Tuned Circuits and Filters
53-220

Feedback Instruments Ltd, Park Road, Crowborough, E. Sussex, TN6 2QX, UK.
Telephone: +44 (0) 1892 653322, Fax: +44 (0) 1892 663719.
email: feedback@feedback-instruments.com website: http://www.feedback-instruments.com

Manual produced from software version: v2.41

Date: 17/12/2013

Feedback Part No. 1160–53220

Product Use

All users must familiarise themselves with the following information.

This product is marked as CE compliant. This means that it complies with the relevant European Directives for this product. In particular the Directives cover Low Voltage, EMC, Machinery, Pressure and electronic waste disposal.

The equipment, when used in normal or prescribed applications and within the parameters set for its mechanical and electrical performance, should not cause any danger or hazard to health or safety.

If, in specific cases, circumstances exist in which a potential hazard may be brought about by careless or improper use, these will be pointed out and the necessary precautions emphasised.

This equipment is designed for use by students as part of the learning process who must be under the supervision of a suitably qualified and experienced person in a laboratory environment where safety precautions and good engineering practices are applied.

By the nature of its intrinsic teaching functionality, parts are visible and accessible that might normally be covered up or encased in an industrial or domestic product. For this reason students attention should be drawn to the need to operate the equipment only in the manner prescribed in the accompanying documentation and supervisors must make students aware of any particular risk. The equipment should not be operated by any person alone.

We are required to indicate on our equipment panels certain areas and warnings that require attention by the user. These have been indicated in the specified way by yellow labels with black printing. The meanings of any labels that may be fixed to the instrument are shown below:

CAUTION -
RISK OF
DANGER

CAUTION -
RISK OF
ELECTRIC SHOCK

CAUTION -
ELECTROSTATIC
SENSITIVE DEVICE

Compliance with the EMC Directive

This equipment has been designed to comply with the essential requirements of the Directive. However, because of the intrinsic teaching function it cannot be electromagnetically shielded to the same extent as equipment designed for industrial or domestic use. For this reason the equipment should only be operated in a teaching laboratory environment where electromagnetic emissions in the immediate area might not be expected to cause adverse effects. In the same way users should be aware that operating the equipment near to an electromagnetic source may cause the experimental results to be outside the range expected.

The Waste Electrical and Electronic Equipment Directive (WEEE)

If this equipment is disposed of it must be in accordance with the regulations regarding the safe disposal of electronic and electrical items and not placed with ordinary domestic or industrial waste.

Product Improvements

We maintain a policy of continuous product improvement by incorporating the latest developments and components into our equipment, even up to the time of dispatch.

All major changes are incorporated into up-dated editions of our manuals and this manual was believed to be correct at the time of printing. However, some product changes which do not affect the teaching capability of the equipment, may not be included until it is necessary to incorporate other significant changes.

Component Replacement

In order to maintain compliance with the Directives all replacement components must be identical to those originally supplied.

Operating Conditions

WARNING:

This equipment must not be used in conditions of condensing humidity.

This equipment is designed to operate under the following conditions:

Operating Temperature	10°C to 40°C (50°F to 104°F)
Humidity	10% to 90% (non-condensing)

Copyright Notice

ACKNOWLEDGEMENTS

Feedback Instruments Ltd acknowledge all trademarks.

MICROSOFT, WINDOWS 8, WINDOWS 7 WINDOWS VISTA, WINDOWS XP, WINDOWS 2000, WINDOWS ME, WINDOWS NT, WINDOWS 98, WINDOWS 95 and Internet Explorer are registered trademarks of Microsoft Corporation.

MATLAB is a registered trademark of Mathworks Inc.

LabVIEW is a registered trademark of National Instruments.

Familiarisation

Objectives

To become familiar with the circuit blocks available on the workboard

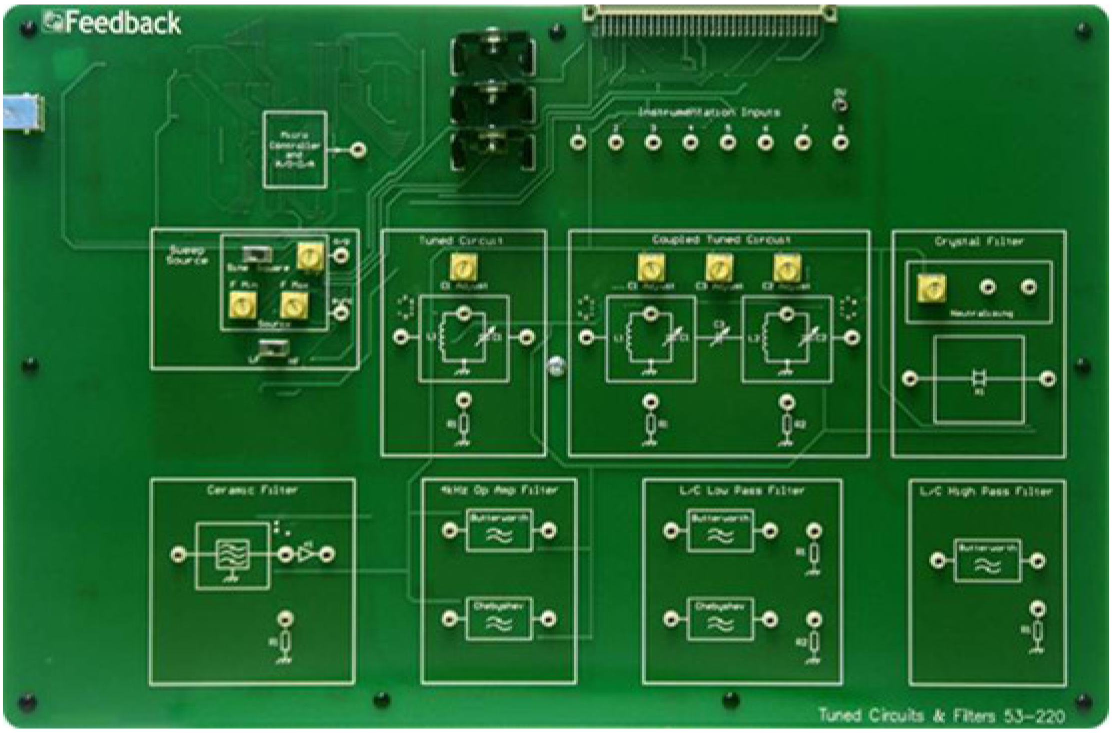
To become familiar with the interconnection of the workboard, terminal and PC

To determine that the set-up is functioning as required

To learn how to navigate the software

The Workboard – an Introduction

The Tuned Circuits & Filters 53-220 workboard contains a number of circuit blocks that may be interconnected in many ways to demonstrate the principles and operation of typical selective and filter circuits used in modern telecommunications equipment.



The workboard is designed to operate with the Real-time Access Terminal (RAT) 92-200, into which it plugs to obtain power and to provide, in conjunction with a personal computer (PC), the instrumentation required by the assignments.

Both the workboard and the RAT require USB connection to the PC.

Interconnection between the various circuit blocks on the workboard is by 2 mm, stackable patch leads. It is recommended that no more than two leads be stacked, as more than this is mechanically vulnerable and can lead to damage of the lead or the workboard.

Practical 1: The Circuits Available

Objectives and Background

This Practical is an exercise to get you conversant with the circuit blocks that are available on the Tuned Circuits & Filters workboard. There is no patching or measurement associated with this Practical.

At this stage, do not worry if you do not understand the description or function of the circuit blocks. As you progress through the assignments their functions and operation should become clearer.

Practical 1: The Circuits Available

Perform Practical

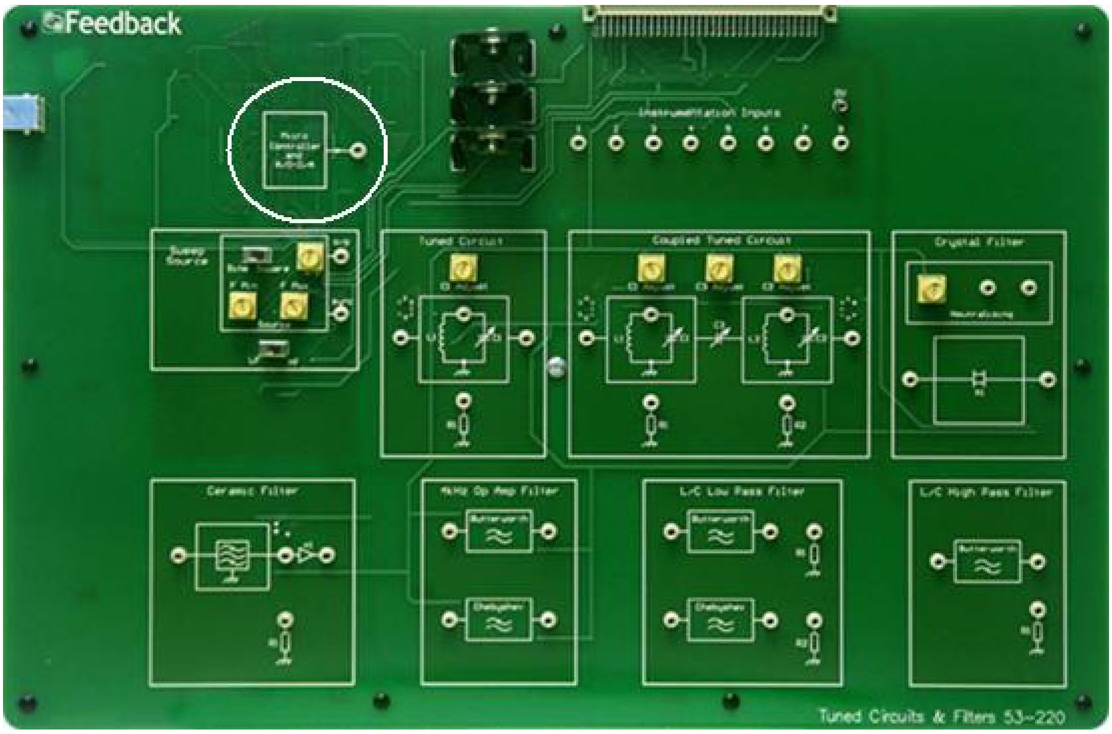
This Practical requires no workboard patching connections and there are no measurements to be taken.

Read through the descriptions below and identify each of the circuit blocks described.

At this stage, do not worry if you do not understand the description or function of the circuit blocks. As you progress through the assignments their functions and operation should become clearer.

The Micro Controller

Towards the top left-hand corner of the workboard you will see the Micro Controller and A/D-D/A circuit block.

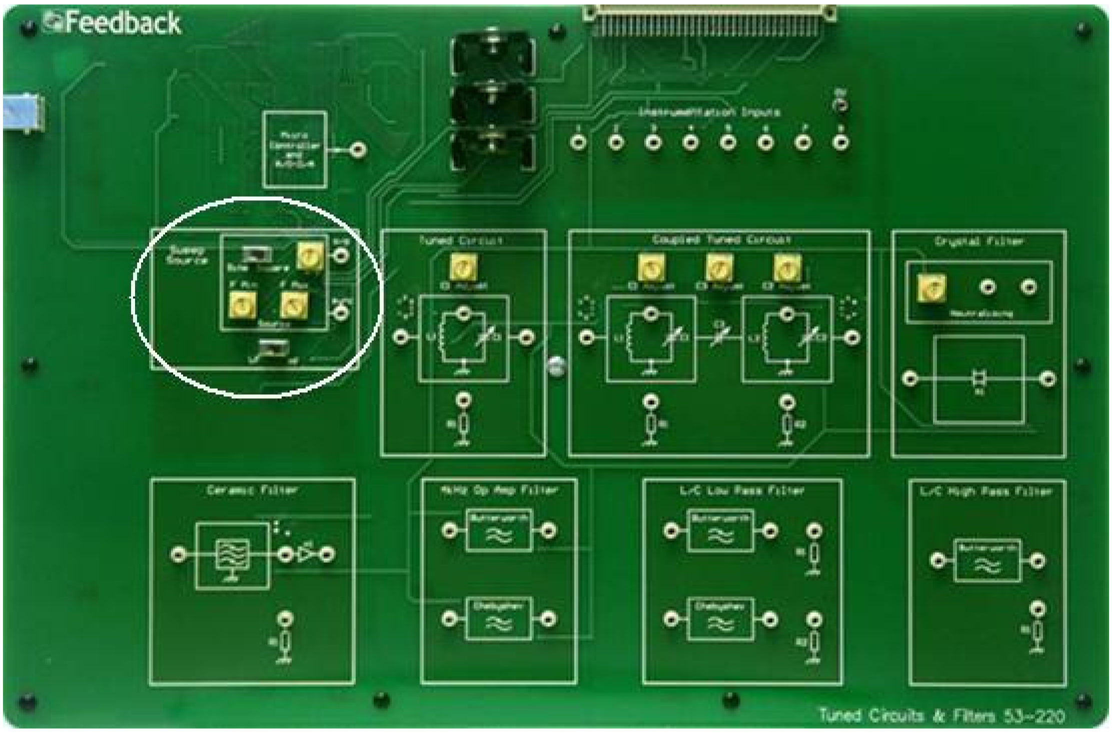


This block contains the circuitry and firmware that provides the modulation source for many of the assignments. It also provides waveforms and timing signals for a number of the assignments.

The Sweep Source

To the right of the Circuit Select switch you will find the Sweep Source circuit block.

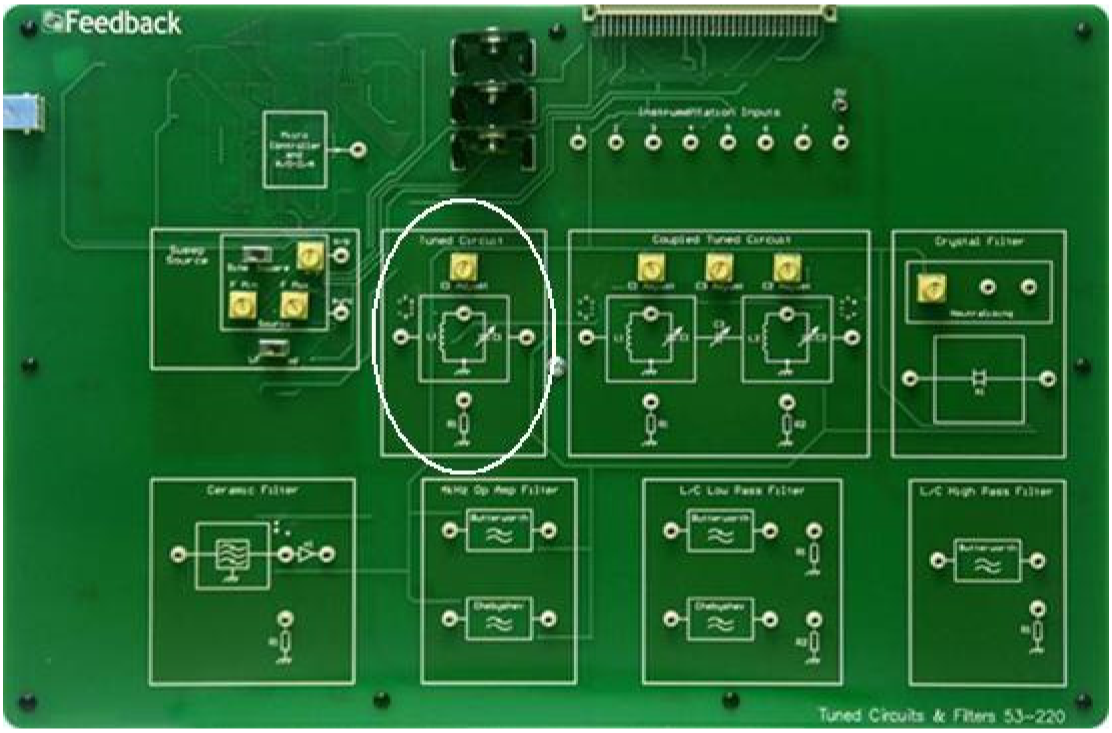
This is a signal source that may be swept in frequency from approximately 400Hz to 8.5MHz, in two ranges, selectable by slide switch. The range of frequency over which the source sweeps is set by the **FMin** and **FMax** controls.



The source can produce either sine or square wave outputs, selectable by slide switch, and the output amplitude is variable using the **o**/**p** control.

The Tuned Circuit

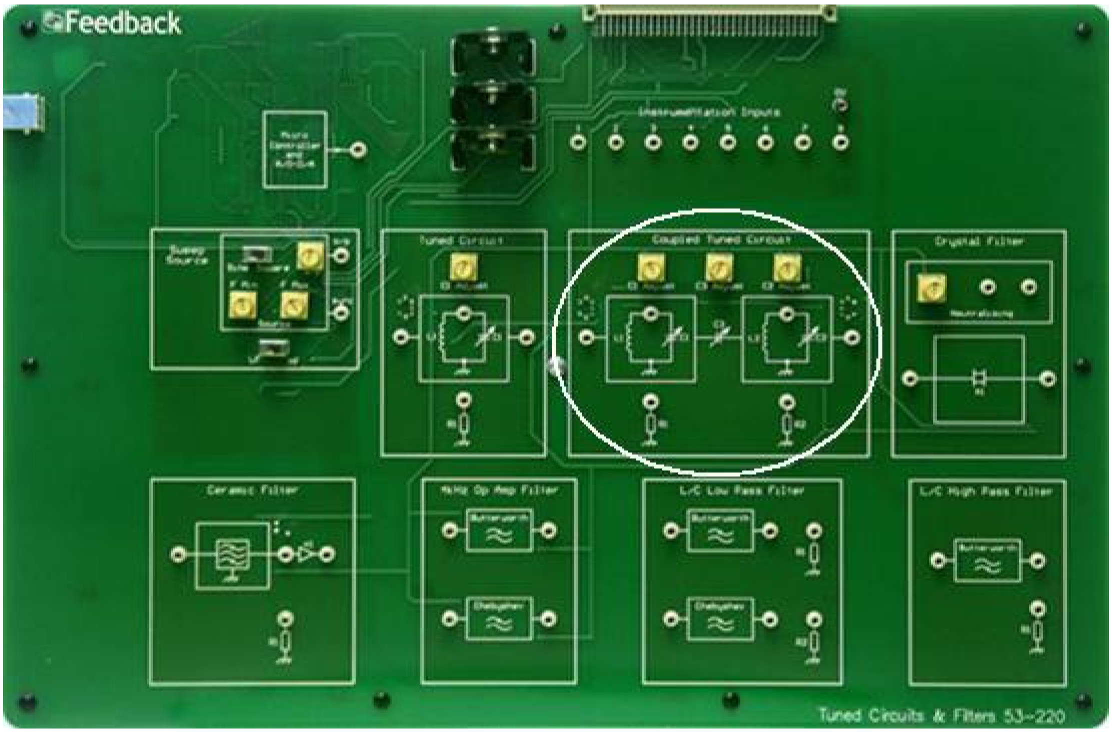
This circuit block is positioned towards the top left centre of the workboard.



This is a single inductor-capacitor tuned circuit that is a form of circuit often used to select a frequency in communications circuits.

The Coupled Tuned Circuit

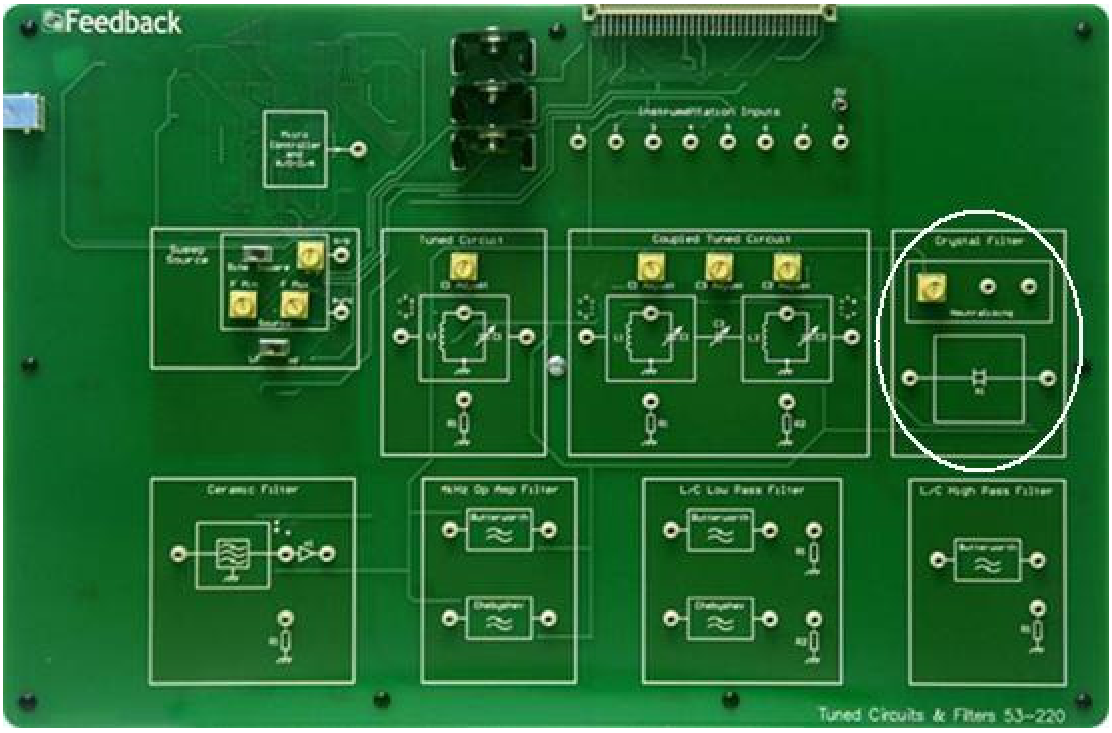
This circuit block is positioned towards the top right centre of the workboard.



This is a connection of two inductor-capacitor tuned circuits that is a form of circuit often used to select a band of frequencies in communications circuits.

The Crystal Filter

This circuit block is positioned to the right-hand centre of the workboard.

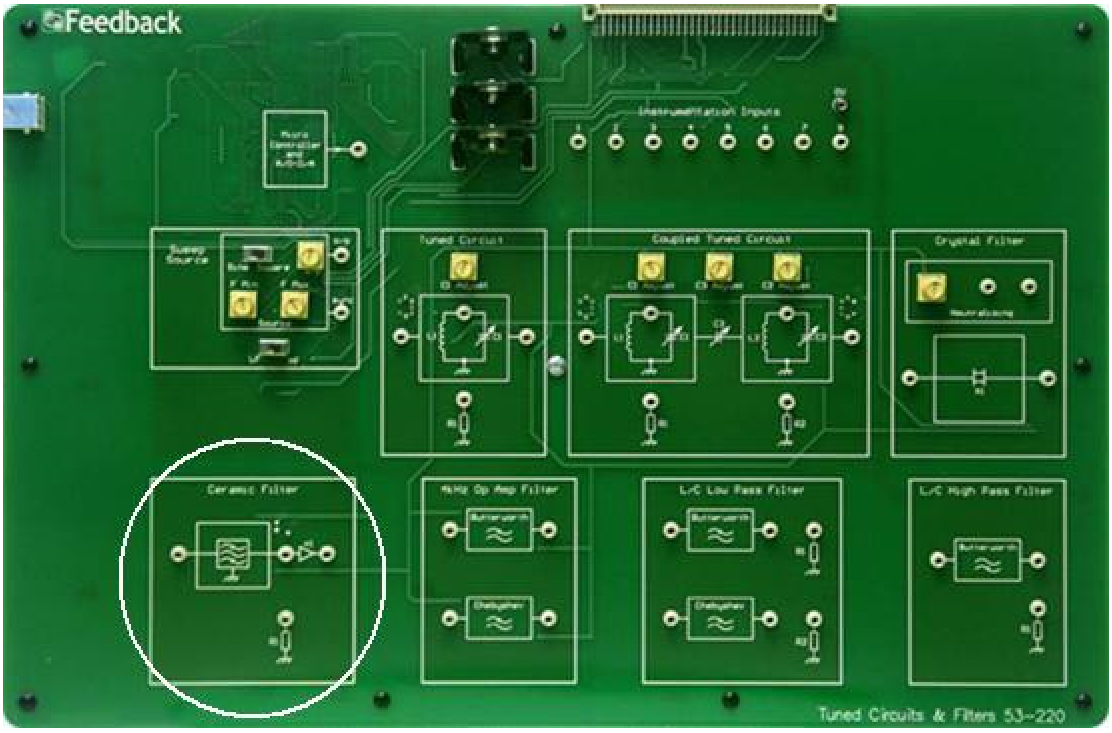


The Crystal Filter is a circuit that has a very selective frequency response. It makes use of the physical properties of the piezoelectric effect of a quartz crystal. It is used in communications circuits to select one frequency and reject others.

You will investigate the properties of the circuit and see its extremely selective response.

The Ceramic Filter

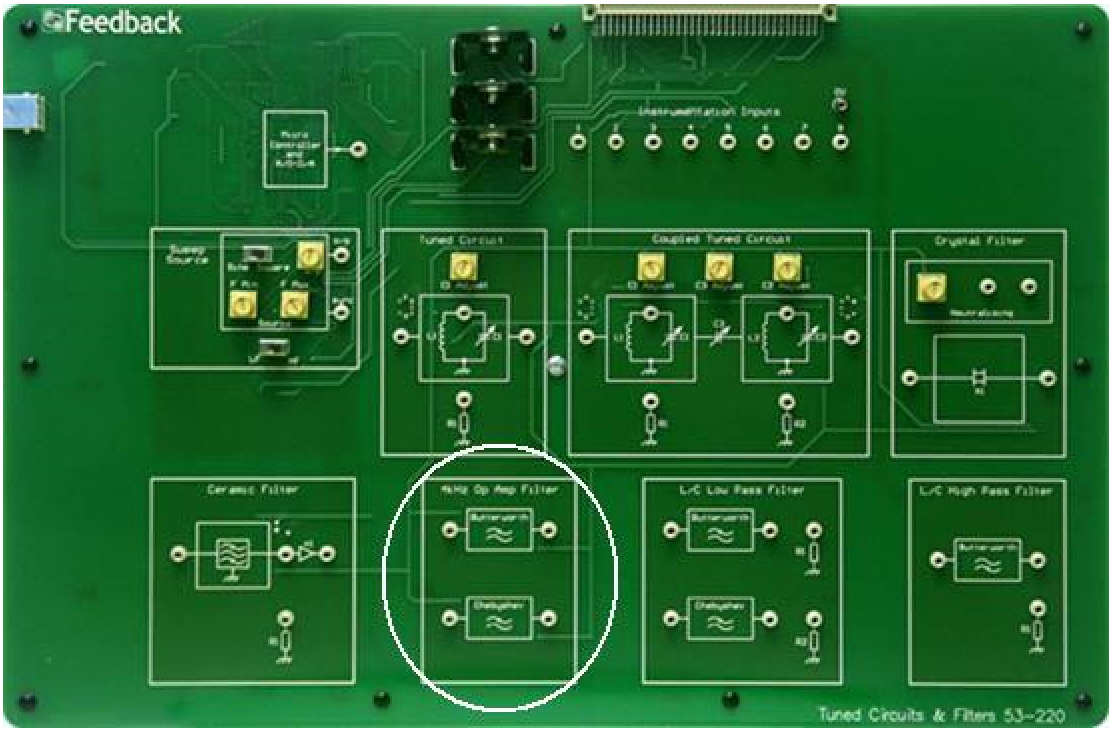
This circuit block is found towards the bottom left-hand corner of the workboard.



A Ceramic Filter circuit is another example of one that uses a piezoelectric device to provide a frequency selective circuit. In this case, the filter produced has a bandpass response. such a filter is often used in radio receivers and other communications applications.

The 4kHz Operational Amplifier Filter

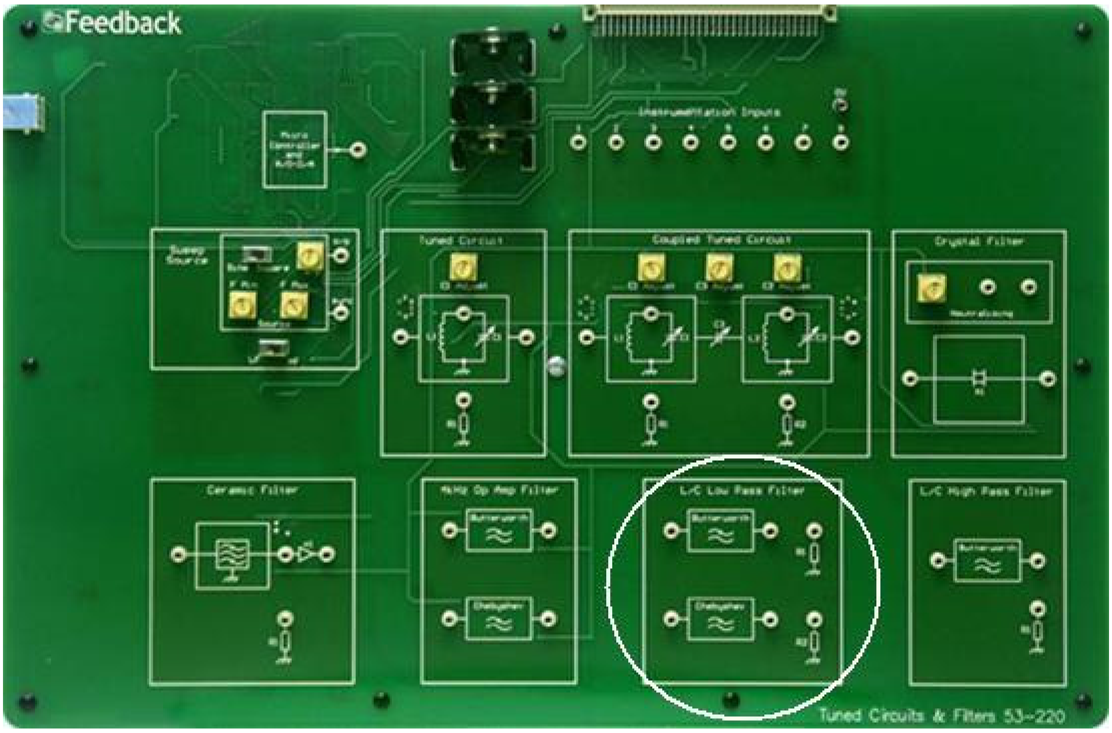
This circuit is found in the lower left-hand centre of the workboard.



There are two circuits within this block: Butterworth and Chebyshev low-pass, operational amplifier-based, active filters. Whilst having nominally the same cut-off frequency, these filters have different characteristics that you will investigate.

The L/C Low Pass Filter

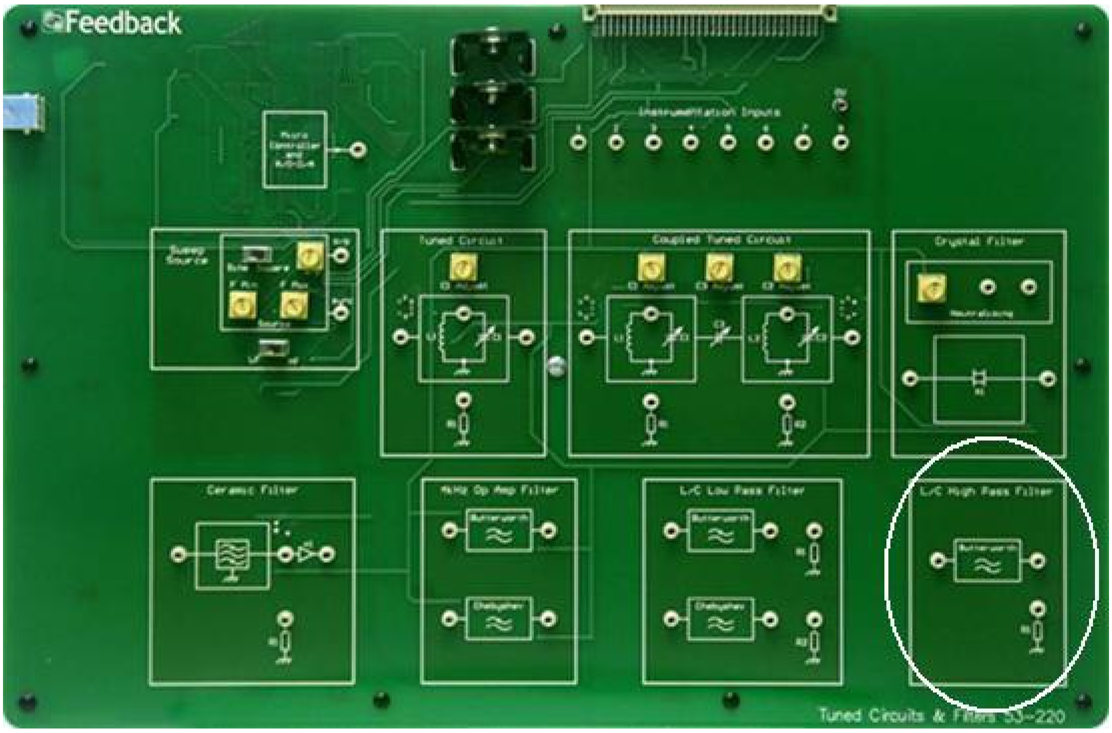
This circuit is found in the lower right-hand centre of the workboard.



There are two circuits within this block: Butterworth and Chebyshev low-pass, L/C (inductance/capacitance)-based, passive filters. Whilst having nominally the same cut-off frequency, these filters have different characteristics that you will investigate.

The L/C High Pass Filter

This is to be found in the bottom right-hand corner of the workboard.

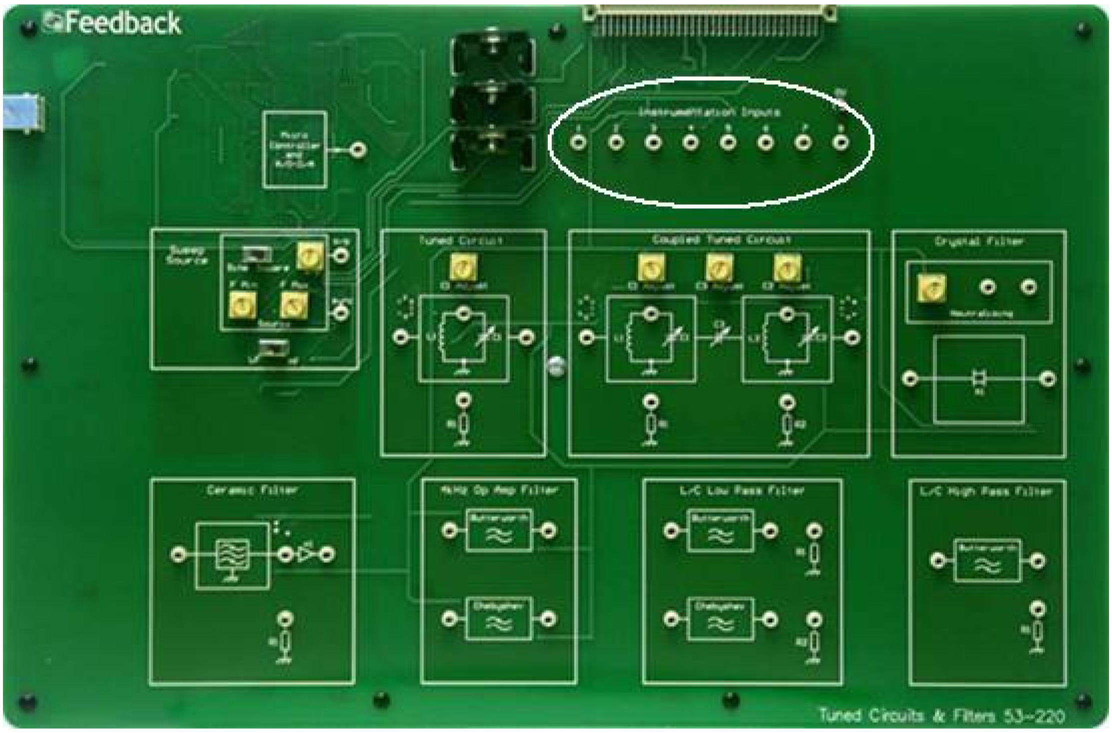


This is an inductance/capacitance-based (L/C) passive high-pass filter circuit.

Instrumentation Inputs

Signals present at any of the sockets available on the workboard may be measured and displayed on a PC using a Real-Time Access Terminal (RAT) and the Espial software that accompanies the product.

The points to be monitored must be patched to the Instrumentation Input sockets that are to be found at the top centre of the workboard. The figures associated with these sockets correspond to the numbers on the monitoring points as seen on the diagrams associated with each Practical activity.



Associated with the Instrumentation Inputs are two switches that switch the gain of the two instrumentation channels.

Practical 2: Connections to the PC

Objectives and Background

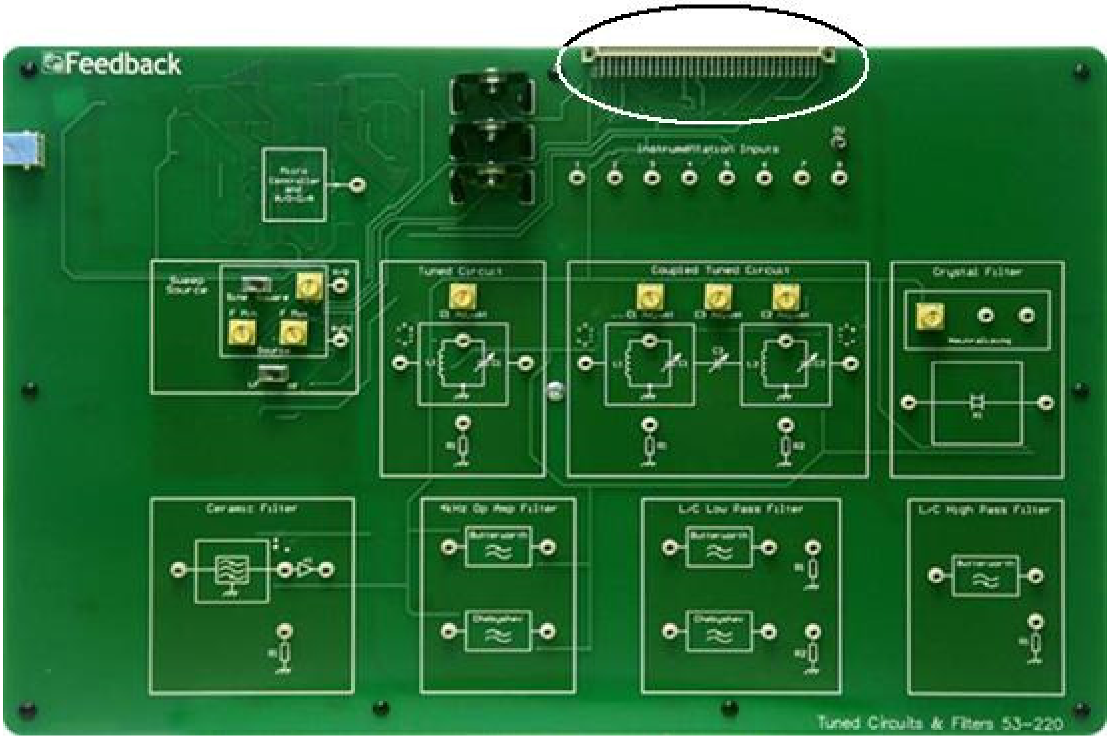
This Practical will familiarise you with the connections required to operate the Tuned Circuits & Filters 53-220 workboard with a PC.

Practical 2: Connections to the PC

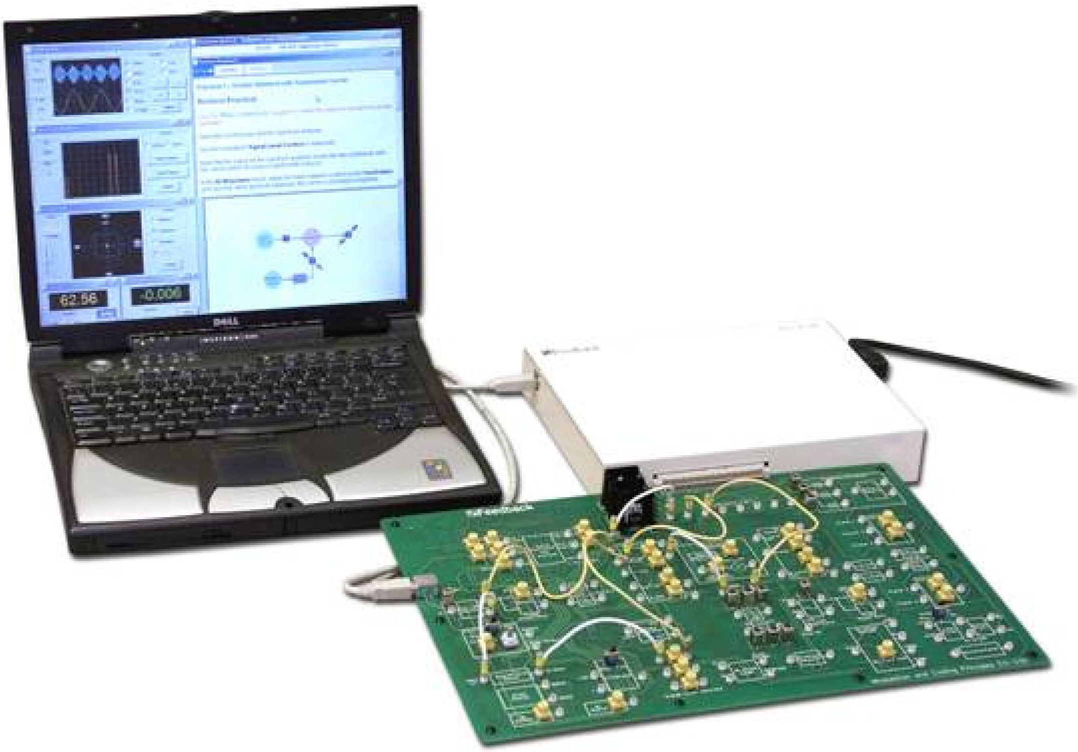
Perform Practical

This Practical requires no workboard patching connections and there are no measurements to be taken.

Identify the multiway connector on the top edge of the workboard.



This connector plugs into its female counterpart on the front edge of the Real-time Access Terminal (RAT) 92-200. The diagram below shows a workboard plugged into a RAT, together with a laptop PC.



Both the RAT and the workboard require USB connection to the PC. They may be USB1 or USB2 ports. If you do not have two available USB sockets on your PC, an external hub will have to be used. It may be either powered or un-powered.

For correct operation the PC must have the relevant Espial software and the RAT and product drivers installed. If it does not, you will need to consult your tutor.

If the Espial software has been installed the workboard and the RAT should automatically be recognised on switch-on and the system will be ready for use.

Practical 3: Operational Check

Objectives and Background

In this Practical you will perform a very simple operational check to confirm that the PC, the RAT and the workboard are communicating with each other and that the set-up is ready to perform a practical Assignment.

