# FG33532A 2-Channel Function/Arbitrary Waveform Generator

**FG33532A series summary:**
Keysight's waveform generators offer users the ability to create and upload their own arbitrary waveform. In this lab, you will learn how you can generate an arbitrary w+aveform signal using Keysight's Waveform Generator. You will also learn to create and sequence arbitrary waveform using Keysight's Benchlink.

**Equipment list:**
- FG33532A 2-channel function/arb waveform generator
- 3x BNC cables
- 1x USB cable
- Oscilloscope or equivalent

**Demo guide conventions:**
- Hard keys represented in the following brackets [].
- Soft keys represented in underline.
- Diagram below are represented by **channel 1** and **channel 2** respectively in the demo procedure.

**Electronic Industrial Solutions Group Center of Excellence (EISG COE)**

# LAB Row 1

# Basic settings & function on function generator

## Setting up and syncing 2 channels:

Synchronizing Signals for Electronics Testing Imagine you are working for a consumer electronics company, and you need to test the synchronization of audio and video signals in a new product. You can use the FG33523A 2-Channel Function/Arbitrary Waveform Generator to generate a sine wave on one channel (representing audio) and a triangle wave on the other channel (representing video), with the sync output used to assure the audio and video stay aligned. By syncing these signals precisely, you can simulate real-world scenarios and ensure your product performs flawlessly, providing a seamless audio-visual experience to your customers.

1. Plug-in and turn on the FG33532A function generator and the scope. Using the three BNC cables, connect chan1 and chan2 of the FG33532A to chan1 and chan2 of the scope. Connect the FG33532A Sync to chan3 of the scope.

2. Press the chan1 button on the oscilloscope and press on impedance, then set the impedance to 1 MΩ (High Z).

3. On the FG33532A unit, press chan1's [Setup] then press Output Load followed by pressing Set to High Z. The chan1 will now be set for high Z output load to match the scope.

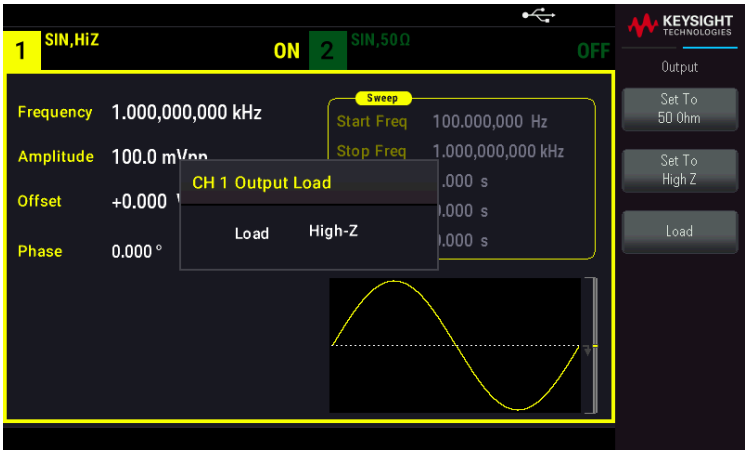



Figure 1 Output High Z setup

4. Press chan1's [On/Off] to turn chan1 output on.

5. Change the amplitude by pressing [Parameters] then Amplitude. Use the number pad to key in [4] followed by Vpp. Chan1 will now output a 4 Vpp sine wave.

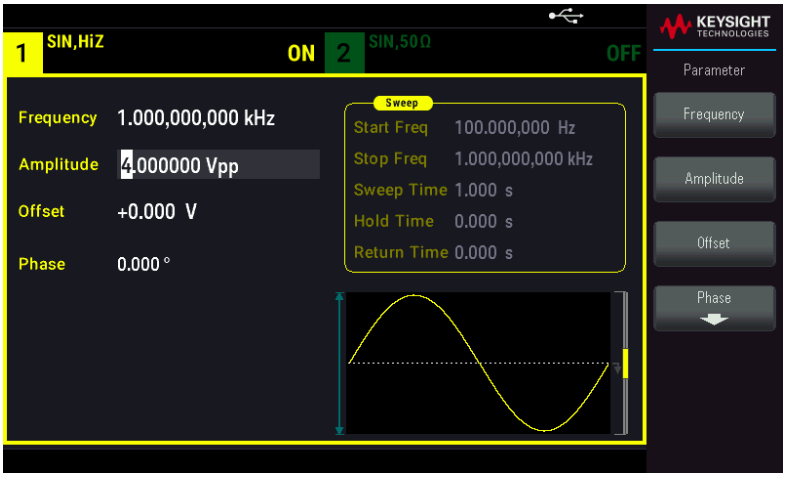



Figure 2 Amplitude Setup

6. Press chan2's [Setup] then press [Waveforms] and then More followed by pressing Triangle. Chan2 will now be set for triangle waveform.

