Combinational logic: Breadboard adders

ENEE 245: Digital Circuits and Systems Laboratory
Lab 1

Objectives

The objectives of this laboratory are the following:

- To become familiar with the Agilent InfiniiVision 2000 X series oscilloscope and its built-in function generation DVM, with which you will learn to acquire, save, and manipulate analog voltage signals. You will also learn to use various cables, probes, and connectors To implement a small combinational logic circuit on breadboard.

- To implement a small combinational logic circuit on breadboard.

In this lab you will gain familiarity with several pieces of test and measurement equipment. The key piece of equipment that you will use is the new digital oscilloscope, with integrated function generator and digital voltage meter.

Mixed Signal Oscilloscope

The oscilloscope that you will be using in ENEE 245 is Agilent InfiniiVision 2000 X – Series Oscilloscope. A complete user's guide can be found in the following URL:

> http://cp.literature.agilent.com/litweb/pdf/75015-97023.pdf

Lab 1 consists primarily of following the steps described in the handout. Make certain that you read the laboratory procedure ahead of time and answer the Pre-lab questions.
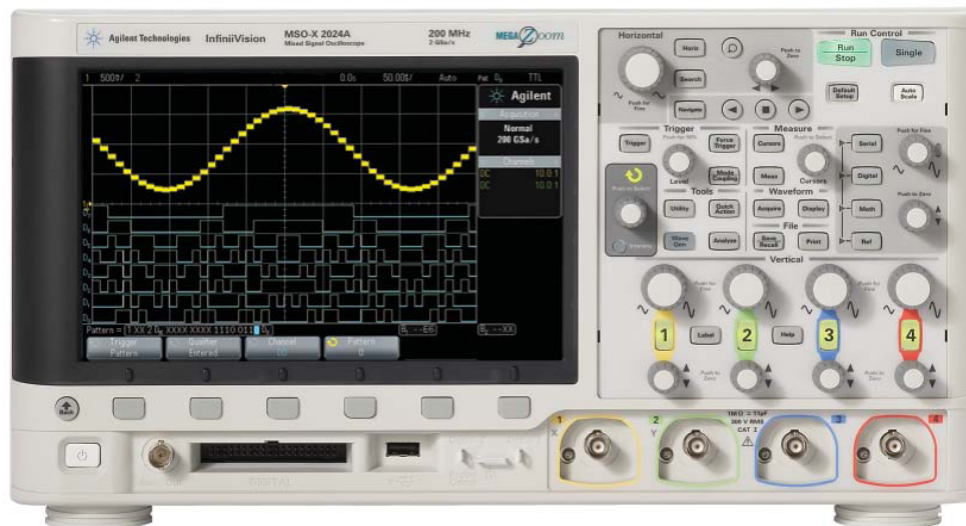

Figure 1. Agilent InfiniiVision 2000 X Oscilloscope

A picture of the oscilloscope front is shown in Fig. 1. The display occupies the left half of the oscilloscope. The color LCD display has a grid that is used to quantify the signal. There are 8 horizontal divisions for voltage and 10 vertical divisions for time. Scales for the traces are given above

the main display. Below the grid is a text region where you can modify the display by pressing the keys beneath the listed options and toggling between the various options. The right half of the oscilloscope front panel contains the input jacks, several knobs, and a number of buttons to control the oscilloscope operation. The controls are divided into the horizontal, analog, measure and run controls, the trigger buttons, and file and waveform options. Each section is described below. Additional buttons, to the right of the MSO screen, are used for digital data acquisition and are not described.

There are four BNC input "channels" for the oscilloscope and one BNC for the external trigger (on the back of the device). Each input can handle a maximum input voltage of 300 V_{RMS} and a frequency of 200 MHz. (This is called the 3-dB analog bandwidth, which is determined by the internal circuit connected to the input. Bandwidths associated with the sampling rates used during the digital conversion process are discussed later.) All channels are identical and any one can be used for any measurement.

The analog signal is continuously converted at certain time intervals to a digital signal and displayed on the LCD. The resolution of the LCD screen is 1024 x 768 (horizontal x vertical) pixels. After converting the signal to a digital representation, the oscilloscope stores it as a sequence of 8 bit numbers, so the absolute accuracy of the measurement is only 1 part in 2^8 = 256! In other words, any signal displayed on the scope can be off by at least ± 0.2% of the full scale value. To minimize the error, we adjust the scale to be as small as possible. To accomplish this, the oscilloscope has an internal amplifier that can be adjusted to produce scales from 1 mV/division to 5 V/div on the vertical axis. Thus, signals ranging over three orders of magnitude can be directly measured. Note that for an x10 probe, the range is from 10 mV/div to 50 V/div. These settings are adjusted in the "analog" section by turning the upper (larger) knob for the appropriate channel. The location of the trace zero is controlled by the lower (smaller) knob.

