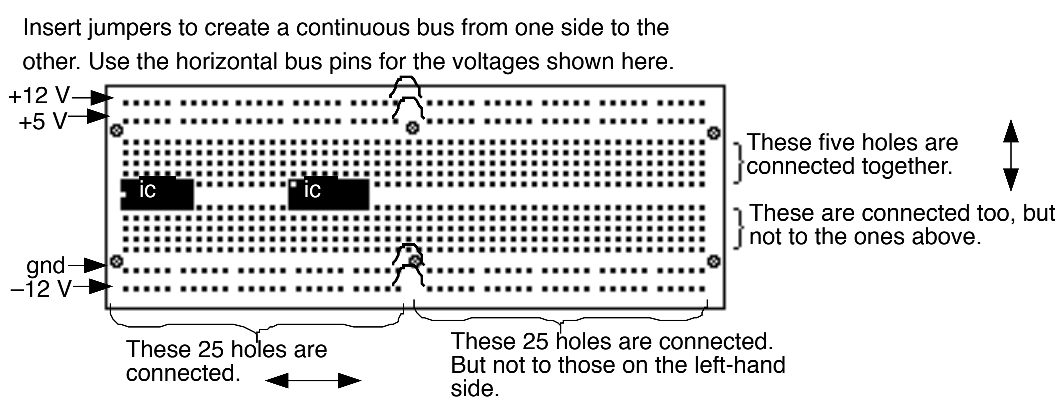
Introduction

The experiments in this lab manual are designed to introduce various aspects of analog electronics starting from the simplest concepts such as Ohm's law and leading to practical electronic circuits including amplifiers, integrated circuits, oscillators, voltage regulators and logic gates. Each lab script is intended for a four-hour lab period. Some students may need more time to complete the labs, especially at the beginning when the equipment is still unfamiliar. Your time can be used more efficiently if you prepare in advance by reading the script and planning the procedure before coming to the lab. There may be homework problems assigned which are intended to be done before the lab in order to prepare.

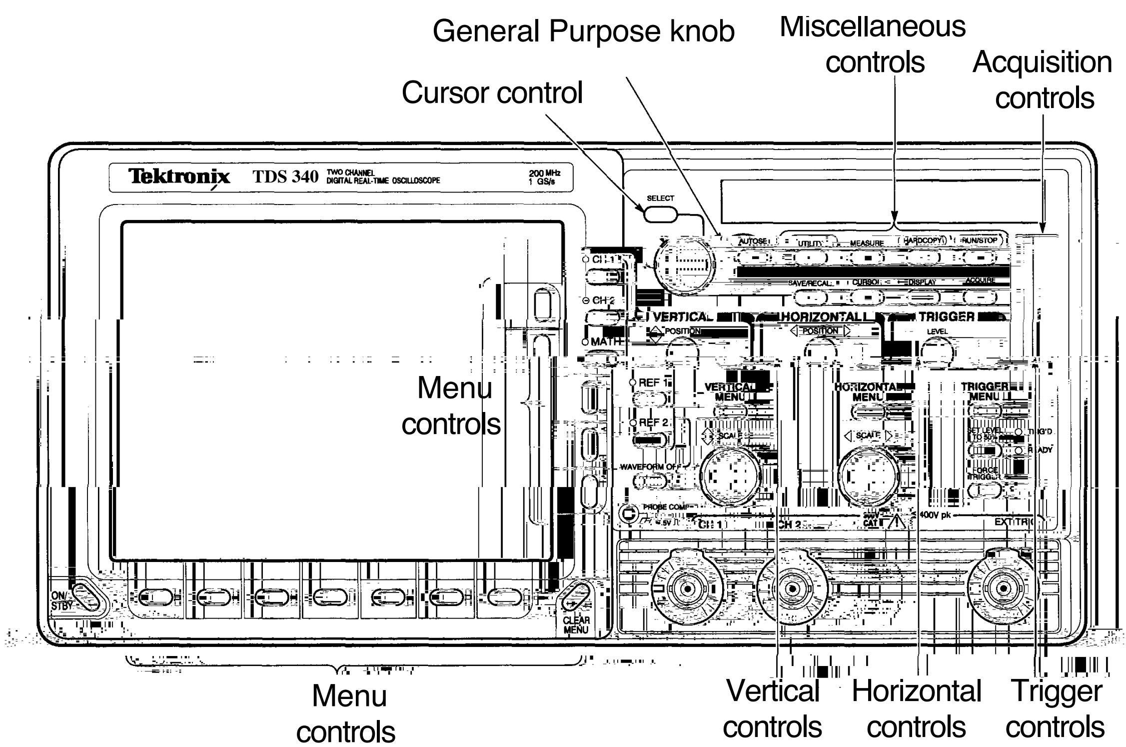
Each workstation in the lab has the necessary equipment: an oscilloscope, a function generator, a multi-meter and an experimental box. The multi-meter can measure voltage, current, resistance and capacitance. The experimental box includes ±12 V power supplies for operational amplifiers and a +5 V supply for logic chips. The independent Anatek variable power supply includes a robust current limiting control. For those circuits built from independent components not using integrated circuit (IC) chips, it is better to use this power supply because it withstands abuse much better than the power supplies in the experimental box.



