Information and instruction manual for the Agilent MSO-X-3024A Oscilloscope

Author Fred Aulich

May 2 2012, rev 2.21

The MSO-X-3024A (Mixed Signal Oscilloscope) *figure 1* is the most recent model oscilloscope brought out by Agilent to debug both digital and analog designs. It has 4 analog channels and 16 digital channels. There are 14 different triggering modes. This allows the user the ability to trigger on a simple edge, a pattern or on a complex serial protocol event such as I2C.

Figure 1

The object of this manual is to familiarize the user with how to use the MSO. We will go through several tutorials and show the user how to label, how to setup different trigger events and analyze the results. The MSO is broken down into two sections. These are the analog section which has 4 analog inputs and the digital section which has 16 inputs split over two 16 pin ribbon cables. Each 16 pin ribbon cable carries 8 digital signal lines and a pairing ground to make 8 digital probe leads. See figure 2 below.

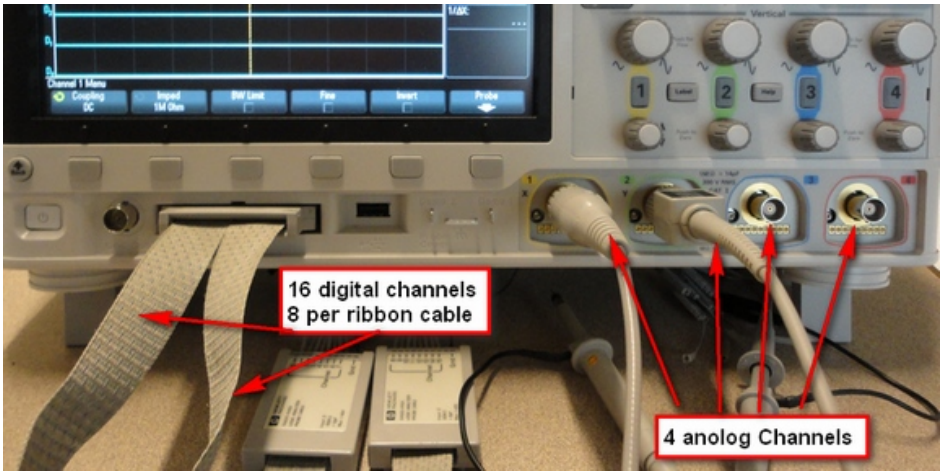


Figure 2

The control panel is divided up into 10 different sections. Each section is used to setup, change or analyze data that is captured in memory. See figure3, to see the 10 different sections.

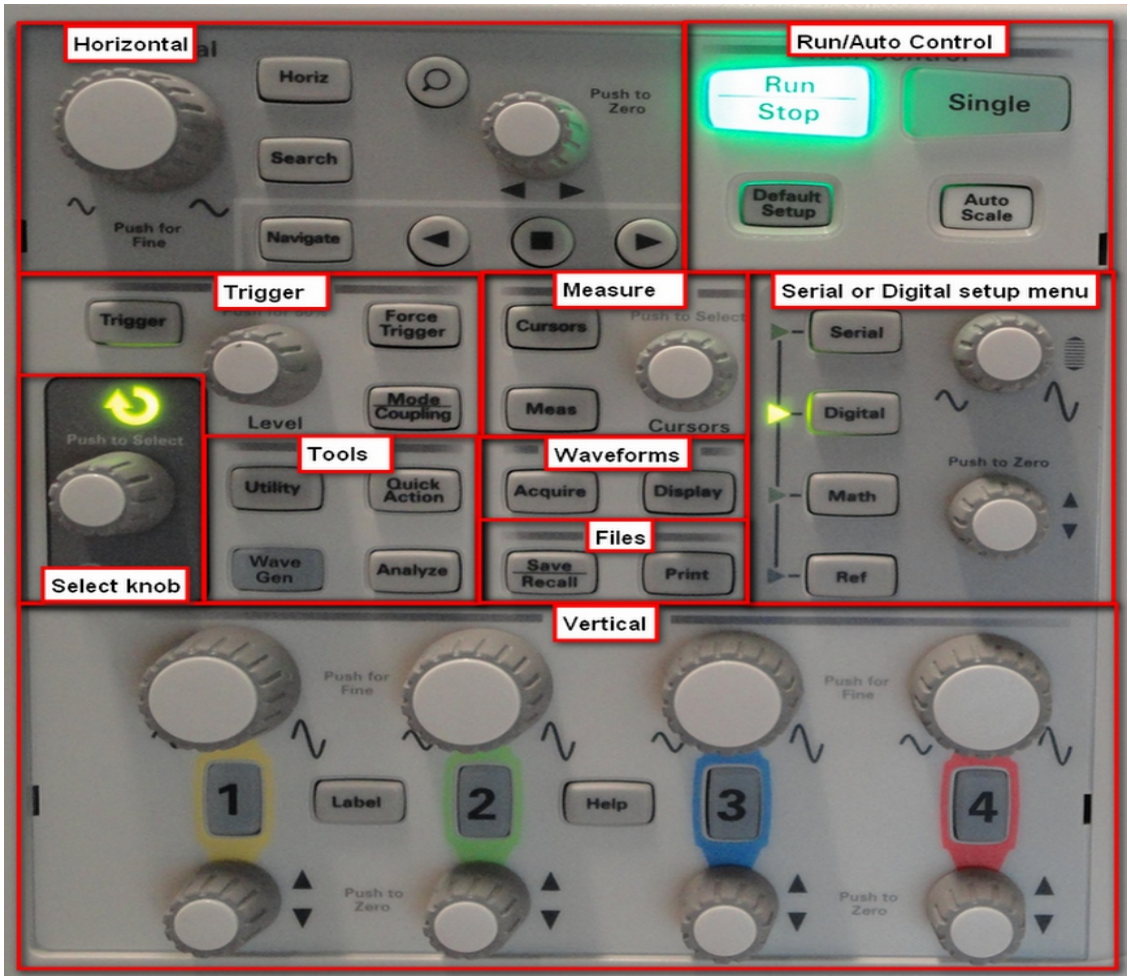


Figure 3

Each section has a different function and is labelled as follows;

1) Horizontal control
2) Run/Auto control
3) Select knob
4) Trigger select
5) Tools
6) Measure
7) Waveform
8) files
9) Serial or digital setup menu
10) Vertical

Horizontal control- 

These buttons and knobs are used to sample and scroll through saved data.

The **Horizontal knob** is used to adjust the number of samples that can be displayed on the screen. Rotating the knob clockwise lower the number of samples displayed on the screen. This will give you more detailed information about what the signal looks like. Rotating the knob counter clockwise increases the number of samples on the screen. In this case more samples are on the screen but they will not be detailed.

If you press the **Horizontal Knob** a screen will pop up that enables fine tune sampling. This means the sample rate will only increase or decrease by 1 sample per division as opposed to the default sample rate per division. Pressing the **Horizontal Knob** again will return the sampling rate back to the default rate per division. As an example, in default sample rate if the initial sample rate is 10us/division in standard mode rotating the **Horizontal Knob** clockwise would result in the following change 20us/division, then 50 us/division, then 100 us/division and then the pattern would start over again. When fine tuning is enabled and the initial sample rate is 10 us/division, if the **Horizontal Knob** is rotated counter clockwise it would increase by 0.4 to 10.4us/division, then 10.8us/division and so on. The opposite is true for clockwise. Fine tune sampling is useful when zooming in on a particular location and wanting to get more detailed information about the signals.

The **Small Knob** allows you to scroll manually through the saved data in memory. Rotating the knob clockwise moves forward through save data. Rotating the knob counter clockwise moves backward through saved data.

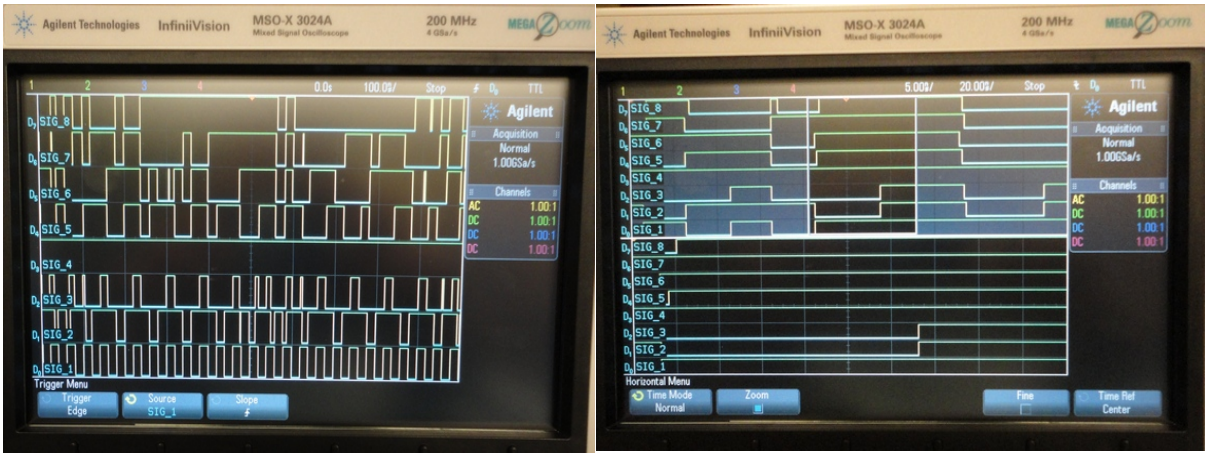
The **Arrow Buttons** are used to automatically scroll through saved data memory.

Pressing the **Forward Arrow Button** initiates forward movement through saved data. If you press the **Forward Arrow button** again it will increase the speed at which the saved data is displayed on the screen.

Pressing the **Backward Arrow Button** initiates backward movement through saved data. If you press the **Backward Arrow button** again it will increase the speed at which the saved data is displayed on the screen.

Pressing the **Stop Button** will stop movement of data at the point the button is pressed.

The **magnification button** is used to magnify a section of the displayed data sample. When the magnification button is pressed the screen is split into two, one showing the full sample and the other a portion of the sample. This allows the user the ability to capture a large amount of signal data and then use the magnifier to zoom in on a certain section of the captured signal data and get a more detailed look at the sample. See figure 4 below.



Full sample **split screen**
(full sample top portion zoom in bottom)

Figure 4

Run/Auto control- These buttons are used to determine if the user wants to display a single sample on the screen or continuously update the screen with new signal data. The **Default Setup** button is used to reset the MSO to the default manufacturing setting. The **Auto Scale** button is used to set the displayed signals to the ideal frequency to view on the screen.

Select Knob- 

This knob is used to scroll or select any menu.

Trigger select- 

These buttons and knob are used to setup and select the different trigger modes. This MSO has 14 different triggering modes. As listed in table 1;

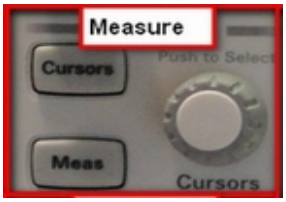
Number	Trigger mode	Description
1	Edge	Trigger on a specified rising or falling edge
2	Edge then Edge	Trigger on specified edge then on another specified edge
3	Pulse Width	Trigger on the pulse of a width which is specified by the user
4	Pattern	Trigger if signals match a pre-determined pattern specified by the user
5	Or	Trigger when all edges are the same as the specified channel (Analog or digital)
6	Fall/Rising time	Trigger on a rising or falling edge and time constant is greater than or less than a predetermined time constant.
7	N*th* edge burst	Trigger after a user specified number of edges have happened.
8	Runt	Trigger on a runt pulse that fails to exceed a high level threshold, low level threshold or both. Time constant is specified by the user.
9	Setup and hold	Triggers when clock/data setup and hold time exceeds a specified time set by the user
10	Video	Trigger on a specified line (odd or even) for video broadcasts (NTSC, PAL, SECAM, PAM-M)
11	USB	Triggers on user specified event, Start of packet, End of packet, Reset or End.
12	Serial1 I2C	Triggers on a user specified event, address with ack, address without ack, start condition, stop condition
13	Serial2 SPI	Trigger on a user specified event, start condition, stop condition or a data pattern during a specified framing period.

14	Serial3UART/RS232	Trigger on Rx or Tx start bit, stop bit or data content

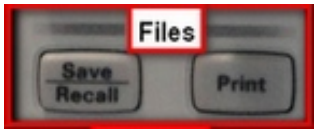
Table 1

What you are trying to sample or debug will determine which trigger mode best suits your needs. The **force trigger** button will generate a trigger and display the data on the screen at the point the button is pressed. The **mode/couple button** is used to select between auto or normal trigger mode.

Tools- These buttons have multiple functions. The **Utility** button bring up a bunch of menus for configuring I/O, file exploration, options to adjust clock, service menu for self test and hardware testing and a quick action menu to save and print data. These menus are rarely used by the user. They are mostly used by technicians servicing the MSO. The **Analyze** button is used to manually set up the trigger levels. There is a wave generator built into this MSO. When you press the **Wave Gen** button a blue light comes on. The user can select up to 11 different waves. They are as follows; Sine, Square, Ramp, Pulse, DC, Noise, Sine Cardinal, Exponential rise, Exponential fall, Cardiac and Gaussian Pulse. All these signals can have the frequency, amplitude and offset settings manually adjusted. The BNC connector labelled **Gen Out** is used to output the signal. It is located to the right of the power switch.