There are additional menu controls for the channel inputs that are accessed by the numbered button between the two knobs. Pushing the channel menu button twice turns off the channel (it is not displayed). The menu has five categories. The signal can be "DC" coupled or "AC" coupled. "AC" coupled means that a circuit is used to "filter out" the DC level so that the average voltage displayed is zero. Signals with high frequency noise can have a low-pass filter applied to attenuate any noise on the input signal above 20 MHz (BW limit button). There is a course/fine adjustment for the voltage scale (Vernier button) that allows 25-30 possible settings between the normal 1-2-5 sequence (e.g. you could have a 5.2 V/div scale). There is a probe button which will bring up a "probe" menu. Finally, the signal can be inverted, i.e. multiplied by minus one.

The time base is used to adjust the average time at which the samples are taken. It can be adjusted from 5 ns/div to 50 s/div via the large knob in the horizontal control section. The small knob in that section can be used to adjust the "zero (delay) time", which corresponds to the time the pulse is "triggered". The Main/Delayed button brings up a menu that we will not use.

The maximum sample rate is actually 2 giga-samples/sec. This means that the closest samples can be in time is about 0.5 ns! This sampling rate puts another upper limit (or "bandwidth") on the maximum signal frequency which can be measured. If the period of a signal were comparable to (or smaller than) this sample rate, the signal could not be adequately measured. This fact seems to be at odds with the fastest sweep rate, since at most 100 data points can be taken over the duration of the sweep. However, it turns out that you don't need too many points to define the frequency of a signal, so the sampling rate is well-matched to the analog bandwidth.

Normal operation is in the repetitive mode. Operation can be interrupted / restarted by pushing RUN/STOP button (which changes color to indicate status). Pushing the "SINGLE" button to the right of the RUN/STOP button will initiate a single shot (once a trigger signal is detected).

Since the data is continuously sampled, one might wonder how the start time for the trace is selected. This is accomplished by the trigger, which is set by the user to sense a particular voltage level on a particular input channel. In addition to the voltage level, the trigger also determines whether the voltage is rising or falling and only fixes the trace when the threshold is reached with the proper slope. Reliable triggering can be a major source of headaches with some signals and there are many triggering options that are available to minimize those headaches. We will only mention a few here. The majority are accessed with the Mode/Coupling button.

The system will run continuously without a trigger signal if "auto" is selected; "normal" is selected to get a trace only when proper triggering has occurred. The coupling is used to trigger out unwanted frequencies (noise). It can block out high frequencies, low frequencies or dc signals. Setting coupling to "DC" uses the complete signal to trigger the scope. The "hold off" time is the time that must pass after one trigger occurs before another one can be detected. This is very useful any time the same voltage level (and slope) occurs more than once in the time range that you are interested in observing. One example is whenever there is "noise" (i.e. a small high frequency signal) superimposed on your desired signal. Another example is when you have a complicated periodic structure and a third is when you are triggering the scope in "single-shot" mode with switches.

The trigger method (edge / pulse width / pattern / etc.) can be changed by pressing the appropriate button– we will typically use edge triggering for analog signals. The selected button is lighted. After selecting "edge," you can select the source (the input channel used for the trigger) and the slope. The trigger level is normally displayed as a small arrow (together with a "T") on the left side of the LCD to aid in setting an adequate trigger voltage.

The oscilloscope has 10 arrays that can be used to store traces from any of the channels. The stored traces can be displayed on the CRT simultaneously with the active "live" traces to facilitate comparisons. The "save/recall" button is used to store and display traces. The memories are called "intern_0" – "intern_9" ; they are located by pushing the Save menu and then the location menu button. The Recall menu and location menu buttons are used to display stored traces.

The oscilloscope can be used to make many measurements on the acquired signals. The measurements can be made manually or automatically. In manual mode, there are "cursors" which are horizontal or vertical lines which can be moved around to measure voltage or time, respectively. Two cursors can be used to measure differences and cursors can be "attached" to a signal to measure both time and voltage (albeit not at the same time – one must switch between cursor types. Cursors can be attached to either signals or math functions, but not stored traces. The cursor button and the resulting menu are used to control these actions.

