Ming Hsieh Department of Electrical Engineering

EE 109L - Introduction to Embedded Systems

Fall 2014 Lab Assignment #2
Test Equipment and the Arduino Uno

1 Introduction

This lab assignment has two purposes:

- Learn how to use test equipment to observe and analyze signals. An important aspect of this is learning how to set up the oscilloscope triggering so signals can be clearly seen.
- Familiarize yourself with the Arduino Uno microcontroller development board. You won't be programming the board in this lab but only looking at signals coming out of it generated by a program already installed in the processor.

For this lab exercise you need to have an Arduino Uno board that has been programmed with a sample program that does several operations using the input and output pins. These can be purchased from the EE-Systems office, or you can provide one to the instructor for programming prior to class.

2 Oscilloscope

An oscilloscope like the Agilent MSOX3024A models (Fig. 1) in VHE 205 are used to show a graph of one or more signals as they change with time. The scope plots the signals showing their voltage level along the vertical axis, and the time of the signal along the horizontal axis. By displaying multiple signals on the same plot, it's possible to see how they are interacting with each other. The scopes in VHE 205 are capable of displaying signals from four analog sources and sixteen digital sources at the same time on the display and are capable of displaying signals with frequencies up to 200MHz.

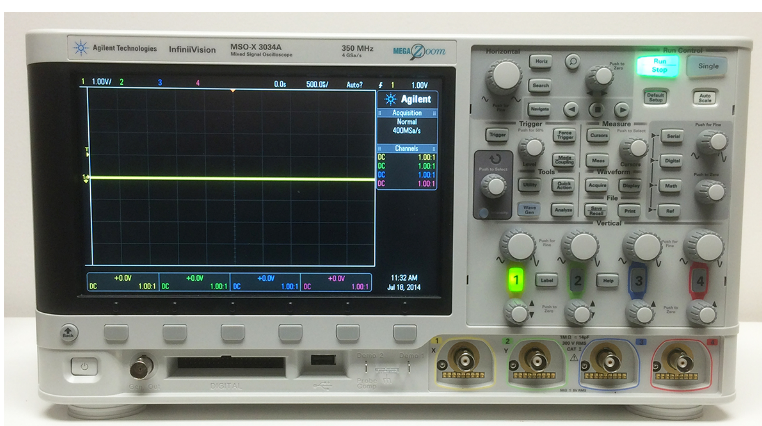

Figure 1: Agilent MSOX3024A oscilloscope

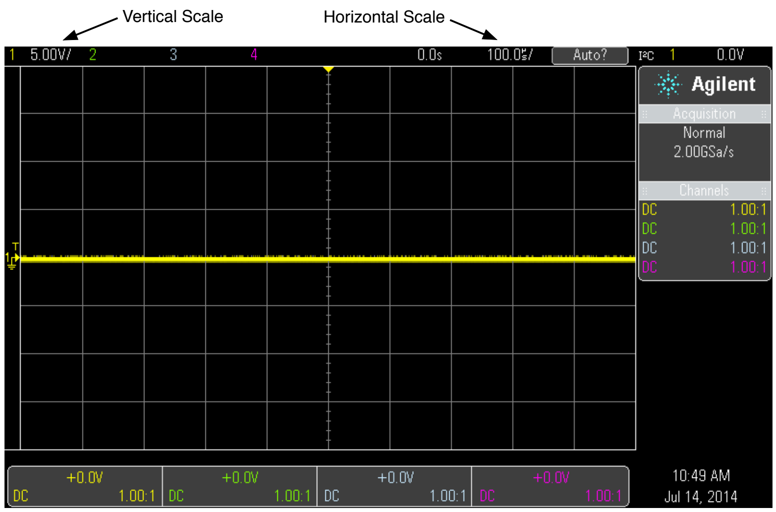

Figure 2: Default display of the Agilent 3000 oscilloscope

Turn on the oscilloscope by pressing the white button in the lower left of the front panel and wait for it to go through a bunch of self tests. In less than a minute it should be ready to go. To get started, press the "Default Setup" button in the upper right portion of the front panel, After it says on the screen that the default setup has been restored, press the "Back" button in the lower left corner. This should restore the scope to the default settings in case someone has altered them. You should see a screen like that in Fig. 2.

