Tektronix®

AWG4162
Arbitrary Waveform Generator

Printable Help

077-1189-00

Tektronix®

AWG4162
Arbitrary Waveform Generator

Printable Help

www.tek.com
077-1189-00

Compiled Help part number: 076-0387-00

Contacting Tektronix

Tektronix, Inc.
14150 SW Karl Braun Drive
P.O. Box 500
Beaverton, OR 97077
USA

For product information, sales, service, and technical support:

- In North America, call 1-800-833-9200.
- Worldwide, visit www.tek.com to find contacts in your area.

Table of Contents

Preface

Your Tektronix AWG4162 Arbitrary Waveform Generator is a convergent waveform generator with full function AFG (Basic) and AWG (Advanced) modes. Basic mode supports basic arbitrary and function waveform generation. Advanced mode has an adjustable sampling rate and supports both DDS mode and arbitrary mode generation, each of which supports sequence, continuous, gated, and trigger modes.

This document describes the Basic Application mode operation.

Documentation

The following table lists related documentation available for your AWG4162. The documentation is available on the Tektronix Web site (www.tek.com/manuals).

Item	Purpose	Location
Compliance and Safety Instructions	Compliance, safety, and basic installation information	Printed and shipped with your instrument
Advanced Application Help	Advanced Application operating information	On instrument and available as a PDF
Basic Application Help	Basic Application operating information	On instrument and available as a PDF
Programmer Manual	Programming syntax and command information for remotely controlling the instrument	Available as a PDF
Service Manual	Instrument servicing procedures and replaceable parts list	Available as a PDF
Specifications and Performance Verification Technical Reference	Instrument specifications and performance verification procedures	Available as a PDF
Declassification and Security Instructions	Describes how to sanitize, secure, and declassify the instrument	Available as a PDF

Getting Started

General Features

- Two working modes
  - Basic (DDS) mode
    - Two analog channels
    - 600 MHz sine waveforms
    - 2.5 GS/s, 14-bit, 16 kpts arbitrary waveforms
    - Amplitude up to 5 Vp-p into 50 Ω load
  - Advanced (Arbitrary) mode
    - Two analog channels
    - 16/32-bit digital channels (optional)
    - 1/16/32/64 Mpts per channel arbitrary waveform memory (optional)
    - Up to 750 MHz bandwidth
    - SFDR < -60 dBc
- Variable sampling rate range from 100 S/s to 2.5 GS/s, with 14-bit vertical resolution, ensures signal integrity in all aspects
- Designed for 100% user-conducted upgrades and configurations, all options activated through SW key
  - Optional and upgradable arbitrary waveform memory up to 64 Mpts for each analog channel and 32 Mbit for each digital channel for long waveforms
  - Optional 16-32 channel digital outputs. Purchasing SW option includes the shipment of digital probe accessory.
- Dual analog channels and up to 32-bit digital channels, ideal for mixed signal circuit designs
- Sync-in and Sync-out interfaces enables the synchronization of multiple units in a daisy chain, to extend the number of output channels
- Digital outputs provide up to 1.25 Gb/s data rate creates high speed digital pattern in parallel
- One marker out for each analog channel for triggering and synchronization
- Three software-configurable output paths fit all test cases
  - Direct DAC mode: 750 MHz bandwidth with differential output
  - AC coupled mode: 750 MHz bandwidth with single ended output for RF applications
  - Amplified mode: 5 Vp-p amplitude 400 MHz bandwidth with differential output
- Full functional sequence with up to 16384 user defined waveforms provides the possibility of generating complex signals with the best memory usage, in the form of loops, jumps, and conditional branches
- Channel 1 and 2 (together with the corresponding digital output channels) can work independently on different sampling clocks and sequences
- Direct communication with RFXpress® for easy waveform generation in RF applications
- Windows based platform with 10.1-in touch screen, front panel buttons, keyboard, and mouse
- Compact form factor, convenient for bench top and portability Removable hard disk guarantees the security of confidential data
- USB 3.0 and LAN interfaces for remote control

Operating Requirements

Power Supply

<table>
<tr><td>Source Voltage and Frequency</td><td>100 to 240 V RMS@ 50-60Hz
115VRMS@400Hz
<table>
<tr><th>Characteristic</th><th>condition</th><th>Min.</th><th>Nom.</th><th>Max.</th><th>Units</th></tr>
<tr><td>Voltage Amplitude</td><td>45-66Hertz</td><td>85</td><td>100-240</td><td>264</td><td>VRMS</td></tr>
<tr><td></td><td>360-440 Hertz</td><td>100</td><td>115</td><td>132</td><td>VRMS</td></tr>
<tr><td>Voltage Wave shape</td><td>All</td><td colspan="3">Sine</td></tr>
</table></td></tr>
<tr><td>Power Consumption</td><td>Maximum: 150W
Measured: 125W</td></tr>
<tr><td>Surge Current</td><td>30 A peak (25°C) for ≤ 5 line cycles, after product has been turned off for at least 30 s.</td></tr>
</table>

Mechanical Characteristics

Net Weight	14.2lbs (6.5 kg)
Net Weight with Package	25.2lbs (11.5kg)
Overall Dimensions	Height: 233 mm Width: 439 mm Depth: 199 mm
Dimensions with Package	Height: 498 mm Width: 457 mm Depth: 574 mm
Clearance	The clearance requirement for adequate cooling is 2.0 in (50.8mm) on the left side (when looking at the front of the instrument) and on the rear of the instrument.

Environmental Characteristics

Temperature	Operating +5 °C to +50 °C (+41 °F to 122 °F) Non-operating -20 °C to +60 °C (-4 °F to 140 °F)
Humidity	Operating 8% to 90% relative humidity with a maximum wet bulb temperature of 29 °C at or below +50 °C, non-condensing Non-operating 5% to 98% relative humidity with a maximum wet bulb temperature of 40 °C at or below +60 °C, non-condensing
Altitude	Operating 3,000 m (9,843 feet) Non-operating 12,000 m (39,370 feet)

Standard Accessories

Item	Description	TPN
Manual	COMPLIANCE AND SAFETY INSTRUCTIONS	071345100
Product CD	Document CD with Browser Including the PDF files of Specs & PV Tech Ref, user manual, programmer, service manual.	063457000
ArbExpress	Application S/W and Instructions	063376310
Power Cable	-	
USB cable	-	174440100
Stylus	For touch panel	119610700
Front protect cover	-	200513000
Accessory Pouch	-	016202900
50Ω SMA Terminator	Male, DC-18GHz; 1 ea / channel	136716200
Certification of Calibration	-	001138701
Tree year warranty	-	

Recommended Accessories

Item	Description	TPN
Pin Header SMA Cable	45 inch	174619300
RMD5000	- Rack mount kit - Instruction sheet (English)	RMD5000
Manual	Service (English) Specs & PV Tech Ref Programmer manual	077-1199-00 077-1197-00 077-1198-00
AWG4HDDE	- Hard disk drive	AWG4HDDE
SMA terminator	50 Ω	136716200
AWG4SYNC	Sync cable; Used for multiple instruments synchronization	AWG4SYNC
RFX100	RFXpress software	RFX100
AWG4DIG16LVDS	16-bit digital output cable; Used for LVDS	AWG4DIG16LVDS
AWG4DIGSCKT	Digital output connector; AWG4k Digital Channel Connector on DUT (Amphenol, U65-B12-40E0C)	AWG4DIGSCKT
TEK-USB-488	GPIB to USB adaptor	TEK-USB-488
HCTEK54	Hard transit case	HCTEK54

Power the Instrument On and Off

Power On

- Insert the AC power cord into the power receptacle on the rear panel.
- Use the front-panel power button to power on the instrument.
- Wait until the system shows windows desktop.
- You have two selections to open the applications:

You can press 

or 

button on front panel to launch one application. You can also click the shortcut icon 

or 

on desktop to launch any one of them.

NOTE. *Only one application can be launched at a time. If you want to launch the other application, first close the one in use.*

Power Off

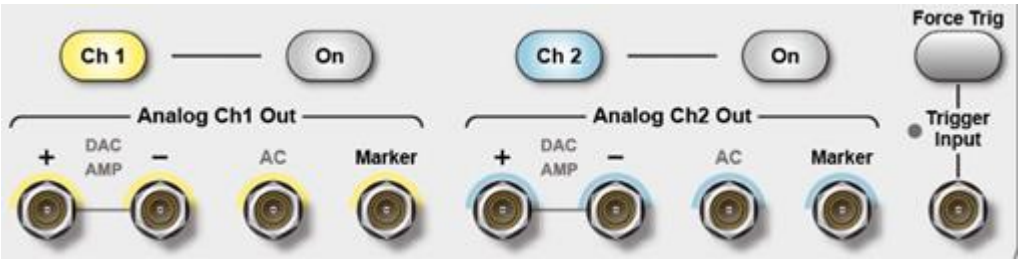
- Close the application in use.
- Press the front-panel power button  to power off the instrument.

  You can also use the Windows menu to shut down the instrument.

Protect Your Instrument from Misuse

Check Input and Output Connectors

When connecting a cable, be sure to distinguish the input connector from the output connectors to avoid making the wrong connection.



The instrument has both input and output connectors on the front panel. When connecting a cable, be sure to distinguish the input connectors from the output connectors.

CAUTION. *Do not short output pins or apply external voltages to Output connectors. The instrument may be damaged.*

CAUTION. *Do not apply excessive inputs over +10 V to Trigger Input connector. The instrument may be damaged.*

CAUTION. *For differential analog output when one connector is used as single-ended output, another connector should be terminated with a 50 Ω terminator.*

Obtaining the Latest Application and Version Releases

The latest version of an optional application that you ordered with your instrument may not be installed on your instrument. The following download location is a fast and easy way to get the latest software version.

To download the latest version of software, go to the home page of the Tektronix Web site (www.tek.com), and locate the Downloads section on that page. Enter the application name in the Search text box, and select Software in the Select Download Type pull-down menu.

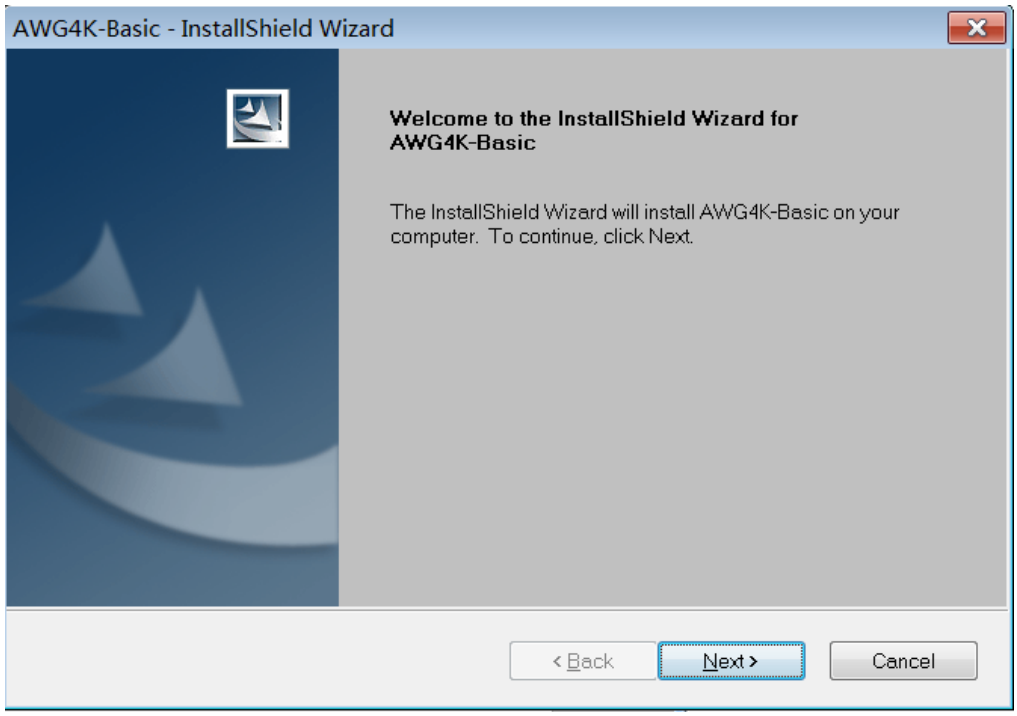
To define the search criteria, use the title of the application in the Search text box. For example, use the keyword AWG4162 to search for and download the latest version of AWG4162 software.

Install Basic APP

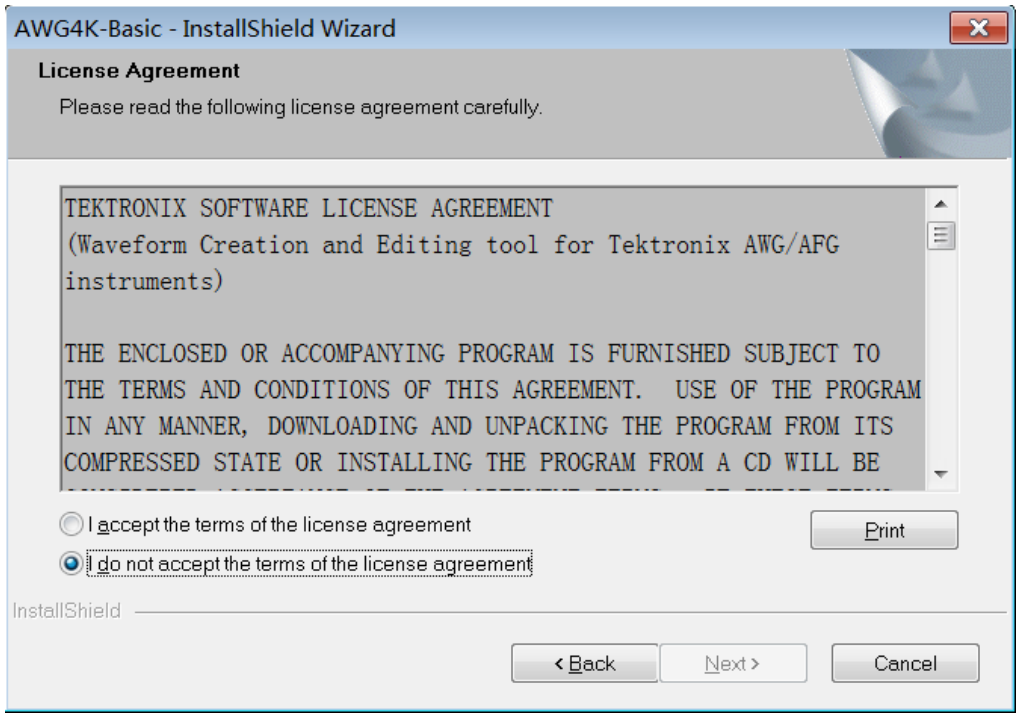
If your instrument has already installed another version of Basic APP, you must first uninstall it. You can find uninstall details in the “Uninstall Basic APP” section.

1. Download Basic APP setup package from Tektronix website and decompress it to instrument’s local disk.

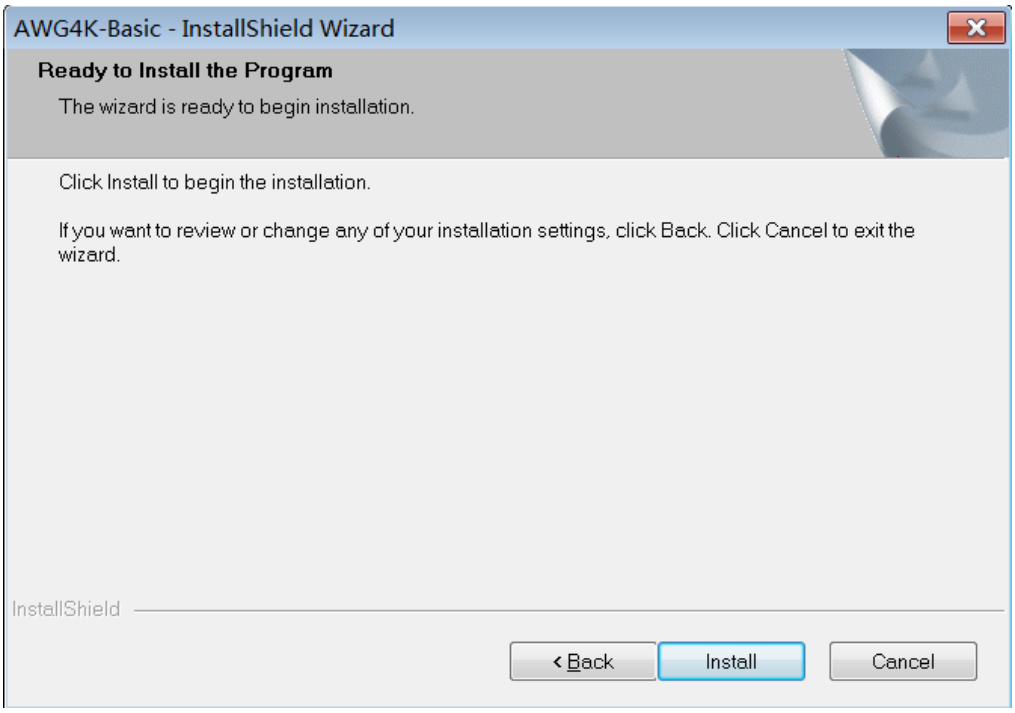
2. Double click setup.exe to start the install. When you see the welcome page click **Next**.



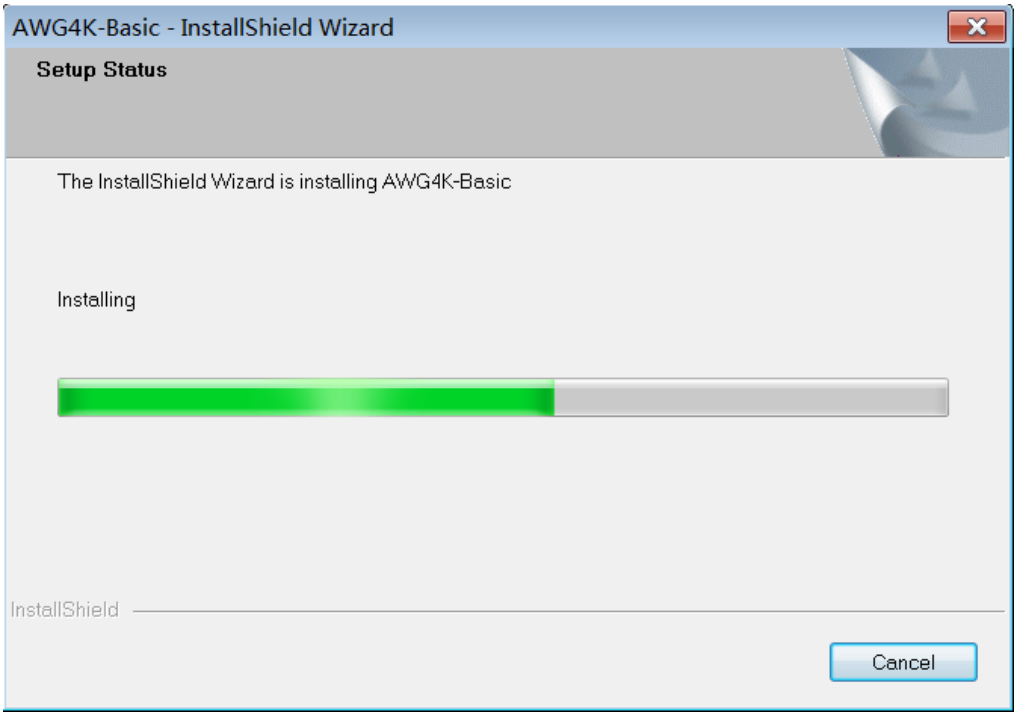
3. Select accept on the License Agreement page and then click **Next.**



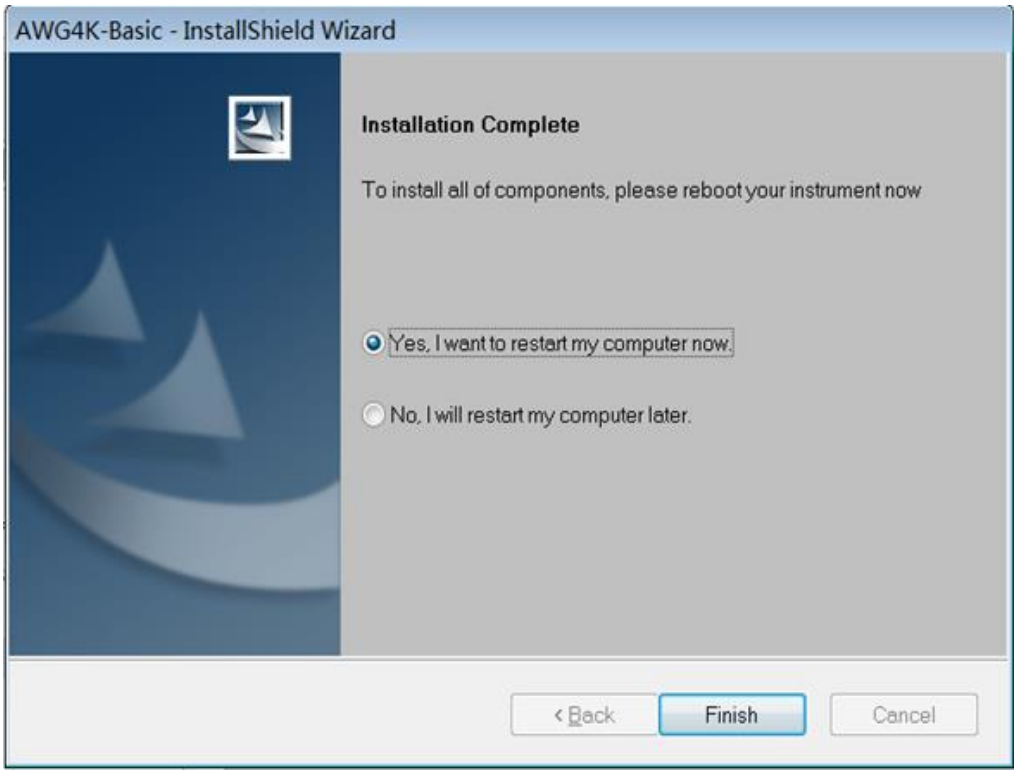
4. Press **Install** to start installation.



5. Installation will begin and the instrument will show installation progress.



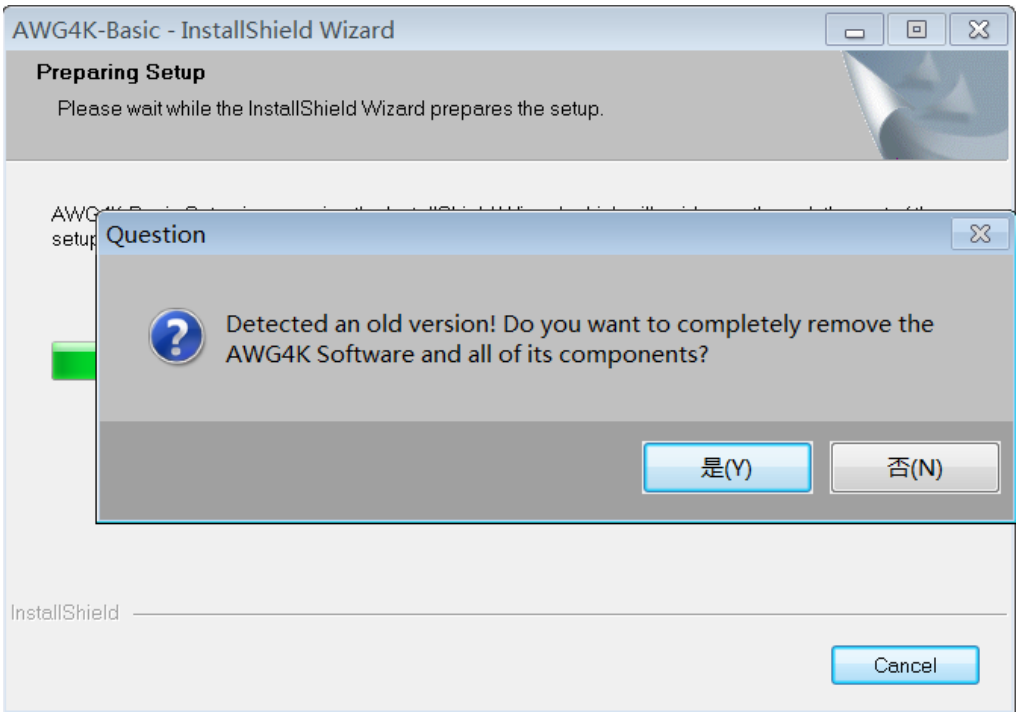
6. When “Installation Complete” appears, press **Finish** to restart the instrument.



Uninstall Basic APP

You can use Basic APP setup package to uninstall the Basic App in following steps.

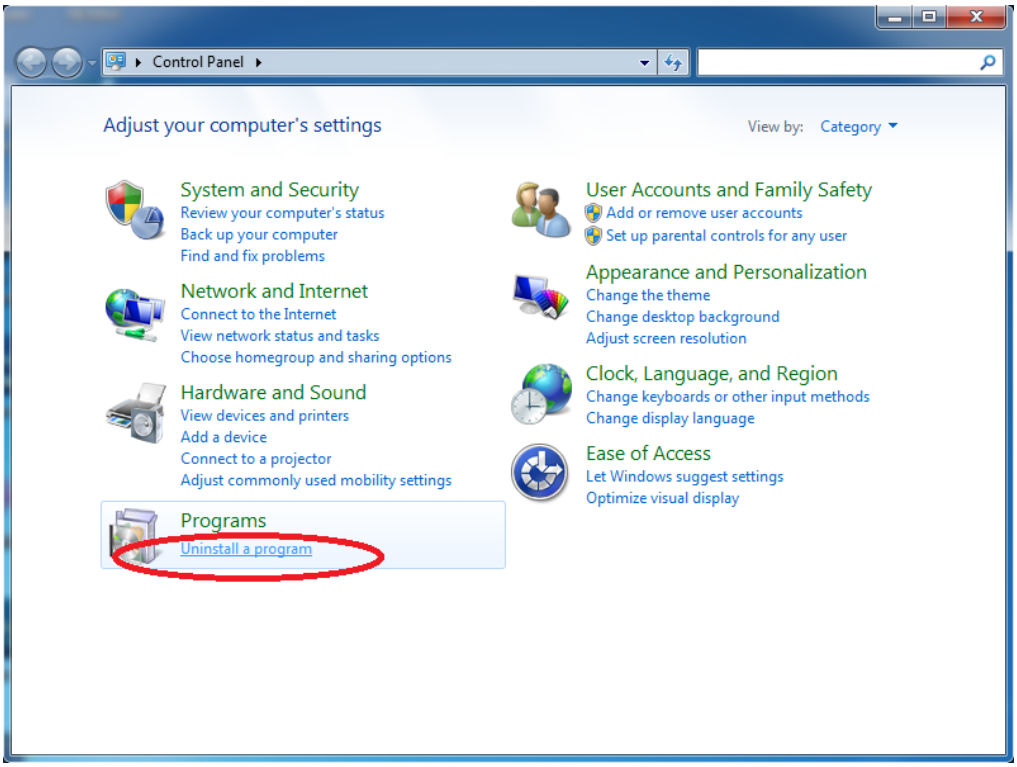
1. Download Basic APP setup package and decompress to instrument’s local disk.
2. Double click setup.exe. The welcome dialog notices you to remove installed version Basic. Select “Yes” to start uninstallation.



3. Uninstallation begins, instrument shows progress and exit automatically.

Besides using setup package, you can also use **Windows Control Panel** tool to do Basic APP uninstallation by followed steps:

1. Enter uninstall page through path: Start → Control Panel → Uninstall a program



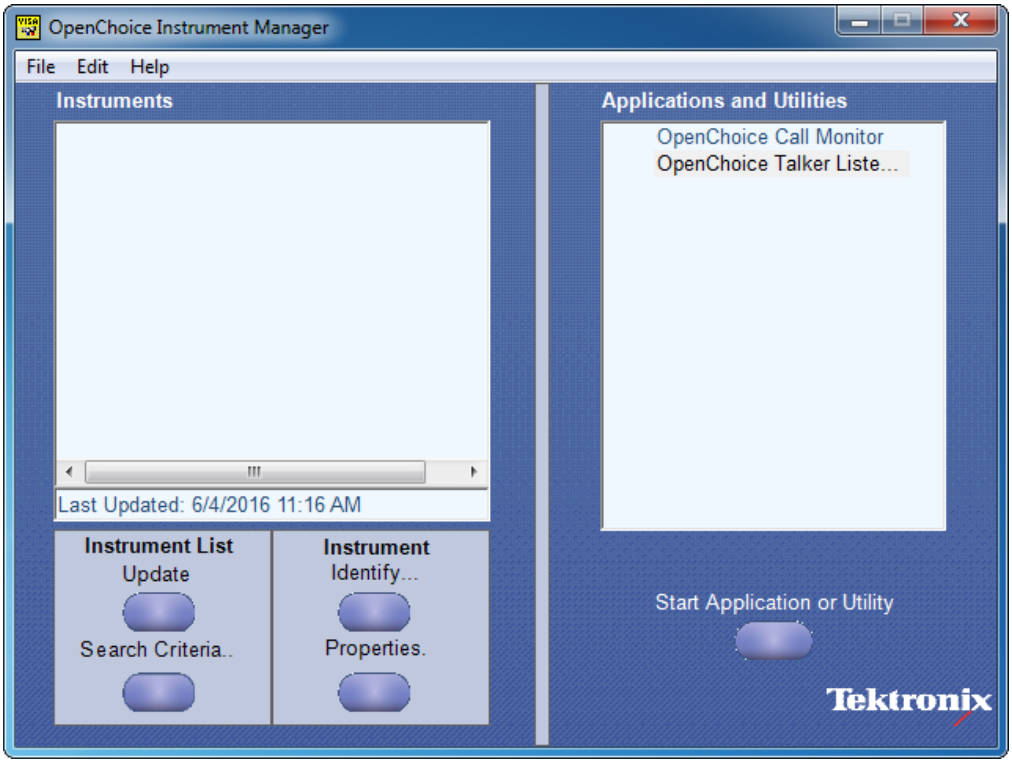
2. In Uninstall or change a program page, please select “AWG4000 Basic” program, and uninstall it.
3. Wait until uninstall is finished.

Remote Control

You can connect your instrument to a network for printing, file sharing, and Internet access, among other functions. Consult with your network administrator and use the standard Windows utilities to configure the instrument for your network. For LAN configuration, use the LAN Configuration dialog box from control panel.

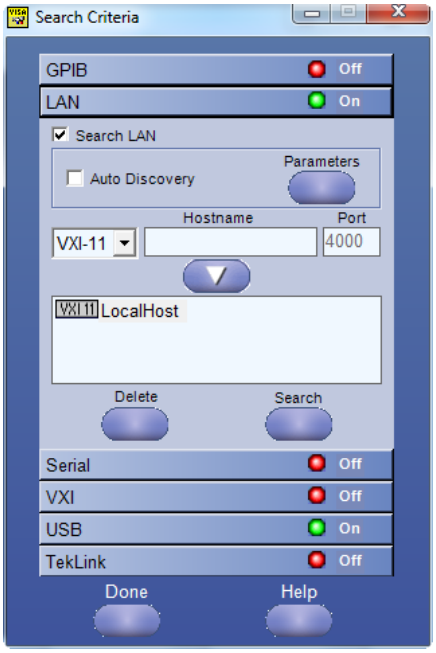
The instrument can be controlled using VXI-11 (LAN) or USBTMC protocols. It allows you to control the instrument remotely by using SCPI commands. Please refer to the AWG4162 programmer manual for a complete description about all available commands. You can follow the next steps to communicate with your AWG4162 instrument:

1. Connect your LAN cable or USB to the instrument.
2. On the Client-PC (IP Address)or AWG4162(LocalHost), launch the Tek OpenChoice Instrument Manager window.

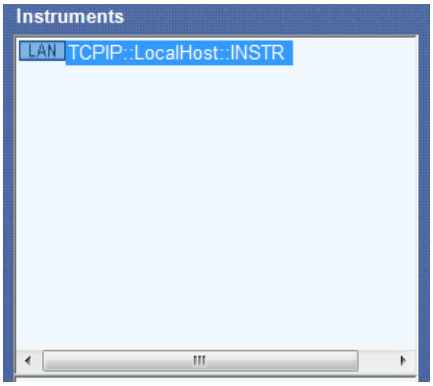


3. Press **Search Criteria…** button 

and enable **LAN** and **USB**. Input IP Address if on Client-PC or LocalHost if on AWG4162 into **Hostname**, then press **Search** for searching optionally. You can also enable **Auto Discovery** for searching all the available instruments connected in LAN. Then press **Done**.

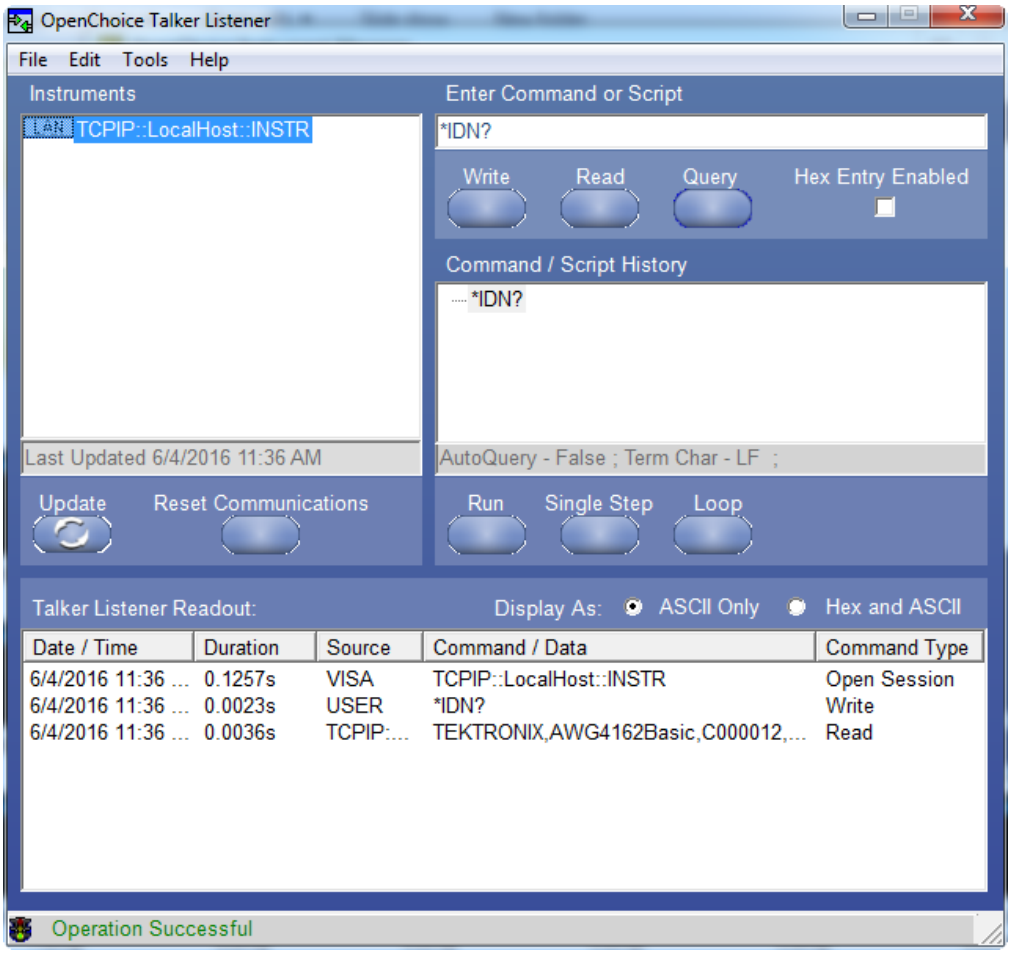


4. Check the **Instruments** list to verify if the AWG4162 has been correctly detected.



5. Press the **Start Application or Utility** button 

to open OpenChoice Talker Listener and send a *IDN? Command.



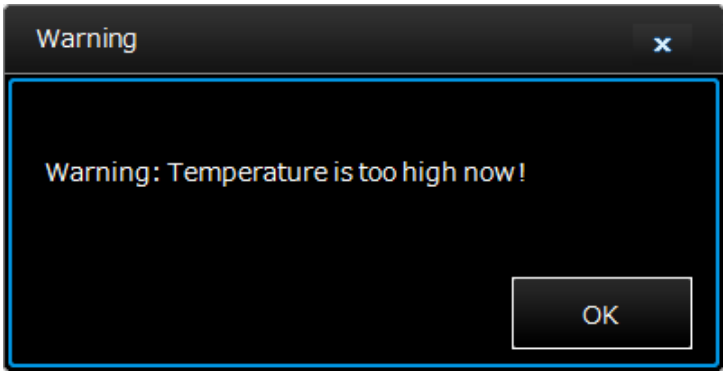
6. The instrument should respond like this:

TEKTRONIX,AWG4162Basic,C0000012,SCPI:99.0,FV:1.0, where C0000012 is the serial number and FV:1.0 is the Application version.

7. You can also load an exist script to run in TekVISA. Please see TekVISA Talk/Listener help for more details.

Overheat Protection

The instrument internal temperature is monitored in AWG4162. A warning message will appear if the internal temperature reaches a threshold level, and the instrument will automatically power off.

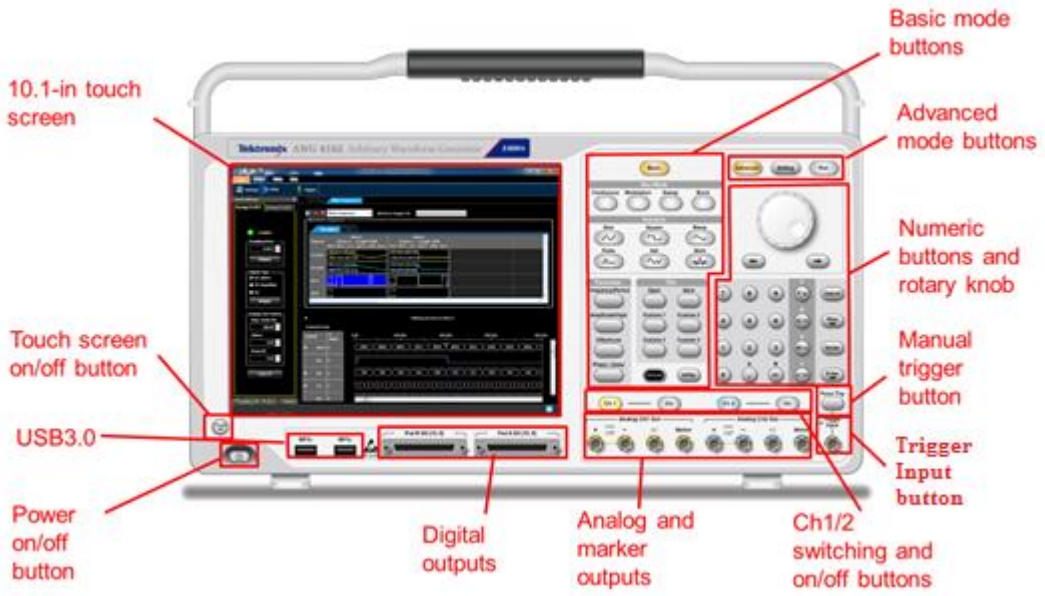


If the warning message appears, check for following conditions:

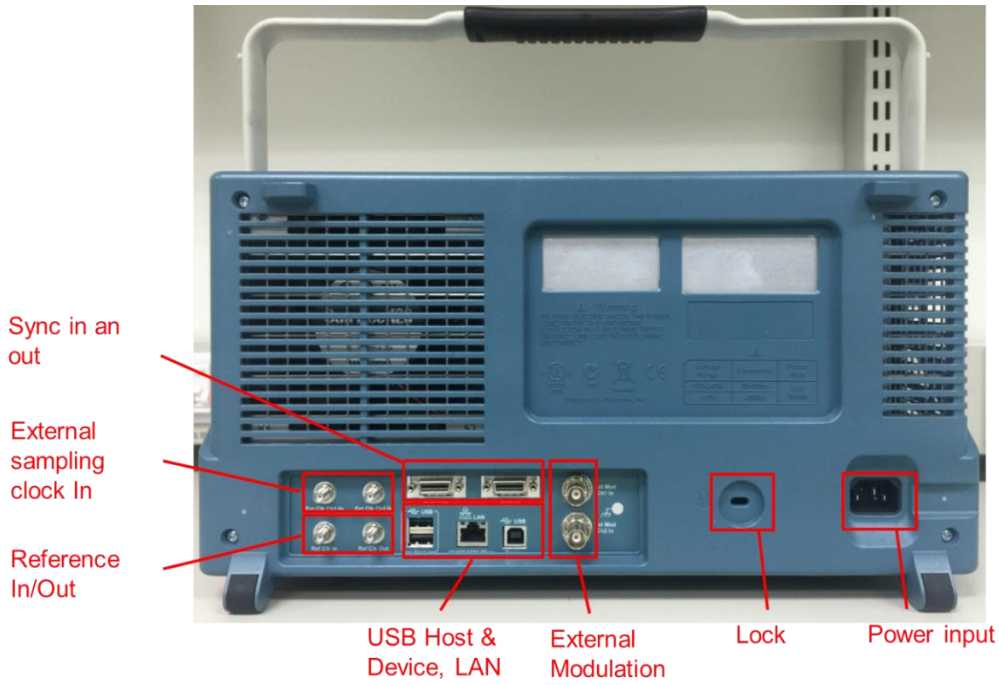
- The ambient temperature requirement is being met.
- The required cooling clearance is being met.
- The instrument fan is working properly.

Getting Acquainted with Your Instrument

Front Panel



Rear Panel



Basic Application Overview

Introduction to Basic Mode

Running the AWG4162 in Basic mode allows you to easily generate function, pulse, and arbitrary waveforms. Select from 12 standard waveforms (Sine, Square, Ramp, Pulse, Sin(x)/x, Noise, DC, Gaussian, Lorentz, Exponential Rise, Exponential Decay, and Haversine). You can also create and save custom setups, define your own arbitrary waveforms, and create modulated waveforms. The following table shows the combination of modulation type and the shape of the output waveform.

Run mode	Sine, Square, Ramp, Arb, Sin(x)/x, Gaussian, Lorentz, Exponential Rise, Exponential Decay, Haversine	Pulse	Noise, DC
Continuous	X	X	X
Modulation			
AM	X		
FM	X		
PM	X		
FSK	X		
PSK	X		
PWM		X	
Sweep	X		
Burst	X	X	

NOTE. *When the instrument outputs an Arb waveform, V_{p-p} of instrument setup indicates the Vp-p value of normalized waveform data. When the instrument outputs Sin(x)/x, Gaussian, Lorentz, Exponential Rise, Exponential Decay, or Haversine, V_{p-p} is defined as twice the value of 0 to peak value.*

Instrument Control

This instrument has a graphical user interface with a flexible waveform edit function. It includes a display screen and touchscreen interface on a Microsoft Windows platform.

You can control instrument operations using the following:

- Front-panel controls
- Menu bar commands
- Touchscreen
- Keyboard and mouse

Touch-screen interface

The touch screen interface is a standard feature of the instrument, which allows you to access menu items and on-screen controls with the touch of a finger. The Touch Screen Off button on the front panel enables or disables this function.

Analysis and Connectivity Support

This Tektronix Windows-based arbitrary waveform generator supports industry-standard software tools, applications and protocols. The integrated Windows desktop enables popular commercial programs or custom-written applications to run on the instrument.

The instrument includes tools that you can install to support data import or export for use with data-analysis tools. The following tools are supported:

- TekVISA

  TekVISA is a library of industry-standard compliant software components, organized according to the standard VISA model established by the VXIplug&play Systems Alliance. Use TekVISA in software to write interoperable instrument drivers to handle communicating between software applications and your instrument.

- VXI-11.2 LAN Server

  The VXI-11.2 LAN Server provides software connectivity between your instrument and remote PCs over an Ethernet LAN. This tool is a client-side component built-in with TekVISA on each remote PC, you must install another copy of TekVISA to make use of its client-side component.

How to Start Basic Mode

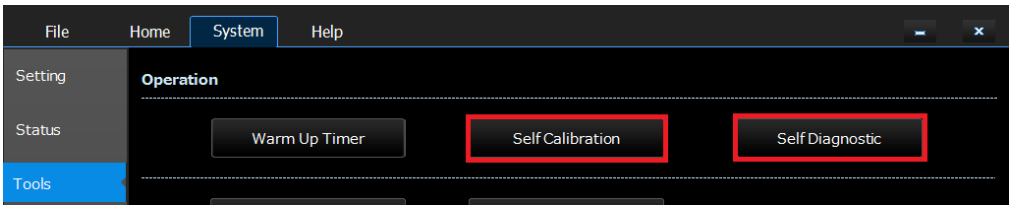
To start Basic mode, first power on the instrument and then push the Basic button 

on the front panel to launch the Basic application. You can also click the Basic icon 

on the desktop to launch.

Perform Instrument Self Calibration and Self Diagnostic

The instrument performs a limited set of hardware tests at power-on. You can also perform the Self Calibration and Self Diagnostic from the System -> Tools menu.



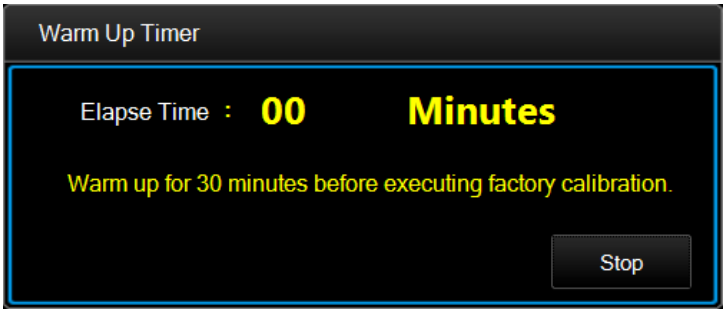
Self Calibration

This calibration primarily checks DC accuracy using the internal calibration routines.

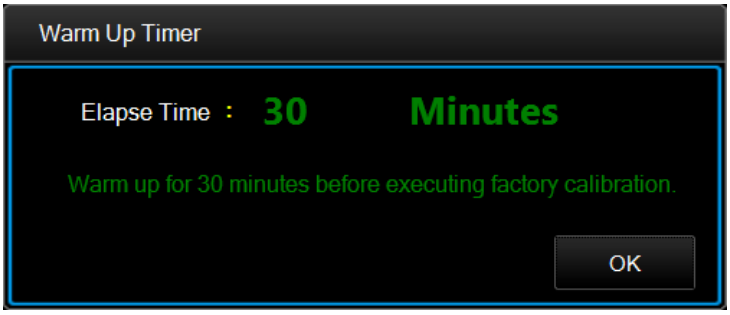
CAUTION. *Do not power off the instrument while executing Self Calibration. If the power is turned off during Self Calibration, data stored in the internal memory may be lost.*

NOTE. *Before executing this operation, allow a 30 minute warm-up period after powering on the instrument, because the calibration is not valid if the instrument does not reach to a valid temperature.*

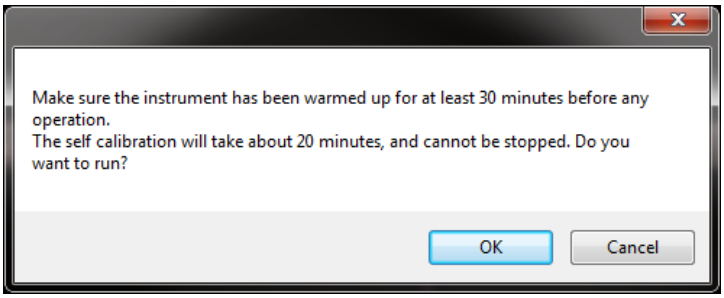
1. Click the **System** tab.
2. Click the **Tools** tab from the left sidebar menu.
3. Click the **Warm up Timer**, and one dialog will pop up to show warm up timer. Wait for 30 minutes. You can press **Stop** to terminate warming up.



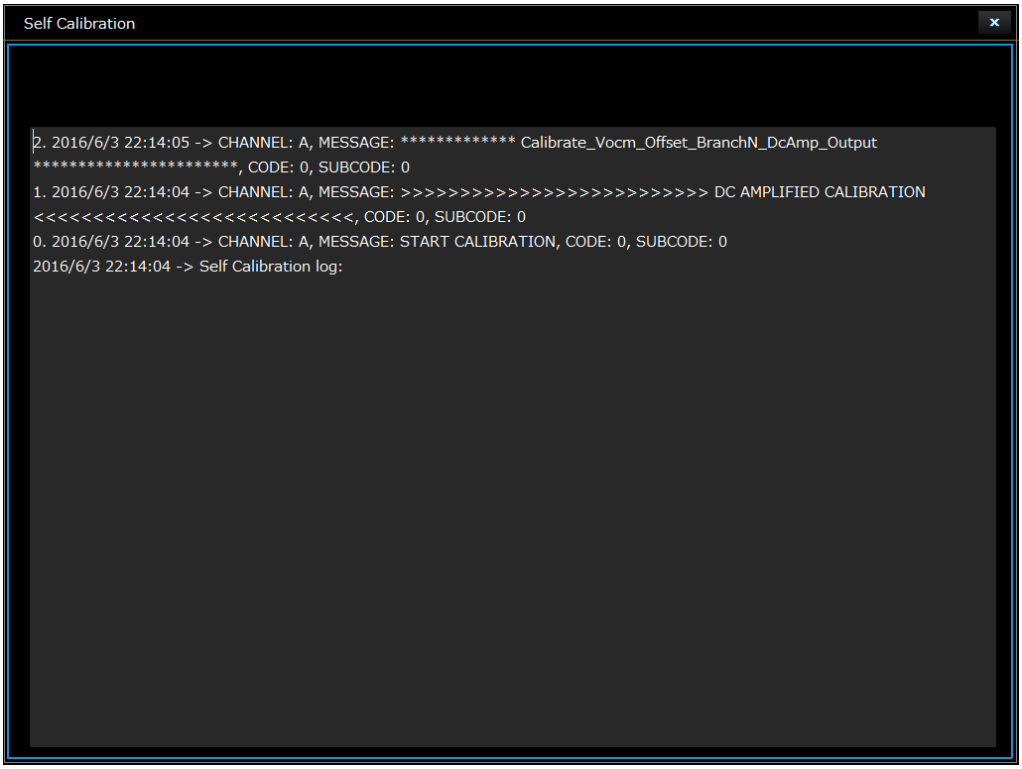
4. When 30 minutes is shown on warm up timer dialog, press **OK**. Wait for about 1 minute until system is not busy.



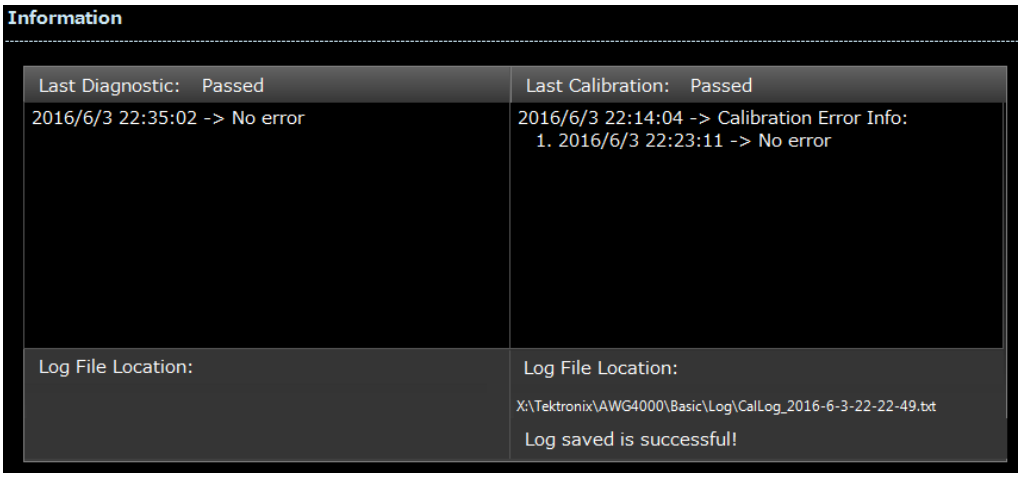
5. Click the **Self Calibration** button and the following dialog will appear.



6. Select **OK** to run the calibration if you have warmed up for 30 minutes, or select **Cancel** to cancel the operation. Self Calibration may take for more than 20 minutes. It can't be stopped during this operation.



7. When the Self Calibration is complete, results are displayed in the Information section of the display under Last Calibration. The log file location is also shown.

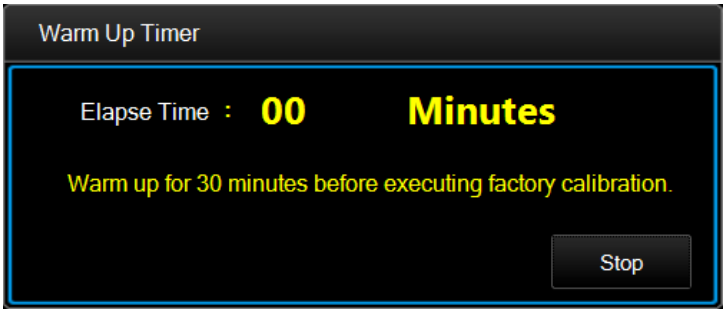


Self Diagnostic

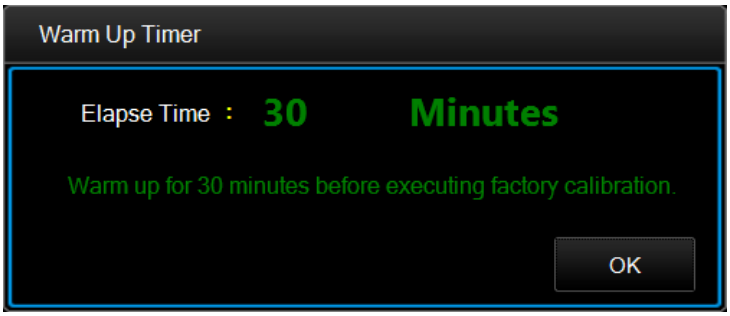
This test verifies that your instrument is operating correctly.

NOTE. *Before executing this operation, allow a 30 minute warm-up period after powering on the instrument, because the calibration is not valid if the instrument does not reach to a valid temperature.*

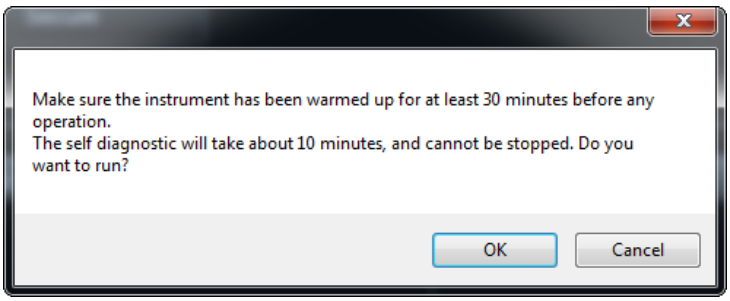
1. Click the **System** tab.
2. Click the **Tools** tab from the left sidebar menu.
3. Click the **Warm up Timer**, and one dialog will pop up to show warm up timer. Wait for 30 minutes. You can press **Stop** to terminate warming up.



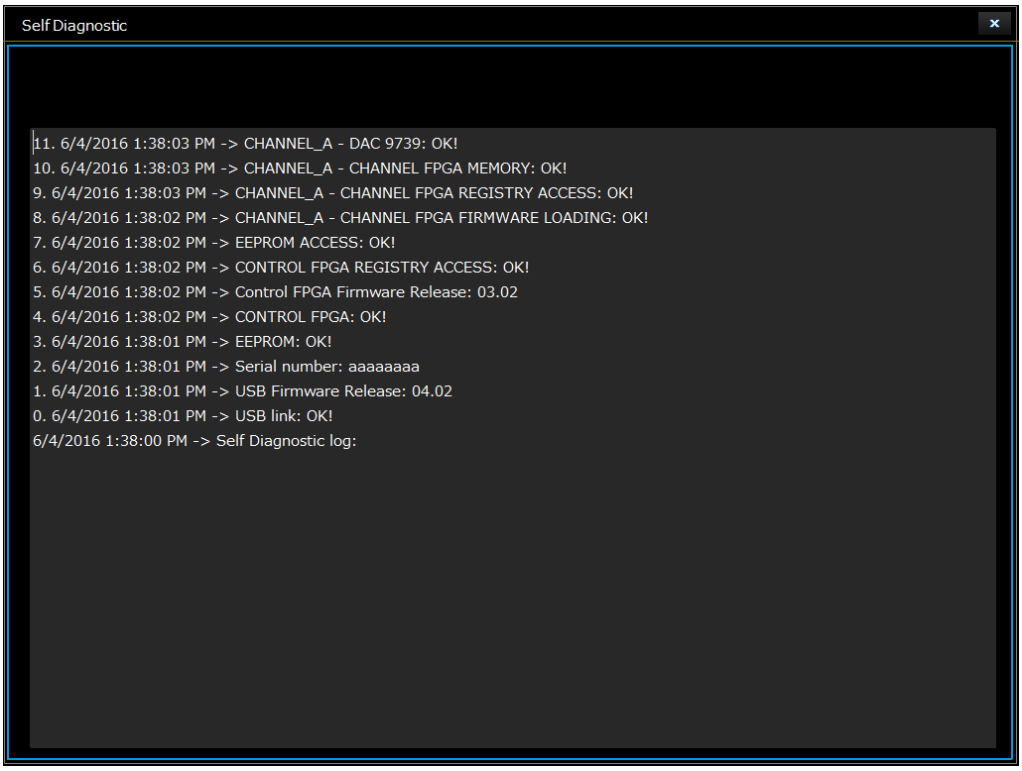
4. When 30 minutes is shown on warm up timer dialog, press **OK**, Wait for about 1minute until system is not busy.



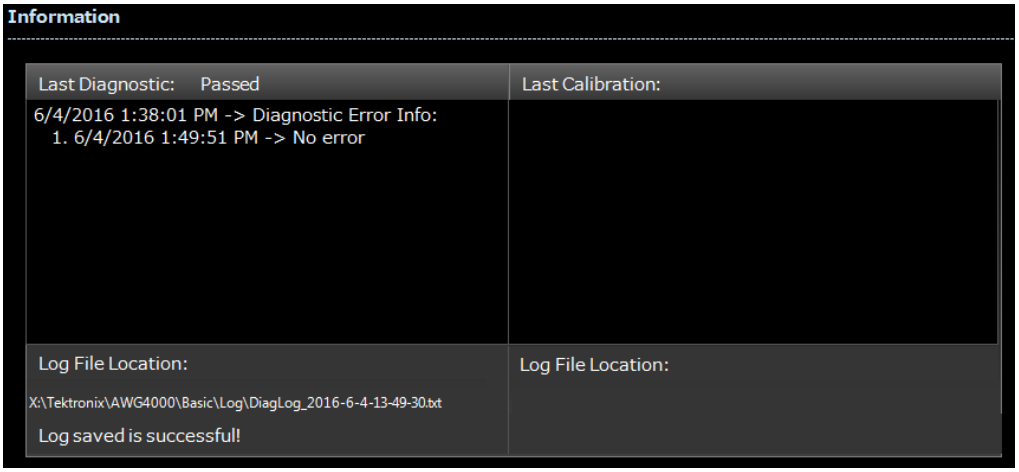
5. Click the **Self Diagnostic** button and the following dialog will appear.



6. Select **OK** to do the diagnostic if you have warm up for 30 minutes, or select **Cancel** to cancel the operation.



7. If diagnostics complete without any errors, the message Passed is displayed in the Information section of the display under Last Diagnostic. The log file location is also shown.



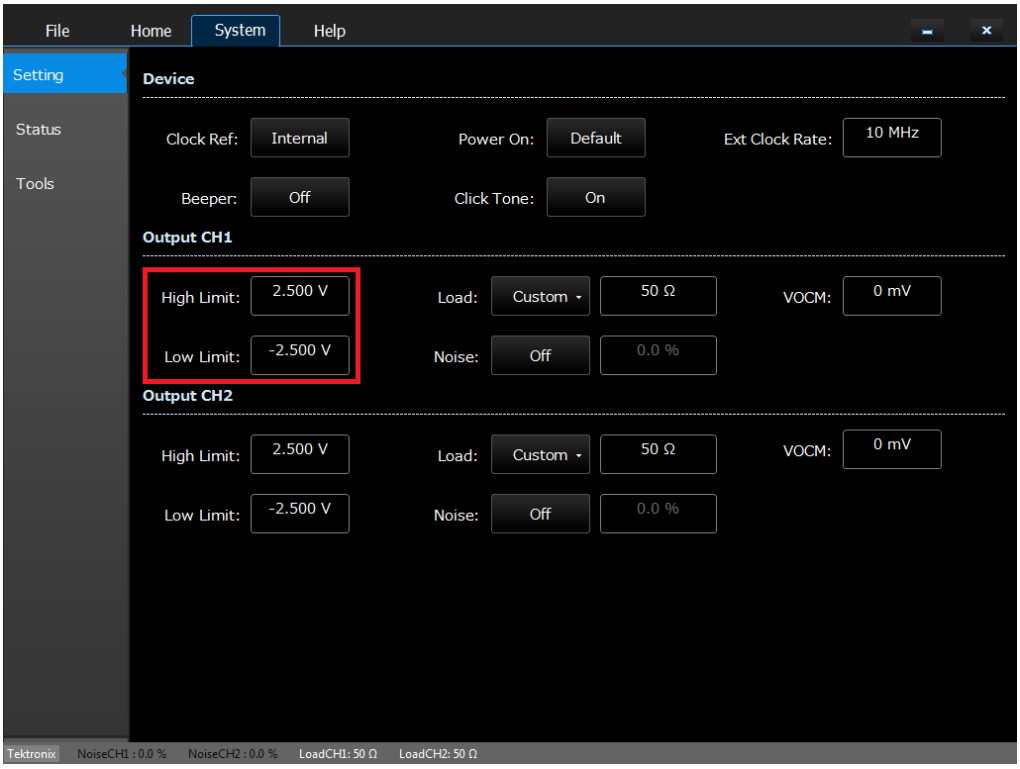
Quick tips

- Allow a 30 minute warm-up period before executing Self Calibration or Self Diagnostic.
- Disconnect all the cables from the instrument when you perform Self Calibration or Self Diagnostic.
- It is recommended that the Self Calibration should be performed along with a periodic check.
- If you need to verify that the instrument meets the warranted specifications, do the complete set of performance verification procedures provided in the Specifications and Performance Verification technical reference manual.
- The Self Calibration will take about 20 minutes. The Self Diagnostic will take about 10 minutes. These operations cannot be stopped.
- Don’t power off the instrument during the Self Calibration or Self Diagnostic operations.

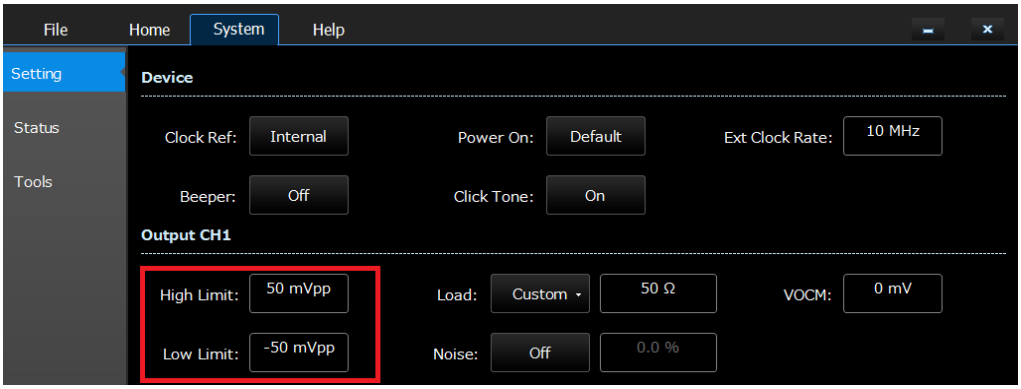
Protect Your DUT from Damage

Use care when you connect the instrument Channel Out to your DUT (device under test). To avoid damage to your DUT, the following preventive measures are provided. Follow these steps to set the limit values for high level and low level.

1. Click the **System** tab and then click the **Setting** tab from the left sidebar menu.
2. In this example, High Limit is set to 2.500 V, and Low Limit is set to -2.500 V.

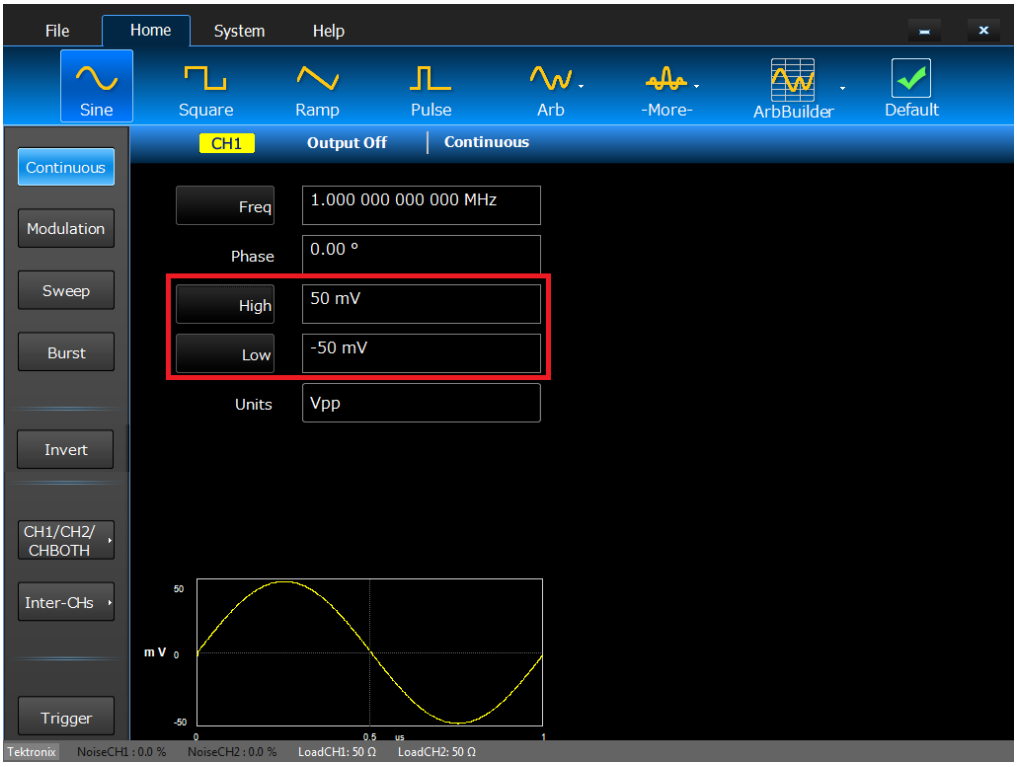


3. Enter 50 mV for **High Limit**, and –50 mV for **Low Limit**.



4. Select the front-panel **Sine** button 

to display the waveform parameter. Confirm that High and Low voltage levels were changed.



NOTE. *You cannot enter any values greater than 50 mV for High level.*

Load Impedance, VOCM and the Output Window

The following table shows the output window (maximum and minimum levels) for a sine waveform when you change the load impedance and VOCM. Window includes Max DC VOCM (50 ohm load: +/- 2.5 V/High Z load: +/- 5.0 V). It depends on the range of amplitude and VOCM. You can read more about load impedance in the Set up Load Impedance topic and VOCM in the Set up VOCM topic.

Frequency	50ohm load, single-ended	high Z load, single-ended
1uHz ~ 350MH	-5V~5V	-10V~10V
350MHz ~ 550MHz	-4V~4V	-8V~8V
550MHz~600MHz	-3.5~3.5V	-7V~7V

Operating Basics

Default Setup

You can return Basic mode to its default settings by clicking the **Default** button 

on the Home tab or the front-panel **Default** button 

. Please see Appendix for details.

Quick tutorial: How to select a waveform and adjust parameters

If you are a beginning user, you can follow the steps described here to get acquainted with how to select a waveform and adjust waveform parameters once the instrument is powered on and running in Basic mode.

1. Connect the power cord, and then push the front-panel power on/off switch  to turn on the instrument.
2. Connect the Analog Ch1 Out 

of the instrument to the oscilloscope input with a cable.
3. Select a waveform. In the image below, **Sine** 

is selected.
4. Select the run mode from the left sidebar menu. In the image below, **Continuous** 

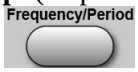
is selected.
5. Push the **Ch1 On** button 

to enable the output.
6. Observe the sine waveform displayed on the oscilloscope screen.
7. Use the front-panel shortcut buttons on the instrument to select a waveform parameter. We push Frequency/Period button 

to select Frequency as a parameter to be changed.
8. You can also set the **Phase** 

, **Ampl** (amplitude) 

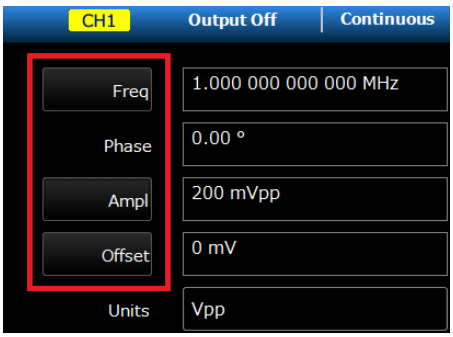
, **Offset** 

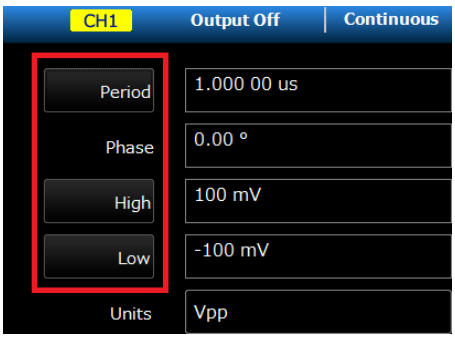
, and **Units**.
9. If you push the Frequency/Period button 

twice, the parameter changes to **Period**. Similarly, you can change the **Ampl** button 

to **High** and the **Offset** button 

to **Low**. Through UI operation, you click on **Freq** button to change the parameter to **Period**. Similarly, you can click on **Ampl** button to **High**, and the **Offset** button to **Low**.

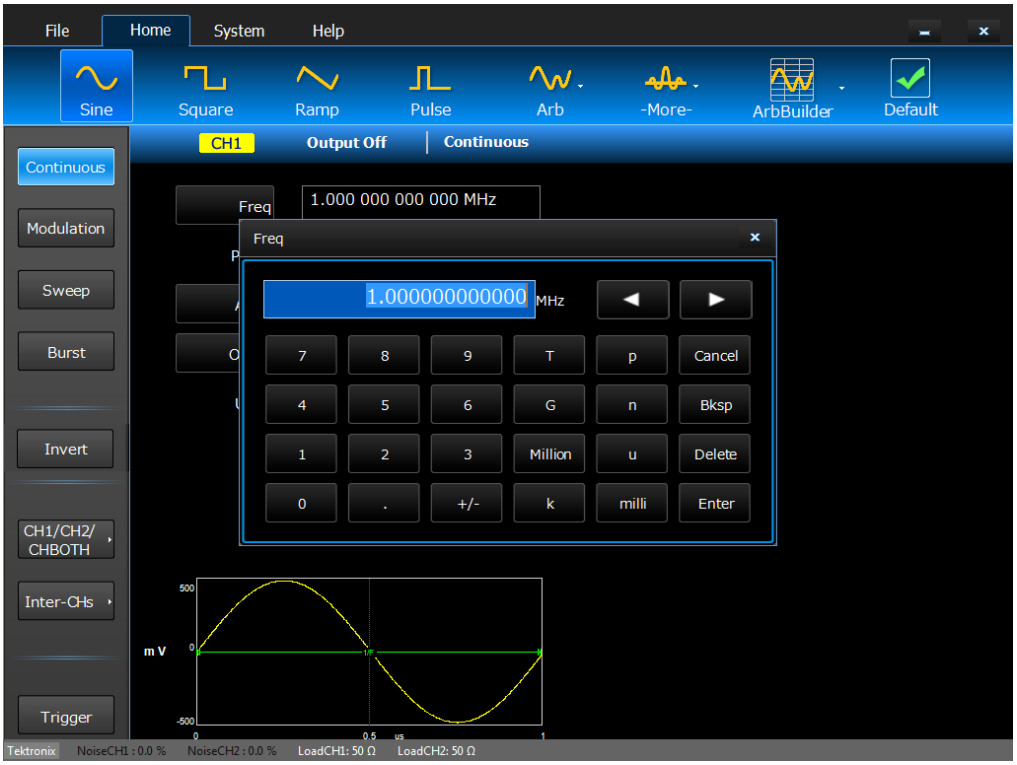




10. Change the waveform parameters using the numeric keypad  or soft keys, the general purpose knob and the arrow keys  , touch screen, or, keyboard and mouse.

Quick tips

- Make sure oscilloscope input impedance is set to 50Ω to observe the correct amplitude, offset or Vocm.

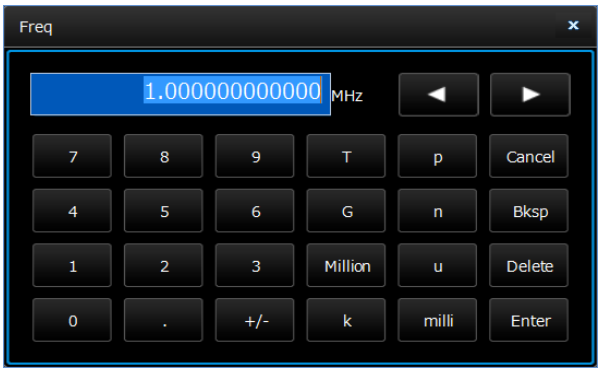


Quick tutorial: How to generate a sine waveform

If you are a beginning user, you can follow the steps described here to generate a continuous sine waveform once the instrument is powered on.

1. Connect the power cord, and then push the front-panel power on/off switch  to turn on the instrument.
2. Start Basic Mode. (See page: How to Start Basic Mode)
3. Connect the Analog Ch1 Out  of the instrument to the oscilloscope input with a cable.
4. Select the **Sine** function  from the top of the **Home** tab.
5. Select the **Continuous** run mode from the left sidebar menu.
6. Push the front-panel **Ch1 On** button  to enable the output.
7. Observe the sine waveform displayed on the oscilloscope screen.

8. To change the frequency, click the number field next to it.
9. To change the frequency value, use the touch panel. For example, click on 2 using the soft keyboard, then click on Units or Enter to complete the entry. You can change the Amplitude, Phase, and Offset values in the same way.



10. You can also change the frequency value using the numeric keypad, the general purpose knob and the arrow keys, or keyboard and mouse.

Quick tips

- Use the front-panel shortcut buttons to quickly select a waveform parameter.
- When you specify a waveform parameter using the shortcut buttons or touch screen selection, an active parameter is displayed in green in the graph area.

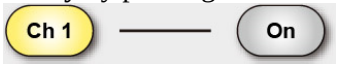
Generate a Continuous Waveform

Continuous run mode sets the generator to continuously output the signal. This is the default run mode.

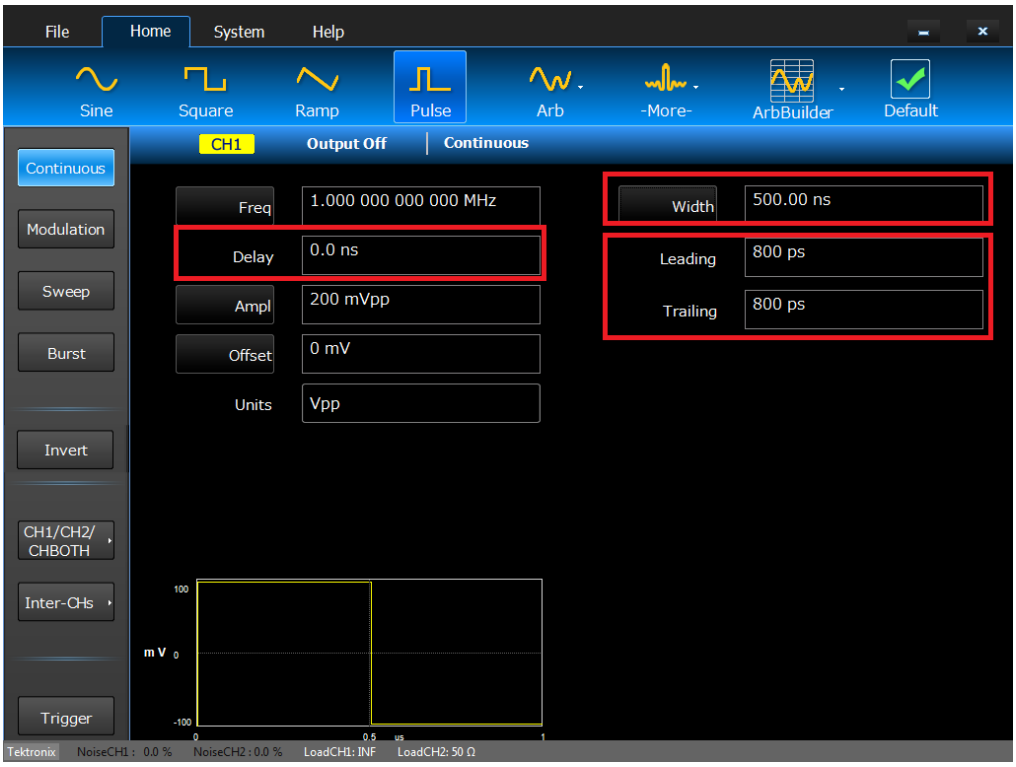
Generate a Pulse Waveform

1. Select the **Pulse** function 

from the top of the **Home** tab.
2. Select the **Continuous** run mode from the left sidebar menu.
3. To change the frequency, click the number field next to it.
4. Click on **Duty** to change the parameter to **Width**.
5. Click on the number field next to **Leading** and **Trailing**.
6. You can set the **Lead Delay** by clicking on the number field next to Delay and adjusting the parameter as needed. You can also select **Lead Delay** by pushing the **Phase/Delay** shortcut button 

.
7. Push the front-panel **Ch1 On** button 

to enable the output.
8. Observe the pulse waveform displayed on the oscilloscope screen.


File
Home
System
Help
Sine
Square
Ramp
Pulse
Arb
-More-
ArbBuilder
Default
Continuous
Modulation
Sweep
Burst
Invert
CH1/CH2/ CHBOTH
Inter-CHs
Trigger
CH1
Output Off
Continuous
Freq
1.000 000 000 000 MHz
Delay
0.0 ns
Ampl
200 mVpp
Offset
0 mV
Units
Vpp
Width
500.00 ns
Leading
800 ps
Trailing
800 ps
100
m V 0
-100
0
0.5 us
1
Tektronix
NoiseCH1 : 0.0 %
NoiseCH2 : 0.0 %
LoadCH1: INF
LoadCH2: 50 Ω

Pulse waveform formulas

The following formulas are applied to leading edge time, trailing edge time, pulse period, and pulse width of pulse waveforms.

lEdge (Leading Edge Time)

tEdge (Trailing Edge Time)

Maximum leading edge time. This value is the minimum of the three in each instance.

Temp1 = 0.8 * 2.0 * width – tEdge;

Temp2 = (period – width) * 0.8 * 2.0 – tEdge;

Temp3 = 1000 s.

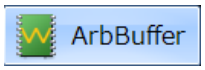
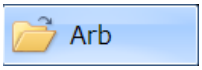
Maximum trailing edge time. This value is the minimum of the three in each instance.

