ME 360: FUNDAMENTALS OF SIGNAL PROCESSING, INSTRUMENTATION AND CONTROL

Lab No. 1 – Introduction to Laboratory Instruments

1. CREDITS

Originated: N. R. Miller, June 1990

Last Updated: S.Y.Olmez and D. Block, August 2020

2. OBJECTIVE

(a) Learn the proper use of the digital multimeter (DMM), function generator, and oscilloscope through simulation.

(b) Measure the frequency response (gain and phase shift) of a first-order, low-pass, RC filter.

(c) Determine an unknown capacitance.

3. KEY CONCEPTS

(a) The oscilloscope and digital multimeter are powerful instruments. The oscilloscope is restricted to voltage measurements (AC and DC) but provides better *qualitative* information by allowing one to observe transient waveforms. The multimeter measures current and resistance as well as voltage and typically yields better *quantitative* information than the oscilloscope.

(b) Except in rare circumstances, voltage measurements are *nonintrusive* (do not materially affect normal circuit operation) whereas current and resistance measurements are *intrusive* (the measuring instrument becomes part of the circuit).

(c) A first-order system is characterized by two dynamic parameters: (i) the steady-state gain K and (ii) the system time constant τ.

(d) A low-pass RC filter consists of a resistor and capacitor in series. The input voltage is applied across both components in series, and the output voltage is measured across the capacitor.

(e) A low-pass, RC filter behaves as a first-order system with K = 1 and τ = RC.

(f) The key features of a first-order system are:

Governing Equation	$\tau \dfrac{dV_{out}(t)}{dt} + V_{out}(t) = K\, V_{in}(t)$
Transfer Function	$\hat{G}(s) = \dfrac{K}{\tau s + 1}$
Steady-state Response	Input: $V_{in}(t) = V_\infty$ where V_∞ is constant with time Output: $V_{out}(t) = K\, V_\infty$
Step Response	Input: $V_{in}(t) = V_\infty$ for $t > 0$ where V_∞ is constant with time Output: $V_{out}(t) = [V_{out}(0) - K\, V_\infty]\exp(-t/\tau) + K\, V_\infty$
Sinusoidal Response	Input: $V_{in}(t) = A\cos(\omega t) = A\cos(2\pi f t)$ where A is amplitude, ω is the angular frequency in rad/s, and f is the frequency in Hz Output: $V_{out}(t) = G(\omega)\, A\cos[\omega t + \phi(\omega)]$ where $G(\omega)$ is gain and $\phi(\omega)$ is phase shift given by $G(\omega) = \dfrac{K}{\sqrt{1 + (\tau\omega)^2}}$ $\qquad \phi(\omega) = -\arctan(\tau\omega)$

(h)   In this simulation, we measure (i) the resistance R, (ii) the gain $G(\omega)$ and (iii) the phase shift $\phi(\omega)$ of a low-pass RC filter, and then use these values to solve for the time constant τ and the unknown capacitance C.

4.   SYNOPSIS OF PROCEDURE

(a)   Perform current measurement on a simple circuit, consisting of a voltage source and the resistor, in Simulink. Compare the scope output to the value on the display. Compare the value on the display with the value calculated using Ohm's Law.

(b)   Perform voltage measurement on a simple circuit, consisting of a voltage source and the resistor, in Simulink.

(c)   Simulate an RC filter in Simulink.

(d)   Determine the gain $G(\omega)$ and phase shift $\phi(\omega)$ of a low-pass RC filter by comparing the input and output sinusoids on Simulation Data Inspector in Simulink.

(e)   Use the gain and phase shift to calculate the filter time constant τ. Use the time constant and the measured resistance R to find the unknown capacitance of the filter capacitor.

5.   BACKGROUND

In this experiment, we consider three of the most useful general-purpose electronic instruments available; namely, the digital multimeter (DMM), the oscilloscope, and the function generator. Background information on each of these instruments appears in Appendices A, B, and C, respectively. Please watch the video provided by your TA to get yourself familiar with these instruments.

This semester, you will have no access to the physical lab. This lab is modified to get you familiar with these instruments by simulating their functions on Simulink. Simulink is a simulation tool, which is an add-on of MATLAB. Future labs will use this tool extensively, so this lab also aims at getting you familiar with it. *If you are using Citrix through a VPN to run MATLAB you should not need to perform these next steps, but just in we have them here in the lab. If you type "ver" (without the quotes) at MATLAB's command prompt you will see all the toolboxes you have access to and Simscape and Simscape Electrical should be listed.* **Before you start this lab, please make sure that you installed Simscape and Simscape Electrical add-ons.** You can do this by clicking on Home->Add-Ons button on the top bar menu and searching for Simscape and Simscape Electrical.

6. PROCEDURE

6.1 Current Measurement in Simulink

6.1.1 Create Simulink Diagram for Current Measurement

(a) In the physical lab, we would wire the circuit as in Figure 1. To simulate this in Simulink, create the diagram as in Figure 2.
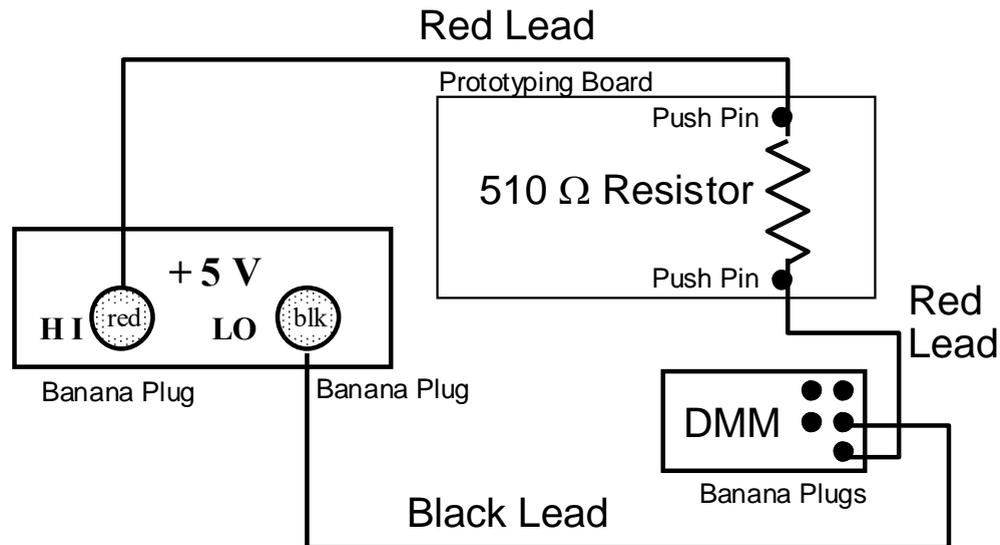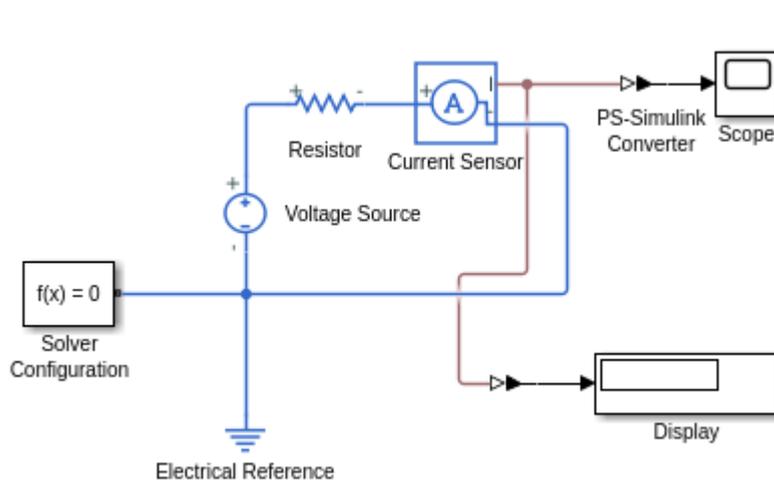