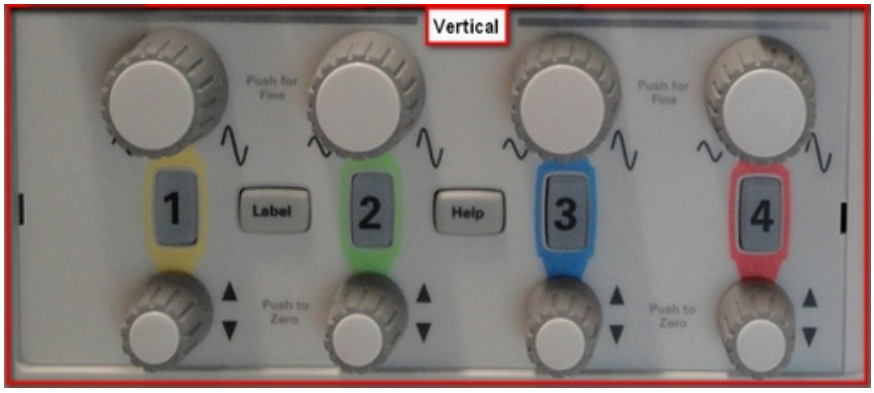
Measure- These buttons and knob are used to measure and analyze the captured data. Selecting the **Cursors** button will allow the user to measure the time or frequency of a signal or clock. The **measure** button allows the user to make measurements of a signal. The **Cursor** Knob is used to manually scroll the cursors **X1** and **X2** forward or backwards on the screen.



Files- The **Save/Recall** button is used to save or recall data from memory. The **Print** button prints what's on the screen to a printer via the Ethernet port or the RS232 port. **Note** that the print capability must be setup by the administrative staff and is currently not available.

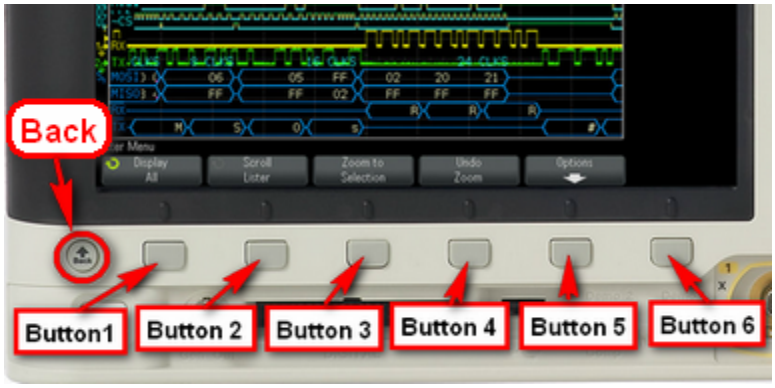
Serial or digital setup menu- 

These buttons and knobs are used to set up digital and serial signals. When the **Digital** button is pressed the user is able to select up to 16 digital channels, create a digital bus up to the number of channels selected and select the threshold from a selected list (TTL, CMOS, ECL or user specified). **Note** that you can create up to two different digital buses. When the **Serial** button is selected the user is able to select and set up to 3 different serial protocols (I2C, SPI, UART/RS232). Note that you can set up two different serial buses. The **Math** button allows the user to add, subtract, divide, multiply and perform several manipulation functions such as square root, FFT and differentiate. The **Ref** button allows the user to set the skew and make measurements .The **top small knob** is used to select any of the displayed digital channels. A selected channel will be highlighted in **RED**. The **bottom small knob** is used to move the selected channel either up or down on the screen. This becomes useful when the user wants to rearrange the signals for better viewing. **Note** it is possible to overlap two or more signals on the same line. This may happen if too many data signals are displayed at the same time.

Vertical- 

These knobs and buttons are used to select up to 4 analog inputs. The buttons labelled **1, 2, 3** and **4** are used to select and unselect the analog channels. The **large knob** is use to adjust the amplitude of the signal on each channel. The **smaller knob** is used to adjust the DC offset for each channel.

Buttons- If you look below the screen you will find 7 buttons.



The button marked **Back** is used to take the user back to the previous menu that you were at. **Buttons 1, 2, 3, 4, 5** and **6** have different meanings depending on which menu you are in. As we do the tutorials you will become more familiar with these buttons.

Probes and **Digital cable**- Two **probes** and accessories can be found in the compartment on top of the MSO. Care should be taken when taking the probes in and out of the compartment. The space is small. Make sure to take tip caps off of probe before doing tutorials or experiments and put back on before returning to the compartment. The two 16 pin **Digital cable** can be found in the lab cabinets. Ask a TA to get the digital cable out for you.

Note: Before continuing it should be mentioned that the following tutorials were created to be done in the University of Toronto DESL Engineering labs. They can be done elsewhere if you own an Altera DE2 Board and have access to an Agilent MSO-X-3024A.

Tutorial 1-Displaying Wave Gen signals

The purpose of this tutorial is to connect an analog probe to the MSO and view the different **Wave Gen** signals on the screen.

Get the following items out of the compartment at the top of the MSO, **Scope Probe to BNC Tip Adaptor, Scope Probe** and **ground clip.** See figure 5 to identify what the items look like.

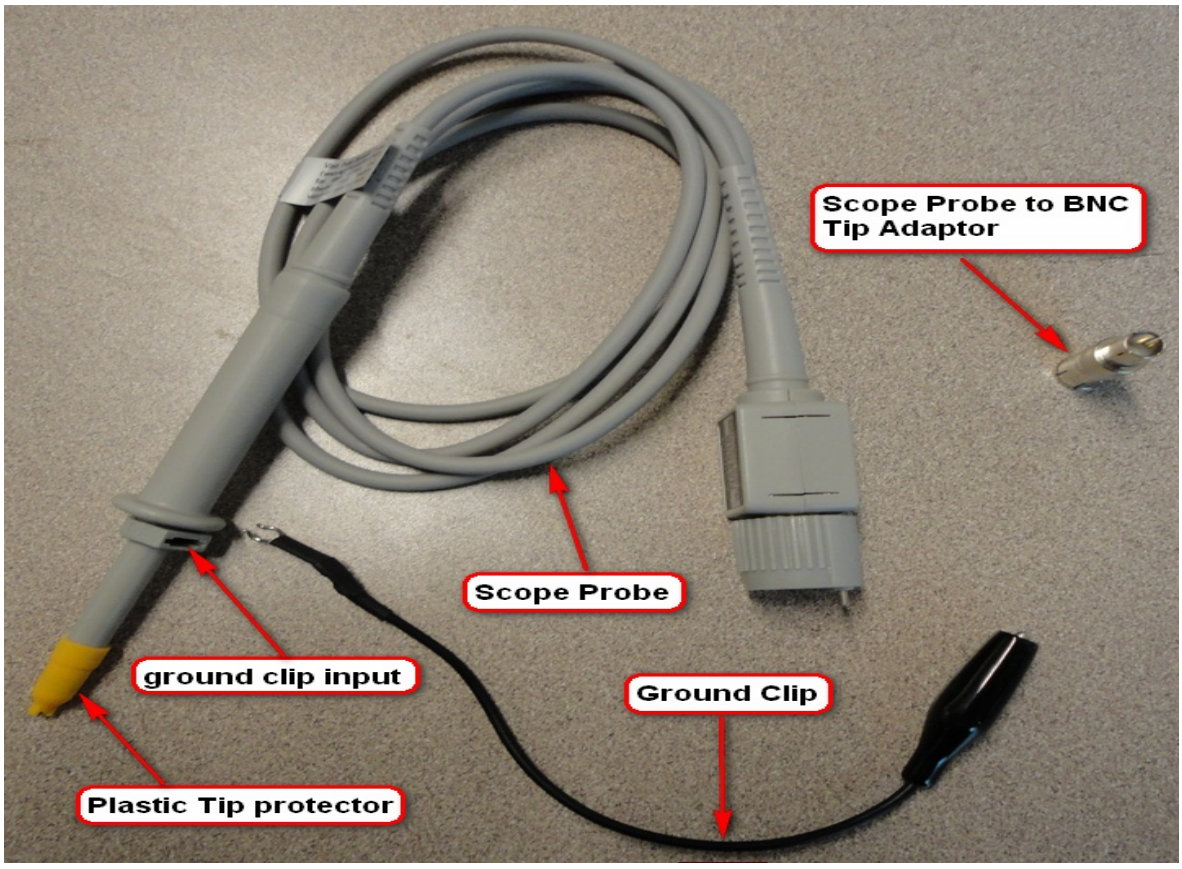


Figure 5

Remove Plastic Tip protector from **Scope Probe.** Insert the **Ground Clip** into the **ground clip input**. See figure 5 to see where the ground clip input is. Connect the **Scope Probe** to **Probe 1** input on the MSO. Insert the **Scope Probe to BNC Tip Adaptor** into the **Gen Out** BNC connector which is located to the right of the power connector. It can only be inserted one way. Insert the Scope probe tip into the **Scope Probe to BNC Tip Adaptor.** Connect the **Ground Clip** to the **Ground Lug** on the MSO**.** See figure 6 to see how the final connection assembly looks like.

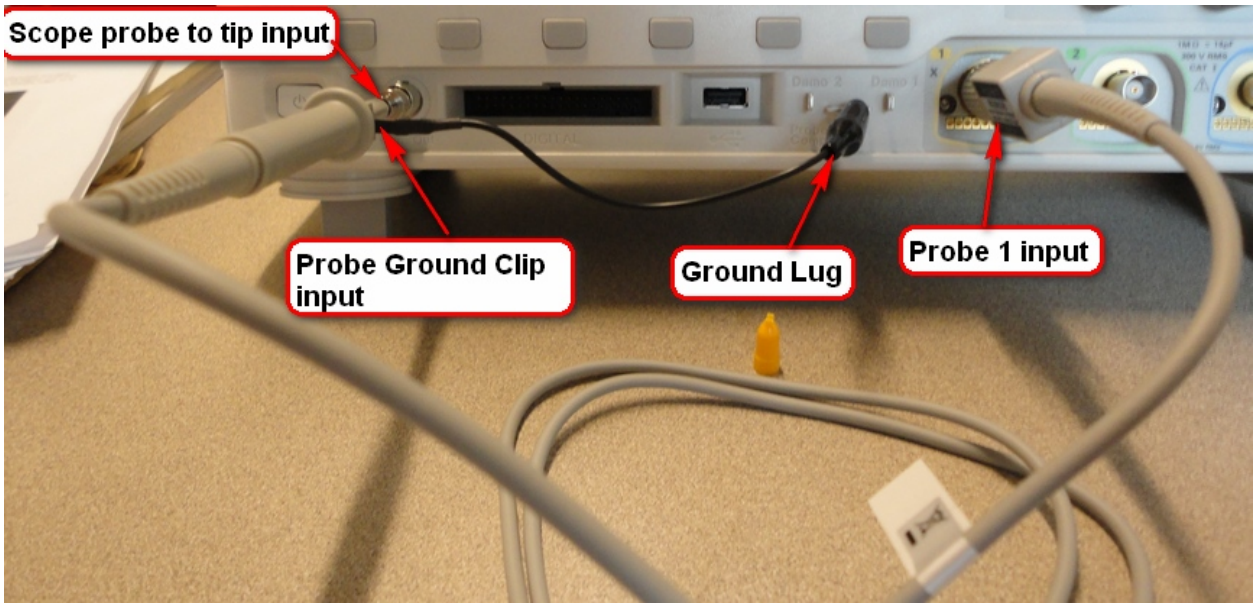
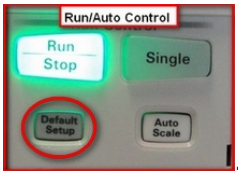
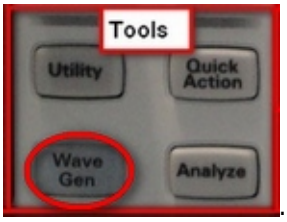


Figure 6

Press the **Default Setup** button 

.

Press the **Wave Gen** button 

. Make sure the blue light comes on.

You will notice that **Buttons 1, 2, 3, 4** and **6** have menus above them. For **Wave Gen** the menus have the following functions as described in table 1.

Button 1	Waveform - Selects the different waveforms
Button 2	Frequency/Period -Adjusts the frequency or period of the waveform. Factory default setting is frequency 1.000Khz.
Button 3	Amplitude - adjusts the amplitude of the waveform. Factory default setting is 500mVpp
Button 4	Offset- Sets the offset to a value defined by the user. The factory default setting is 0.0v
Button 5	Not used
Button 6	Settings- used to set the Wave Gen to default factory settings

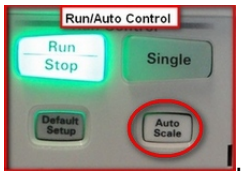
Table2

Press **Button 6** (settings).

Press **Button 6** (Default Wave Gen). This will set the Wave Gen signals all back to factory default setting.

Press the **Wave Gen** Button 

.

Press the **Auto Scale** button 

.

The result should look similar figure 7 below.

