VXG Signal Generator and X-Series Signal Analyzers

M9484C VXG Vector Signal Generator N9042B, N9041B, N9040B, N9032B, N9030B, and N9021B Signal Analyzers

This manual provides documentation for the M9484C and X-Series Signal Analyzers running the Microsoft Windows 10 operating system.

KEYSIGHT

MEASUREMENT GUIDE

Notices

© Keysight Technologies, Inc. 2021- 2026

No part of this manual may be reproduced in any form or by any means (including electronic storage and retrieval or translation into a foreign language) without prior agreement and written consent from Keysight Technologies, Inc. as governed by United States and international copyright laws.

Trademark Acknowledgments

Bluetooth and the Bluetooth logos are trademarks owned by Bluetooth SIG, Inc., U.S.A. and licensed to Keysight Technologies, Inc.

Manual Part Number

M9484-90002

Edition

Edition 1, June 2026
Supersedes: February 2025

Published by:
Keysight Technologies
1400 Fountaingrove Parkway
Santa Rosa, CA 95403

Warranty

THE MATERIAL CONTAINED IN THIS DOCUMENT IS PROVIDED "AS IS," AND IS SUBJECT TO BEING CHANGED, WITHOUT NOTICE, IN FUTURE EDITIONS. FURTHER, TO THE MAXIMUM EXTENT PERMITTED BY APPLICABLE LAW, KEYSIGHT DISCLAIMS ALL WARRANTIES, EITHER EXPRESS OR IMPLIED WITH REGARD TO THIS MANUAL AND ANY INFORMATION CONTAINED HEREIN, INCLUDING BUT NOT LIMITED TO THE IMPLIED WARRANTIES OF MERCHANTABILITY AND FITNESS FOR A PARTICULAR PURPOSE. KEYSIGHT SHALL NOT BE LIABLE FOR ERRORS OR FOR INCIDENTAL OR CONSEQUENTIAL DAMAGES IN CONNECTION WITH THE FURNISHING, USE, OR PERFORMANCE OF THIS DOCUMENT OR ANY INFORMATION CONTAINED HEREIN. SHOULD KEYSIGHT AND THE USER HAVE A SEPARATE WRITTEN AGREEMENT WITH WARRANTY TERMS COVERING THE MATERIAL IN THIS DOCUMENT THAT CONFLICT WITH THESE TERMS, THE WARRANTY TERMS IN THE SEPARATE AGREEMENT WILL CONTROL.

Technology Licenses

The hardware and/or software described in this document are furnished under a license and may be used or copied only in accordance with the terms of such license.

U.S. Government Rights

The Software is "commercial computer software," as defined by Federal Acquisition Regulation ("FAR") 2.101. Pursuant to FAR 12.212 and 27.405-3 and Department of Defense FAR Supplement ("DFARS") 227.7202, the U.S. government acquires commercial computer software under the same terms by which the software is customarily provided to the public. Accordingly, Keysight provides the Software to U.S. government customers under its standard commercial license, which is embodied in its End User License Agreement (EULA), a copy of which can be found at http://www.keysight.com/find/sweula. The license set forth in the EULA represents the exclusive authority by which the U.S. government may use, modify, distribute, or disclose the Software. The EULA and the license set forth therein, does not require or permit, among other things, that Keysight: (1) Furnish technical information related to commercial computer software or commercial computer software documentation that is not customarily provided to the public; or (2) Relinquish to, or otherwise provide, the government rights in excess of these rights customarily provided to the public to use, modify, reproduce, release, perform, display, or disclose commercial computer software or commercial computer software documentation. No additional government requirements beyond those set forth in the EULA shall apply, except to the extent that those terms, rights, or licenses are explicitly required from all providers of commercial computer software pursuant to the FAR and the DFARS and are set forth specifically in writing elsewhere in the EULA. Keysight shall be under no obligation to update, revise or otherwise modify the Software. With respect to any technical data as defined by FAR 2.101, pursuant to FAR 12.211 and 27.404.2 and DFARS 227.7102, the U.S. government acquires no greater than Limited Rights as defined in FAR 27.401 or DFAR 227.7103-5 (c), as applicable in any technical data.

Safety Notices

CAUTION

A **CAUTION** notice denotes a hazard. It calls attention to an operating procedure, practice, or the like that, if not correctly performed or adhered to, could result in damage to the product or loss of important data. Do not proceed beyond a CAUTION notice until the indicated conditions are fully understood and met.

WARNING

A **WARNING** notice denotes a hazard. It calls attention to an operating procedure, practice, or the like that, if not correctly performed or adhered to, could result in personal injury or death. Do not proceed beyond a WARNING notice until the indicated conditions are fully understood and met.

Where to Find the Latest Information

Documentation is updated periodically. For the latest information about this product, including instrument software upgrades, application information, and product information, browse to the following URL:

https://www.keysight.com/us/en/product/M9484C/m9484c.html

Information on preventing instrument damage can be found at:

http://keysight.com/find/PreventingInstrumentRepair

Is your product software up-to-date?

Periodically, Keysight releases software updates to fix known defects and incorporate product enhancements. To search for software updates for your product, go to the Keysight Technical Support website at:

http://www.keysight.com/find/techsupport

1. M9484C VXG Basic Measurements
Overview . . . 12
Equipment Setup . . . 13
Required Software . . . 14
Setting Up Triggers on the X-Series Signal Analyzer . . . 15
Using the M9484C SCPI Recorder Function . . . 16
Using the graphical user interface . . . 16
Generating a Continuous Waveform (CW) . . . 20
On the VXG: . . . 20
On the X-Series Signal Analyzer: . . . 20
On the VXG: . . . 21
On the X-Series Signal Analyzer: . . . 21
On the VXG: . . . 22
On the X-Series Signal Analyzer: . . . 22
Using the equivalent SCPI commands . . . 24
Generating Amplitude Modulation from the Signal Block . . . 26
On the VXG: . . . 26
On the X-Series Signal Analyzer: . . . 29
Using the equivalent SCPI commands . . . 30
Generating Frequency Modulation from the Signal Block . . . 32
On the VXG: . . . 32
On the X-Series Signal Analyzer: . . . 34
Using the equivalent SCPI commands . . . 36
Generating Phase Modulation from the Signal Block . . . 37
On the VXG: . . . 37
On the X-Series Signal Analyzer: . . . 40
Using the equivalent SCPI commands . . . 41
Generating Amplitude Modulation from the Output Modulation Block . . . 42
On the VXG: . . . 42
On the X-Series Signal Analyzer: . . . 44
Generating Frequency Modulation from the Output Modulation Block . . . 46
On the VXG: . . . 46
On the X-Series Signal Analyzer: . . . 48
Generating Phase Modulation from the Output Modulation Block . . . 49
On the MXG: . . . 49
On the X-Series Signal Analyzer: . . . 51
Generating Pulse Modulation from the Output Modulation Block . . . 53
On the VXG: . . . 53
On the X-Series Signal Analyzer: . . . 55
Generating a Pulse Train from the Output Modulation Block . . . 57

Setup Diagram. 58
On the VXG: 58
On the MXR Series Oscilloscope: 61
Using Step Sweep Mode. 65
On the VXG: 65
On the X-Series Signal Analyzer: 67
Using List Sweep Mode. 69
On the VXG: 69
On the X-Series Signal Analyzer: 73
Using the M9484C as a Multifunction Generator 75
Introduction 75
Multifunction Generator Demonstrations 80
Special Function Generators 85
Radio Frequency Modulation Option UNT 98
Applications 108
Conclusion 109
Generating Synchronized Pulse (IQ + Analog) Modulation 110
On the VXG: 110
On the X-Series Signal Analyzer: 113
Generating a Multitone Signal 114
On the VXG: 114
On the X-Series Signal Analyzer: 116
Using the equivalent SCPI commands 118
Setting Up Waveform File Vector Modulation 119
On the VXG: 119
On the X-Series Signal Analyzer: 122
On the VXG: 122
On the X-Series Signal Analyzer: 125
Using the equivalent SCPI commands 126
Creating a Waveform Sequence 127
On the VXG: 127
On the X-Series Signal Analyzer: 132

2. Corrections

Corrections/De-embedding Using PathWave N7653APPC Software 136
Adding Fixture Blocks using s2p Files. 136
Using a Spectrum Analyzer to Make the Corrections Measurement 142
Using a Power Meter to Make the Corrections Measurement 149
Using the Equivalent SCPI Commands. 153
Instrument Nonlinear Correction. 155
Setting Up an INC 1CC 4 GHz, 10 dBm EVM Measurement 155

Setting Up an INC 8CC 28 GHz, 10 dBm EVM Measurement . . . 167
Phase Coherency and Alignment Between Channels . . . 176
Required Options . . . 176
Phase Alignment for Modulated Signal Using an MXR . . . 177
Setting up a Single CW Tone on the VXG . . . 177
Setting up the MXR Series Oscilloscope . . . 180
Channel-to-Channel Adjustment Wizard . . . 182
Setting up a Pulse LFM Chirp Waveform on the VXG . . . 188
Setting Up the VSA to Measure the Chirp . . . 191

3. Advanced Measurements

Channel Bonding . . . 202
Generating a Multi-Carrier DVB-S2X Signal for Satellite TV . . . 203
Generating a Wide Band QPSK . . . 217
Generating a 5G NR Waveform . . . 226
Setting Up 8 Virtual Signal Generators . . . 236
On the VXG: . . . 237
On the Signal Analyzer: . . . 245
Using the VSA Flex Frame to Create a DVB-S2X Signal . . . 247
DVB-S2X PHY Layer Framing . . . 248
Part 1: Generating a DVB-S2X Waveform Using N7623C Signal Studio for Digital Modulation . . . 249
Part 2: Generating a DVB-S2X Waveform Using the N7608APPC Signal Studio for Custom Modulation . . . 269
Creating & Analyzing a 5G NR DL 2x2 MIMO Measurement Using Two UXAs . . . 287
On the VXG . . . 288
Demodulate the 5G NR 2x2 MIMO in X-Apps . . . 299
Demodulate the 5G NR 2x2 MIMO Using VSA: . . . 303
Creating and Analyzing a 4x4 MIMO using a UXR Running VSA Software . . . 310
Equipment Setup . . . 310
Setting up a 1x8 MIMO and PUCCH Fading Example . . . 324
Equipment Setup . . . 326
Configuring 1x8 MIMO . . . 327
Setting Up Fading for PUCCH . . . 333
Setting a 1x4 MIMO and Fading HARQ Example . . . 339
Test Purpose . . . 339
Conditions . . . 339
Equipment Setup . . . 339
Basic Steps . . . 340
Setting up the MXG . . . 341
Setting Up the MXG/VXG . . . 344

Setting a 1x4 SYNC and Fading (5GNR & Custom) Example 357
Overview 357
System Requirements 357
System Setup. 358
Setting Up the VXG for Multi-Instrument Synchronization 360
Setting up Envelope Tracking 383
Key Benefits 383
How ET Works 383
System Requirements 383
Setup 384
On the VXG 384

4. Other X-Series Signal Analyzer Measurements

Setting Up a Group Delay Measurement 398
Setting Up Millimeter-Wave Measurements 409
Setting up a 5G NR Millimeter-Wave Measurement 409
Setting-up a Millimeter-Wave DVB-S2X 64APSK Measurement. 418
Setting-up a Millimeter-Wave 5G NR, 2 GHz BW FR2 Measurement 440
Using the X-Series Analyzer's SCPI Recorder Function 464
Using the graphical user interface 464
On the X-Series Signal Analyzer: 466
Using the X-Series Analyzer's Preload/Unload Function 477
Using Real-time Spectrogram Streaming (Option RRT) for N9032B and N9042B 479
Overview 479
On the MXG/VXG Signal Generator: 480
On the PXA or UXA Signal Analyzer: 483
Using the Rtsa Viewer 489
Conclusion 498

5. Custom Modulation Measurements

Using PathWave E7608APPC to Create a Waveform File then Automatically Configure the Analyzer to View the Results 500
On the MXG: 500
Setup Auto Configure Analyzer 503