Practical 3: Operational Check

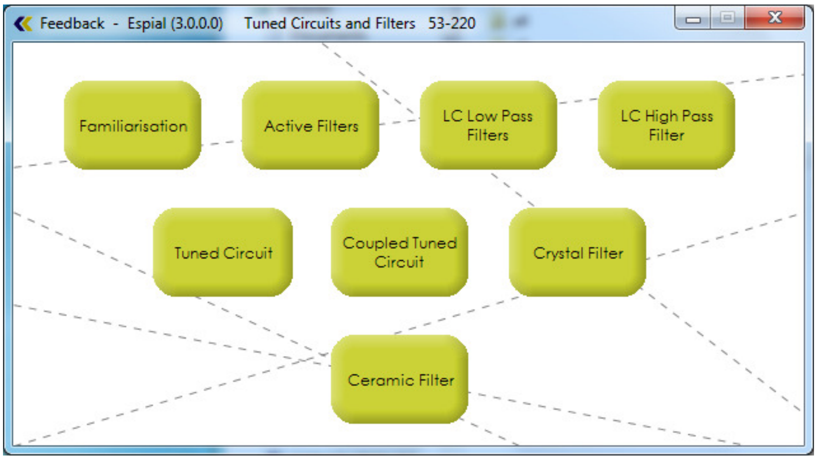
Perform Practical

This Practical requires no workboard patching connections and there are no measurements to be taken.

Ensure that you have connected the equipment as described in Practical 2 of this Assignment.

Ensure that the PC and the RAT are switched on.

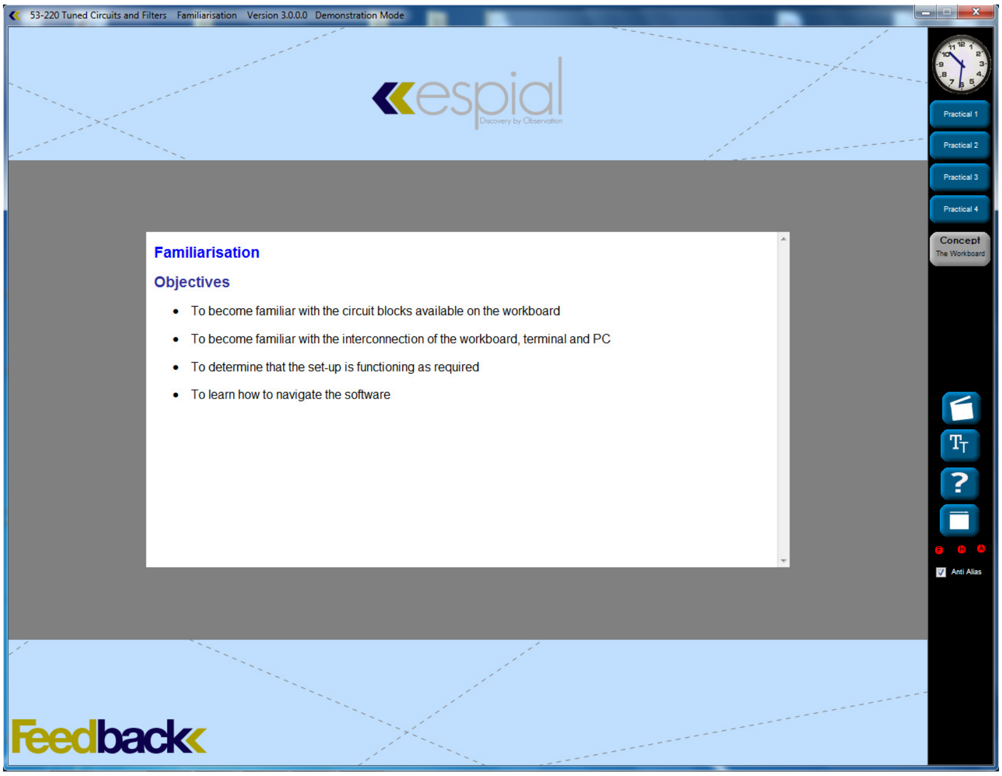
Launch the Espial software associated with the product.



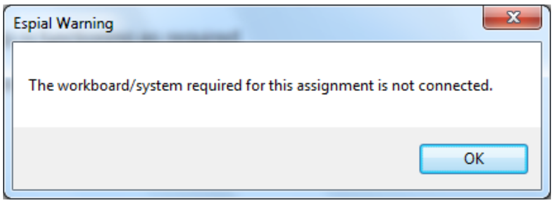
After a Espial Courseware splash screen has been briefly displayed, you should see a window showing all the assignments that are available for the product, of the form shown above. There may be a smaller or greater number of assignments available to you than shown. The precise appearance of this window, such as the choice of colours and how the buttons are arranged, is determined by your tutor. Note that you cannot close this window whilst any assignment is open, and you can have only one assignment open at any time.

To select an assignment to perform, left-click on the appropriate button.

After an Assignment loading dialog has been briefly displayed, the assignment window should appear. The assignment window is full-screen, consisting of a title bar across the top, a side bar at the right-hand edge, and the main working area. Initially the overall objectives for the chosen assignment are shown in the main working area. A typical screen shot is shown below. The precise appearance of the assignment window is determined by your tutor.

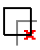


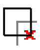
If the hardware has not been connected properly, the following Espial Warning message is immediately displayed on the screen:



If this warning message is shown, you must acknowledge it by clicking the OK button before you can continue. In this event, it is recommended that you resolve the problem before attempting to perform the assignment. You will need to close the assignment, correct the hardware problem and then restart the assignment.

On the screen shot of the assignment window, notice the three red indicators within the side bar. These are marked 'F', 'H' and 'A'. These are warning indicators. If any one of them is visible on your screen then you have a fault condition, as follows:

 indicates that there is a firmware communications error;

 indicates that the hardware is incorrectly connected, probably your workboard is incorrectly connected to your PC, or that the workboard driver is not installed correctly;

indicates that there is a data acquisition error, probably your RAT is incorrectly connected to the PC, or that the RAT driver is incorrectly installed.

If you do not see any of these warning indicators on your screen then your set-up is correct and you may perform any of the Practicals in the assignment. You can still open a Practical when a fault condition exists, but you will not be able to use any test equipment that may be required to perform that Practical. The hardware must be correctly connected before starting an assignment in order to use the test equipment in any of the Practicals within that assignment.

The next Practical takes you through the navigation of the software.

Practical 4: Navigating the Espial Software

Objectives and Background

Although the Espial Laboratory environment is very easy to operate, these notes will help you use all its facilities more quickly.

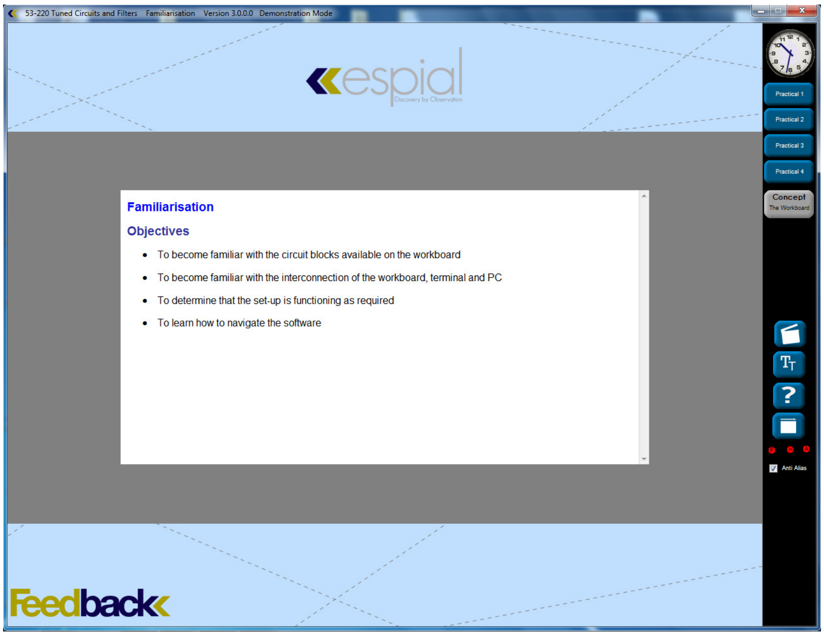
If there is a demonstration assignment, slider controls in the software perform functions that would normally be performed on the hardware. In normal assignments, if the any of hardware systems fail to initialise the system reverts to demonstration mode. This means that none of the test equipment is available.

Practical 4: Navigating the Espial Software

Perform Practical

This Practical requires no workboard patching connections and there are no measurements to be taken.

The assignment window opens when an assignment is launched as described in the previous Practical. The assignment window consists of a title bar across the top, an assignment side bar at the right-hand edge, and the main working area. By default, the overall assignment objectives are initially shown in the main working area whenever an assignment is opened. The assignment window occupies the entire screen space and it cannot be resized (but it can be moved by 'dragging' the title bar, and it can be minimised to the task bar). The title bar includes the name of the selected assignment. The side bar contains the Practicals and any additional resources that are relevant for the selected assignment. The side bar cannot be repositioned from the right-hand edge of the assignment window. An example of an assignment window is shown below.



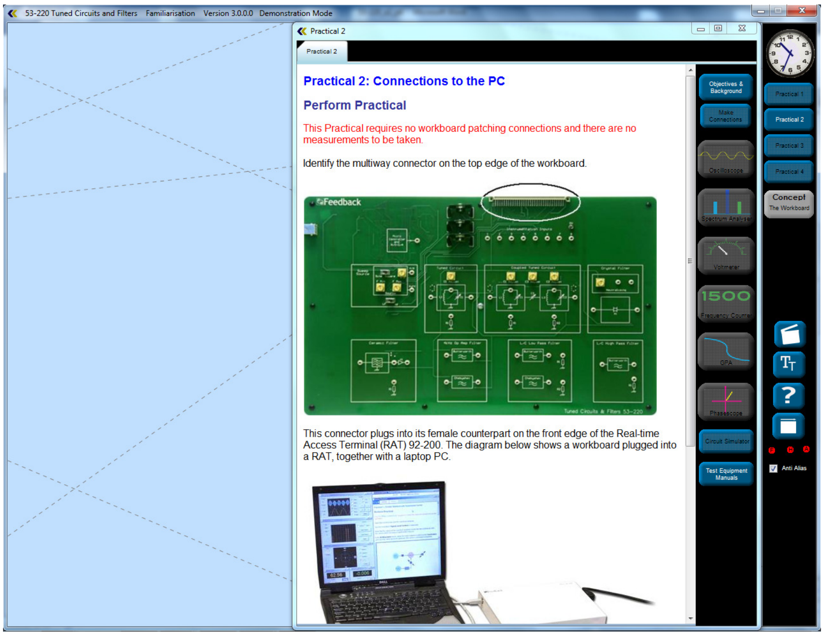
The precise appearance of the assignment window will depend on the 'skin' that has been selected by your tutor. However, the behaviour of each of the buttons and icons will remain the same, irrespective of this.

The clock (if you have one active) at the top of the side bar retrieves its time from the computer system clock. By double clicking on the clock turns it into a stop watch. To start the stop watch single click on the clock, click again to stop the stop watch. Double clicking again will return it to the clock function.

There are a number of resource buttons available in the assignment side bar. These are relevant to the selected assignment. In general, the resources available will vary with the assignment. For example, some assignments have video clips and some do not. However, the Technical Terms, Help and Auto Position buttons have identical functionality in every assignment. You can click on any resource in any order, close them again, or minimise them to suit the way you work.

Practicals are listed in numerical order in the side bar. When you hover the mouse over a Practical button, its proper title will briefly be shown in a pop-up tool-tip. There can be up to four Practicals in any assignment. You can have only one Practical window open at any time.

To perform a Practical, left-click on its button in the assignment side bar. The assignment objectives, if shown in the main working area, will close, and the selected Practical will appear in its own window initially on the right-hand side of the main working area, as shown below. You can move and resize the Practical window as desired (even beyond the assignment window) but its default size and position allows the test equipment to be displayed down the left-hand side of the main working area without overlapping the instructions for the Practical.



Again, the precise appearance of the Practical window can be determined by your tutor but the behaviour of each of the buttons and icons will remain the same, irrespective of this. Whatever it looks like, the Practical window should have icons for the test equipment, together with buttons for Objectives & Background, Make Connections, Circuit Simulator and Test Equipment Manuals. These resources are found in side bar, located on the right-hand edge of the Practical window. The resources will depend on which Practical you have selected. Therefore not all the resources are available in every Practical. If a resource is unavailable, it will be shown greyed out. To open any resource, left-click on its icon or button. Note that when you close a Practical window, any resources that you have opened will close. You may open any resource at any time, provided it available during the Practical. The Circuit Simulator will only be available if you have one loaded.

Note that if the hardware is switched off, unavailable, or its software driver is not installed, all the test equipment is disabled. However, you can open any other window. If you switch on the hardware it will be necessary to close the assignment window and open it again to enable the test equipment.

Resource Windows

These are windows may be moved, resized and scrolled. You may minimise or maximise them. The system defaults to ‘Auto Position’, which means that as you open each resource window it places it in a convenient position. Most resource windows initially place themselves inside the practical window, selectable using tabs. Each one lays over the previous one. You can select which one is on top by clicking the tab at the top of the practical window. You can see how many windows you have open from the number of tabs. If you want to see several windows at once then drag them out of the practical window to where you wish on the screen. If you close a window it disappears from the resources tab bar.

If you want to return all the windows to their default size and position simply click the Auto Position button in the assignment side bar.

Make Connections Window

This movable and resizable window shows the wire connections (2mm patch leads) you need to make on the hardware to make a practical work. Note that some of the wires connect the monitoring points into the data acquisition switch matrix. If this is not done correctly the monitoring points on the practical diagram will not correspond with those on the hardware. The window opens with no connections shown. You can show the connections one by one by clicking the Show Next button or simply pressing the space bar on the keyboard. If you want to remove the connections and start again click the Start Again button. The Show Function button toggles the appearance of the block circuit diagram associated with the Practical.

Test Equipment

The test equipment will auto-place itself on the left of the screen at a default size. You may move it or resize it at any time. Note that below a useable size only the screen of the instrument will be shown, without the adjustment buttons. Each piece of test equipment will launch with default settings. You may change these settings at any time. There is an auto anti-alias feature that prevents you setting time-base or frequency settings that may give misleading displays. If auto anti-alias has operated the button turns red. You can turn off the anti-aliasing feature, but you should be aware that it may result in misleading displays.

You may return to the default settings by pressing the Default button on each piece of test equipment. If you wish to return all the equipment to their original positions on the left of the screen click Auto Position on the side bar of the assignment window.

Note that if you close a piece of test equipment and open it again it returns to its default position and settings.

If you want more information on how a piece of test equipment works and how to interpret the displays, see the Test Equipment Manuals resource in the Practical side bar.

On slower computers it may be noticeable that the refresh rate of each instrument is reduced if all the instruments are open at once. If this is an issue then only have open the instrument(s) you actually need to use.

Test Equipment Cursors

If you left click on the display of a piece of test equipment that has a screen, a green cursor marker will appear where you have clicked. Click to move the cursor to the part of the trace that you wish to measure. If you then move the mouse into the cursor a tool-tip will appear displaying the values representing that position. Note if you resize or change settings any current cursor will be removed.

Perform Practical Window

This window contains the instructions for performing the practical, as well as a block, or circuit, diagram showing the circuit parts of the hardware board involved in the Practical. On the diagram are the monitoring points that you use to explore how the system works and to make measurements. The horizontal divider bar between the instructions and the diagram can be moved up and down if you want the relative size of the practical instruction window to diagram to be different. Note that the aspect ratio of the diagram is fixed.

Information Buttons on Practical Diagrams

On many of the symbols on the diagram you will find a button that gives access to new windows that provide more information on the circuit that the symbol represents. Note that these windows are “modal”, which means that you can have only one open at a time and you must close it before continuing with anything else.

A Further Information point looks like this 

Probes

The practical diagram has probes on it, which start in default positions. These determine where on the hardware the signals are being monitored.

Selecting and Moving the Probes

Probes are indicated by the coloured icons like this .

If this probe is the *selected probe* it then looks like this  (notice the black top to the probe). You select a probe by left clicking on it.

Monitor points look like this 

If you place the mouse over a monitor point a tool-tip will show a description of what signal it is.

You can move the selected probe by simply clicking on the required monitor point. If you want to move the probe again you do not have to re-select it. To change which probe is selected click on the probe you want to select.

You can also move a probe by the normal ‘drag-and-drop’ method, common to ‘Windows’ programs.

Probes and Test Equipment Traces

The association between probes and traces displayed on the test equipment is by colour. Data from the blue probe is displayed as a blue trace. Yellow, orange and green probes and traces operate in a similar way. Which piece of test equipment is allocated to which probe is defined by the practical.

Note that the phasescope shows the relative phase and magnitude of the signal on its input probe using another probe as the reference. The reference probe colour is indicated by the coloured square to the top left corner of the phasescope display.

Practical Buttons

On some Practicals there are buttons at the bottom of the diagram that select some parameter in the practical. These can be single buttons or in groups. Only one of each button in a group may be selected at one time.

Slider Controls

Where slider controls are used you may find you can get finer control by clicking on it and then using the up and down arrow keys on your keyboard.

Active Low-Pass Filters

Objectives

To become familiar with the operation of Active Low-Pass Filters

To appreciate the differences in response between Butterworth and Chebyshev low pass filters

To investigate connecting filters in series

More information on Filters

Filters are devices that pass signals of certain frequencies and block others.

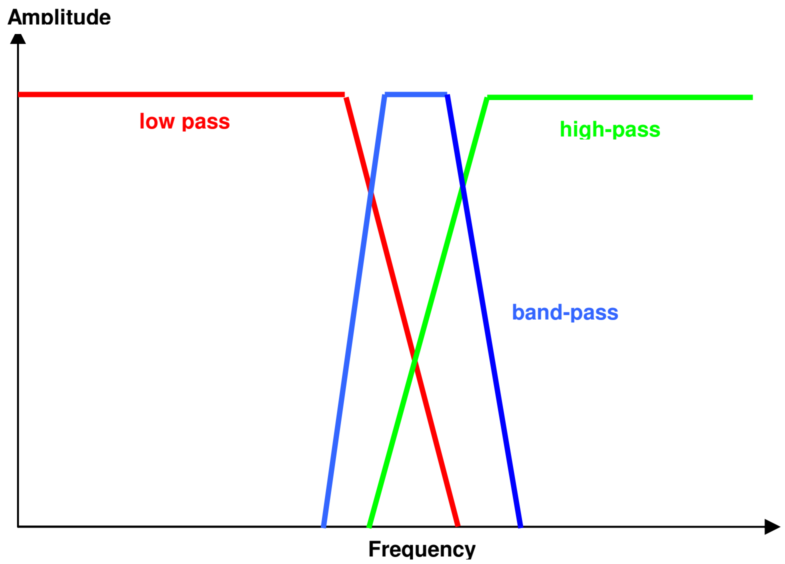
They can be very simple or complex. Until relatively recently, most filters were analogue devices but, with the increased power of digital signal processing (DSP), many filters are now implemented digitally. The mathematics of analogue and digital filters are essentially the same.

Sometimes, analogue filters use only passive components: like capacitors and inductors. However, many operating at low frequency use active components: like operational amplifiers.

Filters divide into several categories depending on the ranges of frequencies they pass. The most common is called a low-pass filter, which passes all signals up to a certain frequency. This frequency is called the cut-off frequency.

A high pass filter only passes signals above its cut-off frequency.

The third type is called a band-pass filter and passes signals between two limits.



As you can see from the diagram, the response of the filter does not fall to zero immediately at the cut-off frequency. The steepness of the response, called the roll-off, is determined by the complexity of the filter. Non digital filters are ultimately limited by losses in the components. Digital filters can have almost ideal responses.

Other considerations in filter design can be:
their input and output impedances,
pass-band loss,
pass-band ripple,

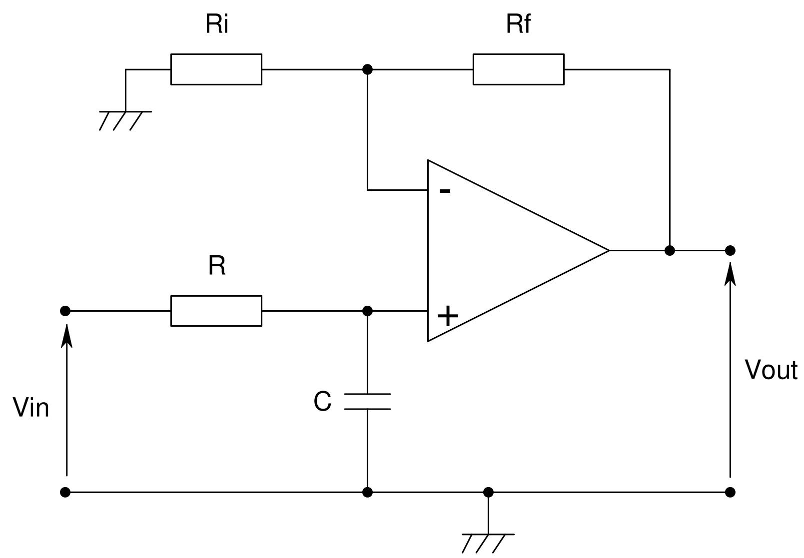
signal delay and
phase response.

The design of both analogue and digital filters is a complex subject that has been made somewhat easier by computer simulation.

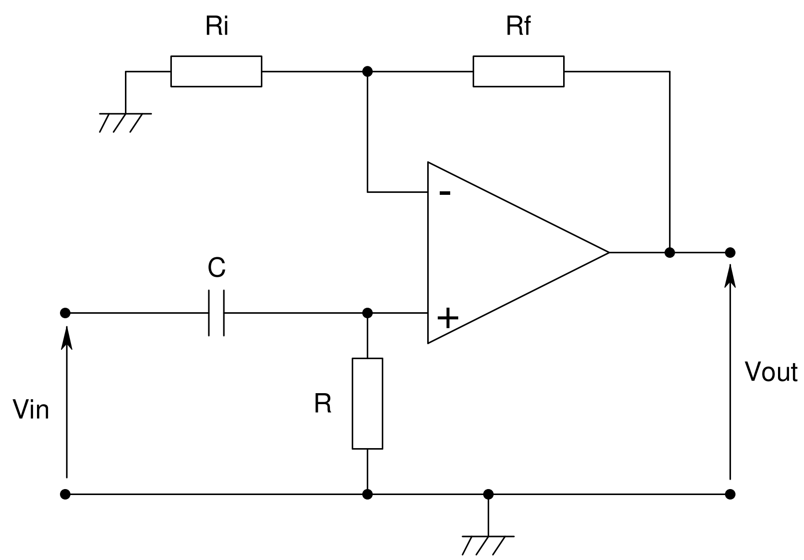
Active Filters

Passive filters have the disadvantage that source and load impedances affect their performance with respect to cut-off and attenuation. The simplest form of first order active filters uses an operational amplifier (Op-Amp) to buffer the RC filter network from variations in load. The response of such a filter should be very similar to that of an ideal first order passive filter.

The circuits of first order active filters for low and high-pass responses are given in the diagram below.



Low-pass



High-pass

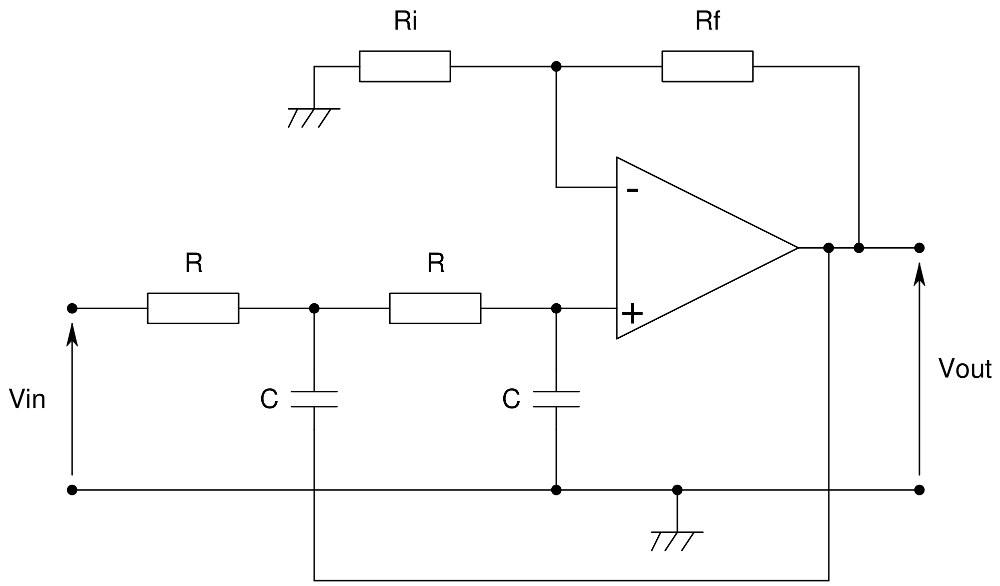
You can notice from these circuits that they comprise a passive first order RC filter followed by an Op-Amp. Note that the Op-Amp circuit is a non-inverting one and that the gain is set by resistors R_f and R_i. For a non-inverting Op-Amp circuit, the gain is given by:

$$\text{gain} = (1 + R_f/R_i)$$

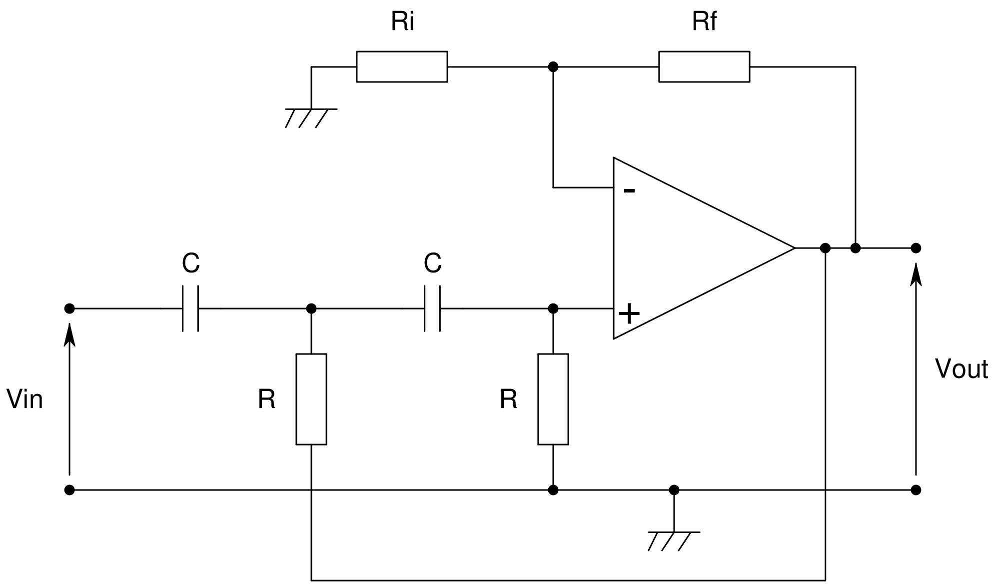
The theory and mathematics for the filters are the same as described for first order passive filters, except that the output is amplified by the Op-Amp. The input impedance of the Op-Amp is extremely high and its output impedance very low, so it acts as a very good buffer circuit for the filter.

Often it is desirable to have a filter response that rolls off more steeply than that achievable with a first order filter. Two first order filters may be connected in series to get a steeper roll-off; however, a more common way of achieving the same result is by using a second order filter circuit.

The circuits of second order active filters for low and high-pass responses are given in the diagrams below.



Low-pass



High-pass

Notice that these second order circuits comprise two passive first order RC networks with feedback from the Op-Amp. This type of circuit is known as a ‘Sallen-Key’ filter, named after its originators. As before, the Op-Amp circuit is a non-inverting one and the gain is set by R_f and R_i resistors. The gain is given by:

$$\text{gain} = (1 + R_f/R_i)$$

Because the two RC filter sections are effectively in series the roll-of of the filter will be at twice the rate of a first-order circuit.

For equal values of R and C in the two sections of the filter, the cut-off frequency for such a circuit is given by $1/2\pi RC$

As well as providing gain and feedback, the input impedance of the Op-Amp is extremely high and its output impedance very low, so it acts as a very good buffer circuit for the filter.

The two active filters on the workboard are based on the low-pass circuit shown above, but have two such stages in series. This gives a fourth-order response for each filter. The operational amplifiers are connected for unity gain.

Practical 1: Butterworth Active Low-Pass Filter

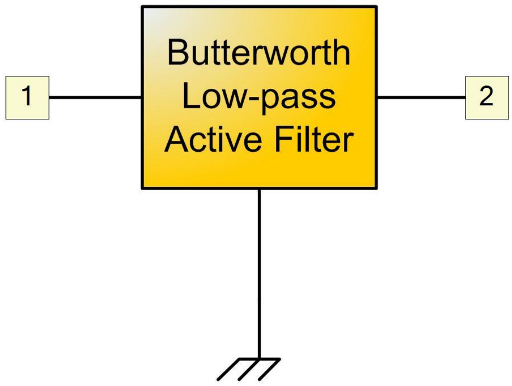
Objectives and Background

The Butterworth filter is the filter type that has the flattest passband magnitude response.

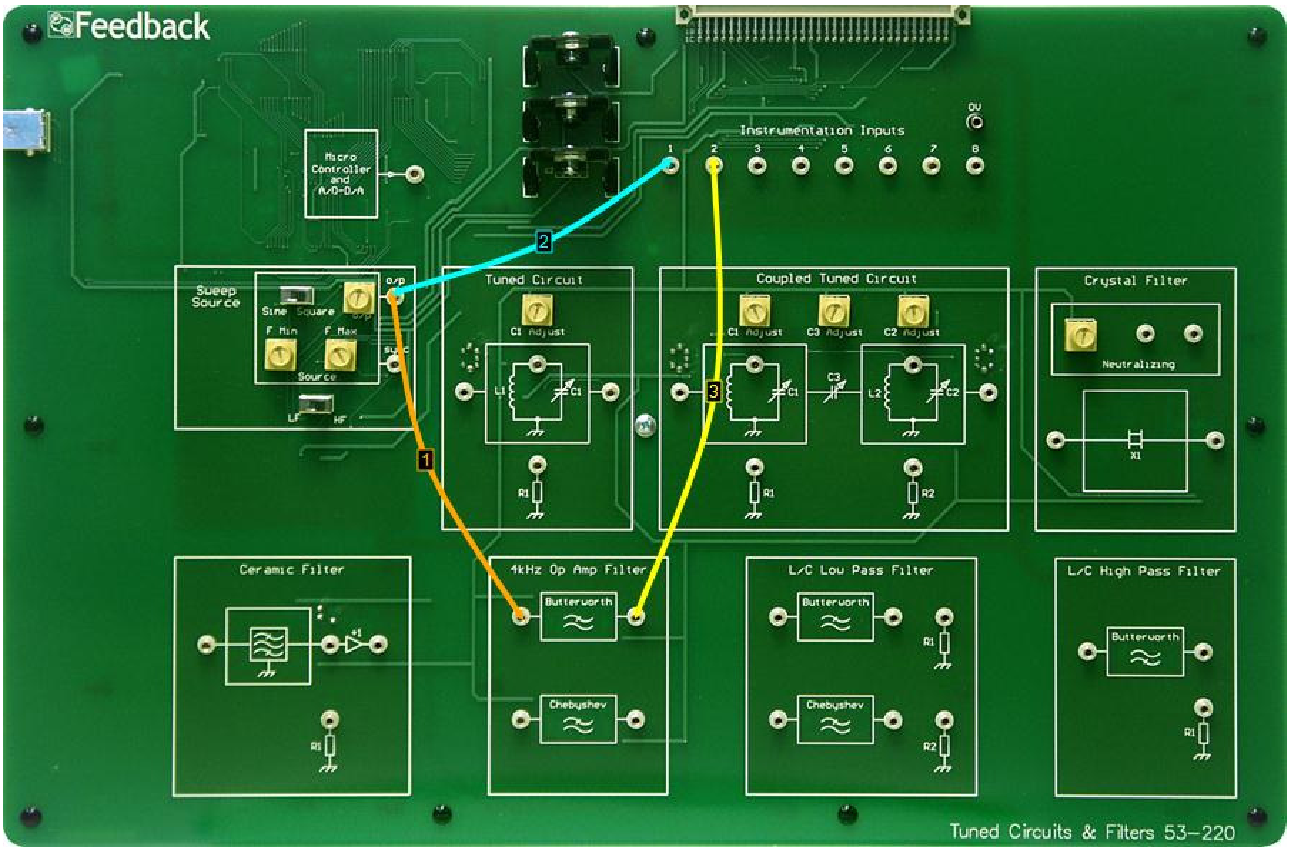
In this Practical you will investigate an active, fourth order, Butterworth filter. You will use the Gain Phase Analyser (GPA) to display both Bode and Nyquist plots of the filter response.

You will measure cut-off frequency of the filter and the rate of attenuation (the roll-off) of the filter in the stopband and relate this to the order of the filter.

Block Diagram



Make Connections Diagram



Practical 1: Butterworth Active Low-Pass Filter

Perform Practical

Use the **Make Connections** diagram to show the required connections on the hardware.

Identify the **4kHz Op Amp Filter** circuit block, located to the left of centre at the bottom of the workboard. Within this block, identify the **Butterworth** filter.

Identify the **CH1** and **CH2** gain switches, located towards the top of the workboard, directly below the **Instrumentation Input** sockets.

Set both CH1 and CH2 switches to **Hi Gain**.

Identify the **Sweep Source** circuit block at the left-hand centre of the workboard. In this block, set the **Sine/Square** switch to Sine and the **LF/HF** switch (just below the Source block, to LF. Set both the **FMin** and **FMax** control to their minimum (fully counter-clockwise) positions. This will set the source frequency to approximately 400Hz.

Set the **o/p** control in the Sweep Source circuit block to maximum (fully clockwise).

Open the Gain-Phase Analyser (GPA). Click on the **Set Max Freq** button on the GPA and use the FMax control on the Sweep Source to set the frequency to approximately 15kHz.

Click on **Plot** on the GPA to plot the Bode response of the filter.

Click **Hi Res** on the GPA (at the bottom centre of the display) to achieve a higher resolution plot.

Notice the flatness of the amplitude response in the passband and the way that the phase changes with frequency. The abrupt change from +180 to –180 degrees on the plot is due to the fact that the plotter cannot distinguish between these two points.

Now click **Nyquist** on the GPA (at the bottom right of the display) and see that the phase does, in fact, smoothly progress through more than 180 degrees.

Use the cursor on the GPA display to measure the phase shift at the frequency at which the magnitude response is –3dB.

Deselect Nyquist, to return to the Bode plot mode.

Measure the rate at which the amplitude decreases above the cut-off of the filter by measuring the attenuation at 5kHz and then at 10kHz. Between these frequencies the roll-off should be relatively linear. Quote this rate of roll-off in dB/octave (doubling of frequency) and also in dB/decade (x10 in frequency). Note that 12dB/octave is nominally 20dB/decade.

This rate should correspond to the number of poles in the filter on the workboard (see the Concept on Active Filters).

Practical 2: Chebyshev Active Low-Pass Filter

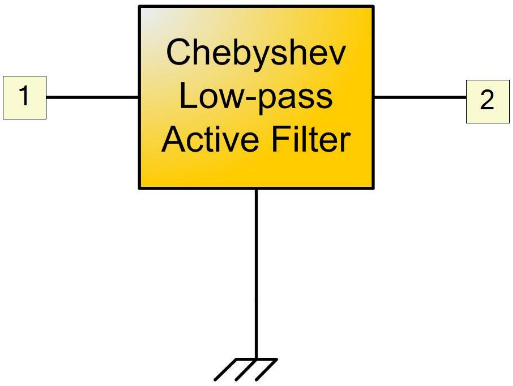
Objectives and Background

The Chebyshev filter has a amplitude response that has a greater rate of roll-off than the Butterwort type; however, it is less flat within the passband.

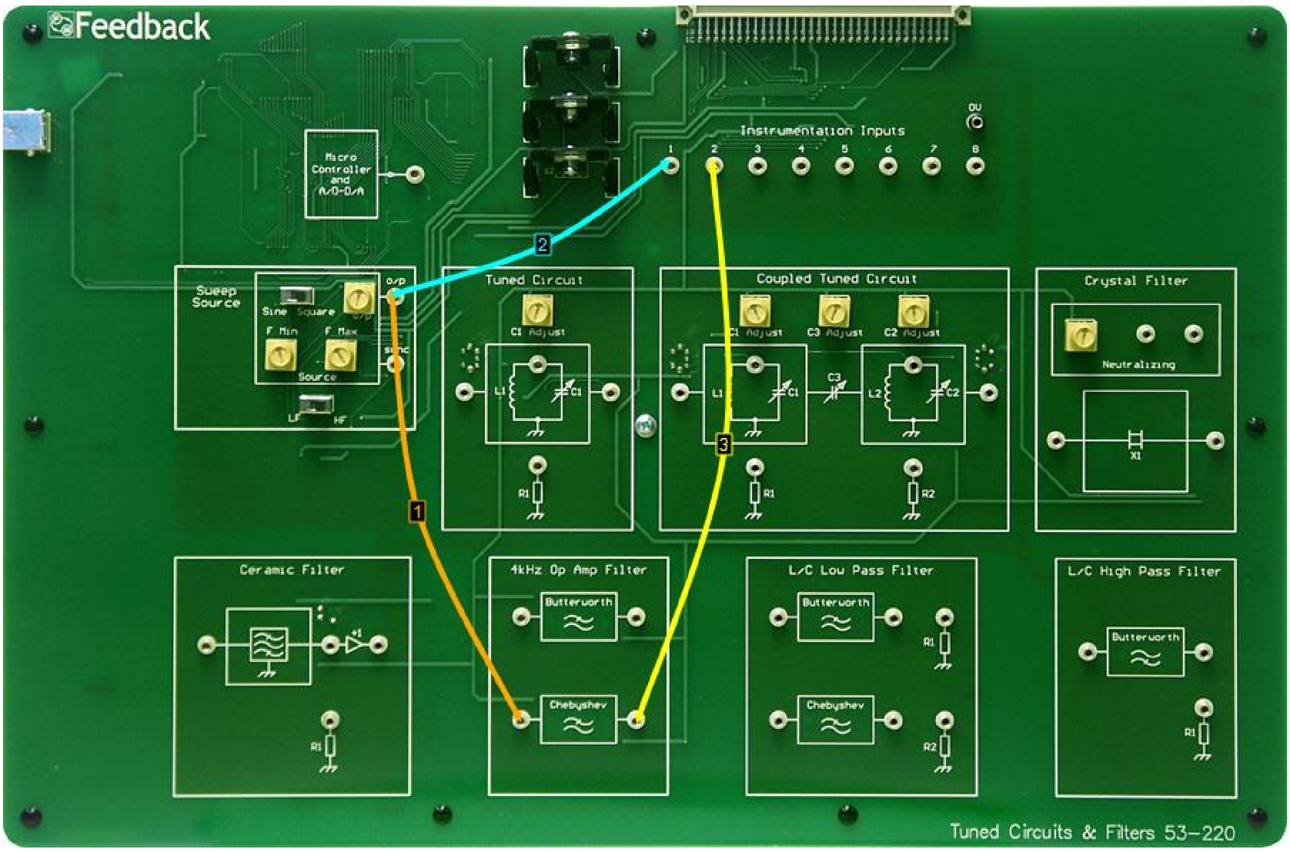
In this Practical you will investigate an active, fourth order, Chebyshev filter, using the Gain Phase Analyser (GPA) to display both Bode and Nyquist plots of the filter response.

You will see the typical passband ripple associated with this type of filter and you will measure the cut-off frequency and the rate of attenuation (the roll-off) of the filter in the stopband and compare these with those found in Practical 1, for the Butterworth type.

Block Diagram



Make Connections Diagram



Practical 2: Chebyshev Active Low-Pass Filter

Perform Practical

Use the **Make Connections** diagram to show the required connections on the hardware.

Within the **4kHz Op Amp Filter** circuit block, identify the **Chebyshev** filter.

Ensure that both **CH1** and **CH2** switches are set to **Hi Gain**.

In the **Sweep Source** block, ensure that the **Sine**/**Square** switch is set to Sine and the **LF**/**HF** switch (just below the Source block) to LF and both the **FMin** and **FMax** control are set to their minimum (fully counter-clockwise) positions.

Also, set the **o**/**p** control in the Sweep Source circuit block to maximum (fully clockwise).

Open the Gain Phase Analyser (GPA). Click on the **Set Max Freq** button on the GPA and use the FMax control on the Sweep Source to set the frequency to approximately 15kHz.

Click on **Plot** on the GPA to plot the Bode response of the filter.

Click **Hi Res** on the GPA (at the bottom centre of the display) to achieve a higher resolution plot.

Notice the ripple in the amplitude response in the passband. Now click **Nyquist** on the GPA (at the bottom right of the display) and see that the phase does, in fact, smoothly progress through more than 180 degrees.

Use the cursor to measure the phase shift at the frequency at which the magnitude response is –3dB.

Deselect Nyquist, to return to the Bode plot mode.

Measure the rate at which the amplitude decreases above the cut-off of the filter by measuring the attenuation at 5kHz and then at 10kHz. Between these frequencies the roll-off should be relatively linear. Quote this rate of roll-off in dB/octave (doubling of frequency) and also in dB/decade (x10 in frequency). Note that 12dB/octave is nominally 20dB/decade.

This rate should be greater than that for the Butterworth filter, seen in Practical 1 (see the Concept on Filters).