Look closely at the screen and note the vertical and horizontal grid lines. There are eight grid spaces vertically and ten horizontally. These are used in conjunction with the vertical and horizontal scale factors to measure the voltages and timing of signals. The first two numbers to look for are the scale factors for the vertical and horizontal axis. Each of the four channels can have a different vertical scale factor and these are shown along the upper left of the screen. In the default settings, only channel 1 (yellow) is turned on, and the display shows that the vertical scale factor is "`5.00V`". This means that each of the grid lines represents 5 volts of signal level. The horizontal scale factor is set to "$100\mu s$" meaning that the signal is plotted with each horizontal grid space representing 100 microseconds of time.

Since there is no input signal connected to channel 1, the plot or "trace" for channel 1 is just a horizontal line across the screen. The position of the trace can be adjusted with the small knob just below the illuminated "1" in the Vertical section of the front panel. Try rotating that knob back and forth and observe how the trace moves up and down on the screen. The position knobs for the four channels are used to spread the four traces out on the screen so you can see them.

Open up the storage compartment on the top of the scope and remove the scope probe that has the yellow markers on the cable. Insert the probe's connector onto the BNC connector for channel one. To make proper contact, the connector needs to be pushed onto the scope's connector fully and then rotated clockwise to lock it in place. You'll know it's locked in place if you can't pull it off without first rotating it counter-clockwise.

The probe has a metal hook at the tip that is exposed by pulling back on the plastic body of the probe. At the lower edge of the scope, hook the probe tip onto the metal tab just right of the USB port that says "`Probe Comp`". The Probe Comp output is normally used to make adjustments to the probe calibration, but here we're just using it as a signal source to put something on the screen. You should now see two yellow lines going across the screen about a half a grid spacing apart. To get a better view of the signal we need to change the vertical scale factor to make the signal bigger on the screen. In the Vertical section of the front panel, just above the illuminated "1", the large knob is the vertical scale adjustment. There is one for each channel. Rotate the one for channel 1 a couple of clicks to change the vertical scale factor to 1.0 volts/division (shown at the top left of the display) and this will move the two yellow lines further apart. You should be able to see a lot of vertical lines running around on the screen between the two horizontal lines. The display should look something like that in Fig. 3.

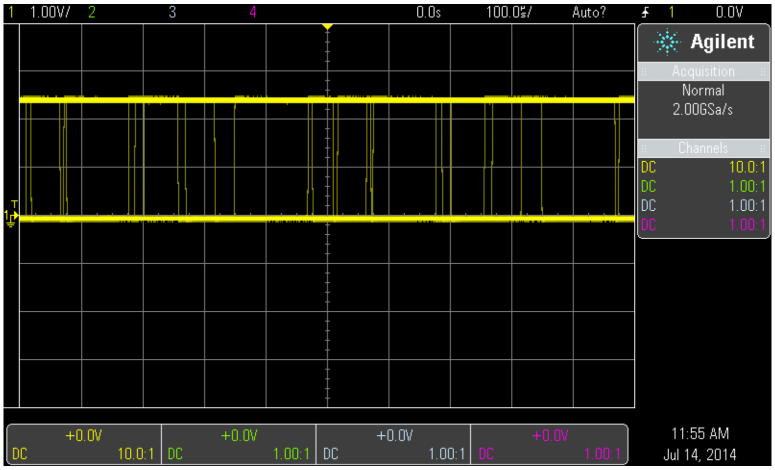

Figure 3: Display with probe connected to Probe Comp output

The next step is to get the display to sit still so we can see what it looks like. To do this we need to set up the triggering on the scope. On an oscilloscope, "triggering" means setting the conditions that makes the scope draw a picture of the signals present on its input channels.

Understanding how triggering works and setting up the triggering conditions is probably the most important aspect of making a scope into a useful diagnostic tool. The scope can be set to trigger when the input signal meets a set of conditions and it will then display the signals on all the channels that occurred before and after that time. It will then wait until the conditions are met again, and then display the signal again. With periodic signals this gives you stable display of the signals since each time they are drawn on the screen they are in the same position. When the scope doesn't know what to trigger on (like now), it just shows a running display of the incoming signal which usually results in the plot of the signal dancing around on the screen. While you are seeing the signal, it probably won't be stable on the screen and it will be difficult to analyze. When the triggering has been adjusted properly a stable display will be shown and you can get a clear look at the shape of the signal.