7. Press the chan2 button on the oscilloscope and press on impedance, then set the impedance to 1 MΩ (High Z).

8. Press chan2's [Setup] then press Output Load and Set to High Z and press [On/Off] to turn chan2 output on.

9. Press [Parameters] then Amplitude and use the number pad to key in [4] followed by Vpp. Chan2 will now output a 4 Vpp triangle wave.

10. Ensure chan1, chan2, and chan3 on the scope are on. Press [Autoscale] on the scope and you should see what's shown below.

11. FG33532A chan1 is in yellow, chan2 is in green, and sync in blue (may vary depending on scope you are using).

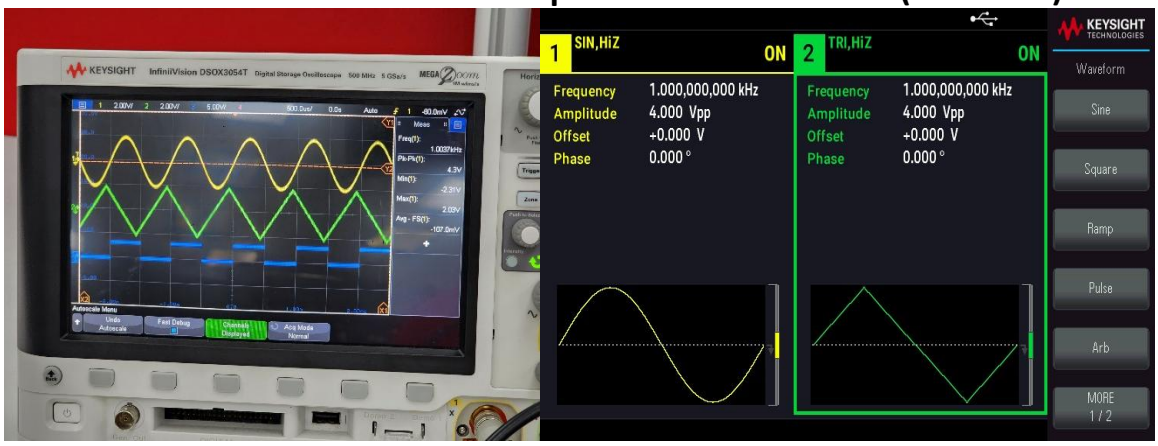



Figure 3 Oscilloscope Output (Unsynchronized)

***You may notice that the chan 1 and chan 2 waveforms in figure 1 are not synchronized. Perform the following to sync them up:***

12. On the FG33532A press chan2's [Setup] then [Parameters] then Phase. Now press Sync Internal - you will see the chan2 waveform sync (go in phase) with the chan1 waveform. The output waveforms are now synchronized.

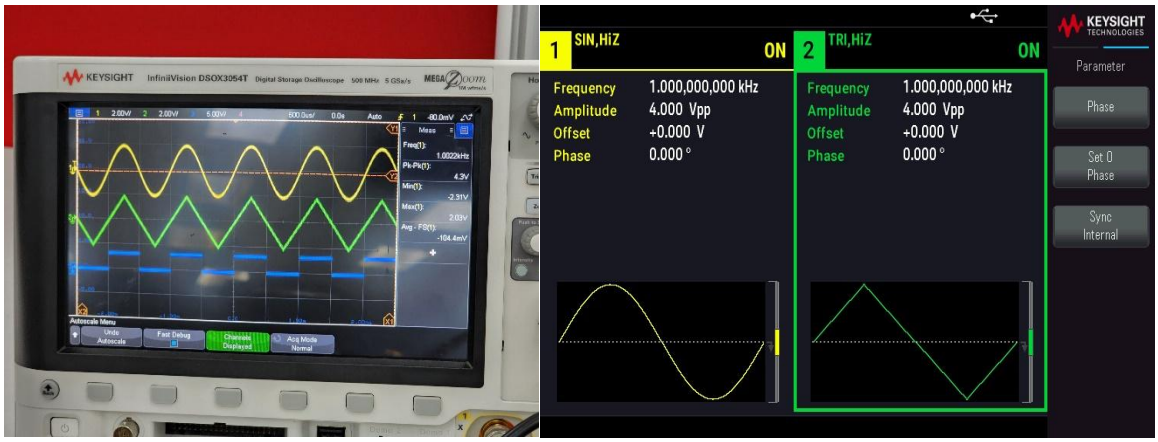



Figure 4 Oscilloscope Output (Synchronized)

# LAB Row 2
# Modulation Practice

**Modulation with focus on the sum features:**

Power Line Noise Simulation for Device Testing In the field of electrical engineering, you may be developing a device that needs to withstand power line noise effectively. Power lines often introduce voltage fluctuations and spikes. To test your device's immunity, you can use the FG33532A 2-Channel Function/Arbitrary Waveform Generator to create a stable carrier signal representing your device's power input. Then, you can use the Sum modulation feature to introduce simulated power line noise, imitating real-world conditions. This helps you verify that your device can operate reliably even in noisy environments.