The breadboard area on the experimental box has holes for component leads, #22 solid wire and IC pins. Don't try to force larger wire into these hole because it will spring them too far and ruin the board. The horizontal rows of holes on the top and bottom of the breadboard are connected together horizontally. The left and right halves are independent. We suggest that you use these horizontal rows for power supply voltages and ground. You may wish to put a jumper wire between the left and right halves so that the voltages are the same across the board. The vertical columns of holes are connected electrically in groups of five along a vertical line. The top and bottom halves are independent. Typically one inserts an IC chip straddling the centre trough. There are then four empty holes for making connections to each IC pin. When you plug ICs into the breadboard, a common convention is to put pin 1 on the left. For other components, make sure the leads are not in the same column of five unless you want them connected together.

The oscilloscope has many knobs and buttons which may be confusing at first. It helps if you read the introductory booklet and manual provided by Tektronix. If nothing seems to be happening press AUTOSTART.

N. Alberding, November 1990
Revised, March 1994
Revised, July 1996
Revised, Nov 1998
Revised, March 2002
Revised, April 2005



LAB 1
Introduction to Equipment and Circuits

The basics of Thévenin's theorem and AC circuits are covered in Physics 231. You may wish to review that material before starting these labs..

Reading: Malvino: Ch. 1.
Hayes and Horowitz: Class 1, Worked Examples and Lab 1.
Note especially "*A preliminary note on* ***procedure.***"
Malvino: Ch. 16, sections 9, 10, 11,16, 17,18, 19, 20.
Hayes and Horowitz: Class 2, Worked Examples and Lab 2.
Read "A Note on Reading Capacitor values", p51, H&H

1. Oscilloscope and function generator

These are a few warm-up exercises in using the Hewlett-Packard function generator and Tektronix Digital Oscilloscope.

a) Show two signals on the oscilloscope display

Adjust the function generator to produce a sine wave set the AMPLITUDE to 2 V_{pp}. Apply the sine wave signal from the function generator to Channel 1 of the oscilloscope. Simultaneously apply the SYNCH output signal of the function generator to Channel 2. Press, in sequence, the oscilloscope's AUTOSET, CLEAR MENU and CH 1 buttons. AUTOSET should configure the scope to measure the signals coming into the inputs. The CLEAR MENU and CH 1 buttons ensure that the display is clean and your next operations will affect the CH 1 display. The green light next to CH 1 should be lit. You should see a 2 V_{pp} signal displayed on the CH 1 trace and a square wave displayed on the CH 2 trace. What is the peak-to-peak voltage of the square wave?