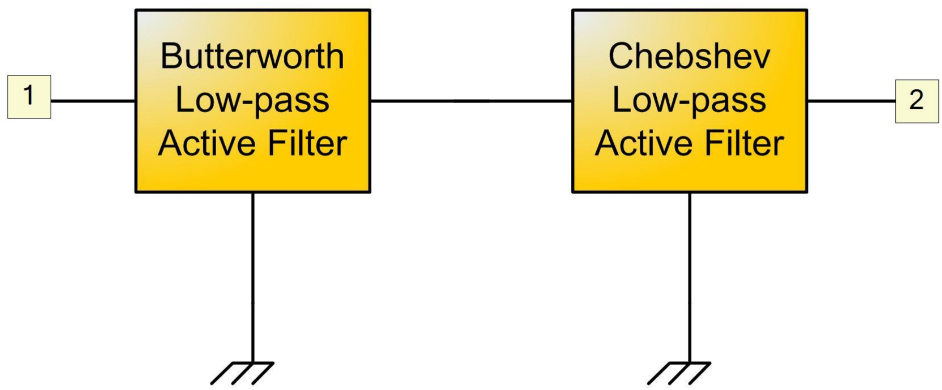
Practical 3: Higher Order Active Filter

Objectives and Background

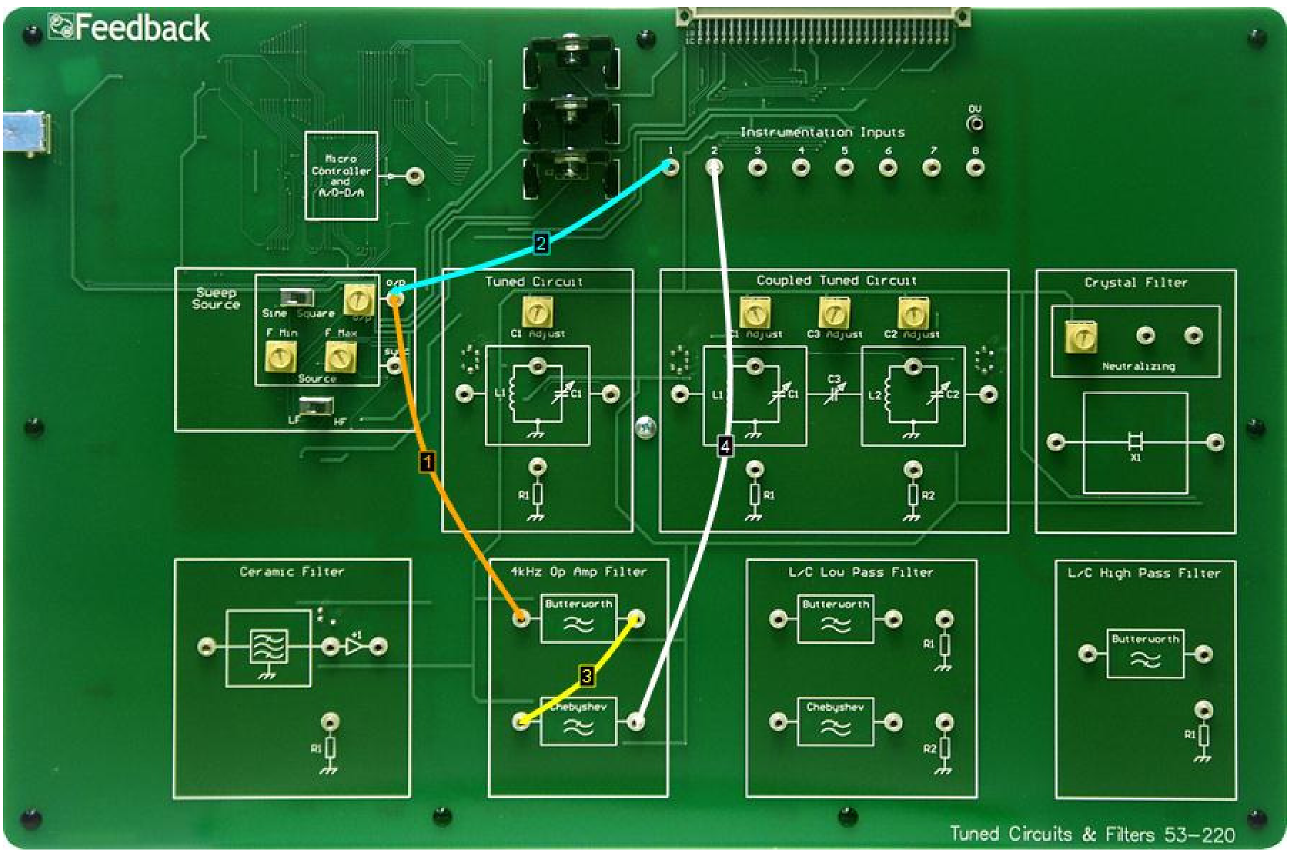
In this Practical you will connect together, in series, the Butterworth and Chebyshev filters that you investigated in Practicals 1 and 2. As each filter is of the fourth order, this effectively makes an eighth order combination.

You will measure the cut-off frequency and the rate of roll-off and you will see that such a combination has a high rate of roll-off whilst maintaining the passband response of the individual filters.

Block Diagram



Make Connections Diagram



Practical 3: Higher Order Active Filter

Perform Practical

Use the **Make Connections** diagram to show the required connections on the hardware.

Keep the switch and control settings the same as for Practical 2.

Open the Gain Phase Analyser (GPA). Click on the **Set Max Freq** button on the GPA and use the **FMax** control on the Sweep Source to set the frequency to approximately 10kHz.

Click on **Plot** on the GPA to plot the Bode response of the filter.

Click **Hi Res** on the GPA (at the bottom centre of the display) to achieve a higher resolution plot.

Notice there is still some ripple in the amplitude response in the passband (due to the Chebyshev part of the circuit). Also, that the phase shift goes through two –180 to +180 degree cycles.

Click **Nyquist** on the GPA (at the bottom right of the display) and see that the phase does, in fact, smoothly progress.

Use the cursor to measure the phase shift at the frequency at which the magnitude response is –3dB.

Deselect Nyquist, to return to the Bode plot mode.

Measure the rate at which the amplitude decreases above the cut-off of the filter by measuring the attenuation at 5kHz and then at 10kHz. Between these frequencies the roll-off should be relatively linear. Quote this rate of roll-off in dB/octave (doubling of frequency) and also in dB/decade (x10 in frequency). Note that 12dB/octave is nominally 20dB/decade.

This rate should be greater than that for either of the individual filters seen in Practicals 1 and 2.

LC Low Pass Filters

Objectives

To become familiar with the operation of LC Low Pass Filters

To appreciate the differences in response between Butterworth and Chebyshev low pass filters

To investigate the effect on the response of terminating filters with their characteristic resistance

To investigate connecting filters in series

More information on Filters

Filters are devices that pass signals of certain frequencies and block others.

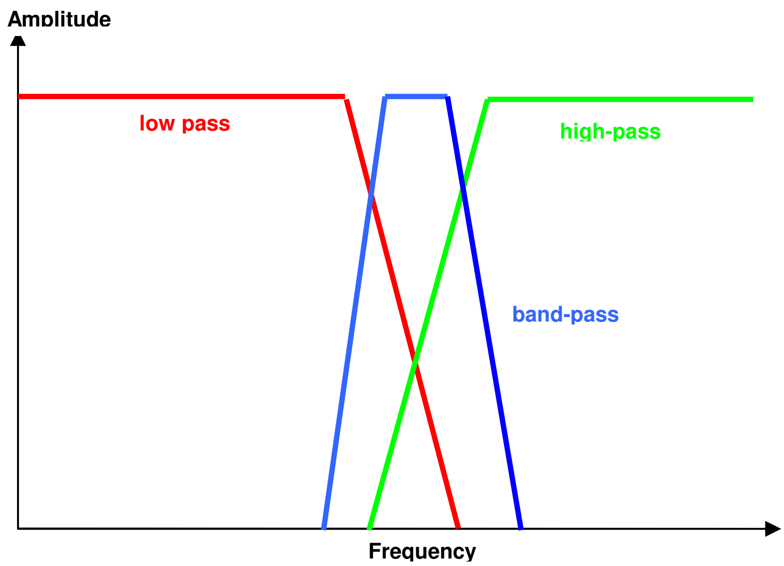
They can be very simple or complex. Until relatively recently, most filters were analogue devices but, with the increased power of digital signal processing (DSP), many filters are now implemented digitally. The mathematics of analogue and digital filters are essentially the same.

Sometimes, analogue filters use only passive components: like capacitors and inductors. However, many operating at low frequency use active components: like operational amplifiers.

Filters divide into several categories depending on the ranges of frequencies they pass. The most common is called a low-pass filter, which passes all signals up to a certain frequency. This frequency is called the cut-off frequency.

A high pass filter only passes signals above its cut-off frequency.

The third type is called a band-pass filter and passes signals between two limits.



As you can see from the diagram, the response of the filter does not fall to zero immediately at the cut-off frequency. The steepness of the response, called the roll-off, is determined by the complexity of the filter. Non digital filters are ultimately limited by losses in the components. Digital filters can have almost ideal responses.

Other considerations in filter design can be:
their input and output impedances,
pass-band loss,
pass-band ripple,

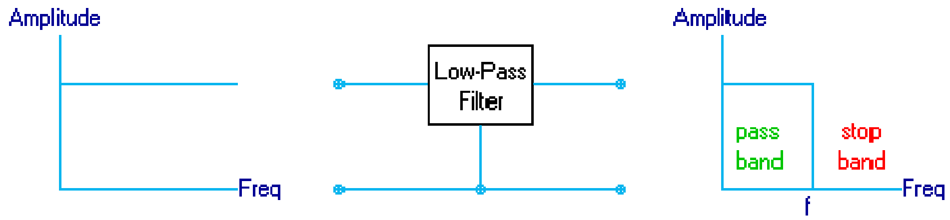
signal delay and
phase response.

The design of both analogue and digital filters is a complex subject that has been made somewhat easier by computer simulation.

LC Filters

A filter is a circuit which passes signals of some frequencies freely, but attenuates at other frequencies. This assignment is about low-pass filters.

Low-pass Filters



An ideal low-pass filter passes signals of all frequencies below a certain value and does not pass signals of frequencies above that value.

The frequency at which the signal starts to be attenuated is called the **cut-off frequency**. Since the cut-off is not quite sharp, the exact value must be based on some measure of how much the signal is reduced. This is typically 3 dB below the level in the pass band.

To pass some frequencies and reject others, a filter must contain components that have impedances that vary with frequency. This means capacitors and/or inductors must be used, often together with resistors whose impedance (resistance) does not vary with frequency.

If the filter does not have any active devices (transistors, IC's, etc.) in its circuit, it is called a **Passive Filter**.

Filters can be classified as passive or active. The assignment includes one filter of each class. **'Active' implies that the circuit contains amplifying devices such as transistors or op-amps. 'Passive' means having no such devices**.

Passive Filters

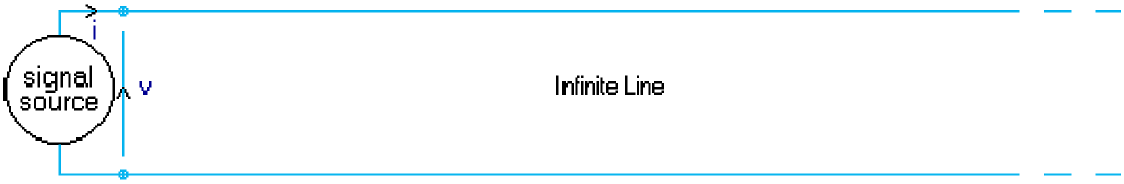
Passive filters consist of networks of inductors and capacitors. They have the advantage of working without power supplies. Their disadvantages are mainly apparent at low frequencies, for which both inductors and capacitors need to be large and expensive. Inductors also tend to have appreciable resistance, making the action of the filter imperfect.

Active Filters

Passive filters consist of networks of inductors and capacitors. They have the advantage of working without power supplies. Their disadvantages are mainly apparent at low frequencies, for which both inductors and capacitors need to be large and expensive. Inductors also tend to have appreciable resistance, making the action of the filter imperfect.

Transmission Line Characteristic Impedance

The concept of **characteristic impedance** first arose in the theory of transmission lines for telegraph purposes (they are now used for carrying RF and other signals).

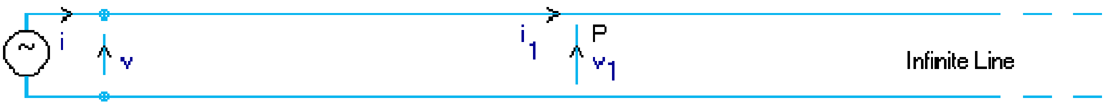


Imagine an infinite transmission line. In practice this means a line so long that a signal takes so long to get to the far end that it virtually never comes back.

Now suppose that we wish to send a signal of voltage v along the line. (v can be a function of time, like V_p sin wt). If that signal is applied at one end of the line, a current i will flow into the line.

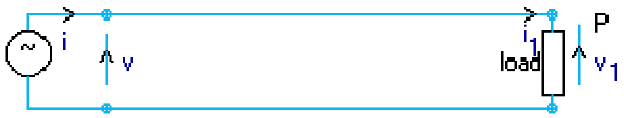
The characteristic impedance, often denoted by Z_0, is then defined as:

$$\mathbf{Z_0 = v / i.}$$



Now let us look at the voltage and current at a point P, some way down the line.

We find them to be v_1 and i_1 respectively. The line remaining is still infinite, so the ratio of voltage to current must again be Z_0, and we can see that Z_0 is also equal to v_1 / i_1.



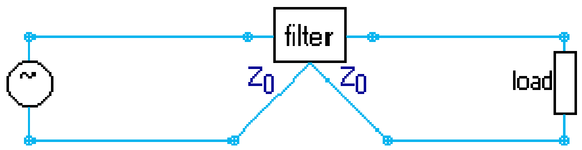
This means that we can cut the line at P and connect an impedance Z_0 at point P without altering the behaviour of the first section of line at all.

The line is now of finite length and has been terminated with Z_0. It is said to be **matched to Z_0, the Characteristic Impedance of the line**.

The important point to be made about matching the load to the characteristic impedance of the line is this :

A properly matched load absorbs all the power which arrives at it, whereas a mismatched load causes 'spare' power to be reflected back towards the signal source.

Filter Matching

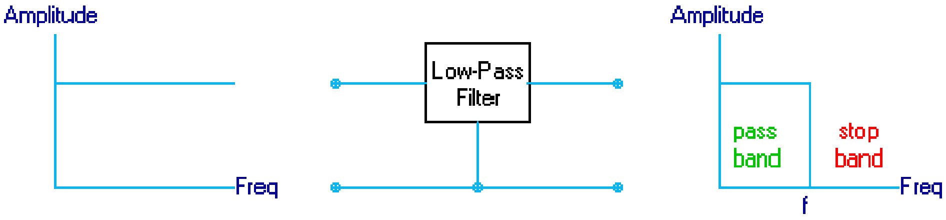


If a filter is inserted into a line, ideally it should have an impedance equal to the line's characteristic impedance at both pairs of terminals. If this condition is realised then the filter causes no reflections.

In practice a resistive load is never a perfect match for a filter at all frequencies, so some reflection always takes place.

Passive Low-Pass Filters

A filter is a circuit which passes signals in a selected range of frequencies freely (the '**passband**'), but attenuates at all other frequencies (the '**stopband**').

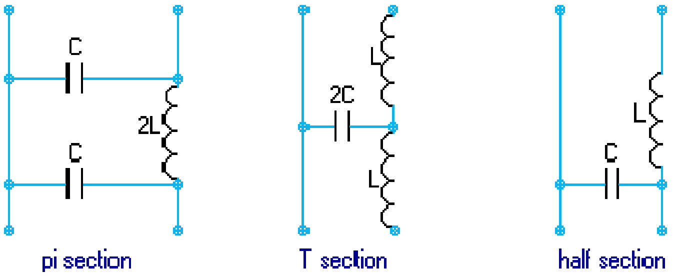


An ideal Low-Pass Filter lets through signals lower than a certain frequency without attenuating them, but does not let through any signal with a frequency above that value. The frequency above which no signals are passed is called the **cut-off frequency**.

A real filter cannot have a response that changes instantly from zero attenuation to infinite attenuation at a certain frequency. In a practical low pass filter the response slopes off at higher frequencies, and the cut-off frequency is defined as being that at which the signal has been attenuated by 3 dB.

To make the filter pass some frequencies and reject others, it must contain components whose impedances vary with frequency; ie, capacitors and inductors.

If the filter does not have any active, amplifying devices in its circuit it is called a Passive Filter. Resistors cannot be used in a passive filter because they would introduce attenuation.



The theory of filters is too complex to explain here. But in simple terms, an elementary low-pass filter has inductance in series with the signal path and capacitance in parallel with it.

As the diagram shows, a filter section can be arranged in a pi or a T form. Either form can be constructed from two identical half-sections.

The **cut-off frequency, ω rad/s**, for any of these filters is given by

$\omega^2 LC = 1$.

The '**design impedance**', which approximates to the characteristic impedance at low frequencies, is Z where

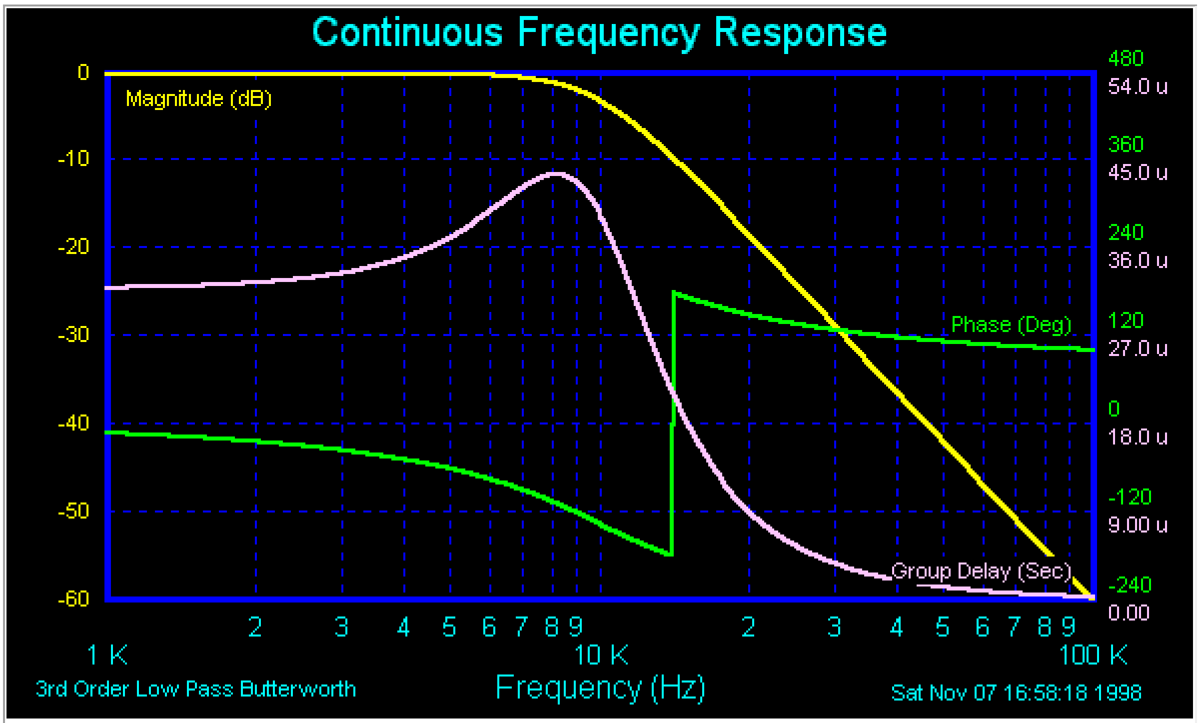
$Z^2 = L/C$

Practical 1: Butterworth LC Low-Pass Filter

Objectives and Background

The Butterworth Filter is the filter type that results in the flattest pass band and contains a moderate group delay.

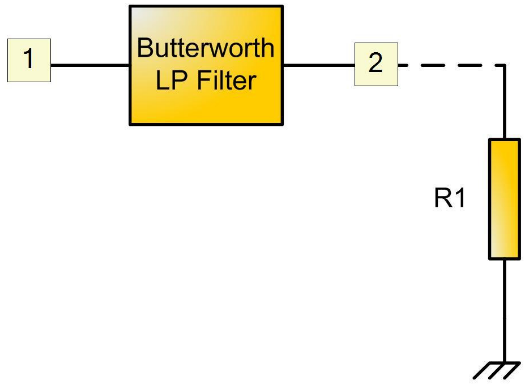
Below is an example of a Butterworth low pass filter response.



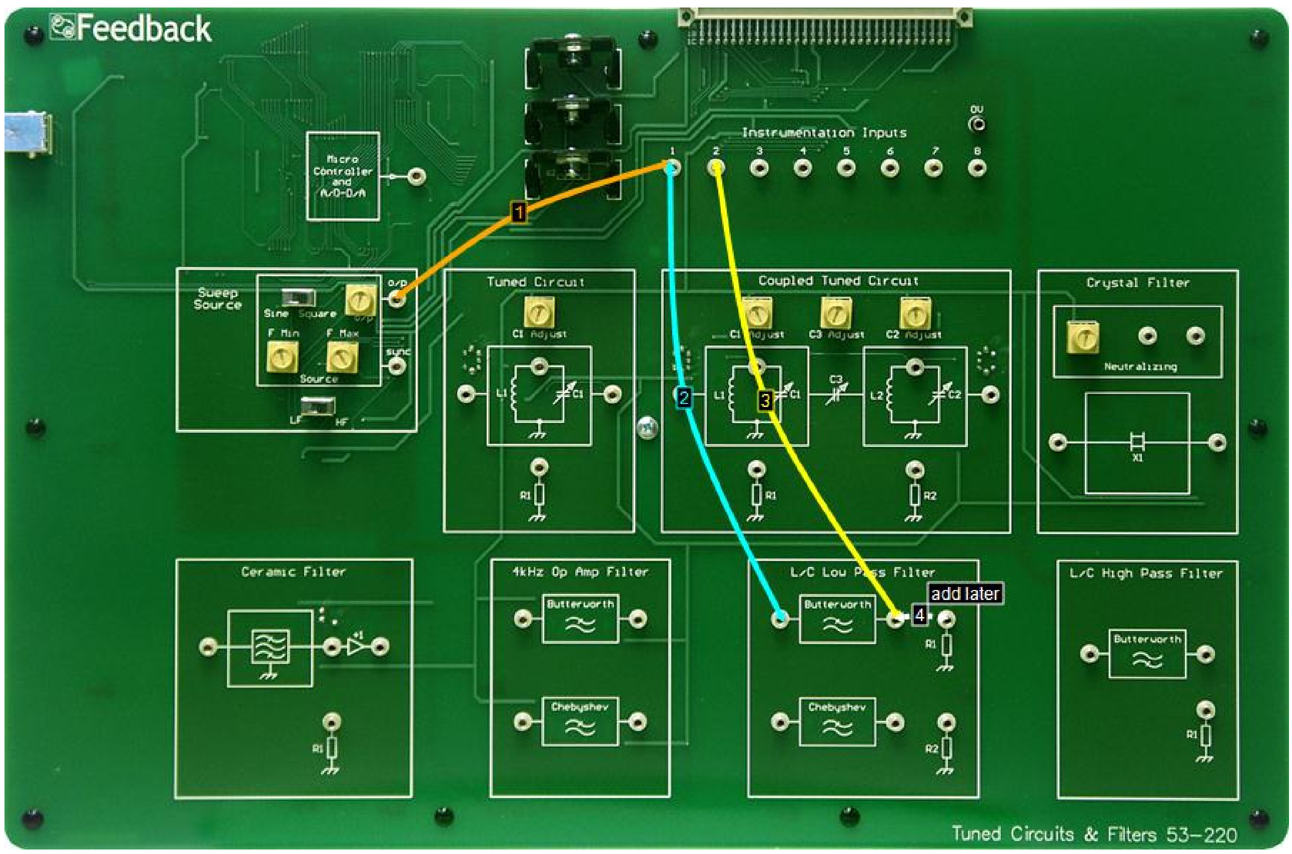
Butterworth Low Pass filter, 10kHz Pass Band Frequency

In this Practical you will measure the response of a Butterworth low pass filter, both with and without termination resistor, and you will see the typical lack of ripple and the rate of cut-off.

Block Diagram



Make Connections Diagram



Practical 1: Butterworth LC Low-Pass Filter

Perform Practical

Use the **Make Connections** diagram to show the required connections on the hardware.

Identify the **L/C Low Pass Filter** circuit block, located to the right bottom of the workboard. Within this block, identify the **Butterworth** filter.

Identify the **CH1** and **CH2** gain switches, located towards the top of the workboard, directly below the **Instrumentation Input** sockets.

Set both CH1 and CH2 switches to **Hi Gain**.

Identify the **Sweep Source** circuit block at the left-hand centre of the workboard. In this block, set the **Sine/Square** switch to Sine and the **LF/HF** switch (just below the Source block, to HF. Set both the **FMin** and **FMax** control to their minimum (fully counter-clockwise) positions. This will set the source frequency to approximately 100kHz.

Open the Gain Phase Analyser (GPA). Click on the **Set Max Freq** button on the GPA and use the FMax control on the Sweep Source to set the frequency to 6MHz.

Click on **Plot** on the GPA to plot the Bode response of the filter.

Click **Hi Res** on the GPA (at the bottom centre of the display) to achieve a higher resolution plot.

Notice the peak of the amplitude response and the way that the phase changes with frequency. The abrupt change from +180 to –180 degrees on the plot is due to the fact that the plotter cannot distinguish between these two points.

Now click **Nyquist** on the GPA (at the bottom right of the display) and see that the phase does, in fact, smoothly progress through more than 360 degrees.

Deselect Nyquist, to return to the Bode plot mode, and connect the terminating resistor **R1**, as shown by connection 4 on the Make Connections diagram.

Notice the difference in response made by adding the terminating resistor. Note the effect on the peaking and on the passband amplitude of the output. This is because there is now a potential divider at the output between the filter's characteristic output impedance and the equal value terminating resistance. Correctly terminating a filter may improve its response, but will always lead to a loss in amplitude.

Measure the rate at which the amplitude decreases above the cut-off of the filter. Also measure the rate of phase change with frequency for the filter.

Practical 2: Chebychev LC Low-Pass Filter

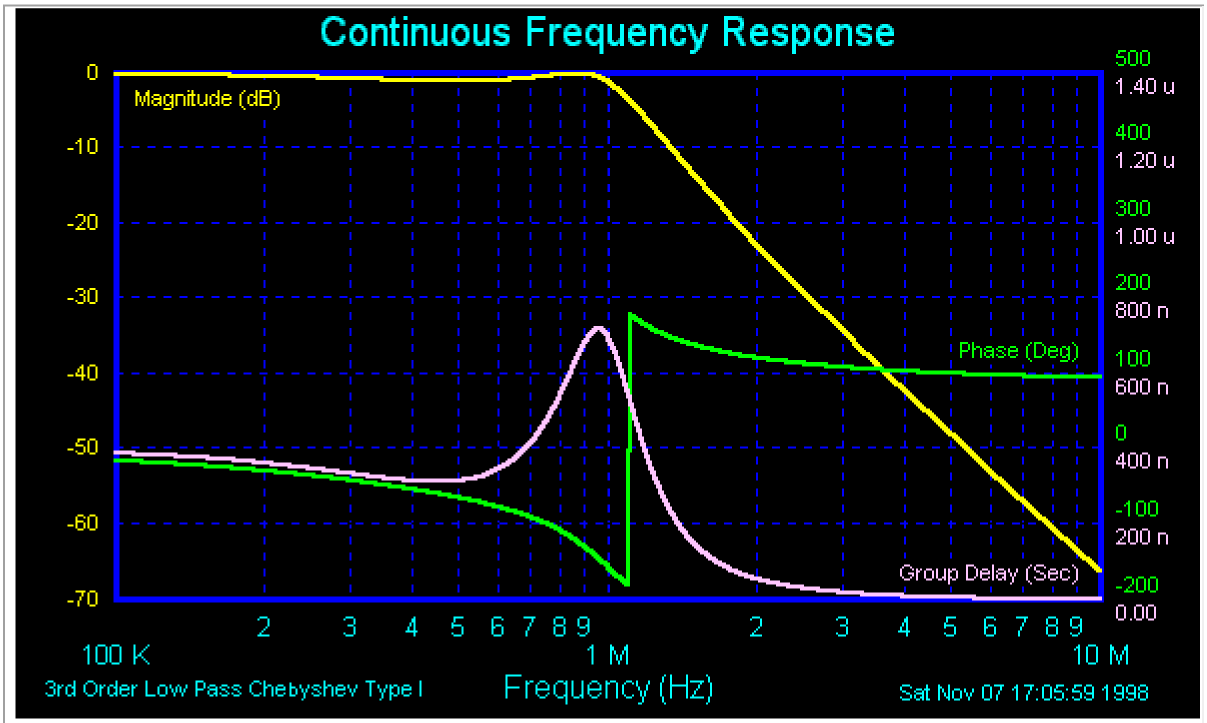
Objectives and Background

The Chebyshev Filter is the filter type that results in the sharpest pass band cut off and contains the largest group delay. The most notable feature of this filter is the ripple in the pass band magnitude.

A standard Chebyshev Filter's pass band attenuation is defined to be the same value as the pass band ripple amplitude. However, Filter Solutions allows the user the option of selecting any pass band attenuation in dB's that will define the filters cut off frequency.

Filter Solutions also offers the user the option of placing user-defined zeros in the stop band.

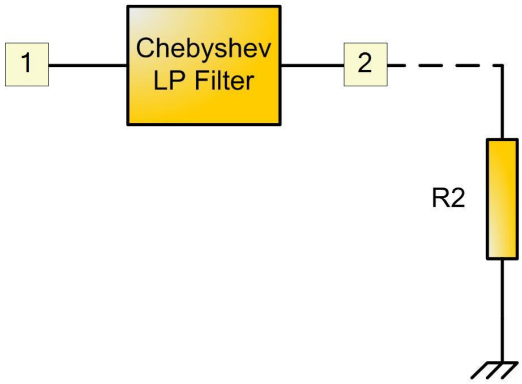
Below is an example of a Chebyshev low pass filter response.



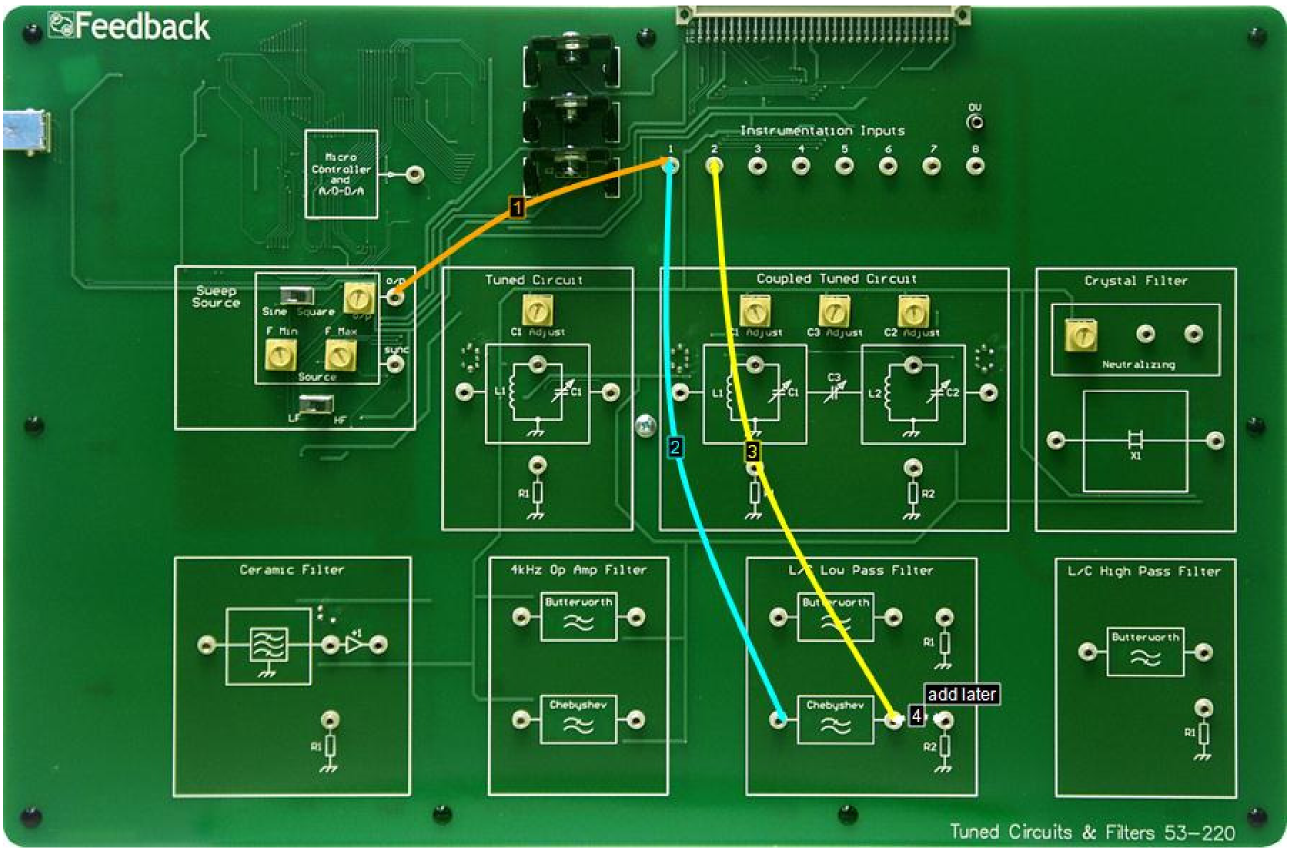
Chebyshev Type I Low Pass filter, 1MHz Pass Band Frequency

In this Practical you will measure the response of a Chebyshev low pass filter, both with and without termination resistor, and you will see the typical ripple and the faster rate of cut-off of this type of filter.

Block Diagram



Make Connections Diagram



Practical 2: Chebyshev LC Low-Pass Filter

Perform Practical

Use the **Make Connections** diagram to show the required connections on the hardware.

Identify the **Chebyshev** filter within **L/C Low Pass Filter** block.

In the **Sweep Source** circuit block, set the **Sine/Square** switch to Sine and the **LF/HF** switch (just below the Source block, to HF. Set both the **FMin** and **FMax** control to their minimum (fully counter-clockwise) positions. This will set the source frequency to approximately 100kHz.

Open the Gain Phase Analyser (GPA). Click on the **Set Max Freq** button on the GPA and use the FMax control on the Sweep Source to a frequency of 6MHz.

Click on **Plot** on the GPA to plot the Bode response of the filter.

Click **Hi Res** on the GPA (at the bottom centre of the display) to achieve a higher resolution plot.

Notice the ripple on the amplitude response and the way that the phase changes with frequency. The abrupt change from +180 to –180 degrees on the plot is due to the fact that the plotter cannot distinguish between these two points.

Now click **Nyquist** on the GPA (at the bottom right of the display) and see that the phase does, in fact, smoothly progress through more than 360 degrees.

Deselect Nyquist, to return to the Bode plot mode, and connect the terminating resistor **R1**, as shown by connection 4 on the Make Connections diagram.

Notice the difference in response made by adding the terminating resistor. Note that the ripple is still evident. This ripple is a characteristic of a Chebyshev filter.

Also notice the effect on the passband amplitude of the output. This is because you have connected the terminating resistor.

Measure the rate at which the amplitude decreases above the cut-off of the filter. Also measure the rate of phase change with frequency for the filter. Compare these two measurements with those found for the Butterworth filter from Practical 1.

Practical 3: Higher Order Filters

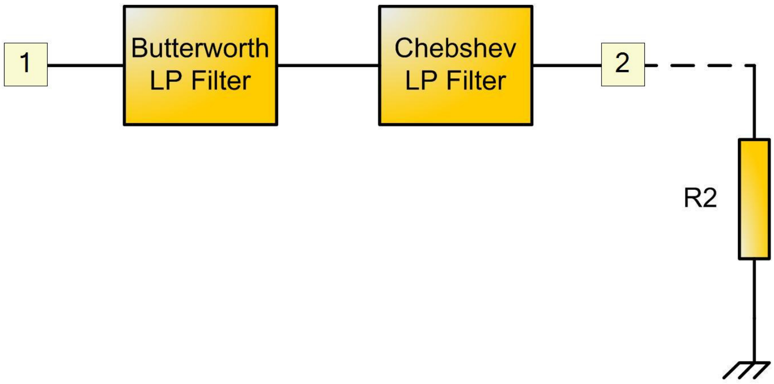
Objectives and Background

In this Practical you will connect in series the filters that you investigated in Practicals 1 and 2.

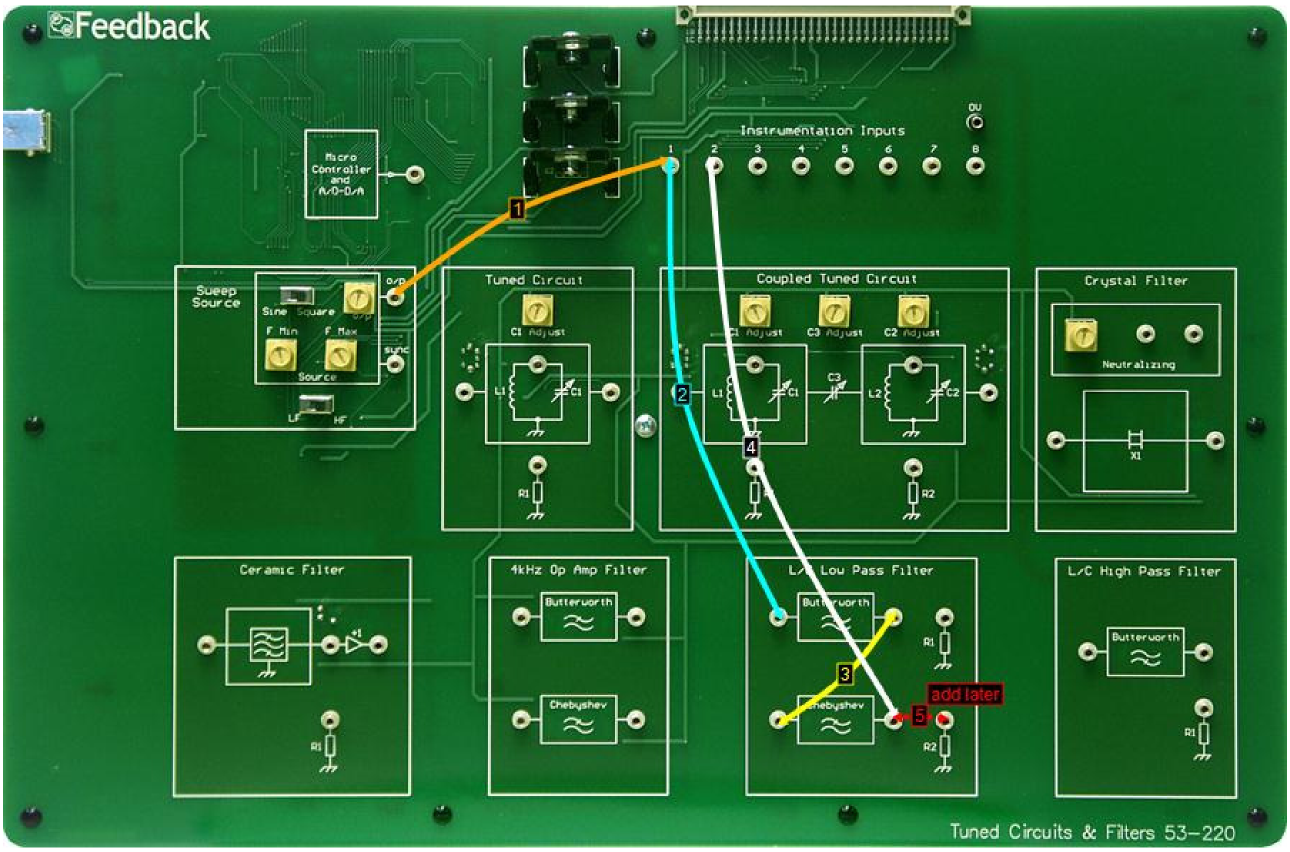
You will find that the rate of fall-off in gain in the stop band increases however the passband attenuation is greater, because of the series connection of filters.

The series combination of two filters as used in the Practical is not optimum. It would be better to design the complete filter as one entity, rather than just connecting together two, already designed, blocks. This would allow the filter component values to be optimised. However, the effects that you will see are representative of those that you would get if a correctly designed filter was used.

Block Diagram



Make Connections Diagram



Practical 3: Higher Order Filters

Perform Practical

Use the **Make Connections** diagram to show the required connections on the hardware.

In the **Sweep Source** circuit block, set the **Sine**/**Square** switch to Sine and the **LF**/**HF** switch (just below the Source block, to HF. Set both the **FMin** and **FMax** control to their minimum (fully counter-clockwise) positions. This will set the source frequency to approximately 100kHz.

Open the Gain Phase Analyser (GPA). Click on the **Set Max Freq** button on the GPA and use the FMax control on the Sweep Source to set the frequency to 6MHz.

Click on **Plot** on the GPA to plot the Bode response of the filter.

Click **Hi Res** on the GPA (at the bottom centre of the display) to achieve a higher resolution plot.

Measure the rate at which the amplitude decreases above the cut-off of the filter. Also measure the rate of phase change with frequency for the filter. Compare these two measurements with those found for the individual filters from Practicals 1 and 2.

Also notice the effect on the passband amplitude of the output.

Now click **Nyquist** on the GPA (at the bottom right of the display) and see that the phase does, in fact, smoothly progress through more than 360 degrees.

LC High Pass Filter

Objectives

To become familiar with the operation of an LC High Pass Filter

To investigate the frequency response of a Butterworth high pass filter

To investigate the effect on the response of terminating the filter with its characteristic resistance

More information on Filters

Filters are devices that pass signals of certain frequencies and block others.

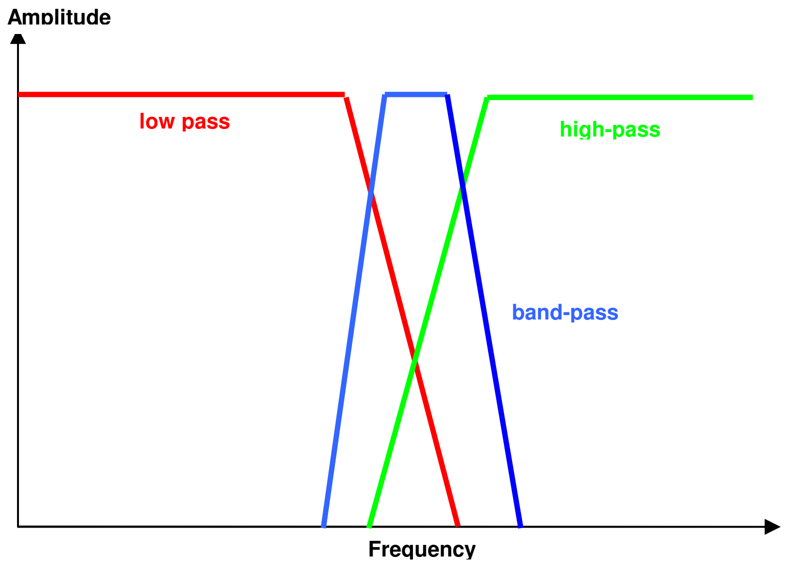
They can be very simple or complex. Until relatively recently, most filters were analogue devices but, with the increased power of digital signal processing (DSP), many filters are now implemented digitally. The mathematics of analogue and digital filters are essentially the same.

Sometimes, analogue filters use only passive components: like capacitors and inductors. However, many operating at low frequency use active components: like operational amplifiers.

Filters divide into several categories depending on the ranges of frequencies they pass. The most common is called a low-pass filter, which passes all signals up to a certain frequency. This frequency is called the cut-off frequency.

A high pass filter only passes signals above its cut-off frequency.

The third type is called a band-pass filter and passes signals between two limits.



As you can see from the diagram, the response of the filter does not fall to zero immediately at the cut-off frequency. The steepness of the response, called the roll-off, is determined by the complexity of the filter. Non digital filters are ultimately limited by losses in the components. Digital filters can have almost ideal responses.

Other considerations in filter design can be:
their input and output impedances,
pass-band loss,
pass-band ripple,

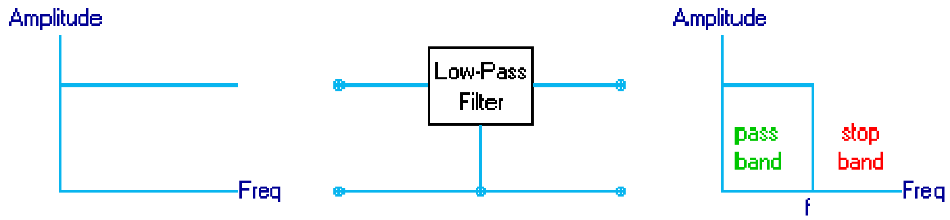
signal delay and
phase response.