6. 5G New Radio (NR) Measurements using X-Apps

5G Waveform, EVM, and ACP Analysis Using X-Applications 508
Setting Up Triggers on the Signal Analyzer using 5G NR Mode 508
Setting Up a 1 CC 28 GHz EVM Measurement 510
Setting up an 8 CC 28 GHz EVM Measurement 517
Setting Up a 1 CC 3.5 GHz ACP Measurement 527

Using PathWave N7631APPC to Create a Waveform File then Automatically Configure the Analyzer to View the Results 532

7. 5G NR Measurements Using the VSA Software

5G Waveform and EVM Analysis Using VSA Software 542
Setting Up a 1 CC 28 GHz EVM Measurement 542
Setting Up an 8 CC 28 GHz EVM Measurement 553
Creating a Basic 5G NR Signal Using PathWave N7631APPC Embedded Software . . . 563
Creating a DL MIMO Signal Using PathWave N7631APPC Signal Generation 583

1 M9484C VXG Basic Measurements

- "Overview" on page 12
- "Equipment Setup" on page 13
- "Required Software" on page 14
- "Setting Up Triggers on the X-Series Signal Analyzer" on page 15
- "Using the M9484C SCPI Recorder Function" on page 16
- "Generating a Continuous Waveform (CW)" on page 20
- "Generating Amplitude Modulation from the Signal Block" on page 26
- "Generating Frequency Modulation from the Signal Block" on page 32
- "Generating Phase Modulation from the Signal Block" on page 37
- "Generating Amplitude Modulation from the Output Modulation Block" on page 42
- "Generating Frequency Modulation from the Output Modulation Block" on page 46
- "Generating Phase Modulation from the Output Modulation Block" on page 49
- "Generating Pulse Modulation from the Output Modulation Block" on page 53
- "Generating a Pulse Train from the Output Modulation Block" on page 57
- "Using Step Sweep Mode." on page 65
- "Using List Sweep Mode." on page 69
- "Using the M9484C as a Multifunction Generator" on page 75
- "Generating Synchronized Pulse (IQ + Analog) Modulation" on page 110
- "Generating a Multitone Signal" on page 114
- "Setting Up Waveform File Vector Modulation" on page 119
- "Creating a Waveform Sequence" on page 127

Overview

The M9484C VXG Vector Signal Generator provides frequency coverage from 9 kHz to 54 GHz, with up to 2.5 GHz RF modulation bandwidth per channel using an internal baseband generator, and up to 5 GHz RF modulation bandwidth with channel bonding.

The measurement examples use an X-Series Signal Analyzer to view the results. A few measurement examples require an N9042B signal analyzer and is called out in those specific measurement examples. For information on using the X-Series Signal Analyzer multi-touch user interface, refer to the Online Help.

CAUTION

Please refer to the VXG data sheet and X-Series Signal Analyzer data sheet to ensure your measurement setup has adequate power.

https://www.keysight.com/us/en/product/M9484C/m9484c.html

NOTE

The software versions used in this measurement guide are:

- **VXG: A.19.00**
- **X-Series Spectrum Analyzers/N9085EM0E: A.41.xx or later**
- **89601 VSA: Version 2025 or later**

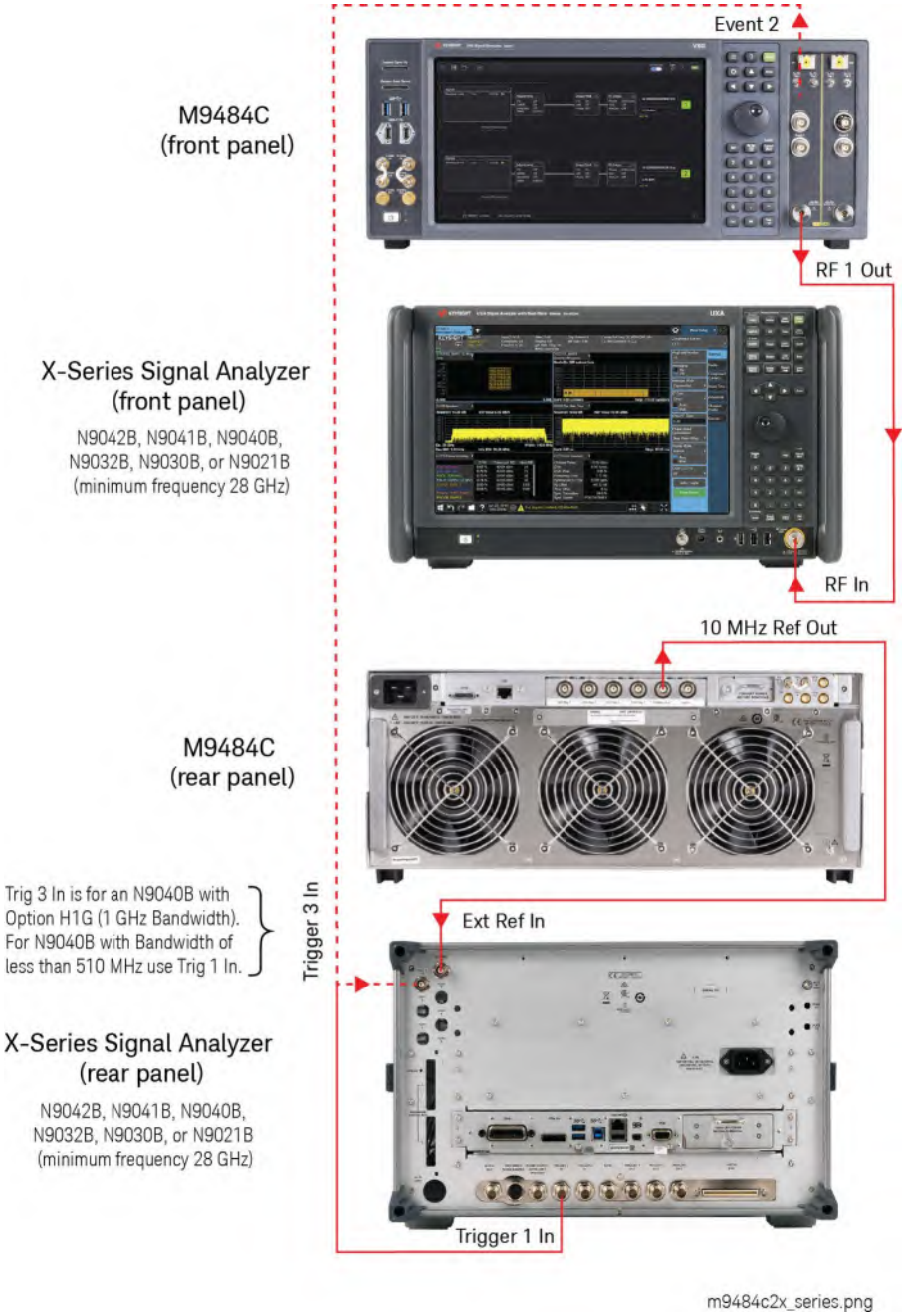
Equipment Setup

– M9484C front panel RF 1 Out to X-Series Signal Analyzer front panel RF In

– M9484C front panel Event 2 to X-Series Signal Analyzer rear panel Trig 1 In

Trig 3 In is used for an N9040B with Option H1G (1 GHz Bandwidth). For N9040B with Bandwidth 510 MHz or less, use Trig 1 In.

– M9484C rear panel 10 MHz Ref Out to X-Series Signal Analyzer rear panel Ext Ref In



m9484c2x_series.png

Required Software

- M9484C
  - N7631APPC - 5GNR
  - N7608APPC - Custom Modulation
  - N7605APOC - 3GPP Real Time/Fading
  - PathWave Automatic Channel Response Correction and S-parameter De-embedding (N7653APPC)

Setting Up Triggers on the X-Series Signal Analyzer

1. From the X-Series Signal Analyzer Menu Panel (on the top right of the display), select **Mode/Meas** > **Spectrum Analyzer** mode.

NOTE

If accessing the instrument via a Remote Desktop connection, select the Screen tab (at the top of the display) to open the Mode/Measurement/View Selector window.

2. Select **Mode Preset** to set Spectrum Analyzer mode to a known state.
3. From the dropdown on the top right, select **Trigger** and set Trigger Source to **Free Run.**

Using the M9484C SCPI Recorder Function

The SCPI Recorder feature allows you to record Signal Generator software GUI interactions as equivalent SCPI commands and save the record/log to a (.txt) file.

NOTE

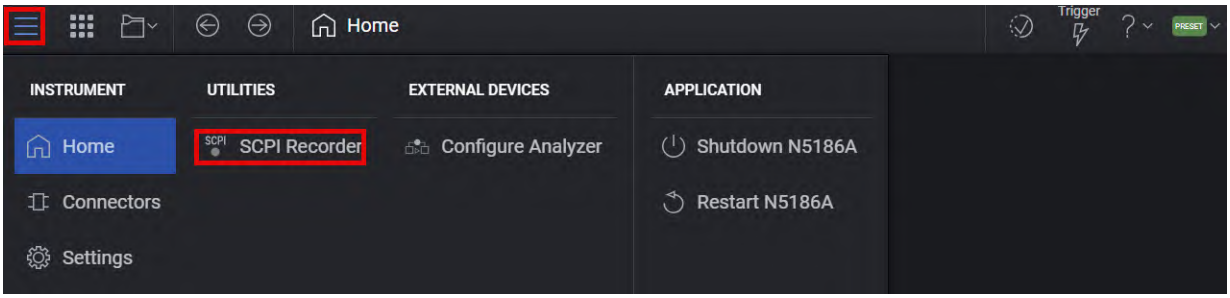
Playback of SCPI recordings is NOT supported in the Pathwave Signal Generator application. External applications that support replaying SCPI commands saved to a .txt file can be used.

For this example, we will generate a simple 5G NR 100 MHz signal.

Using the graphical user interface

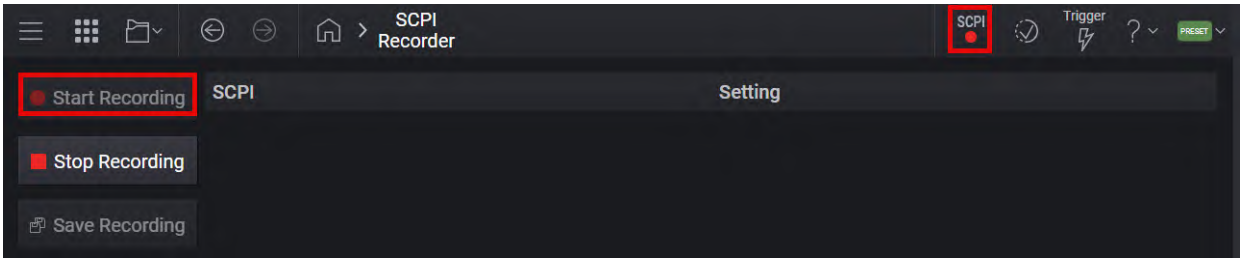
On the VXG:

1. Select **Preset** > **Preset** to set the VXG to a known state.
2. Select the **System** menu > **SCPI Recorder**.

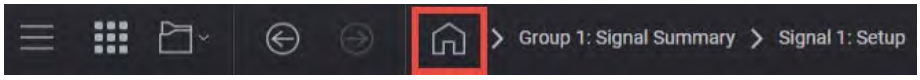


3. Starts recording graphical user interface actions as SCPI commands. Select **Start Recording** to record the graphical user interface actions as SCPI commands.

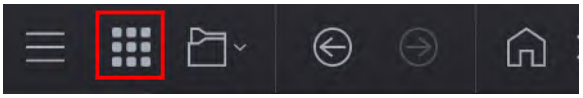
   Notice that a SCPI indicator is displayed on the top of the display.



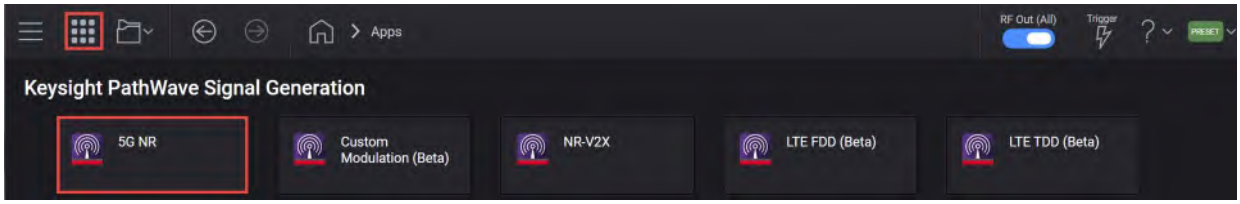
4. Close the Systems menu by selecting the **Home** icon at the top of the display.