Figure 1: Wiring for Current Measurement
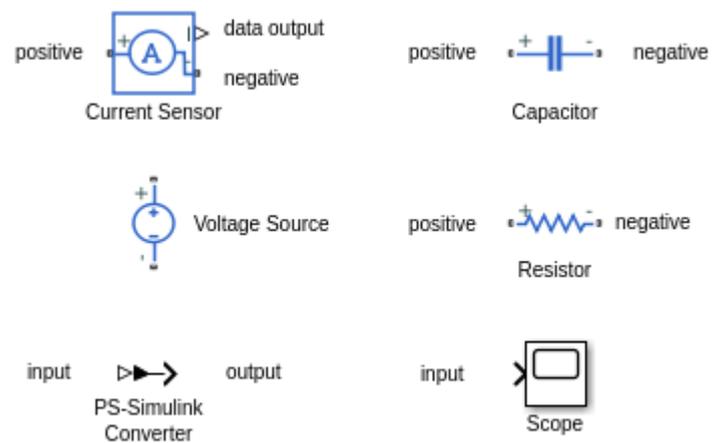

Figure 2: Simulink Diagram for Current Measurement

Figure 3: Simulink Blocks for Current Measurement

Detailed Procedure for Making Current Measurements

- Open MATLAB. Open Simulink by clicking on *Simulink* button in the toolstrip banner. Alternatively, typing *simulink* to your command window.

- On the simulink start page, go to the Simscape drop-down menu and from the Electrical section click *Create Model* as in Figure 4.
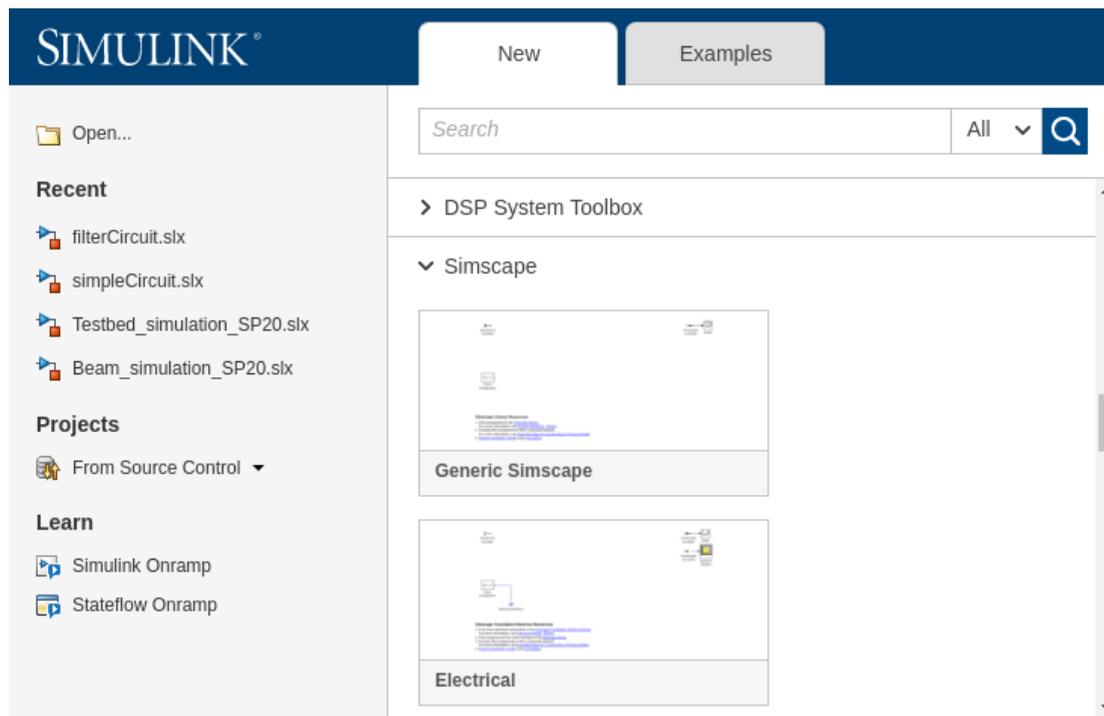
Figure 4: Select Simscape-Electrical in the Simulink start page.

- Click on the Library Browser button in the toolstrip of the model that opens. Navigate to Simscape -> Electrical as in Figure 5.
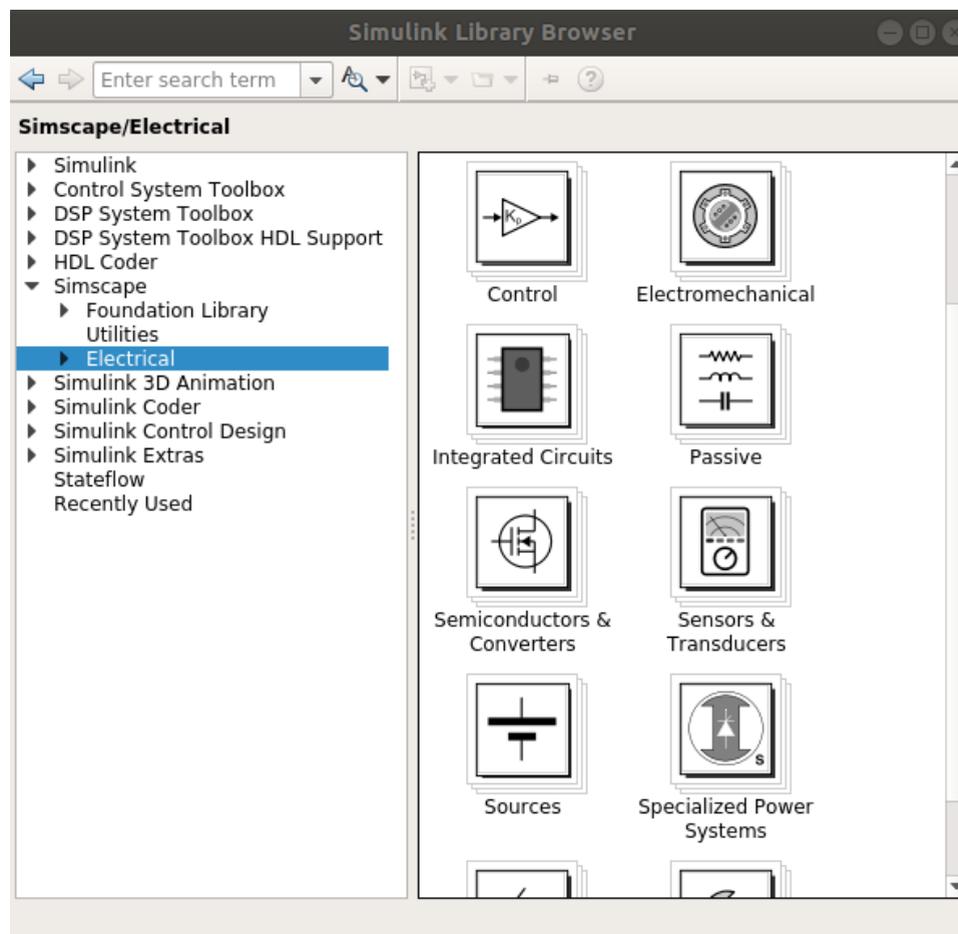

Figure 5: Navigate to Simscape-Electrical in Simulink Library Browser

- Double-Click on Sources. Here you will see various options. Get the block named *Voltage Source* and place it on the Simulink diagram by simply left clicking on it and dragging it to the white space of your Simulink diagram. On the Simulink diagram, double click on the voltage source that you placed. This pops up a *Block Parameters* menu. Set *DC Voltage* to 5 V and click *OK*.