The design of both analogue and digital filters is a complex subject that has been made somewhat easier by computer simulation.

LC Filters

A filter is a circuit which passes signals of some frequencies freely, but attenuates at other frequencies. This assignment is about low-pass filters.

Low-pass Filters



An ideal low-pass filter passes signals of all frequencies below a certain value and does not pass signals of frequencies above that value.

The frequency at which the signal starts to be attenuated is called the **cut-off frequency**. Since the cut-off is not quite sharp, the exact value must be based on some measure of how much the signal is reduced. This is typically 3 dB below the level in the pass band.

To pass some frequencies and reject others, a filter must contain components that have impedances that vary with frequency. This means capacitors and/or inductors must be used, often together with resistors whose impedance (resistance) does not vary with frequency.

If the filter does not have any active devices (transistors, IC's, etc.) in its circuit, it is called a **Passive Filter**.

Filters can be classified as passive or active. The assignment includes one filter of each class. **'Active' implies that the circuit contains amplifying devices such as transistors or op-amps. 'Passive' means having no such devices**.

Passive Filters

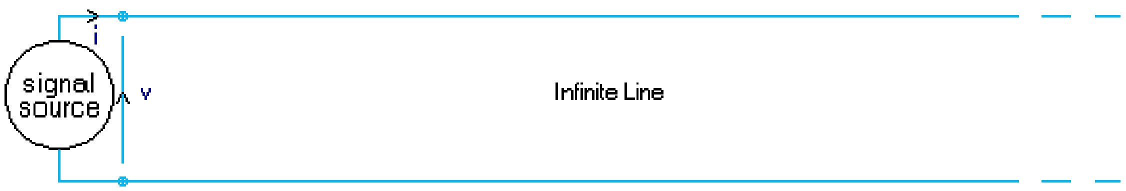
Passive filters consist of networks of inductors and capacitors. They have the advantage of working without power supplies. Their disadvantages are mainly apparent at low frequencies, for which both inductors and capacitors need to be large and expensive. Inductors also tend to have appreciable resistance, making the action of the filter imperfect.

Active Filters

Passive filters consist of networks of inductors and capacitors. They have the advantage of working without power supplies. Their disadvantages are mainly apparent at low frequencies, for which both inductors and capacitors need to be large and expensive. Inductors also tend to have appreciable resistance, making the action of the filter imperfect.

Transmission Line Characteristic Impedance

The concept of **characteristic impedance** first arose in the theory of transmission lines for telegraph purposes (they are now used for carrying RF and other signals).

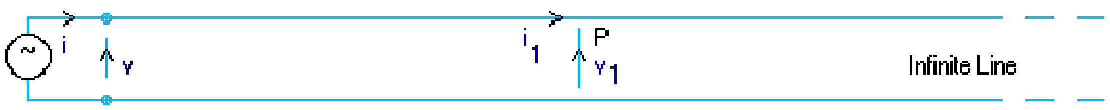


Imagine an infinite transmission line. In practice this means a line so long that a signal takes so long to get to the far end that it virtually never comes back.

Now suppose that we wish to send a signal of voltage v along the line. (v can be a function of time, like V_p sin wt). If that signal is applied at one end of the line, a current i will flow into the line.

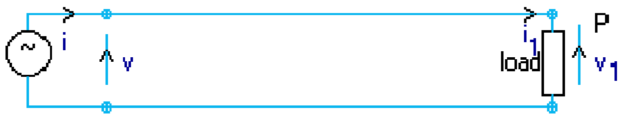
The characteristic impedance, often denoted by Z_0, is then defined as:

$$\mathbf{Z_0 = v / i.}$$



Now let us look at the voltage and current at a point P, some way down the line.

We find them to be v_1 and i_1 respectively. The line remaining is still infinite, so the ratio of voltage to current must again be Z_0, and we can see that Z_0 is also equal to v_1 / i_1.



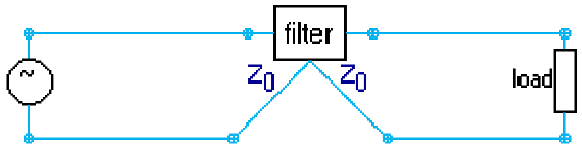
This means that we can cut the line at P and connect an impedance Z_0 at point P without altering the behaviour of the first section of line at all.

The line is now of finite length and has been terminated with Z_0. It is said to be **matched to Z_0, the Characteristic Impedance of the line**.

The important point to be made about matching the load to the characteristic impedance of the line is this :

A properly matched load absorbs all the power which arrives at it, whereas a mismatched load causes 'spare' power to be reflected back towards the signal source.

Filter Matching

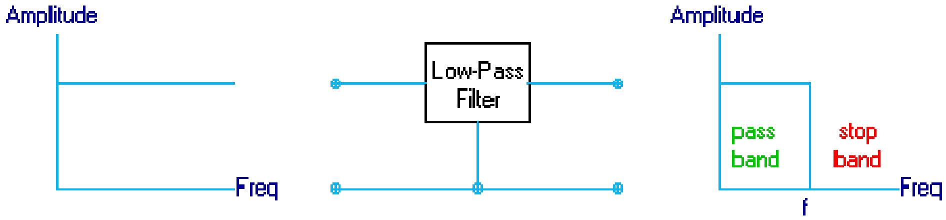


If a filter is inserted into a line, ideally it should have an impedance equal to the line's characteristic impedance at both pairs of terminals. If this condition is realised then the filter causes no reflections.

In practice a resistive load is never a perfect match for a filter at all frequencies, so some reflection always takes place.

Passive Low-Pass Filters

A filter is a circuit which passes signals in a selected range of frequencies freely (the '**passband**'), but attenuates at all other frequencies (the '**stopband**').

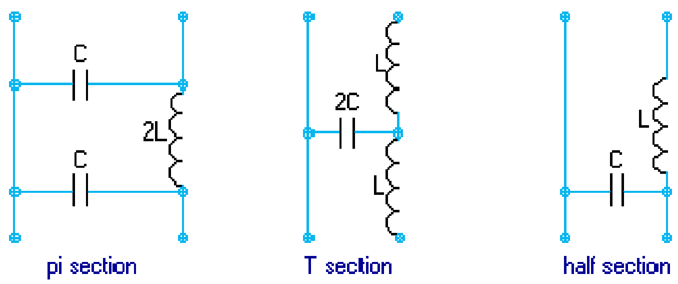


An ideal Low-Pass Filter lets through signals lower than a certain frequency without attenuating them, but does not let through any signal with a frequency above that value. The frequency above which no signals are passed is called the **cut-off frequency**.

A real filter cannot have a response that changes instantly from zero attenuation to infinite attenuation at a certain frequency. In a practical low pass filter the response slopes off at higher frequencies, and the cut-off frequency is defined as being that at which the signal has been attenuated by 3 dB.

To make the filter pass some frequencies and reject others, it must contain components whose impedances vary with frequency; ie, capacitors and inductors.

If the filter does not have any active, amplifying devices in its circuit it is called a Passive Filter. Resistors cannot be used in a passive filter because they would introduce attenuation.



The theory of filters is too complex to explain here. But in simple terms, an elementary low-pass filter has inductance in series with the signal path and capacitance in parallel with it.

As the diagram shows, a filter section can be arranged in a pi or a T form. Either form can be constructed from two identical half-sections.

The **cut-off frequency, ω rad/s**, for any of these filters is given by

$\omega^2 LC = 1$.

The '**design impedance**', which approximates to the characteristic impedance at low frequencies, is Z where

$Z^2 = L/C$

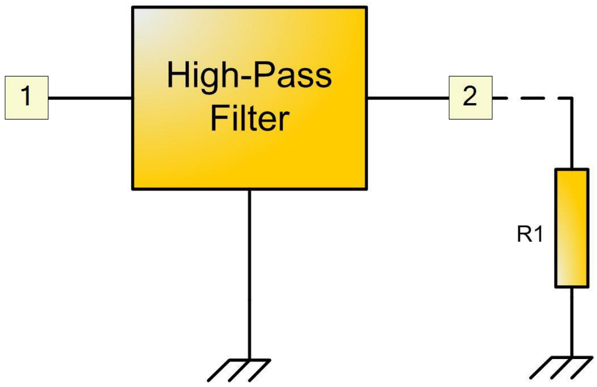
Practical 1: Butterworth LC High-Pass Filter

Objectives and Background

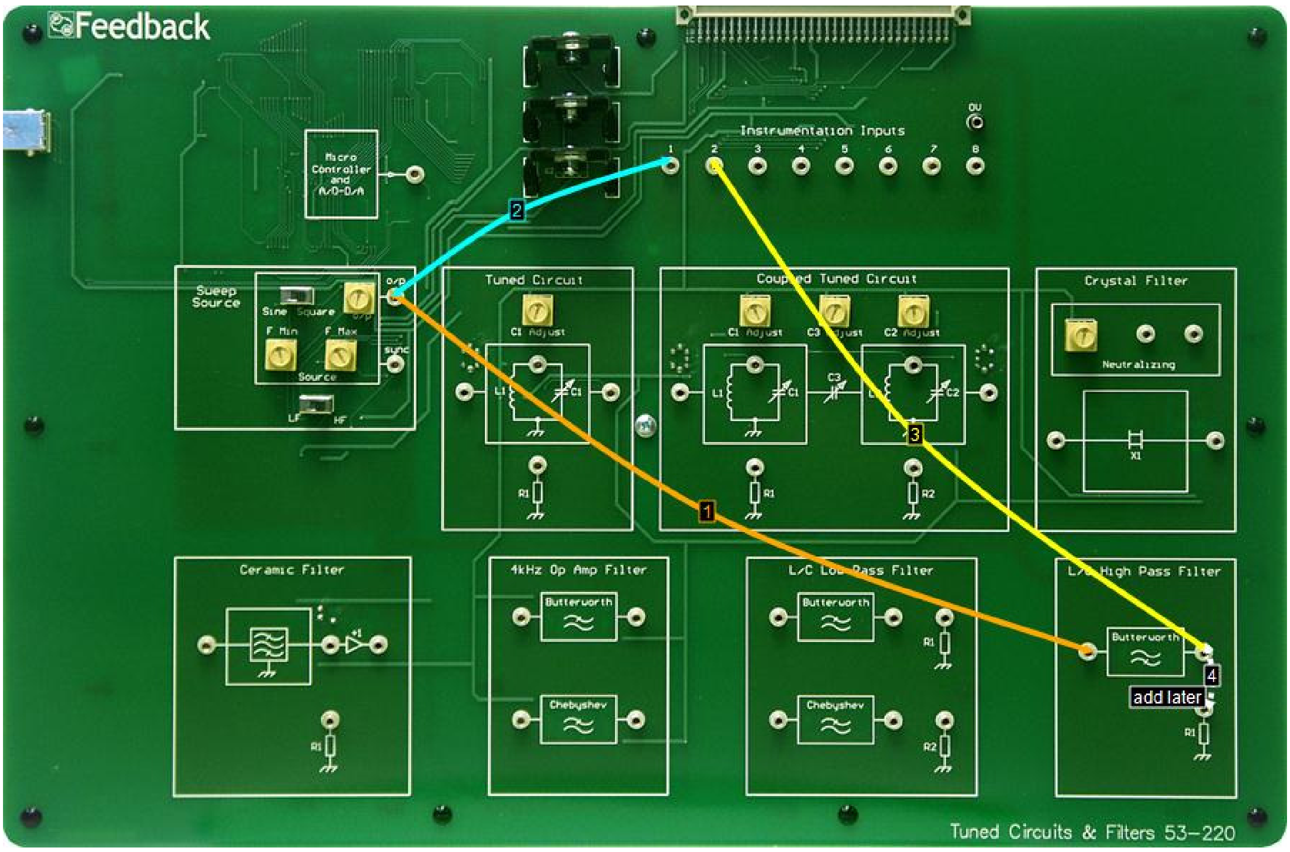
The Butterworth Filter is the filter type that results in the flattest pass band and contains a moderate group delay.

In this Practical you will measure the response of a Butterworth high pass filter, both with and without termination resistor, and you will see the typical lack of ripple and the rate of cut-off.

Block Diagram



Make Connections Diagram



Practical 1: Butterworth LC High-Pass Filter

Perform Practical

Use the **Make Connections** diagram to show the required connections on the hardware.

Identify the **L/C High Pass Filter** circuit block, located in the bottom right-hand corner of the workboard.

Identify the **CH1** and **CH2** gain switches, located towards the top of the workboard, directly below the **Instrumentation Input** sockets.

Set both CH1 and CH2 switches to **Hi Gain**.

Identify the **Sweep Source** circuit block at the left-hand centre of the workboard. In this block, set the **Sine/Square** switch to Sine and the **LF/HF** switch (just below the Source block, to HF. Set both the **FMin** and **FMax** control to their minimum (fully counter-clockwise) positions. This will set the source frequency to approximately 100kHz.

Open the Gain Phase Analyser (GPA). Click on the **Set Max Freq** button on the GPA and use the FMax control on the Sweep Source to set the frequency to 6MHz.

Click on **Plot** on the GPA to plot the Bode response of the filter.

Notice the peak in the amplitude response and the way that the phase changes with frequency. The abrupt change from +180 to –180 degrees on the plot is due to the fact that the plotter cannot distinguish between these two points.

Now click **Nyquist** on the GPA (at the bottom right of the display) and see that the phase does, in fact, smoothly progress through more than 360 degrees.

Deselect Nyquist, to return to the Bode plot mode, and connect the terminating resistor **R1**, as shown by connection 4 on the Make Connections diagram.

Notice the difference in response made by adding the terminating resistor. Note the effect on the peaking and on the passband amplitude of the output. This is because there is now a potential divider at the output between the filter's characteristic output impedance and the equal value terminating resistance. Correctly terminating a filter may improve its response, but will always lead to a loss in amplitude.

Measure the rate at which the amplitude decreases below the cut-off of the filter. Also measure the rate of phase change with frequency for the filter.

Tuned Circuit

Objectives

To become familiar with the operation of an LC Tuned Circuit

To plot the Bode and Nyquist frequency responses for such a circuit

To investigate the effect on the response of connecting a resistor across a tuned circuit

To view the transient response of the circuit

Gain and Phase

Gain

If a signal is applied to an electronic system, the output of that system is unlikely to be exactly the same as the input. For instance, the output is likely to be different in magnitude.

For example, the signal input voltage at the antenna of a radio receiver may be ten microvolts (10μV), whereas the output voltage that drives the loudspeaker in that radio may be ten volts (10V).

The ratio of output signal to input signal (measured in the same units) of a system is referred to as its **gain**.

In the example above, the gain of the radio receiver would be:

$$\frac{10V}{10\mu V} = 1{,}000{,}000$$

Because the units in the equation are those of volts, this gain would be referred to as the **voltage gain** of the receiver.

There are other types of gain that could be quoted for the above receiver. A typical input impedance (resistance) for a radio receiver is 50Ω. Using Ohm's Law, the power applied to that input when 10μV is applied can thus be calculated to be $2x10^{-12}$W and the power output to the speaker to be 6.25W if the loudspeaker's resistance is 16Ω. The **power gain** of the receiver is thus given by:

$$\frac{6.25W}{2x10^{-12}W} = 3.125x10^{12}$$

This is a huge number! More on this later.

Now to calculate the signal currents at the input and the output of the receiver. Again using Ohm's Law, the input current is 0.2μA and the output current is 0.625A. The **current gain** will thus be:

$$\frac{0.625A}{0.2\mu \text{A}} = 3{,}125{,}000$$

Attenuation

For some electronic systems the magnitude of the output signal will be smaller than that of the input signal.

As an example, consider a long telephone line. Due to the resistance of the wire from which the line is constructed there will be loss in the line and the magnitude of the signal at the telephone end of the line (the subscriber's end) will be lower than that at the originating end of the line. It is quite possible for the signal power to be halved in magnitude by the losses in a line. This gives a gain of 0.5 for the line.

Thus gains of less than unity signify **loss** in a system. The term used for this loss is **attenuation**.

The decibel

As can be seen from the two examples of systems given above, the gain of an electronic system may have a value anywhere between a small fraction (if the loss is high) to a huge number (as in the case of the radio receiver).

The use of such a large range of numbers to quantify the gain of systems is, in many ways, very inconvenient. To make things easier, the logarithm of the ratio of output to input is often calculated and quoted.

For example, the logarithm of the power gain of the radio receiver is:

$$\log_{10} 3.125\text{x}10^{12} = 12.5$$

and the logarithm of the gain of the telephone line is:

$$\log_{10} 0.5 = -0.3$$

These are much more convenient numbers to deal with!

Notice that using logs gives positive numbers for gain and negative numbers for loss (attenuation).

The idea of using the log of the ratio was developed in the 19th century to describe the losses associated with long telephone lines. It was initially called the 'transmission unit' and was given the unit name 'the bel', in honour of the telephone pioneer Alexander Graham Bell. However, the bel was rather too large a unit for easy practical use, so the numbers were multiplied by ten and the unit '**decibel**' used.

Thus, this gives the power gain of the radio receiver as:

$$10\log_{10} 3.125\text{x}10^{12} = 125 \text{ decibels}$$

and the power gain of the telephone line as:

$$10\log_{10} 0.5 = -3 \text{ decibels.}$$

These would normally be written as 125dB and –3dB, respectively.

Again, note that positive dBs mean gain and negative dBs mean attenuation.

The units 'bel' or 'decibel' are defined using the ratio of the output to input **power** of a system. Now see what happens if a voltage, or a current ratio is used.

$$\text{Power Gain} = 10\log_{10} \frac{P_{out}}{P_{in}} \text{dB}$$

Now: $P = V^2/R$, therefore

$$\text{Power Gain} = 10\log_{10}\frac{\dfrac{V_{out}^2}{R_L}}{\dfrac{V_{in}^2}{R_{in}}} = 20\log_{10}\frac{V_{out}}{V_{in}}\text{ dB (if } R_L = R_{in}\text{)}$$

So, using a voltage ratio means that you have to use 20 times, instead of 10, to get the correct answer in decibels. You will see later what happens if $R_L \neq R_{in}$.

Also, because $P = I^2R$, a similar result is achieved if a current ratio is used, giving:

$$\text{Power Gain} = 20\log_{10}\frac{I_{out}}{I_{in}}\text{ dB}$$

Now consider a system comprising two parts, the first of which has a power gain of 3 and the second a power gain of 6. To get the total power gain of the system you need to **multiply** the two gains of the parts, giving 18.

Now, in decibels the gains are:

$$10\log_{10} 3 = 4.77\text{ dB}$$

$$10\log_{10} 6 = 7.78\text{ dB}$$

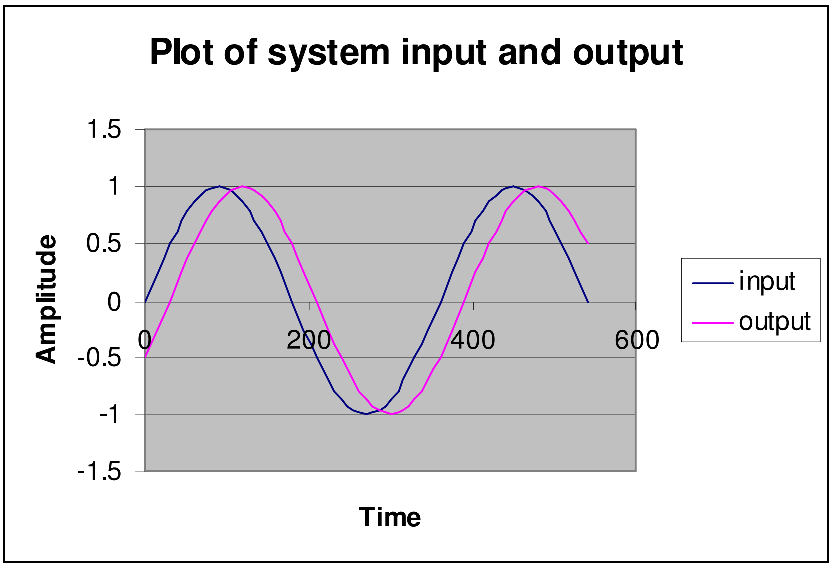
$$10\log_{10} 18 = 12.55\text{ dB}$$

Notice that to get the total gain in dB you just **add** the individual gains in dB – much easier to do than multiplication!

Phase

A second reason for the output of an electronic system to be identical to its input is one of time delay. It takes time for the signals to pass through the system. This time delay may only be parts of a microsecond, but it can have a considerable effect on system performance.

Consider a sinusoidal input signal to a system and the corresponding delayed output (for this example, and simplicity, it is assumed that the system has unity gain). The input and output waveforms may look like the diagram below.



You will see that the output waveform is delayed by a small amount with respect to the input waveform. The delay could be measured in units of time but it is more usual to express it as an angle. This can be done because one cycle of the waveform is equivalent to 2π radians (360 degrees). In the diagram the difference between the two waveforms is 30 degrees.

The difference in degrees (or radians) is referred to as the **phase shift** (or just phase) between the two waveforms.

Frequency Effects

In an electronic system gain and phase are seldom constant with respect to the frequency of the applied signal. Because of this, the system is said to have a **frequency response**. This is just a mathematical, or pictorial (graph) description of how the gain and phase of the system change with frequency.

The frequency response of a system is an important property of that system. Some systems give an increase in gain with increasing frequency. Such a response is called a **high-pass** response.

Some systems have a frequency response with the gain dropping as the frequency increases. Such a response is called a **low-pass** response.

In some systems the gain increases with frequency up to some value and then decreases as the frequency is further increases. Such a response is called a **bandpass** response.

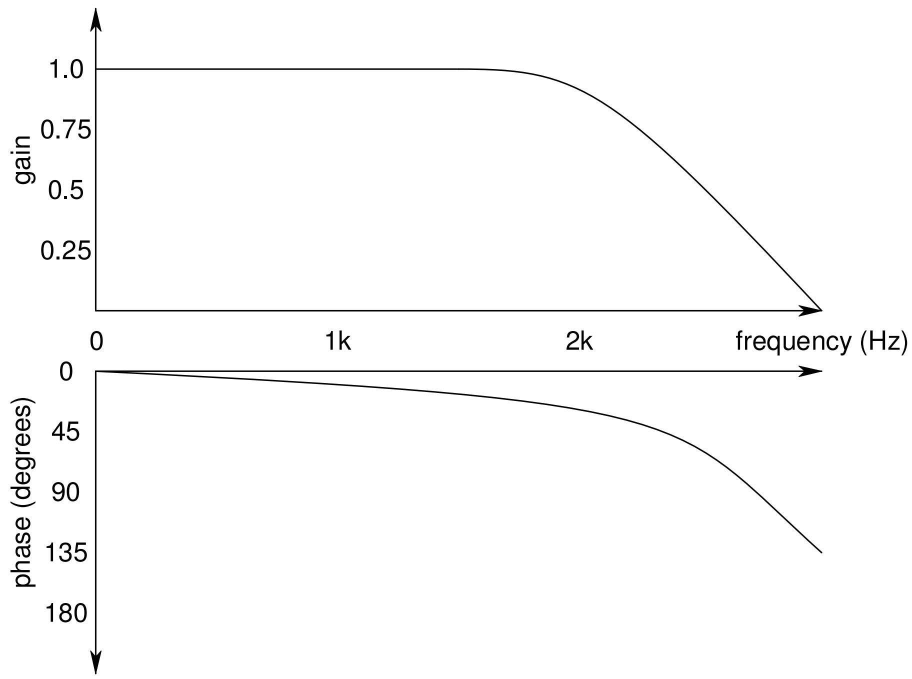
You will be meeting systems with these types of response as you progress through your course.

Plots

One of the most convenient ways of describing the frequency response of a system is in graphical form. This is usually not as accurate as describing it mathematically, but it is often adequate for practical purposes and is normally much easier to see what is happening from a graph, rather than from the mathematics.

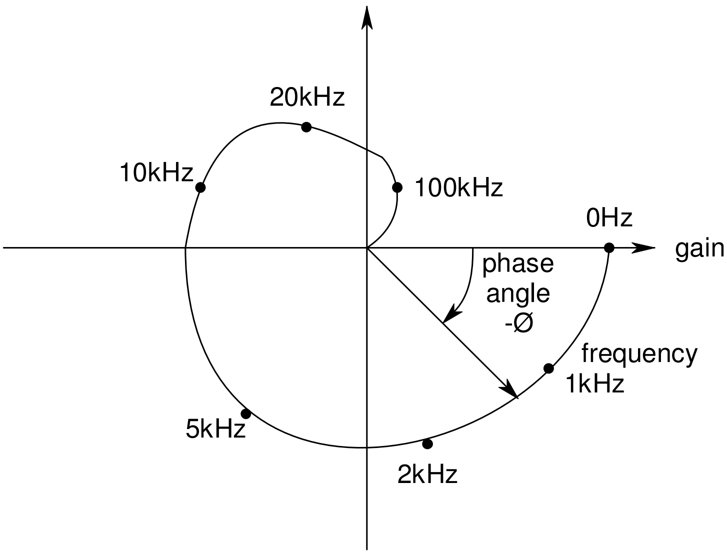
There are two main forms of graphs that are used to show the frequency response of a system. These are named after the persons that devised them and are the **Bode** plot and the **Nyquist** plot.

The **Bode** plot of a system is no more than normal graphs of gain (on the Y axis) against frequency (on the X axis) together with another graph of phase (on the X axis) against frequency. Generally, the gain and the phase curves are plotted on separate axes, one above the other, as shown in the diagram below. The frequency axes are the same for both graphs, so a direct relationship between gain and phase at any required frequency can be made easily.



The second way of displaying the frequency response is by using a vector (or phasor) type plot in which the gain of the system is given by the length of the vector and the phase by the angle of the vector. This type of plot is known as a **Nyquist** plot, after the mathematician who devised it.

An example Nyquist plot of a system is given below.



As you can see, the gain and phase at different frequencies are given by the length and angle (Φ) of the phasor. The corresponding frequencies are usually shown on the plot along the **locus** (the path) of the curve.

Other names for the plots are:

Bode plot – **rectangular** plot

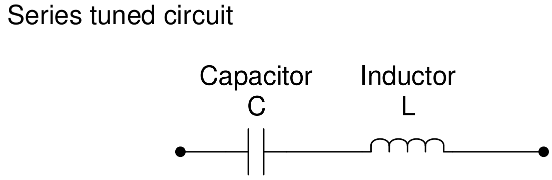
Nyquist plot – **polar** plot

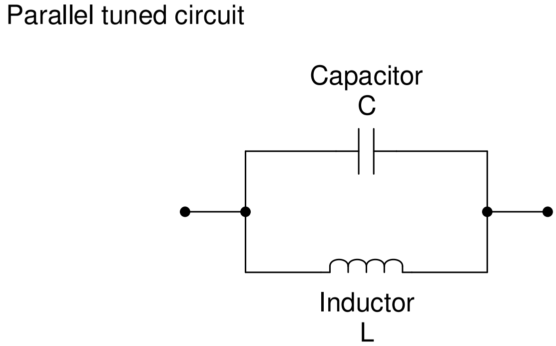
The type of plot that is used depends on the type of system that is being investigated and the properties of the system under investigation. For example, the frequency response of amplifiers and filters normally use the Bode plot form of graph, whereas investigations into control systems and stability use the Nyquist form. However, it is not incorrect to use either form.

You will use both types of plot as you perform assignments using this equipment.

Tuned Circuits

A 'tuned circuit' normally comprises an inductor and a capacitor connected together. These components may be connected either in series or in parallel, as shown below:





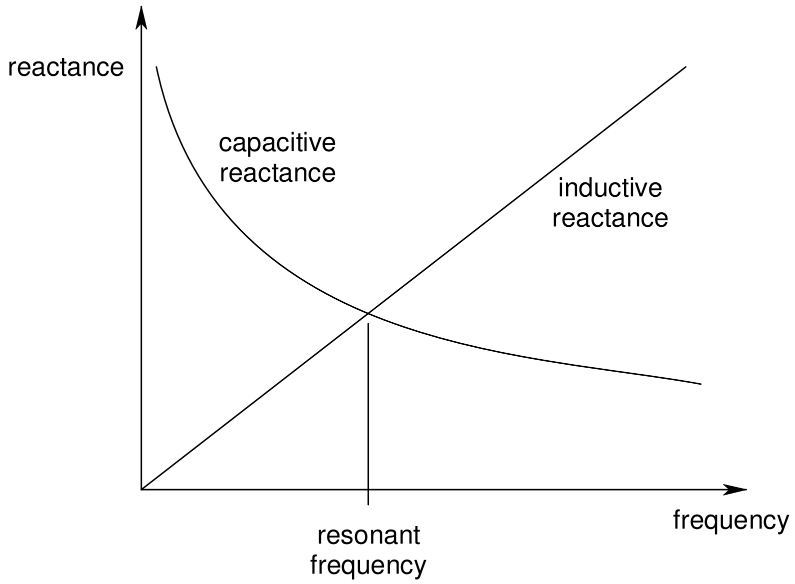
First, consider the series tuned circuit.

At low frequencies the reactance of the inductor will be low and that of the capacitor will be high. Thus the total reactance of the two in series will approximate to that of the capacitor.

Conversely, at high frequencies the reactance of the inductor will be high and that of the capacitor will be low. Thus the total reactance of the two in series will approximate to that of the inductor.

At some frequency in between the reactance of the inductor and that of the capacitor will be equal in value and the total reactance of the two in series will be a minimum. In fact, because the reactance of the inductor and that of the capacitor are of opposite sign, the two will cancel each other and the total reactance will be theoretically zero. The series tuned circuit will look like a short circuit.

The frequency at which the capacitive reactance is equal in magnitude (and opposite in sign) to the inductive reactance is called the **resonant frequency** of the tuned circuit.



The equation for the reactance of a capacitor is:

$$X_C = \frac{1}{2\pi fC}$$

and that for an inductor is:

$$X_L = 2\pi fL$$

These are equal at the resonant frequency f_o, where:

$$\frac{1}{2\pi f_o C} = 2\pi f_o L$$

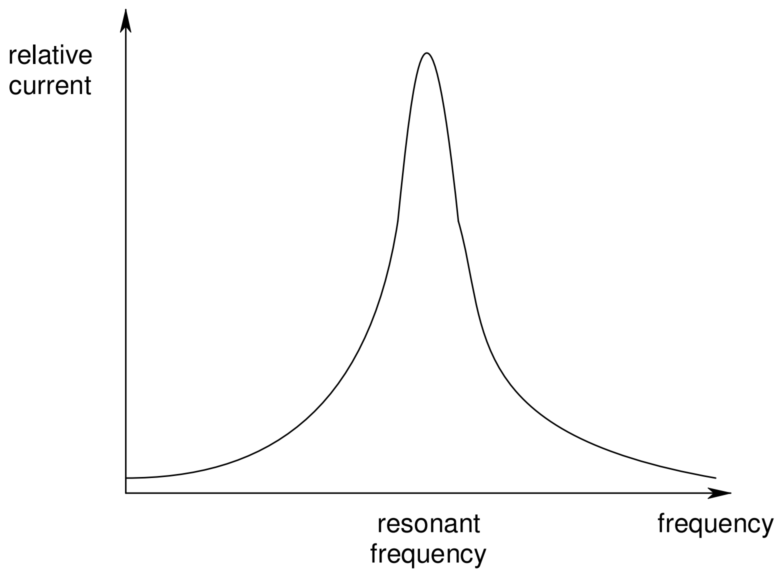
Solving this gives:

$$\boldsymbol{f_o = \frac{1}{2\pi\sqrt{LC}}}$$

the classical equation for the resonant frequency of a tuned circuit.

The same argument and equation applies for a parallel tuned circuit as, at resonance, the reactances are equal.

A typical shape of a curve for the relative current through a series tuned circuit with respect to frequency is as below.

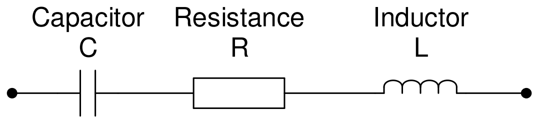


This shows that the circuit is selective. That is, it will pass signals at, or close to, the resonant frequency and reject signals that are not close to this frequency.

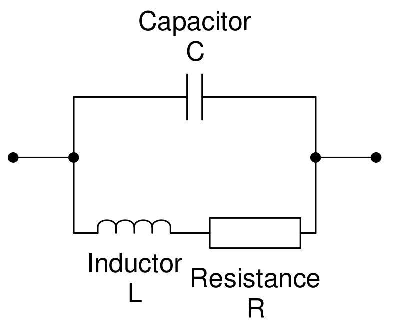
Q and Bandwidth

Any practical tuned (L/C) circuit will have some resistance associated with it. This comes predominantly from that associated with the wire with which the inductor (coil) is wound. This means that the equivalent circuit of a real tuned circuit looks like:

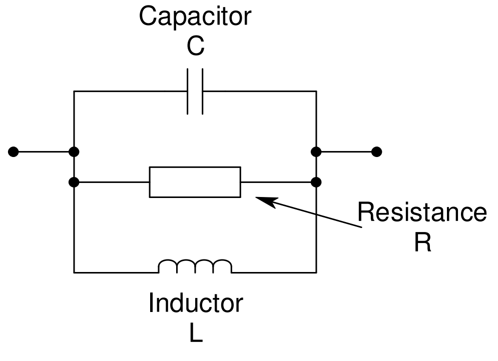
Series tuned circuit



Parallel tuned circuit



After series/parallel transformation



A series to parallel transformation can be performed on the inductor/resistor branch of the parallel tuned circuit to produce a parallel equivalent circuit, as shown in the bottom diagram, above.

The fact that any practical tuned circuit has resistance means that there will be some signal losses associated with that resistance. A factor, known as the Quality Factor, Q, may be quoted to give a numerical value to the 'goodness' of the circuit. The Q is defined as:

$$Q = 2\pi \frac{\text{maximum stored energy}}{\text{energy dissipated per cycle}}$$

For an inductor L, with its associated loss resistance R, this can be shown to be

$$Q = \frac{\omega L}{R}$$

For a capacitor C, with its associated loss resistance R, it is

$$Q = \frac{1}{\omega CR}$$

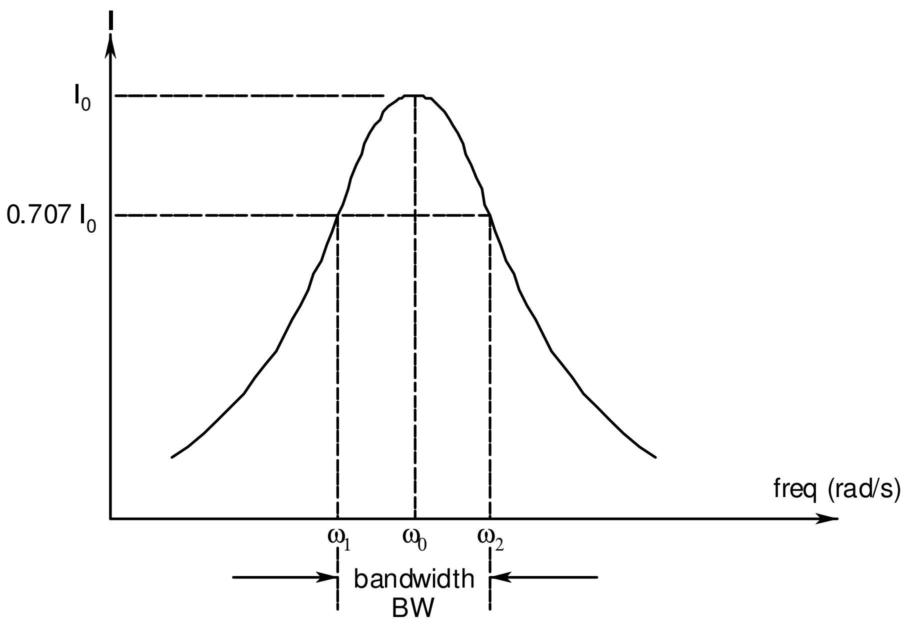
For a series RLC circuit at resonance (ω_0) it is

$$Q_0 = \frac{\omega_0 L}{R} = \frac{1}{\omega_0 CR}$$

And for a parallel RLC circuit at resonance (ω_0) it is

$$Q_0 = \frac{R}{\omega_0 L} = \omega_0 CR$$

A typical plot of current against frequency for a series tuned circuit is shown below.



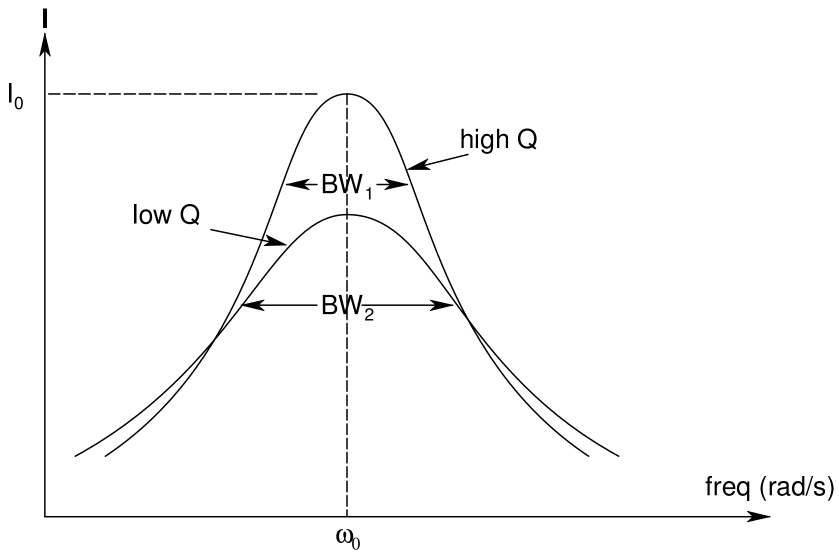
At ω_0 the current is a maximum, I_0. The points at which the current is 0.707 I_0 are shown, at corresponding frequencies ω_1 and ω_2. Since the power delivered to the circuit is I^2R, at

$I = 0.707\ I_0$ the power is one half of the maximum value that occurs at ω_0. Thus the points corresponding to ω_1 and ω_2 are called the half-power points. The frequency difference between the half-power points is called the bandwidth (BW) of the tuned circuit.
i.e.

$$BW = \omega_2 - \omega_1 \text{ (or } f_2 - f_1\text{)}$$

It can be shown that there is a relationship between bandwidth and Q. This is:

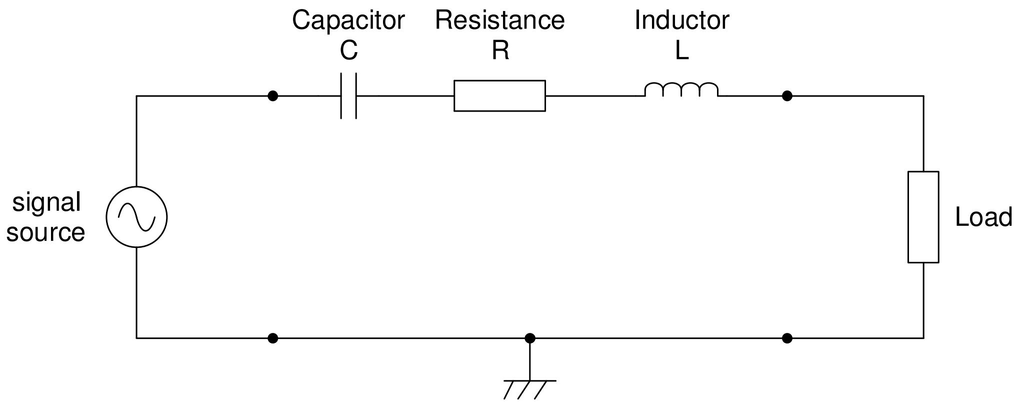
$$\boldsymbol{Q = \frac{f_0}{BW}}$$



The diagram above shows typical curves for high Q and low Q tuned circuits. Note that the low Q circuit has a lower peak (due to higher losses) and has a wider bandwidth than the high Q circuit.

Loading

Consider a series tuned circuit, as shown in the diagram below.

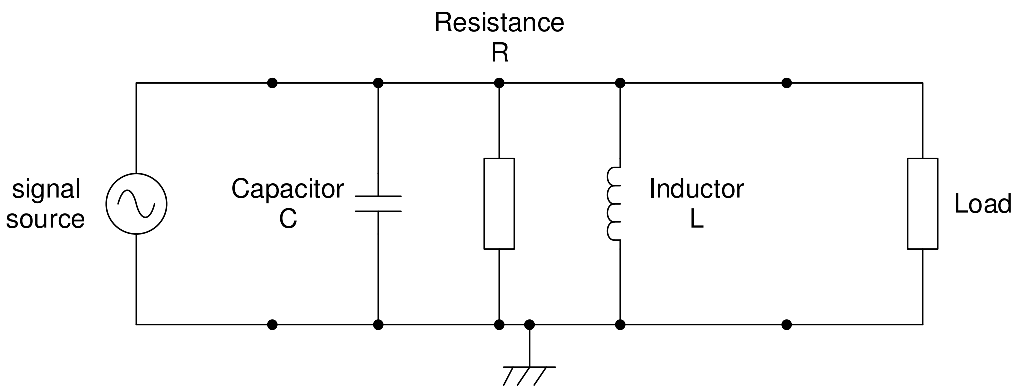


The circuit is being driven from a signal source and a load is connected across the output.

You can see that the load is effectively in series with the tuned circuit and thus the total resistance associated with the complete circuit will be the loss resistance R plus the load resistance.

As the Q of a tuned circuit is dependent on the total resistance associated with that circuit, adding load resistance is going to lower the Q. It also increases the circuit bandwidth.

Now, consider a parallel tuned circuit, as shown below.



You can see that the load is effectively in parallel with the tuned circuit and thus the total parallel resistance associated with the complete circuit will be the parallel combination of the loss resistance R and the load resistance.

Thus the total parallel resistance will be lowered. This will lower the Q and increase the bandwidth.

Practical 1: Frequency Response – Manual Plotting

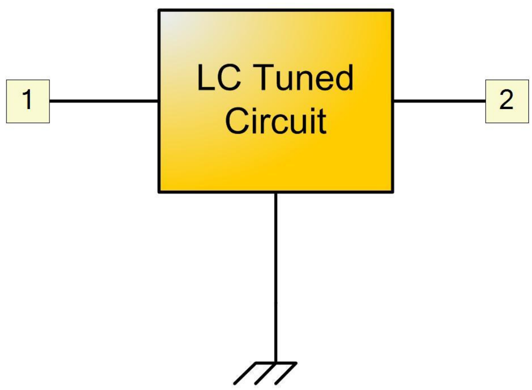
Objectives and Background

The frequency response of a circuit is a ‘picture’ (either mathematical or graphical) of how that circuit behaves as the frequency of the applied input signal is changed.

