# LAB 3: SYSTEM ANALYSIS & DEBUGGING WITH OSCILLOSCOPE AND LOGIC ANALYZER

## INTRODUCTION

The purpose of this lab is to continue to gain experience with designing and testing microcontroller-based systems, and to become familiar with debug tools, including the use of an oscilloscope and a logic analyzer as microcontroller system analysis and debug tools. In the development and debugging of a microcontroller-based system, you will often have a need to examine signals being sent to and from the microcontroller to determine correct data patterns, control signal sequencing, timing, etc.

To that end you will make a few modifications of last week's program, and then perform timing analyses as these programs run on the microcontroller.

## OSCILLOSCOPE OVERVIEW

The Digilent EEBOARD contains the hardware components of several electronic system test instruments: oscilloscope, logic analyzer, waveform generator, pattern generator, and voltmeter. The user interfaces for these instruments are implemented in the *Waveforms* program used in the first lab.

EEBOARD/*Waveforms* provide a four-channel oscilloscope that is simple to use, and yet compares in nature and function to more sophisticated oscilloscopes that you may use later on in your career. The following discussion is a brief introduction to the functions available on this oscilloscope. You will be expected to learn these functions and to set up the oscilloscope for debugging system designs in this and later labs.

Documentation for the *Waveforms* instruments is accessed by clicking on "Help" in the main *Waveforms* window, and then selecting the desired instrument. It is recommended that you read the oscilloscope help information prior to using the oscilloscope in lab. Digilent also provides analog electronics tutorials and lab materials to accompany the EEBOARD on their web site, with examples showing the use of the different *Waveforms* instruments.

http://www.digilent.cc/Classroom/RealAnalog/index.cfm.

*Waveforms* can be downloaded from the Digilent site for use on your own computer. If no EEBOARD is attached, *Waveforms* runs in a "demo mode", to allow you to investigate its features.

### <u>What is a digital oscilloscope?</u>

A digital oscilloscope is not merely an oscilloscope to be used exclusively for digital circuits. The word "digital" refers to the method used to obtain signals to be viewed. Unlike analog oscilloscopes, in which input signals directly control amplifiers that drive the deflection coils of a cathode ray tube (CRT), a digital oscilloscope digitizes its input signals with an analog to digital converter (ADC). The ADC periodically samples the amplitude (voltage level) of the analog input signal and converts each sample to a binary number, storing the binary values in an internal memory. A microprocessor conditions the digital data according to the functions that you request and displays the results on a screen.

The EEBOARD data acquisition circuits convert 40M samples per second on each of four separate input channels into 8-bit values, storing up to 16K data samples in a high-speed memory (buffer), from which they can be uploaded to the attached PC and displayed in a user interface. The *Waveforms* oscilloscope instrument, illustrated in Figure 1, emulates a traditional oscilloscope front panel on your computer, with settings that can be manipulated with the PC mouse and keyboard.