1. Press [System] → Store/Recall → Set to Default to reset the instrument.

2. Chan1 of the FG33532A should be connected to chan1 of the scope with a BNC.

3. Press the chan1 button on the oscilloscope and press on impedance, then set the impedance to 1 MΩ (High Z).

4. Press chan1's [Setup] then Output Load then Set to High Z. FG33532A's chan1 is now set for high Z output load to match the scope. Press [Waveforms] then Square. Press chan1's [On/Off] to turn chan1 output on.

5. Change the amplitude by pressing [Parameters] and then Amplitude. Use the number pad to key in 5 followed by pressing Vpp.

6. Press [Autoscale] on the scope. Adjust the horizontal scale one notch to show 4 to 5 cycles.

7. On the FG33532A, press [Modulate] then press Type to change the modulation type. The FG33532A's modulation menu is now displayed which includes AM, FM, PM, FSK (Frequency Shift Keying), BPSK (Binary Phase Shift Keying), and Sum. The modulation source can be internal, external, or the other channel. This demo will show the Sum modulation feature. Sum allows you to add one of the built-in waveforms or an arb to the carrier signal.

8. Choose the Sum and press Sum Ampl. Enter in [5] and select Percent then go to Sum Freq and enter in [60] and select Hz. The FG33532A is now set up to add a 60 Hz sine wave that is 5% of the carrier amplitude. Press Modulate ON|OFF to turn the modulation on and view the resulting output on the scope. Adjust the horizontal scale to see the modulated carrier.

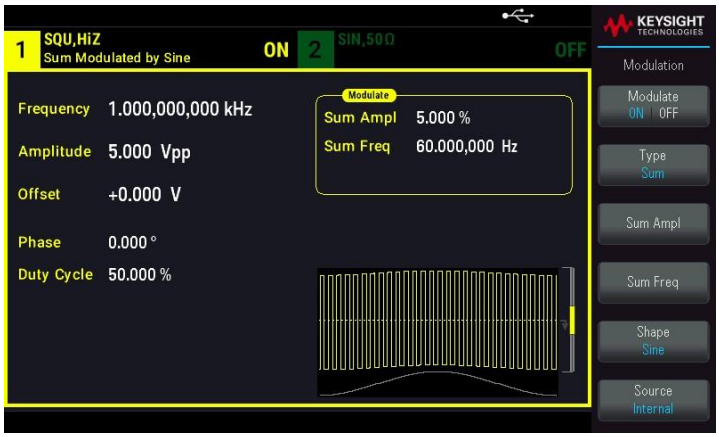



Figure 5 Sum Modulations Setup

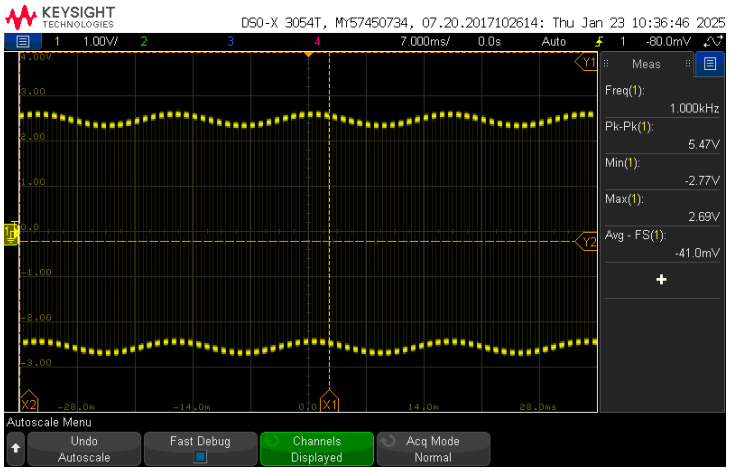



Figure 6 Oscilloscope Output (Sum Modulation)

This could be used to simulate power line noise on the carrier, to test the power line noise immunity of a DUT.

9. Press Shape then More and press Noise. Set Sum Ampl to 15% and Bandwidth to 30 MHz. A user would use this functionality to check their DUT's noise immunity.