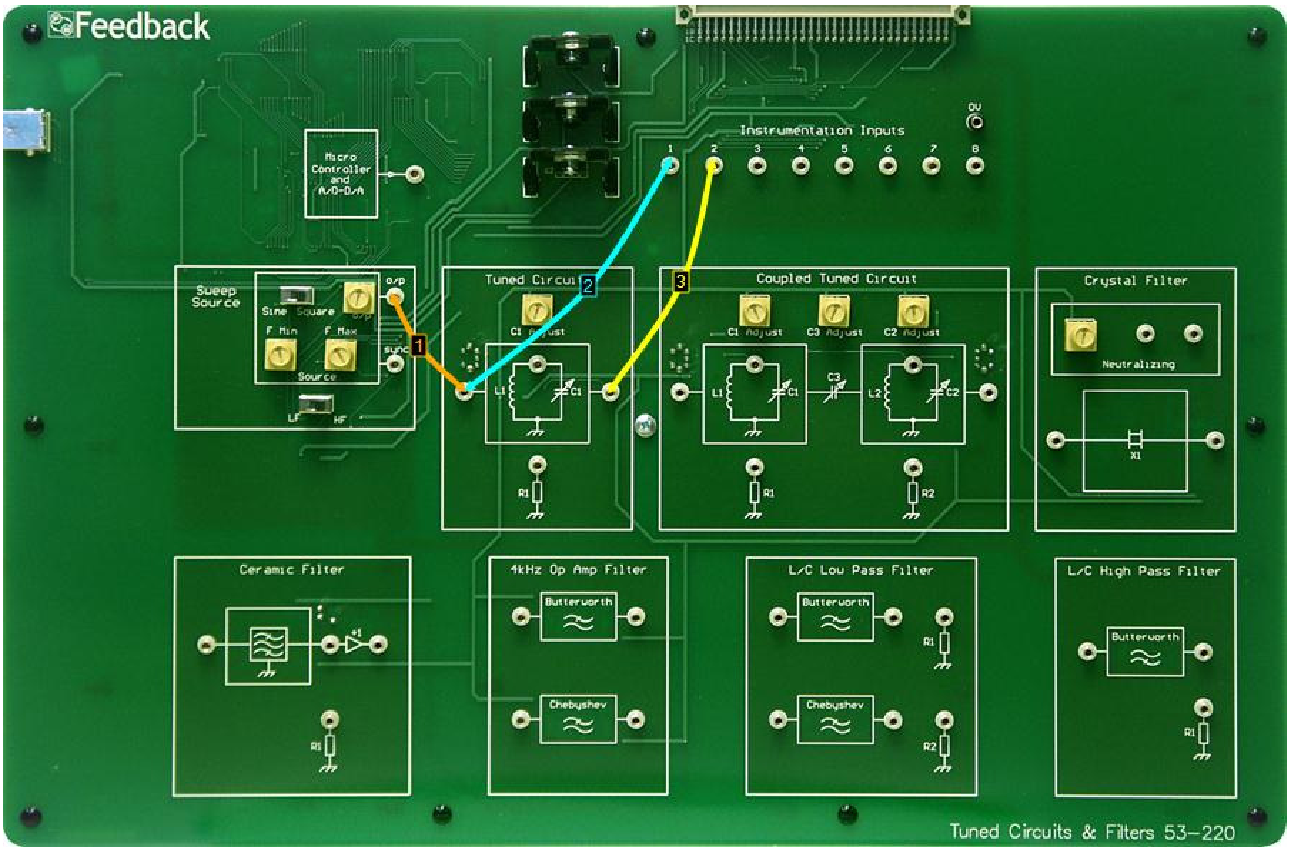
In this Practical you will apply a sine wave input signal to a tuned circuit and you will observe the output signal, using the oscilloscope. You will vary the frequency of the input signal and see how the output responds.

You will find that the control used to vary the frequency is quite coarse at the frequencies of interest in this Assignment and, as the Q of the tuned circuit is reasonably high, the bandwidth of the tuned circuit is narrow. This makes manual investigation of the frequency response rather difficult. However, a more automated plotting method, using the Gain Phase analyser, is investigated in Practical 2.

Block Diagram



Make Connections Diagram



Practical 1: Frequency Response – Manual Plotting

Perform Practical

Use the **Make Connections** diagram to show the required connections on the hardware.

Identify the **Tuned Circuit** block, located to the top left centre of the workboard. Set the **C1 Adjust** control to half scale.

Identify the **CH1** and **CH2** gain switches, located towards the top of the workboard, directly below the **Instrumentation Input** sockets.

Set both CH1 and CH2 switches to **Hi Gain**.

Identify the **Sweep Source** circuit block at the left-hand centre of the workboard. In this block, set the **Sine**/**Square** switch to Sine and the **LF**/**HF** switch (just below the Source block) to HF. Set both the **FMin** and **FMax** control to their minimum (fully counter-clockwise) positions. This will set the source frequency to approximately 100kHz.

Set the **o**/**p** control on the Sweep Source to its maximum (fully clockwise) position.

Open the oscilloscope and the frequency counter. Observe the frequency of the input signal and the amplitudes of the input and output signals.

Slowly increase (rotate clockwise) the FMin control on the Sweep Source and observe the output signal (blue trace). You should find that there is a frequency that gives a sharp peak in output signal. Note the frequency of this peak.

Open the voltmeter and note the peak-to-peak amplitude of the output at the peak frequency.

Reset the FMin control to its minimum position.

Attempt to take readings of peak-to-peak output voltage against frequency for a number of frequencies from about 800kHz to about 1.2MHz. you will not find this easy, as the adjustment of the FMin control is very sensitive.

Plot a graph of your results.

Practical 2: Using the GPA

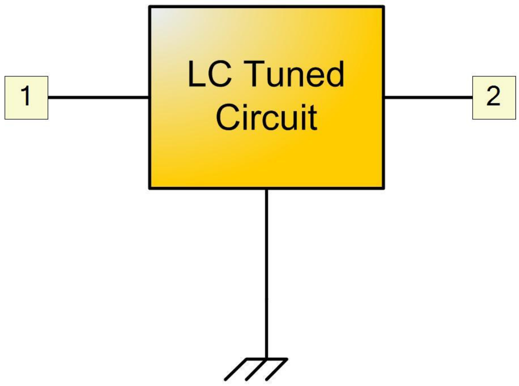
Objectives and Background

In this Practical you will use the Gain Phase Analyser (GPA) to plot the frequency response of the tuned circuit.

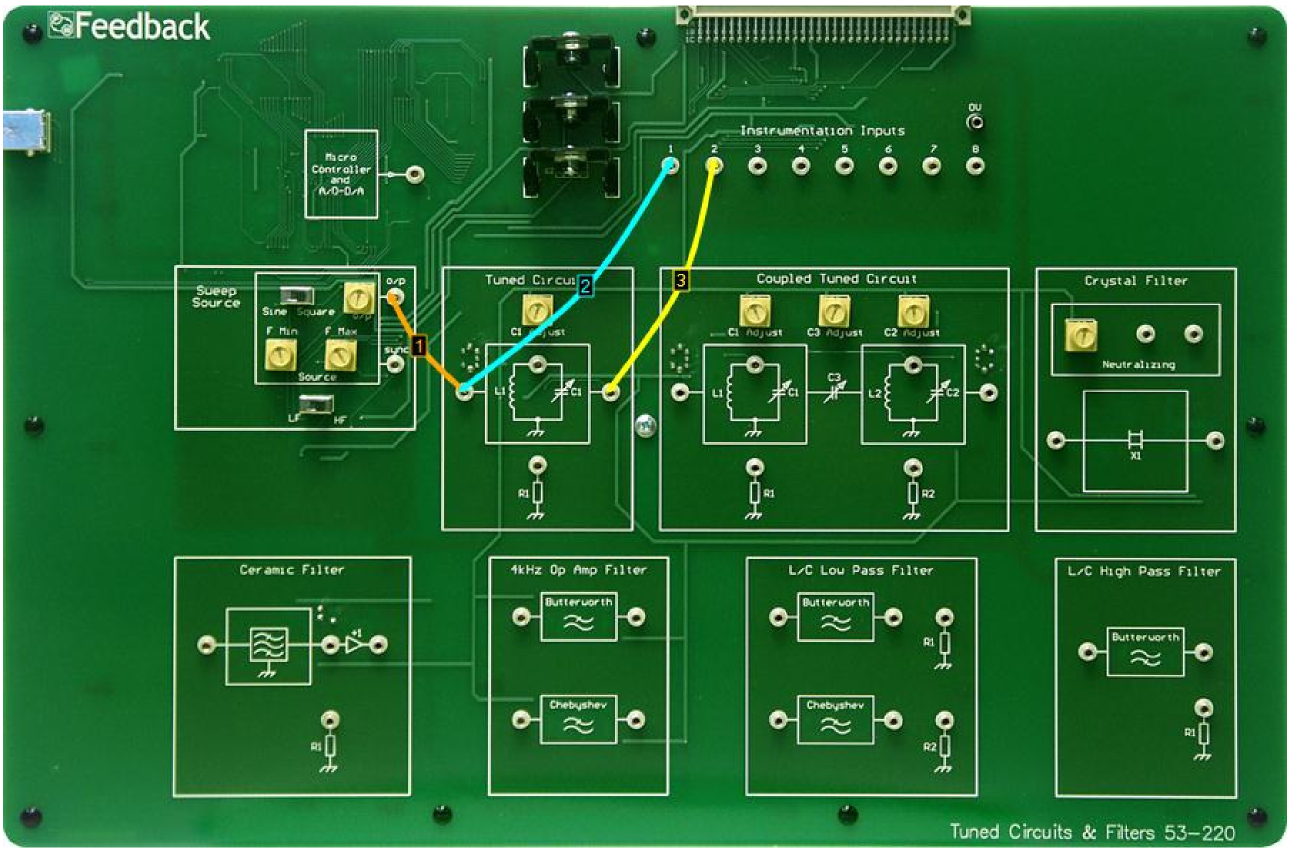
You will see that the GPA can produce plots of both amplitude and phase for the circuit under test (the tuned circuit, in this case) and that these can be displayed in either Cartesian (Bode Plot) form, or polar (Nyquist Plot) form.

You will see how easy it is to acquire these plots and you will observe the effect of changing the value of the capacitor in the tuned circuit.

Block Diagram



Make Connections Diagram



Practical 2: Using the GPA

Perform Practical

Use the **Make Connections** diagram to show the required connections on the hardware.

Ensure that both **CH1** and **CH2** switches are set to **Hi Gain**.

Ensure that the **C1 Adjust** control in the **Tuned Circuit** block is at half scale.

In the **Sweep Source** circuit block, ensure that the **Sine**/**Square** switch is set to Sine and the **LF**/**HF** switch (just below the Source block) to HF. Set both the **FMin** and **FMax** control to their minimum (fully counter-clockwise) positions. This will set the source frequency to approximately 100kHz.

Set the **o**/**p** control on the Sweep Source to its maximum position.

Open the Gain Phase Analyser (GPA). The **Set Min Freq** button on the GPA is already selected by default. Use the FMin control on the Sweep Source to set the frequency to approximately 800kHz.

Click on the **Set Max Freq** button on the GPA and use the FMax control on the Sweep Source to set the frequency to approximately 1.2MHz.

Click on **Plot** on the GPA to plot the Bode response of the circuit.

Click **Hi Res** on the GPA (at the bottom centre of the display) to achieve a higher resolution plot.

Notice the peak of the amplitude response and the way that the phase changes with frequency. Note the phase at the frequency of the amplitude peak.

Now click **Nyquist** on the GPA (at the bottom right of the display). Select **Hi-Res** (at the bottom centre of the GPA) and see how the response is plotted in polar form.

Use the slider bar to the left of the display to position the marker to the zero phase point. Read the marker frequency from the box under the display. Compare this frequency with that of the peak noted from the Bode display.

Deselect Nyquist, to return to the Bode plot mode. Now, alter the position of the C1 Adjust control and note the effect on the frequency response. this control varies the value of capacitor **C1** and thus changes the resonant frequency of the circuit.

Note the range of resonant frequency change.

Practical 3: Loading the Circuit

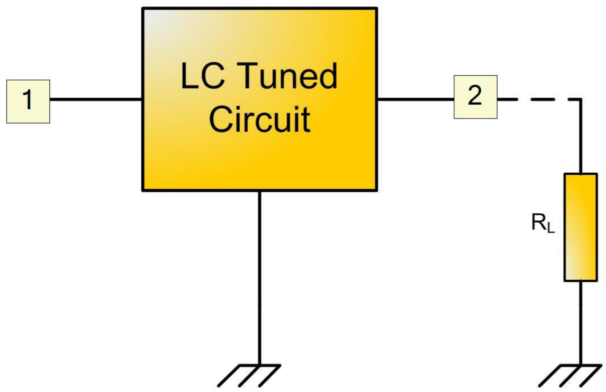
Objectives and Background

In this Practical you will observe the effect that connecting a resistive load across the tuned circuit has on its response. You will see that both the magnitude of the output and the bandwidth of the tuned circuit are changed.

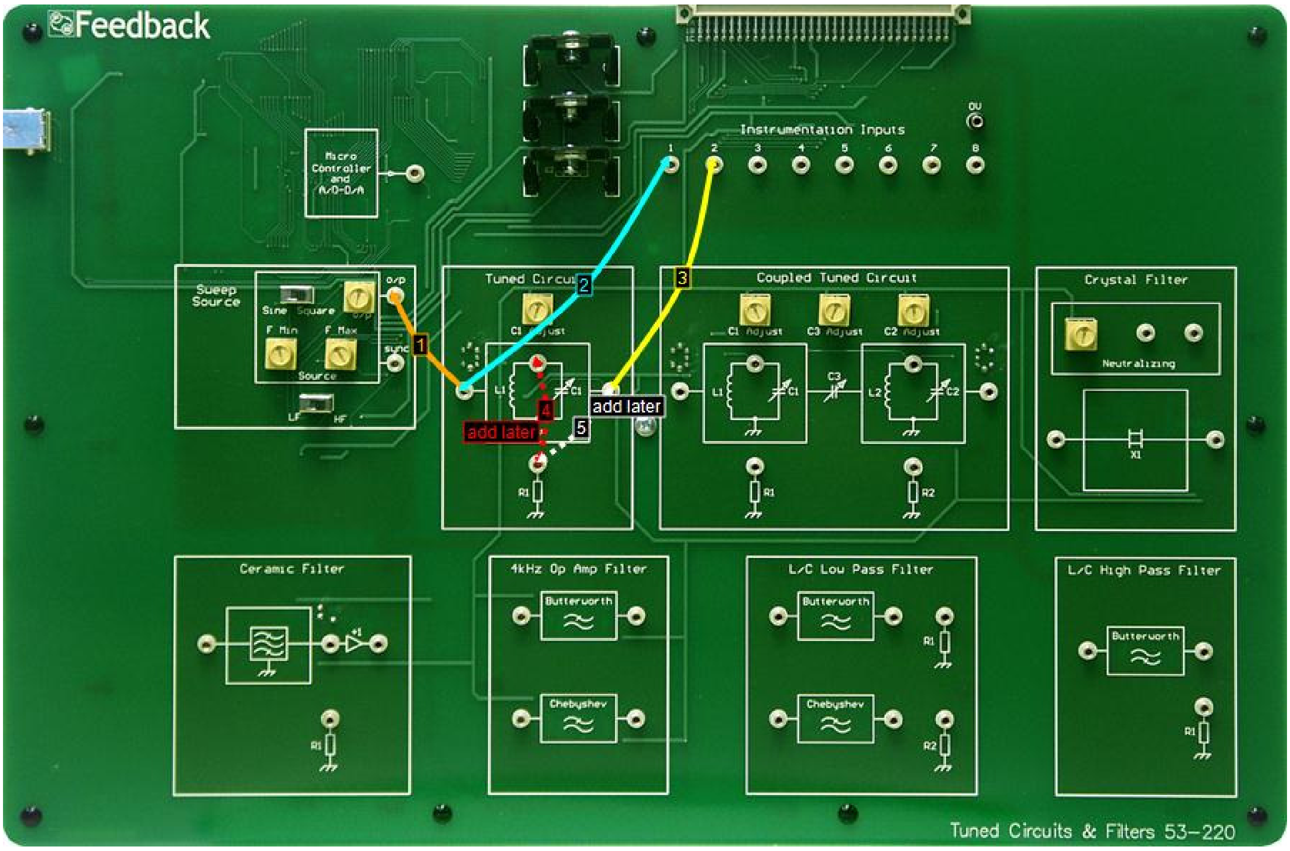
You will measure the bandwidths of the circuit, with and without loading.

You will also see how including a buffer amplifier stage between the tuned circuit and the load can minimise the loading effects.

Block Diagram



Make Connections Diagram



Practical 3: Loading the Circuit

Perform Practical

Use the **Make Connections** diagram to show the required connections on the hardware.

Ensure that both **CH1** and **CH2** switches are set to **Hi Gain**.

Ensure that the **C1 Adjust** control in the **Tuned Circuit** block is at half scale.

In the **Sweep Source** circuit block, ensure that the **Sine**/**Square** switch is set to Sine and the **LF**/**HF** switch (just below the Source block) to HF. Set both the **FMin** and **FMax** control to their minimum (fully counter-clockwise) positions. This will set the source frequency to approximately 100kHz.

Set the **o**/**p** control on the Sweep Source to its maximum position.

Open the Gain Phase Analyser (GPA). The **Set Min Freq** button on the GPA is already selected by default. Use the FMin control on the Sweep Source to set the frequency to approximately 800kHz.

Click on the **Set Max Freq** button on the GPA and use the FMax control on the Sweep Source to set the frequency to approximately 1.2MHz.

Click on **Plot** on the GPA to plot the Bode response of the circuit. Observe the response.

Use the cursor to measure the bandwidth of the tuned circuit. This is the difference between the two frequencies at which the response is 3dB down from the peak amplitude.

Now, add the connection between the load resistor, **R1**, and the top of the tuned circuit, as shown by connection 4 on the Make Connections diagram. This connects R1 directly across the tuned circuit.

Note if there is a change in the shape of the frequency response plot.

Measure the new bandwidth and the new peak amplitude, relative to the peak amplitude with the circuit unloaded.

Now, disconnect the load resistor and observe, again, the unloaded frequency response.

Reconnect the load to the output socket, as shown by connection 5 on the Make Connections diagram. Note if there is a change in the shape of the frequency response plot.

You should observe that there is little, if any, change. This is because there is a buffer amplifier stage built into the circuit between the tuned circuit and the output terminal. The buffer isolates the tuned circuit from load changes. It is often good practice to design in buffer stages if loading effects are likely to be a problem.

Remove the load resistor by removing connection 5.

Practical 4: Transient Response

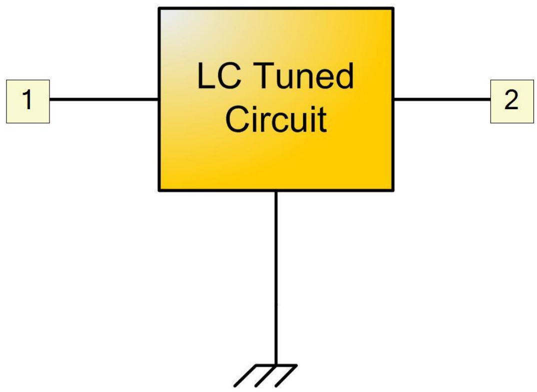
Objectives and Background

In this Practical you will observe the response of the tuned circuit to a step in input. This is called its **transient response**.

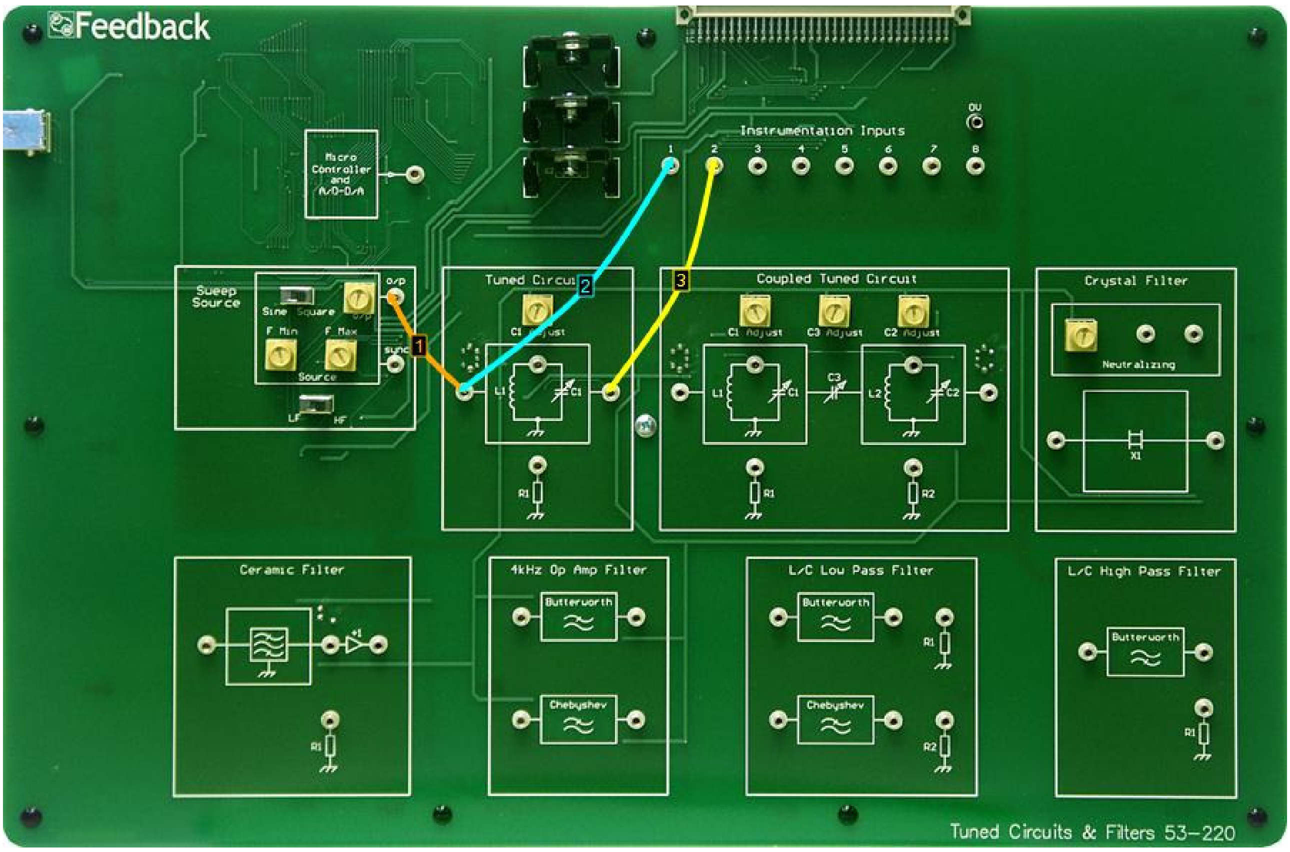
To achieve the required input step, the circuit will be driven with a low frequency square wave signal. The square wave frequency will be very much lower than the resonant (natural) frequency of the tuned circuit.

You will see that the tuned circuit produces an output that is a series of damped sine waves. You will estimate the frequency of these sine waveforms and compare that with the known resonant frequency of the tuned circuit. You will also see if the frequency of the input square wave has any effect on the damped output waveform frequency.

Block Diagram



Make Connections Diagram



Practical 4: Transient Response

Perform Practical

Use the **Make Connections** diagram to show the required connections on the hardware.

Ensure that both **CH1** and **CH2** switches are set to **Hi Gain**.

Ensure that the **C1 Adjust** control in the **Tuned Circuit** block is at half scale.

In the **Sweep Source** circuit block, ensure that the **Sine**/**Square** switch is set to Square and the **LF**/**HF** switch (just below the Source block) to LF. Set the **FMin** control to half scale and set the **FMax** control to its minimum (fully counter-clockwise) positions. This will set the source frequency to approximately 18.5kHz.

Set the **o**/**p** control on the Sweep Source to its maximum position.

Open the oscilloscope and observe the square wave input to the tuned circuit (yellow trace) and the resulting output (blue trace). You may need to increase the size of the oscilloscope display to see the response more clearly.

You should see the classic response of a resonant circuit to a step input, i.e. a damped sine wave.

Change the oscilloscope timebase to 4μs/div and use the cursor to measure the difference in time between successive peaks of the output waveform. Relate this time to the known resonant frequency of the tuned circuit.

Change the frequency of the square wave input and observe the frequency of the damped sine wave output. You should see that it does not change. This is the natural frequency of the tuned circuit.

Coupled Tuned Circuit

Objectives

To become familiar with the operation of a circuit comprising two, coupled LC tuned circuits

To plot the Bode frequency response for such a circuit

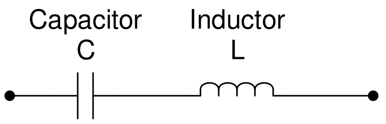
To investigate the effect on the response of varying the coupling between the two tuned circuits

To investigate the effect on the response of connecting a resistor across the tuned circuits

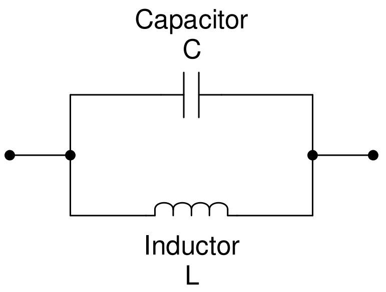
Tuned Circuits

A ‘tuned circuit’ normally comprises an inductor and a capacitor connected together. These components may be connected either in series or in parallel, as shown below:

Series tuned circuit



Parallel tuned circuit



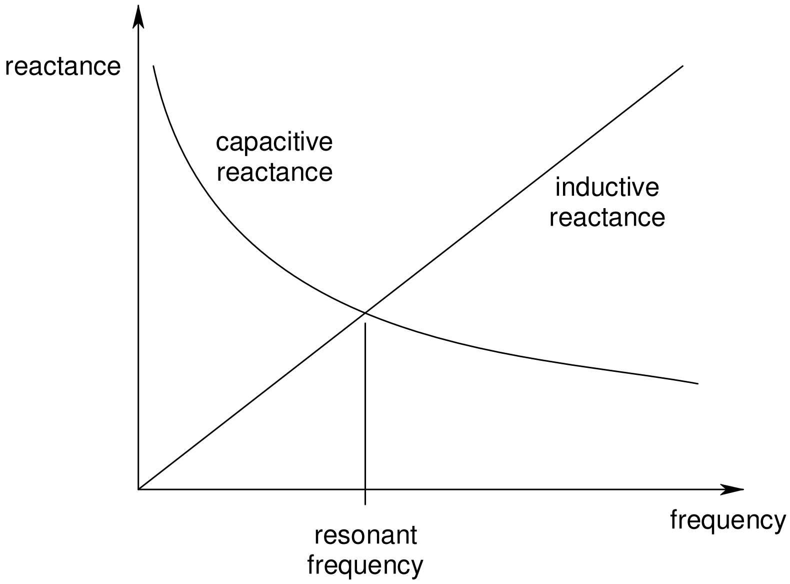
First, consider the series tuned circuit.

At low frequencies the reactance of the inductor will be low and that of the capacitor will be high. Thus the total reactance of the two in series will approximate to that of the capacitor.

Conversely, at high frequencies the reactance of the inductor will be high and that of the capacitor will be low. Thus the total reactance of the two in series will approximate to that of the inductor.

At some frequency in between the reactance of the inductor and that of the capacitor will be equal in value and the total reactance of the two in series will be a minimum. In fact, because the reactance of the inductor and that of the capacitor are of opposite sign, the two will cancel each other and the total reactance will be theoretically zero. The series tuned circuit will look like a short circuit.

The frequency at which the capacitive reactance is equal in magnitude (and opposite in sign) to the inductive reactance is called the **resonant frequency** of the tuned circuit.



The equation for the reactance of a capacitor is:

$$X_C = \frac{1}{2\pi f C}$$

and that for an inductor is:

$$X_L = 2\pi f L$$

These are equal at the resonant frequency f_o, where:

$$\frac{1}{2\pi f_o C} = 2\pi f_o L$$

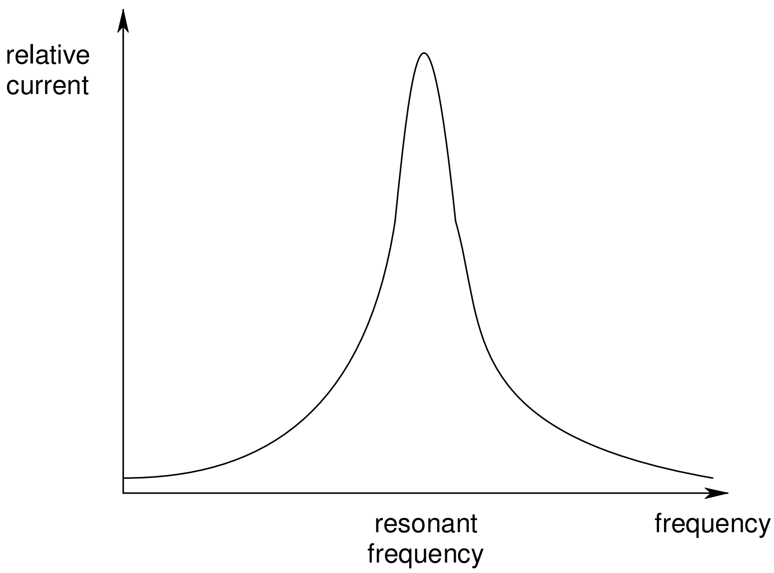
Solving this gives:

$$\boldsymbol{f_o = \frac{1}{2\pi\sqrt{LC}}}$$

the classical equation for the resonant frequency of a tuned circuit.

The same argument and equation applies for a parallel tuned circuit as, at resonance, the reactances are equal.

A typical shape of a curve for the relative current through a series tuned circuit with respect to frequency is as below.



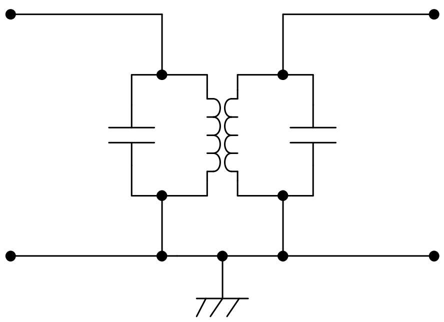
This shows that the circuit is selective. That is, it will pass signals at, or close to, the resonant frequency and reject signals that are not close to this frequency.

Coupled Tuned Circuits

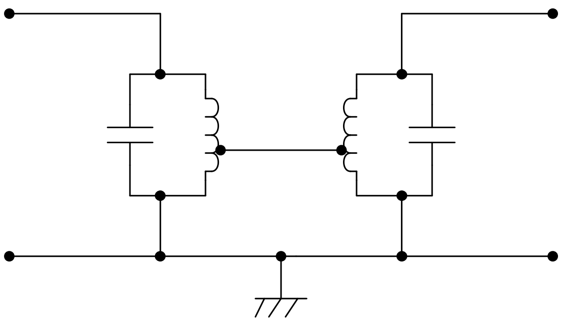
Two or more tuned circuits may be used together to modify the frequency response of the combination, relative to that of a single tuned circuit. The modified response may be more, or less, selective, dependent on how the tuned circuits are connected, or **coupled**, together.

There are many ways to couple tuned circuits, so that the signal from one circuit is transferred to the other. The diagrams below illustrate some of these methods.

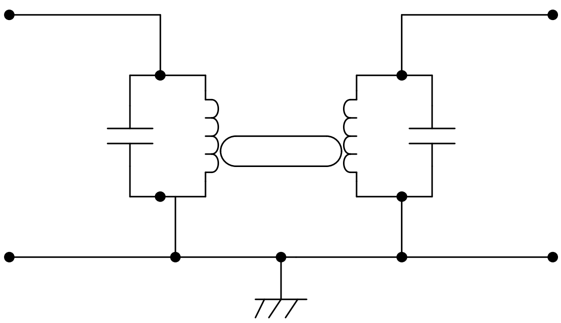
Transformer coupling



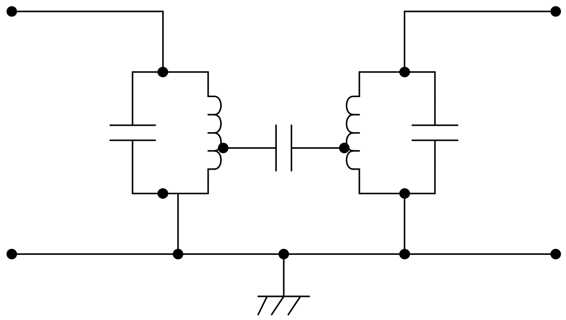
Autotransformer coupling



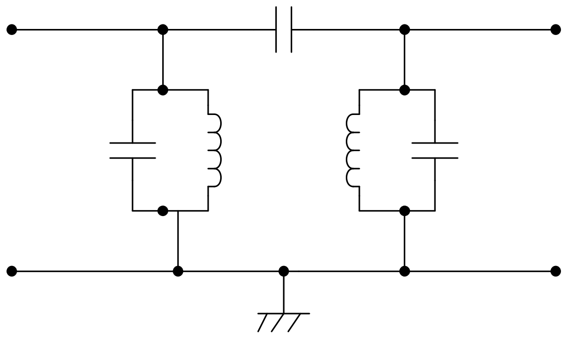
Link coupling



Capacitor coupling



Capacitor top coupling



The coupled tuned circuits on the Tuned Circuits & Filters 53-220 workboard use the capacitor top coupling method.

The amount of energy (signal) that is coupled from one circuit to the other is determined by such factors as the turns ratio (for transformer, autotransformer or link coupling) or the coupling capacitor value (for capacitor or top coupling). The overall shape of the frequency response curve for the coupled circuits depends on the amount of coupling that is used.

If both tuned circuits are adjusted for resonance at the same frequency, the overall selectivity will vary as the coupling is varied.

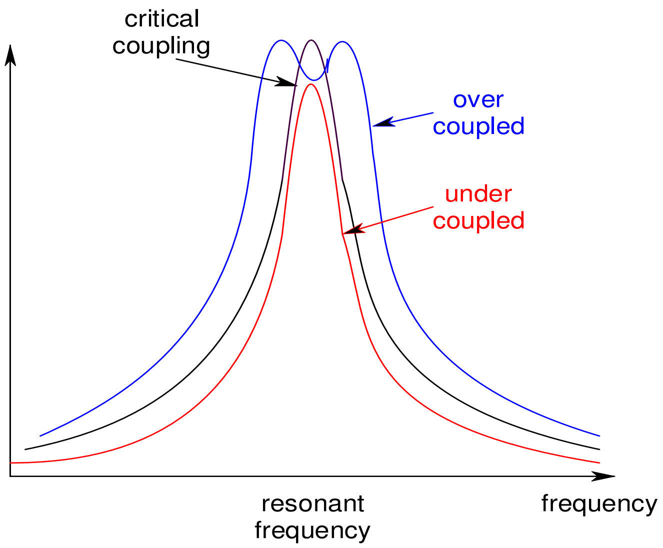
With loose coupling (only a small amount of energy transfer), the overall selectivity will be high and the overall Q approaches the sum of the individual tuned circuit Qs. However, the signal output will be low.

As the coupling is increased the signal output will also increase, but the selectivity will decrease slightly as the secondary circuit begins to load the primary.

There will be a value of coupling where the output signal at the resonant frequency reaches a maximum. This is called **critical** coupling.

If the coupling is increased above this critical coupling level, the output at resonance will start to decrease. However, 'humps' in the response, either side of the resonant frequency, will start to appear and the bandwidth of the response will start to increase. Further increase in coupling will cause a greater dip at resonance and a much wider bandwidth.

This progression is illustrated in the diagram below.



How much coupling to use depends on the application in which the coupled tuned circuit are to be employed. If the requirement is for maximum output at the resonant frequency, then critical coupling is the best choice. If the application requires maximum selectivity, then under coupled circuits should be used. If a bandpass frequency response is required, over coupling may be the answer. However, with over coupled circuits, the magnitude of the dip in the response at the resonant frequency must be taken into account. It may not be desirable to have more than about 1dB dip, although this, again, will depend on the requirements of the application.

Practical 1: Frequency Response

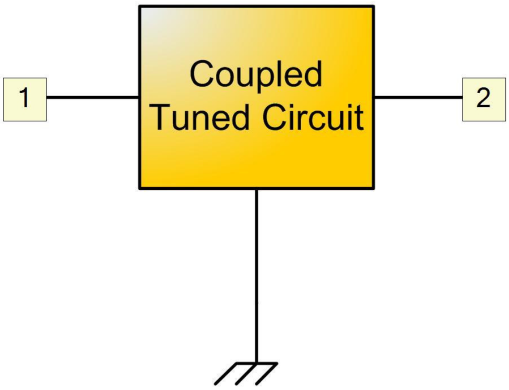
Objectives and Background

In this Practical you will use the Gain Phase Analyser (GPA) to plot the frequency response of the coupled tuned circuit.

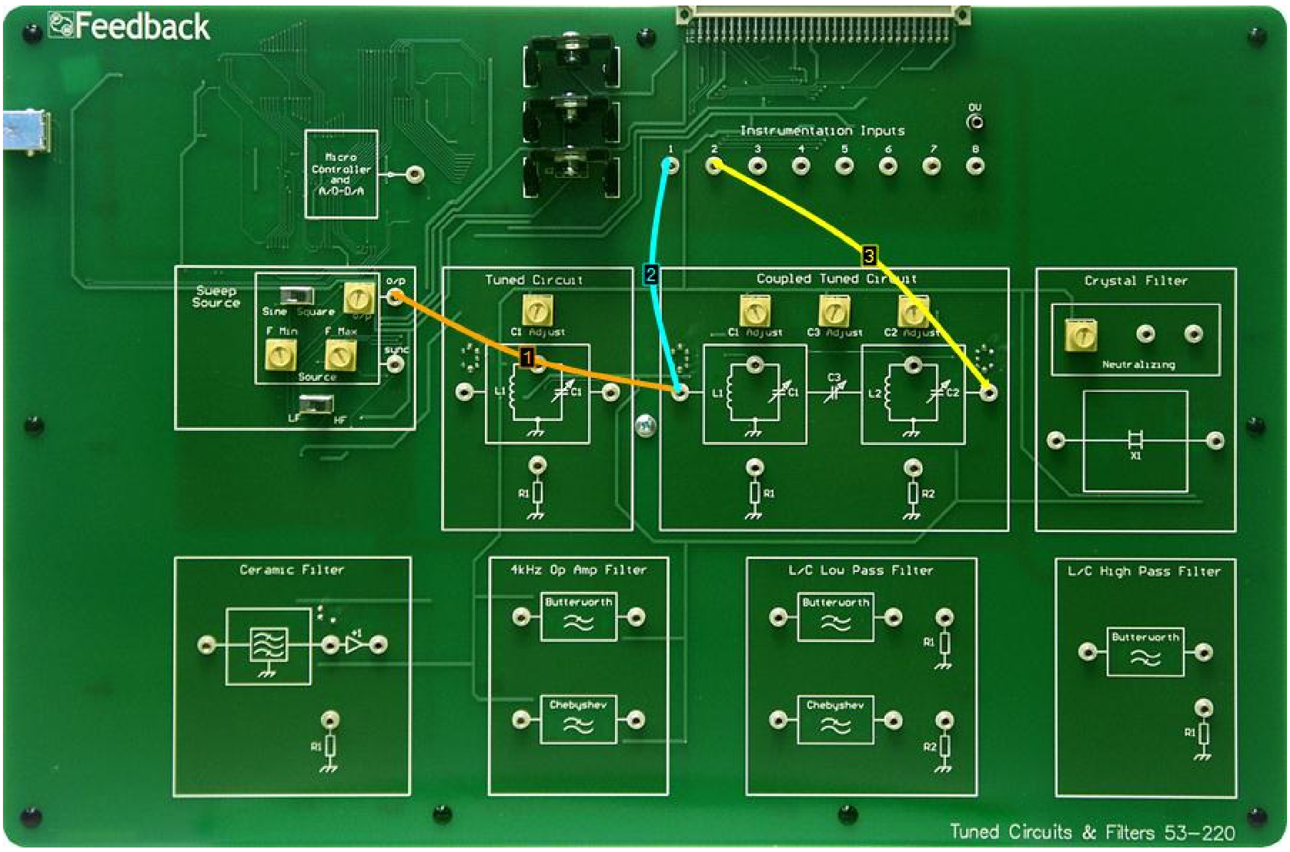
You will see that the GPA can produce plots of both amplitude and phase for the circuit under test (the coupled tuned circuit, in this case) and that these can be displayed in either Cartesian (Bode Plot) form, or polar (Nyquist Plot) form.

You will see how easy it is to acquire these plots and you will observe the effect of changing the values of the resonating capacitors in the tuned circuits.

Block Diagram



Make Connections Diagram



Practical 1: Frequency Response

Perform Practical

Use the **Make Connections** diagram to show the required connections on the hardware.

Identify the **Coupled Tuned Circuit** block, located to the top right centre of the workboard. Set the **C1 Adjust**, **C2 Adjust** and **C3 Adjust** controls to half scale.

Identify the **CH1** and **CH2** gain switches, located towards the top of the workboard, directly below the **Instrumentation Input** sockets.

Set both CH1 and CH2 switches to **Hi Gain**.

Identify the **Sweep Source** circuit block at the left-hand centre of the workboard. In this block, set the **Sine**/**Square** switch to Sine and the **LF**/**HF** switch (just below the Source block) to HF. Set both the **FMin** and **FMax** control to their minimum (fully counter-clockwise) positions. This will set the source frequency to approximately 100kHz.

Set the **o**/**p** control on the Sweep Source to its maximum position.

Open the Gain Phase Analyser (GPA). The **Set Min Freq** button on the GPA is already selected by default. Use the FMin control on the Sweep Source to set the frequency to approximately 800kHz.

Click on the **Set Max Freq** button on the GPA and use the FMax control on the Sweep Source to set the frequency to approximately 1.2MHz.

Click on **Plot** on the GPA to plot the Bode response of the circuit.

Notice the peak of the amplitude response and the way that the phase changes with frequency.

Use the cursor to measure the phase shift at the peak frequency. Try to reason why you get that result.

Now, see the effect of varying C1. Ensure that C1 Adjust, C2 Adjust and C3 Adjust controls are set to half scale and observe the frequency and magnitude of the peak of the response.

Set C1 Adjust to its 3 o'clock position. Again, observe the magnitude and frequency of the peak. Do the same for its 9 o'clock position. Are the results as you might expect?

Reset C1 Adjust to its 3 o'clock position. Observe the peak for C3 Adjust at its 3 and 9 o'clock positions. Explain the responses that you get.