In the Trigger portion of the front panel, press the "Trigger" button and the Trigger Menu will appear at the bottom of the screen (Fig. 4). Pressing the button below one of the options will bring up a list of all the possible settings for that option. Note that the one pressed will also have a green circle with an arrow on it. This indicates that the setting can be adjusted using the knob just to the right of the display screen.

Trigger Type - Determines what type of signal to trigger on. Pressing the button will show all the different triggering types the scope supports. "Edge" triggering is the most common and that's what we will use today. If it doesn't say "`Edge`", use the knob to select it.

Source - Tell which input channel to use for triggering. Set it to channel 1.

Slope - For edge triggering, determines whether to trigger on a signal edge when the signals is going from a low voltage to a high voltage, high to low, either, etc. Set if for a rising edge.

Now that the scope knows to trigger on a rising signal edge on channel 1, we need to tell the scope exactly what voltage level to trigger on. The trigger level is adjusted with "Level" knob in the Trigger section and the value is shown at the top right of the display. Right now it probably says "`0.0V`" and this is the problem. Since the signal goes from 0 volts to about 2.5 volts, there is no rising edge where it is zero volts so the scope is not triggering The scope actually tells us that it's not triggering by putting "`Auto?`" in the top right of the screen. The question mark is the scope's way of saying that it's not triggering.

Rotate the Level knob until the value in the upper right says `1.0V` and the display should freeze and show something like Fig. 5. Note that the "`Auto?`" has changed to "`Auto`".

The trigger level is always indicated on the display with the small "T" and triangle along the left side of the display. When the level is being adjusted it also shows it as a horizontal line through the display. Look

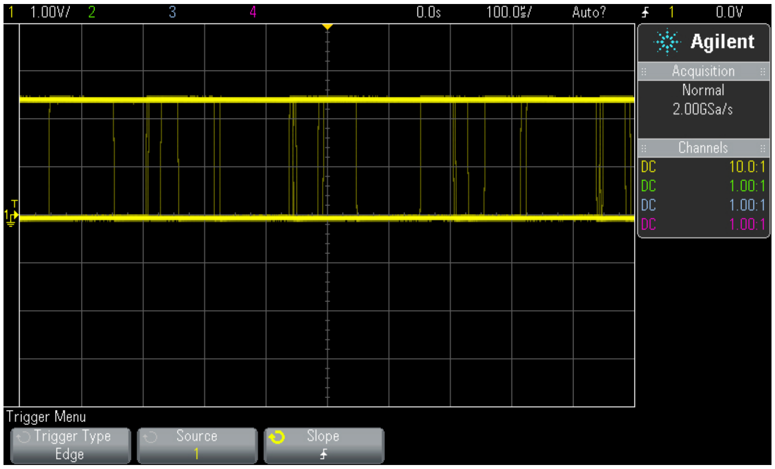

Figure 4: Setting the triggering options

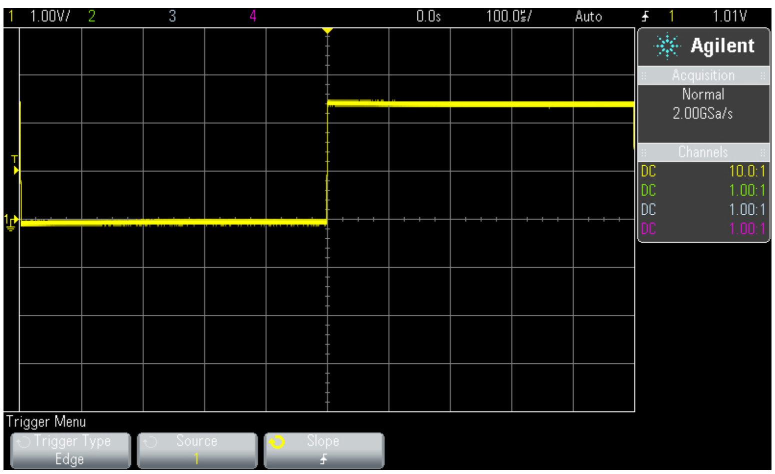

Figure 5: Triggered display, trigger level = 1.0V

at the top center part of the display and there should be small triangle pointing down. This indicator shows the point along the horizontal (time) axis where the trigger condition was met. The default position is in the middle of the screen so you can see what the signal looked like before and after the triggering occurred. To see more of the signal before or after the trigger point, rotate the small knob in the Horizontal section of the controls to move the trigger point (and the displayed signal) back and forth. You can always return the trigger point to the center of the screen by pressing the horizontal position knob.

