RIGOL
UltraVision II Oscilloscope Demo Guide:

RIGOL's new UltraVision II Oscilloscopes are all based on our Proprietary *Phoenix Chipset*, performance HW Architecture, and a next generation User Interface. Across the portfolio they all deliver:

- High Sample Rate
- Long Memory
- Fast Waveform Capture Rate
- 7 Instrument in 1 Capability
- Advanced Analysis Tools
- Powerful Search Capabilities

This guide should help you show/explore these powerful capabilities

Basic Demonstration Hardware Required

Items needed:
- 1 MSO7000 or MSO5000 series oscilloscope
- 2 Probes – PVP3500A
- 1 DS6000 Demo Board
- 1 BNC cable

Optional items:
- DG800, DG1000, DG1000Z, DG900, DG4000 or DG5000 Arb/Function Generator

DS6000 Demo Board

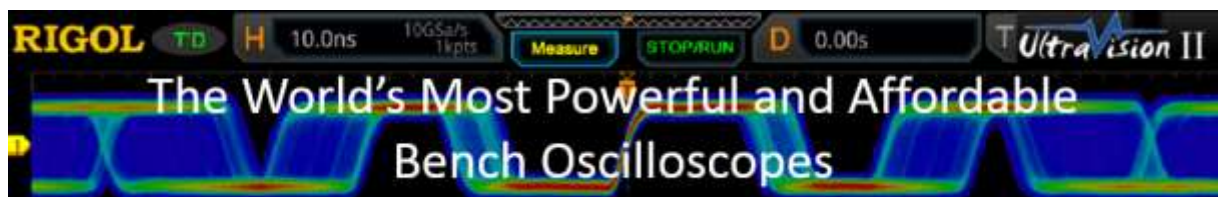**DS7000**

MSO5000

TABLE OF CONTENTS

Initial Setup and Auto Setup

Start by connecting the two probes to channel 1 and channel 2 and connect each to the probe compensation port located in the middle of the front panel of the instrument. This port has the two square loops where you connect the probe tip to the top loop and then the grounding clip to the bottom loop.

- View the square waves by pressing "AUTO". After a few seconds you should see this:

Whenever you get into trouble or don't know what you are looking at you can press "AUTO" and it will find a signal that the scope can trigger on. As another means to be able to reset the instrument back to its default settings enter the instrument's "Utility" menu and then press "System" at this point make certain the "Power On" setting is set to "Default". This way if you ever need to reset the instrument back to factory settings all you have to do is cycle the instrument's power and then it will be back to the original factory settings.

Sampling and Memory Depth

After pressing "AUTO", we can easily demonstrate the sampling and memory depth features.

- Push "CH 2" twice so the light turns off and channel 1 is shown.
- Turn the Horizontal (Scale) knob until time is set to 200 ms / div. This is a rather slow setting and it will look like the almost a yellow rectangle across the entire screen. Then press the "Zoom" button which will enter zoom mode. We can now zoom in on the signal using the Horizontal (Scale) knob. When you zoom down to a 5 us / div window you should see this:

Two things to note in this image:
One: We are looking at just a sliver of our captured waveform (the little black slice between the blue bars on top).
Two: You can see the straight line in the zoom because there are no data points between that time.

*Here is the math for those so inclined: The MSO5000 and the MSO7000 has 500 Mpts per wave in normal mode so in the full window you see 200 ms/div and 10 div = 2 sec of data. Divide that by 500 Mpts and you get a point every 4 nsec or so. Just about what we are seeing above.

- Press "AUTO"
- Turn the horizontal knob to 200ns/div you should see the signal change to this:

- Press "Acquire" then "Mem Depth", then change the memory depth setting from "Auto" to 10K Point and switch between 500M Points and 10K Points.

That curve with 500M Point is much more appropriate. So, if there was an event happening at even µsec timing you can miss it without deep memory especially when you need to capture or trigger on data over a longer period of time.

User Interface

These oscilloscopes come with a touch screen display along with the traditional buttons and knobs to control the instrument, these instruments can also be control via a computer mouse, a web interface and then through UltraScope software.

The main focus of this demo is to cover how to interact with the instrument's touch screen via the four different touch gestures which include Drag, Pinch, Tap, and Draw.

- To begin with connect one of the instruments probes to the probe compensation port on the front of the instrument and then press "AUTO".
- To Drag the signal press anywhere on the main signal display area and then either drag horizontally or vertically.
- To reset the signal press "AUTO".
- To change the signals either vertical or horizontal scale, using a Pinch gesture on the signal display area in either a vertical motion to change the vertical scaling or in a horizontal motion to change the horizontal scaling.
- To reset press "AUTO".
- Tap on the lower left-hand corner of the display on the icon to the left of the channel 1 controls. This will bring up the pop-up navigation menu which allows you to quickly make some measurements, set references, performing pass/fail testing, perform math operations and decode serial communication. See image below:

- Tap on the "Draw Rect" icon in the pop-up menu so the rectangle is highlighted in blue. This control allows you to change from draw to, pinch and drag gestures. With the rectangle function now highlighted Draw a rectangle around your signal and a different pop-up menu will appear and allow you to then Tap on the "Histogram" option. This will then create a histogram view of your signal as defined in the area that you drew out a few moment ago. See the image below:

For more information I have linked to the a User Interface video which demonstrated how to use the instrument's touch screen as well as the Web Interface, mouse controls and UltraScope software.

7 Instruments in 1

These instruments come with seven built in instruments. These include a 4-channel analog oscilloscope, 16-channel logic analyzer, 2-channel signal generator, protocol analyzer, FFT Spectrum Display digital voltmeter, and frequency counter/ totalizer.

