Model 676
High Performance AWG
Expert Rider SW User Manual

Rev 1.4, July 2020

Berkeley Nucleonics Corporation
info@berkeleynucleonics.com | www.berkeleynucleonics.com
2955 Kerner Blvd, San Rafael, CA 94901 | 800-234-7858

Summary

General Safety Summary

Review the following safety precautions to avoid injury and prevent damage to this product or any products connected to it.

To avoid potential hazards, use this product only as specified.

Only qualified personnel should perform service procedures.

To Avoid Fire or Personal Injury

Use Proper Power Cord

Use only the power cord specified for this product and certified for the country of use.

Ground the Product

This product is grounded through the grounding conductor of the power cord. To avoid electric shock, the grounding conductor must be connected to earth ground. Before making connections to the input or output terminals of the product, ensure that the product is properly grounded.

Observe All Terminal Ratings

To avoid fire or shock hazard, observe all ratings and markings on the product. Consult the product manual for further ratings information before making connections to the product.

Power Disconnect

The power cord provides Mains disconnect.

Do Not Operate Without Covers

Do not operate this product with covers or panels removed.

Do Not Operate With Suspected Failures

If you suspect that there is damage to this product, have it inspected by qualified service personnel.

Avoid Exposed Circuitry

Do not touch exposed connections and components when power is present.

Do Not Operate in Wet/Damp Conditions.

Do Not Operate in an Explosive Atmosphere.

Keep Product Surfaces Clean and Dry.

Provide Proper Ventilation

Refer to the manual's installation instructions for details on installing the product so it has proper ventilation.

Safety Requirements

This section contains information and warnings that must be observed to keep the instrument operating in a correct and safe condition. You are required to follow generally accepted safety procedures in addition to the safety precautions specified in this section.

Safety Symbols

Where the following symbols appear on the instrument's front or rear panels, or in this manual, they alert you to important safety considerations.

This symbol is used where caution is required. Refer to the accompanying information or documents in order to protect against personal injury or damage to the instrument.

This symbol warns of a potential risk of shock hazard.

This symbol is used to denote the measurement ground connection.

This symbol is used to denote a frame or chassis connection.

This symbol is used to denote a safety ground connection.

On (Supply). This is the DC power connect/disconnect switch at the back of the instrument.

Off (Supply). This is the DC power connect/disconnect switch at the back of the instrument.

This symbol is used to denote Power. It is located on the front panel and denotes Power On/Off status of the instrument.

This symbol is used to denote Direct Current.

CAUTION
The **CAUTION** sign indicates a potential hazard. It calls attention to a procedure, practice or condition which, if not followed, could possibly cause damage to equipment. If a **CAUTION** is indicated, do not proceed until its conditions are fully understood and met.

WARNING
The **WARNING** sign indicates a potential hazard. It calls attention to a procedure, practice or condition which, if not followed, could possibly cause bodily injury or death. If a **WARNING** is indicated, do not proceed until its conditions are fully understood and met.

EMISSIONS
CISPR 11, Class A, EN61000-3-2:2006, EN 61000-3-3:1995

IMMUNITY
EN 61326-1:2006, IEC 61000-4-2:2001, IEC 61000-4-3:2002,
IEC 61000-4-4:2004, IEC 61000-4-5:2001, IEC 61000-4-6:2003, IEC 61000-4-11:2004

Operating Environment

The instrument is intended for indoor use and should be operated in a clean, dry environment. Before using this product, ensure that its operating environment is maintained within these parameters:
Temperature: +0 ºC to +50 ºC (+32 ºF to 122 ºF)
Humidity (operating) : 8% to 90% relative humidity with a maximum wet bulb temperature of 29ºC at or below +50ºC, (upper limit de–rates to 20.6% relative humidity at +50ºC). Non-condensing.
Humidity (non operating): 5% to 98% relative humidity with a maximum wet bulb temperature of 40ºC at or below +60ºC, upper limit de–rates to 29.8% relative humidity at +60ºC. Non-condensing.
Altitude: Up to 10,000 ft (3,048 m) at or below 30 °C.

AC Power Source

For External AC Adapter: 100 to 240 VAC (+/-10%) at 45-66 Hz; Automatic AC voltage selection;
Installation Category: 300V CAT II
No manual voltage selection is required because the AC Adapter automatically adapts to line voltage.

Power Consumption

</= 120 watts

WARNING - Electrical Shock Hazard

Only use the power cord provided with your instrument.

Calibration

The recommended calibration interval is one year. Calibration should be performed by qualified personnel only.

Cleaning

Clean only the exterior of the instrument, using a damp, soft cloth. Do not use chemicals or abrasive elements. Under no circumstances allow moisture to penetrate the instrument.

Abnormal Conditions

Operate the instrument only as intended by the manufacturer.
If you suspect the instrument's protection has been impaired, disconnect the power cord and secure the instrument against any unintended operation.
The instrument's protection is likely to be impaired if, for example, the instrument shows visible damage or has been subjected to severe transport stresses.
Proper use of the instrument depends on careful reading of all instructions and labels.

WARNING

Any use of the instrument in a manner not specified by the manufacturer may impair the instrument's safety protection.

Environmental Considerations

This section provides information about the environmental impact of the product.

Product End-of-life Handling

Observe the following guidelines when recycling an instrument or component.

Equipment Recycling

Production of this equipment required the extraction and use of natural resources. The equipment may contain substances that could be harmful to the environment or human health if improperly handled at the product's end of life. In order to avoid release of such substances into the environment and to reduce the use of natural resources, we encourage you to recycle this product in an appropriate system that will ensure that most of the materials are reused or recycled appropriately.

The symbol shown to the left indicates that this product complies with the European Union's requirements according to Directive 2002/96/EC on waste electrical and electronic equipment (WEEE).

Operating Requirements

Power Supply

<table>
<tr><td>Source Voltage and Frequency</td><td>100 to 240 VAC ±10& @ 47-63Hz
115VRMS@400Hz
<table>
<tr><td>Characteristic</td><td>condition</td><td>Min.</td><td>Nom.</td><td>Max.</td><td>Units</td></tr>
<tr><td>Voltage</td><td>45-66Hertz</td><td>85</td><td>100-240</td><td>264</td><td>VRMS</td></tr>
<tr><td>Amplitude</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>Sine</td><td></td><td></td><td></td></tr>
</table></td></tr>
<tr><td>Power Consumption</td><td>Maximum: 120W</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	9.7kg
Net Weight with Package	10.7 kg
Overall Dimensions	Height: 160 mm Width: 450 mm Depth: 340 mm
Dimensions with Package	Height: 210 mm Width: 500 mm Depth: 400 mm

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)

Preface

This manual describes the installation and operation of the Arbitrary Waveform Generator Model 676 instruments. Basic operations and concepts are presented in this manual.

Package Contents

The standard Model 676 High Performance AWG package includes the following:

- Model 676 Arbitrary Function Generator instrument
- Power Cord
- Performance/Calibration Certificate
- Model 676 Introduction and Compliance document

Recommended Accessories

Item	Description
RM KIT	Rack Mount Kit
SSD KIT SA	SSD Solid State Drive for Secure Area
WAR EXT	Warranty Extension
CAL. CERT	Certification of Calibration

Introduction

General Features

Two working modes

- **Simple Rider AFG 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
- **Expert Rider (Arbitrary) mode**
  - Two analog channels
  - 16/32-bit digital channels (optional)
  - 1/16/32/64 Mpts per channel arbitrary waveform memory (optional)
  - Up to 1 GHz 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: 1 GHz bandwidth with differential output
- AC coupled mode: 1 GHZ 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

- Windows 10 based platform with 7-in (1024x768) capacitive touch LCD, front panel buttons, keyboard, and mouse
- 3U 19” rackmount form factor, removable hard disk guarantees the security of confidential data
- Ethernet interfaces for remote control

Protect Your Instrument from Misuse

Check Input and 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.

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

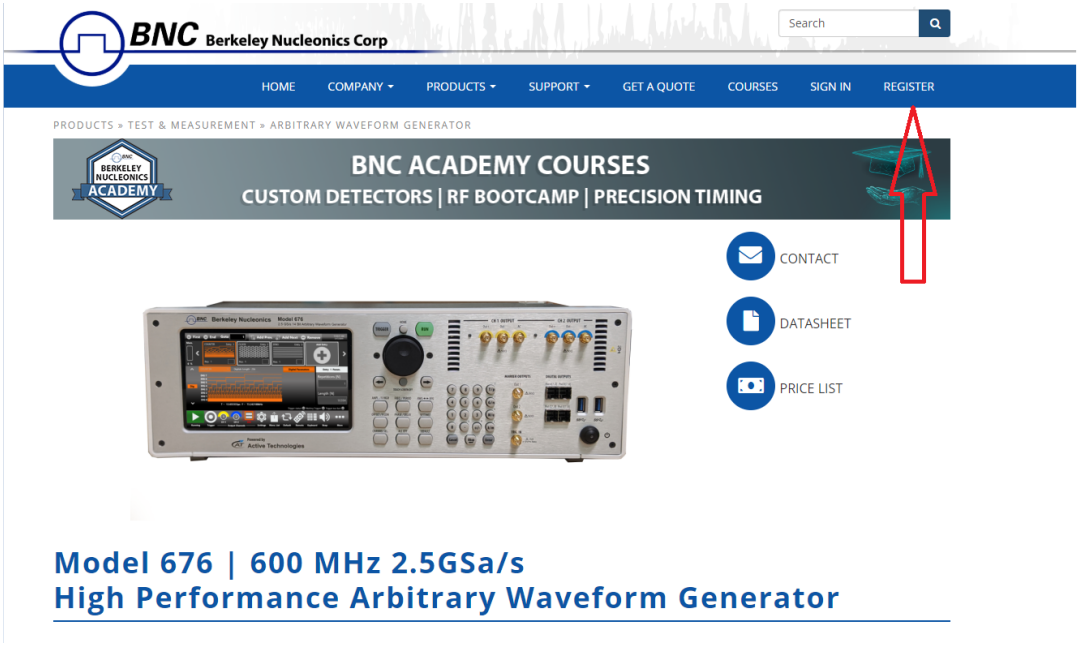
CAUTION. Do not apply excessive inputs over ±15 Vpk to Trigger Input connector. The instrument may be damaged.

CAUTION. The instrument has four **Digital Pods (PodA, PodB, PodC, PodD)** 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.

Obtaining the Latest 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.

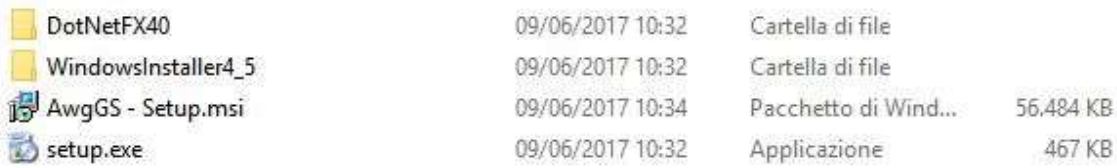
To download the latest version of software, register on the website; go to the home page of the Berkeley Nucleonics website (www.berkeleynucleonics.com), press the Register button in the upper right of your screen.



Install Model 676 Expert Rider Application

If your instrument has already installed another version of the Model 676 Expert Rider app, you must first uninstall it.

1. Download the Model 676 Expert Rider setup package from Berkeley Nucleonics website and decompress it to instrument's local disk.

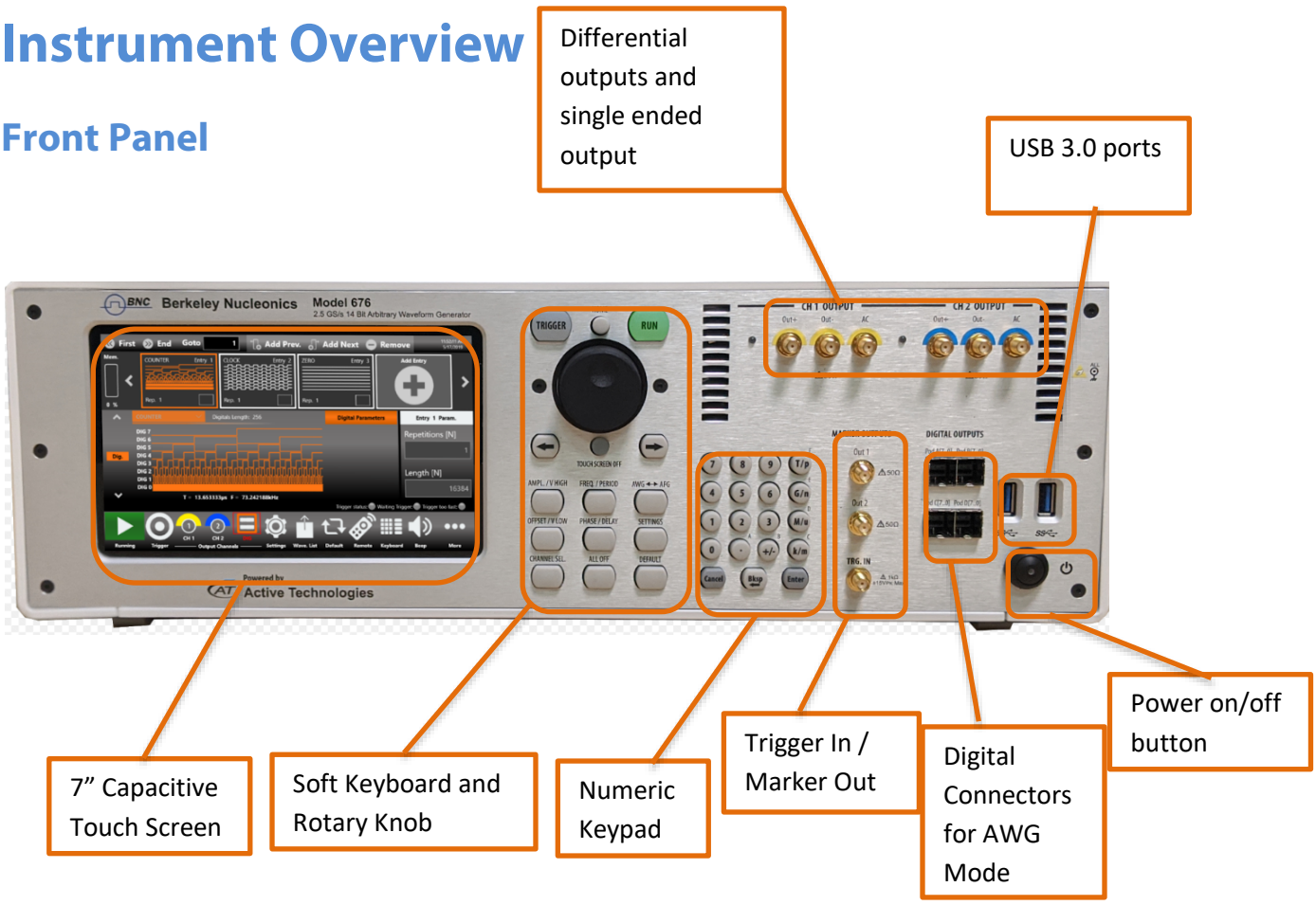


2. Double click on the setup.exe file

3. When the application has been installed, press Return to continue.

Instrument Overview

Front Panel



The Touch screen functionalities and features are described in the Simple Rider AFG Software section.

Analog Channel Outputs

OUTPUT 1 – Differential DC Coupled Analog output 1 (SMA connector): those outputs can work in DC-DIRECT and DC-AMP output mode and it can be selected via software.
AC 1: single ended AC coupled output with 50 Ohm.
OUTPUT 2 – Differential DC Coupled Analog output 2 (SMA connector): those outputs can work in DC-DIRECT and DC-AMP output mode and it can be selected via software
AC 2: single ended AC coupled output with 50 Ohm.

Each connector is coupled at 50 Ohm so in single ended operation the load sees 50 Ohm, instead in differential operation the load sees 100 Ohm.

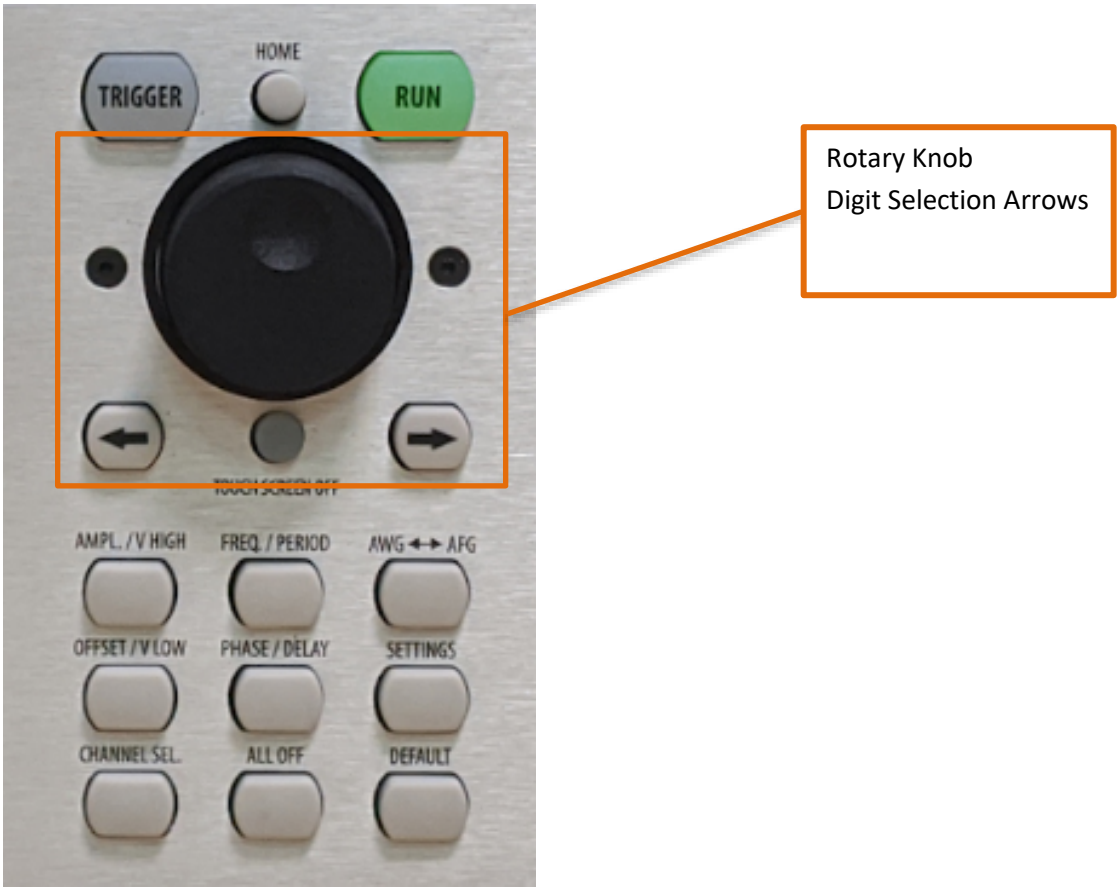
Trigger Input / Marker Output

TRG.IN - SMA input connector for Trigger IN. The **Trigger In** input has a fixed impedance of 1 kOhm and a programmable threshold level in the range -10 V to 10 V.
MARKER 1 / 2 - **SMA Marker Output** connector of the two channels. The **Marker Out** impedance is 50 Ohm and the output voltage range is from 1 V to 2.5 V into 50 Ohm load.

Front Panel Buttons

A physical numeric keypad is available on the front-panel and it can be used instead of the virtual numeric pad.

A useful central knob is available for fine-tuning and adjustments during the on the fly set up operation.
The rotary knob will change the value in continuous, analog fashion.
The → key will move the selected digit to the right and the ← key will move the selected digit to right.
You can keep pressed the rotating knob and rotate it on the right or on the left to change the Delta increment.



Button	Description
HOME	NA
TRIGGER	Use this button to send an internal trigger to the instrument.
RUN	Use this button to start and stop the signal generation. When the button led is **green**, it means that the instrument is in stopped state. If you press the button the instrument will be armed (running state) and the led will turn to **red**: the instrument generates the signal or it is ready to receive the trigger signal.
LEFT ARROW	Once the virtual numeric keypad will be opened, use this button to move left the digit selection cursor.
RIGHT ARROW	Once the virtual numeric keypad will be opened, use this button to move right the digit selection cursor.
TOUCH SCREEN OFF	NA
AMPL./V HIGH	NA.
FREQ/PERIOD	NA
AWG <-> AFG	Use this button to switch between AFG and AWG operation.
OFFSET/V LOW	NA
PHASE/DELAY	NA
SETTINGS	Use this button to open the Settings page
CHANNEL SEL.	Use this button to change the output selection in the user interface

ALL OFF	Use this button to turn off all the outputs.
DEFAULT	NA

Front Panel Numeric Keypad

The physical numeric keypad lets you to set the waveform parameter value and their measure unit. Use the touch panel to select the waveform parameter, each number pressed will be displayed in the display, the Bksp key is provided for deleting erroneous key presses. The [+/-] key will toggle the sign of the number being entered and may be pressed at any time before terminating entry. After the sign and numeric portion of the desired value have been punched in, the pressing of the multiplier button applies the parameter. The Enter button close the virtual keyboard and applying the written value.

When you select a parameter on the user interface, if you press a multiplier button it will automatically set the available range linked to the parameter.

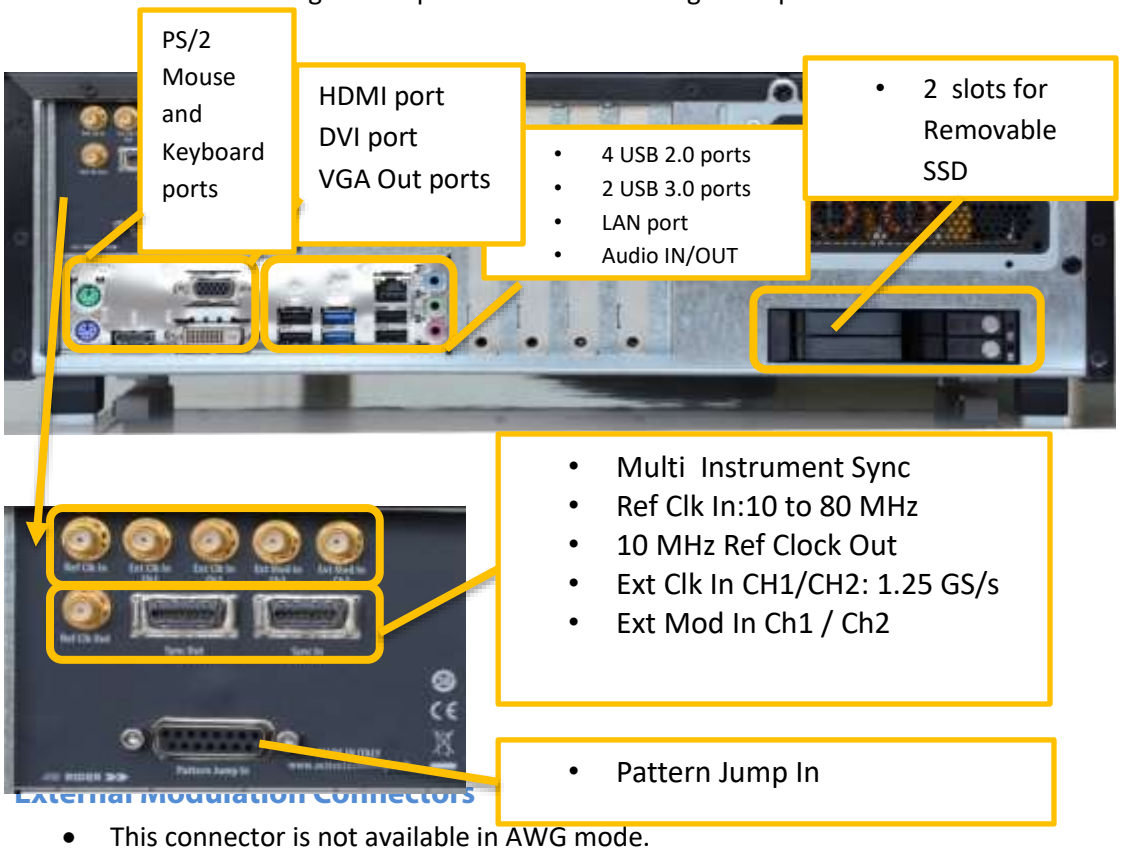
Unit Measure Range Button	Unit Measure Range
T/p	Tera / pico
G/n	Giga / nano
M/u	Mega / micro
k/m	Kilo / milli

As example when you select a Frequency parameter and you press k/m the unit measure range that will be inserted is kHz, if you press M/u it will be inserted MHz, if you press G/n it will be inserted

GHz, if you press T/p nothing will happen because that range is not available for the selected parameter.
If all the two unit of measure of a button are available for a selected parameter (i.e. Mega and Micro), if you press the range button **M/u**, the range will change accordingly between Mega and Micro.

Rear Panel

The callouts on this image correspond with the following descriptions.



External Modulation Connectors

- This connector is not available in AWG mode.

Reference Clock In Connectors

The Model 676 High Performance AWG can use an external clock source to generate the sampling clock at 2.5 GS/s. This feature allows to synchronize the generator with an external clock.
For the specification please refer to the Auxiliary Channels section.

Reference Clock Out Connector

This connector generates the low frequency clock used as reference for the sampling clock. If the clock source is internal it produces a signal at 10 MHz, but if the source is external it generate the reference frequency.
For the specification please refer to the Auxiliary Channels section.

Sync In/Sync Out Connectors

The purpose of those connectors is to connect and synchronize together multiple instruments: up to 4 instruments can be linked together.

Expert Rider AWG Software

The Model 676 High Performance AWG instrument includes a 7” capacitive touch screen based on a Microsoft Windows platform.
You can control instrument operations using the following:

- Keyboard, mouse and front-panel soft keys (recommended for Expert Rider AWG)
- Touch Screen and Front-panel soft key controls

This chapter will give you on overview about using Expert Rider AWG UI.

The Expert Rider AWG software can manage the instrument as a powerful Arbitrary Waveform Generator. The software is project based. Two type of project exist: Single Sequencer projects or Multi Sequencer projects. The two kind of projects differ in the way the analog/digital resources are managed:

- **Single Sequencer** projects: in this kind of projects all analog/digital resources are managed synchronously, also in the case where multiple board 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 in contrast with the “Multi Sequencer” where each analog channel has its own sequencer.
- **Multi Sequencer** projects: in this kind of projects each analog output can be configured completely independently each other. Each analog output behaves like a completely independent device with a single output channel.
  This mode is called “Multi Sequencer” because each analog channel has its own sequencer. Even if each channel can be configured independently the channels can share trigger and synchronization signals.

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.

In Arbitrary mode it is possible to define different waveforms and decide, by means of a sequencer, the sequence how these waveforms are generated at the outputs of the device.
Since the Model 676 High Performance AWG has both analog and digital outputs there are three different kind of

waveforms that can be defined: **analog** waveforms, **digital** waveform or **mixed** (analog/digital) waveforms.

Digital Data

The Model 676 High Performance AWG 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 the customer.

The digital bus mode can be configured as: Low Speed or High Speed.
Low Speed Mode: 32 bits are available on POD A,POD B,POD C and POD D (8 bits per Pod). The digital outputs 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: 16 bits are available on POD A and POD C. The digital outputs 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) and Pod C 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 out from the front panel SMA and its maximum update rate is 156.25 MHz.
If the selection is digital, the marker is connected to the Digital Pod A/PodC and it is available through the digital connector PIN.

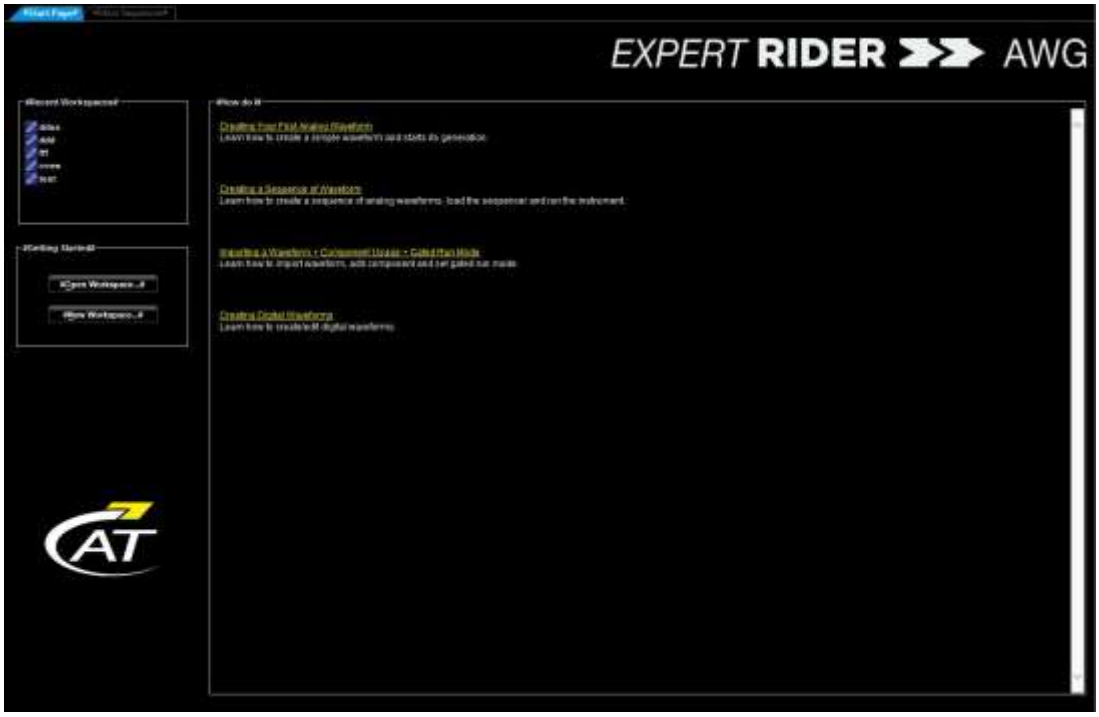
Expert Rider Workspace

The Model 676 High Performance AWG software workspace consists of two main elements:

- The main document editing area, shown on the right side on the following screen-shot.
- The Model 676 High Performance AWG Control Navigation Tree, shown on the left side on the following screen-shot.

As mentioned, when you open the Model 676 High Performance AWG environment, the most common initial tasks are displayed on the Start Page screen for easy selection in a special view, called the **How do I** section.

The **How do I section** contains links to step-by-step explanations of standard functions and setups using your new product.



These titles along with others provided here and in other locations of this documentation provide accurate descriptions for regular tasks.
The common tasks included in the Start Page are the following:

- "How do I" Scenario Details
- Recent Workspaces
- Open Workspace…
- New Workspace…

Interface and Display

The user interface is shown when you **open an existing project** or **create a new one**. Numbered callouts on this image correspond with the following interface section descriptions.

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

	**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.
	**Settings** - Use this button to access more detailed Model 676 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.
	**Remote/Local Mode** - Use this button to open the Remote Mode window that 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.
	**License –** Use this button to manage the option installation.
	**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 display area mainly provides information on the output sequence.
3. **Waveform List / Quick Settings**– This display area gives access to the Waveform List, and to the Quick Settings tools.
   - **WAVEFORM LIST → Waveforms TAB.** This TAB contains the list of the arbitrary waveforms that the user added to the project.

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.
	**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 1/2 or 1/4** 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 mouse drag & drop.
  This TAB contains the list of the available subsequences and the toolbar on the right allows you to Add/Edit them.

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 projects with the same configuration.
  As example if you create a Mixed Predefined waveform in Single sequencer 8/16 digital bits, it will be available on all the projects with the same features, but it will not be available in Multi Sequencer projects or Single Sequencer 16/32 digital bits ones.
  Predefined Analog waveforms will be available in all the user projects, while predefined Digital waveforms will be available in the user projects that will match the same digital configuration (8/16 or 16/32).
  You can add a Predefined waveform to the Sequence Area by drag & drop with

mouse.

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.

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

PLEASE NOTE THE FOLLOWING:

- All of the panels are dockable; meaning they snap into convenient screen positions adjacent to other panels. Move individual panels by clicking the panel's top side, holding, and dragging with your mouse.

Setup Examples and Common Tasks

The following examples provide a quick way to learn standard Model 676 High Performance AWG signal setups and common tasks:

1. **Arbitrary Mode Single Sequencer Setup Example** (on page 34)
2. **Creating a New Workspace** (on page 40)
3. **Opening an Existing Workspace** (on page 41)

"How do I" Scenario Details

In addition to the **Setup Examples and Common Tasks**, the end of this manual also contains scenarios with detailed steps for performing typical tasks and setups using Model 676 High Performance AWG as shown in the **How do I** section on the Welcome screen (shown when first launching the software).

PLEASE NOTE THE FOLLOWING:

- Some more specific steps are required around Creating a New Workspace for each scenario. Details are provided for those scenarios when necessary.

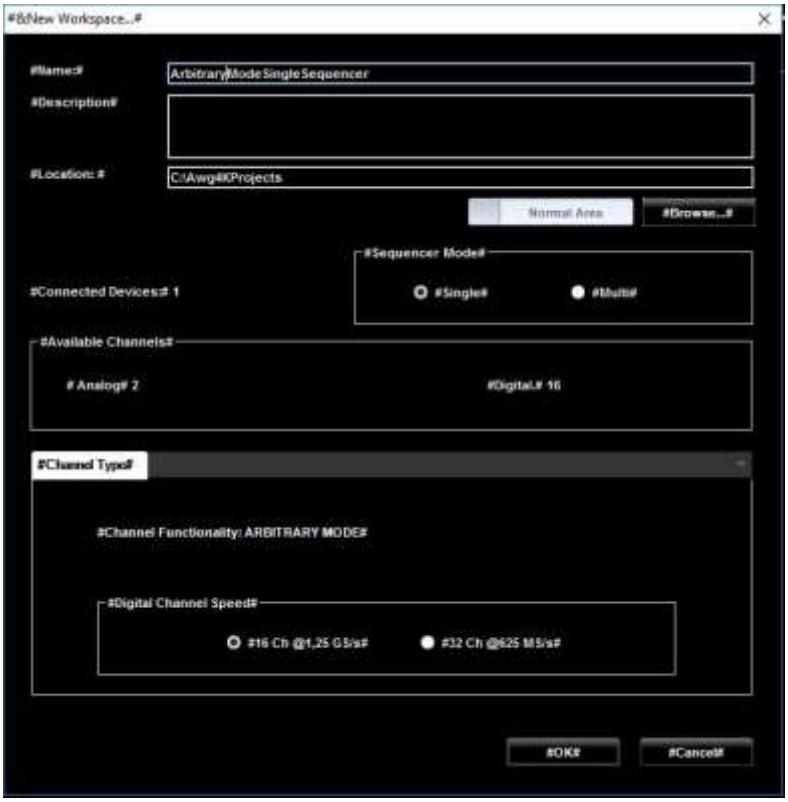
Arbitrary Mode Single Sequencer Setup Example

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

1. Type the Workspace name
2. Select **Single** as Sequencer Mode
3. Select **Arbitrary** as Channel Functionality
4. Select **16Ch@1.25 GS/s** as Digital Channel Speed.

   Note: four Mini SAS HD connectors provide 8 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 max. update rate of 625MSps or with half channels (16) at 1.25Gsps.

5. Click **OK**.

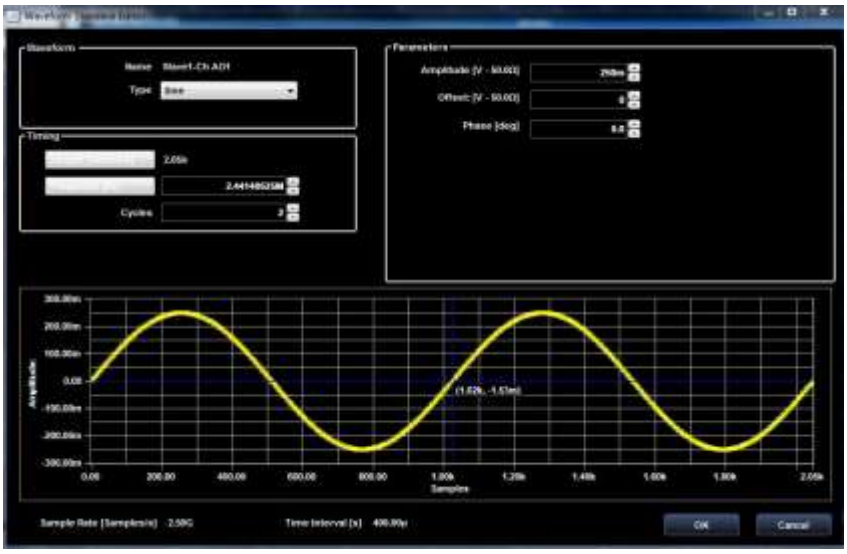


6. Click the **New Mixed Waveform** button.

7. 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.