- In the Simulink Library Browser, go to Simscape->Electrical->Passive. Get *Resistor* and place it on your diagram. Double click and set its value to 510 Ω. Set its tolerance to 5% and tolerance application to random tolerance with uniform distribution.

- Go to Sensors & Transducers, get *Current Sensor* and place it on your diagram.

- Go to Simscape -> Utilities, get a *PS-Simulink Converter* and place it on your diagram. We need this converter whenever we want to convert a physical signal into a Simulink output signal. Electrical and mechanical signals are characterized as physical signals in Simulink and they are denoted with red, solid arrows. In order to examine these signals using Simulink blocks such as *Scope* and *Display*, we need to convert these to Simulink signals. This can be done by using this *PS-Simulink Converter* block.

- Go to Simulink->Sinks and drag *Display* to the diagram. Click on the input of this block. A red, dashed arrow will appear. Connect this to the output of *PS-Simulink Converter* that you placed in the previous step.

- Connect *Electrical Reference* to the negative side of *Voltage Source.* To do this, click on the negative side of the *Voltage Source.* A red, dashed arrow will appear. Connect the other side of the arrow to the blue line which connects *Electrical Reference* and *Solver Configuration.*

- Connect the positive side of *Voltage Source* to the positive side of *Resistor.* To do this, click on the positive side of the *Voltage Source.* A red, dashed arrow will appear. Connect this arrow to the positive side of the *Resistor.*

- Connect the negative side of *Resistor* to the positive side of *Current Sensor.* Connect the negative side of *Current Sensor* to *Electrical Reference.*

- Connect the data link of *Current Sensor* to *Scope* and *Display*. To do this, click on the data output on the *Current Sensor* block. A red, dashed arrow will appear. Connect this to the input of *PS-Simulink Converter,* which is connected to the *Scope.* At this point, your diagram should look as in Figure 2.

6.1.2   Measure Current and Compare with Calculated Current

(b)   Using the theoretical value of resistance, calculate the expected current.

(c)   Hit *Run* button. You should see a horizontal line with a value of ~9.8 mA in your scope (9.8 x 10^{-3}) . Note that you do not have much of a resolution here. This is typically the case with the oscilloscope in the physical lab setting. It is a very powerful instrument to visualize signals, but it does not have as good as a resolution compared to DMM. You will see a value with higher resolution on *Display*. This is the case of DMM in a physical lab setting. Compare the value on the DMM to the theoretical value of resistance.

6.2 Voltage Measurement in Simulink

(a)   Now that you know how to measure the current, it is the time to demonstrate your knowledge by doing a simple voltage measurement. Remove the *Current Sensor* from the diagram. You can do this simply by clicking on the block and hitting *Delete* button on your keyboard. Find *Voltage Sensor* from the library. Wire your circuit in order to measure the voltage across the resistor. Note that voltage sensor should be wired in parallel. Submit the plot generated by *Scope* and your Simulink diagram.

6.3 Simulating Gain and Phase Shift of Low-Pass, RC Filter

6.3.1   Connect Resistor and Capacitor in Series and Place the Sensors

(a)   In the physical lab, we would wire the circuit as in Figure 6. To simulate this in Simulink, create the diagram as in Figure 7.
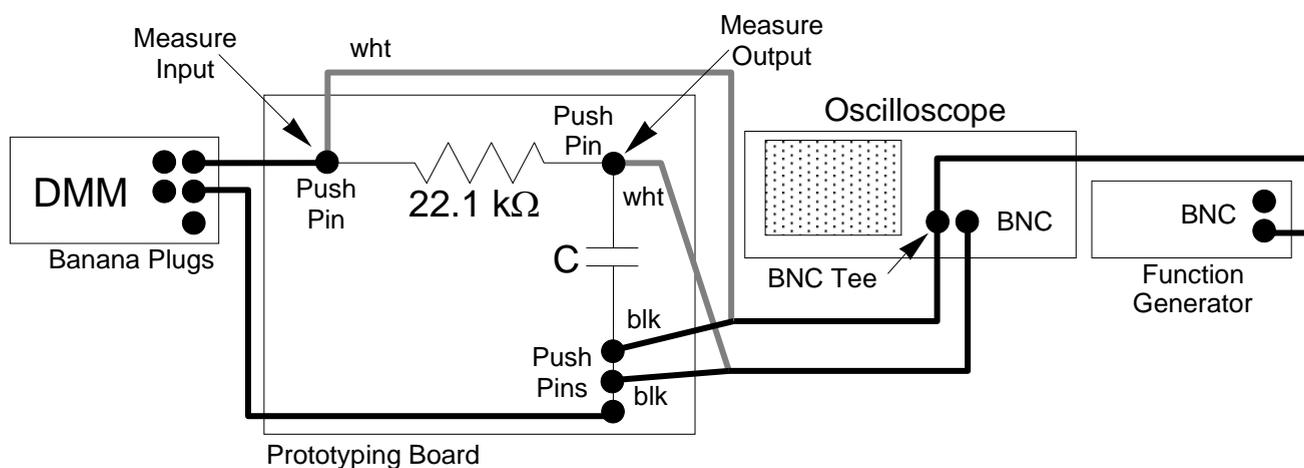
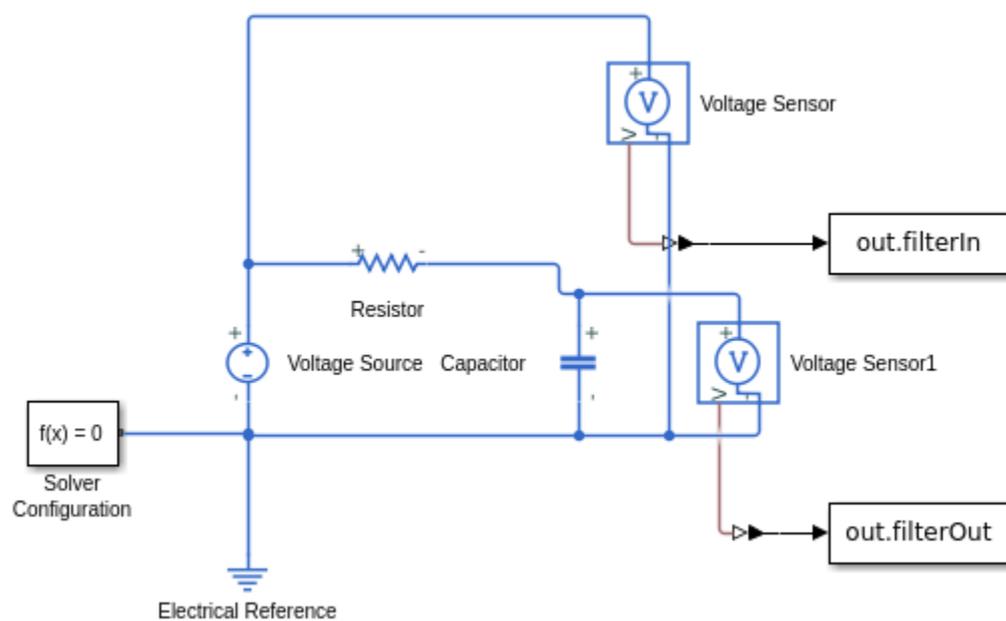**Figure 6: Wiring for Measuring Gain and Phase Shift of Low-Pass, RC Filter**

Figure 7: Simulink Diagram for Simulating Gain and Phase Shift of Low-Pass, RC Filter

Detailed Procedure for Creating the Diagram

- Open MATLAB. Open Simulink by typing *simulink* to your command window. This pops the simulink start page.

- On the simulink start page, go to Simscape menu and select Electrical again.

- Click on the Library Browser button. Navigate to Simscape -> Electrical.