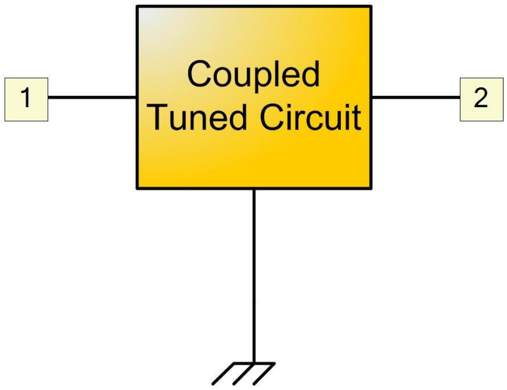
Practical 2: Coupled Response

Objectives and Background

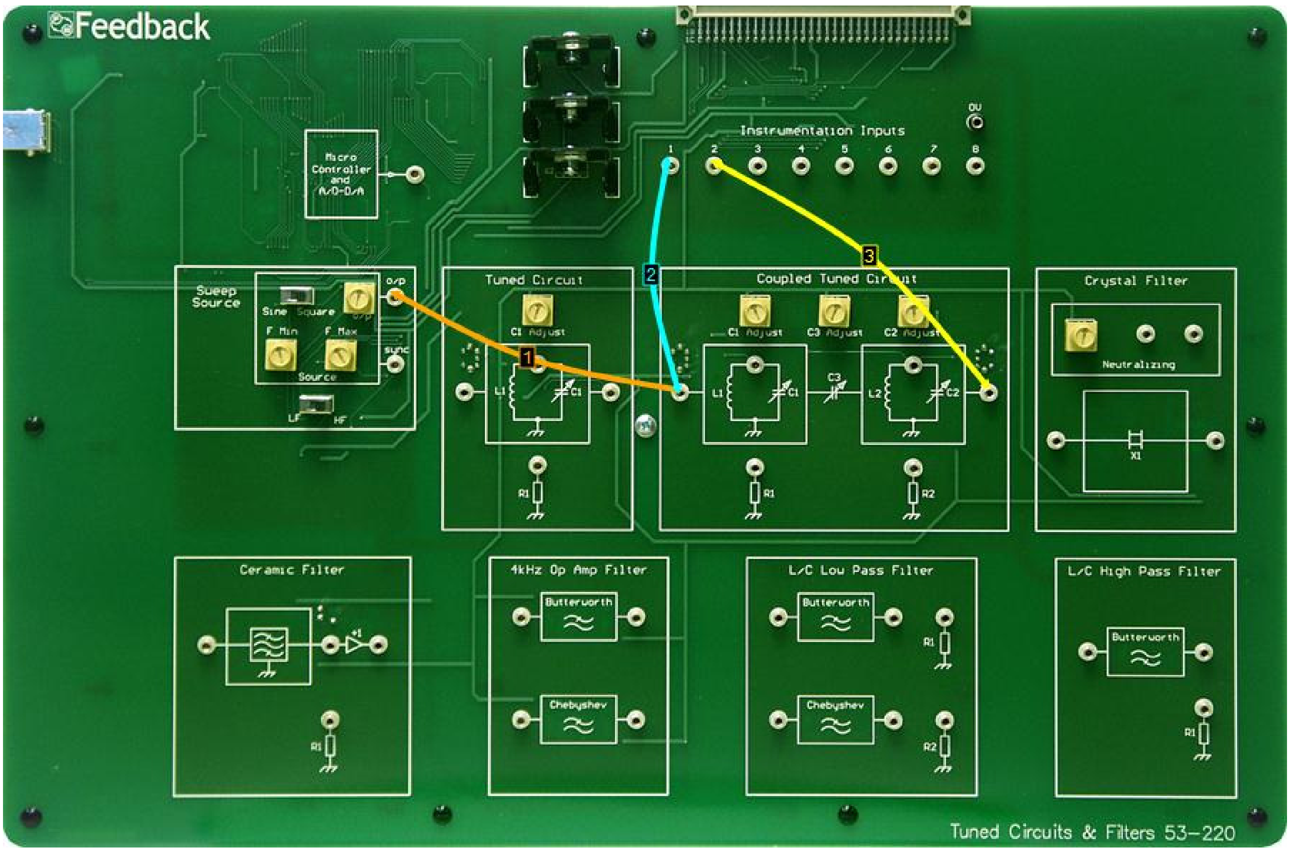
In this Practical you will investigate the effect that the coupling capacitor has on the frequency response of the circuit.

You will measure the response with the tuned circuits both under- and over-coupled and also determine the response for the critically coupled case.

Block Diagram



Make Connections Diagram



Practical 2: Coupled Response

Perform Practical

Use the **Make Connections** diagram to show the required connections on the hardware. The connections are the same as for Practical 1.

Ensure that both **CH1** and **CH2** switches are set to **Hi Gain**.

Ensure that the **C1 Adjust**, **C2 Adjust** and **C3 Adjust** controls in the **Coupled Tuned Circuit** block are set to half scale.

In the **Sweep Source** circuit block, ensure that the **Sine**/**Square** switch is set to Sine and the **LF**/**HF** switch (just below the Source block) to HF. Set both the **FMin** and **FMax** control to their minimum (fully counter-clockwise) positions. This will set the source frequency to approximately 100kHz.

Set the **o**/**p** control on the Sweep Source to its maximum position.

Open the Gain Phase Analyser (GPA). The **Set Min Freq** button on the GPA is already selected by default. Use the FMin control on the Sweep Source to set the frequency to approximately 800kHz.

Click on the **Set Max Freq** button on the GPA and use the FMax control on the Sweep Source to set the frequency to approximately 1.2MHz.

Click on **Plot** on the GPA to plot the Bode response of the circuit.

The response that you see is with a mid value of the coupling capacitor, **C3**. A smaller value of C3 will decrease the coupling between the two tuned circuits, a larger value will increase the coupling.

Turn the C3 Adjust control fully counter-clockwise to its minimum position. This minimises the coupling between the two tuned circuits. Observe the response. This is the type of response that you get with ‘loose’ coupling between tuned circuits. Measure the magnitude and bandwidth of the response.

Now, turn the C3 Adjust control fully clockwise to maximise the coupling between the two tuned circuits. Observe how the response changes. This is the type of response that you get with ‘tight’ coupling between tuned circuits (also known as ‘over-coupling’). Measure the magnitude and bandwidth of the response.

Set the C3 Adjust control back for minimum coupling. Increase the coupling slowly until the response *just* shows signs of the ‘double hump’ associated with over-coupling. Reduce the coupling very slightly from this point, so there is just one peak. This is the coupling position that gives maximum signal transfer between the tuned circuits. This is known as

Critical Coupling. You should see that the response has maximum magnitude and has wider bandwidth with a slightly flat top.

Measure the magnitude and bandwidth of the response and compare these with your other results.

Practical 3: Loading the Circuit

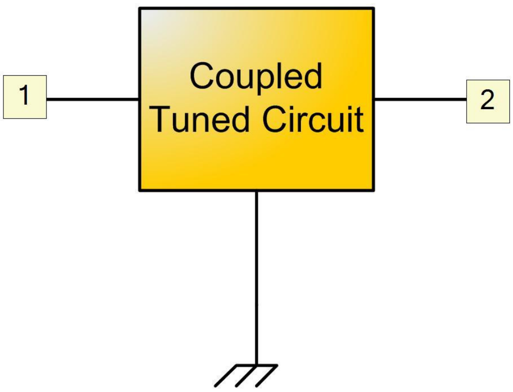
Objectives and Background

In this Practical you will observe the effect that connecting resistive loads across the tuned circuits have on the response of the coupled tuned circuit. You will see that both the magnitude of the output and the bandwidth of the tuned circuit are changed.

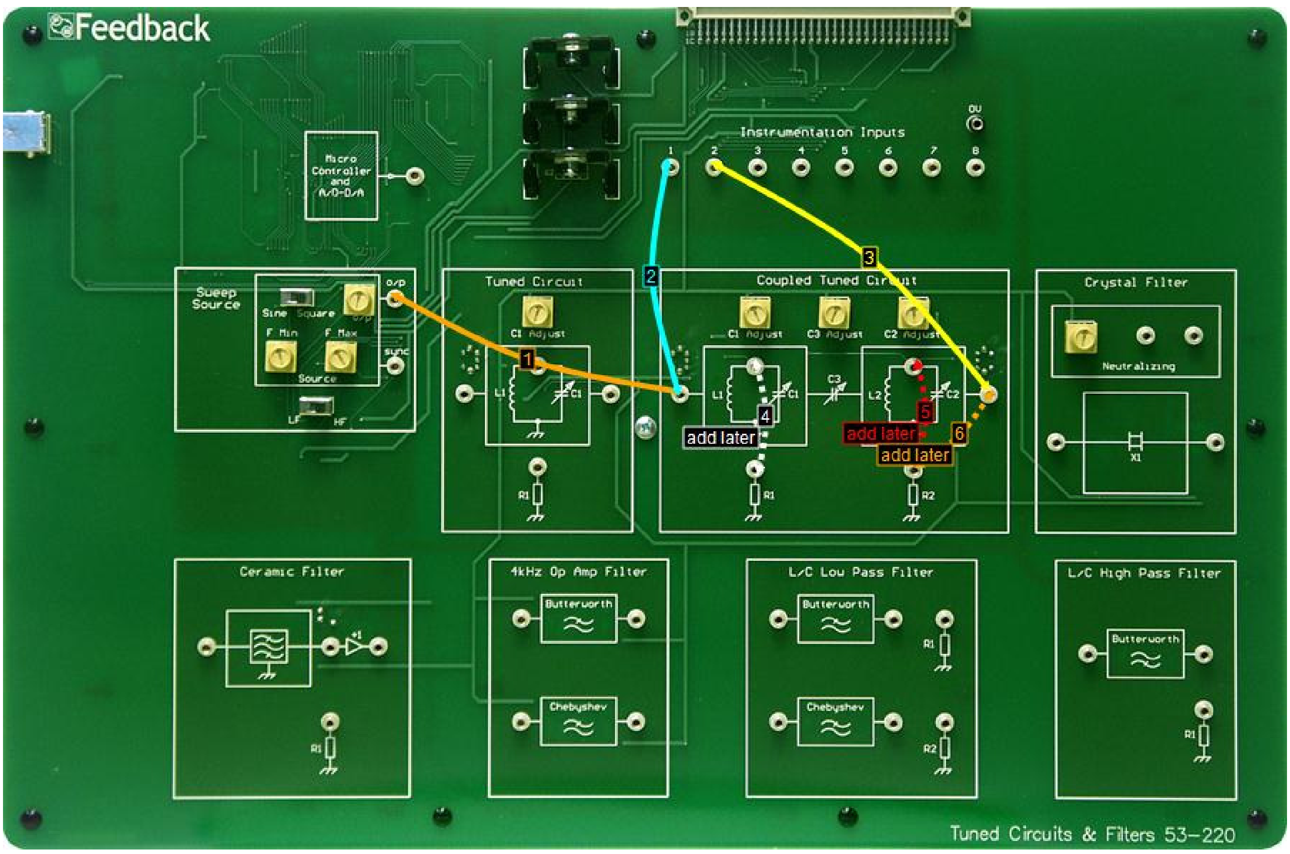
You will measure the bandwidths of the circuit, with and without loading.

You will also see how including a buffer amplifier stage between the tuned circuit and the load can minimise the loading effects.

Block Diagram



Make Connections Diagram



Practical 3: Loading the Circuit

Perform Practical

Use the **Make Connections** diagram to show the required connections on the hardware. The connections are the same as for Practicals 1 and 2.

Ensure that both **CH1** and **CH2** switches are set to **Hi Gain**.

Ensure that the **C1 Adjust**, **C2 Adjust** and **C3 Adjust** controls in the **Couple Tuned Circuit** block are set to half scale.

In the **Sweep Source** circuit block, ensure that the **Sine/Square** switch is set to Sine and the **LF/HF** switch (just below the Source block) to HF. Set both the **FMin** and **FMax** control to their minimum (fully counter-clockwise) positions. This will set the source frequency to approximately 100kHz.

Set the **o/p** control on the Sweep Source to its maximum position.

Open the Gain Phase Analyser (GPA). The **Set Min Freq** button on the GPA is already selected by default. Use the FMin control on the Sweep Source to set the frequency to approximately 800kHz.

Click on the **Set Max Freq** button on the GPA and use the FMax control on the Sweep Source to set the frequency to approximately 1.2MHz.

Click on **Plot** on the GPA to plot the Bode response of the circuit. The response that you see is that for the tuned circuits unloaded.

Now, connect the load resistor **R1** across the first tuned circuit, as shown by connection 4 in the Make Connections diagram. Note the change in response of the circuit.

Remove connection 4 and connect load resistor **R2** across the second tuned circuit, as shown by connection 5 on the Make Connections diagram. Note the new response.

These connections show that loading either of the tuned circuits has a detrimental effect on the response of the coupled tuned circuits, as a whole.

Now, connect the load directly to the output, as shown by connection 6 in the Make Connections diagram. Note any change in the response.

There is a buffer amplifier connected between the second tuned circuit and the output socket.

Crystal Filter

Objectives

To become familiar with the operation of a piezoelectric Crystal Filter

To appreciate the form of the frequency response of a typical, single crystal filter

To investigate the effect on the response of using a neutralizing capacitor

Gain and Phase

Gain

If a signal is applied to an electronic system, the output of that system is unlikely to be exactly the same as the input. For instance, the output is likely to be different in magnitude.

For example, the signal input voltage at the antenna of a radio receiver may be ten microvolts (10μV), whereas the output voltage that drives the loudspeaker in that radio may be ten volts (10V).

The ratio of output signal to input signal (measured in the same units) of a system is referred to as its **gain**.

In the example above, the gain of the radio receiver would be:

$$\frac{10V}{10\mu V} = 1{,}000{,}000$$

Because the units in the equation are those of volts, this gain would be referred to as the **voltage gain** of the receiver.

There are other types of gain that could be quoted for the above receiver. A typical input impedance (resistance) for a radio receiver is 50Ω. Using Ohm's Law, the power applied to that input when 10μV is applied can thus be calculated to be $2x10^{-12}$W and the power output to the speaker to be 6.25W if the loudspeaker's resistance is 16Ω. The **power gain** of the receiver is thus given by:

$$\frac{6.25W}{2x10^{-12}W} = 3.125x10^{12}$$

This is a huge number! More on this later.

Now to calculate the signal currents at the input and the output of the receiver. Again using Ohm's Law, the input current is 0.2μA and the output current is 0.625A. The **current gain** will thus be:

$$\frac{0.625A}{0.2\mu\mathrm{A}} = 3{,}125{,}000$$

Attenuation

For some electronic systems the magnitude of the output signal will be smaller than that of the input signal.

As an example, consider a long telephone line. Due to the resistance of the wire from which the line is constructed there will be loss in the line and the magnitude of the signal at the telephone end of the line (the subscriber's end) will be lower than that at the originating end of the line. It is quite possible for the signal power to be halved in magnitude by the losses in a line. This gives a gain of 0.5 for the line.

Thus gains of less than unity signify **loss** in a system. The term used for this loss is **attenuation**.

The decibel

As can be seen from the two examples of systems given above, the gain of an electronic system may have a value anywhere between a small fraction (if the loss is high) to a huge number (as in the case of the radio receiver).

The use of such a large range of numbers to quantify the gain of systems is, in many ways, very inconvenient. To make things easier, the logarithm of the ratio of output to input is often calculated and quoted.

For example, the logarithm of the power gain of the radio receiver is:

$$\log_{10} 3.125\text{x}10^{12} = 12.5$$

and the logarithm of the gain of the telephone line is:

$$\log_{10} 0.5 = -0.3$$

These are much more convenient numbers to deal with!

Notice that using logs gives positive numbers for gain and negative numbers for loss (attenuation).

The idea of using the log of the ratio was developed in the 19th century to describe the losses associated with long telephone lines. It was initially called the 'transmission unit' and was given the unit name 'the bel', in honour of the telephone pioneer Alexander Graham Bell. However, the bel was rather too large a unit for easy practical use, so the numbers were multiplied by ten and the unit '**decibel**' used.

Thus, this gives the power gain of the radio receiver as:

$$10\log_{10} 3.125\text{x}10^{12} = 125 \text{ decibels}$$

and the power gain of the telephone line as:

$$10\log_{10} 0.5 = -3 \text{ decibels.}$$

These would normally be written as 125dB and –3dB, respectively.

Again, note that positive dBs mean gain and negative dBs mean attenuation.

The units 'bel' or 'decibel' are defined using the ratio of the output to input **power** of a system. Now see what happens if a voltage, or a current ratio is used.

$$\text{Power Gain} = 10\log_{10} \frac{P_{out}}{P_{in}} \text{ dB}$$

Now: $P = V^2/R$, therefore

$$\text{Power Gain} = 10\log_{10}\frac{\frac{V_{out}^2}{R_L}}{\frac{V_{in}^2}{R_{in}}} = 20\log_{10}\frac{V_{out}}{V_{in}}\,\text{dB (if } R_L = R_{in})$$

So, using a voltage ratio means that you have to use 20 times, instead of 10, to get the correct answer in decibels. You will see later what happens if $R_L \neq R_{in}$.

Also, because $P = I^2R$, a similar result is achieved if a current ratio is used, giving:

$$\text{Power Gain} = 20\log_{10}\frac{I_{out}}{I_{in}}\,\text{dB}$$

Now consider a system comprising two parts, the first of which has a power gain of 3 and the second a power gain of 6. To get the total power gain of the system you need to **multiply** the two gains of the parts, giving 18.

Now, in decibels the gains are:

$$10\log_{10} 3 = 4.77\text{ dB}$$

$$10\log_{10} 6 = 7.78\text{ dB}$$

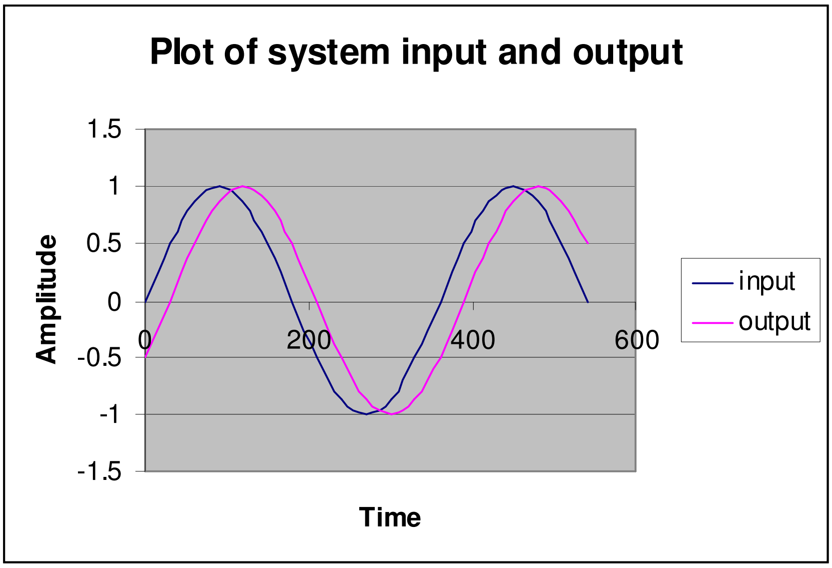
$$10\log_{10} 18 = 12.55\text{ dB}$$

Notice that to get the total gain in dB you just **add** the individual gains in dB – much easier to do than multiplication!

Phase

A second reason for the output of an electronic system to be identical to its input is one of time delay. It takes time for the signals to pass through the system. This time delay may only be parts of a microsecond, but it can have a considerable effect on system performance.

Consider a sinusoidal input signal to a system and the corresponding delayed output (for this example, and simplicity, it is assumed that the system has unity gain). The input and output waveforms may look like the diagram below.



You will see that the output waveform is delayed by a small amount with respect to the input waveform. The delay could be measured in units of time but it is more usual to express it as an angle. This can be done because one cycle of the waveform is equivalent to 2п radians (360 degrees). In the diagram the difference between the two waveforms is 30 degrees.

The difference in degrees (or radians) is referred to as the **phase shift** (or just phase) between the two waveforms.

Frequency Effects

In an electronic system gain and phase are seldom constant with respect to the frequency of the applied signal. Because of this, the system is said to have a **frequency response**. This is just a mathematical, or pictorial (graph) description of how the gain and phase of the system change with frequency.

The frequency response of a system is an important property of that system. Some systems give an increase in gain with increasing frequency. Such a response is called a **high-pass** response.

Some systems have a frequency response with the gain dropping as the frequency increases. Such a response is called a **low-pass** response.

In some systems the gain increases with frequency up to some value and then decreases as the frequency is further increases. Such a response is called a **bandpass** response.

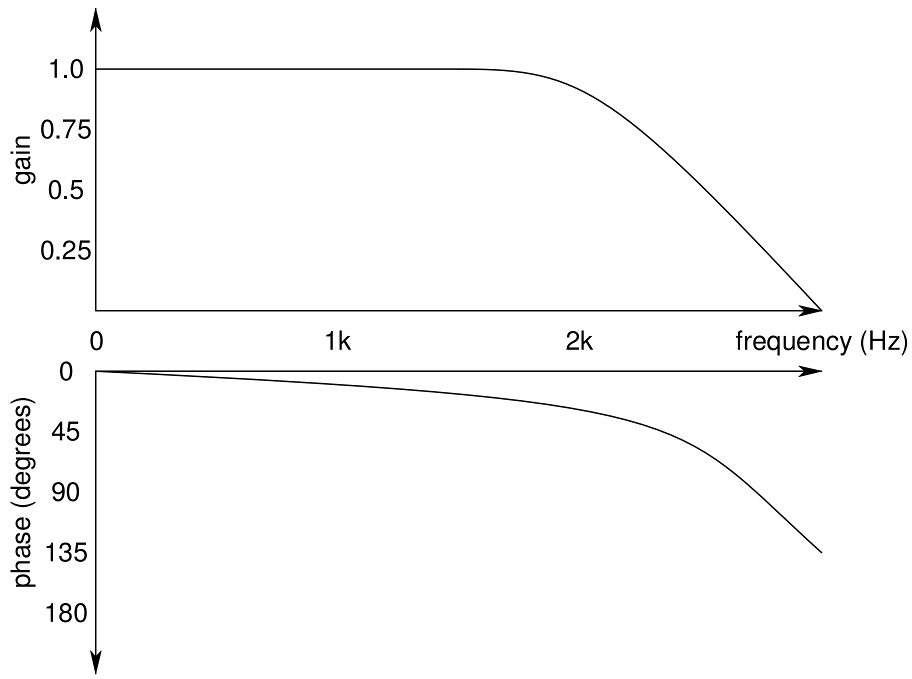
You will be meeting systems with these types of response as you progress through your course.

Plots

One of the most convenient ways of describing the frequency response of a system is in graphical form. This is usually not as accurate as describing it mathematically, but it is often adequate for practical purposes and is normally much easier to see what is happening from a graph, rather than from the mathematics.

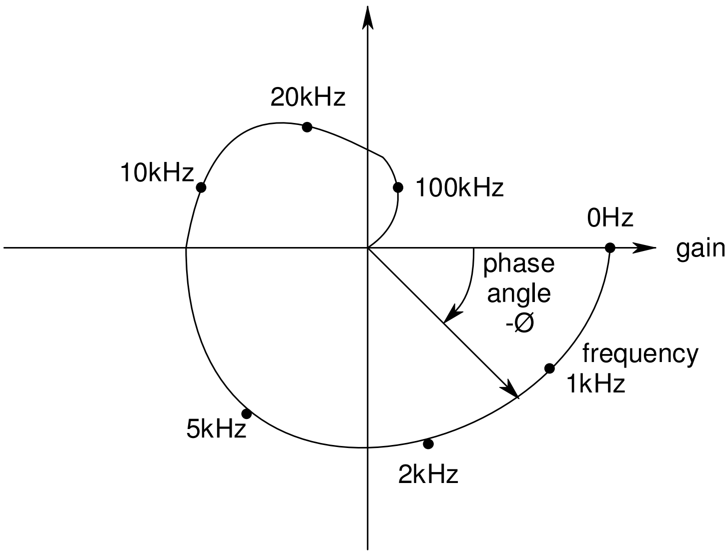
There are two main forms of graphs that are used to show the frequency response of a system. These are named after the persons that devised them and are the **Bode** plot and the **Nyquist** plot.

The **Bode** plot of a system is no more than normal graphs of gain (on the Y axis) against frequency (on the X axis) together with another graph of phase (on the X axis) against frequency. Generally, the gain and the phase curves are plotted on separate axes, one above the other, as shown in the diagram below. The frequency axes are the same for both graphs, so a direct relationship between gain and phase at any required frequency can be made easily.



The second way of displaying the frequency response is by using a vector (or phasor) type plot in which the gain of the system is given by the length of the vector and the phase by the angle of the vector. This type of plot is known as a **Nyquist** plot, after the mathematician who devised it.

An example Nyquist plot of a system is given below.



As you can see, the gain and phase at different frequencies are given by the length and angle (Φ) of the phasor. The corresponding frequencies are usually shown on the plot along the **locus** (the path) of the curve.

Other names for the plots are:

Bode plot – **rectangular** plot

Nyquist plot – **polar** plot

The type of plot that is used depends on the type of system that is being investigated and the properties of the system under investigation. For example, the frequency response of amplifiers and filters normally use the Bode plot form of graph, whereas investigations into control systems and stability use the Nyquist form. However, it is not incorrect to use either form.

You will use both types of plot as you perform assignments using this equipment.

Crystal Filters

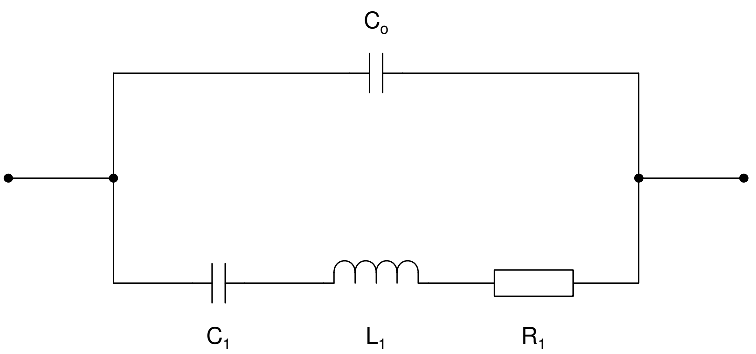
Certain crystals are called **piezoelectric** when they exhibit a relationship between mechanical strain (tension or compression) and voltage across their surfaces. Specifically, when compressed or pulled, a piezoelectric crystal will build up alternate charges on opposite faces, thus acting like a capacitor with an applied voltage. A current, called **piezoelectricity**, can then be generated between the faces. On the other hand, when subjected to an external voltage, the crystal will expand or contract accordingly.

This effect is put to use in several ways, one of which is in **quartz crystal filters**. When a crystal of quartz is correctly cut and mounted, it can be made to bend if a voltage is applied to it. When the voltage is removed the quartz crystal will generate a voltage as it returns to its original shape. The result is that the crystal behaves like a circuit composed of a capacitor, an inductor and a resistor. This is equivalent to a **tuned circuit**, with a resonant frequency determined by the dimensions and properties of the crystal. The crystal will have a very precise resonant frequency and thus very precise filters may be designed using crystals in circuit instead of (or as well as) a conventional LC circuit.

Such crystal filter circuits are common in communication electronics, where the selection of precise frequencies is important. Quartz has the advantage that its dimensions change very little with temperature; hence a crystal filter can be made very thermally stable.

Quartz crystals for filters can be made for frequencies from a few tens of kHz to several tens of MHz.

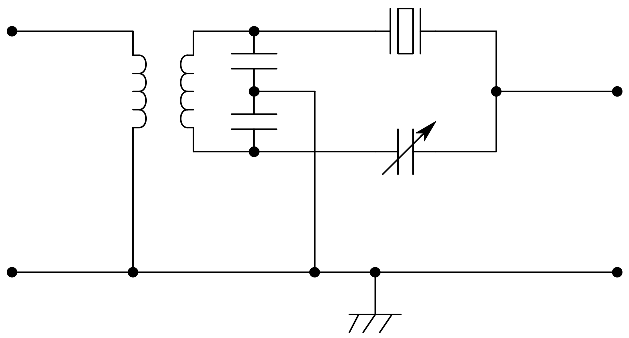
The equivalent circuit for a quartz crystal is shown below.



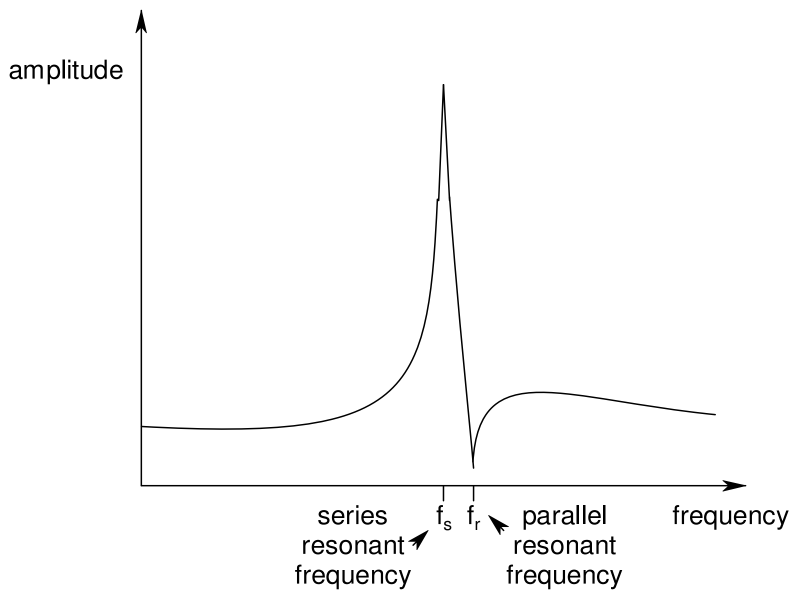
As you can see, there is a series resonant branch comprising C_1, L_1 and R_1. These are really the electrical equivalents of the mechanical properties of the crystal: C_1 represents the compliance of the crystal, L_1 its inertia and R_1 the losses due to lattice friction within the crystal. The electrical equivalent values are an extremely small value for C_1 (parts of a pF), a large value for L_1 (perhaps several Henries) and, as the losses are very low, a very low value for R_1 (parts of an Ohm). This means that the **Q** of the circuit is **very high** (much higher than a conventional LCR resonant circuit) and thus its resonant frequency is very precisely defined.

There is also the parallel branch of C_2. This is a real capacitor – the mounting capacitance of the electrodes and connection wires of the crystal – and is a few pF in practice. There will be a second parallel resonant frequency of the total crystal assembly, including this capacitance. These two resonances are very close together, with the series resonance being generally only a kHz or two below the parallel resonance.

Most filter circuits use the series resonant properties of the crystal, where it has very low impedance at its series resonant frequency, very high impedance to signals at its parallel resonant frequency and moderately high impedance to signals at other frequencies. Thus only signals at the series resonant frequency will be transferred.



The diagram above shows a simple, single crystal, filter circuit. The variable capacitor in the circuit is used to adjust the exact frequency of the parallel resonance of the crystal. This is normally referred to as the 'phasing' or 'neutralising' control.



The diagram above shows a typical frequency response of a simple, single crystal, filter. Notice the extremely narrow bandwidth of the main peak and also the notch in the response. The position of the notch (the parallel resonant frequency, f_r) is variable using the capacitor.

Practical 1: Frequency Response

Objectives and Background

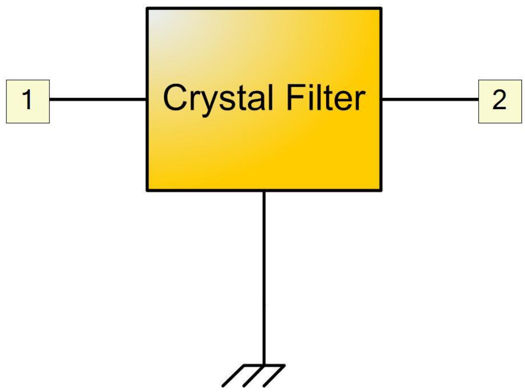
In this Practical you will use the Gain Phase Analyser (GPA) to plot the frequency response of the single-crystal filter circuit.

You will see that the GPA can produce plots of both amplitude and phase for the circuit under test (the crystal filter, in this case) and that these can be displayed in either Cartesian (Bode Plot) form, or polar (Nyquist Plot) form.

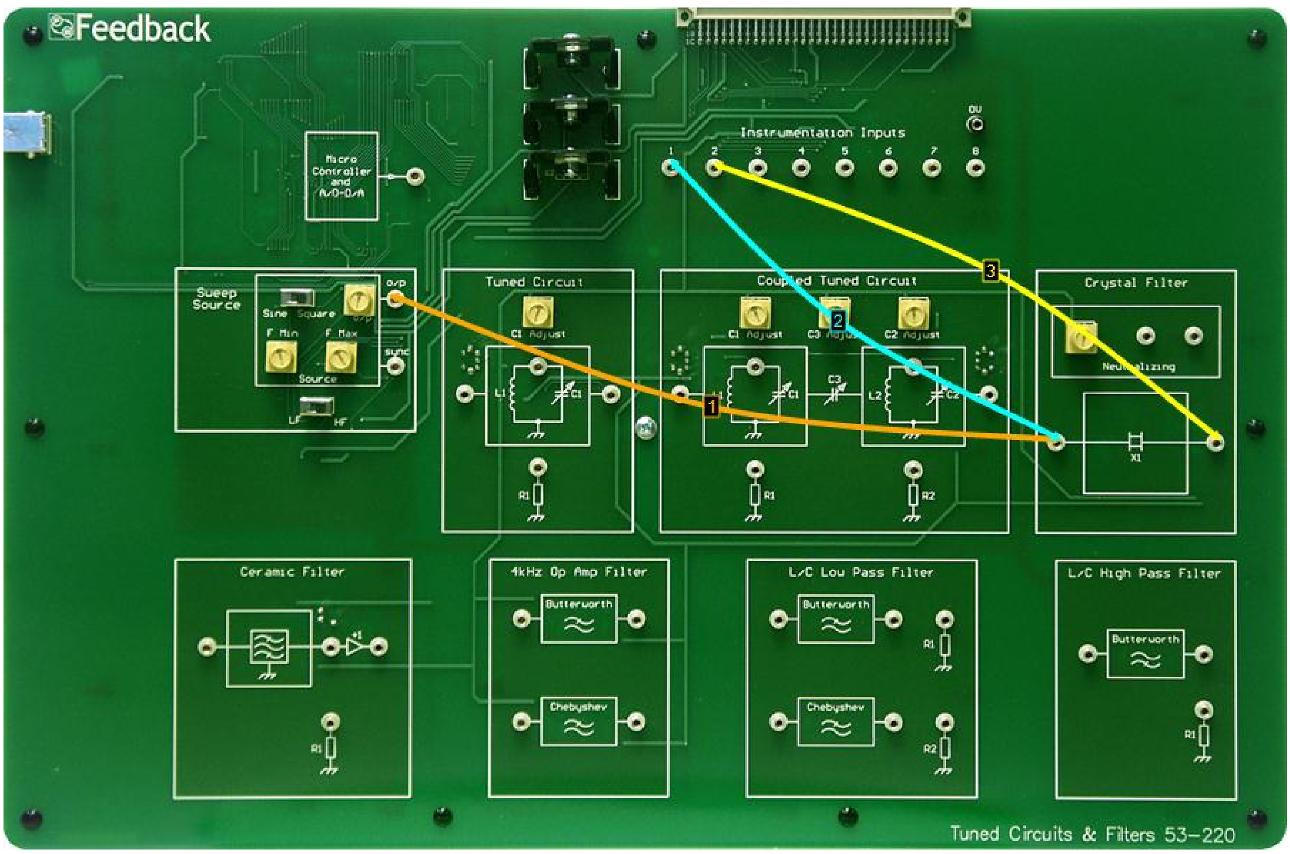
You will see that the extreme sharpness of the frequency response makes the setting of the start and stop frequencies of the GPA rather critical.

You will observe the series and parallel resonances of the crystal and you will measure the frequency difference between them.

Block Diagram



Make Connections Diagram



Practical 1: Frequency Response

Perform Practical

Use the **Make Connections** diagram to show the required connections on the hardware.

Identify the **Crystal Filter** circuit block, located to the top-right of the workboard. Set the **Neutralizing** control to half scale.

Identify the **CH1** and **CH2** gain switches, located towards the top of the workboard, directly below the **Instrumentation Input** sockets.

Set both CH1 and CH2 switches to **Hi Gain**.

Identify the **Sweep Source** circuit block at the left-hand centre of the workboard. In this block, set the **Sine**/**Square** switch to Sine and the **LF**/**HF** switch (just below the Source block) to HF. Set both the **FMin** and **FMax** control to their minimum (fully counter-clockwise) positions. This will set the source frequency to approximately 100kHz.

Set the **o/p** control on the Sweep Source to its maximum position.

Open the Gain Phase Analyser (GPA). The **Set Min Freq** button on the GPA is already selected by default. Use the FMin control on the Sweep Source to set the frequency to approximately 1.75MHz.

Click on the **Set Max Freq** button on the GPA and use the FMax control on the Sweep Source to set the frequency to approximately 1.95MHz.

Click on **Plot** on the GPA to plot the Bode response of the circuit.

Examine the magnitude plot for the crystal. Compare your practical plot with the theoretical one given in the concept on Crystal Filters.

See if you can re-adjust the start and stop frequencies of the sweep to give you a better frequency resolution of the response.

Use the cursor to measure the series and parallel resonant frequencies of the crystal. Determine the frequency difference between them.

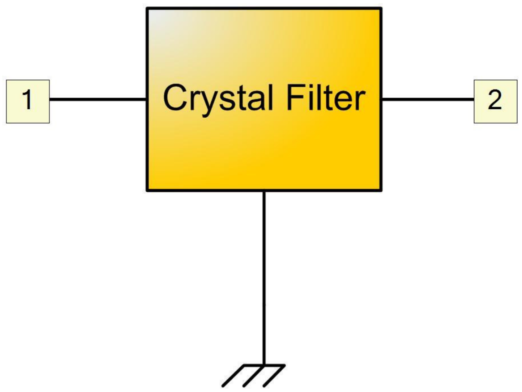
Practical 2: Neutralisation

Objectives and Background

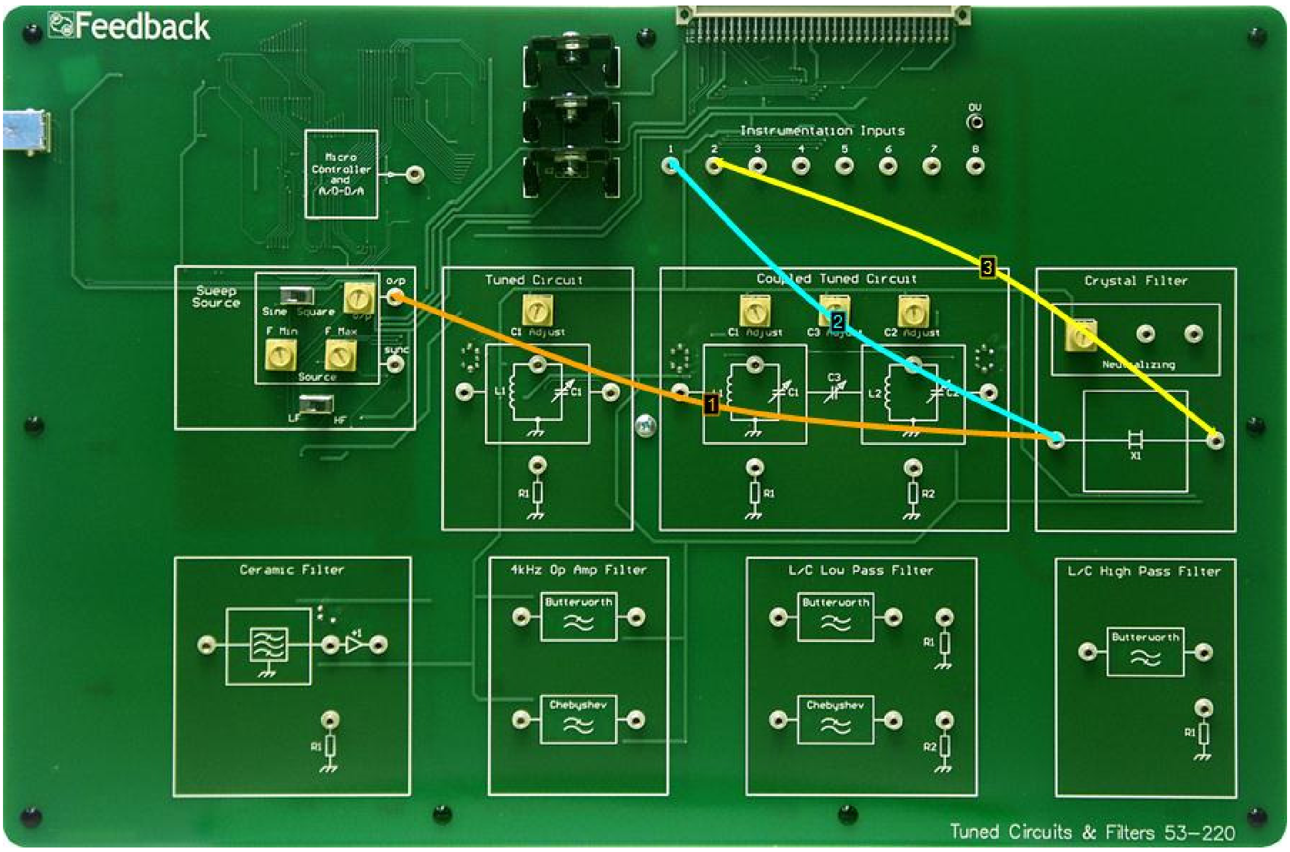
In this Practical you will investigate the effect of the neutralizing control on the response of the filter.

The bandwidth of a crystal filter is very narrow and the plot on the GPA must be done in high resolution mode to be able to distinguish between series and parallel resonances. This means that the time taken for each sweep of the GPA is significant. Thus to see the effect of varying the neutralization can be a slow, painstaking procedure.

Block Diagram



Make Connections Diagram



Practical 2: Neutralisation

Perform Practical

Use the **Make Connections** diagram to show the required connections on the hardware. The connections are the same as for Practical 1.

Ensure that both **CH1** and **CH2** switches are set to **Hi Gain**.

Ensure that the **Neutralizing** control in the **Crystal Filter** circuit block is set to half scale.

In the **Sweep Source** circuit block, ensure that the **Sine**/**Square** switch is set to Sine and the **LF**/**HF** switch (just below the Source block) to HF. Set both the **FMin** and **FMax** control to their minimum (fully counter-clockwise) positions. This will set the source frequency to approximately 100kHz.

Set the **o**/**p** control on the Sweep Source to its maximum position.

Open the Gain Phase Analyser (GPA). The **Set Min Freq** button on the GPA is already selected by default. Use the FMin control on the Sweep Source to set the frequency to approximately 1.8MHz.

Click on the **Set Max Freq** button on the GPA and use the FMax control on the Sweep Source to set the frequency to approximately 1.9MHz.

Ensure that the **Hi Res** box on the GPA is ticked.

Click on **Plot** on the GPA to plot the Bode response of the circuit.

Observe the parallel resonance part of the response. This is the notch in the response, to the right (higher in frequency) of the series resonant peak.

Now, adjust the Neutralizing control and see if there is any change in the response. You will have to change the position of the control slightly and then wait for a new sweep of the plot to see any difference.

With very careful adjustment, you should be able to find a position of the control that almost flattens out the notch.

You may also find that too much neutralization will lower the series resonance peak.