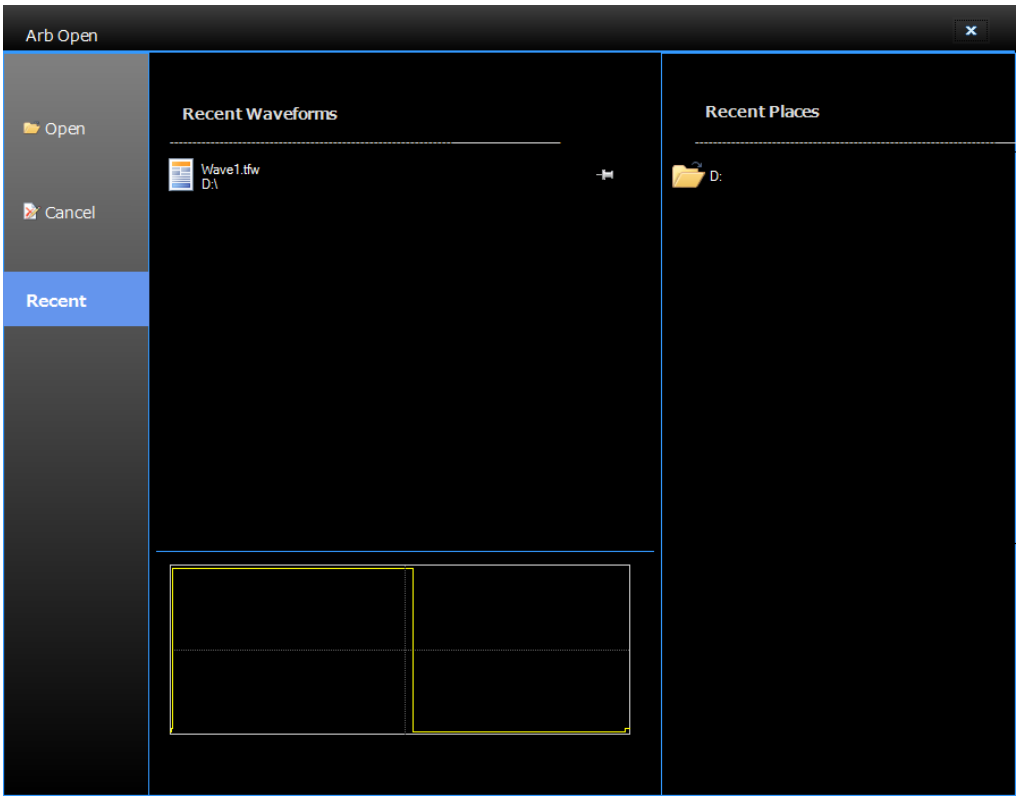
Temp1 = 0.8 * 2.0 * width – lEdge;

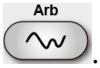
Temp2 = (period – width) * 0.8 * 2.0 – lEdge;

Temp3 = 1000 s.

Generate an Arbitrary Waveform

1. Select the **Arb** function  from the top of the **Home** tab.
2. Select **ArbBuffer**  from the **Arb** drop down menu to recall a previous internal arbitrary waveform or select **Arb**  to recall a stored arbitrary waveform.



3. You can also recall waveforms by pushing the front-panel **Arb** button  .
4. The default internal arbitrary waveform is Sine.


File
Home
System
Help
Sine
Square
Ramp
Pulse
Arb
-More-
ArbBuilder
Default
ArbBuffer
Arb
CH1
Output Off
Continuous
Modulation
Sweep
Burst
Invert
CH1/CH2/ CHBOTH
Inter-CHs
Trigger
Freq
1.000 000 000 000 MHz
Phase
0.00 °
Ampl
1.000 Vpp
Offset
0 mV
Units
Vpp
Shape
ArbBuff1
500
0
-500
m V
0
0.5
us
1
Tektronix
NoiseCH1 : 0.0 %
NoiseCH2 : 0.0 %
LoadCH1: 50 Ω
LoadCH2: 50 Ω

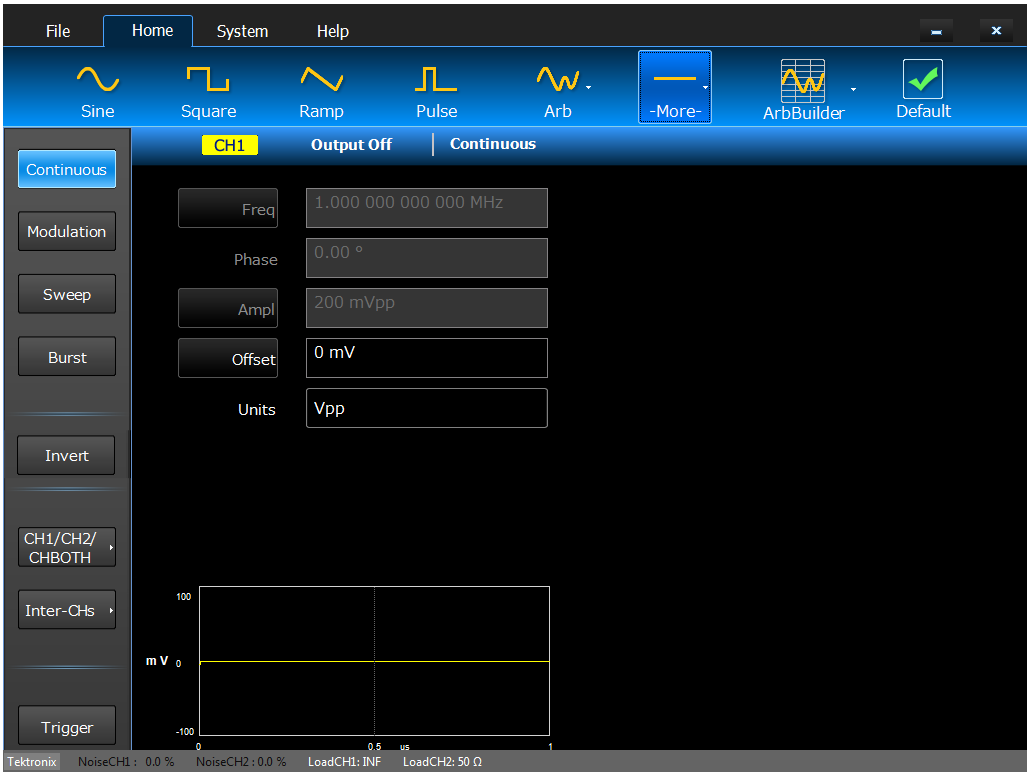
Generate Noise and DC

1. Select the **Noise** function 

by clicking on **More** 

from the top of the **Home** tab.
2. You can set waveform parameters for Noise.
3. Select the **DC** 

to display DC parameters.



Quick tips

- You cannot modulate, sweep or burst noise or a DC waveform.

Generate a Burst Waveform

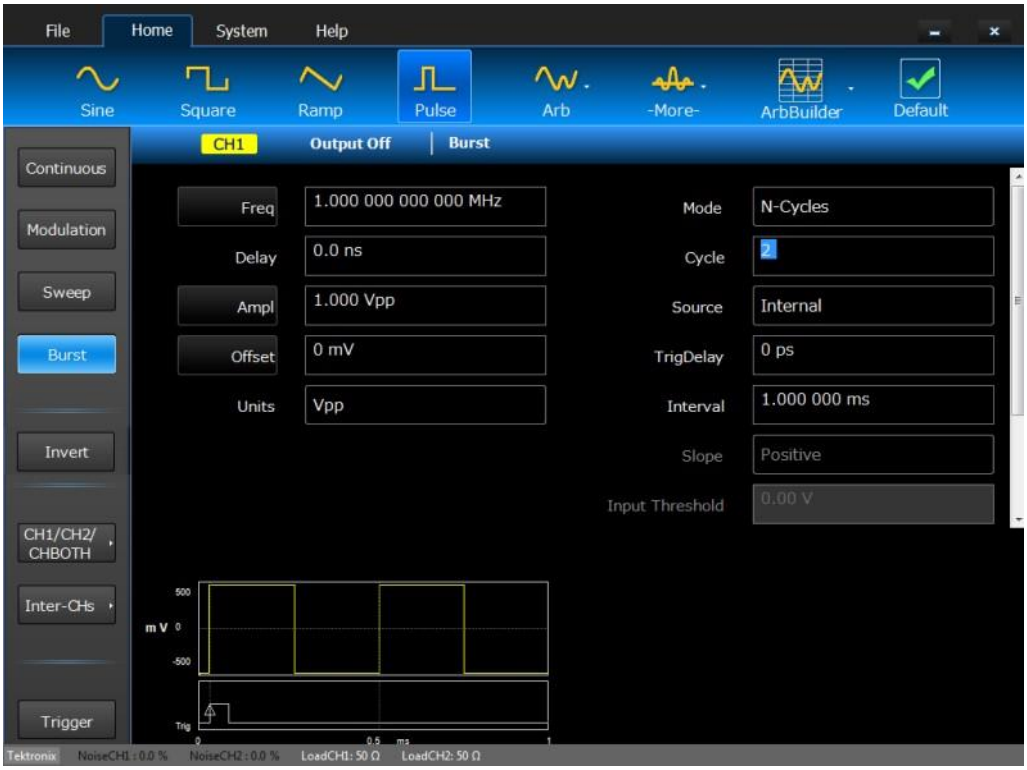
The instrument can output a burst using standard waveforms such as sine, square, ramp, and pulse, or arbitrary waveforms. The instrument allows you to use triggered and gated burst modes as follows:

To Generate a Triggered Burst Waveform

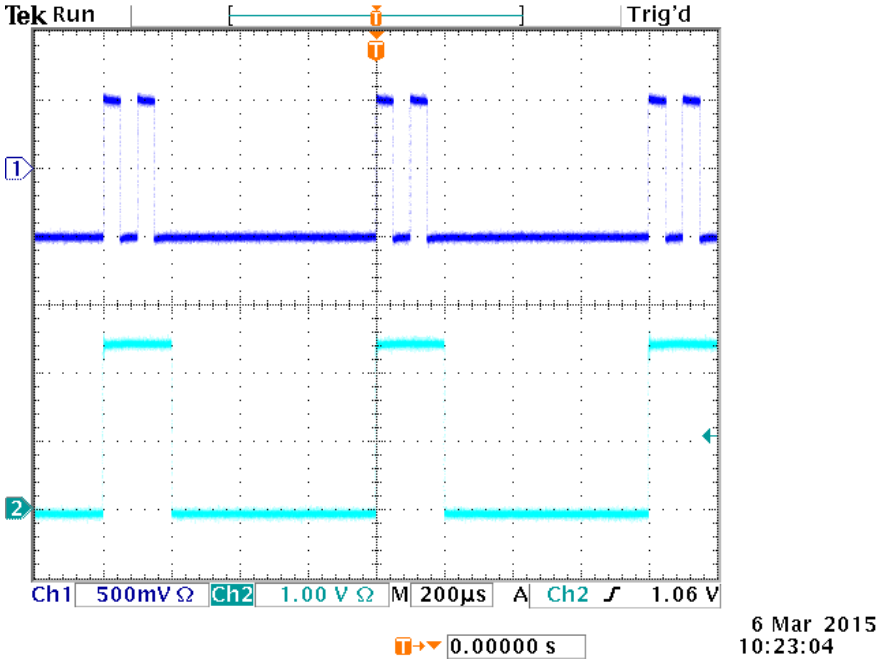
A specified number (burst count) of waveform cycles is output when the instrument receives a trigger input from the internal trigger source, an external trigger source, a remote command, or the manual trigger button.

1. Select the **Pulse** function 

from the top of the **Home** tab.
2. Select the **Burst** run mode from the left sidebar menu.
3. Confirm that 1-Cycle, N-Cycles, or Inf-Cycles is selected which means triggered burst mode is enabled. To generate a double pulse, set the **Mode** to N-Cycles and the burst count to 2.



4. An example of a double pulse and a trigger output signal is shown as below

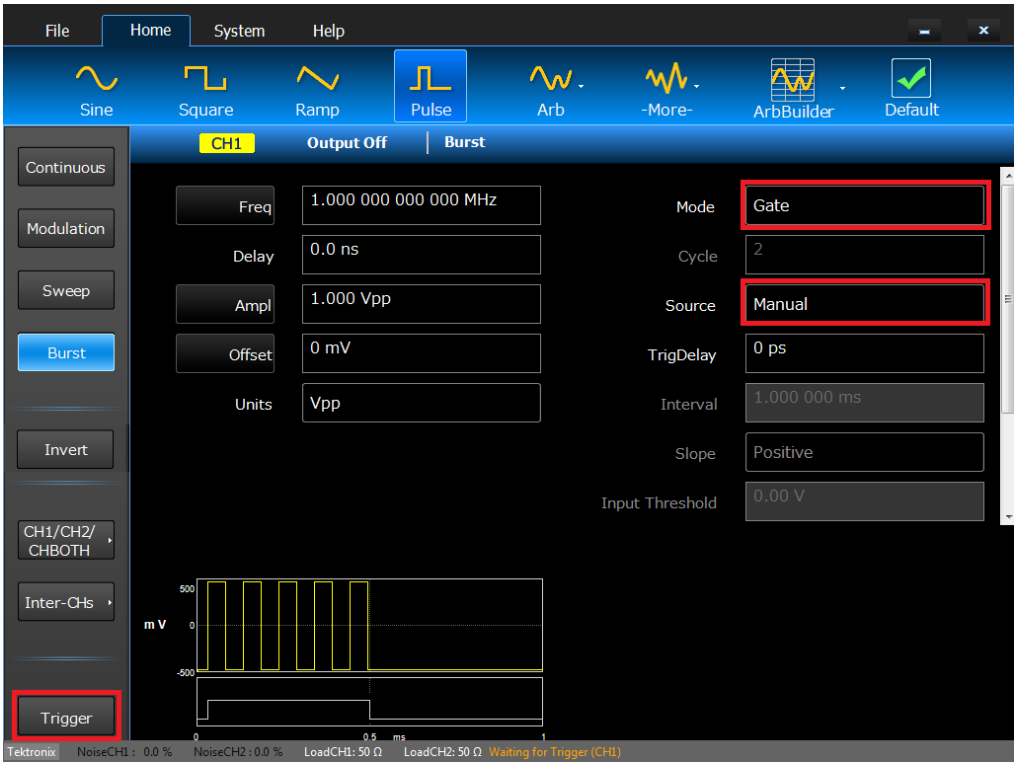


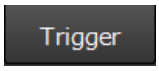
To Generate a Gated Burst Waveform

This outputs a continuous waveform when an effective gate signal is applied externally, when the manual trigger button is depressed, when a remote command is applied, or during 50% of the selected internal trigger interval.

In the gated burst mode, the output is enabled or disabled based on the internal gate signal or an external signal applied to the front-panel Trigger Input connector. While the gate signal is true or the front-panel Force Trig button is pushed in, the instrument outputs a continuous waveform.

1. Select the **Burst** run mode from the left sidebar menu.
2. Select **Gate** in the **Mode** field which means gated burst mode is enabled.
3. Select **Manual** from the Source drop down menu to enable the manual trigger.



4. Click on the **Trigger** button 

or push the front-panel Force Trig button 

.
5. Observe the gated burst waveform displayed on the oscilloscope screen.

Quick tips

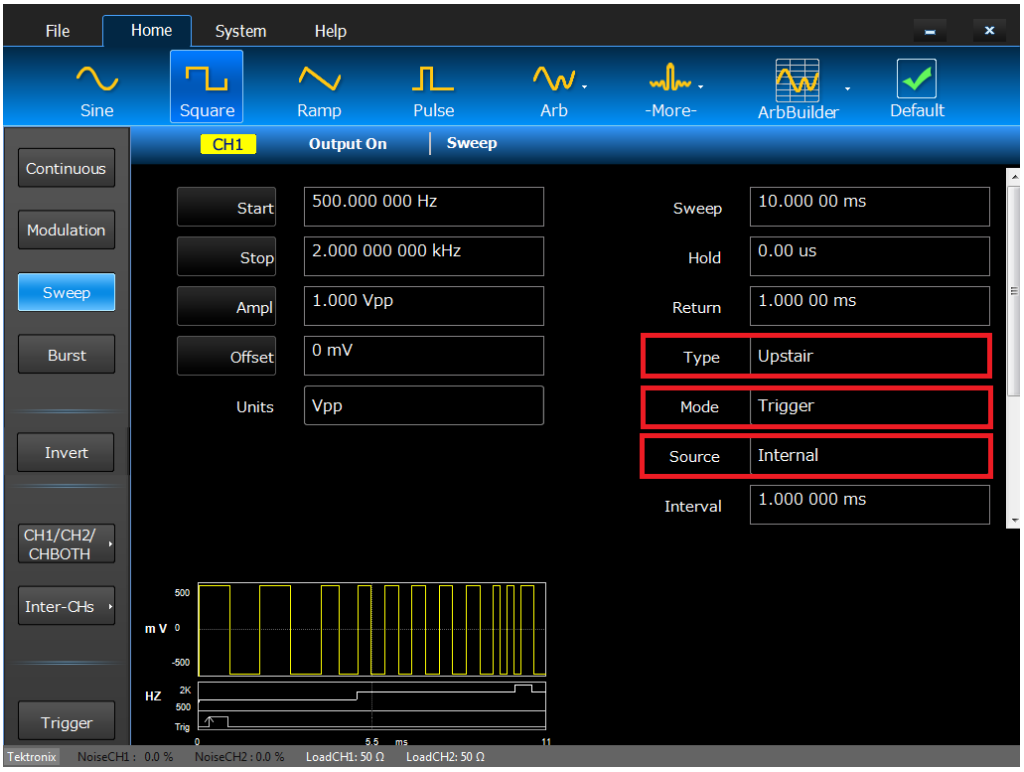
- Use the front-panel shortcut buttons to quickly select a waveform parameter.
- The instrument provides the following three trigger sources for Burst mode:
  - Internal or external trigger signal.
  - Manual trigger (Force trigger).
  - Remote command.
- Once Gate is selected, burst count parameters are ignored.

Sweep a Waveform

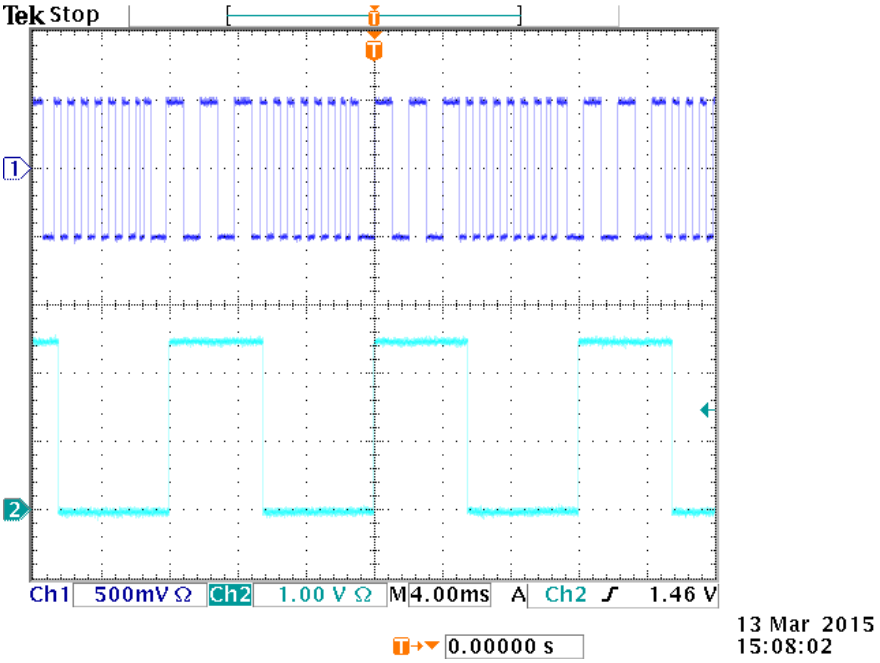
Sweep outputs a waveform with the output signal frequency varying in sweep type of Linear, Logarithm, Upstair, and User Defined. You can set the following Sweep parameters:

- Start frequency: the beginning value of the frequency sweep.
- Stop frequency: the end value of the frequency sweep.
- Sweep time: affects the length (time) of the measurement.
- Return time: the amount of time from Stop Frequency to Start Frequency.
- Center frequency: the frequency half way between the start and the stop frequencies.
- Frequency span: the boundary of the frequency display.
- Hold time: the amount of time that the frequency must remain stable after reaching the stop frequency.

1. Select a waveform function from the top of the **Home** tab.
2. Select the **Sweep** run mode from the left sidebar menu.
3. Specify the **Start** frequency, **Stop** frequency, **Sweep** time, **Hold** time, and **Return** time, as desired. When you click on the **Start** frequency button, it toggles to the **Center** frequency. When you click on the **Stop** frequency button, it toggles to the **Span** frequency.
4. Click on the sweep **Mode** field and select **Trigger** or **Repeat**.
5. Select the trigger source from the **Source** drop down menu.



6. This is a sample oscilloscope screen. The top is a sample of a sweep waveform. The bottom is a trigger output signal.



Quick tips

- For frequency sweep, you can select a Sine, Square, Ramp, More, or Arbitrary waveform. Pulse, DC, and Noise waveforms cannot be selected.
- Once the sweep is selected, the frequency is swept from the sweep start to the sweep stop frequencies.

- If a start frequency is lower than a stop frequency, the instrument sweeps from the low frequency to the high frequency.
- If a start frequency is higher than a stop frequency, the instrument sweeps from the high frequency to the low frequency.
- If you want to return to the Sweep menu after selecting other menus, push the front-panel Sweep button again.

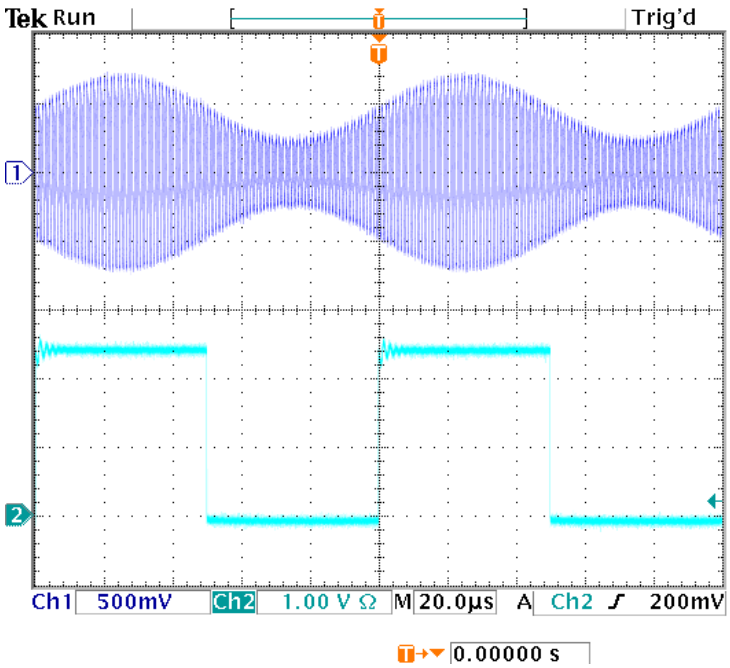
Modulate a Waveform

To Output an AM Waveform

Amplitude modulation (AM) is a technique that varies the amplitude of the carrier waveform.

1. Select a waveform function from the top of the **Home** tab. This will be the carrier waveform.
2. Select the **Modulation** run mode from the left sidebar menu.
3. Specify the modulation type as AM by clicking on the **Type** field and selecting **AM** from the drop down menu.

4. Select modulation source, set modulation frequency, select shape, and set modulation depth.
5. This is an example amplitude modulation waveform displayed on an oscilloscope screen.



Quick Tips

- You can output frequency modulation (FM) or phase modulation (PM) waveforms in the same way. You cannot select Pulse, Noise, or DC as a carrier waveform.
- You can select an internal or external signal as an AM source. If you select an external source and set the modulation depth to 120%, the output will be at the maximum amplitude when a ±1 V_{p-p} signal is applied to the rear panel Ext Mod Ch1 In or Ext Mod Ch2 In connector.
- You can select a modulation shape from the internal memory or Local/USB `memory`.
- The following equations show the output amplitude of AM, FM, and PM modulation (in this example, sine waveform is used for carrier waveform and modulation waveform):

AM: Output(V_{p-p})=$\frac{A}{2.2}\left(1+\frac{M}{100}\sin\left(2\pi fmt\right)\right)\sin\left(2\pi fct\right)$$\left(1+\frac{M}{100}\sin\left(2\pi fmt\right)\right)$

FM: Output(V_{p-p}) = $A\sin\left(2\pi\left(fc+D\sin\left(2\pi fmt\right)\right)t\right)$

PM: Output(V_{p-p}) = $A\sin\left(2\pi fct+2\pi\frac{P}{360}\sin\left(2\pi fmt\right)\right)$

Carrier amplitude	A [V_{p-p}]
Carrier frequency	fc [Hz]
Modulation frequency	fm [Hz]
Time	t [sec]
AM Modulation depth	M [%]
FM Deviation	D [Hz]
PM Deviation	P [degree]

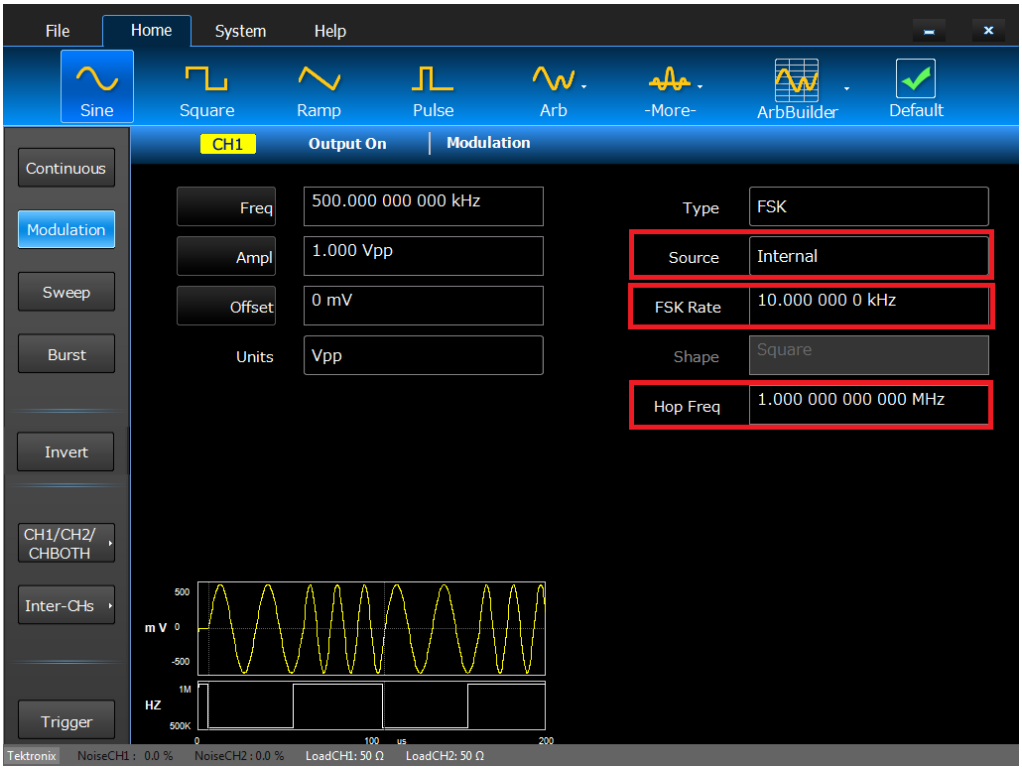
- The following table shows relationship between modulation depth and maximum amplitude for AM modulation waveform (internal modulation source is selected):

Depth	Maximum amplitude
120%	A (V_{p-p})
100%	A (V_{p-p}) * 0.909
50%	A (V_{p-p}) * 0.682
0%	A (V_{p-p}) * 0.455

To Output an FSK Waveform

Frequency Shift Keying (FSK) modulation is a modulation technique that shifts the output signal frequency between two frequencies: the carrier frequency and hop frequency. The AWG4162 generates a phase continuous FSK signal.

1. Follow the steps described in the To Output an AM Waveform procedure to display the modulation type drop down menu. (See Modulate a Waveform.) In this example, select **FSK** as the modulation type.
2. The FSK parameter setting screen is displayed. Select Internal or External as **FSK Source**.
3. If you select Internal, you can set the **FSK Rate**. If you select External, the **FSK Rate** is ignored.



4. Set **Hop Frequency**. Carrier waveform frequency shifts to the Hop frequency with the specified FSK rate, and then returns to the original frequency.

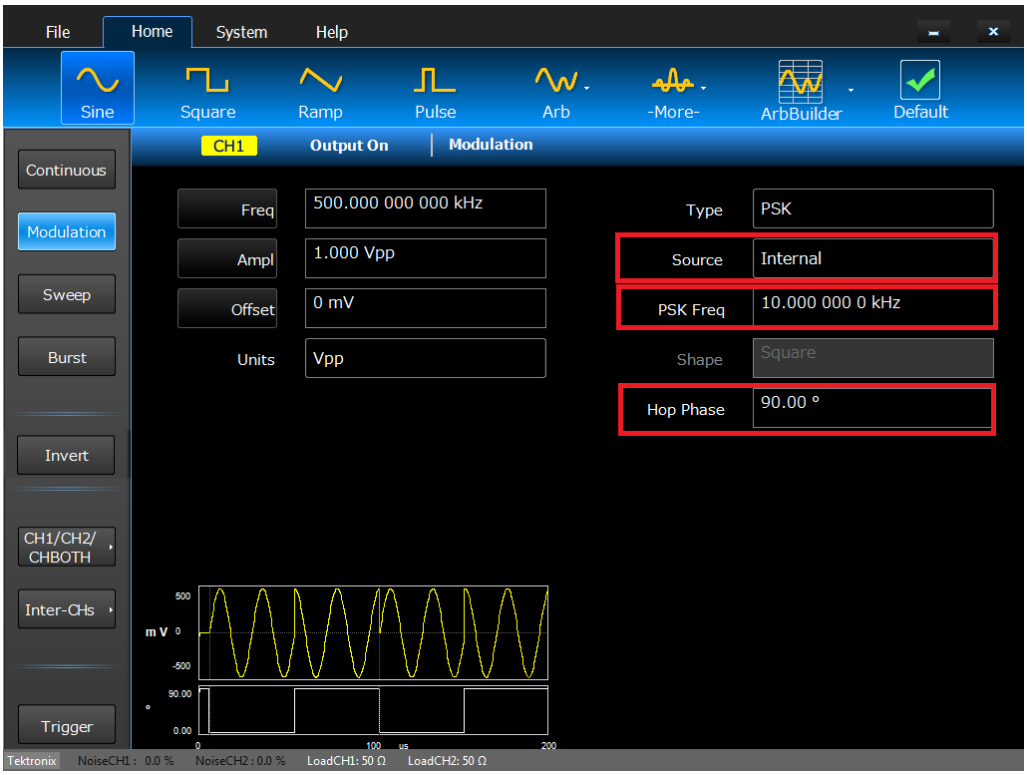
To Output a PSK Waveform

Phase Shift Keying (PSK) modulation is a modulation technique that shifts the output signal phase between two phases: the carrier phase and hop phase.

1. Follow the steps described in the To Output an AM Waveform procedure to display the modulation type drop down menu. (See Modulate a Waveform.) In this example, select **PSK** as the modulation type.

2. The PSK parameter setting screen is displayed. Select Internal or External as **PSK Source**.

3. If you select Internal, you can set the P**SK Frequency**. If you select External, the **PSK Frequency** is ignored.



4. Set **Hop Phase**. Carrier waveform phase shifts to the **Hop Phase** with the specified **PSK Frequency**, and then returns to the original phase.

To Output a PWM Waveform

Follow these steps to output a PWM waveform.

1. Select the **Pulse** function 

from the top of the **Home** tab to display the pulse parameter setting screen.

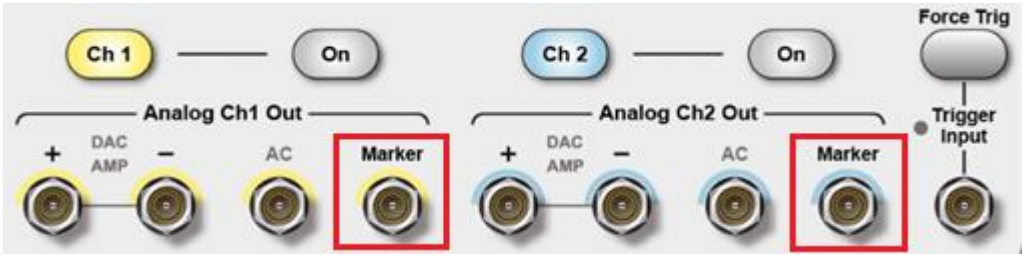
2. Select the **Modulation** run mode from the left sidebar menu and the modulation type will be specified as PWM automatically. Select the **PWM Source**.

3. Set the **PWM Frequency**, select the Modulation Shape, and set the **Deviation** (pulse width deviation).

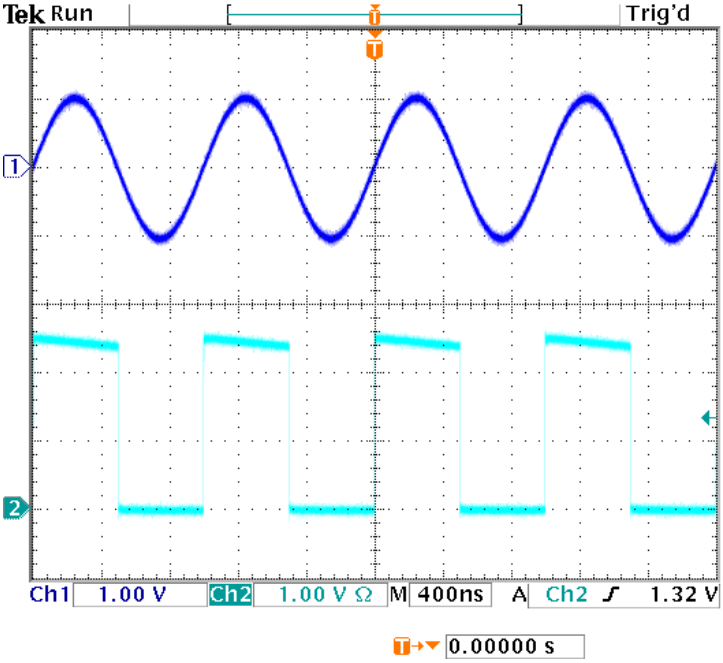
Marker Out

The Marker out signal of the instrument is linked to run mode and function selected in the two channels respectively.

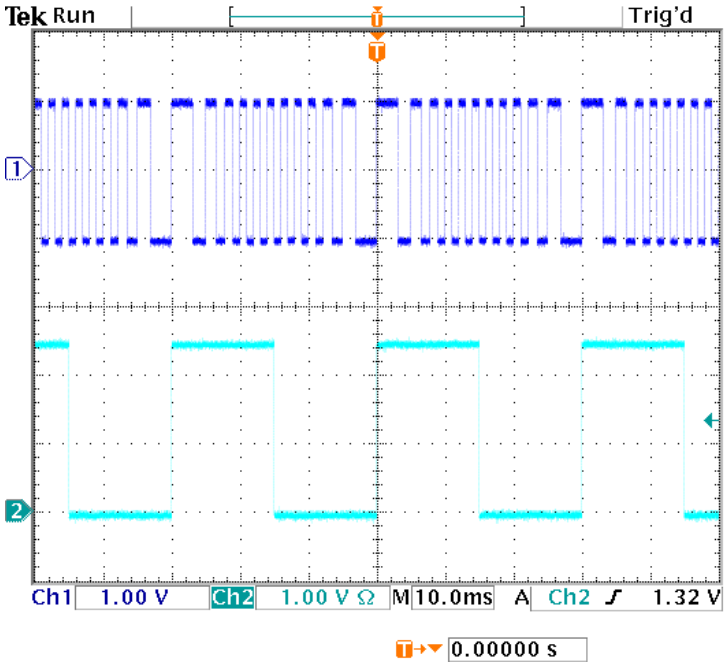
1. Connect the front-panel Marker out connector and the external Trigger Input connector of the oscilloscopes. The Marker out connector provides the trigger signal for oscilloscopes.



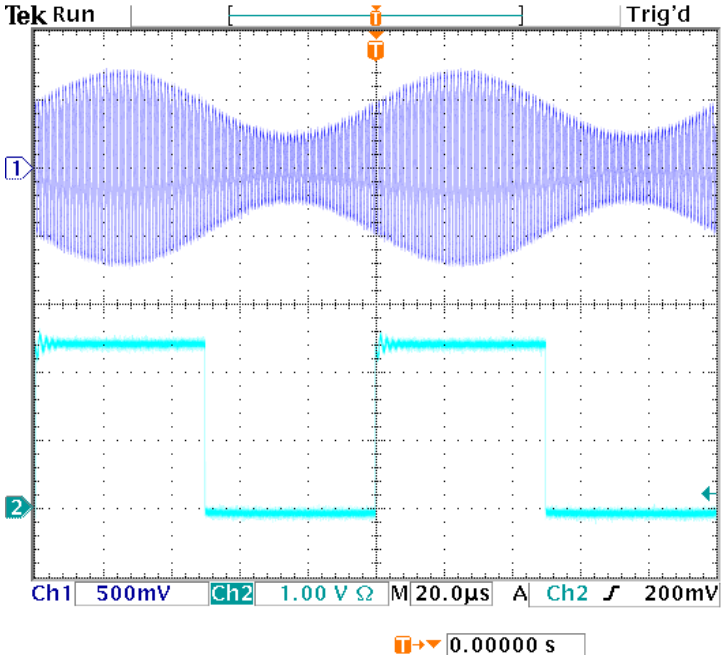
2. Continuous mode: The Marker out is a square waveform and the rising edge at the start of each waveform period. When an output frequency is higher than 156.25 MHz, some restrictions are applied. See the Quick Tips below.



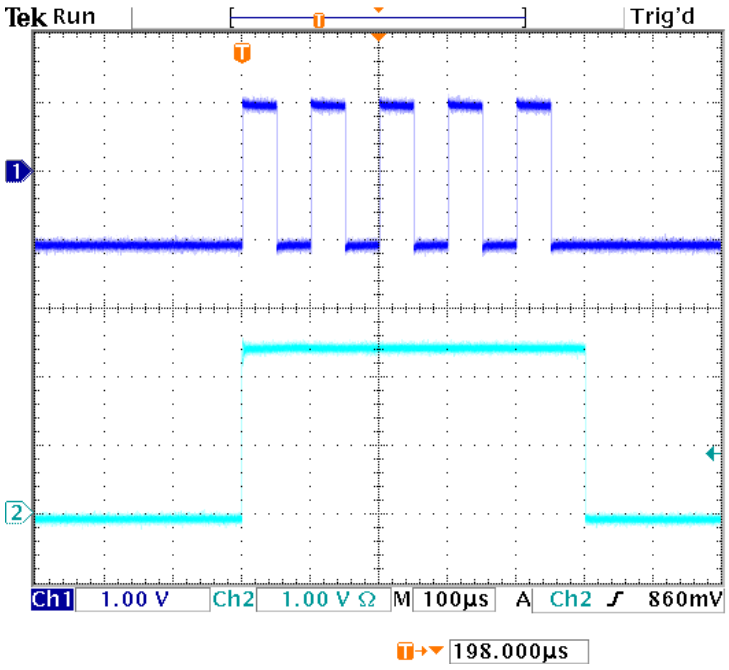
3. Sweep mode: When the **Repeat** or **Trigger** sweep mode and trigger source are selected, the Marker out is a square waveform and the rising edge at the start of each sweep.



4. Modulation mode: When internal modulation source is selected, the Marker out is a square waveform of the same frequency as the modulating signal. When an external modulation source is selected, the Marker out is disabled.



5. Burst Mode: When internal trigger source is selected, the Marker out is a square waveform and the rising edge at the start of each burst period. When an external trigger source is selected, the Marker out is high during the time the trigger input is high.



Quick Tips

- The relationship of frequency between Marker Out and Analog Output (AO):

Start Frequency	*Stop Frequency*	*Marker Frequency*
0	100 MHz	Marker f = AO f
100 MHz	200 MHz	Marker f = AO f / 2
200 MHz	400 MHz	Marker f = AO f / 4
400 MHz	600 MHz	Marker f = AO f / 8

NOTE. *The maximum frequency of Marker Out signal is 156.25 MHz.*

NOTE. *When the instrument outputs a modulation waveform, Marker Out signal cannot be output if you select External as the modulation source.*

- The instrument provides the following three trigger sources for Burst mode:
  - Internal or external trigger signal.
  - Manual trigger (Force trigger).
  - Remote command.

Adjusting Parameters of Two Channel Signals

To Align Phase

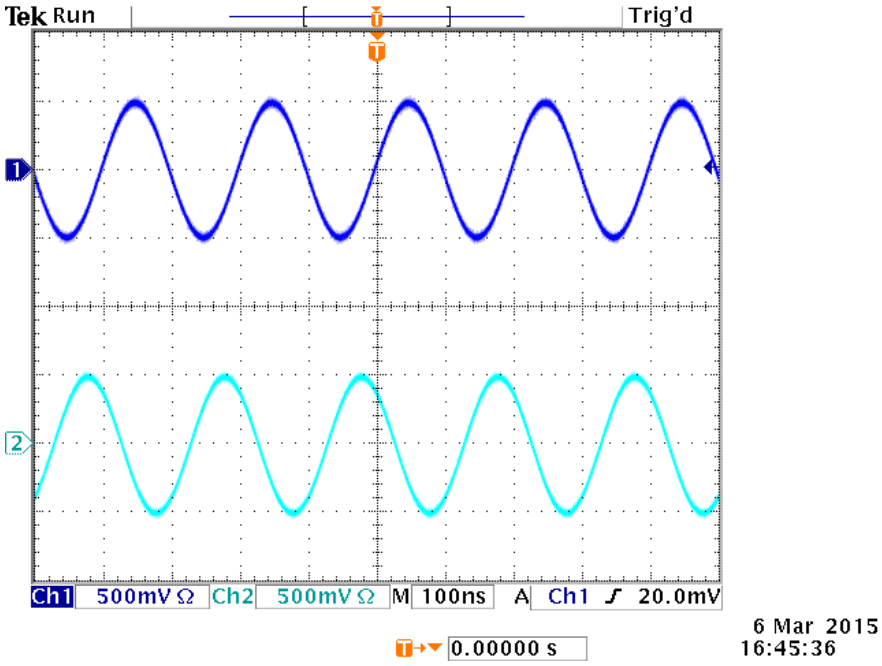
The AWG4162 uses a phase continuous method to change frequency. When you change a frequency of one channel, it will affect the phase relationship between the two channels.

For example, the instrument is generating a 5 MHz sine waveform for both CH1 and CH2 and the phase is adjusted between the two channels. If you change the CH2 frequency to 10 MHz and then return it to 5 MHz, the CH2 phase does not return to its initial condition. To adjust the phase relationship between the two channels, you need to stop signal generation and restart it. The instrument provides an Align Phase function to adjust the phase relationship.

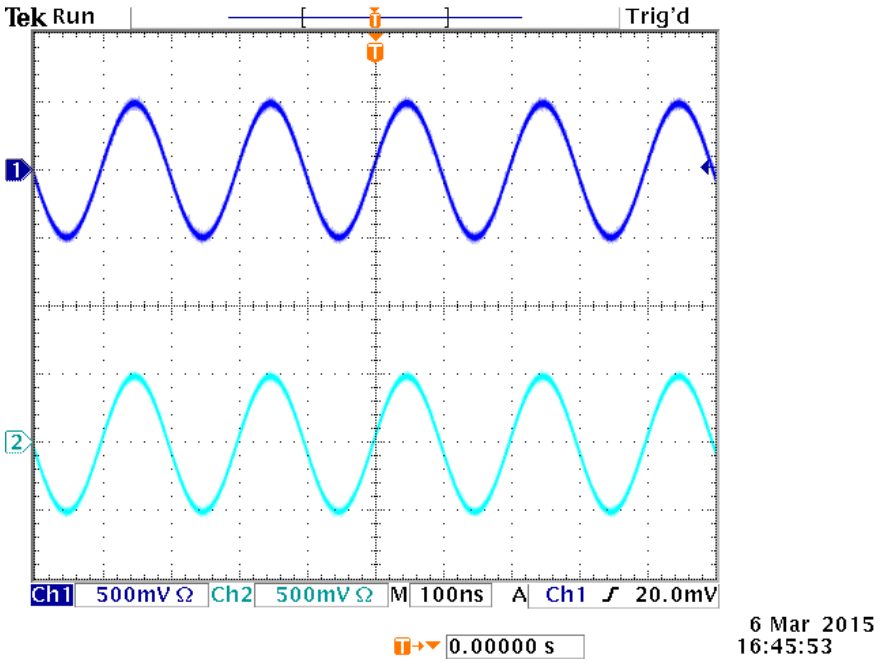
1. Set the instrument to generate a continuous sine wave at 5 MHz for CH1 and CH2. Confirm that both phases are set to 0 degrees.

2. View both channels at once by clicking the **CH1/CH2/CHBOTH** button and selecting **CHBOTH**.

3. Change the CH1 frequency to 10 MHz, and then back to 5 MHz. In this state, the CH2 phase does not return to its initial condition.



4. To align the phase of two channel signals, push the **Inter-CHs** button and select **Align Phase**. The instrument will stop signal generation, adjust the phases of both channels, and then automatically restart signal generation.



To Match Amplitude

To set the CH1 amplitude and CH2 amplitude to the same level, follow these steps:

1. To set both channels to the same amplitude, select the channel that has the desired amplitude. The selected channel will have a colored rectangle around it in the status area.

2. Click the **Inter-CHs** button and select **Amplitude CH1=CH2**. The instrument will stop signal generation, set the amplitude of both channels to match the selected channel, and then automatically restart signal generation.

To Match Frequency/Period

To set the CH1 frequency and CH2 frequency to the same level, follow these steps:

1. To set both channels to the same frequency, select the channel that has the desired frequency. The selected channel will have a colored rectangle around it in the status area.

2. Click the **Inter-CHs** button and select **Frequency CH1=CH2**. The instrument will stop signal generation, set the frequency of both channels to match the selected channel, and then automatically restart signal generation.

Set up Load Impedance

The output impedance of the AWG4162 is 50 Ω (in case of single-ended output). If you connect a load other than 50 Ω, the displayed Amplitude, Offset, and High/Low values are different from the output voltage. To make the displayed values same as output voltage, you need to set load impedance as follows:

1. Select the **System** tab and then select **Setting** from the left sidebar menu.
2. Click on **Load** in the desired channel to view the drop down menu.
3. To adjust the load impedance, select one of the following:
   - **50** to set the load impedance to 50 Ω.
   - **High Z** to set the load impedance to approximate to infinite. When dBm is specified for the output amplitude units, the amplitude units setting is automatically changed to Vpp if you select high.
   - **Custom** allows you to set the load impedance to a value of 1 Ω to 1 MΩ.
4. The load value is displayed in the Status menu.

Quick tips

- Load impedance is applied to the amplitude, offset, high/low level, and VOCM settings.

Set up VOCM

VOCM is “Voltage Output Common Mode” between + and – channels. In the case of differential output, the negative output DC VOCM is the same value of the DC VOCM of the positive output.

1. Select the **System** tab and then select **Setting** from the left sidebar menu.
2. Click on the number field next to the **VOCM** in the desired channel.

Quick tips

- VOCM isn’t related to what amplitude setting.
- Maximum of VOCM is related to the load impedance as following:

	AWG4162
(50ohm load, single-ended)	-2.5V ~ +2.5V
(high Z load, single-ended)	-5V ~ +5V

Invert Waveform Polarity

You can use the Invert button on the left sidebar menu to invert the polarity of a generated waveform. The following example shows how to get a differential signal using the invert function with a continuous sine wave.

1. Set up the instrument to generate a continuous sine wave on CH1.
2. Set the frequency of CH1 to a desired value.
3. Click the **CH1/CH2/CHBOTH** button and select **CHBOTH** to view CH1 and CH2 simultaneously.
4. Click on the **Inter-CHs** button and select **Frequency CH1=CH2** to set the CH2 frequency to match CH1.
5. Click **CH2** in the CH2 status bar.
6. Click on the **Invert** button on the left sidebar and notice that the CH2 waveform becomes inverted.

7. Push the front-panel **Ch1 On** button 

to enable the output.

Quick tips

- See the Quick tutorial: How to generate a sine waveform topic for a quick tutorial on getting started generating this waveform.
- See the Adjusting parameters of two channel signals topic to read about how to quickly set the frequency of one channel to match the other.

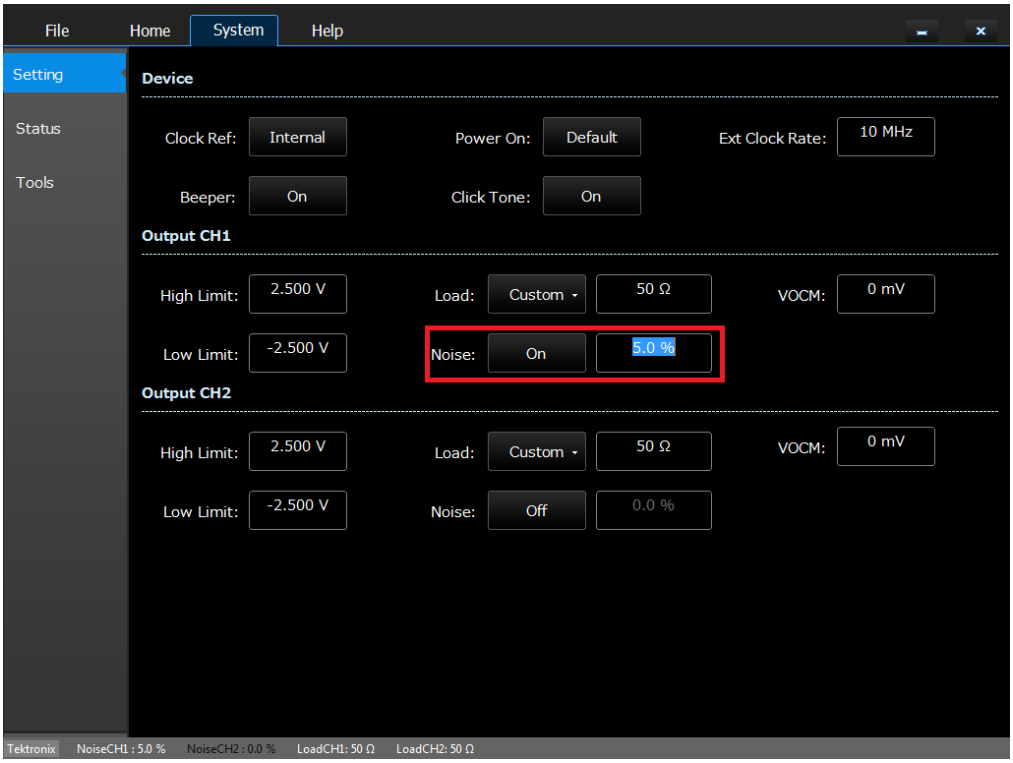
Add Noise

You can add the internal noise signal to a waveform using the following procedure. In this example, a continuous sine wave is used.

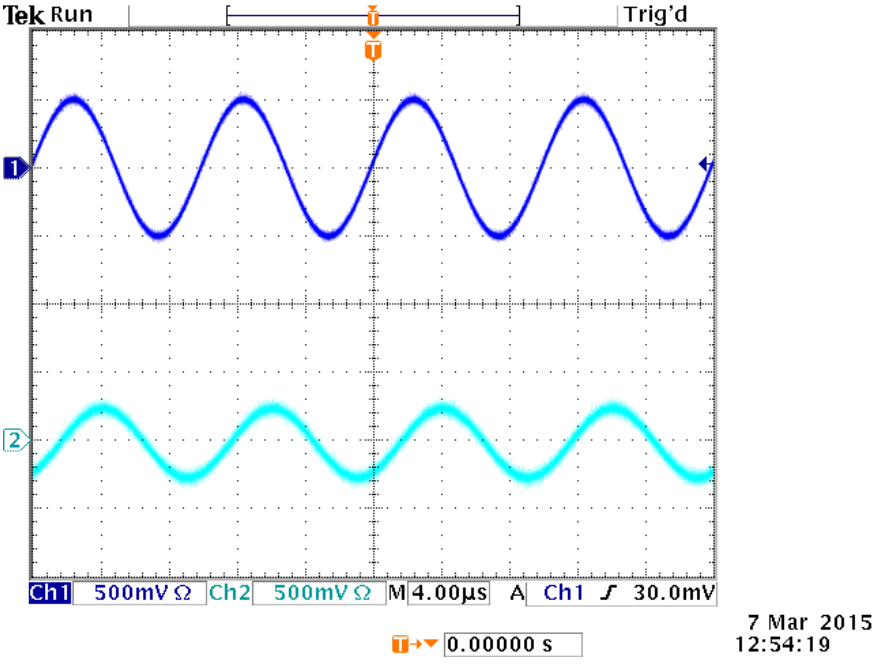
1. Select the **Sine** function 

from the top of the **Home** tab.
2. Select the **Continuous** run mode from the left sidebar menu.
3. Select the **System** tab and then select **Setting** from the left sidebar menu.
4. Click on **On/Off** button next to **Noise** in the desired channel to turn on noise add function.
5. Click on the number field next to the **Noise** parameter on the AWG4162 and adjust it as desired. **Noise Level** can't be modified when Noise is **Off**.

6. Observe the waveform adding noise displayed on the oscilloscope screen.

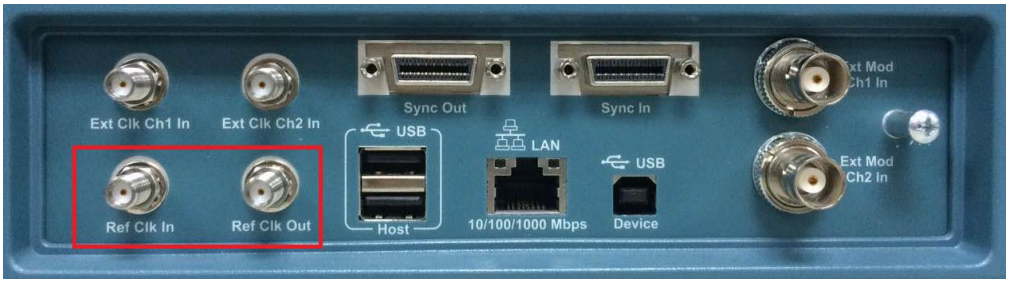


7. The top waveform is the one before adding noise. The bottom waveform is the one after adding noise. To avoid overflow by noise addition, the amplitude of the output signal is automatically halved.



Reference Clock Input

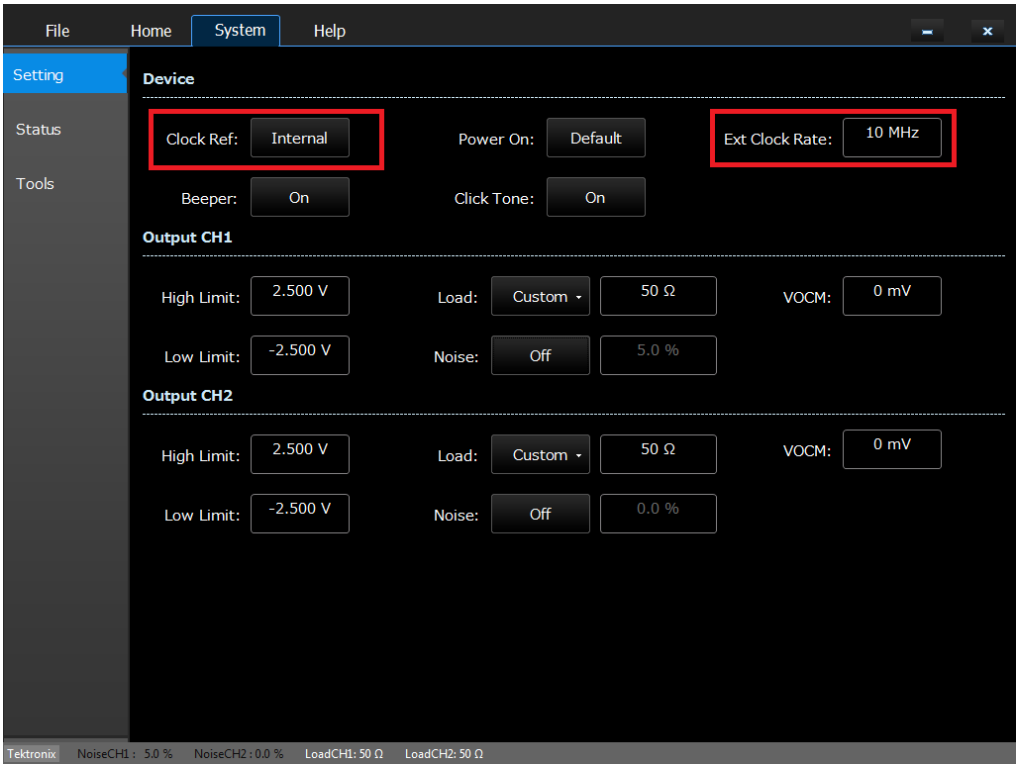
1. The Reference Clock Input (Ref Clk In) and the Reference Clock Output (Ref Clk Out) connectors are provided on the AWG4162 rear panel.



2. The instrument can use the internal or external source as a reference clock. To select a reference clock, push the front-panel **Utility** button 

and then select **Setting** from the left sidebar menu.

3. Click on **Clock Ref** to toggle between Internal and External.



Quick Tips

- The instrument can use the internal source or an external source as a reference clock. When the internal reference is activated, a 10 MHz reference clock is output on the rear panel Ref Clk Out connector.
- When the reference clock input is activated, the rear panel Ref Clock Input connector is used as the input for an external reference clock.

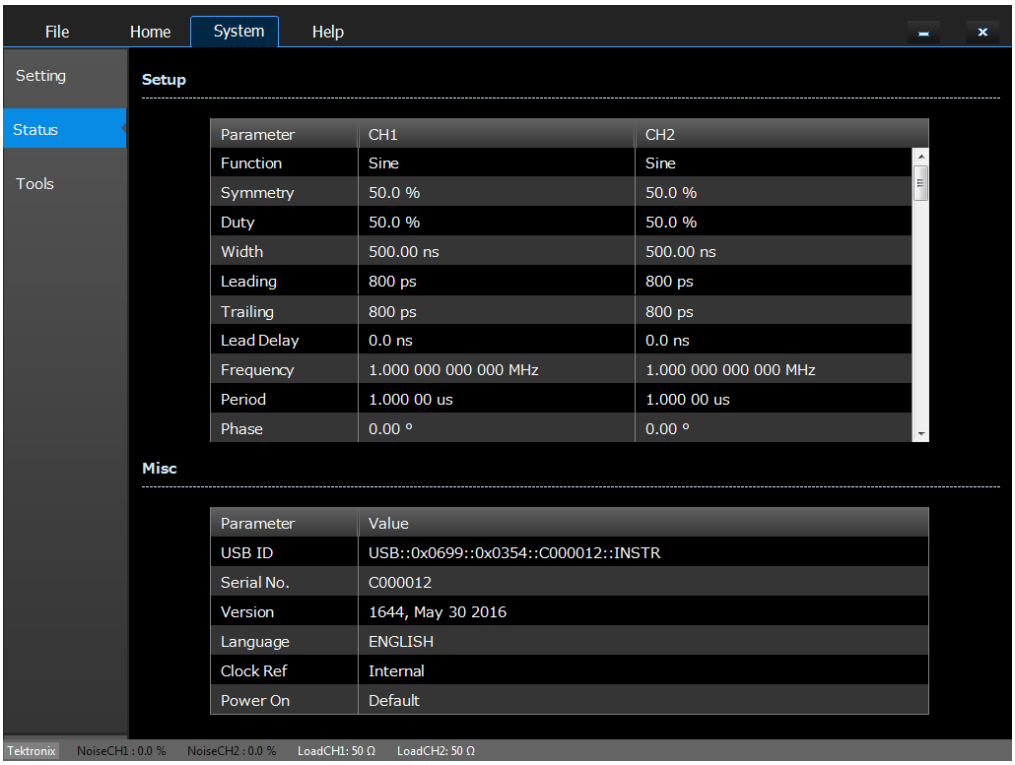
Utility Menu

Push the front-panel **Utility** button to display the **System** Tab. The **System** Tab provides access to utilities used by the instrument such as system related menus, Self Calibration, and Self Diagnostics.

1. Push the front-panel **Utility** button 

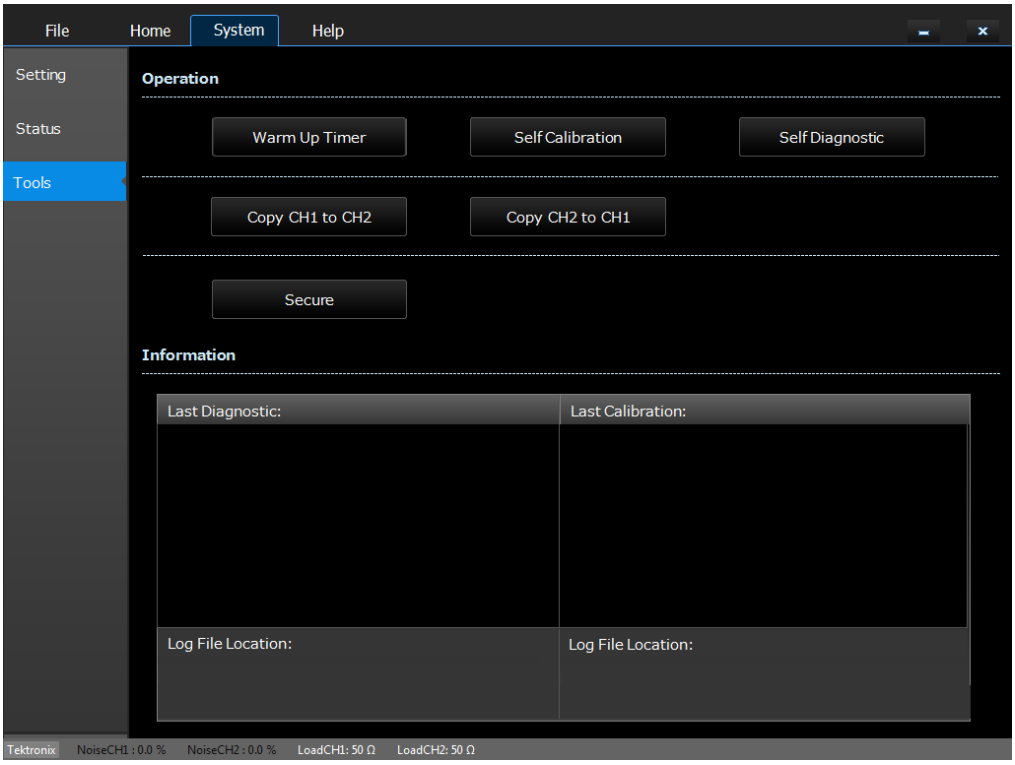
to display the **System** Tab. Select the **Setting** from the left sidebar menu to display the system related menus.
2. **Clock Ref**. (See Reference Clock Input.)
3. You can select the instrument **Power On** setting.
4. You can modify the **Ext Clock Rate** if select **Clock Ref** source as External.
5. Click on **Beeper** to toggle the beep sound Off and On.
6. Click on **Click Tone** to toggle the click tone Off and On.
7. You can modify **High Limit** and **Low Limit** for the desired channel.
8. **Load** Impedance. (See Set up Load Impedance.)
9. **Noise**. (See Add Noise.)
10. **VOCM**. (See Set up VOCM.)

Select the **Status** from the left sidebar menu to display the instrument status.



Select the **Tools** from the left sidebar menu.

11. **Warm Up Timer** is used to calculate the warm up time.
12. **Self Calibration** (See Perform Instrument Self Calibration and Self Diagnostic.)
13. **Self Diagnostic**. (See Perform Instrument Self Calibration and Self Diagnostic.)
14. You can copy the waveform parameter of one channel to another channel by clicking on **Copy CH1 to CH2** or **Copy CH2 to CH1**.
15. **Secure**. (See Erase Custom Waveform Files.)

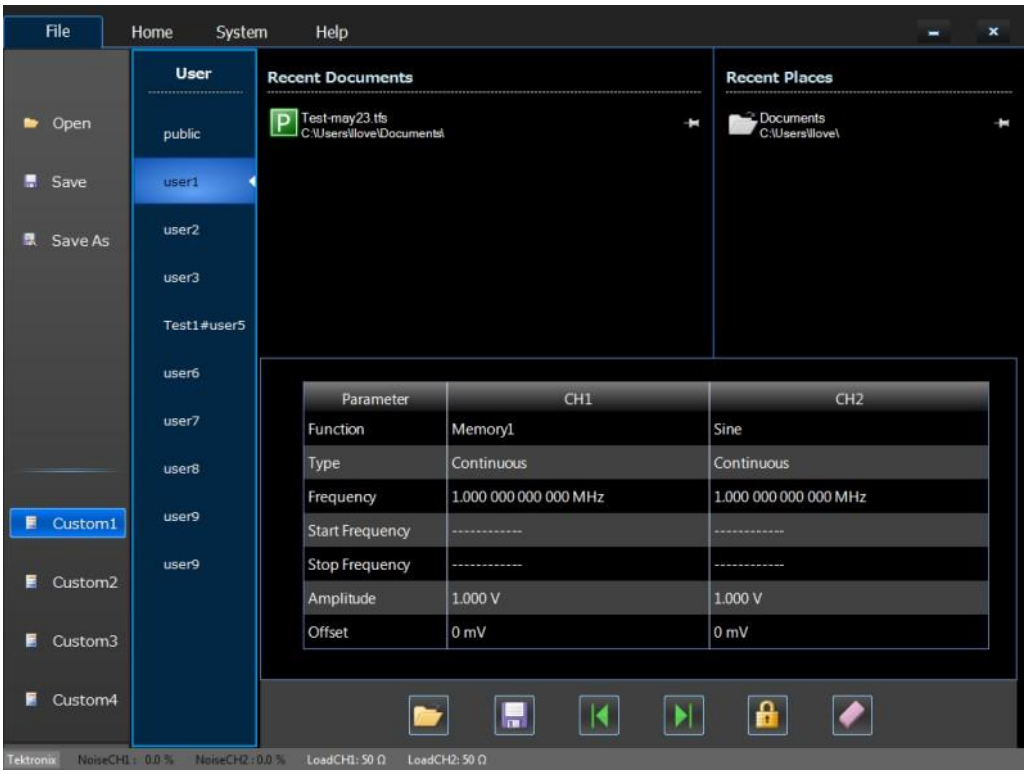


Save/Recall a Custom Setup

You can save up to four custom setups in the instrument internal memory. They are saved as Custom 1 through 4 under the File tab. You can save more setups to the Local Disk or a USB memory device.

Save a Custom Setup by UI

1. Click the File tab.



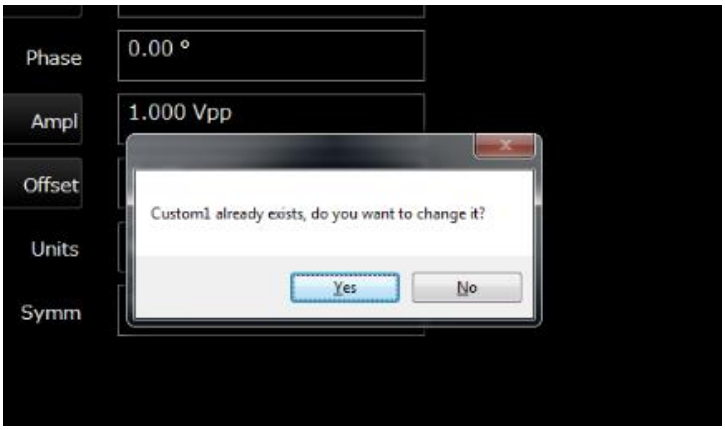
2. Click the desired Custom button which you want to save.
3. Click the save icon  to save.

Save a Custom Setup by Front Panel.

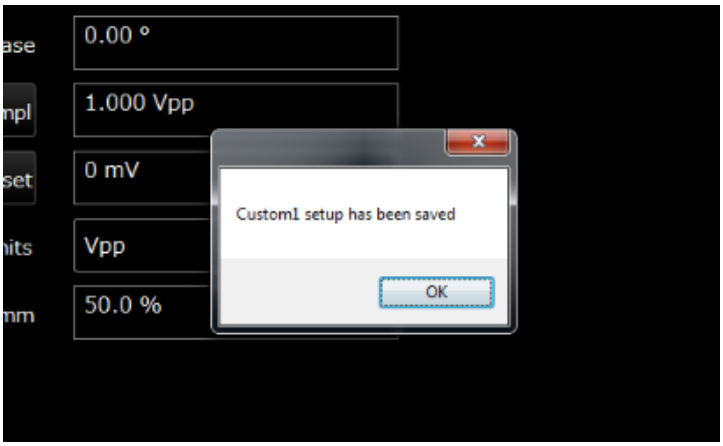
1. Hold down the **Custom1** button 

in front panel file area.

2. A warning dialog will show if the custom1 file exist.

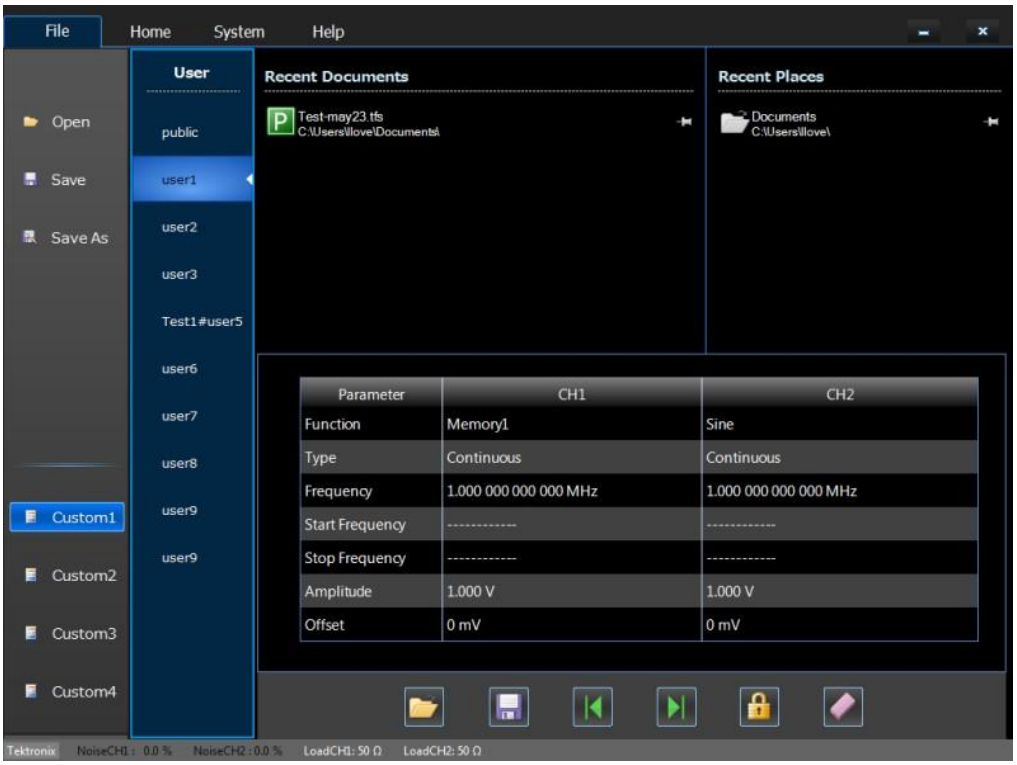


3. Click **Yes**.
4. An information dialog will show “Custom1 setup has been saved”.



Recall a Custom Setup by UI

1. Click the **File** tab.
2. Click the desired Custom button to view the waveform parameters and make sure it is the desired waveform. The parameters appear in the bottom half of the display.



3. Click  to load the waveform file. You will now see the waveform on the AWG4162 display.

Recall a Custom Setup by Front Panel

1. Push the **Custom1** button 

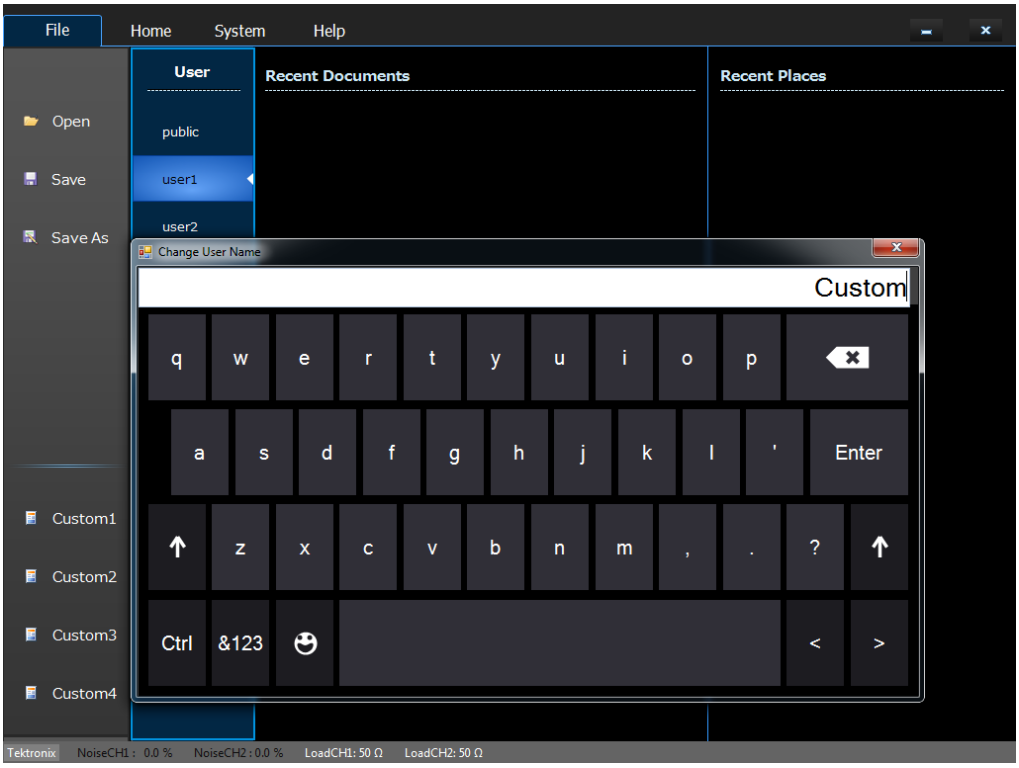
in front panel file area.
2. You will see the custom1 waveform display if custom1 exist. A warning dialog will show if the custom1 doesn't exist.

Quick tips

- Click  to lock a custom waveform file from being erased or edited.
- Click  to erase the selected custom waveform file.
- Click  to quickly navigate the four available custom files in the menu.
- Click  to open a custom waveform file.
- Click  to save a custom waveform file.

Save/Recall a Custom Setup for Different User by UI

1. Click the **File** tab.
2. Click the desired **User** button to view the **Recent Documents** and **Recent Places** for different user. You can rename the User 1-9 by double click.



Save a Screen Image

You can save a screen image of the instrument. Do the following steps:

1. Set the display to show the screen you want to save as image. Then simultaneously push the two arrow keys  underneath the rotary knob on the front panel.
2. A message appears on the screen, indicating that the screen image was saved.

Quick Tips

- Image files are saved in the path "D:\Tektronix\AWG4000\Basic"
- Image files are saved as .BMP format. The instrument gives all files created by the instrument the default name yyyy-mm-dd-hh-mm-ss.BMP.

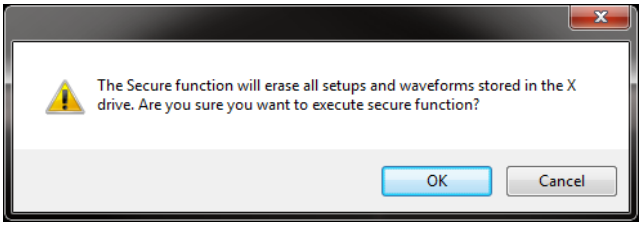
Erase Custom Waveform Files

Erase instrument setups and waveforms from memory

You can also erase all instrument setups and waveforms from the instrument internal memory using the following procedure.

NOTE. *You can restore the instrument to its default settings at any time without erasing memory by pressing the Default button located in the top right corner of the Basic mode display.*

1. Click the **System** tab.
2. Click the **Tools** tab in the right sidebar menu.
3. Click the **Secure** button and the following dialog will appear.



4. Select **OK** to erase all setups and waveforms stored in hard drive X:\, or select **Cancel** to cancel the operation.

ArbBuilder

You can create a waveform using the ArbBuilder tool. You can create new waveforms by selecting from a list of standard functions, using the Equation Editor (See Create a Waveform by Equation.), or drawing a waveform (See Draw a Waveform with ArbBuilder.).

You can open ArbBuilder from the Home tab by selecting **New** 

or **Edit** 

from the drop down menu.

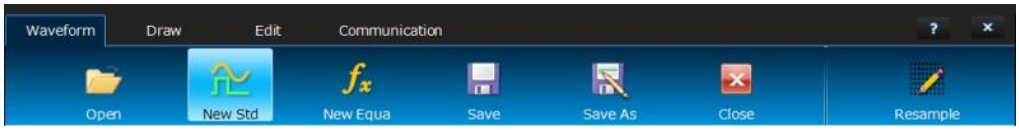
Create a Standard Waveform

1. Select **ArbBuilder** > **New** 

or Edit 

from the Home tab.
2. Select **New Std** 

from the Waveform tab.



3. Select a standard waveform from the **Function** drop down menu. You can choose Sine, Square, Triangle, Pulse, Noise, DC, Exponential Rise, Exponential Decay, Sin(x)/x, Sweep, Multi Tone, Lorentz waveform.
4. Adjust the vertical and horizontal parameters as desired.
5. Click the **Preview** button to view the waveform.

6. Click the **OK** button to view the waveform, or click **Cancel** to cancel and exit the window.
7. Click the **SAVE** button  or SAVE AS button  to save the waveform, or click **Close** button  to close the waveform.
8. Click table to turn to communication page.
9. Click **Send to CH1**  or **Send to CH2**  under communication tab to send this waveform to the channel 1 or channel 2.

Create a Waveform by Equation

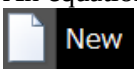
You can generate a waveform by equation editor. You can use this function in the ArbBuilder tool.

NOTE. *The Equation editor processes all file inputs and outputs in the current working directory. The current working directory must have read and write access or the equation file will not compile.*

Equation Editor Overview

The Equation editor is an ASCII text editor that allows you to create, edit, load, and compile equation waveform definitions into a waveform using the Waveform Programming Language (WPL). Use WPL to generate a waveform from a mathematical function, perform calculations between two or more waveform files, and use loop and conditional branch commands to generate waveform values. Compile the equation file to generate the described waveforms.

The equation editor processes all file inputs and outputs in the current working directory. The current working directory must have read and write access. Compilation may be dependent on the available memory and other resources of the instrument.

An equation file is a text file that you create and edit in the equation editor. Select **ArbBuilder** > **New**  **or Edit**  **Waveform** tab and click the **New Equa**  icon to open the equation editor.

The following table describes the screen elements of the equation editor.

Element	Description
Toolbar	Provides edit operations, such as open, save, cut, copy, and paste.
File name	The file name to which the equation or text is written, or the name of the file being edited. The instrument appends the default .equ file extension to all Equation editor files.
Equation	The area where you enter text and/or equation information.
Output	Displays the status of the compilation. If the compilation fails, then the application displays an error message. If the compilation is successful, then the application displays “Compiled Successfully”.
Command List	Keypad of math functions, numbers, and letters for creating equations.
Preview	Display of waveform graph after compile.
Compile	Button that compiles the currently loaded or edited equation file. The status of the compilation is displayed in the output window.
Settings	Provides controls for adjusting range and points.
OK and Cancel buttons	Use these buttons to save and exit (OK) or cancel and exit (Cancel) the Equation Editor window.