- Click on Sources. Get *Voltage Source* and place it on the Simulink diagram. On the Simulink diagram, double click on the voltage source that you placed. This pops up a *Block Parameters* menu. Set *AC voltage peak amplitude* to 2V and *AC voltage frequency* to 1kHz. Click *OK*.

- Add a *Resistor* and a *Capacitor* (also in Passive menu) to your diagram. Set resistance to 22.1 kΩ and capacitance to 10nF. Set tolerance application of the capacitor to random tolerance with uniform distribution. Set capacitance tolerance to 20%.

- Go to Sensors & Transducers, get two *Voltage Sensor* and place them on your diagram.

- Place two *PS-Simulink Converter* from Simscape->Utilities to your diagram.

- Go to Simulink->Sinks and drag two *To Workspace* to the diagram. Set the name of one to *filterIn* and set the name of the other to *filterOut*. Set *Limit data points to last* to 512 and *Sample time* to 0.00001 for both.

- Connect the blocks as in Figure 7. Here, note that two overlapping links are connected only if there is a solid dot in their intersection. Otherwise they are not connected. *Solver Configuration* box comes as default whenever you launch Simscape-Electrical from your Simulink start page. It contains the properties of the solver

- Adjust your *Stop Time* to 0.01 seconds in the toolstrip

6.3.2 Gain and Phase Shift Measurement

(b) Run the simulation. Plot the input and output signals. Calculate the gain and phase shift of the RC filter. Calculate the capacitance.

Detailed Procedure for Measuring Gain and Phase Shift

- Hit Run and wait until the simulation is complete.

- Go to your MATLAB console and run the following

  plot(out)

  This will pop up a *Simulation Data Inspector*.

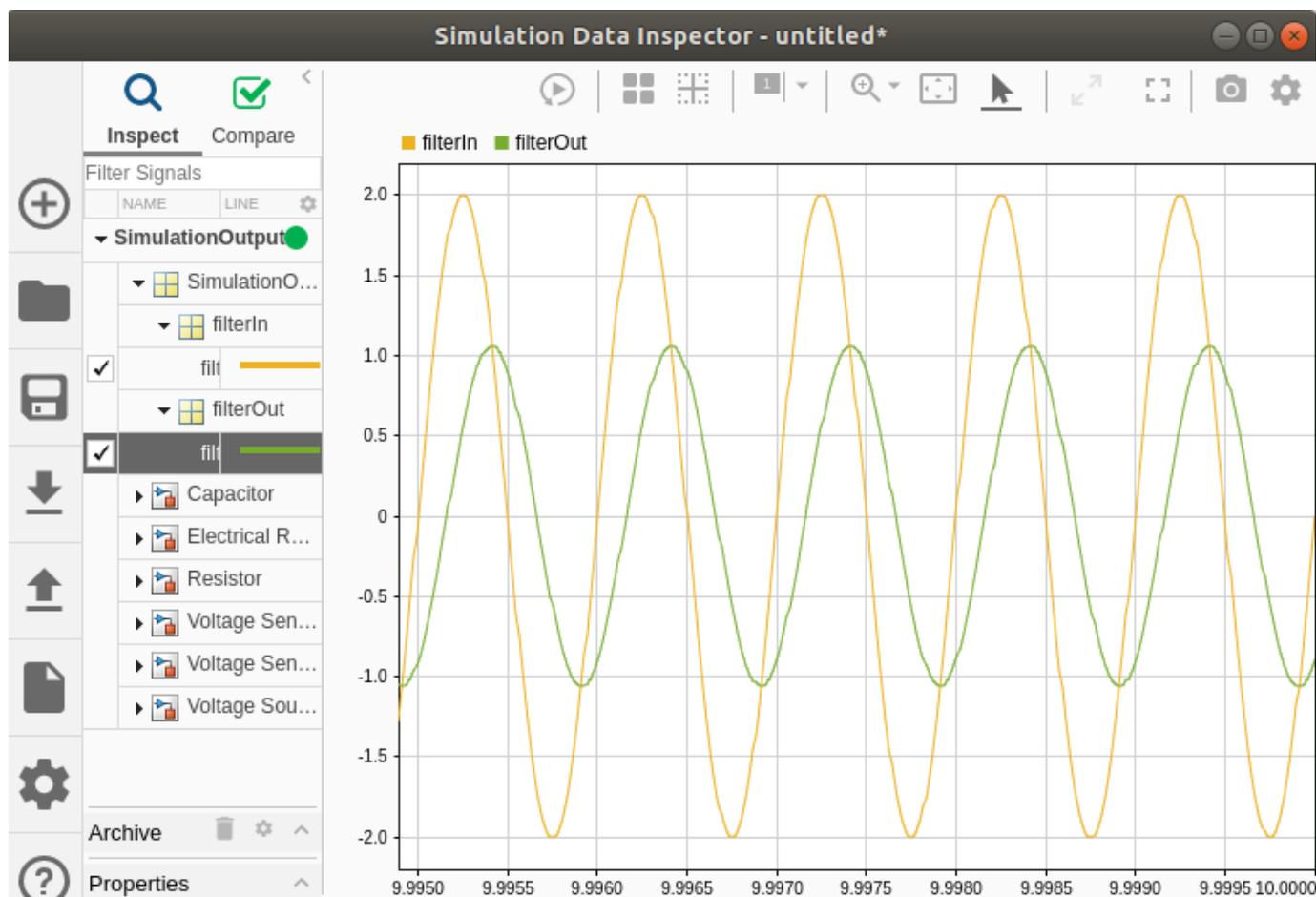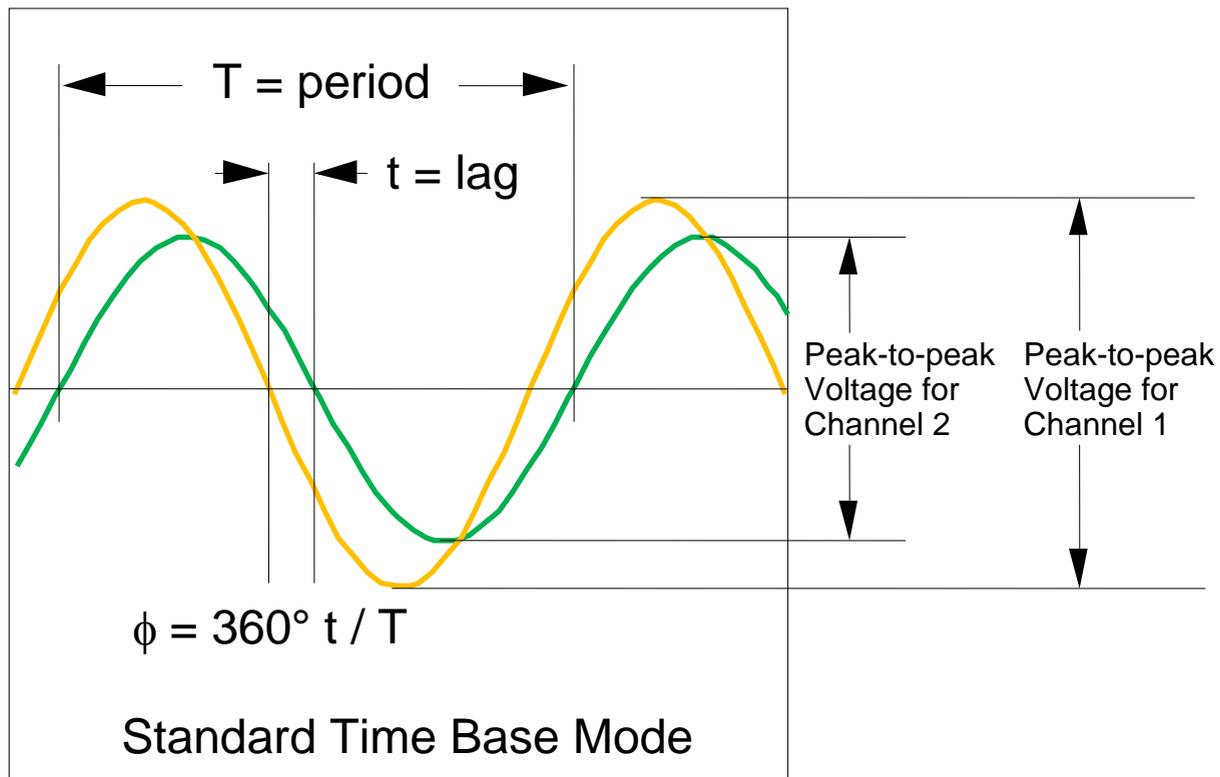- Select *filterIn* and *filterOut*. At this point, it should look like as in Figure 8.