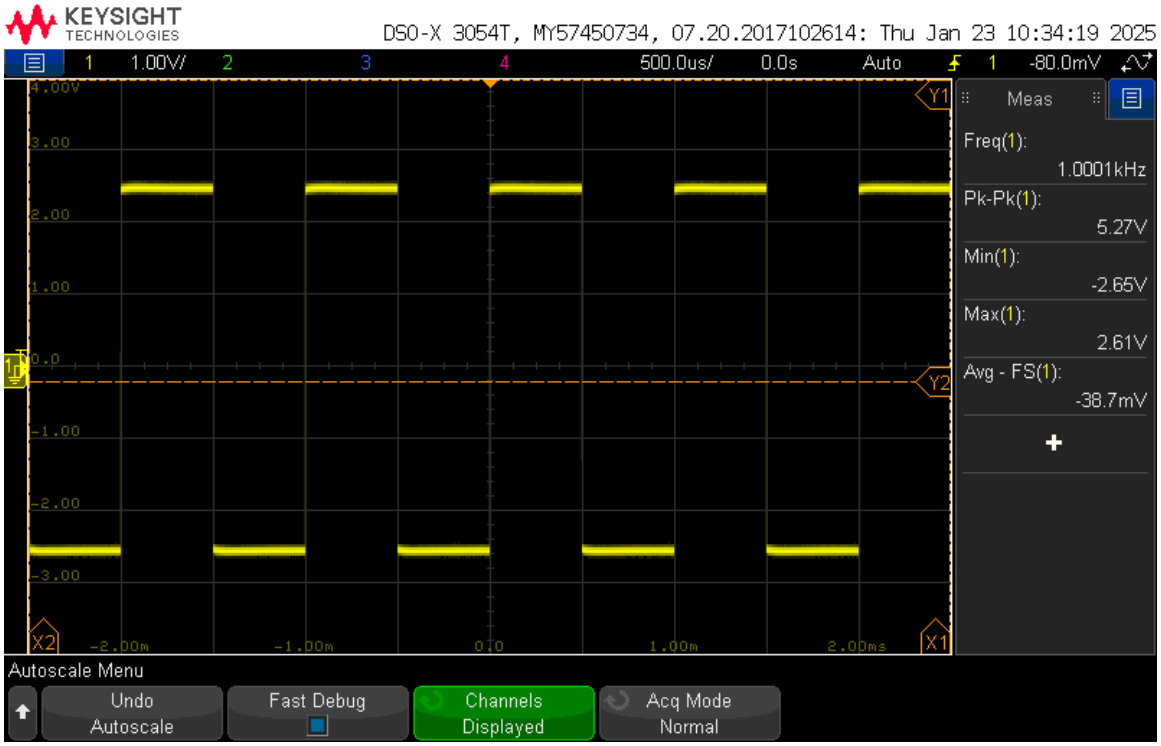



Figure 7 Carrier with noise BW of 30 MHz at 15% amplitude

# LAB Row 3

# Generating Arbitrary Waveform Using A Function Generator

The arbitrary waveform generator (AWG) function allows you to set up your function generator to supply a customized complex signal for the specific application. In R&D, AWGs are used to test prototypes. Another common application is troubleshooting, injecting known signals into a circuit and observing the output.

A specific use for function generator is in double pulse testing by producing the precisely timed voltage pulses required to evaluate the switching performance of power semiconductor devices like MOSFETs and IGBTs. In this testing method, the function generator creates two distinct pulses with adjustable widths and a controlled dead-time between them, enabling the study of the device's turn-on, turn-off, and dynamic behaviors under load. These signals are then used to drive the gate of the device under test (DUT), while the resulting voltage and current waveforms are analyzed to measure switching losses, reverse recovery effects, and other critical characteristics. By integrating seamlessly with oscilloscopes and other measurement tools, function generators ensure accurate and repeatable testing, which is essential for optimizing the performance of power devices in real-world applications.

First set up the device by connecting the function generator to an oscilloscope. The connection is simple, just attach the oscilloscope probe to the function generator output as shown in the diagram below. The oscilloscope will allow us to observe the output waveform and verify the AWG functionality.


Figure 8 Diagram Connection

**Electronic Industrial Solutions Group Center of Excellence (EISG COE)**
**Setting up the FG33532A to Load in Built-in Arbs**

1. To access the waveform menu, press [**Waveform**].

2. Press the **Arb**, followed by the **Arbs**.

3. Press **Select Arb** and select the internal storage, here there are 9 built-in arbitrary waveform.

4. Select any of them, in this case select the cardiac arb.

5. Set the **Sample Rate** to 225MSample/s and the **Amplitude** to 4 Vpp.

6. Set the oscilloscope [**Horizontal**]scale to 600ns/div and the [**Vertical**] scale to 1V/div.

7. Press [**ON**] on the function generator and press run on the oscilloscope to visualize the built-in arbitrary waveform.