Hint: The function generator's output voltage display depends on the Z_{out} setting of the function generator. Z_{out} can be set to either 50 Ω or infinity. If the setting is 50 Ω then the displayed output voltage will be 1/2 the voltage produced by the function generator, because it assumes it is driving a 50 Ω load where the output voltage would be split evenly between its 50 Ω output impedance and a 50 Ω load. If Z_{out} is infinity then the display will show the full voltage produced by the function generator, which is the voltage applied to a very high impedance load. For this course the Z_{out} setting should be infinity. The Z_{out} setting does not affect the actual output of the function generator, it only changes the numerical display.

b) Change the scale and position of the waveforms

Turn the SCALE knob under VERTICAL. Notice how the display changes. The V/DIV legend beneath the display reflects the scale changes. Adjust the CH 1 scale to 1 V/div. Press CH 2 and investigate the VERTICAL SCALE adjustment as before and leave CH 2 on 1 V/div. Press CH 1. Play with the VERTICAL POSITION knob.

Turn the HORIZONTAL SCALE knob and note how the displayed waveform changes. The legend beneath the display reflects the change in sweep rate. Move the trace left and right with the HORIZONTAL POSITION control.

After you have changed a few settings you should be able to return to the original configuration by pressing AUTOSET. The result of AUTOSET depends on the signals which input to the oscilloscope, so if you have changed the function generator output in any way AUTOSET will result in a different configuration. Furthermore, not all of the oscilloscope's function settings are reconfigured by AUTOSET. It is possible to save all settings of particular configuration for later recall from the oscilloscope's internal memory. See the User's Manual for how to do this.

c) Investigate the difference between AC, DC, and GND input coupling.

Press VERTICAL MENU. Observe the signals when you choose AC, DC and GND on the menu. Normally you use DC, even when you are measuring AC signals. The purpose of the AC coupled input is to subtract a DC offset from a signal so that you can magnify the alternating component. Add a DC offset to the signal by pulling the OFFSET button on the function generator. You should notice that AC coupling subtracts the offset from the displayed waveform.

> Avoid AC coupling unless you need to subtract an offset—at low frequencies AC coupling can distort the signal's display. AC coupling puts a high-pass filter on the input to remove the DC offset. To see this, put the coupling on AC and decrease the function generator's frequency until the signal starts to appear smaller in amplitude. After you finish put the frequency back to its original value.

d) Learn the operation of the scope's sweep and trigger controls.

Press TRIGGER MENU. Make sure trigger source is CH1. Vary the level control to observe the effect of changing the trigger level. There is a floating T on the screen to show you where the triggered position of the input signal is displayed. There is also an arrow on the right-hand side if the screen to indicate the trigger voltage level. If either of these indicators are not visible they may have been turned off. Consult the user's manual or an instructor to find out how to turn them on again.

Change the trigger source to CH 2, which displays the SYNCH signal. What effect does changing the trigger level have now?

Change the trigger slope from positive to negative. Note the difference.

There's a button labelled "Set Level to 50%" which is handy to quickly stabilize a signal on the screen when you don't know where the trigger level should be set.

Reconnect the SYNCH signal from the function generator to EXT TRIG. Select EXT TRIG for the trigger source. This frees CH 2 for observing another signal while still allowing the trigger signal to come from SYNCH of the function generator.

e) Learn to measure frequency, assuming that the horizontal time base is accurately calibrated. Centre the displayed waveform about a horizontal line. Measure the period from zero crossing to zero crossing and calculate the frequency. Compare with the value obtained using the MEASURE menu and from the function generator readout.

f) Generate Lissajous figures by applying two signals of different frequencies to CH 1 and CH 2 (Use the transformer for one signal.) Choose the DISPLAY menu and switch to the XY display instead of the YT display. If the HORIZONTAL SCALE setting is vastly inappropriate for the signal being displayed, the Lissajous figure may be incomplete. To see this, try varying HORIZ SCALE when displaying a Lissajous figure to make sure at least one full cycle is being displayed.

g) Invert a signal using the VERTICAL menu and **add, subtract, multiply and divide two signals applied to the two channels** using the MATH functions. Use the function generator for one signal, and the PROB COMP signal for another.

h) Printing

You can print the scope display on a printer using the HARDCOPY function. This is useful for recording results to put in your lab book. Before using HARDCOPY you must ensure that the output port and printing options are correctly configured. Use the UTILITY — I/O menu to select the hardcopy port (e.g., Centronics), Layout (e.g., portrait) and Hard Copy format (e.g., Epson printer. BMP file).