The component menu contains the items used to set the time range as well as functions, operators, variables, constants, syntax items and characters. You can use these items to create equations and enter comments.

Component	Symbol	Meaning	Example
Syntax Items	( )	These are parentheses — (and) — for specifying the order of operations. Each opening (left) parenthesis must be paired with a closing (right) parenthesis. When there are two arguments – for example, range, max, min – they are separated with a , (comma).	NA
Variables. Here are the variables that can be used in an equation.	t x v	Time from the head of that range() statement Variable taking on a value from 0.0 to 1.0 within that range ( ) Variable showing the current value of the waveform data at that position statement	NA
Operators	+, –, *, /	These add, subtract, multiply, or divide the components. The priorities are the same as usual for these four operators – * and / have priority over + and –.	NA
	^	Expresses exponents. Only integers can be raised to a higher power. ^ has the same priority as * and /. Therefore, parentheses are required to give priority to multiplication.	pi * (2^3) * x where 2^3 = two raised to the third power.
Comment	#	Comments are preceded by a number sign (#). When a number sign is entered, all characters after that until the end of the line are treated as a comment. All of the items in the component menu can be used in a comment.	NA
Characters	a–z, %, $, &, @, A, _	The characters available in the component menu are the letters of the alphabet (a–z) and several symbols (%, $, &, @, A and _). These are used in comments.	NA

Other items	pi, e, k, =,		NA
	pi	The circumferential ratio.	NA
	e	Exponent (for an implied 10). The range for numbers expressed in this scientific notation is from |5.9e–39| to |3.4e38|.	1e6=1,000,000, 1e–3=0.001
	k	The k0–k9 can be specified; these are constants that may be used in equations. Specifying a new value for the same k# replaces the old value with the new one. If no constant is defined for k, this value will be automatically set to 0.	NA
	=	Equals sign. = is used with k constants.	k0=2*pi
		Ends the line for the range or equation; inserting a return ( ) in the middle of the line partitions it.	NA
Functions	sin(, cos(	The arguments for these trigonometric functions are in radians.	range(0,100 s)cos(2*pi*x) Example: range(0,100 s)sin(2*pi*1e4*t)
	exp(, log(, ln(	Exponential function, common log function, natural log function. The log and ln arguments must be positive.	range(0,50 _s) 1–exp(–5*x) range(50 _s,100 _s) exp(–5*x) Example: range(0,100 _s) log(10*(x+0.1)) Example: range(0,100 _s) ln(2*(x+0.2))
	sqrt(	The square root; the argument must be a positive value.	range(0,100 _s) sqrt(sin(pi*x))
	abs(	The absolute value.	range(0,100 _s) abs(sin(2*pi*x))
	int(	Truncates the fraction to obtain the integer.	range(0,100 _s) int(5*sin(2*pi*x))/5
	round(	Rounds off the fraction to obtain the integer.	range(0,100 _s) round(5*sin(2*pi*x))/5
	norm(	Normalizes the range specified with range() and scales the amplitude values so that the maximum absolute value is 1.0 (that is, a value of +1.0 or –1.0). The norm() statement comprises an entire line.	range(0,100 _s) sin(2*pi*x)+rnd()/10 norm()
	max(min(	Takes the larger of two values. Takes the smaller of two values.	range(0,100 _s) sin(2*pi*x) range(0,50 _s) min(v,0.5) range(50 _s,100 _s) max(v,–0.5)

	range(	The equation must specify the time domain. If the time domain is not defined, this is an error. The time domain is specified with range(). When making a new equation file, range(0, is input in the first line of equation. Next, the time is specified. This setting is valid until the next range(item is specified. With the first range() specification, any number of lines of equation can be input. Text written after the range() on the same line is invalid. Here is the format for the range(item. range(Equation starting time, Equation ending time)	range(0,1ms) Time range sin(2*pi*x) Equation
	rnd (integer from 1 to 16,777,215)	When an argument is specified, generates a random number sequence using that argument as the initial value. If the argument is omitted, 1 is used.	range(0,100 _s) rnd(2)/3
	diff(	Differentiates the function over the range specified with range(). Specified with diff(). The diff() comprises an entire line.	range(0,33 _s) –0.5 range(33 _s,66 _s) 0.5 range(66 _s,100 _s) –0.5 range(0,100 _s) diff()
	integ(	Integrates the function over the range specified with range(). Specified with integ(). The integ() comprises an entire line. After integ(), specify normalization (norm()) as necessary.	range(0,33 _s) –0.5 range(33 _s,66 _s) 0.5 range(66 _s,100 _s) –0.5 range(0,100 _s) integ() norm()
	mark (marker1 or marker2)	Sets the marker for the range set with range(). After compiling, there is no marker display, but the set marker can be verified with the waveform editor. The mark() statement comprises an entire line. For example, when mark(1) is input, nothing else can be input on that line.	NA

Use the equation editor Units menu to specify the units for the parameters or variables used in the equation. The following table lists the units that you can use and their descriptions.

Unit	Meaning
m	milli (e^{-3})
u	micro (e^{-6})
n	nano (e^{-9})
p	pico (e^{-12})
s	second
,	comma separator
K	Kilo (e^{3})
M	Mega (e^{6})

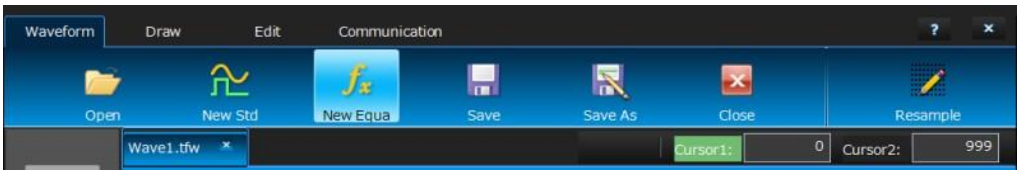
Use the selection menu to confirm, backspace, or clear the equation. The following table lists the units that you can use and their descriptions.

Button	Meaning
Enter	Confirms the selection and moves to the next line of the equation
BKSP	Backspaces over the last character. Works like the backspace key on the keyboard
CLR	Clears the entire equation

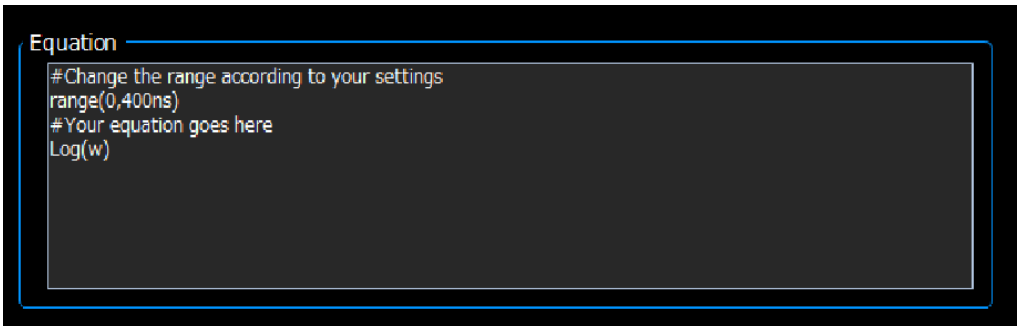
Create a Waveform by Equation Editor

1. Select **ArbBuilder > Edit > Waveform** tab and click the **New Equa** icon 

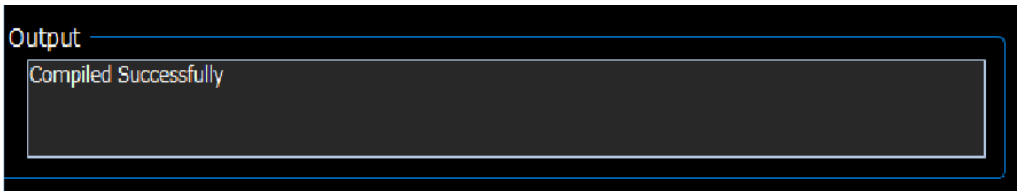
to open the equation editor.

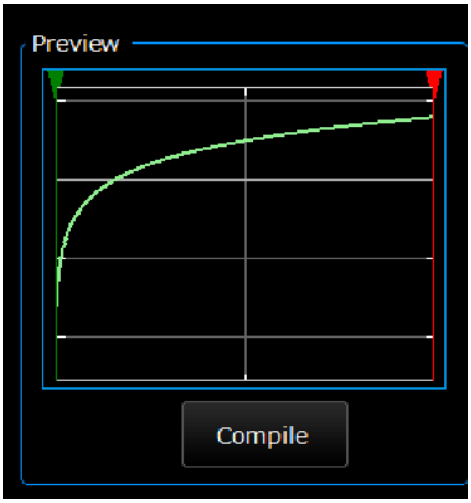


2. In the Equation Editor window, enter text to form a waveform equation. For example, type “Log(w)” in equation input box.



3. Click **Compile** button to generator a waveform in the preview box. You will see “Compiled Successfully” information in the Output box and a log waveform display in the Preview box. If you enter an invalid equation, the Output Box will show error warning message and the error line in the equation input box will turn to red.





4. Click the **OK** button to display the waveform in editable mode, or click **Cancel** button to cancel the operation.
5. You will see a log waveform. You can edit, save or send it to channel 1 or 2.

NOTE. *The equation editor supports only the basic 7-bit ASCII character set. The maximum length of a single string is 256 characters, including spaces. Concatenate strings by entering a colon character (:) at the end of a line. The maximum length, that is, sum of all string lengths is 1000.*

Save a Waveform Equation

1. Create a waveform equation by following step 1 – 3 in the "Create a waveform equation".
2. Click  button in the equation editor toolbar.
3. A Windows **save as** dialog box will show up. Enter a filename and save. The equation file will be saved in the .eqa format file.

Open an Equation File

1. In the Equation Editor window, click  in the equation editor toolbar.
2. Select an existing waveform equation file and click **Open**.

Edit an Equation File

In the Equation Editor window, you can do the following to modify equations:

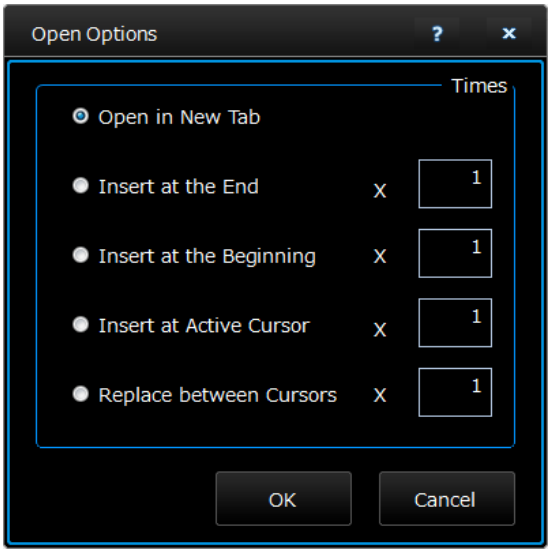
- Cut. Highlight the part of the equation you want to cut and then click this icon .
- Copy. Highlight the part of the equation you want to copy and then click this icon .
- Paste. Highlight the part of the equation you want to paste and then click this icon .
- Directly edit the text using a keyboard or the Command List keypad.

Open a Waveform with ArbBuilder

1. Select **ArbBuilder** > **New or Edit** to open the ArbBuilder tool.
2. Select **Open button** 

from the Waveform tab.
3. The windows file open dialog box pop up. Please select an existing waveform file. ArbBuilder supports several formats of waveform, such as .wfm, .pat, .txt, .tfw and .isf. The formats supported by ArbBuilder.
4. You can open the files with 5 methods:

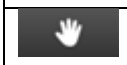
- **Open in New Tab**: You can open the file in a new tab.
- **Insert at the End**: You can insert the new file at the end of current waveform.
- **Insert at the Beginning**: You can insert the new file at the beginning of current waveform.
- **Insert at Active Cursor**: There are two cursors on the waveform view. Click the cursor you want to insert new waveform. Then click **OK**, the new file at the cursor of current waveform.
- **Replace between Cursors**: There are two cursors on the waveform view. The new waveform will be inserted between the two cursors.



5. Click **OK**, the file will be opened.

Draw a Waveform with ArbBuilder

1. Open a waveform. You can open a standard waveform (See Create a Standard Waveform.).
2. Click the **Draw** tab.
3. Use the icons in the **Draw** toolbar to create your waveform.

	Redo is used to restore the previous state.
	Undo is used to restore the next state.
	Zoom In is used to enlarge the waveform graph.
	Zoom Out is used to shrink the waveform graph
	Horizontal Zoom In is used to enlarge the waveform graph in horizontal direction.
	Fit To Window is used to restore the default state of waveform.
	Move With Hand is used to move the waveform by hand.

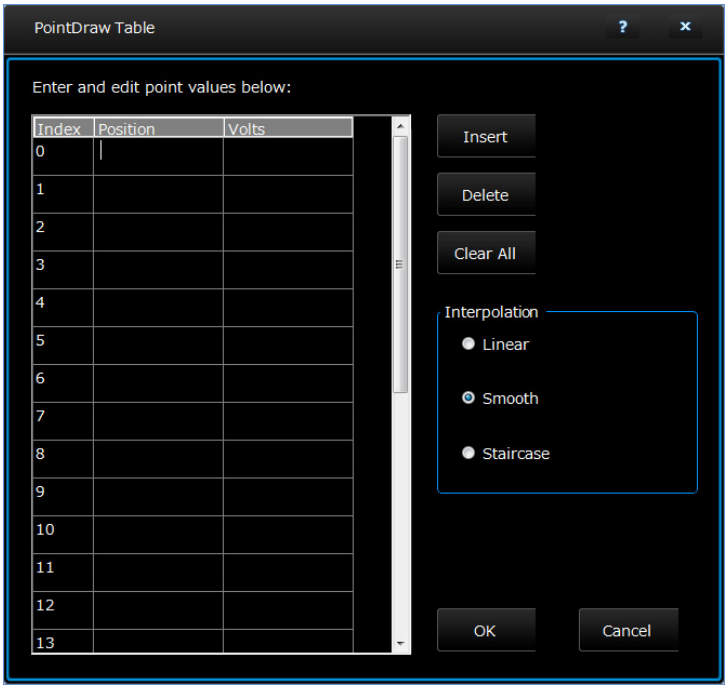
4. **Freehand**  means you can draw waveform on both horizontal and vertical points. **Horizontal**  means you can draw waveform on horizontal direction. **Vertical**  means you can draw waveform on vertical method. **Point**  means you can draw some points and the points can be interpolated in the way of linear, smooth or staircase.

5. If you select **Point** icon 

, then the **Point Draw** toolbar appears. You can click on the graph, select points and then use the toolbar icons to edit them. You can use tools in the following table.

6. If you click **Table** icon 

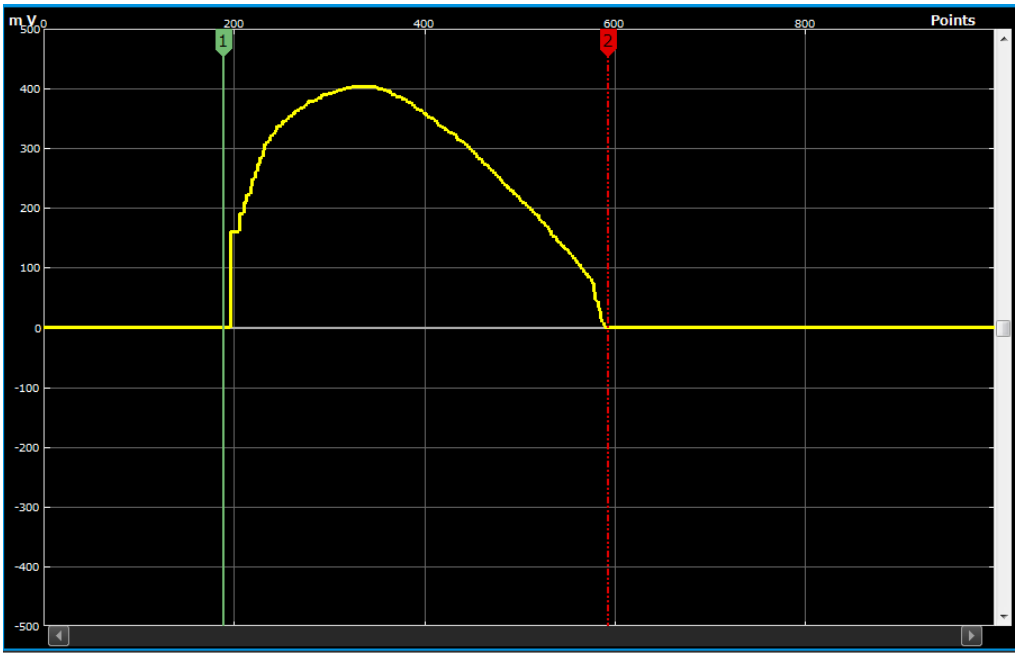
, the **Point Draw Table** window will open. You can enter points directly into this table and click **OK** to save. You can Insert, Delete or Clear all the data points.



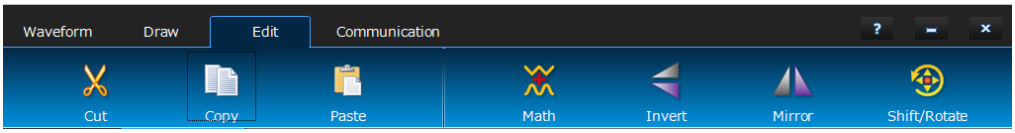
7. You can select from the interpolation  menu and apply one of the three selections to the waveform. **Linear**: Points can be connected with straight line. **Smooth**: Points can be connected with smooth line. **Staircase**: Points can be connected with staircase.

Edit a waveform with ArbBuilder

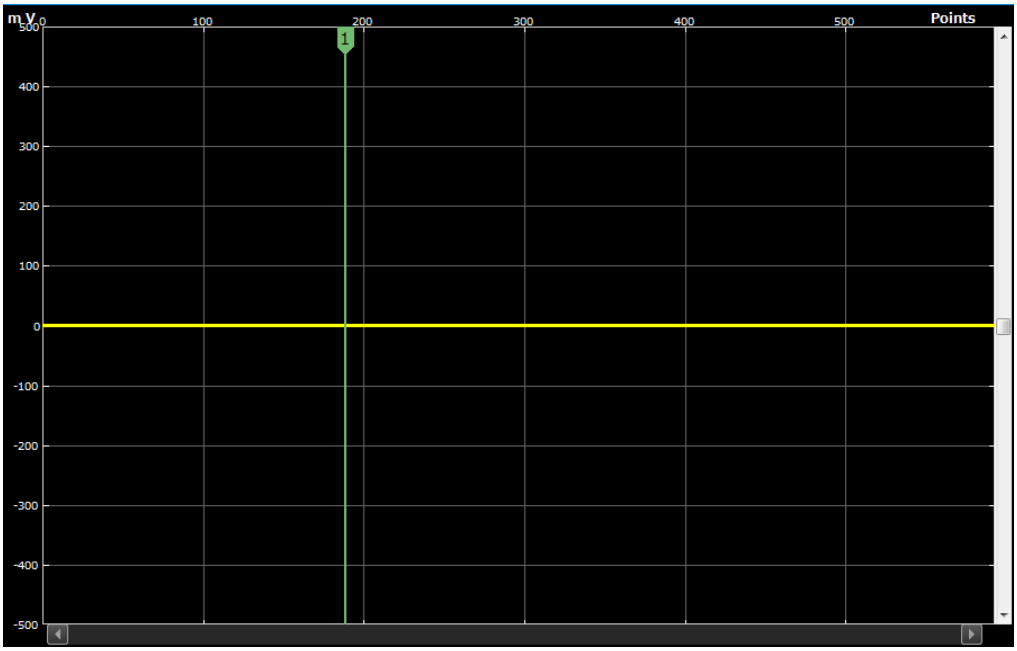
1. Open a waveform. Please see Open a Waveform with ArbBuilder.



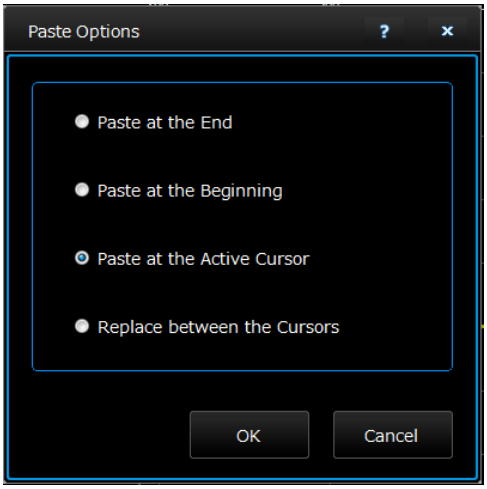
2. Click **ArbBuilder->Edit**. You will see the edit toolbar.



3. **Cut** . It is used to cut a piece of waveform data between the two cursors.



4. **Copy** . It is used to copy a piece of waveform data between the two cursors. The waveform will not be changed.
5. **Paste** . If the **Paste** button is clicked, a dialog box is pop up. There are four paste options.

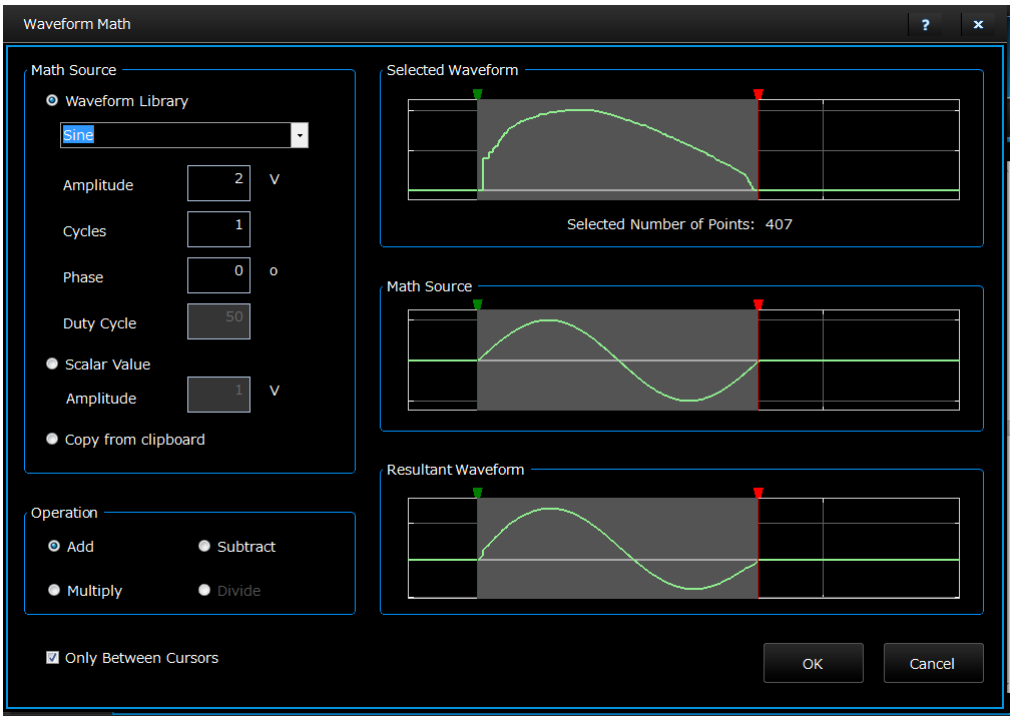


Item	Description
Paste at the End	Paste a waveform segment at the end of a waveform
Paste at the Beginning	Paste a waveform segment at the beginning of a waveform
Paste at the Active Cursor	Paste a waveform segment behind an active cursor.
Replace between the Cursors	Replace the waveform between the cursors by a waveform segment.

6. Math

It is used to make an arithmetic for a waveform. The arithmetic contains four operations which are addition, subtraction, multiplication and division. When the **Math** button 

is clicked, a dialog box is pop up.

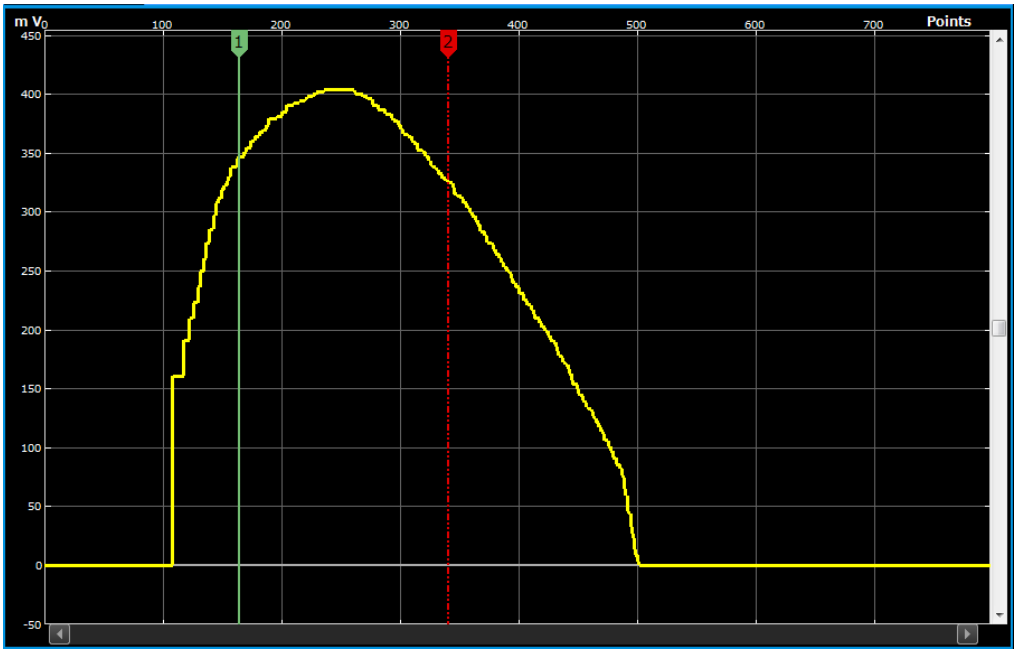


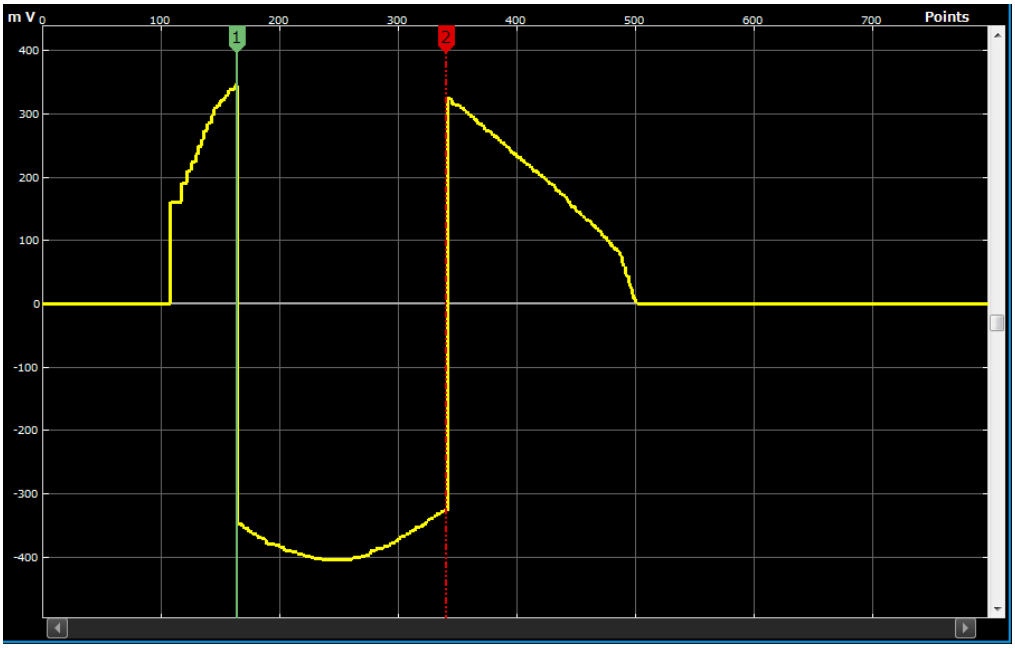
There are three kinds of math sources which are waveform library, scalar value and copy from clipboard.

Item	Description
Waveform Library	There are seven kinds of waveform which are Sine, Square, Triangle, Pulse, Noise, Exponential Rise, Exponential Decay.
Scalar Value	An amplitude value set by the user
Copy from clipboard	A piece of waveform from Copy or Cut operation.

7. **Invert** 

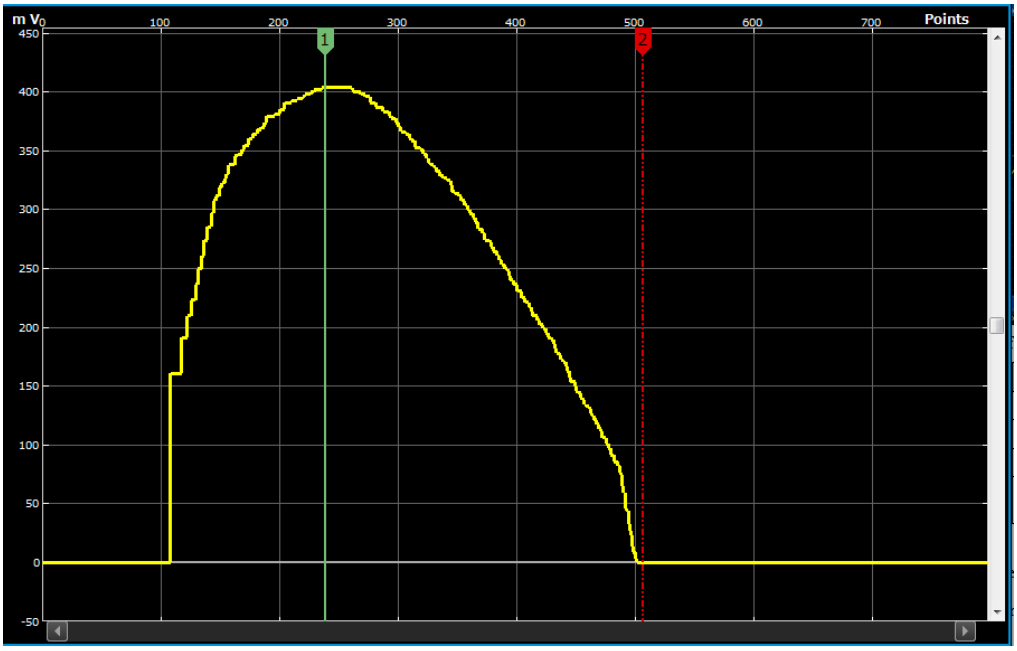
. It is used to overturn the waveform between the cursors in the vertical direction.

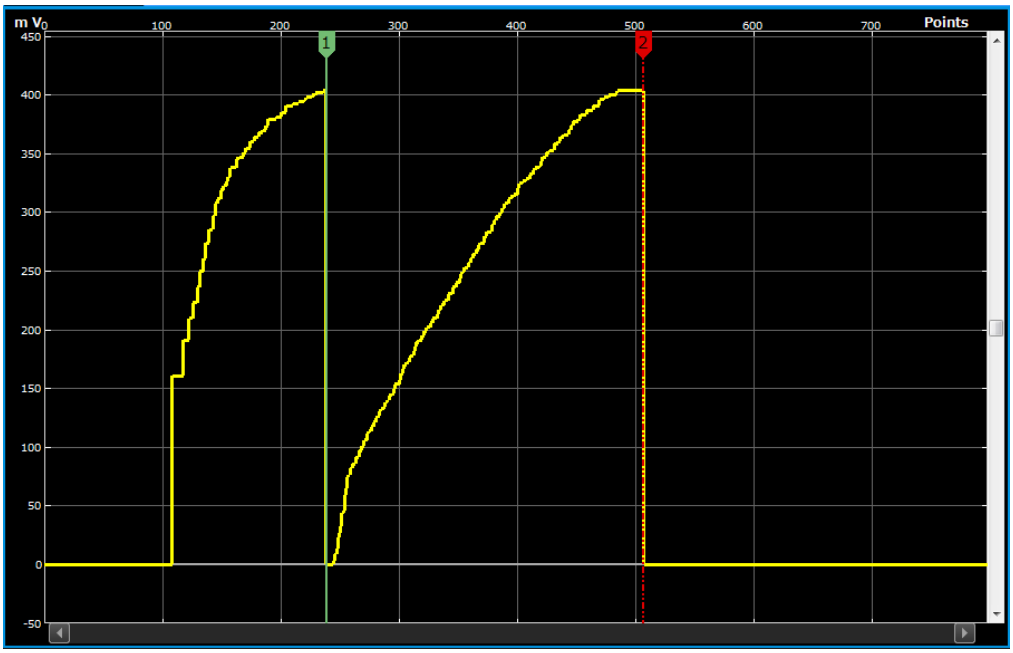




8. **Mirror** 

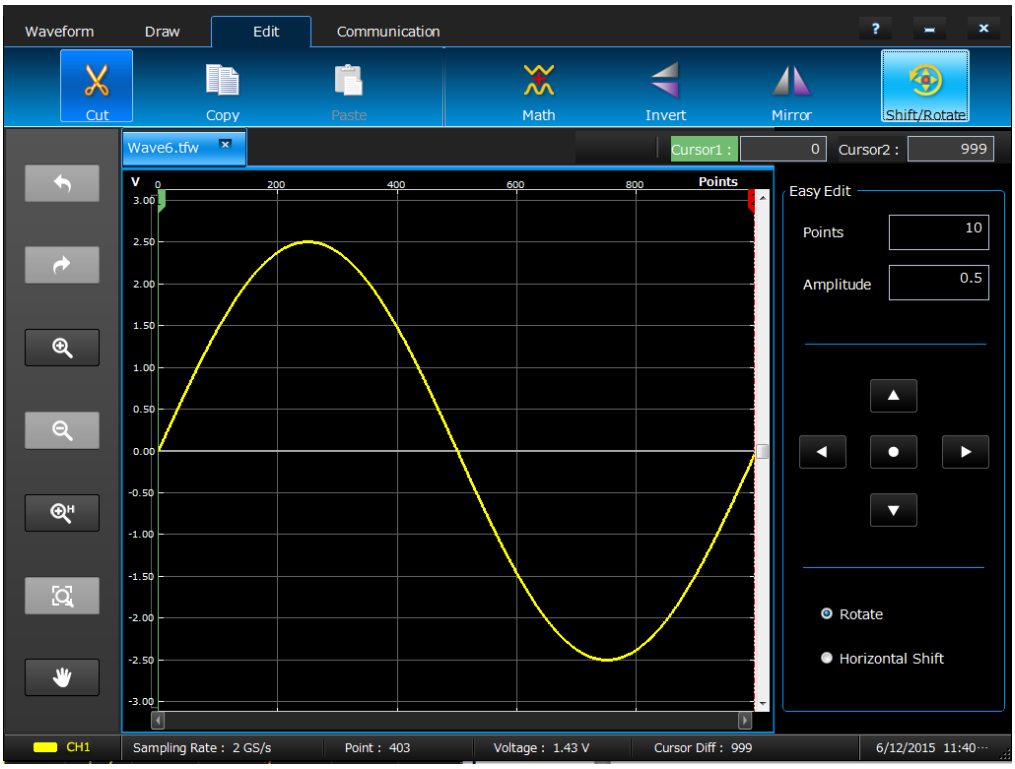
. It is used to overturn the waveform between the cursors in the horizontal direction.





9. **Shift/Rotate** 

. When the user clicks the **Shift/Rotate** button, a panel is pop up in the right of main panel zone. The waveform display zone will shrink.



Item	Description
Points	Set the point number which is moved each time
Amplitude	Set the amplitude which is moved each time
Rotate	Move the waveform periodically
Horizontal Shift	Move the waveform and use zero to fill the data after the waveform

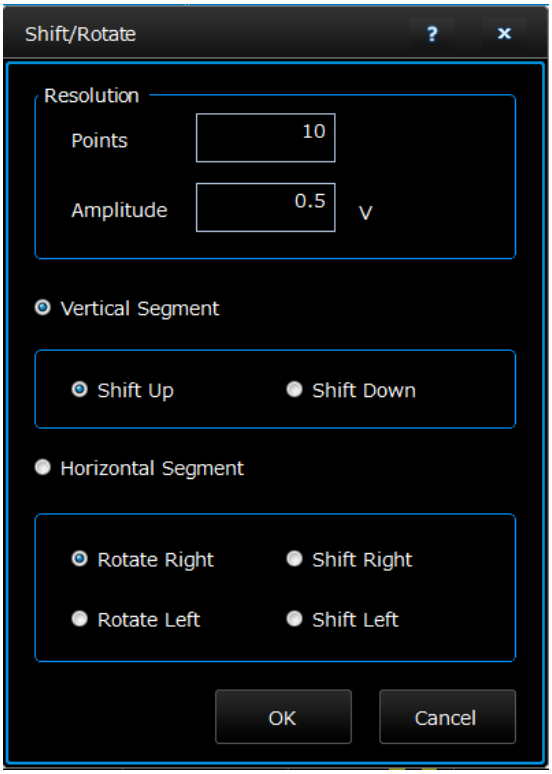
 Shift left is used to move the waveform left.

 Shift right is used to move the waveform right.

 Shift up is used to move the waveform up.

 Shift down is used to move the waveform down.

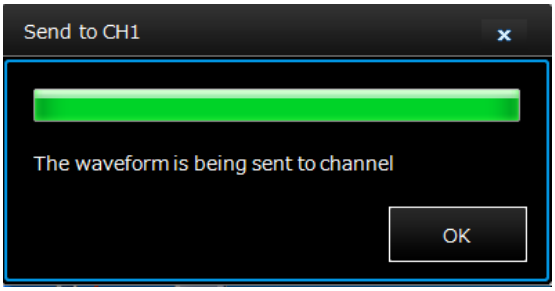
 Shift/Rotate can pop up a dialog and realize the shift/rotate function all the same.



Send a Waveform to CH1/CH2

1. Click **Communication** in the ArbBuilder.

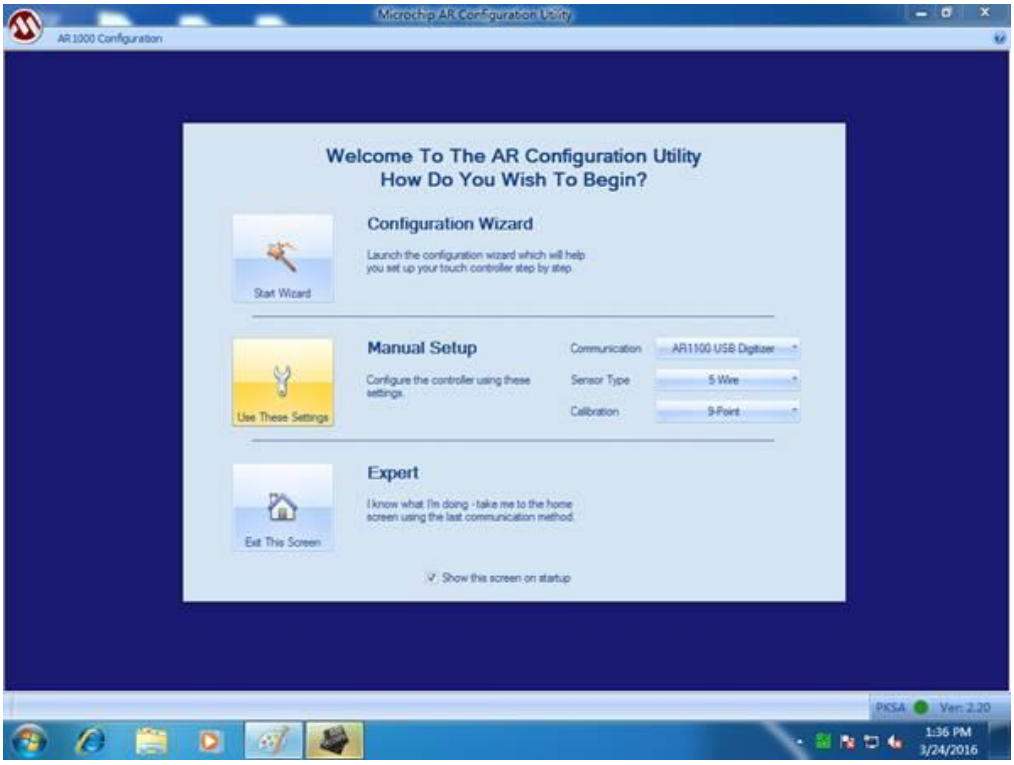
2. Click **Send to CH1**  or **Send to CH2**  . You will see a progress bar. The waveform is sent to Basic.



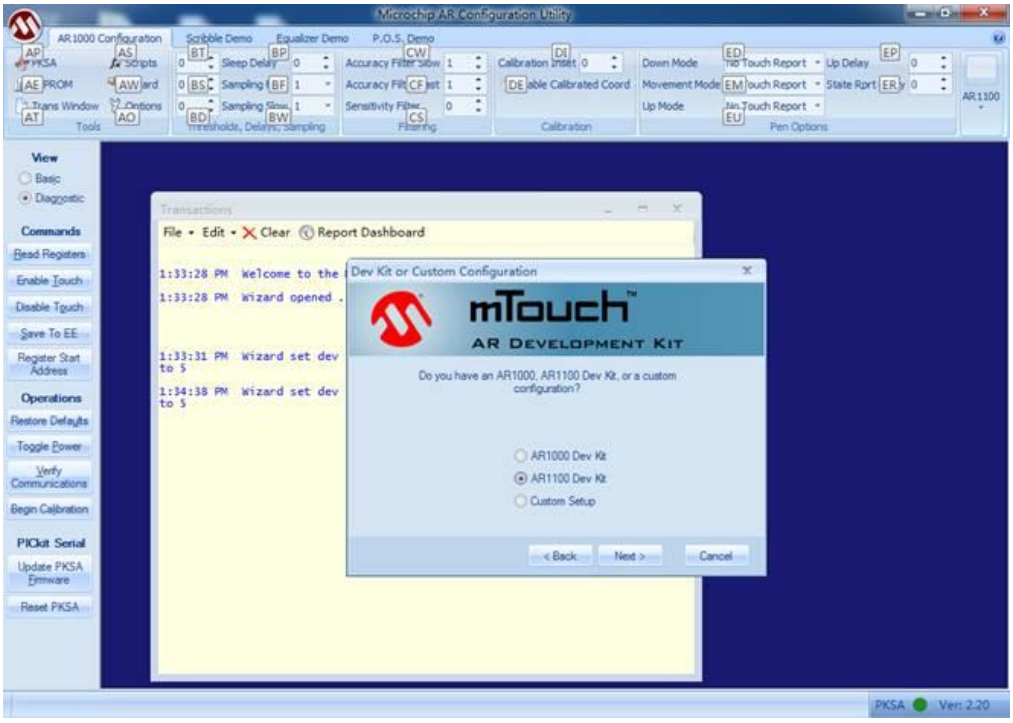
Appendix

Touch Panel Calibration

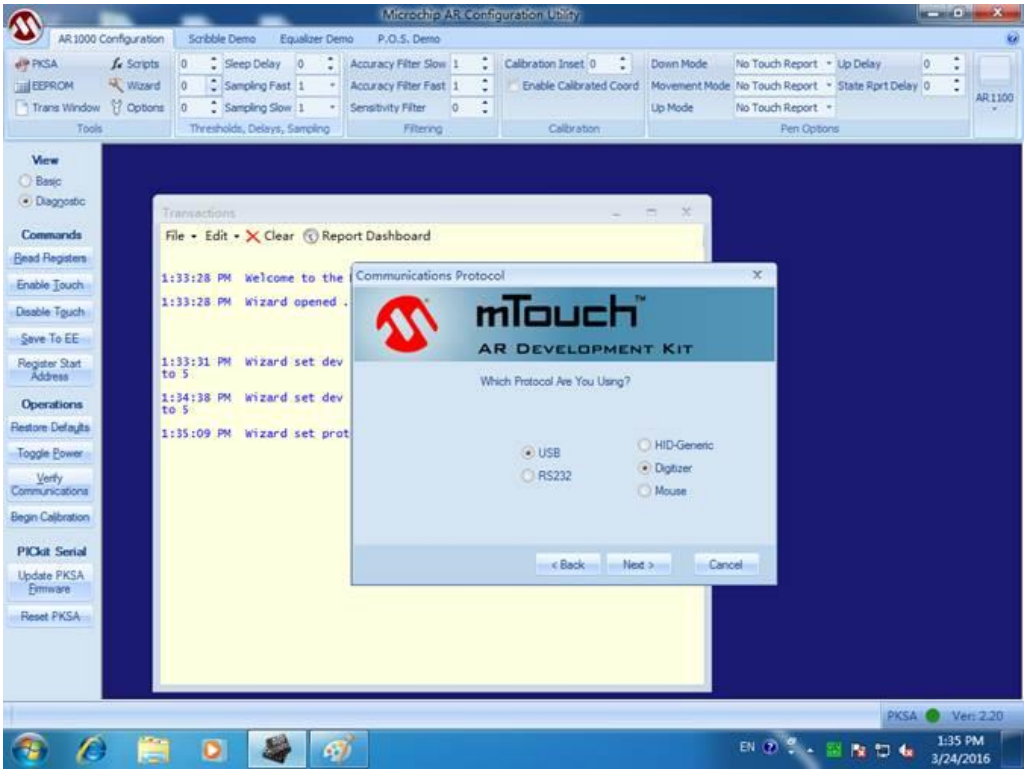
1. Double click “Microchip AR Configuration Utility” on the desktop.
2. Choose “Configuration Wizard”.



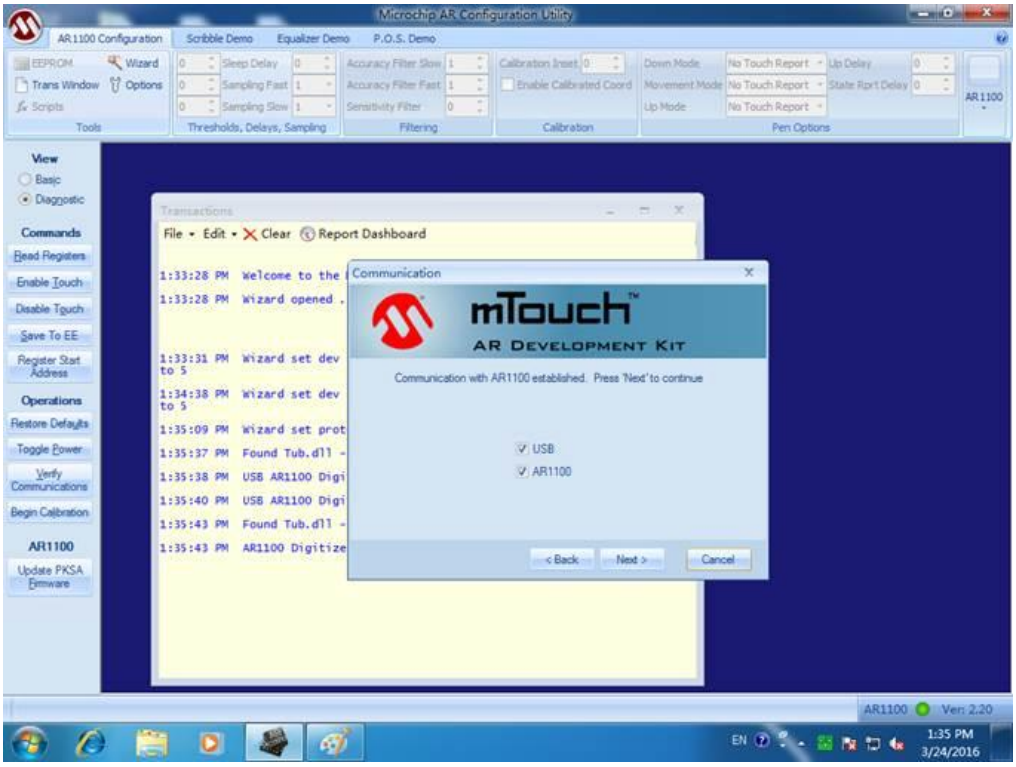
3. After click next will show below dialog, please choose “AR1100 Dev Kit”.



4. After that it will show the dialog as below, please choose “USB” and “Digtizer”, and please wait.

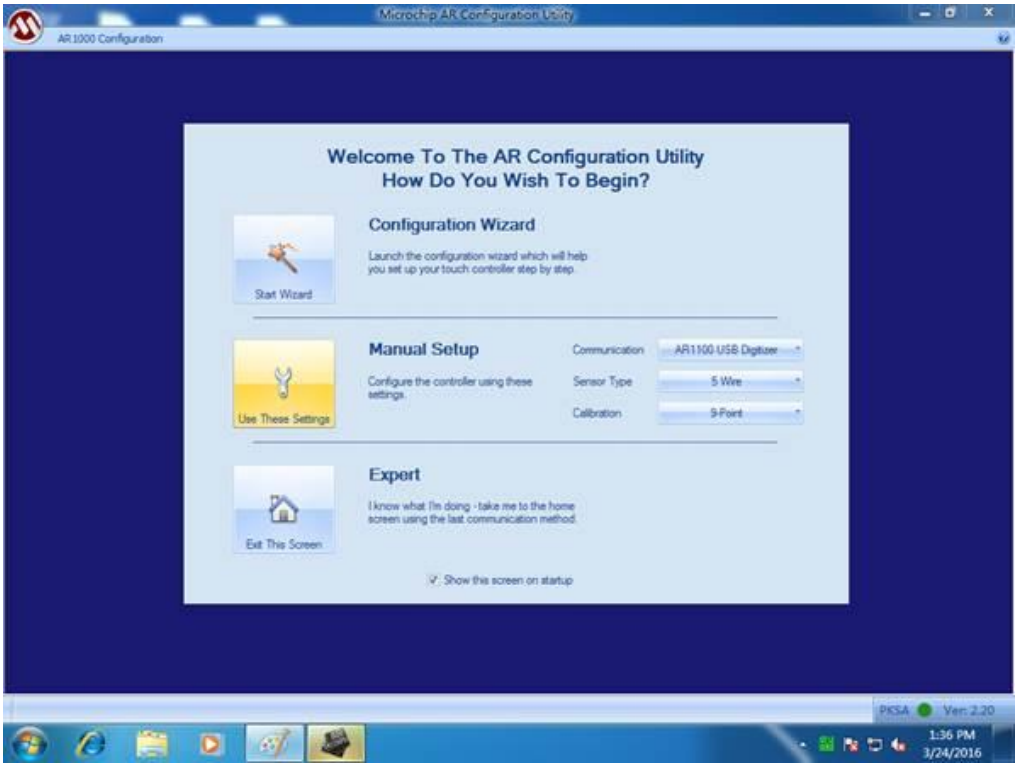


5. Please click next until it shows below dialog, after that please click “Next”, and finished.

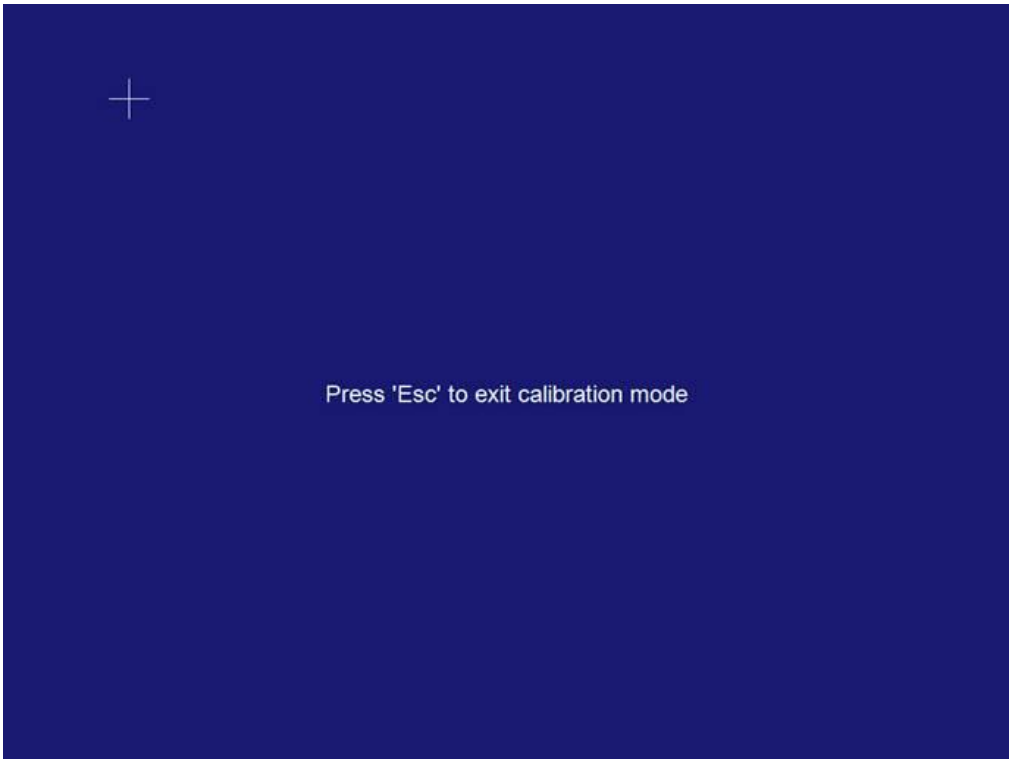


6. Please close this application.

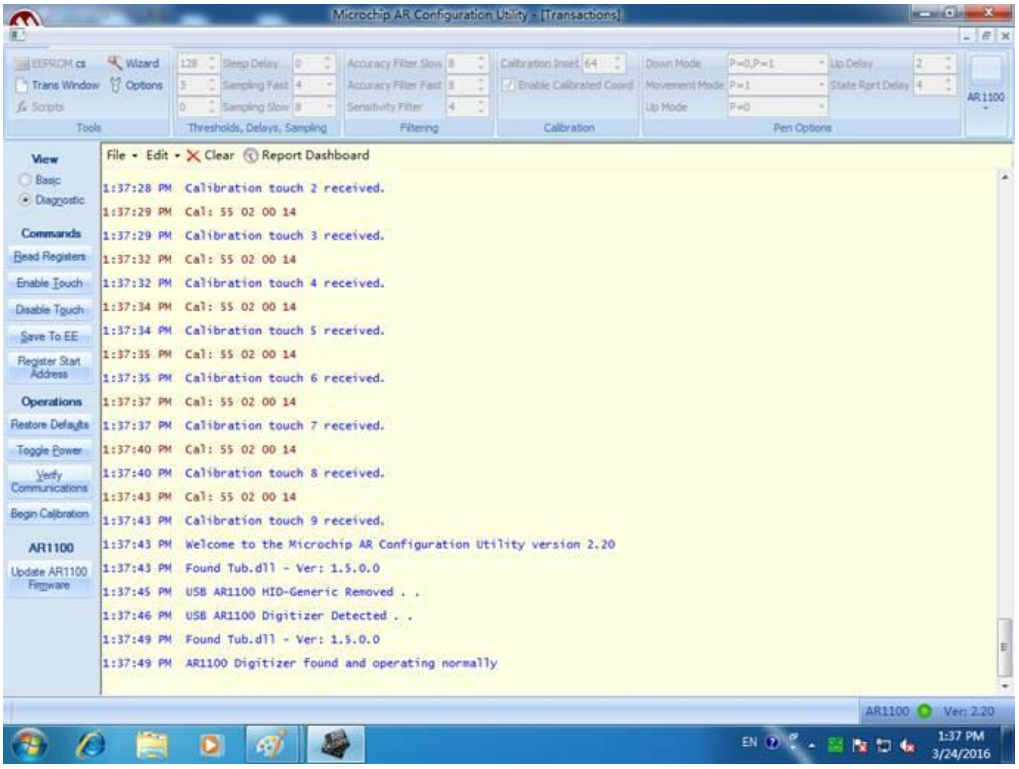
7. Please double click “Microchip AR Configuration Utility” on the desktop again, and choose “Manual Setup”.



8. It will show below picture that let you click the plus to calibrate touch panel.



9. After touch all the pluses , when it will show below picture means you can close application and complete the calibration.



Default Settings

The following table lists the settings that are restored when you push the front-panel **Default** button or send SCPI command “*RST”.

Menu or System		Default setting
Output configuration	Function	Sine
	Frequency	1.000 000 000 00 MHz
	Amplitude	1.000 Vp-p
	Offset	0 mV
	Symmetry (Ramp)	50.0%
	Duty (Pulse)	50.0%
	Output Units	Vp-p
	Output Impedance	50Ω
	Output Invert	Off
	Output Noise Add	Off
	Output High Limit	2.500 V
	Output Low Limit	-2.500 V
	VOCM	0 mV
Modulation	Modulation Waveform	10.00 kHz, Sine (except FSK,PSK)
	Modulation Waveform	10.00 kHz, Square (FSK,PSK)
	AM Depth	50.0%
	FM Deviation	1.000 000 MHz
	PM Deviation	90.0 □
	FSK Hop Frequency	1.000 000 MHz
	FSK Rate	10.000 000 0 kHz
	PWM Deviation	5.0%
	PSK Frequency	10.00 kHz
	PSK Hop Phase	90.0 □
Sweep	Sweep Start Frequency	100.000 kHz
	Sweep Stop Frequency	1.000 000 MHz
	Sweep Time	10 ms
	Sweep Hold Time	0 ms
	Sweep Return Time	1 ms
	Sweep Type	Linear
	Sweep Mode	Repeat
	Sweep Source	Internal
	Trigger Slope	Positive
	Trigger Interval	1.000 ms
	Input Threshold	0.00 V
	Step Num	1
Burst	Burst Mode	Triggered
	Burst Count	5
	Trigger Source	Internal
	Trigger Delay	0.0 ps
	Trigger Interval	1.000 000 ms
	Slope	Positive
	Input Threshold	0.00 V
	Phase	0.00°
System-related settings	Clock Reference	Internal
	Ext Clock Rate	10 MHz

AWG4162
Arbitrary Waveform Generator
Advanced Application

Contents

Preface

This document contains information about how to use the Advanced Application of the Tektronix AWG4162 Arbitrary Waveform Generator, along with basic operations and concepts.

Documentation

The following table lists related documentation available for your AWG4162. The documentation is available on the Document CD and on the Tektronix Web site (www.tektronix.com/manuals).

Item	Purpose	Location
Compliance and Safety Instructions	Compliance, safety, and basic installation information	Printed and shipped with your instrument
Advanced Application Help	Advanced Application operating information	Instrument and available as a PDF at www.tek.com/manuals
Basic Application Help	Basic Application operating information	Instrument and available as a PDF at www.tek.com/manuals
Programmer Manual	Programming syntax and command information for remotely controlling the instrument	Instrument and available as a PDF at www.tek.com/manuals
Service Manual	Instrument servicing procedures and replaceable parts list	Instrument and available as a PDF at www.tek.com/manuals
Specifications and Performance Verification Technical Reference	Instrument specifications and performance verification procedures	Instrument and available as a PDF at www.tek.com/manuals
Declassification and Security Instructions	Describes how to sanitize, secure, and declassify the instrument	Instrument and available as a PDF at www.tek.com/manuals

Getting Started

General Features

- Two working modes
  - Basic (DDS) mode
    - Two analog channels
    - 600 MHz sine waveforms
    - 2.5 GS/s, 14-bit, 16 kpts arbitrary waveforms
    - Amplitude up to 5 Vp-p into 50 Ω load
  - Advanced (Arbitrary) mode
    - Two analog channels
    - 16/32-bit digital channels (optional)
    - 1/16/32/64 Mpts per channel arbitrary waveform memory (optional)
    - Up to 750 MHz bandwidth
    - SFDR < -60 dBc
- Variable sampling rate range from 100 S/s to 2.5 GS/s, with 14-bit vertical resolution, ensures signal integrity in all aspects
- Designed for 100% user-conducted upgrades and configurations, all options activated through SW key
  - Optional and upgradable arbitrary waveform memory up to 64 Mpts for each analog channel and 32 Mbit for each digital channel for long waveforms
  - Optional 16-32 channel digital outputs. Purchasing SW option includes the shipment of digital probe accessory.
- Dual analog channels and up to 32-bit digital channels, ideal for mixed signal circuit designs
- Sync-in and Sync-out interfaces enables the synchronization of multiple units in a daisy chain, to extend the number of output channels
- Digital outputs provide up to 1.25 Gb/s data rate creates high speed digital pattern in parallel
- One marker output for each analog channel for triggering and synchronization
- Three software-configurable output paths fit all test cases
  - Direct DAC mode: 750 MHz bandwidth with differential output
  - AC coupled mode: 750 MHz bandwidth with single ended output for RF applications
  - Amplified mode: 5 Vp-p amplitude 400 MHz bandwidth with differential output
- Full functional sequence with up to 16384 user defined waveforms provides the possibility of generating complex signals with the best memory usage, in the form of loops, jumps, and conditional branches
- Channel 1 and 2 (together with the corresponding digital output channels) can work independently on different sampling clocks and sequences
- Direct communication with RFXpress® for easy waveform generation in RF applications
- Windows based platform with 10.1-in touch screen, front panel buttons,keyboard, and mouse
- Compact form factor, convenient for bench top and portability Removable hard disk guarantees the security of confidential data
- USB 3.0 and LAN interfaces for remote control

Operating Requirements

Power supply

Source voltage and frequency 100 to 240 Vrms @ 50 - 60 Hz

115 Vrms @ 400 Hz

Power consumption max 150 W

Surge current 30 A peak (25 °C) for ≤ 5 line cycles, after product has been turned off for at least 30 s

Physical characteristics

Weight (typical)

Net weight 6.5 kg (14.2 lbs)

Net weight with packaging 11.5 kg (25.2 lbs)

Dimensions

Height 233 mm (9.17 in.)

Width 439 mm (17.28 in.)

Depth 199 mm (7.82 in.)

Dimensions with packaging(typical)

Height 498 mm (19.61 in.)

Width 457 mm (17.99 in.)

Depth 574 mm (22.60 in.)

Clearance ≥50.8 mm (2.0 in.) on left and rear sides of the instrument

Temperature

Operating +5 °C to +50 °C (+41 °F to 122 °F)

Non-operating -20 °C to +60 °C (-4 °F to 140 °F)

Humidity

Operating 8% to 90% relative humidity with a maximum wet bulb temperature of 29 °C at or below +50 °C, non-condensing

Non-operating 5% to 98% relative humidity with a maximum wet bulb temperature of 40 °C at or below +60 °C, non-condensing

Altitude

Operating 3,000 m (9,843 feet)

Non-operating 12,000 m (39,370 feet)

Standard Accessories

Item	Description
Manual	COMPLIANCE AND SAFETY INSTRUCTIONS
Product CD	Document CD with Browser Including the PDF files of Specs & PV Tech Ref, user manual, programmer, service manual.
Power Cable	-
USB cable	-
Stylus	For touch panel
Front protect cover	-
Accessory Pouch	-
50Ω SMA Terminator	Male, DC-18GHz; 1 ea / channel
Certification of Calibration	-
Three year warranty	-

Recommended Accessories

Item	Description
Pin Header SMA Cable	45 inch
RMD5000	- Rack mount kit
	- Instruction sheet (English)
Manual	Service (English)
	Specs & PV Tech Ref
	Programmer manual
AWG4HDDE	- **Hard disk drive**
SMA terminator	50 Ω

AWG4SYNC	**Sync cable**; Used for multiple instruments synchronization
RFX100	RFXpress software
AWG4DIG16LVDS	**16-bit digital output cable**; Used for LVDS
AWG4DIG16TTL	**16-bit digital output adaptor**; LVDS to TTL/CMOS
AWG4DIGSCKT	**Digital output connector**; AWG4k Digital Channel Connector on DUT (Amphenol, U65-B12-40E0C)
TEK-USB-488	GPIB to USB adaptor
HCTEK54	Hard transit case

Powering the Instrument On and Off

Power On

1. Insert the AC power cord into the power receptacle on the rear panel.
2. Use the front-panel power button to power on the instrument.
3. Wait until the system shows windows desktop.
4. You have two selections to open the applications:

You can press the “Basic” or “Advanced” Buttons on front panel to launch one application. You can also click the shortcut icons on desktop to launch any one of them.

NOTE:

Only one application can be launched at same time. If you want to launch the other application, please close the first one at first.

Power Off

1. Please close the application that you are using at first.
2. Use the front-panel power button to power off the instrument.

Protect Your Instrument from Misuse

Check Input and Output connectors

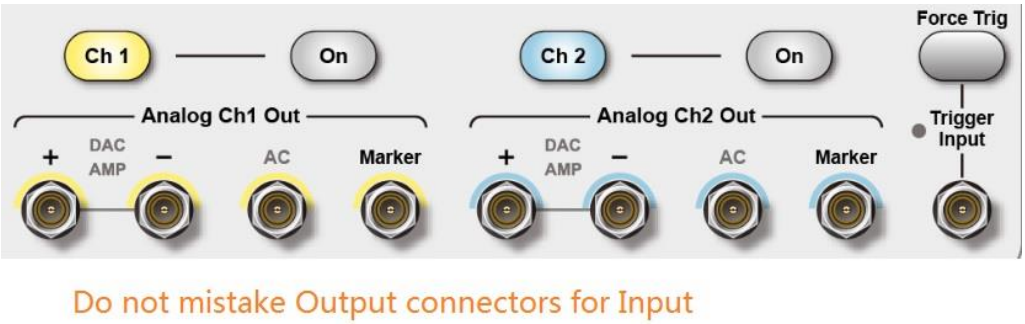


Figure 1 Output connectors

The instrument has both input and output connectors on the front panel. When connecting a cable, be sure to distinguish the input connectors from the output connectors.

No Hot Plugging

Figure 2 Digital pods

The instrument has two **Digital Pods** on the front panel and **Sync In, Sync out** connector on the rear panel. Please be noted: No Hot Plugging for these connectors when connecting a cable on these connectors.

Update Your Instrument Firmware

Install Advanced APP

If your instrument has already installed another version of Advanced APP, you must uninstall it firstly. You can find uninstall details in next section “Uninstall Advanced APP”.

1. Download Advanced APP setup package from Tektronix website and decompress it to instrument’s local disk.
2. Double click setup.exe, start to install. You will see the welcome page, press **Next**.

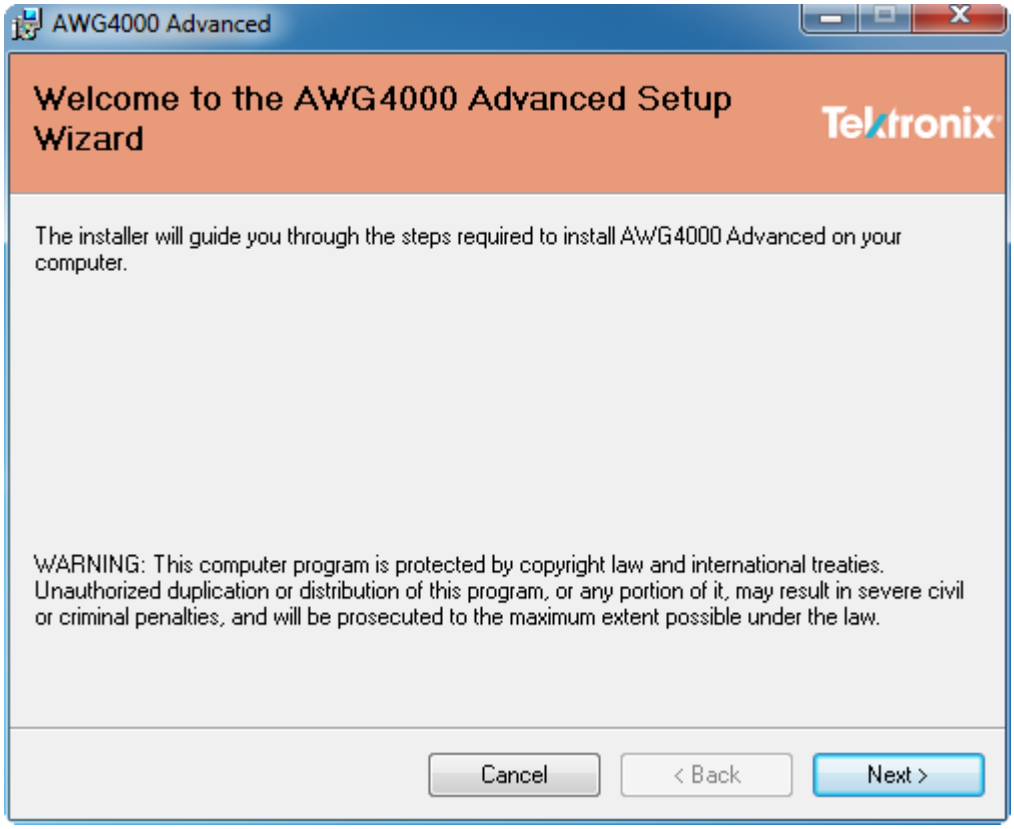


Figure 3 Welcome page

3. Select the path that you want to install your application by pressing **Browse** button, choose the person who have the right to use the application, then click **Next**.

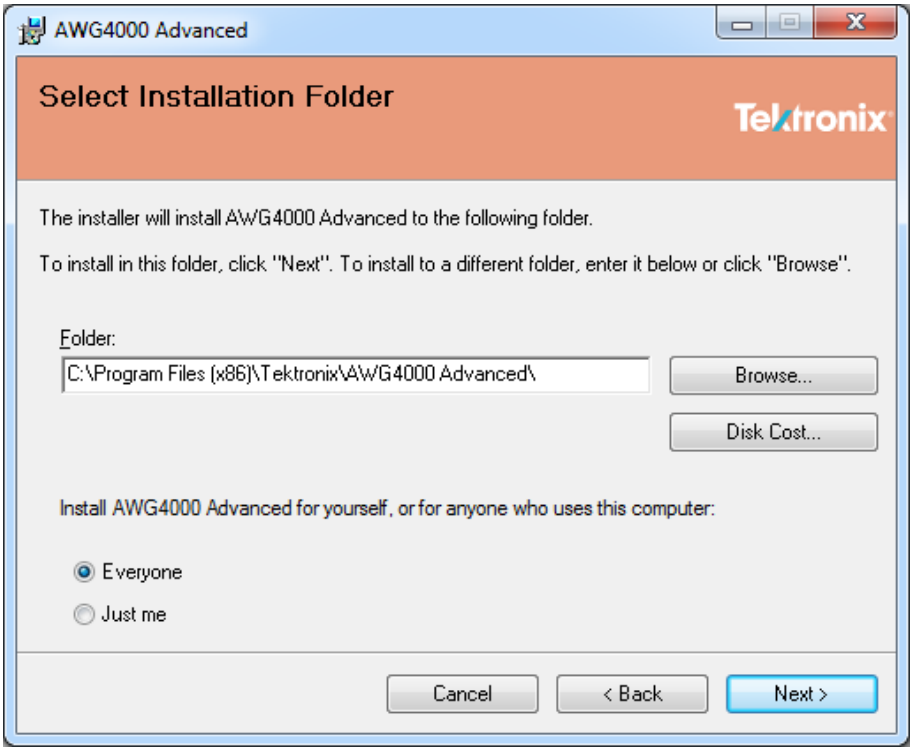


Figure 4 Select installation folder

4. Press **Next** to start installation.

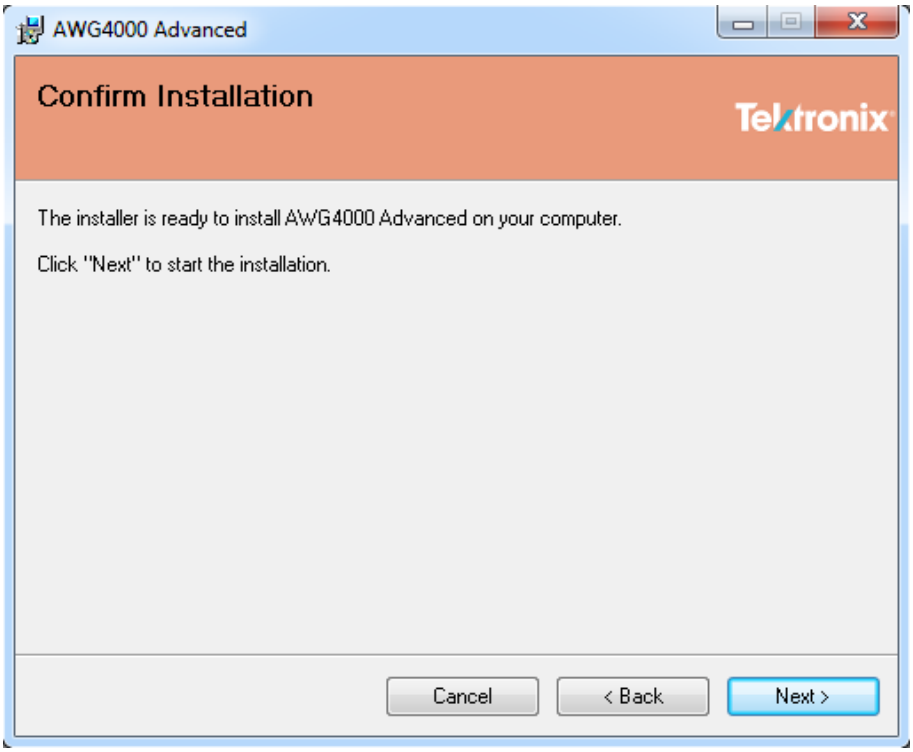


Figure 5 Confirm installation

5. Installation begins, instrument shows installation progress, please wait.

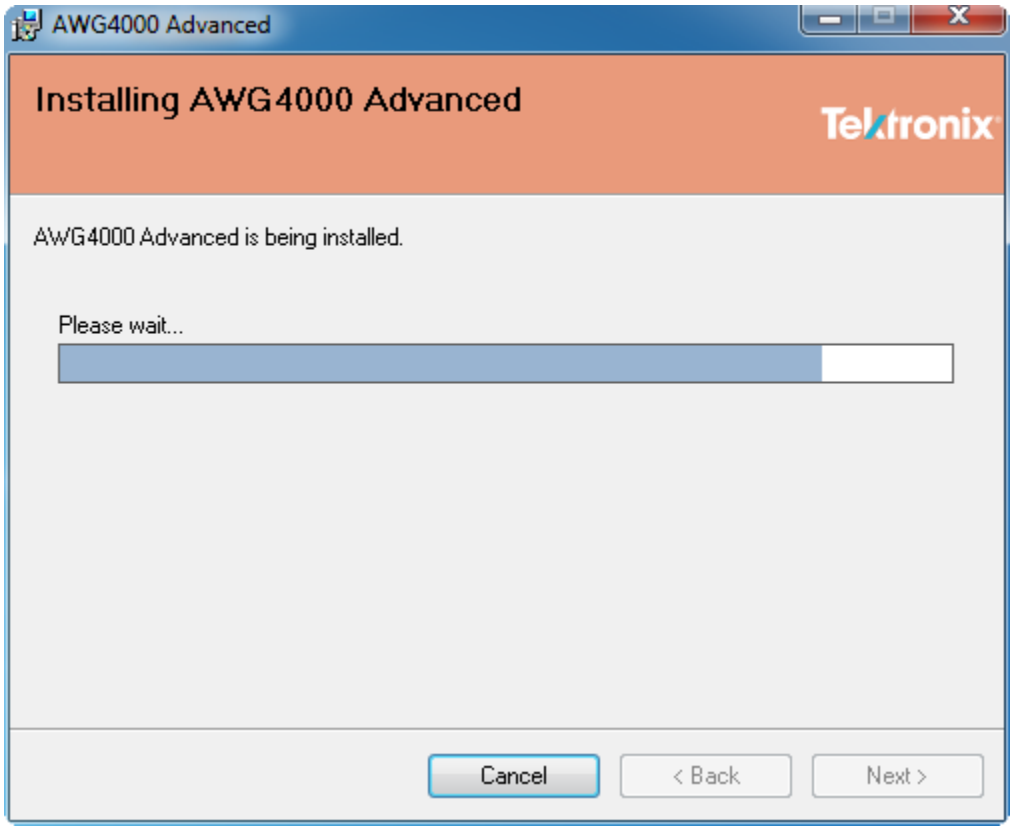


Figure 6 Installation in progress

6. Wait until instrument shows “Installation Complete”, then press **Close** to finish the installation.

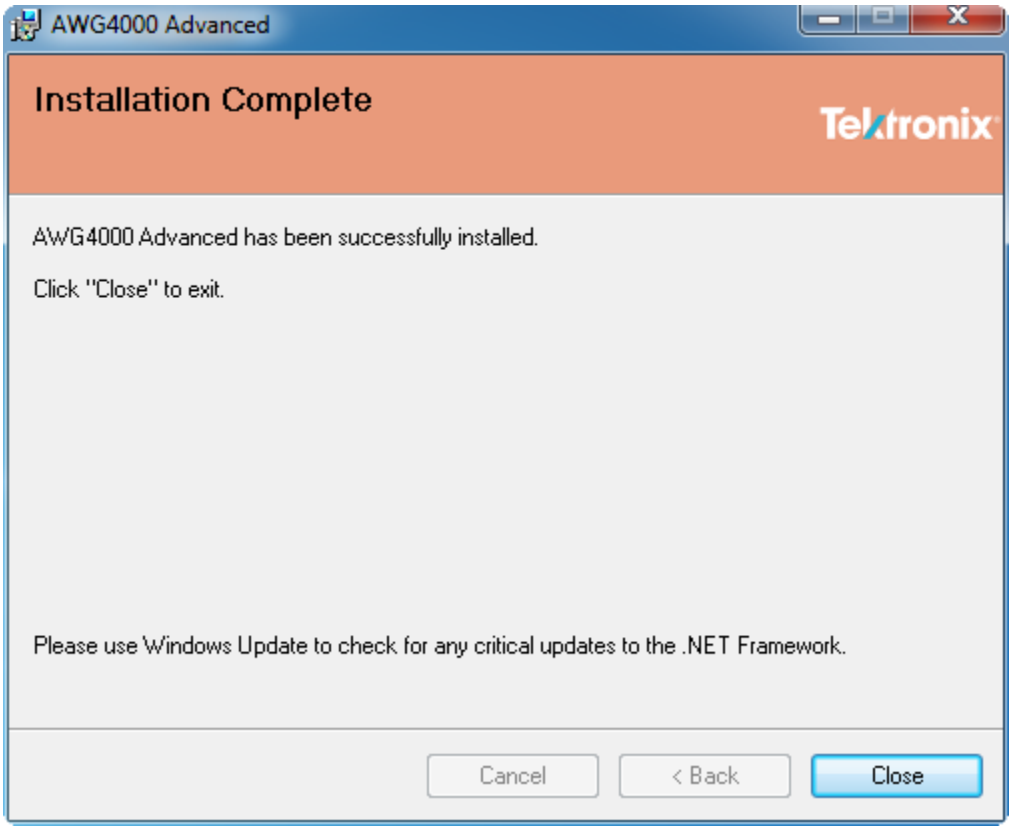


Figure 7 Installation complete

Now you can find installed Advanced APP icon in the Start Menu and on the desktop.

Uninstall Advanced APP

You can use Advanced APP setup package to uninstall the firmware in following steps:

1. Download Advanced APP setup package and decompress to instrument's local disk.
2. Double click setup.exe. The welcome dialog notices you to select whether repair or remove installed version Advanced. Select "Remove AWG4000 Advanced" and press **Finish** to start uninstallation.