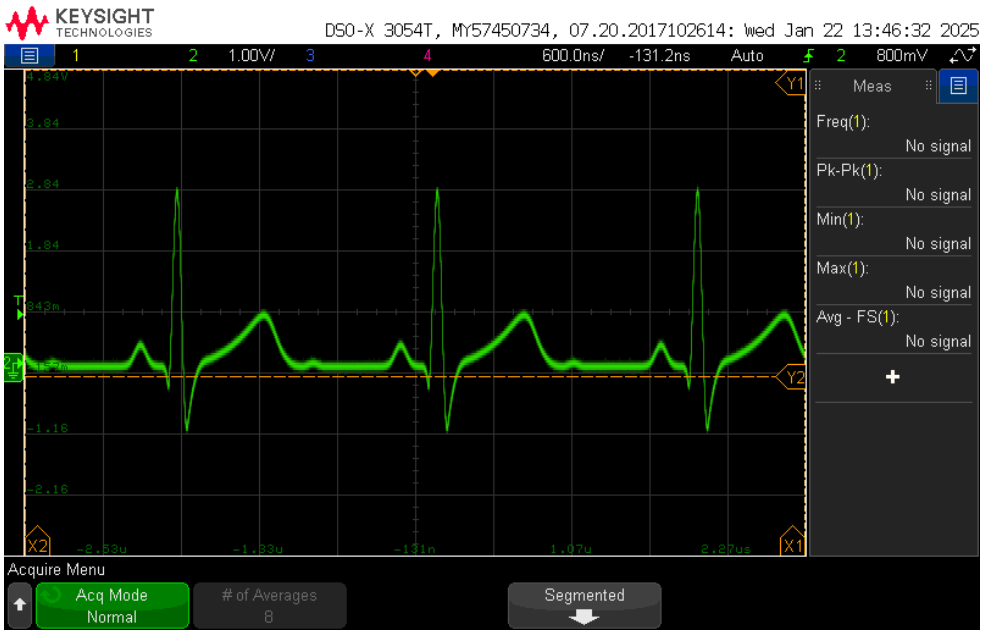



Figure 9 Arbitrary Waveform oscilloscope view

# LAB Row 4

# Double Pulse Test Waveform Generation Using Keysight Benchlink Waveform Builder Pro App

Double pulse test is a standard method used to characterize the dynamic switching performance of power transistors or power stages in ICs. This technique is widely used in power electronics to measure switching losses, voltage overshoots, recovery characteristics, and overall transient behavior under specific conditions. Technically, the double pulse test focuses on analyzing the behavior of power devices during critical switching events: turn-on and turn-off. These events are fundamental in understanding energy losses, thermal stress, and reliability in high-performance applications.

The double pulse test involves applying a controlled sequence of two pulses to the device under test (DUT). The waveform generated simulates real operational conditions, enabling detailed performance characterization. The first pulse magnetizes the load inductor to a target current level for simulating realistic operation. To prevent incomplete recovery that could distort measurements or damage the DUT, there is a recovery period after the first pulse. Following the brief off period, the second pulse measures how the device behaves under nominal current flow, capturing critical switching parameters like switching losses, peak voltages, and recovery times.