There is a program which allows downloading the oscilloscope display to a computer attached to the oscilloscope through the IEEE-488 interface. You can either print this file on the printer or include it in a document. Instructions should be provided in the lab.

2. Frequency Sweep Analysis of a Parallel LC Circuit

In this exercise you will use the frequency sweep feature of the function generator to generate a graphical display of the frequency response of a parallel LC filter

a) Construct the parallel resonant circuit of Fig. 1.1, The parallel LC circuit is often called a "tank." Choose *R* to be several times larger than the impedance of the *LC* circuit at the resonant frequency. Choose *L* and *C* so put the resonant frequency between 10 kHz and 100 kHz.

Measure the internal resistance of the inductor, R_i. The internal resistance that you measure is a series resistance, but in a parallel resonant circuit you can model it as a resistance in parallel with *L* with a value of $\omega_o^2L^2/R_i$ where ω_o is the resonant frequency. (For example, if $L = 30$ mH, $\omega_o = 10$ kHz and $R_i= 10\Omega$, then $\omega_o^2L^2/R_i \approx$ 9 kΩ.) Thus the expected *Q* can be calculated.

After constructing the circuit manually tune the function generator to locate the resonant frequency and 1/2 power points. Make sure the observed behaviour agrees tolerably well with what you expect before proceding.

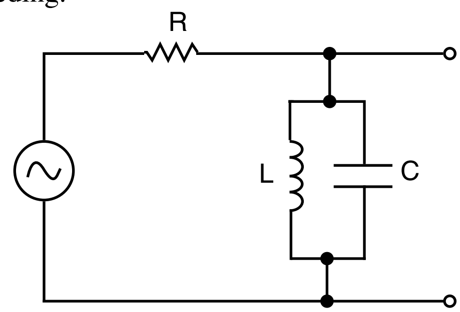


Fig. 1.1
A parallel *LC* Circuit. A large *R* in series with an AC voltage course (the signal generator) simulates an AC current source.

But be careful of grounding! *Both ground clips of the scope probes are connected together within the scope. They both go to earth ground.* *Make sure that they are not connected to different points of your circuit.*

b) Set up the frequency sweep of the signal generator to display the frequency response and print it.

Instead of changing the frequency manually, you can use the frequency sweep capability of the function generator. In frequency sweep mode, the function generator gradually changes the frequency output from f_1 to f_2 passing through all intermediate frequencies. The sweep time, t_{SW}, determines how long it takes to pass through the frequency range from f_1 to f_2 . After it reaches f_2, it abruptly returns to f_1 and repeats the sweep. If the sweep mode is linear then the frequency change from f_1 is proportional to the time that has passed from the start of the sweep:

$$f = f_1 + (t/t_{sw})\,(f_2 - f_1)$$

If the sweep mode is logarithmic, then the frequency is proportional to the exponential of the time and the rate at which it sweeps is proportional to frequency:

$$f = f_1\, e^{kt} \quad \text{where } k = \ln(f_2/f_1)/t_{SW}\,.$$

In logarithmic mode, for example, if the sweep is from 10 Hz to 1000 Hz it covers the range from 10 Hz to 100 Hz in the same amount of time that it covers the range from 100 Hz to 1000 Hz. In linear mode it covers the range from 10 Hz to 100 Hz in the same time that it covers the range from 100 Hz to 190 Hz and it would take 111 times longer to go from 100 Hz to 1000 Hz.

To set the sweep mode on the function generator follow these steps:

- Press SHIFT MENU to enter the function generator menus.
- Using the **>** key select B: SWP MENU.
- Pressing ↓ gives 1:START F.
- You can now specify the start of the frequency sweep using the knob.
- Use **>** to select other submenus such as 2:STOP F, 3:SWP TIME and 4: SWP MODE (LINEAR) OR (LOG). We suggest that START F should be near zero, and STOP F should be high enough to conveniently capture the frequency range of interest.
- After you have entered in all the numbers depress SHIFT SWEEP to enter sweep mode. The output of the function generator should now be sweeping through the specified frequencies.