The last step is to change the horizontal scale factor, also known as the "time base" so we can see more of what the signal looks like. Rotate the large knob in the Horizontal section of the front panel to change the time per division (shown at the top of the display) until it says $500\mu s$. We can now see that the signal (Fig. 6) is a square wave that repeats every $1000\mu sec$, or $1msec$. From the vertical scale we can see that the signal goes between zero volts and +2.5 volts.

Try playing with the controls (horizontal time base, vertical scale, horizontal position, vertical position, trigger level) to get a feel for how they affect what you see on the screen. Note that using the vertical position control to move the trace up and down on the screen does not affect the triggering level, it moves with the trace.

3 Function Generator

The Probe Comp test point on the scope is a convenient place to get a simple signal to look at but we want to see more interesting signals. In most electronics labs at USC, a separate piece of test equipment called a function generator or a waveform generator is used to produce a wide variety of signals. In VHE 205 the Agilent oscilloscopes contain a built-in waveform generator that can produce signals of several different shapes and this will be used in this lab assignment.

From the podium at the the front of the room, get one of the black cables that has a BNC connector on one end, and the other end is a pair of white and black wires with pins on the end. Attach the cable to the BNC connector just to the right of the scope's on/off button that's labeled "`Gen Out`". If you haven't done so already, disconnect the scope probe from the Comp Probe tab and connect it to the pin on the end of the white wire. Also attach the black alligator clip on the scope probe to the pin on the end of the black wire.

In the Trigger portion of the controls, press the "`Wave Gen`" button to bring up the wave generator options. You should see something like in Fig. 7. The tabs along the bottom are used to change the type and shape of the wave that is being produced. The effect of changing these parameters is illustrated in Figures 8, 9 and 10.

Waveform - Selects what type of signal to produce. Sine, square, ramp and pulse waves are the most commonly used but others are also available.

Frequency - The frequency of the output wave in Hertz (Hz) or cycles per second.

Amplitude - The amplitude of the wave is given as the "peak-to-peak" voltage (V_{PP}), the voltage between the most negative part of the wave and the most positive part it. Other function generators may give the amplitude as the peak voltage, the voltage from the center of the wave to the top of the positive part of the wave. The peak-to-peak voltage will be twice the peak voltage. When using a function generator you always need to determine which way the amplitude is characterized.

Offset - Adjusting the offset voltage moves the waveform voltage up or down without otherwise changing its shape. If a signal with an amplitude of $4.0V_{PP}$, covering a range from $-2.0V$ to $+2.0V$, is given an offset of $+1.0V$, then the signal will cover a range from $-1.0V$ to $+3.0V$.

As seen from the labels on the tabs, the wave generator is producing a 1kHz sine wave with amplitude of $0.5V_{PP}$ and no offset. Note that the scope is not triggering since the trigger level is still set to $1.0V$ and this is higher than the maximum amplitude of the waveform. Press the button below the Amplitude label and then rotate the knob just right of the screen to increase the amplitude to $3.0V$. This should cause the scope to start triggering and you should get something like Fig. 11.

With the 1 KHz sine wave displayed on the scope, use the Waveform menu to change it to some other type of wave. Take a look at a few of them and when done change it back to the sine wave. Now try altering