Figure 8: Input and Output Signals in Simulation Data Inspector

T = period

t = lag

Peak-to-peak Voltage for Channel 2

Peak-to-peak Voltage for Channel 1

$\phi = 360° \, t / T$

Standard Time Base Mode

- Locate *Show/hide data cursors* button in the top menu. Select two cursors.

- Measure peak-to-peak voltage of both signals. Calculate the gain by using the following formula:

$$G = \frac{V_{pout}}{V_{pin}}$$

where V_{pout} is the peak-to-peak voltage of the output signal and V_{pin} is the peak-to-peak voltage of the input signal.

- Measure the time difference between two peaks. Calculate the phase shift by using the formula

$$\phi_d = 360° \, \tau_{delay} / T_{in}$$

where T_{in} is the period of the input signal.

- Calculate the capacitance based on phase shift

$$C_d = \frac{T_{in} \tan(\phi_d)}{2\pi R}$$

- Calculate the capacitance based on gain

$$C = \frac{T_{in}}{2\pi R} \sqrt{\frac{1 - G^2}{G^2}}$$

7. INSTRUCTIONS FOR THE REPORT

7.1. Format

You are expected to write a **brief** report demonstrating your work and your understanding of the concept. The report should have the following:

- Must be in pdf format.
- Name the file as 'Lastname_Firstname_360_Lab1.pdf'.
- Use either 11pt or 12pt font, Arial/Calibri/Times New Roman, single spaced.
- 1'' margins.
- Number all of your pages.
- Add a title page.

Submission instructions will be provided by the instructors.

7.2. Content

Your report should include all of the following:

- Using the information provided in the accompanying video walkthrough and the attached appendices, answer the following concept questions in a section titled "Concept Questions".
    1. In this experiment, what measurements are made using the digital multimeter?
    2. What measurements are made using the oscilloscope?
    3. Which of these instrument has the higher resolution and why?
    4. What measurement requires you to make series conections with the DMM leads?
    5. How does the information obtained from the oscilloscope differ from the DMM?
    6. What is the amplitude (peak-to-peak) of the voltage used in the example video by Dan Block?
    7. What four "standard" waveforms can the function generator produce?
    8. What are the resitance and tolerance values for the mil-spec coding 2212Q?

From your *Simulink* procedures in section 6, you should have or be able to generate the following:

- Submit your scope and display reading for the current measurement one. Was the value on the display different than the one you calculated? If it was different, why?
- Throughout the simulations, we used uniform distribution as the distribution of randomly selected resistance/capacitance values. Does this make sense? Would Gaussian distribution be a better choise? Explain your reasoning.
- Submit the Simulink diagram for voltage measurement.
- When measuring Voltage across the resistor, did the tolerance of the resistor effect the Voltage measurement? If there was a second resistor in series, would the tolerance have any effect when measuring voltage across one of them? Explain your reasoning.
- Submit your plot from gain and phase measurement part along with your calculations of gain, phase shift and capacitance.
- Are the periods of input and output signals the same for the RC filter?
- If the frequency of the input signal is increased, how would this affect the gain of the RC filter (increase or decrease)? Explain your reasoning.

Appendix A. Digital Multimeter (DMM)

The digital multimeter, or DMM for short, is the most common and widely used electronic instrument available today. It can measure both AC and DC voltage and current, resistance and signal frequency. Digital multimeters come in a wide variety of styles and performance levels including battery-powered handheld units intended primarily for field use and AC-powered bench-top units found primarily in the laboratory. Accuracy, measurement capability, and price vary between the different performance levels within each category. Bench-top units are typically more accurate but more costly than handheld instruments although the high end of handheld units overlaps the low end of bench-top models.

The instrument used in our laboratory is the HP 34401A, a 6-1/2 digit, high-performance, digital multimeter.

A.1 General Characteristics of the HP 34401A Digital Multimeter

The general characteristics of the HP 34401A are summarized below. Where appropriate, the specifications of this instrument are compared with those of other high-end units for illustrative purposes.

(a)   Definitions: *Resolution* for a DMM is the smallest difference in measurement the meter can display.

Accuracy for a DMM is how close the displayed measurement is to the true value.

(b)   The maximum resolution of this instrument is achieved in the 6-1/2 digit setting. In this setting the maximum resolution is 1.2 million counts. Or in other words, there are 1.2 million possible measurement readings that the DMM can display given 6-1/2 digits. On the 1-V scale this means that readings between -1.199999 and +1.199999 are possible. A reading of 0.999999 corresponds to 6 full digits of resolution, and the 20 % overrange capability is said to provide an additional 1/2 digit of resolution. This 1/2 digit by convention is the most significant digit of the reading and it can only be a zero or a one. Manufacturers of DMMs came up with this 1/2 digit terminology. In truth there is only $\log_{10}$ (1200000) = 6.08 digits of resolution. Resolution may also be described in terms of percent, parts per million (ppm), or number of bits. Thus, we have

$$\text{resolution as a fraction of full scale} = \text{1 part in 1,200,000} \approx \text{1 ppm} = 0.0001 \text{ \%}$$

$$\text{number of digits of resolution} = \log_{10} (1200000) = 6.08 \text{ digits} \approx \text{6-1/2 digits}$$

$$\text{number of bits of resolution} = \log_2 (1200000) + \text{1 sign bit} = 21.19 \approx \text{21 bits}$$

Thus, a resolution of 1,200,000 counts, 6-1/2 (6.08) digits, 0.0001 %, 1 ppm, and 21 bits are all roughly equivalent. In contrast, a typical handheld digital multimeter has 3-1/2 or 4-1/2 digits of resolution with the typical upper limit being 50,000 counts. Thus, the resolution of this meter is roughly 30 times better than the best handheld digital multimeter. Typical computer-based data acquisition systems have resolutions ranging from about 8 to 18 bits with values between 12 and 16 bits predominant. Thus, this meter is roughly 10 times better than the best analog-to-digital conversion board and at least 30 times better than the typical board.

The resolution of this instrument can be set at 4-1/2 digits, 5-1/2 digits (the default), or 6-1/2 digits. Lower resolution is used to improve readability or for faster scanning when remote data transfer is used.

(c)   Accuracy is far more important than resolution; however, accuracy depends on many factors external to the instrument itself. Accuracy is typically given as the sum of two terms: (i) relative error often expressed as a percent of reading and (ii) absolute error expressed either as a fixed value or as a percent of range. A trait of nearly all digital multimeters is that highest accuracy is realized for DC voltage measurement because additional factors contribute to error for the other measurements. The accuracy of this instrument for DC voltage measurement may be generally rated at 0.005 % of reading (relative error) plus 0.001 % of full scale (absolute error). This accuracy is roughly 10 times better than the best handheld multimeter.

(d) The following measurements are all possible with the HP 34401A multimeter.