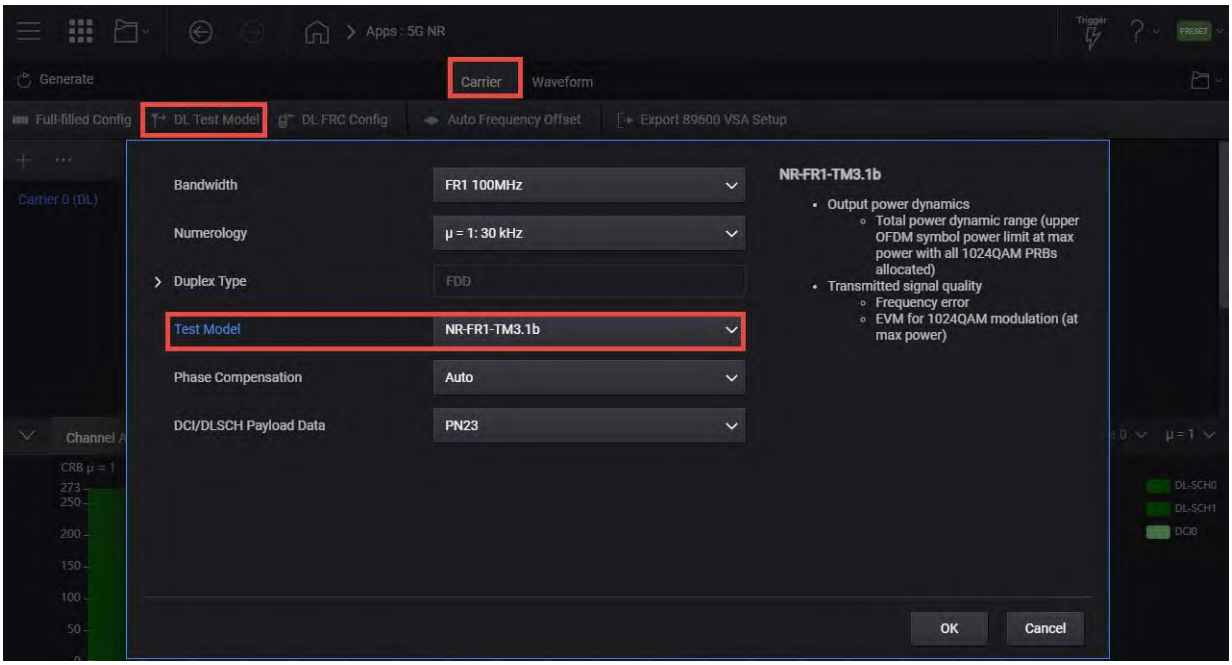
5. In the Output area, set Frequency to **3.5 GHz** and Power to **0 dBm**.
6. Select the **Radio Apps** block to open the mode selection panel.



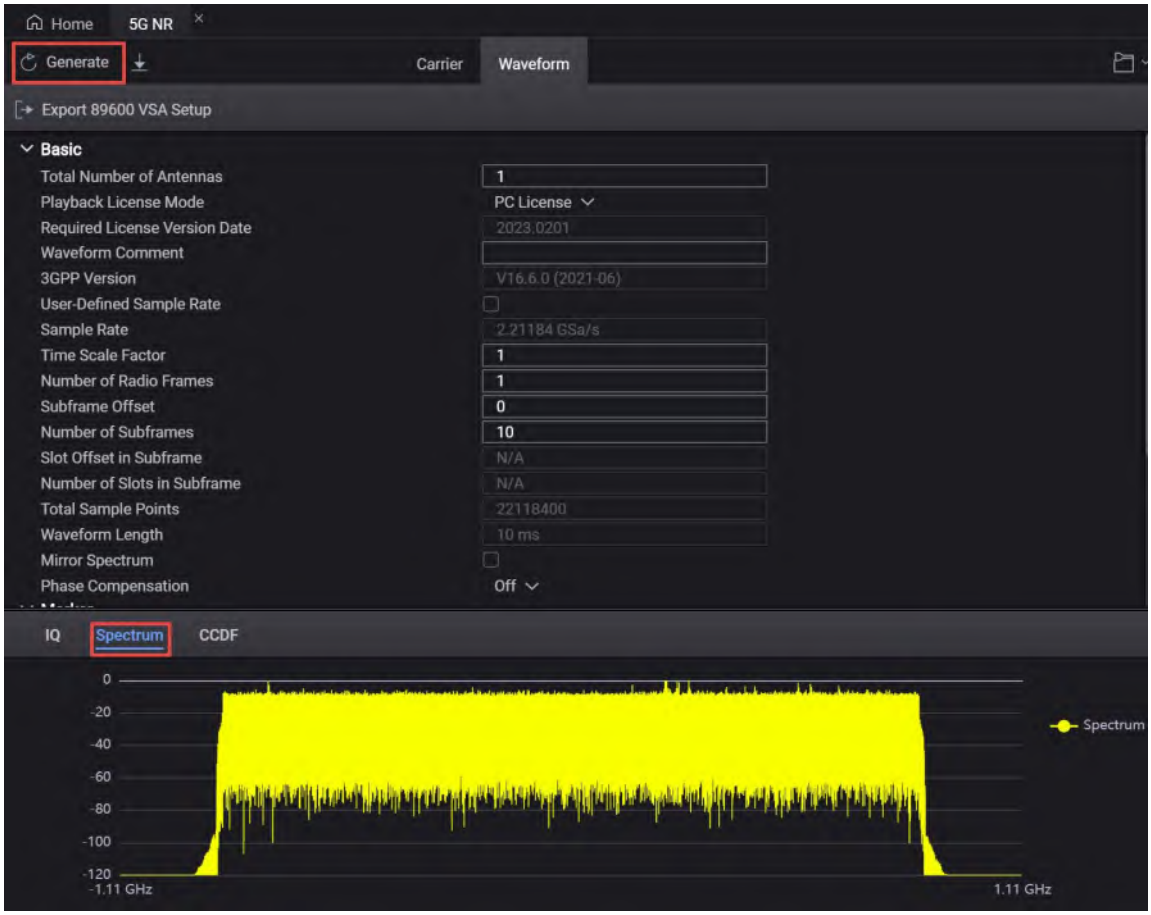
7. Select **5G NR/5G-Advanced**.



8. Select the **Carrier** tab > **Test Model** and set Test Model to **NR-FR1-TM3.1b**, leave the remain settings at their default values, and select **OK**.



9. Select the **Waveform** tab, then **Generate** to generate the Waveform, and then select **Home** to exit the setup panel.

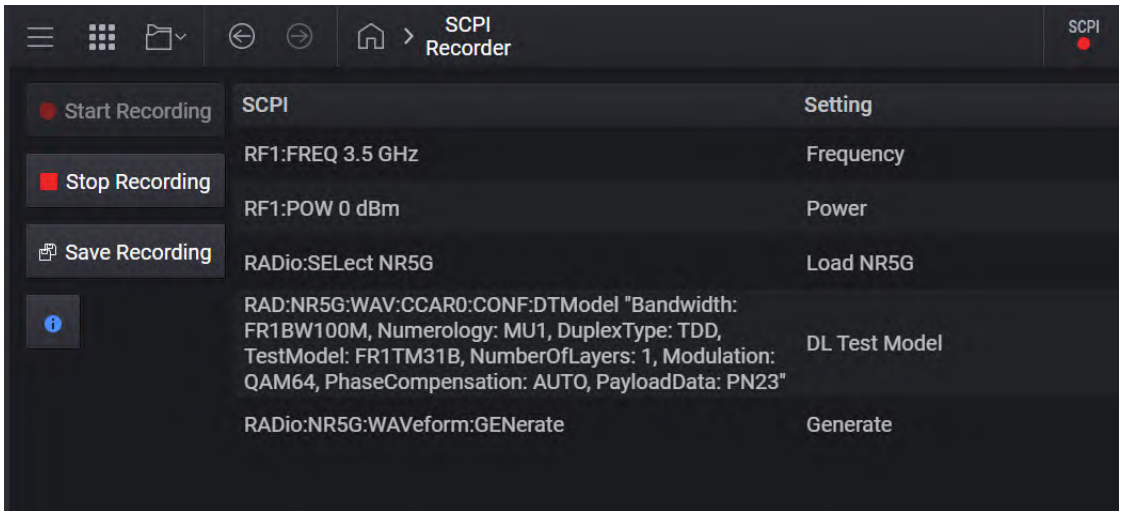


10. Set RF Out to **On** by selecting the numbered channel indicator switch.

This enables the RF Out for the indicated channels, in this case Channel 1 if using a multi-channel VXG.

11. Open the **System** menu > **SCPI Recorder**.

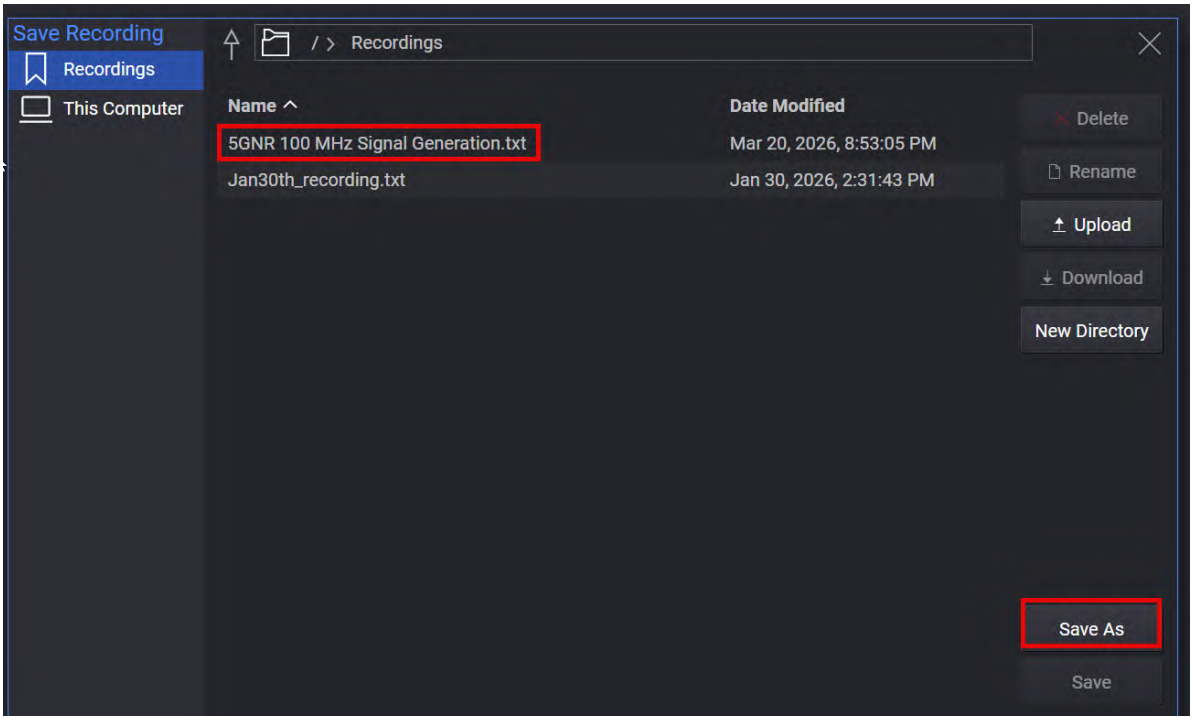
The SCPI commands are displayed for the parameters specified above



12. Select Stop Recording. A dialog box comes up where you can Save the recording. Select **Stop & Save Recording**.

    Once SCPI recording is stopped, you may either save the SCPI recording using this button or discard it. Saving or discarding the recording will clear all previously recorded commands displayed on screen.

13. In the Save Recording window, select **Save As** and specify a file name for this recording



Generating a Continuous Waveform (CW)

This procedure will demonstrate the amplitude and frequency accuracy of the VXG at RF and µW frequencies.