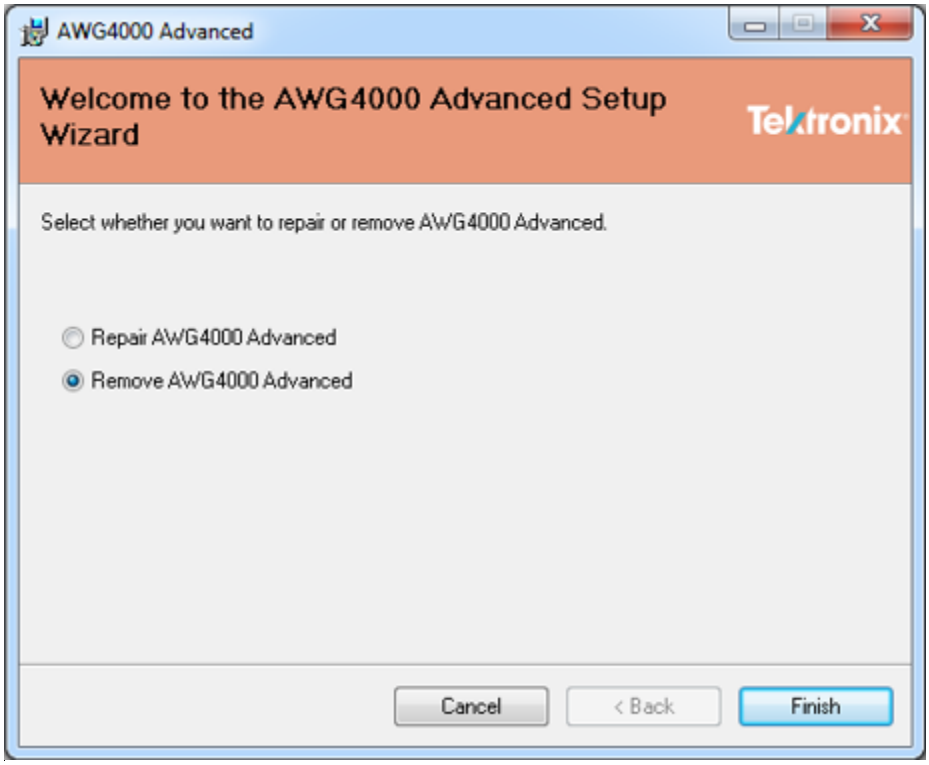


Figure 8 Uninstalling firmware

3. Uninstallation begins, instrument shows installation progress, please wait.

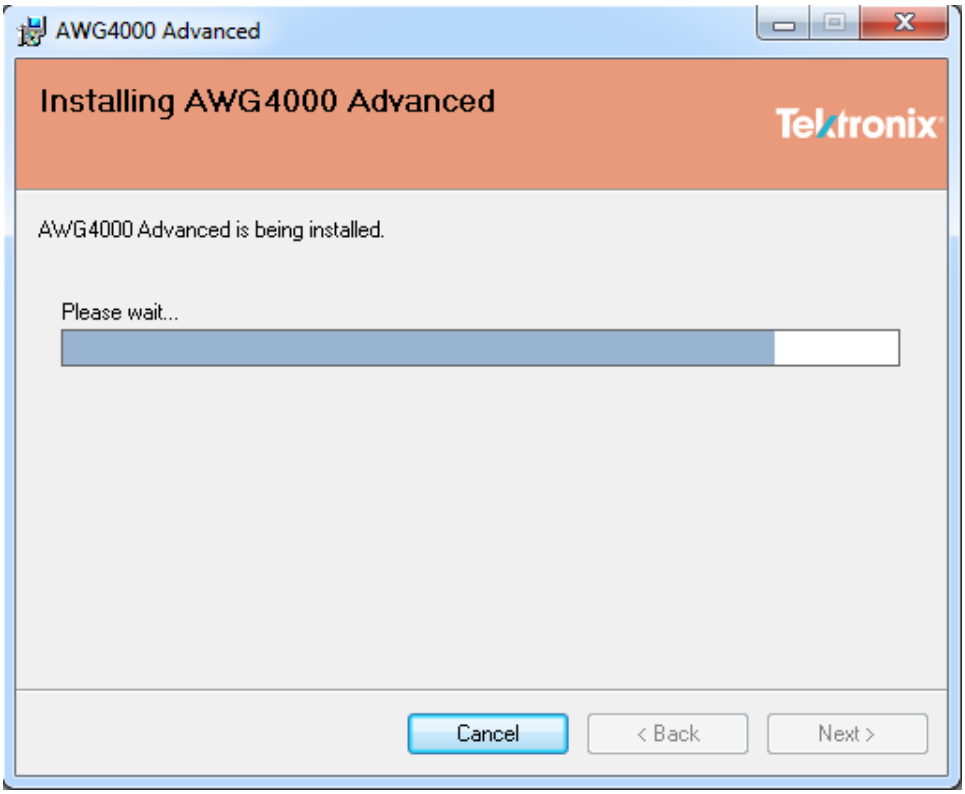


Figure 9 Uninstallation in progress

4. Wait until instrument shows “Uninstallation Complete”, then press **Close** to finish the uninstallation.

Besides using setup package, you can also use Windows Control Panel tool to do Advanced APP uninstallation by followed steps:

1. Enter uninstall page through path: Start → Control Panel → Uninstall a program

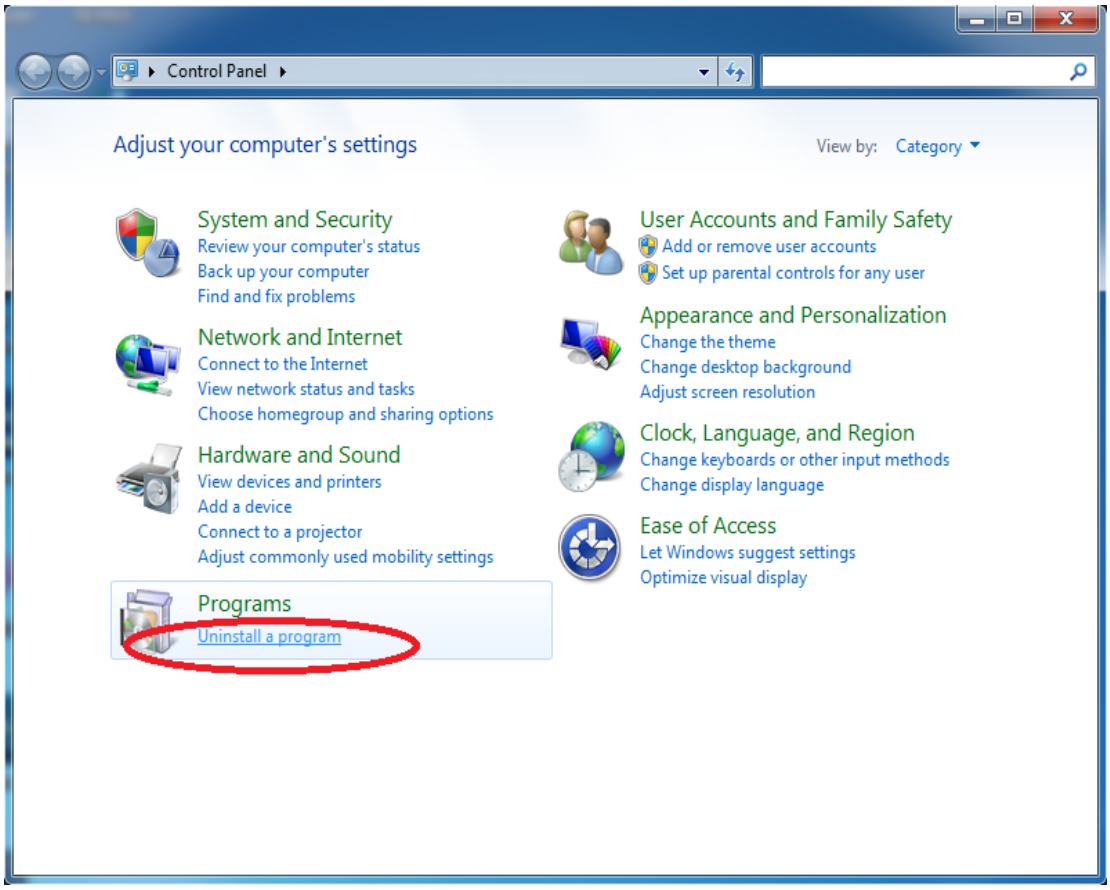


Figure 10 Uninstall a program in control panel

2. In Uninstall or change a program page, please select “AWG4000 Advanced” program, and uninstall it.
3. Wait until uninstall finished.

Remote Control

You can connect your instrument to a network for printing, file sharing, Internet access, among other functions. Consult with your network administrator and use the standard Windows utilities to configure the instrument for your network. For LAN configuration, use the LAN Configuration dialog box from control panel.

In addition, if the instrument is under remote mode, it allows you to control the instrument remotely by using SCPI commands. Please refer to the AWG4162 SCPI commands manuals for a complete description about all available channels.
The instrument can be controlled using VGPIB, VXI-11 (LAN) or USBTMC protocols. You can follow the next steps to enable the servers and start to communicate with your AWG4162 instrument:

1. Connect your LAN cable or USB to the instrument.
2. Press the **Remote/Local Mode** button in the Utility Tab, as shown in the following figure.

Figure 11 Utility tab

3. The VGPIB/LAN/USB-TMC configuration panel will open, as shown in the following figure.

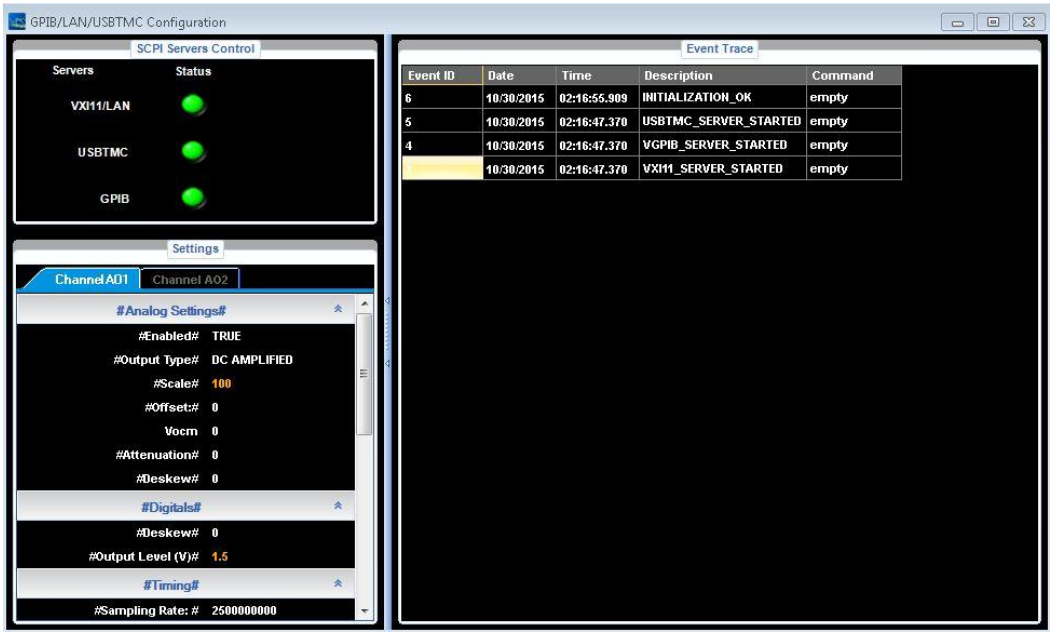


Figure 12 VGPIB/LAN/USB-TMC configuration panel

4. On the Client-PC launch the Tek OpenChoice Instrument Manager window , as shown in the following figure.

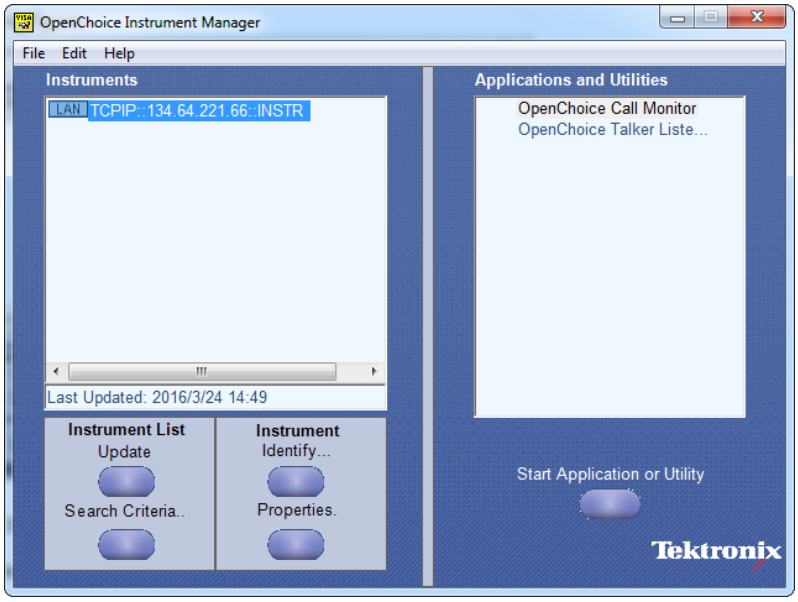


Figure 13 OpenChoice Instrument Manager

5. Press *Search Criteria…* button and enable VXI/LAB/USB/GPIB, as shown in the following figure.

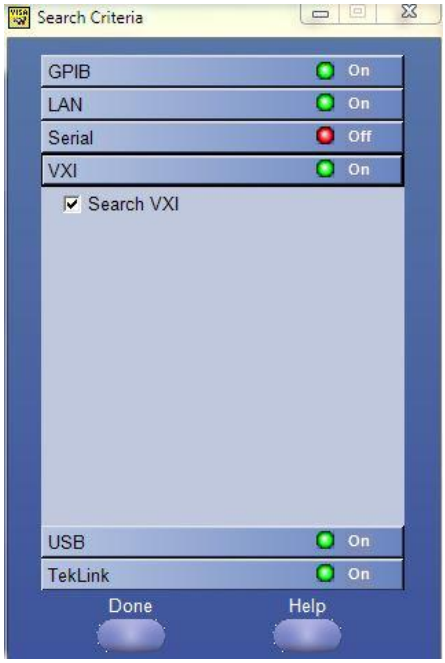


Figure 14 Search Criteria

7. Check the *Instruments* list to check the if the AWG4162 has been correctly detected.
8. Press the *Start Application* button to open OpenChoice Talker Listener and send a *IDN? Command.

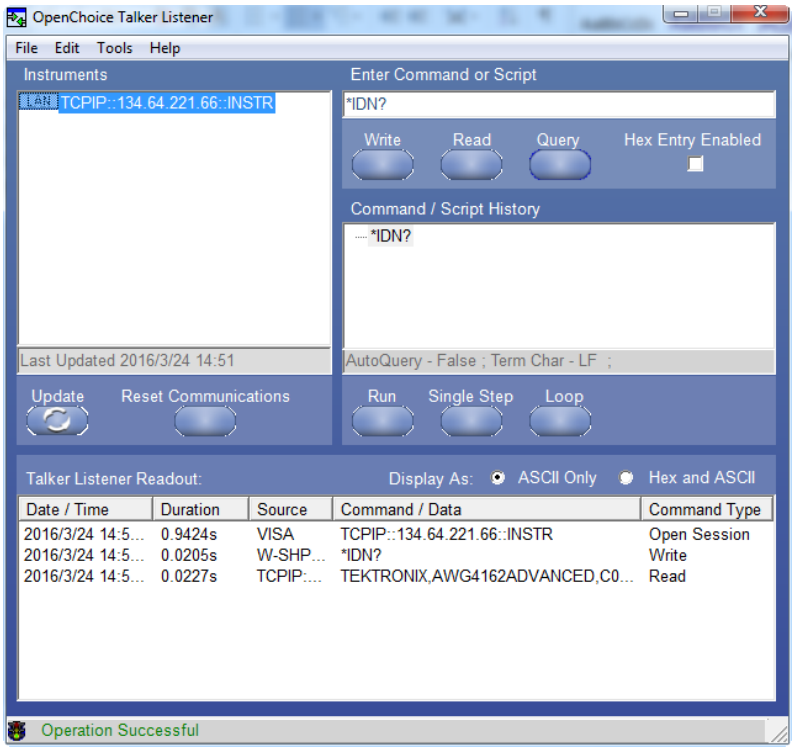


Figure 15 OpenChoice Talker Listener

9. The instrument should respond to the string:
   TEKTRONIX,AWG4162ADVANCED,CO000003,SCPI:99.0,FV:ARB 5.3
   where CO000003 is the serial number and ARB 5.3 is the firmware version.

11. You can check also the result in the Remote/Local form, as shown in the following figure.

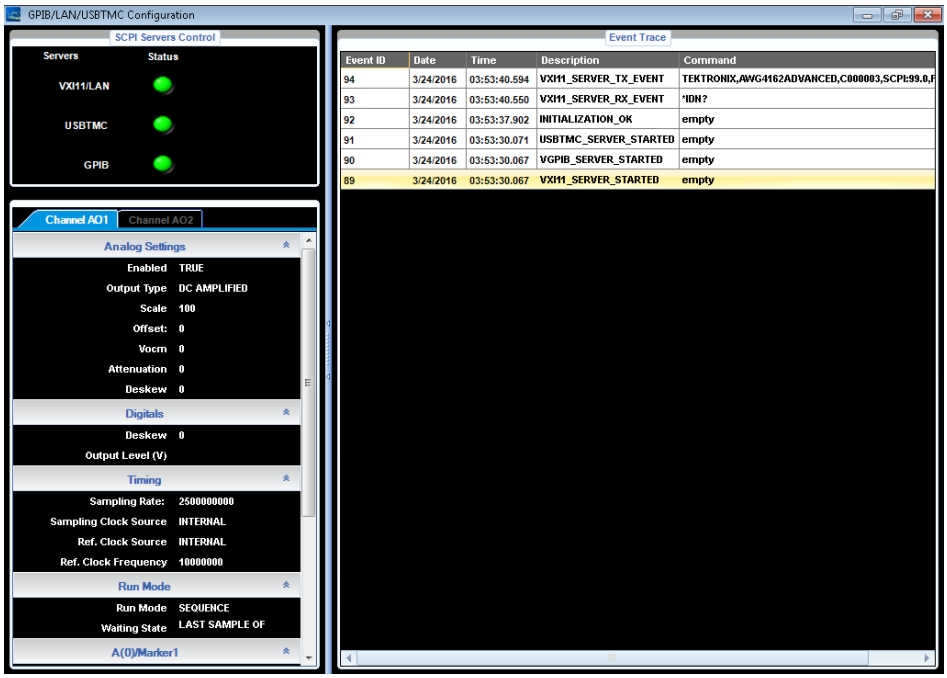


Figure 16 Result in the Remote/Local form

13. Using the OpenChoice Taker Listener window you can also load a script (SampleImport.txt) , as shown in the following figure.

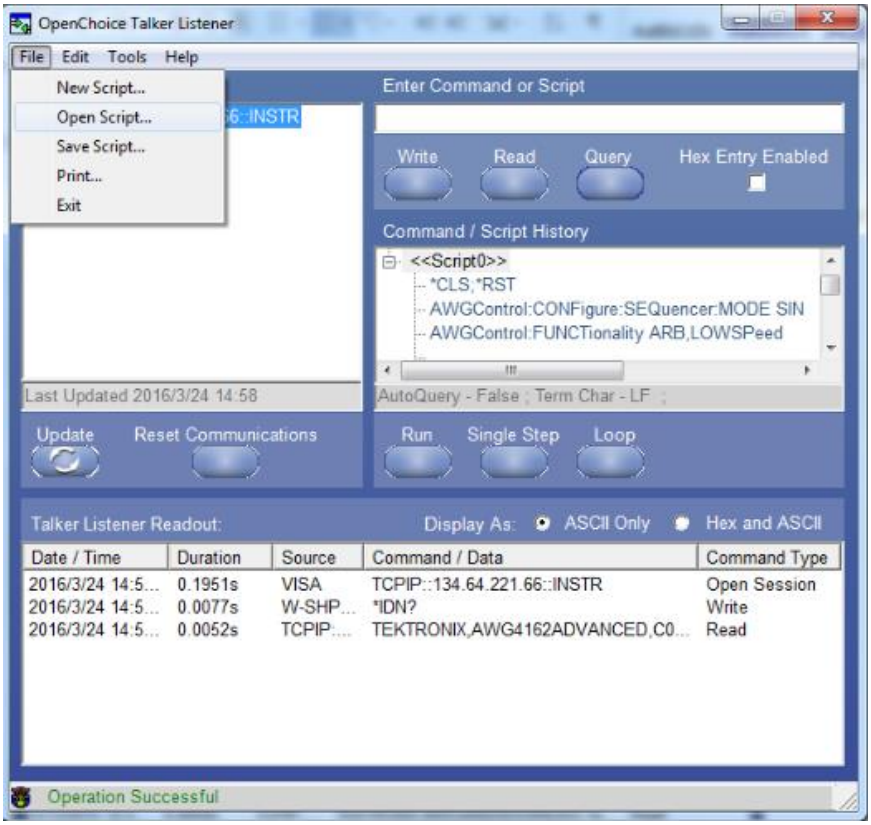


Figure 17 Load a script in OpenChoice Taker Listener

14. Press the Run button to run the script and send the SCPI commands to the instrument, as shown in the following figure.

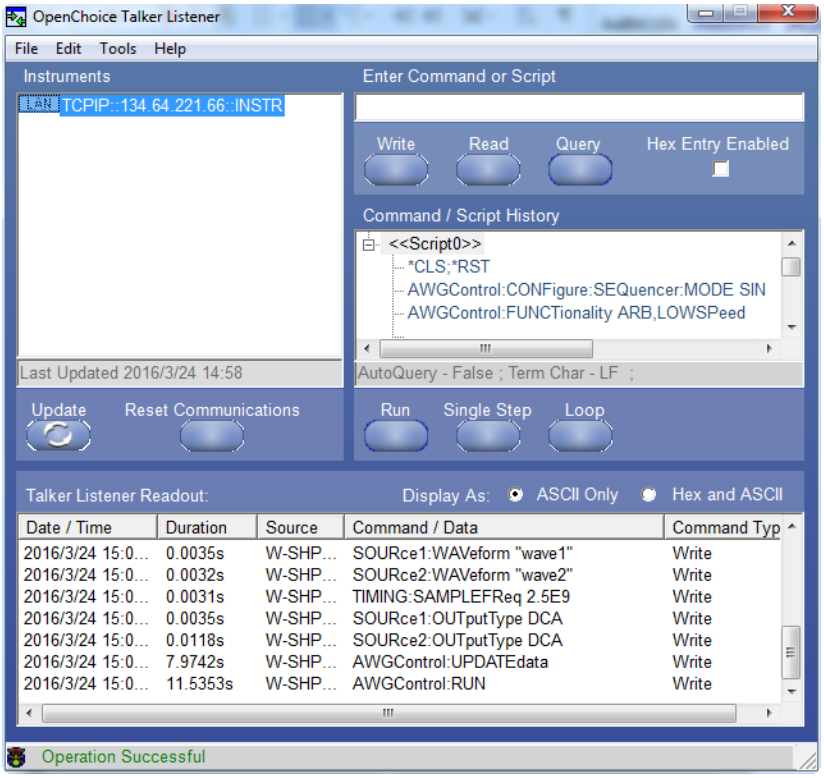


Figure 18 Run a script in OpenChoice Taker Listener

16. You can also check the SCPI commands in the Remote/Local form, as shown in the following figure.

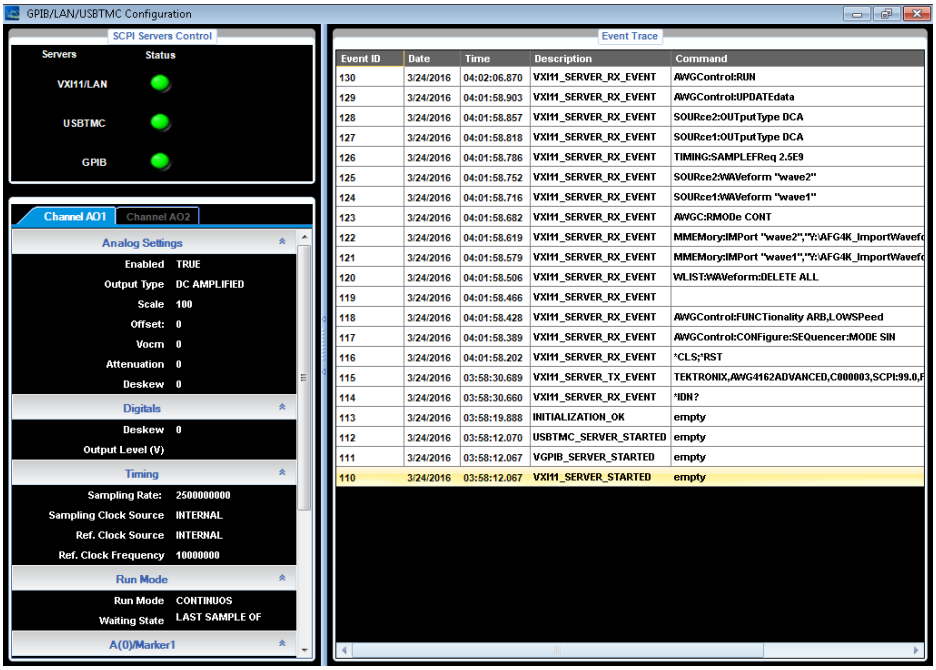


Figure 19 Check the SCPI commands in the Remote/Local form

17. The instruments settings will be updated in the Channel AO1/ Channel AO2 tabs.

Overheat Protection

The instrument internal temperature is monitored in AWG4162. A warning message will appear if the internal temperature reaches a threshold level, and signal output will automatically turn off. If the warning message appears, check for following conditions:

- The ambient temperature requirement is being met.
- The required cooling clearance is being met.
- The instrument fan is working properly.

Getting Acquainted with Your Instrument

Front Panel Overview

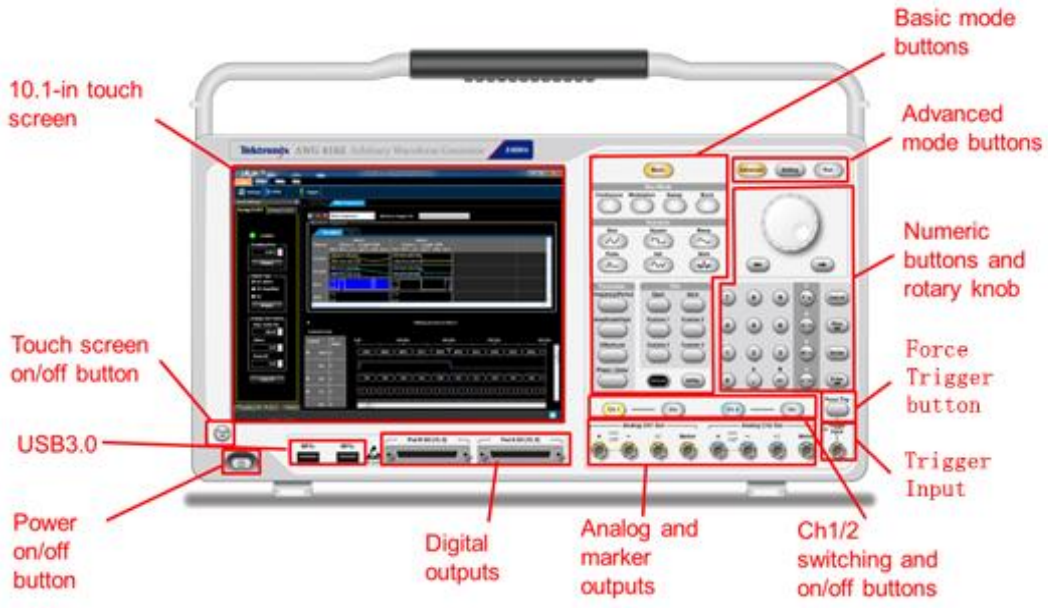


Figure 20 Front panel overview

Rear Panel

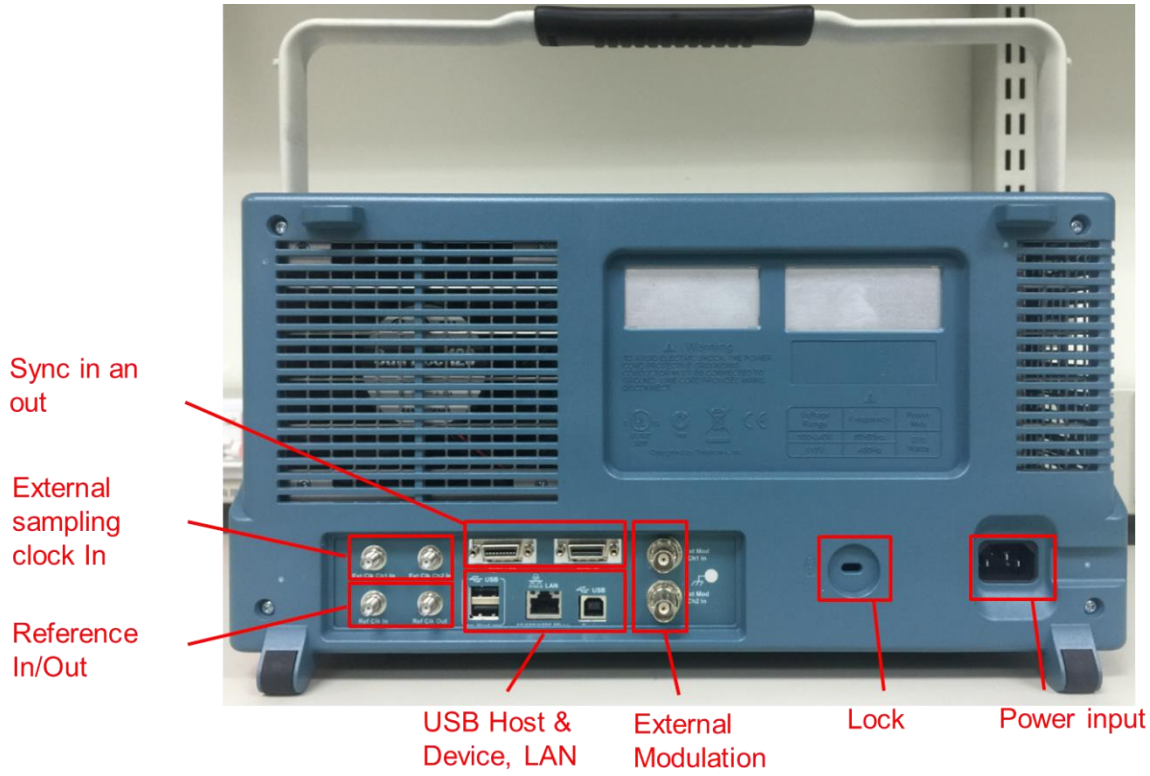


Figure 21 Rear panel overview

Advanced UI Basics

This chapter gives explanation of some basic concepts for the advanced UI functions:

Sequence

Sequencing is often used for two purposes

a) To save memory space by repeating a single waveform stored in memory, instead of storing many copies in memory

b) To generate a complex series of waveforms by means of loop, wait, jump when certain triggering events happen.

A sequence contains a series of entries, in which a user can put a mixed waveform with 2 analog and 32 digital waveforms in parallel. Users can program execution rules of each entry to define how they are executed in the sequence, including

a) Wait Event – Entry waits to start until the event happens

b) Repetition times

b) Go to – the next entry after the entry repetition expires

c) Jump Event – the next entry to jump to when the event happens, even repetition doesn't expire

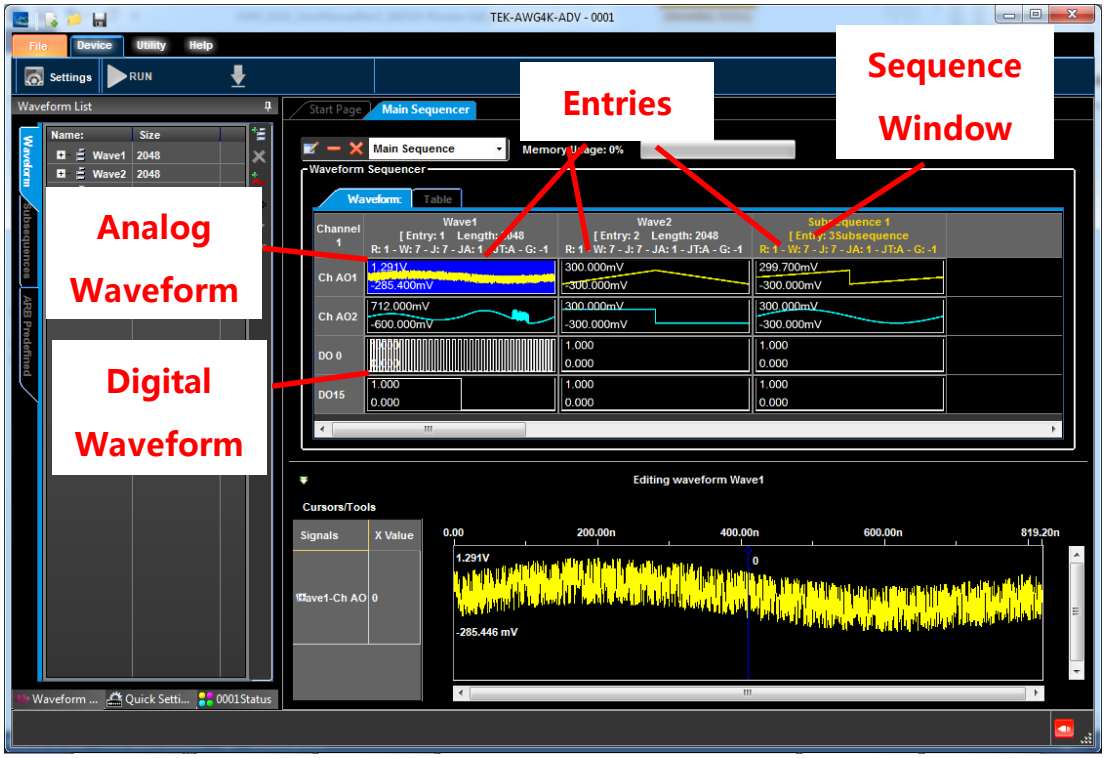


Figure 22 Sequence

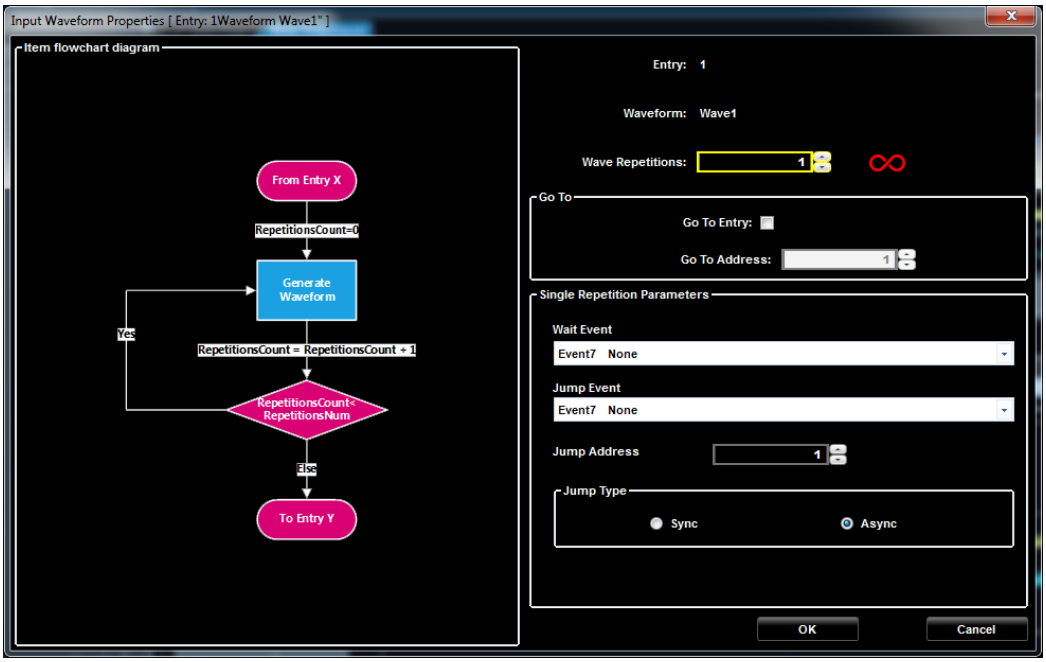


Figure 23 Editing sequence

SubSequence

A subsequence is a combination of entries pre-defined by a user, which can be called and put in the main sequence just as an entry.

Entry

In Sequence mode, entry is used to set repetition and event parameters of every waveform element, every waveform element has one entry.

Segment

A Segment contains one or more Components, all of the same length, combined by means of the elementary Add, Subtract, Multiply operations.
If one Segment contains more Components, the following formula will be applied:
Segment = (Component1 (Add/Sub/Multiply)Component2) Add Component 3 Add Component 4 Add Component N

Component

A Component is the basic element for the construction of a Segment. Each Component may be represented by a standard waveform (DC Level, Sine, Cosine, Exponential, Triangle, Rectangle, Ramp, Pulse, Sync, Sawtooth, Sweep), by a Formula, or its component samples can be loaded from a text file.

Workspace

The AWG4162 software can manage the instrument as a powerful arbitrary waveform generator. The software is project based. There are two types of projects: Single Sequencer and Multi Sequencer. The two kinds of workspace differ in the way how the analog/digital resources are managed:

Single Sequencer

In this workspace all analog/digital resources are managed synchronously, even when multiple boards are connected together in a daisy chain configuration.
This mode is called “Single Sequencer” because there is only one sequencer which manages multiple output channels.

Multi Sequencer

In this workspace each analog output can be configured independently from each other. Each analog output behaves as an independent device with a single output channel.
This mode is called “Multi Sequencer” because each analog channel has its own sequencer. However, the channels can share trigger and synchronization signals, even if each channel is configured independently.

Digital Data

The AWG4162 has up to 32 digital lines that can be configured as a powerful pattern generator. The available digital lines number depends on the Digital Option loaded by you.

The digital bus can be configured in two modes:
Low Speed Mode: 16 bits are available on PodA and PodB. The digital output sampling rate is ¼ of the analog sampling rate, so the length of the digital samples must be ¼ of the analog waveform length.
High Speed Mode: 8 bits are available on PodA and PodB. The digital output sampling rate is ½ of the analog sampling rate, so the length of the digital samples must be ½ of the analog waveform length.

If the digital option is not available, the user can modify in the Waveform Editor the digital lines Pod A DO(0) a nd Pod B DO (0). Those lines represent Marker 1 and Marker 2 output signals.

The marker outputs sampling rate depends on the High Speed/Low Speed mode selection.
The marker type can be analog or digital:

- When the analog marker is selected, the signal comes from the front panel SMA and its maximum update rate is 156.25 MHz.
- When the digital marker is selected, the marker is connected to the Digital Pod A/Pod B and it is available through the digital connector PIN.

Note that if there is no digital option, the marker is in high speed mode. In addition, with option DO16, Marker 2 speed will follow the speed set for Marker 1.

Mixed (analog/digital) waveforms

Since some constrains must be applied for only analog and digital waveforms in the same sequencer entry, it is suggested to work with mixed waveforms if not specified otherwise.

Advanced UI Introduction

Start Page

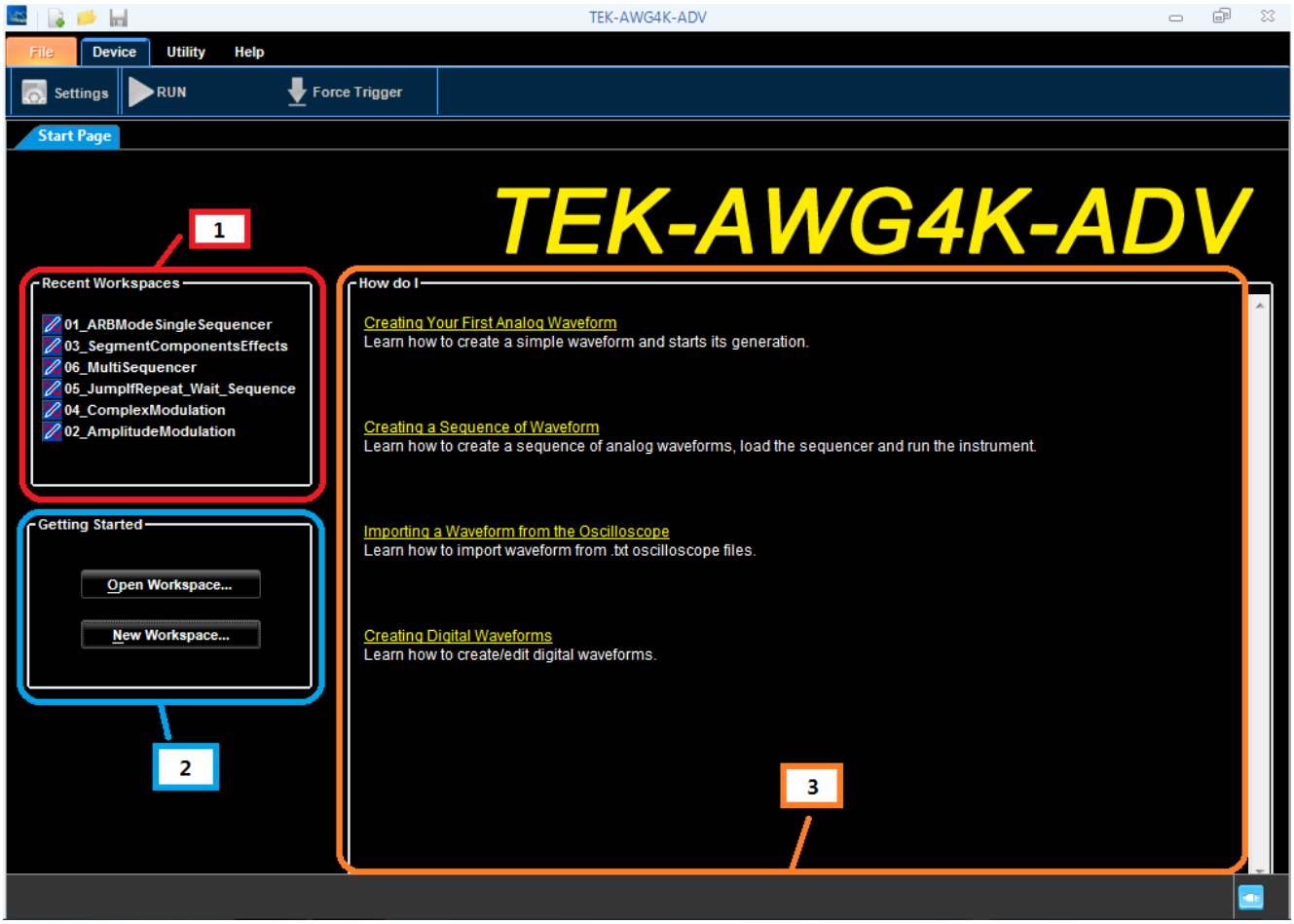


Figure 24 Start page

When you start the Advanced APP, it enters Start Page first. The Start Page includes 3 areas:

1. Recent Workspaces:
   List name of recent open workspaces.
2. Getting Started:
   Press “Open Workspace” button can open one certain existed workspace.
   Press “New Workspace” button can create a new workspace.
3. How do I
   In this area, it provides several simple examples, which help you to learn how to use Advanced APP.

Self-Diagnostic

NOTE:

Before performing this procedure, allow a 30 minute warm-up period after powering on the instrument, because the diagnostic is not valid if the instrument does not reach to a valid temperature. You can go to Basic application –> System -> Tools -> Warm Up Timer to do the warm-up.

You can run the internal diagnostics using the Utility menu item.

Figure 25 Utility menu

Take the following procedures to run the internal diagnostics:

1. Press The Diagnostic button
2. The Diagnostic dialog box appears, as shown in the following figure.

Figure 26 Diagnostic dialog box

3. Click Start button to start the diagnostics.

The diagnostics takes for about 10 minutes to complete. If diagnostic failure occurs, run self-calibration. If failures still exist, please contact your local Tektronix service person.

Self-Calibration

Self-calibration uses internal calibration routines to check electrical characteristics such as DC Direct and DC AMP analog level/offset and then adjust the internal calibration constants if necessary.

NOTE:

Before performing this procedure, allow a 30 minute warm-up period after powering on the instrument, because the calibration is not valid if the instrument does not reach to a valid temperature. You can go to Basic application –> System -> Tools -> Warm Up Timer to do the warm-up.

Figure 27 Utility menu

Take the following procedures to do the self-calibration:

1. Press The Self Calibration button 
2. The Self-calibration dialog box appears, as shown in the following figure.

Figure 28 self-calibration dialog box

3. Check the **Verify** box if you want to also verify the calibration parameters.
4. Press **Start**.
   The calibration takes for about 10 minutes to complete. The message “Calibration process completed:SUCCESS” will pop up when the calibration finishes. If failure occurs, please contact your local Tektronix service person.

Home Page

When you open an existing project or create a new one, the following figure is displayed.

Figure 29 Project interface

The numbered interface sections are explained as follows.

1. **Ribbon Menu Bar and Toolbar-** provides menu access to device settings, utilities, and online help.

- **File:**

	**New Workspace - Use this button to create a new workspace.**
	**Open Workspace** - Use this button to open an existing workspace.
	**Save Workspace** - Use this button to save newly-created or edited workspaces.

- **Device:**

	**Settings - Use this button to access more detailed AWG4162 option settings.**
	**RUN/STOP** - This button first loads setting parameters and the waveforms into the instrument, and then it starts/stops the waveform generation for all enabled channels selected from the Channels Selection button.
	**Force Trigger** – Clicking this button makes the instrument generate an internal trigger signal and forces an event on the selected channels/pods.

- **Utility:**

	**Remote/Local Mode - Use this button to open the Remote Mode window** to **let the user control the instrument by remote connection.**
	**Self Calibration –** Use this button to perform a self-calibration of the instrument.
	**Load Factory** – Use this button to reload the factory calibration parameters.
	**Diagnostic** – Use this button to perform the instrument internal diagnostic.

- **Help:**

	**License – Use this button to manage the option (see Option Installing part).**
	**Online Help –** Use this button to open the online help.
	**About** - Use this button to open the "About" window and retrieve information such as the software, dll and firmware version.

2. **Sequence Area** - This section mainly provides information on the output sequence.
3. **Waveform List / Quick Settings / Status Area** – This section gives access to the Waveform List, Quick Settings tools, and the instrument Status.
   - **WAVEFORM LIST → Waveforms TAB.** This tab contains the list of the arbitrary waveforms that the user added to the project.

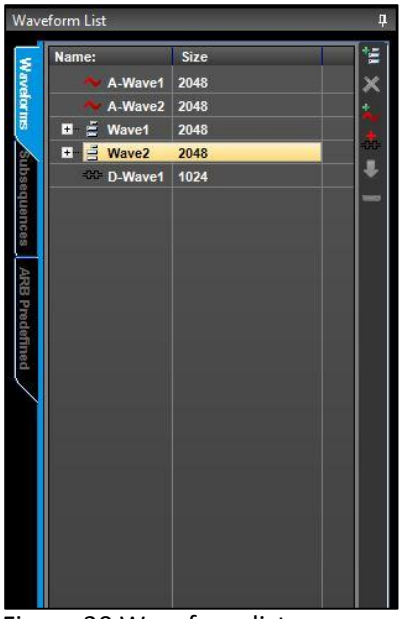


Figure 30 Waveform list

- The Waveforms TAB has a toolbar on the aside allowing you to Add, Delete or Copy the waveforms.

Icon	Action	Arbitrary Mode
	New Mixed Waveform	Use this button to add a Mixed waveform to the list. The waveform editor opens and there you can define at the same time the output waveforms for all the analog and digital available resources.
	Edit Waveform	Use this button to edit an existed waveform.
	New Analog Waveform	Use this button to add and edit an analog waveform to the list.
	New Digital Waveform	Use this button to add and edit the stimulus for the digital outputs. **Note**: the digital waveform length **must be ½** or ¼ of the analog waveform length in the same sequencer entry.
	Copy To Predefined	Use this button to copy the selected waveform to the Predefined list.
	Delete Waveform	Use this button to delete an existing waveform.

QUICK TIPS:

Double click on an existing waveform to open the **Editing Waveform Window**.

- **WAVEFORM LIST → Subsequences TAB**
  It is possible to create a subset of waveforms identifying a Subsequence that can be placed into a Sequencer entry by dragging & dropping.
  This TAB contains the list of the available subsequences and the toolbar on the right allows you to Add/Edit them.

Figure 31 Sequence list

Icon	Action	
	**New Subsequence**	Use this button to create a Subsequence: the new subsequence opens in the Sequence Area and you can add waveforms to it simply by drag & drop them from the Waveform TAB. Click on the Main Sequence button to exit from Subsequence edit mode; the created subsequence will appear on the Subsequences list.
	**Edit Subsequence**	Use this button to edit an existing subsequence. Select the Sequence entry from the list and press the button to edit it in the Sequence Area.

- **WAVEFORM LIST → Predefined TAB**
  This TAB contains the predefined waveforms: predefined waveforms cannot be edited, but once they have been copied to the user waveform list they can be

opened and edited.
Predefined waveforms are available on the user workspace with the same configuration.

If you create a Mixed Predefined waveform in Single sequencer DO16 digital bits, it will be available on all the workspace with the same features, but it will not be available in Multi Sequencer workspace or Single Sequencer DO32 digital bits ones.
Predefined Analog waveforms will be available in all the user workspace, while predefined Digital waveforms will be available in the user workspace that will match the same digital configuration (DO16 or DO32).
You can add a Predefined waveform to the Sequence Area by dragging & dropping.

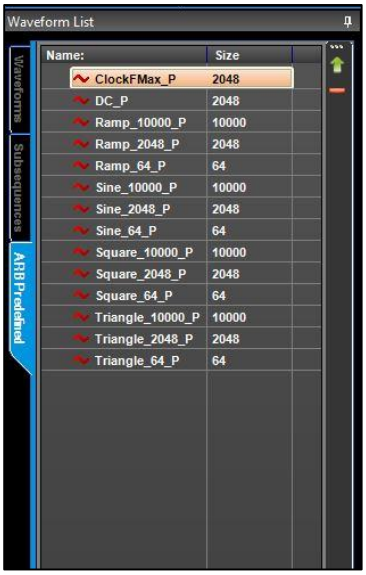


Figure 32 Predefined waveform list

Icon	Action	
	**Copy to Project Waveform**	Use this button to copy a Predefined waveform to the user waveform list: the copied waveform can be opened and edited.
	**Delete Predefined Waveform**	Remove and delete a predefined waveform from the list.

- **QUICK SETTINGS**
  This TAB gives the user quick access to the most useful instrument settings like Sampling Rate, Output type and Analog Out Control.

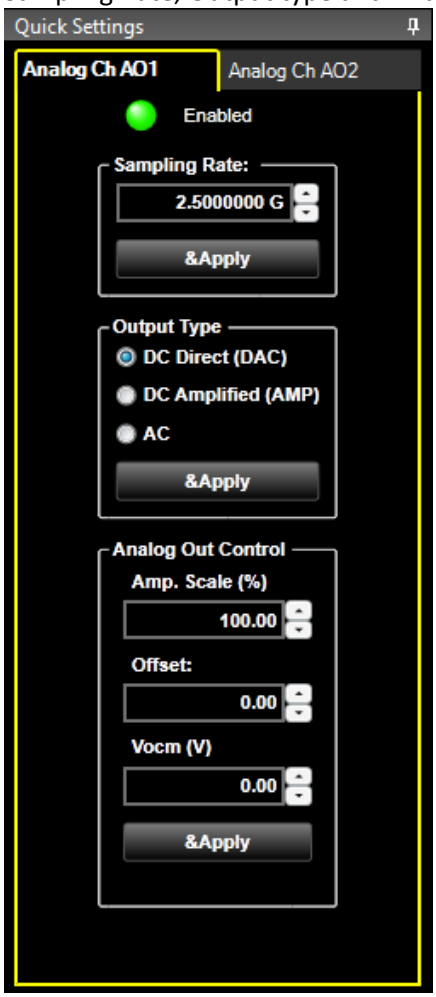


Figure 33 Quick settings

- **DEVICE STATUS**
  This area of the screen displays channel functionality information as follows.
  - **Type** - Shows the channel functionality

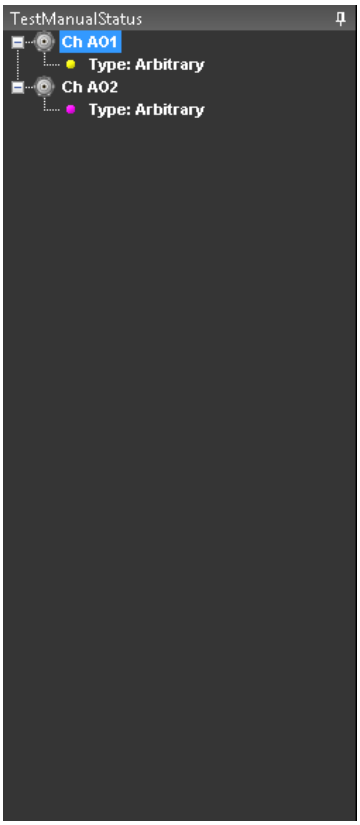


Figure 34 Device status

4. **Waveform Display Area** -This area displays the waveform that you selected in the Waveform Table List or in the Sequence Area.

NOTE:

All of the panels in the interface are dockable. You can move individual panels by clicking the panel's top side, holding, and dragging with your mouse.

Settings

Use **Settings** to control the channel settings of the instrument. Access Settings by double clicking on the  button in the main toolbar.

The **Settings** screen is divided into the following tabs:

1. **The Analog Ch AO1/AO2 Tab**
2. **Digital Channels Tab**

3. **The Timing Tab**
4. **The Run Mode Tab**
5. **The Events Tab**
6. **The Dynamic Jump Tab**

Setting panel is displayed differently by projects:

- In Single Sequencer workspace, one Settings panel is available to manage all the channels capabilities.
- In Multi Sequencer workspace, one Settings panel per analog channel is available to manage the single channel capabilities: those control panels are called the *Master Settings* and the *Slave Settings* panels.

Settings - The Run Mode Tab

The **Run Mode** tab is shown by default when the **Settings** screen is opened.
Use the Run Mode tab to define the generation mode for the edited sequence.

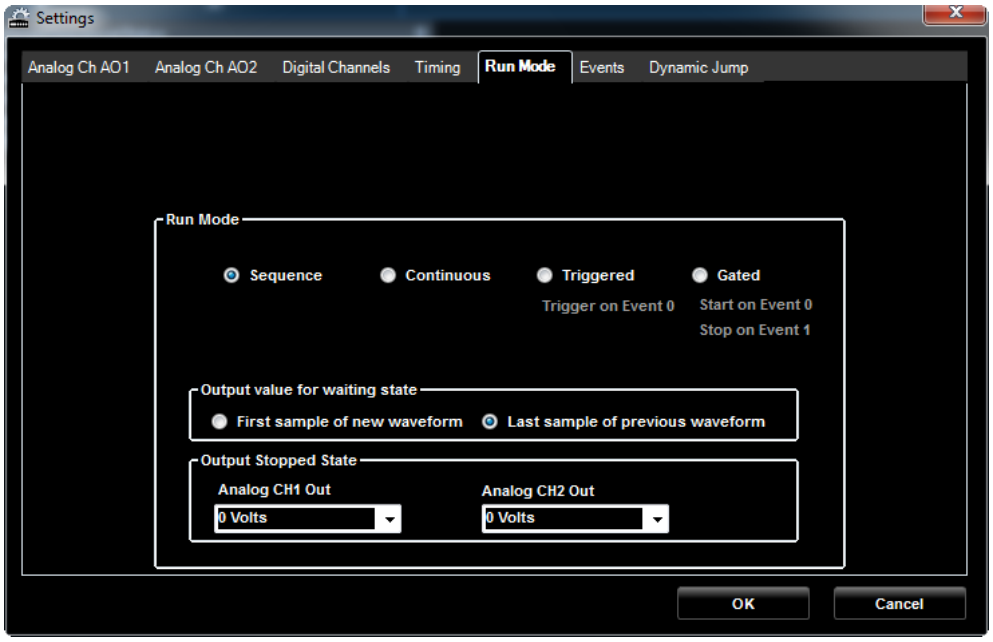


Figure 35 Run mode tab

The AWG4162 instrument supports the following four run modes:

- **Sequence** – Multiple waveforms can be output in the order specified in the Sequence Window.
- **Continuous** – A continuous waveform is output. Only one entry is allowed in the Sequence Window.
- **Triggered** – A waveform is output once when the instrument receives a trigger signal. The instrument will wait for the next trigger signal after outputting the waveform. Only one

entry is allowed in the Sequence Window.
The *Event 0* is the trigger signal and it can be set in the Events Tab.

- **Gated** – The waveform is generated when Event 0 occurs (by default Event 0 = Force Trigger button is pressed) and stopped when Event 1 occurs (by default Event 1 = Force Trigger button is released). Only one entry is allowed in the Sequence Window.
  The *Event 0 and Event 1* are the gate signals (Start/Stop) and they can be set in the Events Tab.

NOTE:

In Sequence and Triggered mode the instrument waits for an event before a waveform is output. It is possible to select the output value for the waiting state between the last sample of the current waveform or the first sample of the next waveform.

Settings - The Analog Ch Tab (ARB Mode)

You can use the **Analog Ch AO1/AO2** tab to quickly set the parameters of analog channels.

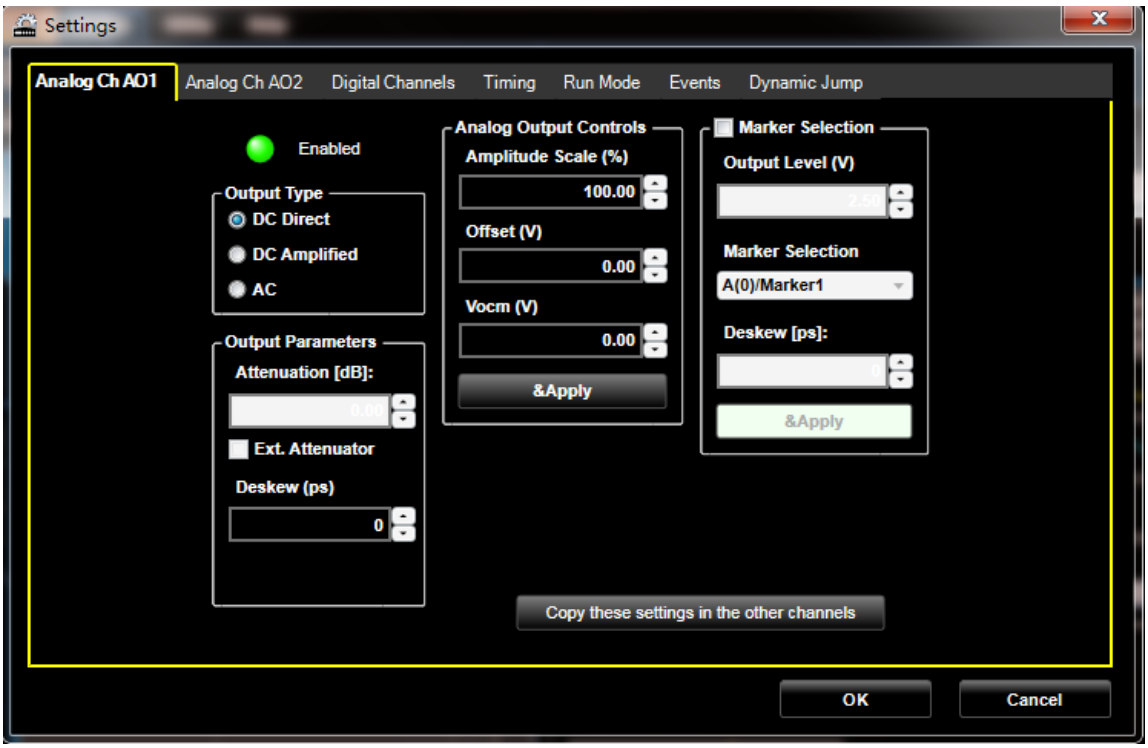


Figure 36 Analog Channel Tab

- **Enabled**: To enable a channel output, click the **Enabled** button.
- **Output Type**: select the output type (DC Direct, DC Amplified or AC) of the selected channel.

- **Analog Output Controls**: you can set parameters for the amplitude scale in %, offset and Vocm. Press the **Apply** button to confirm the changes.
    - **Amplitude Scale(%):** the amplitude scale is a real-time parameter that can adjust the waveform amplitude while the instrument is running and it is applied to all the waveforms contained in the sequencer.
      The default value is 100% and it means that the sequencer waveforms will not be scaled, the maximum value is 200% and it means that the waveform amplitude is multiplied by two and power multiplied by four. The minimum value is 0% and it means that the waveform amplitude is zero.

NOTE:

Limitations of output amplitude range apply here.

    - **Offset:** this parameter applies a differential offset to the selected output. It is not available on AC output.
    - **Vocm(V):** this parameter applies a common mode voltage on positive and negative outputs of the selected channel. It is not available on AC output.

- **Output Parameters:**
    - **Deskew (ps):** this parameter can set a fine delay between the analog to analog output channels in order to realign the outputs with a resolution of about 10 ps at 2.5 GS/s.

    - **Manual Attenuator**: this sets the attenuation type to manual on the DC AMP output. By default the attenuation type is automatic. If you check the checkbox before “Manual Attenuator”, Manual attenuator is enabled.
      Different rules should be considered for automatic attenuation and manual attenuation.
      Using automatic attenuation, the output will not be attenuated and it helps improve the output signal quality for low level signals.
      Using Manual attenuation, the output will be attenuated according to customer’s setting.
      The attenuation has effect on DC Amplified output values and on the offset level. It does not have an effect on Vocm value.

    - **Attenuation [dB]:** if you selected the DC Amplified output, you can apply a programmable attenuation.

- **Marker Selection :**

Marker Selection checkbox : When Marker output is enabled, the signal comes out from the front panel SMA and its maximum update rate is 156.25 MHz at 2.5G S/s.

When Marker output is enabled, the following parameters can be set:

- **Marker Output Level**: Set the Marker Out *Output level (V)* .
- **Marker Selection**: The drop-down list contains Marker / Low / High: it means that you can associate to the Marker Out connector the marker digital signal you can edit in the Waveform Editor Window or an always low/high level signal.
- **Deskew(ps):** this parameter can set a fine delay between the Marker and analog output channel in order to realign the outputs with a resolution of about 78 ps at 2.5GS/s.
  The skew between analog/digital /Marker depends on the sampling frequency.
  The software automatically calculates the maximum value in relationship to the sampling rate you set in the Timing Tab.

NOTE:

The "Analog CH" deskew of CH1 also have impact on CH1 digital channels, so with Analog deskew parameter, you can shift the entire CH1 outputs (Analog+Digital) in relationship to the CH2.

If you move the Digital skew (Digital channels Tab) of CH1, you can shift the digital channels in relationship to the Analog Waveform of CH1, so in this way the digital will be shifted from the analog.

Settings - Digital Channels Tab

Use the **Digital Channels** tab to set a fine deskew on the digital lines, assign values to the output pins and grouping digital lines.
You can generate a digital pattern to test digital devices such as serial and parallel DACs or to emulate protocols. Two Infiniband 12x connectors provide 16 bit LVDS digital outputs each for a total of 32 LVDS outputs.

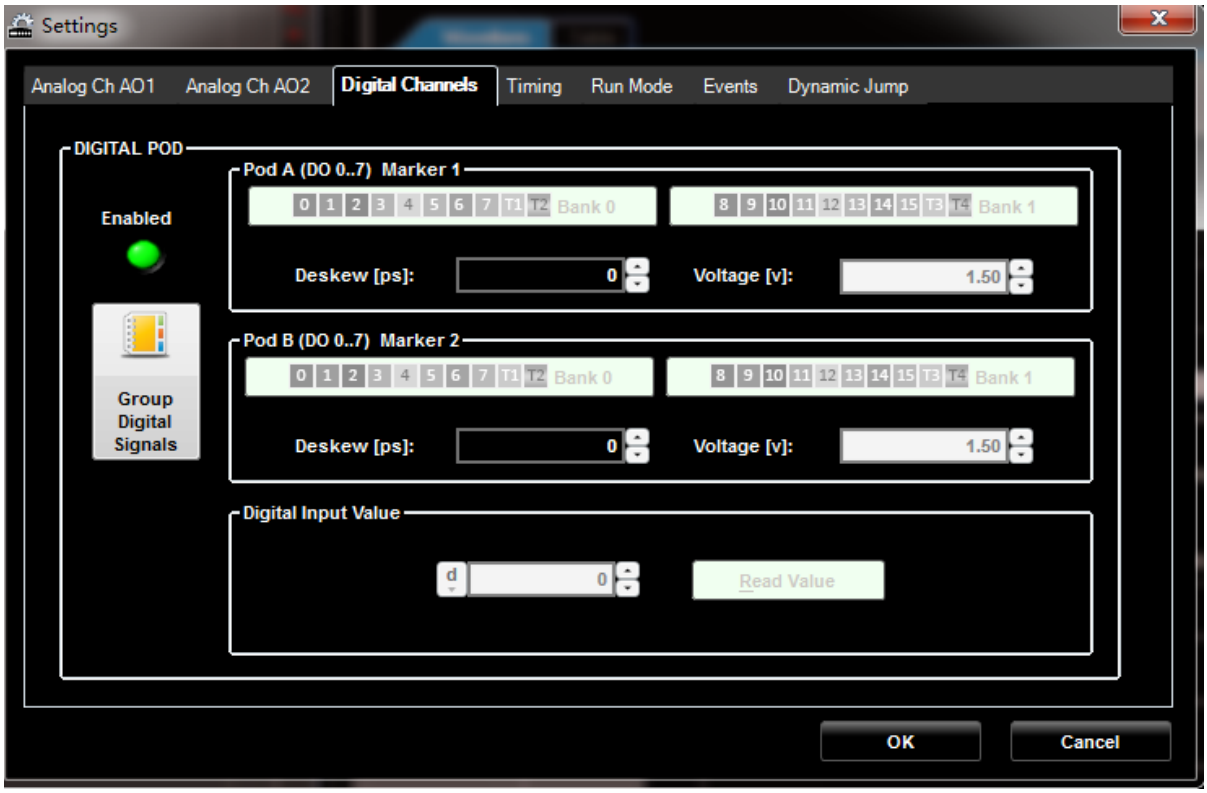


Figure 37 Digital channel tab

- **Enabled**: to enable the digital output channels, click the **Enabled** button.
- **Deskew(ps) :** this parameter can set a fine delay between the digital channels in order to realign the analog and digital outputs with a resolution of about 78 ps at 2.5G S/s.
  The skew between analog/digital channels depends on the sampling frequency.
- **Group Digital Signals:** You can change the name of the signals and create / rename buses pressing the  button, the following figure is displayed.

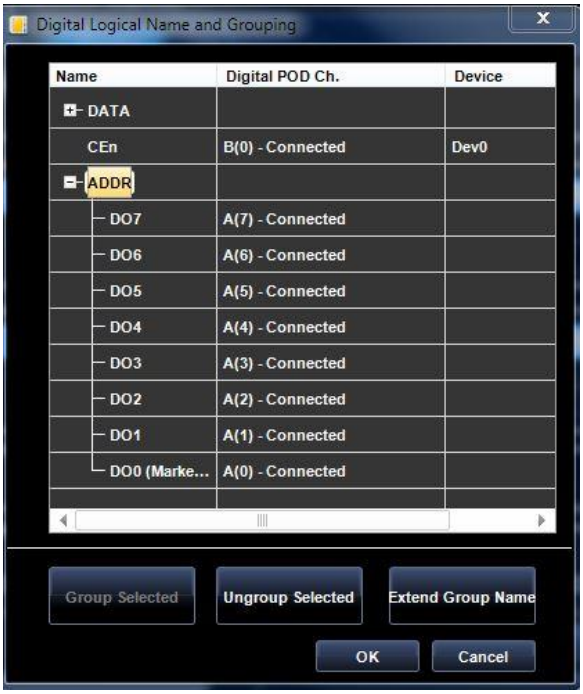


Figure 38 Digital Logical Name and Grouping window

- The first column displays the digital channels logical names that can be assigned by the user.
- The second column displays the *digital pod channel* that it is associated to the *digital logical name*.
  "Connected" means that the Digital Probe is connected to the digital channel.
- The third column displays the device name associated to the digital channel.
- Left Click + SHIFT to select multiple DO single digital lines, then press the **Group Selected** button to make a bus.
- Double Click on single signal or bus name to rename it.
- Select a bus and press the **Extend Group Name** button to extend the root name to the single lines of the bus.
- Select a bus and press the **Ungroup Selected** button to ungroup a bus into single lines.

Settings - Timing Tab

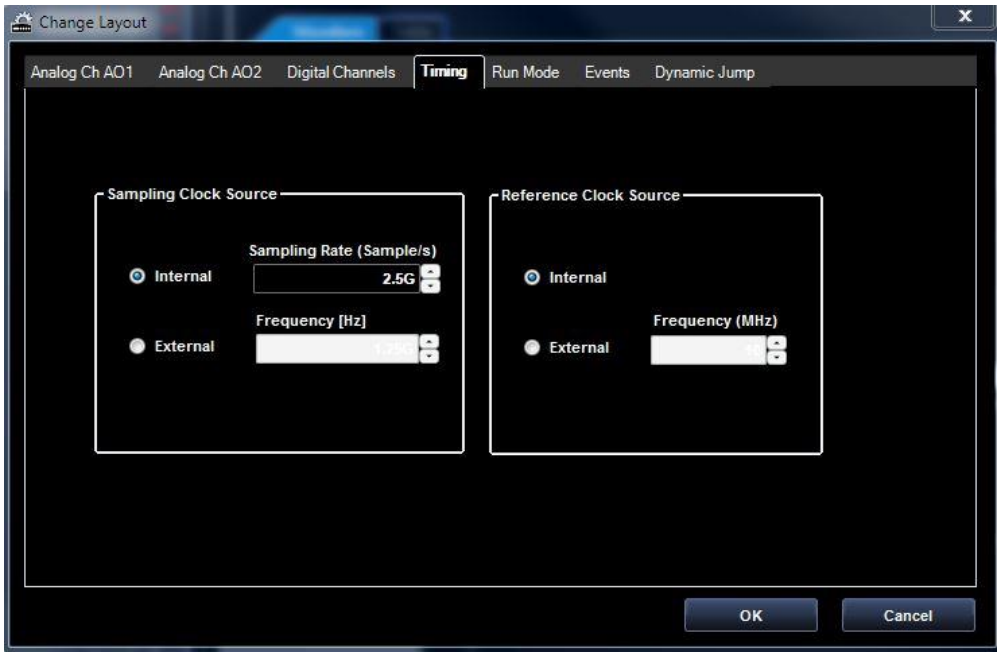


Figure 39 Timing tab

- **Sampling Rate:** the sample rate can be set to from 100 S/s to 2.5 GS/s for Arbitrary mode channels
- **Sampling Clock Source**: it may be set to Internal or External.
  If **Internal** is selected, the sampling clock signal is generated internally.
  If **External** is selected, the clock signal from the EXT. CLOCK IN SMA connector is used.
  When the External Clock is selected, an External Clock Frequency must be set for your clock signal (1.25 GHz to 2.5 GHz).

NOTE:

- *In the Single Sequencer project, the AO1 is the Master channel and it propagates the sampling clock to the slave channel AO2, so the EXT. CLOCK IN CH1 is the only one input available as External clock.*
- *In a Multi Sequencer project, both EXT. CLOCK IN SMA inputs are available as External clock, because the two channels works independently, or ch2 clock can be set to "Lock to Master Clock" so both channels shares the same sampling clock.*

- **Reference Clock Source**: you can select the Reference Source (Internal or External), the Reference Source is selectable only when the Sampling Clock Source is set to Internal.

Settings - Events Tab (Single Sequencer)

The AWG4162 has Event Jump and Wait Event (*Input Waveform Properties window*) functions which change the generation sequence using an event signal.
Additionally, Triggered and Gated run mode execution depends on Event 0 and Event 1.

Use the **Events** tab to set the Events, Trigger IN and Timer parameters.

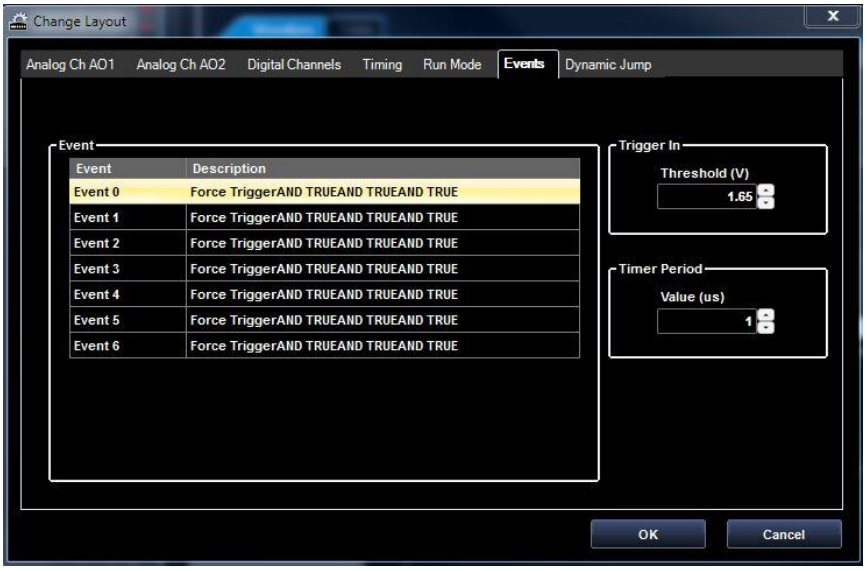


Figure 40 Events tab

- **Event**: you can configure up to seven events (Event 0…Event 6) and each event is a logic combination between four operands (*Operand 1..Operand 4*) and three operators(*Operator 1..Operator 3*).
  The *Event table* contains the event columns and the description columns that give a summary of the event setting.

Double click on an Event table row to open the *Event Editor*; the editor gives you access to the available operands and operators.

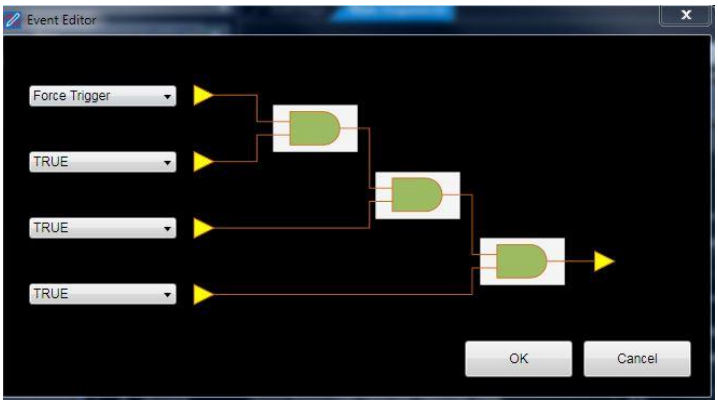


Figure 41 Events editor

The logic combination evaluation formula is:
EventN= (Operand1 ***Operator1*** *Operand2)****Operator2*** *Operand3)****Operator3*** *Operand4)*

The possible operations include **AND**, **OR**, **XOR**, **NAND**, **NOR**, and **XNOR** and the possible operands include **False, True, Trigger IN, Timer, Force Trigger, Not Trigger IN, Not Timer, Not Force Trigger, A(0)/Marker1, B(0)/Marker2**.
The Event Manager detects the level of the Operand and it executes the selected logic operation.
The Sequencer will trigger on the edge of the resulting equation.

The parameters of the Events tab are explained in the following:

- **Trigger IN**: the event is received by the dedicated SMA Connector on the instrument front panel.
- **Timer**: the event is a pulse generated by a time counter you can set in the Event Tab. If you select “Not Timer” the pulse will be inverted.
- **Force Trigger**: the event is generated by a software trigger pressing the button in the main toolbar.
- **A(0)/Marker1, B(0)/Marker2**: the marker event can only be used for jump in sequence and stop event in gate. In other case, marker event can't take effect. B(0)/Marker2 is available in Multi-Sequencer workspace only.
- **NOT: the** “Not” gives the user the possibility to invert the signal polarity. For example “Not Trigger In” event takes effect on the falling edge of the Trigger In signal.
- **Trigger IN - Threshold(V):** use this field to select the Trigger IN threshold voltage level.
- **Timer Period - Value (uS):** use this field to set the value of the time counter in us.

Settings - Events Tab (Multi Sequencer)

In Multi Sequencer workspace, the Event TAB parameters are the same of the Single Sequencer workspace except for *Using Master Event* led placed on the Slave Settings panel.

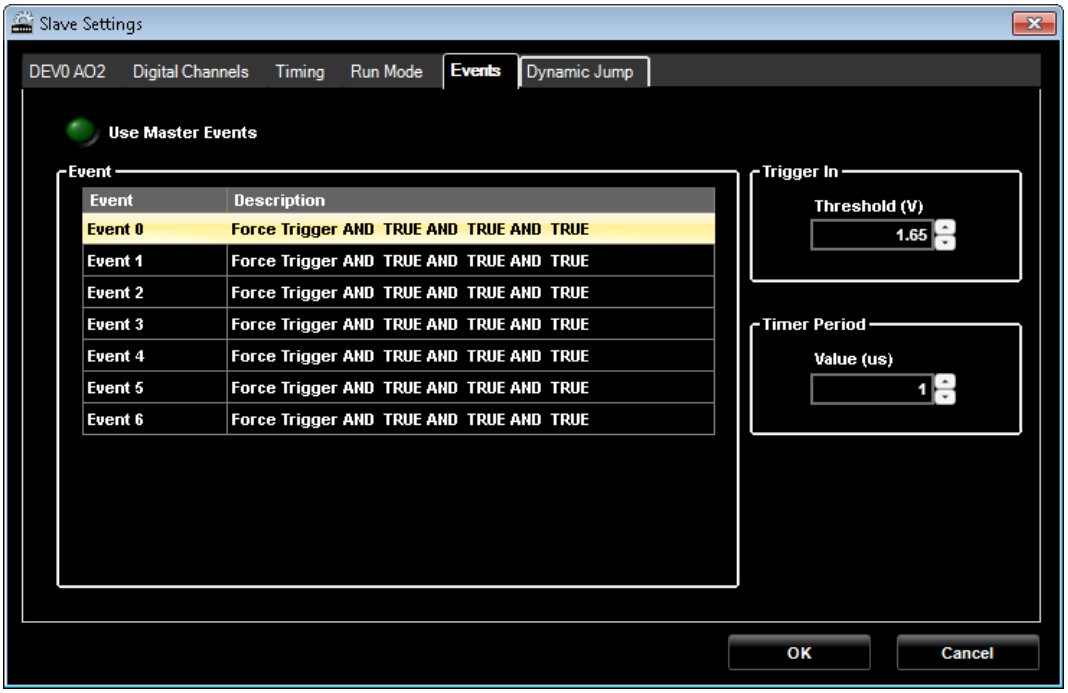


Figure 42 Events tab (Multi sequencer)

If you enable the *Use Master Events* control, the Slave event list will be disabled and the Slave channel will use the events of the master channel to control the generation sequence.

Settings - Dynamic Jump

Use this tab to modify the execution flow of the sequencer by forcing a specific entry to be executed.