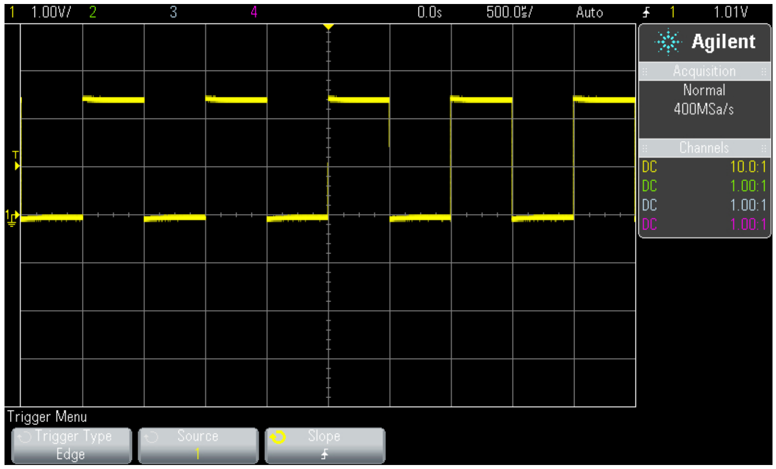

Figure 6: Horizontal scale adjusted for 500μsec per division

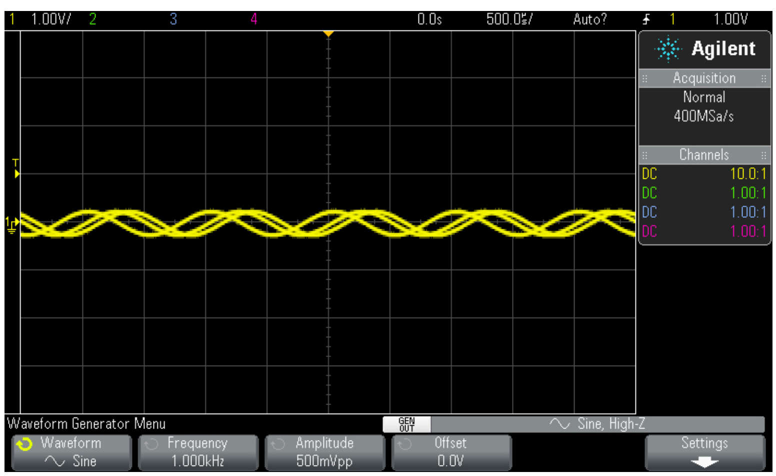

Figure 7: Wave generator options

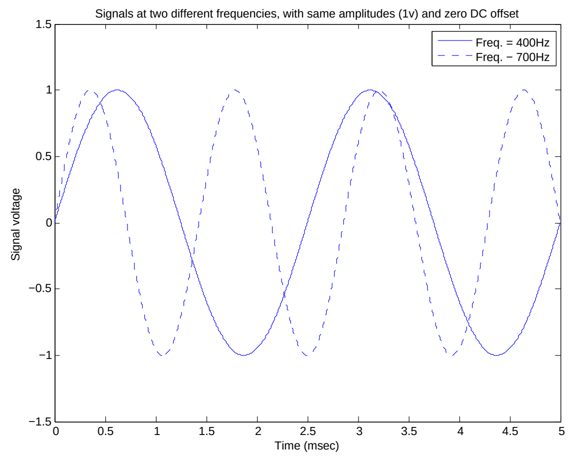

Figure 8: Two signals with different frequencies

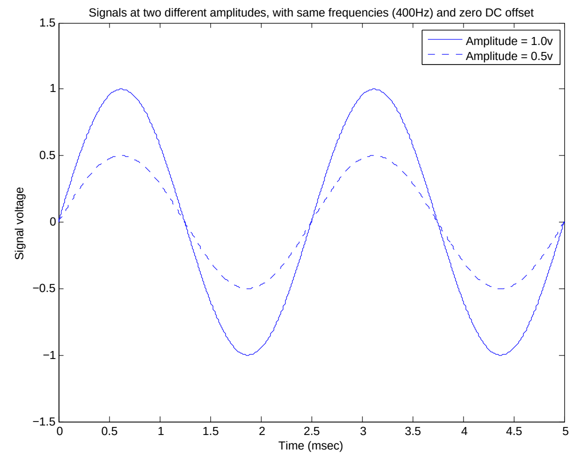

Figure 9: Two signals with different amplitudes

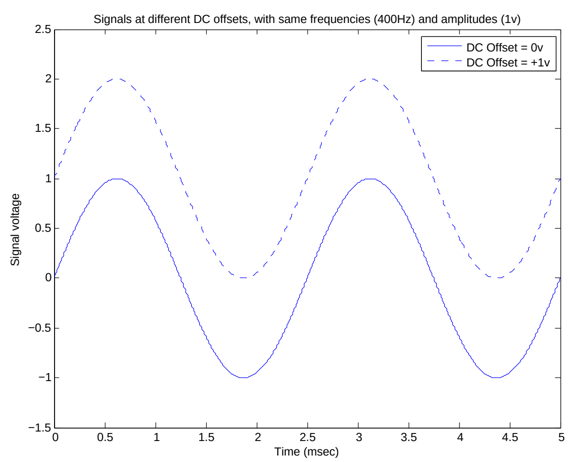

Figure 10: Two signals with different DC offsets

the frequency and amplitude settings of the waveform generator. Observe how the picture of the waveform changes on the scope. If the waveform on the scope gets too big or too small, use the vertical and horizontal scale knobs to change the settings. Confirm that the frequency and amplitude voltage that the function generator is set for matches (more or less) what you are seeing on the scope.