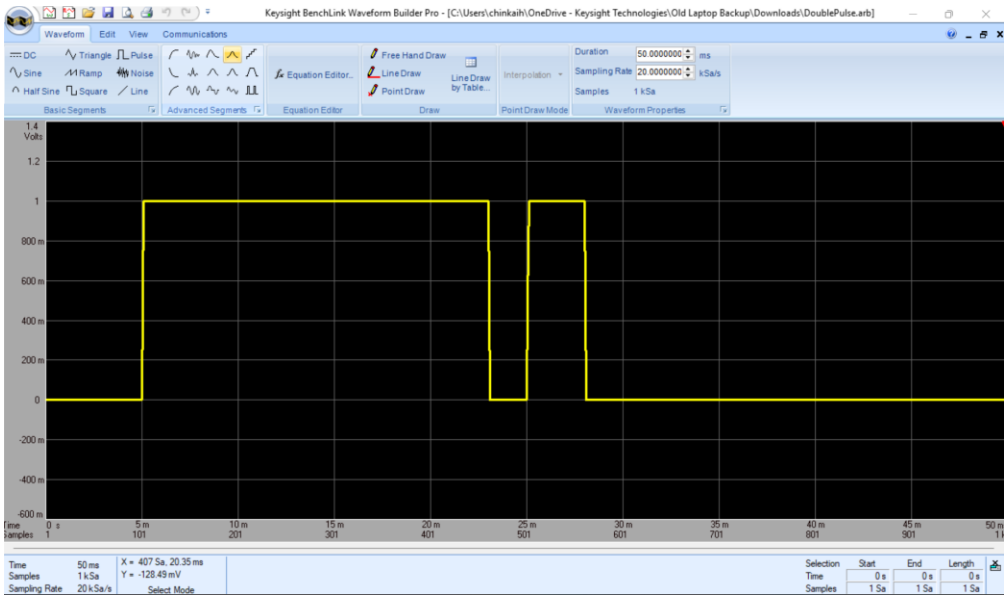


Figure 10 Double Pulse Waveform

1. Install the Benchlink Waveform Builder App in Benchvue.

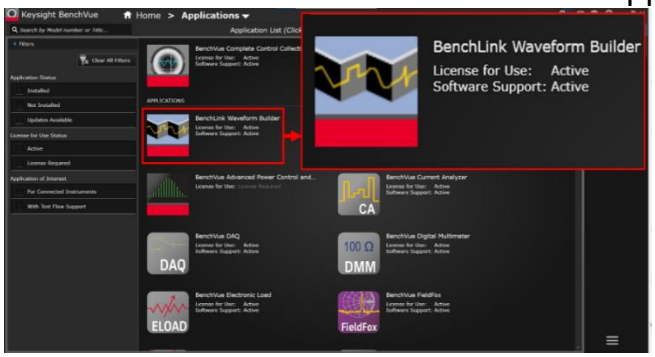



Figure 11 Benchlink Waveform Builder App

2. Launch the Benchlink software and press **New Waveform** and select **FG33532A** as instrument model. Set the duration to 50ms and sampling rate to 20kSa/s. Press **OK** to continue.

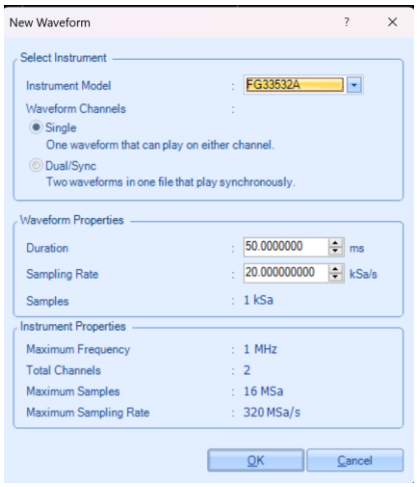



Figure 12 New Waveform

3. Click on **DC** segment to create an initial state of Play. Click OK.

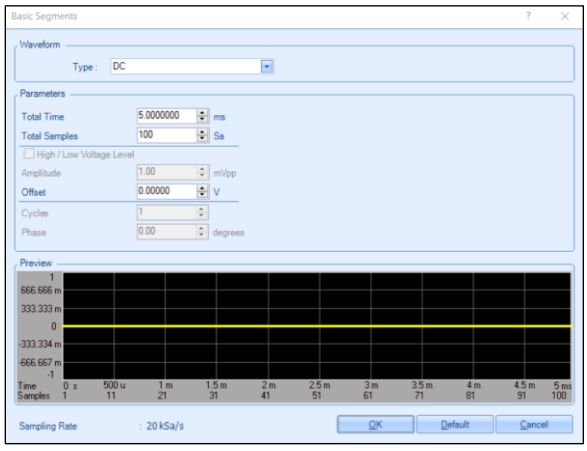



Figure 13 DC Parameter

4. Click on **Pulse** to generate the first pulse with a pulse width of 18ms and total time of 20ms. Click **OK**.

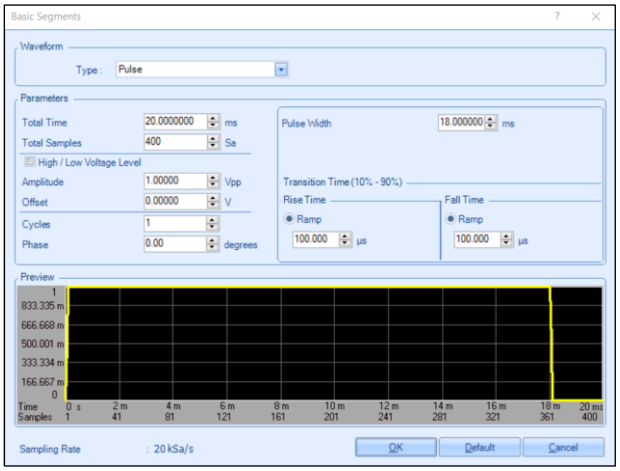



Figure 14 First Pulse Parameters

5. After creating the initial pulse, we create a second instance of a pulse to simulate the secondary pulse of a double pulse waveform. Click on **Pulse** again. Set the total time to 25 ms, and the pulse width is set to 3 ms. Click **OK**.

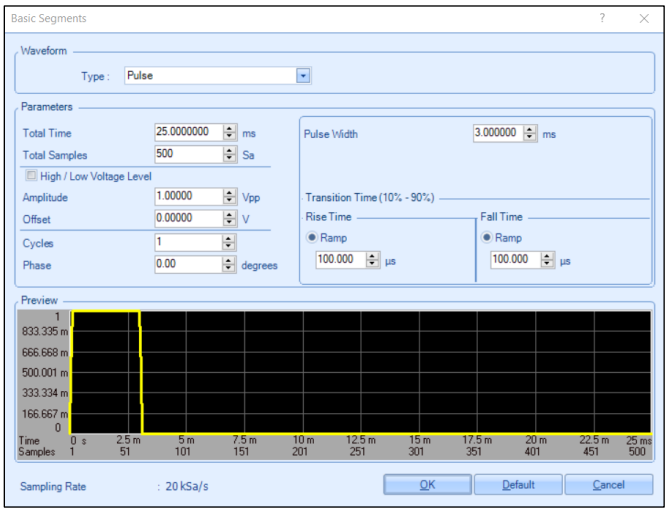
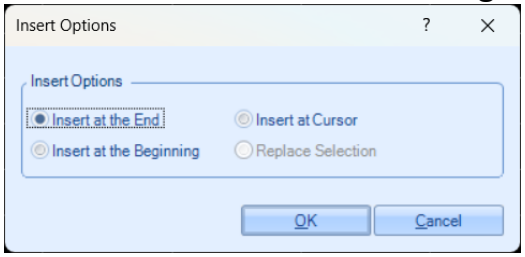