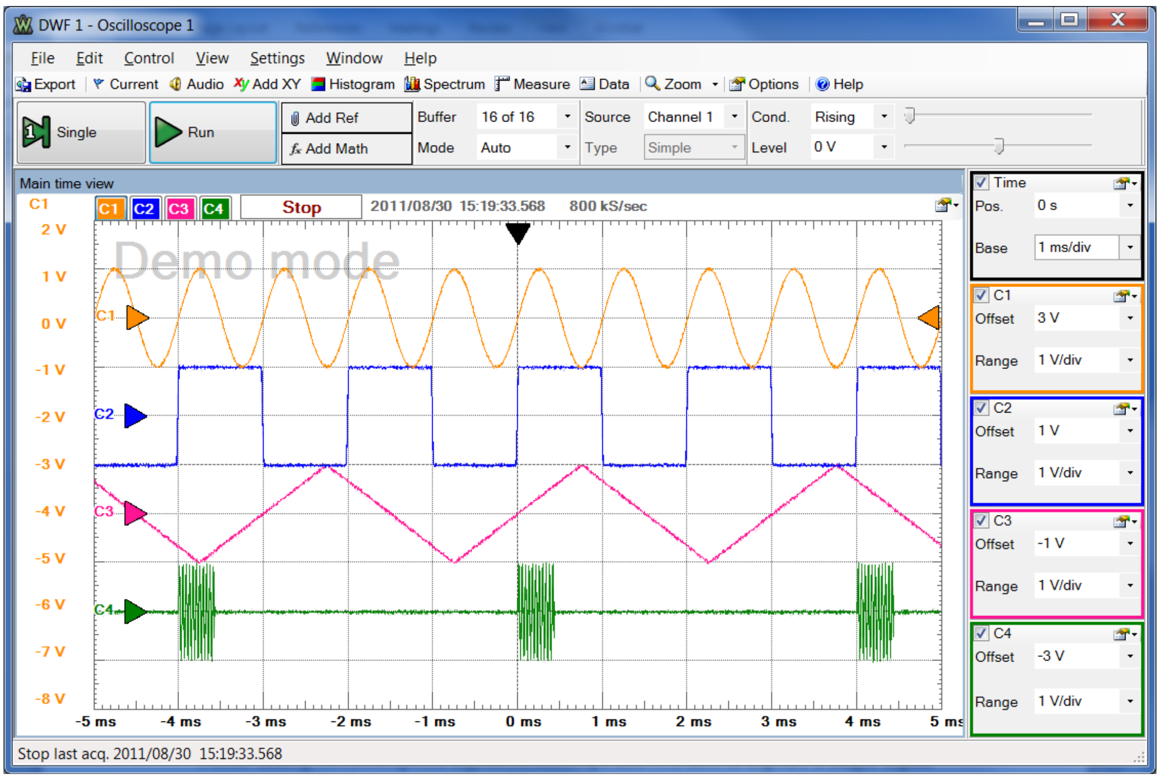

Figure 1. EEBOARD/*Waveforms* oscilloscope instrument (shown in demo mode).

Unlike analog oscilloscopes, a digital oscilloscope allows a user to capture a single signal trace, zoom in and out to view a portion or all of the captured data, use cursors to make measurements on waveforms, save waveforms to disk and recall them at a later time for analysis, save the oscilloscope setup and restore it later, and capture the display for inclusion in reports.

The following is a list of EEBOARD oscilloscope features (from EEBOARD data sheet.)

- 4 channels, 40MSa/sec sampling frequency
- 70MHz analog input stage bandwidth
- 9MΩ /10pF input impedance
- **+/-20V input range**
- AC/DC coupling
- 10 bits Analog to Digital converter
- 0.8mV to 40mV/LSB resolution
- Input protection up to +-200V
- Up to 16KSa buffer depth
- Advanced triggering(edge, pulse, transition types and hysteresis, holdoff parameters)
- Channels filtering: average, decimate, min/max
- FFT, XY, and Histogram functions
- Recording and audio functions
- Advanced data measurements for each channel and global measurements
- Export data and waveform options

## Connecting the Oscilloscope to the Circuit

Four different signals can be captured and displayed simultaneously, captured via "probes" connected between circuit nodes and the SCOPE connector (channels 1-4), shown in Figure 2. Recall that voltage is the potential difference between two points. In most cases, one of these points is a common reference (ground). Therefore, each SCOPE channel must be connected to the circuit with two wires: one between the node to be measured and one of the connectors marked "AC" (for AC-coupling) or "DC" (for DC-coupling), with the second wire connected to ground (the holes with the dash below and arrow above them.) In contrast, a traditional oscilloscope probe, pictured in Figure 3, has a probe to touch the signal to be measured and a clip to connect to ground; these are connected to the instrument with a BNC connector.

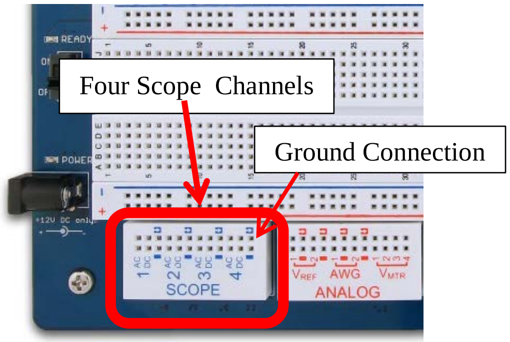


**Figure 2. Oscilloscope connections on the EEBOARD.**

**Figure 3. Typical oscilloscope probe**

## Setting Up and Using The Waveform Display

Waveforms are displayed on the oscilloscope as two-dimensional graphs of voltage (vertical axis) vs. time (horizontal axis). Controls are provided to set trigger conditions and format the axes. All displayed signals have common horizontal settings (time base), but each signal has separate vertical settings, allowing it to be displayed in a manner that best facilitates study of the signal. As shown in Figure 1, on the right side of the oscilloscope window, are four channel configuration toolboxes, labeled C1-C4, whose colors correspond to the displayed waveforms. Clicking in the small box to the left of the channel number enables and disables display of that channel, allowing removal of selected signals from the display to facilitate studying the other signals.