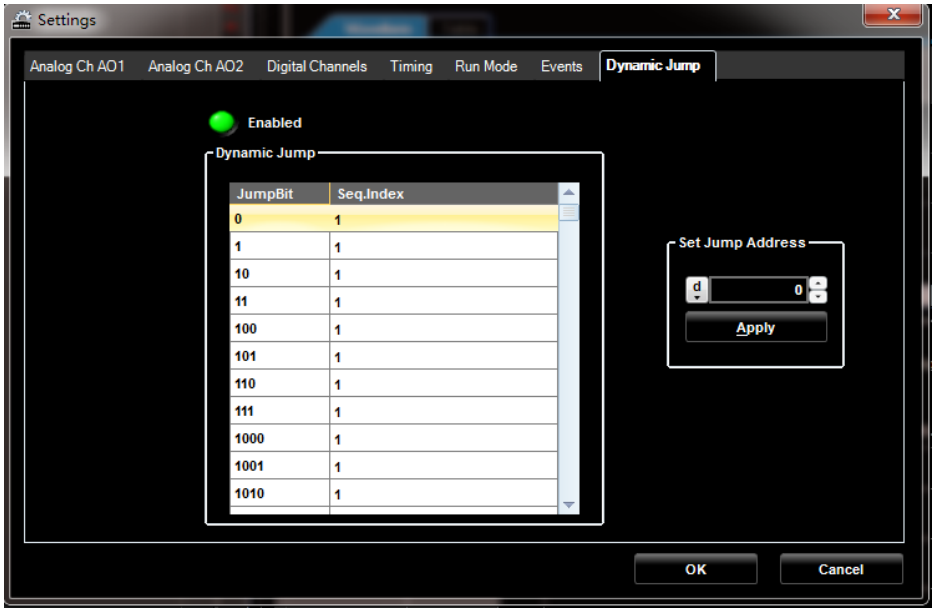


Figure 43 Dynamic jump tab

Take the following procedures to modify the sequencer:

1. Specify the relationship between the Set Jump Address value and the entry of the sequencer in the *Dynamic Jump* table.
2. Use the “Set Jump address” field to select one of the available entries in the JumpBit column.
3. Press the Apply button.
4. The **Enabled** button enable/disable the dynamic jumps.
5. If change the value in table “Dynamic Jump”, it won’t take effect until press ok to close setting; Then reopen setting to operate "Set Jump Address"

Waveform Window

You can use the **Editing Waveform Window** menu to create a new analog/digital waveform or modify an existing waveform.
The AWG4162 instrument has two analog outputs and up to 32 digital channels that the user can configure according at his needs.

Take the following steps to create a new waveform or edit a waveform:

1. Use the Waveforms TAB toolbar buttons to create a New Mixed / Analog / Digital Waveform .
   PLEASE NOTE THE FOLLOWING:
   - The *Mixed Waveform* is the easiest way to create a new entry for the Sequencer because it inserts in the same entry both analog and digital channels correctly formatted.
   - If you want to modify an existing waveform you can double click it on the *Waveforms TAB* to open **the Editing Waveform** window.

2. The **New Waveform** window is displayed as shown in the following figure. Type the name of the waveform and choose the samples length. You can insert the Length in *Samples* or *Time*. Click **OK** to confirm.

Figure 44 New waveform dialogue box

Note: Waveform length must be multiple of 64(<320 points) or 16 (>=320 points).

3. The **Editing Waveform Window** is displayed as shown in the following.

Figure 45 Editing waveform window

Numbered callouts on this image correspond with the following interface section descriptions.

1. Analog/Digital Waveform Graph Tools
2. Mixed Signal Waveform Editor
3. Data Editor

Cursors Tools

Cursors/Tools TAB gives you access to the following functions:

Cursors

Cursors are useful to identify and enlighten waveform data for improved organization and viewing.

Clicking the Cursors Markers button on the *Editing Waveform main toolset* shows or hides the marker window.

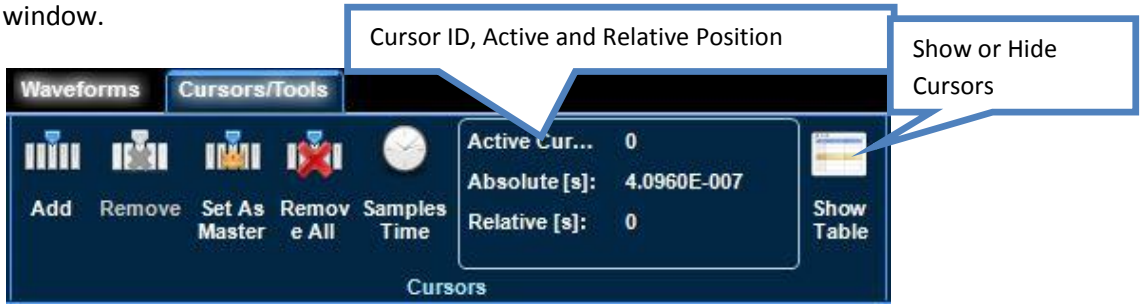


Figure 46 Tool menu

Other field values on the toolbar show the **Active** (or ID) of the currently selected cursor, and its **Absolute** and **Relative** positions.
When Cursors are turned on, all of the cursors present in the *Waveform Editing Window* are listed inside the **Cursor** screen.

Master	Id	Abs Pos	Rel Pos	Sync
	0	409.60...	0	
	1	409.60...	0	
	2	675.02...	265.420...	
	3	174.48...	-235.11...	
	4	409.60...	0	

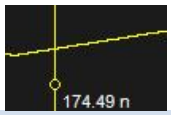
Figure 47 Cursor list

The **Master Cursor** is the one labeled with the following icon.

Relative positions are calculated from the master cursor position.
The master cursor automatically moved during a data search operation to show relative results.
Change the master cursor by selecting the new marker in the cursor window and clicking the Master Cursor icon in the *Waveform Editing Window* toolbar.

Cursor screen columns show the progressive cursor identifiers, the absolute time position (the time distance between the cursor position and the start of the acquisition) and the relative time position (the time distance between the cursor and the master cursor). Any time one of

the cursor is moved, all the values are automatically updated and shown.
The following functions are used on Cursor.

	**The Add button puts a new cursor in the visualization area.**
	The **Remove** button eliminates the marker selected in the Cursor screen.
	Move a marker by clicking and dragging a selected cursor.
	Remove all cursor by clicking the **Clear all cursor** button.

NOTE:

- *You can also perform many of the aforementioned functions by right clicking inside the Cursor screen and choosing from the list of functions shown.*
- *You can remove all cursors except for one.*
- *You can create as many cursors as needed.*

Go To a Selected Target

The Go to field on the *Editing Waveform main toolset* contains multiple functions on its right side drop-down. The functions allow you to select the position where the master marker is going to be moved within the visualization area.
The Go to functions include:

	**Go to time - Moves the master cursor to the time position specified in the text field to the left of the control.**
	**Go to start samples** - Moves the master cursor and visualization area to the start of the acquisition.
	**Go to end samples** - Moves the master cursor and visualization area to the end of the acquisition.
	**Cursor n** - Centers the visualization area on the cursor/marker n (position specified in the text field to the left of the control).
	You can move the selected cursor to the middle of the current visualization by clicking the **Move active cursor here** button.

Search

Searching can be done from the *Editing Waveform Window*. It is also available in the aforementioned search section regarding the **Waveform View** screen.
You can search for a specific bus, signal, rising, or falling edge value.

Activate the search option by clicking the **Search Settings** button  .
The Search Settings window is shown as follows. You can input search criteria here.

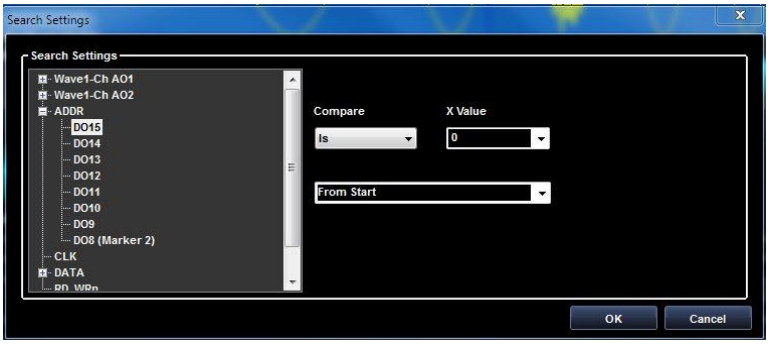


Figure 48 Search settings window

In the **Signal Type** search list on the right side of the Search Settings window, all defined analog/digital signals and busses are shown. Select the signal or bus and then provide a specific value for the search.

NOTE:

*The **Compare** and **Value** fields change depending on the Signal Type selected in the search list.*

Use the Compare field to select between the following Search logic operators:

- **=** or **Is** - Find the equivalent value.
- **!=** or **Is not** - Find the unequal value.
- **>** - Find values greater than the one specified (on digital channels only available if a bus is selected).
- **<** - Find values less than the one specified (on digital channels only available if a bus is selected).

On digital channels use the **Value** field to provide the specific value or edge on which to search. If one channel is selected, the **Value** field has the following options:

- **0** - Searches for a logic **0**.
- **1** - Searches for a logic **1**.
- **Rise** - Searches for a Rising Edge trigger. 
- **Fall** - Searches for a Falling Edge trigger.
- **Change** - Searches for any trigger edge. 
- **HiZ**

- **Violations** – Searches for any timing violations on Analog Marker. The Analog Marker maximum update rate is 156.25 MHz; in case of timing violations, a warning symbol appears next to the digital waveform label.

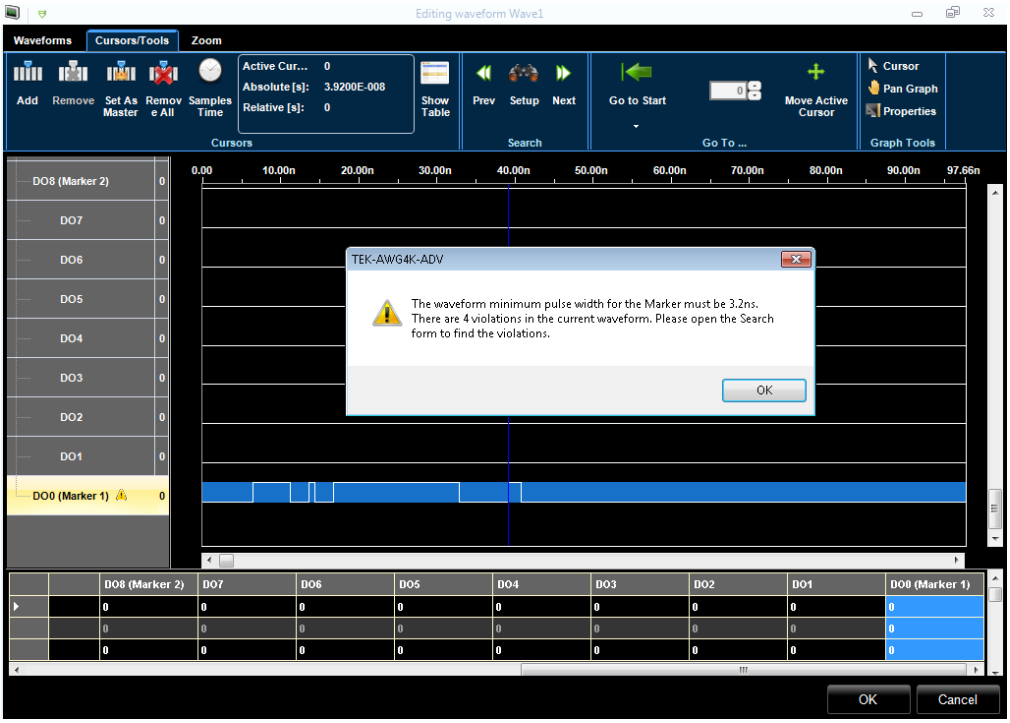


Figure 49 Search result window 1

Figure 50 Search result window 2

To cancel the timing violations, the following procedure should be applied:

1. Open the Search window and select the digital waveform where the violations occur.
2. Select Violations and press OK.
3. Press the Next button to move the cursor on the timing violation.
4. Remove the timing violation manually filling the values with '0's or '1's.
5. Remove all the timing violations on the Marker waveform.
6. Please note that it is possible to start the waveform generation even if all the violations are not solved, but the analog marker will work out of specification.

The **From Start** button can be used to specify where the search starts within your data generation. Possible options include:

- **From Start** - Starts the search from the beginning of the waveform.
- **From End** - Starts the search from the end of the waveform.
- **Master Marker** - Starts the search from the Master Marker position.

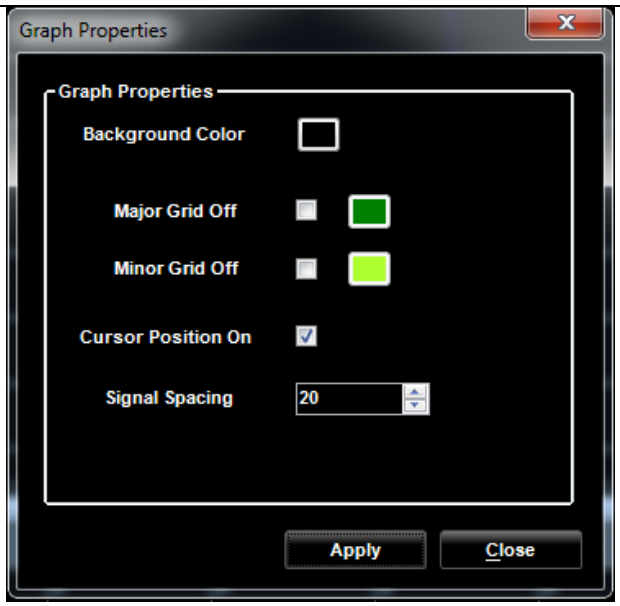
You can select criteria on the Search Settings screen and click the **OK** button. The results are then shown on the **Editing Waveform Window**.
You can use the **Search Backward** or **Search Forward** buttons to navigate through your search results.

NOTE:

As you navigate through your search results, the master cursor is updated to the subsequent values in your results.

Graph Tools

	**Cursors/Tools TAB: This button allows changes the mouse function for the graphic area to cursors/markers movement.**
	*Cursors/Tools TAB:* The hand tool allows you to dragging inside the graph area.
	*Cursors/Tools TAB:* You can change the properties of the graph display area. Click the **Waveform View Settings** button and the **Graph Property** screen is shown.



Changes can be made as follows:

- The **Background Color** can be changed as desired.
- Change colors and turn the **Major** and **Minor Grids** on or off and change their line coloring.
- **Cursor Position** indicators can be turned on or off.

This button switches the X-axis representation between **number of samples** to **seconds**. Default values are optimized based on the selection made.

Zoom Tools

Zoom TAB gives you access to the following zoom functions:

	**Zoom In: Auto zoom in function.**
	Zoom Out: Auto zoom out function.
	Zoom Manual: This button allows zooming in on a selected rectangle of the graph. Click and drag inside the graph area to create your zoom rectangle.
	Zoom All: This button resets all activated zooms
	Cursors: This button allows changes the mouse function for the graphic area to cursors/markers movement.

Analog Waveform Graph Tools

AWG4162 instrument handles Analog Waveforms, Segments, and Components as explained in the next parts.

Analog Waveform

An analog waveform is a sequence of elementary segments and it contains the temporal order by which the segments are generated.
You can add a standard waveform simply by pressing the **Edit** button and choosing a basic waveform like DC Level, Sine, Increase Ramp, Triangle, Sawtooth, Rectangle etc. The AWG4162 software will display a waveform made of one segment.

If you need to generate more advanced waveforms, you should add more segments to your waveform.

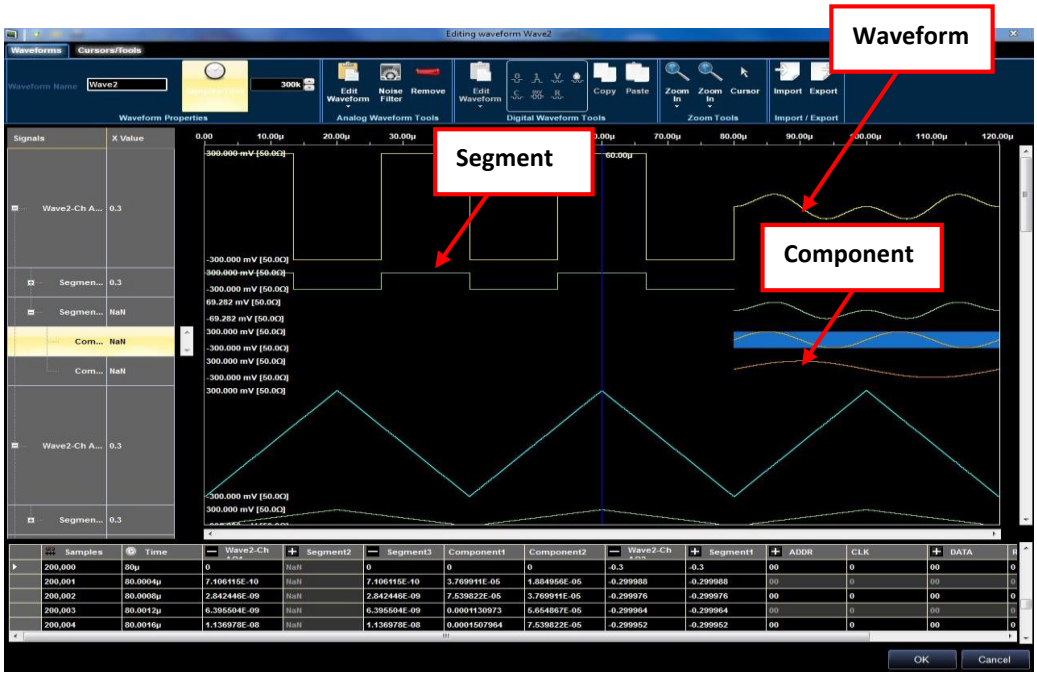


Figure 51 Analog waveform

Segment

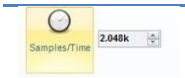
A Segment contains one or more Components, all of the same length, combined by means of the elementary Add, Subtract, Multiply operations.
If one Segment contains more Components, the following formula will be applied:

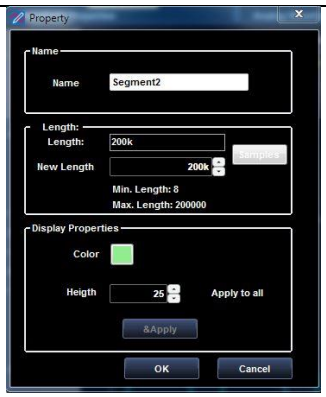
Segment = (Component1 (**Add/Sub/Multiply**)Component2) **Add** Component 3 **Add** Component 4 **Add** Component N

Component

A Component is the basic element for the construction of a Segment. Each Component may be represented by a standard waveform (DC Level, Sine, Cosine, Exponential, Triangle, Rectangle, Ramp, Pulse, Sync, Sawtooth, Sweep), by a Formula, or its component samples can be loaded from a text file.

In any case, the samples of a Component are calculated/loaded as a function of the AWG4162 frequency (sample rate) and of the length of the component itself (number of points).

	**This button allows changes the waveform length. Click on the Samples/Time button to change the waveform length visualization from samplers to time.** **PLEASE NOTE THE FOLLOWING:** • **In Arbitrary Mode the allowed waveform length is 64 to 64M samples in multiple of 64 for < 320 samples or in multiple of 16 for >= 320 samples.** • **All segments and components will be re sampled.**
	*Vertical Zoom Auto scale* function for the selected analog waveform.
	*Vertical Zoom Auto zoom in* function for the selected analog waveform.
	*Vertical Zoom Auto zoom out* function for the selected analog waveform.
	Press the **Edit** button to open the *Waveform Standard Editor Window* and create a basic waveform like DC Level, Sine, Increase Ramp, Triangle, Sawtooth, or Rectangle. Click the arrow to open the pop-up menu: select the standard waveform (Sine, Triangle, Square,etc.) as short-cut for the Waveform Standard Editor Window.
	Press the **Remove** button to remove the selected Segment / Component of the waveform. This button is active only if more than one segment/component exists in the current waveform.
	Right-click on the Waveform/Segment/Component to open the context menu. Select **Properties** to open the waveform/segment properties window and change the waveform display parameters like color, plot height or resize the segment length.



Press the **Effect** button to open the *Effects Settings and Parameters* window and add noise, filtering to your analog waveforms.

Waveform Standard Editor Window

Press the **Edit** button and the Waveform Standard Editor Window is shown.
This window shown as follows allows you to edit standard waveforms, segments and components parameters.

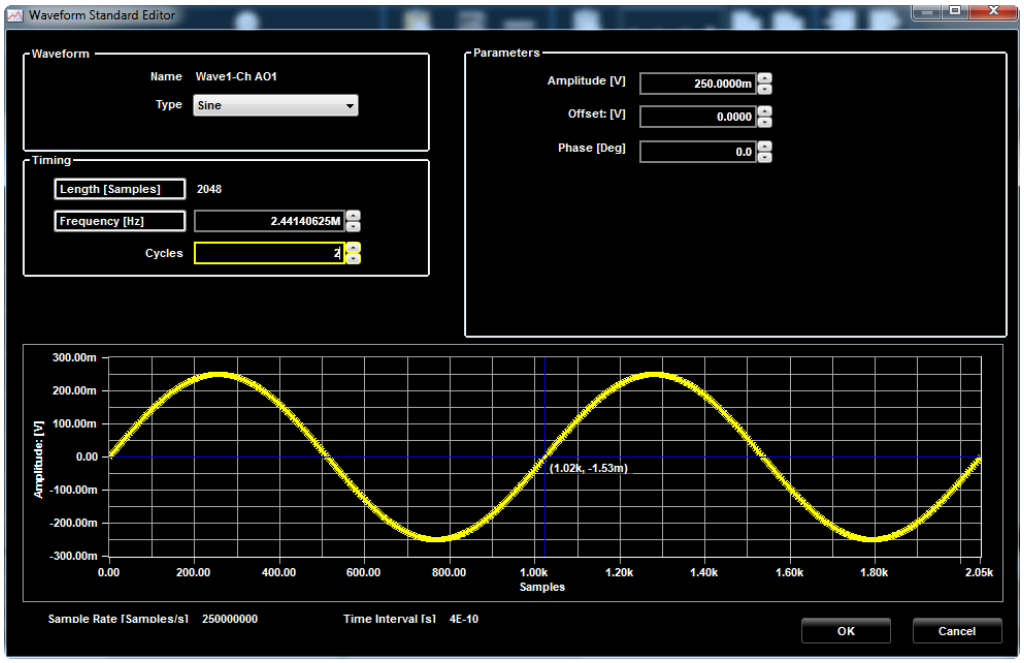
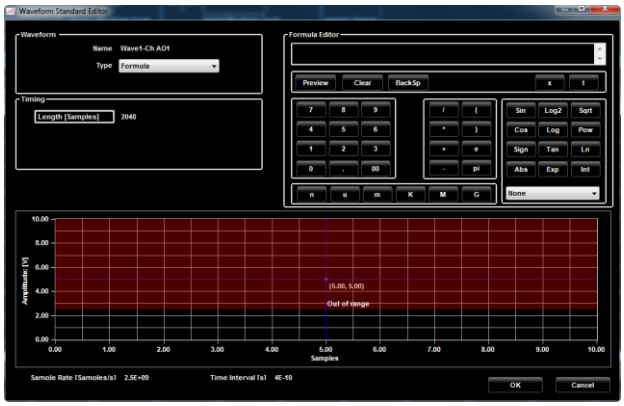
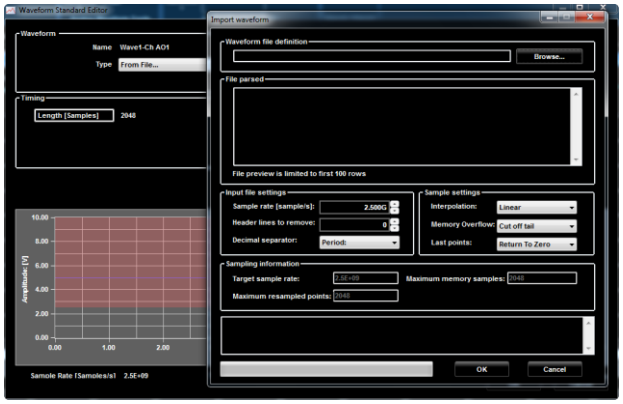


Figure 52 Waveform standard editor window

The **Type** menu allows selecting the waveform among a list of possible signals or functions. Depending on the selected Type, different parameter may be edited. The different possibilities include the following:

Type	Available Parameters
DC Level	Offset [V]
Sine	Frequency[Hz/cycles], Amplitude[V], Phase[°], Offset[V],Cycles
Cosine	Frequency[Hz/cycles], Amplitude[V], Phase[°], Offset[V], Cycles
Triangle	Frequency[Hz/cycles], Amplitude[V], Phase[°], Offset[V],Cycles
Rectangle	Frequency[Hz/cycles], Amplitude[V], Phase[°], Offset[V], Duty Cycle [%],Rise time[s], Fall time[s], Cycles
Saw tooth	Frequency[Hz/cycles], Amplitude[V], Phase[°], Offset[V], Cycles
Increase Ramp	Amplitude[V], Offset[V]
Decrease Ramp	Amplitude[V], Offset[V]
Pulse	Amplitude[V], Delay[s], Width[s], Offset[V]
Sinc	Amplitude[V], Offset[V], Peak Position[s], Lobe Width[s],

Exponential	Frequency[Hz/cycles], Vo[V], Vinf[V], Time Constant[s]
Sweep	Amplitude[V], Offset[V], Start Frequency[Hz], Stop Frequency[Hz]
Formula	Calculator Window 
From File	Explorer Window 
Custom	Insert the sample values by editing the table entries

	
PRBS (PRBS pattern)	Amplitude[V], Offset[V],Equation, Invert

The Formula type allows defining the waveform by means of a mathematical expression.
The waveform is edited by using the Formula Editor window that can be activated by clicking the Edit Formula button. The mathematical expression can be a function of time or a function of samples by using the **t** or **x** variables, respectively. The software verifies, in run time, that the component to be edited does not exceed the limits for the selected output and that the formula syntax is correct. In case of error, an error indication is shown in the Error message indicator.

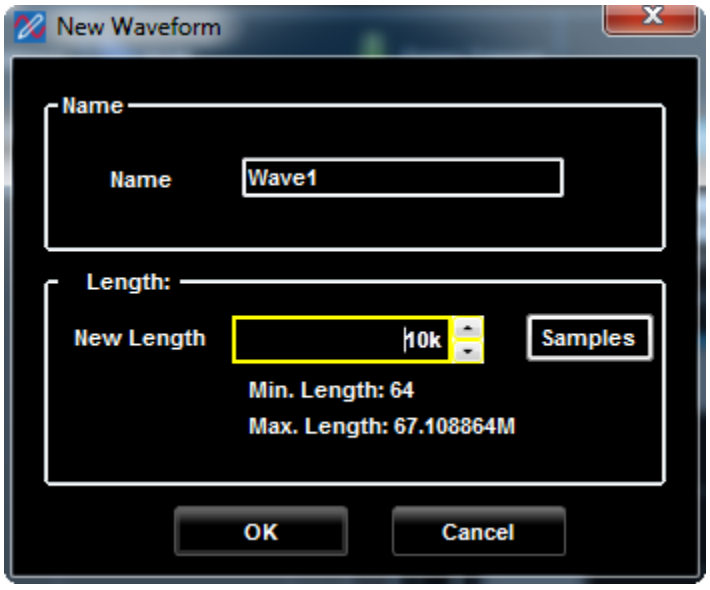


Figure 53 New waveform

Effect Settings and Parameters Window (Analog Waveforms Only)

Press the **Effect**  button and the *Effects Settings and Parameters Window* is shown. On the *Settings* Tab for each waveform you can select to add a noise effect, a filter, noise then filter (noise and filter), filter then noise (filter and noise) .

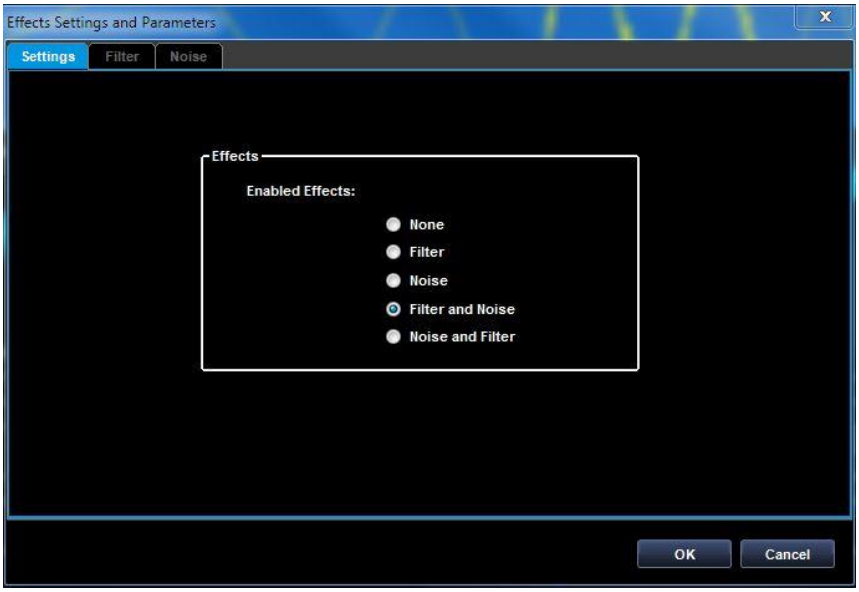


Figure 54 Effect settings and parameters

Filter Settings TAB

This tab allows you to apply a digital filter to the selected waveform.

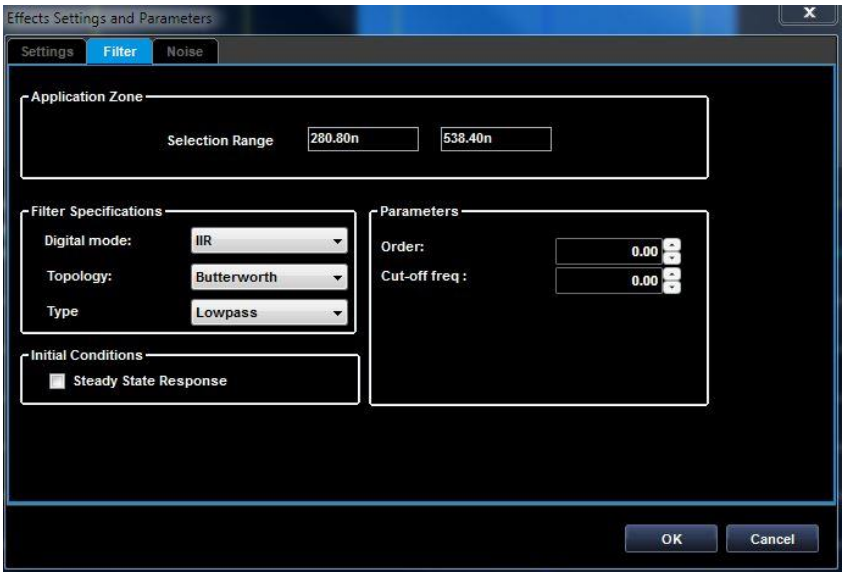


Figure 55 Filtering waveform

Formula waveform view

When creating standard analog and digital waveforms, **sine** and **square wave (rectangular)** types are commonly used (alone or in combination with other waveshapes).
However, when waveform creation cannot be accomplished using simple combinations of standard waveform types, importing waveforms from a measurement instrument, file, or creating it analytically using equations or formulae are ideal methods.

Create waveform with Formulas

This section of the manual shows you how to create various waveforms using formulas in AWG4162.

It covers the standard steps used to create an advanced waveform component using formulas. Remaining topics show example formulas producing various waveform types.

Similar to most other tasks in AWG4162, you must first create a workspace with Arbitrary Waveform Generator as the Operating Mode.

If you already have a workspace open, be sure it meets the aforementioned requirements before proceeding with the following steps:

1. Click the **New Mixed Waveform** button. 
   The **New Waveform** window is shown. Type the name of the waveform “Wave1” and choose 10000 for the samples length of the waveform. Click **OK** to confirm.

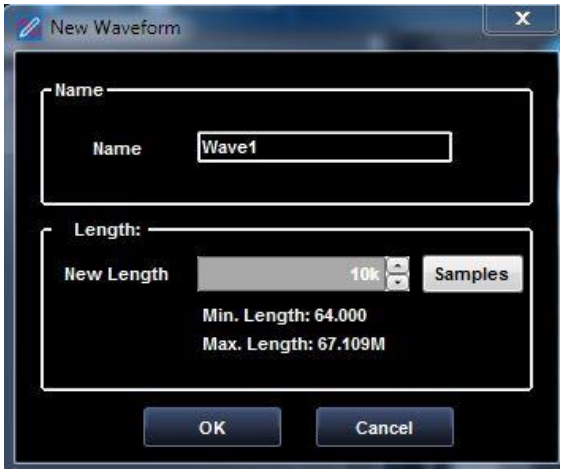


Figure 56 New waveform

3. The Editing Waveform Window is shown. Select the waveform Wave1 and click on the Edit 

button.

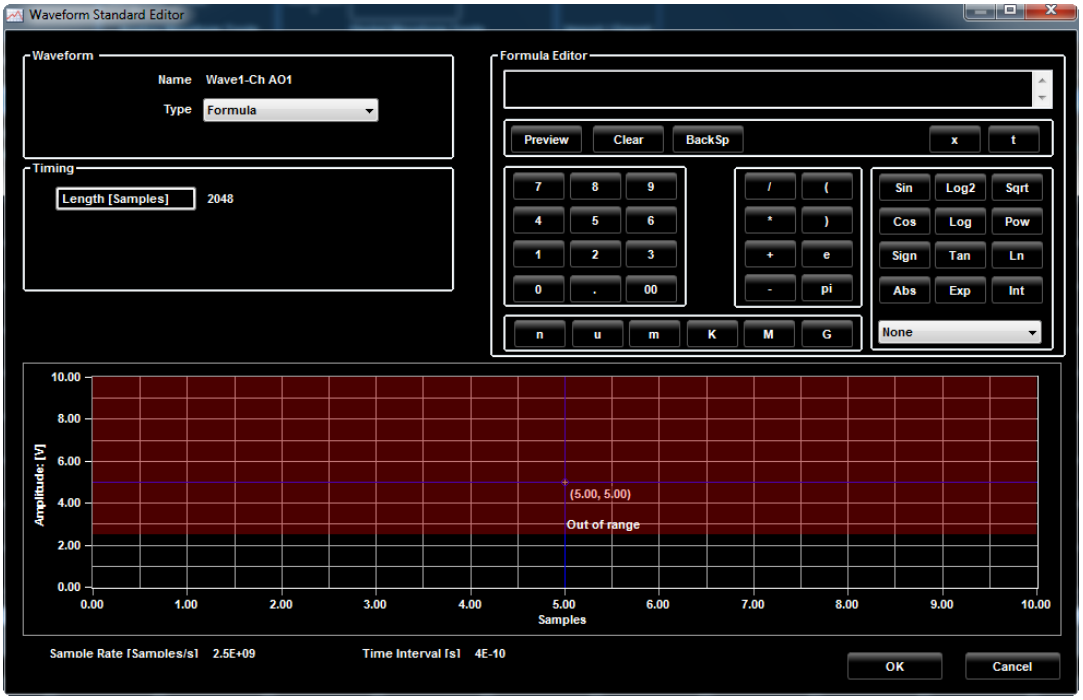


Figure 57 Digital waveform editor

4. On the Type drop-down list select *Formula*. The Formula Editor is then shown on the right. The editor helps building your waveform analytically using equations. Your equation can be based on time(**t**) or samples(**x**).

   Please pay attention to the following:

   - Your formula is shown in the text box as it's built.
   - Numeric values can be entered from the keypad along with n (nano), µ (micro), m (milli), K (kilo),M (Mega), and G (Giga) multipliers.
   - Equations can be based on the functions **Sin, Cosine, Log base 2, Log Base 10, Pow** (rise to a power**), Square Root, Sign, Tan, Ln** (Natural Log), **Abs, Exp, Integer, ArcSine, Arc Cosine, Arc Tan, Ceiling**, and **Floor** along with the basic arithmetic operators **+** (addition), **-** (subtraction), ***** (multiplication), and **/** (division).
   - The **Preview** button compiles your formula and renders it on the graph above the Component Definition dialog.
   - The **OK** button saves your formula and exits from the **Waveform Standard Editor** window.

At this point, your newly-created waveform (made using formulas) is now saved as a waveform. You can create additional components and/or add your newly-created component to the sequencer and output your waveform from AWG4162.

Exponentially Decaying Sine Wave

An exponentially decaying 2 MHz sine wave is shown in the following figure. The actual formula used here is Exp(-t/E-6)*Sin(2*3.14592*2*E6*t).

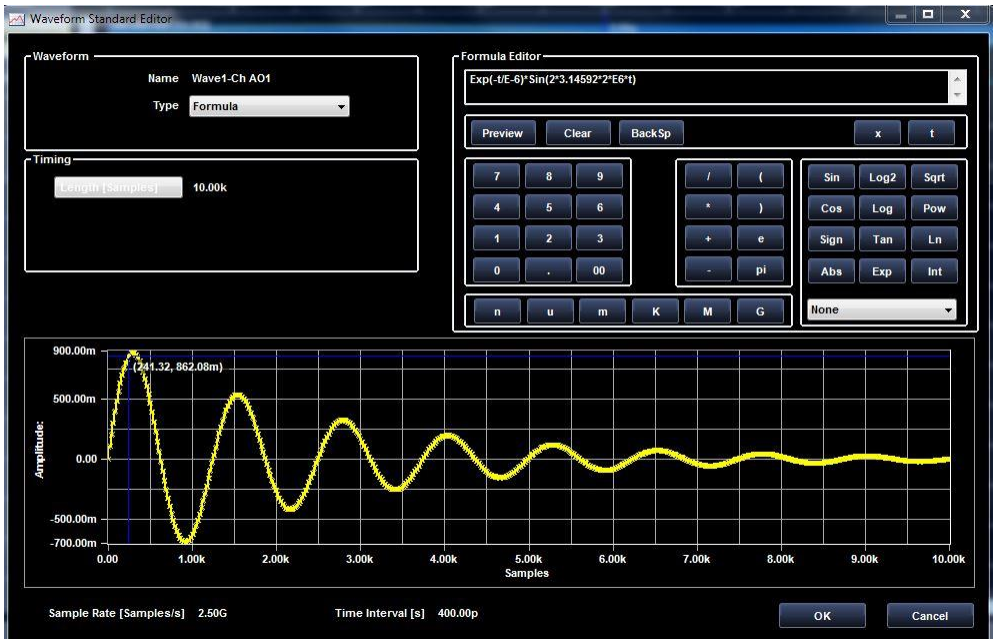


Figure 58 Exponentially decaying sine wave

Formula's General Format

$V*Exp(-t/T_c)*Sin(2*pi*t*F_s)$

Where

- T_c – Time Constant in seconds
- F_s - Sine wave frequency in Hertz
- V – Signal amplitude in Volts peak

Ramp

A ramp waveform is shown in the following figure. The actual formula used here is 0.2*E6*t.

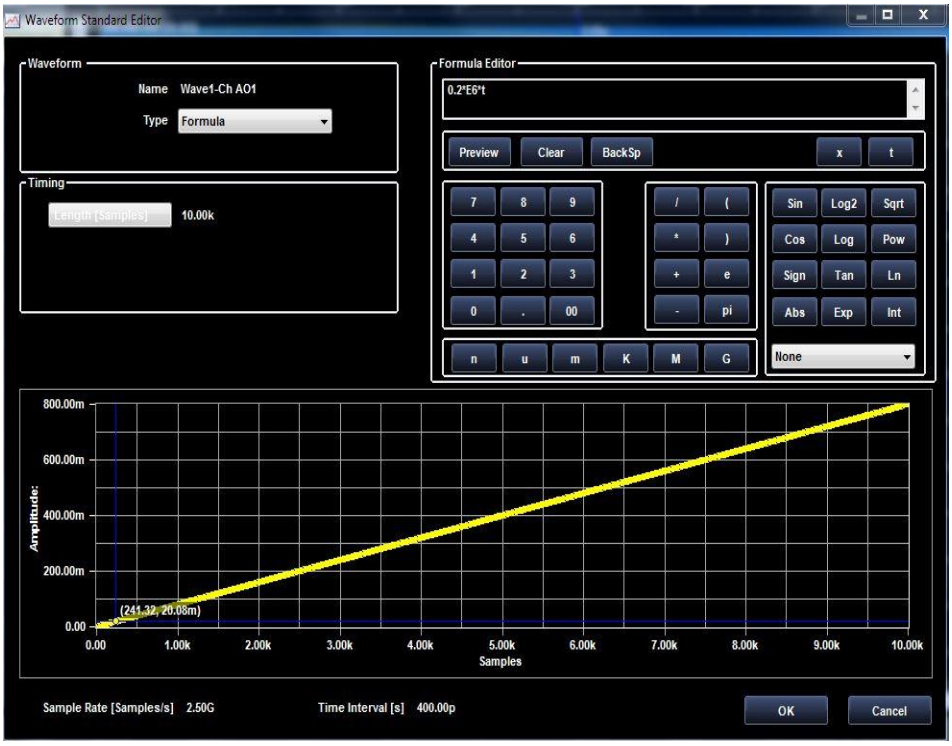


Figure 59 Ramp waveform

Formula's General Format

$A*t$

Where A – Slope of the ramp in Volts/second.

Rising Exponential

A rising exponential waveform. The actual formula used here is 1-Exp(-t/(1*E-6)).

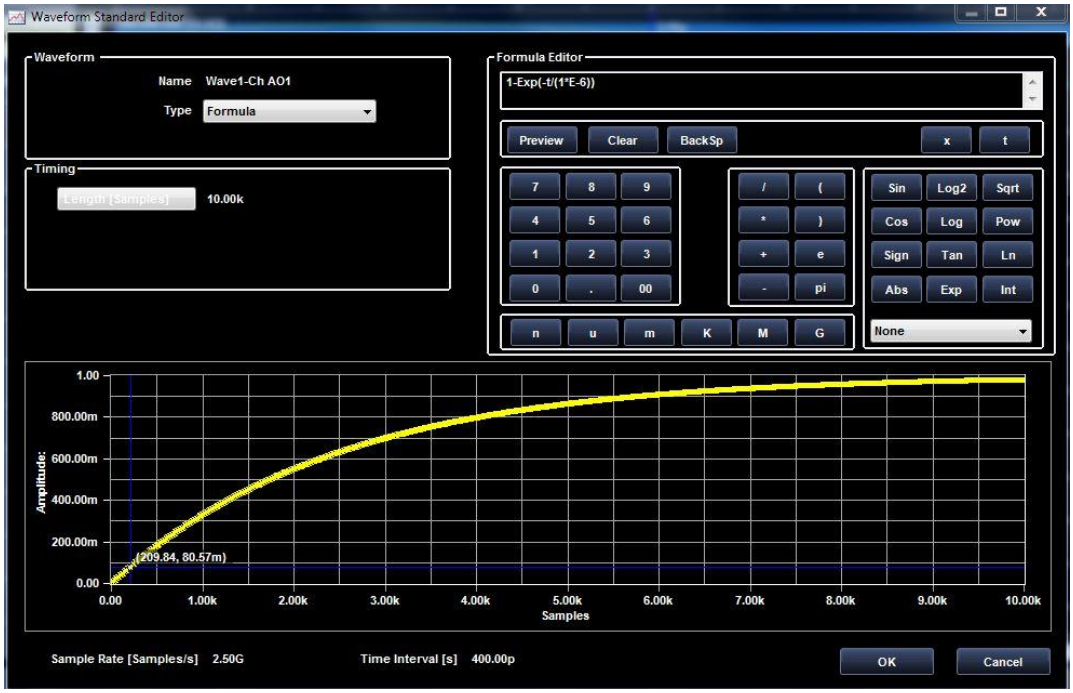


Figure 60 Rising exponential waveform

Formula's General Format

$1\text{-}Exp(-t/T_c)$

Where T_c – Time Constant in seconds.

Decaying Exponential

A decaying exponential waveform is shown as follows. The actual formula used here is Exp(-t/(1*E-6)).

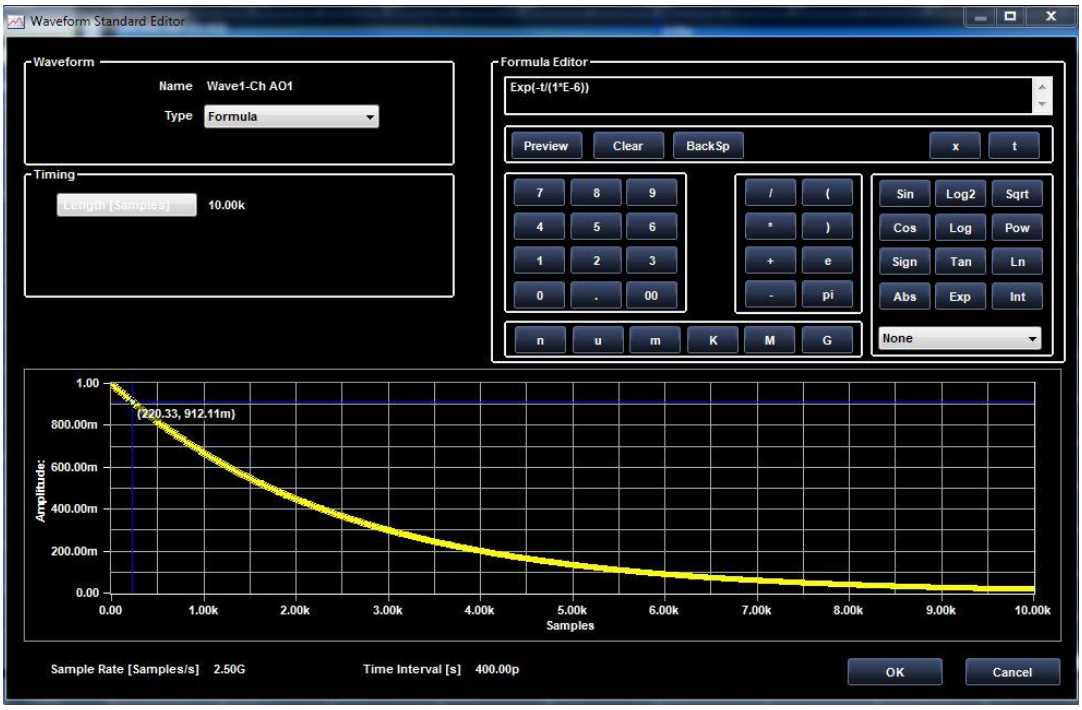


Figure 61 Decaying exponential waveform

Formula's General Format

$Exp(-t/T_c)$

Where T_c – Time Constant in seconds.

Sine

A linear amplitude sweep of a 1 MHz sine waveform is shown as follows. The actual formula used here is Sin(2*3.141592*2*E6*t).

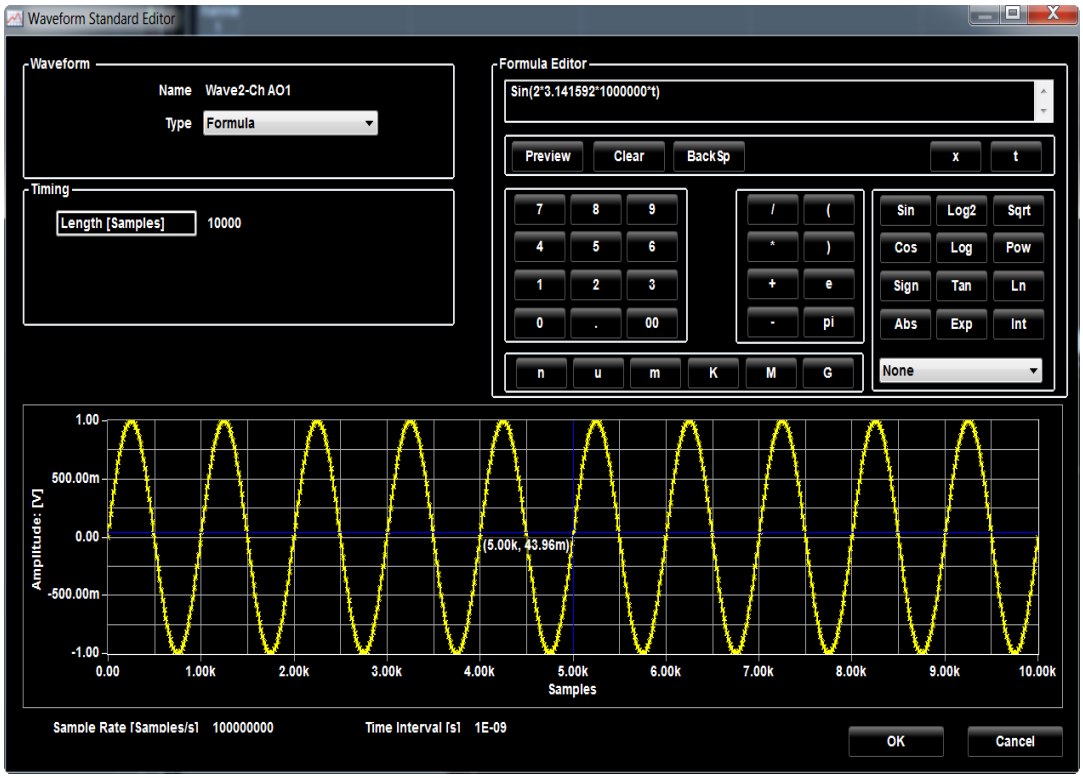


Figure 62 Sine waveform

Formula's General Format

$V*Sin(2*pi*t*F_s)$

Where

- F_s – Sine Wave frequency in Hertz.
- V – Signal amplitude in Volts peak.

Linear Amplitude Sweep of a Sine Wave

A Sine waveform is shown as follows. The actual formula used here is 0.2*E6*t*Sin(4*3.141592*E6*t).

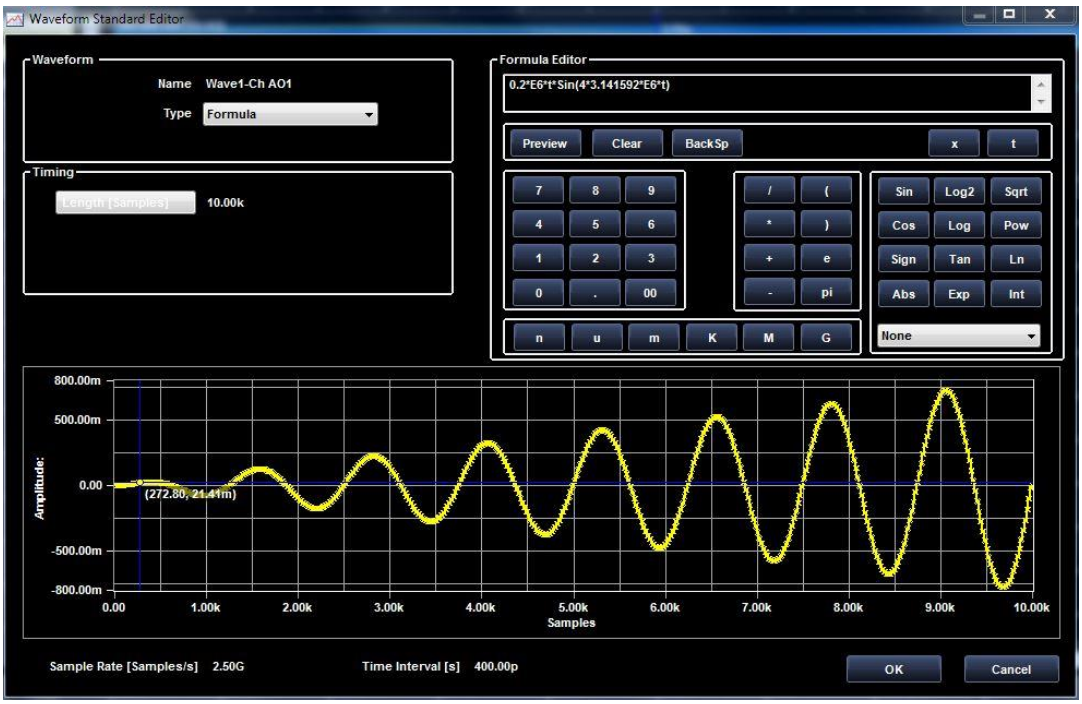


Figure 63 Sine waveform

Formula's General Format

$(A*t)\ *Sin(2*pi*t*\ F_s)$

Where

- F_s – Sine Wave frequency in Hertz.
- A – Slope of the ramp in Volts/second.

Frequency Modulation

A frequency modulation waveform is shown in the following figure. The actual formula used here is Sin(2*3.141592*2*E6*t+2*Cos(2*3.141592*0.4*E6*t)).

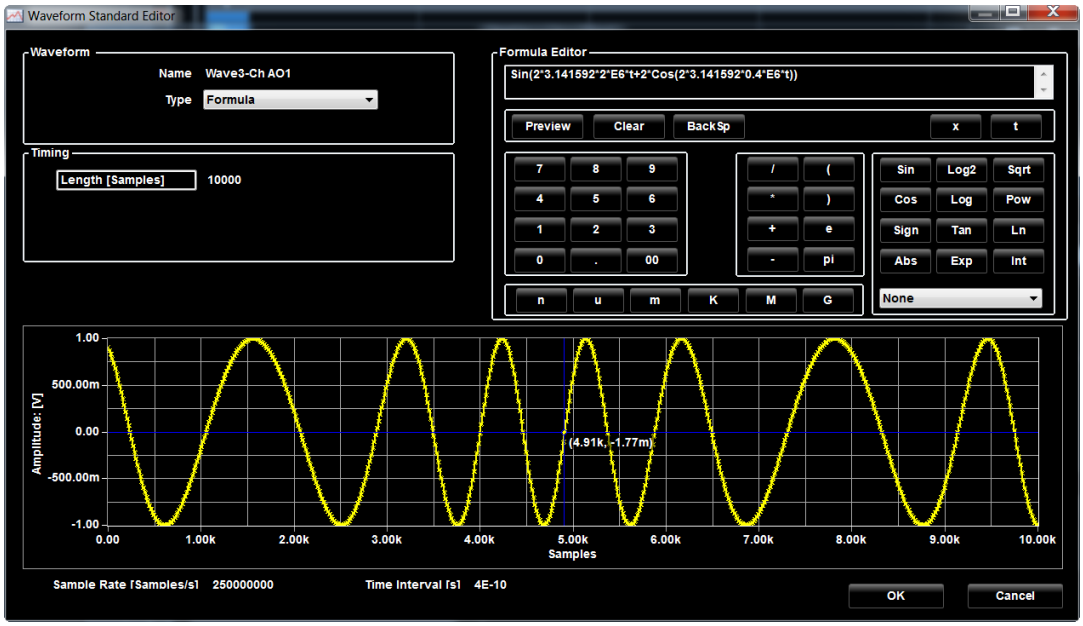


Figure 64 Frequency modulation waveform

Formula's General Format

$$V*Sin\ (2*pi*t*F_c+(F_D/F_M)*Cos(2*pi*t*F_M))$$

Where

- F_c – Carrier frequency in Hertz.
- F_D – Frequency deviation in Hertz.
- F_M – Modulation frequency in Hertz.
- V – Signal amplitude in Volts peak.

Phase Modulation

A phase modulation waveform is shown as follows. The actual formula used here is Sin(2*3.141592*2*2*E6*t+(3.141592*Sin(2*3.141592*0.4*E6*t))).

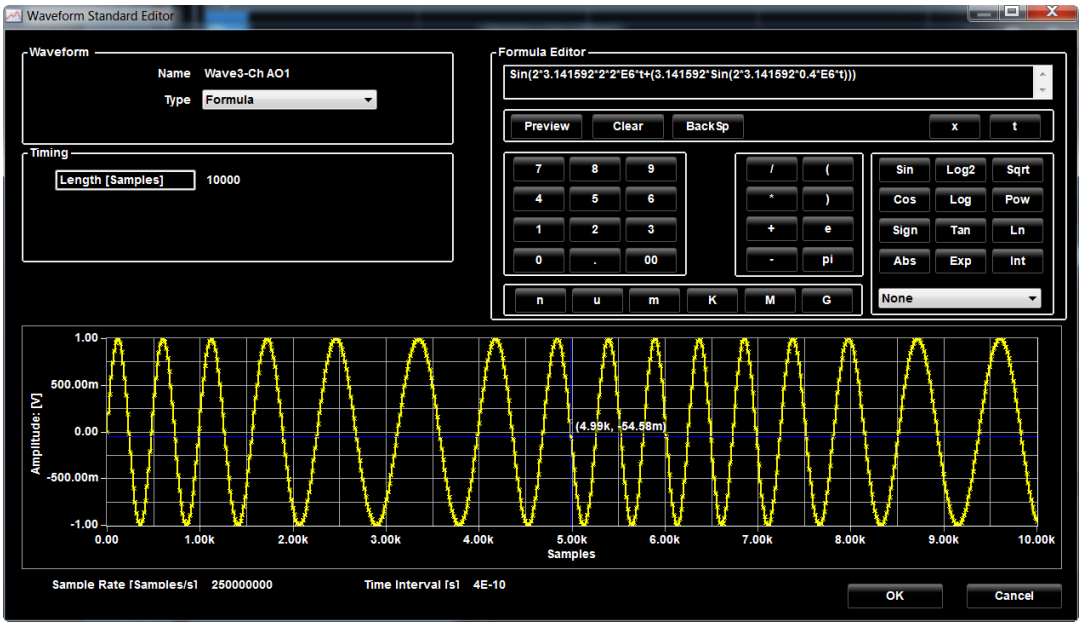


Figure 65 Phase modulation waveform

Formula's General Format

$$V*Sin((2*pi*t*F_C+ K*Sin(2*pi*t*F_M))$$

Where

F_C – Carrier frequency in Hertz.

K – Peak phase excursion in radians.

F_M – Modulation frequency in Hertz.

V – Signal amplitude in Volts peak.

Lin A linear frequency sweep waveform

Linear Frequency Sweep waveform is shown as follows. The actual formula used here is Sin(3.141592*(2*t*E6+((8*E6 -1*E6)/(10*E-6))*Pow(t,2))).

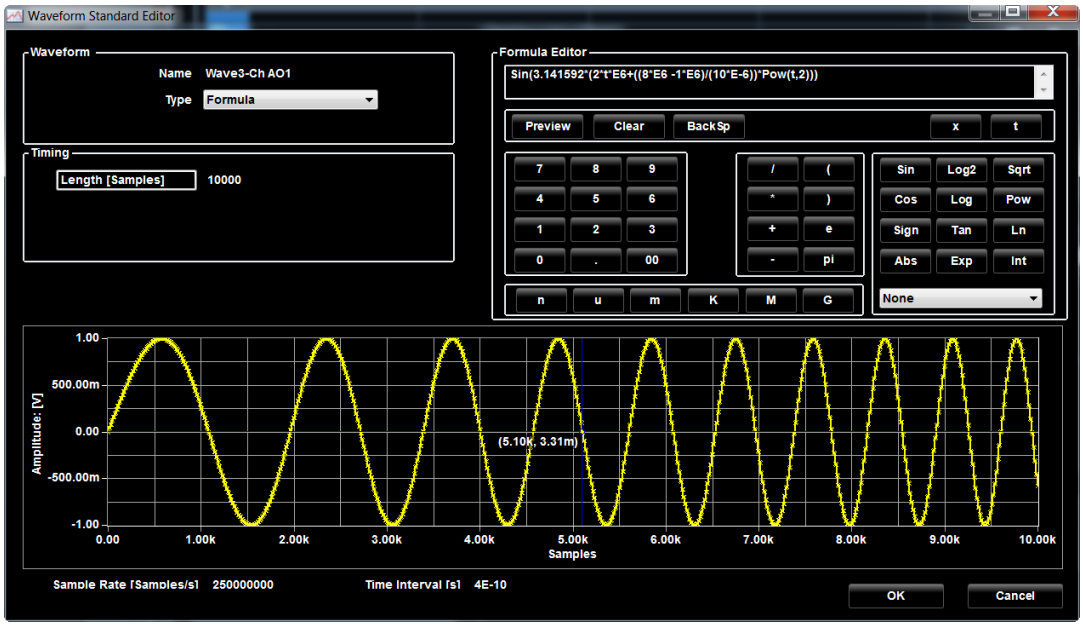


Figure 66 Linear frequency sweep waveform

Formula's General Format

$$V*Sin(pi*(2*t*F_S+((F_E-F_S)/T_S)*T\hat{}2))$$

Where

- F_S – Start frequency in Hertz.
- F_E – End frequency in Hertz.
- T_S – Sweep duration in seconds.
- V – Signal amplitude in Volts peak.

Gaussian Pulse

A Gaussian pulse waveform is shown as follows. The actual formula used here is Exp(-(8)*Pow(((t-2*E-6)/(E-6)),2).

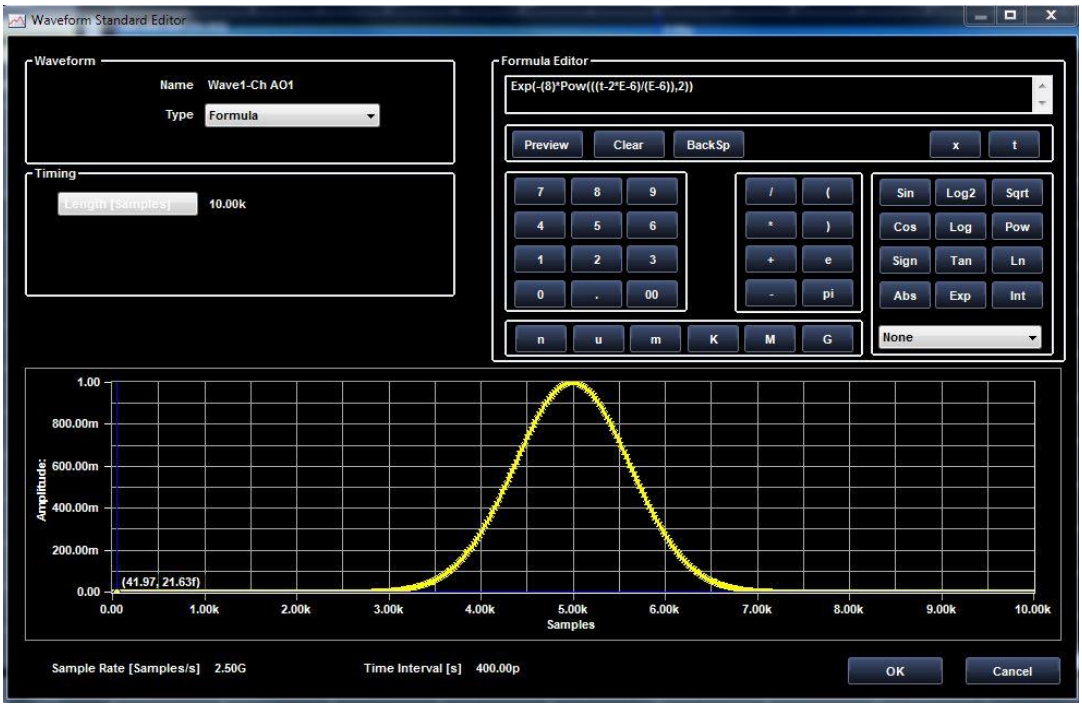


Figure 67 Gaussian pulse waveform

Formula's General Format

$V*Exp(-(1/2)*((T-T_M)/T_\sigma)^2$

Where

- T_M – Time location of the mean of the Gaussian pulse.
- T_σ – Half width point of Gaussian pulse corresponds to the standard deviation **σ**.
- V – Signal amplitude in Volts peak.

Amplitude Modulated Sine

An amplitude modulated sine waveform is shown as follows. The actual formula used here is 0.5*Sin(2*3.141592*2*E6*t)*(1+0.75*Cos(2*3.141592*0.2*E6*t)).

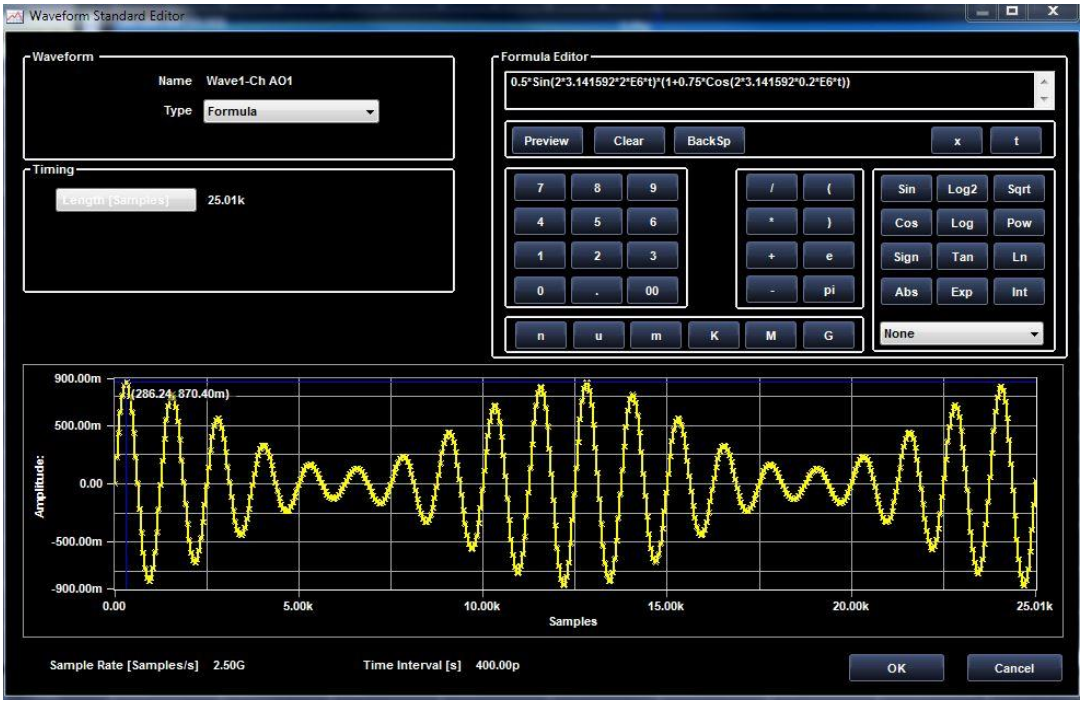


Figure 68 Amplitude modulated sine waveform

Formula's General Format

$$V*Sin(2*pi*t*F_s)*(1+K*Cos(2*pi*t*F_M))$$

Where

- F_s – Sine wave frequency in Hertz.
- F_M – Modulation frequency in Hertz.
- K – Modulation index, 0<K<1.
- V – Signal amplitude in Volts peak.

Full-Wave Rectified Sine

A full-wave rectified sine waveform is shown as follows. The actual formula used here is Abs(Sin(2*3.141592*E6*t)).

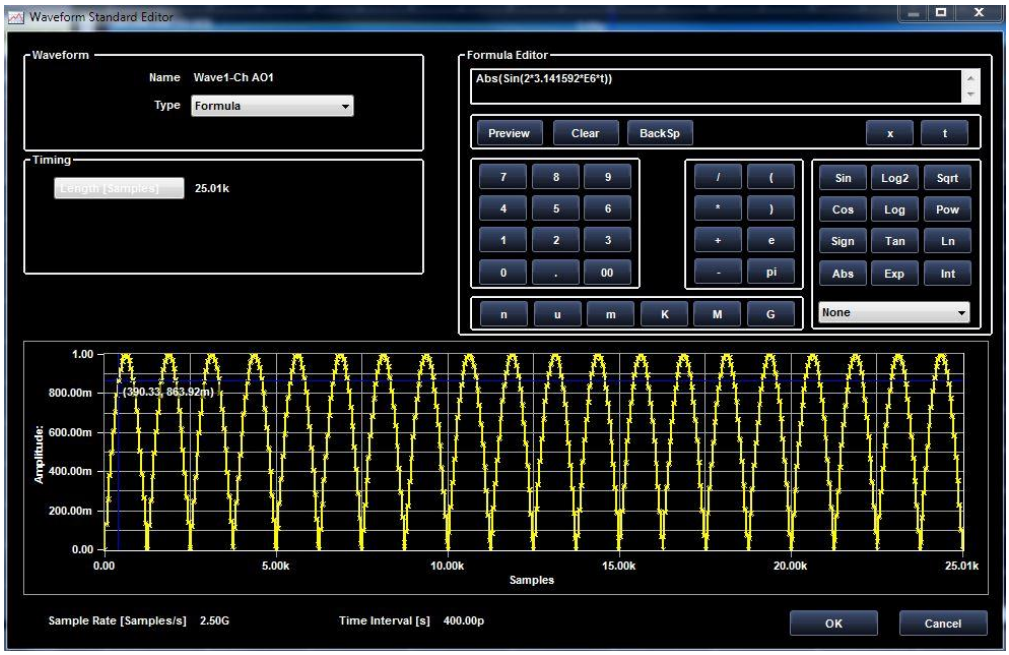


Figure 69 Full-wave rectified sine waveform

Formula's General Format

$$V*Abs(Sin(2*3.141592*F_s*t))$$

Where

- F_s – Sine wave frequency in Hertz.
- V – Signal amplitude in Volts peak.

Filter

This section allows you to select whether the digital filtering is to be applied to the entire waveform or to a limited part.
If you click on the waveform name in the *Analog Waveform Graph* Viewer/Editor, the selection range indicators will display the entire waveform limits.
If you need to apply the filter to a limited part, click and drag inside the graph area to create a rectangle delimiting the waveform section to be filtered. Their position is shown in the same Application Zone section.

FILTER Specifications

This section allows setting all the characteristics of the filter.

- **Digital Mode IIR (Infinite Impulse Filter)** - Bessel, Butterworth, Chebyshev, Inverse Chebyshev, and Elliptic.
- **Digital Mode FIR (Finite Impulse Filter)** - EquiRipple, Kaiser, and Windowed.
- **Type** - Low Pass, High Pass, Band Pass, Band Stop, and General.
- **Initial Condition** – Steady State Response means the output is in *steady-state*, since the input has fully engaged the filter.

Type, Topology and filter Order options depend on the specific filter characteristics.
Click **OK** button and the set filtering options are applied to the waveform. A preview of the filter/noise effects on the waveform is shown in the graph area.
You can remove a filter by clicking the current waveform and selecting the **None** option on the Setting tab.

Noise Settings Tab

This tab allows you to apply noise to the selected waveform.

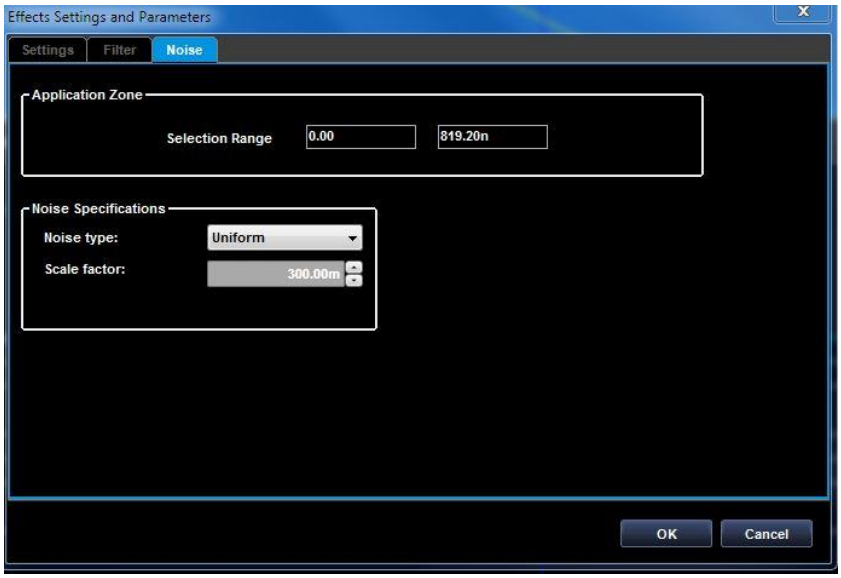


Figure 70 Noise setting tab

The Noise setting tab is divided into Application Zone, Noise Specification, and Parameters sections.

NOISE Application Zone

This section allows selecting whether noise is to be applied to the entire waveform or to a limited part of it.
If you click on the waveform name in the *Analog Waveform* Editor, the selection range indicators will display the entire waveform limits.
If you need to apply the noise to a limited part, click and drag inside the graph area to create a rectangle delimiting the waveform section. Their position is shown in the same Application Zone section.

Noise Specifications

This section allows setting all the noise characteristics.
Noise Type - Use this drop-down to select the noise type applied to the waveform. Options include **Gaussian**, **Uniform**, and **White**.
Depending on the selected noise type, specific parameters (**Standard Deviation** or **Amplitude**) are enabled together with the **Scale Factor** field, for increasing/decreasing noise intensity.
Click the **OK** button and the noise options set are applied to the waveform. A preview of the

noise effects on the waveform is shown in the graph area.

Remove a filter by clicking the current waveform and selecting **None** on the **Settings** Tab.

Import / export

Import of Analog Waveform

Data import functions allow you to use waveform data created outside the arbitrary waveform generator. You can import data to create a new waveform or to replace existing waveform data.

The arbitrary waveform generator supports the following file formats:

The supported file formats are:

- ISF
- TXT - Tab separated value file
- CSV – Comma separated value file
- WFM – Tekscope Series waveform(Depend on scope model)
- PAT – AWG Series pattern file(Depend on AWG model)
- TFW – AFG3000 Series waveform file format
- RFD – RFXpress file format
- MAT – Matlab .mat file format.

The Matlab file format needs to follow this format:

NumPoints *= 2400; %Waveform length*

t = (0:1:NumPoints-1)'; %Define t vector

*waveform = single(sin(2*pi*1/NumPoints*t)); %Create single sinewave*

%% Save Waveform

Waveform_Name_1 *= 'SINE'; %Name Waveform*

Waveform_Data_1 *= waveform; %Assign waveform data*

Waveform_Sampling_Rate_1 *= 2.4e9; %You can specify sample rate in S/s*

Waveform_Amplitude_1 *= 0.300; %and amplitude in V*

*save('SingleCycleSine', '*_1', '-v7.3'); %**Save all variables ending in _1 to .mat file***

How to Import an Analog or Mixed Waveform (TXT Files)

- Select the Analog Waveform associated to the Channel AO1 or Channel AO2 and press the Import  button.

The import form is open, as shown in the following figure.

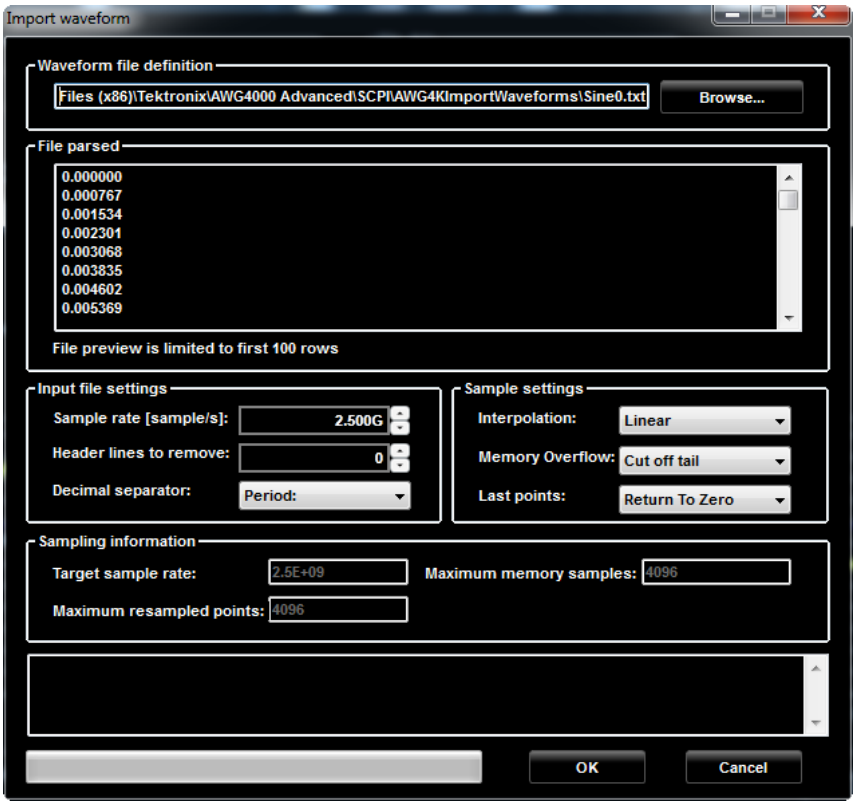


Figure 71 Import waveform

The following options are available on the Import Waveform form:

- Sample rate (sample/s): it is sampling rate of the points that will be imported
- Header lines to remove: the import file could have some header lines that needs to be removed before importing it; And you also can remove some points from the start of import file.
- Decimal separator: it can be or ,
- Interpolation: if the file sampling rate is different from the instrument sampling rate, you can choose the Interpolation method between Linear,Coerce and Polinomial.
- Memory Overflow: if the import file has more points than the current waveform, the overflow points will be cut off(“Cut off tail”)
- Last Points: Return to Zero means that the last points of the waveform will be zero in case the imported waveform is shorter than the actual total samples points. Otherwise if Last value is selected, it will maintain the last waveform sample.

If the selected TXT file has only one column, it will be interpreted as an analog signals and so only the analog part will be imported applying the options above. If the selected TXT file has more than one column, the following cases can happen: TXT Mixed Waveform File with Header (#AO1,#0,#1…..,#n)	The first column contains the analog data, while the other columns will be interpreted as the digital lines. Please note that the digital lines that will be imported depends on the number of lines present in the current project. **Example 1:** if the header is #AO1,#0,#1…..,#15, but the project has 32 digital lines, only the first 16 digital lines will be imported according to the header and the other ones will remain as is. **Example 2:** if you try to import a Single Sequencer header on a Multi-Sequencer project, the analog channel will be imported in the selected Multi-Sequencer channel (no matter if header is #AO1 or #AO2) and the digital channels will be imported on the selected Multi-Sequencer digital lines. Please note that in Multi-Sequencer, the digital lines will be always marked as #0,….,#15 or #0,….,#8, so digital lines with #16,……,#31 (Single Sequencer) should start from #0.
.TXT file: 2 columns without header	The first column will be imported as analog data, the second column will be imported as Marker data relative to the selected channel (AO1 or AO2)
TXT file: 3 columns without header	The first column will be imported as analog data, the second column will be imported as Marker data relative to the selected channel (AO1 or AO2). The third column will be ignored.
TXT file: more than 3 columns without header	The first column will be imported as Analog data, the other columns will be imported as digital lines. #Analog,#D0,#D1,#D2,#D3,#D4…….

IMPORT of Digital Waveforms

Select a digital line and press the Import  button.

The Import file format for digital waveforms is a comma separated value file where each column represent the samples of one digital channel.

The first row of the file is a header that represent the number of the digital channel (#0,#1,#2……..,#15) associated to the logical name.

Please not that if digital lines are not matchable in the header line, the digital bit can't

be imported successfully; you need to remove the digital header line numbers to import correctly the file.

The digital data import format is.txt ; the file is comma separated and it can have an header on the first line.
The Import Waveform window will open, please see as below.