(i) DC voltage in five ranges (100 mVDC, 1 VDC, 10 VDC, 100 VDC, and 1000 VDC),

(ii) two-wire resistance measurement in seven ranges (100 Ω, 1 kΩ, 10 kΩ, 100 kΩ, 1 MΩ, 10 MΩ, and 100 MΩ),

(iii) four-wire resistance measurement, a technique to eliminate lead resistance effects in small resistance measurements, over the same seven resistance ranges,

(iv) DC current measurement in four ranges (10 mA, 100 mA, 1 A, 3 A),

(v) continuity test that produces an audible beep sound when the resistance measured drops below a specified low value, (used for testing circuit connections without having to look at the displayed resistance reading)

(vi) diode voltage drop with a fixed 1 mA test current and 1 V range with beep for values between 0.3 and 0.8 V,

(vii) RMS AC voltage in five ranges (100 mVAC, 1 VAC, 10 VAC, 100 VAC, and 750 VAC),

(viii) RMS AC current in four ranges (10 mA, 100 mA, 1 A, 3 A),

(ix) frequency from 3 Hz to 300 kHz with an input signal between 100 mVAC and 750 VAC, and

(x) period from 0.33 µs to 0.33 s with an input signal between 100 mVAC and 750 VAC.

Appendix B. Oscilloscope

The oscilloscope displays voltage waveforms allowing one to easily detect glitches and other waveform anomalies that are virtually impossible to identify with other laboratory instruments. The displayed pattern on the screen of the oscilloscope is known as the *trace*. An oscilloscope can also be used to make AC and DC voltage measurements although typically not as accurately as with a digital multimeter. (A digital oscilloscope like the one in the laboratory combines the elements of both the traditional oscilloscope and the DMM although the accuracy of the DMM function is less that of the dedicated DMM in the laboratory.) Our discussion of oscilloscope principles is divided into three broad areas: (a) horizontal sweep rate and time-base operation, (b) vertical voltage scale, and (c) time-base triggering.

B.1 Horizontal Sweep Rate and Time-base Operation

To display transient waveforms (horizontal mode = main or delayed), time is shown on the horizontal axis and voltage is shown on the vertical axis of the oscilloscope screen. The horizontal scale is calibrated in units of time per screen division, and this value is known as the *time base* or *horizontal sweep rate*. The typical oscilloscope screen is divided into 10 horizontal and 10 vertical divisions. Thus, a sweep rate of 2 ms per division means one division corresponds to 2 ms, and the maximum time slice of a continuous waveform that can be displayed is 20 ms at this sweep rate.

A modern, digital oscilloscope has two functional units. One is devoted to measuring the voltage of the input signal at a given sample interval and storing the values in memory. The other is devoted to displaying the latest time slice of voltage readings stored in memory.

If the scope samples its data without some kind of synchronization the displayed waveforms will start at random voltage locations making the display very hard to read or even useless. Triggering remedies this problem for a periodic waveform by synchronizing the time slices so that they form what appears to be a nearly steady pattern on the screen. Even though the trace may show some jitter, it is generally stable enough for analysis by a human observer. This issue is more fully discussed in Section B.3.

The Agilent DSO6012A oscilloscope at your station also has a "roll" mode (Main/Delayed Menu = Roll) in which voltage measurements are continuously shifted from right to left so that the most recent segment of the waveform is always displayed. This mode tracks the waveform continuously much like a chart recorder, a device that records voltage on paper from a long roll (thus the name "roll" mode). The paper moves at a fixed rate while a pen moves back and forth across the width of the paper marking the instantaneous voltage and thus providing a continuous record of the voltage. Roll mode on the oscilloscope at your station works much the same way except that only that portion of the waveform that fits on the screen can be viewed.

The time base can be set at discrete values such as 2 ms/div, 5 ms/div, and so on. It is not uncommon to have 25 or more such discrete settings ranging from 1 µs/div or less to 10 s/div or more. In addition to these discrete settings, the sweep rate can also be varied between these discrete settings. The oscilloscope at your station uses a knob to adjust this sweep rate. The knob normally varies the time-base setting in large steps; then, in "vernier mode", the same knob varies the time-base setting in smaller steps between the large-step values.

The above discussion assumes that the internal time base drives the horizontal sweep. In "XY" mode (horizontal mode = XY), the Channel 1 or X input defines the horizontal scale, and the Channel 2 or Y input defines the vertical scale. XY mode is used to display the phase shift between two sinusoids with a Lissajous pattern in this experiment.

B.2 Vertical Deflection and Voltage Scale

The oscilloscope at your station has two input channels. As noted above, both channels can be displayed on the vertical axis with time on the horizontal axis, or else Channel 1 or X can drive the horizontal axis and Channel 2 or Y can drive the vertical axis (XY mode). Each channel may be AC coupled or DC coupled. *AC coupling* means that only the AC component of the signal is displayed; the DC portion is filtered out. *DC coupling* means that *both*

the AC and DC components are displayed. *Coupling only affects whether the DC component of the waveform is displayed. The AC component is displayed in both modes.*

Each channel also has a position knob that allows the waveform to be moved up or down on the screen for improved viewing. The knob changes the vertical position corresponding to 0 V. The oscilloscope at your station automatically shows the zero point ("⊉") along the left-hand edge of the screen.

The oscilloscope at your station uses a high-speed analog-to-digital converter to realize the above noted oscilloscope functions. Because discrete values are stored in memory, an on-screen cursor can be used to read out voltages and times. This data can be transferred to a computer in digital form for further processing or printing. If the voltage falls outside the range of the measurement, an overrange reading is recorded, and no trace is displayed for that portion of the waveform.

B.3 Time-base Triggering

Triggering establishes t = 0 for the horizontal time base. Triggering options provide flexibility in displaying different waveforms. Important considerations here are the (a) trigger source and coupling, (b) trigger level, (c) trigger slope, and (d) trigger delay. Trigger source simply determines which signal generates trigger events. Trigger source options include (a) one of the two input signals ("1" or "2'), (b) a separate trigger signal "external", or (c) the AC power line ("line"). Trigger coupling can be AC or DC. As before, AC coupling means that the DC component of the trigger signal is filtered out. The trigger level sets the voltage at which potential trigger events are produced. Trigger slope specifies whether positive-going or negative-going transitions generate trigger events. Trigger delay is the time delay between the occurrence of a trigger event and the start of the horizontal sweep.

The oscilloscope at your station also has a special feature known as "trigger holdoff". Trigger holdoff keeps the trigger from re-arming for a user-specified length of time ranging from 200 ns to 13.5 s.

With this background on basic terminology, we examine some of the more common situations to see what type of triggering works best.

(a) Periodic Waveforms

For periodic waveforms, the source is almost always the waveform itself. Successive time slices of the waveform are superimposed to produce an apparently continuous pattern on the screen. "Normal" triggering ensures that successive slices lie on top of one another by starting each trace at the same point on the waveform as shown in Fig. B.1.