The SCOPE can capture signals in the range +/- 0.8 mvolts to +/- 20 volts. **Please do not attempt to measure signals that exceed +/- 20 volts to avoid damaging the EEBOARD!**

## Coupling

*AC coupling* removes any DC offset from a displayed signal before it is sampled, resulting in a signal that alternates around 0 volts. This is useful when a small amplitude signal is superimposed on a DC level. Removing the DC level allows the “interesting” part of the signal to be expanded to fill the display.

*DC coupling* results in the signal being measured and displayed in its entirety. This is preferred for digital signals, which vary between voltages corresponding to logic 0 and logic 1. While the coupling mode for each channel is user-selectable via the front panel on most oscilloscopes, the EEBOARD has separate connectors in the PROBE block, labeled ***AC*** and ***DC***. Each signal must be connected with a wire to either the ***AC*** or a ***DC*** connector, to use the corresponding coupling mode.

## Triggering

A waveform is produced on an analog oscilloscope by sweeping an electron beam horizontally across the screen at a fixed rate, while deflecting the beam vertically with the acquired signal. A digital oscilloscope appears to do likewise, although a “sweep” is generated from acquired data samples stored in its memory. A sweep begins when the oscilloscope is “triggered” by some user-specified event. A trigger signal can be derived from one of the signals being measured or from a separate trigger input. In *Waveforms*, a trigger condition is defined by the following drop-down lists in the Trigger toolbar immediately above the signal display area.

- ***Mode*** – Selects the type of triggering.
  - **Normal:** the acquisition is triggered only on the specified condition. The oscilloscope only sweeps if the input signal reaches the set trigger point.
  - **Auto:** when the trigger condition does not appear in approximately 2 seconds (configurable in Options), the acquisition is started automatically.
  - **None:** the acquisition is started immediately, without a trigger.

- ***Source*** **–** selects the source of the trigger signal, normally one of the signals acquired on channels 1-4.

- ***Type*** **–** the type of trigger event to be detected. In this lab, the most common option is "**Edge"**, which causes acquisition to be triggered by a rising or falling edge of the source signal.

- ***Condition (Cond.)*** **–** select rising or falling edge of the source signal to activate the trigger

- ***Level*** **–** select the voltage level at which to activate the trigger while the signal is rising/falling.

  For example, to capture a digital signal that switches between 0V and 5V, one might trigger on the rising edge of the signal as it passes through the 50% point of 2.5V.

## Horizontal Display Settings

Referring to Figure 1, the "Time" toolbar in the top right section of the *Oscilloscope* window contains two control settings, "Base" and "Pos", which configure the horizontal waveform display. These settings are common to all four channels, so that the user can determine and compare signal values within a common time frame.

### Base ("Time Base" or Time/Division)

The time base (time/division) is the amount of time represented by each horizontal division on the display grid, and thus determines the time duration of the displayed signal. In Figure 1, the time base is set to 1ms/div. This is reflected in the times displayed along the bottom of the screen (the x axis). All displayed waveforms have a common time base, allowing values to be compared at specific points in time. A time base should be selected that allows "interesting features" of a waveform to be conveniently studied. For example, channels C1, C2 and C3 can be easily examined in Figure 1, but one might wish to use a smaller time base to "spread out" (zoom in on) one of the sinusoidal bursts displayed on channel C4.

### Position (Pos)

The position setting controls the horizontal position of the trigger condition (time 0), relative to the center of the display area. An analog oscilloscope begins plotting signals on the screen, from left to right, when a specified "trigger condition" is detected. A digital oscilloscope, however, samples signals continuously, saving the samples in memory. Thus a digital oscilloscope can display both pre-trigger and post-trigger data. Changing the trigger position on the screen allows the user to conveniently focus on signals before and/or after the trigger. The trigger position can be changed in the drop down list in the Time toolbox, or by dragging the black triangle found above the time 0 position.