- To activate the analog channels, press any of the numbered channel buttons in the vertical section of the display or tap on the bottom channel sections on the touch screen.
- To activate the digital channels, press the "LA" button or the "L" section at the bottom of the display.
- To activate the signal generators either press "GI" or "GII" either on the front panel of the instrument or on the bottom of the touch screen. They will then allow you to then select which waveform you would like to create, its frequency, amplitude, offset, start phase, and addition channel settings like modulation.
- To activate the protocol analyzer, press the "Decode" button then you will be able to choose if you would like to decode a signal either on the digital channels or on the analog channels. Additionally, you will be able to select from either decoding a parallel, RS232, I2C, SPI, LIN, CAN, FlexRay, I2S or 1553B protocols.
- To activate the digital voltmeter press "Measure", then "Analyze" then "DVM" and then turn it "ON".
- To activate the frequency counter/ totalizer press the "Measure" button, then "Counter" and then turn it "ON"

For more information I have linked to a video covering the 7 Instruments in 1 as well the next seven pages of this demo guide will cover them in more detail.

Logic Analyzer

The logic analyzer allows you to view up to 16 digital channels at once along with group them and label them to make your measurements easier to view.

First turn off channel 1 by pressing "CH 1" twice and then nothing will be displayed on the screen.

- To activate the logic analyzer, press the "LA" button or the "L" section at the bottom of the display. Note that the LA channels will allow you some vertical scaling control within the LA menu, but the horizontal scaling is controlled through the horizontal control knobs.
- Once you have entered the logic analyzer menu you can choose which channels you would like to turn on by pressing the "On/Off" section and then with in there you can either turn on or off each bank of eight channel or you can turn on or off specific channels. Turn "OFF" channels "D15-D8"
- This is also where you can group your channels together by pressing "Group" and then you can select one of the four groups and then you can select which channels you would like to group together. Group channels D0-D2 together, and then choose another group and group D5-D7 together.
- Then press "Back" and then press "Group On/Off" to turn on both groups.
- Press the "Back" button.
- To change the channel arrangement, press "Arrange". Notice how this will change how the channels will be positioned on the display either from D0-D7 or D7-D0.
- To change the vertical scaling of the channel press "Size" and then select "Large as your desired size. This will then fill the entire display with the eight channels.
- Press "Label" and then turn "Display" ON. You will now see the channel labels being displayed on the left-hand side of the display, and you can also change these labels. To change the labels press "Select" and choose your channel, then you can select a predefined label from the "Library" or you can create your own custom label by pressing "Label" and then you type one in with the onscreen keyboard.
- Add a "CLK" label to channel "D0" and then label "D7" with your name.

Signal Generator

The signal generator allows you to both create standard waveforms but also allows you to easily create some custom waveforms as well as replicate a captured signal on the oscilloscope.

- To activate the logic analyzer, press either "GI" or "GII" on either the front panel of the instrument or at the bottom of the screen.
- To begin with to select your desired waveform press "Wave" and then you can select form several different built in waveforms as well as load arbitrary waveforms that you either created externally or by copying one of the scope analog channels.
- Within the waveform menu you can set the functions frequency, amplitude, offset, starting phase.
- To modulate, burst or sweep the signal, you will need to press "Settings", then "Type" and then select your desired advanced feature.

- To create the signal below connect "GI" to "Channel 1" on the instrument with a BNC cable.
- Select "Sinc" as your waveform by pressing "Wave" and then "Sinc".
- Then change the signals frequency to 10 kHz. You can do this by either pressing on "Frequency" on the touch screen and then using the pop-up number pad or you can use the keys and then use the intensity knob to change the frequency settings.
- Press "AUTO" and you will see an image like the one below.

Protocol Analysis

Protocol Analysis allows you to quickly decode serial communication. The oscilloscope is capable of triggering on and decoding parallel, RS232, I2C, SPI, LIN, CAN, FlexRay, I2S, and 1553B.

For this example, you will be decoding an I2C signal on the DS6000 demo board. Connect a probe connected to "Channel 1" on your oscilloscope to the "I2C_SCL" loop on the board. Then connect a probe connected to "Channel 2" to "I2C_SDA" loop on the board and then press "AUTO".

- Now that you have your probes connected to the two signals you will need to setup the correct triggering for the instrument. To do this press the "Menu" button in the triggering section on the front panel of the instrument. This is the right most "Menu" button.
- Change the "Type" to "I2C" which you do by pressing "Type".
- Make certain that "SCL" is set to "CH1" and "SDA" is set to "CH2".
- Change your "Horizontal" scaling to 100 us. Which you can do that by rotation the horizontal scale knob counter clockwise.