Starting at the left and moving from left to right in the direction of increasing time, we see that a trigger event occurs when the voltage crosses the trigger level in the positive going direction. This point is indicated by the "X" in Fig. B.1. After a user-specified trigger delay (which is usually zero), the horizontal sweep starts. The linear motion of the trace across the screen is indicated by the saw-tooth waveform shown below the input waveform. The second crossing of the trigger level (indicated by ●) is ignored because the trace is still active. The trace is still active for the third trace as well. The slope is wrong for the fourth crossing even though the trace is no longer active. Finally, the fifth crossing has the correct slope, and the trace is no longer active so a trigger event is generated. This process is repeated continuously, and the successive time slices are superimposed to produce a continuous, ideally stationary pattern on the screen.
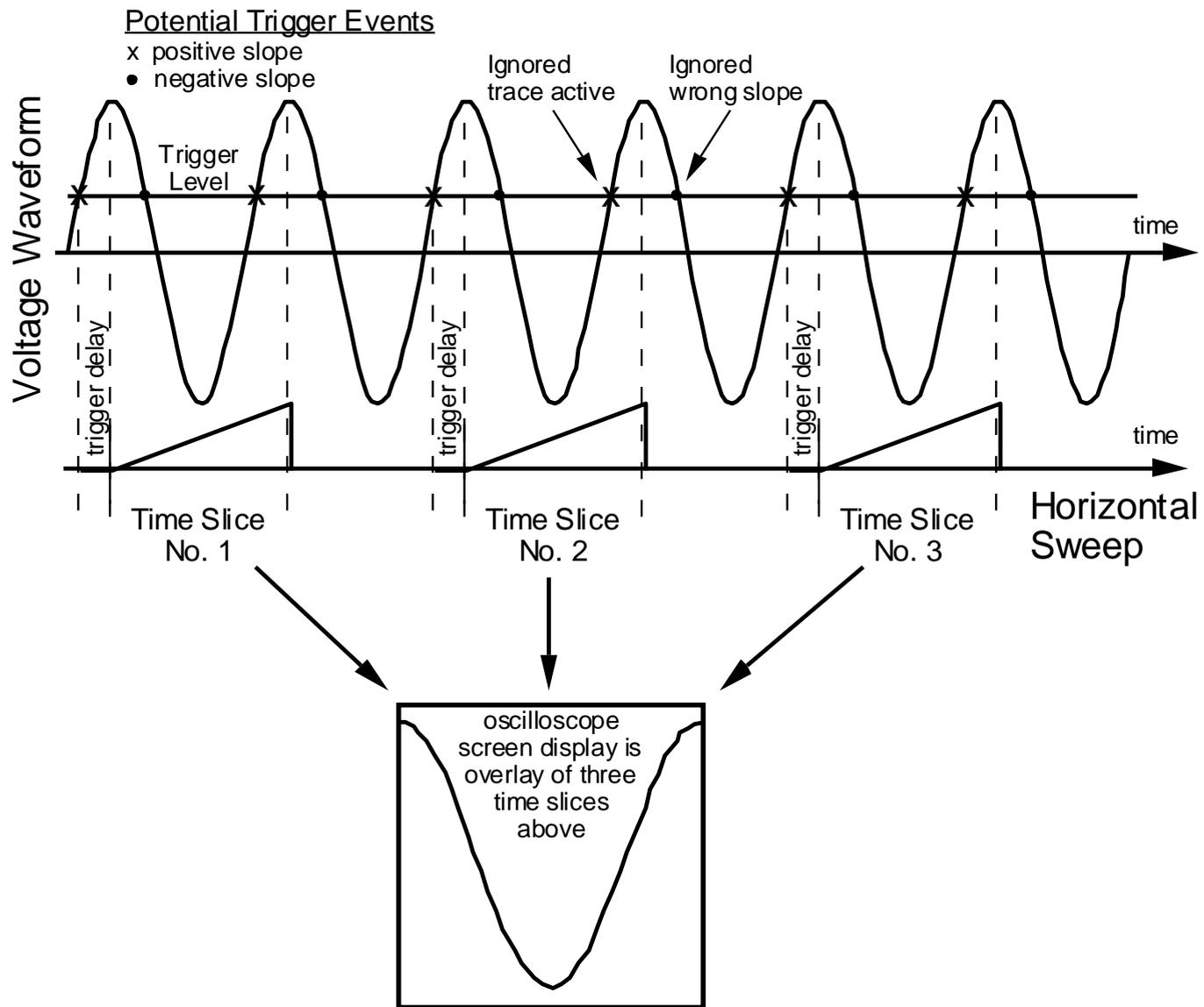
Figure B.1   Illustration of how time-base triggering produces a steady pattern on the oscilloscope screen.

The effect of trigger slope is shown In Fig. B.2. The trigger level is the same in both displays, and the time base is set so that one full period of waveform is displayed. The only difference is that the display on the left is triggered by a positive-going transition whereas the display on the right is triggered by a negative-going transition.

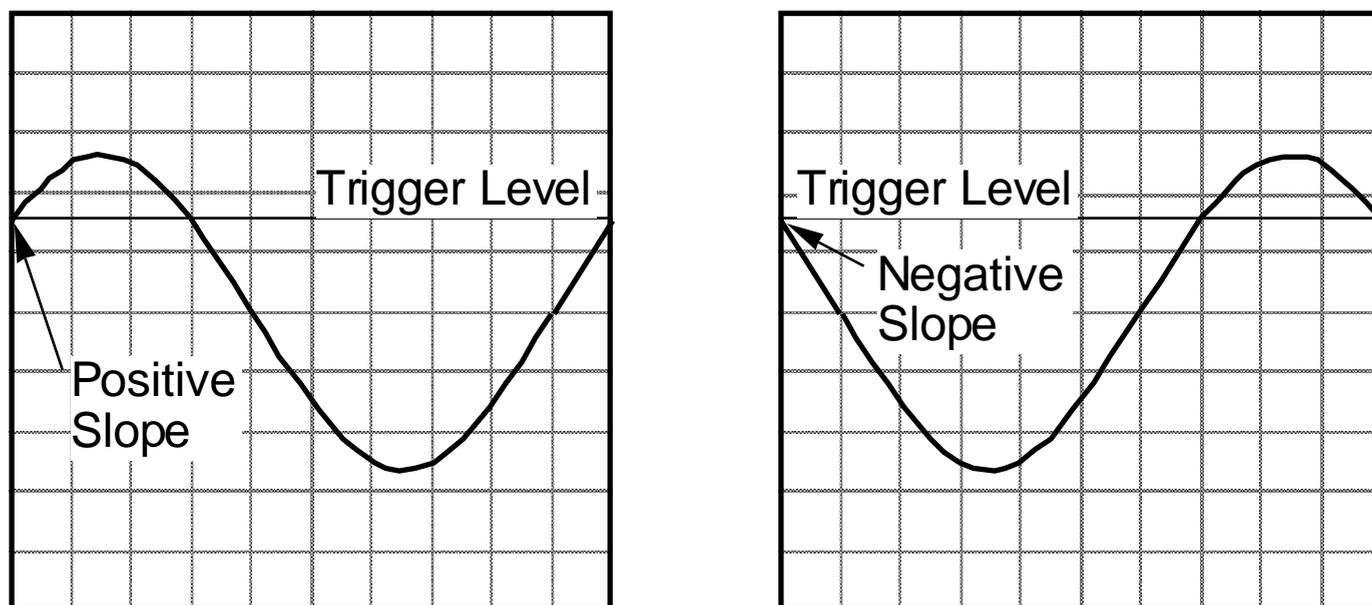Figure B.2   Effect of trigger slope on waveform display.

The Agilent DSO6012A oscilloscope displays the last digitized waveform until the next trigger event occurs so that rare events are not lost. This behavior can be potentially confusing when the trigger level is too high or too low. One may mistake the last captured waveform for a valid instantaneous waveform. The totally static nature of

the image on the screen is the first clue that an incorrect trigger level is being used. The trigger-mode indicator in the upper right-hand corner of the display also flashes. The "Display" then "Clear Display" buttons manually erase the screen and thus remove the captured waveform.

(b)   DC Signals

DC signals do not produce trigger events and thus cannot trigger the horizontal trace by themselves. However, because superposition of successive traces is not an issue, any means of triggering the trace is satisfactory. The 60 Hz, AC, power-line voltage provides a convenient source of trigger events for this case. This method of triggering the time base is called "line triggering". If line triggering is used for periodic waveforms, the result will be an incoherent parade of random time slices across the screen known as "garbage".

Both AC and DC waveforms can be viewed in "auto" triggering mode. Auto triggering automatically generates a trigger event if the trigger signal does not generate a trigger event within a certain length of time.

(c)   Single-event Triggering

Single-event triggering is used for nonperiodic, transient waveforms. A single event is captured and stored in memory. In single-sweep mode, the time base can be started by a trigger event in the normal manner or by manually pressing a key on the oscilloscope.