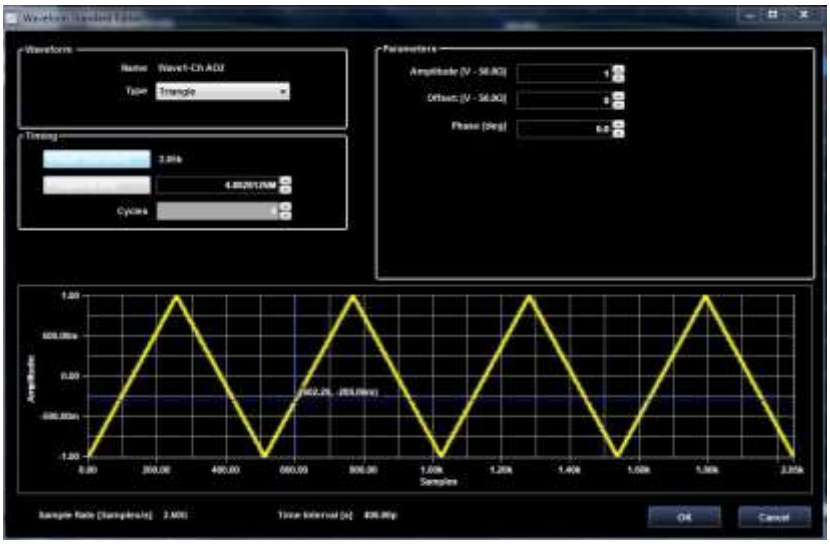
8. The **Editing Waveform** window is shown. Select the waveform Wave1-0 and click on the **Edit**  button

9. The **Waveform Standard Editor** is shown. Choose a sine waveform with the following specs:
    - Cycles: 2
    - Amplitude[V]: 250mV



Press **OK** button.

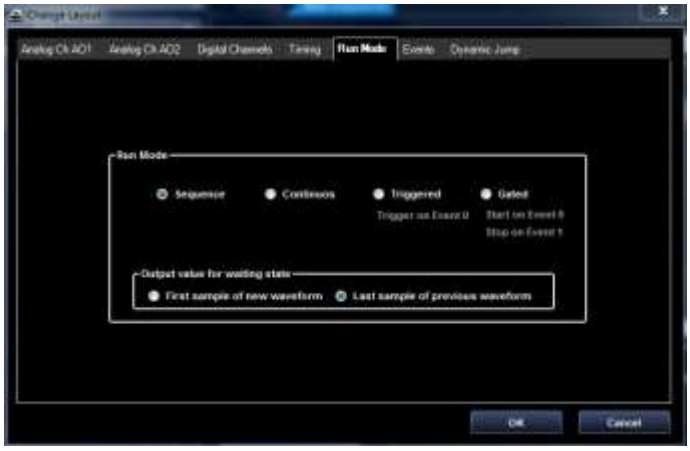
10. Select the waveform Wave1-1 and click on the **Edit**  button.
11. The **Waveform Standard Editor** is shown. Choose a triangle waveform with the following specs:
    - Cycles: 4
    - Amplitude[V]: 1V



Press **OK** button.

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

    - Select **Sequence** as Run Mode

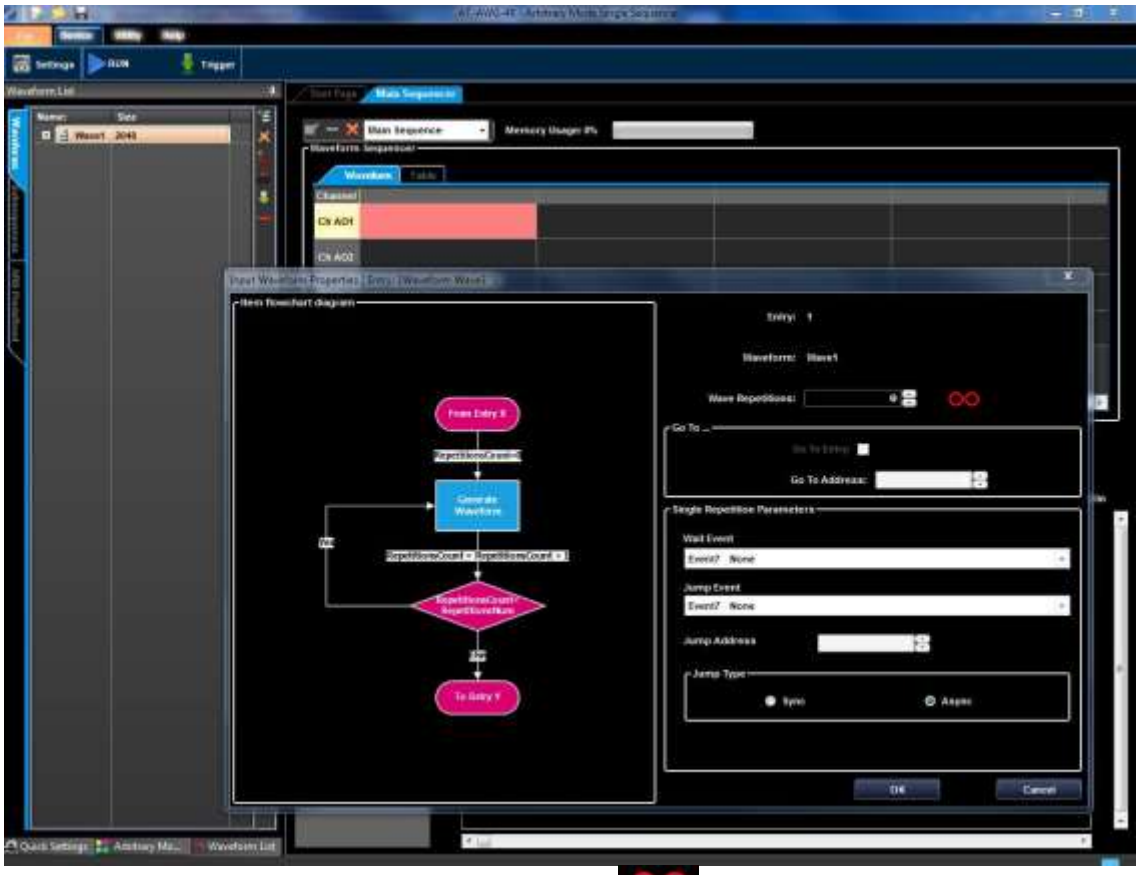


- Select the *Analog Ch AO1* TAB and select **DC Direct** as Out Type

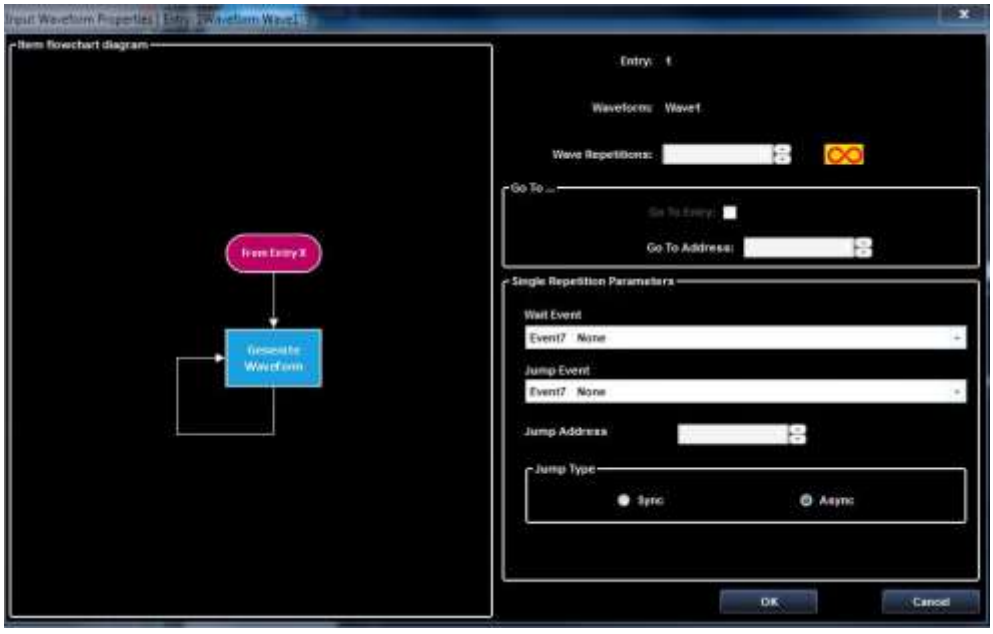
- Select the *Analog Ch AO2* TAB and select **DC Amplified** as Out Type

Press **OK** button.

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



15. 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.



Press **OK** button.

16. The Sequence Area will show now the Wave1 inserted in the first cell. Selecting one of the waveform with the mouse, it will appear on the *Waveform Display Area* placed

below.

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

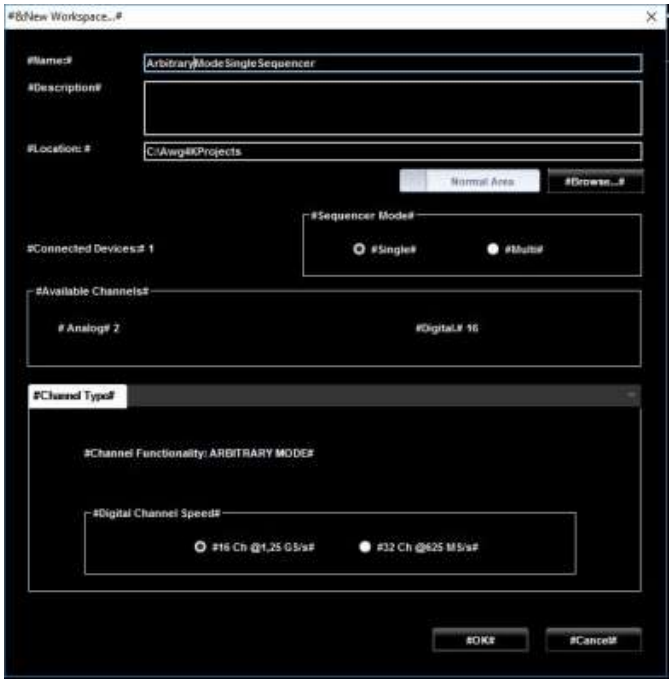
    Note: Once the instrument has started, Sequence Run Mode with infinite repetitions, repeats *Wave1* until the **Run/Stop** button is clicked again.

    The software loads the waveforms into the Model 676 High Performance AWG hardware and then generates waveforms.
18. **Wave1** ports to the **CH1/CH2 SMA output**, which can be connected to an oscilloscope for signal analysis.

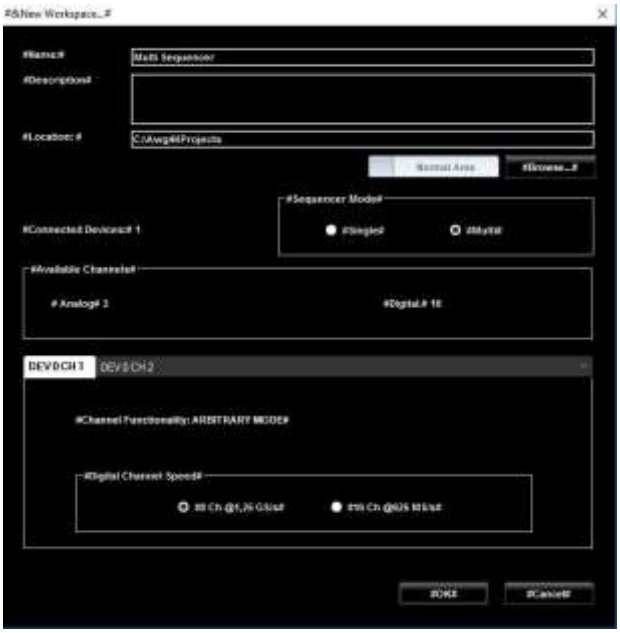
Creating a New Workspace

1. From the Start Page, click the **New Workspace** button.

2. The New Workspace form is shown and displays the connected Model 676 High Performance AWG hardware on the **Connected Devices** section.
3. Now, provide a **Name** and **Description** for your new **Workspace**.



4. Select the **Sequencer Mode**
   - **Single Sequencer** projects: in this kind of projects all analog/digital resources are managed synchronously, also in the case where multiple board are connected together in a daisy chain configuration.
   - **Multi Sequencer** projects: in this kind of projects each analog output can be configured completely independently each other. Each analog output behaves like a completely independent device with a single output channel.



5. Select the **Digital Channel Speed**: 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.

Opening an Existing Workspace

1. Open preexisting workspaces by clicking the **Open Workspace** toolbar button. The Open Workspace screen is shown.
2. The Open Workspace screen automatically navigates to the Awg4KProjects folder. Select the desired workspace and click **Open**.

Examples Project

1. The Model 676 High Performance AWG setup automatically installs under the folder C:\Program Files (x86)\Active Technologies\EXPERT RIDER AWG 4000\DemoProjects several demo projects that can help you to understand more in depth all the instrument features.
2. Please note that the Demo Projects 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.
   In that you can open them in DEMO Mode turning off the Model 676 High Performance AWG.

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** (on page 44)
2. **Digital Channels Tab** (on page 47)
3. **The Timing Tab** (on page 48)
4. **The Run Mode Tab** (on page 43)
5. **The Events Tab** (on page 49)
6. **The Dynamic Jump Tab**(on page 52)

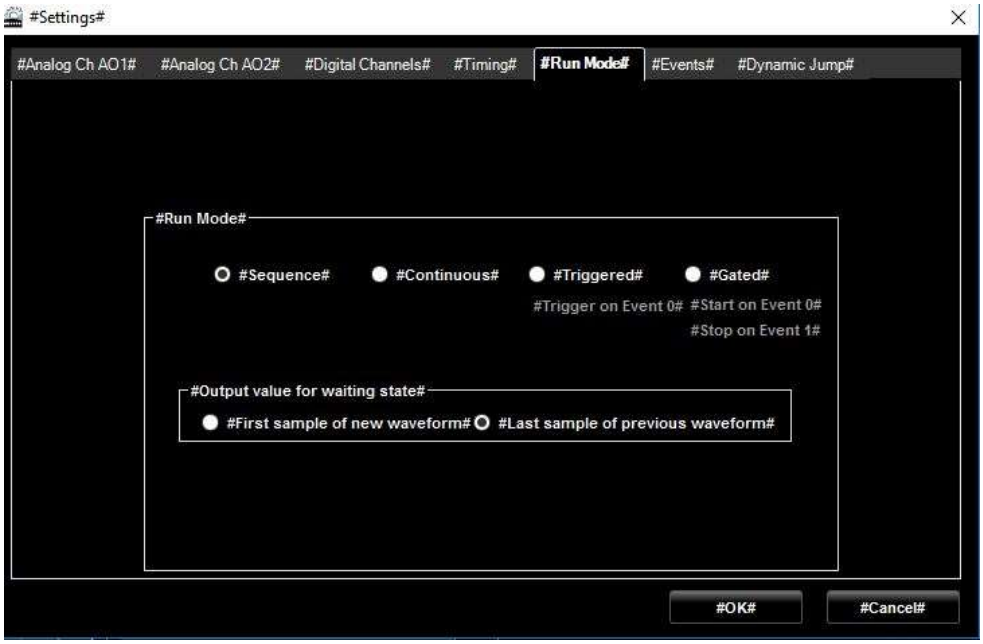
PLEASE NOTE THE FOLLOWING:

- In Single Sequencer projects, one Settings panel is available to manage all the channels capabilities.
- In Multi Sequencer projects, 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.



The Model 676 High Performance AWG supports the following four run modes:

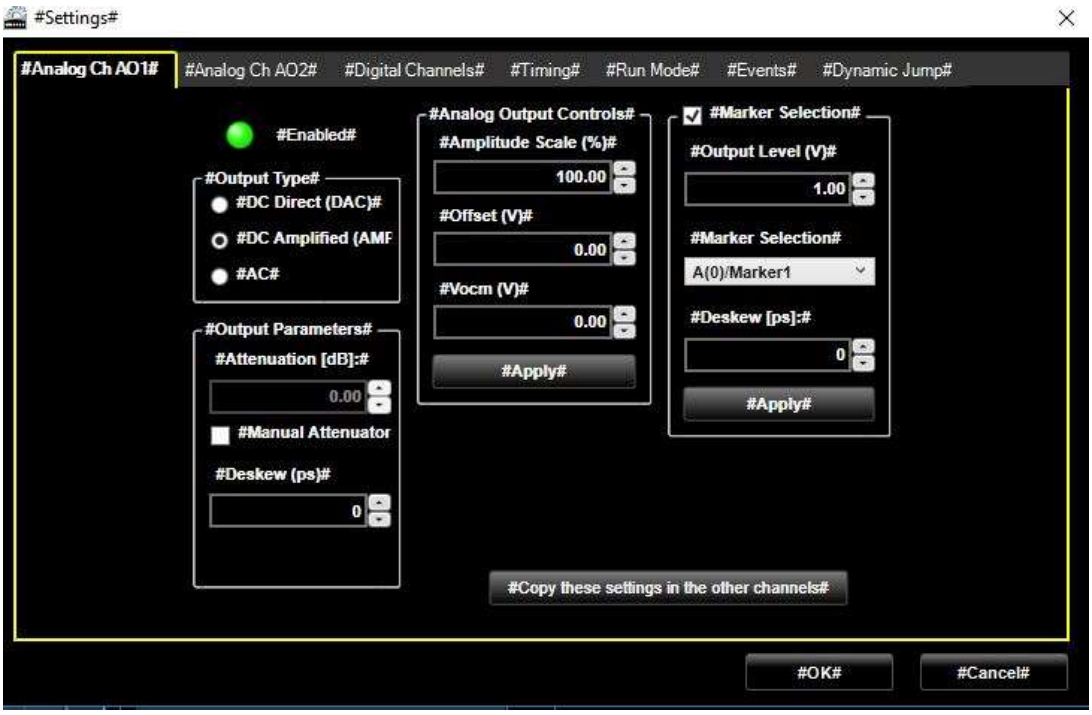
- **Sequence** – Multiple waveforms can be output in the order specified in the Sequence Window.
- **Continuous** – A continuous waveform is output. Only one entry is allowed in the Sequence Window.
- **Triggered** – A waveform is output once when the instrument receives a trigger signal. The instrument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

PLEASE NOTE THE FOLLOWING:

- In Sequence and Triggered mode the instrument can wait 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)

Use the **Analog Ch AO1/AO2** tab to quickly access the output type, amplitude, de skew and trigger out parameters of the selected channel.



- **Enabled**: to enable a channel output, click the **Enabled** button.
- **Out 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 run-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 four. The minimum value is -200% and and it means that the waveform amplitude is divided by four.
- **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.
- **Deskew(ps):** this parameter can set a fine delay between the output channels in order to realign the outputs with a resolution of about 10 ps.
  The maximum value you can set as skew between analog/digital channels depends on the sampling frequency you set; the software automatically calculates the maximum value in relationship to the sampling rate you set in the Timing Tab.

  Please note that the “Analog CH” deskew of CH1 will have impact also on CH1 digitals 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 Wavefrom of CH1, so in this way the digitals will be shifted from the analog.

- **Manual Attenuator**: it sets the attenuation type to manual on the DC AMP output.By default the attenuation type is automatic.

  Different rules should be considered for automatic attenuation and manual attenuation.

  1. Automatic attenuation rule: only the 20dB fixed output attenuator can be switched on/off. The attenuator is switched on when the output voltage swing is <= 100mVpp and the Vocm is within +-250mV. In all other situation no attenuation will be applied. Using automatic attenuation, it will improve the output signal quality with low level signals.
  2. Manual attenuation rule: any attenuation value in the range 0dB to 40dB can be programmed.
     Attenuations >= 20dB can only be applied if the Vocm is within +-250mV. The voltage at the output will be lower than the programmed waveform according to the attenuation applied.
     The attenuation has effect on DC Amplified output values and on the offset level. It has no effect on Vocm value.

- **Attenuation[dB]:** if you selected the DC Amplified output, you can apply a programmable attenuation between 0 and 40 dB.

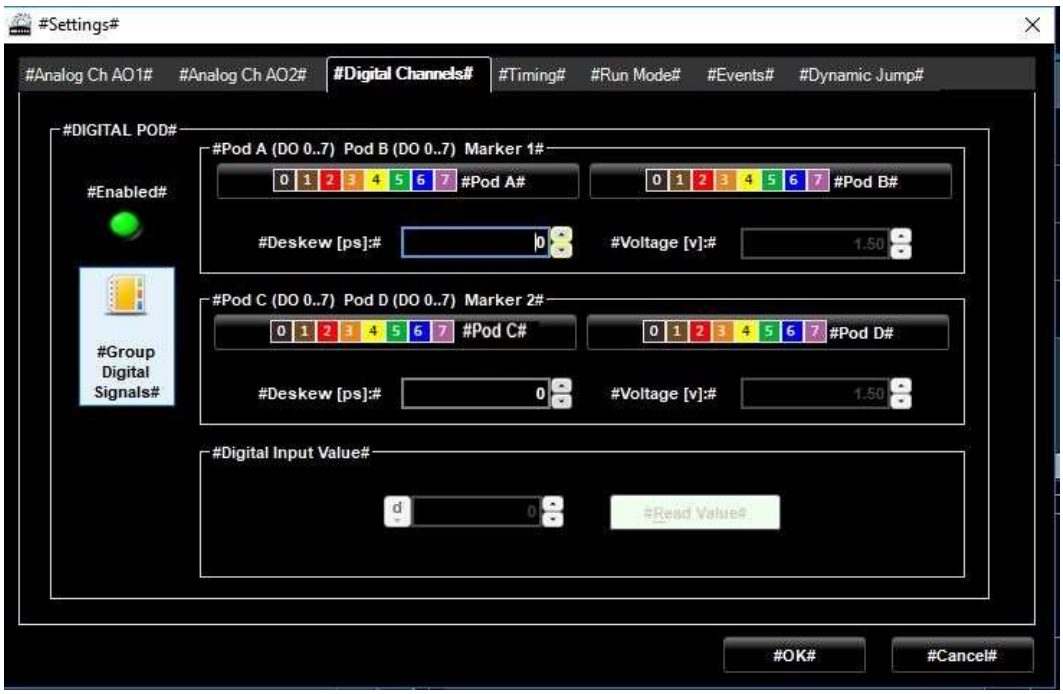
- **Marker Selection:** the marker type can be analog or digital; when the analog marker is selected the signal comes out from the front panel SMA and its maximum update rate is 156.25 MHz.
- This checkbox enables the Analog Marker and the Marker Out section. If you uncheck the Marker Selection, the marker type will be set to Digital.
  If the selection is digital, the marker is connected to the Digital Pod A(0)/PodB(0) and it is available through the digital connector PIN.

  Marker Out: set the Marker Out *Output level (V)* and the *Marker Selection* associated to the Marker Out. 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.

The *Deskew(ps)* value sets the skew between the marker and the analog waveform of the selected channel.
Press the Apply button to confirm the changes.

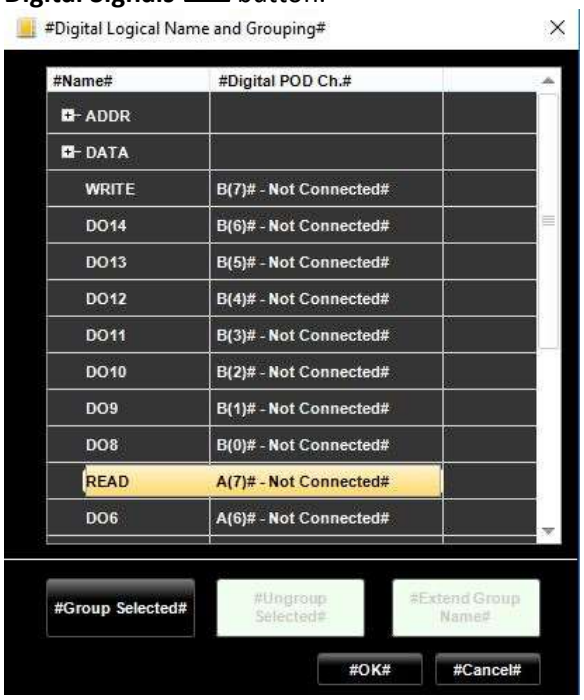
Settings - Digital Channels Tab

Use the **Digital Channels** tab to set a fine de skew on the digital lines, assign values to the output pins, grouping digital lines.
You can generate a digital pattern to test digital devices such as serial and parallel DACs or to emulate protocols. Four Mini SAS HD connectors provide 16 bit LVDS digital outputs (in High Speed Mode) each for a total of 32 LVDS outputs (in Low Speed Mode).



- **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.
  The skew between analog/digital channels depends over the sampling frequency.
- **Level(V):** this parameter sets the output voltage level (in Volt) of the Digital Probe. Please note that it will take effect on the Model 676 High Performance AWG instruments with installed Digital Option and with the probe connected.

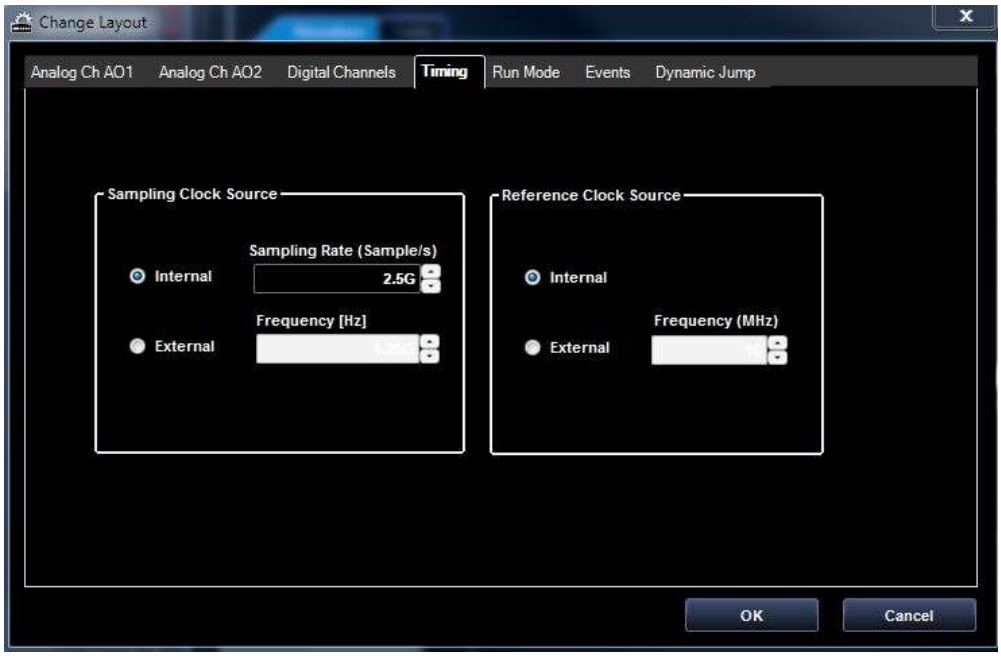
- **Digital Signals** button.



The *Digital Logical Name and Grouping Window* is shown.

- The first column displays the digital channels logical names that can be assigned by the user.
- The second column displays the *digital output 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.
- Keep pressed the left mouse button 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

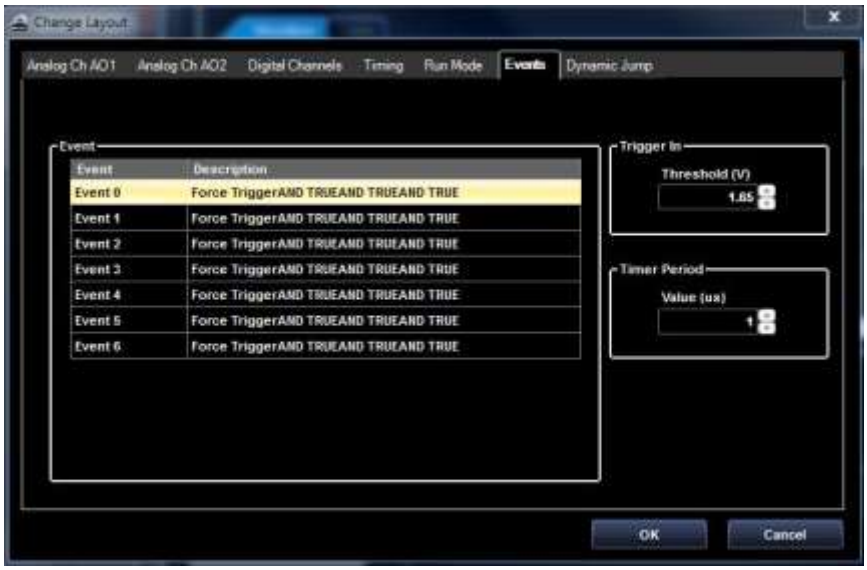


1. **Sampling Rate:** the sample rate can be set to the following ranges
   - The 100 S/s to 2.5 GS/s for Arbitrary mode channels
2. **Sampling Clock Source**: it may be set to Internal or External.
   If **Internal** is selected, the sampling clock signal is generated internally.
   If **External** 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).
   PLEASE NOTE THE FOLLOWING:
   - 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.
3. **Reference Clock Source**: you can select the Reference Source (Internal or External), the Reference Source is selectable only when the Clock Source is set to Internal.

Settings - Events Tab (Single Sequencer)

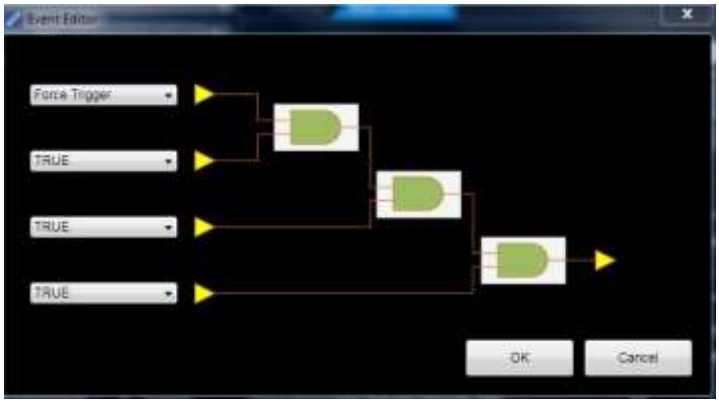
The Expert Rider AWG has Event Jump and Wait Event (*Input Waveform Properties window*) functions which change the generation sequence using an event signal.
Additionally Triggered and Gated run mode execution depend on Event 0 and Event 1.

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



- **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 gives 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.



  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.

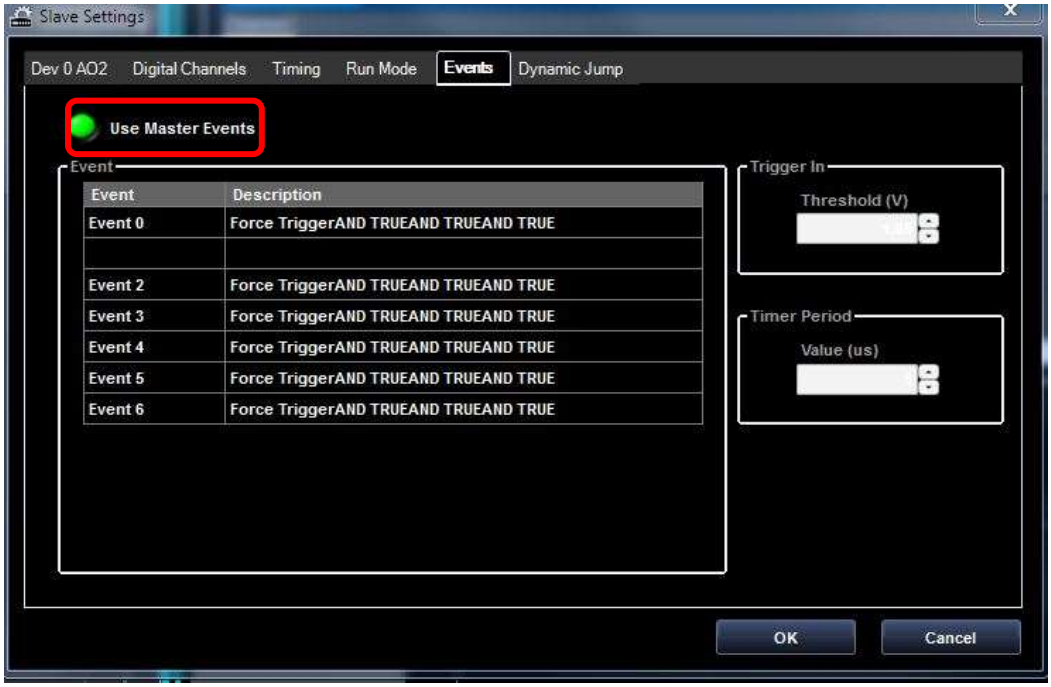
PLEASE NOTE THE FOLLOWING:

- **Trigger IN**: the event is received by the dedicated SMA Connector on the instrument front panel.
- **Timer**: the event is a pulse with a duration of 40 ns 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 projects 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 microseconds.

Settings - Events Tab (Multi Sequencer)

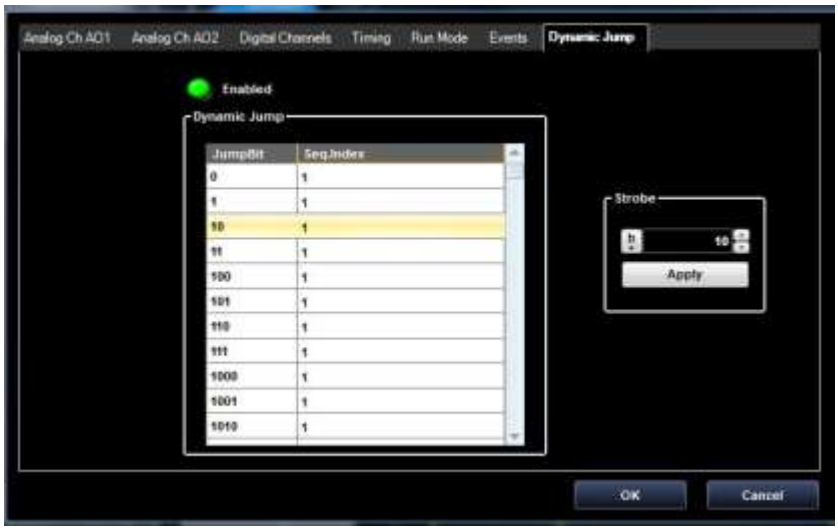
In Multi Sequencer projects, the Event TAB parameters are the same of the Single Sequencer projects except for *Use Master Event* led placed on the Slave Settings panel.



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.



This can be done by following these guidelines:

1. Specify the relationship between the "Strobe" value and the entry of the sequencer in the *Dynamic Jump* table.
2. Use the Strobe 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.

Editing Waveform Window

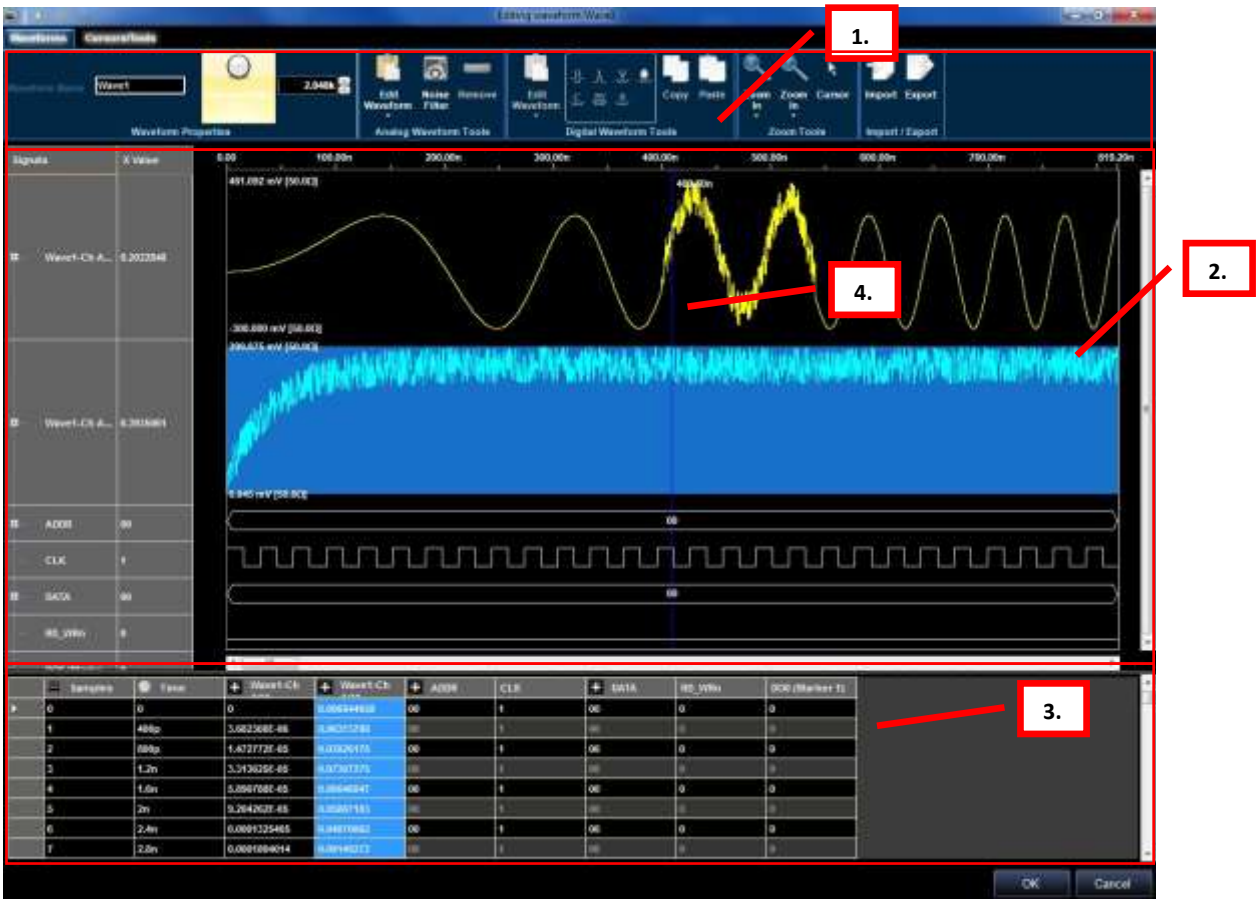
Use the **Editing Waveform Window** menu to create a new analog/digital waveform or modify an existing waveform.
The Model 676 High Performance AWG has two analog outputs and up to 32 digital channels that the user can configure according at his needs.