Ceramic Filter

Objectives

To become familiar with the operation of a Ceramic Filter

To appreciate the form of the frequency response of a typical Ceramic Filter

To investigate the effect on the response of resistive loading the filter

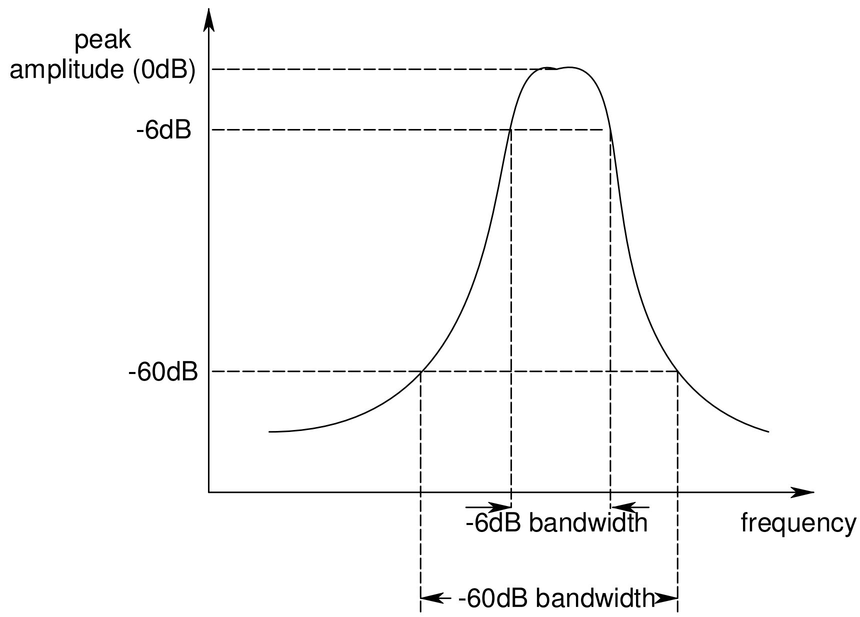
Ceramic Filters

Piezoelectric effects are not confined to quartz crystals. Certain ceramics, such as lead zirconate titanate (PZT), exhibit piezoelectric properties and are also used in communications and electronics.

Small discs of PZT, which resonate in the radial dimension, can form selective filters in much the same way as quartz. An advantage of PZT over quartz is one of cost. A disadvantage is that PZT has considerably lower Q.

The simplest ceramic filters use just one resonator (similar to the simple, single crystal, quartz filter), but numbers of resonators can be coupled together to form filters of the required bandwidth and shape factor.

The shape factor of a filter is normally defined as the ratio of the bandwidths at two defined attenuations from the peak of the frequency response (often at –6dB and –60dB down from the peak magnitude). This is illustrated below.



$$SF = \frac{BW_{-60dB}}{BW_{-6dB}}$$

For instance, if the bandwidth at –6dB is 3kHz and that at –60dB is 6kHz, the shape factor would be 2.

Such a filter might be useful in an IF (intermediate frequency) stage in a radio receiver for receiving SSB (single sideband, suppressed carrier, amplitude modulated) signals that have a communications speech bandwidth of approximately 3kHz.

Ceramic filters tend to be more economical than crystal filters, but have lower temperature stability and often have greater pass-band attenuation.

Crystal Filters

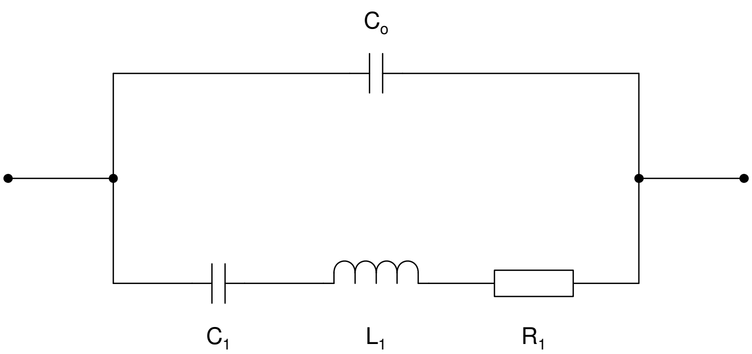
Certain crystals are called **piezoelectric** when they exhibit a relationship between mechanical strain (tension or compression) and voltage across their surfaces. Specifically, when compressed or pulled, a piezoelectric crystal will build up alternate charges on opposite faces, thus acting like a capacitor with an applied voltage. A current, called **piezoelectricity**, can then be generated between the faces. On the other hand, when subjected to an external voltage, the crystal will expand or contract accordingly.

This effect is put to use in several ways, one of which is in **quartz crystal filters**. When a crystal of quartz is correctly cut and mounted, it can be made to bend if a voltage is applied to it. When the voltage is removed the quartz crystal will generate a voltage as it returns to its original shape. The result is that the crystal behaves like a circuit composed of a capacitor, an inductor and a resistor. This is equivalent to a **tuned circuit**, with a resonant frequency determined by the dimensions and properties of the crystal. The crystal will have a very precise resonant frequency and thus very precise filters may be designed using crystals in circuit instead of (or as well as) a conventional LC circuit.

Such crystal filter circuits are common in communication electronics, where the selection of precise frequencies is important. Quartz has the advantage that its dimensions change very little with temperature; hence a crystal filter can be made very thermally stable.

Quartz crystals for filters can be made for frequencies from a few tens of kHz to several tens of MHz.

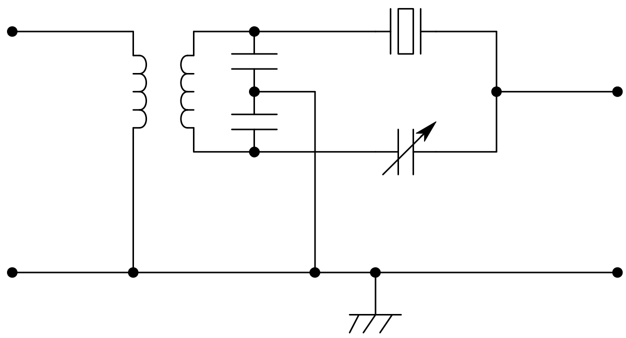
The equivalent circuit for a quartz crystal is shown below.



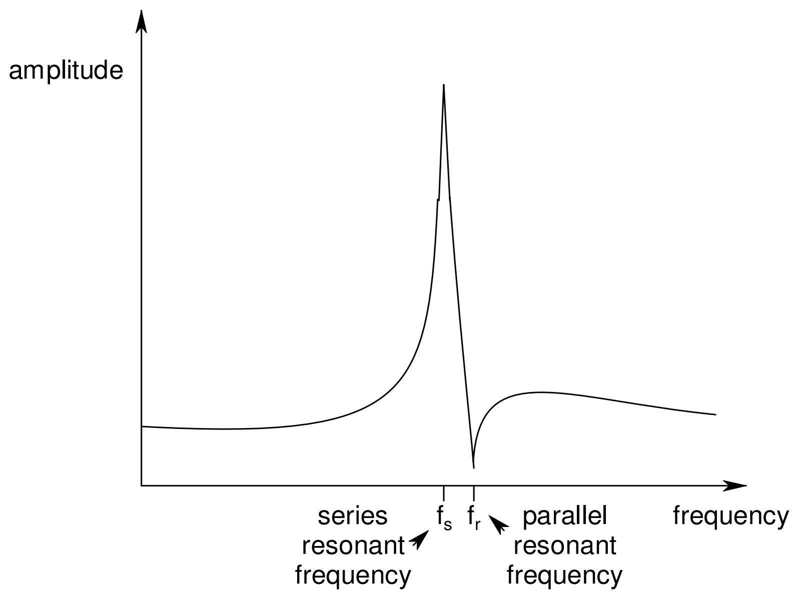
As you can see, there is a series resonant branch comprising C_1, L_1 and R_1. These are really the electrical equivalents of the mechanical properties of the crystal: C_1 represents the compliance of the crystal, L_1 its inertia and R_1 the losses due to lattice friction within the crystal. The electrical equivalent values are an extremely small value for C_1 (parts of a pF), a large value for L_1 (perhaps several Henries) and, as the losses are very low, a very low value for R_1 (parts of an Ohm). This means that the **Q** of the circuit is **very high** (much higher than a conventional LCR resonant circuit) and thus its resonant frequency is very precisely defined.

There is also the parallel branch of C_2. This is a real capacitor – the mounting capacitance of the electrodes and connection wires of the crystal – and is a few pF in practice. There will be a second parallel resonant frequency of the total crystal assembly, including this capacitance. These two resonances are very close together, with the series resonance being generally only a kHz or two below the parallel resonance.

Most filter circuits use the series resonant properties of the crystal, where it has very low impedance at its series resonant frequency, very high impedance to signals at its parallel resonant frequency and moderately high impedance to signals at other frequencies. Thus only signals at the series resonant frequency will be transferred.



The diagram above shows a simple, single crystal, filter circuit. The variable capacitor in the circuit is used to adjust the exact frequency of the parallel resonance of the crystal. This is normally referred to as the ‘phasing’ or ‘neutralising’ control.



The diagram above shows a typical frequency response of a simple, single crystal, filter. Notice the extremely narrow bandwidth of the main peak and also the notch in the response. The position of the notch (the parallel resonant frequency, f_r) is variable using the capacitor.

Practical 1: Frequency Response

Objectives and Background

In this Practical you will use the Gain Phase Analyser (GPA) to plot the frequency response of the ceramic filter circuit.

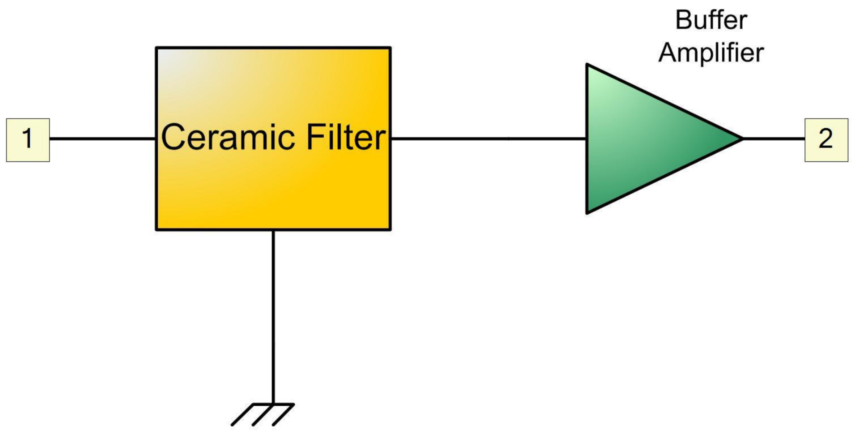
You will see that the GPA can produce plots of both amplitude and phase for the circuit under test (the ceramic filter, in this case) and that these can be displayed in either Cartesian (Bode Plot) form, or polar (Nyquist Plot) form.

You will see that the frequency response is that of a band-pass filter with only a few kilohertz bandwidth. This makes the setting of the start and stop frequencies of the GPA rather critical.

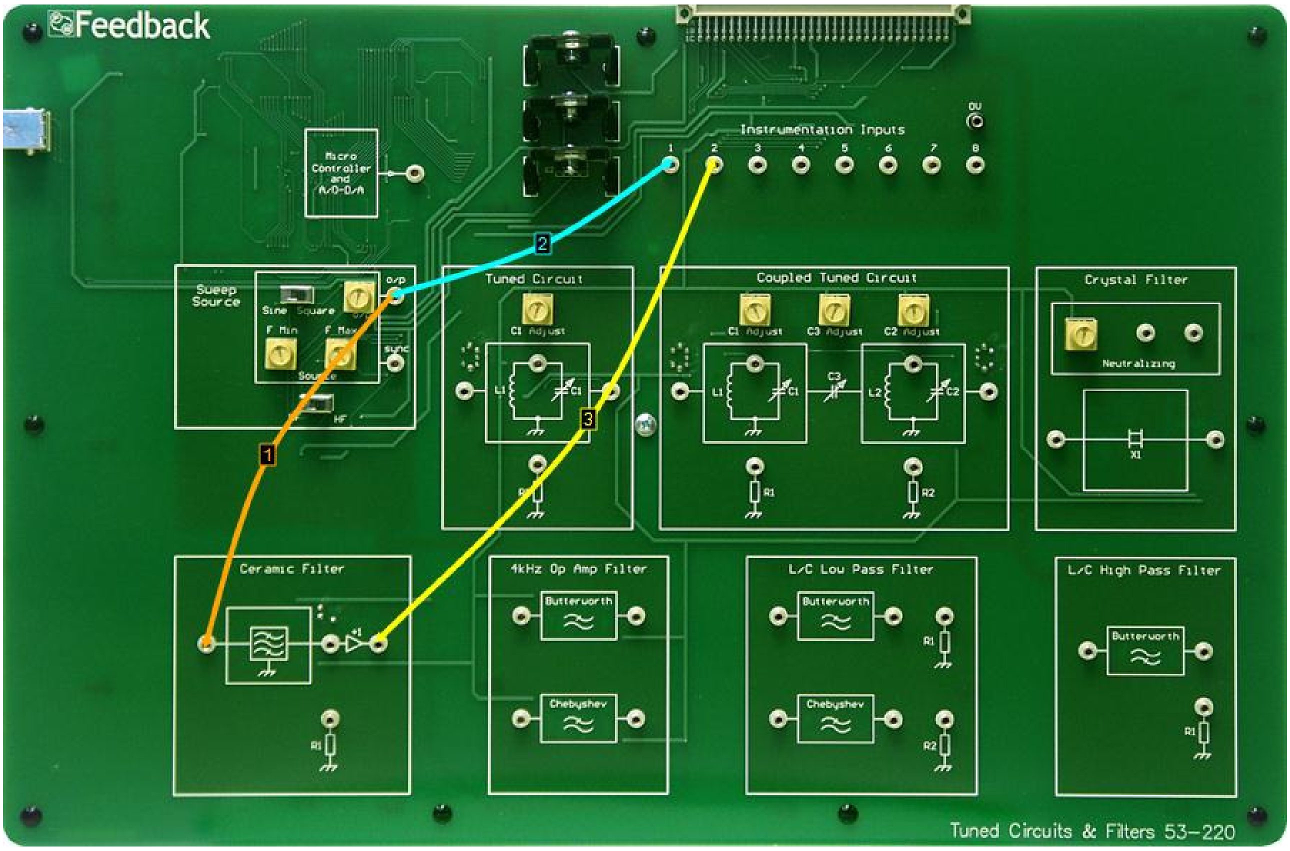
You will observe the typical passband ripple associated with a multi-element ceramic filter and also the steep sides to the cut-off part of the response.

You will measure the filter bandwidth and its shape factor.

Block Diagram



Make Connections Diagram



Practical 1: Frequency Response

Perform Practical

Use the **Make Connections** diagram to show the required connections on the hardware.

Identify the **Ceramic Filter** circuit block, located to the bottom left-hand corner of the workboard.

Identify the **CH1** and **CH2** gain switches, located towards the top of the workboard, directly below the **Instrumentation Input** sockets.

Set both CH1 and CH2 switches to **Hi Gain**.

Identify the **Sweep Source** circuit block at the left-hand centre of the workboard. In this block, set the **Sine**/**Square** switch to Sine and the **LF**/**HF** switch (just below the Source block) to HF. Set both the **FMin** and **FMax** control to their minimum (fully counter-clockwise) positions. This will set the source frequency to approximately 100kHz.

Set the **o**/**p** control on the Sweep Source to its maximum position.

Open the Gain Phase Analyser (GPA). The **Set Min Freq** button on the GPA is already selected by default. Use the FMin control on the Sweep Source to set the frequency to approximately 400kHz. Do not exceed 420kHz

Click on the **Set Max Freq** button on the GPA and use the FMax control on the Sweep Source to set the frequency to approximately 500kHz. Do not set it to less than 480kHz.

Click on **Plot** on the GPA to plot the Bode response of the circuit.

Examine the magnitude plot for the filter. Compare your practical plot with the theoretical one given in the concept on Ceramic Filters.

See if you can re-adjust the start and stop frequencies of the sweep to give you a better frequency resolution of the response.

Note that the filter has a much wider passband than the single crystal filter.

Note the ripple in the passband of the filter. Also, note the steep sides of the filter response. Both of these characteristics are quite typical for a general purpose ceramic filter.

Use the cursor to measure the bandwidths of the filter at the –3dB and –30dB points down the response. Note that it may be difficult to determine the –3dB bandwidth accurately, due to the passband ripple present.

Calculate the ratio between the –30dB and –3dB bandwidths. This is known as the *shape factor* of the filter.

(Note: the shape factor of a bandpass filter is often defined for attenuations other than –3dB and –30dB. For instance, a professional grade filter for use in the intermediate frequency stages of a communications receiver would have a stop-band that might be greater than –80dB. For such a filter the –6dB and –60dB points may be used to specify the filter shape factor. To be sure of a filter's performance, always read the manufacturer's specification!).

Note the phase part of the Bode response. Again, the multiple excursions in phase between +180 and –180 degrees are typical of a multi-element ceramic, or crystal, filter. For the majority of applications these rapid changes of phase are not significant.

Practical 2: Terminating the Filter

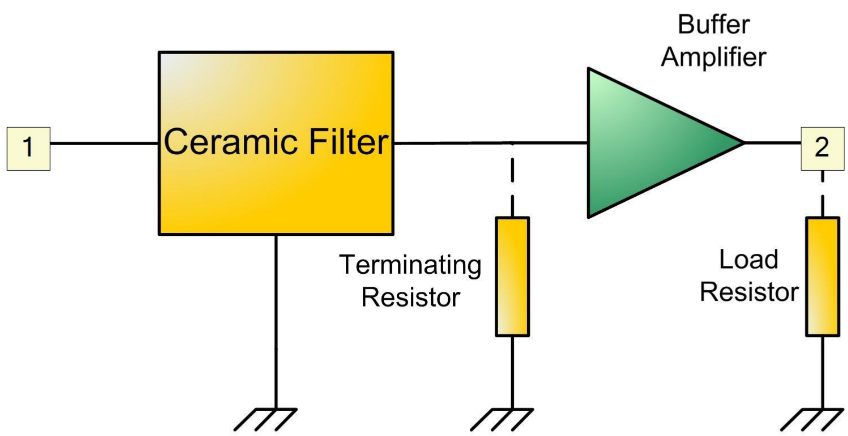
Objectives and Background

In this Practical you will investigate the effects of terminating the filter in its optimum load resistance.

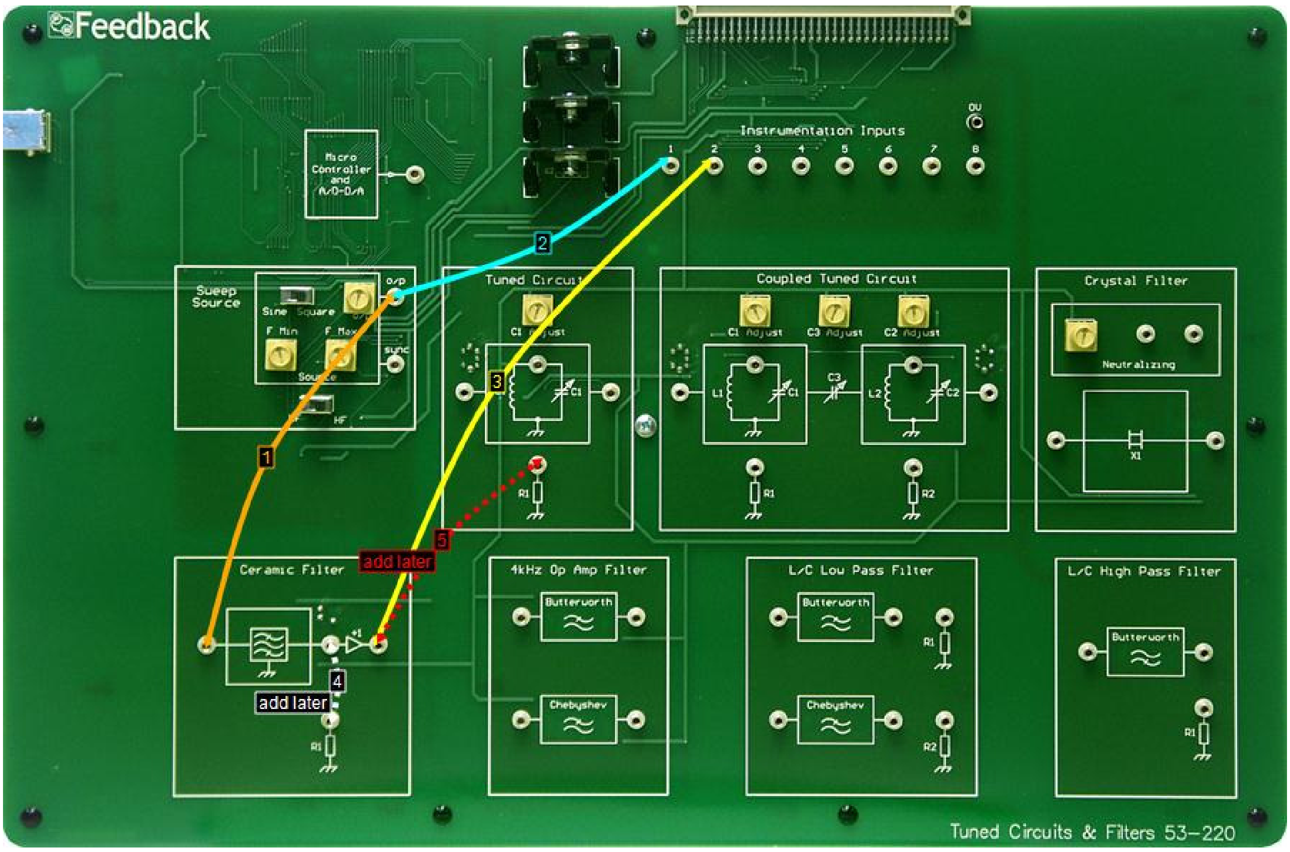
You will see that the passband ripple is improved and that the stop-band attenuation is slightly greater.

You will also see the effect of following the filter with a buffer amplifier and how it isolates the filter from further load changes.

Block Diagram



Make Connections Diagram



Practical 2: Terminating the Filter

Perform Practical

Use the **Make Connections** diagram to show the required connections on the hardware. The connections are the same as for Practical 1.

Ensure that both **CH1** and **CH2** switches are set to **Hi Gain**.

In the **Sweep Source** circuit block, ensure that the **Sine**/**Square** switch is set to Sine and the **LF**/**HF** switch (just below the Source block) to HF. Set both the **FMin** and **FMax** control to their minimum (fully counter-clockwise) positions. This will set the source frequency to approximately 100kHz.

Set the **o**/**p** control on the Sweep Source to its maximum position.

Open the Gain Phase Analyser (GPA). The **Set Min Freq** button on the GPA is already selected by default. Use the FMin control on the Sweep Source to set the frequency to approximately 400kHz. Do not exceed 420kHz

Click on the **Set Max Freq** button on the GPA and use the FMax control on the Sweep Source to set the frequency to approximately 500kHz. Do not set it to less than 480kHz.

Click on **Plot** on the GPA to plot the Bode response of the circuit. Note, again, the passband ripple associated with the filter.

Ceramic filters are designed to operate with a known load. This is generally known as the *terminating resistance*. In the case of the filter used on the workboard, the optimum value for the terminating resistor is 2kΩ. The resistor **R1** is of this value.

Connect the terminating resistor R1 as shown by connection 4 on the Make Connections diagram.

Observe the new plot on the GPA and how it differs from the un-terminated case.

Notice the decrease in passband ripple and the slightly better stop-band attenuation. In most applications, filters should be terminated by their optimum load resistance to achieve the best response.

As shown on the schematic on the workboard, the filter is followed by a buffer amplifier. Connecting a load to the output of the buffer should not affect the filter response. Verify this by connecting a load resistor as shown by adding connection 5 on the Make Connections diagram.

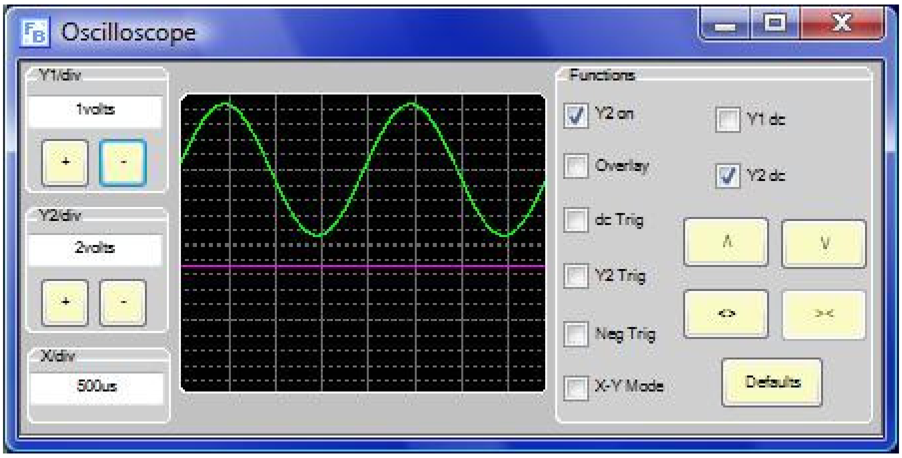
Using the Test Equipment

General Notes

Any of the instruments can be resized and moved at any time using conventional 'drag-and-drop' mouse techniques. If you make an instrument small enough then only the display area will be shown; you must increase its size again in order to restore the controls. If you close any of the instruments and open them again they will return to their default settings. Each instrument has a Defaults button which returns the equipment to its default settings (equivalent to closing and re-opening the instrument). If you want to return all the instruments (and any other resource windows) to their default size and position simply click the Auto Position button in the assignment side bar.

Some instruments allow you to place a cursor (by clicking the mouse) at any position on their display; the cursor reveals information regarding the point at which it is located. You will have to reactivate this cursor each time you change the settings, size or position of the instrument.

The Oscilloscope



The Espial oscilloscope has many of the functions that you would find on a conventional or computer-driven scope. Its fundamental purpose is to show varying waveforms plotted against time. It is a dual trace scope, which means that it can display two separate waveforms at the same time.

The Y (voltage) axis is set to a default value by the practical for each channel, but you may change it by using the + button for more volts/div and the - button for less volts/div. Either only one channel can be displayed or both channels. The Y2 Show tick box determines whether the second channel is shown. In two-channel mode, if the Overlay box is ticked, the two traces are superimposed on the same scale as for one trace. If Overlay is not ticked the display area is divided into two and each trace is displayed half-size. The Y1 dc and Y2 dc tick-boxes determine if the inputs are dc coupled or not (ac coupled). If the signal has a large dc offset then ac coupling can be useful.

The X (time) axis is set to a default value by the practical but you may change it by using the ^ button for a faster timebase and the v button for a slower timebase. The <> and >< buttons provide a means of further expanding the trace if the highest, or lowest, timebase is in use. If you have the X scale expanded and select a lower timebase speed then the X scale automatically returns to its default setting.

An anti-alias feature automatically switches the time-base speed up if you select a rate that may produce a misleading display due to aliasing. If this feature has increased the timebase rate then the ^ button is coloured red.

The oscilloscope can also be operated in X-Y mode, where data from channel 1 is in the vertical axis and data from channel 2 is in the horizontal axis. Because the oscilloscope is a digital sampling scope, in X-Y mode the time base settings are still relevant and determine the sampling rate for both channels. Also in X-Y mode the traces have persistence and stay on the screen longer than one trace refresh.

Note that you can switch off the anti-aliasing feature from the main laboratory screen.

Triggering takes place when the selected trace crosses the zero volt level. If the Y2 Trig box is ticked, then the trigger source is Channel 2. Otherwise, Channel 1 is used. The Neg trig box enables only negative transitions to trigger the scope. Normally only positive ones do.

If the signal has a large dc offset, ac triggering can be useful.

You can return to the default settings by pressing the Default button. The Auto Position button on the Espial laboratory window moves **all** the test instruments back to their default positions and sizes on the screen but does not affect their settings.

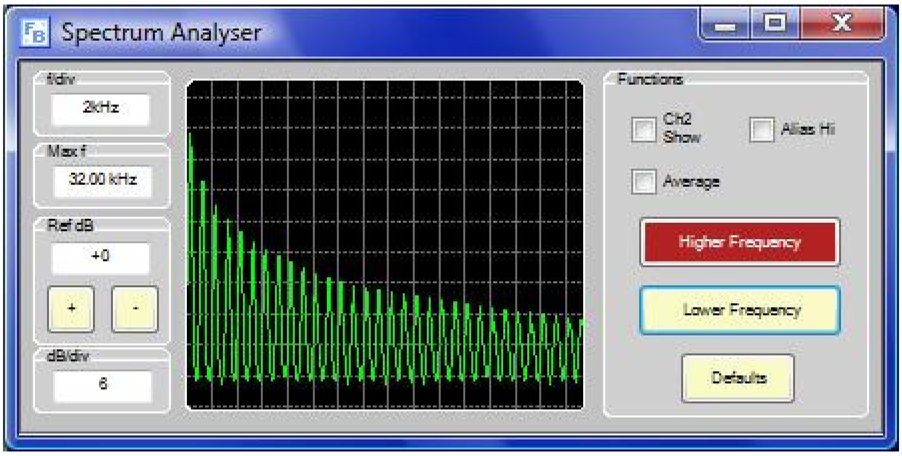
A cursor is available to make more accurate measurements. Left click on the display area to activate it. The green cursor can be moved to anywhere on a waveform. Move the mouse away and back into it to allow a tool-tip window to open with the measurement data displayed for that point.

You have to reactivate the cursor if you change the settings, size or position of the oscilloscope.

By right clicking on the display an options box appears. The options available are:

Print Display – Sends image to the default printer.
Export Display to File – Opens a window enabling the name for the file you wish to use to be entered and the directory where to save the file can be selected.
Export Display to File (reverse colours) - Opens a window enabling the name for the file you wish to use to be entered and the directory where to save the file can be selected.

The Spectrum Analyser



The spectrum analyser enables you to look at signals in the frequency domain. In common with many modern test instruments, it uses DSP to transform time domain data into frequency domain data. The mathematics to do this is called a Fourier transform.

The Y (amplitude) scale is calibrated in Decibels relative to an arbitrary dotted line near to the top of the screen. The dB scale is linear and the number of dB per division is shown in the box. The Y (amplitude) axis is set to a default value by the practical, but you may change it by using the + button or the - button to change the Ref dB value higher or lower. The minimum level that you can see is determined by the assignment, and ultimately by the noise in the system.

The analyser has the capability of showing two channels at the same time. Click Ch2 Show button to show channel 2 as well as channel 1.

The X (frequency) axis is calibrated in MHz, kHz or Hz per division, as appropriate. The default scale is set by the practical but you may change it by using the Higher Frequency and Lower Frequency buttons.

The anti-alias feature will operate if you try to set the frequency too low. The Higher Frequency button is shown red if this feature has increased the frequency. Note that if a new frequency component appears such as noise, the anti-alias feature may operate suddenly. The Alias Hi tick-box allows you to increase the threshold at which the anti-alias feature operates. This allows signals to be examined that have larger amounts of harmonic content. The default setting for this is off.

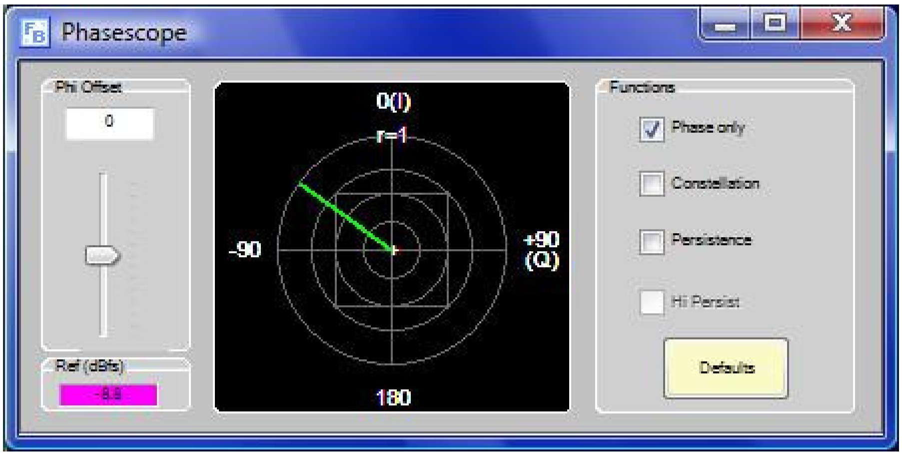
A cursor is available to make more accurate measurements. Left click on the display area to activate it. The green cursor can be moved to anywhere on a waveform. Move the mouse away and back into it to allow a tool-tip window to open with the measurement data displayed for that point.

You will have to reactivate the cursor if you change the settings, size or position of the spectrum analyser.

By right clicking on the display an options box appears. The options available are:

Print Display – Sends image to the default printer.
Export Display to File – Opens a window enabling the name for the file you wish to use to be entered and the directory where to save the file can be selected.
Export Display to File (reverse colours) - Opens a window enabling the name for the file you wish to use to be entered and the directory where to save the file can be selected.

The Phasescope



The Phasescope is a special instrument that compares two signals in phase and amplitude (magnitude). The two signals are referred to as the reference and the input. The display is in polar format, i.e. the phase is in the form of a circle and the amplitude as the radius. The use of a circle is possible because phase is a continuous function repeating every 360 degrees. The display can be seen as Polar, as the one orthogonal axis represents the real component and other the imaginary part. The convention here is that the real axis is the X axis, which means that zero degrees is straight up or at 12 on a clock face. +90 degrees is at 3 on the clock face and –90 at 9. It is important to note that in terms of phase +180 degrees is the same as –180 degrees.

The radius scale has one circle at radius = 1 (the outermost circle) i.e. the two signals are of the same amplitude. Further inner circles are at 0.707, 0.5 and 0.25.

The circle at 0.5 has a square associated with it, the corners of which are at 0.707. This represents the case when two orthogonal vectors of amplitude = 0.5 are added.

In some cases only the phase is of interest so, if you click the Phase Only box, the radius is set to 1.

The conventional display is that of a vector i.e. a line joining the point to the centre. However, in some cases it is much easier to interpret the display if only a point is drawn. Where the amplitude and phase is varying between discrete values they are shown as a pattern of dots resembling stars, hence the term constellation display. This mode can be selected by ticking the Constellation box. In constellation mode, the persistence of the display can be varied. By selecting the Persistence tick box, traces stay on the screen for a number of trace refreshes before being removed. By selecting Hi Persist this time is extended.

If the two signals are of different frequencies the result is a continuously rotating vector, rotating at a rate equal to the difference in frequency. The direction depends on the sign of the frequency difference. If the rate is fairly fast, the instrument may only be able to show a limited number of discrete values.

In many cases the reference input will not be at exactly zero degrees with respect to the theoretical zero degrees of the input signal. This causes the display to be rotated. In some cases this may be important to know, but where it is not the Phi Offset control gives the ability to rotate the display for easier interpretation.

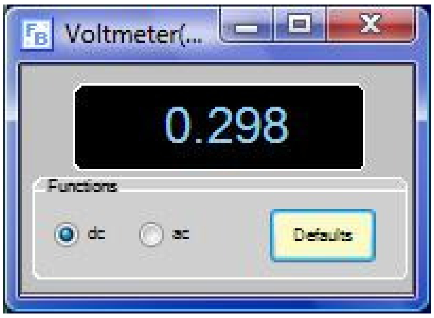
The coloured indicator (Ref Ch) to the top left of the display tells you which probe is being used as the reference channel.

A cursor is available to enable more accurate measurement. Click the display to use it.

By right clicking on the display an options box appears. The options available are:

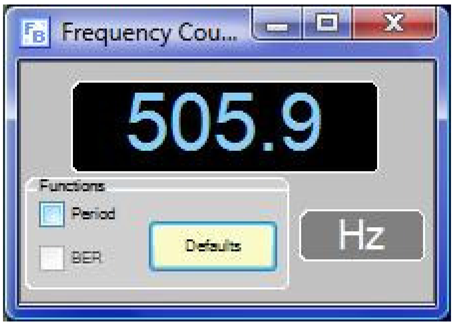
Print Display – Sends image to the default printer.
Export Display to File – Opens a window enabling the name for the file you wish to use to be entered and the directory where to save the file can be selected.
Export Display to File (reverse colours) - Opens a window enabling the name for the file you wish to use to be entered and the directory where to save the file can be selected.

The Voltmeter



The meter is simply an ac and dc voltmeter that displays the value in digital form. It can be used in ac mode by clicking ac p-p, in which case the value represents the peak to peak value. If the waveform has a high crest factor the results can be slightly surprising. In dc mode, if there is an ac component present, the average value is displayed.

The Frequency Counter



This has the facilities of a conventional frequency meter/counter. It will display in either frequency or time. If the input amplitude is too low a warning message will be displayed.

Like all frequency counters, it can produce misleading results if the waveform is complex or contains many frequencies.

The Gain Phase Analyser (GPA)

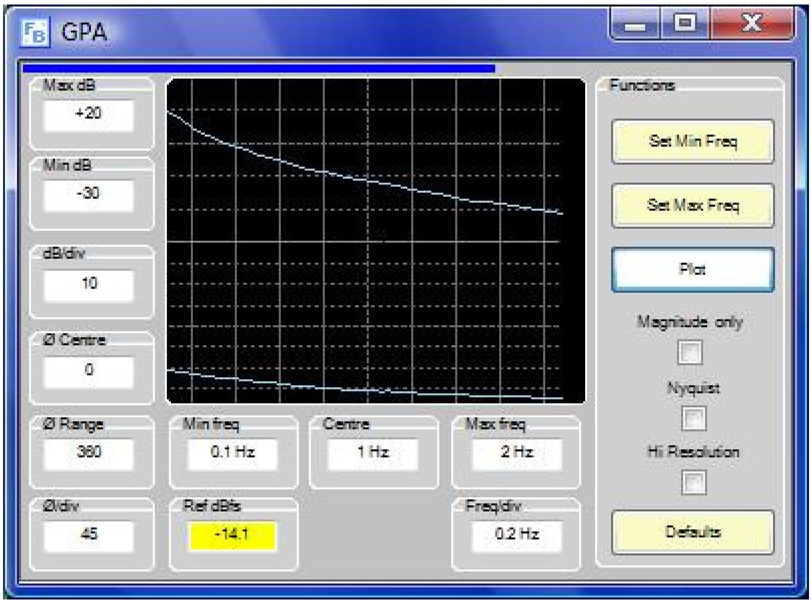
The GPA displays graphically the gain and phase characteristics between a selectable range of frequencies, either as Bode plots (the default mode) or as a Nyquist plot.

Whenever the GPA is opened, the Set Min Freq button on the GPA is initially selected. The minimum frequency that you set in the circuit whilst this button is selected will represent the low end of the range to be plotted. After setting the minimum frequency, select the Set Max Freq button and adjust the maximum frequency in the circuit to the high end of the range. When either the Set Min Freq or Set Max Freq buttons are selected, the frequency will be shown numerically on the GPA display. Finally, select the Plot button to plot the gain and phase between the frequencies you have chosen. Note that the GPA takes time to plot at low frequencies.

You can verify (and change) the minimum or maximum frequencies at any time by selecting the Set Min Freq or Set Max Freq buttons as required (and adjusting the low or high frequency if desired). You will then need to select the Plot button again. Note that only one of these three buttons can be selected at any time.

Tick the Hi Resolution box to get a better frequency resolution of the plot, although plotting will take longer.

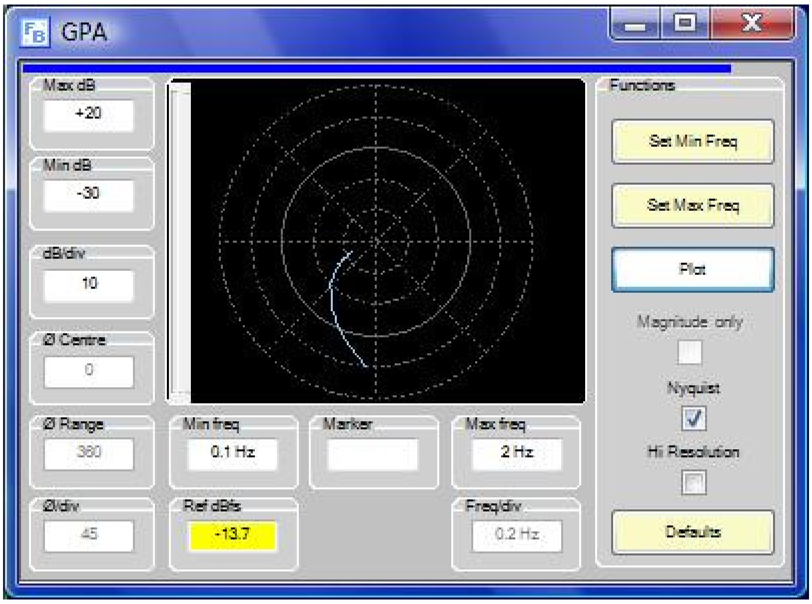
Bode plots are the most widely used means of displaying and communicating frequency response information. Bode diagrams are presented as two separate graphs: one showing magnitude and one showing phase, both plotted against frequency. Because the axes are logarithmic, they condense a wide range of frequencies (horizontal axis) and a wide range of gains (vertical axis) into the graphical area. In Bode plots, commonly encountered frequency responses have a shape that is simple and easy to recognise.



A Bode diagram shows two traces, representing the magnitude and phase, from which you can see variations with frequency. You are able to measure the gain and phase at different frequencies by clicking the cursor on the GPA display. It is much easier to use the GPA than the oscilloscope for this purpose.

Tick the Magnitude only box if you prefer to display the gain trace without the phase trace. This box is disabled when you select a Nyquist plot.

The Nyquist plot of a system is simply the polar representation of the Bode plots. This plot combines the magnitude and phase on a single graph, with the frequency as a parameter along the curve. Nyquist plots are particularly helpful for stability analysis in control system design.



On the GPA, tick the Nyquist box to create a Nyquist plot (you should already have set the minimum and maximum frequencies as described above). Use the vertical slider bar at the left-hand side of the GPA display to move the cursor to any point along the plotted curve. Then read off the frequency, magnitude and phase by hovering the mouse over that cursor. The convention here is that the zero degrees radial is at 3 on a clock face.

Untick the Nyquist box to return to Bode plot mode. You can switch between Bode and Nyquist modes at any time.

By right clicking on the display an options box appears. The options available are:

Print Display – Sends image to the default printer.
Export Display to File – Opens a window enabling the name for the file you wish to use to be entered and the directory where to save the file can be selected.
Export Display to File (reverse colours) - Opens a window enabling the name for the file you wish to use to be entered and the directory where to save the file can be selected.

Espial System Help

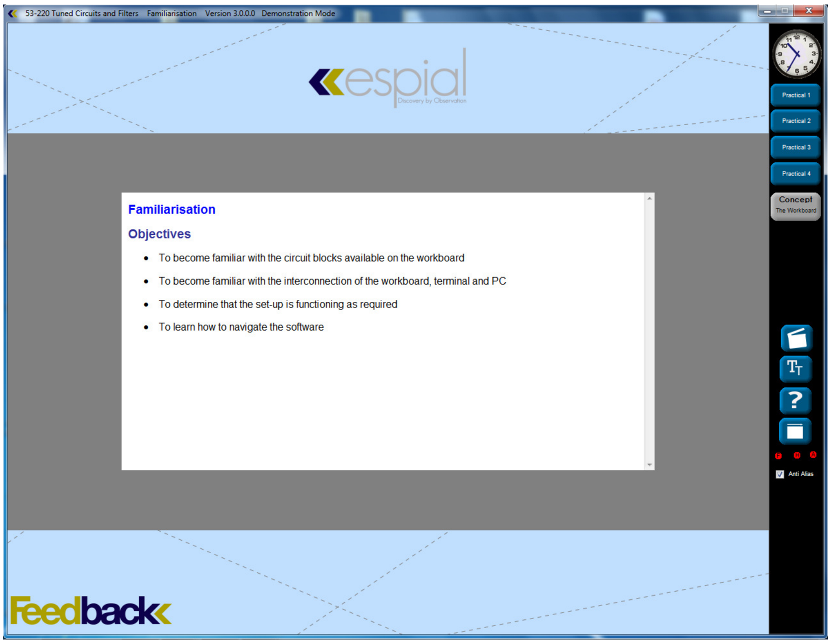
Although the Espial environment is very easy to operate, these notes will help you use all its facilities more quickly.