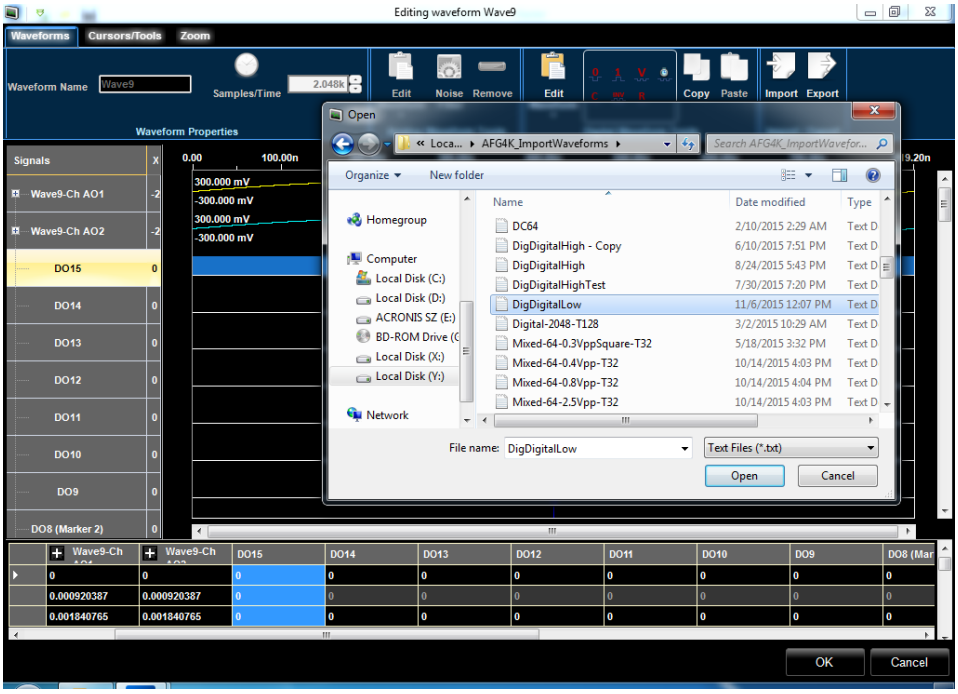


Figure 72 Import Waveform window

Export of Analog, Digital and Mixed Waveforms

Press the Export button to open *the Export Waveform* window:

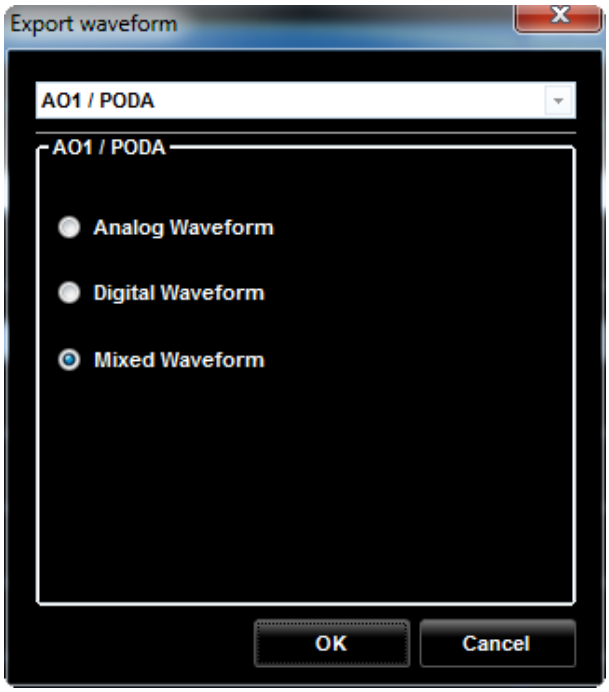


Figure 73 Export waveform

This window lets you to export the Analog, Digital or Mixed Waveform relative to the channel 1 (A01/POD A) or channel 2 (A02/POD B).

When you select “Analog Waveform”, the Export file format for analog waveforms is a comma separated value file (only one column) where the column represent the samples of the selected analog channel.
The first two rows of the exported file is a header that represent the sample rate and the number of samples (# Sample rate: 2500000000 # Samples: 2048).
The exported values representation is double.

If you select “Digital Waveform”, the Export file format for digital waveforms is a comma separated value file where each column represent the samples of one digital channel.
The first row of the file is a header that represent the number of the digital channel (#0,#1,#2……..,#15) associated to the logical name.

Example 1(POD A 16 digital lines):

#0,#1,#2,#3,#4,#5,#6,#7,#8,#9,#10,#11,#12,#13,#14,#15

0,0,0,0,0,1,0,0,0,0,0,0,0,0,0,0

0,0,0,0,0,1,0,0,0,0,0,0,0,0,0,0

. . . . . .

. . . . . .

0,0,0,0,0,1,0,0,0,0,0,0,0,0,0,0

0,0,0,0,0,1,0,0,0,0,0,0,0,0,0,0

Example 2 (POD B 16 digital lines):

#16,#17,#18,#19,#20,#21,#22,#23,#24,#25,#26,#27,#28,#29,#30,#31

0,0,0,0,0,1,0,0,0,0,0,0,0,0,0,0

0,0,0,0,0,1,0,0,0,0,0,0,0,0,0,0

. . . . . .

. . . . . .

0,0,0,0,0,1,0,0,0,0,0,0,0,0,0,0

0,0,0,0,0,1,0,0,0,0,0,0,0,0,0,0

If you select “Mixed Waveform”, the Export file format for mixed waveforms is a comma separated value file where the first column represents the analog data and the other columns represent each the samples of one digital channel.
The first row of the file is a header that represent the analog waveform and number of the digital channel associated to the logical name.

SINGLE SEQUENCER HEADER format:

- AO1/POD A – 8 bit mode -
  #AO1,#0,#1,#2,#3,#4,#5,#6,#7
- AO1/POD A – 16 bit mode -
  #AO1,#0,#1,#2,#3,#4,#5,#6,#7,#8,#9,#10,#11,#12,#13,#14,#15
- AO2/POD B – 8 bit mode -
  #AO2,#16,#17,#18,#19,#20,#21,#22,#23
- AO2/POD B – 16 bit mode -
  #AO2,#16,#17,#18,#19,#20,#21,#22,#23,#24,#25,#26,#27,#28,#29,#30,#31

MULTI SEQUENCER HEADER format:

- AO1/POD A – 8 bit mode -

#AO1,#0,#1,#2,#3,#4,#5,#6,#7

- AO1/POD A – 16 bit mode -
  #AO1,#0,#1,#2,#3,#4,#5,#6,#7,#8,#9,#10,#11,#12,#13,#14,#15
- AO2/POD B – 8 bit mode -
  #AO2,#0,#1,#2,#3,#4,#5,#6,#7
- AO2/POD B – 16 bit mode -
  #AO2,#0,#1,#2,#3,#4,#5,#6,#7,#8,#9,#10,#11,#12,#13,#14,#15

Please note that if you will try to import single sequencer mixed/digital waveforms in multi sequencer workspace, some mismatches in digital lines can happens.
In particular if you will try to import #AO2,#16,……,#31 (Single Sequencer) in Multi Sequencer project, you have to cut the header to import also the digital lines, otherwise they will be skipped.
Please note that the import/export functions can become slow with large amount of data to import or export.

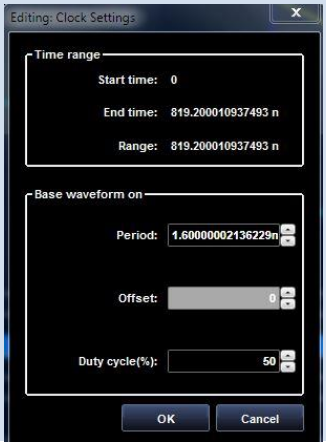
Digital Waveform Graph Tools

This toolbar contains several commands for use on digital waveforms as follows:

	**Signal/bus to 0.**
	Signal/bus to 1.
	Signal/bus to Arbitrary Value. Arbitrary Value allows overwriting a node value over the selected waveform, waveform interval, or across one or more nodes or groups. Overwrite a node value using the following steps: 1. Select a node or a bus and click the **Value** button on the Digital Editor toolbar. The **Arbitrary Value** dialog box appears.  2. In the Radix button, select the radix type. 3. Specify the new value you want overwritten in the Numeric or named value box. 4. Click OK.

Clock Editor for selected signal.
The Clock feature can be used to automatically generate the clock wave, rather than drawing each clock triggering pulse.

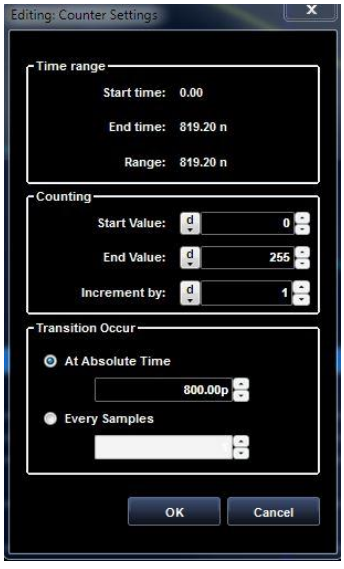
The start and end time of a clock signal can also be selected.



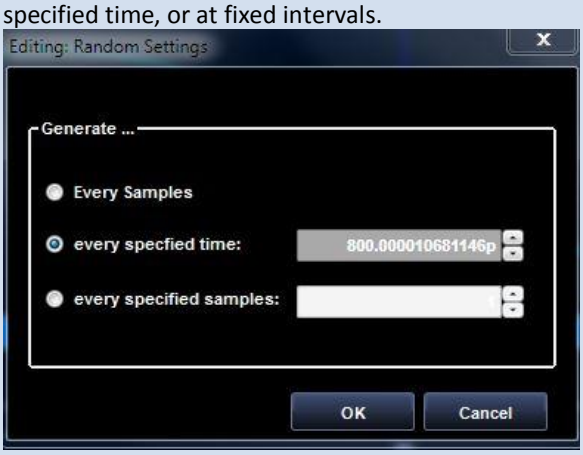
Counter Editor for selected bus.
The counter editor applies a count value to a bus which increments the value of the bus by a specified time interval.

Instead of manually editing the values for each node, the Counter editor automatically creates the counting values for buses.

You can also specify a starting value for a bus and the time interval for increments.



Random Value for signal/bus.
Random Value allows generating random node values over the selected waveform, waveform interval, or across one or more

	nodes or groups. Random node values can be generated for each grid interval, a specified time, or at fixed intervals. 
	Invert signal/bus value.
	Copy Waveform. Select the entire waveform clicking on the signal/bus name on the left column or select a portion of it with mouse selection. Press the **Copy Waveform** button to copy the waveform.
	Paste Waveform. Paste the copied waveform into a selected area of the graph (mouse selection) or from the start of another waveform.

Mixed Signal Waveform Editor

The **Mixed Signal Waveform Editor** screen is used to create or edit *analog* and *digital* waveforms in a graphic or tabular format.

Single signals are visualized as analog or digital signals, while grouped signals are represented as buses.

Analog Waveform Editor

Arbitrary edited waveforms can be generated as they have been set.

Think of a waveform as a list of **Segments**, where each segment can contain one or more **Components**, all of the same length, combined by means of the elementary Add, Subtract, Multiply operations.

Each waveform may be constituted by an arbitrary number of segments and each segment can have its own length.
In Arbitrary mode each waveform must be constituted by a multiple number of samples of 64 for <320 samples length or in multiple of 16 for >= 320 samples.

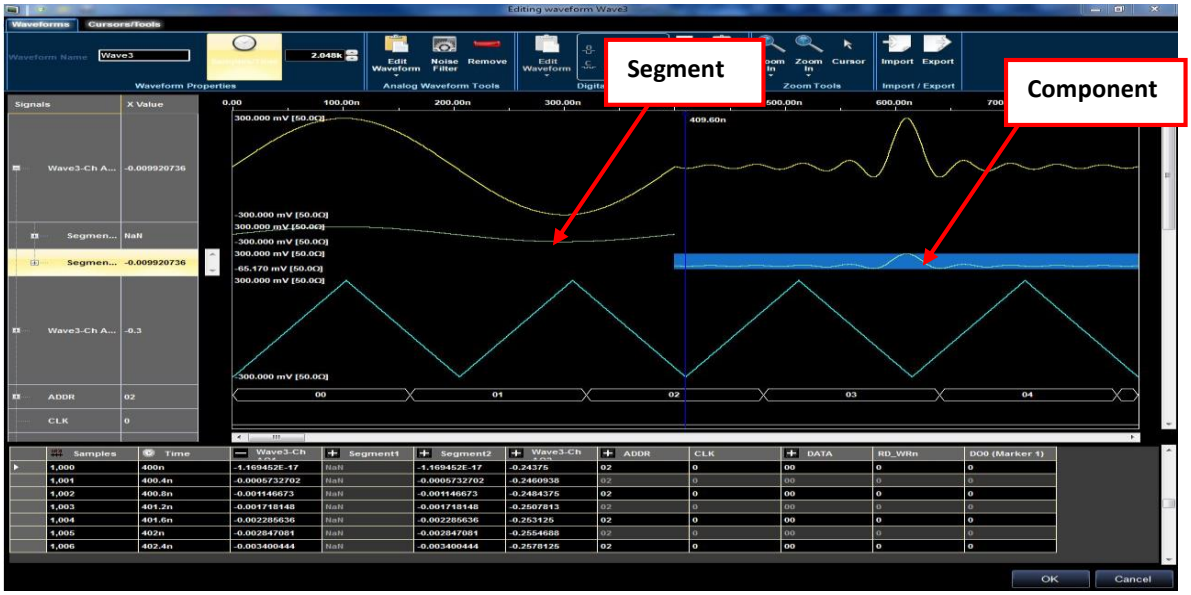


Figure 74 Waveform segments

The waveform in the picture above is the composition of two segments. Each segment is made of one component.

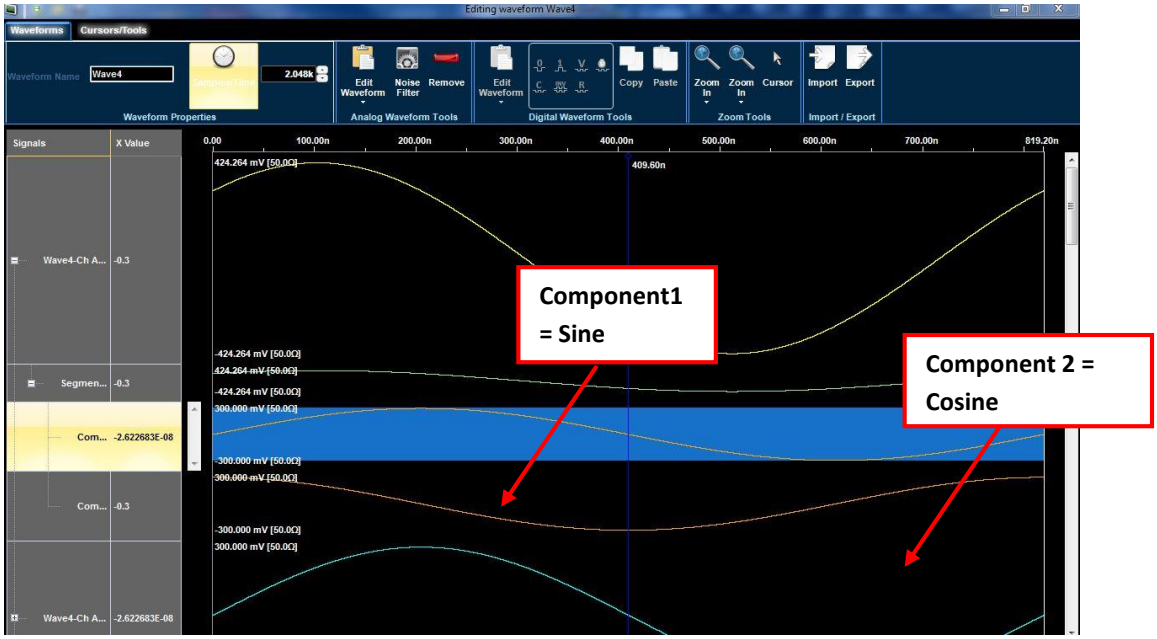


Figure 75 Waveform components

The waveform in the picture above is made of one segment. The segment is the composition of two Components multiplied together: Segment1 = Component1 * Component2.

Common operations can be directly performed on waveforms, segments and components (**Selection**, **Left Click**, **Right Click**). Drag and Drop operation between analog waveforms is not allowed.

LEFT CLICK AND SELECTION

- Left Click on the analog *Signals cell* to select the entire waveform that will be enlightened in blue.
- Click and drag inside the graph area to create a rectangle delimiting the waveform section. You can add *Effects* on the selection.
- Left Click on the Signals cell tree item to open/close the Segments of the waveform.
- Left Click on the Segment tree item to open/close the Components of the single segment.
- Resize the signal amplitude by dragging the line between a signal name cell.
- To the right of every analog signal, a number indicates the value the signal at the time position of the master cursor.

RIGHT CLICK

A Right Click on a Waveform tree item activates a pop-up menu, with functions depending on a Waveform or Segment selection as follows.

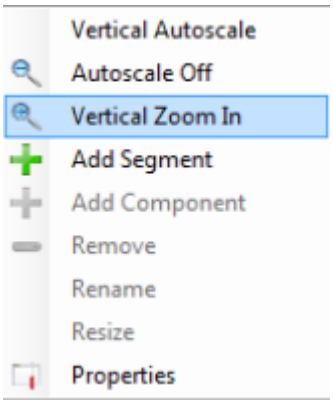


Figure 76 Right-click pop-up menu

- **Vertical Autoscale On** – Enable the auto scale function;
- **Autoscale Off** – Select this option to disable the waveform auto scale. it will appear a vertical scrollbar on the right side of the Signals cell you can use to scroll up and down the selected waveform.
- **Vertical Zoom In** – Auto zoom in function.
- **Add Segment** – To add a segment you have to increment the waveform length or resize/delete the existing segments.
  For example if you have a 10k samples waveform made of one segment and you need to add a second segment to it, you can perform the following operations:
  - Right click on the existing segment to activate the pop-up menu and select *Property*. The Segment1 Property window is shown: resize the Segment1 length

from 10K to 5K.

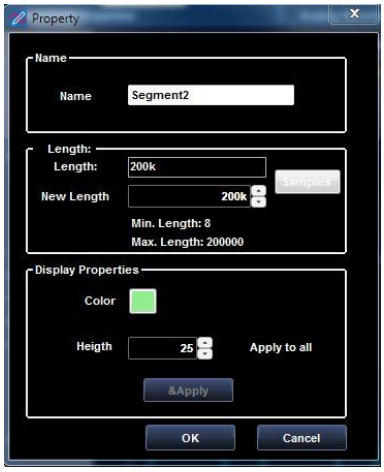


Figure 77 Property window

- Right click on the Waveform or on the existing Segment to activate the pop-up menu and select Add Segment.
  The Segment2 Property window is shown, insert the new segment length and press OK to add it to the waveform.
  PLEASE NOTE THE FOLLOWING:
  If you select the *Add Segment* option from the Segment pop-up menu, the new one will be added at the end; if you select the *Add Segment* option from the Waveform pop-up menu, the new one will be added at the beginning.

- **Add Component –** Add a Component at the selected Segment.
- **Remove –** Remove the selected Segment or the Component.
- **Rename –** Rename the selected Segment or the Component.
- **Resize –** Resize the selected Component.

To create an Advanced Analog Waveform:

- Follow the steps 1-6 of the Arbitrary Mode Single Sequencer Setup Example.
- The **New Waveform** window is shown. Type the name of the waveform “Wave1” and choose 10k for the samples length of the waveform. Click **OK** to confirm.

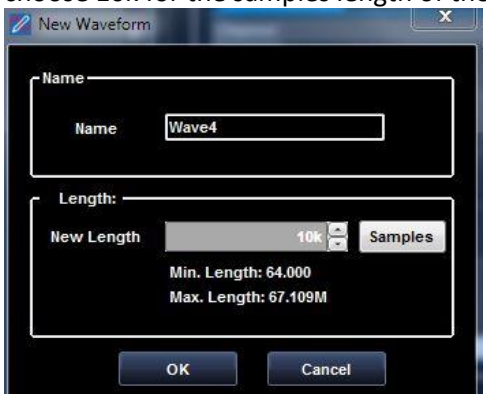


Figure 78 New waveform window

- The **Editing Waveform Window** is shown. Right click on Segment1 of the Wave1-0 to open the pop-up menu.

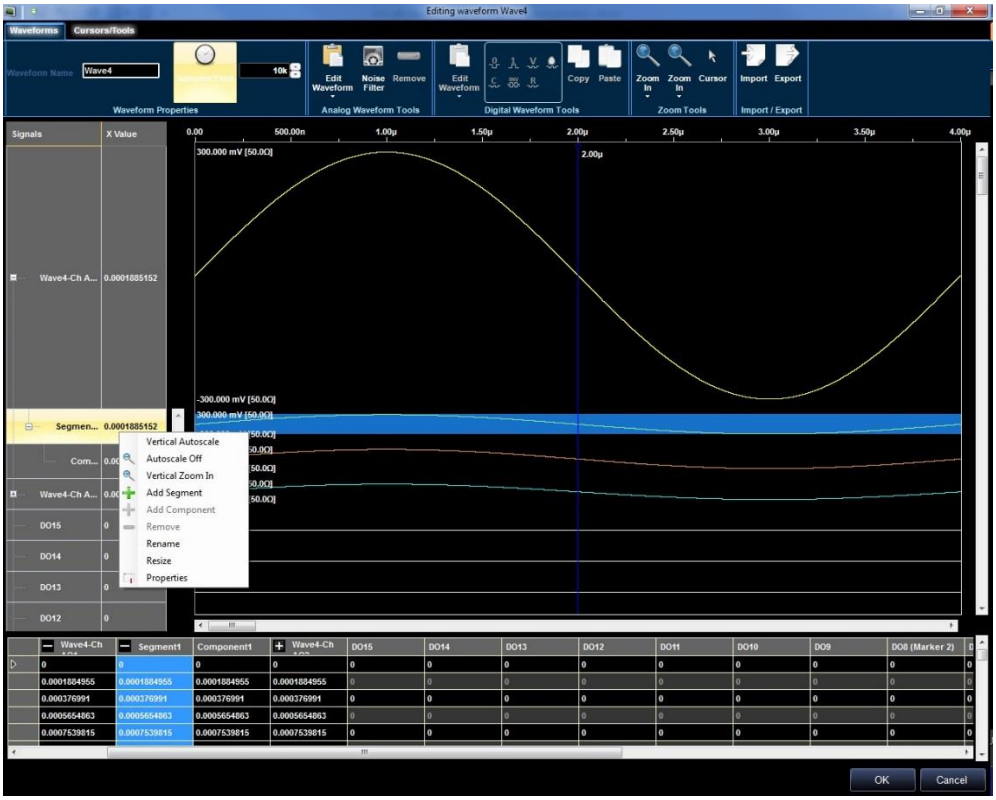


Figure 79 Editing waveform window

- Select **Properties** on the pop-up menu.
- Change the segment length: insert 2.5k in the New Length field.

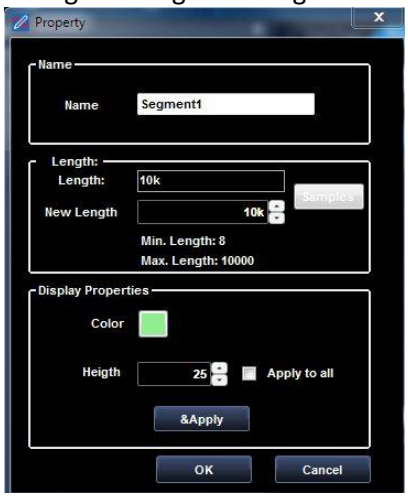


Figure 80 Property window

- The Wave1-0 and the Segment1 will be resampled to the new length.

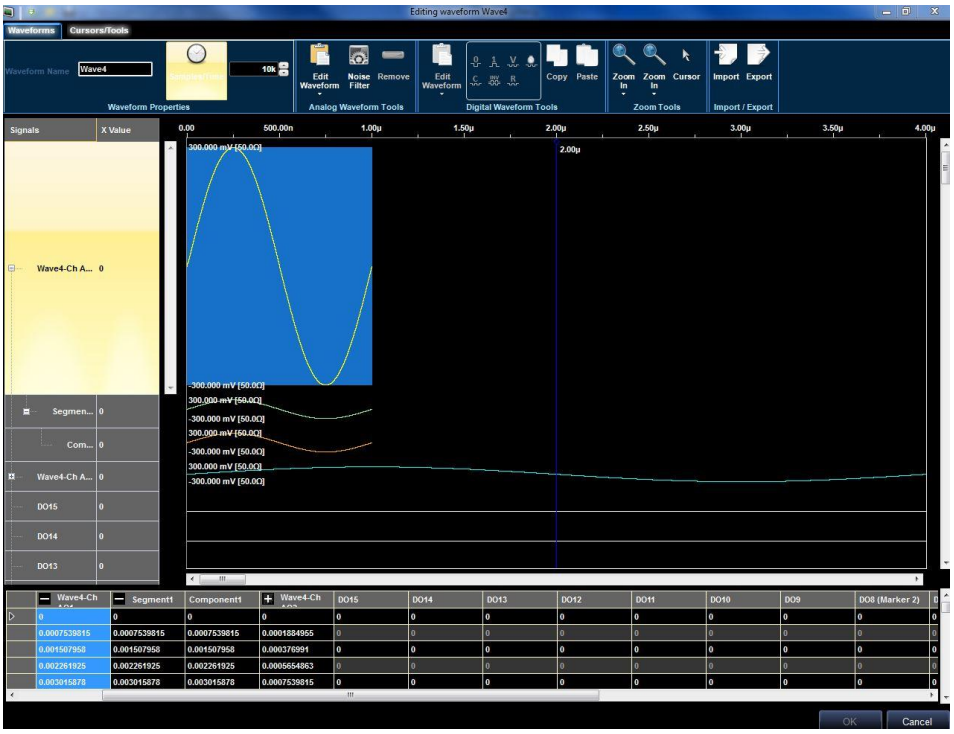


Figure 81 Waveform editing

- Select the Segment1 and click the **Edit** button on the toolbar, the Standard Waveform Editor window is sho

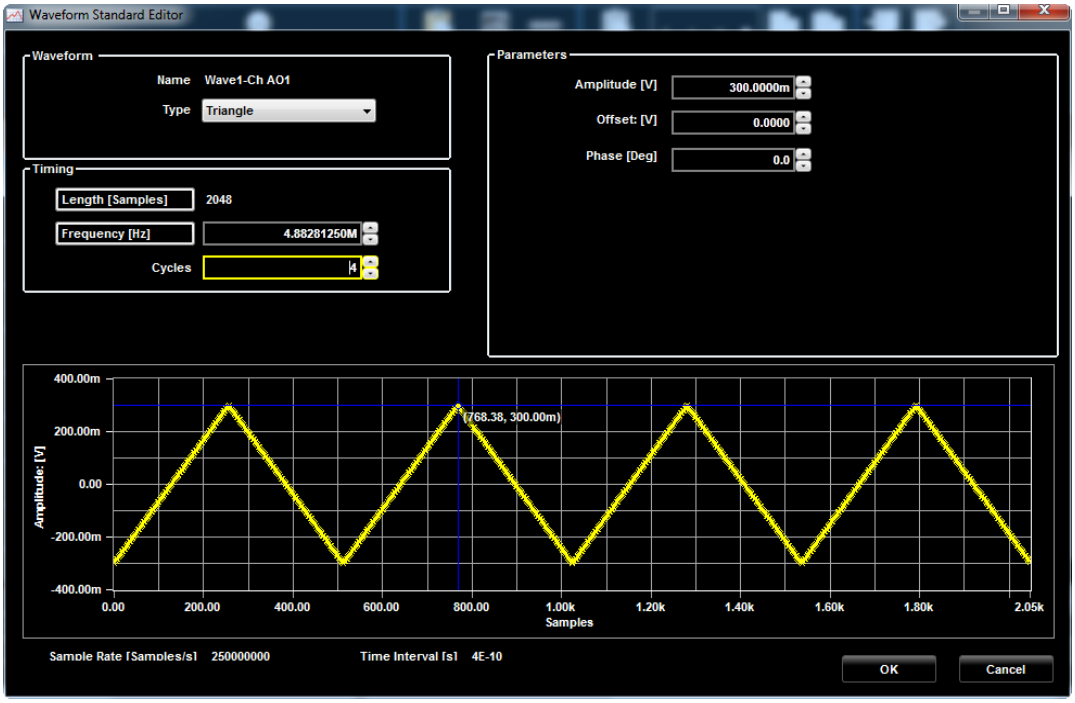


Figure 82 Standard Waveform Editor

- Select Triangle as waveform Type and 4 as number of Cycles.
- Click **OK**.
- Right click on the Segment1 of Wave 4-0 to open the pop-up menu and select **Add Segment**.

Figure 83 Add segment

- The Property window is shown. Select 2.5k as Segment2 length and change the color.
- Click OK.
- Right click on the Segment2 of Wave4-0 to open the pop-up menu and select Add Segment.
- The Property window is shown.
- Select 5k as Segment3 length and change the color.
- Click OK.
- Select the Segment3 and click the Edit button on the toolbar.

- The Waveform Standard Editor is shown. Select Sinc as waveform Type. Click OK.

Figure 84 Waveform standard editor

- Right click on the Segment3 of Wave4-0 to open the pop-up menu and select **Add Segment**.
- Right click on the Component1/Segment2 of Wave4-0 to open the pop-up menu and select **Add Component**

Figure 85 Add component

- Select the Component 2 and press the Edit Waveform button. The Standard Waveform Editor window is shown.

Figure 86 Editing waveform

- Select Cosine as waveform Type and Multiply as Operation.
- Press OK.
- The **Advanced Waveform** is ready to use and can be inserted in the Sequencer.

Digital Waveform Editor

AWG4162 can be configured to work as a powerful Digital Pattern Generator.
In this working mode AWG4162 provides the capability to emulate standard serial or parallel bus transitions or custom digital interfaces for system debugging and characterization.

Pay attention to the following:

- The Max vector memory depth is 32Mpoints /Ch.
- In Arbitrary Mode the maximum update rate is 1.25GS/s on 16 channels workspace and 625 MS/s on 32 channels workspace.
- Alignment between Digital and Analog Channels: by using the de skew controls on the *Settings Tab,* it is possible to realign the analog channels with a resolution of about 10ps and the digital channels with a resolution of about 78ps.

Single signals are visualized as digital signals, while grouped signals are represented as buses.

When you create a New Mixed or Digital Waveform you have single digital signals at your disposal; the number and the names of those signals depend on the initial Project setup.
You can change the name of the signals and create / rename buses pressing the **Group Digital Signals** button on the Digital Channels Settings Tab.

Figure 87 Digital waveform editor
Common operations can be directly performed on digital single signals or buses (**Selection**, **Left**

Click, **Right Click**). Drag and Drop operation between digital waveforms is not allowed.

LEFT CLICK AND SELECTION

- Left Click on the *Signals cell* to select the entire digital single signal or bus that will be enlightened in blue.
- Click and drag inside the graph area to create a rectangle delimiting the digital waveform section. You can apply one of the operations described in the Digital Waveform toolbar on the current selection or on the entire waveform.
- Left Click on the Signals cell tree item to open/close the bus.
- Resize the digital signal amplitude by dragging the line between a signal name cell.
- To the right of every analog signal, a number indicates the value the signal or bus at the time position of the master cursor.

RIGHT CLICK

A Right Click on a Digital Waveform tree item activates a pop-up menu, with functions depending on a Waveform or Segment selection as follows.

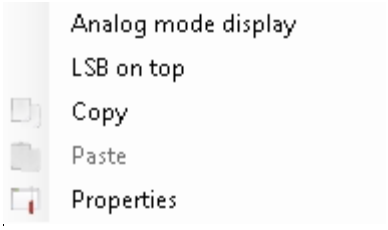


Figure 88 Pop-up menu

- **Analog mode display** - This option (available only for the buses) will represent a bus as an analog waveform. This is useful for example if an ADC or a DAC has to be tested.
- **Properties-** Open the digital waveform Property Window. You can use it to change signals/buses colors, plot height and the display format of the bus value.

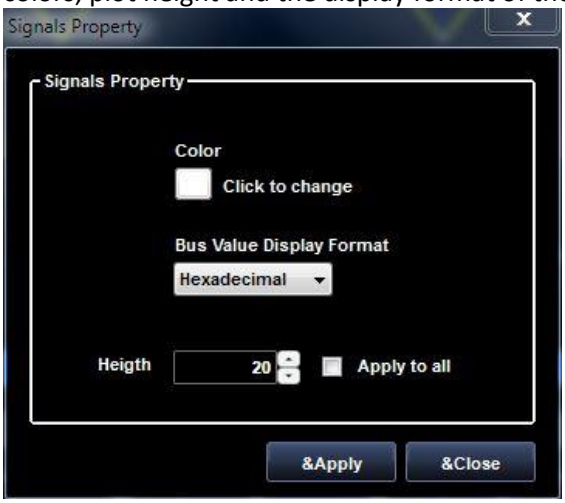


Figure 89 Signals property

- **LSB on Top** - Bus values are calculated with MSB (Most Significant Bit on Top) by default. Select LSB to have the Least Significant Bit on Top instead.
- **Copy -** Copy Waveform. Select the entire waveform clicking on the signal/bus name on the left column or select a portion of it with mouse selection.
- **Paste -** Paste Waveform. Paste the copied waveform into a selected area of the graph

(mouse selection) or from the start of another waveform.

Data Editor

The Data Editor can be used to edit analog/digital signals and bus values in tabular format. Data is visualized numerically in columns when using the Data Editor.
The Data Editor contains the following two additional columns:

- **Samples** - Contains the progressive number of the samples generated.
- **Time** - Contains the progressive absolute time of every sample.

BUS

A bus node is indicated by the **Expandable Bus** icon shown to the left of the bus name (in the columns on the right of the screen).
Expanding an *analog waveform*, the Segments are shown. Expanding a Segment, the Components are shown.
Expanding a *digital waveform*, the single digital lines are shown.

Double click the bus name to open the bus.

Once a bus node is opened, the **Expanded Bus** icon is shown instead of the **Expandable Bus** icon. Double click the bus name again to close the bus.

Common operations can be directly performed on table values (**Selection**, **Left Click**, **Right Click**). Drag and Drop operation between table columns is not allowed.

LEFT CLICK AND SELECTION

- Left Click on the signal name to select the entire analog/digital single signal or bus that will be enlightened in blue.
- Click and drag inside the table cells to create a rectangle delimiting the analog/digital waveform section.
  You can apply one of the operations described in the Analog/Digital Waveform toolbar on the current selection or on the entire waveform.
- Keep pressed the left mouse button on single cell to edit the value.
- Resize the column amplitude by dragging the line between a signal name.

RIGHT CLICK

A Right Click on a table cell activates a pop-up menu, with functions depending on a Analog/Digital Waveform selection as follows.

Figure 90 Pop-up menu by right-click

You can change table values by pressing Digital Editor Waveform toolbar buttons or right clicking on a table item to open the context menu.

To Export data, select the **Export data…** right click menu item or click on the export menu icon  .

The digital data will be exported into a .txt file; the file is comma separated with an header on the first line.

To Import data, select the **Import data…** right click menu item or click on the export menu icon  .

The digital data import format is .txt ; the file is comma separated with an header on the first line.

Please pay attention to the following:

- The disabled cells in the Digital single signals/buses are not editable.
- The changes made on the table cells are also shown on the Mixed Waveform Editor directly above it.

You can change table values by pressing Analog Editor Waveform toolbar buttons, changing the values on the data grid. As shown in the following figure, right-click on a table item to open the context menu using the *Select Waveform …* item; if you select some values on the data grid and use the *Edit…* option, the Waveform Standard Editor window will open and you can change the values directly on the table.

Effects… menu item will open the Filter & Noise window and *Search…* item will open the "Search Settings" window.

To Export data, select the **Export data…** right click menu item or click on the export menu icon  .

Figure 91 Right-click on an item

The analog data will be exported into a .txt file; the file is comma separated with an header on the first line that represents the Sample rate and the number of samples.

 # Sample rate: 2.5E+09

 # Samples: 16000000

To Import data, select the **Import data…** right click menu item or click on the import menu icon  .

Sequence Window

It is sometimes necessary to create long waveform files to fully implement a DUT test. Where portions of the waveforms are repeated, a waveform sequencing function can save you a lot of memory-intensive waveform programming.

The Sequencer allows you to select which edited waveforms are generated, their sequence, number of repetitions and the generation conditions.

Sequencer is used for mainly the following two purposes:

- Output longer waveform than hardware memory.
- Change the output waveform quickly following conditions.

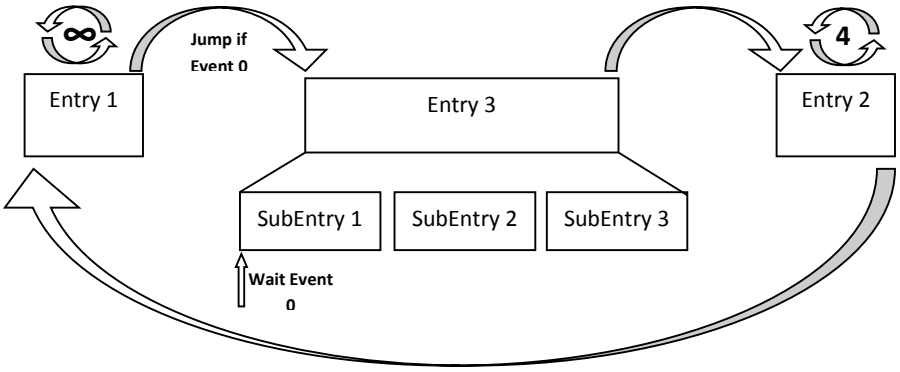


Figure 92 Sequence

A sequence is made of multiple entries; each **entry** contains analog and digital signals properly formatted.
It is possible to create a subset of entries identifying a Subsequence that can be placed into the Sequencer entry by mouse drag & drop. In the picture above the entry 3 of the sequencer is replaced by a Sub-Sequence with 3 entries.

Pay attention to the following:

- The digital waveform length **must be 1/2 or 1/4** of the analog waveform length in the same sequencer entry.
- A subsequence can be placed into the sequencer in the same way as a waveform by dragging and dropping the SubSequence from the Subsequences TAB into the sequencer.
- The Subsequence entry does not have the Wait Event option.
- If you select the Jump Event option in the main subsequence entry and the first element of the subsequence has Wait Event or Jump If Event selection, the main sequence has the higher priority on all the entries of the Subsequence.
- To create a new SubSequence press the “New Subsequence” button located in the toolbar aside the SubSequences tab.
- A Subsequence can be edited exactly in the same way as the main sequence.
- It is not possible to have a Subsequence with just one entry.
- The maximum number of entries in the sequencer is 16384.

When the waveform or the subsequence is dropped into the sequencer a dialog box, as shown in the following. You can specify the properties of the new sequencer entry that is going to be created.

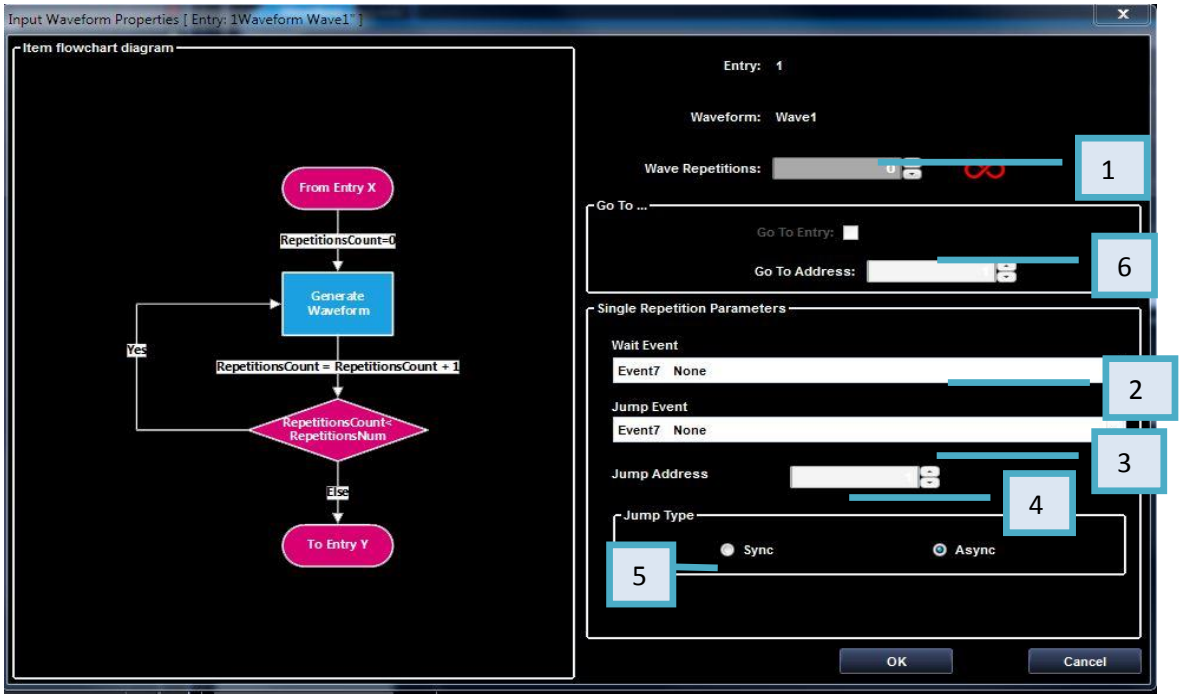


Figure 93 Input Waveform Properties

Input Waveform Properties

1. An entry can be repeated if the **Wave Repetitions** is specified.
   Press the ∞ button for *infinite* repetitions.
2. If an event is selected in the **Wait Event** drop-down list (Event0..Event7 are available), the sequencer will wait for the Event N before generating a waveform.
3. **Jump Event** changes the sequencing of the waveform by the event specified in the *Jump Event* drop-down list. Event0...Event7 are available.
   If the Jump Event has been set inside a *Subsequence entry*, when the event occurs the Sequencer will exit from the Subsequence and it will continue the entry generation in the main sequence.
4. **Jump Address** sets the number of the entry at which the Sequencer will jump when the event occurs.
5. **Jump Type:** if Sync is selected and the event occurs, the sequencer waits for the end of the current waveform before executing the Jump.
   If Async is selected, the sequencer executes the Jump as soon as the event occurs.
6. **Go to Address:** if Jump Event is not selected (Event7 None), the sequencer will execute the next entry after the completion of the current one.
   You can change the execution order pressing the checkbox to activate the Go to Address control and type the next entry address.

Pay attention to the following:

- The infinite repetitions have the priority over the Go To Address.
- It is possible to set Wait Event AND Jump If Event conditions in the same entry; the Jump If condition is evaluated **after** that the Wait Event has occurred.
- The Jump instruction cannot be evaluated during the transition between the current and the next entry.
- The header of the entry contains the settings of the Input Waveform Properties window.

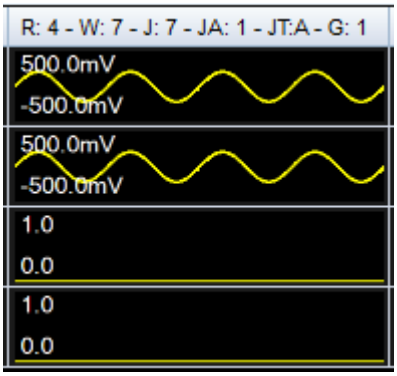


Figure 94 Waveform settings

R :4 = 4 repetitions
W :7 = Wait for Event 7 (None)
J: 7 = Jump if Event 7 (None)
JA:1 = Jump Address 1
JT:A = Jump Type Asynchronous
G:1 = Go To Address 1

Main Sequencer window

When Sequence is selected in the Run Mode, multiple waveforms can be output in the order specified in the Sequence Window. The Sequence Window displays the analog and digital waveforms that will be generated.

Figure 95 Single main sequencer

Single Sequencer workspace has one sequencer called Main Sequencer that controls all analog/digital channels waveform generation.

Multi Sequencer workspace has two Sequence Windows: the first one is relative to the Analog Output 1, the second one is relative to the Analog Output 2. Each one works independently by the other and has its own list of available waveforms in the Waveform TAB.

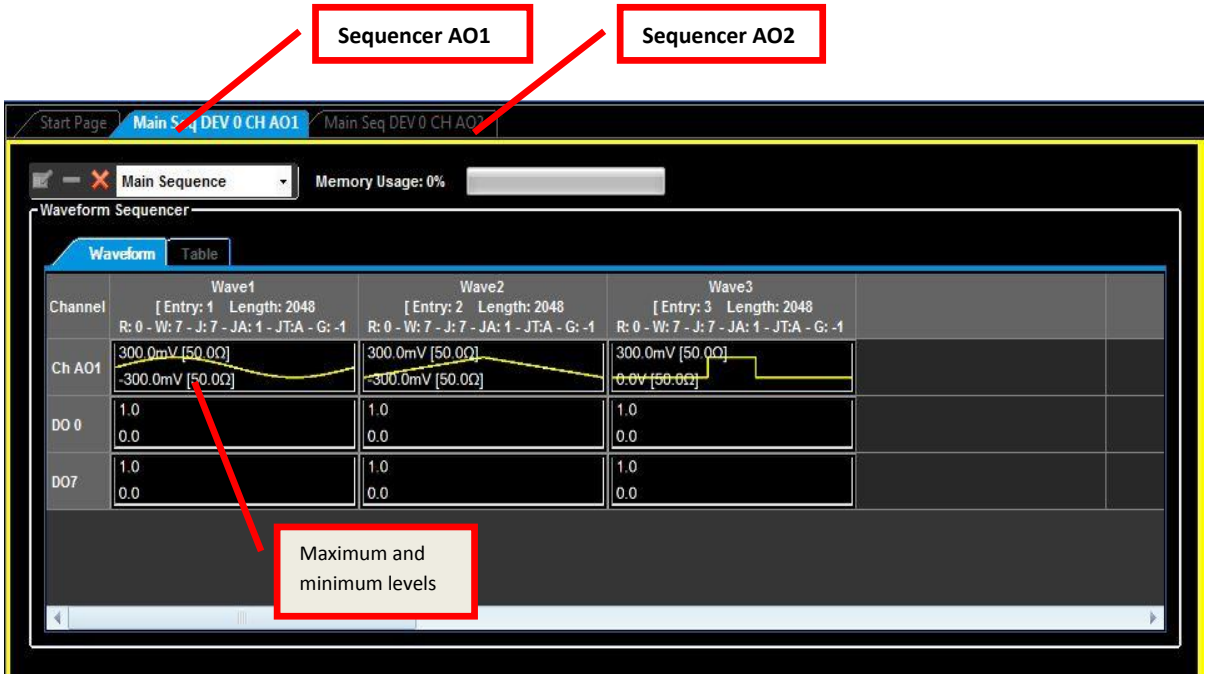


Figure 96 Multi main sequencer

Pay attention to the following:

1. If you click on a cell in the Sequence window, the selected waveform is displayed in the Waveform View window. The background color of the selected cell changes in blue.
2. If you click on the header of the cell in the Sequence window, the entry is selected and all the waveforms inside it are displayed in the Waveform View window. The background color of the selected entry changes in blue.
3. Drag a waveform from the Waveform List window and drop it to a cell of the Sequence window to insert it in the sequencer.
4. To edit a waveform: after selecting a cell in the Sequence window, right click to open the pop-up menu and select *Edit Waveform* to open the *Editing Waveform Window*.
5. When you insert a waveform in a sequencer cell, its maximum and minimum levels are displayed on the left margin of the cell.
6. When Continuous, Triggered, or Gated is selected in the Run mode, the Sequence window contains one waveform only.

Waveform and Table View of the Sequencer

NOTE:

The sequencer provides both a Waveform and Table view of your sequenced waveforms. Both views provide the same data; however, Waveform provides a graphical representation of your sequence, while the Table shows a list.

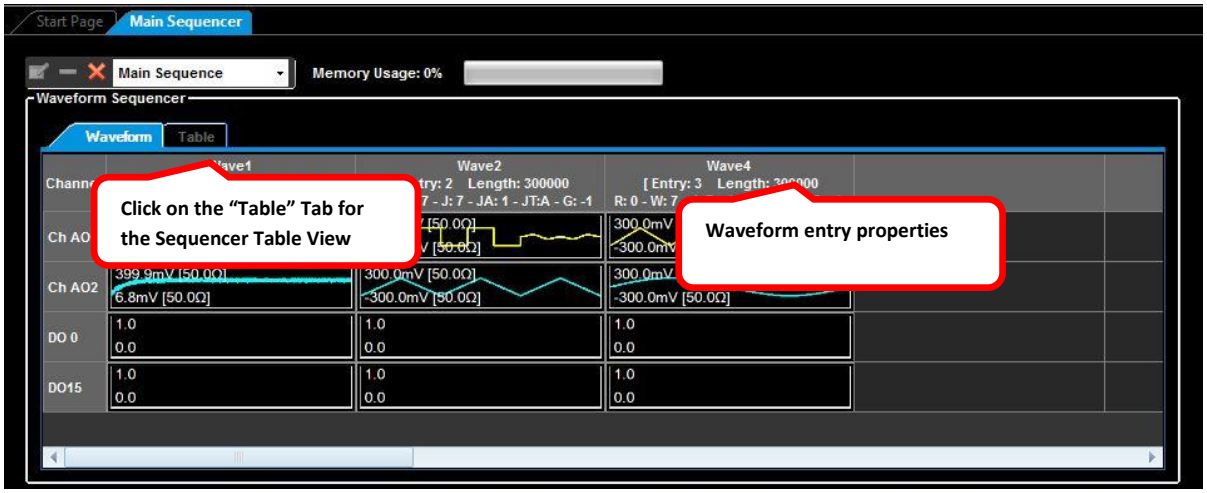


Figure 97 Table view of the sequencer

Editing a Sequence

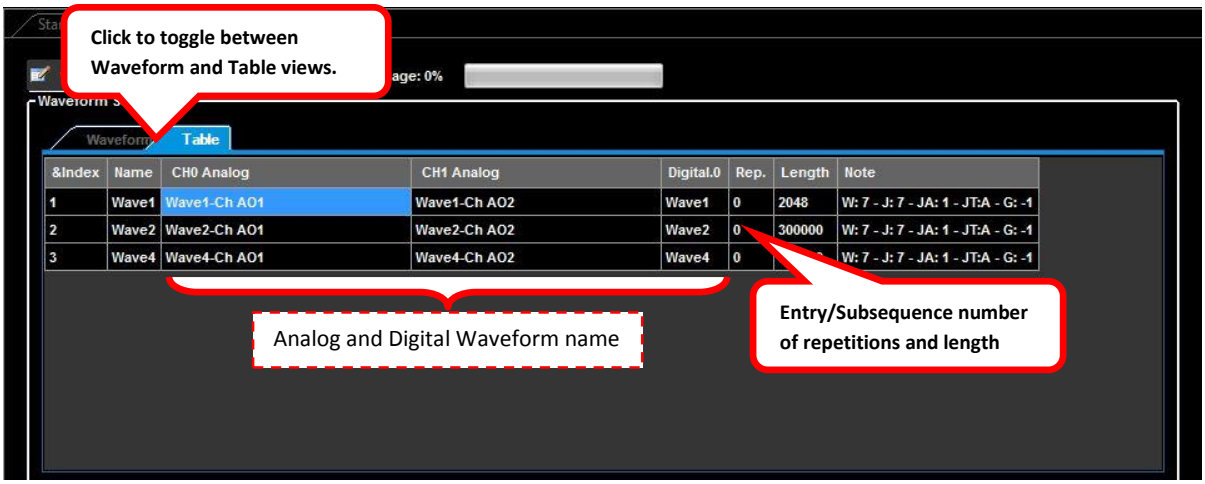
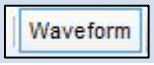


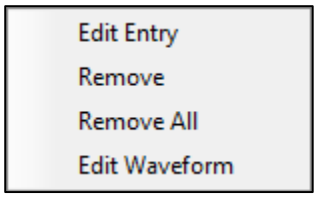
Figure 98 Sequence editor

The Sequence Window allows you to select waveforms from the Waveforms tab for generation by the AWG4162 instrument.
Drag a waveform from the Waveform TAB to a cell of the Sequence window to insert it in the sequencer. **Remove** waveforms from the Sequencer using the Sequence Window Toolbar.

Sequence Window Toolbar

	**Edit Entry – opens the *Input Waveform Property window***
	**Remove selected entry –** removes all the waveforms in the selected entry from the sequencer.
	**Remove all –** removes all the entries from the sequencer.
	**Waveform/Table** – switches the sequencer visualization from waveform to table style.
	**Memory Usage** – indicates the percentage of available memory for waveform generation.

A Right Click on a Waveform cell activates a pop-up menu



- **Edit Entry**: opens the *Input Waveform Properties* window to change the current entry properties.
- **Remove:** removes all the waveforms in the selected entry from the sequencer.
- **Remove all:** removes all the entries from the sequencer.

7. **Edit Waveform:** opens the *Editing Waveform Window* to make changes on the selected waveform.

How to Create a Sequence

Prerequisites: 03_JumpIfRepeat_Wait_Sequence project is ready.

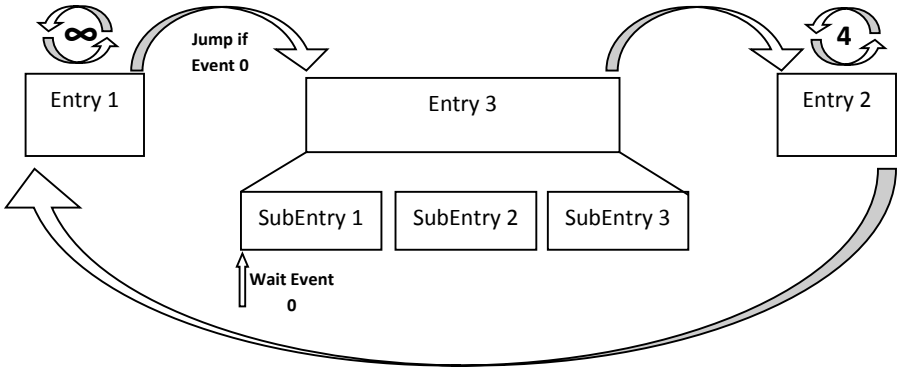


Figure 99 Sequence editor

1. Open the demo Jump project 03_JumpIfRepeat_Wait_Sequence from C:\Program Files (x86)\Tektronix\AWG4000 Advanced\DemoProjects. (This is default Advanced APP install path. If user install Advanced APP in other path, then you should open form InstallPath\ DemoProjects)
2. Press the  button in the Sequence Window toolbar to remove all the sequencer entries.

3. On the Waveforms tab, drag the Square waveform and drop it in the first entry of the sequencer.

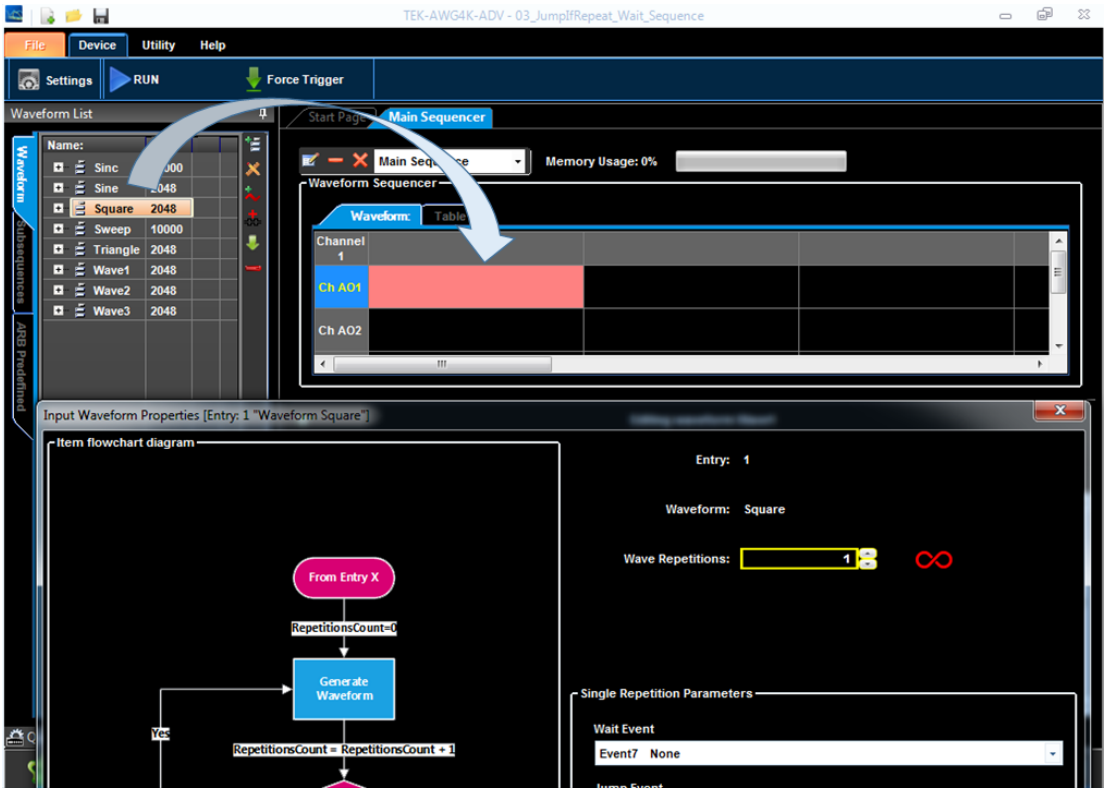


Figure 100 Dragging the waveform

4. On the *Input Waveform Properties* window select infinite repetitions and Event0 (Force Trigger AND True AND True AND True) in the Jump Event drop-down list.
   The Jump Event and Address control is disabled because there are not entries in the sequencer yet. Will change it in step9.
   Press OK to confirm.

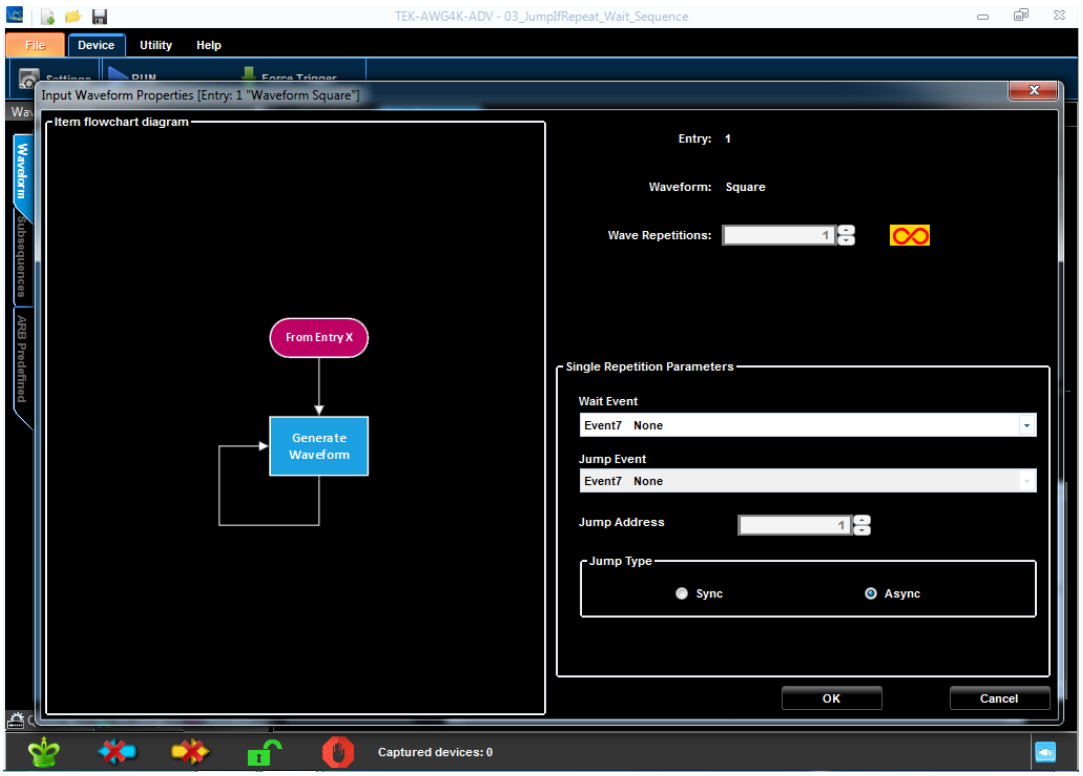


Figure 101 Edit waveform entry 1

5. On the Waveforms tab, drag the Sine waveform and drop it in the second entry of the sequencer.

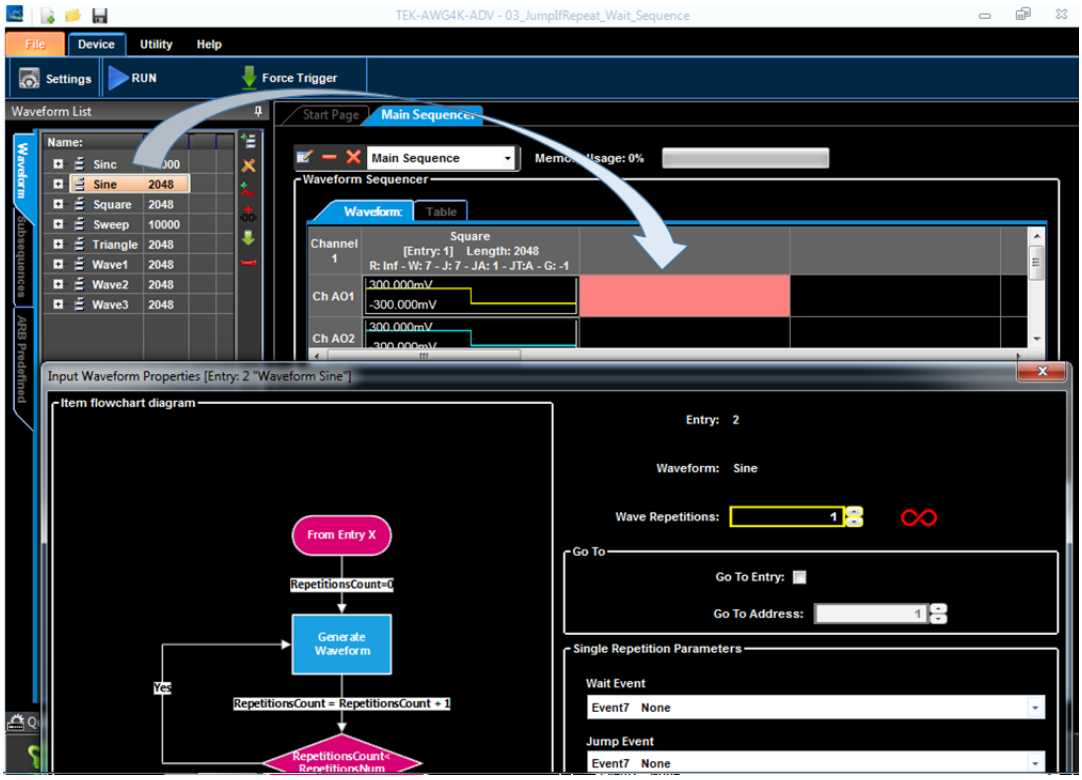


Figure 102 Dragging the waveform

6. On the *Input Waveform Properties* window select 4 repetitions, mark the Go To Address checkbox and type 1 as Go To Address value. Press OK to confirm.

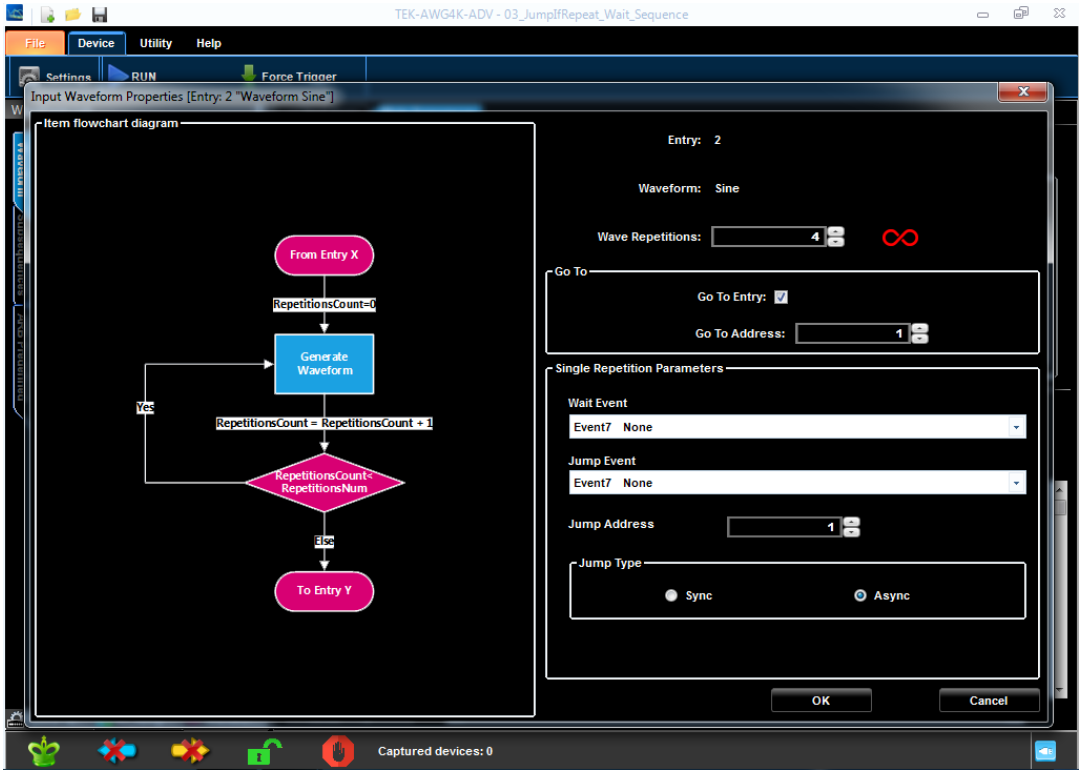


Figure 103 Edit waveform entry 2

7. On the Subsequences TAB, drag the Subsequence1 and drop it in the third entry of the sequencer.

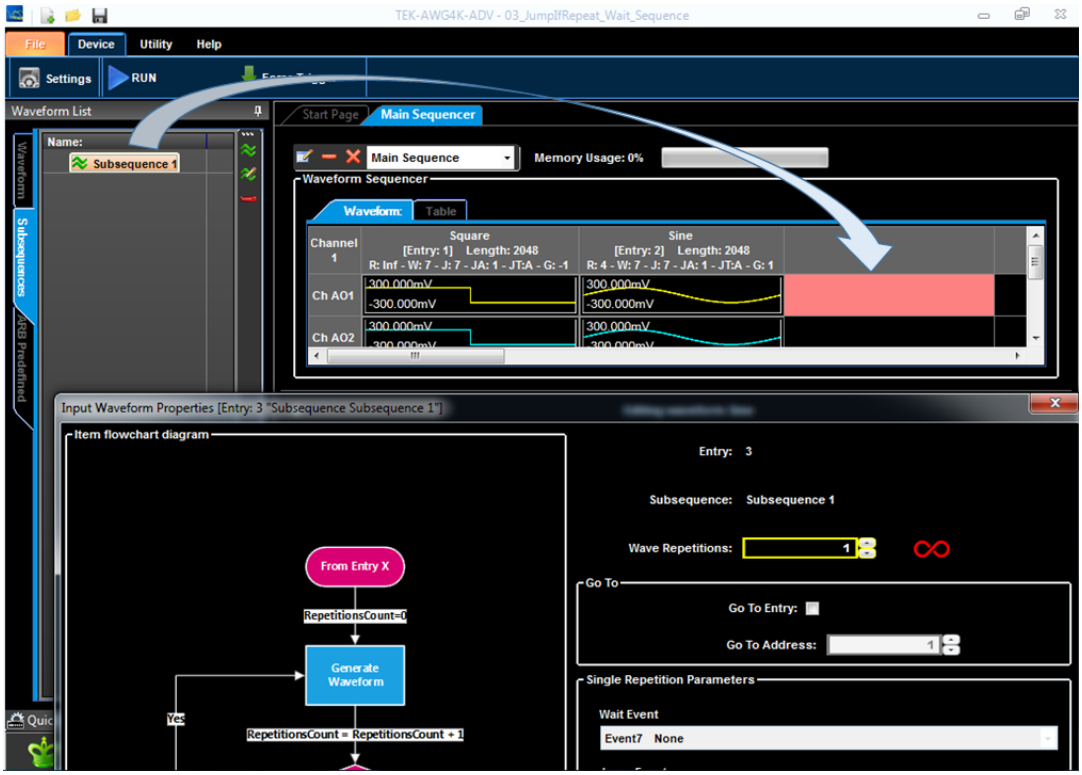


Figure 104 Dragging the Subsequence

8. On the *Input Waveform Properties* window select 1 repetitions, mark the Go To Address checkbox and type 2 as Go To Address value. Press OK to confirm

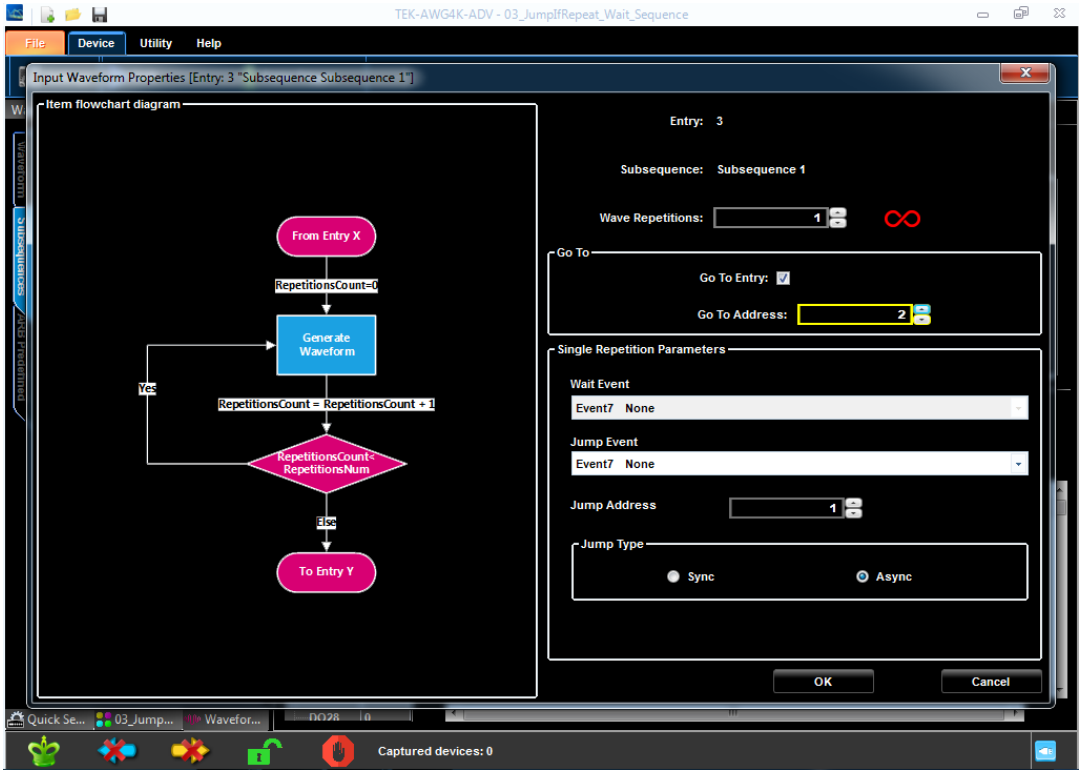


Figure 105 Edit waveform entry 3

9. Now the sequence is complete but you need to modify the properties of the Entry 1; double click on the Entry 1 or right click to open the pop-up menu and select Edit Entry.
   Set Event0 (Force Trigger AND True AND True AND True) in the Jump Event drop-down list. Modify the Jump Address field and type 3 as address of the jump. Press the OK button to confirm.

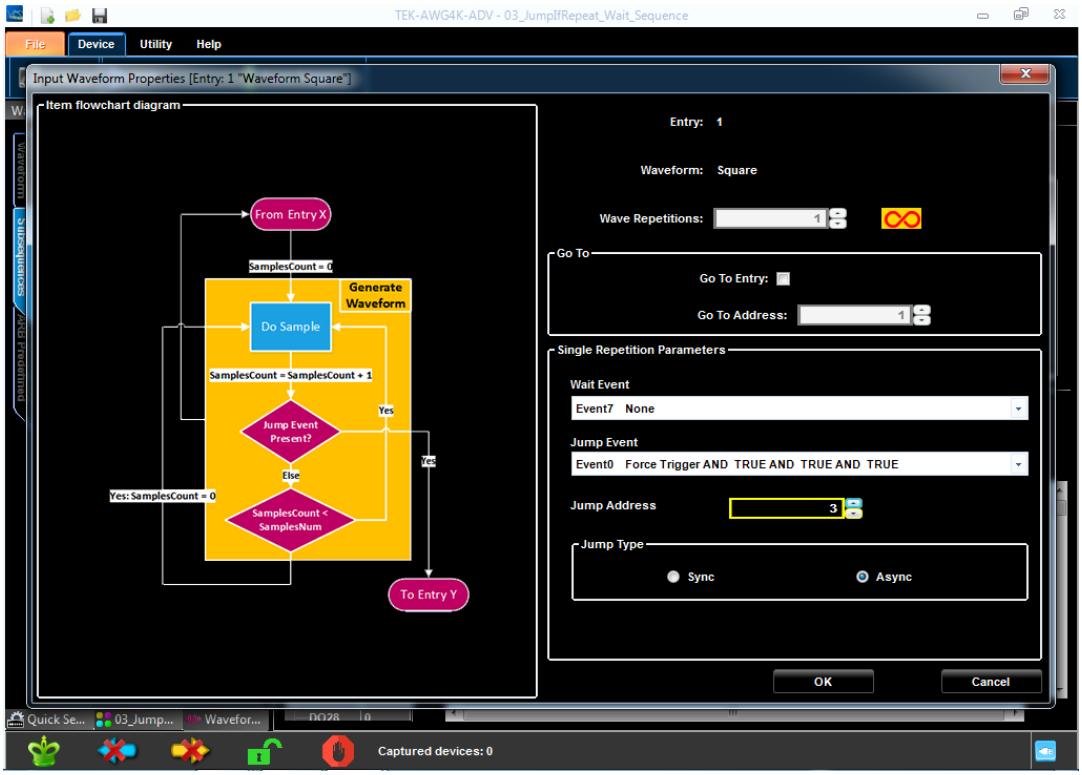
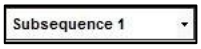


Figure 106 Edit waveform entry 1

10. The last step is to set wait event 0 to the subentry 1 of Subsequence 1.

    Press the 

Sequence/Subsequence button to select Subsequence 1.

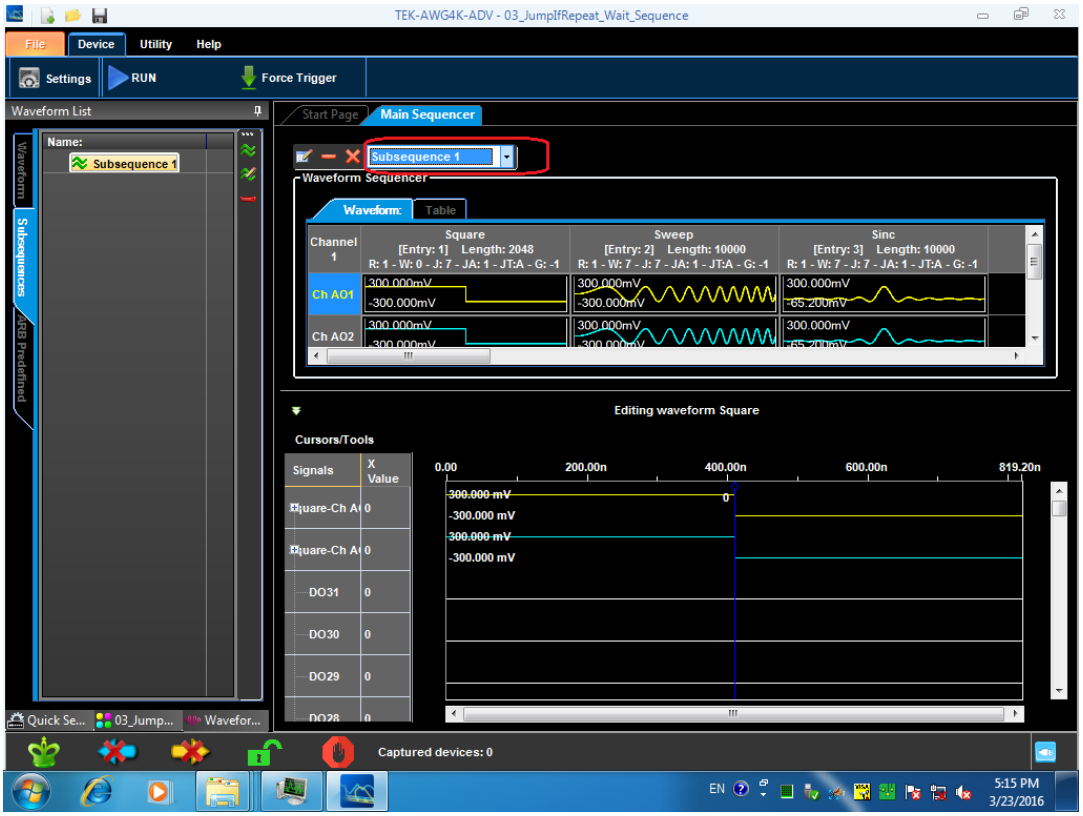


Figure 107 Waveforms of Subsequence 1

Then double click on the Entry 1 or right click to open the pop-up menu and select Edit Entry.

11. On the *Input Waveform Properties* window, set Event0 (Force Trigger AND True AND True AND True) in the Wait Event drop-down list. Press OK to confirm

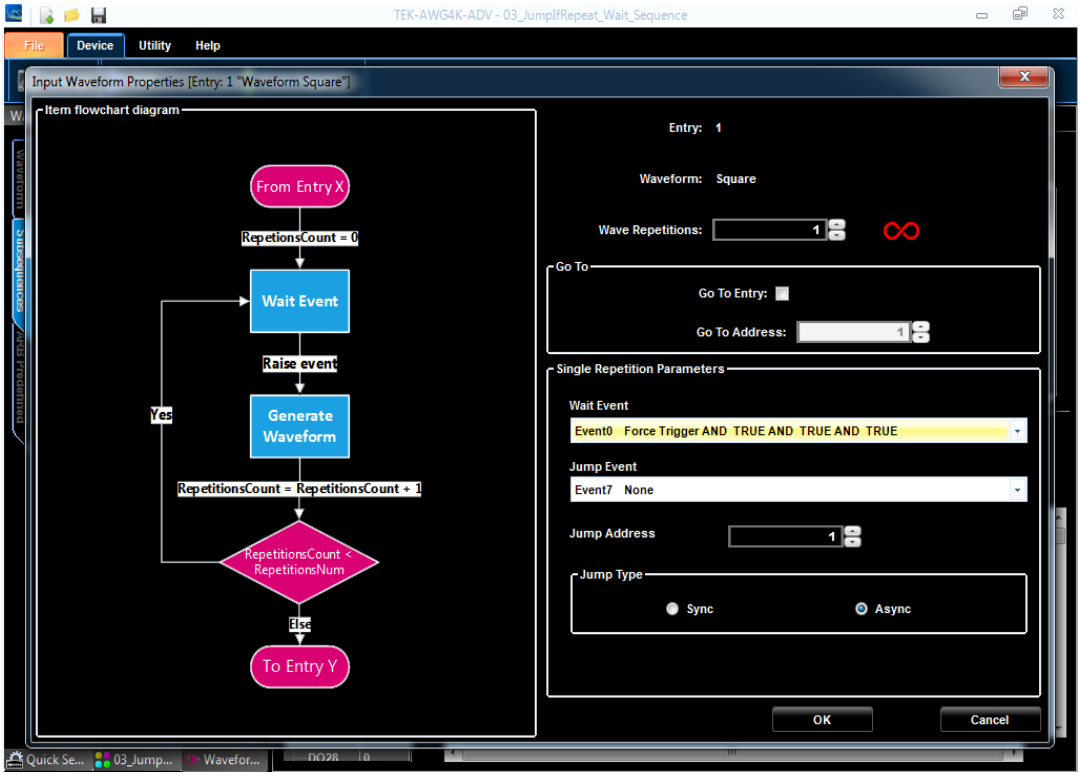


Figure 108 Edit waveform subentry 1

How to Create a Subsequence

Prerequisites: 03_JumpIfRepeat_Wait_Sequence project is ready.