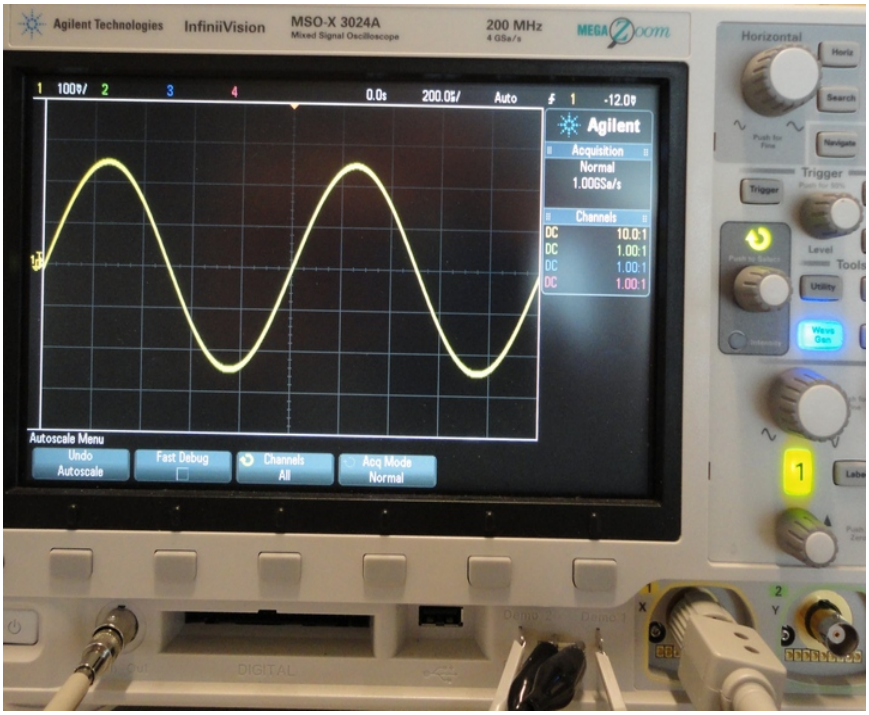
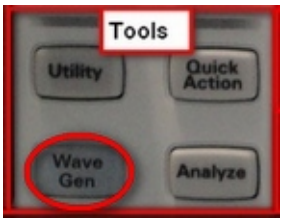


Figure 7

Press the **Wave Gen** button 

.

Press **Button 1**.As you rotate the **Select Knob** 

, a different waveform will appear on the screen.

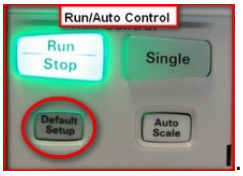
Rotate the **Horizontal Knob** 

to adjust the number of samples displayed on the screen.

You should now be able to connect and assemble an analog probe and display **Wave Gen** signals on the display. This is ends the tutorial.

Tutorial 2- Labelling and positioning

The purpose of this tutorial is to learn how to label both analog and digital channels, select, unselect and position any channel on the screen.

Before starting this tutorial we will set the MSO to the factory default setting. Press the **Default Setting** button 

.

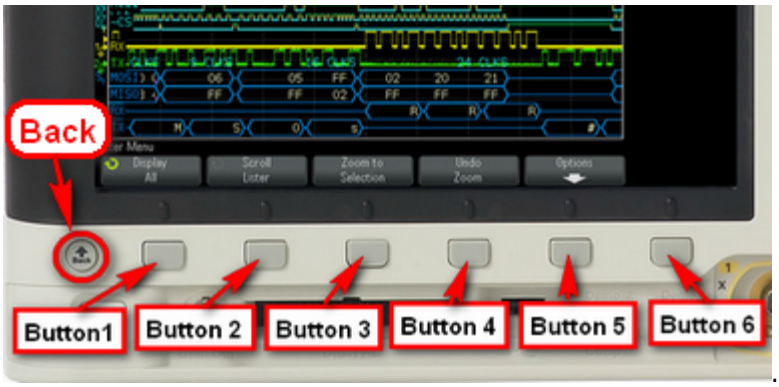
Analog Channels;

We will be changing the name of Analog **Channel 1** to **"PROBE_1"**

Press analog channel **1** button 

until the yellow light is on. This indicates that channel "**1**" is selected. All other channels should be off.

Press the **Label** button. You will notice that **Buttons 1, 2, 3, 4, 5** and **6** have menus above them. For the **Label** section, the buttons are defined by table 3;



.

Button 1	Select which channel to label
Button 2	Used to spell the new name for the selected channel
Button 3	Used to save letter, number or symbol at present location and move through label.
Button 4	Delete letter at the position in the label you are editing
Button 5	Save new label for the channel being edited
Button 6	Is a library of pre named labels that you can choose from

Table3

Press **Button 1**. Rotate the **Select Knob** 

until 1:1(yellow) is selected. Press the **Select Knob**. On the screen above scope probe channel 1 you will see the label **NEW LABEL = "NO_LBL"**. This is where the new name will be edited and saved.

Press **Button 2**, rotate the **Select Knob** until the letter "**P"** is selected**.** Press the **Select Knob**. This will save the letter and advance to the next location.

You can also press **Button 3 (enter)** to enter the letter at the present location in the label editor.

Spell out the rest of the name **"PROBE_1"**.

Once the name has been spelled out, press **Button 5** to save the new label. **Note** once **Button 5** has been pressed the newly edited name appear beside scope probe channel 1. You should also notice that in the label editor the name PROBE_1 has incremented to PROBE_2. This is helpful to speed up labelling, if you plan on creating a bus or use the same label name for all channels but increment the number to distinguish labels. The result should look like figure 8.

Figure 8

All analog channels can have their label renamed using the procedure just discussed in this tutorial.

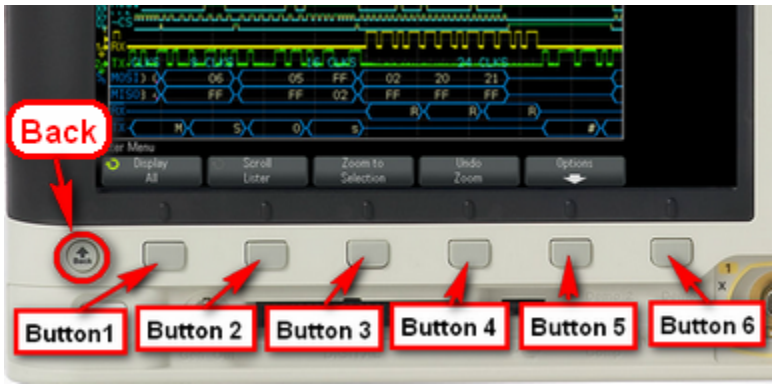
Labelling Digital Channels

Before starting the digital labelling tutorial make sure all analog channels are unselected. All analog channel (**1, 2, 3** or **4**) should not be highlighted.

Press the **Digital** button 

.

Just like the analog channels the 6 buttons have their own unique functions. They are defined in table 4.



Button 1	Scale size of signals. Depending on the number of signals being displayed it will determine which scale to use
Button 2	Individually select and unselect a digital channel
Button 3	Enable or disable digital channels 8 through 15 (group)
Button 4	Enable or disable digital channels 0 through 7 (group)
Button 5	Set threshold level for all selected digital channels
Button 6	Create a bus. Group channels to form a bus. I.E. address bus, or data bus

Table4

For this exercise we will only label the first 8 channels.

Press **Button 2** and rotate the **Select knob**. Press the **Select knob** at any channel. This will either select or unselect an individual channel. When the channel is selected the box will become blue.

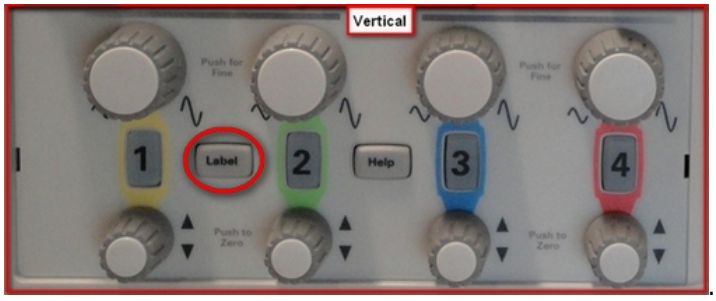
Button 3 and **Button 4** will select or unselect digital channels 0 through 15.

Each time **Button 3** is pressed, it will either select or unselect channels 8 to 15.

Each time **Button 4** is pressed, it will either select or unselect channels 0 to 7.

Make sure channel 0 to 7 are selected and channel 8 to 15 are unselected.

Continue to press **Button 1** until the **maximum scale** size has been reached.

Press **Label** button 

.

Press **Button 1**. Rotate the **Select knob** 

so each of the channels in table 5 gets selected and the label name gets changed accordingly. The procedure for labelling each digital channel is the same as was done for analog channels in the previous exercise. **Note** you must delete the present name in the editor in order to create a new label name. Also remember that once the name has been created the numerical value increments. So you only have to create the first name **SIG_0** all the other names will get generated automatically. All you need to do is select and apply the new name. All channels should be labelled as indicated in table 5.

Digital Channel 0	SIG_0
Digital Channel 1	SIG_1
Digital Channel 2	SIG_2
Digital Channel 3	SIG_3
Digital Channel 4	SIG_4
Digital Channel 5	SIG_5
Digital Channel 6	SIG_6
Digital Channel 7	SIG_7

Table 5

Once labelling is completed the result should look like figure 9.

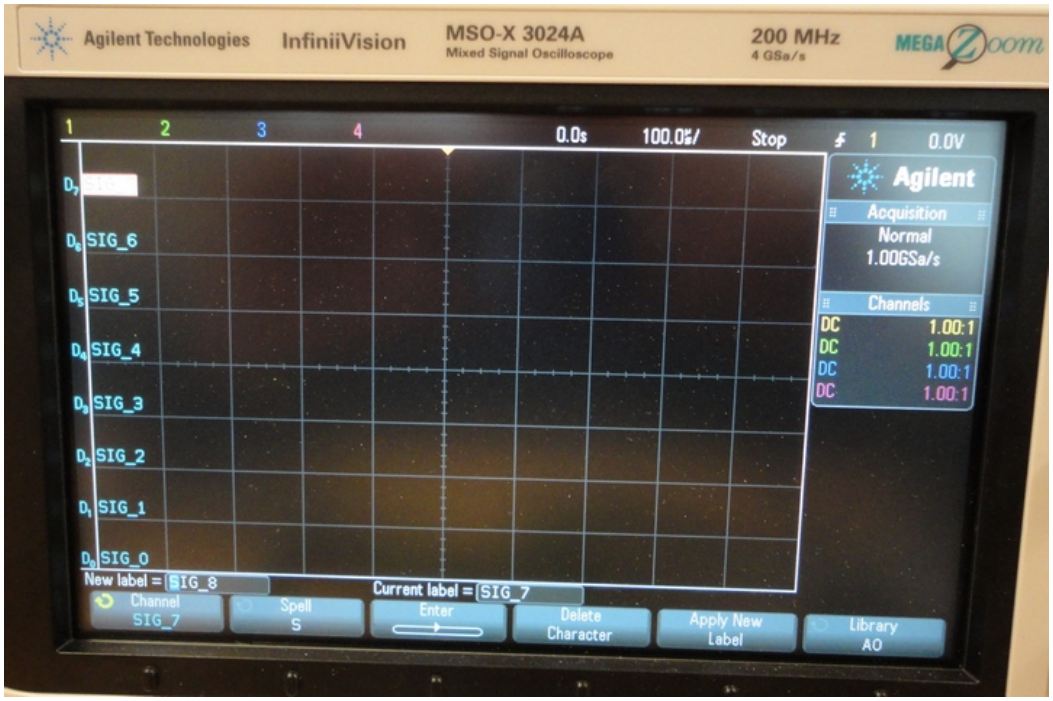


Figure 9

Press the **Label button**. Note that all the label names disappear from the screen. Press the **Label button** again; you will notice that the label names appear again. This is an option if you do not want to see the label names once they have been created.

How to select and unselect position and order digital channels on the screen

Press the **Digital** button 

.

Press **button 2.** Rotate the **Select Knob** until **D6:SIG_6** has been selected. Press the **Select Knob**.

Each time you press the **Select Knob** you will either select or unselect **D6:SIG_6**. As you do this the channel will appear or disappear from the screen. Unselect channels **D6: SIG_6** and **D7:SIG_7**.

Rotate the **top small knob** 

. Rotate the knob to the left or right. You will notice that as you rotate the knob a channel becomes highlighted in RED.

Rotate the **top small knob** so **D5:SIG_5** is highlighted in RED.

Rotate the **bottom small knob** 

. You will notice that as you rotate the knob the RED highlighted channel moves up or down on the screen depending on which way the knob is turned. **Note** that as you move the channel, you can end up having two channels on top of each other. Make sure to try and avoid this. For the purpose of this tutorial and for future ones move **D5:SIG_5** to the top most position on the screen. Align all the signals to look like figure 10 below.