The automatic measurements are made by pushing the "Quick Meas" button. Four measurements can be displayed at a time. On the resulting menu, "Source" determines the signal source and "Select" gives a complete table of measurements. Possibilities include peak-to-peak, maximum, minimum, RMS, standard deviation, or average voltages, period, frequency, rise time, fall time, and positive and negative pulse widths. Measurements can be performed only on active signals (channels which are currently being displayed on the scope) and math. The "Measure" button takes the data and displays it.

Simple math (multiplication, subtraction, differentiation, integration and FFTs) can be performed on active signals, but not on stored reference signals. Multiplication and subtraction are only for channels 1 and 2. There is an "acquire" menu button that allows one to just sample an input or average the signal (between 2 and 65536 traces can be averaged). Peak detect also can be accessed from the acquire button and is used to look for glitches.

Function Generator

The function generator that we use in ENEE 245 is a wave generator integrated into the oscilloscope. It can produce sine, square, and triangle wave pulses at a wide range of frequencies. The peak-to-peak voltage range is approximately 20 V.

To access the Waveform Generator Menu and enable or disable the waveform generator output on the front panel BNC, press the [Wave Gen] key. The waveform generator output is always disabled when the instrument is first turned on.
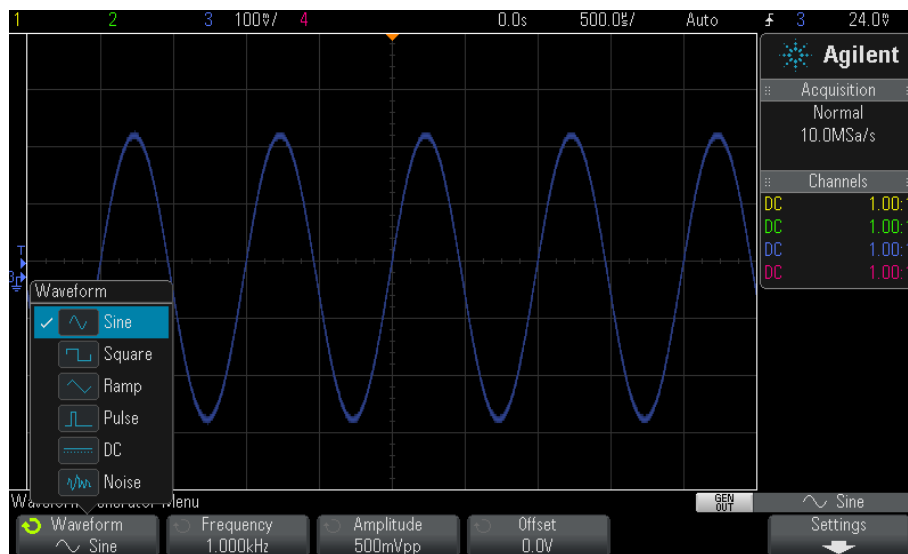

Figure 2. Waveform generation from the Oscilloscope

The Waveform button selects the output waveform. In ENEE 245, the most frequently used waveform will be the square wave, as you are designing only digital circuits. A pulse can be regarded as a square wave except that its duty cycle is not 50%. The Frequency and Amplitude knobs select the frequency and amplitude of the waveform. The limit of the output is software-controlled by the oscilloscope, and is therefore free of unwanted high-frequency voltage drop, which is likely to occur in classical function generators, where the amplitude of the output signal drops dramatically at the 3db point of the device, which is usually around 1MHz. Other settings are available for advanced users. The Offset button can be used to add an average voltage to the periodic signals.

The output signal (in sine mode) can be modeled by the nonideal AC source shown in Fig. 3. The two rightmost terminals in the figure are connected to the output BNC. Both terminals are "floating" which means that neither is connected to ground (if needed, you must make the ground connections in your circuits). When connected to a high impedance source, the output voltage will essentially be equal to Vg, but if the load impedance is 50Ω, the output voltage will be cut in half. If the load becomes too small, the function generator WILL SUSTAIN DAMAGE and will become inoperable. Be careful to make sure that the waveform output is never accidentally shorted out!
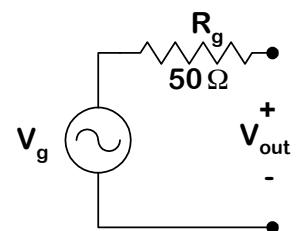