- Now that you are triggering on the I2C signal press "Decode" button and then press "Decode1" and then select your "Bus Type" to be "I2C".
- Press "Copy Trig" and then switch "Bus Status" to "ON". You will start to see the instrument decoding the signal and displaying this in the middle of the display.
- Currently the signal is being decoded in a hex format, but this can easily be changed to either decode hex, binary, decinormal or ASCII. To do this press "Display" and then press "Format" and change it to "ASCII". This will then decode the signal to read RIGOL. See the image below.
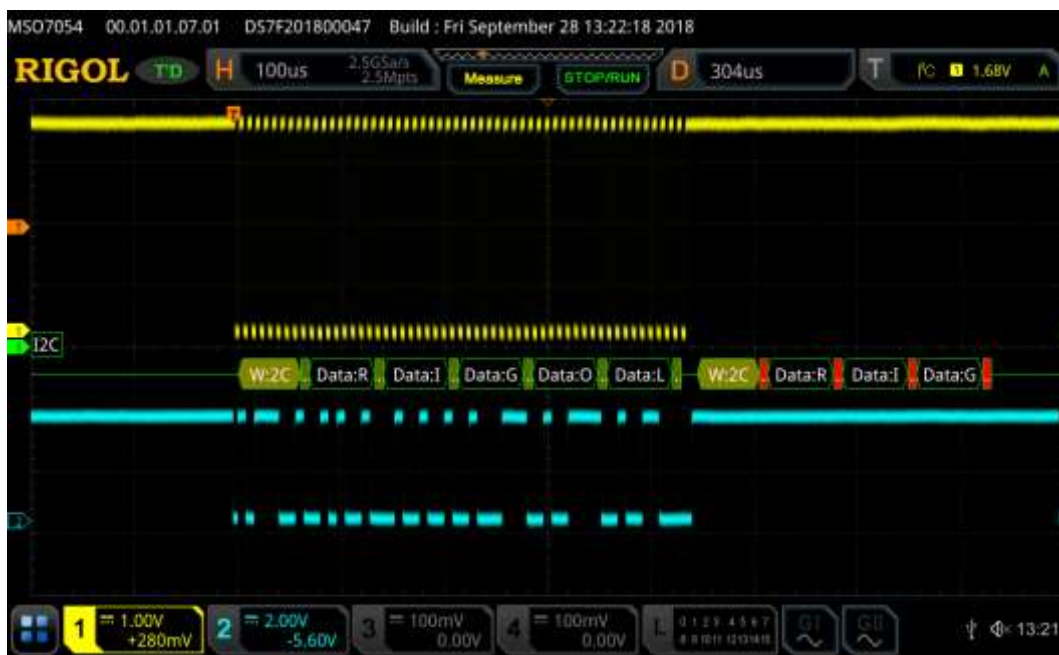
For more information I have linked to a video covering Protocol Analysis.

Advanced Protocol Analysis

Within the decode menu there are a couple of different ways to view the decoded signal and move around on the signal. To start off with perform all the sets in the Protocol Analysis demo.

- First change the horizontal scale to be 1.ms by twisting the "Horizontal Scale" encoder counter clockwise.
- From the "Decode" menu press "Display"
- In the "Display" menu there are several different settings first you can change the decode format, then you can move the position of the decoded line, then there is the ability to label the signal and then finally the event table.
- Press "Event Table" and then turn ON the "Event Table". At this point it will be displayed in the middle of the screen.
- Change the format of the event table to ASCII by pressing "Format" and then select "ASC".
- Change the view to payload, to do this press "View" and then select "Payload". Here you will see the decoded signal being displayed in a matrix which this is a great way to view a serial signal all at once to quickly allow you to see if there is a problem with the serial signal.
- Next change "View" to "Packets". In this view you can scroll on the menu to view all of the data packets, in this case "RIGOL" you can see the time that it was sent in relationship to the trigger point, weather it is a read or write packet then address of the packet, the data and then if it was acknowledged.
- Another useful aspect of the event table is the jump to capability. First press "SINGLE" then select the "6th" packet by pressing on it. Notice that there are two dotted lines both above and below that packet and then press "Jump to". This will cause the instrument to automatically move the 6th packet into the middle of the display so that you can zoom in on it.
- Exit out of the event table by pressing the "X" in the upper righthand corner of the window and then change the horizontal scaling to be 100 us by rotating the "Horizontal Scale" knob clock wise.


For more information I have linked to a video covering the Protocol Analysis

Digital Voltmeter

These instruments also comes with a digital voltmeter which allows you to easily take voltage measurements while also monitoring the signal on the display.

Reset the oscilloscope and then connect an analog probe to channel 1 and then connect it to the probe compensation port.

- Press "AUTO"
- Press "Measure" then press "Analyze" and then "DVM". This will bring you to the digital voltmeter menu and allow you to change the channel that the digital voltmeter is monitoring, then the mode of the tool and then limits.
- Make certain that the digital voltmeter is set to monitor channel 1 and then turn on the digital voltmeter by pressing "DVM"
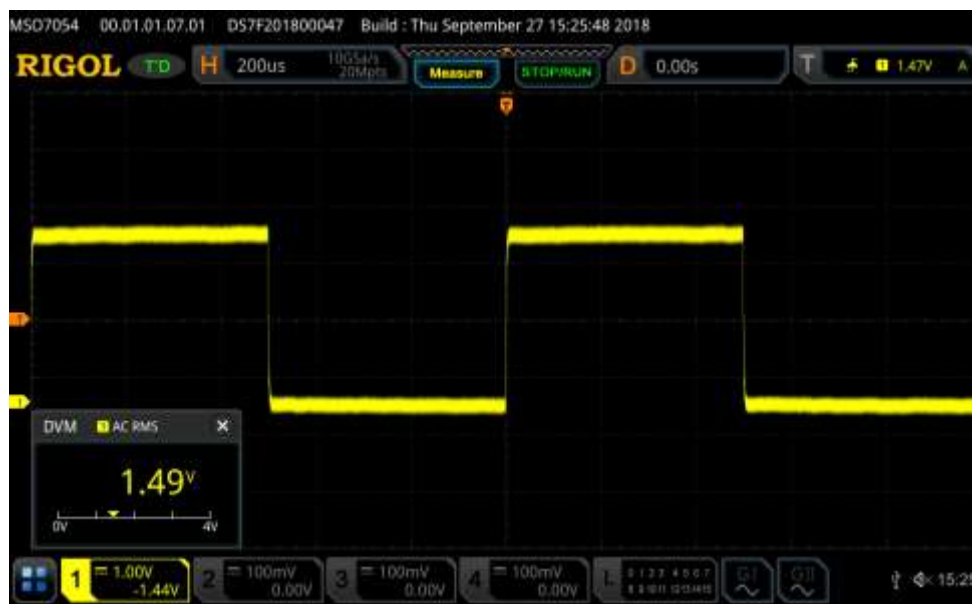
Frequency Counter and Totalizer

These oscilloscopes also comes with a Frequency Counter and Totalizer which allows you to easily measure a signals frequency, period, or count the number of trigger events that have occurred.