Figure 15 Second Pulse Parameters

6. Click **Insert at the end** for all segments. Click OK.



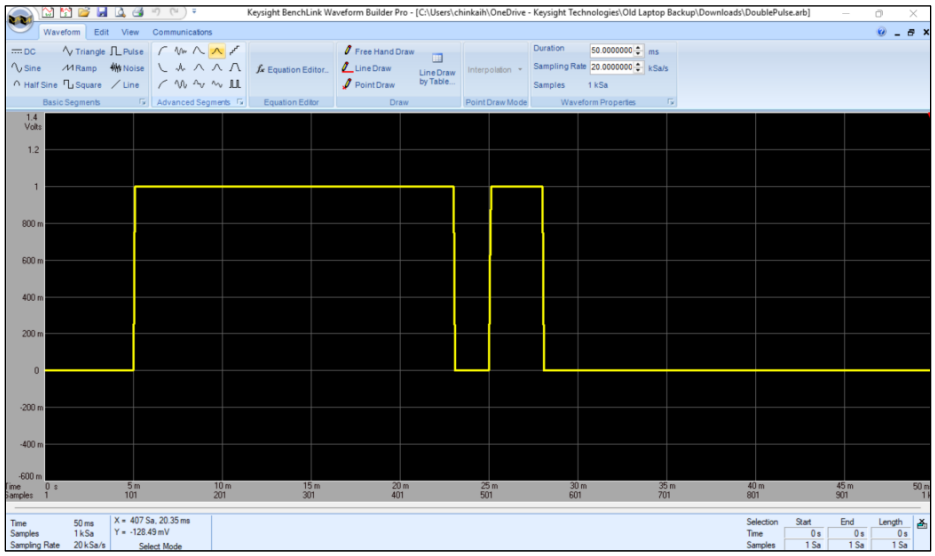


Figure 16 Double Pulse Signal

7. To send the arbitrary waveform to the function generator click on **Communications** in top left corner and press **Connect.**

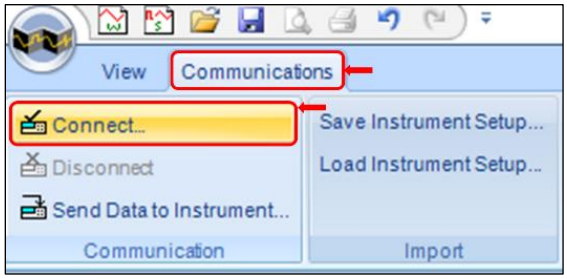



Figure 17 Connecting to Function Generator

8. Connect your computer to the function generator using a USB cable, once connected the function generator will pop up in the select instrument window. Select the function generator and click on **Connect**.

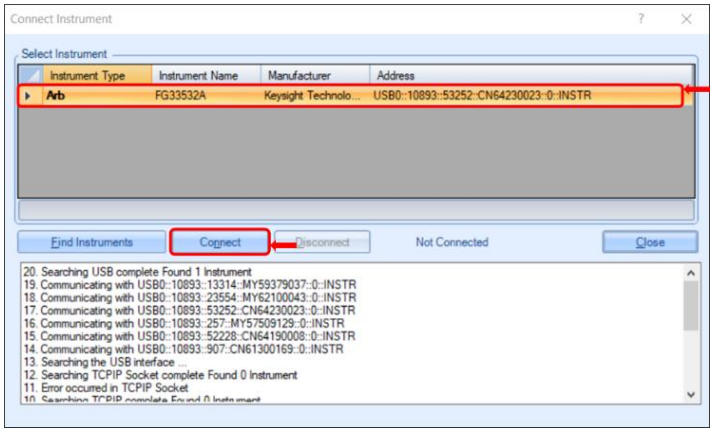



Figure 18 Connecting to Function Generator

9. Close the select instrument window and click **Send Data To Instrument**.

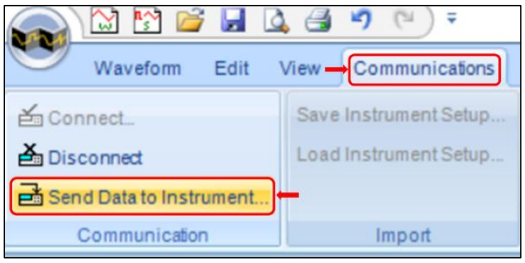



Figure 19 Connecting to Function Generator

10. Ensure that the correct waveform is selected, before clicking **Send** the High Level is set to <u>5 V</u> to simulate binary condition where 5 V is 'HIGH' logic and 0 V is 'LOW' logic.