Figure 3. Equivalent circuit

Breadboard

A breadboard is used to facilitate the construction of circuits by providing a means for making fast temporary connections between components. A typical breadboard is shown in Fig. 1.4. There are two lines of holes which run horizontally along the top and 8 lines (4 pairs) that run vertically along the sides (indicated by the red and blue lines). The holes in each set are connected together electrically by a metal strip that runs underneath the plastic. These holes are often used to make connections to power supplies (e.g. ± 12 V, 5 V, or ground). Between each pair of vertical columns of holes there are two horizontal rows each comprised of 5 holes that are electrically connected. The gap between the sets of holes is the proper width to allow DIPs to be inserted (and the hole spacing vertically corresponds to the pin spacing on a DIP).
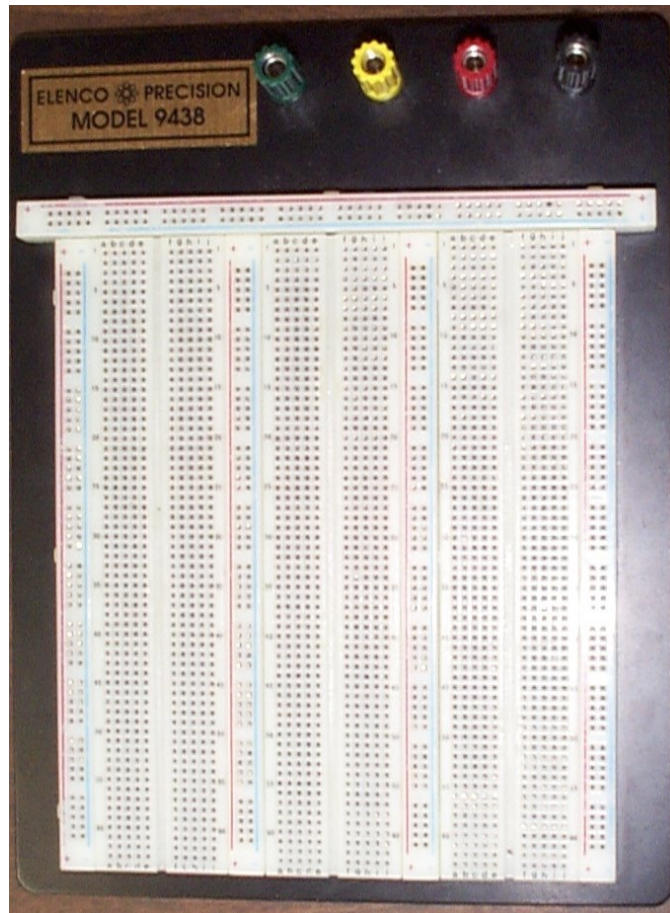

Figure 4. A typical breadboard.

Half Adder

The half adder performs one-bit addition operation. Its inputs are two operands: A and B, and the outputs are the sum S, and the carry-out Cout. The logic expression is:

S = A xor B

C = A and B

The Truth Table of a half adder is as follows:

A \ B	0	1		A \ B	0	1
0	0	1		0	0	0
1	1	0		1	0	1

$$S = AB' + A'B = A \text{ xor } B \qquad\qquad C = AB$$

A straight-forward way to implement a half adder will be to use an XOR gate and an AND gate. However, we can convert the above two expressions into NAND expressions:

$$S = AB' + A'B = ((AB')' (A'B)')'$$

$$C = ((AB)')'$$

The chips you will be are using in this lab are 4069 and 4011. The datasheet of these two devices can be found on the websites:

http://www.fairchildsemi.com/ds/CD/CD4069UBC.pdf

http://www.nxp.com/documents/data_sheet/HEF4011B.pdf

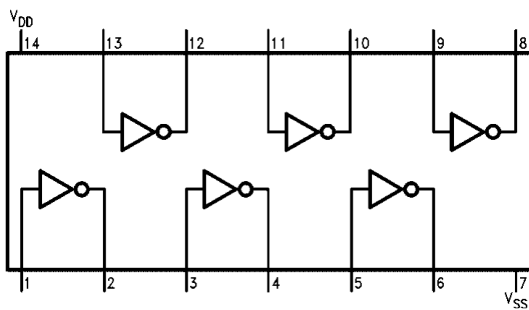The pin diagrams of the two chips are shown in Figure 5 and Figure 6.
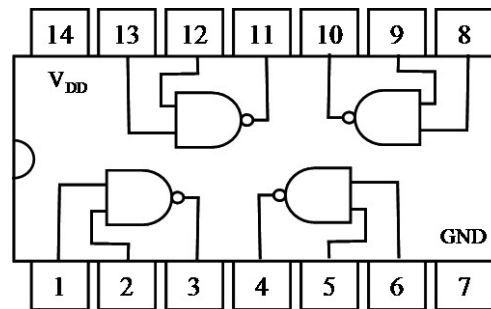

Figure 5. 4069 Inverter Pin Diagram.