- Press "Measure" and then press "Counter". This will bring you to the frequency counter and totalizer menu and will allow you to change the channel that the tool is monitoring, and then measurement that you wish to use at that time.
- Make certain that channel 1 is selected and that Frequency is selected as the "Measurement" of choice. Then turn on the counter by pressing "Counter". You will then see an image similar to the one below.

- To count the total number of trigger events that have occurred change the measurement type to totalizer. To do this press "Measure" and then select "Totalizer".

FFT

All of our oscilloscopes come built in with an FFT capability to allow you to easily view your captured signal in the frequency domain.

Reset the oscilloscope and then connect an analog probe to channel 1 and then connect it to the probe compensation port.

- Press "AUTO"
- Press "Math", then select your Math Channel then you will need to change the "Operator" to be FFT and then you will need to turn the "Operation" on.
- Select "CH 1" as the source
- Set your offset to be 20 dBV and your scale to be 10 dBV.
- To change the instruments displayed span you will need to press "More" and then you can change the "Start" frequency and "End" frequency. In the Image below, I used a "Start" frequency of 0 Hz and an "End" frequency of 50 kHz.
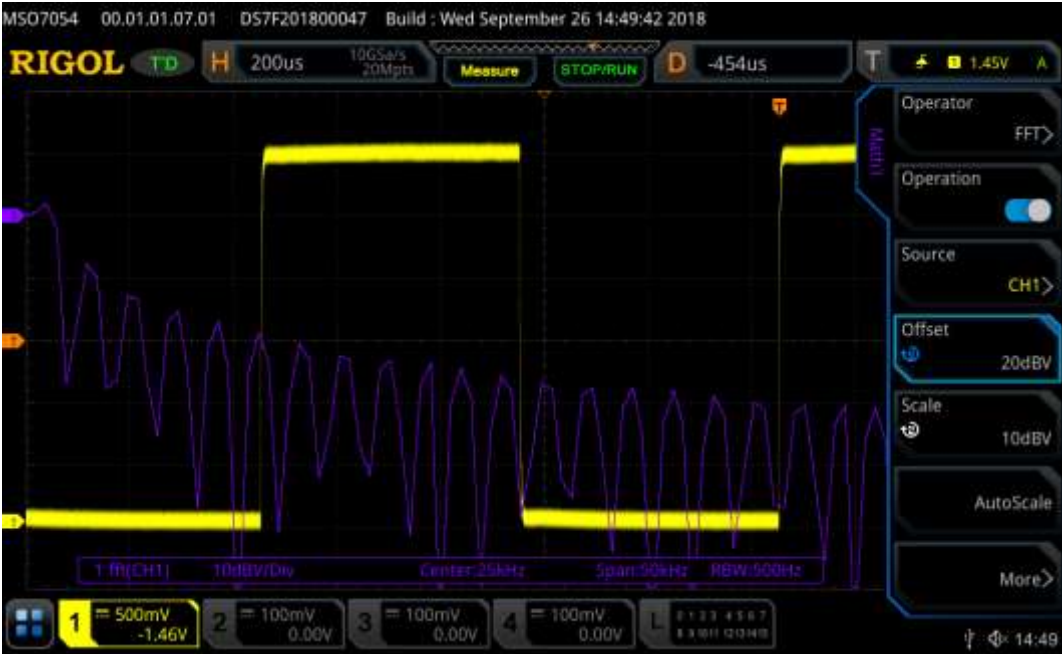
Enhanced FFT

The FFT measurement has some advanced features built in, these include the ability so show the FFT in a frequency intensity display add labels to the signal along with easily change the signals span, scaling and display up to four different FFTs at once. To demonstrate some of the advanced features you will need to use a waveform generator either the built-in generator or with an external waveform generator.

- Set up the waveform generator so that it is a producing a square wave with a frequency of 5 MHz and an amplitude of 5 Vpp.
- Press "Auto" on the oscilloscope and then change the horizontal scaling so that it has a horizontal scaling of 20 us and then change the vertical scaling so that it has a vertical scale of 5 V and then move the signal so that it is on the upper part of the display.
- Press "Math", then select your Math Channel (Note that this instrument can have up to four different math operations operating at once and you can perform math operation of pervious math operations allowing you to perform some advance math functions), then you will need to change the "Operator" to be FFT and then you will need to turn the "Operation" on.
- Select "CH 1" as the source.
- Set your offset to be 20 dBV and your scale to be 20 dBV.
- To change the instruments displayed span you will need to press "More" and then you can change the start frequency and end frequency. In the Image below, I used a "Start" frequency of 0 Hz and an "End" frequency of 100 MHz.
- To add a label to the FFT and display it in a frequency intensity display you will need to press "More".
- To add a label, you can either select a label from the built in "Library" or you can create you own custom label by pressing "Label" and then use the on screen keyboard to enter you desired label. Once one has been selected turn the "Display" ON.
- To the FFT display the signal in a frequency intensity display mode press "Color" and turn it ON. It will take some time for the FFT to propagate the display but after a sort period of time you will have an image similar to the one below.
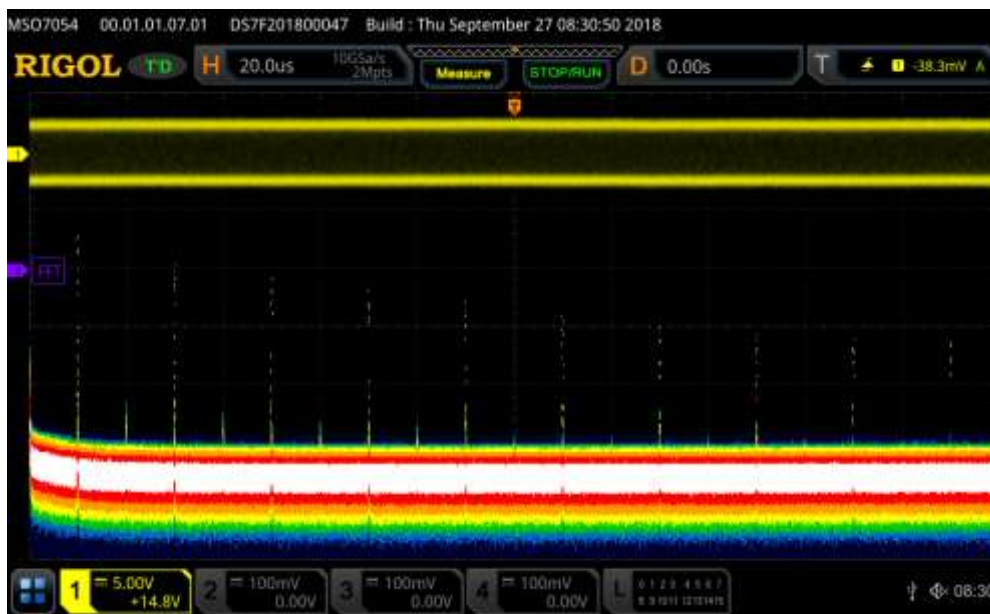
For more information I have linked to a video covering the Enhanced FFT Capabilities of the instrument.