Appendix C. Function Generator

A function generator produces a periodic waveform with a user selectable shape, frequency, amplitude, offset, and modulation. The output voltage is given by

$$V_{out}(t) \;=\; V_{DC} + V_{AC}(t) \;=\; V_{offset} + \frac{V_{pp}}{2} w(t) \;=\; V_{offset} + A\, w(t)$$

where

$V_{out}(t)$ = generator output voltage [V],

V_{DC} = V_{offset} = DC component of voltage waveform or offset voltage [V],

$V_{AC}(t)$ = AC component of voltage waveform [V],

$V_{p\text{-}p}$ = peak-to-peak AC voltage [V] = 2 A,

A = amplitude of AC voltage [V] = $V_{p\text{-}p}$ / 2,

$w(t)$ = periodic waveform normalized so that w(t + T) = w(t), min w(t) = -1, and

max w(t) = +1 for all t,

T = period of waveform [s], and

f = 1/T = frequency of waveform [Hz].

A few important terms are further explained below.

Waveform

The function generator at your station can produce the following standard waveform shapes as well as a user-specified "arbitrary" waveform.

Standard Waveform Shapes: sinusoidal   square wave   triangular   ramp

Frequency

Frequency is the rate at which the periodic waveform repeats. Frequency f is expressed in Hz = 1/(seconds) and is the inverse of the waveform period T.

Amplitude

Amplitude can be expressed in terms of the peak-to-peak voltage $V_{p\text{-}p}$, the single-sided amplitude A, or as a root-mean-square (RMS) value V_{rms}.

DC Offset

A periodic waveform may have a constant (DC) offset in addition to the time-varying (AC) component.

Modulation

Modulation means that the amplitude, or frequency, or both are varied by a second waveform. Amplitude modulation is commonly abbreviated AM as in AM radio stations. Similarly, frequency modulation is commonly abbreviated FM as in FM radio stations. Other forms of modulation such as phase modulation are used in communications and other industries. Our use of the function generator is limited in scope and does not involve modulation.

Appendix D.   Resistance Coding Schemes

EIA Color-band Resistance Coding Scheme

Band 1 = First Digit
Band 2 = Second Digit
Band 3 = Multiplier
Band 4 = Tolerance
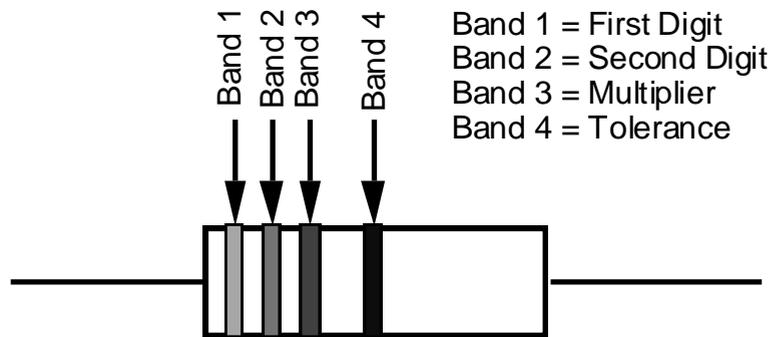
Example
Brown = 1
Black = 0
Red = 100
Red = 2 %

$1000\ \Omega \pm 20\ \Omega$

Resistance Color Code

Digit	Color	Multiplier	No. of Zeros
	Silver	0.01	-2
	Gold	0.1	-1
0	Black	1	0
1	Brown	10	1
2	Red	100	2
3	Orange	1K	3
4	Yellow	10K	4
5	Green	100K	5
6	Blue	1M	6
7	Violet	10M	7
8	Gray		
9	White		

Tolerance Code

Color	Tolerance
None	20 %
Silver	10 %
Gold	5 %
Red	2 %
Brown	1 %
Green	0.5 %
Blue	0.25 %
Violet	0.10 %
Gray	0.05 %

(a)   The nominal resistance code consists of two or more bands, the typical number being four for standard resistors with 2-% and 5-% tolerance and five for so-called precision resistors with 1-% tolerance. The last band, which may be set slightly apart from the rest, generally signifies resistance tolerance. The next-to-last band is the multiplier and gives the number of zeros following the digits. Occasionally, extra bands after the tolerance band give characteristics like the temperature coefficient of resistance.

(b)   Resistance values are usually restricted to the digit combinations given below followed by an appropriate multiplier.

Tolerance	Standard Digit Values
10 % and 20 %	10, 12, 15, 18, 22, 27, 33, 39, 47, 56, 68, 82
2 % and 5 %	10, 11, 12, 13, 15, 16, 18, 20, 22, 24, 27, 30, 33, 36, 39, 43, 47, 51, 56, 62, 68, 75, 82, 91
1 %	100, 102, 105, 107, 110, 113, 115, 118, 121, 124, 127, 130, 133, 137, 140, 143, 147, 150, 154, 158, 162, 165, 169, 174, 178, 182, 187, 191, 196, 200, 205, 210, 215, 221, 226, 232, 237, 243, 249, 255, 261, 267, 274, 280, 287, 294, 301, 309, 316, 324, 332, 340, 348, 357, 365, 374, 383, 392, 402, 412, 422, 432, 442, 453, 464, 475, 487, 499, 511, 523, 536, 549, 562, 576, 590, 604, 619, 634, 649, 665, 681, 698, 715, 732, 750, 768, 787, 806, 825, 845, 866, 887, 909, 931, 953, 976

Examples:

(i) A 2-% tolerance resistor for which the first two bands are yellow and violet may be 4.7 Ω, 47 Ω, 470 Ω, 4.7 kΩ, 47 kΩ, 470 kΩ, 4.7 MΩ, 47 MΩ, or 470 MΩ depending on the third band or multiplier.

(ii) The standard resistance value nearest 500 Ω is 470 Ω in a 10-% or 20-% tolerance resistor, 510 Ω in a 2-% or 5-% tolerance resistor, and 499 Ω in a 1-% tolerance resistor.

Military Specification (Mil-Spec) Coding Scheme

(a) A resistor may alternatively be labeled with a military-type identifier like "RN55D". This specification determines the resistor type, package size, lead configuration, and temperature coefficient. In this case, "RN" signifies a fixed, precision, metal-film resistor, "55" denotes a specific package type and lead configuration, and "D" indicates a temperature coefficient of 100 ppm/°C. The two most common package codes are "55" and "60". Also, "C" denotes a temperature coefficient of 50 ppm/°C, and "E" denotes a temperature coefficient of 25 ppm/°C.

(b) The resistance value is coded with N characters with N typically being 4 or 5. Except as noted below, the first $N-1$ characters are significant digits in the resistance value, and the final character signifies the number of trailing zeros. Thus, "1003" means "100" followed by three zeros which gives "100 000" or 100 kΩ. The letter R may be used instead of a decimal point with the number of trailing zeros omitted. Thus, "49R9" denotes a 49.9 Ω resistor. The letter K may be used to denote both the decimal point and to signify that the value is in kΩ. Thus, "49K9" denotes a 49.9 kΩ resistor. A few more examples serve to illustrate this coding scheme.

R100 = 0.100 Ω 1R00 = 1.00 Ω 10R0 = 10.0 Ω 1000 = 100 Ω 1002 = 10 kΩ

10K0 = 10.0 kΩ 100K = 100 kΩ 2212 = 22.1 kΩ 51R0 = 51.0 Ω 1004 = 1.00 MΩ

(c) A resistance tolerance code may be given after the resistance value as follows: J = 5 %, F = 1 %, D = 0.5 %, C = 0.25 %, B = 0.1 %, A = 0.05 %, Q = 0.02 %, T = 0.01 %, V = 0.005 %, and Y = 0.001 %.

(d) Resistance values are typically restricted to the same standard values listed for the EIA Color-coding Scheme.