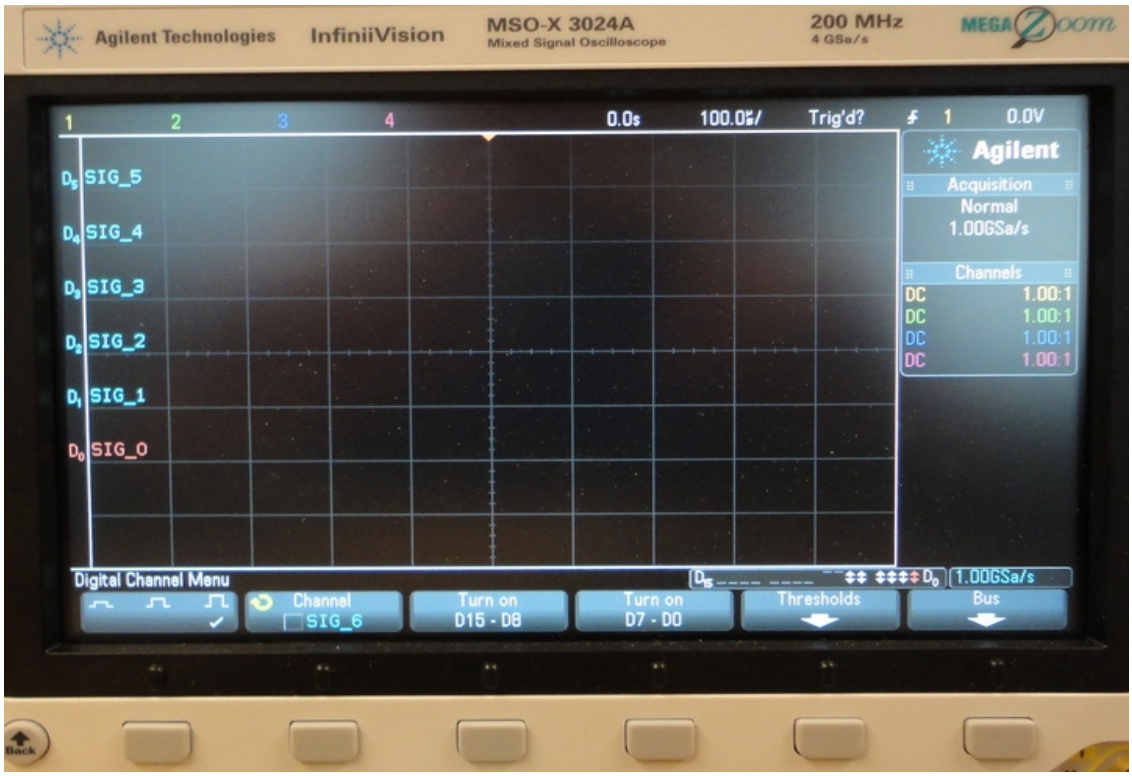


Figure 10

This concludes the tutorial 2. You should now be able to label both analog and digital channels and position channels anywhere on the screen.

Tutorial 3-Simple triggering

The purpose of this tutorial is to set up an edge trigger and display the result on the screen. Once the results have been captured we will manipulate and magnify the results.

Note-The label names for each channel, for this tutorial, have been adopted from the results of tutorial 2.

Note-For this tutorial we will be using the **Altera DE2 board,** so you need to be familiar with **Quartus** and how to download a file to the **Altera DE2 board**.

Note- Before starting this tutorial, get a digital cable from a TA or the lab manager.

Connect the 40 pin digital cable header to the 40 pin input on the MSO. Connect the digital leads to JP1 (GPIO_0) as in table 6 below. Once connected it should look like figure 11. **Make sure power is off when making these connections.**

GPIO_0-1	Channel 0
GPIO_0-2	Channel 1
GPIO_0-3	Channel 2
GPIO_0-4	Channel 3
GPIO_0-5	Channel 4
GPIO_0-6	Channel 5
GPIO_0-7	Channel 6
GPIO_0-8	Channel 7
GPIO_0-12	Ground pin

Table 6

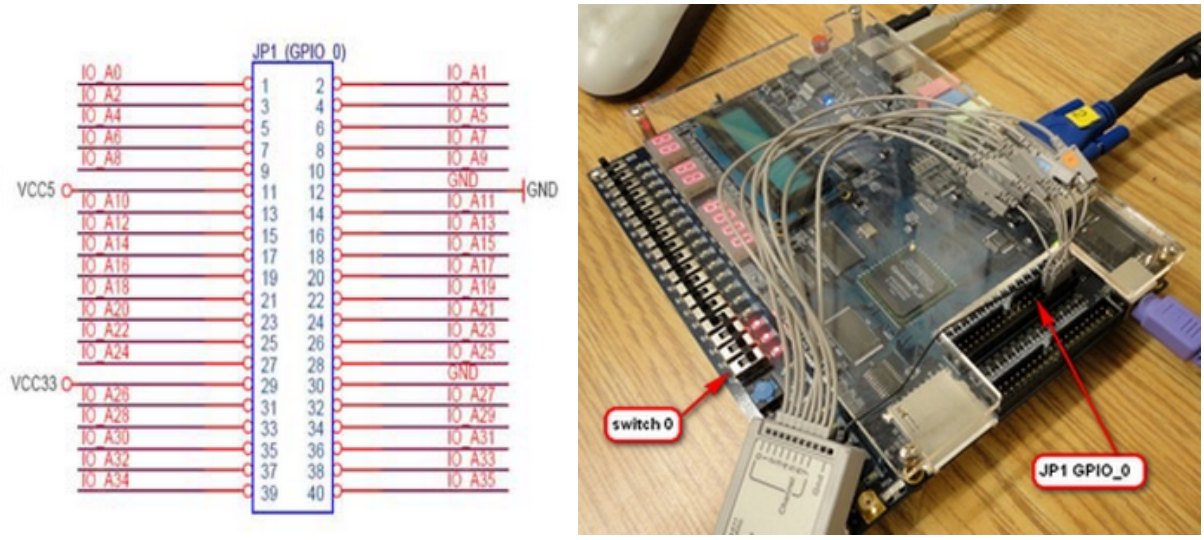


Figure11

Using internet explorer, go to the DESL home page ***www-ug.eecg.utoronto.ca/desl/index.html***.

Select -*Equipment>Test>MSO_tutorial.sof.* Save the file to your **home directory** or **desktop**.

Open ***Quartus-*** select tools>*programmer*> Add file.

Select MSO_tutorial.sof from the saved location. Download to **Altera DE2 board**.

Make sure **switch 0** and **1** on the **Altera DE2 board** are in the up position and all other switches are in the down position.

Note- **switch 0** is the right most slider switch on the **Altera DE2 board**.

Setting up for Edge Trigger

For the purpose of this part of the tutorial we will be using digital channels **0** to **5**

Press the **Trigger** button 

.

Press **Button 1.** Rotate **Select Knob** until **Edge is** selected**.** Press the **Select Knob.**

For **Edge** triggering, only 2 buttons are used as described in table 7.

Button 2	Used to select Channel
Button 3	Select Rising or Falling edge

Table 7

Press **Button 2**. Rotate the **Select Knob** until **D0_SIG_0** is selected. Press the **Select Knob.**

Press **Button 3**. Rotate the **Select Knob** until **Falling** is selected. Press the **Select Knob.**

Rotate the **Horizontal knob** 

and change the horizontal frequency of the display to **20.0 ns/division**. **Note-** If you press the **Horizontal Knob** you will enable fine tune mode. This will change the division in increments of 1 as appose to the default mode. For more information on this feature go to page 3 of this manual.

The result should look similar to figure 12 below.

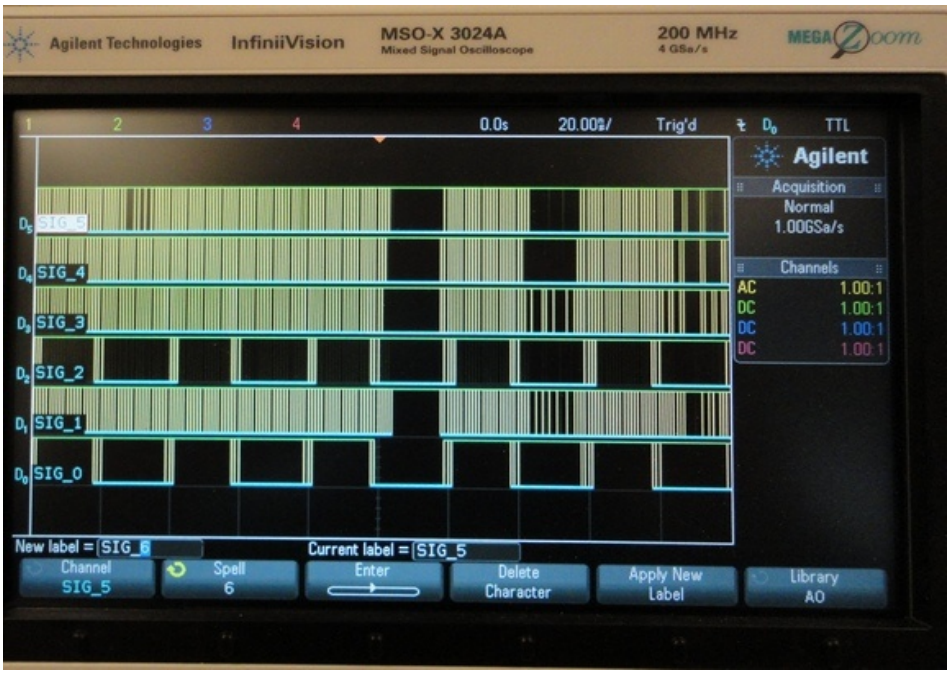


Figure 12

This gives you a continuous update on the screen of what all the signals are doing.

Press the **mode/coupling** button 

.

Press **Button 1**. Rotate the **Select Knob** until **Normal** is selected. Press the **Select Knob**. This ensures that the trigger will happen only when the trigger requirements are met.

To capture a single event press the **Single** button 

. This will give you a single event which will capture the first **falling edge** of **D0:SIG_0.** The trigger capture point is the solid orange arrow at the top of the display. It should be at the top middle of the display. The result may look similar to figure 13 below.

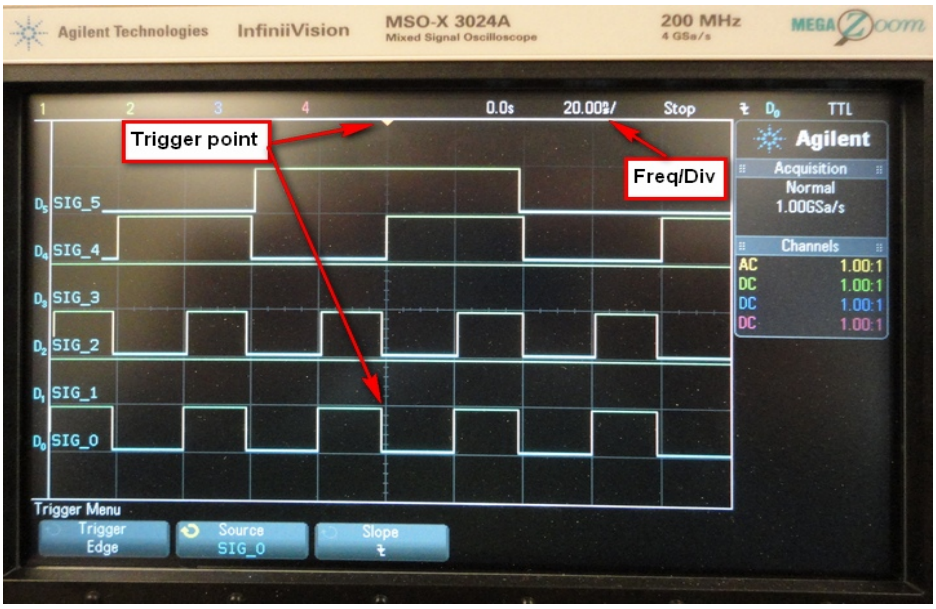


Figure 13

Each time you press the **Run/Stop** button 

and then the **Single** button 

you will get a somewhat different result on the screen but it will still be triggered on the falling edge of SIG_0.

This concludes this section of the tutorial. You now should understand how to setup and trigger on an edge. Try to trigger on different trigger modes.

Manipulating and Magnifying captured data

Now we can further examine the signals.

Rotating the **Horizontal** knob 

, this will allow us to zoom in or zoom out of the captured data. Rotating the **Horizontal Knob** counter clockwise increases the amount of signal data you can view on the screen, but it becomes less detailed. Rotating the **Horizontal Knob** clockwise will decrease the amount of signal data you can view on the screen, but it is more detailed. Note there is a limit as to how much you can increase or decrease the view. Once the limit has been reached, a prompt will come up on the screen.

There are several different ways we can scroll through the data.

Rotating the **Small Knob** 

allows the user to manually scroll through the signal data. Rotating the **Small Knob** clockwise moves the solid **orange** arrow forward through saved data memory. You will also notice that as you rotate it tell you how much delay you are adding from the centre trigger point. The delay value is displayed at the top of the screen. Note the value is negative due to the fact that this is the delay after the centre trigger point. Rotating the **Small Knob** counter clockwise will move the solid **orange** arrow backwards through saved data memory.

Note the delay is positive to indicate how much delay must be added before the centre trigger point. Adding delays may come in handy if you are more interested in what happens before or after the trigger event. If you press the **Small Knob** the solid orange arrow will automatically move back to centre trigger point. An alternative for scrolling through data is to press the arrow buttons. Data will automatically scroll forward or backward through saved data memory.

Pressing the **Right Arrow** 

button allows the user to scroll forward through data memory. If you keep pressing the **Right Arrow** button, it will increase the speed at which saved data is scrolled forward across the screen. The maximum speed is reached after 3 presses.

Pressing the **Stop** button 

automatically stops scrolling.

Pressing the **Left Arrow** 

button scrolls data backwards through saved data memory. If you keep pressing the **Left Arrow** key, it will increase the speed at which the saved data is scrolled backwards across the screen. The maximum speed is reached after 3 presses.

Press the **Magnify** button 

, you will get a split screen which looks like figure 14.