## Vertical Display Settings (Channel Configuration)

The channel configuration toolbar contains boxes labeled C1, C2, C3 and C4, which each contain two settings that configure the vertical display of their respective waveforms. Clicking the box next to the channel number displays or removes that channel from the screen.

### Range (Volts/Division)

The amplitude of a signal is indicated by its vertical height on the display and the number of volts represented by each division on the display grid. The "range" (volts per division) for each channel is selected from its drop-down ***Range*** box. The y axis of the display is labeled with voltages for one selected channel. In Figure 1, the displayed voltages are for Channel 1. Clicking on the colored triangle next to the channel name at the left side of the window changes the display to that channel's voltages. Each channel can have a different setting for volts per division, allowing each waveform to be displayed with the most convenient setting to facilitate analysis. **Again, note that the maximum input range is +/- 20 volts!**

### Offset (Vertical Position)

For convenience, a waveform can be shifted up or down on the display by changing the "vertical offset" from the drop-down list labeled ***Offset***. The offset can also be changed by dragging up or down the colored triangle next to the channel name at the left of the display. For example, one might want to compare two waveforms by shifting one up or down to superimpose it on the other.

## Starting and Stopping Data Acquisition and Saving/Printing Waveforms

### Run/Stop and Single Trace Operation

In the top left portion of the oscilloscope window is the Acquisition Toolbar, which contains the ***Run/Stop*** and ***Single*** buttons. The oscilloscope can be triggered one time or repetitively. Clicking on the Single button causes a single waveform to be acquired and displayed when the triggering condition is met. Then then data acquisition stops until the user again clicks on either ***Single*** or ***Run***. Performing a single acquisition allows aperiodic signals to be studied (one-time events, perhaps). "Normal" (repetitive) triggering, initiated and halted by the ***Run*** button, operates continuously, producing a new sweep every time the trigger condition is met. This works best when the acquired signal is periodic, otherwise a different waveform will be displayed on each sweep, resulting in what appears to be an unstable signal.

### Saving/Restoring Data and Configuration Settings

Once you have a good setup for your oscilloscope, you may want to save it and then restore it in a later lab. From the menu bar, select ***File > Save Oscilloscope Project*** to save your settings, and ***File > Save Acquisition Data*** to save the current data. You will be prompted for a file name to save/restore. **(You should save these on your H: drive.)** Select ***File > Open*** to restore saved settings and data.

You can also export data or screenshots from the *File* menu, and copy text and images via the *Edit* menu. The latter is especially useful for capturing images to include in reports.

## OVERVIEW OF LOGIC ANALYZER OPERATION

In an embedded system it is often desirable to monitor the logic states of multiple digital signals to determine if the system is operating as expected. A *logic analyzer* captures and displays signal logic states (0 and 1), rather than analog voltage levels, over a designated time interval. As with an oscilloscope, probes are connected to the signals to be monitored and the states of those signals are sampled and captured over a period of time. Sampling is performed on transitions of an internal or external clock. The state of each signal line is interpreted as 0 or 1, according to whether the sampled voltage level is below or above a threshold value. The captured states are stored in a memory, from which all or part of the data can be displayed as a timing diagram or in tabular form. Captured signals can be displayed individually or as multi-signal buses (such as microcontroller I/O ports).

The EEBOARD logic analyzer comprises 32 signal connections and a few dedicated trigger and clock connections on the Digital 1-4 blocks at the top of the board, plus the *Waveforms Logic Analyzer* instrument. Note that these connections are shared with the *Static I/O* instrument; signals can be used either or both of these instruments. The following is a list of EEBOARD logic analyzer features (from EEBOARD data sheet.) **It is recommended that you review the *Waveforms* "help" information on the logic analyzer prior to the start of lab.**

- 32 digital pins (shared with Digital Signal Generator and Static I/O)
- 100MSamples/sec
- Internal/external clock
- Up to 16K samples per pin buffer depth
- Trigger options
- Save signals values option
- Customized visualization for each signal or bus
- Tabular data visualization

The *logic analyzer* instrument displays one waveform per signal or bus, as illustrated in Figure 4. A *bus* is group of signal lines, with the displayed numeric value determined by treating the signals as bits of a binary number. In Figure 4, "Bus 0", labeled "Digital" in the waveform area, comprises the 8 digital inputs DIO 27-20, and the binary number represented by these signals is displayed as a sequence of decimal numbers. In Figure 4, the bus has been "expanded" (by clicking on the + sign to the left of the label "Bus 0") into its individual binary signals [0] – [7], where [0] is the least significant bit (LSB) of the displayed number and [7] is the most significant bit (MSB).