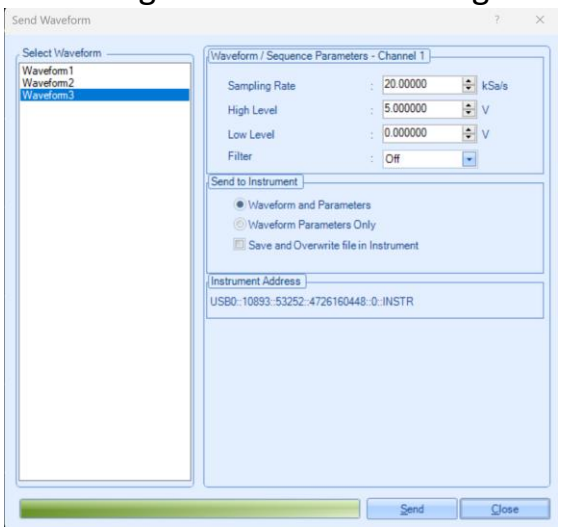



Figure 20 Connecting to Function Generator

11. Once the send is complete the function generator should now be showing the double pulse arbitrary waveform.

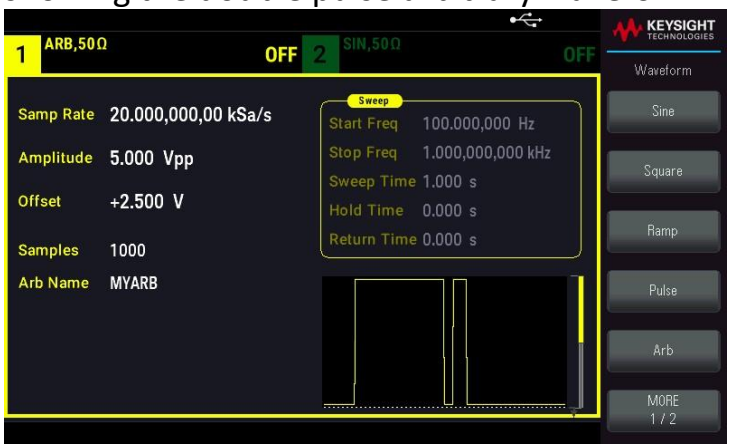



Figure 21 Function Generator with Double Pulse Signal

12. Connect the oscilloscope to the function generator using a probe.

Figure 22 Instrument setup

13. Press **On/Off** on the function generator to start supplying the waveform.
14. Press on **1** then Coupling to set the oscilloscope coupling to DC.
15. Press **Start** on the oscilloscope to start the capturing waveform.
16. Press **Stop** on the oscilloscope to observe the captured waveform in detail.
17. Adjust the x-axis and y-axis scaling using the Vertical and Horizontal knob to get a clearer look at the waveform.

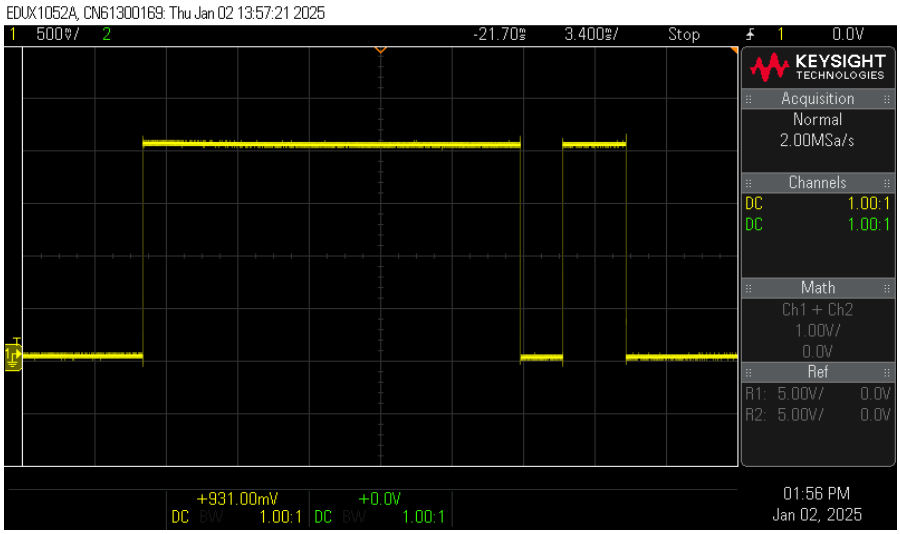



Figure 23 Double pulse signal captured from oscilloscope