Now try adjusting the offset of the waveform generator. As a DC voltage is added to the sine wave, the display on the scope should move upwards. Adjust the vertical scale on the scope so you can see how far it moves. From the settings of the vertical scale measure how much DC voltage is present in the signal and compare this with the DC offset setting on the function generator.

Adjust the frequency back to 1kHz, the amplitude back to $3V_{PP}$ and the offset back to $0V$. From the rack at the front right of the room get a pair of red and black clip leads like you used in a previous lab. Plug them into your multimeter and set the multimeter to measure DC Volts. Without disconnecting the waveform output from the scope probe, also hook the red and black test leads from the multimeter to the wires coming coming from the waveform output of the scope (red → white, black → black). You should have the white wire from the waveform output connected to both the scope probe and the multimeter red lead, and the three black leads from the waveform generator, the scope and the multimeter connected together.

On the multimeter, select "AC V" to measure AC voltage. The multimeter should now indicate the AC voltage coming from the scope's waveform generator. However multimeters do not measure the peak voltage or peak-to-peak voltage but rather the RMS (root mean square) voltage. The relationship between V_{RMS}

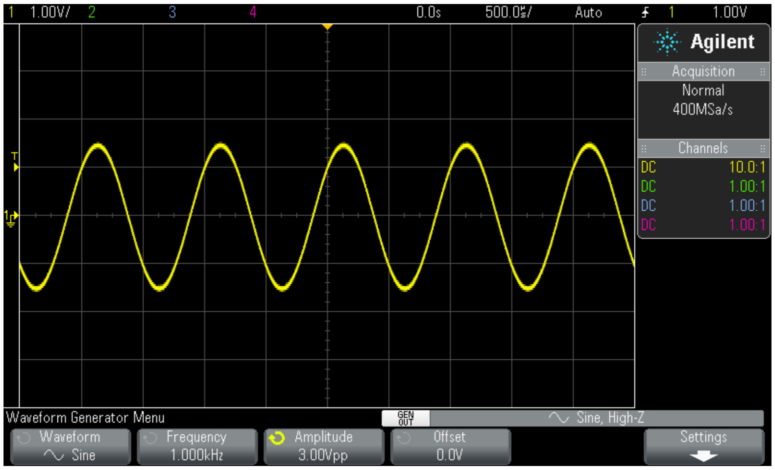

Figure 11: Scope triggering on a 3Vpp sine wave

and V_{Peak} for a sine wave is

$$V_{RMS} = \frac{1}{\sqrt{2}} V_{Peak} = \frac{1}{\sqrt{2}} \frac{V_{Peak-to-Peak}}{2}$$

Based on the peak-to-peak voltage that you have set on the function generator use the above equation to calculate the V_{RMS} of the function generator output.

Question 1: What AC voltage did you measure with the multimeter, and what is the predicted V_{RMS} as given by the equation?

Switch the multimeter to measuring DC volts. The reading on the multimeter should be zero since the offset is set to zero volts. Adjust the offset voltage on the function generator to something between 1 and 2 volts and see if the multimeter sees the same DC voltage.

Question 2: How much DC offset did you add and what does the multimeter say the DC voltage is?

With the offset voltage added to the signal, check the AC voltage on the multimeter.

Question 3: How much did the AC voltage change when you added a DC offset to the signal?

4 Arduino Uno

The Arduino Uno is a microcomputer development board based on an Atmel ATmega328P eight-bit microcontroller. Along with the microcontroller the board has connectors to allow solderless connection to the ATmega328P input and output pins, a clock oscillator, a second microcontroller for implementing a USB interface, and other related components. As can be seen in Fig. 12, the Uno has 13 I/O ports along the top that are labeled "Digital (PWM-)". These are referred to as ports D0 through D13 in the notes below. The six I/O ports at the lower right are labeled "Analog In" on the board and these are referred to as ports A0 through A5 below.

Plug the USB cable into a port on your computer or on the back of the iMac and then plug it into the USB connector on the Arduino board. In this lab the USB cable is only being used to provide power to the Arduino. In future labs the USB cable will also be used to download programs to the Arduino. Soon after power is applied to the Arduino an LED near the D13 I/O pin should start to blink about once each second.