Measurements

Start from an AUTO setup on a Square wave and a Sine wave signal. You can create these signals either using the built-in waveform generator or with an external generator.

- Press "Measure" then press "Add" on the right side of the display. Now the Quick Measurement Menu appears.
- You are able to change between the different categories by either pressing on the tabs in the menu or by changing the "Category" section on the menu on the right side of display. This will allow you to pick between vertical and horizontal measurements along with use the other comparison measurements and then analyze the signal.
- You can change which channel is associated with the measurements by toggling between channel 1 and channel 2 by changing the source on the right side of the display

- To show all measurements for one channel you can do this by pressing "Measure" and then "All Measure" and you can select to display either all of the measurements on channel 1 or channel 2.

Cursors

You can also set up a single measurement like "Frequency" and then set the cursor to "Measure". The "Measure" cursor takes the last measurement and places the cursor on the screen to show how it is making that measurement. This is an easy and great way to align cursors and highlight characteristics of a signal for review or reporting.

- Press "Measure", press "Remove" and then press "Remove All" Delete all of your previously set measurements, then add "Frequency" as the only measurement. It should display on the screen. Then press "Cursor", then set mode to "Measure" and turn on "Indicator". It should now look like this:

The cursor is locked to the zero-crossing showing the time between consecutive waves.

- You can also show the "Track" and "Manual" modes of the "Cursor".
- Use the "Intensity Knob" to move the cursor position.
- In "Manual" mode note that you can use the soft keys or touchscreen to highlight Cursor A, Cursor B or Cursor AB. This determines which cursors will change with the "Intensity Knob". If you haven't shown the Recording and Playback yet, this is a great time to show the Sinc or Sine wave signals and you can use the cursors on the recorded data. That is a great way to do post-analysis of signals.

Creating a Reference

- Press AUTO and press SINGLE
- Press "Ref" (Next to the "LA", "Math" and "Decode" buttons). Select "SaveToRef". Note that you can save and reload References from internal or external memory. This is a great way to compare traces between scopes or to use as a guide for verification testing. Once saved, the reference will appear as an orange trace that you can then visually compare to live signals on any channel. It looks like this:


- You can also change the references colors by pressing "More" and then you can change the colors and even add a label for the reference. This is a rather useful feature if you are using multiple references at once or if you are rapidly changing between references.

Persistence View

Persistence allows you to view how the signal changes over time on a scope. This is most useful when viewing signals that are dynamic such as a modulated waveform. To increase the signals persistence, press "Display" and then press "Intensity" and select "Infinite".

Use a waveform generator to create an AM modulated sine wave. To create the image below I use an AM modulated sine wave that was created on one of the waveform generator channels on the front of the oscilloscope, then I set the initial frequency to be 3.8 kHz and then a modulating frequency of 50 kHz which will cause the viewed signal to rapidly move on the display.

You will need to set the horizontal scaling to be 100 us and then you will need to adjust the trigger point by rotating the encoder in the trigger section on the front panel so that the trigger point is slightly above the center of the scale.
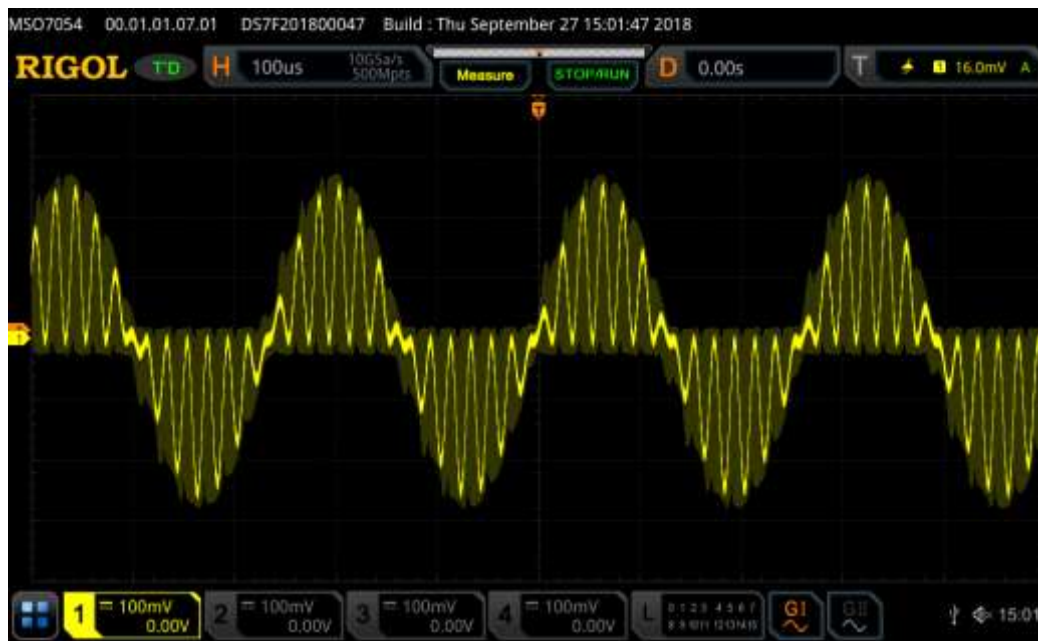
In the brighter yellow we can see the signal that is being captured at the exact moment in time and then then darker yellows showing the modulation envelope shows were the signal has been while the scope has been capturing the signal.