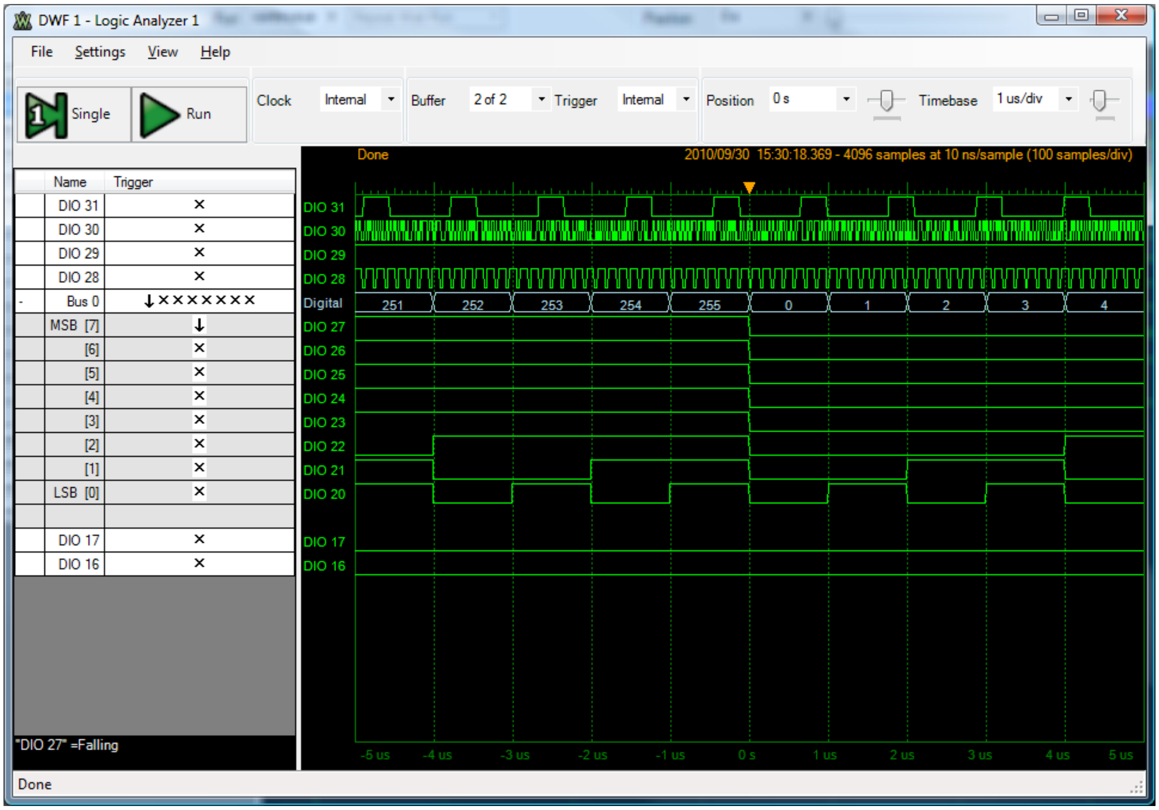

**Figure 4.** ***Waveforms Logic Analyzer*** **main window**

## <u>Connecting Signals to the Logic Analyzer</u>

To probe digital signals with the logic analyzer, connect wires between the nodes to be examined and the numbered connectors in the EEBOARD DIGITAL 1-4 blocks. Connector number K is displayed as “DIO K” in the logic analyzer window. It is also good practice to connect a ground point in your circuit to the ground connector (indicated by the dash above and arrow below the holes) on each DIGITAL block used, to ensure that the digital signal voltages are measured relative to a common ground.