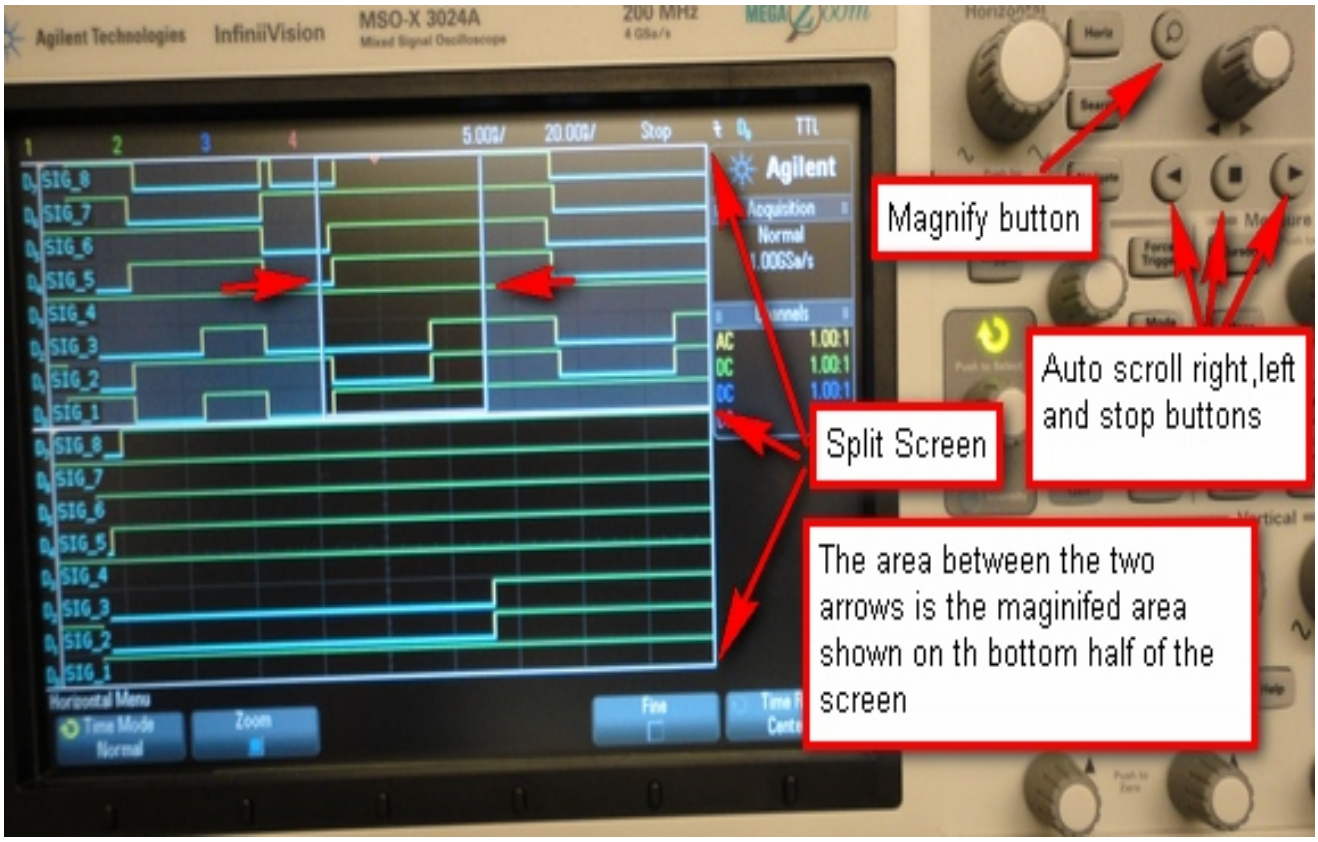


Figure 14

The display gets split into two frames. The top half is the fully captured saved signal data. The bottom half of the frame shows the magnified area between the two arrows as seen in figure 14.

By rotating the **Horizontal knob**  we can increase or decrease the magnified viewing area of the bottom frame. There is a limit as to the maximum and minimum magnification area. Use the **smaller** knob or the **arrow** buttons to scroll through the saved signal data.

Press the **Magnify** button  until there is no split screen.

You should now be familiar with how to view, magnify and scroll through the displayed signals on the screen. This concludes this portion of the tutorial.

Tutorial 4 - Creating a bus

For this tutorial we will be creating a bus and setting up a trigger event using the created bus and display the result on the screen

Note - Before starting this tutorial make sure you have connected the digital cable to the **Altera DE2 board.** To find out how to properly do this, go to **page 20** of this manual.

Note-The label names for each channel, for this tutorial, have been adopted from the results of tutorial 2.

In some cases when debugging code you may want to see what address or data value happens at the trigger point. We will learn how to generate a bus in this part of the tutorial. For the purpose of this tutorial we will create a 4 bit bus.

Press the **Digital** button 

.

Press **Button 6 (Bus)**. In this section there are 5 button menus **1**, **2**, **3**, **4** and **5** as described in table 9.

Button 1	Select Bus 1 or 2. You can assign up to two buses
Button 2	Select the channels that will be associated with the bus
Button 3	Select or unselect D15-D8 group
Button 4	Select or unselect D7-D0 group
Button 5	Select whether the base value displayed for the bus will be in HEX or Binary

Table 9

Press **Button 1**. Rotate the **Select Knob** until **Bus 1** is selected. Press the **Select Knob**. The little box beside the label **Bus1** should change colour to blue. This creates a bus. If you look at the bottom of the screen just below **DO: SIG_0** you will see a new signal channel called **B1 BUS1**. This is the new bus label we just created.

Press **Button 2**. Rotate the **Select Knob** until **D0:SIG_0** is selected. Press the **Select Knob**. Each time you press the **Select Knob** the channel gets selected or unselected**.** The little box beside the label **D0:SIG_0** should become blue when selected.

Select **D0:SIG_0**, **D1:SIG_1, D2:SIG_2 and D3:SIG_3**. Make sure all the other channels are unselected. We have now created our 4 bit bus.

Press the **Trigger** button 

.

Press **Button 1.** Rotate **Select Knob** until **Edge is** selected**.** Press the **Select Knob.**

Press **Button 2**. Rotate the **Select Knob** until **D0_SIG_0** is selected. Press the **Select Knob.**

Press **Button 3**. Rotate the **Select Knob** until **Falling** is selected. Press the **Select Knob.**

Rotate the **Horizontal knob** 

and change the horizontal frequency of the display to **20.0 ns/division**.

Press the **mode/coupling** button 

.

Press **Button 1**. Rotate the **Select Knob** until **Normal** is selected. Press the **Select Knob**.

Press the **Single** button 

, you will notice the bus below the saved data signal will have HEX values. This is the 4 bit HEX values representing the 4 bit bus just created. The screen should look similar to figure 15 below.

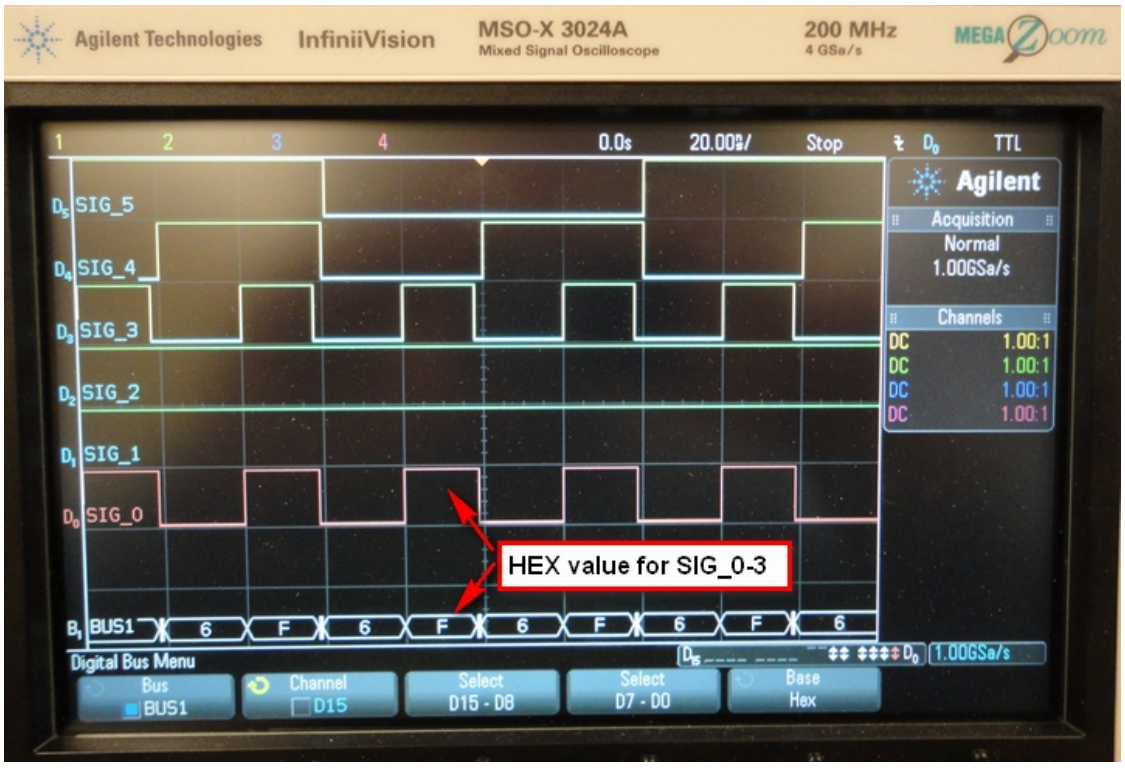


Figure 15

Note that as the signal changes states, a new HEX value is generated.

Each time you press the **Single** button 

new data and corresponding HEX values will be displayed on the screen. You will also notice at times there is a section where the HEX value will not be displayed.

Rotate the **Horizontal knob** 

. As you zoom in on an area you get a more detailed look at the signals and their corresponding HEX values.

Now we will trigger on a **Pattern** instead of an **Edge**.

Press the **Trigger** button 

.

Press **Button 1.** Rotate **Select Knob** until **Pattern is** selected**.** Press the **Select Knob.**

For **Pattern** triggering, buttons **2, 3** and **4** are used as described in table 10.

Button 2	Used to select qualifier, Entered Value, Greater Than, Less Than, In Range, Out of Range
Button 3	Select channel or Bus
Button 4	Set individual channel to active "1", active "0", or don't care

Table 10

For this example we will be triggering a pattern using the entered value of **Bus 1**.

Press **Button 2**. Rotate the **Select Knob** until **Entered** is selected. Press the **Select Knob.**

Press **Button 3**. Rotate the **Select Knob** until **Bus 1** is selected. Press the **Select Knob.** Note **button 4** changes meaning and 2 new button menus **5** and **6** appear**.** They are described in table 11.

Button 4	Digit – not used in this model
Button 5	Selects the HEX data value to be triggered on.
Button 6	Keeps track of number of digits used depending on size of HEX value.

Table 11

Press **button 5.** Rotate the **Select Knob** until the HEX value of **"A"** appears. Press the **Select Knob.**

Rotate the **Horizontal knob** 

and change the horizontal frequency of the display to **10.0 ns/division**. Press the **Small Knob** to ensure that you have centre trigger point on the display. Rotate **Small Knob** counter clockwise to add 6.200 ns of delay to trigger point.

Press the **Single** button 

. The result will look similar to figure 16 below.

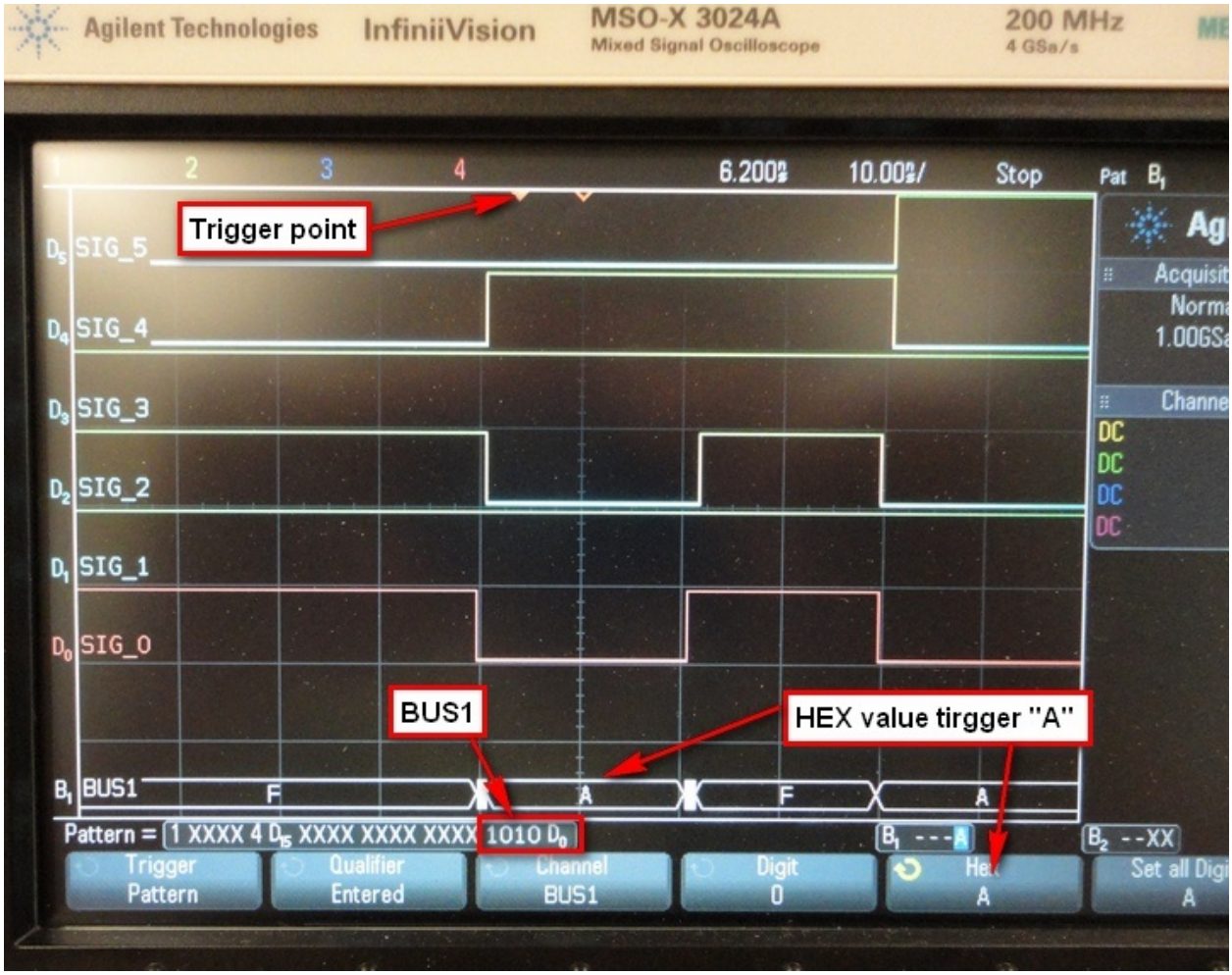


Figure 16

This is the end of this tutorial. You now know how to create a bus and then create a trigger event using this bus.