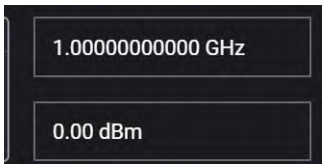
NOTE

Ensure the equipment and triggers are properly configured. Refer to "Equipment Setup" on page 13 and "Setting Up Triggers on the X-Series Signal Analyzer" on page 15.

On the VXG:

1. Select **Preset** > **Preset** to set the instrument to a known state.
2. In the Output area, set Frequency to **1 GHz** and Power to **0 dBm**.

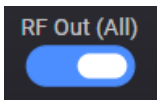
   These values are coupled to CW Frequency and Total Power (RMS) in the corresponding RF Output Block.



3. Set RF Out to **On** by selecting the numbered channel indicator switch.

   This enables the RF Out for the indicated channels, in this case Channel 1 if using a multi-channel VXG.

4. **For multi-channel instruments only:** In the top right corner of the display, set RF Out (All) to **On** by selecting the switch.



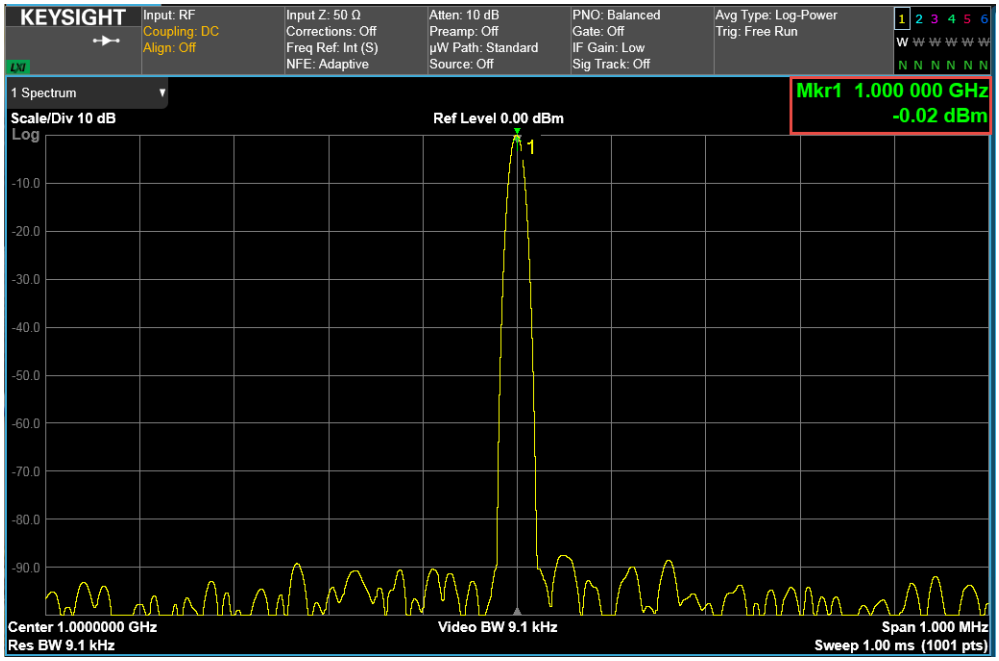
NOTE

In order to turn on RF for any channel, both the RF Out for the specific channel (for example, Channel 1 or Channel 2), and RF Out All must be turned on.

On the X-Series Signal Analyzer:

1. Select **Mode Preset** to set Spectrum Analyzer mode to a known state.
2. From the Menu Panel, select **Frequency** and set Center Frequency to **1 GHz** and Span to **1 MHz**.
3. Select **Peak Search**.

Observe the accuracy of the amplitude and frequency of the signal.



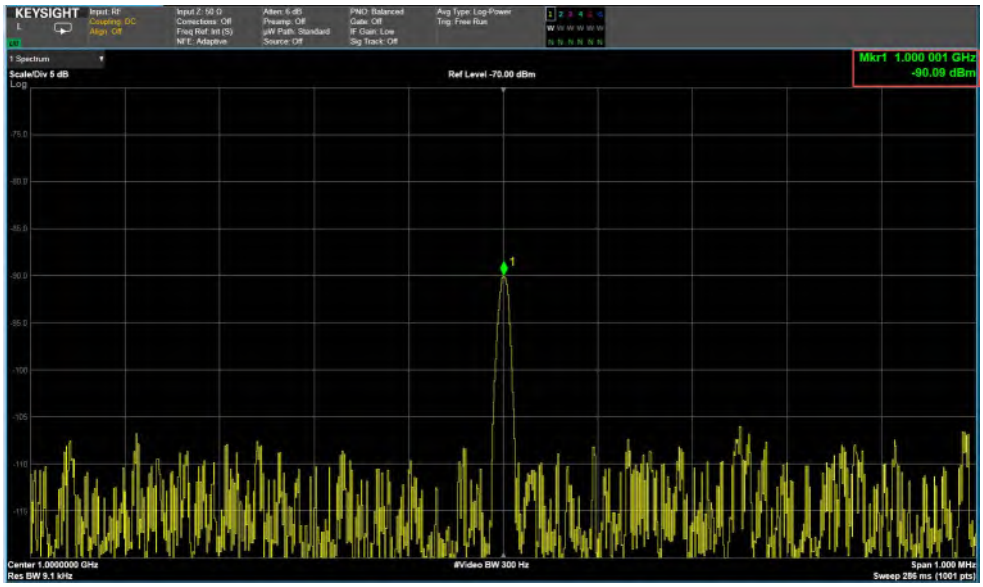
On the VXG:

1. Set Output 1 Power to **–90 dBm.**
2. Ensure that **RF Out** is On.

On the X-Series Signal Analyzer:

1. Select **Amplitude** and set Ref Level to **–70 dBm** and Scale/Div **5 dB**.
2. Select **BW** and set Video BW to **300 Hz**.
3. Select **Peak Search**.

Observe the frequency and amplitude accuracy of the VXG at low power levels.

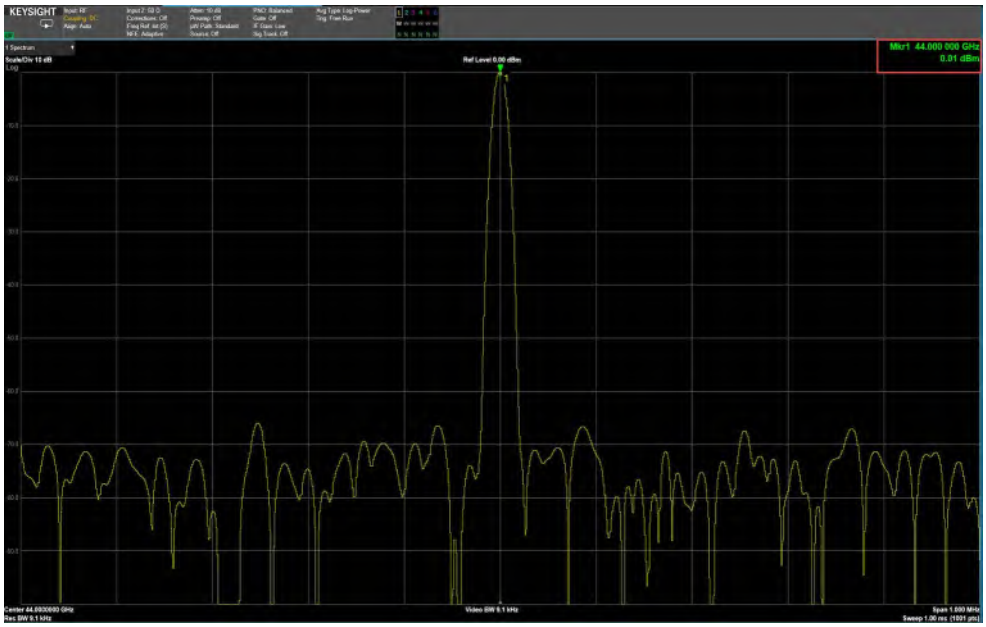


On the VXG:

1. Set Frequency to **44 GHz** and Power to **0 dBm**.

On the X-Series Signal Analyzer:

1. Select **Frequency** and set Center Frequency to **44 GHz**.
2. Select **Amplitude** and set Ref Level to **0 dBm** and Scale/Div to **10 dB**.
3. Select **BW** > **Video BW** and set to **Auto**.
4. Select **Peak Search**.



Observe the frequency and amplitude accuracy at high frequency levels.

Using the equivalent SCPI commands

Creating a CW signal on Channel 1.

On the VXG:

```
SYSTem:PRESet
RF1:FREQuency:CW 1GHZ
RF1:POWer:AMPLitude 0dBm
RF1:OUTPut:STATe ON
```

For multi-channel instruments, set RF Out (all) to On.

```
RFALl:OUTPut ON
```

On the X-Series Signal Analyzer:

```
FREQuency:CENTer 1GHZ
FREQuency:SPAN 1MHZ
CALCulate:MARKer1:MAXimum
```

On the VXG:

```
RF1:FREQuency:CW 1GHZ
RF1:POWer:AMPLitude -90dBm
RF1:OUTPut:STATe ON
```

On the X-Series Signal Analyzer:

```
DISPlay:WINDow1:TRACe:Y:RLEVel -70
DISPlay:WINDow1:TRACe:Y:PDIVision 5
BWIDth:VIDeo 300Hz
CALCulate:MARKer1:MAXimum
```

On the VXG:

```
RF1:FREQuency:CW 44GHZ
RF1:POWer:AMPLitude 0dBm
```

On the X-Series Signal Analyzer:

```
FREQuency:CENTer 44GHZ
DISPlay:WINDow1:TRACe:Y:RLEVel 0
DISPlay:WINDow1:TRACe:Y:PDIVision 10
BANDwidth:VIDeo:AUTO ON
```

```
CALCulate:MARKer1:MAXimum
```

Generating Amplitude Modulation from the Signal Block

NOTE The VXG must have the N7642APPC PathWave Signal Generation for IQ Based AM, FM, Phase license installed.

NOTE Ensure the equipment and triggers are properly configured. Refer to "Equipment Setup" on page 13 and "Setting Up Triggers on the X-Series Signal Analyzer" on page 15.

AM is applied inside the signal generation chain, before RF up-conversion. It is part of the analog/baseband modulation functions.

- Use Signal block AM when AM represents power behavior of the transmitter or channel rather than modulation content.
- Use Output Modulation block AM when AM represents power behavior of the transmitter or channel rather than modulation content. Refer to "Generating Amplitude Modulation from the Output Modulation Block" on page 42.

Key Characteristics:

- Modulates the envelope of the carrier mathematically, as part of the waveform
- Preserves RF output calibration and ALC behavior downstream
- Supports clean, repeatable AM depth and frequency with good spectral fidelity
- Best when AM is part of the intended signal content (e.g. standards testing, receiver sensitivity, and distortion measurements)

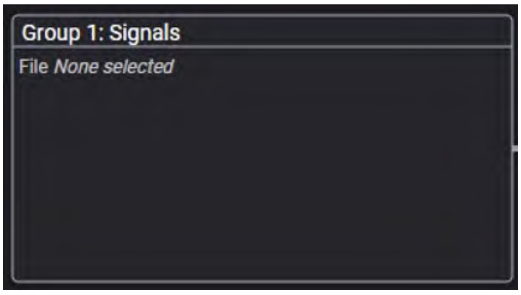
Use Signal block AM when AM is part of the signal you are generating.

Follow the steps below for AM analog modulation using the internal or an external I/Q modulation source.

On the VXG:

1. Select **Preset** > **Preset** to set the instrument to a known state.
2. Set Frequency to **20 GHz** and Amplitude to **0 dBm**.