If you would like to use one or more external signals to trigger data acquisitions, the EEBOARD provides four “trigger” connections (T1-T4 in DIGITAL 2-3). In addition, a clock connection (CLK in DIGITAL 4) allows a signal from your system to provide the data acquisition clock, instead of the EEBOARD internal clock.

## Triggering a Data Acquisition

In system testing, one is often interested in determining what the system was doing around the time of some specific "event". Typical events might be an interrupt request signal from a device, arrival of a particular data pattern at an input port, writing data to an output port, etc. The logic analyzer captures and stores data samples continuously, while examining the data for the occurrence of a designated event, referred to as a "trigger". A trigger defined as the states of one or more signal lines, and can be internal or external. When the logic analyzer detects a trigger condition, it displays data from the buffer, including data prior to and following the trigger. The desired trigger condition is selected from the ***Trigger*** drop-down box immediately above the display area. The most common options are the following

- **None**: Acquisition is performed immediately after pressing the *Single* or *Run* button.
- **Internal**: Acquisition is performed when the defined logic analyzer trigger condition is met.
- **Ext 1**: Acquisition is performed when External Trigger 1 (T1) line is raised
- **Ext 2**: Acquisition is performed when External Trigger 2 (T2) line is raised
- **Ext 3**: Acquisition is performed when External Trigger 3 (T3) line is raised
- **Ext 4**: Acquisition is performed when External Trigger 4 (T4) line is raised

To use Ext 1-4, an external signal must be connected to trigger inputs T1-T4 in EEBOARD blocks DIGITAL 2 and 3.

An "internal" trigger is a user-defined pattern on one or more of the 32 digital inputs. To define an internal trigger condition, you may assign any of the following values to each digital input:

x no trigger
0 0 pattern value
1 1 pattern value
↓ falling edge
↑ raising edge
↕ any edge

The overall trigger condition is the logical AND of all pattern conditions (0,1,X) combined with the logical OR of edge conditions (rising edge on DIO 1, any edge on DIO 2, etc.) For example, a trigger condition might be the occurrence of the 8-bit pattern 00000000 at an output port, or a rising edge on an interrupt request signal. The initial default condition for each input is "X" (no trigger). To change this right click on the current trigger value of that input, and select the desired trigger from the pop-up menu.

## Performing Data Acquisition

To acquire and display data, a clock source and time base must be selected, and then either the ***Single*** or ***Run*** button pressed.

### Sampling Clock

Either the 100MHz internal clock, or an external signal connected to CLK in the EEBOARD DIGITAL 4 block, may be used as the data acquisition sampling clock. One sample of each input signal is taken per clock period and stored in a buffer. The clock source is selected from the ***Clock*** drop-down box immediately above the left side of the display area.

### Time Base

The "time base" is the time per division in the display grid. This is selected via the ***Timebase*** drop-down box immediately above the right side of the display area. In Figure 4, the selected time base is 1us per division. Note that time is displayed along the x axis of the screen, relative to the position of the trigger event, with negative values indicating times prior to the trigger and positive values indicating times after the trigger. The time base can be changed to zoom in on a specific event, or zoom out to view multiple events. The position of the trigger point on the display can be changed by dragging the orange arrow immediately above the trigger point, or by selecting the desired position from the ***Position*** drop-down box immediately above the display area.

### Initiating Data Acquisition

To begin data acquisition, click the ***Run*** or ***Single*** button in the upper left corner of the screen. The "Run" option performs continuous data acquisition, periodically updating the display with new information each time the selected trigger condition is met. The "Single" option captures and displays a single set of data when the trigger condition is met, and then stops data acquisition.

### Data Visualization

To facilitate studying captured data, it may be helpful to create buses and/or assign names to signals on the display. Signal names and states are displayed in the "Signals Definition Grid" on the left side of the window. A name can be assigned to a signal by clicking in the "Name" column at the left of the screen, which puts the grid into edit mode, and then entering the desired name. (The default names are DIO 0-31).

The most efficient way to organize signals into a bus is to select those signals by left-clicking on the first one and then control-left clicking on each successive signal. Then right click on the last selected signal name, and from the pop-up menu select ***Insert > Buses > New Bus From Selected Signals***. This adds a new horizontal line to the display. For example, "Bus 0" in Figure 4 was created from DIO 20-27. Clicking the "+" to the left of the bus name expands the bus to individual signals; clicking again collapses the bus to the single row. It helps to connect signals to EEBOARD DIO inputs in the order in which you would like them displayed.