Tutorial 5- Cursors

The purpose of this tutorial is to enable the cursors and using them to measure the displayed results

Note - Before starting this tutorial make sure you have connected the digital cable to the **Altera DE2 board.** To find out how to properly do this, go to **page 20** of this manual.

Note-The label names for each channel, for this tutorial, have been adopted from the results of tutorial 2.

Press the **Trigger** button 

.

Press **Button 1.** Rotate **Select Knob** until **Edge is** selected**.** Press the **Select Knob.**

Press **Button 2**. Rotate the **Select Knob** until **D0_SIG_0** is selected. Press the **Select Knob.**

Press **Button 3**. Rotate the **Select Knob** until **Falling** is selected. Press the **Select Knob.**

Rotate the **Horizontal knob** 

and change the horizontal frequency of the display to **20.0 ns/division**.

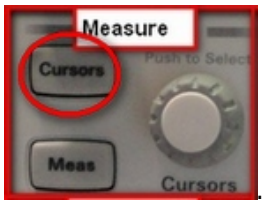
Press the **mode/coupling** button 

.

Press **Button 1**. Rotate the **Select Knob** until **Normal** is selected. Press the **Select Knob**.

Press the **Single** button 

.

Press the **Cursors** button 

. Menus buttons **1**, **2**, **3** and **4** will appear. They are explained in table 12.

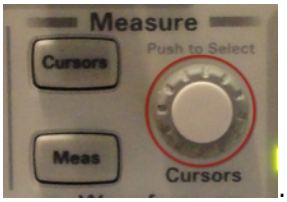
Button 1	Mode –select Manual, Track Wave, Binary or Hex
Button 2	Source – selects if cursor is tracking digital values or bus value
Button 3	Cursor – select X1, X2 or both together
Button 4	Units – seconds , Frequency, phase or ratio

Table 12

Notice that the cursors **X1** and **X2** will appear in the lower right of screen.

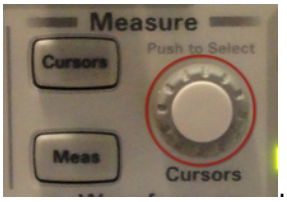
Press **Button 1**. Rotate the **Select Knob** until **Manual** is selected. Press the **Select Knob**. This means we can manually move either cursor **X1**, **X2** or **both** together.

Press **button 3**. Rotate the **Select Knob** until **X1** is selected**.** Press the **Select Knob**.

Rotate the **Cursors Select Knob** 

. Depending on which way you rotate the **Cursor Select Knob** the **X1** cursor moves forward or backward across the screen.

Press **button 3**. Rotate the **Select Knob** until **X2** is selected**.** Press the **Select Knob**.

Rotate the **Cursors Select Knob** 

. Depending on which way you rotate the cursor **Select Knob** the **X2** cursor moves forward or backward across the screen.

Press **button 3**. Rotate the **Cursors Select Knob** until **X1 X2 linked** is selected. When you rotate the Cursor Knob both **X1** and **X2** will move together.

The above action allows the user to move the cursors anywhere on the screen and make measurements. Look at figure 17.

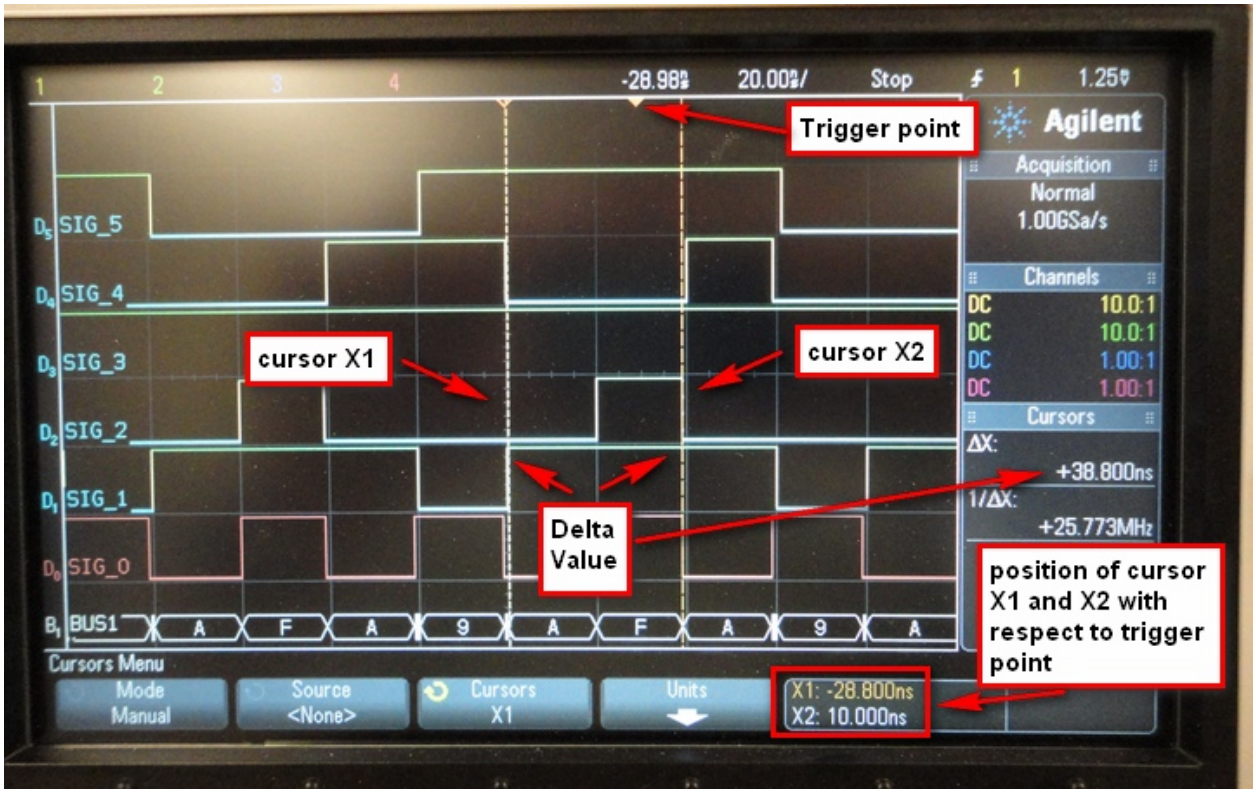
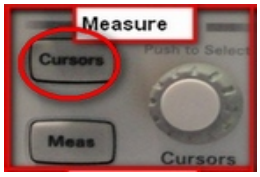
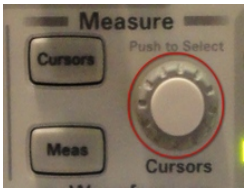


Figure 17

In figure 17 a measurement is being made of **D4**:**SIG_4**. The **Delta value**, distance between two cursors, is **38.800** ns, cursor **X1** is located **-28.800** ns from the trigger point and cursor **X2** is located **10.000** ns from the trigger point. It should be noted that sometimes when you press the **Cursors** button



the cursors may not appear on the screen. The reason for this is that they are off screen. By rotating the **Cursors Select Knob**



forward or backward they will eventually appear on the screen.

Press the **cursor** button 

until both cursors are removed from the screen.

You should now be able to enable the cursors and be able to make measurements of any signal on the screen. This concludes this tutorial.

Tutorial 6 **-Setting up Serial Triggering**

The purpose of this tutorial is to learn how to setup and trigger a serial bus. For this tutorial we will focus on the I2C serial bus protocol.

Note - Before starting this tutorial make sure you have connected the digital cable to the **Altera DE2 board.** To find out how to properly do this, go to **page 20** of this manual.

Press the **Digital** button 

.

Press **Button2.** Select **D6** and **D7** unselect everything else including **BUS1**. When completed only two signals should be displayed on the screen, **D6: SIG_6** and **D7:SIG_7**.

Press the **Serial** button 

. 5 new buttons menus **1**, **2**, **3**, **4** and **5** will appear and are explained in table 13;

Button 1	Serial Bus select
Button 2	Select different serial protocol modes (I2C,SPI,UART/RS232)
Button 3	Signals (defines which data signals will represent serial data and serial clock)
Button 4	Address Size (whether 7 bit or 8 bit)
Button 6	Lister (show saved data results as a listing)

Table 13

Press **Button 1**. Rotate the **Select Knob** until ***Serial* 1** is selected. Press the **Select Knob**. A blue light is highlighted in box beside **Serial 1**.

Press **Button 2.** Rotate the **Select Knob** until **I2C** is selected. A blue highlight and check mark will appear over **I2C.**

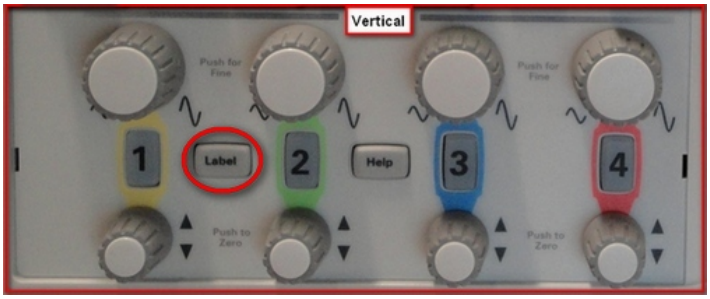
Press **button 3 (signals)**, 4 new button menus 1, 2, 4 and 5 will appear as in table 14;

Button 1	SCL (signal that will represent the serial clock)
Button 2	Threshold value for Serial clock
Button 4	SDA(signal that will represent the serial data value)
Button 5	Threshold value for the serial data

Table 14

Press **Button 1.** Rotate the **Select Knob** until **D6** is selected. Press the **Select Knob**.

Press **Button 4**. Rotate the **Select Knob** until **D7** is selected. Press the **Select Knob.**

Press **label** Button 

if there are no labels beside **D6** and **D7**. **D6** should be labelled **SCL1** and **D7** should be labelled **SDA1**.

Now **D6** and **D7** have been set up and labelled. The next step is to setup the serial triggering.

Press the **Trigger** button 

.

Press **Button 1**. Rotate the **Select Knob** until **Serial1 (I2C)** is selected. Notice that there are two serial bus trigger menus, Serial1 and Serial2.

Press **Button 2.** Rotate the **Select Knob** until the trigger protocol is set to **Start Condition**. Press the **Select Knob.** This means that when **SDA1** generates a start pulse (high to low transition on **SDA1**) a trigger will be generated.

Press **mode/coupling** button 

.

Press **Button 1**. Rotate the **Select Knob** until **Normal** is selected. Press the **Select Knob**. This ensures that when a trigger happens it will happen at the point that the trigger event occurs.

Rotate the **Horizontal knob** 

and change the horizontal frequency of the display to **200.0 us/division**.

Rotate the **Small Knob** 

counter clockwise so the solid **orange** arrow adds 718 us of delay before the centre trigger point. This will mean that when the trigger event happens it will start saving data 718 us before the centre trigger point. So we are more interested in saving trigger information after the trigger event as opposed to trigger information before the trigger event.

Set **all** switches on the Altera DE2 board to the **down** position. Move **switch 0** of the Altera DE2 board to the **up** position.

Press the **Single** button 

.

Move **switch 1** of the DE2 board to up position. This should trigger an event. The result should look like figure 18.