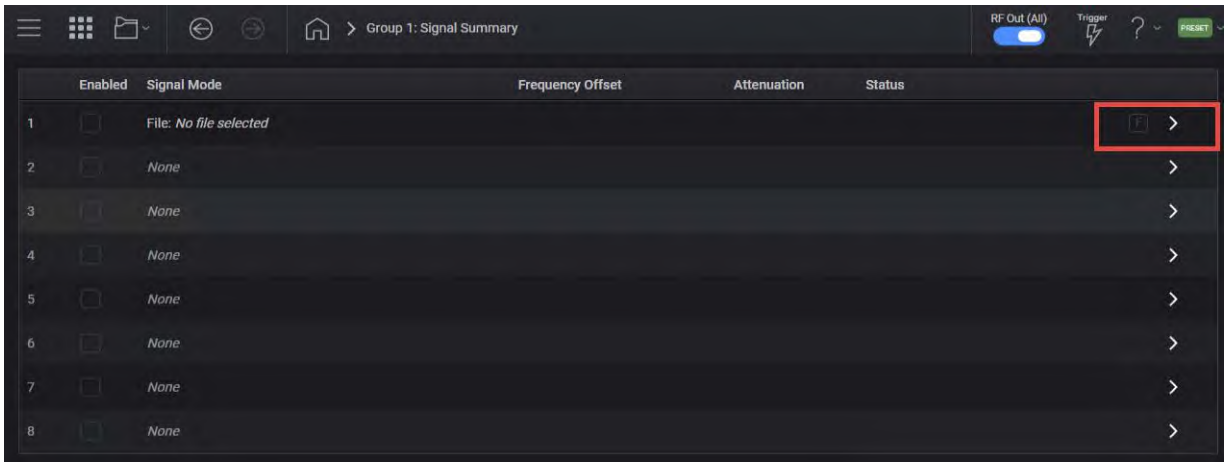
3. Select the **Signal** block.



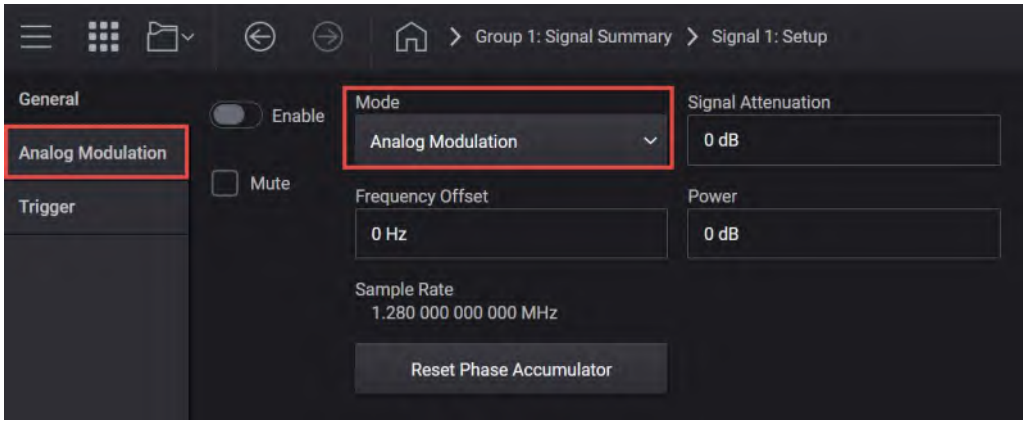
4. Select the arrow for Signal 1 to open the Signal Setup window.

NOTE

This screen is only accessible if Option M9484C-8SG (8 virtual signal generators) is installed. For all other option configurations, continue to the next step.



5. In the **Mode** dropdown, select **Analog Modulation** and then select the **Analog Modulation** tab in the left pane.



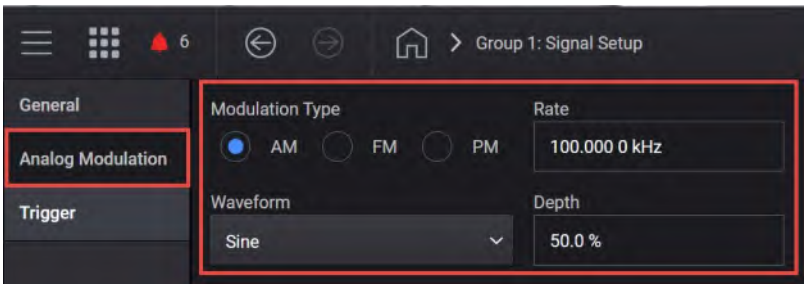
6. In the Analog Modulation Signal Setup:

a. Set Modulation Type to **AM**.

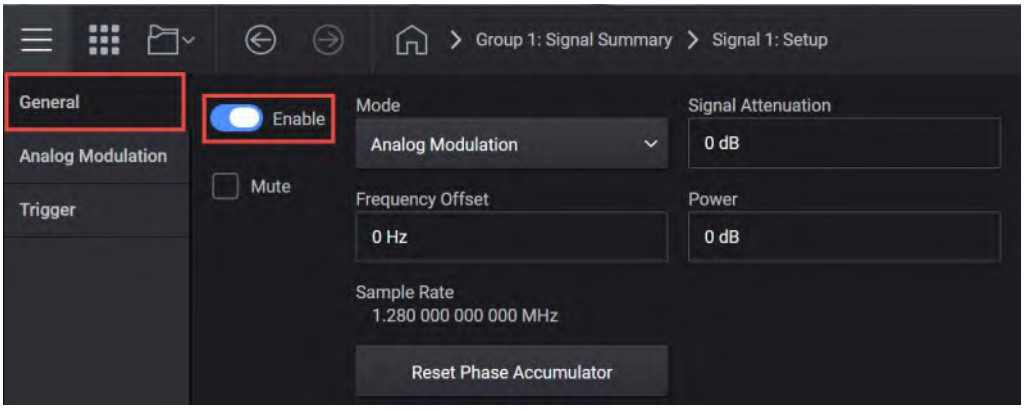
b. Set Waveform to **Sine**.

c. Set Rate to **100 kHz**.

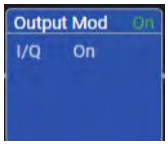
d. Set Depth to **50%**.



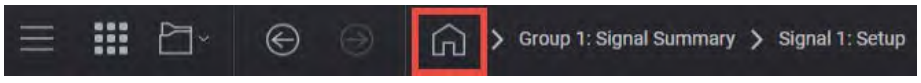
7. Select the **General** tab > **Enable**.



Selecting Enable automatically turns on both Output Modulation and Internal I/Q Modulation as displayed in the Output Modulation block.



8. Close the Vector Modulation Signal Setup by selecting the **Home** icon at the top of the display.

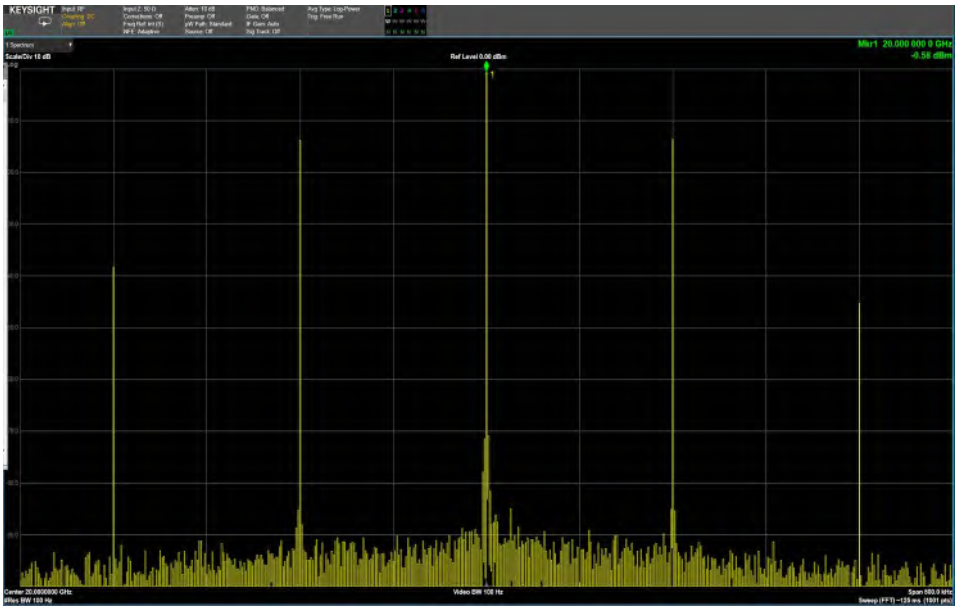


9. Set RF Out to **On** by selecting the numbered channel indicator switch.

This enables the RF Out for the indicated channels, in this case Channel 1 if using a multi-channel VXG.

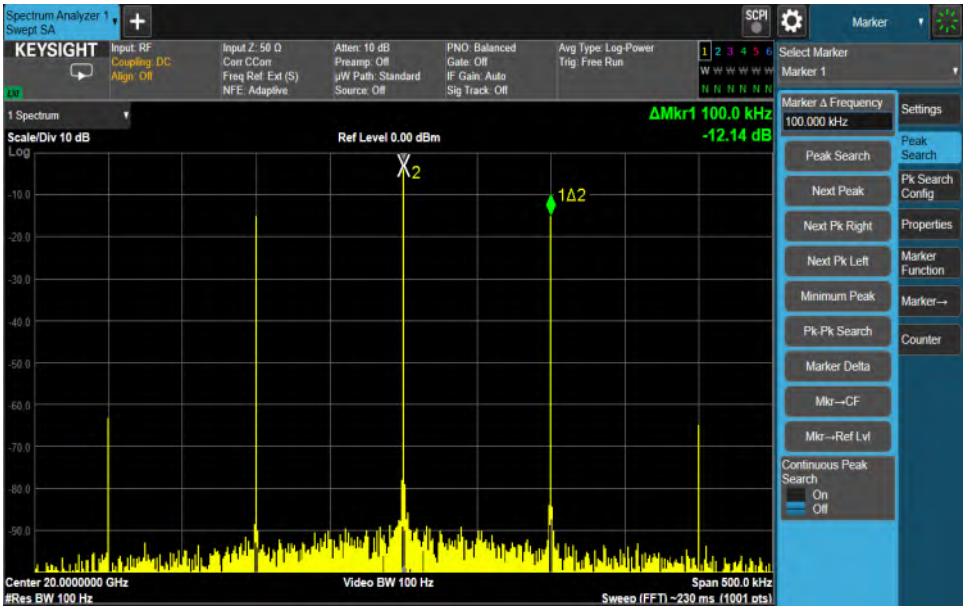
On the X-Series Signal Analyzer:

1. Select **Mode Preset** to set Spectrum Analyzer mode to a known state.
2. Select **Frequency** and set Center Frequency to **20 GHz** and Span to **500 kHz**.
3. Select **BW** and set Res BW to **100 Hz**.



4. Select **Peak Search**.
5. Use markers to measure sideband power relative to the center frequency by selecting **Marker Delta**. Select **Next Pk Right** until the second marker is at the next highest peak.

The Delta Marker should be approximately –12 dB for 50% AM.



Using the equivalent SCPI commands

On The VXG:

```
SYSTem:PRESet
RF1:FREQuency:CW 20GHZ
RF1:POWer:AMPLitude 0dBm
SIGNal:MODE AMODulation
SIGNal1:AMODulation:TYPE AM
SIGNal1:AM:SHAPe SINE
SIGNal1:AM:FREQuency 100KHZ
SIGNal1:AM 50
SIGNal1 ON
RF1:OUTPut:STATe ON
```

For multi-channel instruments, set RF Out (all) to On.

```
RFALl:OUTPut ON
```

On the X-Series Signal Analyzer:

```
FREQuency:CENTer 20GHZ
FREQuency:SPAN 500KHZ
BANDwidth 100 Hz
```

```
CALCulate:MARKer1:MODE DELTa
```

```
CALCulate:MARKer1:MAXimum:RIGHt
```

Repeat the above command until the marker is at the next highest peak.

To retrieve the delta marker:

```
CALCulate:MARKer1:Y?
```

Generating Frequency Modulation from the Signal Block

NOTE The VXG must have the N7642APPC PathWave Signal Generation for IQ Based AM, FM, Phase license installed.

NOTE Ensure the equipment and triggers are properly configured. Refer to "Equipment Setup" on page 13 and "Setting Up Triggers on the X-Series Signal Analyzer" on page 15.

- **Signal block FM** is inside the signal generation chain, before the RF upconversion. FM is mathematically applied to the carrier during waveform creation. It changes the instantaneous frequency of the signal itself according to the modulation waveform.
- **Output Modulation block FM** is in the RF output stage, after upconversion and signal generation. It dynamically pulls the RF carrier frequency around its nominal value and does not modify the waveform or baseband signal. See "Generating Frequency Modulation from the Output Modulation Block" on page 46.