If there is a demonstration assignment, slider controls in the software perform functions that would normally be performed on the hardware. In normal assignments, if the any of hardware systems fail to initialise the system reverts to demonstration mode. This means that none of the test equipment is available.

The Assignment Window

The assignment window opens when an assignment is launched. If you are reading this you have already found the help button in the side bar of the assignment window!

The assignment window consists of a title bar across the top, an assignment side bar at the right-hand edge, and the main working area. By default, the overall assignment objectives are initially shown in the main working area whenever an assignment is opened. The assignment window occupies the entire screen space and it cannot be resized (but it can be moved by 'dragging' the title bar, and it can be minimised to the task bar). The title bar includes the name of the selected assignment. The side bar contains the practicals and any additional resources that are relevant for the selected assignment. The side bar cannot be repositioned from the right-hand edge of the assignment window. An example of an assignment window is shown below.



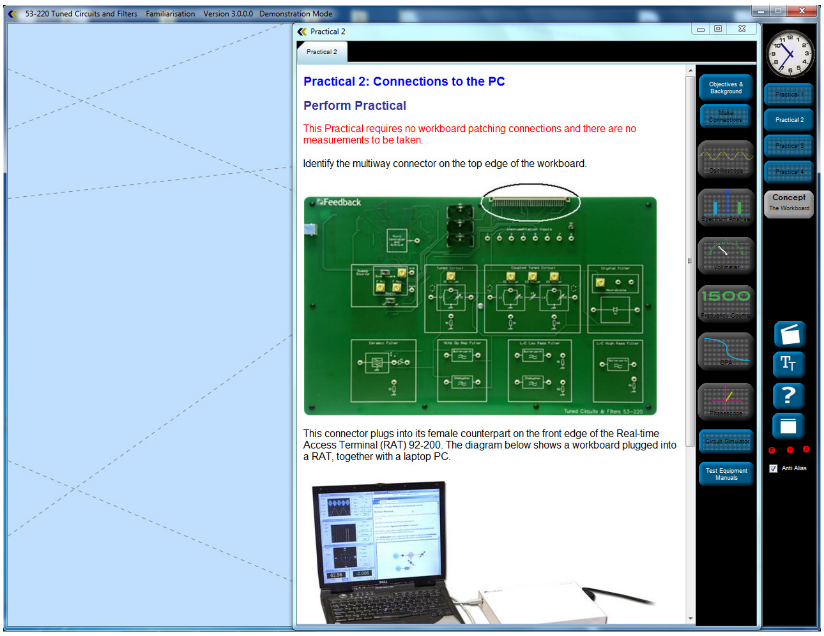
The precise appearance of the assignment window will depend on the ‘skin’ that has been selected by your tutor. However, the behaviour of each of the buttons and icons will remain the same, irrespective of this.

The clock (if you have one active) at the top of the side bar retrieves its time from the computer system clock. By double clicking on the clock turns it into a stop watch. To start the stop watch single click on the clock, click again to stop the stop watch. Double clicking again will return it to the clock function.

There are a number of resource buttons available in the assignment side bar. These are relevant to the selected assignment. In general, the resources available will vary with the assignment. For example, some assignments have video clips and some do not. However, the Technical Terms, Help and Auto Position buttons have identical functionality in every assignment. You can click on any resource in any order, close them again, or minimise them to suit the way you work.

Practicals are listed in numerical order in the side bar. When you hover the mouse over a practical button, its proper title will briefly be shown in a pop-up tool-tip. There can be up to four practicals in any assignment. You can have only one practical window open at any time.

To perform a practical, left-click on its button in the assignment side bar. The assignment objectives, if shown in the main working area, will close, and the selected practical will appear in its own window initially on the right-hand side of the main working area, as shown below. You can move and resize the practical window as desired (even beyond the assignment window). However, its default size and position is designed to allow the test equipment to be displayed down the left-hand side of the main working area without overlapping the instructions for the practical.



Again, the precise appearance of the practical window can be determined by your tutor but the behaviour of each of the buttons and icons will remain the same, irrespective of this. Whatever it looks like, the practical window should have icons for the test equipment, together with buttons for Objectives & Background, Make Connections, Circuit Simulator and Test Equipment Manuals. These resources are found in side bar, located on the right-hand edge of the practical window. The resources will depend on which practical you have selected. Therefore not all the resources are available in every practical. If a resource is unavailable, it will be shown greyed out. To open any resource, left-click on its icon or button. Note that when you close a practical window, any resources that you have opened will close. You may open any resource at any time, provided it available during the practical. The Circuit Simulator will only be available if you have one loaded.

Note that if the hardware is switched off, unavailable, or its software driver is not installed, all the test equipment is disabled. However, you can open any other window. If you switch on the hardware it will be necessary to close the assignment window and open it again to enable the test equipment.

Resource Windows

These are windows may be moved, resized and scrolled. You may minimise or maximise them. The system defaults to 'Auto Position', which means that as you open each resource window it places it in a convenient position. Most resource windows initially place themselves inside the practical window, selectable using tabs. Each one lays over the previous one. You can select which one is on top by clicking the tab at the top of the practical window. You can see how many windows you have open from the number of tabs. If you want to see several windows at once then drag them out of the practical window to where you wish on the screen. If you close a window it disappears from the resources tab bar.

If you want to return all the windows to their default size and position simply click the Auto Position button in the assignment side bar.

Make Connections Window

This movable and resizable window shows the wire connections (2mm patch leads) you need to make on the hardware to make a practical work. Note that some of the wires connect the monitoring points into the data acquisition switch matrix. If this is not done correctly the monitoring points on the practical diagram will not correspond with those on the hardware. The window opens with no connections shown. You can show the connections one by one by clicking the Show Next button or simply pressing the space bar on the keyboard. If you want to remove the connections and start again click the Start Again button. The Show Function button toggles the appearance of the block circuit diagram associated with the practical.

Test Equipment

The test instruments will auto-place themselves on the left of the screen at a default size. You may move or resize any instrument at any time. Note that below a useable size only the screen of the instrument will be shown, without the adjustment controls. Each piece of test equipment will launch with default settings. You may change these settings at any time. There is an auto anti-alias feature that prevents you setting time-base or frequency settings that may give misleading displays. If auto anti-alias has operated the button turns red. You can turn off the anti-aliasing feature, but you should be aware that it may result in misleading displays.

You may return to the default settings by pressing the Default button on each piece of test equipment. If you wish to return all the equipment to their original positions on the left of the screen click Auto Position on the side bar of the assignment window.

Note that if you close a piece of test equipment and open it again it returns to its default position and settings.

If you want more information on how a piece of test equipment works and how to interpret the displays, see the Test Equipment Manuals resource in the practical side bar.

On slower computers it may be noticeable that the refresh rate of each instrument is reduced if all the instruments are open at once. If this is an issue then only have open the instrument(s) you actually need to use.

Test Equipment Cursors

If you left click on the display of a piece of test equipment that has a screen, a green cursor marker will appear where you have clicked. Click to move the cursor to the part of the trace that you wish to measure. If you then move the mouse into the cursor a tool-tip will appear displaying the values representing that position. Note if you resize or change settings any current cursor will be removed.

Practical Window

This window contains the instructions for performing the practical, as well as a block, or circuit, diagram showing the circuit parts of the hardware board involved in the practical. On the diagram are the monitoring points that you use to explore how the system works and to make measurements. The horizontal divider bar between the instructions and the diagram can be moved up and down if you want the relative size of the practical instruction window to diagram to be different. Note that the aspect ratio of the diagram is fixed.

Information Buttons on Practical Diagrams

On many of the symbols on the diagram you will find a button that gives access to new windows that provide more information on the circuit that the symbol represents. Note that these windows are “modal”, which means that you can have only one open at a time and you must close it before continuing with anything else.

A Further Information point looks like this 

Probes

The practical diagram has probes on it, which start in default positions. These determine where on the hardware the signals are being monitored.

Selecting and Moving the Probes

Probes are indicated by the coloured icons like this .

If this probe is the *selected probe* it then looks like this  (notice the black top to the probe). You select a probe by left clicking on it.

Monitor points look like this 

If you place the mouse over a monitor point a tool-tip will show a description of what signal it is.

You can move the selected probe by simply clicking on the required monitor point. If you want to move the probe again you do not have to re-select it. To change which probe is selected click on the probe you want to select.

You can also move a probe by the normal ‘drag-and-drop’ method, common to ‘Windows’ programs.

Probes and Test Equipment Traces

The association between probes and traces displayed on the test equipment is by colour. Data from the blue probe is displayed as a blue trace. Yellow, orange and green probes and traces operate in a similar way. Which piece of test equipment is allocated to which probe is defined by the practical.

Note that the phasescope shows the relative phase and magnitude of the signal on its input probe using another probe as the reference. The reference probe colour is indicated by the coloured square to the top left corner of the phasescope display.

Practical Buttons

On some practicals there are buttons at the bottom of the diagram that select some parameter in the practical. These can be single buttons or in groups. Only one of each button in a group may be selected at one time.

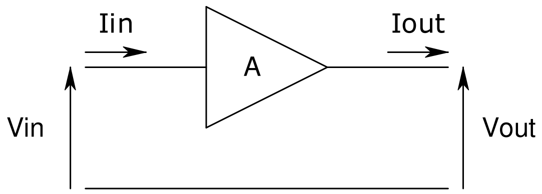
Slider Controls

Where slider controls are used you may find you can get finer control by clicking on it and then using the up and down arrow keys on your keyboard.

Amplifiers

Gain

The block diagram of a general amplifier is given below.



The gain of the amplifier is denoted by the symbol A. The definition of A is given by:

$$\text{Gain, A} = \frac{(output\ power)}{(input\ power)}$$

This is called the **power gain** of the amplifier.

If a circuit does not have power gain, then it is not an amplifier!

As you can see, there is present input current and input voltage and, at the output, the corresponding output current and voltage. As power is the product of current and voltage, this gives the expression for the gain:

$$A = \frac{Vout.Iout}{Vin.Iin}$$

Now:

$\frac{Vout}{Vin} = A_V$, the **voltage gain** of the amplifier

and:

$\frac{Iout}{Iin} = A_I$, the **current gain** of the amplifier

Therefore:

$$A = A_V \, . \, A_I$$

Note: that it is quite possible for an amplifier to have a voltage gain of less than one; however, its corresponding current gain must be high enough to give a power gain greater than unity for it to be classed as an amplifier. This also works the other way: if the current gain is less than one, the voltage gain must be high enough.

Going back to the expression for gain of:

$$A = \frac{Vout.Iout}{Vin.Iin}$$

This can be further re-arranged by defining two more relationships:

$$\frac{Vout}{Iin} = G_m$$, the **transresistance gain** of the amplifier

and:

$$\frac{Iout}{Vin} = R_m$$, the **transconductance gain** of the amplifier

Giving, therefore:

$$A = G_m \,.\, R_m$$

Using a similar argument to before, to qualify as an amplifier, it is quite possible for either one or the other of these terms to be less than one providing the product of the two is greater than unity.

Types of Amplifier

The input signal to an amplifier may be a current or it may be a voltage. Therefore, this gives rise to two types of amplifier: the current input amplifier and the voltage input amplifier.

Each of these types of amplifier may be further sub-divided, as each can give a current output or a voltage output.

There are, therefore, four general forms of amplifier.

Different applications require amplifiers to have different properties of amplification.

An example

Consider a fibre optic communications system with a great distance between transmitter and receiver. Because of the attenuation of light along the length of fibre it is often necessary to compensate for this loss by having ‘repeater’ circuits at intervals along the fibre. The purpose of these circuits is to detect the incoming light signal, convert it into an electrical signal that can be amplified and then used to drive a secondary light source that provides a regenerated signal for transmission further down the cable.

A typical sensor for the input of such a repeater is a photodiode. This will produce a current that is proportional to the light intensity. A typical output device might be a light-emitting diode, or perhaps a laser diode, which also needs a current signal to drive it. The amplifier within the repeater must thus take the small current output from the photodiode and amplify it to drive the LED or laser.

Therefore, the amplifier required has to be a **current amplifier**. The requirements for such a circuit are:

Its input should affect the signal current from the photodiode as little as possible,

It should have output circuitry that maximises the current transfer out of the amplifier,

It should have current gain.

To achieve this, it should have **as low an input resistance as possible** and its output should look like an ideal current source (i.e. it should have **as high an output resistance as possible**).

Another example

Consider a communications receiver system. Within the receiver there are circuits that perform functions such as high frequency amplification of the signals, the selection of the required signal (filters) and frequency translation of the signal to a lower frequency (mixers). Because of current limitations on the operating speed of analogue-to-digital converters and other digital circuitry, these functions are normally performed by analogue circuits. However, it is common practice to convert high frequency signals to a much lower frequency (often a few tens of kHz) so that such functions as demodulation or decoding and the final signal processing can be done using digital techniques. DSP (digital signal processing) circuits are widely used to do this.

Typically, DSP chips require input voltage signals of a few volts amplitude. The output signal from the analogue part of the receiver system may be only tens of millivolts in amplitude. An amplifier is therefore required to 'bridge this gap'.

Therefore, this amplifier has to be a **voltage amplifier**. The requirements for such a circuit are:

Its input should affect the signal voltage from the analogue circuitry as little as possible,

It should have output circuitry that maximises the voltage transfer out of the amplifier,

It should have voltage gain.

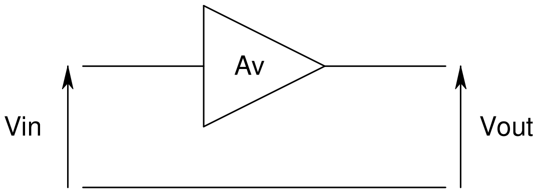
To achieve this, it should have **as high an input resistance as possible** and its output should look like an ideal voltage source (i.e. it should have **as low an output resistance as possible**).

You can imagine that the requirements for the two example amplifiers above will result in completely different circuitry to satisfy them.

The two other types of amplifier also find uses in electronics and communications systems. For example, a field effect transistor amplifier is an example of a **transconductance amplifier** and **transresistance amplifiers** are widely used in audio mixers and digital-to-analogue converters.

The Voltage Amplifier

A **voltage amplifier** is one to which an input voltage is applied and an output voltage results. The block diagram for such a system is given below.



The **voltage gain** of such an amplifier is given by

$$\text{Voltage gain,} \quad A_v = \frac{V_{out}}{V_{in}}$$

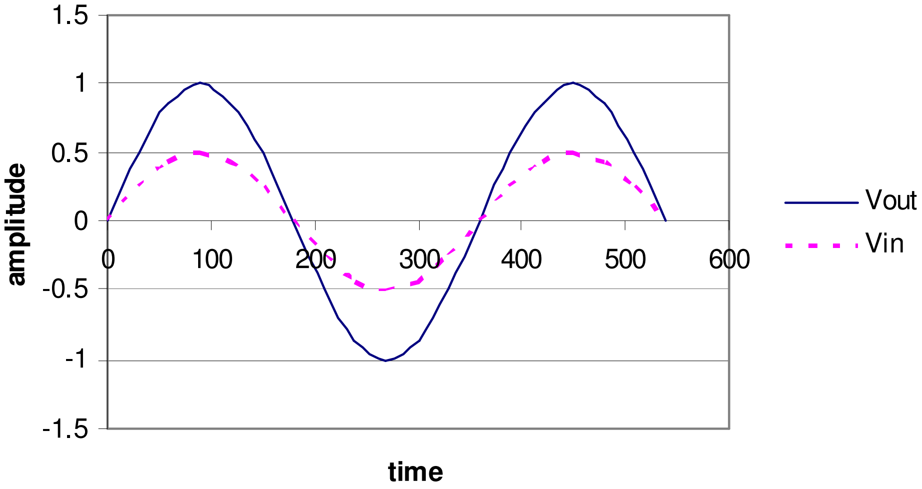
Ideally, connecting a voltage amplifier into a circuit should not affect the input voltage in any way. To achieve this, the input impedance of the amplifier should be as high as possible.

Also, ideally, connecting a load onto the output of the amplifier should not change the output voltage in any way. For this to happen, the output impedance of the amplifier should be as low as possible.

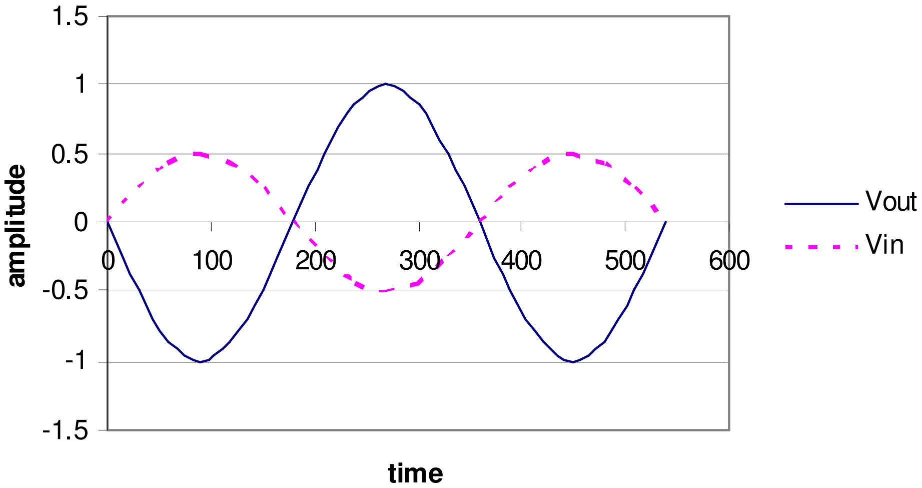
The ideal properties for a voltage amplifier are thus:

Voltage gain	Yes
Input impedance	Very high
Output impedance	Very low

The output voltage may follow the input voltage directly, or it may be inverted in polarity. This is illustrated below:



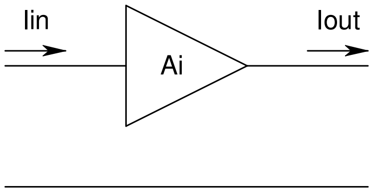
This shows the output following the input directly. The output is said to be **in phase** with the input. Another name for such an amplifier is a **non-inverting amplifier**. Note, also, that the output is amplified with respect to the input.



This shows the output inverted with respect to the input. The output is said to be in **anti-phase** with the input. Another name for such an amplifier is an **inverting amplifier**. Note, also, that the output is amplified with respect to the input.

The Current Amplifier

A **current amplifier** is one to which an input current is applied and an output current results. The block diagram for such a system is given below.



The **current gain** of such an amplifier is given by

$$\text{Current gain,}\quad A_i = \frac{I_{out}}{I_{in}}$$

Ideally, connecting a current amplifier into a circuit should not affect the input current in any way. To achieve this, the input impedance of the amplifier should be as low as possible.

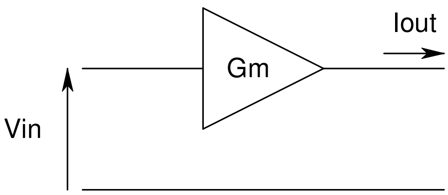
Also, ideally, connecting a load onto the output of the amplifier should not change the output current in any way. For this to happen, the output impedance of the amplifier should be as high as possible.

The ideal properties for a voltage amplifier are thus:

Current gain	Yes
Input impedance	Very low
Output impedance	Very high

The Transconductance Amplifier

A **transconductance amplifier** is one to which an input voltage is applied and an output current results. The block diagram for such a system is given below.



The **transconductance gain** of such an amplifier is given by

$$\text{Transconductance gain,}\quad G_m = \frac{I_{out}}{V_{in}}$$

Ideally, connecting a voltage amplifier into a circuit should not affect the input voltage in any way. To achieve this, the input impedance of the amplifier should be as high as possible.

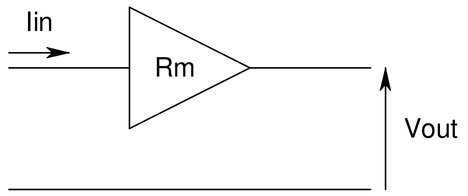
Also, ideally, connecting a load onto the output of the amplifier should not change the output current in any way. For this to happen, the output impedance of the amplifier should be as high as possible.

The ideal properties for a transconductance amplifier are thus:

Transconductance gain	Yes
Input impedance	Very high
Output impedance	Very high

The Transresistance Amplifier

A **transresistance amplifier** is one to which an input current is applied and an output voltage results. The block diagram for such a system is given below.



The **transresistance gain** of such an amplifier is given by

$$\text{Transresistance gain,} \quad R_m = \frac{V_{out}}{I_{in}}$$

Ideally, connecting a transresistance amplifier into a circuit should not affect the input current in any way. To achieve this, the input impedance of the amplifier should be as low as possible.

Also, ideally, connecting a load onto the output of the amplifier should not change the output voltage in any way. For this to happen, the output impedance of the amplifier should be as low as possible.

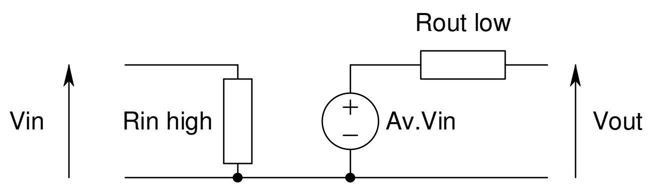
The ideal properties for a transresistance amplifier are thus:

Transresistance gain	Yes
Input impedance	Very low
Output impedance	Very low

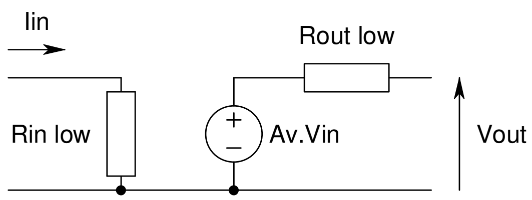
Summary of Amplifier Properties

A summary of the properties of the four types of amplifier, together with an equivalent circuit for each, is given below.

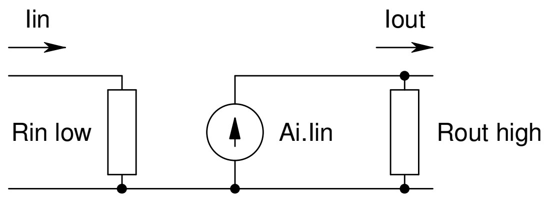
Amplifier Type	Gain Required	Input Resistance	Output Resistance
Voltage Amplifier	Voltage A_V	Very high	Very low
Current Amplifier	Current A_I	Very low	Very high
Transconductance Amplifier	Transconductance G_m	Very high	Very high
Transresistance Amplifier	Transresistance R_m	Very low	Very low



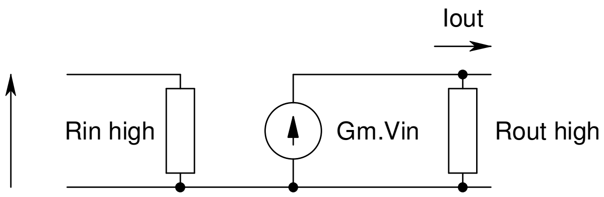
Voltage Amplifier



Transresistance Amplifier



Current Amplifier



Transconductance Amplifier

Gain, decibels and Input and Output Resistances

The following section uses some mathematics to explain how the decibel is used to describe the gain of an amplifier. Although it may look complex, working through it should make things much clearer. There is absolutely no need to commit these relationships to memory or be able to reproduce them. The important point is that you appreciate that this apparently complex concept is based on nothing more than Ohm's law and logarithms.

In the Introduction on Gain and phase, the power gain of a system in decibels was defined as:

$$10\log_{10}\frac{P_{out}}{P_{in}}\ \text{dB}$$

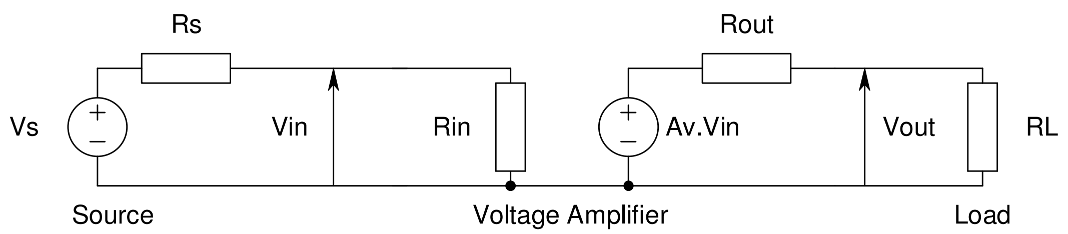
Expressing gain in dB should essentially be confined to the power gain, as the definition of the dB relates to power only. However, it was shown that the above expression became:

$$20\log_{10}\frac{V_{out}}{V_{in}}\ \text{dB} \qquad \textbf{if } \mathbf{R_L = R_{in}}.$$

So, in practice, the voltage gain of an amplifier is often expressed in dB, using this expression. Unfortunately, users often do not appreciate that normally R_L and R_{in} are normally **not** equal and that this inequality affects the gain.

By investigating the equivalent circuit of the amplifier, including its source and load, you can now see what happens if R_L and R_{in} are not equal.

The full equivalent circuit of a voltage amplifier, together with source and load, is given below.



From this:

$$P_{in} = \frac{V_{in}^2}{R_{in}} \text{ and } P_{out} = \frac{V_{out}^2}{R_L}$$

Therefore:

$$\text{Power gain} = \frac{P_{out}}{P_{in}} = \frac{V_{out}^2}{V_{in}^2}\cdot\frac{R_{in}}{R_L}$$

Expressing this is dB gives:

$$\text{Power gain} = 10\log_{10} \frac{P_{out}}{P_{in}} = 10\log_{10} \frac{V_{out}^2}{V_{in}^2} \cdot \frac{R_{in}}{R_L} \quad \text{dB}$$

$$= 20\log_{10} \frac{V_{out}}{V_{in}} + 10\log_{10} \frac{R_{in}}{R_L} \quad \text{dB}$$

From this expression you can see that, for a voltage amplifier, if the input and load resistances are **not** equal, the power gain in dB is **no longer** just 20 times the log of the voltage gain. There is the second term that is dependent on the ratio of those resistances.

If $\frac{R_{in}}{R_L} = 1$, then $10\log_{10} \frac{R_{in}}{R_L}$ will be zero and the power gain in dB will be the same as the voltage gain in dB.

If $\frac{R_{in}}{R_L} < 1$, then $10\log_{10} \frac{R_{in}}{R_L}$ will be negative and the power gain in dB will be less than the voltage gain in dB.

If $\frac{R_{in}}{R_L} > 1$, then $10\log_{10} \frac{R_{in}}{R_L}$ will be positive and the power gain in dB will be higher than the voltage gain in dB.

This shows that the gain of an amplifier depends on its input and load resistances. This is an important point to understand.

A similar analysis can be done for the other types of amplifier. The equations that result are given below.

Current amplifier:

$$\text{Power gain} = 20\log_{10} \frac{I_{out}}{I_{in}} + 10\log_{10} \frac{R_L}{R_{in}} \quad \text{dB}$$

$$\text{i.e. Power gain} = 20\log_{10} A_i + 10\log_{10} \frac{R_L}{R_{in}} \quad \text{dB}$$

Transconductance amplifier:

$$\text{Power gain} = 20\log_{10} \frac{I_{out}}{V_{in}} + 10\log_{10} R_L.R_{in} \quad \text{dB}$$

$$\text{i.e. Power gain} = 20\log_{10} G_m + 10\log_{10} R_L.R_{in} \quad \text{dB}$$

Transresistance amplifier:

$$\text{Power gain} = 20\log_{10} \frac{V_{out}}{I_{in}} + 10\log_{10} \frac{1}{R_L.R_{in}} \quad \text{dB}$$

$$\text{i.e. Power gain} = 20\log_{10} R_m + 10\log_{10} \frac{1}{R_L.R_{in}} \quad \text{dB}$$

These also show that the gain of an amplifier depends on its input and load resistances.

Thus the dB value of the power gain and that of the voltage gain (or current, transconductance or transresistance gain, depending on the type of amplifier) will only be equal when the input and load resistances are equal.

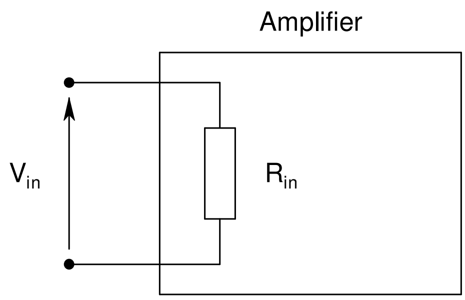
As you can now see, stating the voltage or current gain of an amplifier in decibels without some knowledge of the input resistance and the output load can well be misleading.

However, the use of $20\log_{10} \frac{V_{out}}{V_{in}} \cdot$ has been adopted in practice as a convenient definition of the voltage gain of an amplifier (in dB), regardless of the magnitudes of the input and output resistances. This is a case of engineering "slang", and worse, is often done by those who do not appreciate how misleading it can be.

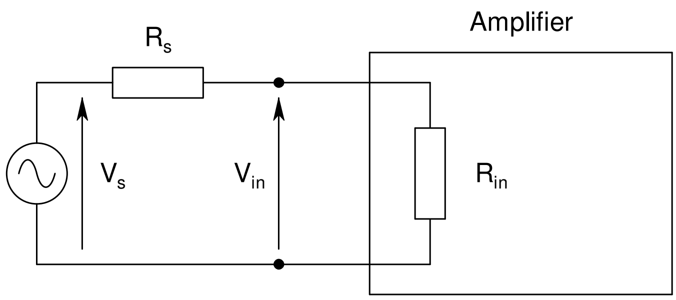
Determining Input and Output Resistances of Voltage Amplifiers

Input Resistance

The input resistance of an amplifier can be represented by a resistance R_{in} connected internally across its input terminals, as shown in the diagram below.



If a signal voltage source is connected across the input terminals the circuit becomes:

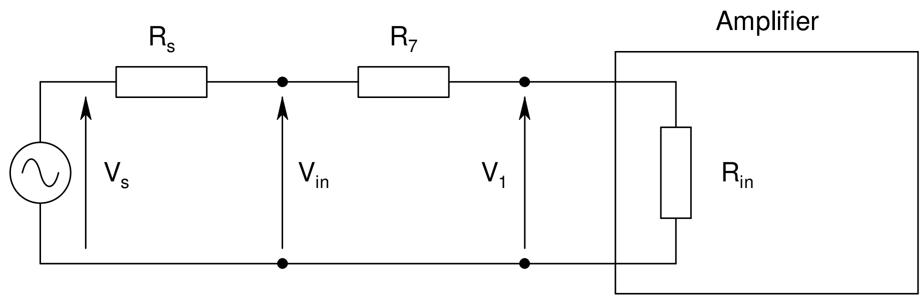


You can now see that there will be a potential divider circuit comprising the resistors R_s and R_{in} associated with the input circuit. The relationship between V_s and V_{in} will be determined by the relative values of the two resistors and can be calculate using Ohm's law.

V_s is the open circuit voltage produced by the signal source. You can determine this by just measuring the source voltage with no circuit connected to the source.

R_s is the resistance of the source. The source supplied on the workboard (the Sweep Source) has been designed to have a relatively low resistance output. However, you do not know exactly its value. So how can you determine R_{in}?

Suppose a much larger value resistor was connected in series with the source. The circuit now becomes:



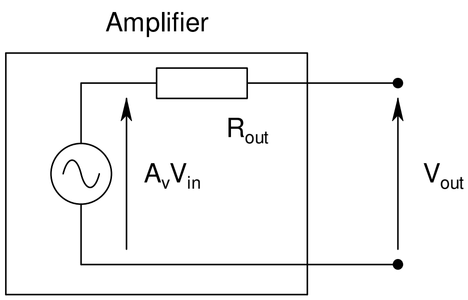
Now, both V_{in} and V_1 can be measured and, if the value of R_7 is known, the value of Rin can be determined using the normal potential divider formula:

$$\frac{V_1}{V_{in}} = \frac{R_{in}}{R_7 + R_{in}}$$

The value of the resistance R7 on the workboard is 10kΩ.

Output Resistance

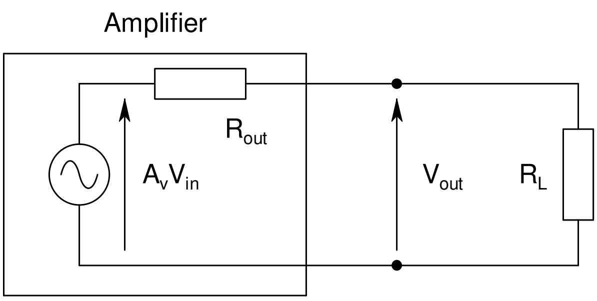
A somewhat similar technique can be used to determine the output resistance of the amplifier. The output circuit of the amplifier can be represented as an equivalent voltage source that has a value dependent on the input voltage, V_{in}, and the voltage gain of the amplifier, A_v. This voltage source is in series with the output resistance of the amplifier, R_{out}. This is shown in the following diagram.



V_{out} is the output voltage with nothing connected to the output terminals – the open circuit output voltage.

Because there is nothing connected across the output there will be no current flowing in R_{out}. So the voltage drop across R_{out} will be zero. Thus V_{out} will be the same as A_vV_{in}. You can measure this value (call it V_{out1}).

If a load resistor is now connected across the output the circuit becomes:



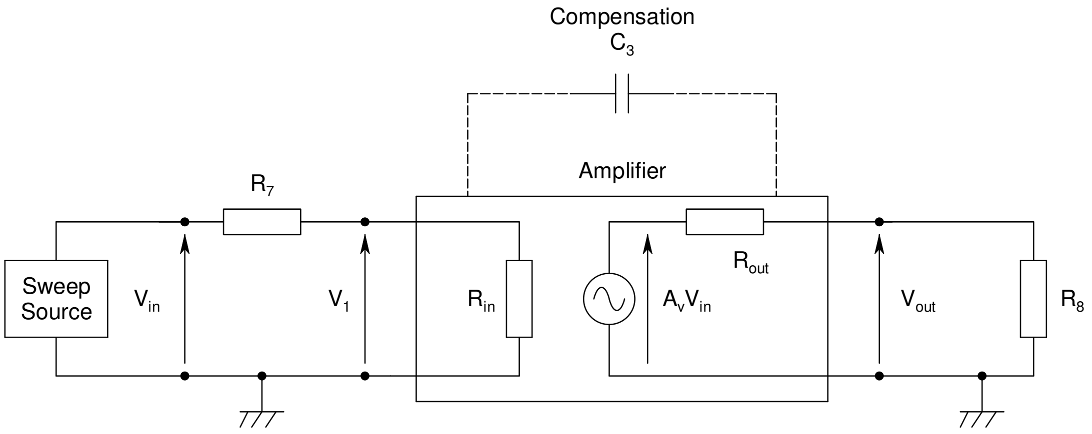
There will now be current flowing in the output circuit and thus there will be a voltage drop across R_{out}. So V_{out} will be lower than before. You can measure this new value of V_{out} (call it V_{out2}).

If R_L is known, R_{out} can be determined using a potential divider relationship again:

$$\frac{V_{out1}}{V_{out2}} = \frac{R_{out} + R_L}{R_L}$$

R_L is shown as R_8 on the workboard and the value of R_8 is 220Ω.

The full equivalent circuit representing the block on the workboard thus becomes:

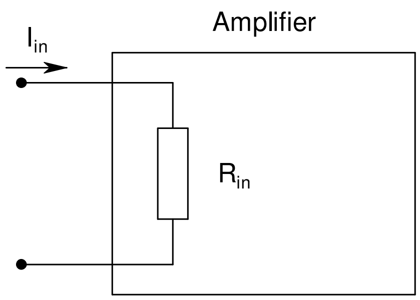


The function of the compensation capacitor C_3 will become evident when you perform the Practical.

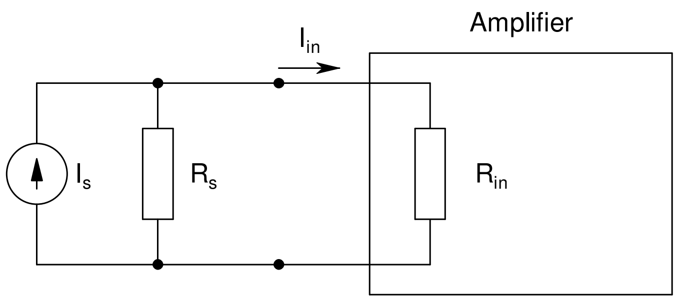
Determining Input and Output Resistances of Transresistance Amplifiers

Input Resistance

The input resistance of an amplifier can be represented by a resistance R_{in} connected internally across its input terminals, as shown in the diagram below.



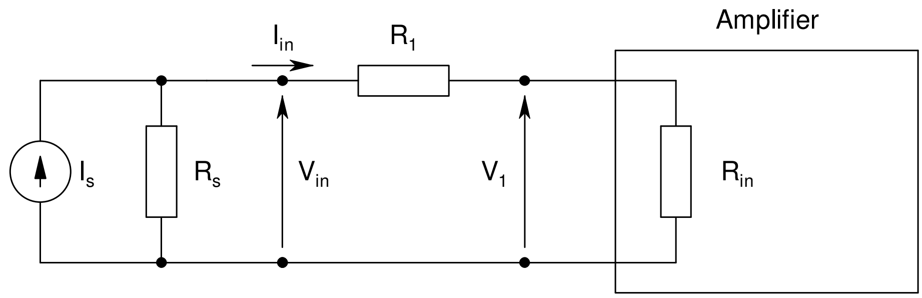
If a signal current source is connected to the input terminals the circuit becomes:



You can now see that there will be a current divider circuit comprising the resistors R_s and R_{in} associated with the input circuit. The relationship between I_s and I_{in} will be determined by the relative values of the two resistors and can be calculate using Ohm's law.

R_s is the resistance of the source. The source supplied on the workboard (the Sweep Source) has been designed to have a relatively high resistance output. However, you do not know exactly its value. So how can you determine R_{in}?

Suppose a resistor was connected in series with the source. The circuit now becomes:



The current I_{in} flows through both resistor R_1 and the input resistance R_{in}. There will be a voltage drop across R_1 due to this current.

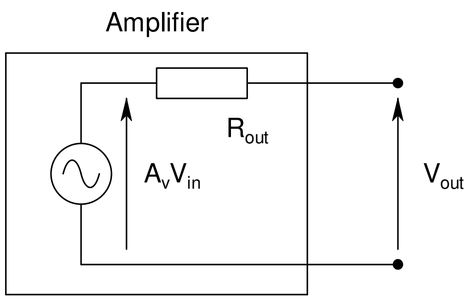
Now, both V_{in} and V_1 can be measured and, if the value of R_1 is known, the value of Rin can be determined using the normal potential divider formula:

$$\frac{V_1}{V_{in}} = \frac{R_{in}}{R_1 + R_{in}}$$

The value of the resistance R_1 on the workboard is 10kΩ.

Output Resistance

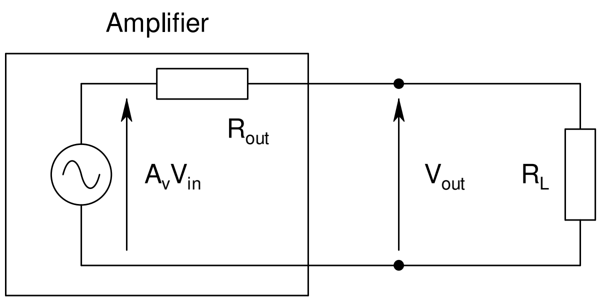
A somewhat similar technique can be used to determine the output resistance of the amplifier. The output circuit of the amplifier can be represented as an equivalent voltage source that has a value dependent on the input voltage, V_{in}, and the voltage gain of the amplifier, A_v. This voltage source is in series with the output resistance of the amplifier, R_{out}. This is shown in the following diagram.



V_{out} is the output voltage with nothing connected to the output terminals – the open circuit output voltage.

Because there is nothing connected across the output there will be no current flowing in R_{out}. So the voltage drop across R_{out} will be zero. Thus V_{out} will be the same as A_vV_{in}. You can measure this value (call it V_{out1}).

If a load resistor is now connected across the output the circuit becomes:



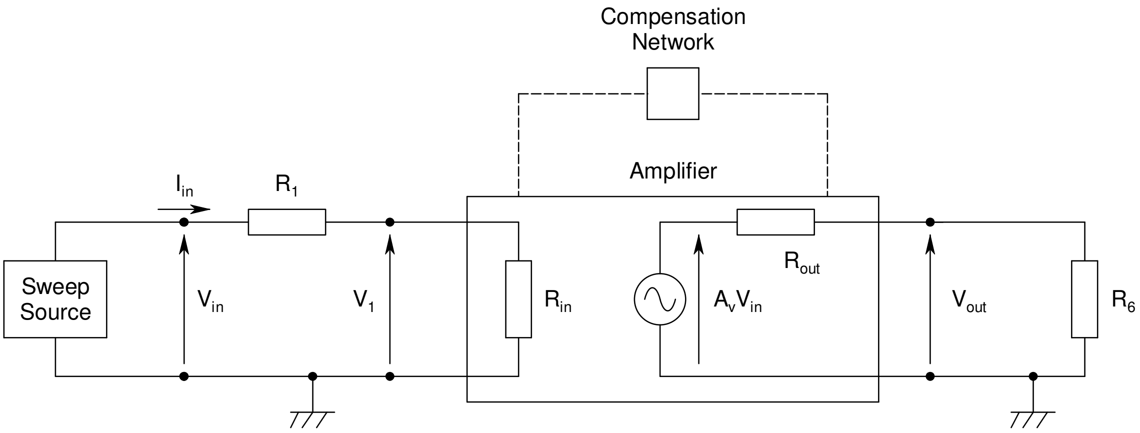
There will now be current flowing in the output circuit and thus there will be a voltage drop across R_{out}. So V_{out} will be lower than before. You can measure this new value of V_{out} (call it V_{out2}).

If R_L is known, R_{out} can be determined using a potential divider relationship again:

$$\frac{V_{out1}}{V_{out2}} = \frac{R_{out} + R_L}{R_L}$$

R_L is shown as R_6 on the workboard and the value of R_6 is 220Ω.

The full equivalent circuit representing the block on the workboard thus becomes:



The function of the compensation network will become evident when you perform the Practical.

More information on Signal Sources

A signal source is usually some kind of generator that produces a signal output with no signal input. An oscillator is an example of a signal source. Of course, the amplitude, frequency and waveshape may take many values, or it could be simply a constant dc voltage. Usually, we differentiate between a dc source that is used as a power supply and that used for a signal in a circuit.

Sources may be completely autonomous, i.e. they have no inputs at all, simply an output. However, many sources have inputs that control an output parameter such as amplitude or frequency. These are called control inputs. Some may have synchronising inputs that allow the output phase or frequency to be locked to an input signal. To be regarded as a true source the output should continue when all the control signals are removed.