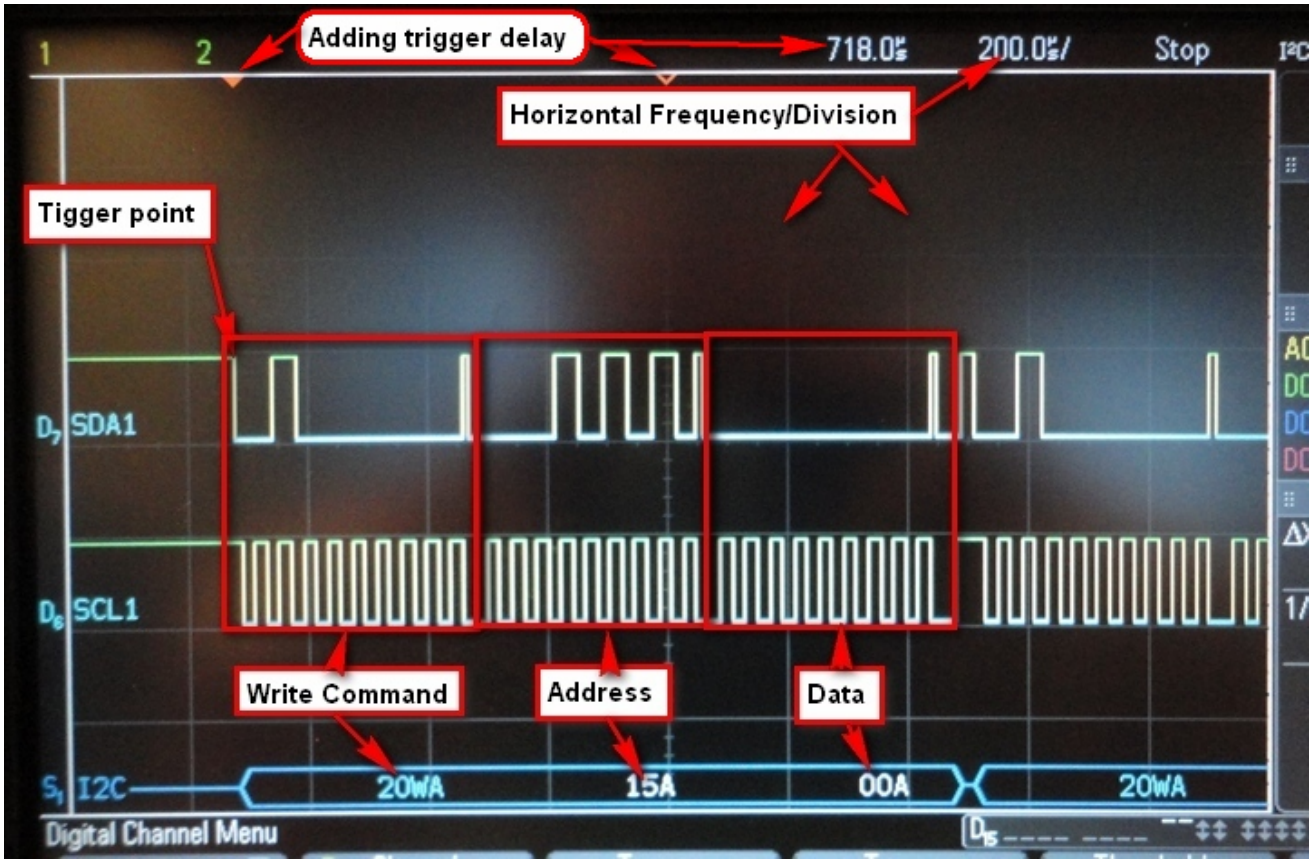


Figure 18

Figure 18 shows the serial clock and serial data and the HEX representation for the **Write Command**, the **Address** and the **Data** value.

Rotate the **Horizontal knob**  and change to horizontal frequency of the display to **5 ms/division**. This will increase the amount of data that you can see on the screen but it will not be as detailed.

Press the **Serial** button  .

Press **Button 6 (Lister).** A new set of menus will appear. Press **Button 1**. Rotate the **Select Knob** until **Serial 1(I2C)** is selected. Press the **Select Knob**. This will enable the user to generate a listing of the HEX values displayed on the screen when we next trigger an event. You will notice a split screen has appeared.. The top half of the screen will show a listing and the bottom half is the serial data and bus. The next time a **Start** trigger event happens a listing of all the serial command/address/data will be gnerated and displayed on the screen. To do this first move **switch 1** on the Altera DE2 board to the **down** position. Make sure that **Switch 0** is in the **up** position.

Press **Small Knob** 

so the solid **orange** arrow moves to the centre trigger position.

Press the **Single** button 

. Move **switch 1** to the up positon. This should trigger an event and the result should look simular to figure 19 below.

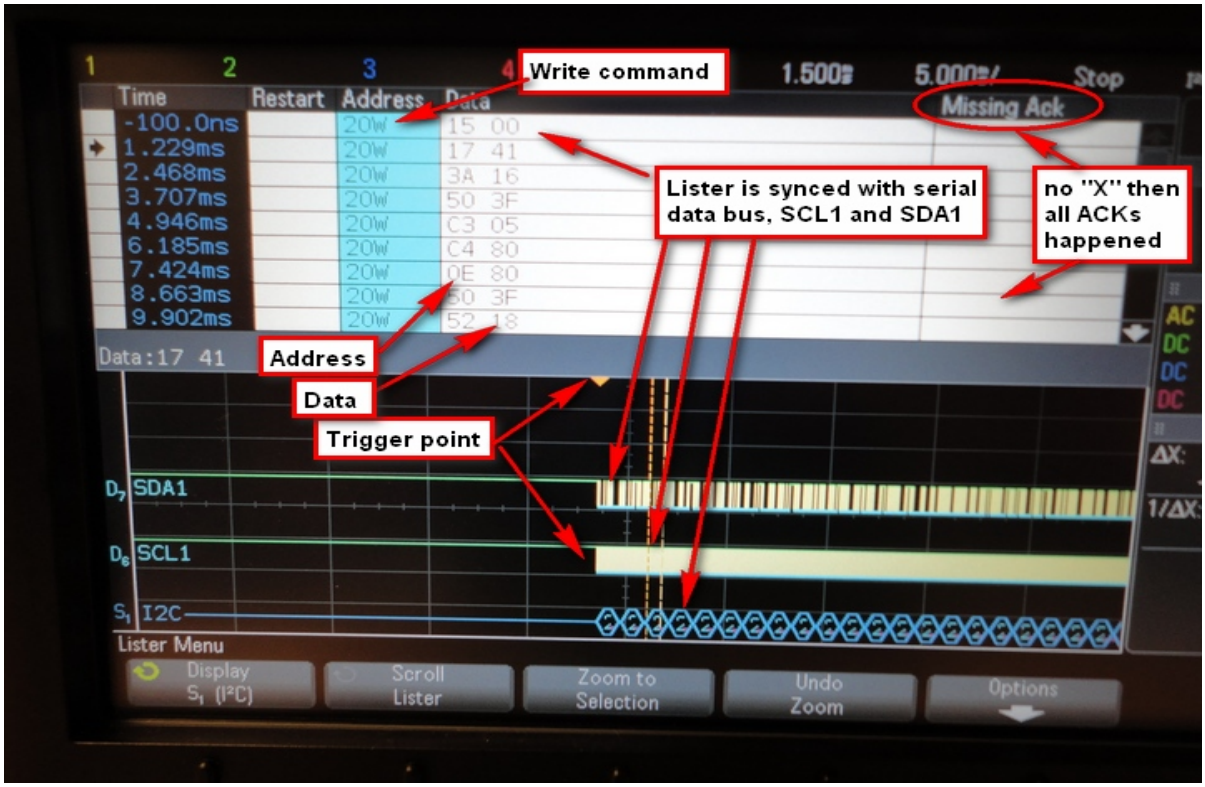


Figure 19

Note that Lister shows time between each Command/Address/Data. The **"W"** in **20W** under address refers to the fact that this is a write command. For a read command you would see an **"R"**. If any of the Read/Write cycles are missing an **Ack**, a **"X"** mark would show up in the **Missing Ack** field. An **Ack** is defined as an acknowledgement from the slave device that it has received the sent data.

Press the **small knob** or the **Right** and **Left Arrow** buttons to scroll through the saved serial data memory.

Trigger on a serial **Address**, rather than **Start**.

Press the **Trigger** button 

.

Press **Button 2**. Rotate the **Select Knob** until **Frame (Address Start Add 7:Write:Ack:data)** is selected. Press the **Select Knob**. Once selected it should be highlighted in blue with a check mark to the left of it. **Button 2** should have the label "Trigger: Write 7".

Button 3 should have the label "Address 0xXX".

Button 4 should have the label "Data 0xXX".

Press **Button 3.** Rotate the **Select Knob** and change the address to **0x20**.

Press **Button 4.** Rotate the **Select Knob** and change the data value to **0x52**.

Rotate the **Horizontal knob** 

and change the horizontal frequency of the display to **200 us/division**.

Move **switch 1** on the Altera DE2 board to the **down** position.

Rotate **Small Knob** 

clockwise so the solid orange arrow moves forward 64 us. This will move the trigger point -64 us from the centre trigger position.

Press the **Single** button 

.

Move **switch 1** to the up positon. This should trigger an event and the result should look similar to figure 20.

Figure 20

There are many different trigger possiblities with the MSO 3000. These 6 tutorials should give you a lot of insight as to the powerful debugging possiblities available. Take time to learn how to use and setup the different trigger protocols. If you have any questions you can email gizmotech@eecg,utoronto.ca for help.

Agilent also provides built in demos

These are built in demos of different triggering demos that are provide by the manufacture Agilent Technologies.

Read instruction on page 9 on how to assemble scope probes if not familiar with them.

Before trying these built in training demos the following must be done. Get the following items out of the top compartment of the MSO.

- both scope probe ground clips
- both scope probe tip clips
- both scope probes

see figure 21 to see what the three items look like.

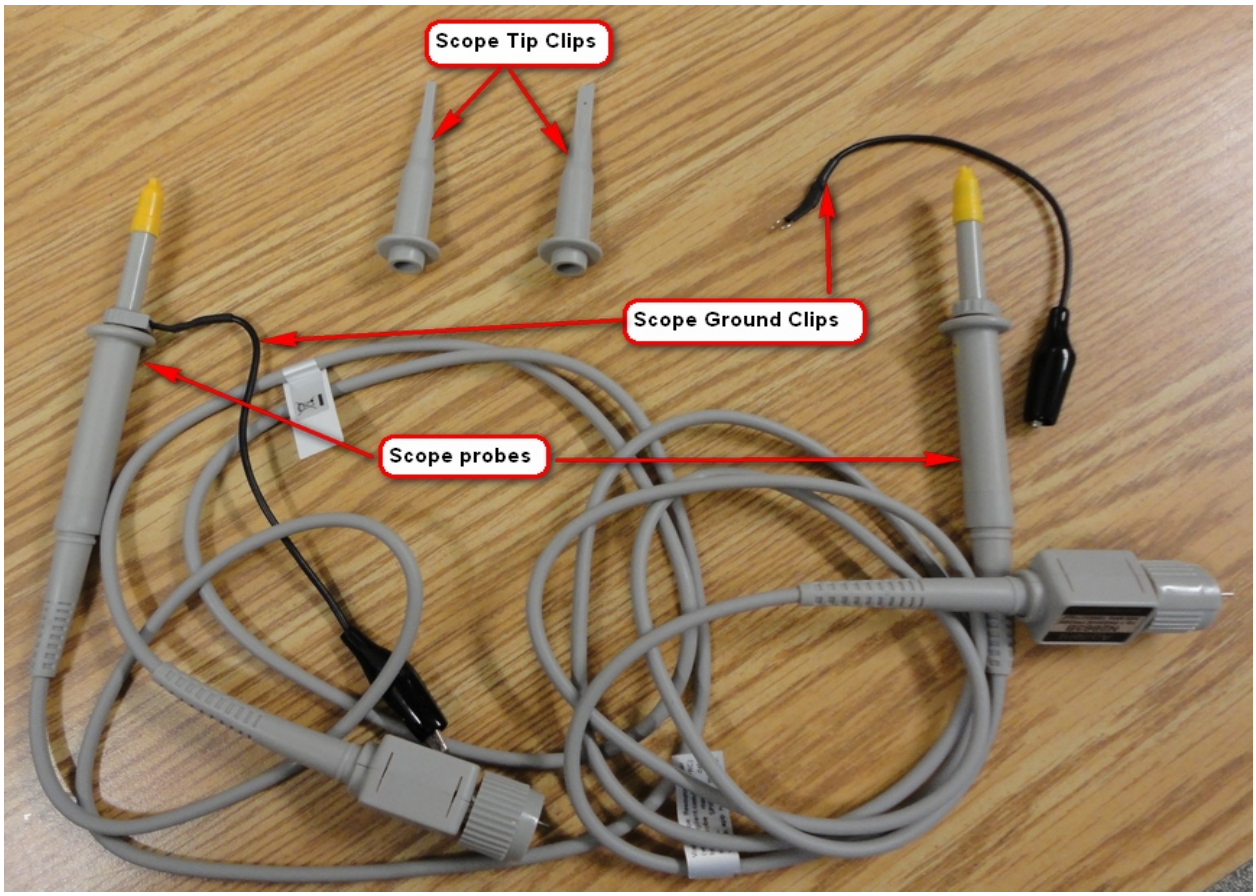
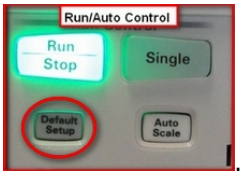
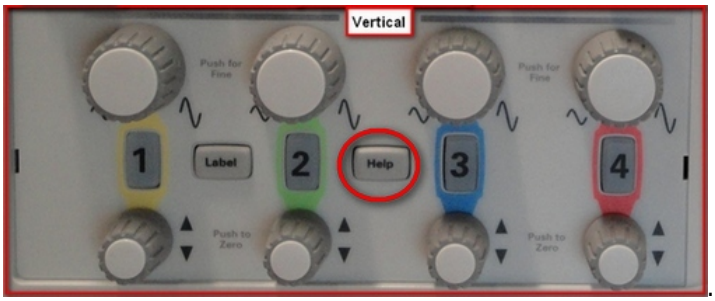


Figure 21

Remove the **orange** tip protective cover from the Scope probes. Insert the Scope probe clips into the tip of the Scope probes. Connect the ground clips into the ground clip input on the scope probe if it is not already done. Connect the scope probes into analog channel **1** and **2**. Connect the ground clip to the ground lug on the front panel of the MSO.