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 shown. Type the name of the waveform and choose the samples length. You can insert the Length in *Samples* or *Time*.Click **OK** to confirm.

3. The **Editing Waveform Window** is shown. 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**

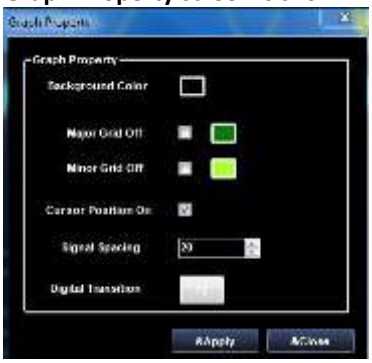


Waveform Graph Tools

When viewing a waveform in the Waveform Editor, Model 676 High Performance AWG main toolset is provided and gives you access to the following functions:

	*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.
	*Waveforms TAB:* Auto zoom in function.
	*Waveforms TAB:* Auto zoom out function.
	*Waveforms TAB:* This button allows zooming in on a selected rectangle of the graph. Click and drag inside the graph area to create your zoom rectangle.
	*Waveforms TAB:* This button resets all activated zooms
	*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:

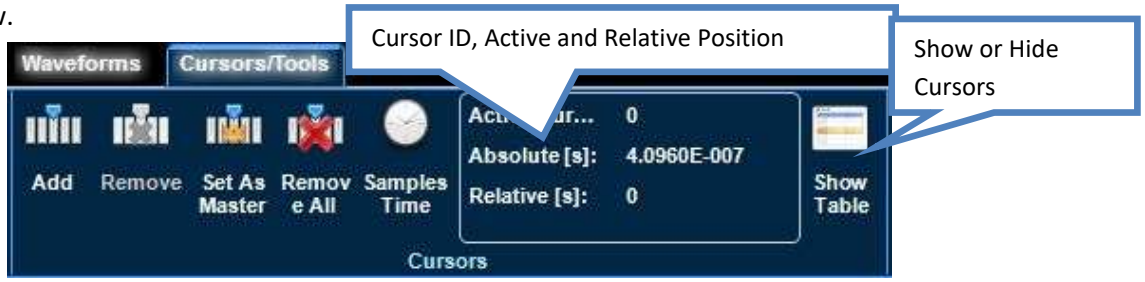
- The **Background Color** can be changed as desired.
- Change colors and turn the **Major** and **Minor Grids** on or off and change their line coloring.
- **Cursor Position** indicators can be turned on or off.

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

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.



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	

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.

PLEASE NOTE THE FOLLOWING:

- 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:

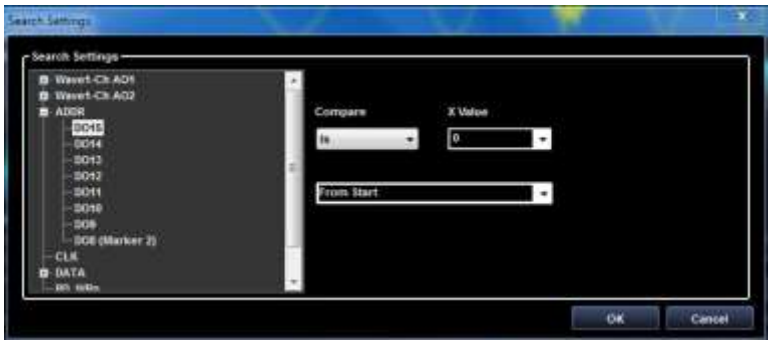
	**Go to time** - Moves the master cursor to the time position specified in the text field to the left of the control.
	**Go to start samples** - Moves the master cursor and visualization area to the start of the acquisition.
	**Go to end samples** - Moves the master cursor and visualization area to the 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 also is 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 and used to provide your search criteria



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.

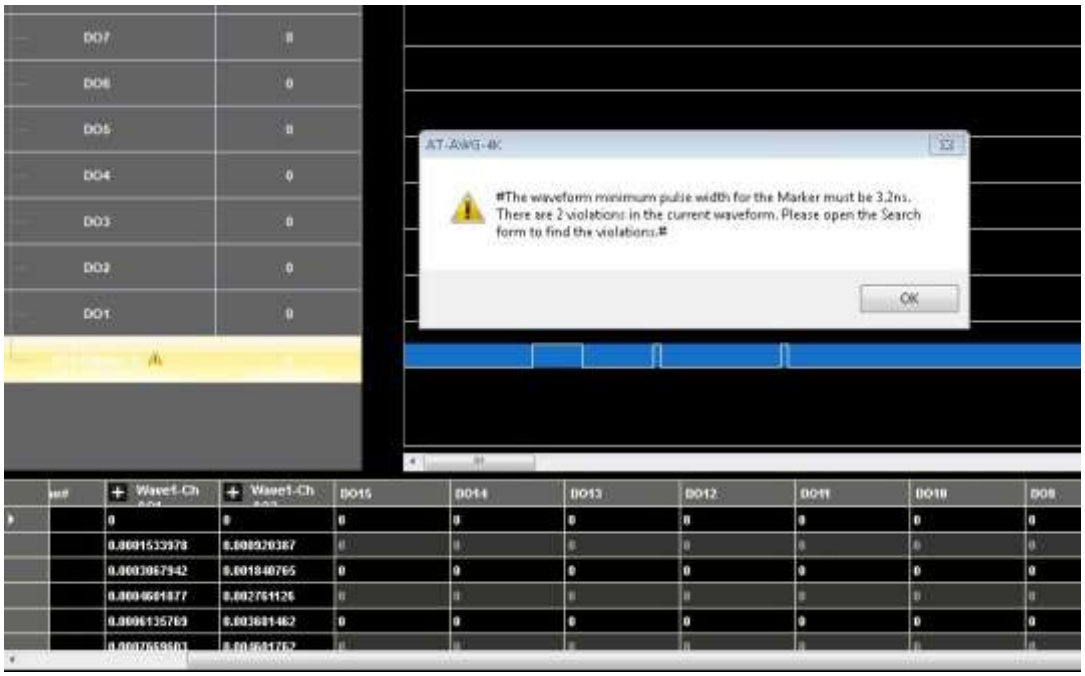
Note: Depending on the Signal Type selected in the search list,
the **Compare** and **Value** fields contains different options.

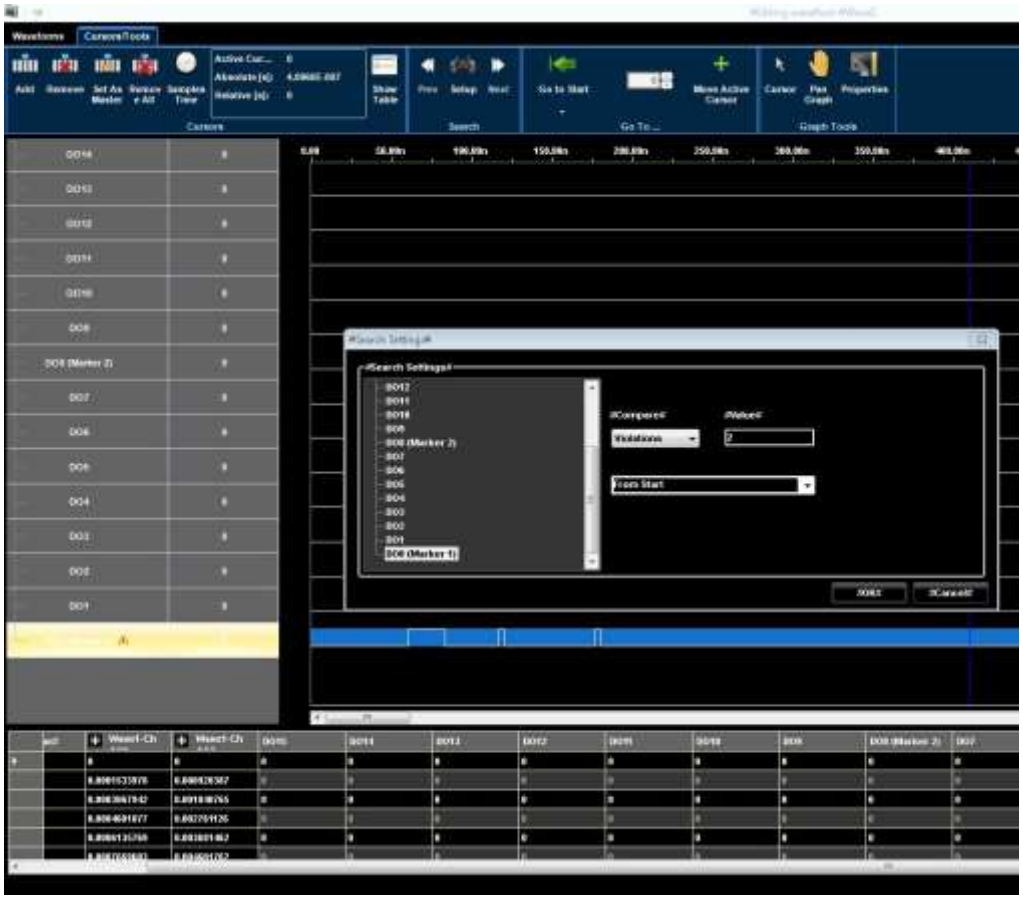
Use the Compare field to select between the following search logic operators:

- **=** 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. 
- **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.





The following procedure should be applied to cancel the timing violations:

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.

Select criteria on the Search Settings screen and click the **OK** button. The results are then shown on the **Editing Waveform Window**.
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.

Analog Waveform Graph Tools

The Model 676 High Performance AWG handles Analog Waveforms, Segments, and Components in the following manner.

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 choose a basic waveform like DC Level, Sine, Increase Ramp, Triangle, Sawtooth, Rectangle etc. The Expert Rider AWG 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 (see section on page 78).

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

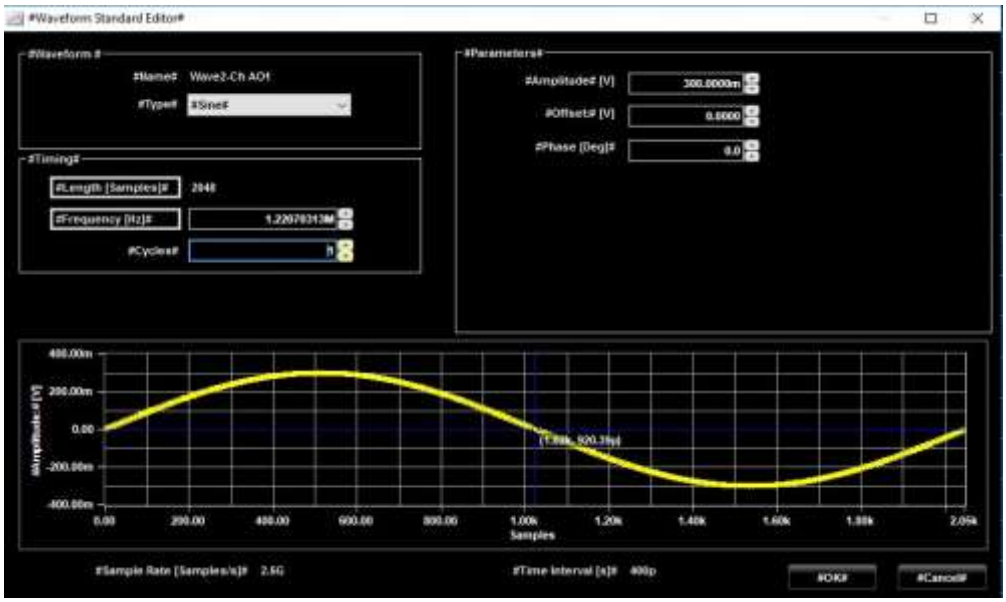
Component

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 Model 676 High Performance AWG 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.

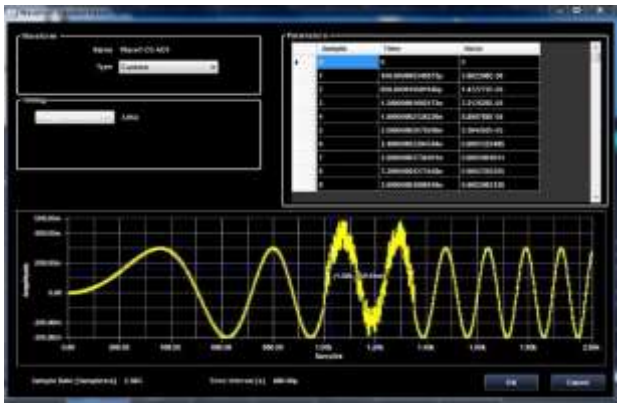
Waveform Standard Editor Window



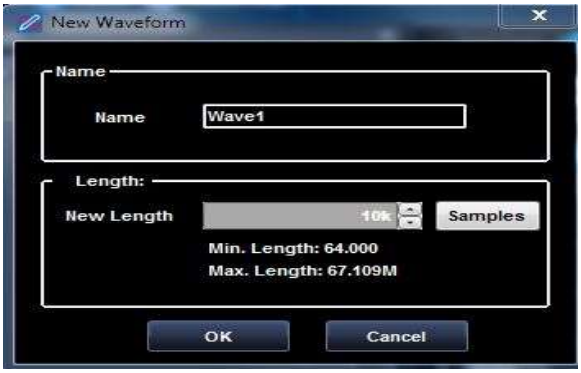
Press the **Edit** button and the Waveform Standard Editor Window is shown.
This window allows editing standard waveforms, segments and components parameters.

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]
Cosine	Frequency[Hz/cycles], Amplitude[V], Phase[°], Offset[V]

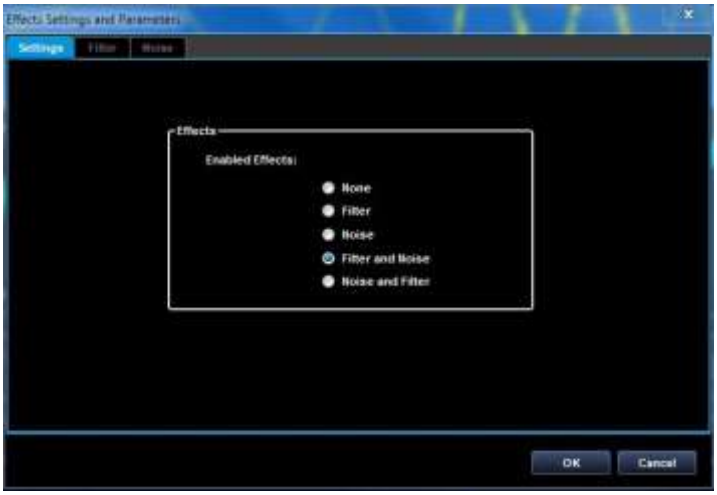
Triangle	Frequency[Hz/cycles], Amplitude[V], Phase[°], Offset[V]
Rectangle	Frequency[Hz/cycles], Amplitude[V], Phase[°], Offset[V], Duty Cycle [%]
Saw tooth	Frequency[Hz/cycles], Amplitude[V], Phase[°], Offset[V]
Increase Ramp	Amplitude[V], Offset[V]
Decrease Ramp	Amplitude[V], Offset[V]
Pulse	Amplitude[V], 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.



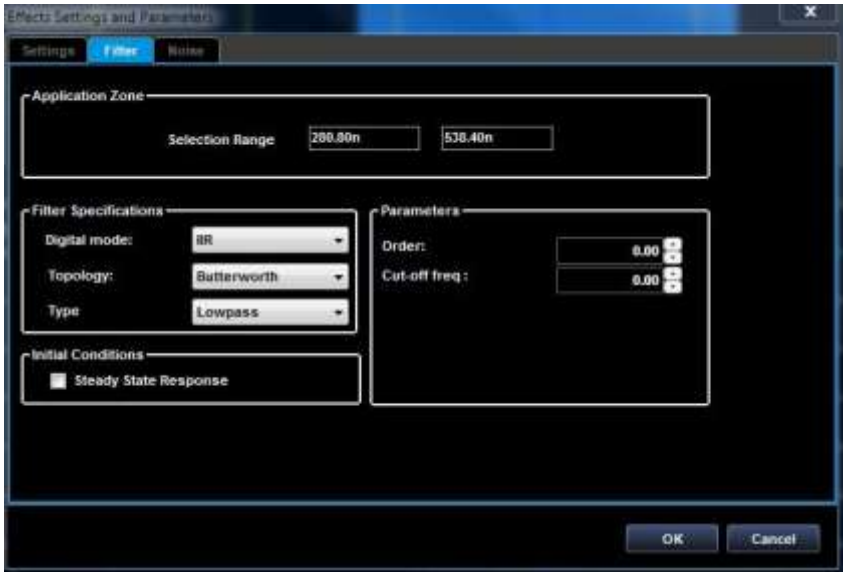
Effects Settings and Parameters Window (Analog Waveforms Only)

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) .



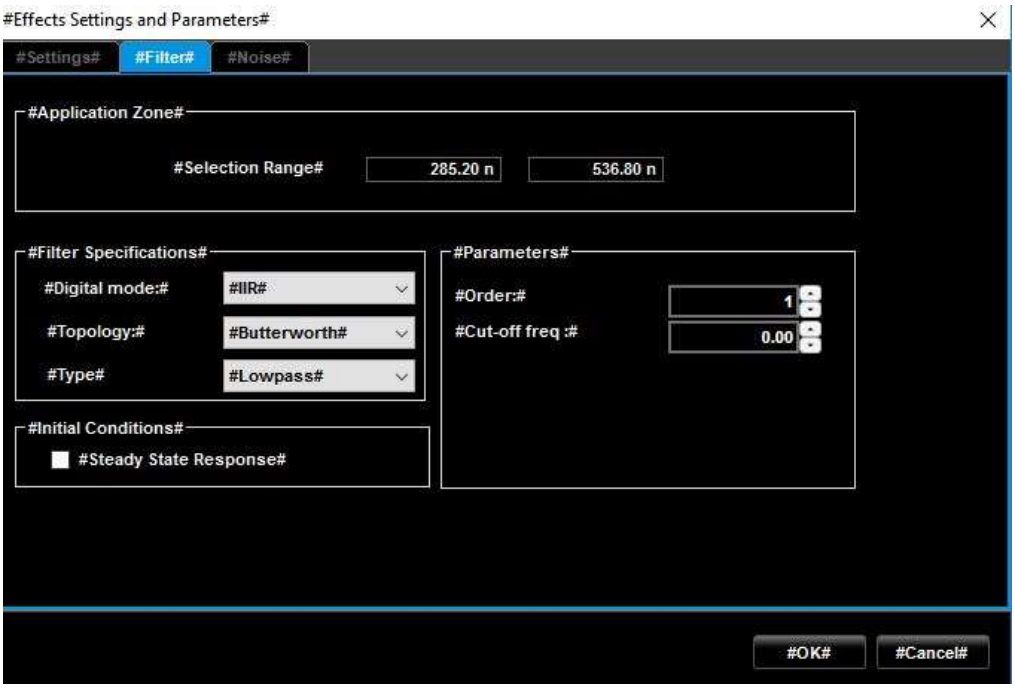
Filter Settings TAB

This tab allows applying a digital filter to the selected waveform.



FILTER

This section allows selecting 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.

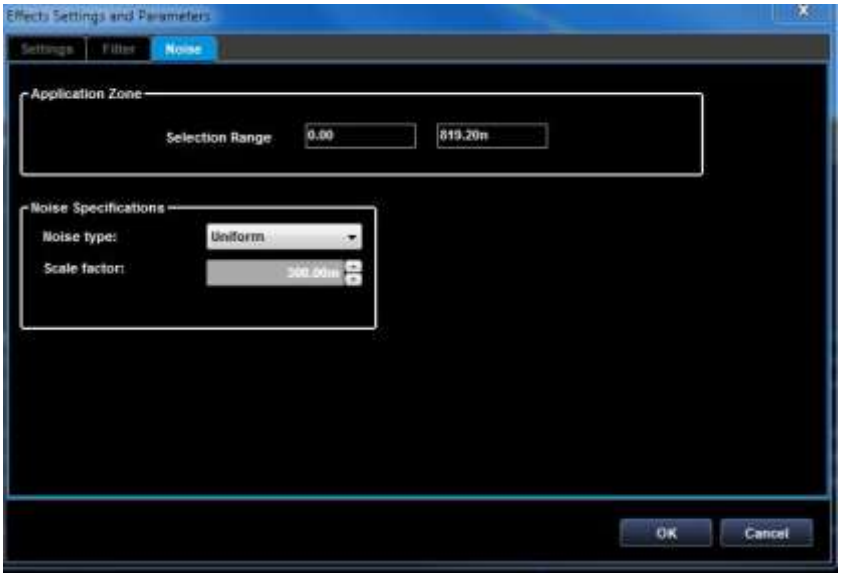
- **Digital Mode IIR (Infinite Impulse Filter)** - Bessel, Butterworth, Chebyshev, Inverse Chebyshev, and Elliptic.

- **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 the **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

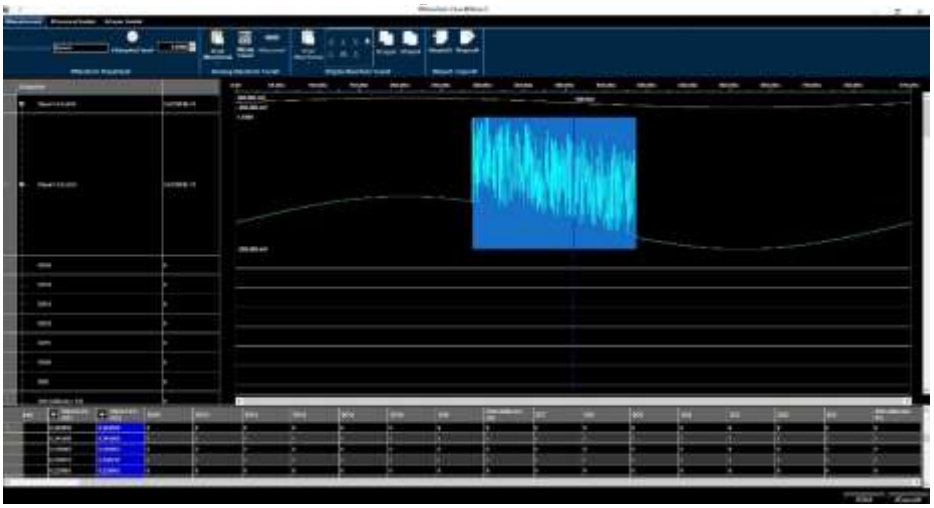
This TAB allows applying a digital noise to the selected waveform.



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

NOISE Application Zone

This section allows selecting whether the digital 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 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:

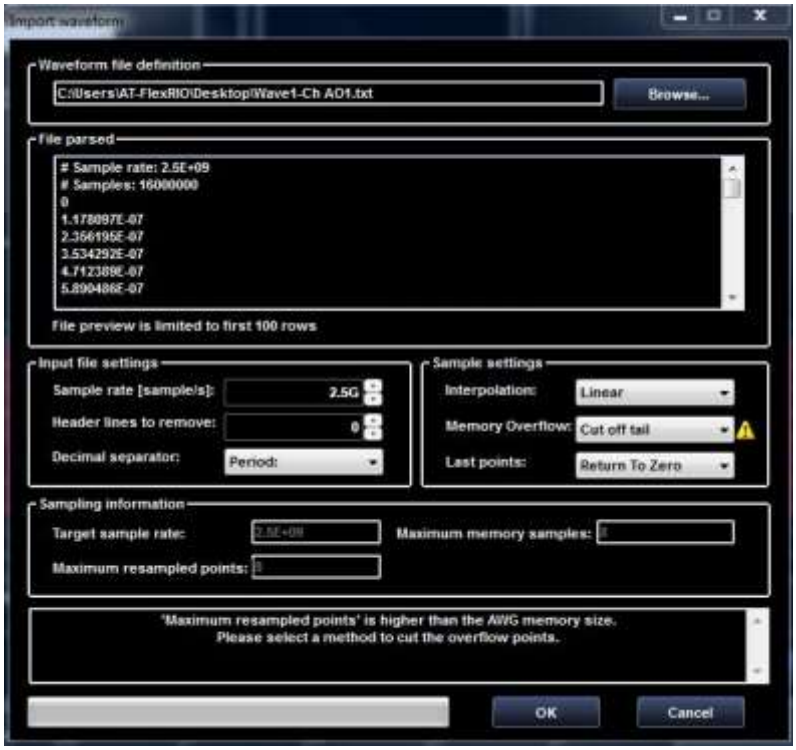
The supported file formats are:

- TXT - Tab separated value file
- CSV – Comma separated value 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 will open



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.
- 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, you can choose to cut off the last points (“Cut off tail”) or the first points (“Cut off head”)
- 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…….

EXPORT of Analog,Digital and Mixed Waveforms

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

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

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

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

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

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 projects, 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.

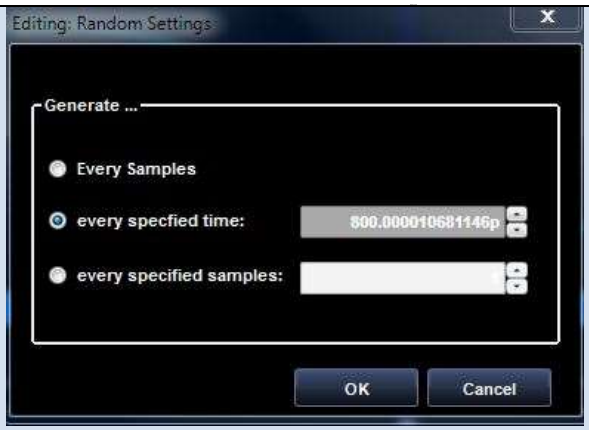
Please note that the import/export functions can become slow with large amount of data to import or export.

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 Waveform: edited waveforms can be generated as they have been set.

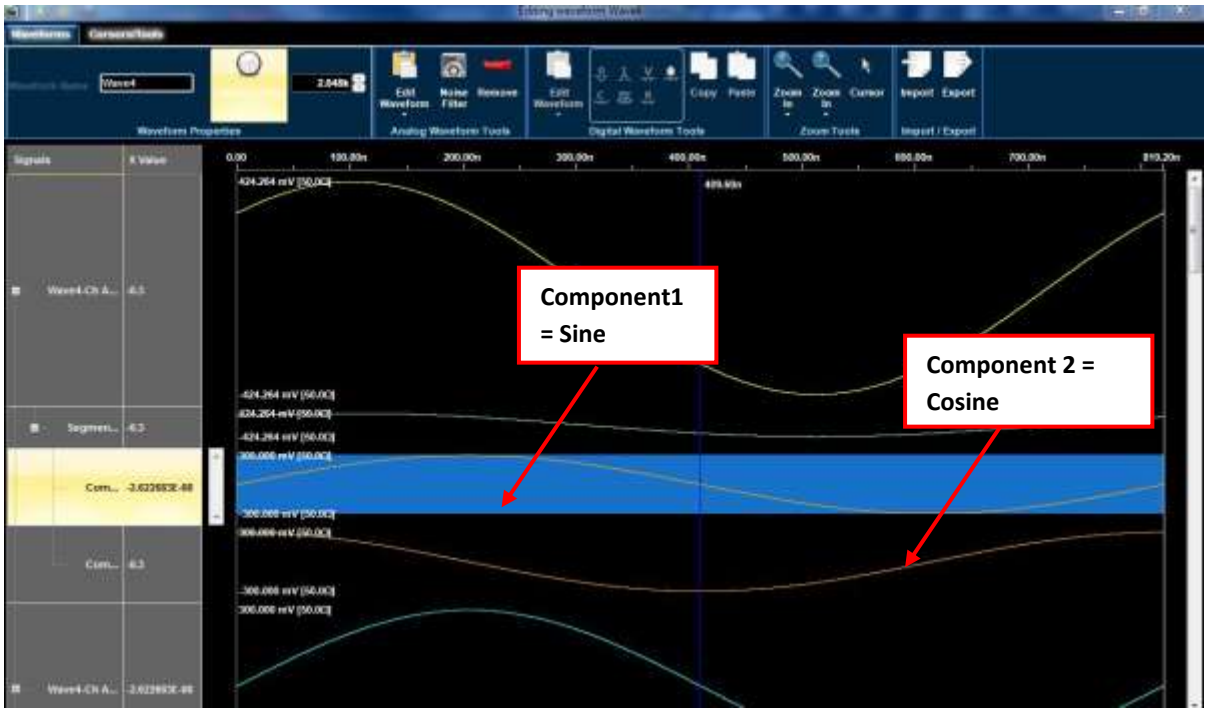
Think of a waveform as a list of **Segments**, where each segment can contain one or more **Components**, all of the same length, combined by means of the elementary
Add, Subtract, Multiply, Divide operations.

Each waveform may be constituted by an arbitrary number of segments and each segment can have its own length.

PLEASE NOTE THE FOLLOWING:

- 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.

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



The waveform in the picture above is made of one segment. The segment is the composition of two Components multiplied together: Segment1 = Component1 * Component2.
You can use this technique for example to generate **IQ** modulated signals.

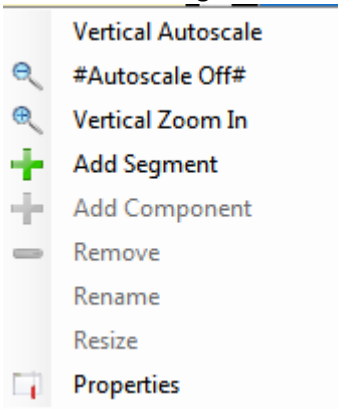
Common operations can be directly performed on waveforms, segments and components (**Selection**, **Left Click**, **Right Click**). Drag and Drop operation between analog waveforms is not allowed.

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.



- **Vertical Autoscale On** – Enable the auto scale function; 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.
- **Autoscale Off** – Select this option to disable the waveform auto scale.
- **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.

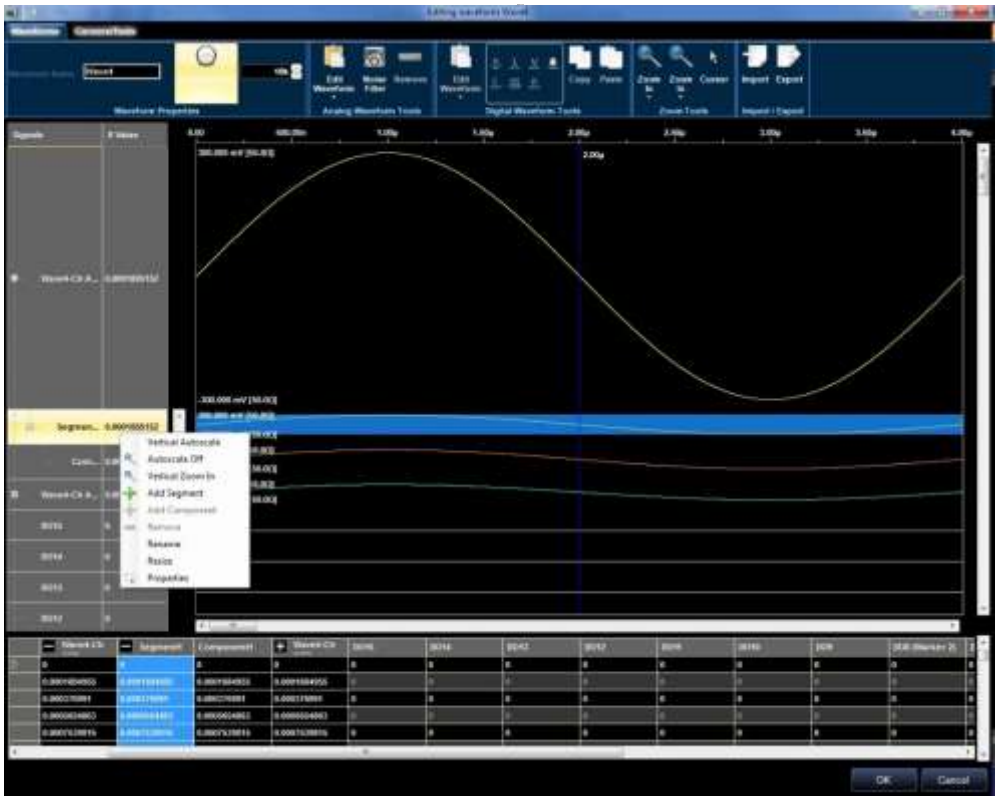
- Right click on the Waveform or on the existing Segment to activate the pop-up menu and select Add Segment.
  The Segment2 Property window is shown, insert the new segment length and press OK to add it to the waveform.
  PLEASE NOTE THE FOLLOWING:
  If you select the *Add Segment* option from the Segment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

How to create an Advanced Analog Waveform

- Follow the steps 1-6 of the Arbitrary Mode Single Sequencer Setup Example on page 34
- 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.

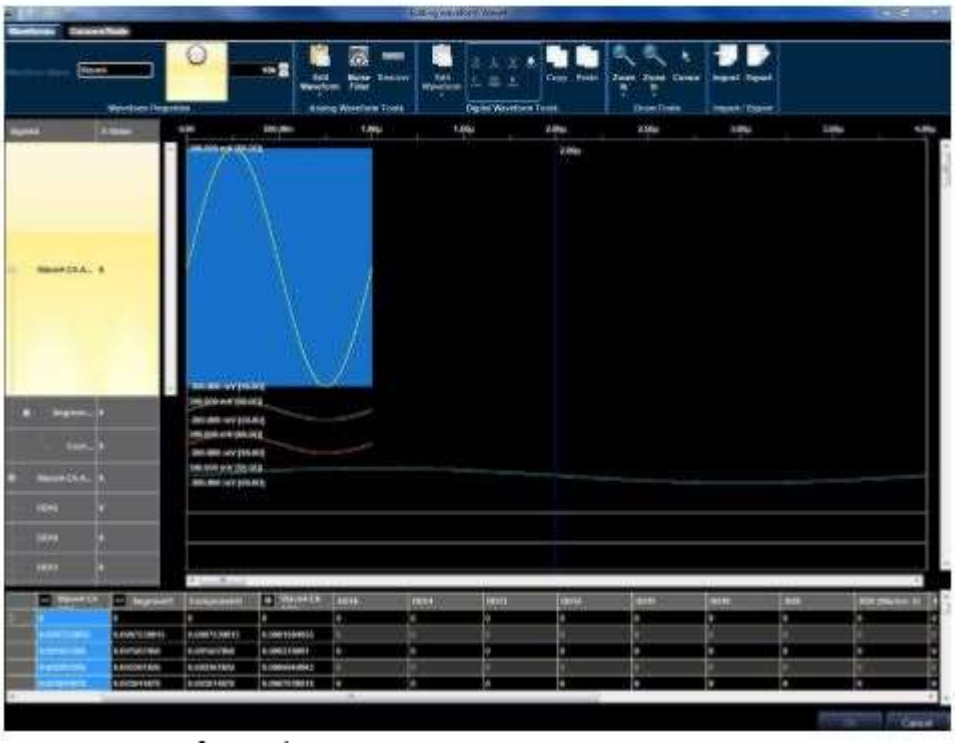
- The **Editing Waveform Window** is shown. Right click on Segment1 of the Wave1-0 to open the pop-up menu.