Key Characteristics:

- FM sidebands are part of the generated signal
- Clean, deterministic, and highly repeatable
- Independent of RF output leveling, ALC, or hardware settling
- Preserves modulation accuracy and spectral predictability

Typical Use:

- FM demodulation tests
- Sensitivity and distortion measurements
- Standards-based or algorithmic FM testing

Use Signal block when FM is what you want to transmit.

Follow the steps below for setting up FM, which is applied at the Signal block stage, by directly varying the RF output power.

On the VXG:

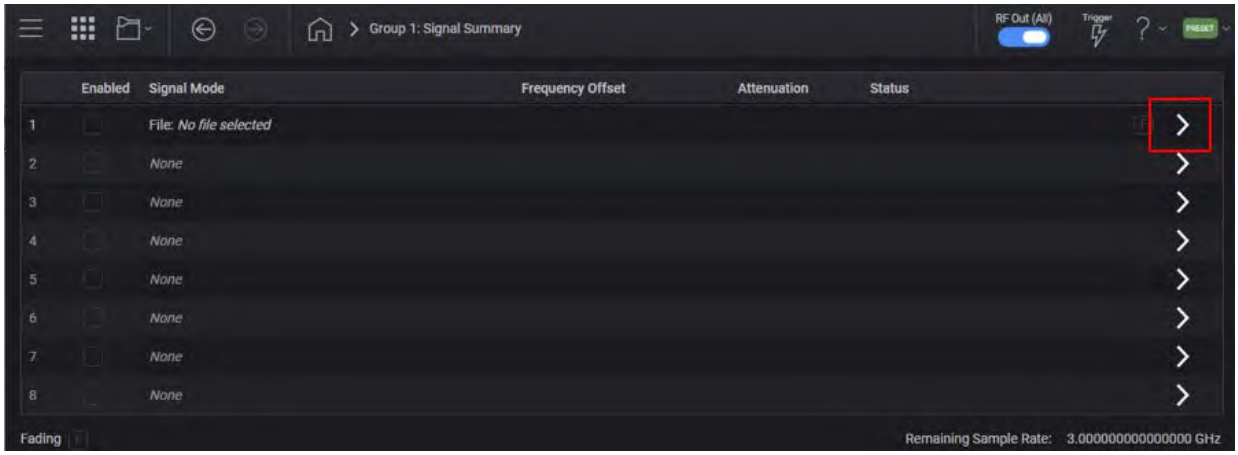
1. Select **Preset** > **Preset** to set the instrument to a known state.
2. Set Frequency to **20 GHz** and Amplitude to **0 dBm**.

3. Select the **Signal** block.

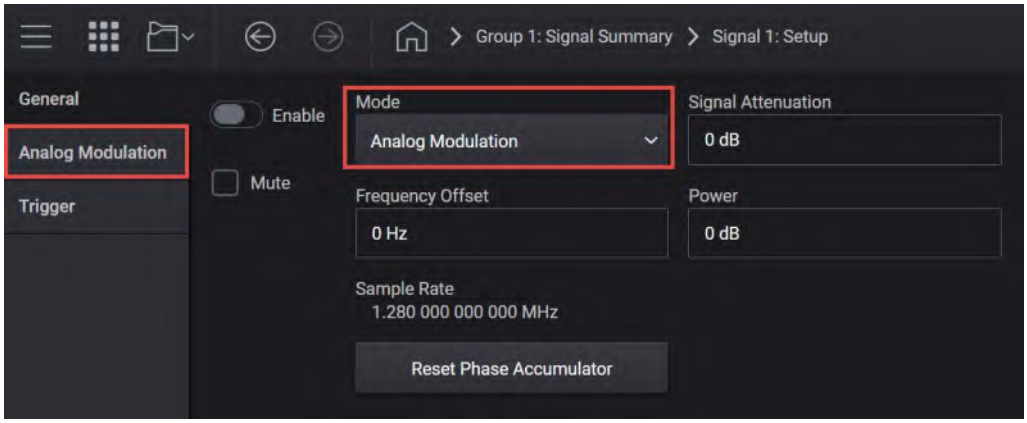
4. Select the arrow for Signal 1 to open the Signal Setup window.

NOTE

This screen is only accessible if Option M9484C-8SG (8 virtual signal generators) is installed. For all other option configurations, continue to the next step.



5. In the Mode dropdown, select **Analog Modulation** and then the **Analog Modulation** tab.

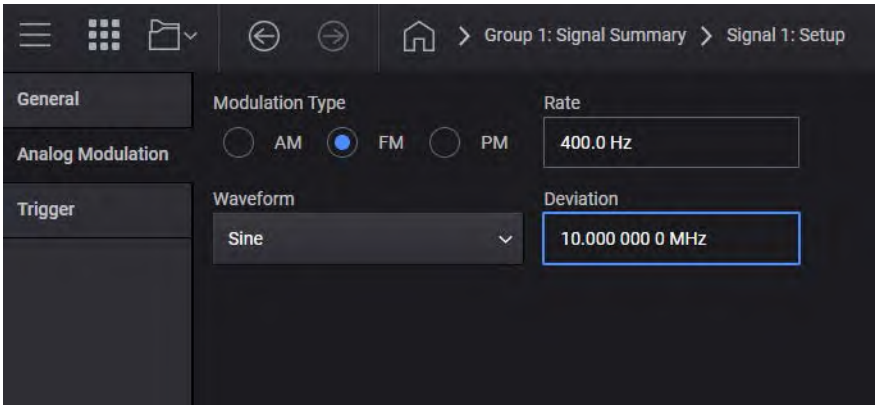


   a. Set Modulation Type to **FM**.

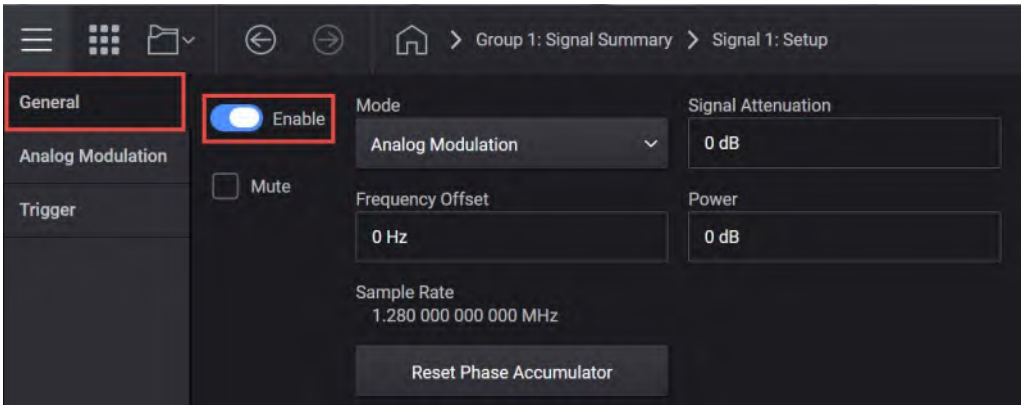
b. Set Waveform to **Sine**.

c. Set Rate to **400 Hz**.

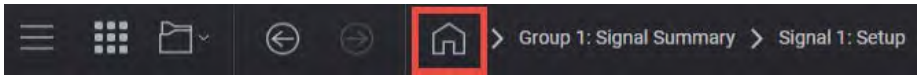
d. Set Deviation to **10 MHz**.



6. Select the **General** tab > **Enable**.



7. Close the Vector Modulation Signal Setup by selecting the **Home** icon at the top of the display.



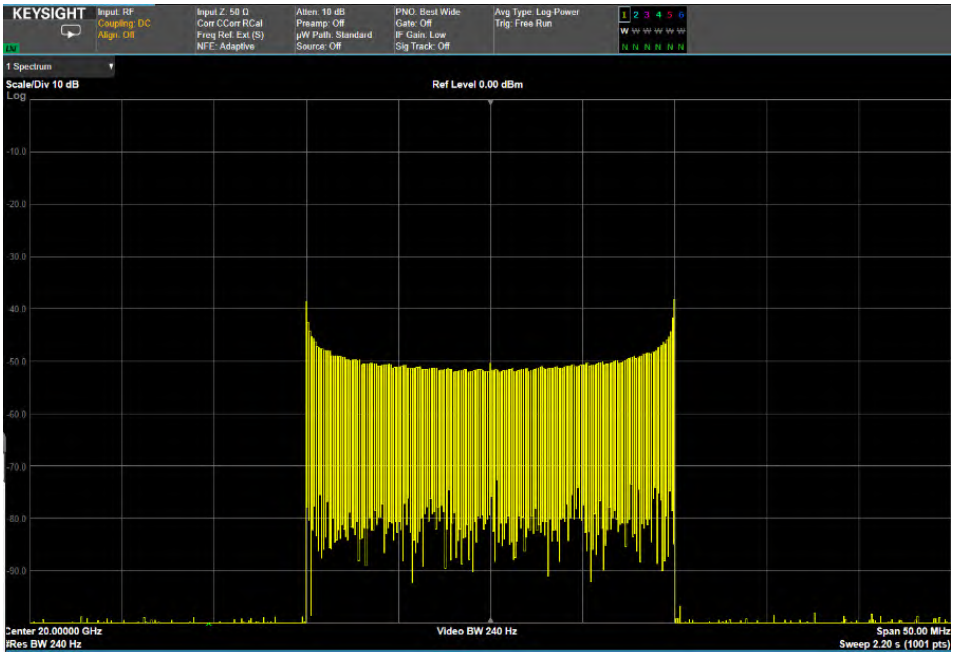
8. Set RF Out to **On** by selecting the numbered channel indicator switch.

This enables the RF Out for the indicated channels, in this case Channel 1 if using a multi-channel VXG.

On the X-Series Signal Analyzer:

1. Select **Mode Preset** to set Spectrum Analyzer mode to a known state.

2. Select **Frequency** and set Center Frequency to **20 GHz** and Span to **50 MHz**.
3. Select **BW** and set Res BW to **240 Hz**.



Using the equivalent SCPI commands

On the VXG:

```
SYSTem:PRESet

RF1:FREQuency:CW 20GHZ

RF1:POWer:AMPLitude 0dBm

SIGNal:MODE AMODulation

SIGNal1:AMODulation:TYPE FM

SIGNal1:FM:SHAPe SINE

SIGNal1:FM:FREQuency 400HZ

SIGNal1:FM 10MHZ

SIGNal1 ON

RF1:OUTPut:STATe ON
```

For multi-channel instruments, set RF Out (all) to On.

```
RFALl:OUTPut ON
```

On the X-Series Signal Analyzer:

```
FREQuency:CENTer 20GHZ

FREQuency:SPAN 50MHZ

ACPower:BANDwidth 240 Hz
```

Generating Phase Modulation from the Signal Block

NOTE The VXG must have the N7642APPC PathWave Signal Generation for IQ Based AM, FM, Phase license installed. The X-Series Analyzer must have N9063EM0E Analog Demodulation Measurement license installed.

NOTE Ensure the equipment and triggers are properly configured. Refer to "Equipment Setup" on page 13 and "Setting Up Triggers on the X-Series Signal Analyzer" on page 15.

- **Signal block PM** is inside the signal generation chain, before the RF upconversion. It applies true phase modulation to the carrier as part of the signal definition. The instantaneous carrier phase is mathematically modulated by the PM waveform.
- **Output Modulation block PM** is in the RF output stage, after upconversion and signal generation. It dynamically pulls the RF carrier frequency around its nominal value and does not modify the waveform or baseband signal. See "Generating Pulse Modulation from the Output Modulation Block" on page 53.

Key Characteristics:

- PM sidebands are intrinsic to the waveform
- Spectrally clean and deterministic
- Fully repeatable and independent from the RF hardware behavior
- Does not interact with RF leveling or synthesizer loops

Typical Use:

- PM or PSK-related receiver testing
- Demodulation accuracy and sensitivity measurements
- Standards-based testing where PM is intentional signal content
- Educational or analytical PM measurements

Use Signal block when PM is what you want to transmit.