Press **Default setup** button 

.

Press the **Help** button 

. 5 new menus buttons **1**, **2**, **3**, **4** and **5** will appear. They are defined in table 15.

Button 1	Getting started- tips on using the MSO
Button 2	Using Quick Help-Help menu information
Button 3	About Oscilloscope – information on model, what software is installed and version code.
Button 4	Language – Select preferred language
Button 5	Training signals – on line training demos

Table 15

Press **button 5** (**training signals**). Connect scope probes as described on the screen, **Scope probe1** to **demo connection 1** and **scope probe 2** to **demo connection 2**. The result should look like figure 22.

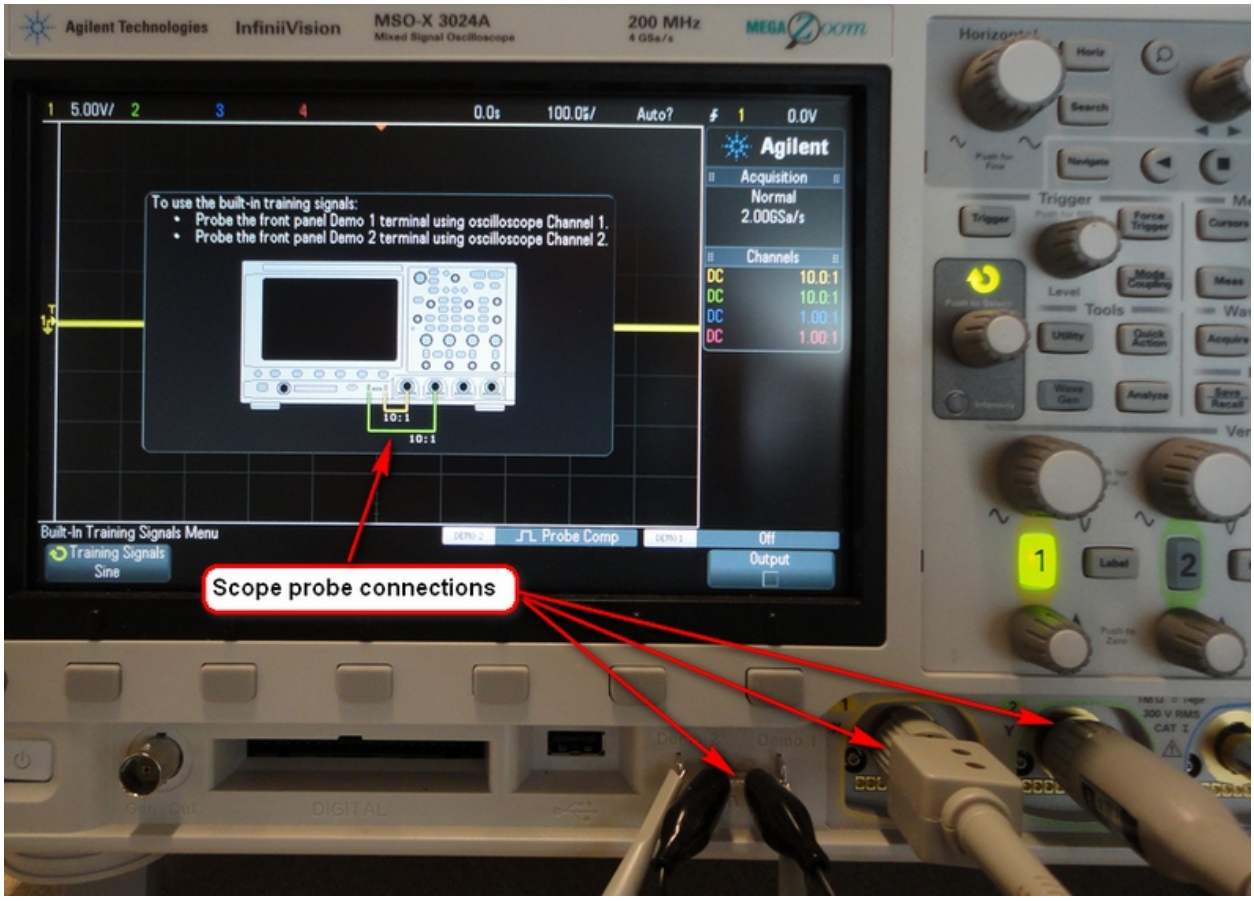


Figure 22

There are 2 button menus **1** and **6**. They are defined in table 16.

Button 1	Training Signals – selects all the different demos
Button 6	Output- enables or disables demos

Table 16

Press **Button 1,** rotate the **Select Knob** until **Edge then Edge** is selected. Press **Select Knob**.

Press **Button 2** (Auto Setup). This automatically configures probe **1** to be signal **A** and probe **2** to be signal **.** For this demo the tigger is set to happen on the 3 edge of **B** and the delay is 2.00ns. See figure 23.

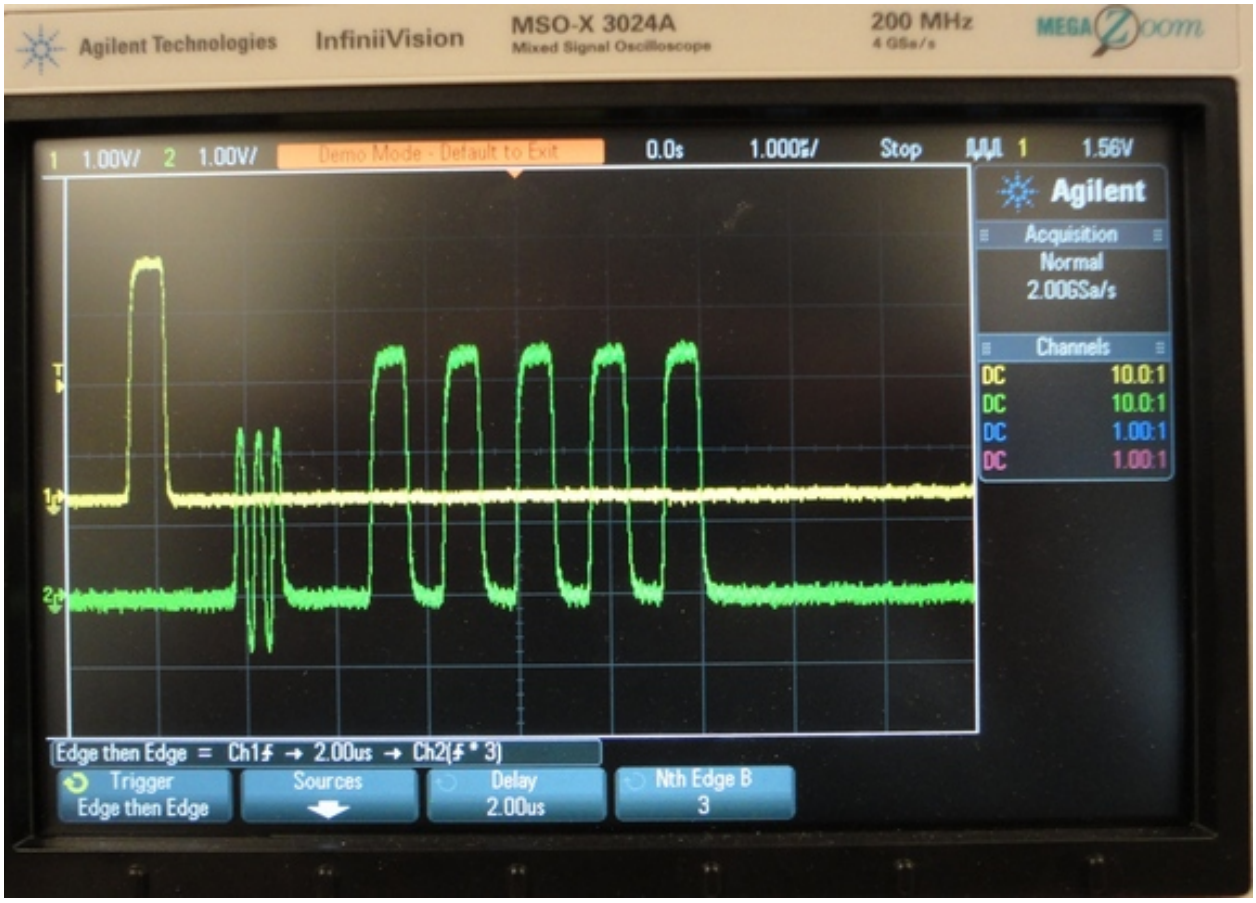
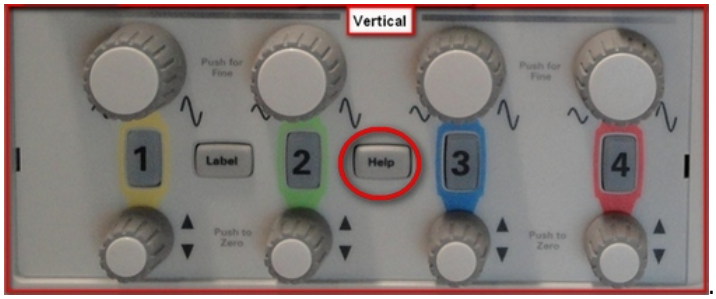


Figure 23

Press the **Single** button 

. Each time the **Single** button is pressed**,** you will get a somewhat different result on the screen.

To change to a different demo press the **Help** button



Press **button 5**(training Signals).

Press **button 1** and rotate the **Select Knob** until the desired demo is selected. Try **I2C**.

Press **button 2** (auto setup). This automatically configures probe **1** to be **SCL1**(clock) and probe **2** to be **SDA1**(data). The result should appear like figure 24.

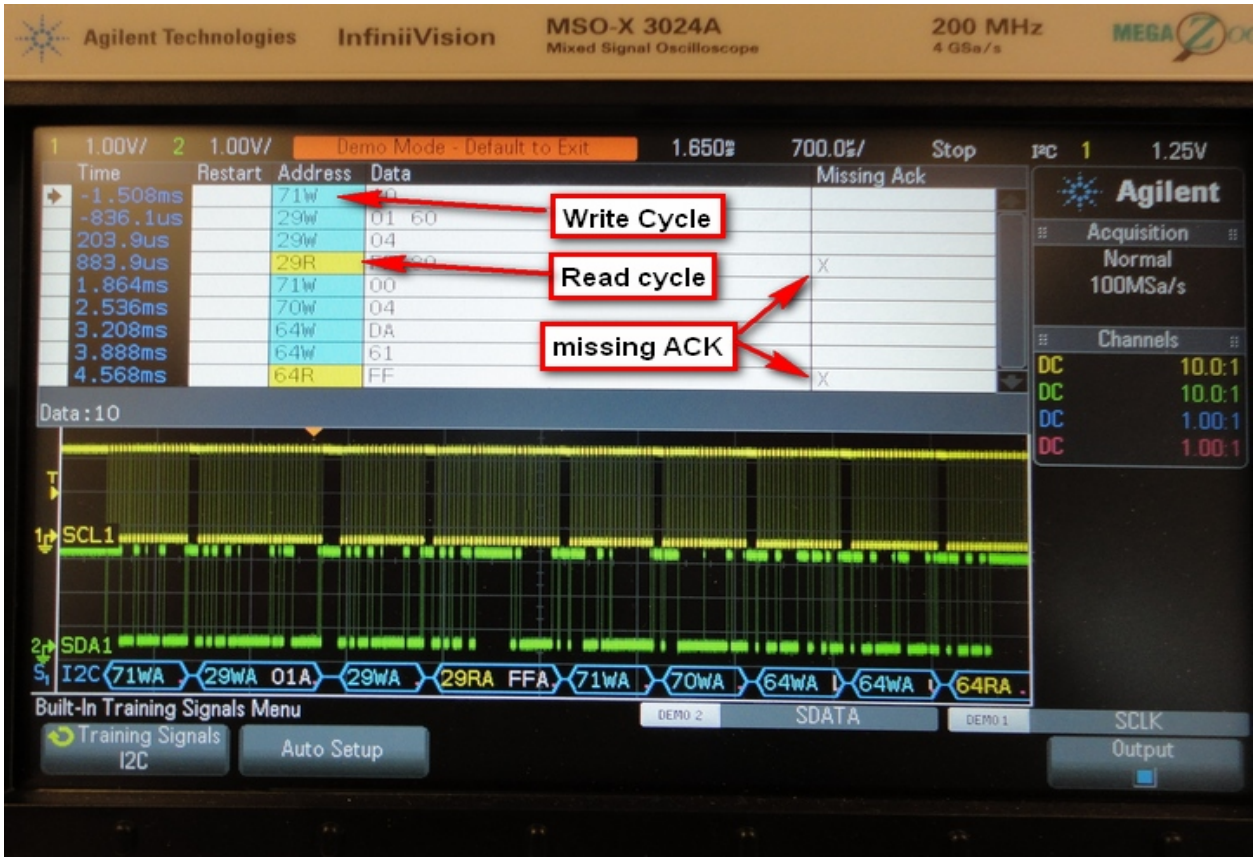


Figure 24

Press the **Single** button 

. Each time the **Single** button is pressed you will get a different result. Note that this demo gives examples of a **read** cycle, a **write** cycle and a cycle with a missing **ACK**. There is a **"X"** beside the **read** cycle containing the missing **ACK**.

This concludes the demo.