷II .
   The LCD will display the “D” logo.

2. To turn the player **off**, press and hold ▷II .
   The LCD displays “*Bye Bye!!*”.

Main Menu

The main menu gives you access to the different function modes of the player.

1. To enter the Main Menu, hold down the `Menu` button.

	The Play Music mode is the only mode we need. To select it, navigate with the joystick and then press ▷II .

2. To exit the Main Menu, hold the `Menu` button.

Music Playback Controls

Use the following controls during music playback.

Key Action	Function
Press ▷II	Play / Pause music playback.
Press ▷II ←	Play the previous track.
Press ▷II →	Play the next track.
Hold ▷II ←	Reverse through the current track.
Hold ▷II →	Fast-forward through the current track.
Press ▷II ↑	Decrease the volume level.
Press ▷II ↓	Increase the volume level.