Follow the steps below for setting up PM, which is applied at the RF output stage, by directly varying the RF output power.

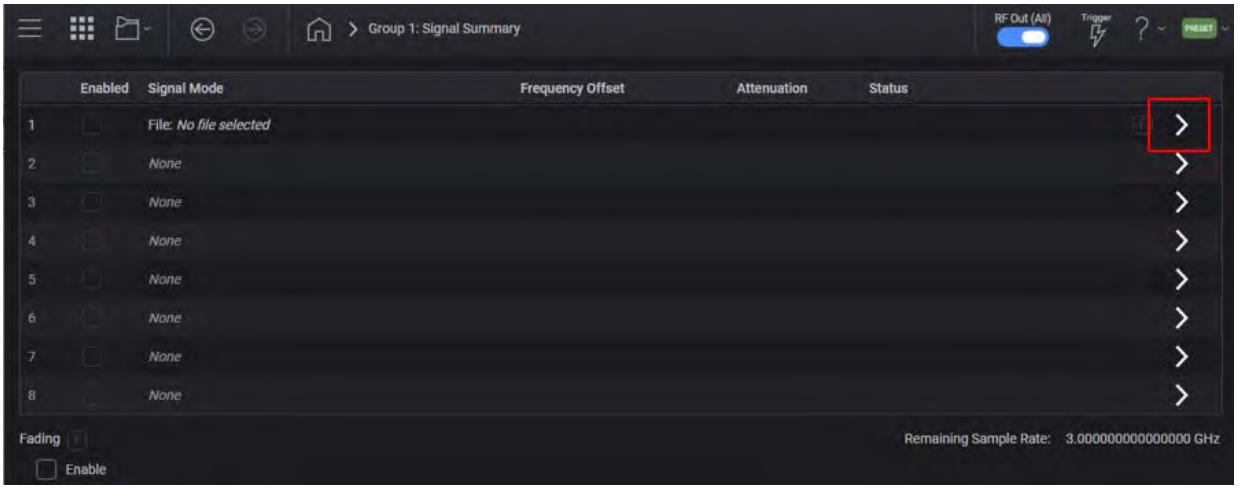
On the VXG:

1. Select **Preset** > **Preset** to set the instrument to a known state.
2. Set Frequency to **20 GHz** and Amplitude to **0 dBm**.

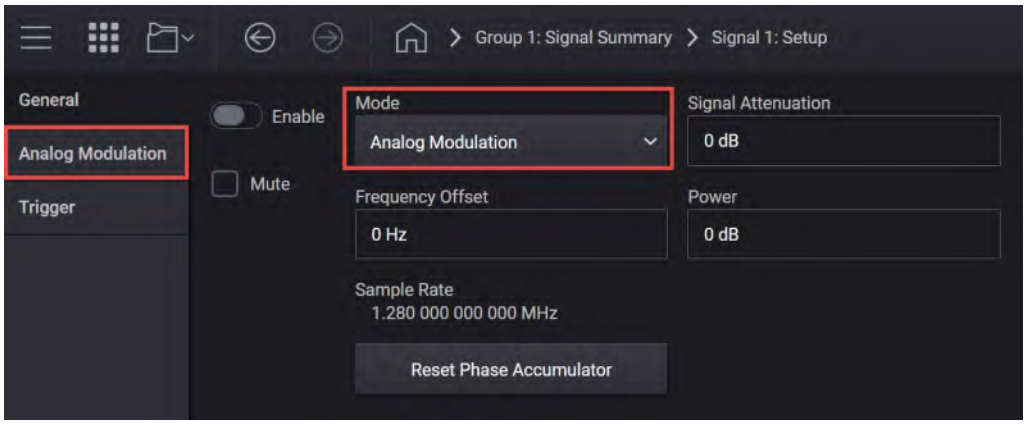
3. Select the **Signal** block to open.

4. Select the arrow for Signal 1 to open the Signal Setup window.

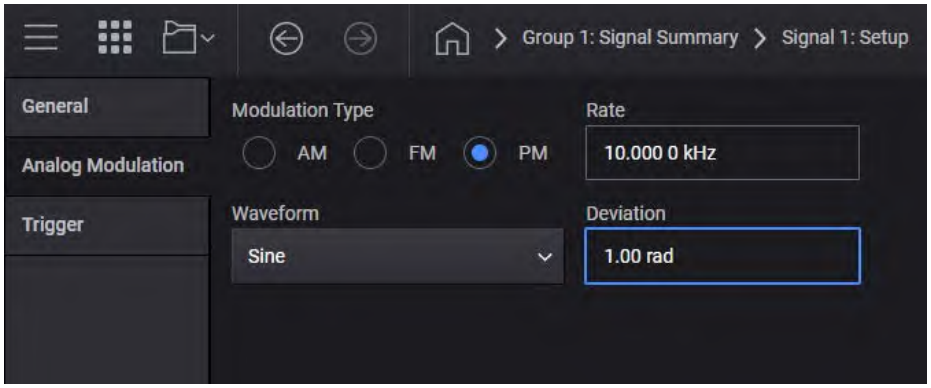
NOTE This screen is only accessible if Option M9484C-8SG (8 virtual signal generators) is installed. For all other option configurations, continue to the next step.



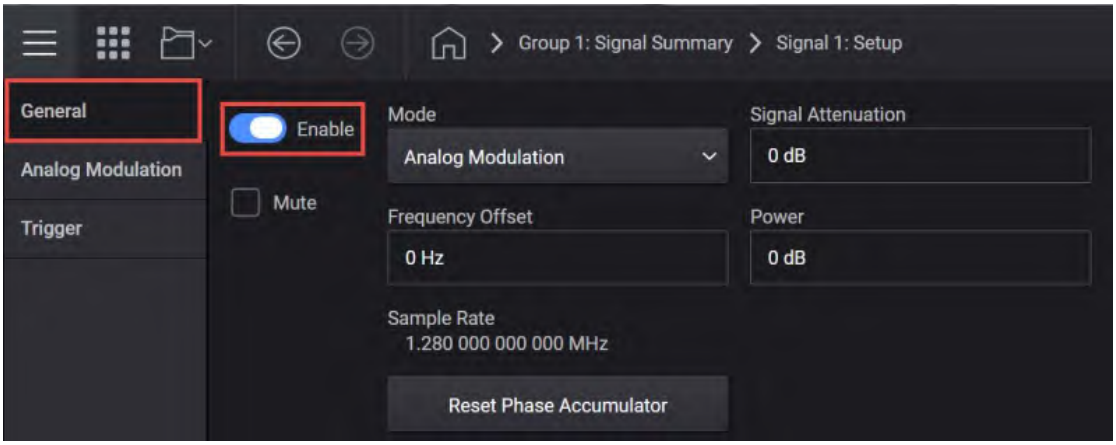
5. In the Mode dropdown, select **Analog Modulation** and then the **Analog Modulation** tab.



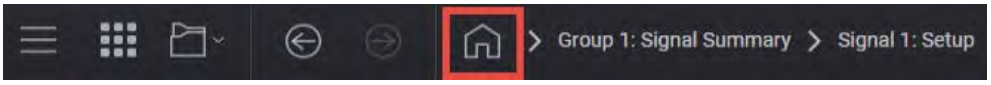
6. In the Analog Modulation Signal Setup:
   a. Set Modulation Type to **PM**.
   b. Set Waveform to **Sine**.
   c. Set Rate to **10 kHz**.
   d. Set Deviation to **1 rad**.



7. Select the **General** tab > **Enable**.



8. Close the Vector Modulation Signal Setup by selecting the **Home** icon at the top of the display.



9. Set RF Out to **On** by selecting the numbered channel indicator switch.

   This enables the RF Out for the indicated channels, in this case Channel 1 if using a multi-channel VXG.

On the X-Series Signal Analyzer:

1. From the Menu Panel (on the top right of the display), select **Mode/Meas** > **Analog Demod** mode.

NOTE

If accessing the instrument via a Remote Desktop connection, select the Screen tab (at the top of the display) to open the Mode/Measurement/View Selector window.

2. Select **Mode Preset** to set Spectrum Analyzer mode to a known state.
3. From the Menu Panel (on the top right of the display), select **Mode/Meas** > **Analog Demod** mode > **PM** Measurement > **Quad** View.

NOTE

If accessing the instrument via a Remote Desktop connection, select the Screen tab (at the top of the display) to open the Mode/Measurement/View Selector window.

4. Select **Frequency** and set Center Frequency to **20 GHz**.

Using the equivalent SCPI commands

On the VXG:

```
SYSTem:PRESet

RF1:FREQuency:CW 20GHZ

RF1:POWer:AMPLitude 0dBm

SIGNal:MODE AMODulation

SIGNal1:AMODulation:TYPE PM

SIGNal1:PM:SHAPe SINE

SIGNal1:PM:FREQuency 10KHZ

SIGNal1:PM 1

SIGNal1 ON

RF1:OUTPut:STATe ON
```

For multi-channel instruments, set RF Out (all) to On.

```
RFALl:OUTPut ON
```

On the X-Series Signal Analyzer:

```
INSTrument:CONFigure:ADEMOD:PM

SYSTem:PRESet

DISPlay:VIEW:ADVanced:SELect "QUAD"

FREQuency:CENTer 20GHZ
```

Generating Amplitude Modulation from the Output Modulation Block

NOTE The VXG must have Option UNT installed.

NOTE Ensure the equipment and triggers are properly configured. Refer to "Equipment Setup" on page 13 and "Setting Up Triggers on the X-Series Signal Analyzer" on page 15.

- Use Signal block AM when AM represents power behavior of the transmitter or channel rather than modulation content. Refer to "Generating Amplitude Modulation from the Signal Block" on page 26.
- Use the Output Modulation block AM when AM represents power behavior of the transmitter or channel rather than modulation content.

Key Characteristics:

- Effectively a power-level modulation, not a waveform-level modulation.
- Interacts with ALC, power limits, and RF output bandwidth.
- AM bandwidth and linearity are limited by the RF output hardware and ALC settings.
- Can slightly affect spectral purity compared to Signal Block AM.

Typical Use:

- Simulating a power-level modulation, not a waveform-level modulation
- Stressing DUT AGC behavior
- Situations where you want to modulate the output power itself, not the signal definition.

Use Output Modulation block AM when AM represents power behavior of the transmitter or channel rather than modulation content.

Follow the steps below for setting up AM, which is applied at the RF output stage, by directly varying the RF output power.

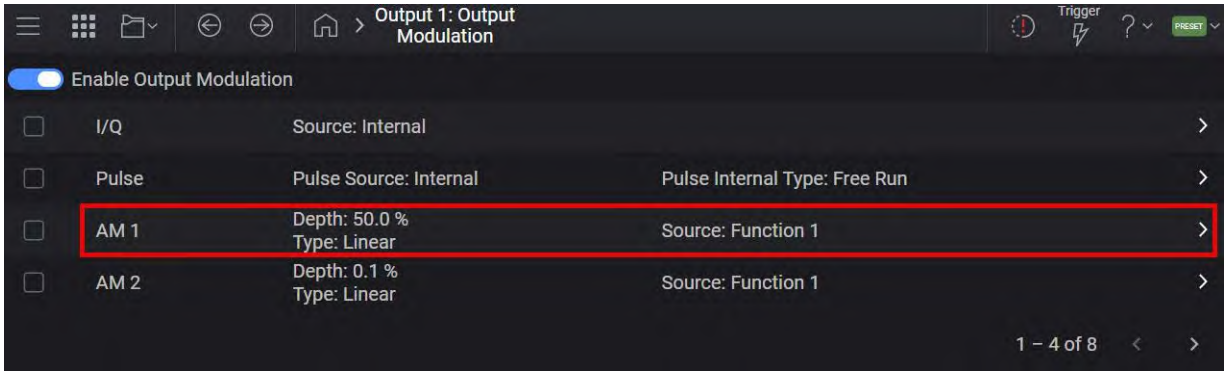
On the VXG:

1. Select **Preset** > **Preset** to set the instrument to a known state.
2. Set Frequency to **20 GHz** and Amplitude to **0 dBm**.