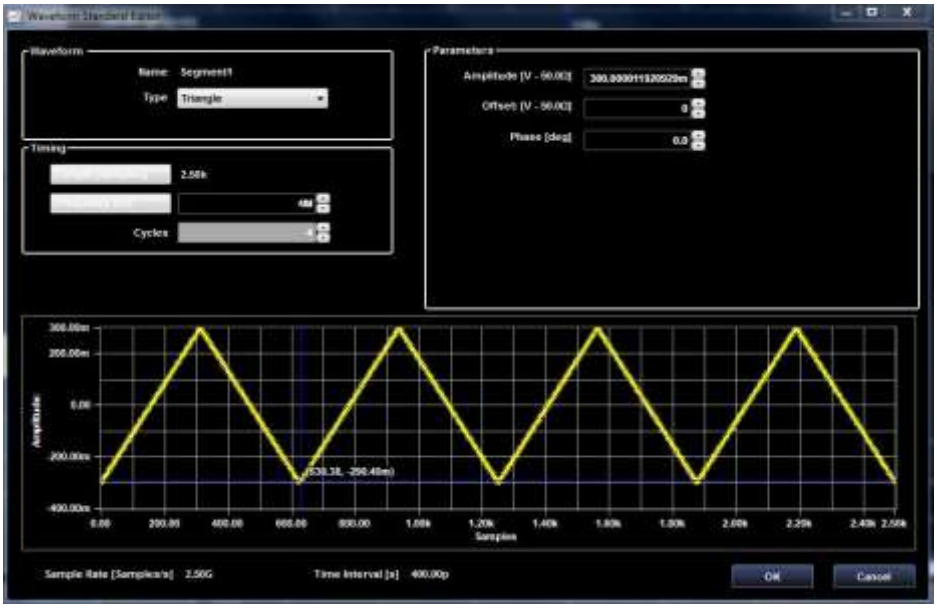
Select **Properties** on the pop-up menu.

- Change the segment length: insert 2.5k in the New Length field.

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



- Select the Segment1 and click the **Edit** button on the toolbar.
- The Standard Waveform Editor window is shown.



Select Triangle as waveform Type and 4 as number of Cycles.
Click **OK**.

- Right click on the Segment1 of Wave4-0 to open the pop-up menu and select **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**.

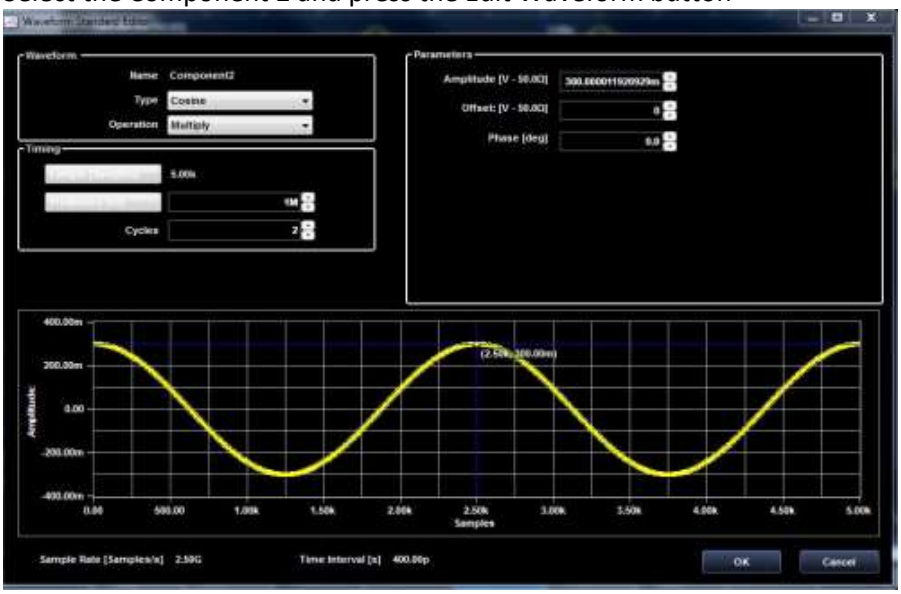
- Select the Segment3 and click the **Edit** button on the toolbar.

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

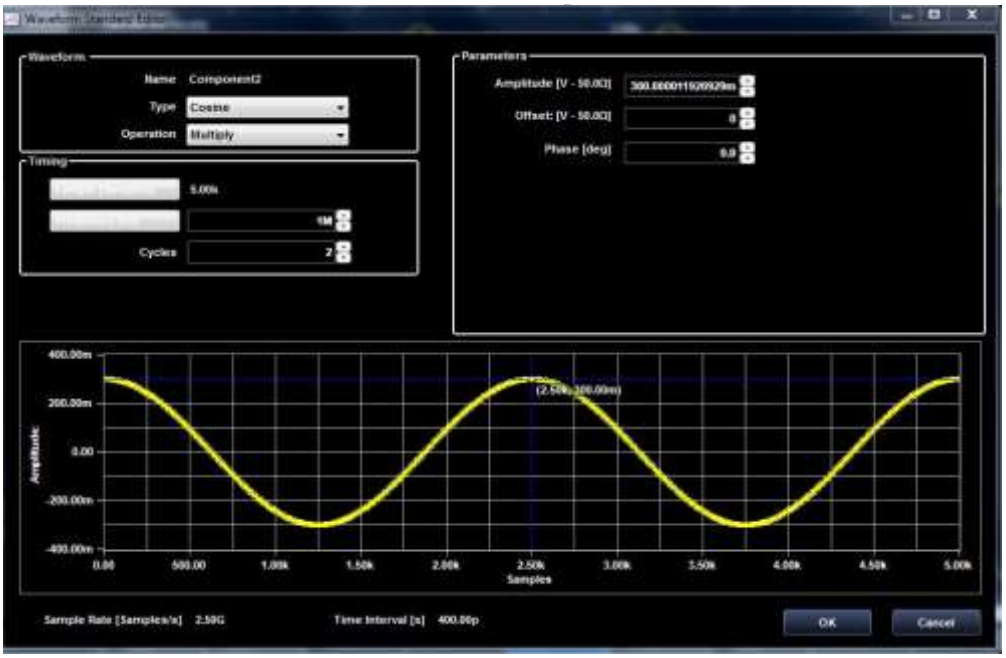
- Right click on the Segment3 of Wave4-0 to open the pop-up menu and select **Add Segment**.

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

- Select the Component 2 and press the Edit Waveform button



- The Standard Waveform Editor window is shown. 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

Model 676 High Performance AWG can be configured to work as a powerful Digital Pattern Generator. In this working mode Model 676 High Performance AWG provides the capability to emulate standard serial or parallel bus transitions or custom digital interfaces for system debugging and characterization.

PLEASE NOTE THE FOLLOWING:

- The vector memory depth is 32Mpoints /Ch.
- In Arbitrary Mode the maximum update rate is 1.25GS/s on 16 channels projects and 625 MS/s on 32 channels projects.
- Alignment between Digital and Analog Channels: by using the de skew controls on the *Settings Tab* (on page 47) 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.

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.

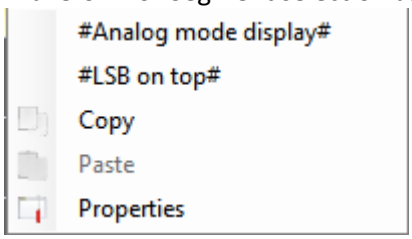
LEFT CLICK AND SELECTION

- Left Click on the analog *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.

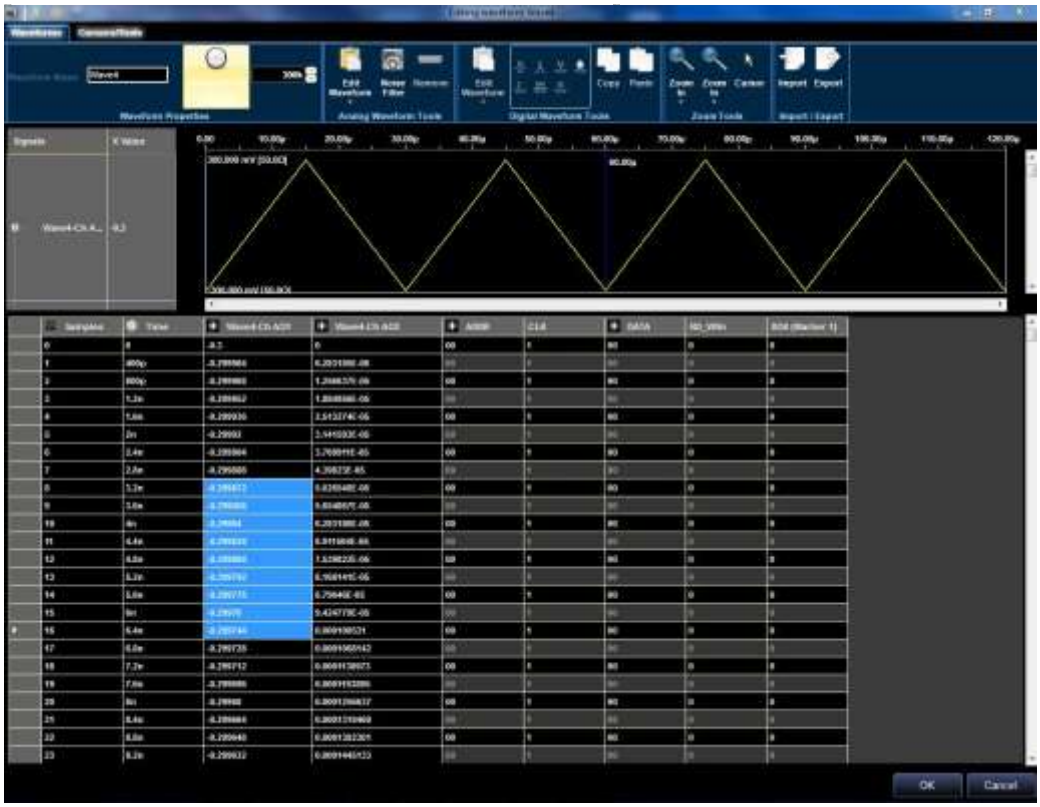


- **Analog mode display** - This option (available only for the buses) will represent a bus as an analog waveform. This is useful for example if an ADC or a DAC has to be tested.
- **Properties-** Open the digital waveform Property Window. You can use it to change signals/buses colors, plot height and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

- **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.

BUSSES

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.

You can change table values by pressing Digital Editor Waveform toolbar buttons or writing directly the value on the cell 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 a .csv or .txt ; the file is comma separated with an header on the first line.
PLEASE NOTE THE FOLLOWING:

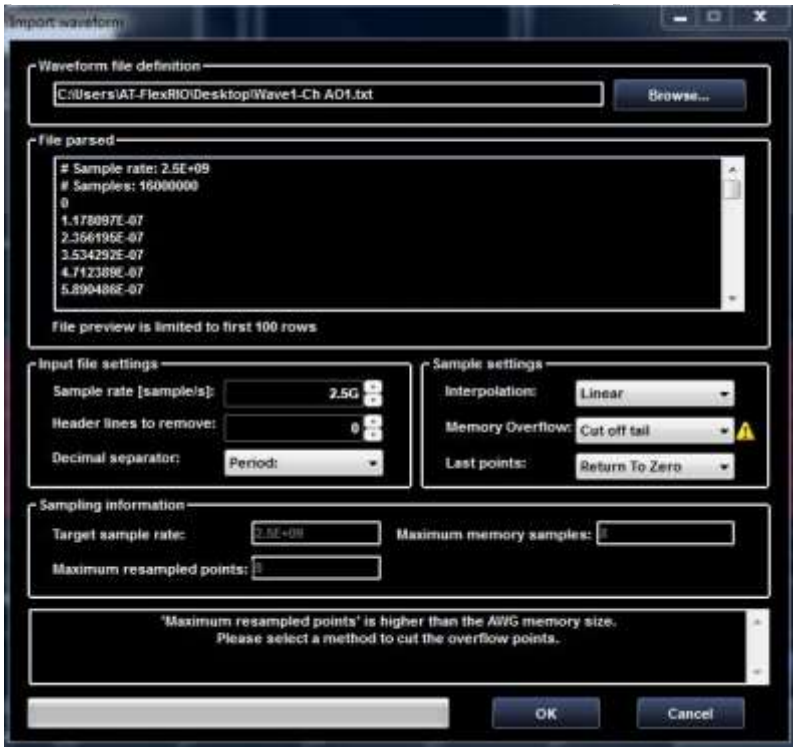
- The disabled cells in the Digital single signals/buses are not editable.
- The changes made on the table cells are also shown on the Mixed Waveform Editor directly above it.

You can change table values by pressing Analog Editor Waveform toolbar buttons, changing the values on the data grid; 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  .
The analog data will be exported into a .csv 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

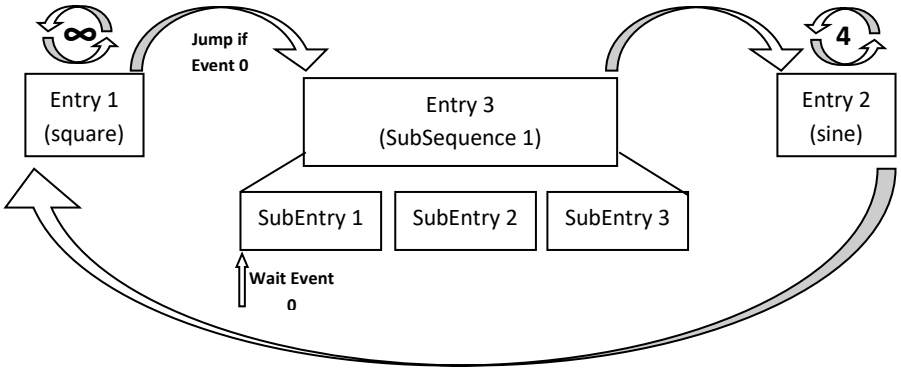
To Import data, select the **Import data…** right click menu item or click on the import menu icon  .
The digital data import format is a .csv or .txt ; the file is comma separated and it can have an an header on the first line.
The Import Waveform window will open, please refer to the Import File section for the explanation about this form.


Import waveform
Waveform file definition
C:\Users\AT-FlexRIO\Desktop\Wave1-Ch AD1.txt
Browse...
File parsed
Sample rate: 2.5E+09
Samples: 16000000
0
1.178897E-07
2.356195E-07
3.534292E-07
4.712389E-07
5.890486E-07
File preview is limited to first 100 rows
Input file settings
Sample rate [sample/s]:
2.5G
Header lines to remove:
0
Decimal separator:
Period
Sample settings
Interpolation:
Linear
Memory Overflow:
Cut off tail
Last points:
Return To Zero
Sampling information
Target sample rate:
Maximum memory samples:
Maximum resampled points:
'Maximum resampled points' is higher than the AWG memory size.
Please select a method to cut the overflow points.
OK
Cancel

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.



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 SubSequence with 3 entries.

Please note the following:

- In **ARB Mode** 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 opens called *Input Waveform Properties* to specify the properties of the new sequencer entry that is going to be created.

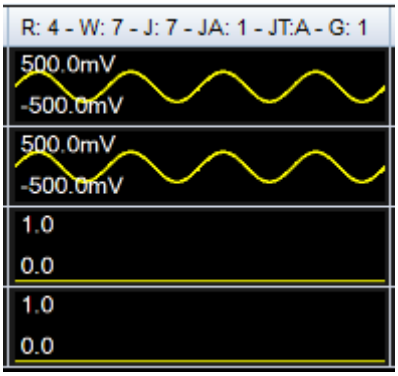
Input Waveform Properties

1. An entry can be repeated if the **Wave Repetitions** is specified.
   The maximum number of repetitions is 2097150.
   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.

PLEASE NOTE 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.



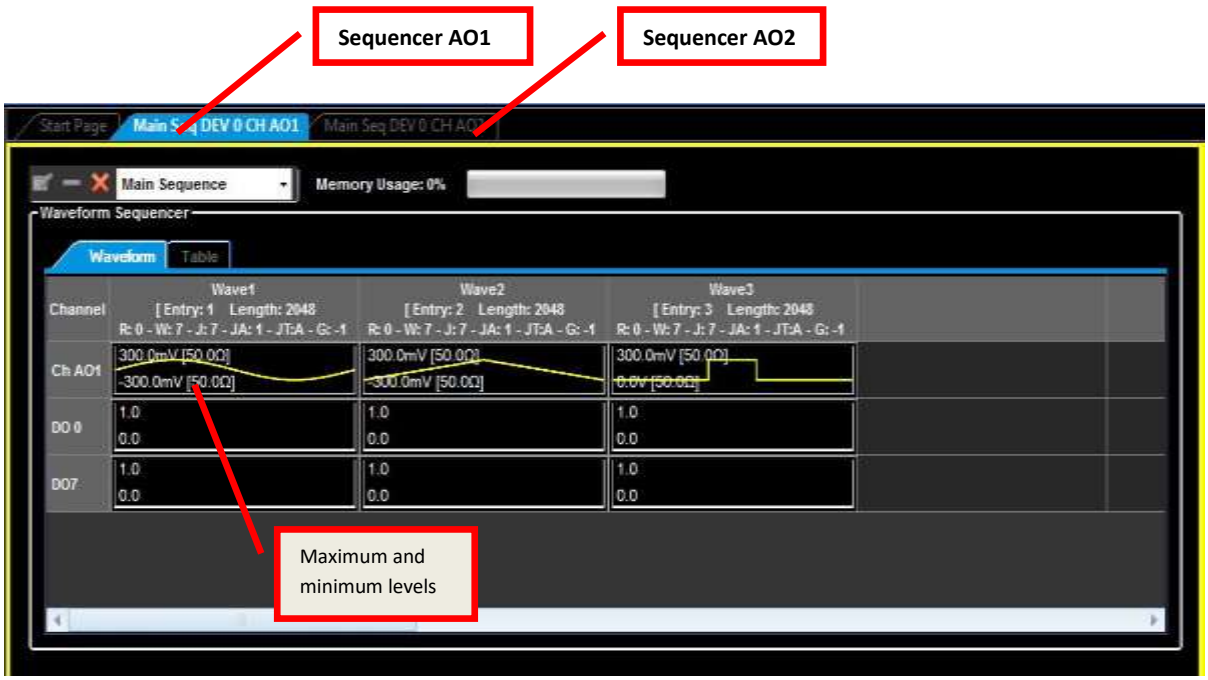
R :4 = 4 repetitions
W :7 = Wait for Event 7 (None)
J: 7 = Jump if Event 7 (None)
JA:1 = Jump Address 1
JT:A = Jump Type Asynchronous
G:1 = Go To Address 1

Sequence in Arbitrary mode

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

generated.

Single Sequencer projects have one sequencer called Main Sequencer that controls all analog/digital channels waveform generation.
Multi Sequencer projects have 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.



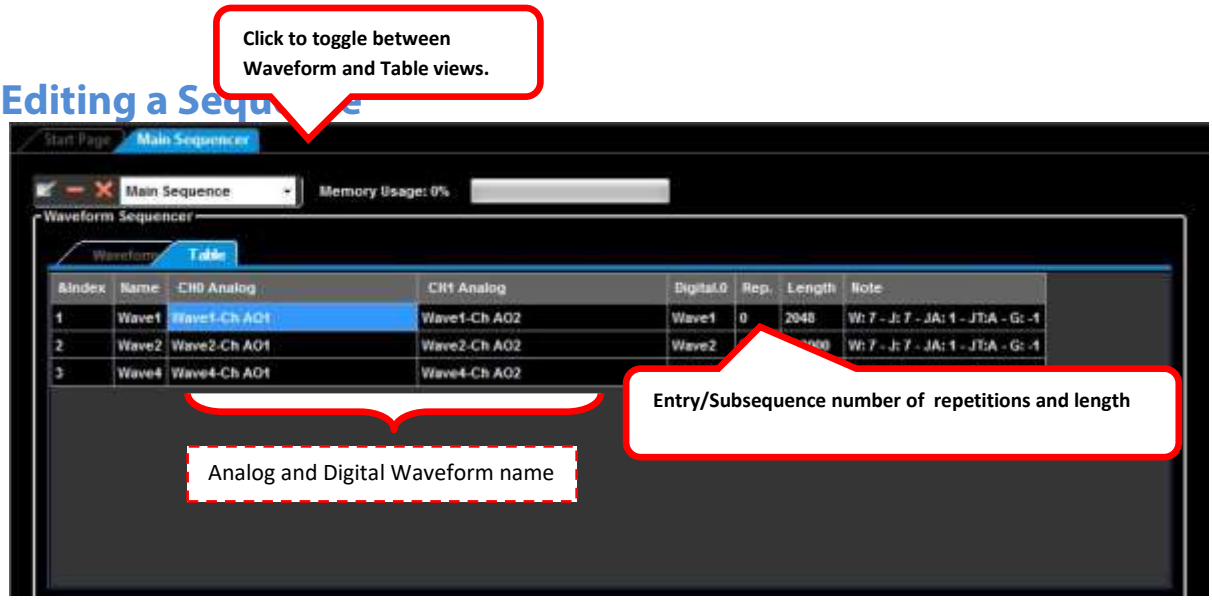
PLEASE NOTE THE FOLLOWING:

- 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.
- 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.
- Drag a waveform from the Waveform List window and drop it to a cell of the Sequence window to insert it in the sequencer.
- 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.*
- When you insert a waveform in a sequencer cell, its maximum and minimum levels are displayed on the left margin of the cell.
- When Continuous, Triggered, or Gated is selected in the Run mode, the Sequence window contains one waveform onl

Waveform View of the Sequencer

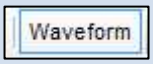
Note: The sequencer provides both a **Waveform** and **Table** view of your sequenced waveforms. Both views provide the same data; however, Waveform provides a graphical representation of your sequence, while the Table shows a list.

Table View of the Sequencer

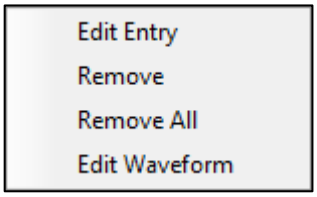


The Sequence Window allows you to select waveforms from the Waveforms TAB for generation by the Model 676 instrument.
Drag a waveform from the Waveform TAB to a cell of the Sequence window to insert it in the sequencer. **Remove** waveforms to the Sequencer using the Sequence Window Toolbar.

Sequence Window Toolbar

	**Edit Entry** – click a waveform and press this button to open 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.

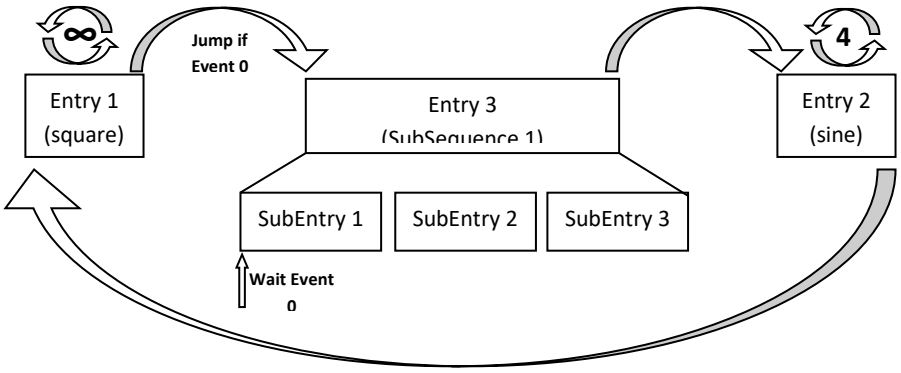
A Right Click on a Waveform cell activates a pop-up menu



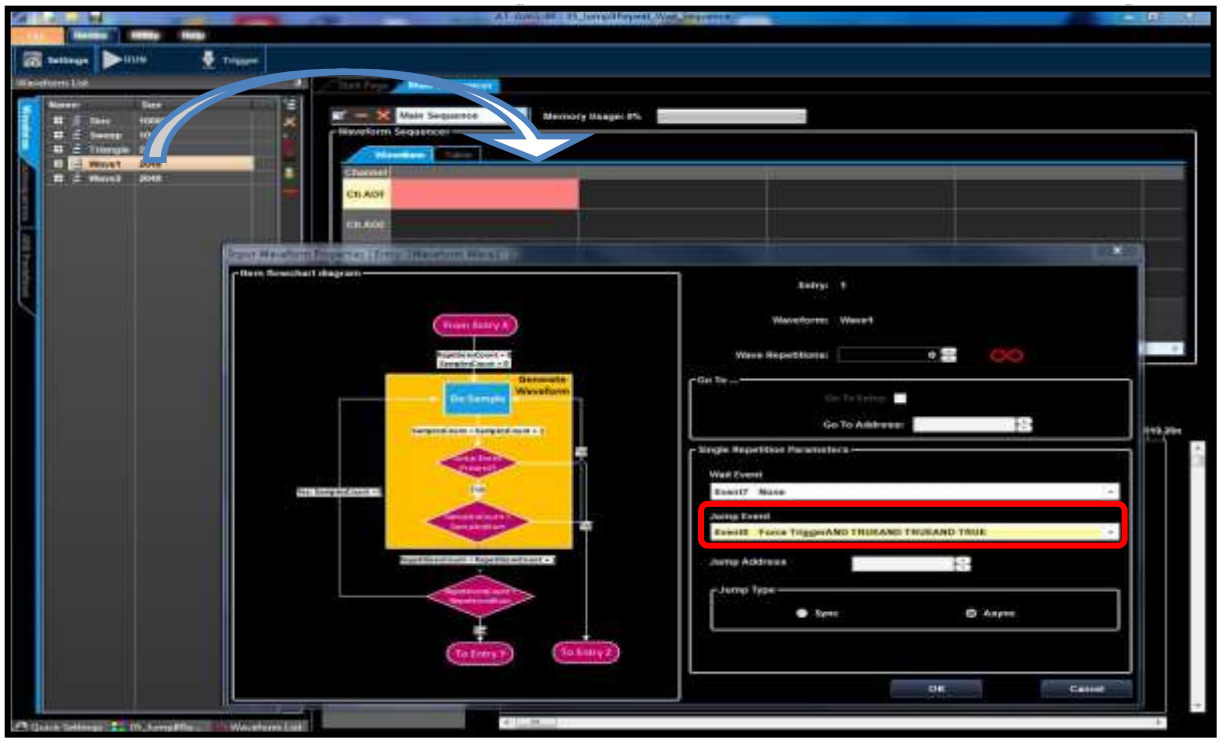
- **Edit Entry**: opens the *Input Waveform Properties* window to change the current entry properties.
- **Remove:** removes all the waveforms in the selected entry from the sequencer.
- **Remove all:** removes all the entries from the sequencer.
- **Edit Waveform:** opens the *Editing Waveform Window*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

How to Create a Sequence

Prerequisites: JumpIf_Repeat_WaitSubsequence project

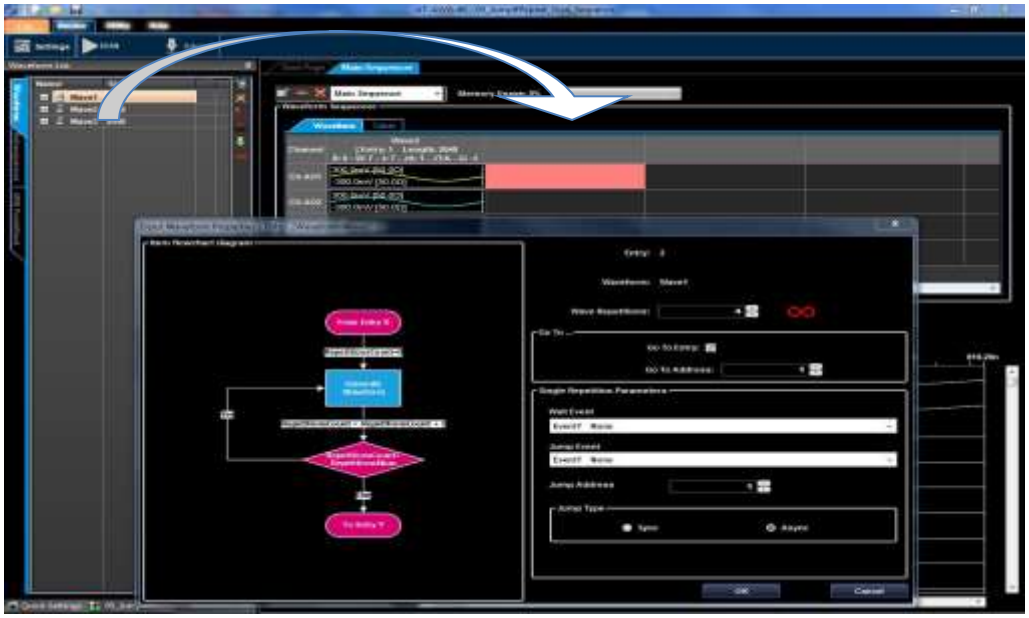


1. Open the JumpIf_Repeat_WaitSubsequence project
2. Press the  button in the Sequence Window toolbar to remove all the sequencer entries.
3. On the Waveforms TAB, drag the Square waveform and drop it in the first entry of the sequencer.



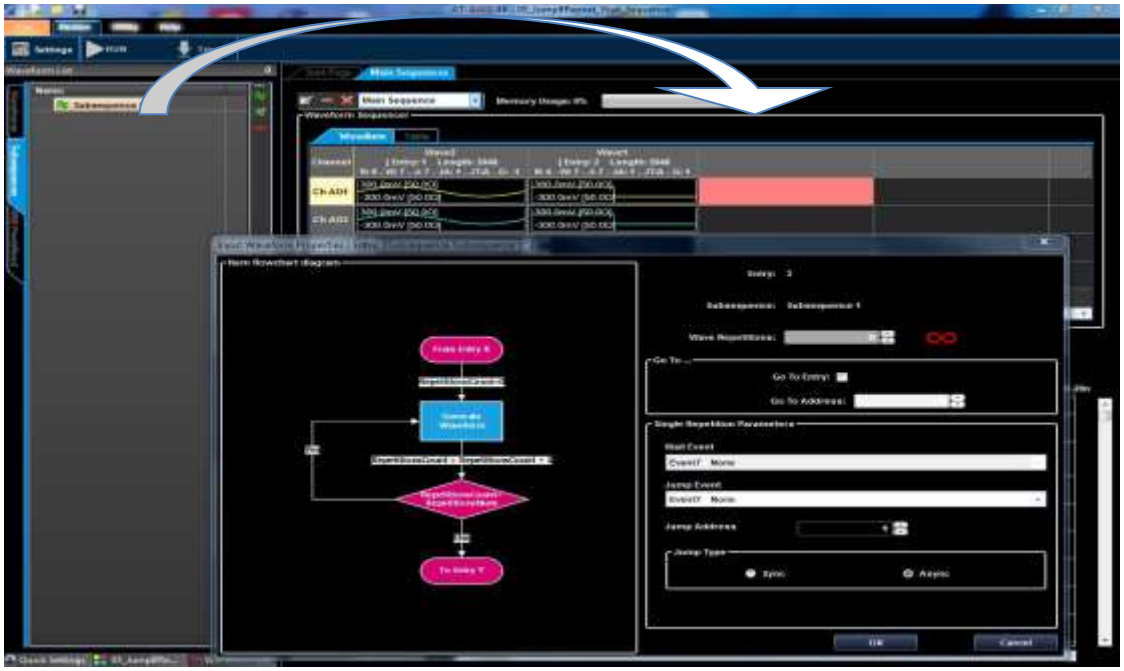
On the *Input Waveform Properties* window select infinite repetitions and Event0 (Force Trigger AND True AND True AND True) in the Jump Event drop-down list.
The Jump Address control is disabled because there are not entries in the sequencer yet.
Press OK to confirm.

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

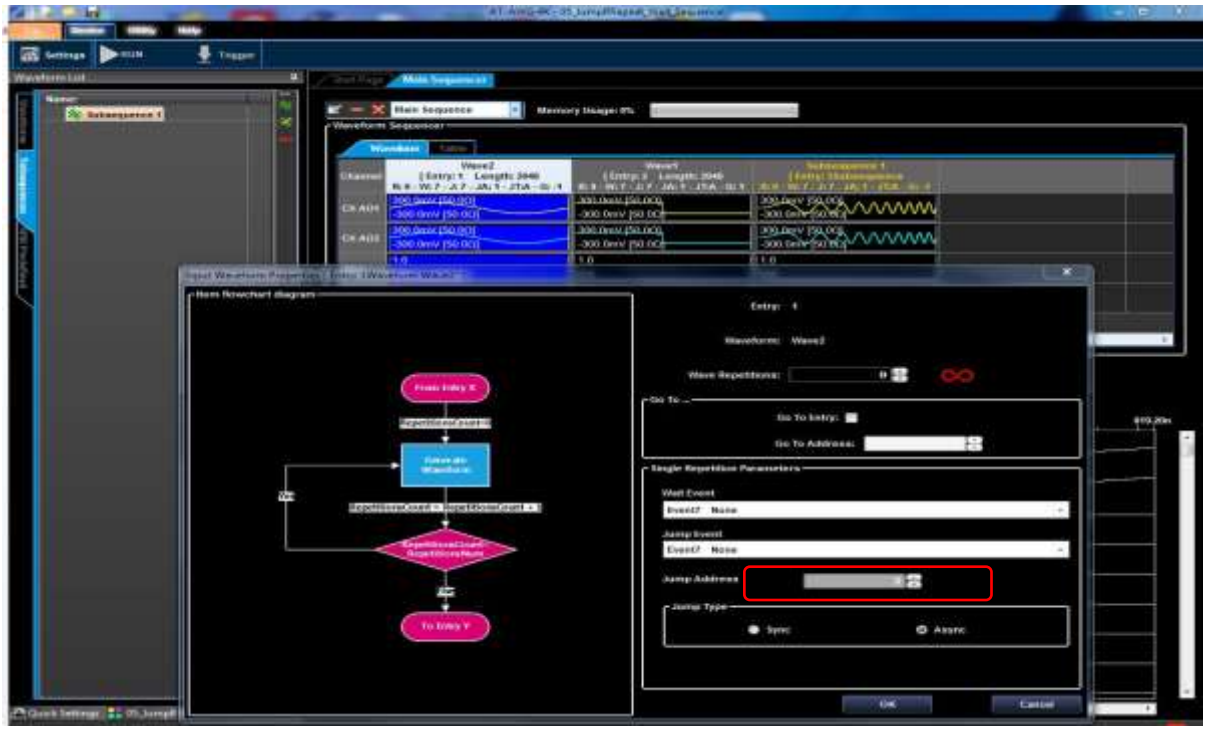


On the *Input Waveform Properties* window select 4 repetitions, mark the Go To Address checkbox and type 1 as Go To Address value. Press OK to confirm.

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



6. 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.



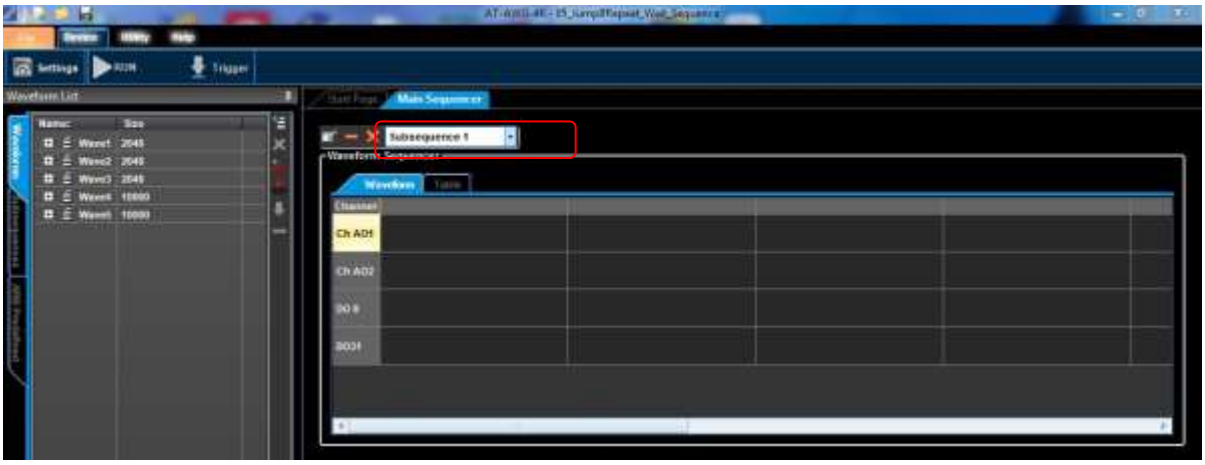
Modify the Jump Address field and type 3 as address of the jump. Press the OK button.