Color Grade View

Color Grade allows you to view your signals in reference to how often the signal is in a particular location. This is most useful when viewing signals that are dynamic. p" and then in the "Display" menu press "More" then press "Color Grade" so that it is switched on.

Use a waveform generator to create an AM modulated sine wave. To create the image below I use an AM sine wave that was created on one of the waveform generators channels built into the instrument, then I set the initial frequency to be 25 MHz and then the modulating frequency to be 50 kHz which will show a stable AM signal. To cause the signal trigger in the middle of the screen move the trigger encoder up on the screen so that it is triggering on the top of the AM signal.

Histogram

The histogram function allows you to visually analyze modulations, noise, jitter and other signal characteristics with the customizable histogram display. Easily characterize signal behavior versus time or amplitude and display both visual histogram as well as a statistical data associated with the reference area.

Use a waveform generator to create an AM modulated sine wave. To create the image below I use an AM sine wave that was created on one of the waveform generators channels built into the instrument, then I set the initial frequency to be 25 MHz and then the modulating frequency to be 50 kHz which will show a stable AM signal. To cause the signal trigger in the middle of the screen move the trigger encoder up on the screen so that it is triggering on the top of the AM signal.

- Tap on the lower left-hand corner of the display on the icon to the left of the channel 1 controls. This will bring up the pop-up navigation menu which allows you to quickly make some measurements, set references, performing pass/fail testing, perform math operations and decode serial communication.
- Tap on the "Draw rect" icon so that the rectangle is highlighted in blue.
- Once this is done draw a box over one of the AM signals lobes.
- Once you have drawn the box press "Histogram". This will not only display the histogram but also bring you to the histogram menu.
- In this menu you can choose the type either if you want it to be over time or amplitude of the signal, the source that you would like to view and the scaling of the histogram.
- Turn statics on. This will not only display the histogram on the screen but also tell you the results in the box that you originally drew.
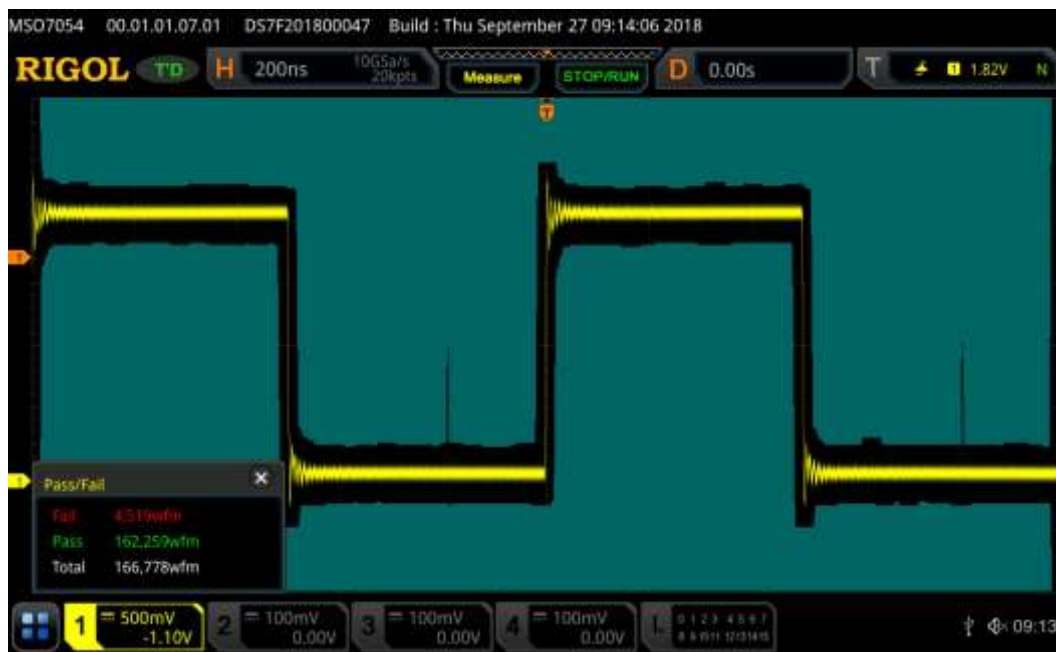

For more information I have linked to a video covering Histogram Analysis.

PASS/FAIL Measurements

You can use Pass/Fail measurements for a number of applications. With this scope family you can set it up to stop or continue when there is a failure, you can have the instrument make a beep or take a screen shot when a failure occurs. You can create masks with varying error budgets around a target signal, and you can even use the record mode to record all the failures as frames. In this way, you could set up a test over night, let it run, and come back and have all the failure states recorded for you.

Connect an analog probe from channel 1 to the Glitch_CLK" signal on the DS6000 Demo board.