## PRE-LAB ASSIGNMENT

1. Study this lab description. If necessary, review the procedure for editing, assembling, and downloading from Labs 1 & 2, and make any necessary corrections to the program from Lab 2.
2. Modify the program from Lab 2 to continuously display on LEDs the states of the two switches being used to control the counting function. For this, connect an LED to Port T pin PT6, to display the state of the switch connected to PA0, and connect an LED to Port T pin PT7 to display the state of the switch connected to PB4. Displaying this information should not interfere with the count being displayed via PT3-PT0, and vice-versa.

## LAB PROCEDURE

***Please ensure that you record all observations and measurements in your engineering notebooks. Some of this data might be appropriate for inclusion in subsequent reports.***

The lab procedure is to load and execute the program from Lab 2, modified as described above, and capture the microcontroller inputs/outputs on the logic analyzer. Verify that the correct sequence of values is being generated, and measure the exact rate at which the count is being changed. Then, use the oscilloscope to measure the rate at which each bit of the count is being changed.

1. As specified in Table 2, use wires to connect the indicated microcontroller signals to the DIGITAL blocks on the EEBOARD.

2 Using the procedure described above, configure *Waveforms* to assign the signals listed in Table 2 to buses as follows:

   a. Create a bus to display PortT bits 0-3, connected to EEBOARD DIO 8-11.

b. Microcontroller port pins PA0, PT6, PB4 and PT7 are inputs and outputs that we want to study. These will be kept as individual signals. Assign these as indicated in Table 2.

**Table 2: Connections from Circuit to EEBOARD DIGITAL Block**

| Logic Analyzer Signal # | MC9S12C32 Signal Name (Display Name) | Signal Function |
|---|---|---|
| DIO 8 | Port T – bit 0 (PT0) | Count bit 0 |
| DIO 9 | Port T – bit 1 (PT1) | Count bit 1 |
| DIO 10 | Port T – bit 2 (PT2) | Count bit 2 |
| DIO 11 | Port T – bit 3 (PT3) | Count bit 3 |
| DIO 12 | Port A – bit 0 (PA0) | Switch 1 |
| DIO 13 | Port T – bit 6 (PT6) | LED showing switch 1 state |
| DIO 14 | Port B – bit 4 (PB4) | Switch 2 |
| DIO 15 | Port T – bit 7 (PT7) | LED showing switch 2 state |

3 In the toolbars, select appropriate values for Clock (internal), Trigger (internal), and Timebase, so that you can capture and display a complete counting sequence on Port T, with a count of 0000 defined as a trigger condition.

4 You are now ready to perform a data acquisition. Your program should be running. To start the data acquisition, click on the ***Single*** button to capture and display a single set of data.

5 You can vary the amount of information displayed by changing the Timebase box. Experiment with different values of this value, noting the effects on the displayed information. Record your observations of the results, and especially note and record the time delay between changes of the count displayed on the LEDs. This will give you some information that you can relate to the timing of the instructions executed in your "delay" function. Note that the logic analyzer sampling rate (100MHz) determines the accuracy of the resulting data, while the Time Base determines the amount and level of detail displayed in the window.

6 Open the *Waveforms Oscilloscope* instrument. Wire Port T bits 0 and 1 to the DC connections for channel 1 and 2 in the SCOPE block, and wire a ground point in your circuit to the ground connectors for channels 1 and 2. For best results, configure both channels for 2 volts per division and adjust the time base until you see a good waveform. Sketch what you see, noting signal frequency, period, and amplitude. From this information, determine the exact time duration of your "delay" function.

The oscilloscope and logic analyzer can be left connected on your project board. Keep in mind in later labs that both instruments can be beneficial to you in debugging your circuits. Feel free to "play around" with them and get a feel for what they can do for you.

**LAB REPORT (For the next schedule report.)**

1. Brief statement of the goals of the lab.

2. Describe your observations of the operation of your program, including the measured logic analyzer and oscilloscope information described above. You should especially note the measured execution time of your delay function.

3. Brief statement of what was learned/accomplished in this lab.