1. Open the demo Jump project 03_JumpIfRepeat_Wait_Sequence from C:\Program Files (x86)\Tektronix\AWG4000 Advanced\DemoProjects. (This is default Advanced APP install path. If user install Advanced APP in other path, then you should open form InstallPath\ DemoProjects)
2. On the Sequences TAB press the  New Subsequence button.
3. The new empty subsequence is shown on the Main Sequencer Window (Subsequence 2)

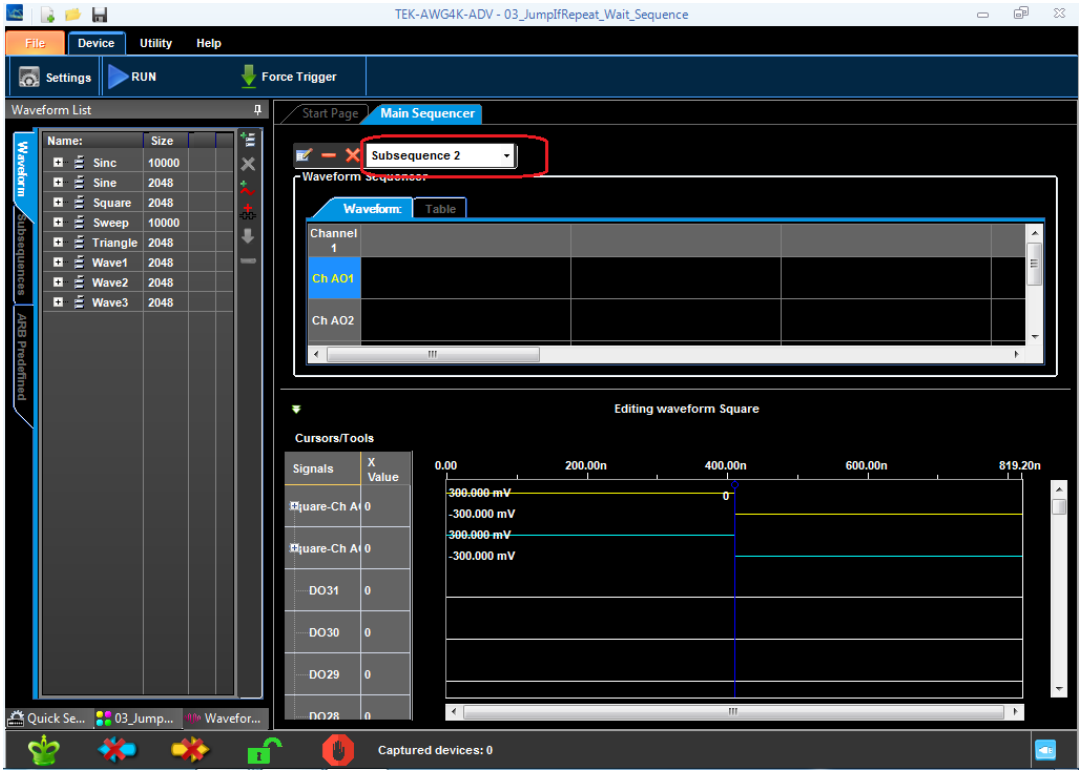


Figure 109 New empty subsequence window

4. Drag a waveform from the Waveform TAB to a cell of the Sequence window to insert it in the sequencer. The destination sequencer cell of the selected waveform will be enlightened in red.
   On the *Input Waveform Properties* window, select 1 repetitions. Press OK to confirm

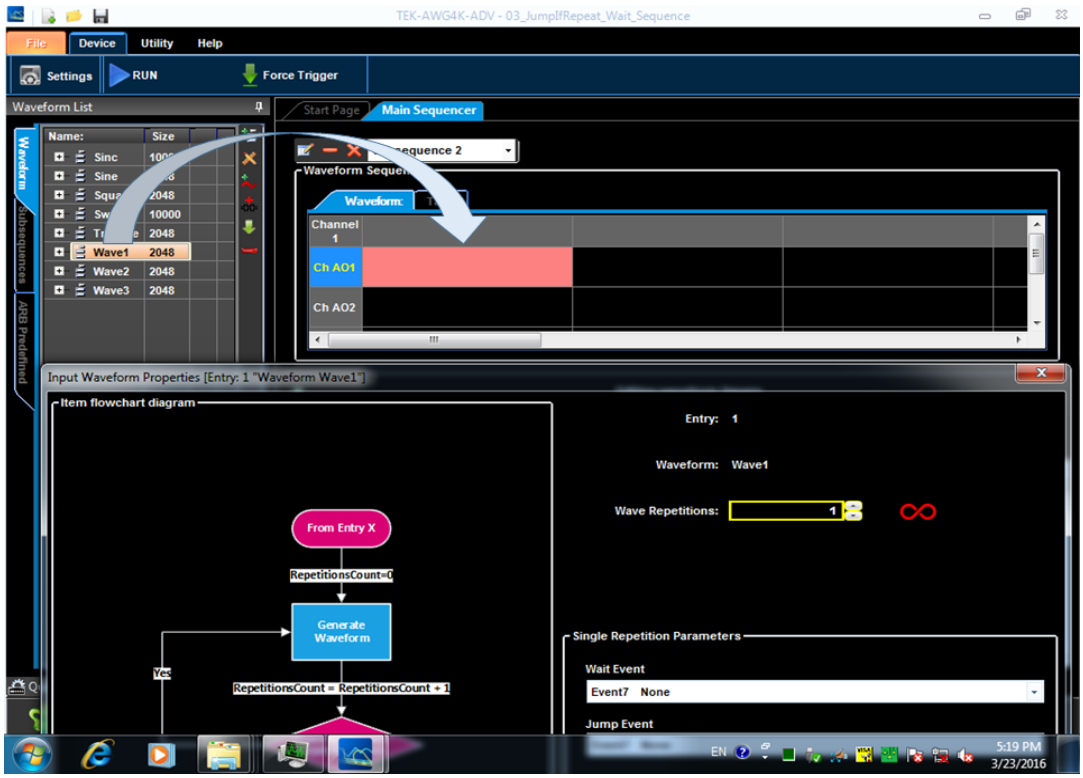
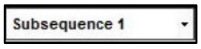


Figure 110 Dragging the waveform

5. Repeat the last step Square and Sync waveforms. The Subsequece2 is made of three entries: Wave1(Square), Sweep and Sinc Mixed waveforms.

Figure 111 Waveforms of subsequence 2

6. Press the 

Sequence/Subsequence button to close the subsequence, confirm the changes and return to the main sequencer.
The Subsequence2 will appear in the Sequences TAB and now you can insert it into the Main Sequencer.

Waveform View Window

A selected element in the Sequence Window is displayed in the *Waveform View* window.
To the left of every signal, there are two columns: the first one called *Signals* displays the name and the root icon to open/close the segment/component/bus elements, the second one called *Value* displays a number indicating the value the signal at the time position of the master cursor.

1. If you select a single cell, you can use the Waveform View window to display analog and digital waveforms.

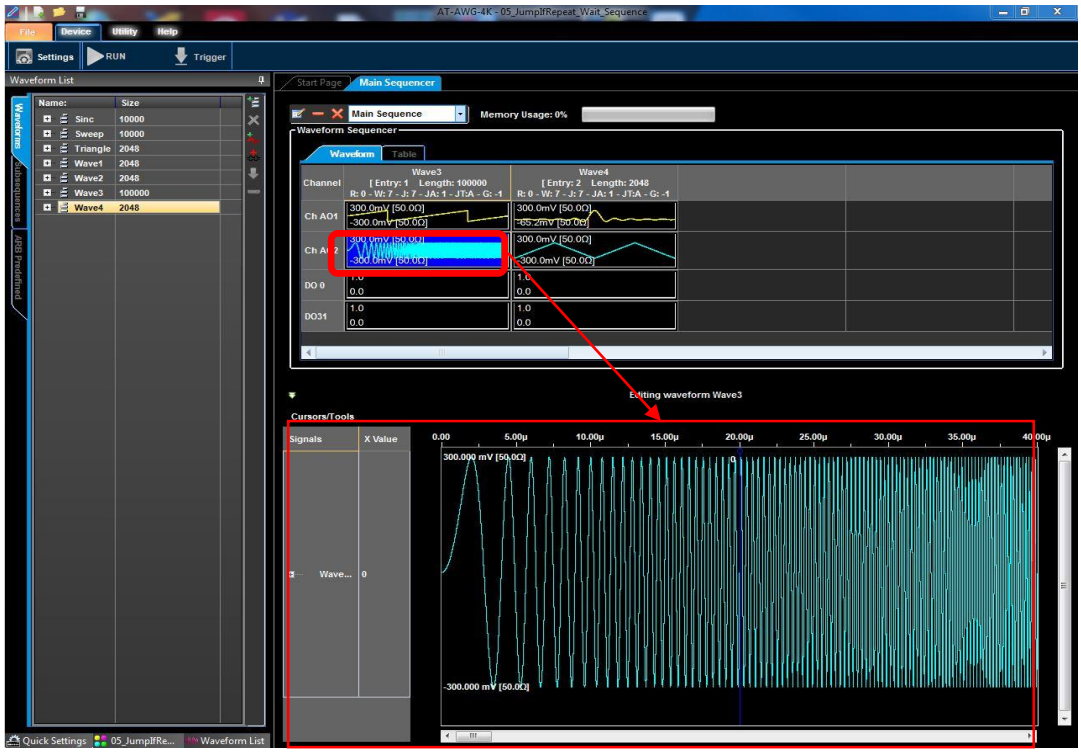


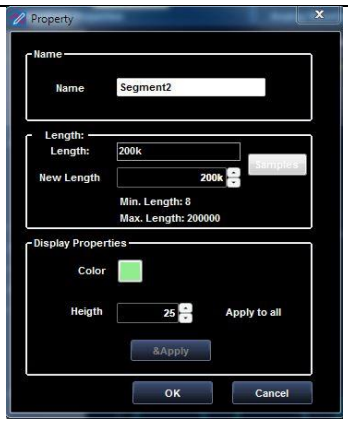
Figure 112 View the waveforms

2. If you click on the header of the cell in the Sequence window, the entry is selected and all the waveforms inside it are displayed in the Waveform View window, as shown in the following.

Figure 113 View the waveforms

When viewing a waveform in the Waveform View Window, you have access to the following functions:

	**This button allows changes the mouse function for the graphic area to cursors/markers movement.**
	The hand tool allows you to dragging inside the graph area.
	Auto zoom in function.
	Auto zoom out function.
	This button allows zooming in on a selected rectangle of the graph. Click and drag inside the graph area to create your zoom rectangle.
	This button resets all activated zooms
	You can change the properties of the graph display area. Click the **Waveform View Settings** button and the **Graph Property** screen is shown.



Changes can be made as follows:

- The **Background Color** can be changed as desired.
- Change colors and turn the **Major** and **Minor Grids** on or off and change their line coloring.
- **Cursor Position** indicators can be turned on or off.

This button switches the X-axis representation between **number of samples** to **seconds**. Default values are optimized based on the selection made.

Tutorials

Scenarios with detailed steps for performing typical tasks and setups using AWG4162 are shown in the **Examples.**

How do I Examples

PLEASE NOTE THE FOLLOWING:

- Before performing any of the scenarios, you must first make sure you've **correctly powered on your instrument**, and **launched the AWG4162 software** as explained in the **Getting Started** section.
- Some more specific steps are required around Creating a New Workspace for each scenario. Details are provided for those scenarios when necessary.

With the aforementioned prerequisites completed, you can perform the following scenarios:

1. **Creating Your First Analog Waveform**
2. **Creating a Sequence of Waveforms**
3. **Importing a Waveform + Component Usage + Gated Run Mode**
4. **Creating Digital Waveforms**

Creating Your First Analog Waveform

After you have powered on the instrument, launch the software and use the system menu bar or the toolbar to create a New Workspace.

1. Create a Single Sequencer project with Arbitrary Waveform Generator as the Operating Mode.
2. Click the New Mixed Waveform button.
   

3. The New Waveform window is shown. Type the name of the waveform “Wave1” and choose 2048 for the samples length of the waveform. Click OK to confirm.

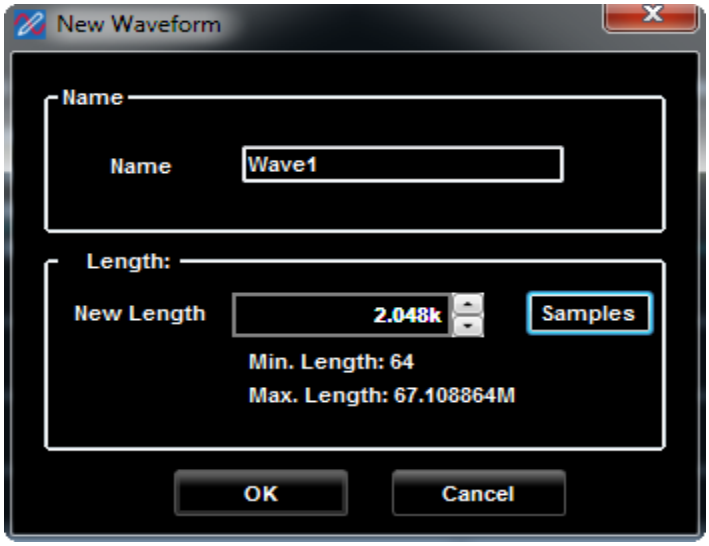


Figure 114 New waveform settings

4. The Editing Waveform window is shown. Select the waveform Wave1-Ch AO1 and click on the Edit Waveform button

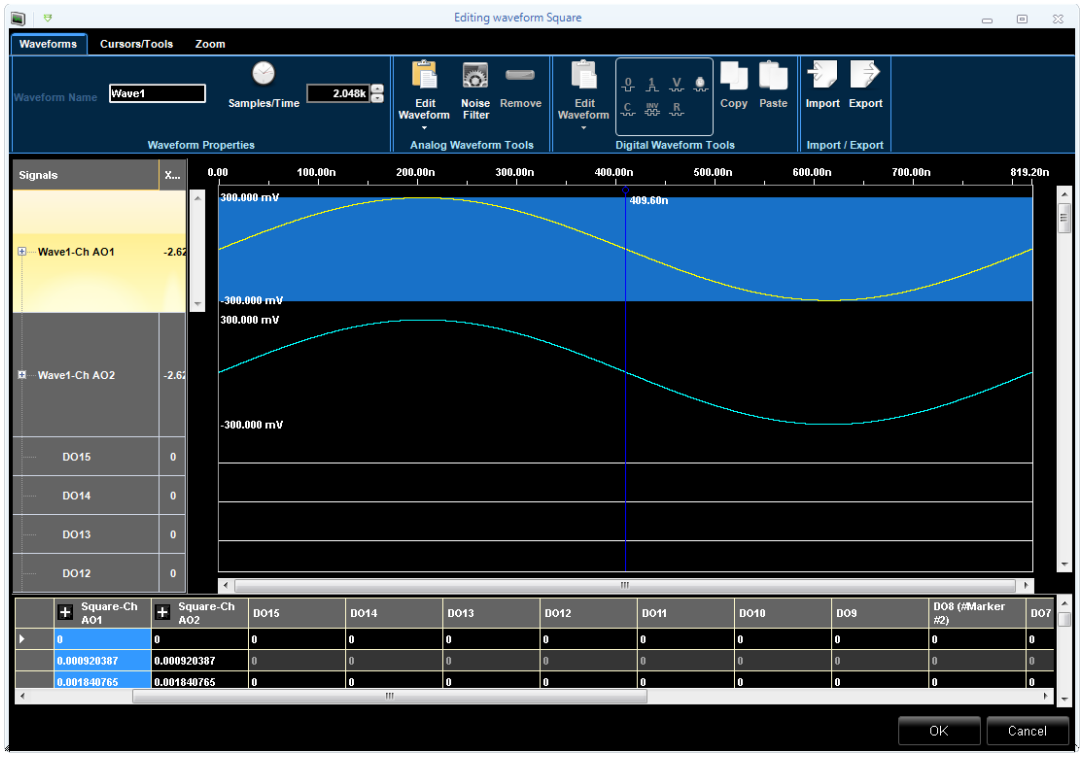


Figure 115 Editing waveform window

5. The Waveform Standard Editor is shown. Choose a sine waveform with the following specs:

- Cycles: 2
- Amplitude[V]: 250mV

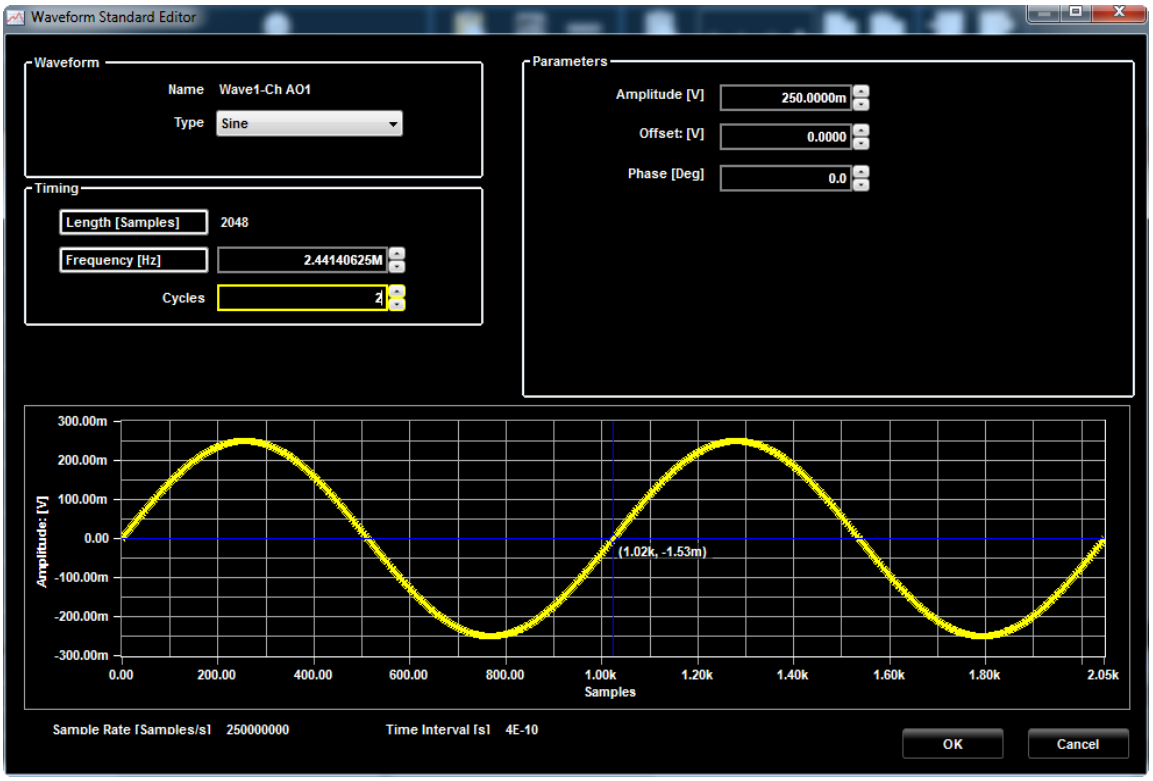
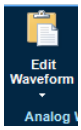


Figure 116 Standard waveform editor

6. Press OK button.

7. Select the waveform Wave1-Ch AO2 and click on the Edit Waveform 

button.

8. The Waveform Standard Editor is shown. Choose a rectangle waveform with the following specs:

- Cycles: 4
- Amplitude[V]: 300m

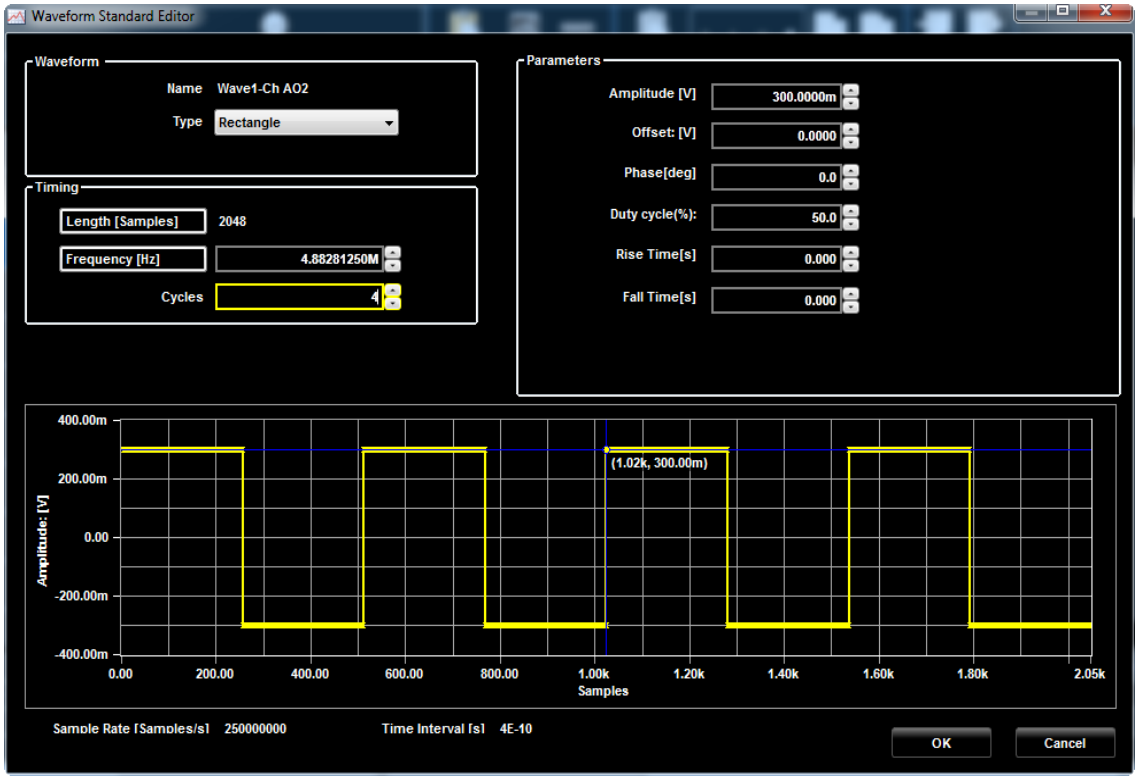


Figure 117 Standard waveform editor

9. Press OK button.
10. Press OK button on the Editing Waveform window.
11. On the main toolbar, press the Settings 

button.
12. On the Settings pop-up screen's Run Mode tab, select Continuous.

13. Click OK.

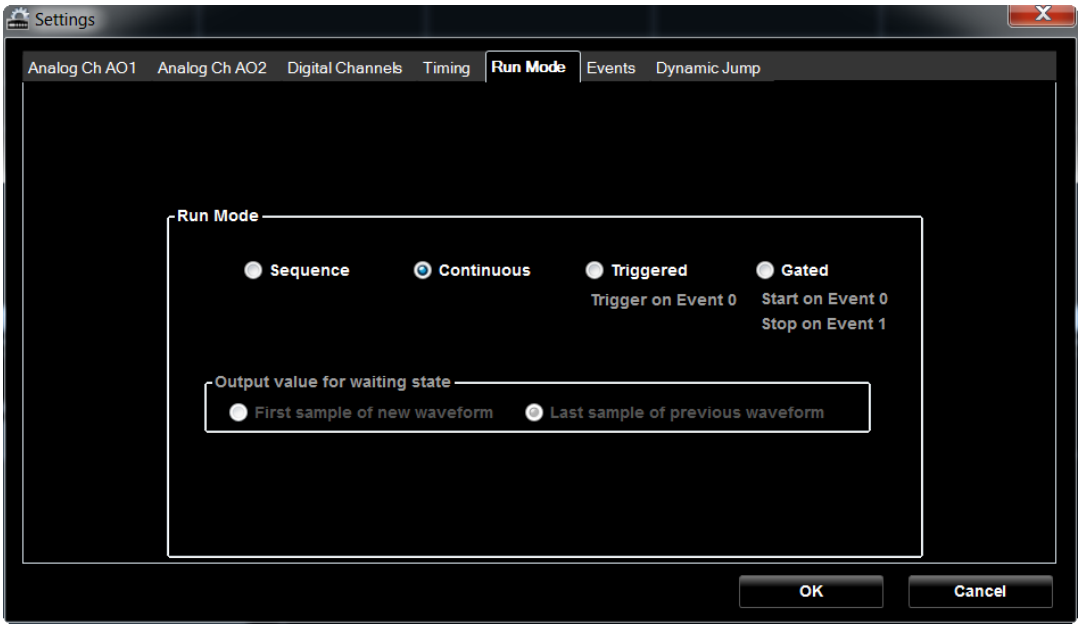


Figure 118 Setting screen

14. Drag the Wave1 from the Waveform Area to the first cell of the Sequence Area (the selected cell is highlighted).

Figure 119 Sequence window

15. Now, press the Run/Stop button on the AWG4K toolbar.

The software loads the waveforms into the AWG4K instrument and starts generating the waveforms. Wave1 is generated on the AO1 and AO2 SMA outputs. You can connect an oscilloscope to this output and analyze the signals.
You can stop generating waveforms by pressing the **Run/Stop** button again.

Creating a Sequence of Waveforms

After you have powered on the instrument, launch the software and use the system menu bar or the toolbar to create a New Workspace.

1. Create a Single Sequencer project with Arbitrary Waveform Generator as the Operating Mode.

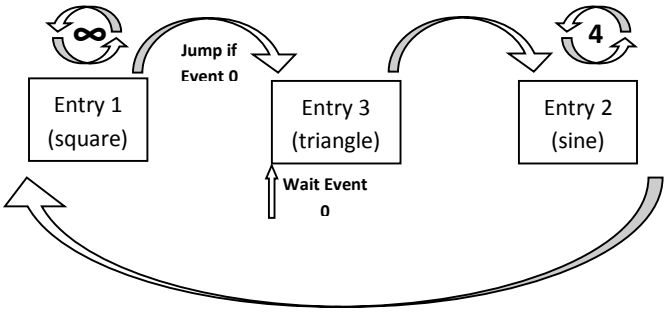


Figure 120 Single sequencer project

2. Add three waveforms to our Waveforms tab list.
3. Assign the following specifications for waveforms 1, 2, and 3 on the Waveform Standard Ed itor dialg box.
   Waveform 1 - Square waveform Type, 2048 Samples, 1 Cycle and 300 mv Amplitude.
   Waveform 2 - Sine waveform Type, 2048 Samples, 1 Cycle and 300 mVolt Amplitude.
   Waveform 3 – Triangle waveform Type, 2048 Samples, 1 Cycle and 300 mVolt Amplitude.
4. Click the New Mixed Waveform button  .
5. The New Waveform window is shown. Type the name of the waveform “Square” and choose 2048 for the samples length of the waveform. Click OK to confirm.

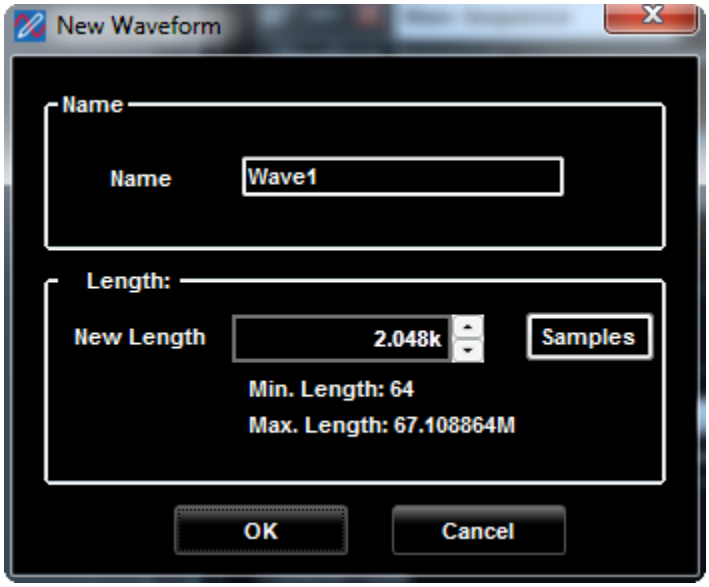


Figure 121 New waveform window

6. The Editing Waveform window is shown. Select the waveform Square-0 and click on the Edit 

button

Figure 122 Edit waveform window

7. The Waveform Standard Editor is shown. Choose a rectangle waveform with the following specs:

- Cycles: 1
- Amplitude[V]: 300mV

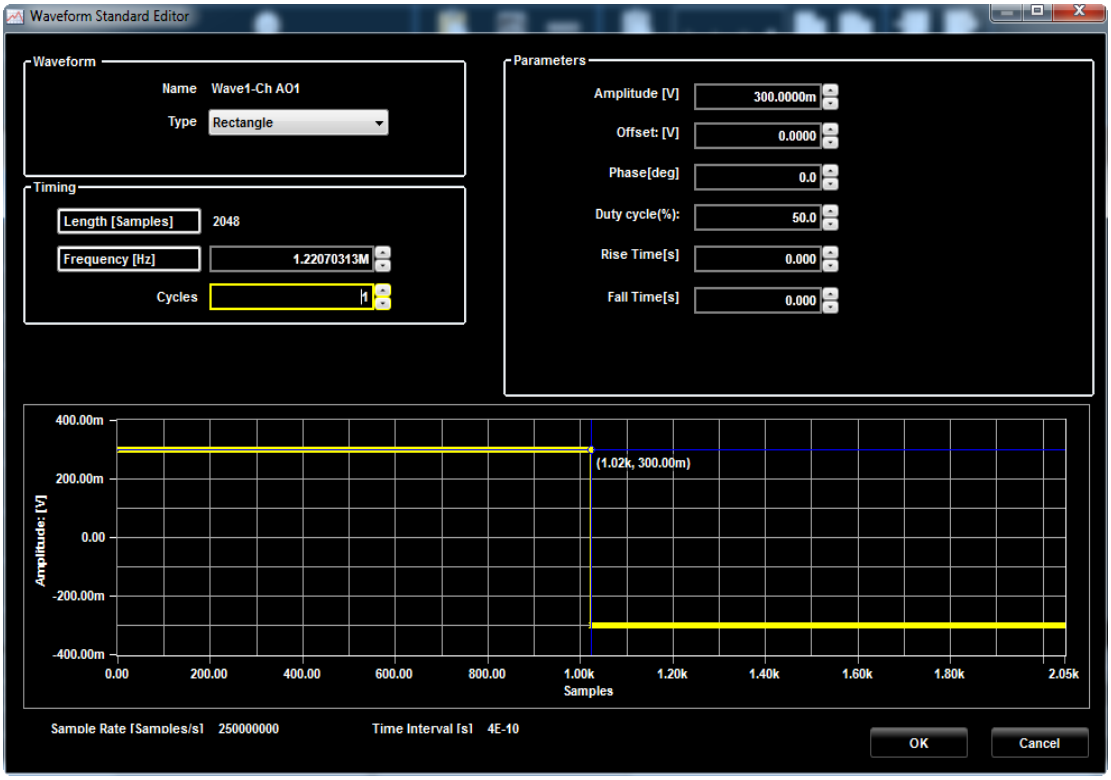


Figure 123 Waveform standard editor

8. Press OK button.
9. Press OK button on the Editing Waveform Window to close and confirm.

Figure 124 Edit waveform

10. Repeat the steps 4 ~ 9 to create the Sine and Triangle waveforms. The *Square, Sine* and *Triangle* will appear on the Waveform tab.
11. On the main toolbar, press the Settings 

button. On the Settings pop-up screen's Run Mode tab, select Sequence.

Click OK.

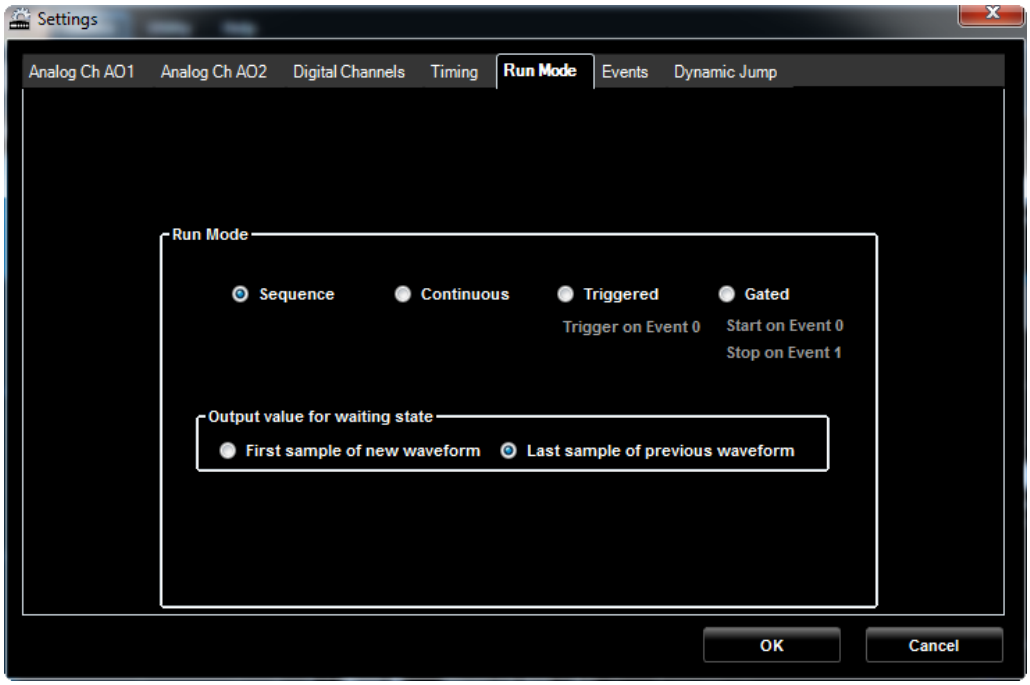


Figure 125 Setting screen

12. On the Waveforms tab, drag the Square waveform and drop it in the first entry of the sequencer.
13. On the Input Waveform Properties window select infinite repetitions.
14. The Jump Address control is disabled because there are not entries in the sequencer yet.
15. Press OK to confirm.

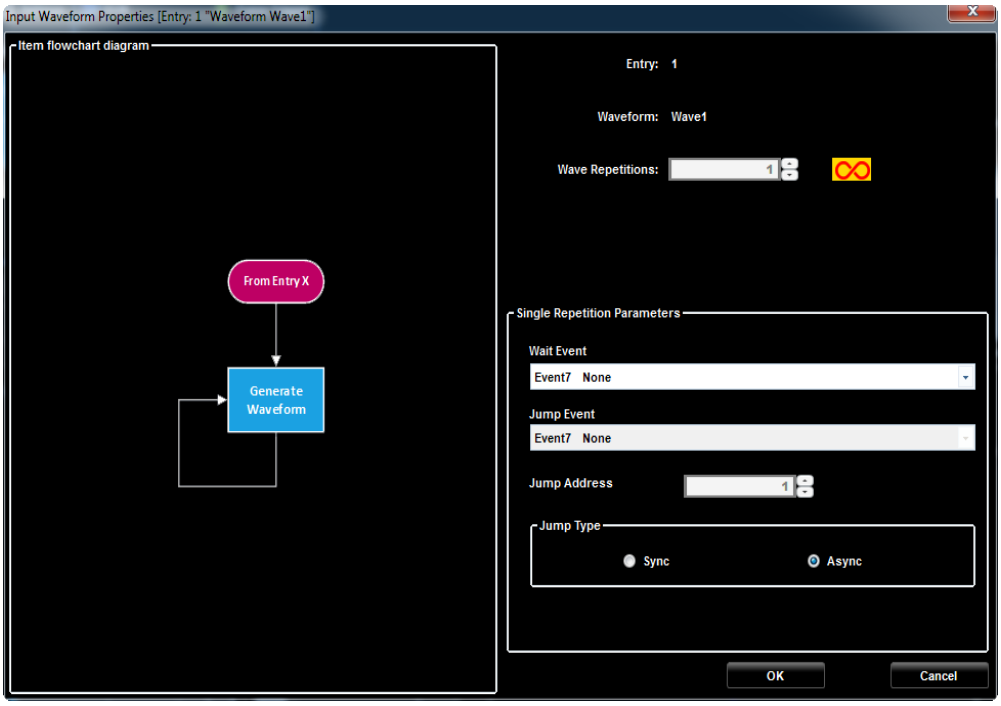


Figure 126 Input waveform properties

16. On the Waveforms tab, drag the Sine waveform and drop it in the second entry of the sequencer.
17. On the Input Waveform Properties window select 4 repetitions, mark the Go To Address checkbox and type 1 as Go To Address value. Press OK to confirm.

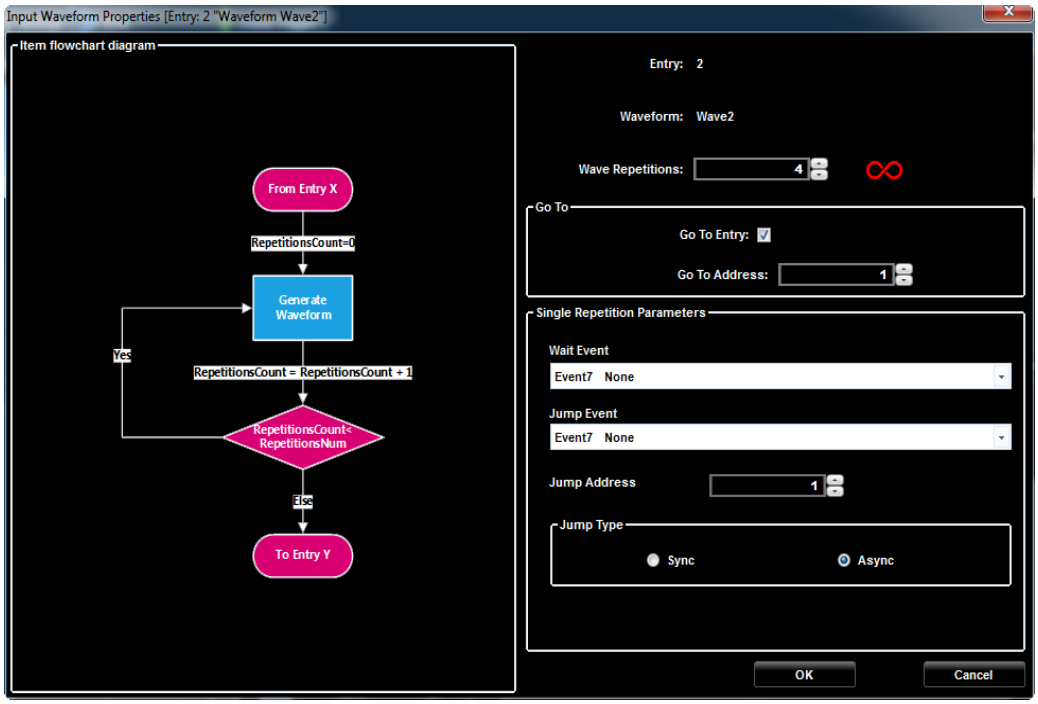


Figure 127 Input waveform properties

18. On the waveforms TAB, drag the Triangle waveform and drop it in the third entry of the sequencer.
19. On the Input Waveform Properties window select Wait Event0 and mark the Go To Address checkbox and type 2 as Go To Address value. Press OK to confirm.

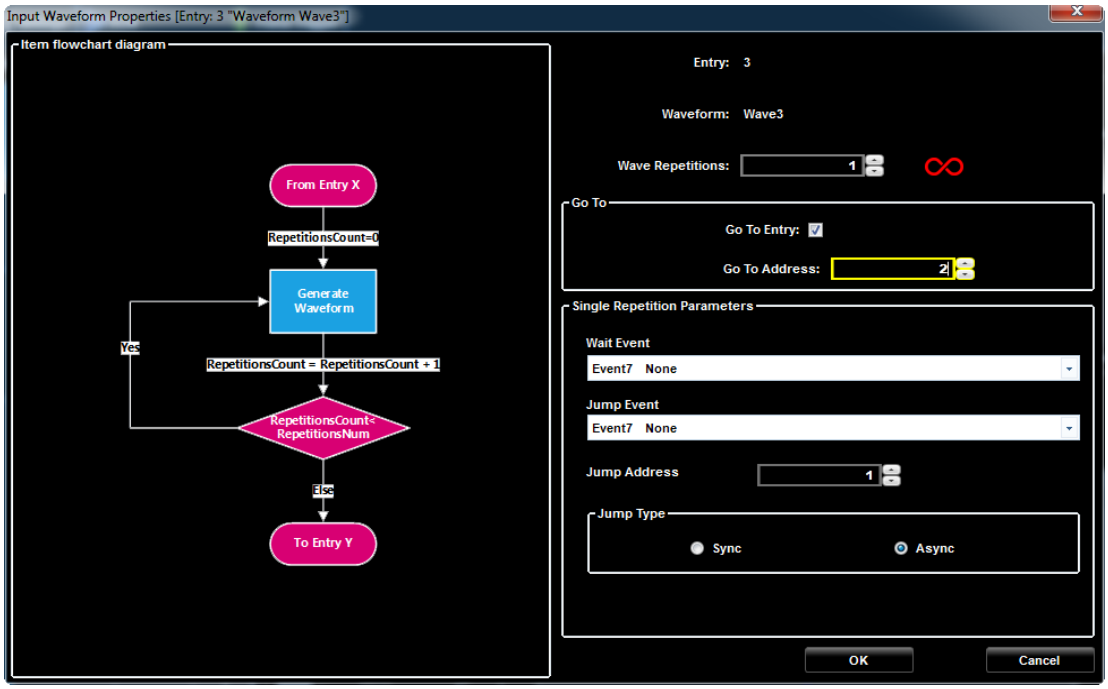


Figure 128 Input waveform properties

20. Now the sequence is complete but you need to modify the properties of the Entry 1; double click on the Entry 1 or right click to open the pop-up menu and select Edit Entry.

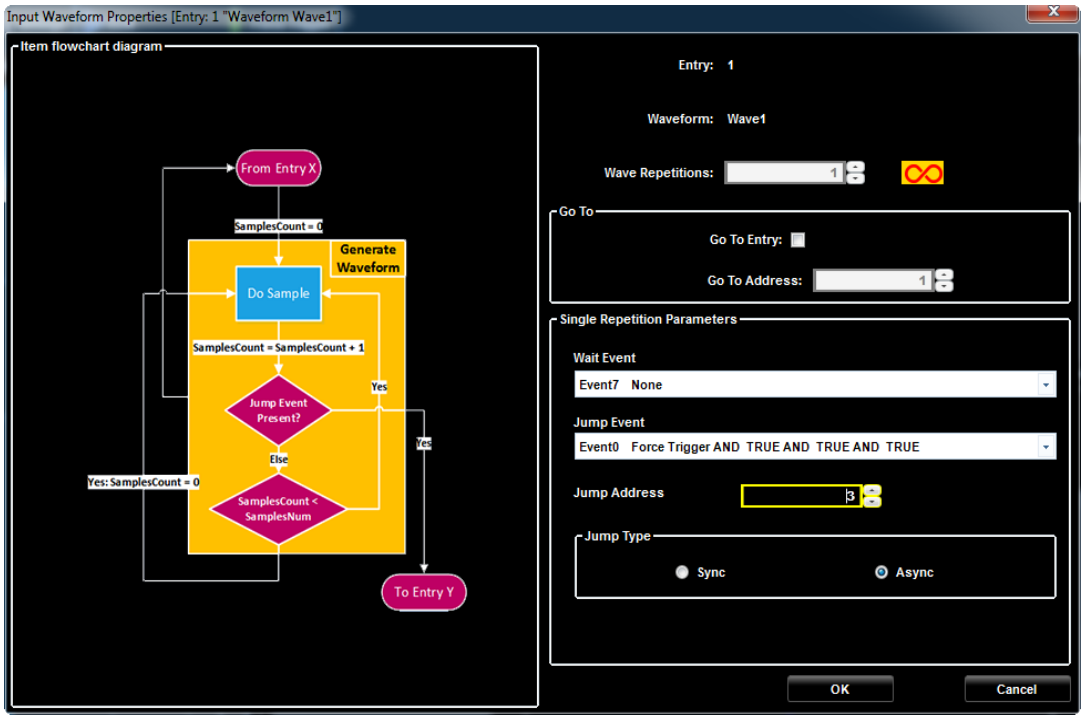


Figure 129 Input waveform properties

21. Modify the Jump Address field and type 3 as address of the jump. And set Event0 (Force Trigger AND True AND True AND True) in the Jump Event drop-down list. Press the OK button.
22. Now, press the Run/Stop button on the AWG4K toolbar.

23. The software loads the waveforms into the AWG4K instrument and starts generating the waveforms. The sequence is generated on the AO1 SMA output. You can connect an oscilloscope to this output and analyze the signal
24. Stop generating waveforms by pressing the Run/Stop button again.

Importing a Waveform + Component Usage + Gated Run Mode

Import a Waveform by first creating a project with Arbitrary Waveform Generator Mode.
If you already have a workspace open, be sure it meets the aforementioned requirements before proceeding with the following steps:

1. Click the New Mixed Waveform button. 
   The NEW Waveform window is shown. Type the name of waveform"Wave1" and choose 2048 for the samples length of the waveform. Chick OK to confirm.

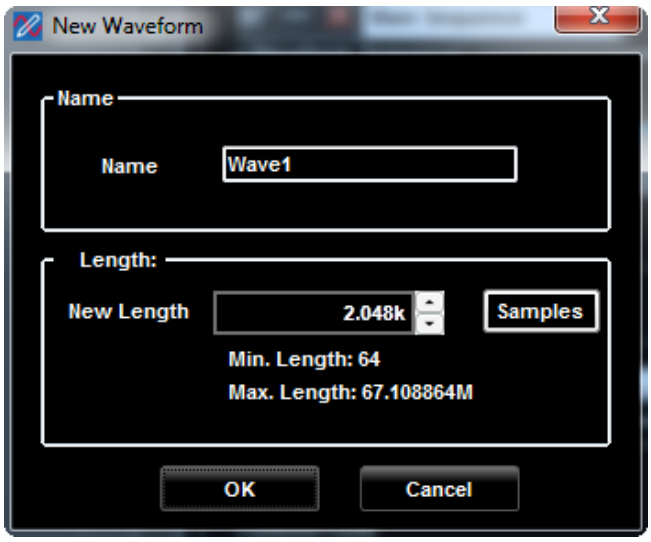
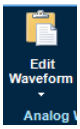


Figure 130 New waveform

2. The Editing Waveform Window is shown. Select the waveform Wave1 and click on the Edit Waveform 

button.

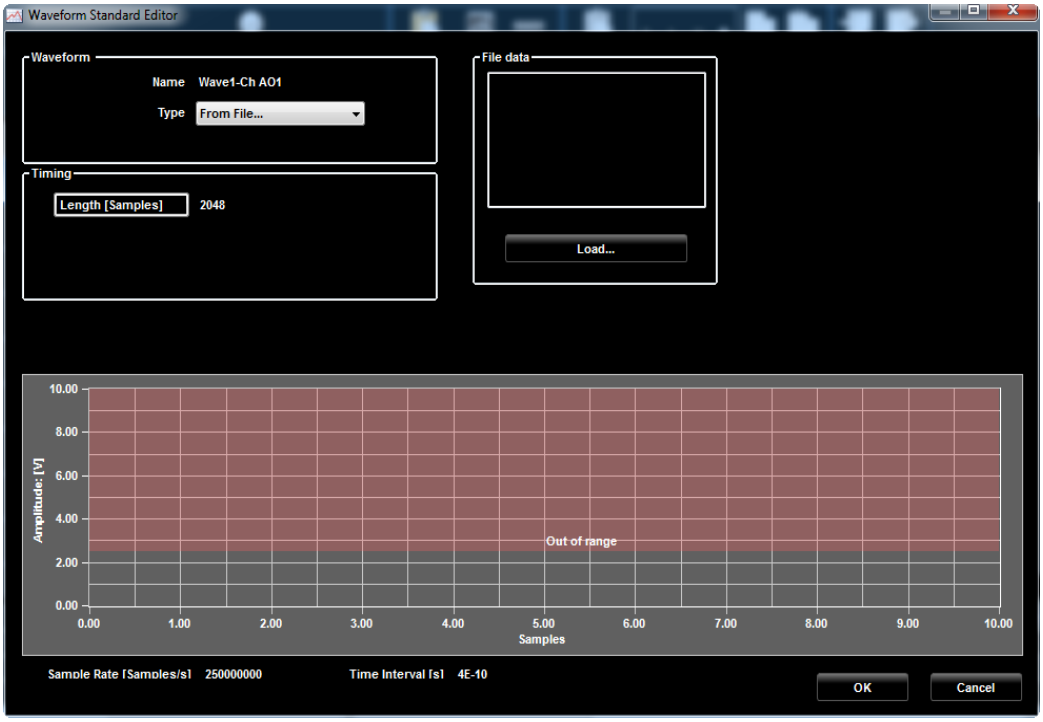


Figure 131 Editing waveform window

3. On the Type drop-down list select From File… and press the Load… button.
   The Import waveform pop-up screen is shown as follows.

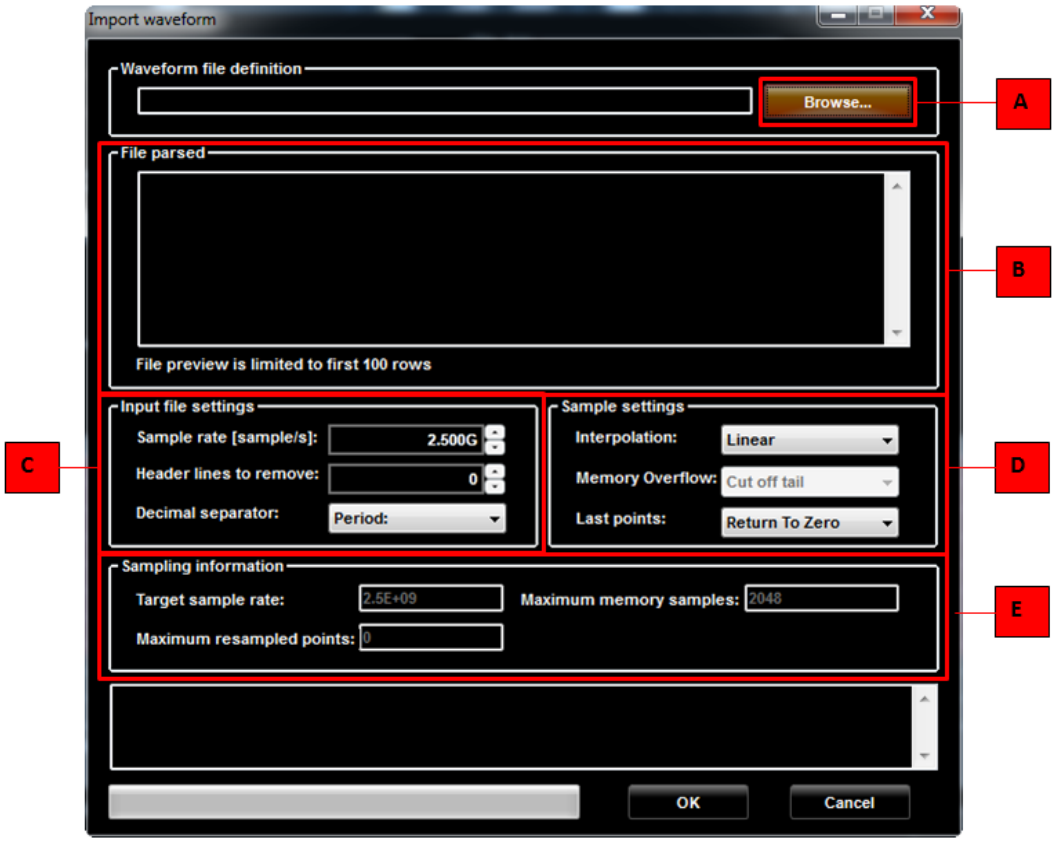


Figure 132 Input waveform properties

Labels A-E on this image corresponds with the following descriptions:

A. Click the Browse button and locate your desired .txt file for import.

B. The File parsed section then shows the first 100 samples of the imported waveform.

C. If desired, Input file settings allow for Sample rate, Header lines to remove, and Decimal separator changes to the imported file.

D. Use the Sample setting fields to adjust parameters and rules.

E. If the imported waveform has a sample rate greater than the AFG4K target sampling rate (2.5 GS/s), the waveform points are re-sampled and shown on the Sampling information fields.

4. Browse from C:\Program Files\Tektronix\AWG4000 Advanced\SCPI\AWG4KImportWaveforms\Sine0.txt file

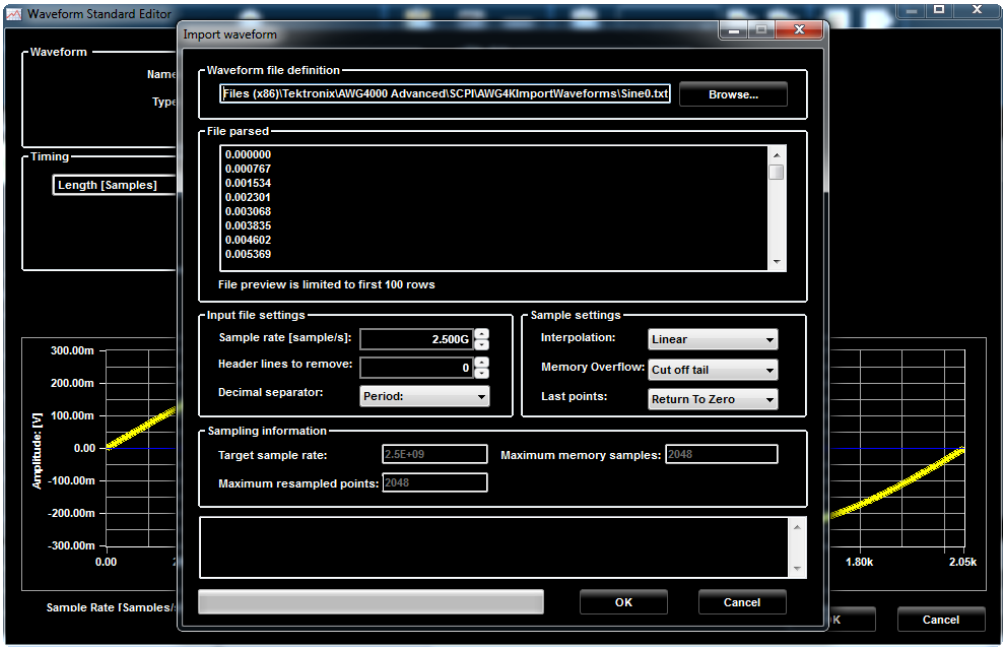


Figure 133 Browse waveform

5. Click OK to confirm. Waveform Standard Editor window is shown and it displays the imported waveform.

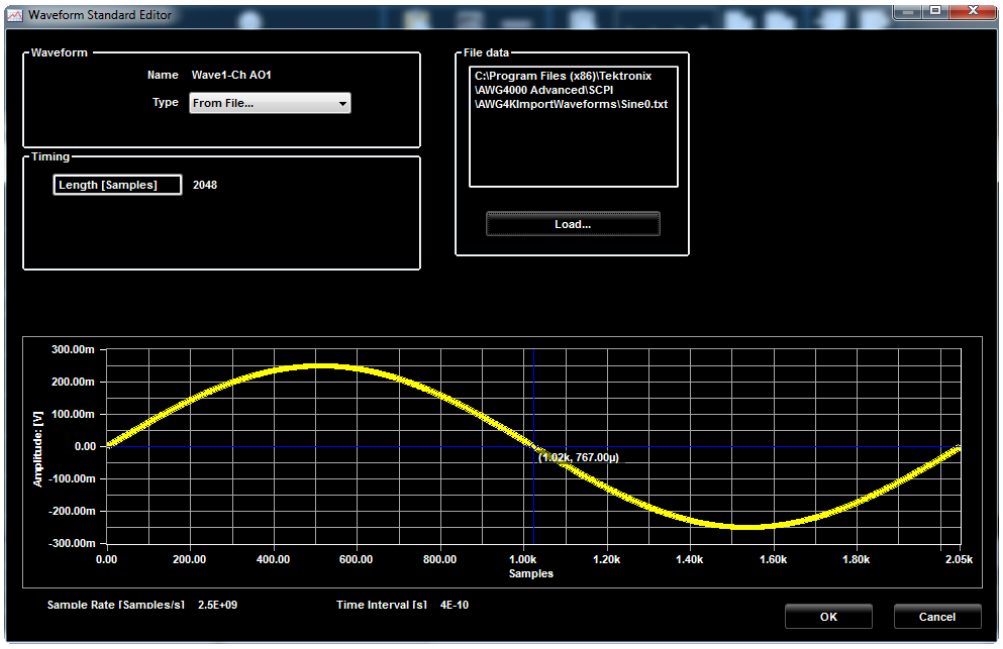


Figure 134 Waveform standard editor

In the following steps we will use a DC component to multiply it by a constant and change current waveform voltage. Click OK to confirm.

6. Right click on the Wave1 Component1 to display the pop-up menu and select Add Component.

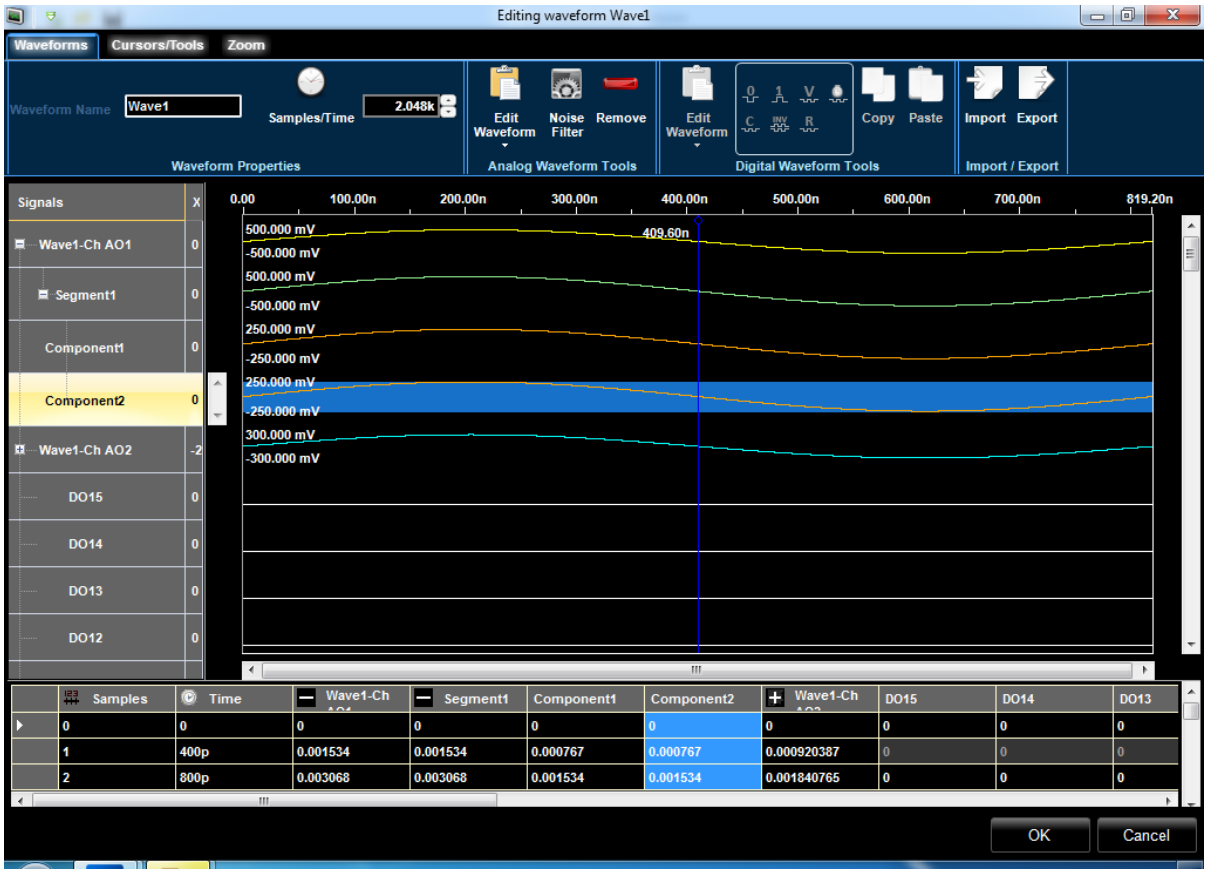


Figure 135 Edit waveform
Component2 is added to Wave1-Ch AO1

7. To Component1, select Multiply in the Operation field.
   To Component2, select DC Level as Type and 1.2 in the Offset[V] field as multiplication factor.

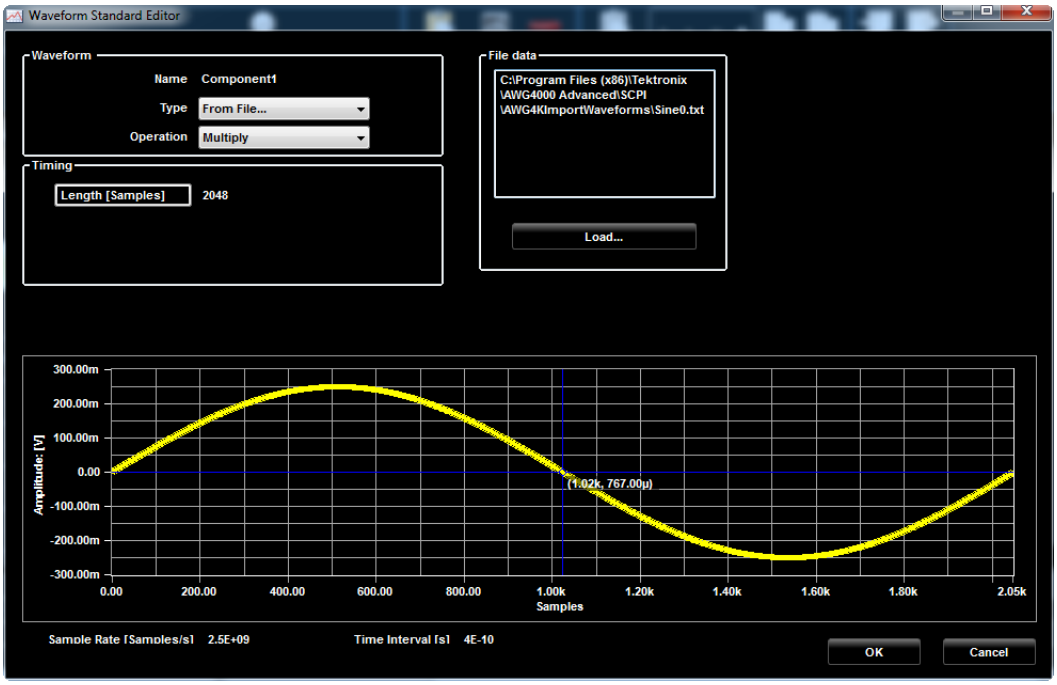


Figure 136 Edit waveform

The Wave1-Ch AO1 = Component1*Component2. The waveform amplitude of Wave1-Ch AO1 has been increased by a 1.2x factor.

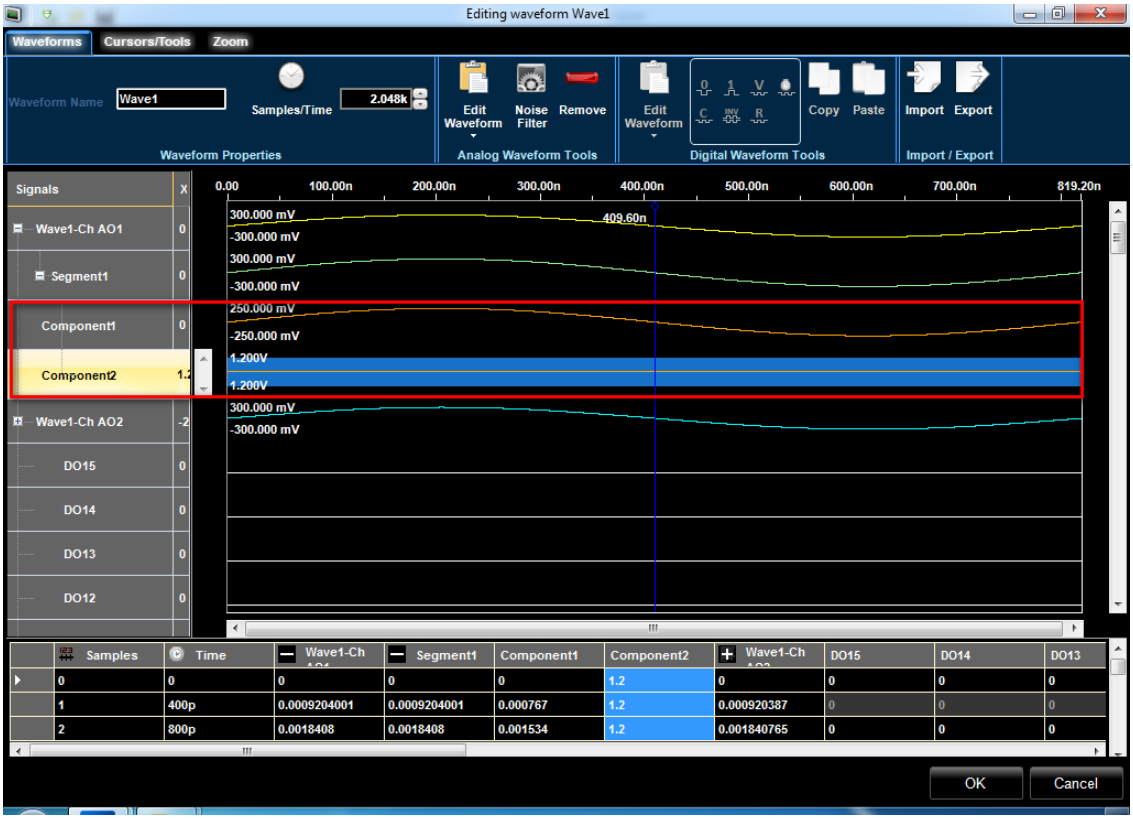


Figure 137 Edit waveform

8. Click OK to confirm
9. On the Settings pop-up screen's Run Mode tab, select Gated.
10. Click OK.

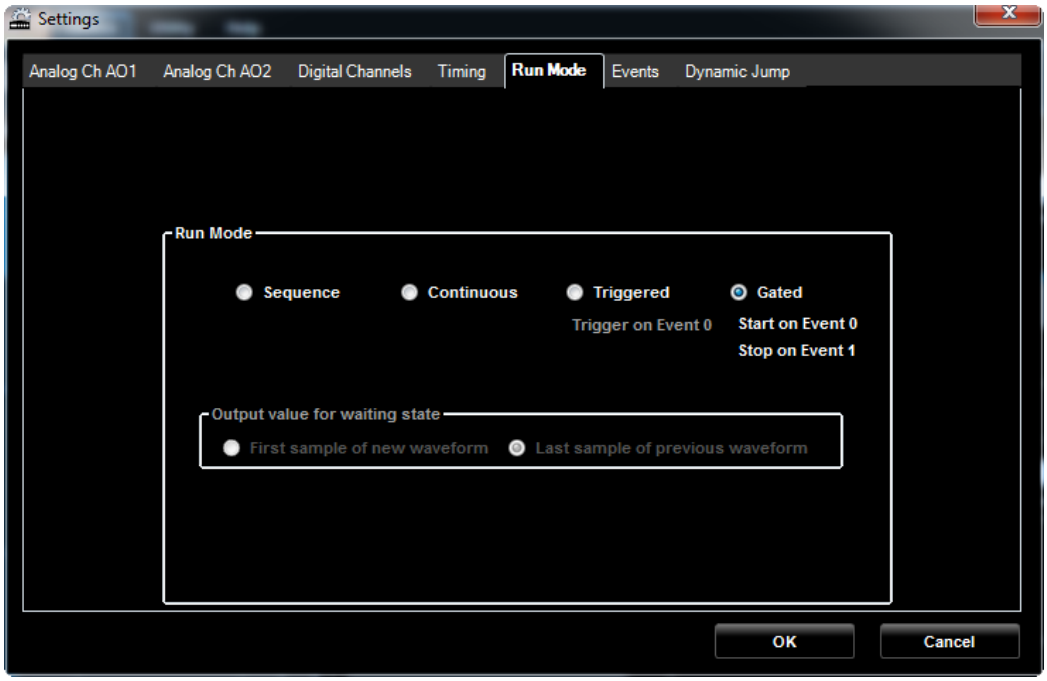


Figure 138 Setting screen

11. Drag the Wave1 from the Waveform Area to the first cell of the Sequence Area (the selected cell is highlighted).

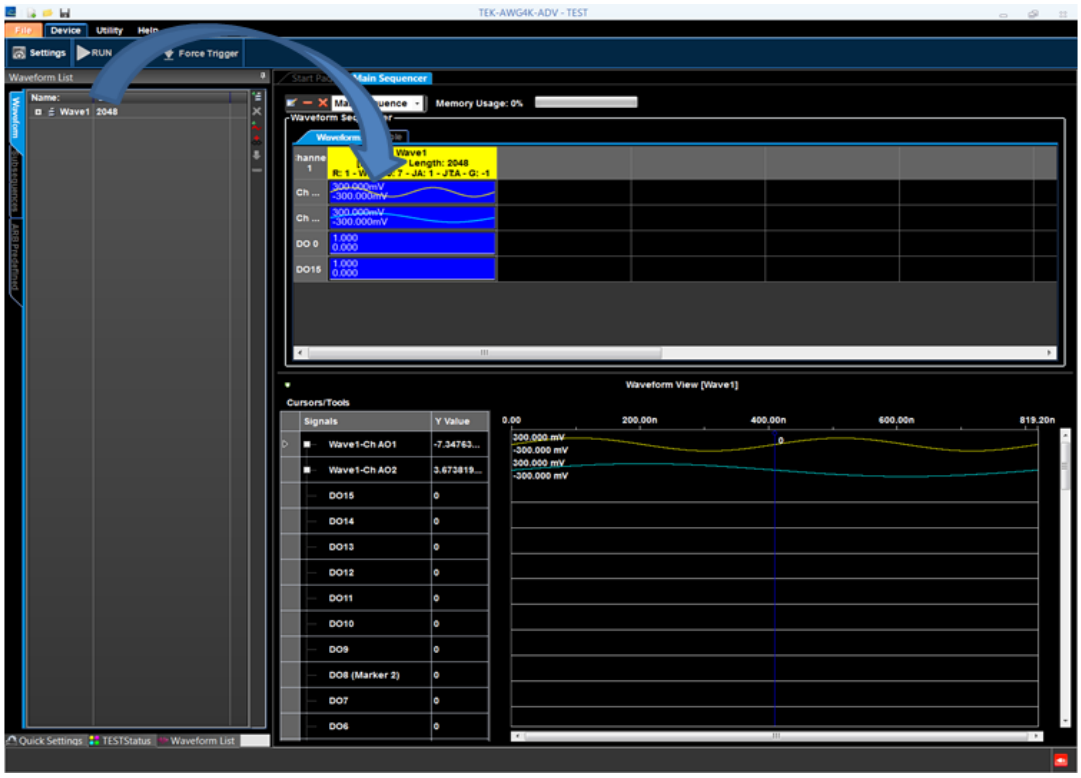


Figure 139 Edit waveform sequence

12. Now, press the Run/Stop button on the AFG4K Toolbar. The software loads the waveforms into the AFG4K instrument.

13. Keep pressed the ForceTrigger button to start the waveform generation.
    Wave1 is generated on the AO1 and AO2 SMA outputs.
14. Release the ForceTrigger button to stop the waveform generation.
    You can connect an oscilloscope to this output and analyze the signals.
15. Stop the instrument by pressing the Run/Stop button again.

Creating Digital Waveforms

The AWG4K may optionally be configured to work as a powerful Digital Pattern Generator. When the AWG4K runs in this mode it can emulate standard serial or parallel bus transitions or custom digital interfaces for system debugging and characterization.

After you have powered on the instrument , launch the software and use the menu bar to create a New Workspace.

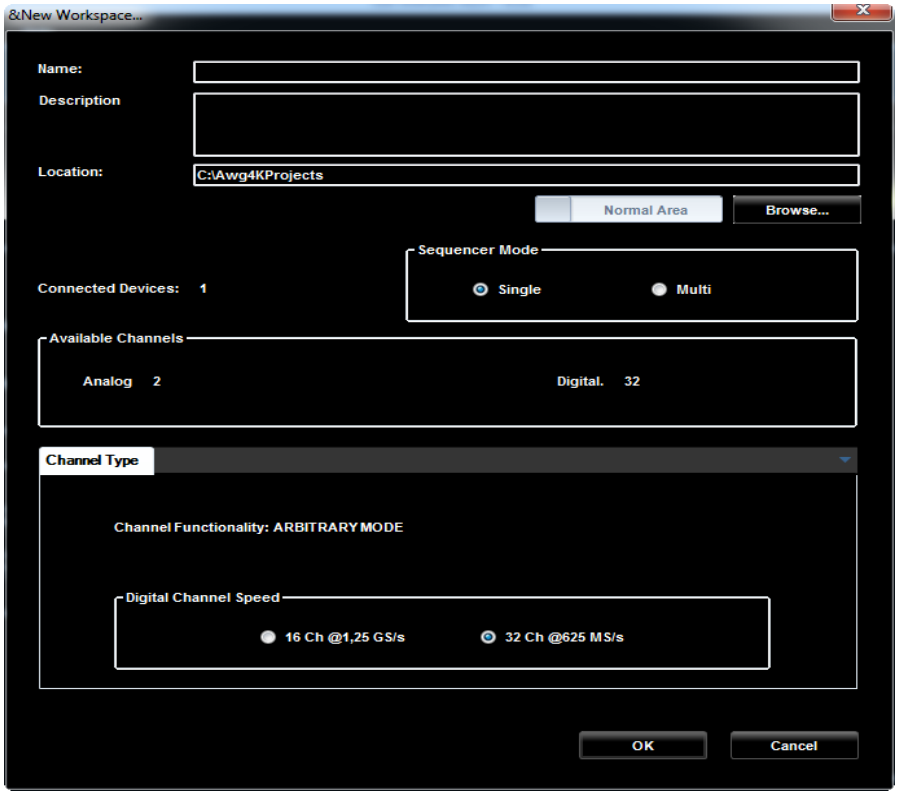


Figure 140 New workspace

NOTE:

The following steps are done with Pattern Generator operation enabled.

1. Type the Workspace name.
2. Select Single as Sequencer Mode.
3. Select Arbitrary as Channel Functionality.
4. Select 32Ch@625 MS/s as Digital Channel Speed.

NOTE:

Two Infinite band 12x connectors provide 16 bit LVDS digital outputs each for a total of 32 LVDS outputs. These digital outputs can be software configured to operate in different ways.

In ARB mode it is possible to operate with all of the 32 channels with a maximum update rate of 625MSps or with half channels (16) at 1.25Gsps.

5. Click OK.
6. On the main toolbar, press the Settings button. Then click the Group Digital Signals button.

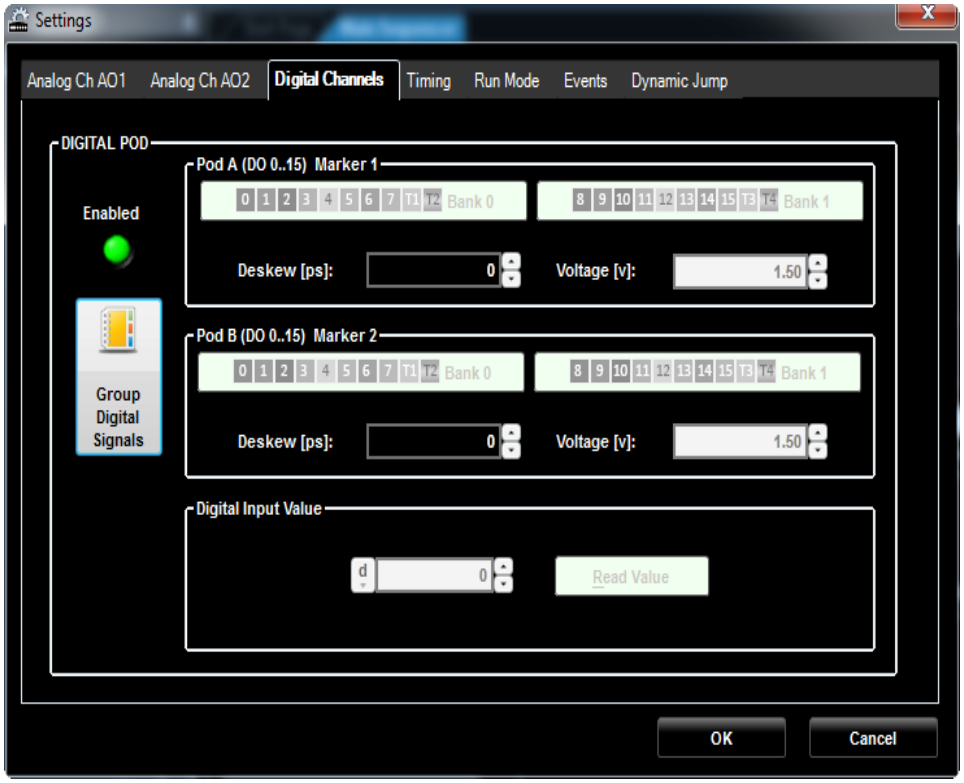


Figure 141 Settings

7. The Digital Logical name and Grouping window is shown.

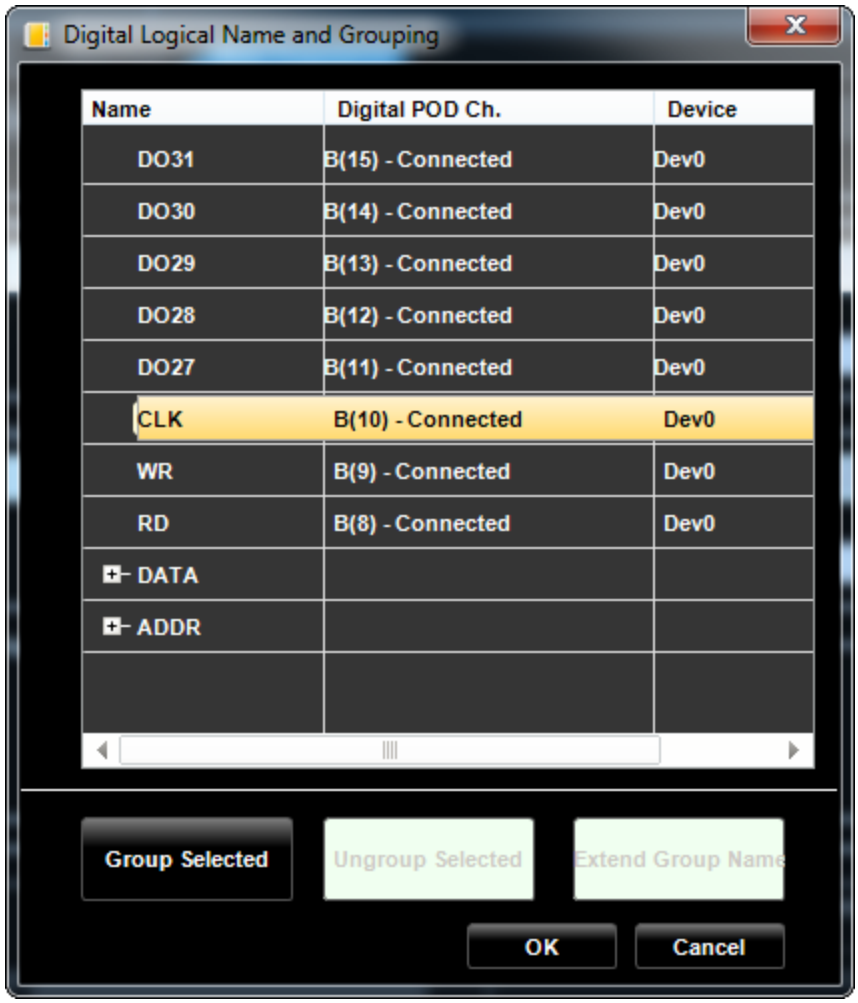


Figure 142 Digital Logical name and Grouping window

Now, let's make a bus by selecting DO0..DO7 on the channel list on the left, and then clicking the Group Selected. Keep pressing the left mouse button on the bus root name to change its name to ADDR.