- Press "Utility" and then press "PassFail".
- Set "Enable" to ON.
- To change the mask parameters, press "Mask", and then "Create", you can then change the Masks X parameters and Y Parameters and even select if you would like to mask effect the full screen or between two cursor points. Once this is done press "Create".
- You can turn on the information display by first pressing "Back" and then press "Information" to turn on the display.
- At this point you can then turn on the mask by pressing "Operate" to begin your testing.

For more information I have linked to a video covering Pass/Fail Testing.

Record Mode

This mode can be used to capture a large number of single waveforms and allows you to play back and/or analyze them.

Connect an analog probe from channel 1 to the Glitch_CLK" signal on the DS6000 Demo board.

- To activate this mode press "Utility" and then "Record".
- Press "Record" and a recording will start.
- By pressing "Play" you can view the recording that you just took from the beginning of the capture.
- You can also view and then change the current frame that you are viewing by pressing "Current" and then either entering in the desired frame with the touchscreen or you can easily scroll through the frames by using the intensity knob.
- You can also jump to the beginning and end of the recording by pressing "Jump To".
- You can also navigate through the recording by using the "Navigation" buttons located below the "Display" button.
- For more setting in record mode press "More" and within this menu you are able to change the number of frames you would like to record for and the time between each frame.
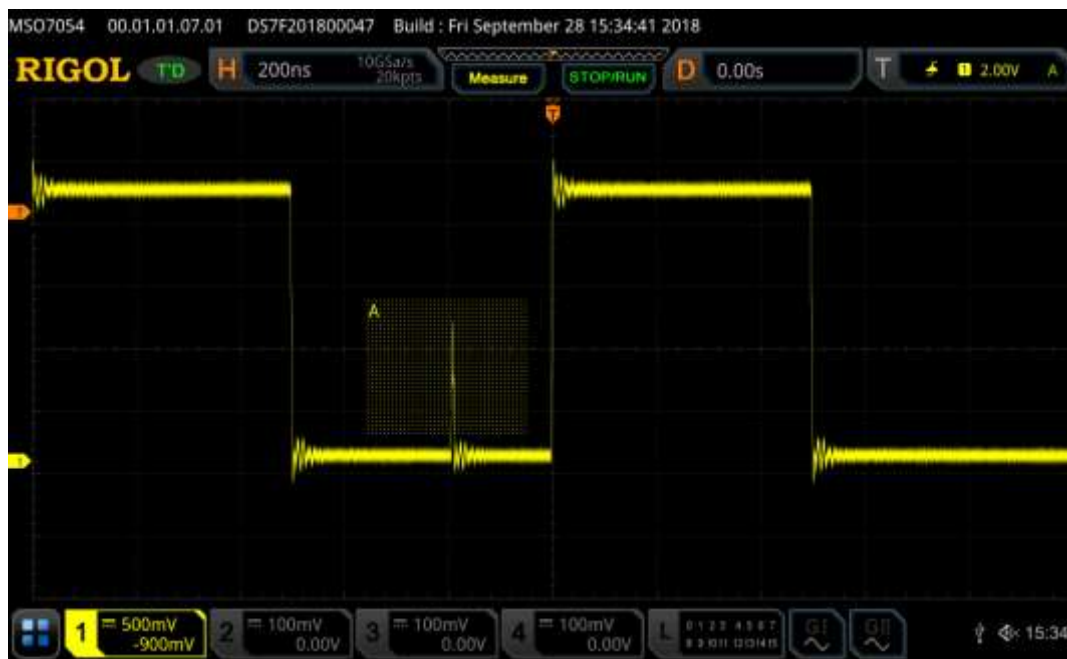

For more information I have linked to a video covering Record Mode Analysis.

Zone Trigger

The zone trigger capability of the instrument can be useful for triggering on none standard trigger events or if you would like an easy way to trigger on or exclude part of a signal

Connect an analog probe from channel 1 to the Glitch_CLK" signal on the DS6000 Demo board.

- With your signal now connected to the oscilloscope press "Auto"
- Tap on the lower left-hand corner of the display on the icon to the left of the channel 1 controls. This will bring up the pop-up navigation menu which allows you to quickly make some measurements, set references, performing pass/fail testing, perform math operations and decode serial communication.
- Tap on the "Draw rect" icon so that the rectangle is highlighted in blue.
- Now tap anywhere outside of that menu to close it.
- To draw your zone trigger, draw a square in the section of the square wave that is in the same place as in the image below. Don't include the square wave with the signal. Tap "Trigger zone A"
- This will bring you to the zone trigger menu, this instrument can use to two different zone triggers at once in addition to the already standard trigger that the scope is using. You can then choose to include or exclude the pulse by changing the "Zone A" setting between "Intersect" and "Not Intersect". (Note that if you need to move the zone trigger you can drag it around on the screen by pressing on it and then dragging it.) See image below.

For more information I have linked to a video covering Zone Triggering.

Search and Navigation

Search and Navigation is a great feature to easily move around on long complex signals with multiple trigger events. The Search and Navigation feature can be used when either an edge, pulse, runt, slop, RS232, I2C or SPI signal is being trigger on and can allow you to select a desired trigger point.

For this demonstration I connected the oscilloscope to the "SPI_MOSI" signal on the DS6000 Demo board with one of the PVP2350A probes.

- Once the probe is connected press the "Auto" button, this will cause the instrument to trigger on the edge of the signal. It will not be using a serial trigger at this time.
- Change the horizontal scaling so that it reads 50 ms.
- Press "Search" then set the "Threshold" with the "Intensity Knob" so that the orange dotted line is in the middle of the signal.
- Press "Back" and then turn the "Search" ON.
- Notice at the top of the display there are now a large number of white triangles and one red triangle these indicate all of the trigger events that the search and navigation function is recognizing.
- Press the "Stop/Run" button and then turn ON the "MarkTable". This will cause the Marker Table to appear on the screen.
- You are able to both scroll and then tap on any of the marked events and then the scope will automatically jump to them and they will then be marked in a red arrow.
- For instance, press on the 11th event and then change your horizontal scaling to be zoom in on the signal.