The microcontroller input and outputs pins are all connected to one of the four black connectors along the edges of the boards. To connect the scope probes to one of these signals, take a short (1-2 inch) piece of wire from your project box and strip about 1/4" off each end. Hook the probe clip to one end of the wire, and you can then insert the other end down into one of the holes in the black connectors. The same needs

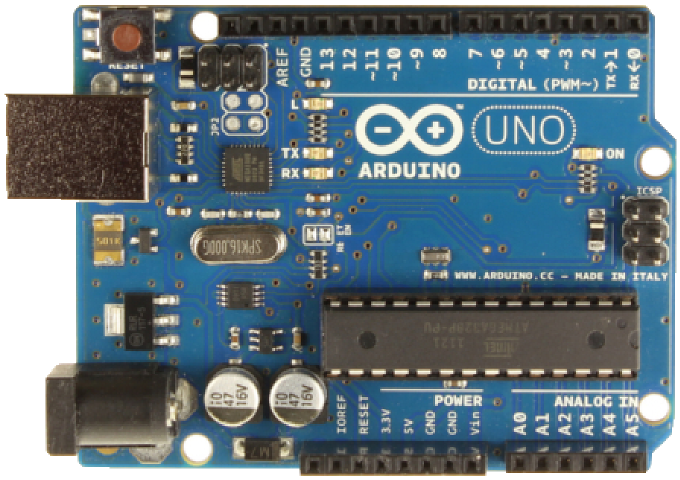

Figure 12: Arduino Uno R3

to be done to establish a connection between the Arduino's ground and the ground connector of the scope probe. The wire from the probe's ground clip should be inserted into one of the ground connections on the board (labeled "GND").

4.1 Outputs A0 and A1

Use the oscilloscope to observe the action of bits A0 and A1. Both of these should be oscillating at different frequencies.

Question 4: Which bit is oscillating fastest, and which is oscillating slowest?

Question 5: Fill out the table on the lab report page for both of the above signals showing the time it spends in each state, the period, the frequency and the duty cycle. The duty cycle of a signal is the ratio (expressed as a percentage) of the time the signal is in the high state to the period of the signal.

The time it takes a signal to go from the 0 to the 1 state, or from the 1 to the 0 state, is called the transition time of the signal. The transition time can vary greatly depending on the technology used in the circuit, the type and size of the load on the signal, etc. For this question, assume the transition time is defined as is the time it takes the signal to go between 1.0 Volts and 3.0 Volts (or vice versa).

Question 6: Examine the bit that is oscillating fastest and determine the $0 \rightarrow 1$ and $1 \rightarrow 0$ transition times.

Question 7: After the signals make the transition from 1 to 0 and 0 to 1 and are stable in one of these states, what voltages are the bits at when in the 0 state and the 1 state?

4.2 Output A2

Use the scope to observe the pulse signal being output on A2. Unlike the periodic signals on A0 and A1, the signal on A2 is not periodic. It appears at random times and the pulse width varies. To see signals like this the scope has to take a single acquisition of the signal and then hold it so it can be viewed. Set up the triggering of the scope to trigger on the input channel connected to A2 and then press the "`Single`" button in the upper right to capture one event and display it. Change the horizontal rate setting to expand the pulse out to where you can see its shape. Repeat the capture several times and determine the range of the pulse width.

Question 8: What is the minimum and maximum width of the A2 pulse that you observed.?

5 Lab Report

Question 1: What is the measured AC voltage and the predicted V_{RMS} as given by the equation?

Measured: ______________ Calculated: ______________

Question 2: How much DC offset did you add and what does the multimeter say the DC voltage is?

Setting on generator: ______________ DC voltage measured: ______________

Question 3: How much did the AC voltage change when you added a DC offset to the signal?

____

Question 4: Which bit is oscillating fastest, and which is oscillating slowest?

Fastest: ______________ Slowest: ______________

Question 5: Fill out the following table for bits A0 and A1. The duty cycle of a signal is the ratio (expressed as a percentage) of the time the signal is in the high state to the period of the signal.

Bit	Time in 0 state	Time in 1 state	Period	Frequency	Duty cycle
A0					
A1					

Question 6: Determine the $0 \to 1$ and $1 \to 0$ transition times in nanoseconds.

$0 \to 1$: ______________ $1 \to 0$: ______________

Question 7: What voltages are the bits at when in the 0 state and the 1 state?

0 state voltage: ______________ 1 state voltage: ______________

Question 8: What is the range of the width of the A2 pulse? ______________