How to Create a Subsequence

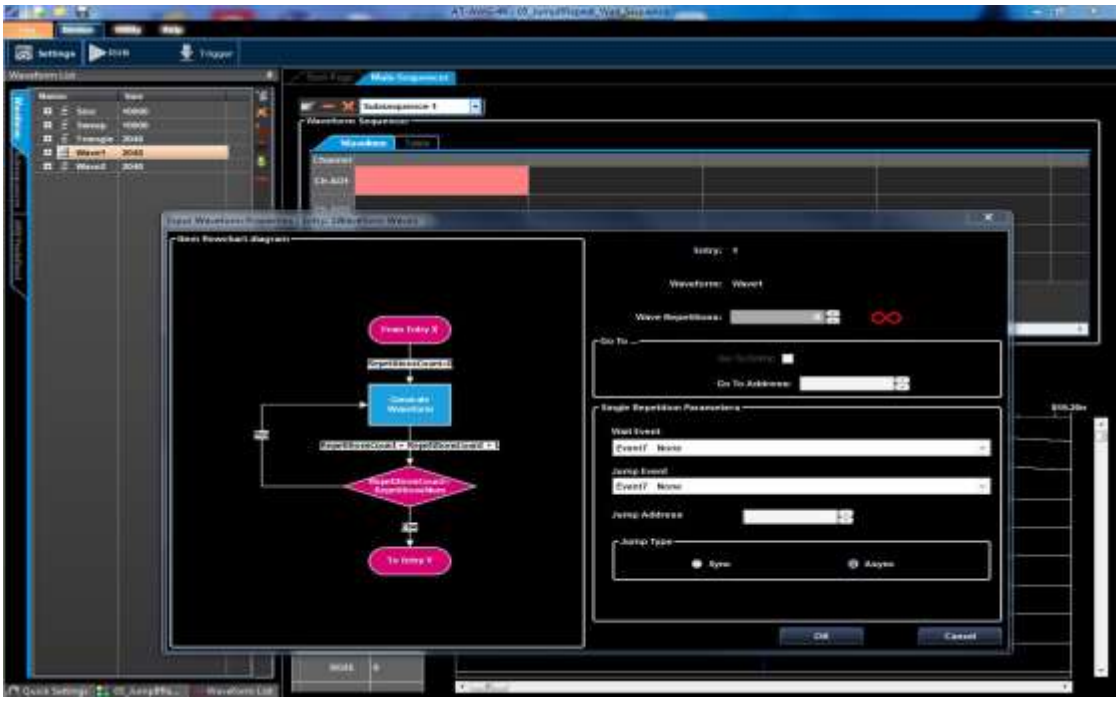
Prerequisites: JumpIf_Repeat_WaitSubsequence project

- Open the JumpIf_Repeat_WaitSubsequence project

- On the Sequences TAB press the  New Subsequence button.
- The new empty subsequence is shown on the Main Sequencer Window



- 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.



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

- 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.

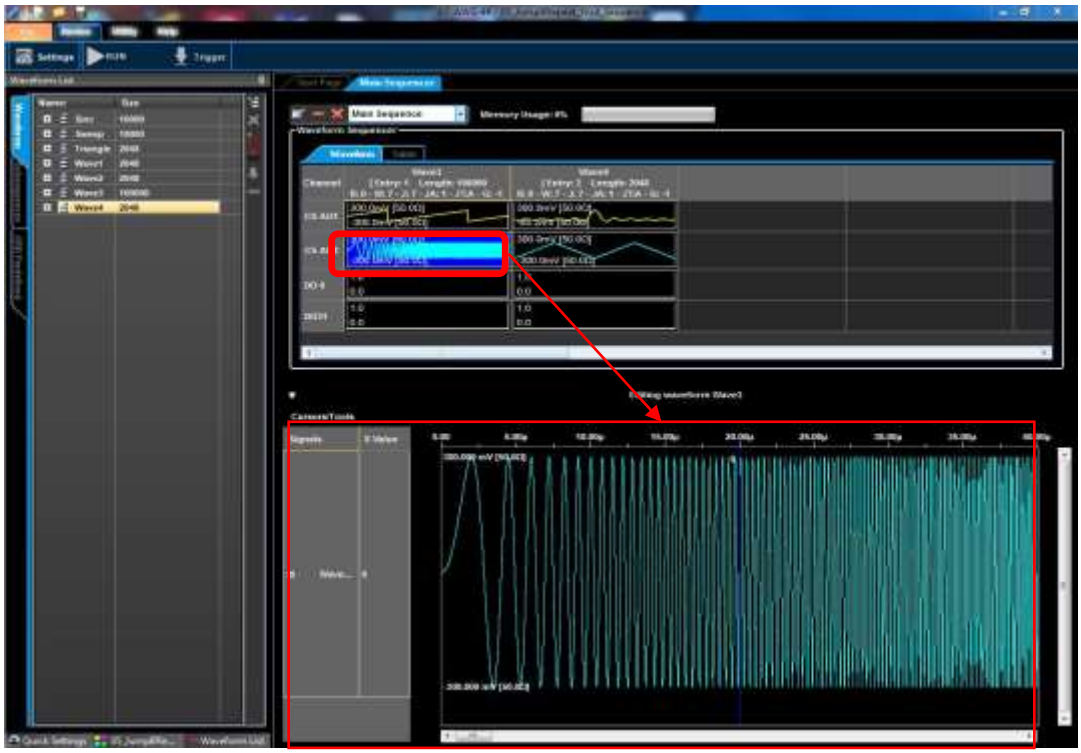
Waveform View Window

A selected element in the Sequence Window or in the Carrier Graph is displayed in the *Waveform View* window.

Important Note: this window is only available when the instrument is connected to an external monitor.

To the right 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.

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



- 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.

Waveform View Window Tools

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:

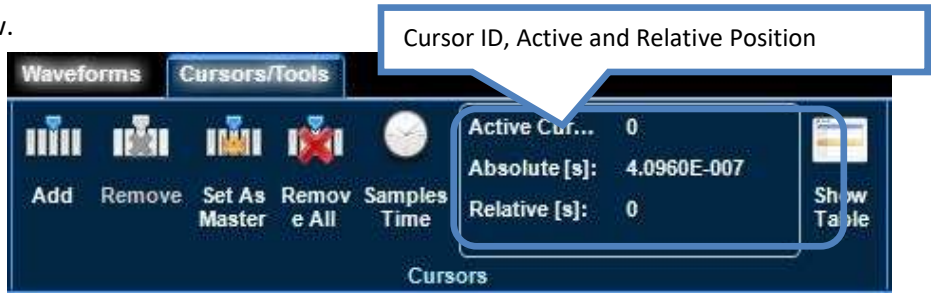
- The **Background Color** can be changed as desired.
- Change colors and turn the **Major** and **Minor Grids** on or off and change their line coloring.
- **Cursor Position** indicators can be turned on or off.

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

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.



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 View 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	

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 cursor in the cursor window and clicking the Master Cursor icon in the *Waveform View 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 cursor selected in the **Cursor** screen.
	Move a cursor by clicking and dragging a selected cursor.
	Remove all cursors by clicking the **Clear all cursors** button.

PLEASE NOTE THE FOLLOWING:

- 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 *Waveform View Window 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 *Waveform View Window*. It also is available in the aforementioned search section regarding the **Waveform View** screen.

You can search for a specific bus, signal, rising, or falling edge value depending on the selected waveform is analog or digital.

Activate the search option by clicking the **Search Settings** button  .
The *Search Settings* window is shown and used to provide your search criteria

Note: Depending on the Signal Type selected in the search list, the **Compare** and **Value** fields contains different options.

Select the signal or bus and then provide a specific value for the search

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.

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.

Select criteria on the Search Settings screen and click the **OK** button. The results are then shown on the **Waveform View Window**.
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.

Tutorials

Scenarios with detailed steps for performing typical tasks and setups using Model 676 High Performance AWG 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 Expert Rider SWG 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.

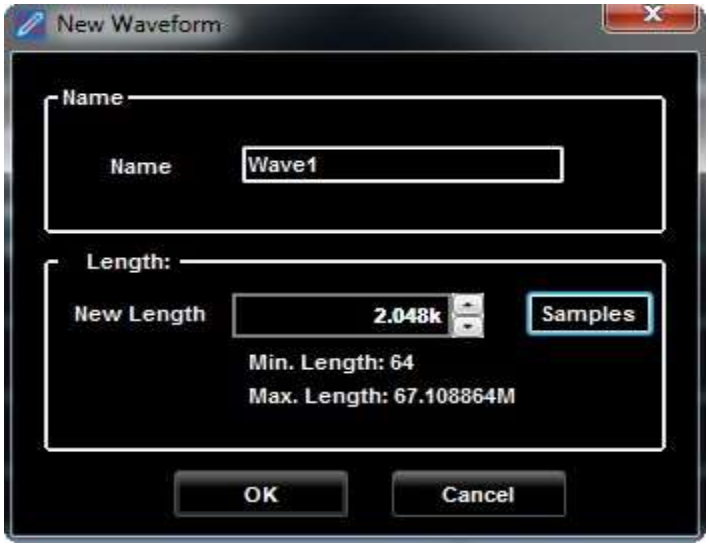
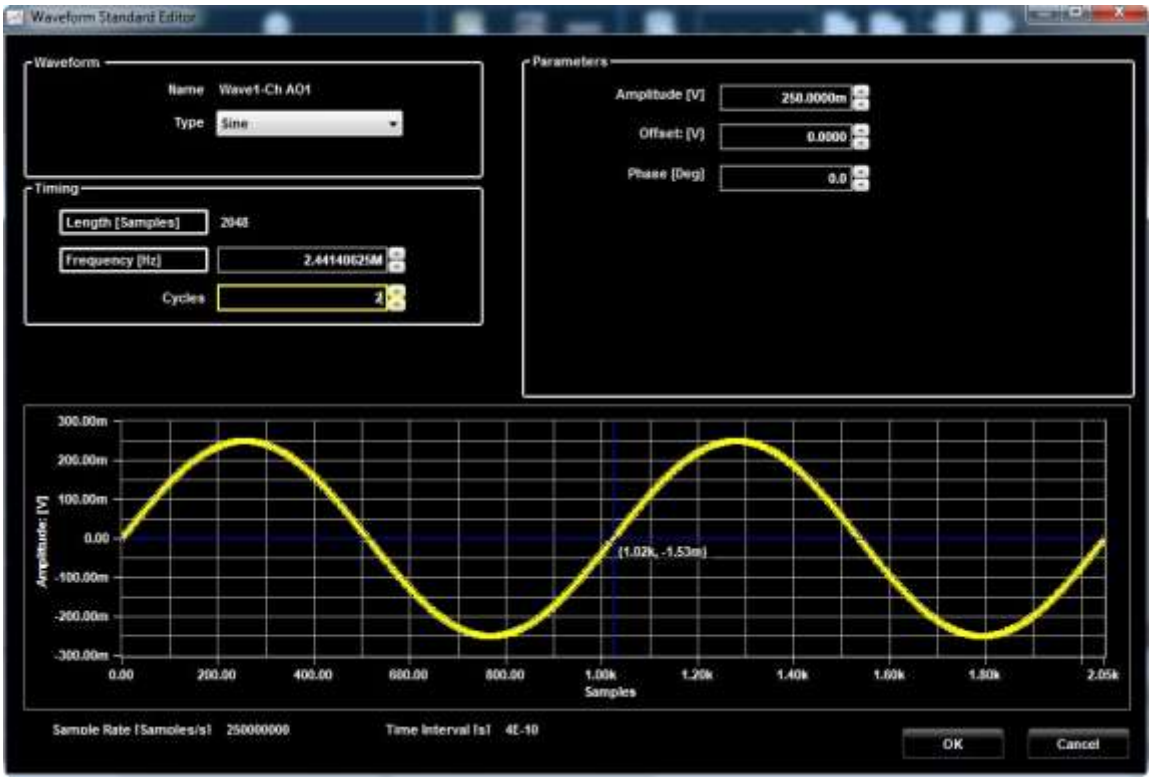


Figure 114 New waveform settings

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

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

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



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

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

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

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

The software loads the waveforms into the Model 676 High Performance AWG 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.

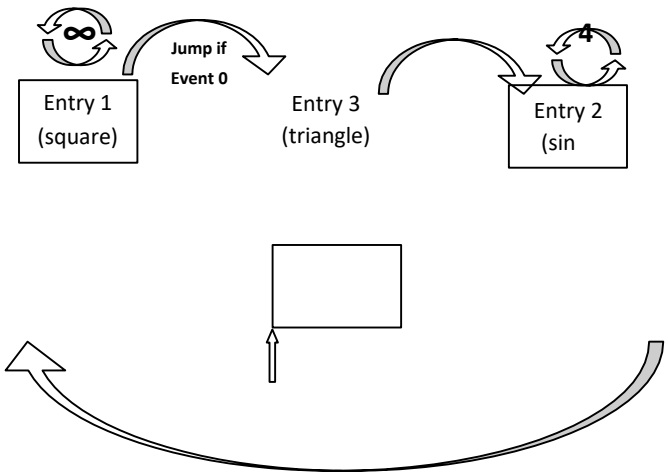


Figure 120 Single sequencer project

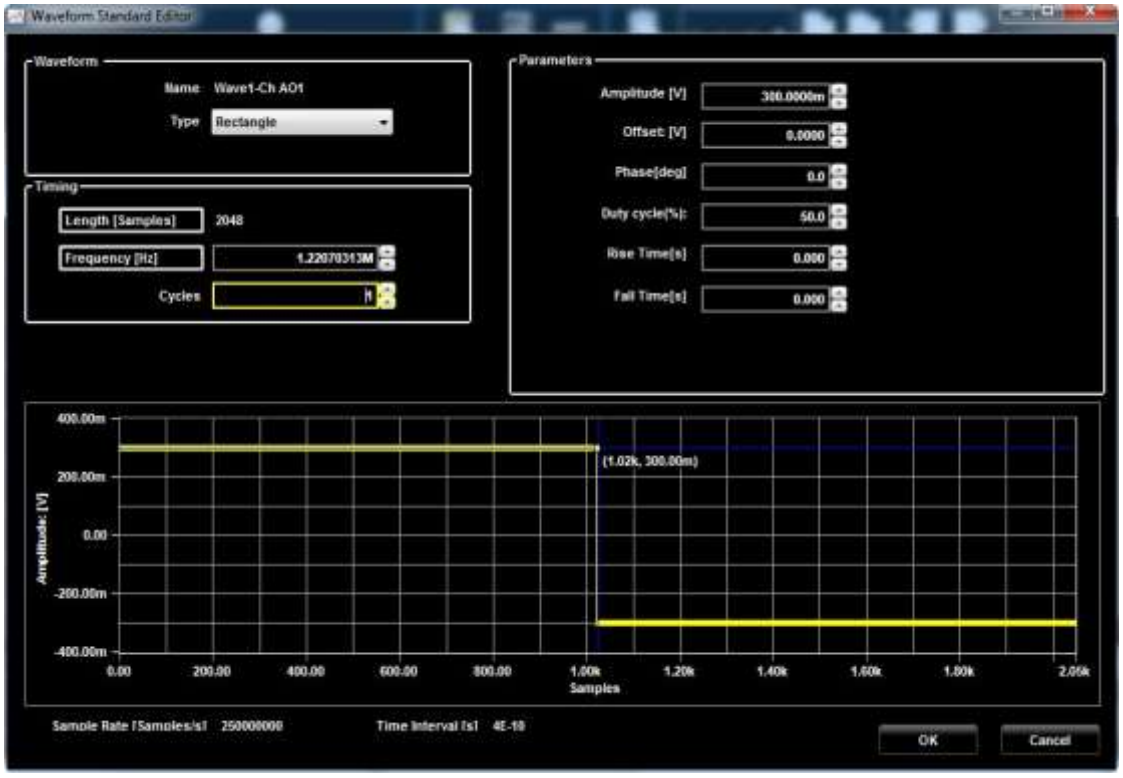
2. Add three waveforms to our Waveforms tab list.
3. Assign the following specifications for waveforms 1, 2, and 3 on the Waveform Standard Ed itor dialg box.
   Waveform 1 - Square waveform Type, 2048 Samples, 1 Cycle and 300 mv Amplitude.
   Waveform 2 - Sine waveform Type, 2048 Samples, 1 Cycle and 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.

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

button

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

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



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

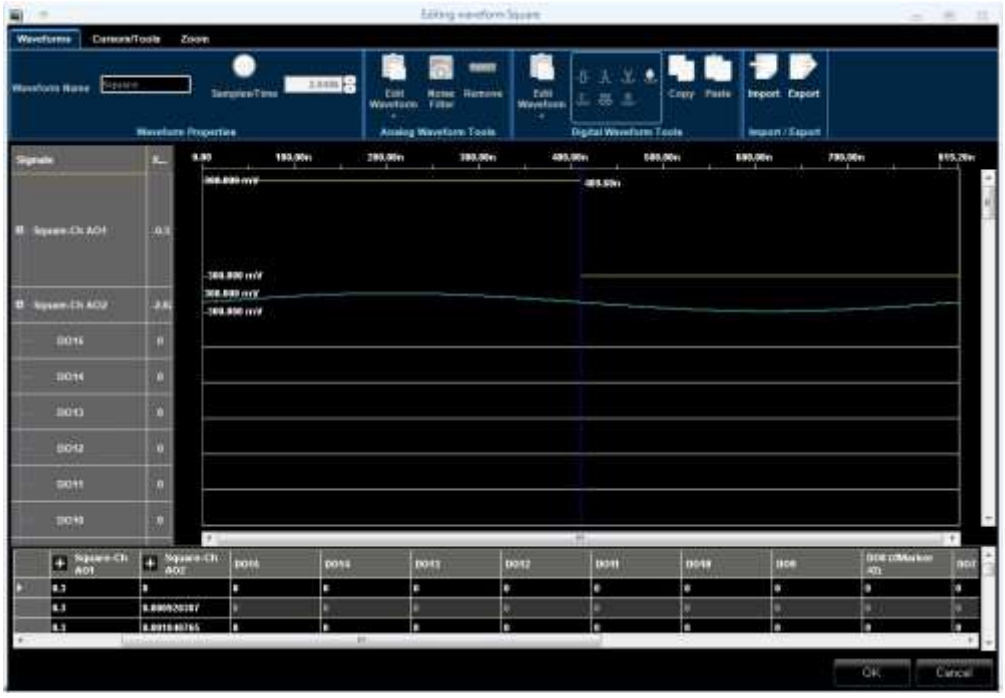
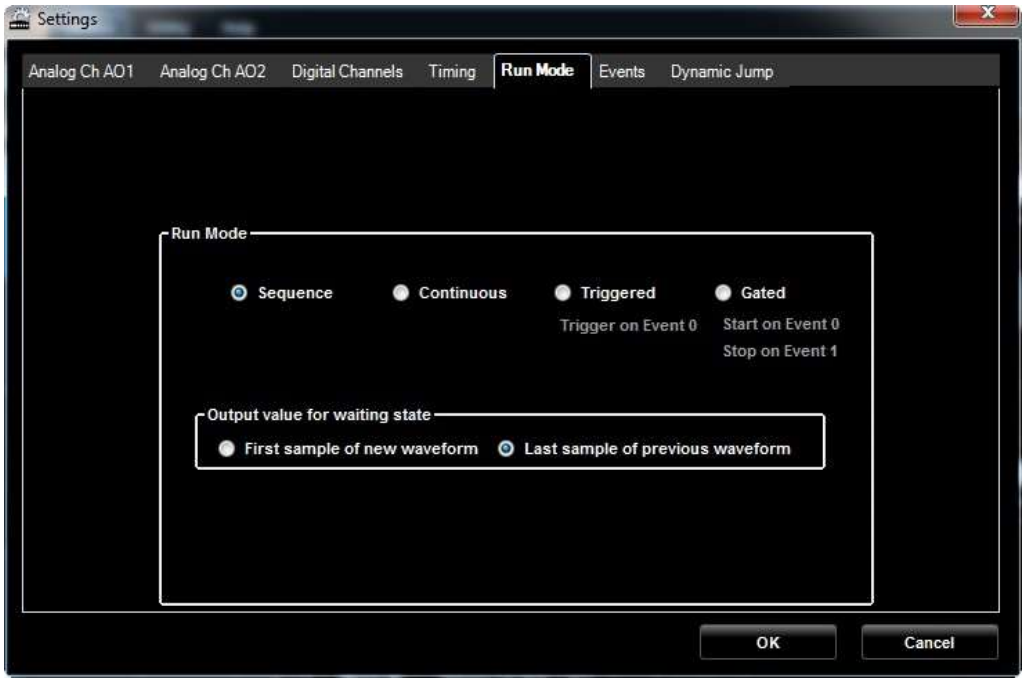


Figure 124 Edit waveform

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 

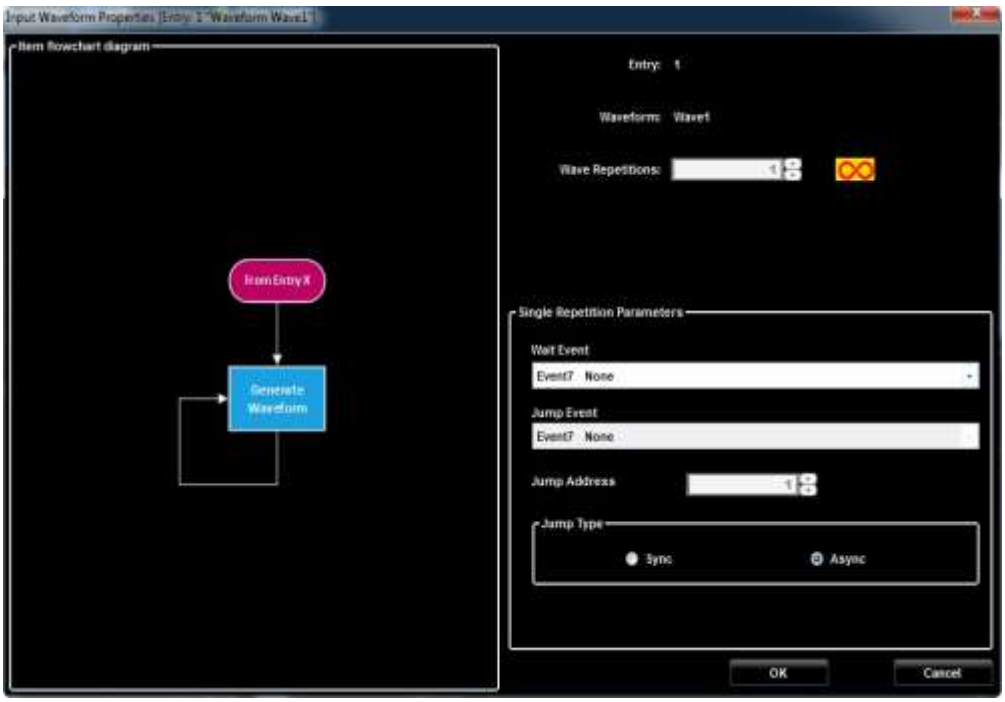
button. On the Settings pop-up screen's Run Mode tab, select Sequence.



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.



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.

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.

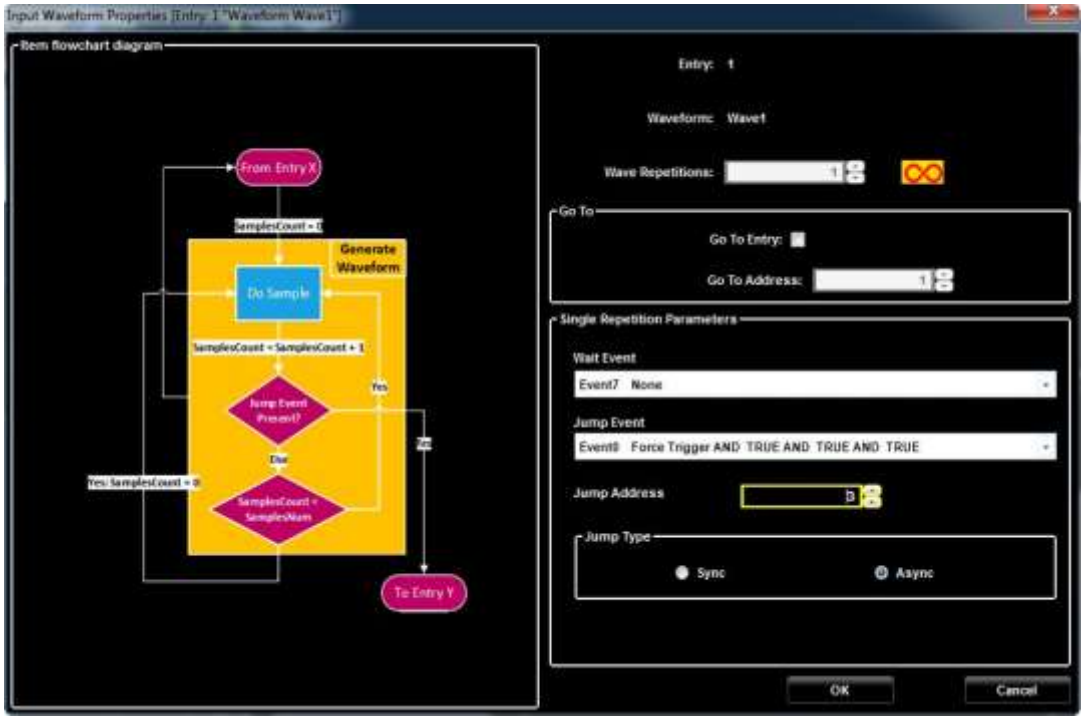


Figure 129 Input waveform properties

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.

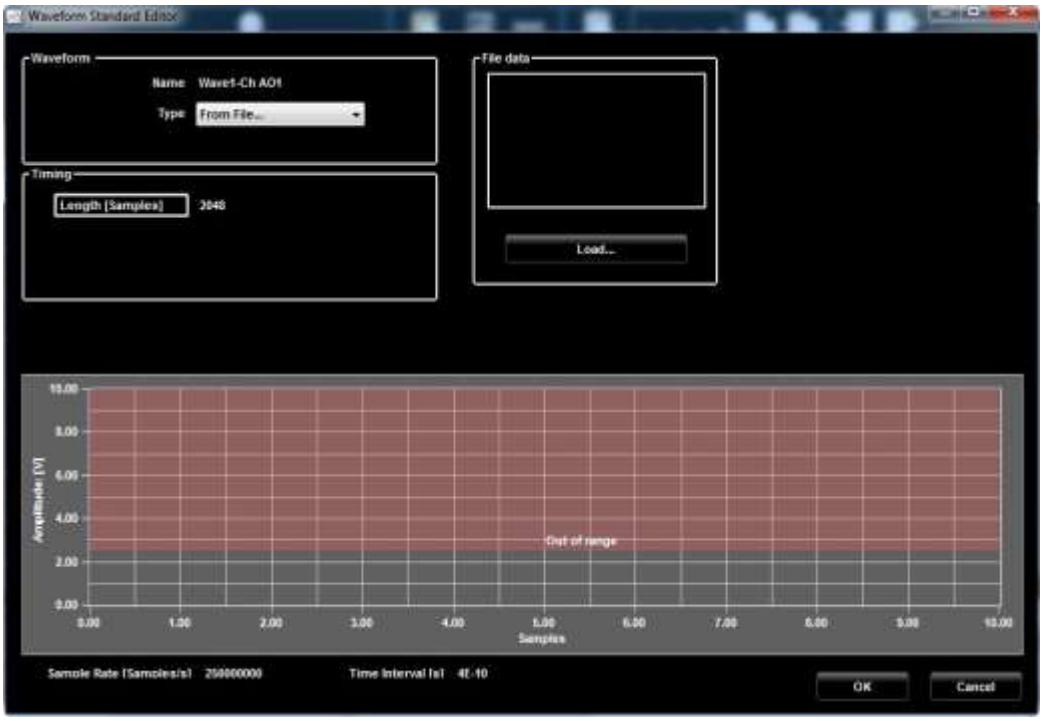
23. The software loads the waveforms into the Model 676 High Performance AWG instrument and starts generating the waveforms. The sequence is generated on the AO1 SMA output. You can connect an oscilloscope to this output and analyze the signal
24. Stop generating waveforms by pressing the Run/Stop button 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:

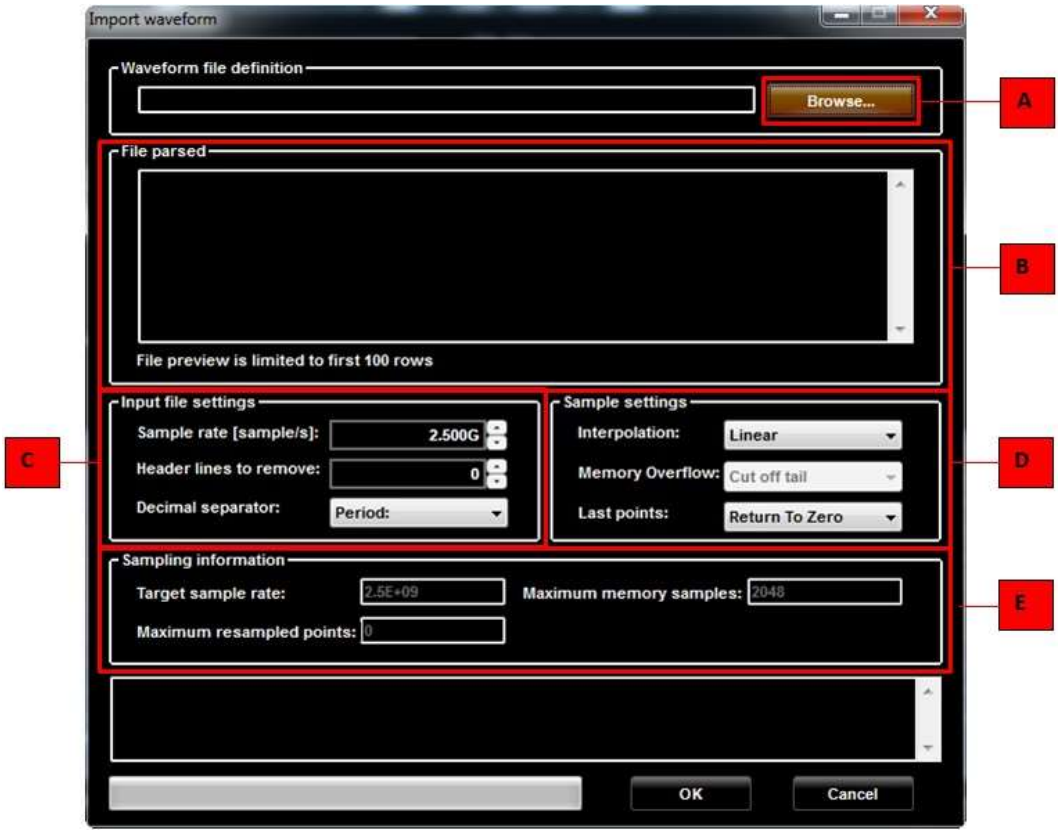
1. Click the New Mixed Waveform button. 
   The NEW Waveform window is shown. Type the name of waveform"Wave1" and choose 2048 for the samples length of the waveform. Chick OK to confirm.

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



Waveform  button.

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.



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.

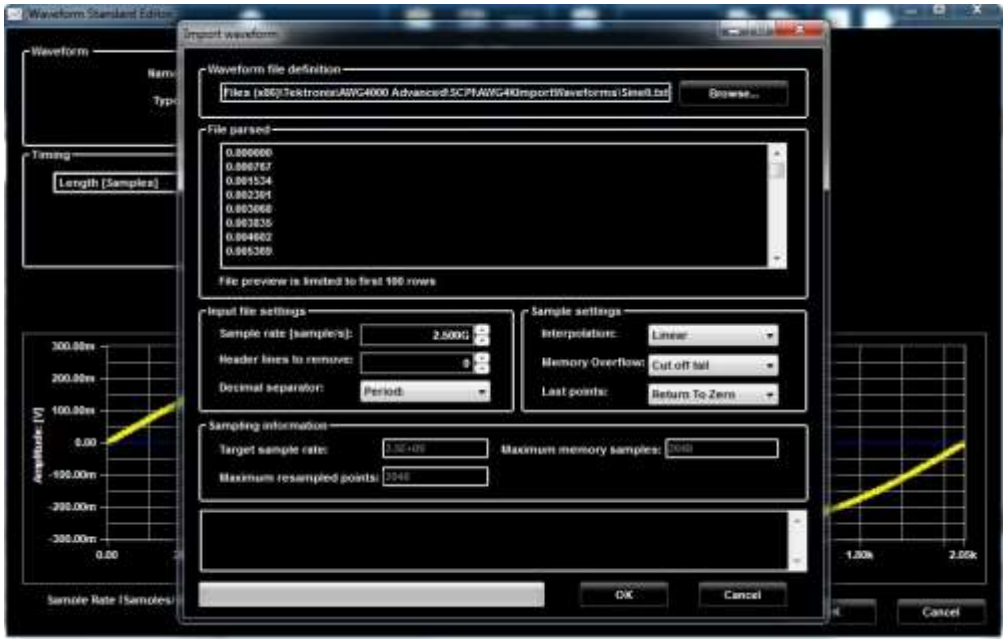
B. The File parsed section then shows the first 100 samples of the imported waveform.

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.

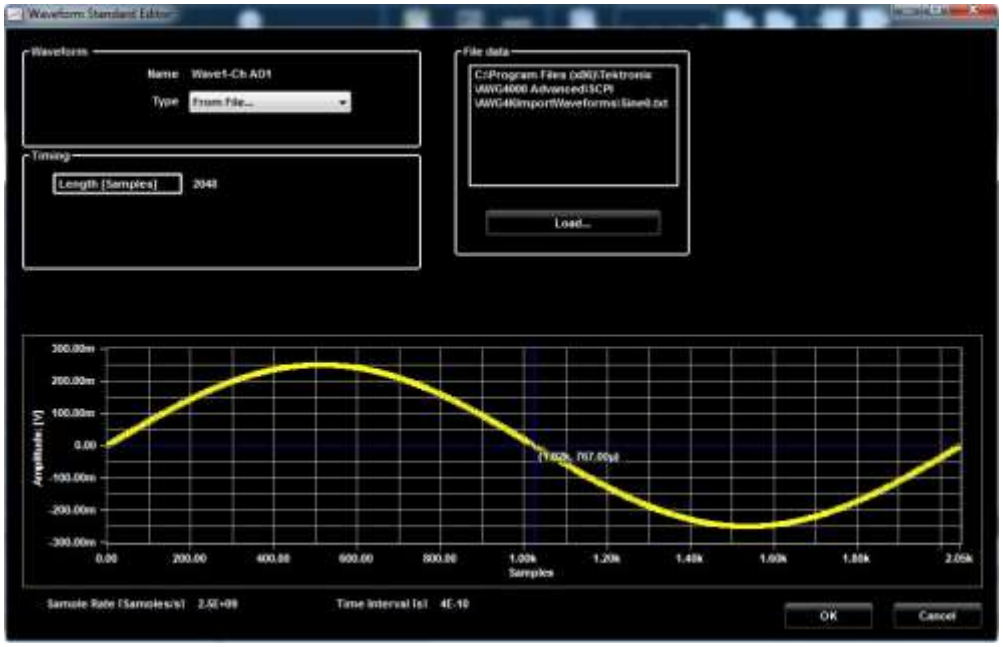
E. If the imported waveform has a sample rate greater than the Model 676 High Performance AWG target sampling rate (2.5 GS/s), the waveform points are re-sampled and shown on the Sampling information fields.

4. Browse from C:\Program Files (x86)\Active Technologies\EXPERT RIDER AWG 4000\SCPI\AWG4KImportWaveforms\Sine0.txt file



Browse waveform

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



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.

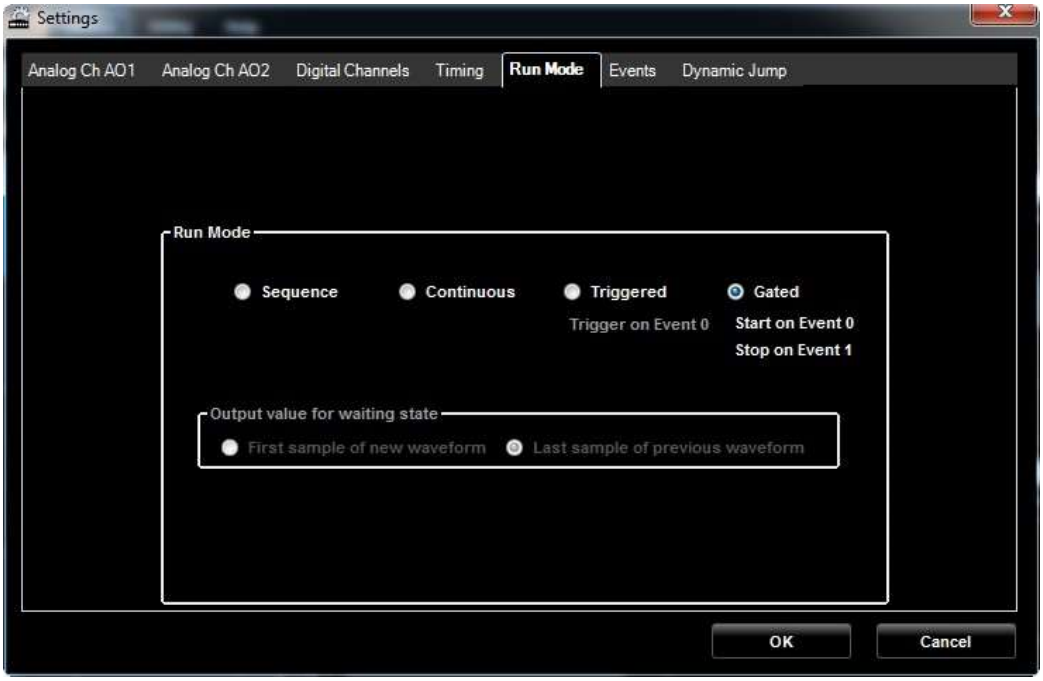
   To Component2, select DC Level as Type and 1.2 in the Offset[V] field as multiplication factor.