Repeat the procedure for DO8..DO23 and change the group name to DATA.

Rename the DO24 to RD, the DO25 to WR and the DO26 to CLK.

8. Click the New Mixed Waveform button  .
9. The New Waveform window is shown. Type the name of the waveform “Wave1” and choose 10K for the samples length of the waveform. Click OK to confirm.

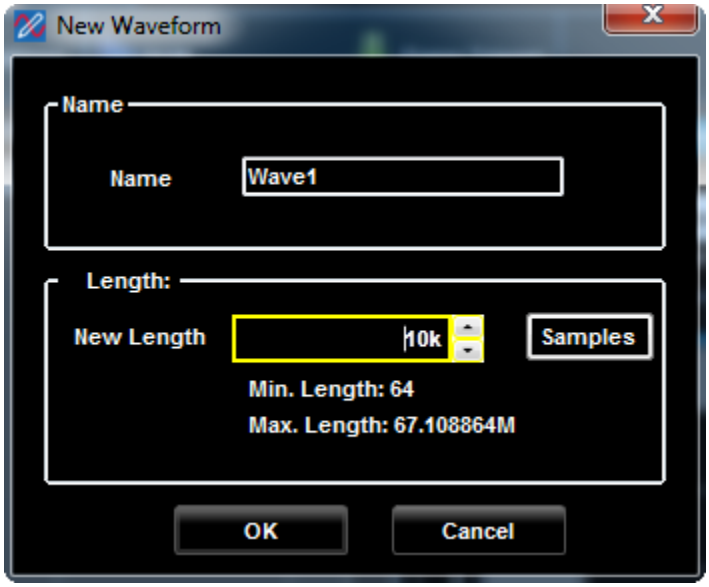


Figure 143 New waveform

10. The Editing Waveform Window is shown.

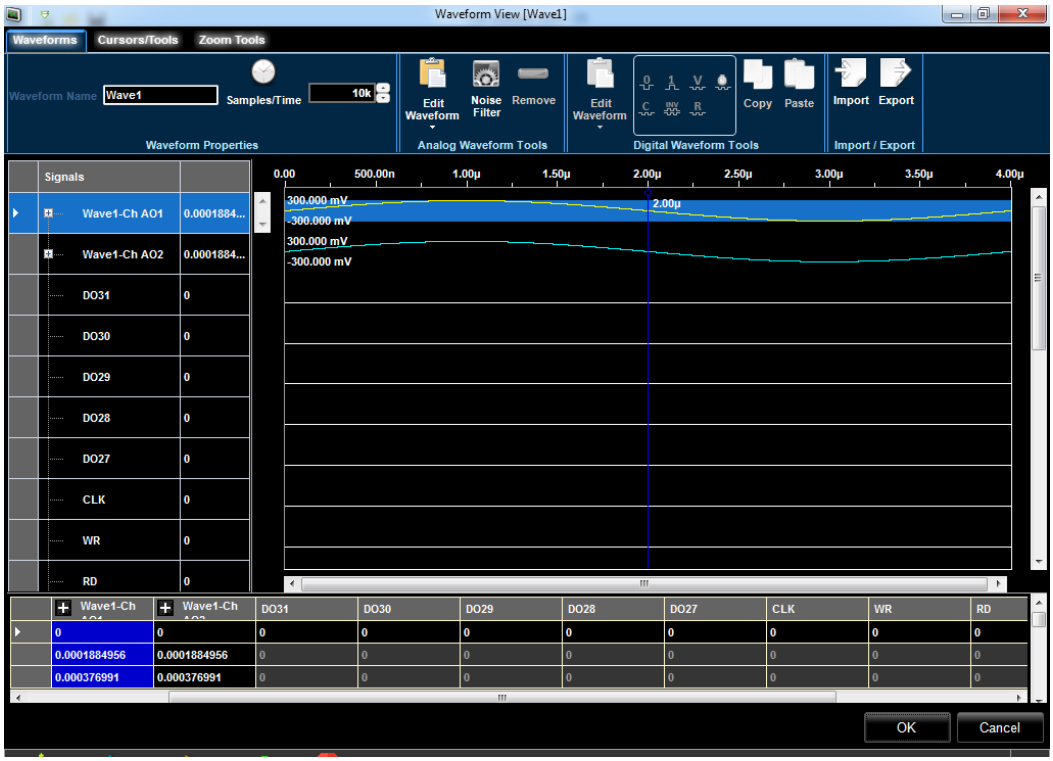


Figure 144 Editing waveform window

11. Select the CLK row and use the Clock Button to modify settings on the corresponding pop-ups.

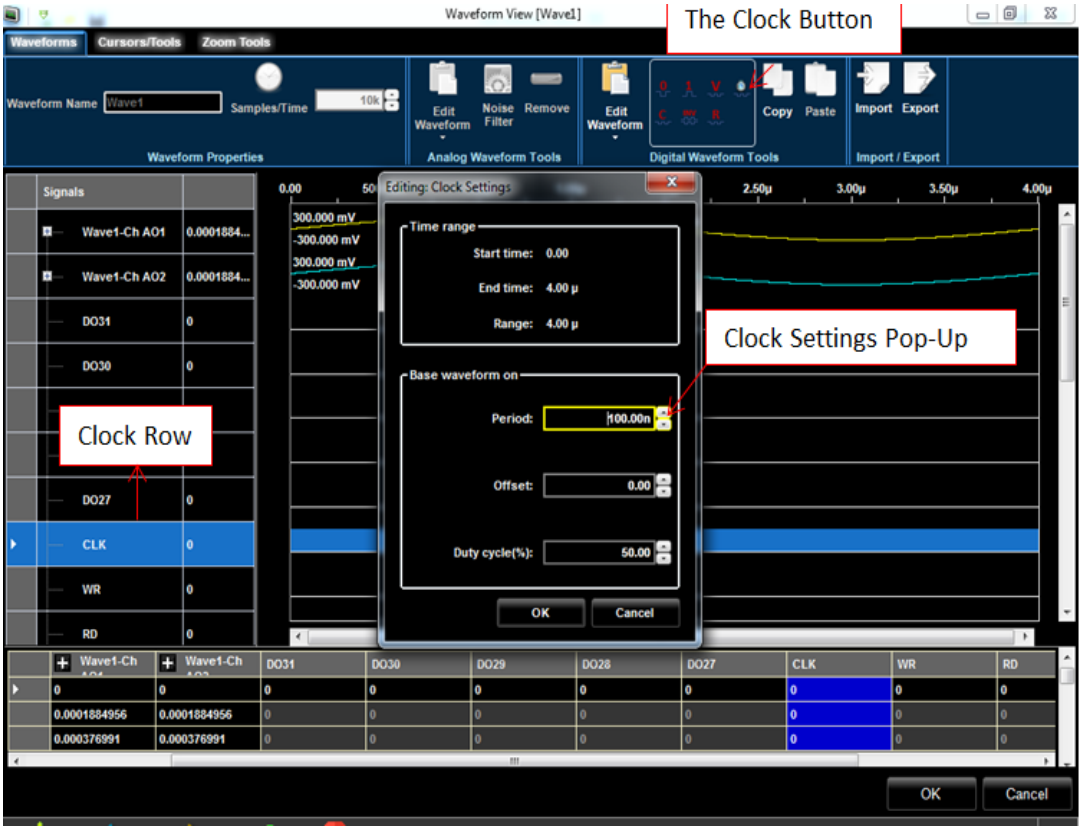


Figure 145 Editing waveform

12. Now, define WR and RD signals similarly, by selecting a portion of the digital channel graph, and clicking the desired button from the following choices:

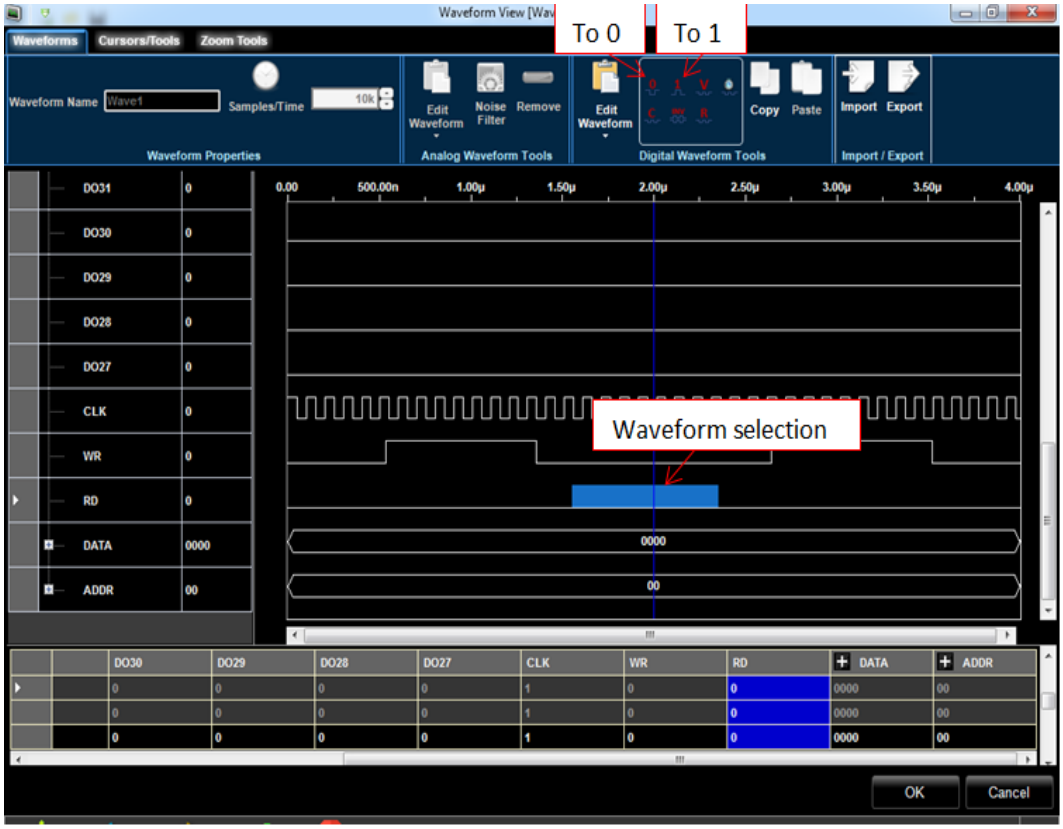


Figure 146 Editing waveform

13. Now define the ADDR bus by selecting the ADDR row and use the Counter Button to modify settings on the corresponding pop-ups.

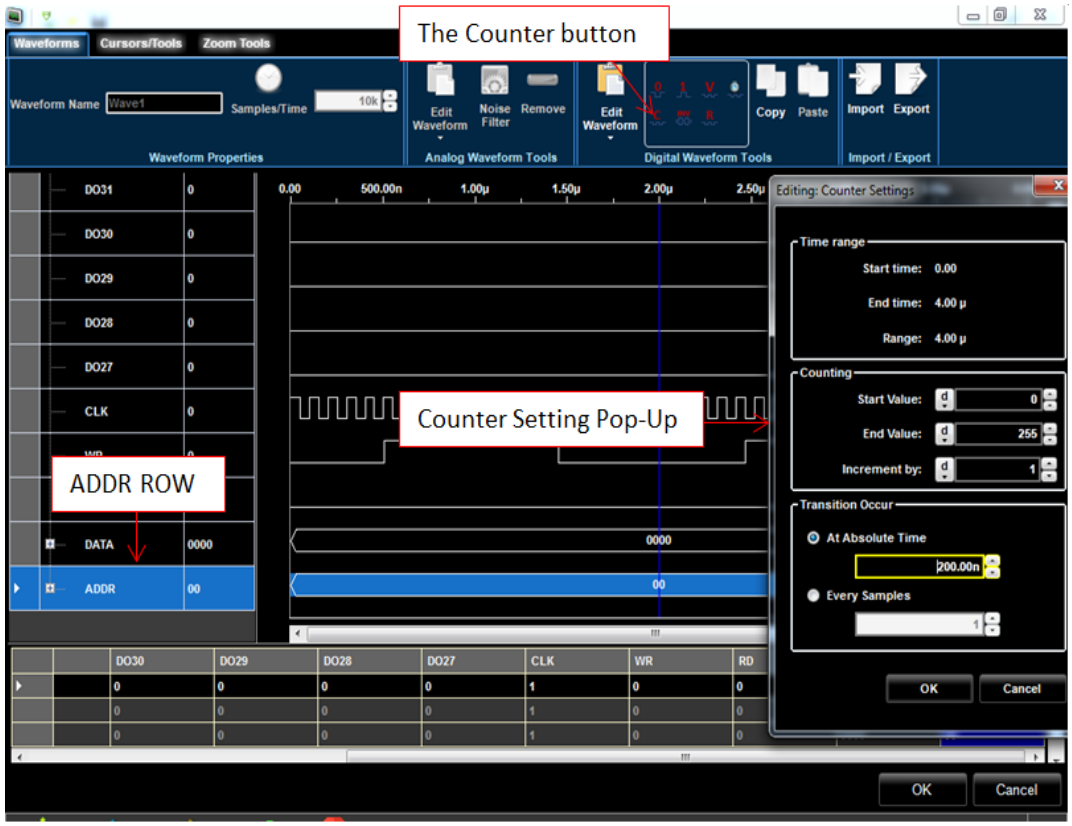


Figure 147 Define the ADDR bus

14. Now, define DATA bus signals, by selecting a portion of the data bus graph, and clicking the value button:

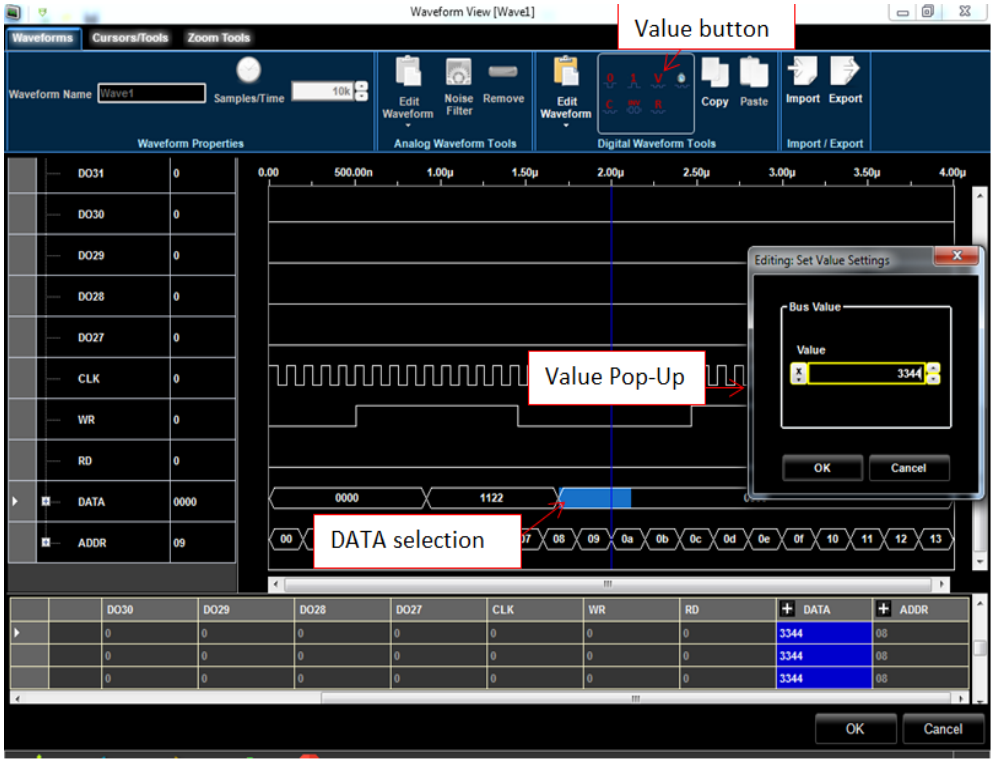


Figure 148 Define the DATA bus

15. Click OK to close the Editing Waveform Window.
16. On the Settings pop-up screen's Run Mode tab, select Continuous. Click OK.

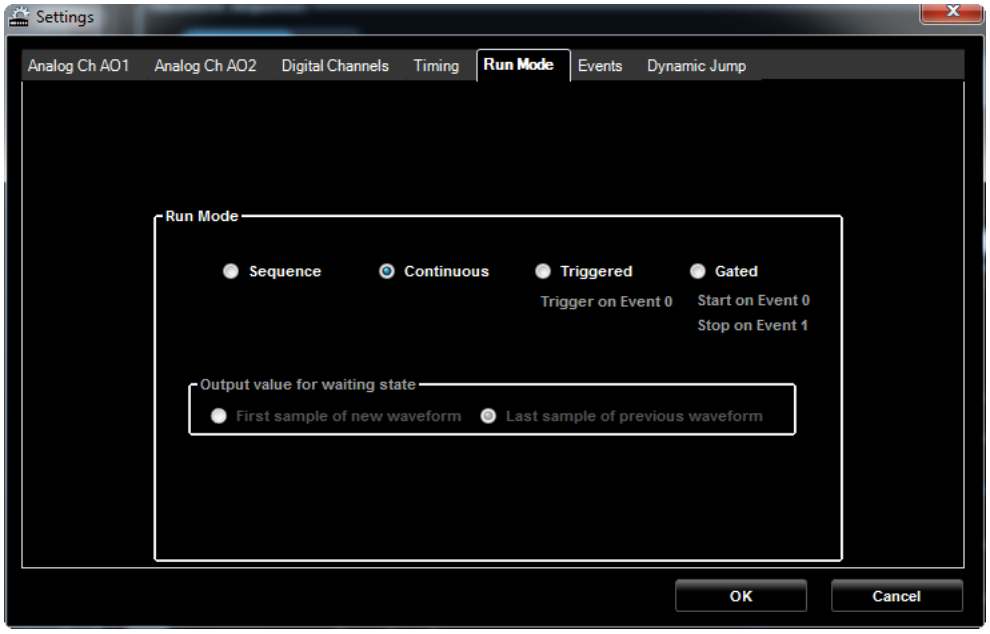


Figure 149 Editing waveform

17. Drag the Wave1 from the Waveform Area to the first cell of the Sequence Area (the selected cell is highlighted).

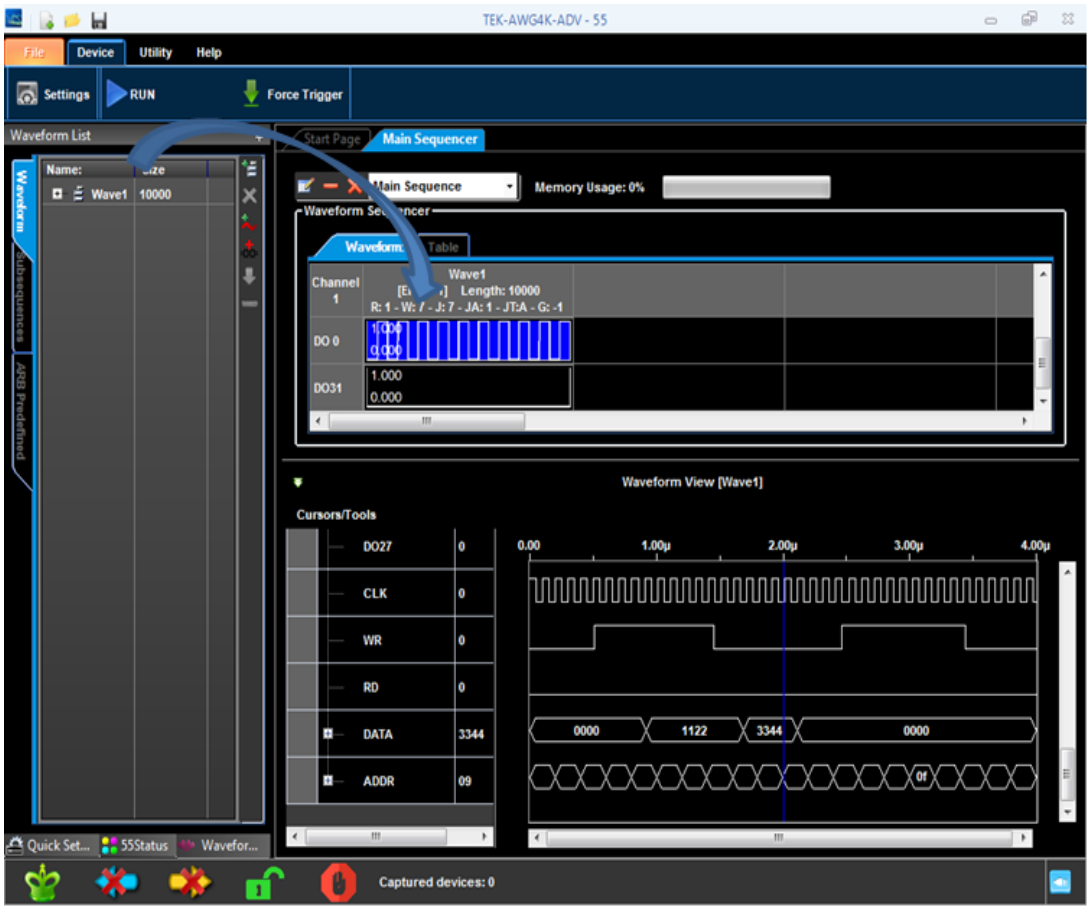


Figure 150 Drag Wave1

18. Press the Run/Stop button on the AWG4K toolbar.

The software loads the waveforms into the AWG4K instrument and starts generating the waveforms.

Wave1 is generated on the Pod A and Pod B probes. You can connect a logic analyzer and analyze the generated signals.

Wave1 analog signals are generated on AO1 and AO2 SMA outputs.

You can connect an oscilloscope to this output and analyze the signals.

19. Stop generating the waveforms by pressing the Run/Stop button again.

Common Tasks Examples

In addition to these **How do I Scenario Details**, there are some common tasks for your reference.

1. **Creating a New Workspace**
2. **Opening an Existing Workspace**

Pay attention to the following:

Some more specific steps are required when creating a new workspace for each scenario. Details are provided for those scenarios when necessary.

Create a New Workspace

1. Click File->New Work Space
2. Type the Workspace name
3. Select Location. There are two locations: Normal Area and Security Area. If you select security area, when you execute “Secure” function in Basic App, all the data in security area will be erased completely.
4. Select Single as Sequencer Mode
5. Select 16Ch@1.25 GS/s as Digital Channel Speed.

NOTE:

Two Infiniband 12x connectors provide 16 bit LVDS digital outputs each for a total of 32 LVDS outputs. These digital outputs can be software configured to operate in different ways. The digital channels are available with Digital Option installed only.

It is possible to operate with all of the 32 channels with a maximum update rate of 625MSps or with half channels (16) at 1.25Gsps.

6. Click **OK**.

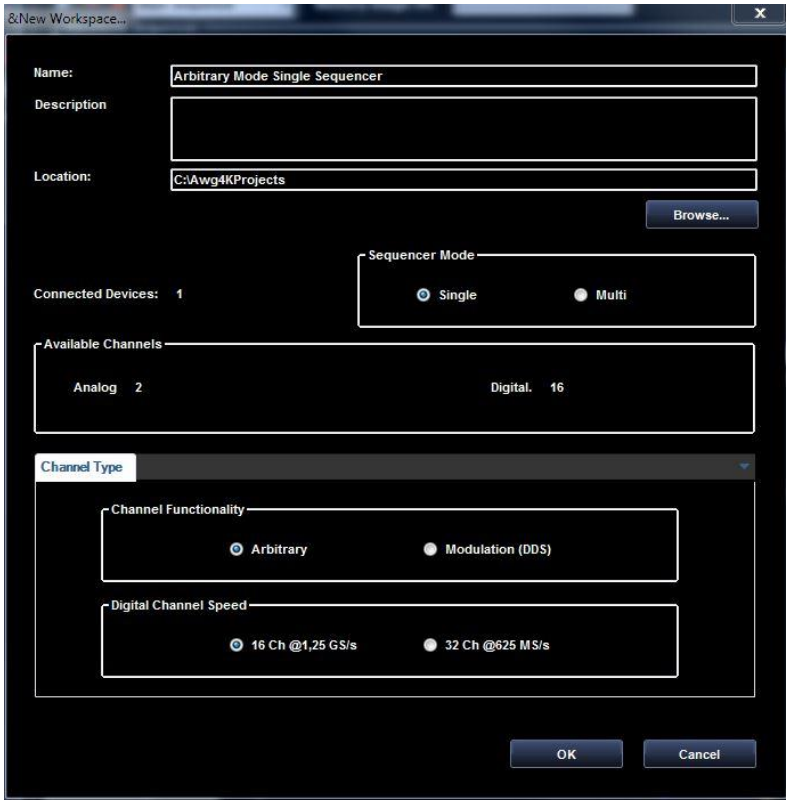


Figure 151 Setting the new waveform

7. Click the **New Mixed Waveform** button.

8. The **New Waveform** window is shown. Type the name of the waveform “Wave1” and choose 2048 for the samples length of the waveform. Click **OK** to confirm.

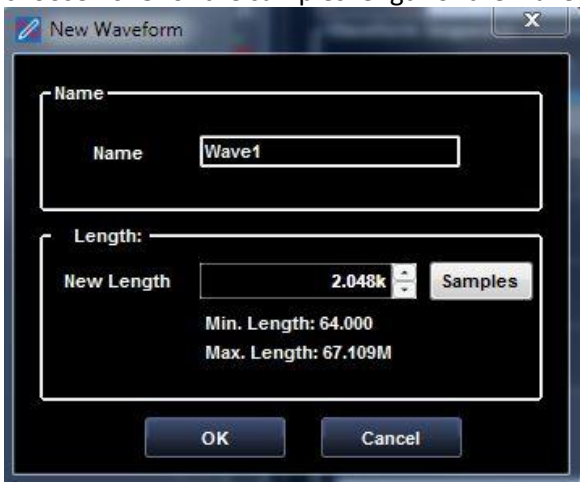


Figure 152 New waveform

9. The **Editing Waveform** window is shown. Select the waveform Wave1-0 and click on the **Edit**  button.

Figure 153 Editing waveform

10. The **Waveform Standard Editor** is shown. Choose a sine waveform with the following specs:
    - Cycles: 2
    - Amplitude[V]: 250mV
11. Press **OK** button.
12. Select the waveform Wave1-1 and click on the **Edit**  button.
13. The **Waveform Standard Editor** is shown. Choose a triangle waveform with the following specs:
    - Cycles: 4
    - Amplitude[V]: 1V

Figure 154 Waveform standard editor

14. Press **OK** button.
15. Press **OK** button on the **Editing Waveform** window; the *Wave1* will appear on the Waveform TAB.
16. Click the  **Settings** button. The Settings window is shown.

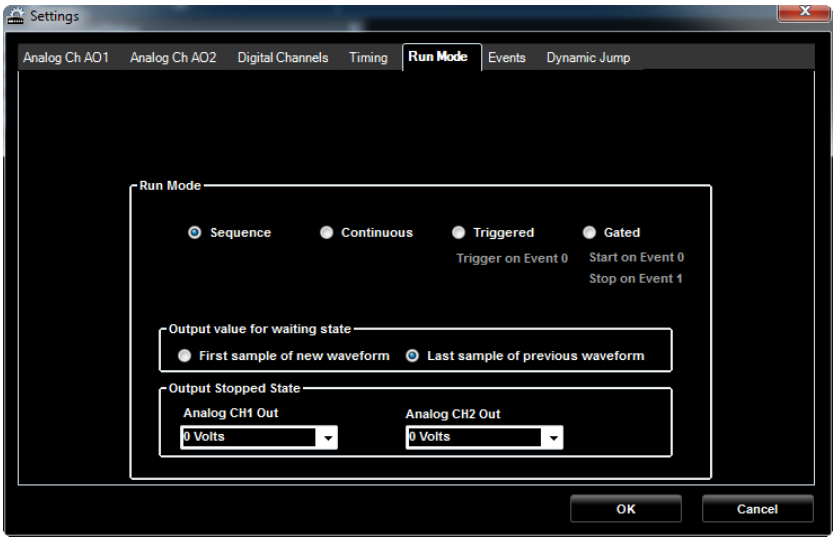


Figure 155 Settings window

17. Select **Sequence** as Run Mode
18. Select the *Analog Ch AO1* TAB and select **DC Direct** as Out Type

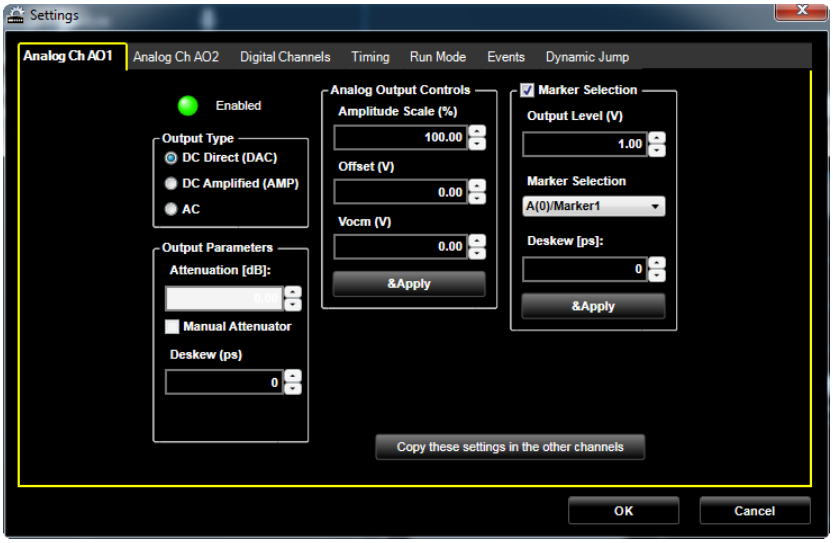


Figure 156 Settings window

19. Select the *Analog Ch AO2* TAB and select **DC Amplified** as Out

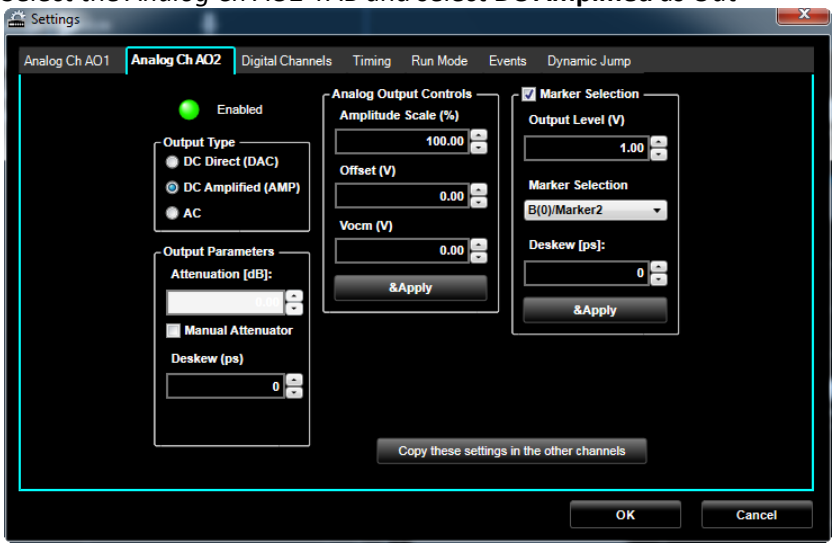


Figure 157 Settings window

20. Press **OK** button.
21. Drag the *Wave1* from the Waveform Area to the first cell of the Sequence Area (the selected cell is highlighted).
22. The **Input Waveform Properties** is shown. Click the  button to *set infinite repetitions* on Wave1. The item flowchart will help you to understand the correct behavior of the instrument.

Figure 158 Input window properties

23. Press **OK** button.
    The Sequence Area shows now the Wave1 inserted in the first cell. If you select one of the waveform, it appears on the *Waveform Display Area* placed below.

25. Press the **Run/Stop** toolbar button.

NOTE:

Once the instrument has started, Sequence Run Mode with infinite repetitions, repeats Wave1 until the Run/Stop button is clicked again.

The software loads the waveforms into the AWG4162 hardware and then generates waveforms.

26. **Wave1 is** exported to the **CH1/CH2 SMA output**, which can be connected to an oscilloscope for signal analysis. The output are 50ohm single ended or 100ohm differential, oscilloscope termination must be set accordingly to ensure proper observation.

Open an Existing Workspace

1. Open existing workspaces by clicking the **Open Workspace** toolbar button. The Open Workspace screen is shown.
2. The Open Workspace screen automatically navigates to the Awg4KWorkspace folder. Select the workspace and click **Open**.

Demo Project

1. The AWG4162 setup automatically installs under the folder C:\Program Files(86)\Tektronix\AWG4000 Advanced\DemoWorkspace several demo workspace that can help you to understand more in depth all the instrument features.
2. Note that the Demo Workspace are configured for a full optional instrument (64MS/CH and 32 DIOs), so if you try to open them with a connected instrument with less options, you will receive an error message.

Option Installation

There are some options. They relate with Memory size and Digital bits:

MEM64
MEM32
MEM16
MEM01

DO32
DO16

Use the License Manager dialog box to enable the upgrades that you purchased from Tektronix for your instrument. For the most current list of upgrades, go to www.tektronix.com or contact your local Tektronix representative.

1. Press **Help** > **License** button to open the License Manager dialog box.
2. The dialog box will show you the current license codes loaded into the instrument.

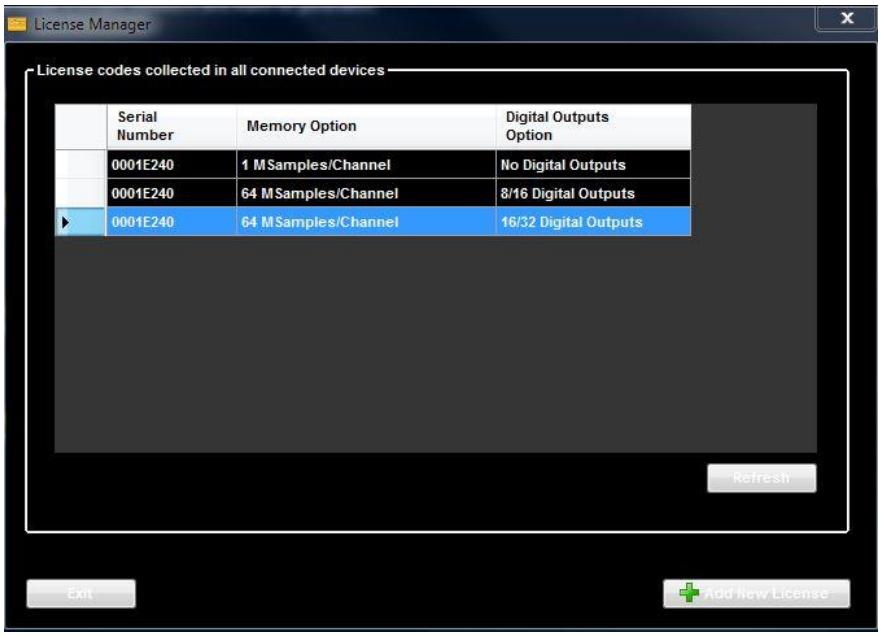


Figure 159 License codes

3. Press the Add New License to open Add New License Code dialog box.

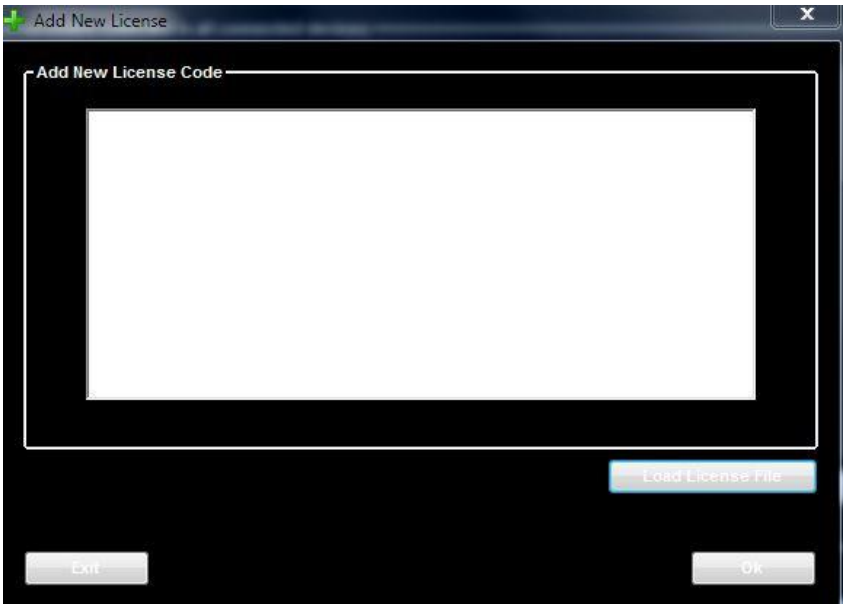


Figure 160 Add a new license

4. Enter the Option Installation Key provided by Tektronix or load it form file using the Load License File button.

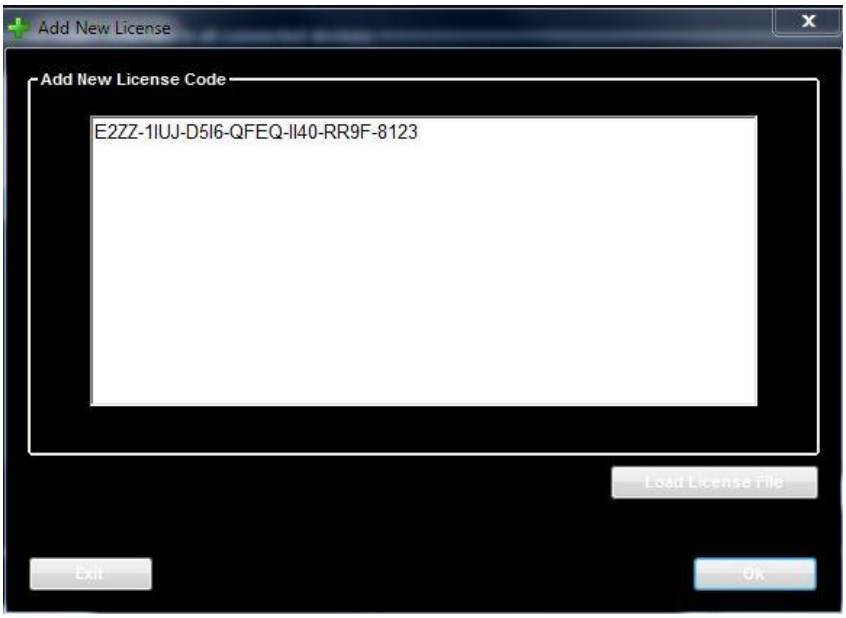


Figure 161 Load a license

5. Press the OK button to load the new license code into the instrument.

Multi-Instrument System

You can connect two AWG4162 units to have a system with 4 synchronized analog channels and up to 64 digital channels.

In order to set up a Multi-Instrument system you must first:

- Turn off the instruments.
- Select the instrument that you want to use as Master, the other unit will be considered as Slave.
- Using the Sync Cable, connect the Master **Sync Out** connector to the Slave **Sync In** connector you can find on the rear of the instruments.

Figure 162 Rear of the instrument

Figure 163 Connecting units

- Turn on the instruments.
- Launch AWG4162 Advanced software

NOTE:

1. *The Multi-Instrument System is available with Single Sequencer workspace only.*
2. *Before connecting or disconnecting the sync Cable, you must turn off the instruments.*
3. *The waveform length and sequencer must be the same in all the instruments (master+slave) connected using the Multi-Instrument System sync feature.*
4. *The external sampling clock and external reference clock are available in Master device only.*

The following steps describe the steps that you should perform to set up a Multi-Instrument project and start the generation on two devices.

1. On Master and Slave unit, launch the Advanced software and create a Single Sequencer project.
2. The communication between Master and Slave has started: on the Master device the Capture button  is enabled.

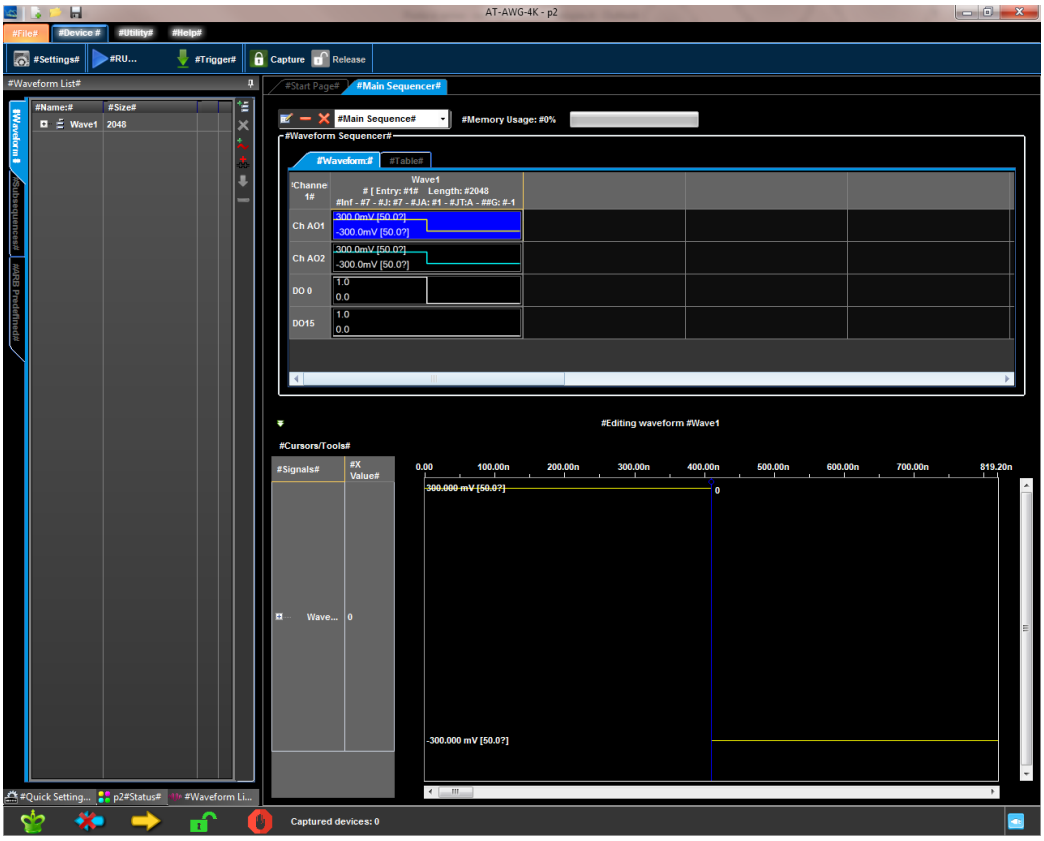


Figure 164 Master device

3. Create one Mixed waveform (square wave, 0.6Vpp, 2048 points) on Master and Slave units.

4. Add the “Wave1” waveform into the sequencer.

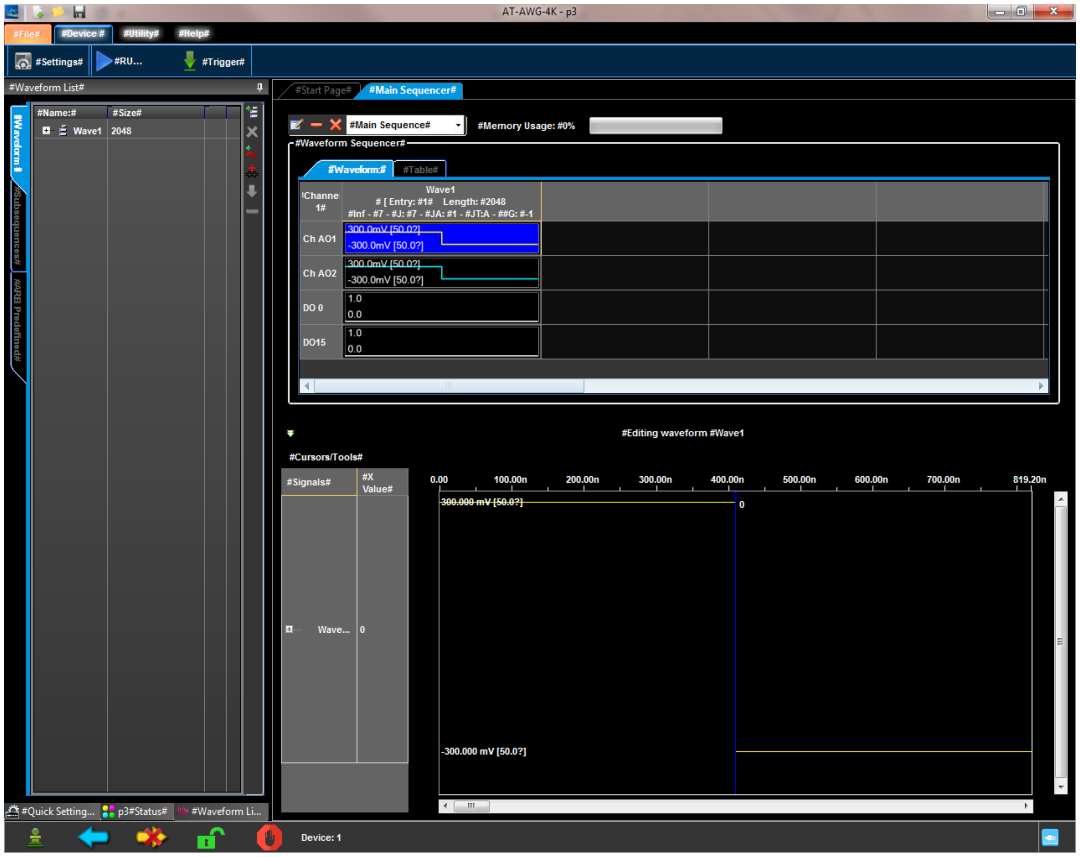


Figure 165 Slave device

5. Even if Sync Cable has been connected, at the start-up the two devices works independently and you can use them as two separate devices; in this case the “crown” icon  will appear both in Master and  icon will appear Slave unit on the Multi-Instrument bottom status bar. The Slave unit has not been captured yet, so  icon will appear on the status bar.

   On the Multi-Instrument bottom status bar there are icons that represents the status of the Multi-Instrument system.

	It represents the Master device.
	It represents the Slave device.
	There is another unit connected backward to the current instrument.
	There is another unit connected forward to the current instrument.
	There is not another unit connected backward to the current instrument.

	There is not another unit connected forward to the current instrument.
	The Slave device has been captured by the Master.
	The Slave device has not been captured by the Master.
	The instrument is stopped.
	The instrument is running and the waveform generation has started.

6. Press the capture button on the Master device to link the two instruments:

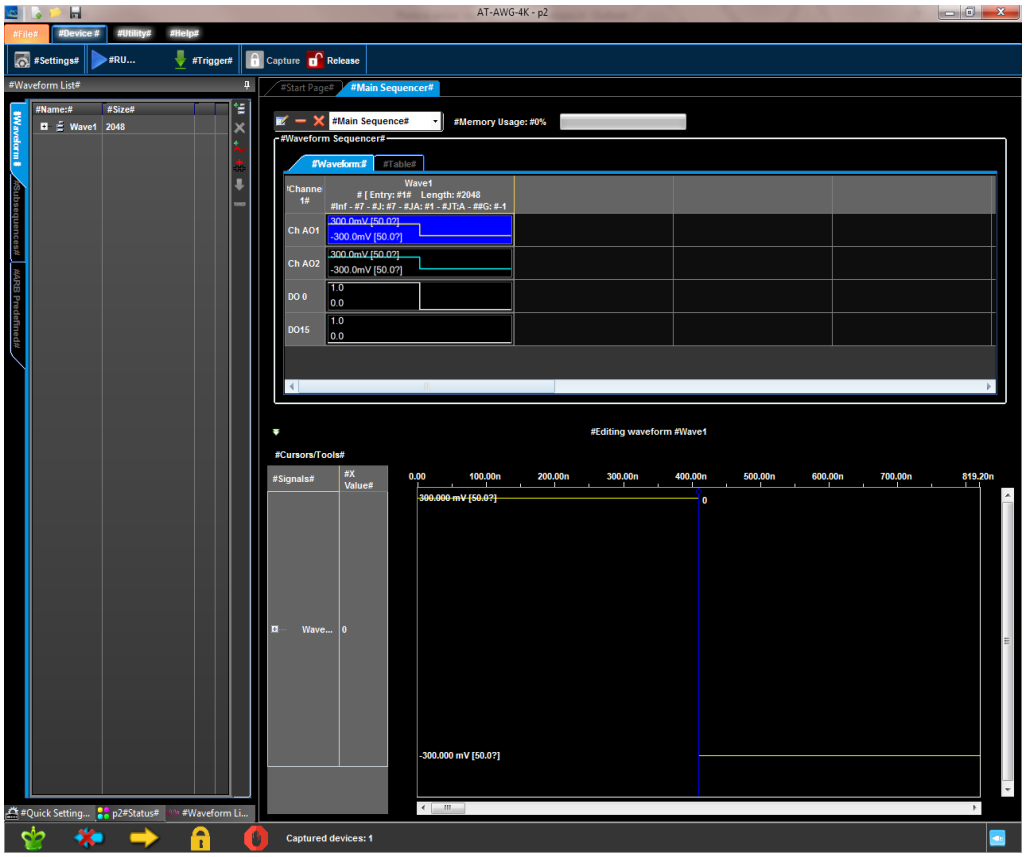


Figure 166 Capturing a slave device

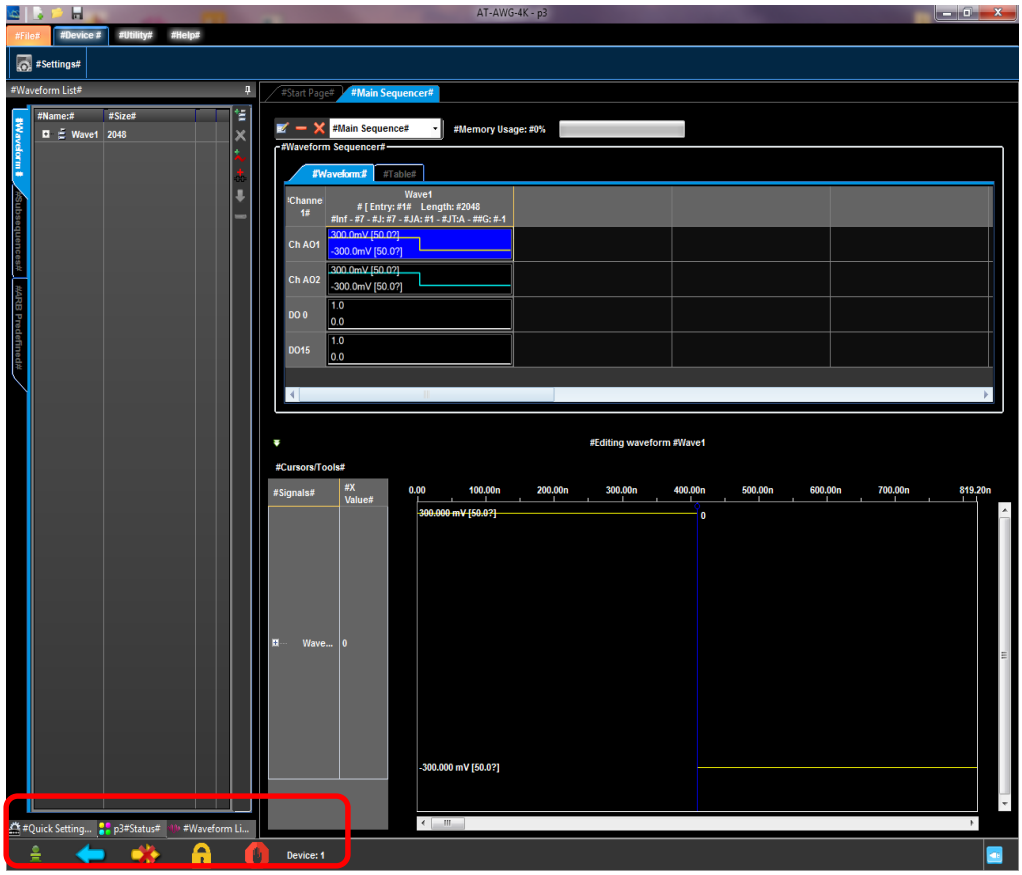
7. 

Figure 167 A slave device captured

If you check the Salve Multi-Instrument Status bar, you will notice that the  icon indicates that you are on the Slave device, the blue arrow tell you that there is a device connected backward (the Master) and the  icon indicates that the Slave has been captured by the Master. The Slave instrument is stopped  .

The two instruments now works linked together and the four channels will be synchronized; the Run/Stop button on the Slave device disappears because the Run/Stop will be controlled by the Master, as shown in the following figure, moreover the Timing settings and the Events settings are not available in the Slave since the sampling rate and the events are both Master controlled.

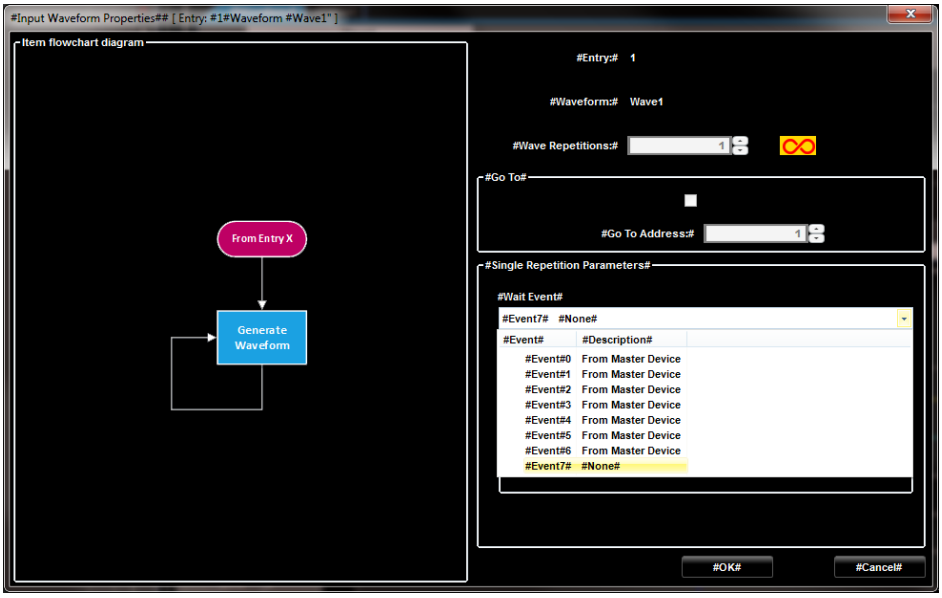


Figure 168 A slave device controlled

8. Press the RUN button on the Master device, the "Load configuration …" form will appear: this form informs you that the Slave instrument (Device 1) is loading its configuration in terms of settings, sequencer and waveforms and as soon as it will be ready, the Start button will be enabled.
   Please note that the "load configuration" procedure may fail due to wrong settings on the slave device, but this will not block the possibility to start the instruments indeed.

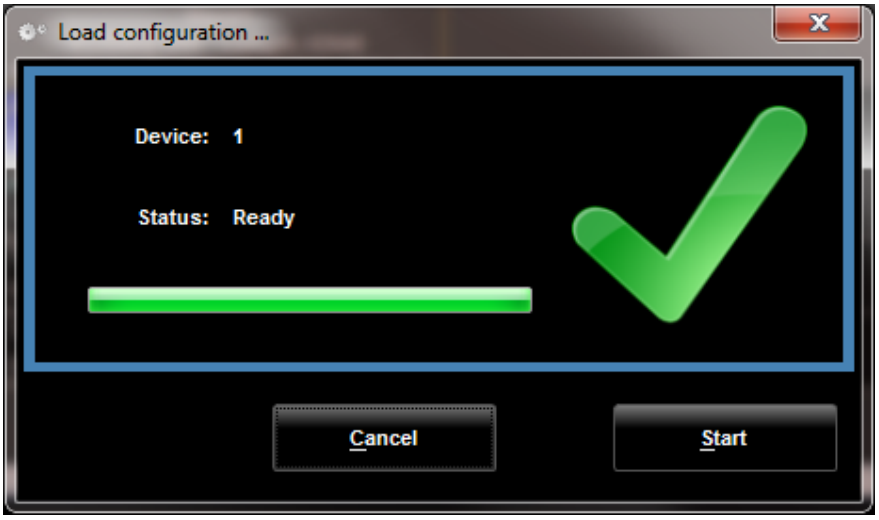


Figure 169 Load configuration

9. Press the Start button to start the waveform generation on Master and Slave device.

   If you need the instruments to work as two independent devices, you should press the Release button  on the Master device. After you press it, the Run/Stop button, the events and the timing settings will appear again on the Slave unit.

Appendix

A. Digital Outputs

The AWG4000 can output 16-bit (from Pod A) or 32-bit (from Pod A and B) of digital patterns with option DO16 or DO32. All bits are differential pairs in LVDS. The digital outputs can be configured as high speed or low speed mode in the Advanced Mode application.

In high speed mode, only the 8 lsb's (D0-D7) are available on each pod. The bit rate is half of the sampling rate (for example, 1.25 Gb/s at 2.5 GS/s sampling rate).

In low speed mode, all 16 bits on each pod are available. The bit rate is a quarter of the sampling rate (for example, 625 Mb/s at 2.5 GS/s sampling rate).

To ensure the best signal integrity when transmitting such high speed digital signals, a customized digital cable (AWG4DIG16LVDS) and the corresponding connector (AWG4DIGSCKT) mounted on the DUT are required.

The AWG4DIGSCKT is an off-the-shelf part from Amphenol with part number U65-B12-41E0C, which a user can either buy from Tektronix, Amphenol or its distributors.

The below graph and table show the PCB layout and pin definitions of the AWG4DIGSCKT mounted on the DUT. Please follow the manufacturer's guidelines for PCB and mechanical designs

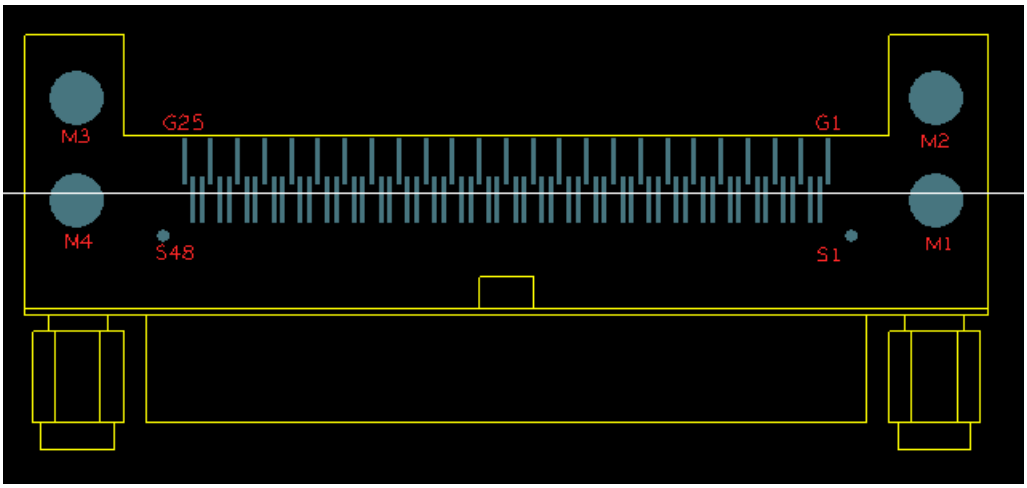


Figure 170 PCB layout

Pin	Description
S1	NC
S2	NC
S3	NC
S4	NC
S5	NC
S6	GND
S7	NC
S8	GND
S9	D15_P
S10	D15_N
S11	D14_P
S12	D14_N
S13	D13_P
S14	D13_N
S15	D12_P
S16	D12_N
S17	D11_P
S18	D11_N
S19	D10_P
S20	D10_N
S21	D9_P
S22	D9_N
S23	D8_P
S24	D8_N
S25	D7_P
S26	D7_N
S27	D6_P
S28	D6_N
S29	D5_P

S30	D5_N
S31	D4_P
S32	D4_N
S33	D3_P
S34	D3_N
S35	D2_P
S36	D2_N
S37	D1_P
S38	D1_N
S39	D0_P
S40	D0_N
S41	NC
S42	GND
S43	NC
S44	GND
S45	NC
S46	NC
S47	NC
S48	NC
G1~G25	GND
M1~M4	GND

Note: All NC pins MUST be left floating

B.Touch Panel Calibration

1. Double click “Microchip AR Configuration Utility” on the desktop
2. Choose “Configuration Wizard”

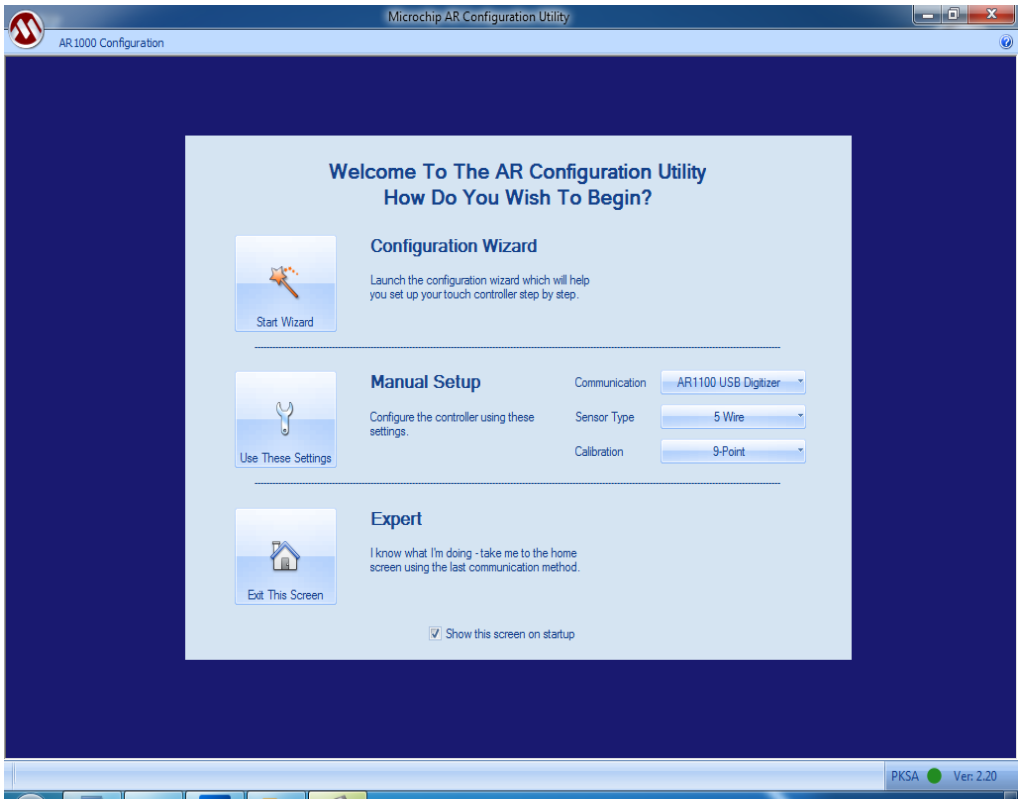


Figure 171 Microchip AR Configuration Utility

3. After click next will show below dialog, please choose “AR1100 Dev Kit”

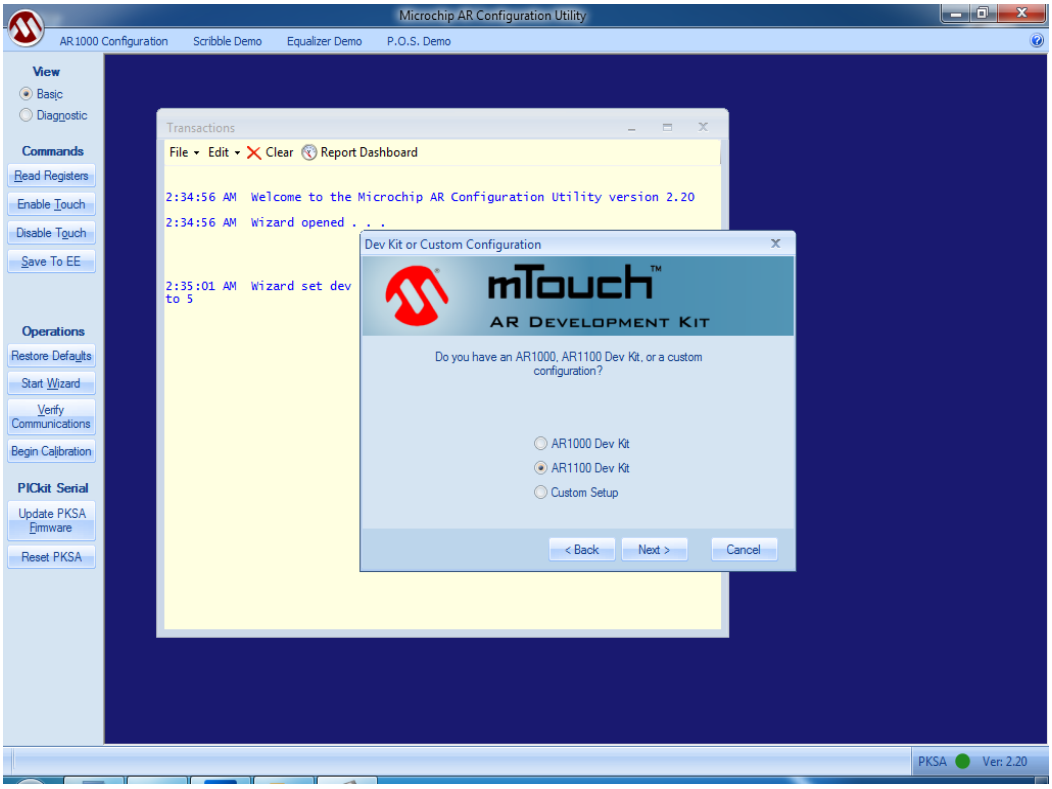


Figure 172 Choose “AR1100 Dev Kit”

4. After that will show the dialog as below, please choose “USB” and “Digtizer”, and please wait.

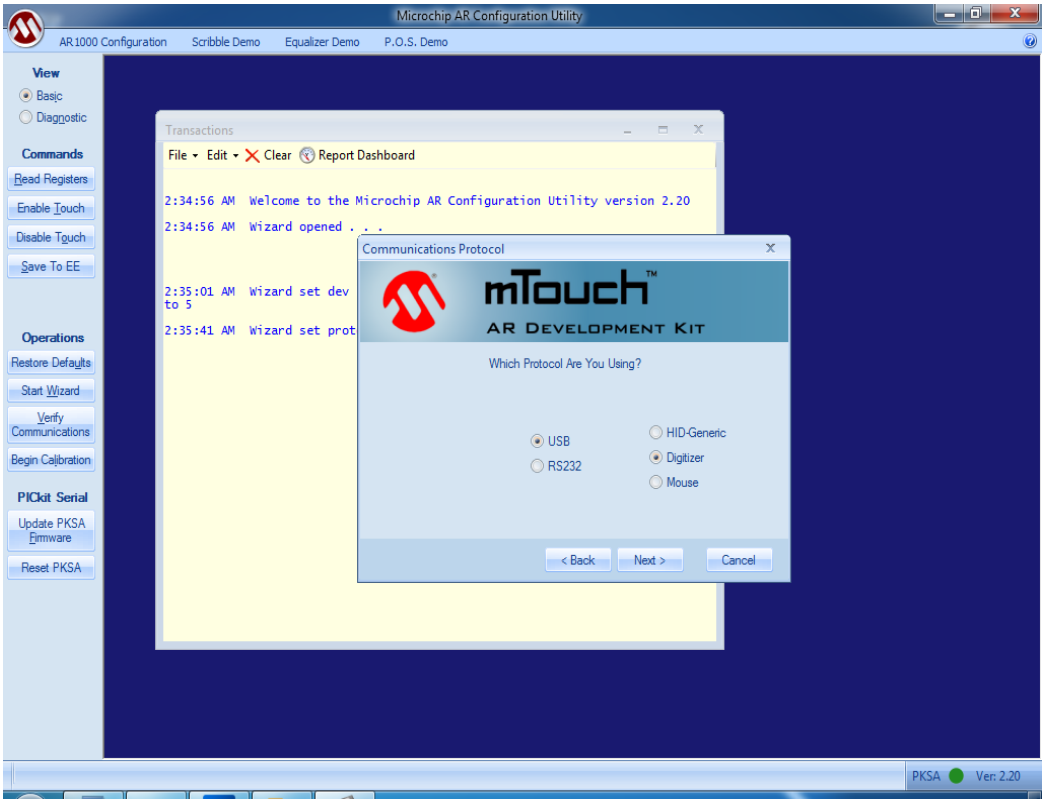


Figure 173 Choose “USB” and “Digtizer”

5. Please click next until show below dialog, after that please click “Next”, and Finished

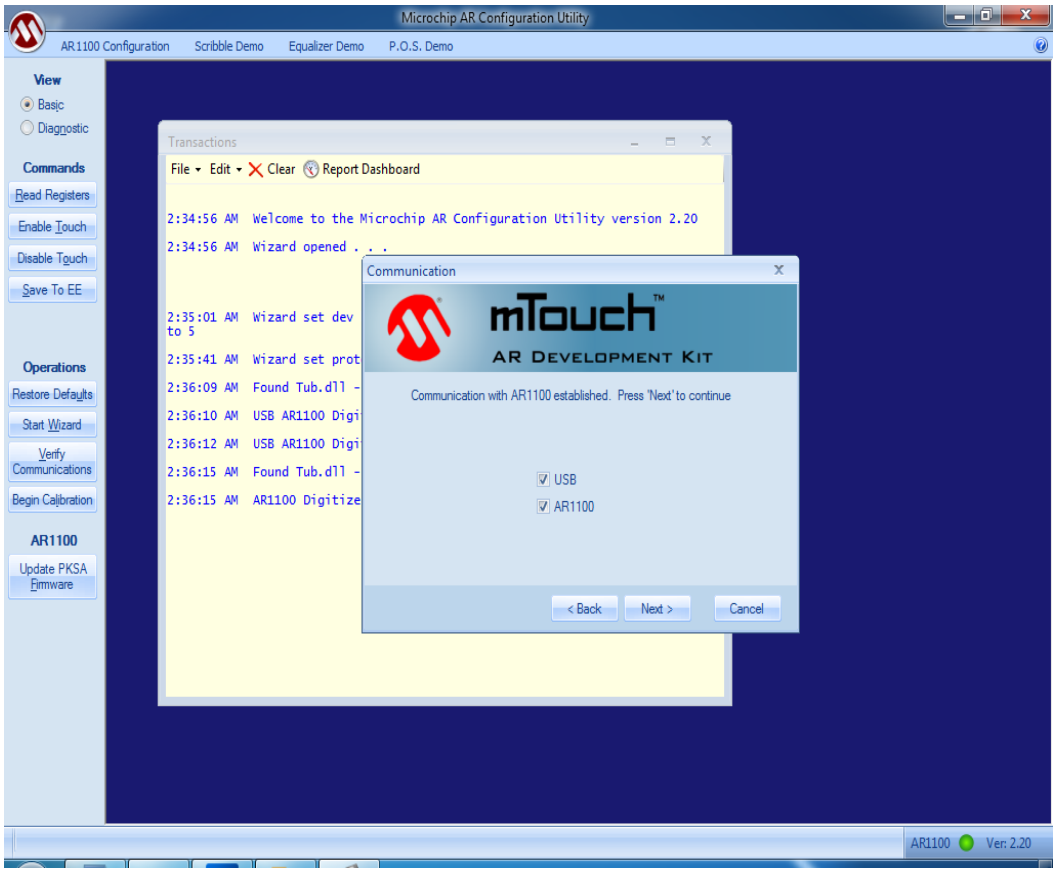


Figure 174 Finish configuration

6. Please close this application.

7. Please double click “Microchip AR Configuration Utility” on the desktop again, and choose “Manual Setup”

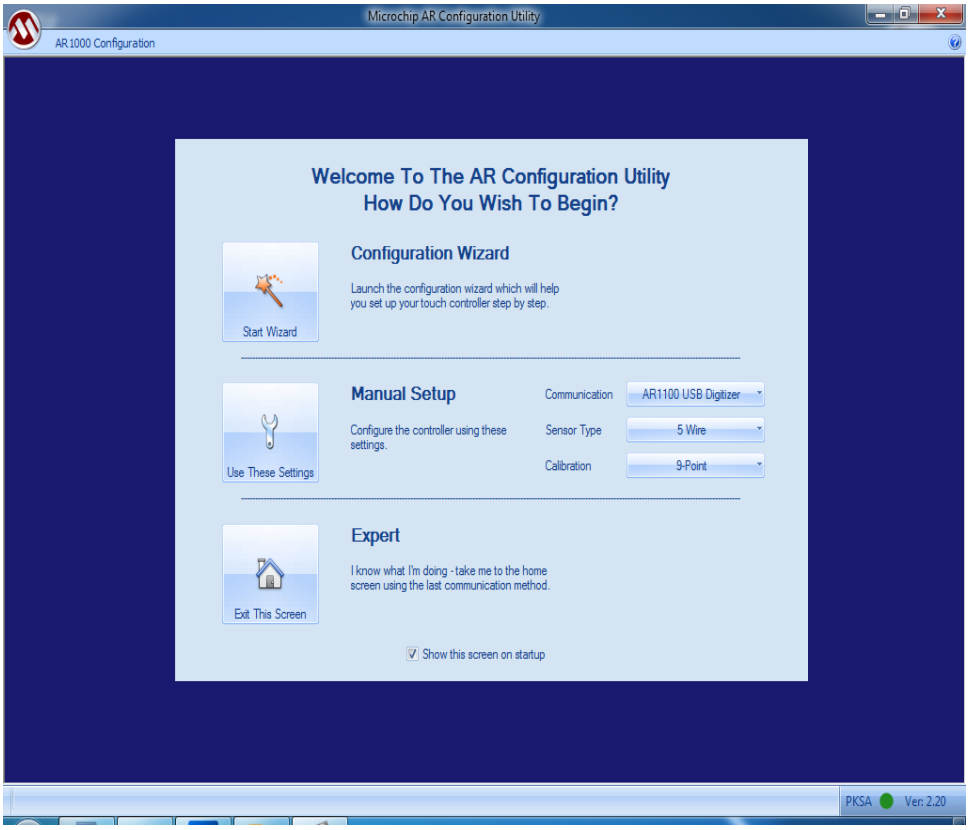


Figure 175 Manual setup

8. It will show below picutre that let you click the plus to calibrate touch panel.

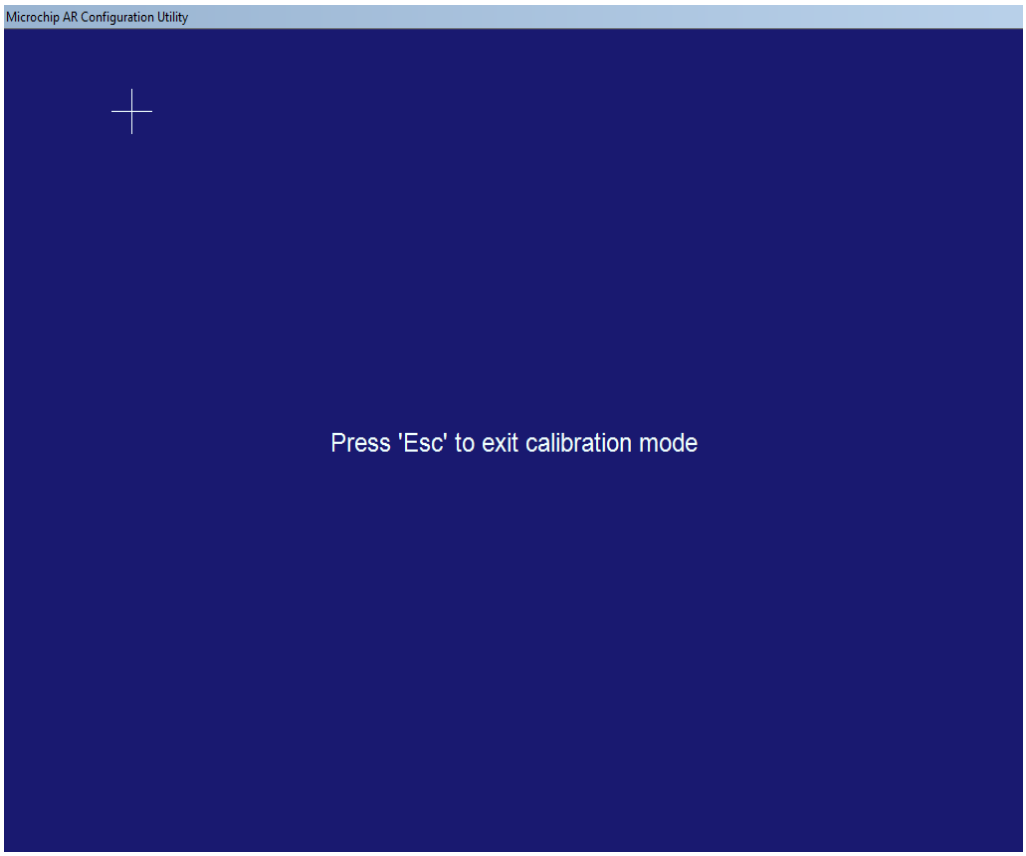


Figure 176 Click the plus to calibrate touch panel

9. After touch all the pluses , when it will show below picutre means you can close application and complete the calibration.

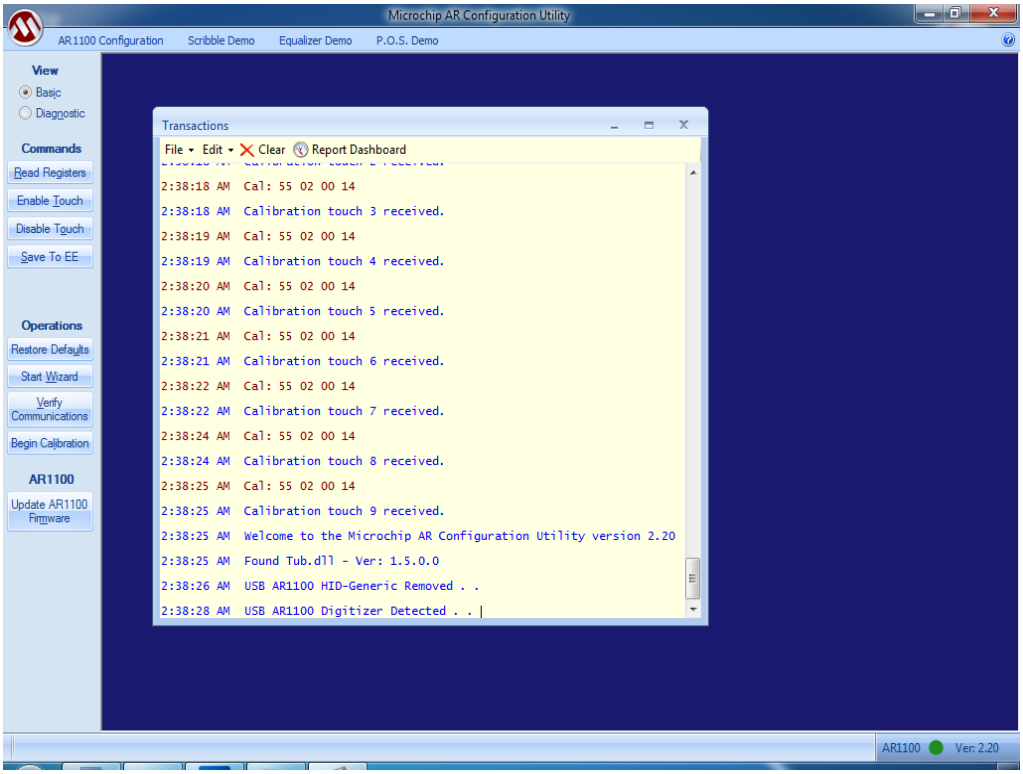


Figure 177 Complete the calibration