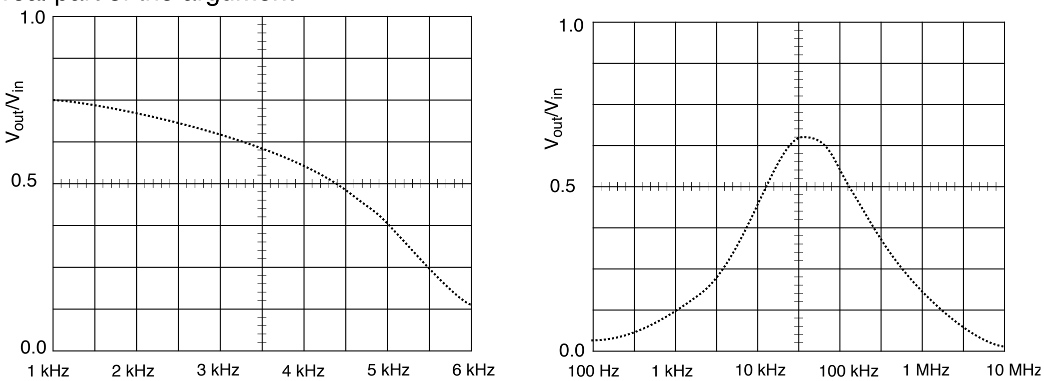
You can display a picture of the frequency response of a circuit on the oscilloscope in the following way:

- Trigger on the SYNCH signal from the function generator .
- Position the trigger point near the left-hand side the of the display.
- Adjust the function generator's sweep time and the scope's HORIZ SCALE settings so that the end of the sweep is at the right-hand side of the display. You should choose the sweep time to be slow enough to allow for at least one cycle to take place before the frequency changes significantly.
- Adjust the channel's zero volts position to the bottom of the display.
- Increase the amplitude of the function generator's output so that a frequency response graph fills the display.
- You can fill in the trace by using the "envelope mode." Choose ENVELOPE from the ACQUIRE-MODE menu so that the screen will display

the accumulation of several sweeps. The number of sweeps accumulated is controlled by the General Purpose Knob.

b) Use the display to find the resonant frequency, the bandwidth and Q of the circuit.
c) Investigate what happens when you load the output with a load resistor. Explain.

Note: The parallel LRC circuit is analogous to the series LRC circuit. Whereas the series circuit is drive with an alternating voltage source, the parallel circuit is driven by a current source. The calculations in both cases correspond with each other if, in the parallel case, one uses admittance (complex conductance) to calculate current instead of using impedance to calculate voltages. The function generator is a voltage source, but you can make an approximate current source by placing a relatively large resistor in series with its output. (The signal generator behaves as a current source if R is much larger than the magnitude of the impedance of the parallel LC circuit.) Also be aware that the internal series resistance of the inductor, R_i, can be approximated as a parallel resistance near the resonant frequency for purposes of analysis: $R_{parallel} = 1/\Re(1/Z_{LC})$ where $Z_{LC} = R_i + j\omega L$, where $\Re$ stands for the real part of the argument.



Linear Sweep Fig. 1.2 **Logarithmic Sweep**

On the left is a hypothetical display of a linear frequency sweep with f_1=1 kHz and f_2=6 kHz .
On the right is another display made by a logarithmic sweep with f_1=100 Hz and f_2=10 MHz.
Note that by assiduously choosing t_{sweep} and the horizontal scale a convenient frequency scale is produced.

3. Fourier Analysis

Drive the circuit of Fig. 1.1 with a square wave and carefully observe the frequency response of the output voltage. You will get peaks in the output sine wave response at the circuit's resonant frequency and at certain lower frequencies that have harmonics at the resonant frequency. This is a sort of backward Fourier analysis. The first few terms of the Fourier expansion of a square wave should be roughly related to the peak frequencies and amplitudes. Try using the sweep generator to display a series of peaks at once.

4. Cascaded Filters

Build a band pass filter by cascading a high pass and a low pass filter. To avoid the filters affecting each other, use the following design criterion: the output impedance of the first filter should be about 1/10 the input impedance of the second filter. Only resistive impedances are considered. (The resistor values should be 1:10.) Measure its frequency response.