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.



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

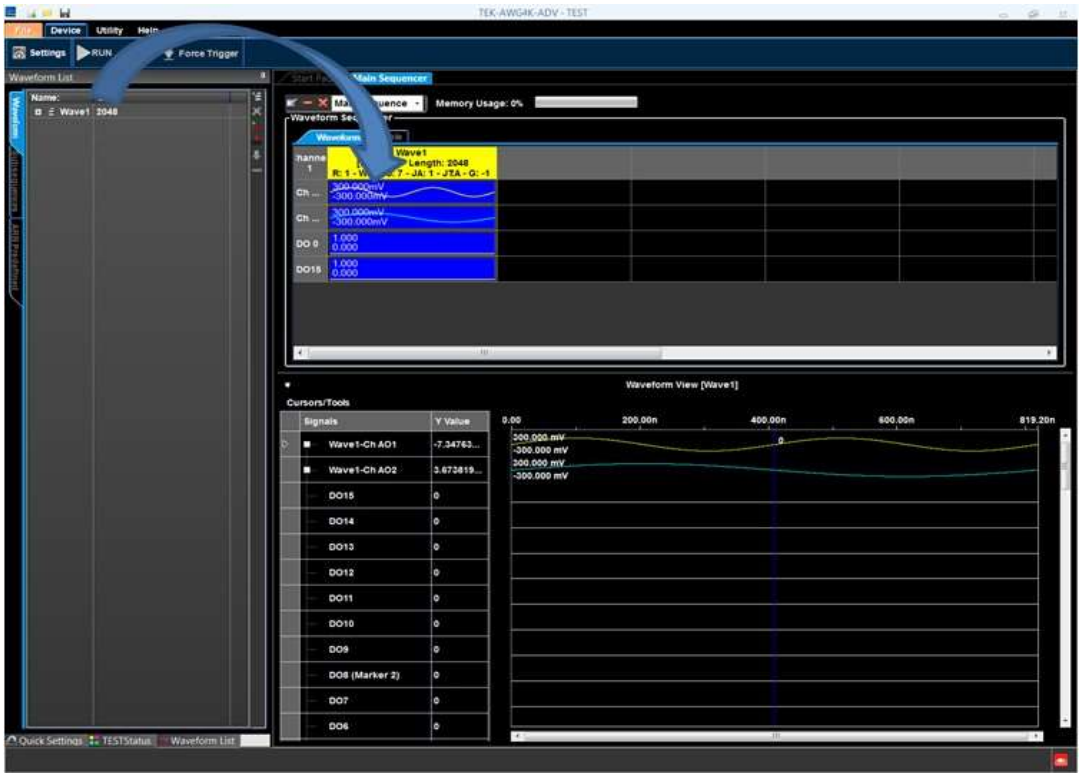


Figure 139 Edit waveform sequence

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 Model 676 High Performance AWG may optionally be configured to work as a powerful Digital Pattern Generator. When the The Model 676 High Performance AWG 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.

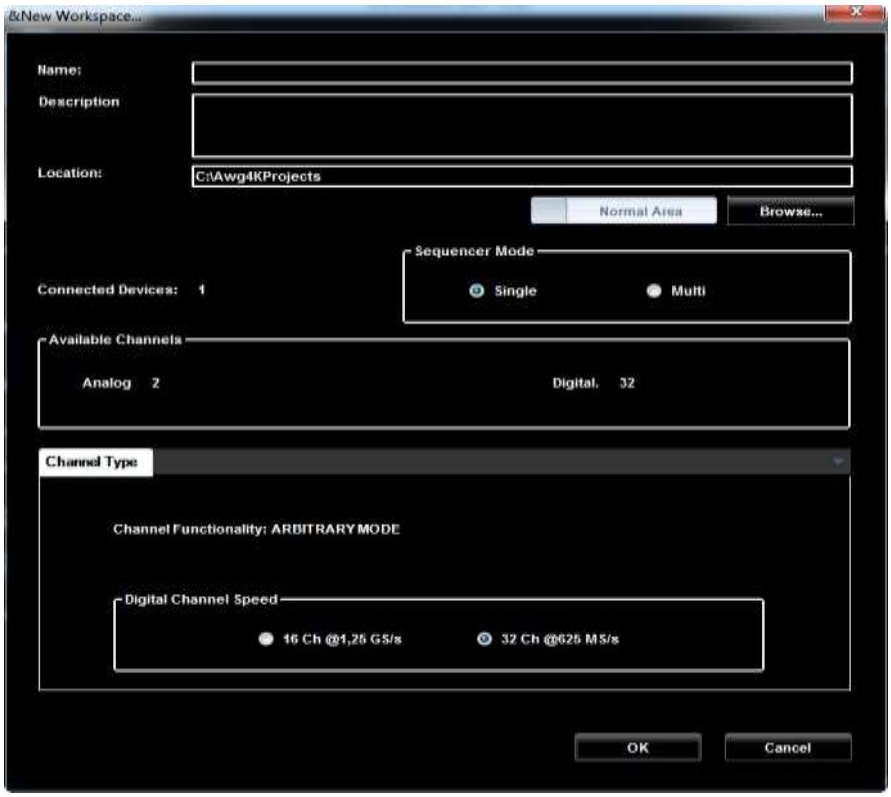


Figure 140 New workspace

IMPORTANT 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.

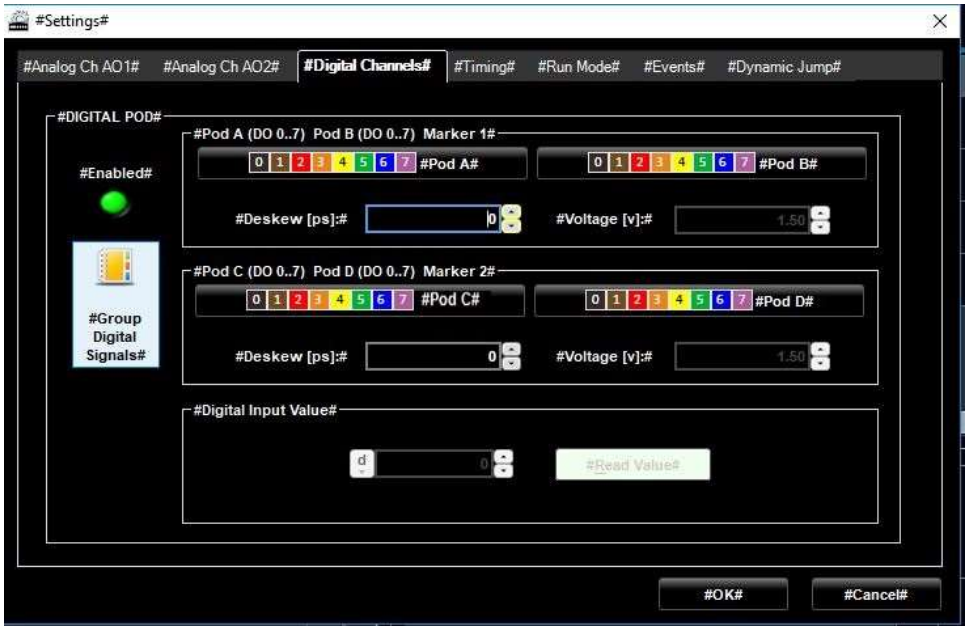
NOTE:

Four Mini-SAS HD connectors provide 8 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.

For slower applications, LVDS to LVTTL are available as well

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



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

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.

New waveform

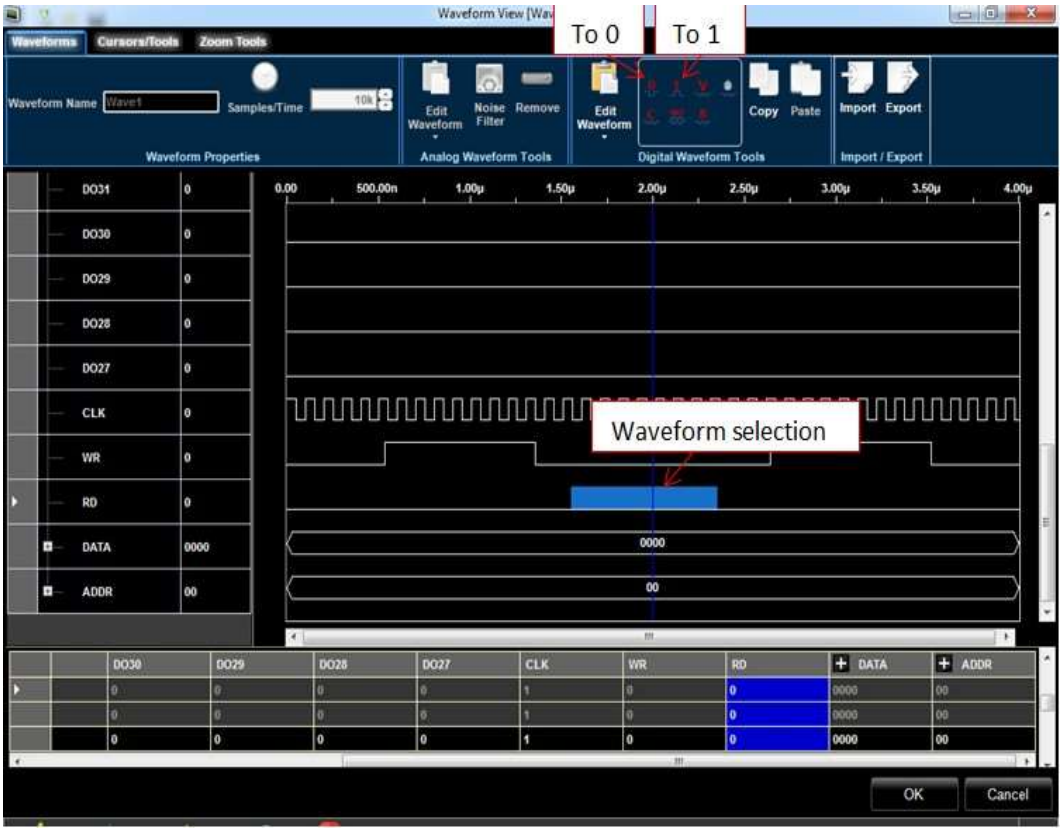
10. The Editing Waveform Window is shown.

Editing waveform window

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

Editing waveform

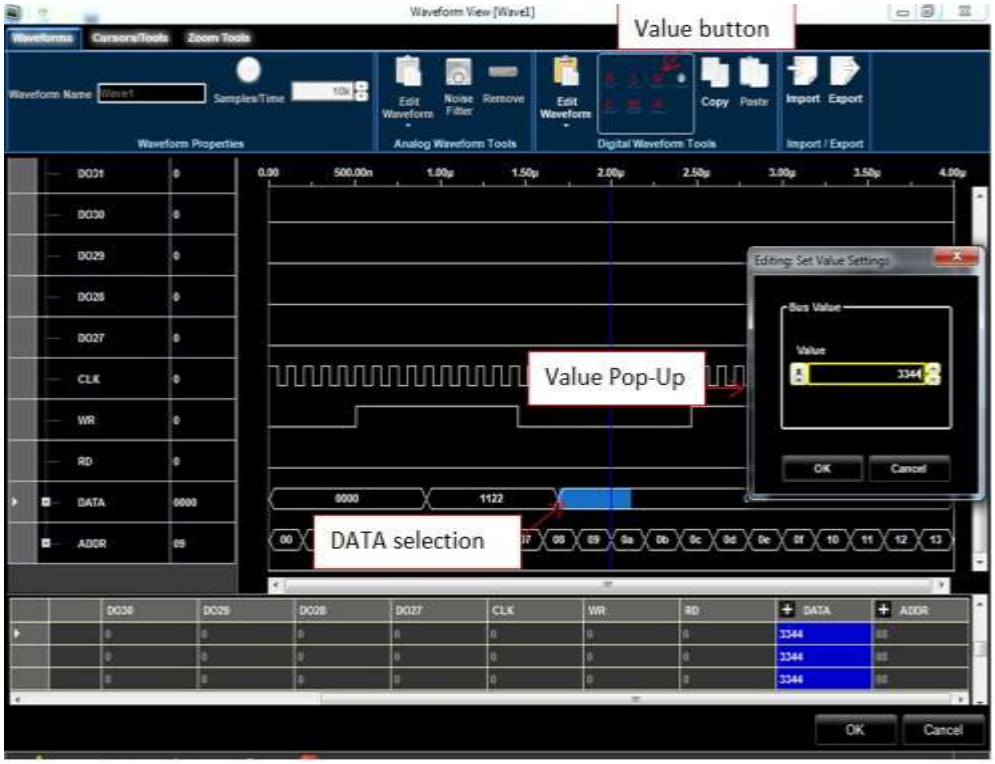
12. Now, define WR and RD signals similarly, by selecting a portion of the digital channel graph, and clicking the desired button from the following choices:



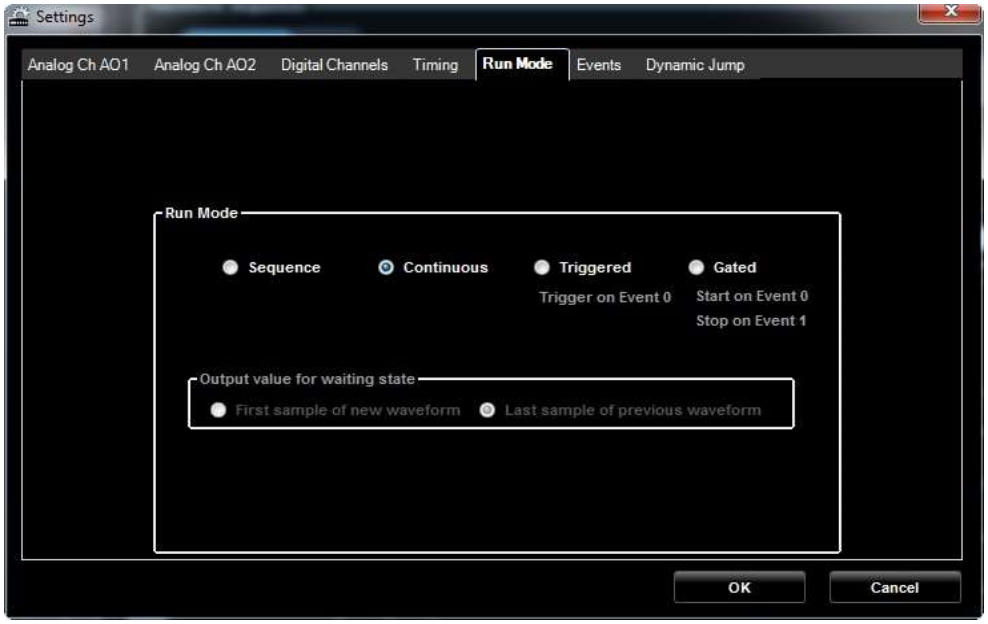
13. Now define the ADDR bus by selecting the ADDR row and use the Counter Button to modify settings on the corresponding pop-ups.

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:



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



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

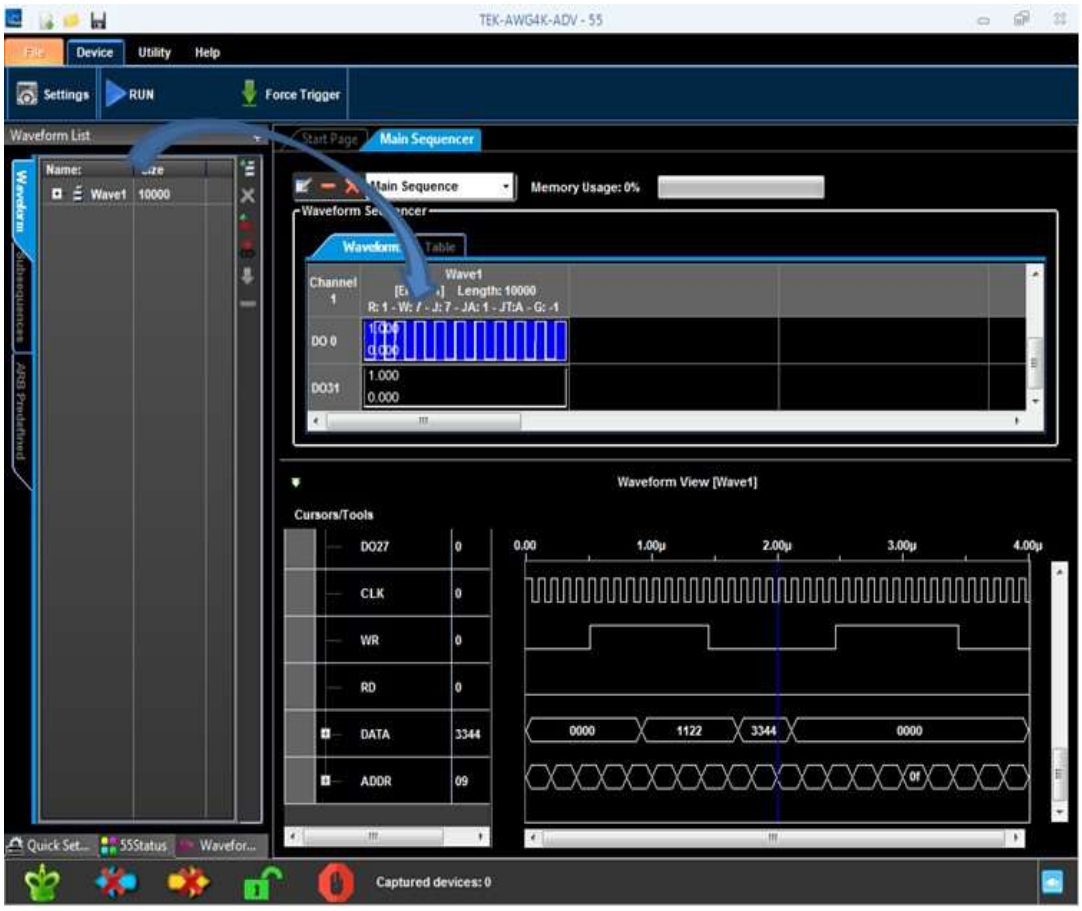


Figure 150 Drag Wave1

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 Expert Rider AWG App.
   You should buy the "SSD Solid State Drive for Secure Area Kit" and store all the sensible data in this second hard disk
   IMPORTANT: before returning the instrument to the service center, you should remove this second hard disk.
4. Select Single as Sequencer Mode
5. Select 16Ch@1.25 GS/s as Digital Channel Speed.

NOTE:

Four Mini-SAS HD connectors provide 8 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**.

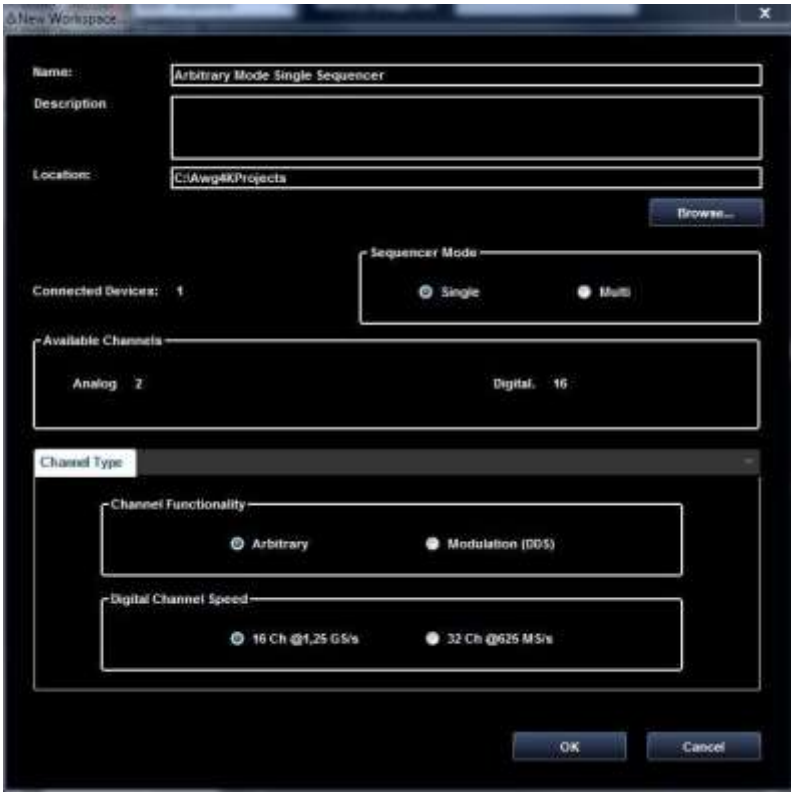


Figure 151 Setting the new waveform

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

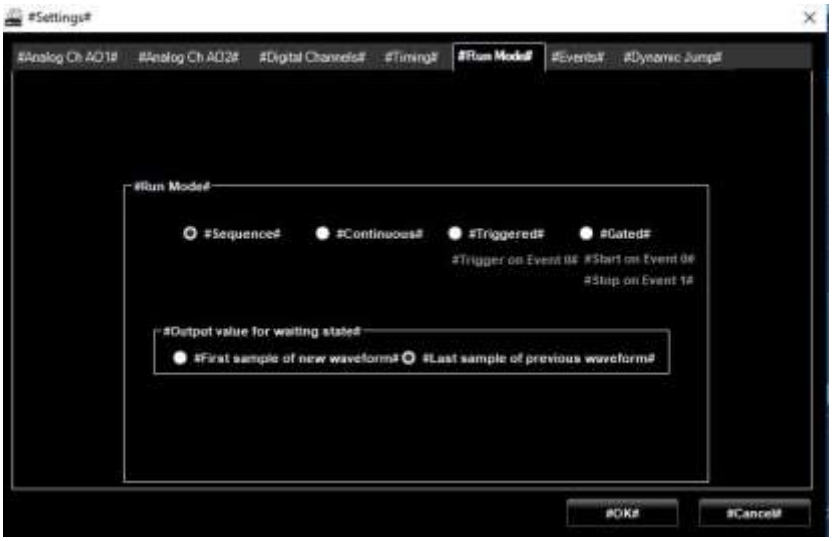
10. The **Waveform Standard Editor** is shown. Choose a sine waveform with the following specs:
    - Cycles: 2
    - Amplitude[V]: 250mV
11. Press **OK** button.
12. Select the waveform Wave1-1 and click on the **Edit**  button.
13. The **Waveform Standard Editor** is shown. Choose a triangle waveform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



Settings window

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

Settings window

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

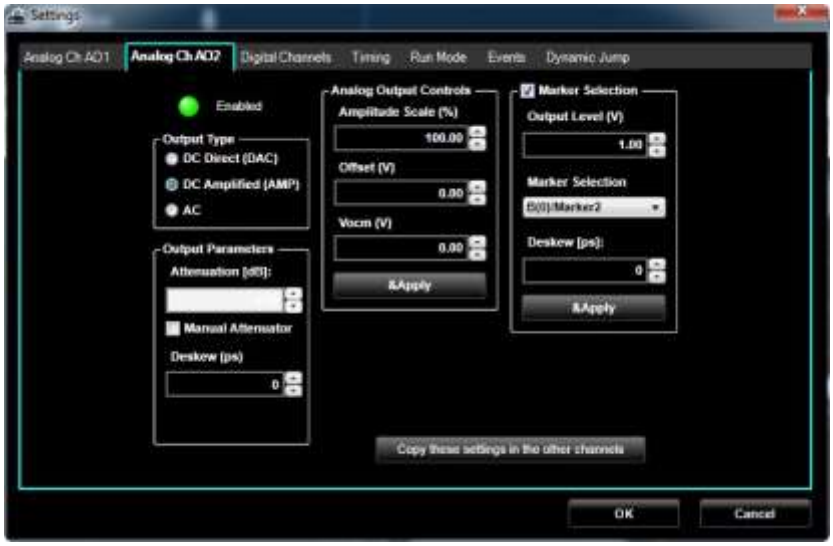


Figure 157 Settings window

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.

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.

24. 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 Model 676 High Performance AWG 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 AWG4022 setup automatically installs under the folder C:\Program Files (x86)\Active Technologies\EXPERT RIDER AWG 4000\DemoProjects 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.

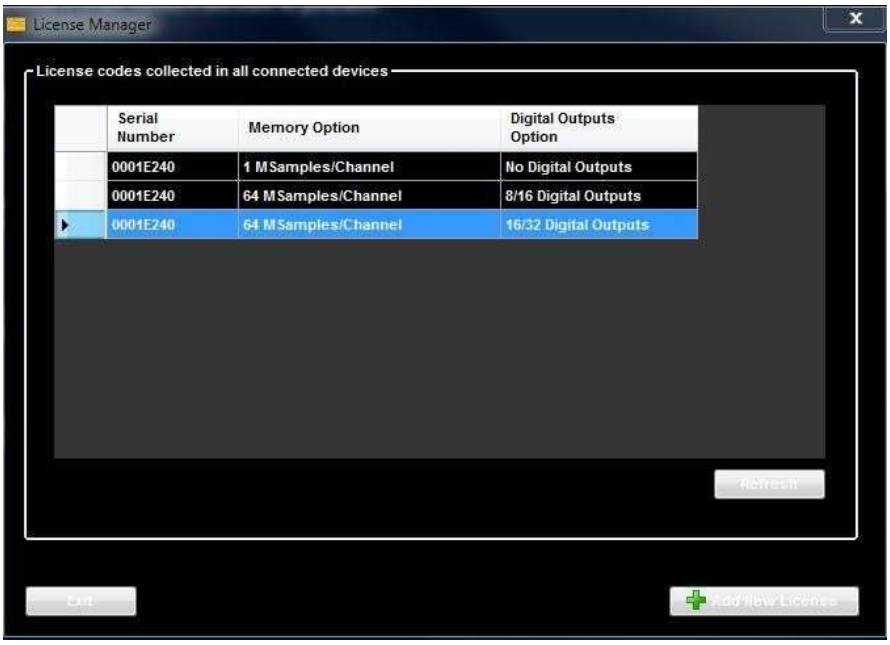
Option Installation

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

MEM64
MEM32
MEM16
MEM01
DO32
DO16
WIDE BAND OPTION (it enables Direct DAC and AC outputs)

Use the License Manager dialog box to enable the upgrades that you purchased from Berkeley Nucleonics for your instrument. For the most current list of upgrades, contact your local Berkeley Nucleonics 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.



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 Berkeley Nucleonics 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 four Model 676 High Performance AWG units to have a system with 8 synchronized analog channels and up to 128 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.

Rear of the instrument

Connecting units

- Turn on the instruments.
- Launch the Expert Rider SWG 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.

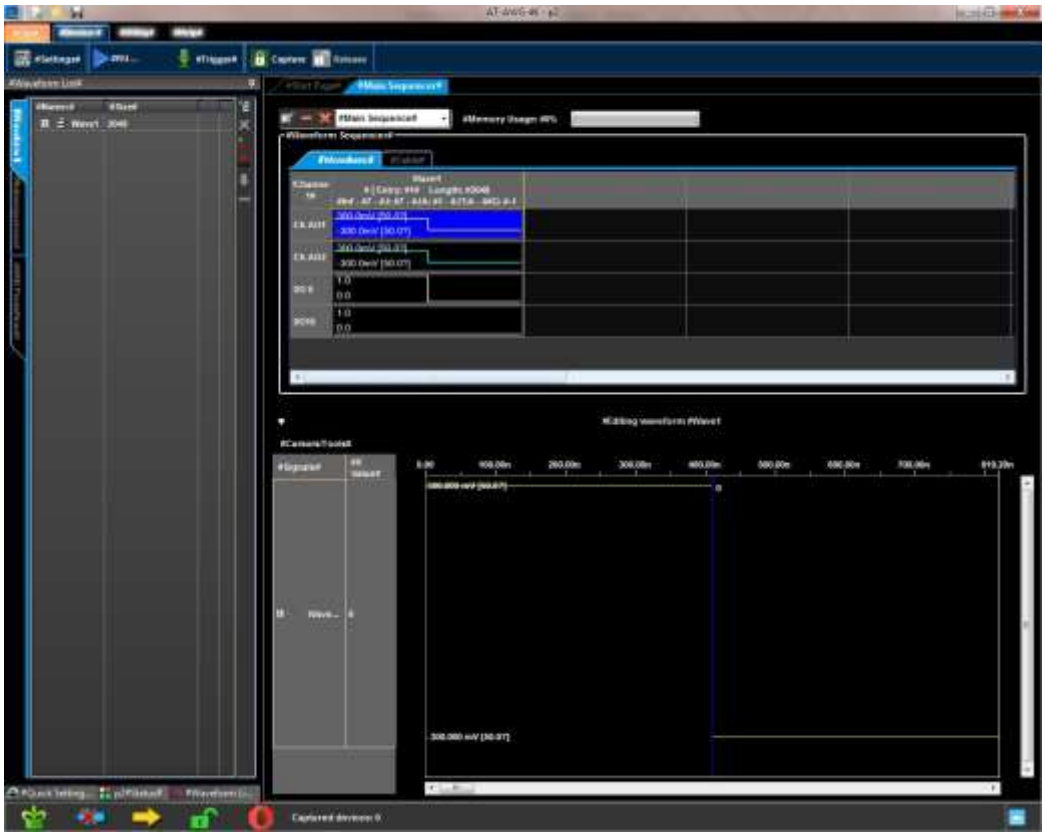


Figure 164 Master device

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

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

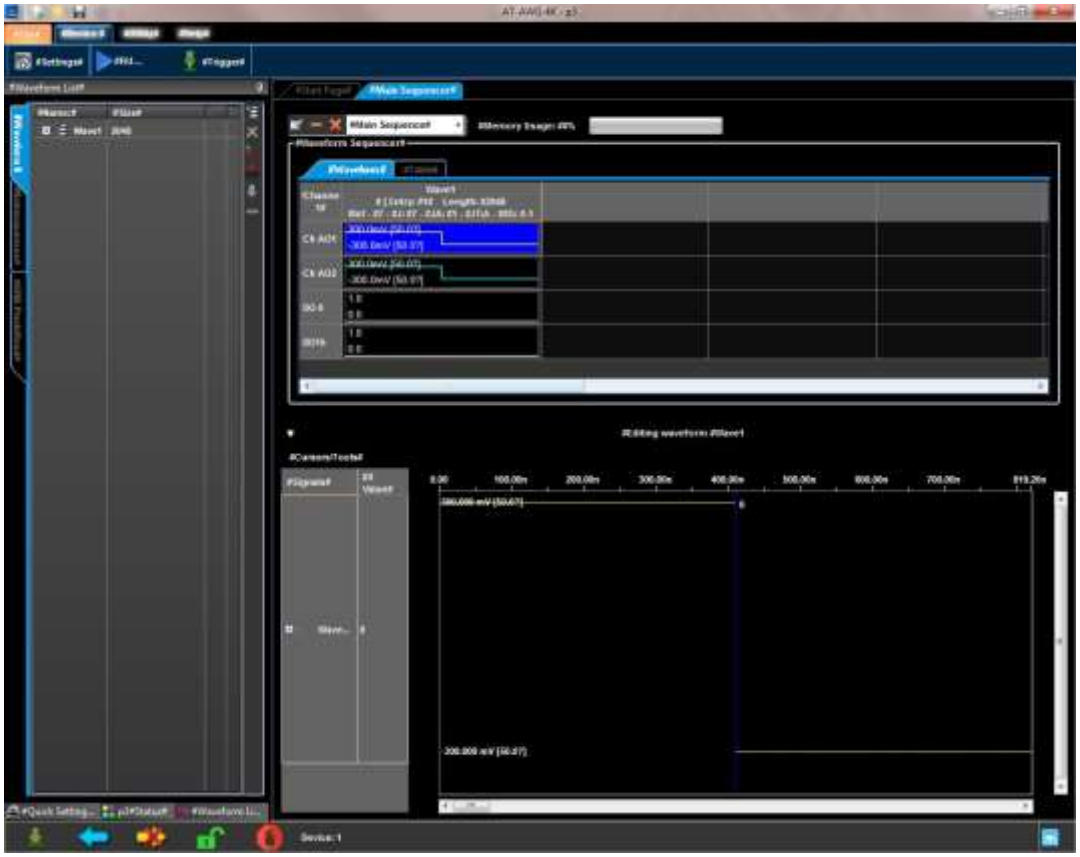


Figure 165 Slave device

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:

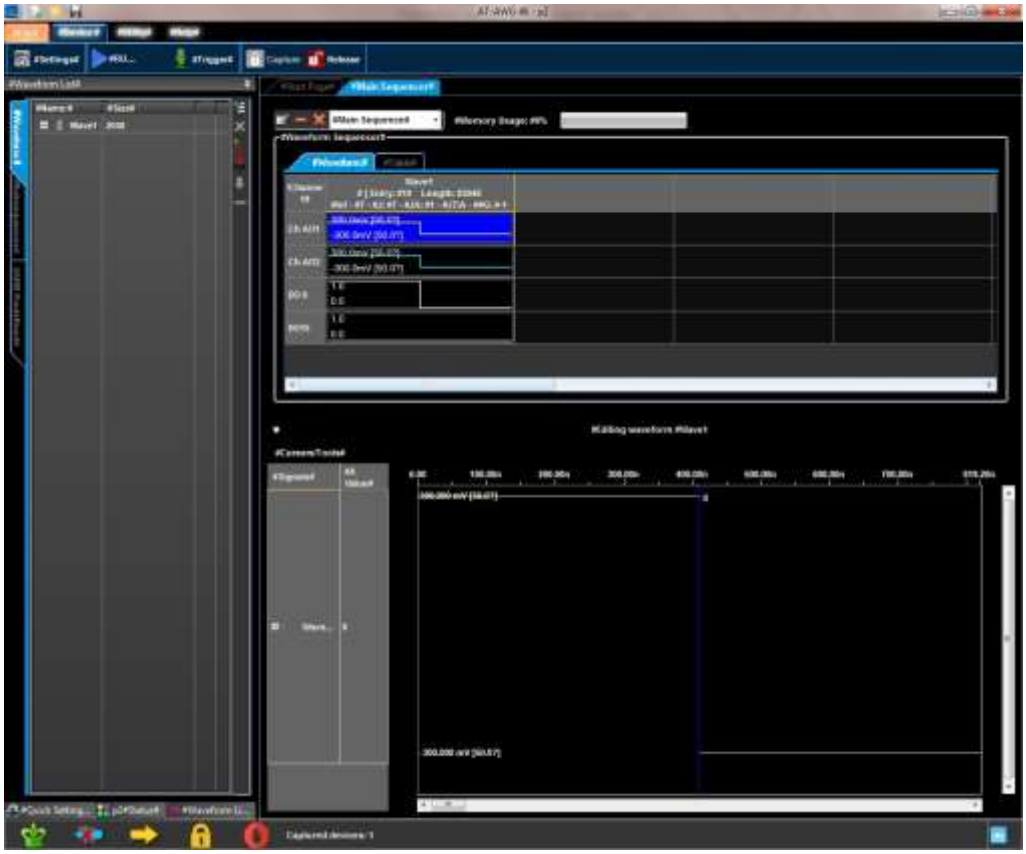
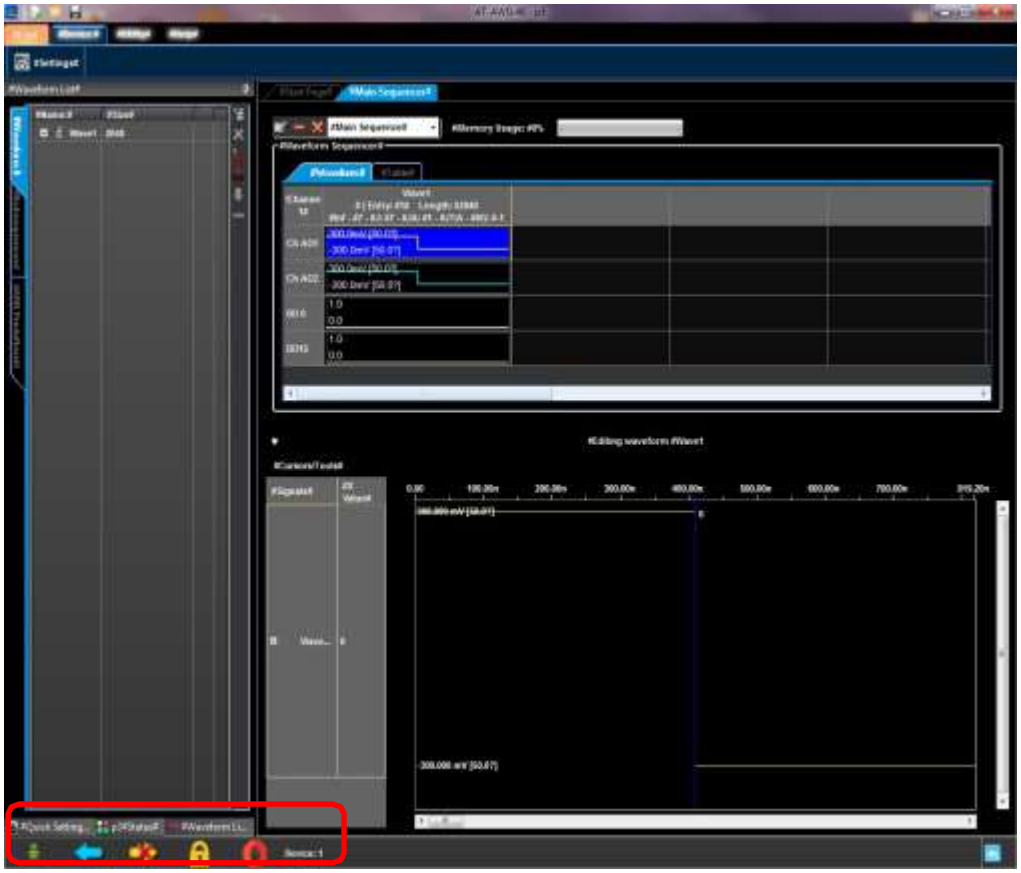


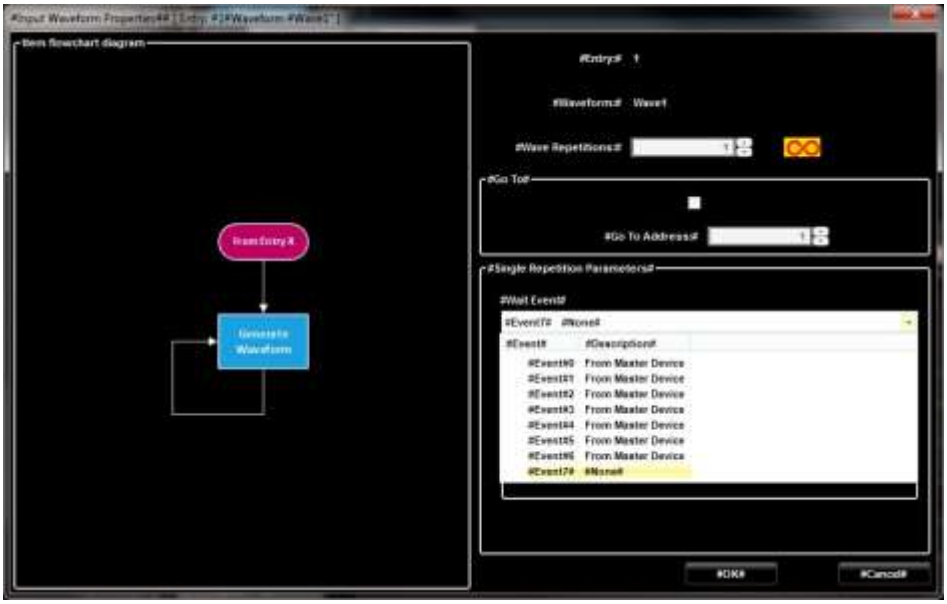
Figure 166 Capturing a slave device



A slave device captured

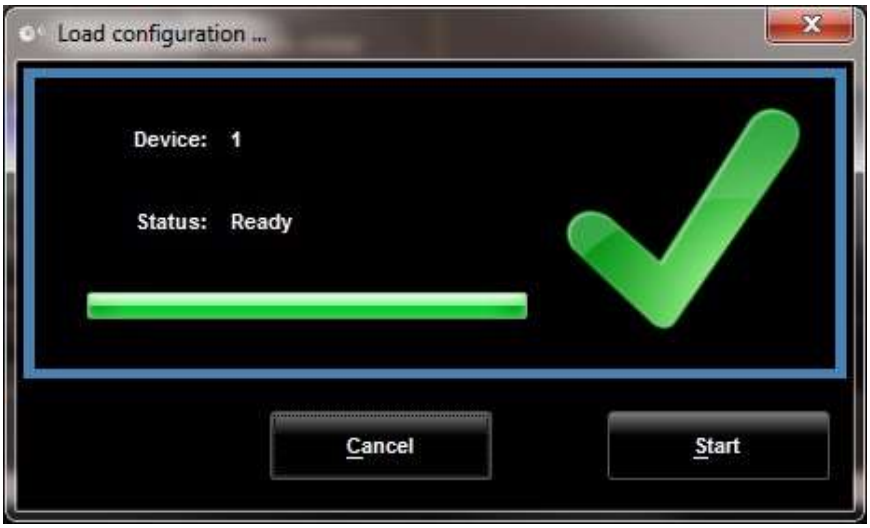
If you check the Salve Multi-Instrument Status bar, you will notice that the  icon indicates that you are on the Slave device, the blue arrow tell you that there is a device connected backward (the Master) and the  icon indicates that the Slave has been captured by the Master. The Slave instrument is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



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.