Figure 6. 4011 NAND Pin Diagram.

Pre-Lab Preparation

Part I – Using the Oscilloscope

There are no calculations or simulations required for this part. However, make certain that you do the pre-lab reading and answer the pre-lab questions before you attend lab.

Part II – Half Adder

1. Derive the two NAND expression of half adder.

2. Using PSPICE, draw the wiring diagram for a half adder based on the available chips. Note that PSPICE knows about chips and their pinouts, so your job will be simplified tremendously.

3. Use PSPICE to simulate the circuit.

Pre-Lab Questions

1. What is the key sequence you must use to measure the frequency on channel 1?

2. How do you measure the voltage difference between two arbitrary points in time?

3. How do you store a waveform from channel 1 to the internal memory (what is the key sequence)?

4. How do you store a waveform from channel 1 to a memory stick (what is the key sequence)?

5. How do you make a paper copy of an oscilloscope trace?

6. What is the difference between "AC Coupling" and "DC Coupling"?

7. What is the difference between a digital oscilloscope and an analog oscilloscope?

8. What is the purpose of the "trigger" for an oscilloscope?

9. What is the relative resolution of an oscilloscope with an 8-bit D/A converter? … with a 12-bit D/A converter? … with a 16-bit D/A converter?

10. What is the uncertainty of the measurement of a VOLT1: _____ signal on the oscilloscope when the vertical scale is set to 2V / division, given that it uses an 8-bit D/A converter?

11. What is the uncertainty of the measurement of a VOLT0: _____ signal on the oscilloscope when the vertical scale is set to 5V / division, given that it uses an 8-bit D/A converter?

In-Lab Procedure

Bring flash drives to store your traces.

Ask the TA questions regarding any procedures about which you are uncertain.

Turn off all power supplies any time that you make any change to the circuit.

Do NOT apply more than 5 V to the circuit at any time.

Arrange your circuit components neatly and in a logical order.

Compare your breadboard carefully with your circuit diagram before applying power to the circuit.

Complete the following tasks:

Part I – Using the oscilloscope

1. Power up the oscilloscope. When the oscilloscope is ready, press the [WaveGen] button to turn on the function generation. Connect the "Gen Out" to one of the BNC cables. Adjust the function generator to 1MHz, use a rectangular wave, and from the displayed waveform, adjust the peak-to-peak voltage to 1V.

2. Using the measurement function of the oscilloscope, measure the exact frequency and peak-to-peak voltage of the square wave.

3. Acquire the waveform using the data acquisition software on the work station Desktop Computer.

4. Reference the directions in the "single pulse" section of Appendix D to acquire a zoomed-in view of 1MHz square wave.

5. Adjust the function generator offset voltage and observe what happens to the shape and location of the square wave. Use both "AC coupling" and "DC coupling".

Part II – Half Adder

1. Connect the fixed 5 V DC supply to ground and connect the supply to the breadboard.

2. Construct the half adder circuit using the hardware provided.

3. Set the function generator frequency to 100kHz. Treat the function generator as one input (A) and pull the other input (B) to Vcc. Show the outputs on the oscilloscope screen and verify the correctness of your circuit. Pull the input B to ground and repeat the process. Increase the input frequency to 1MHz and higher to observe any differences of output. Save the outputs as experiment results.

Post-Lab Report

Write up your circuit, schematic, and lab procedure. Mention any difficulties encountered during the lab. Describe any results that were unexpected and try to account for the origin of these results (i.e. explain what happened). In addition, answer the following questions:

Part I – Using the Oscilloscope

1. Describe the difference between the "dots" display and the "vectors" display setting.

2. To measure the frequency of a repetitive signal, how many periods are required? Is this independent of the location of the signal on the scope trace (hint: does the position of the minimum affect the ability to measure the frequency?)

3. How many measurements can be displayed at one time?