For more information I have linked to a video covering Search and Navigation Analysis

Roll Mode for Slower Moving Signals

You need a waveform generator to create an arbitrary file like the ECG file that was created below. I recommend using the built-in waveform generator creating an ECG signal and then an output frequency of 1 Hz to demonstrate this mode.

Roll mode is a nice option for slower moving signals that helps you follow it visually.
To enter roll mode press "Acquire" and then change the "Timebase Mode" to "Roll". Once your timing is set you should see something like this.

XY Mode

XY mode is a way to look at 2 signals and show how they interact. To activate, press "Acquire" then change the "Timebase Mode" to "XY"

To demonstrate this viewing mode, you need to use a two channel waveform generator and then turn both outputs on and have them be producing sine waves with the same frequency. Align the phase of both channels and then change the start phase of one of the two channels. You will notice as you change the phase you will start to see an ellipse on the screen. To create the image below I used the two built in waveform generator channels both creating a sine wave with a frequency of 10 kHz and then I change the start phase of channel 2 so that it was 60 degrees out of phase.
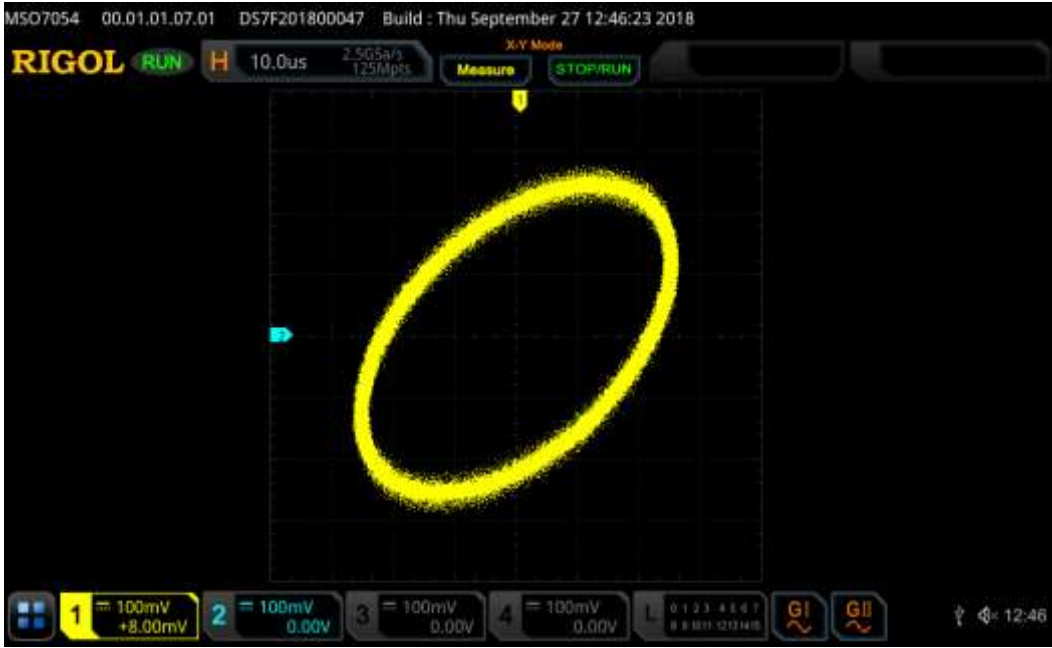
The ellipse below is generated in XY mode where both signals are sine waves and channel 1 is on the X axis and channel 2 is on the Y axis. This is one way to measure the phase between 2 sine waves. To complete the calculation follow the instructions from the manual shown below. There are a number of uses for XY mode. This is just one example.
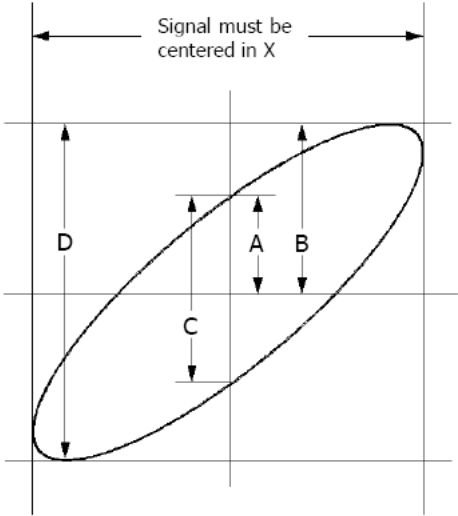

Figure 3-6
Ellipse method to observe the phase difference

Sinθ = A/B or C/D, where θ = phase shift (in degrees) between the two signals.
From the formula above, you could get:

θ = ±arcsine (A/B) or ±arcsine (C/D)

If the main axis of the ellipse is at I and III quadrant, θ must be in the range of (0~π/2) or (3π/2~2π). If the main axis is at II and IV quadrant, θ must be in the range of (π/2~π) or (π~3π/2).

Consequently, if you change the phase deviation between the two Generator Outputs on the waveform generator you can alter the orientation of the ellipse. If you hook it up to 2 different generator outputs that are on different instruments and not synced the ellipse will change shape over time.

Measurements Between Channels

Another way to measure the phase between two signals is by using the phase measurement in the measurement menu.

- Press "Measure" then "Add".
- Make certain that "Source A" and "Source B" are set up to be the two channels that you using to view the two sine wave signals.
- Change the Category tab in the pop-up menu to be "Other" then press "Phase (1 rising edge – 2 rising edge)". This will then display in the bottom of the display the phase difference between the rising edge of one channel to the rising edge of the second channel. See image below.