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.

Creating Waveforms Using Formulas

Overview

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. This section of the manual shows you how to create various waveforms using formulas in Model 676 High Performance AWG.
The next topic in this section covers the standard steps used to create an advanced waveform component using formulas. Remaining topics show example formulas producing various waveform types.

Steps to Creating Advanced Waveform Components Using Formulas

Similar to most other tasks in Model 676, 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.

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

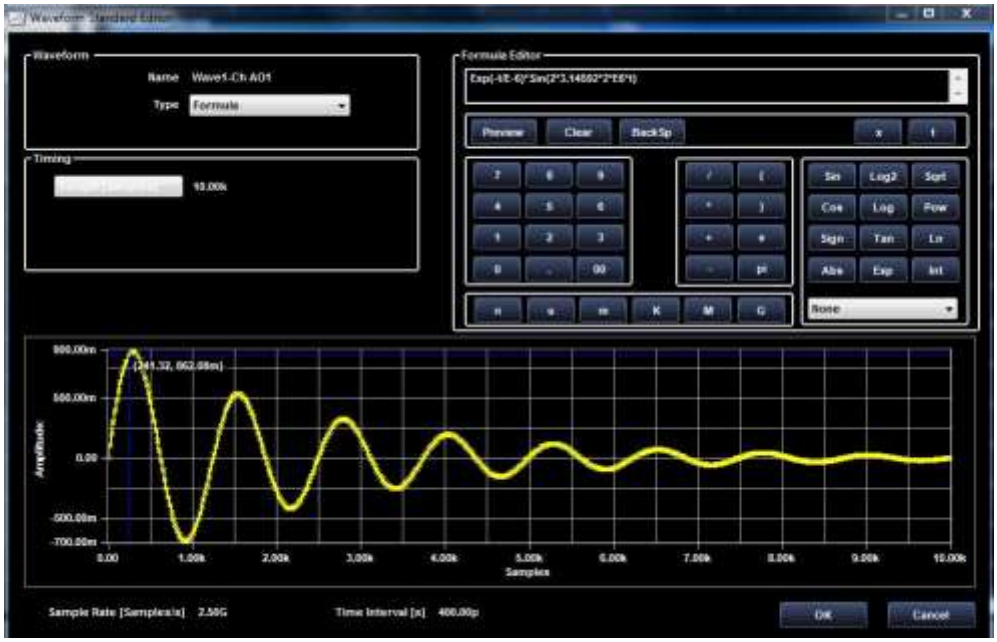
button.

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 NOTE THE FOLLOWING:

- Your formula is shown in the text box as its 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.
From here you can create additional components and/or add your newly-created component to the sequencer and output your waveform from Model 676
The next set of topics show example formulas producing various waveform types.



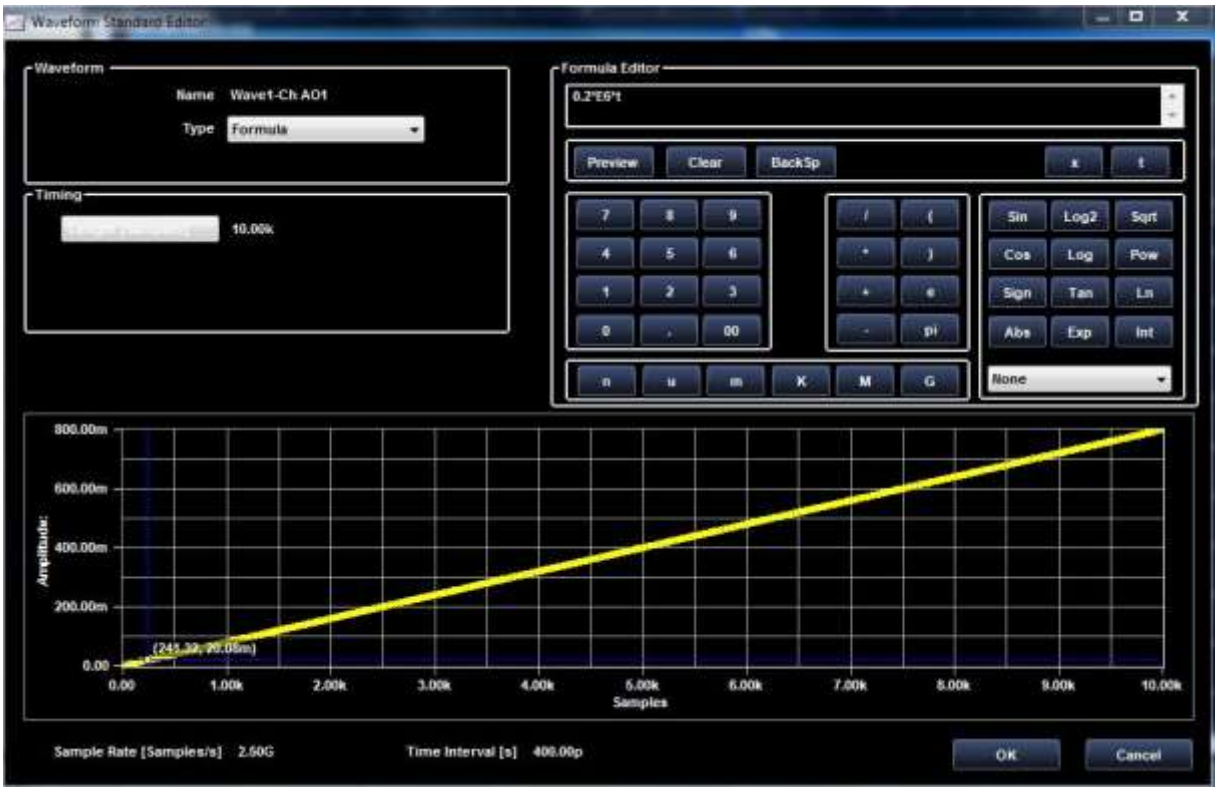
An exponentially decaying 2 MHz sine wave.
The actual formula used here is Exp(-t/E-6)*Sin(2*3.14592*2*E6*t).

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

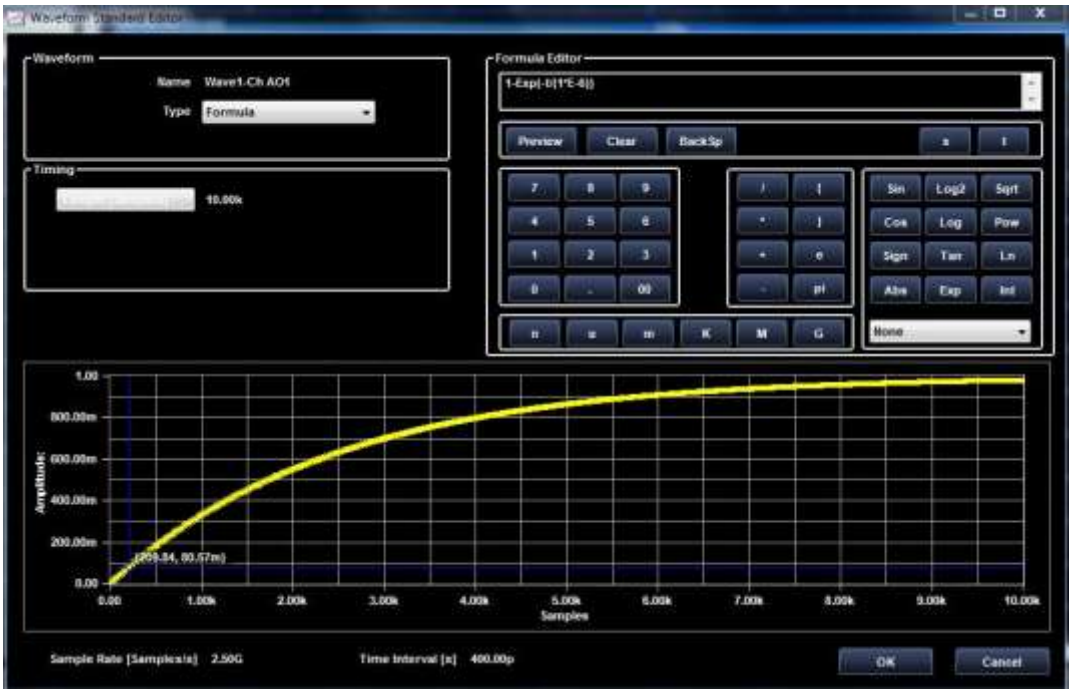


A ramp waveform. The actual formula used here is 0.2*E6*t.

Formula's General Format

$A*t$

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

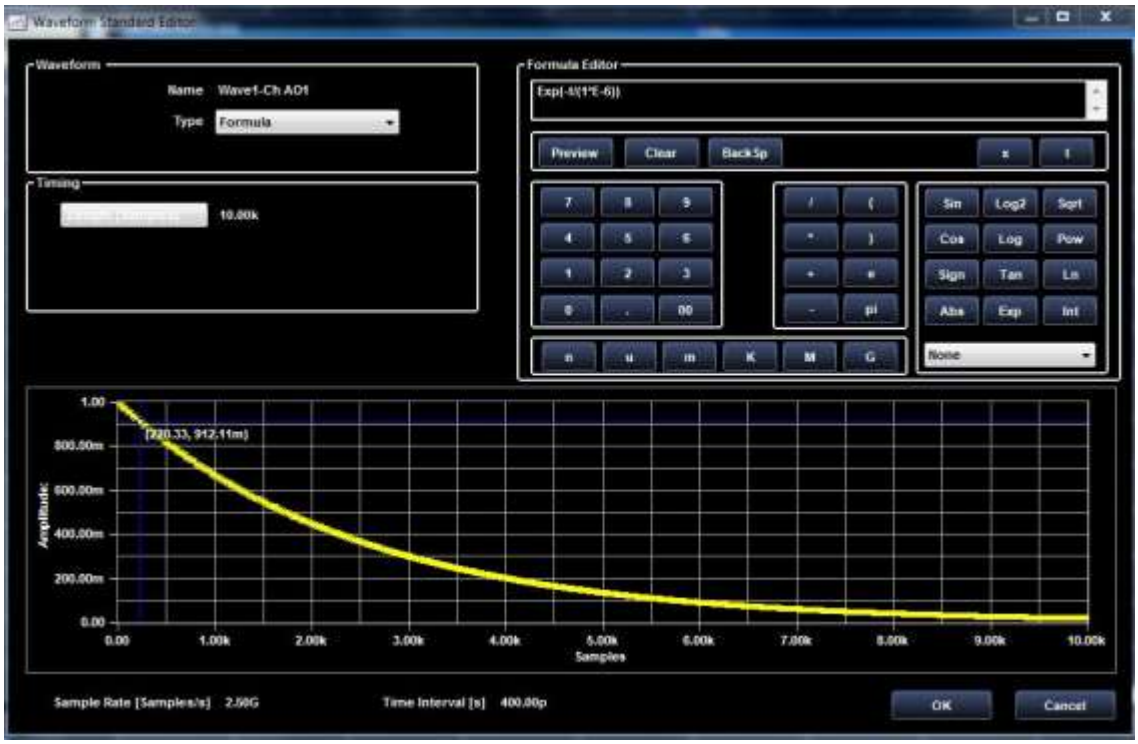


A rising exponential waveform. The actual formula used here is 1-Exp(-t/(1*E-6)).

Formula's General Format

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

Where T_c – Time Constant in seconds.



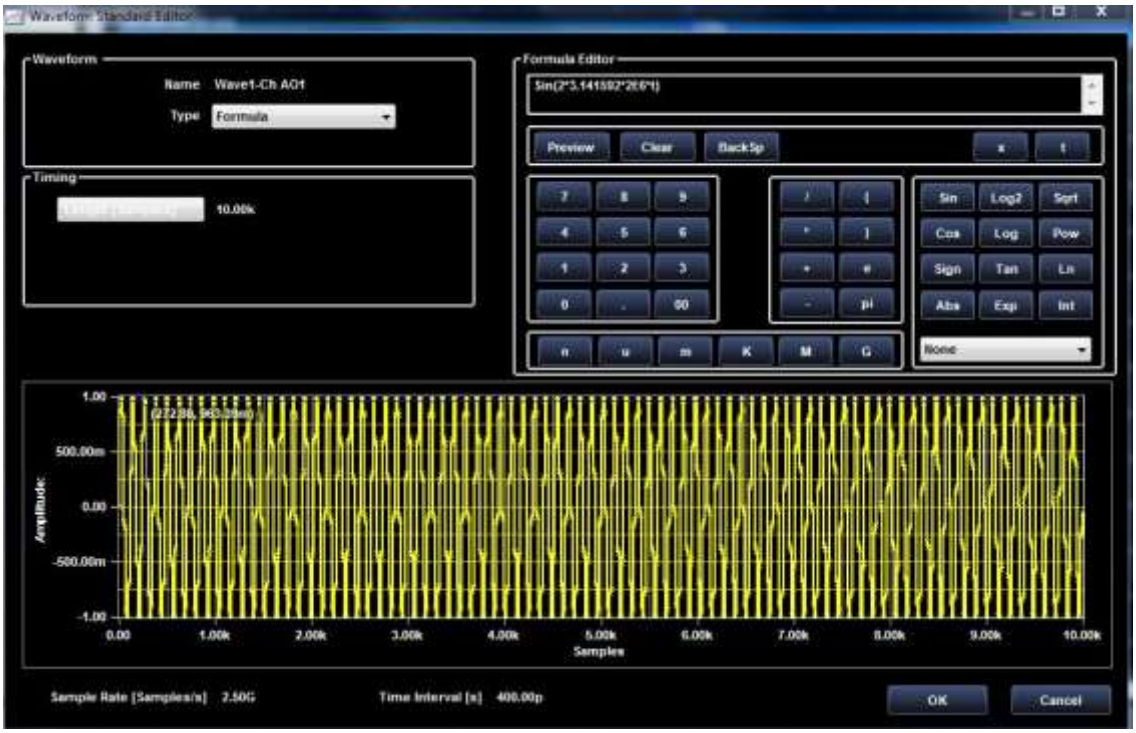
A decaying exponential waveform. The actual formula used here is Exp(-t/(1*E-6)).

Formula's General Format

$Exp(-t/T_c)$

Where T_c – Time Constant in seconds.

A linear amplitude sweep of a 1 MHz sine waveform. The actual formula used here is



Sin(2*3.141592*2E6*t).

Formula's General Format

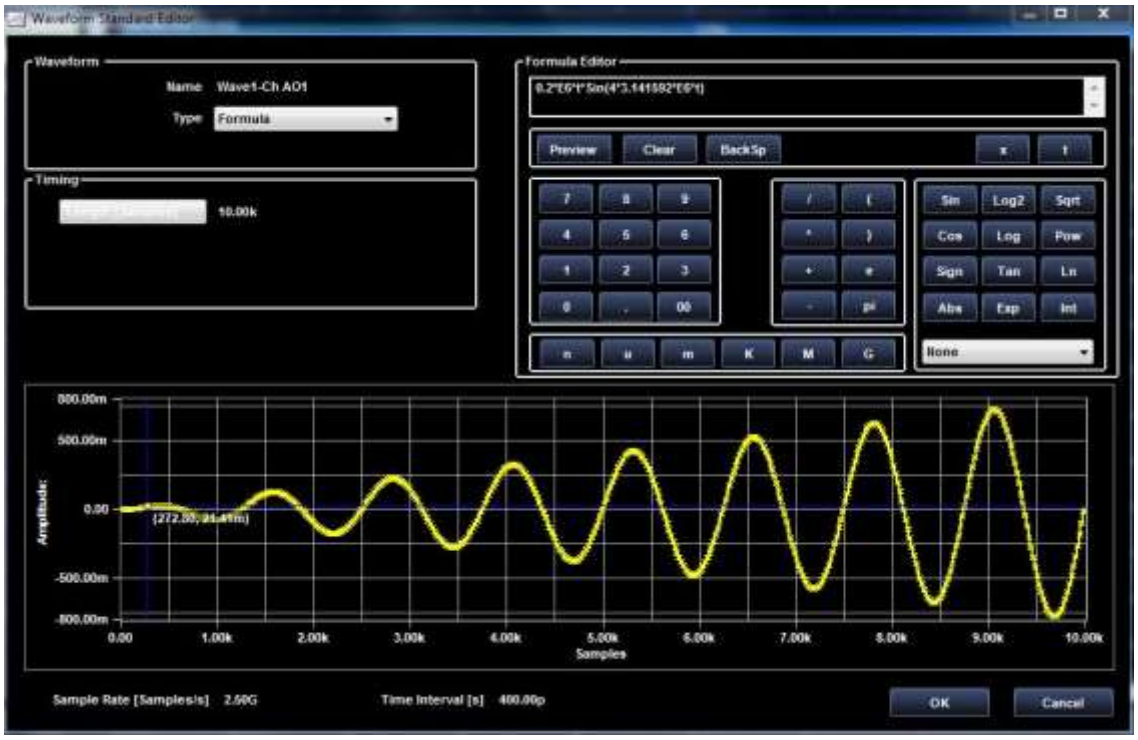
$V*Sin(2*pi*t*F_s)$

Where

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

Linear Amplitude Sweep of a Sine Wave

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

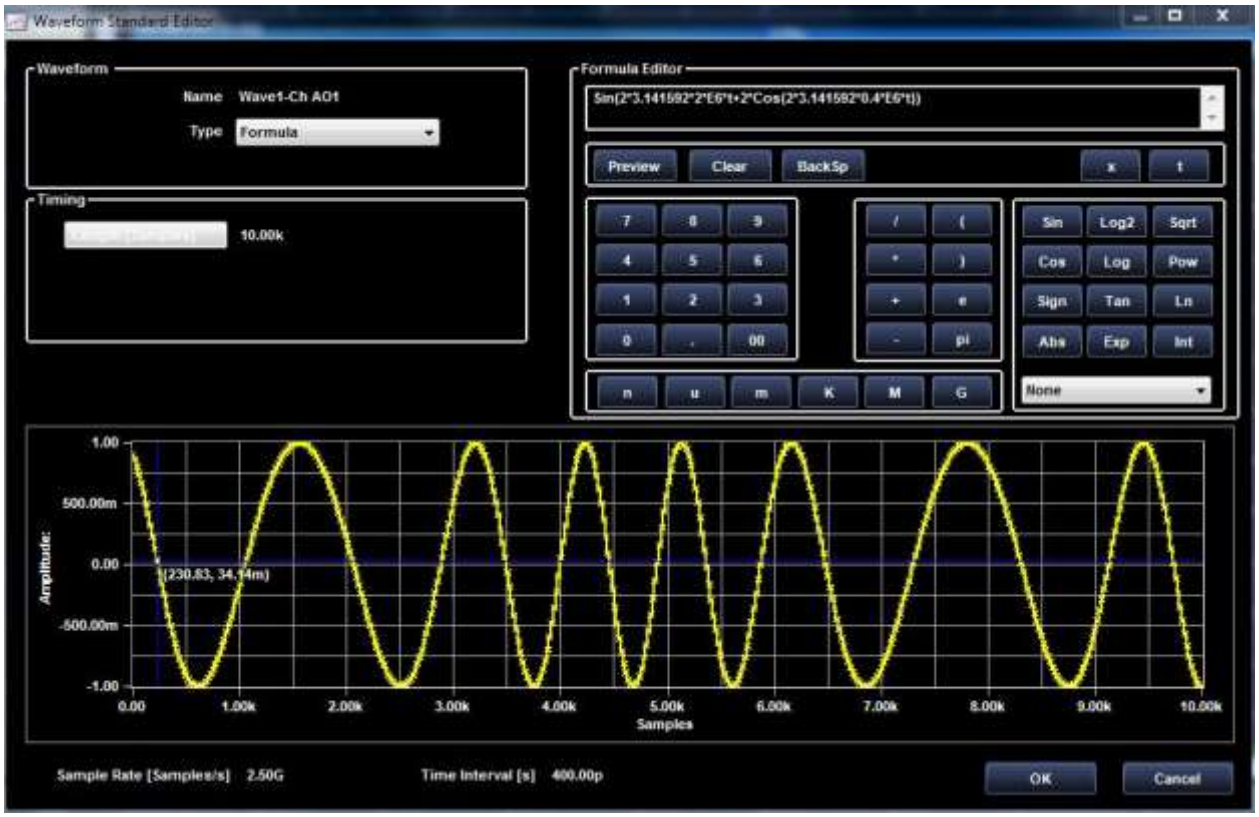


Formula's General Format

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

Where

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



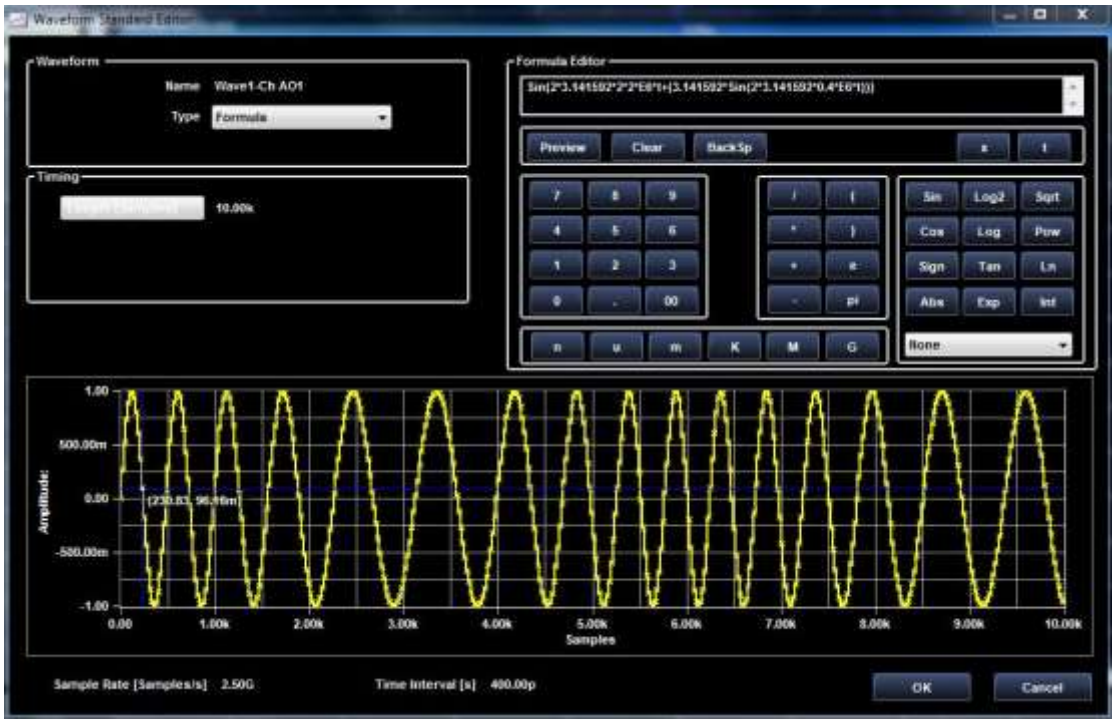
A frequency modulation waveform. The actual formula used here is Sin(2*3.141592*2*E6*t+2*Cos(2*3.141592*0.4*E6*t)).

Formula's General Format

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

Where

- F_c – Carrier frequency in Hertz.
- F_D – Frequency deviation in Hertz.
- F_M – Modulation frequency in Hertz.



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

Formula's General Format

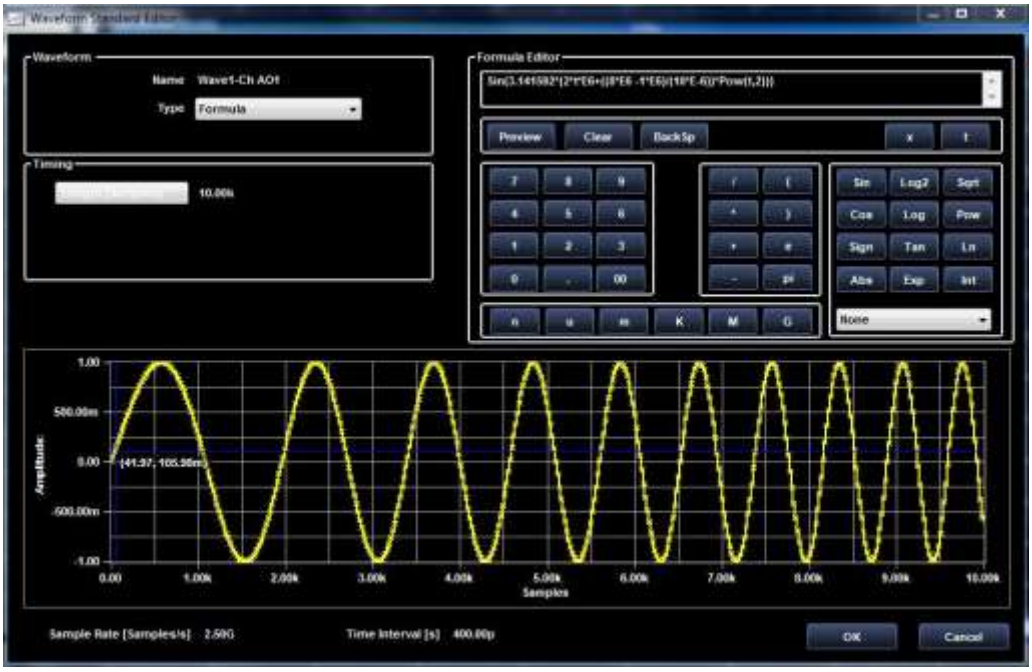
$$\mathrm{Sin}((2*\mathrm{pi}*t*F_C + K*\mathrm{Sin}(2*\mathrm{pi}*t*F_M))$$

Where

F_C – Carrier frequency in Hertz.

K – Peak phase excursion in radians.

F_M – Modulation frequency in Hertz.



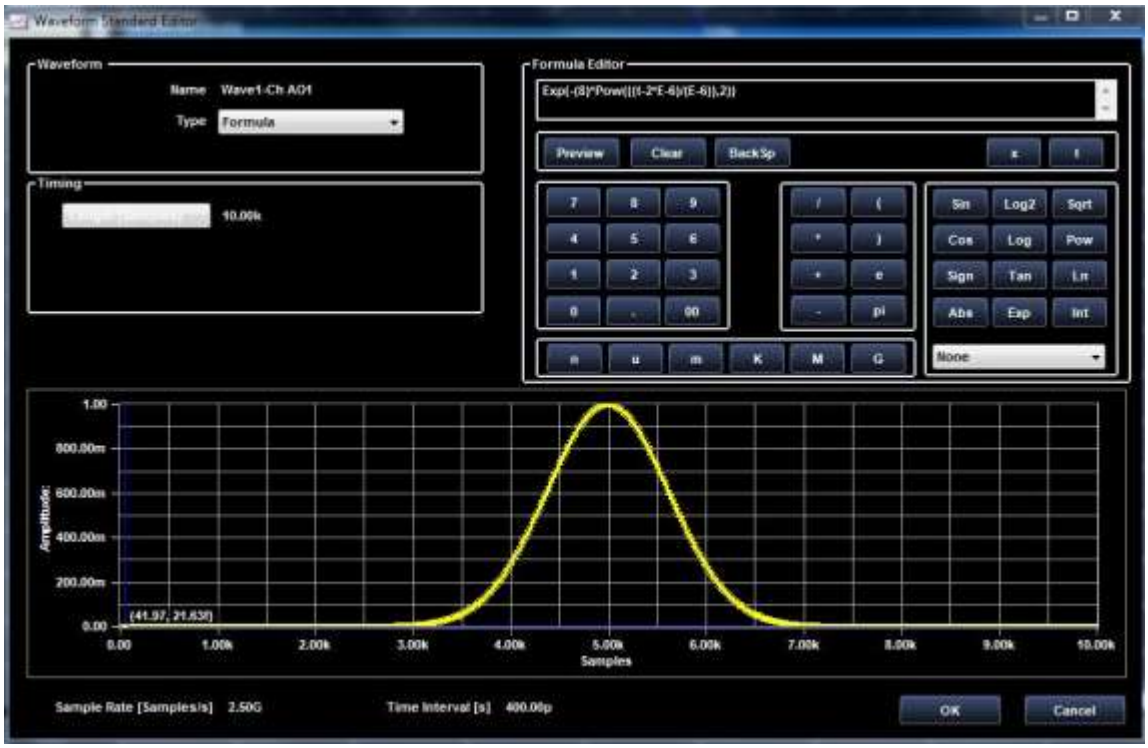
A linear frequency sweep waveform.
The actual formula used here is Sin(3.141592*(2*t*E6+((8*E6 -1*E6)/(10*E-6))*Pow(t,2))).

Formula's General Format

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

Where

- F_S – Start frequency in Hertz.
- F_E – End frequency in Hertz.
- T_S – Sweep duration in seconds.



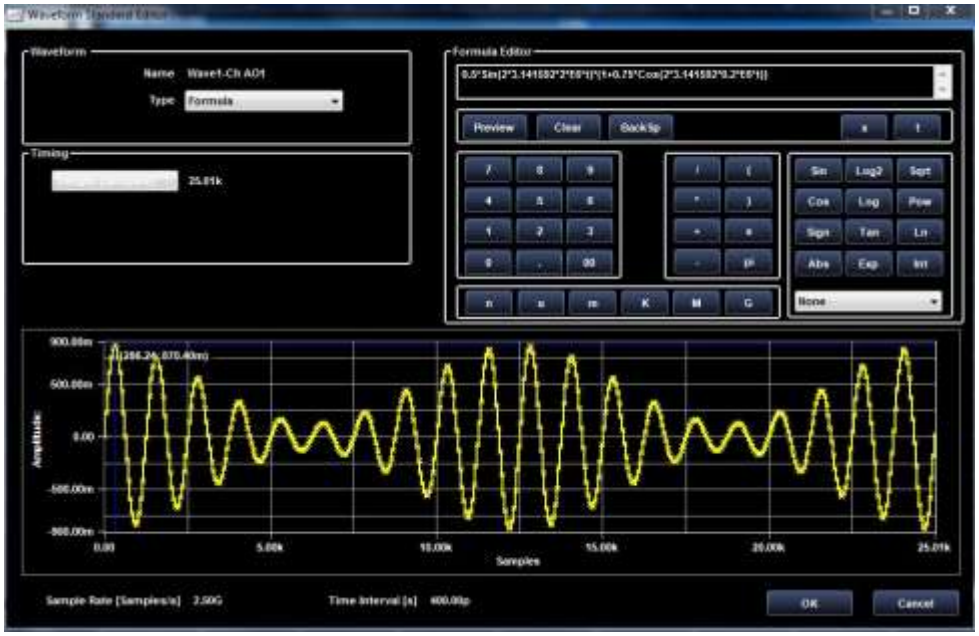
A Gaussian pulse waveform. The actual formula used here is Exp(-(8)*Pow(((t-2*E-6)/(E-6)),2).

Formula's General Format

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

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**.



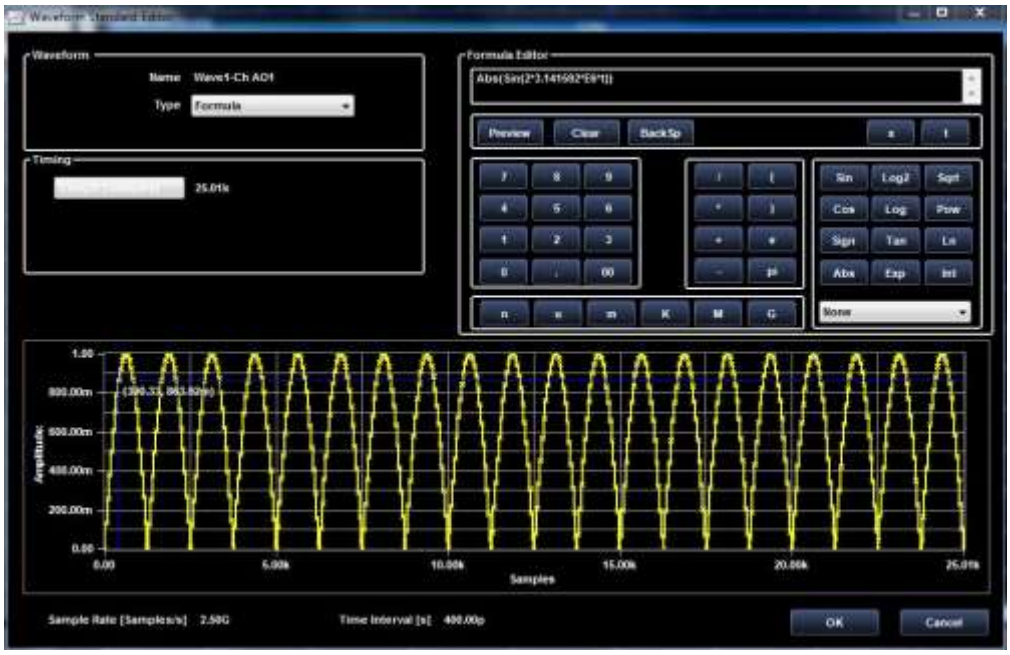
An amplitude modulated sine waveform. The actual formula used here is 0.5*Sin(2*3.141592*2*E6*t)*(1+0.75*Cos(2*3.141592*0.2*E6*t)).

Formula's General Format

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

Where

- F_s – Sine wave frequency in Hertz.
- F_M – Modulation frequency in Hertz.
- K – Modulation index, 0<K<1.



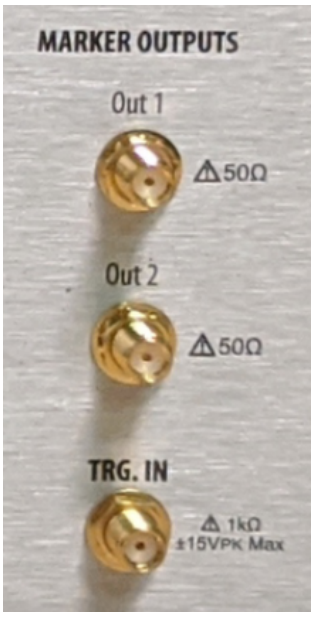
A full-wave rectified sine waveform. The actual formula used here is Abs(Sin(2*3.141592*E6*t)).

Formula's General Format

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

Where F_s – Sine wave frequency in Hertz.

Auxiliary Channels



Marker Out

The Marker Out generates an analog pulse that can be aligned with the analog waveform using the Waveform Editor. To set the Marker Out parameters refer to the Settings Page.

Marker Out Specification	Value
Connector	SMA on the Front Panel
Number of connectors	two, one for each analog output
Output impedance	50 Ω
Output level (into 50 Ω)	1 V to 2.5 V
Resolution	10 mV
Accuracy (typical)	±(2% setting + 10 mV)
Variable delay control	0 to 60606 ps
Resolution	78 ps
Accuracy (typical)	±(10% of setting + 140 ps)
Rise/fall time (10% to 90%, 2.5 V, typical)	800 ps
Jitter RMS	155 ps

Trigger In

The Trigger In (TRG. IN connector on the front panel) can be used to control the entry generation. To know how to set the trigger parameter or the Run Mode refer to the inherent manual chapters.

Trigger In Specification	Value
Connector	SMA on the Front Panel
Number of connectors	one for all analog output
Input impedance	1.1 kΩ
Slope/Polarity	Positive or negative selectable
Input damage level	< -15 V or > +15 V
Threshold control level	-10 V to 10 V
Resolution	50 mv
Threshold control accuracy (typical)	±(10% of |setting| + 0.2 V)

Input voltage swing	0.5 V_{p-p} minimum
Minimum pulse width	12 ns
Initial trigger / gate delay to Analog Output	332.8 ns ±400 ps

Reference Clock Input

When the "Clock Source" is set on "External" from the "Device Settings" page, an internal clock generator takes the signal on "Reference Clock Input" SMA connector as reference to generate the DAC clock signal at 2.5 GHz.

Reference Clock Input	Value
Connector type	1 SMA on the Rear Panel
Number of connectors	one for all analog output
Input impedance	50 Ohm, AC coupled
Input voltage range	-5 dBm to 4 dBm sine or square wave
Damage level	+8 dBm or ±15 V_{DC} Max
Input Frequency range	10 MHz to 80 MHz

External Modulation Input

Not available in Model 676 High Performance AWG.

Reference Clock Output

This connector output the clock signal at 10 MHz or the Reference Clock Input if the Clock source is External.

Reference Clock Input	Value
Connector	SMA on the Rear Panel
Number of connectors	one for all analog output
Output impedance	50 Ohm, AC coupled
Frequency	10 MHz or External Reference Clock input
Accuracy	± 1.0 x 10^{-6}
Aging	± 1.0 x 10^{-6}/year
Jitter (rms, typical)	11.5 ps

Remote Control

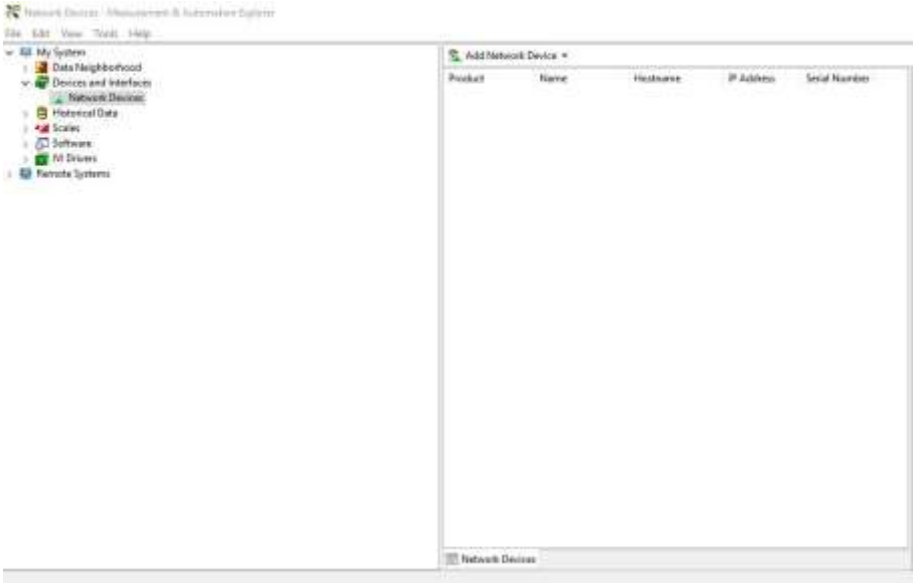
You can connect your instrument to a network for printing, file sharing, and Internet access, among other functions. Consult with your network administrator and use the standard Windows utilities to configure the instrument for your network.
The instrument can be controlled using VXI-11 (LAN) protocol. It allows you to control the instrument remotely by using SCPI commands. Please refer to the programmer manual for a complete description about all available commands.
You can follow the next steps to communicate with your Model 676 High Performance AWG instrument.

- **IMPORTANT NOTE:** the SCPI commands are available for Expert Rider AWG mode in Single Sequencer.

NI-VISA

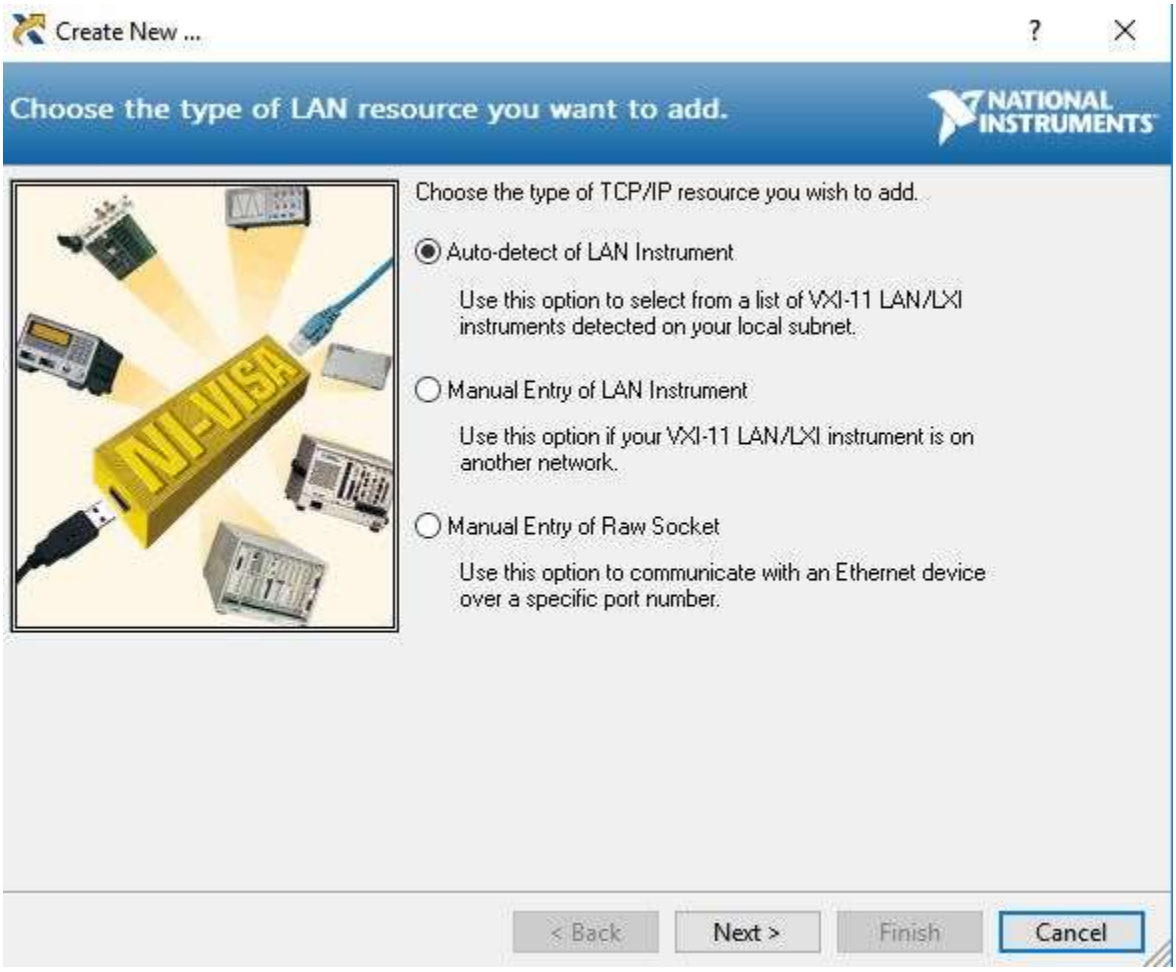
VISA provides the programming interface between the hardware and development environments such as Visual Studio .NET, LabVIEW, LabWindows/CVI, Measurement Studio for Microsoft Visual Studio and MatLab. NI-VISA is the National Instruments implementation of the VISA I/O standard. NI-VISA includes software libraries, interactive utilities such as NI I/O Trace and the VISA Interactive Control, and configuration programs through Measurement & Automation Explorer for all your development needs.

1. Connect your LAN cable to the instrument.
2. On the Client-PC you must install the latest NIVISA package that you can find here http://search.ni.com/nisearch/app/main/p/bot/no/ap/tech/lang/it/pg/1/sn/catnav:du,n8:3.1637,ssnav:sup/
3. Launch the Expert Rider AWG software and press the button Remote/Local on the command bar. Check if the SCPI VXI/LAN is turned on.
4. Launch the NI-MAX tool on the Client-PC

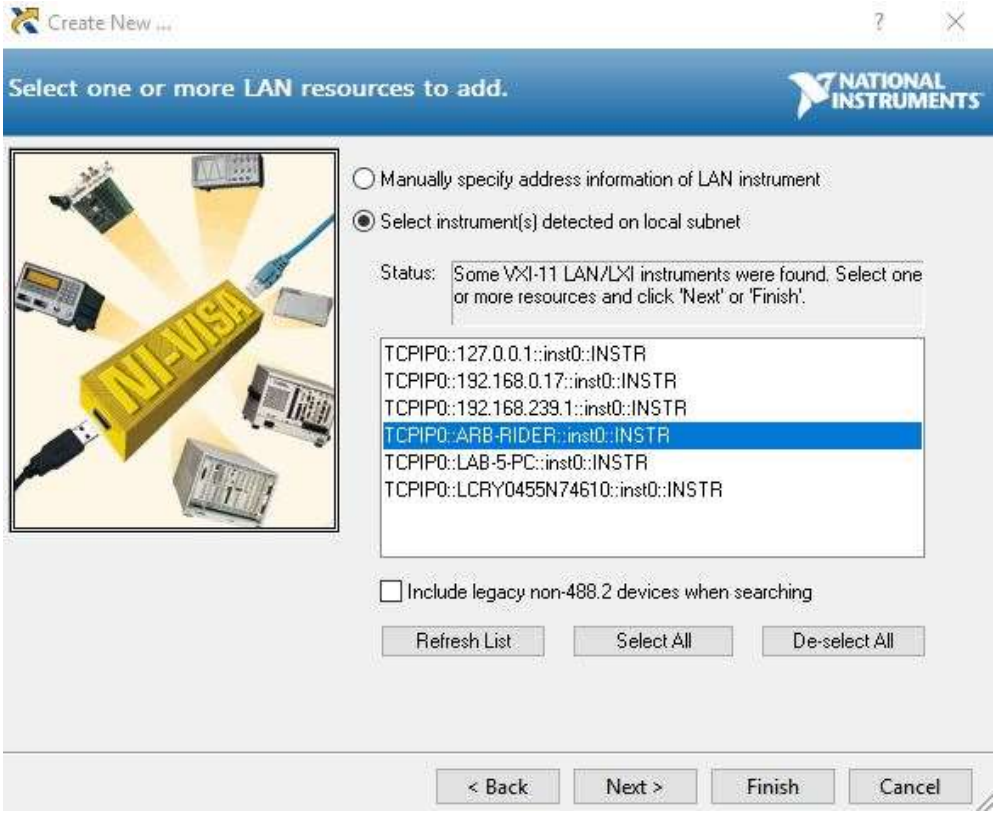


Press Add Network Device → VISA TCP/IP Resource…

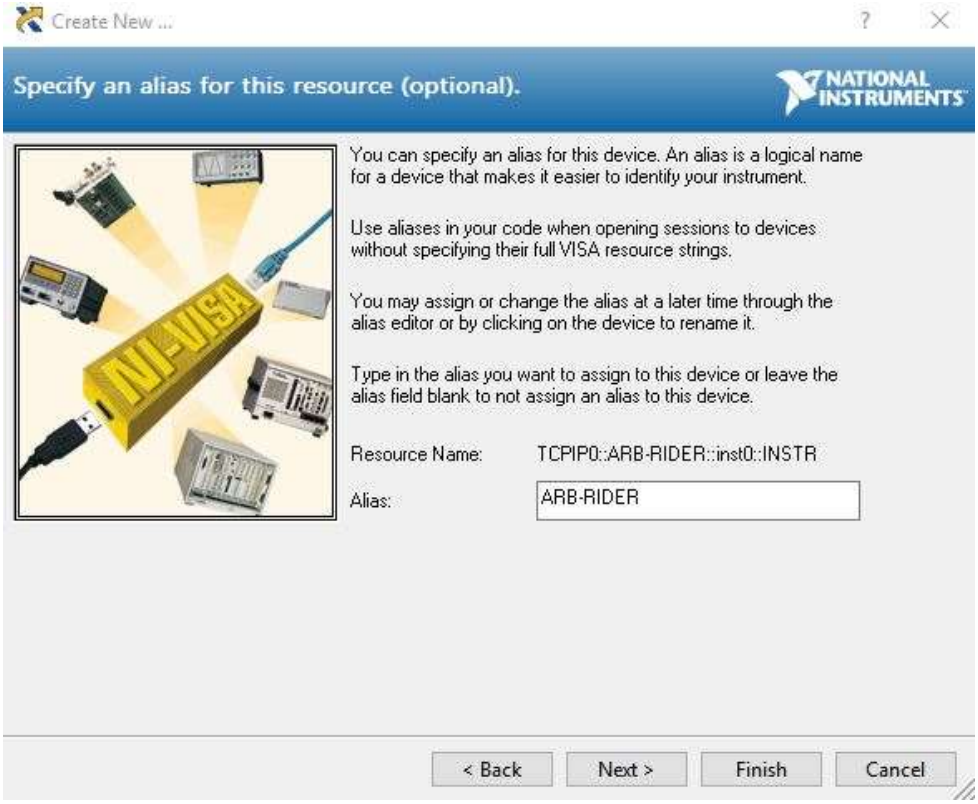
5. Select Auto-detect of LAN Instrument



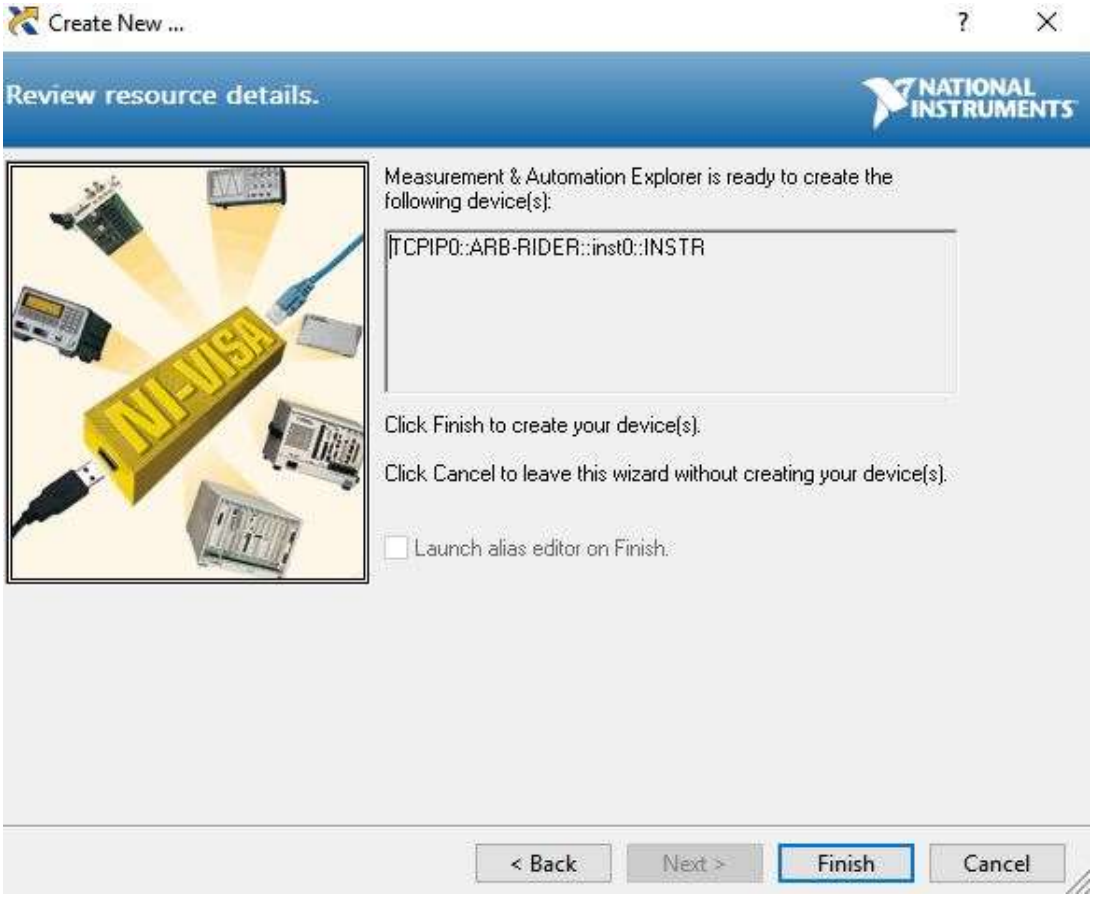
6. The panel will retrieve the discovered instruments on the LAN network, you should select the ARB-RIDER series one.



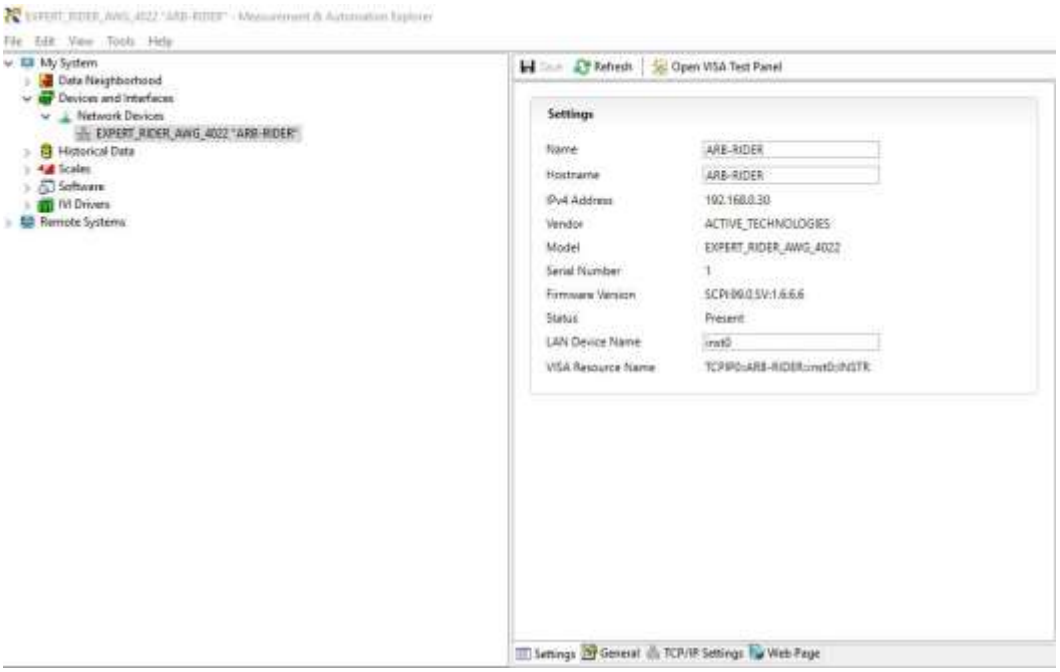
7. Specify an Alias for the selected resource



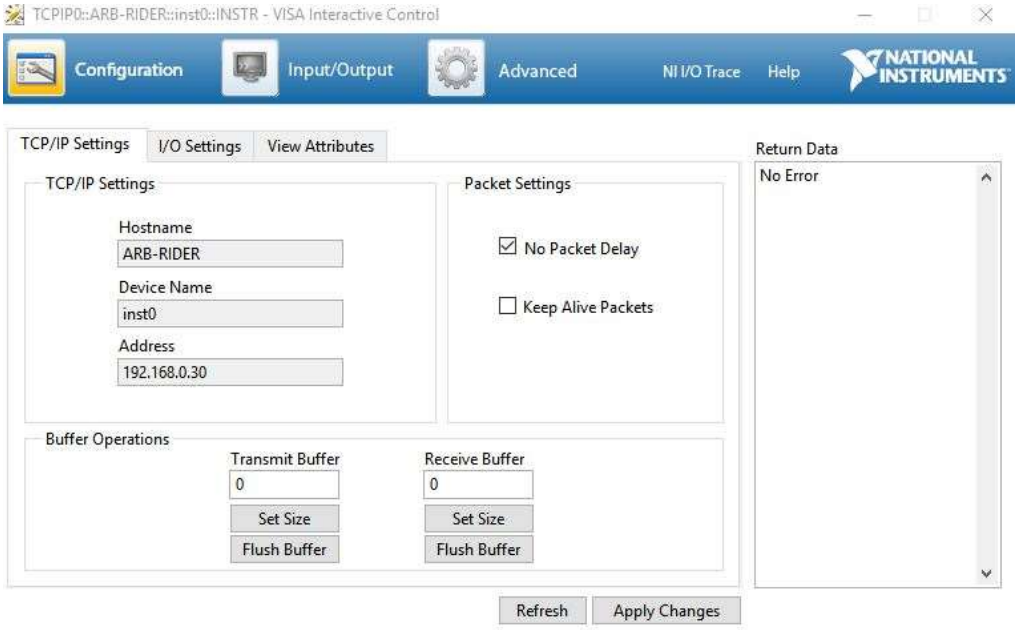
8. Press Finish



9. The AR resource will be available in the Network Devices list

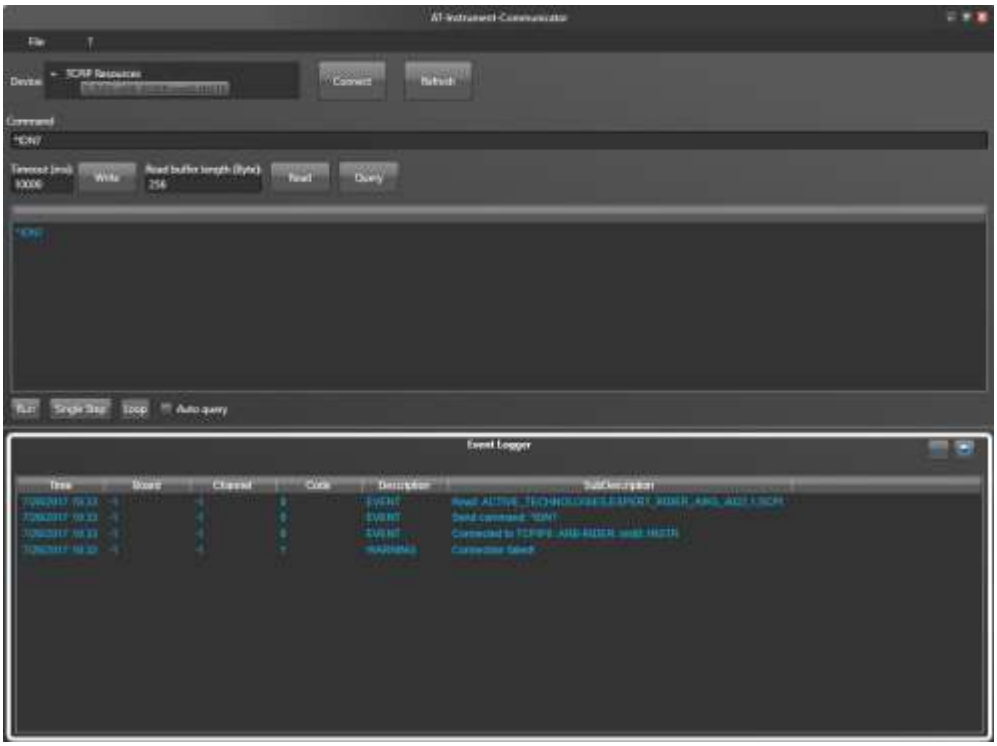


10. Now you can use send the AWG Mode SCPI commands to the Model 676 High Performance AWG resource using the NI Visa Test Panel or the AT-Instrument-Communicator



11. On the Client-PC (IP Address) or Model 676 High Performance AWG instrument (Local Host), launch the AT-Instrument-Communicator tool

AT Instrument Communicator



The instrument VXI-11 LAN Server provides software connectivity between your instrument and remote PCs over an Ethernet LAN.
The AT-Instrument-Communicator software is a client-side component tool that uses NI-VISA on each remote PC, you must install a copy of NIVISA to make use of this client-side component (please follow the Prerequisite steps).

1. On the Client-PC launch the *AT Instrument Communicator* setup and install the software.
2. Select the ARB-RIDER resource on the Device list
3. Press the Connect button
4. If the instrument connection will be established, the SCPI command button will be enabled.
5. Write *IDN? in the command
6. Press the Query button
7. In the Event Logger list, the instrument should respond like this:
   ACTIVE TECHNOLOGIES, ACTIVE TECHNOLOGIES, EXPERT_RIDER_AWG4022, 253A0001,SCPI:99.0,SV:1.6.6.4
8. A command script is a list of SCPI commands (one command for each line) saved in a txt file; you can send a command script using the File → Load Script menu item.
9. Refer to the SCPI programmer manual for a complete list of all available commands.

Remote Desktop

The Expert Rider software interface can be accessed by an external PC using the Microsoft Windows 10 Remote Desktop feature.
Please follow the procedure here to configure the Remote Desktop https://support.microsoft.com/en-us/help/4028379/windows-how-to-use-remote-desktop

The Model 676 PC User-Name is: Arb Rider
The password is: 1234

Appendix

A. Digital Outputs

The Model 676 High Performance AWG can output 16-bit or 32-bit 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 DO32, Pod A and Pod C are available. 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 DO32, Pod A, Pod B, Pod C and Pod D are available. The bit rate is a quarter of the sampling rate (for example, 625 Mb/s at 2.5 GS/s sampling rate).

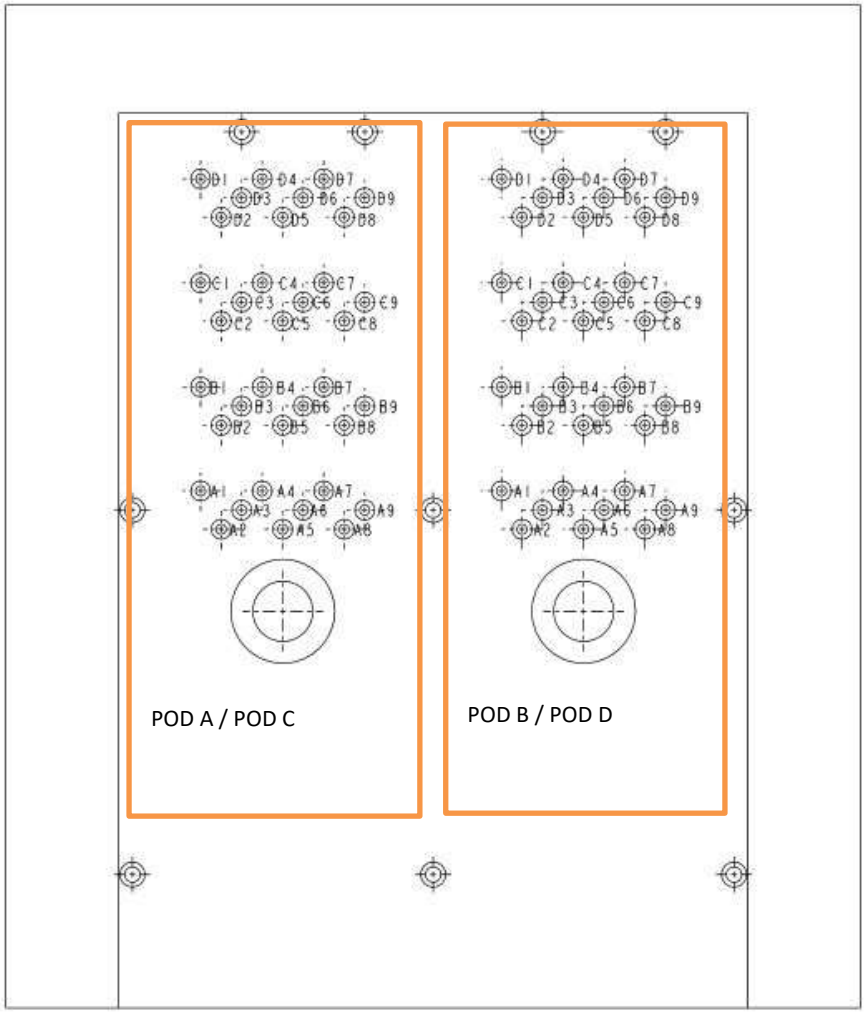
In high speed mode DO16, Pod A is available. The bit rate is half of the sampling rate (for example, 1.25 Gb/s at 2.5 GS/s sampling rate).

In low speed mode DO16, Pod A, Pod B are available. The bit rate is a quarter of the sampling rate (for example, 625 Mb/s at 2.5 GS/s sampling rate).

The Model 676 High Performance AWG has two Mini SAS HD connectors

with the following pinout

PIN OUT DESIGNATIONS



POD A / PODC	Connection	AT-LVDS-SMA8 Mini SAS HD to 16 SMA cable (8 LVDS output)
A1	+12Vcc	NA
A2	+12Vcc	NA
A3	GND	SMA Ground
A4	DO7_P	RX1+
A5	DO7_N	RX1-
A6	GND	NA
A7	DO0_P	RX3+
A8	DO0_N	RX3-
A9	GND	SMA Ground
B1	CS1 (RESERVED)	NA
B2	+12Vcc	NA
B3	GND	SMA Ground

B4	DO6_P	RX0+
B5	DO6_N	RX0-
B6	GND	SMA Ground
B7	DO1_P	RX2+
B8	DO1_N	RX2-
B9	GND	SMA Ground
C1	+5Vcc	NA
C2	+5Vcc	NA
C3	GND	SMA Ground
C4	D5_P	TX1+
C5	D5_N	TX1-
C6	GND	SMA Ground
C7	D2_P	TX3+
C8	D2_N	TX3-
C9	GND	SMA Ground
D1	SCL	NA
D2	SDA	NA
D3	GND	SMA Ground
D4	D4_P	TX0+
D5	D4_N	TX0-
D6	GND	SMA Ground
D7	D3_P	TX2+
D8	D3_N	TX2-
D9	GND	SMA Ground

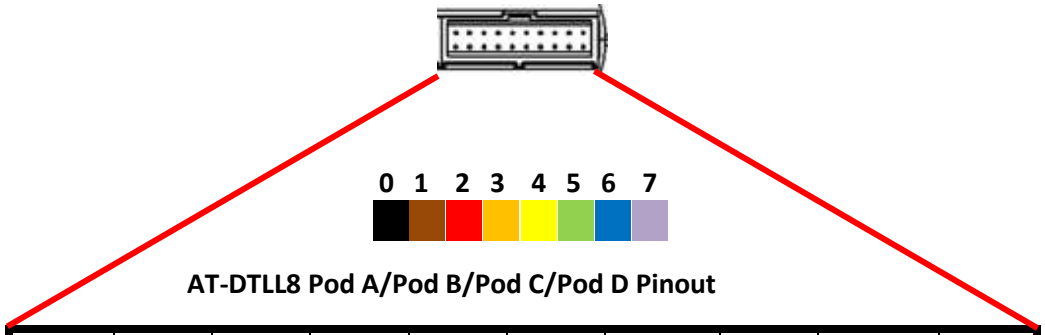
POD B / POD D	**Connection**	
A1	+12Vcc	NA
A2	+12Vcc	NA
A3	GND	SMA Ground
A4	DO15_P	RX1+
A5	DO15_N	RX1-
A6	GND	SMA Ground
A7	DO8_P	RX3+
A8	DO8_N	RX3-
A9	GND	SMA Ground
B1	CS2(RESERVED)	NA
B2	+12Vcc	NA
B3	GND	SMA Ground
B4	DO14_P	RX0+
B5	DO14_N	RX0-
B6	GND	SMA Ground
B7	DO9_P	RX2+
B8	DO9_N	RX2-
B9	GND	SMA Ground
C1	+5Vcc	NA
C2	+5Vcc	NA
C3	GND	SMA Ground
C4	D13_P	TX1+
C5	D13_N	TX1-
C6	GND	SMA Ground
C7	D10_P	TX3+
C8	D10_N	TX3-
C9	GND	SMA Ground

D1	SCL	NA
D2	SDA	NA
D3	GND	SMA Ground
D4	D12_P	TX0+
D5	D12_N	TX0-
D6	GND	SMA Ground
D7	D11_P	TX2+
D8	D11_N	TX2-
D9	GND	SMA Ground

To ensure the best signal integrity when transmitting such high speed digital signals, a customized digital cable and the corresponding connector Mini SAS HD to 16 SMA connector (Berkeley Nucleonics item code: AT-LVDS-SMA8**)** can be bought from Berkeley Nucleonics.

B. 8 bit LVDS to LVTTL converter for Rider series

Berkeley Nucleonics developed an 8 bit LVDS to LVTTL converter (Berkeley Nucleonics item code: AT-DTLL8) that can be used to convert LVDS differential signals to LVTTL single ended signals with a software programmable threshold from 0.8V to 3.8V.
The AT-DTLL8 probe bit rate is 125 Mbps@0.8V and 400 Mbps@3.6V.
Important Note: when the customer will buy the digital option, a customized digital cable will be provided in the package.
The 8 bit LVDS to LVTTL converter can be bought as an additional option or the customer can build its own converter.



AT-DTLL8 Pod A/Pod B/Pod C/Pod D Pinout

Ch.0	Ch.1	Ch.2	Ch.3	Ch.4	Ch.5	Ch.6	Ch.7	n.c.	n.c.
GND	GND	GND	GND	GND	GND	GND	GND	GND	GND