3. Select the **Out Mod** block.

4. Select the arrow button for **AM1**.

   AM1 and AM2 are two separate amplitude versus time control paths. Each path can be enabled or disabled independently and driven by its own modulation source and parameters. When both are enabled, they are applied multiplicatively to the RF output power.



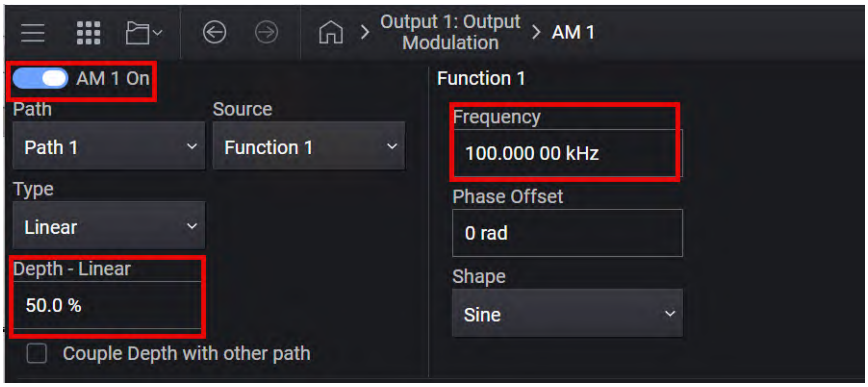
5. In the AM1 dialog box:

   a. Set Shape to **Sine**.

   b. Set Frequency to **100 kHz**.

   c. Set Depth-Linear to **50%**.

   d. Set AM1 **On**

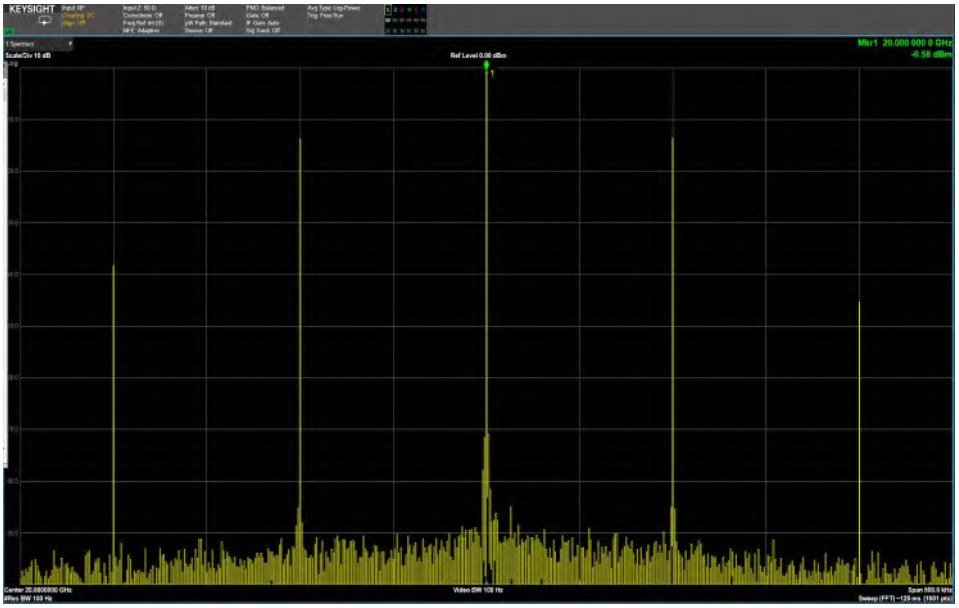


6. Set RF Out to **On** by selecting the numbered channel indicator switch.

This enables the RF Out for the indicated channels, in this case Channel 1 if using a multi-channel VXG.

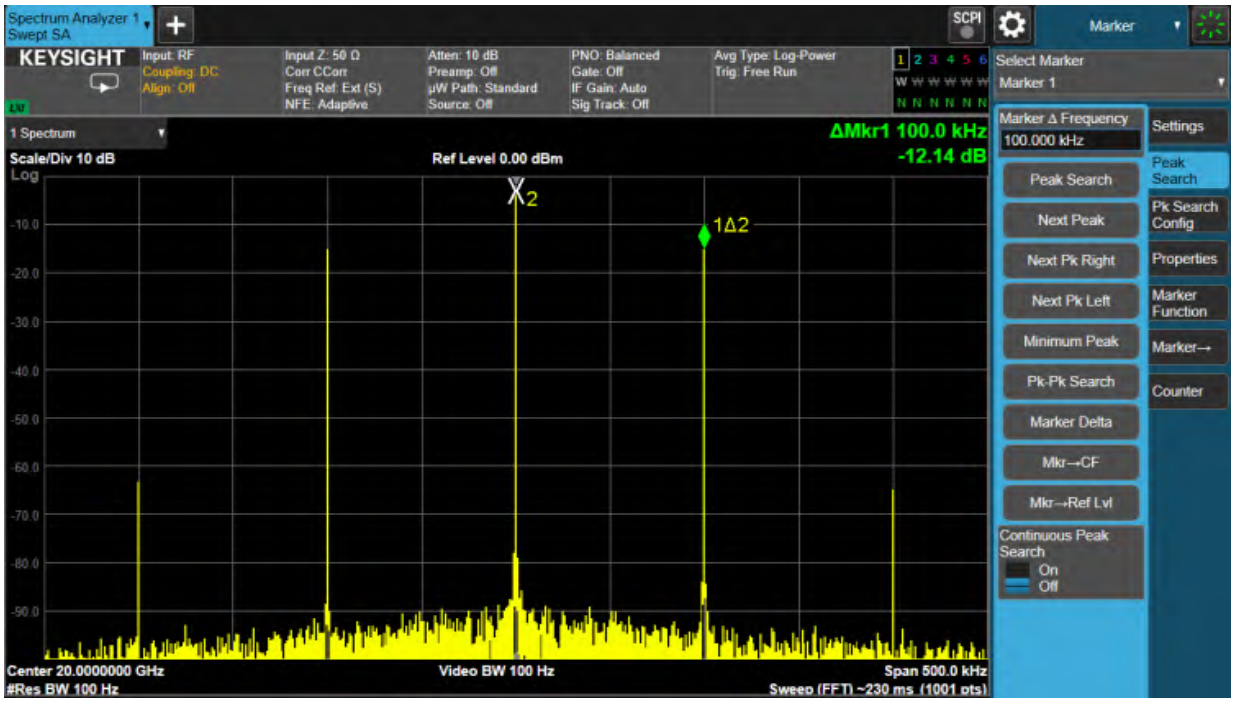
On the X-Series Signal Analyzer:

1. Select **Mode Preset** to set Spectrum Analyzer mode to a known state.
2. Select **Frequency** and set Center Frequency to **20 GHz** and Span to **500 kHz**.
3. Select **BW** and set Res BW to **100 Hz**.



4. Select **Peak Search**.
5. Use markers to measure sideband power relative to the center frequency by selecting **Marker Delta**. Select **Next Pk Right** until the second marker is at the next highest peak.

   The Delta Marker should be approximately –12 dB for 50% AM.


Spectrum Analyzer 1
Swept SA
KEYSIGHT
Input: RF
Coupling: DC
Align: Off
Input Z: 50 Ω
Corr CCorr
Freq Ref: Ext (S)
NFE: Adaptive
Atten: 10 dB
Preamp: Off
µW Path: Standard
Source: Off
PNO: Balanced
Gate: Off
IF Gain: Auto
Sig Track: Off
Avg Type: Log-Power
Trig: Free Run
SCPI
Marker
Select Marker
Marker 1
1 Spectrum
ΔMkr1 100.0 kHz
-12.14 dB
Scale/Div 10 dB
Ref Level 0.00 dBm
Log
-10.0
-20.0
-30.0
-40.0
-50.0
-60.0
-70.0
-80.0
-90.0
1Δ2
Center 20.0000000 GHz
#Res BW 100 Hz
Video BW 100 Hz
Span 500.0 kHz
Sweep (FFT) ~230 ms (1001 pts)
Marker Δ Frequency
100.000 kHz
Peak Search
Next Peak
Next Pk Right
Next Pk Left
Minimum Peak
Pk-Pk Search
Marker Delta
Mkr→CF
Mkr→Ref Lvl
Continuous Peak Search
On
Off
Settings
Peak Search
Pk Search Config
Properties
Marker Function
Marker→
Counter

Generating Frequency Modulation from the Output Modulation Block

NOTE The VXG must have Options UNT, ST6, and 600 installed.

NOTE Ensure the equipment and triggers are properly configured. Refer to "Equipment Setup" on page 13 and "Setting Up Triggers on the X-Series Signal Analyzer" on page 15.

- **Signal block FM** is inside the signal generation chain, before the RF upconversion. FM is mathematically applied to the carrier during waveform creation. It changes the instantaneous frequency of the signal itself according to the modulation waveform. See "Generating Frequency Modulation from the Signal Block" on page 32.
- **Output Modulation block FM** is in the RF output stage, after upconversion and signal generation. It dynamically pulls the RF carrier frequency around its nominal value and does not modify the waveform or baseband signal.

Key Characteristics:

- Represents frequency instability, not modulation content
- Interacts with RF synthesizer behavior and output hardware
- Limited by RF output dynamics
- Can stress DUT tracking, AFC, or PLL loops

Typical Use:

- LO drift or wander simulation
- AFC/PLL stress testing
- Emulating real transmitter frequency instability
- Environment or vibration-induced frequency errors

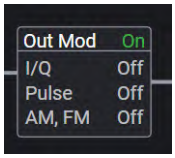
Use Output Modulation block when FM represents "hardware behavior."

Follow the steps below for setting up FM, which is applied at the RF output stage, by directly varying the RF output power.

On the VXG:

1. Select **Preset** > **Preset** to set the instrument to a known state.
2. Set Frequency to **20 GHz** and Amplitude to **0 dBm**.

3. Select the **Out Mod** block.



4. Select the arrow button for **5-8**, then the right arrow for **FM1**.

   FM1 and FM2 are two separate serial frequency modulation paths in the RF output stage. Each path can be enabled or disabled independently and varies the instantaneous RF frequency around the carrier. When both are enabled, their deviations are added together at the RF output. They exist so that you can layer multiple frequency-error effects without touching the signal waveform.

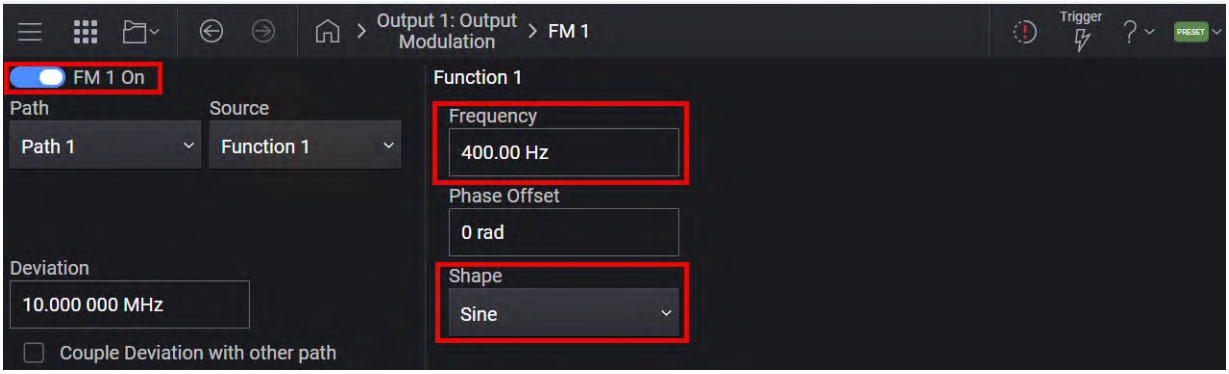
5. In the FM1 dialog box.

   a. Set Waveform to **Sine**.

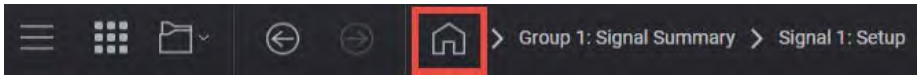
   b. Set Frequency to **400 Hz**.

   c. Set Deviation to **10 MHz**.

   d. Set FM 1 **On**



6. Close the Vector Modulation Signal Setup by selecting the **Home** icon at the top of the display.

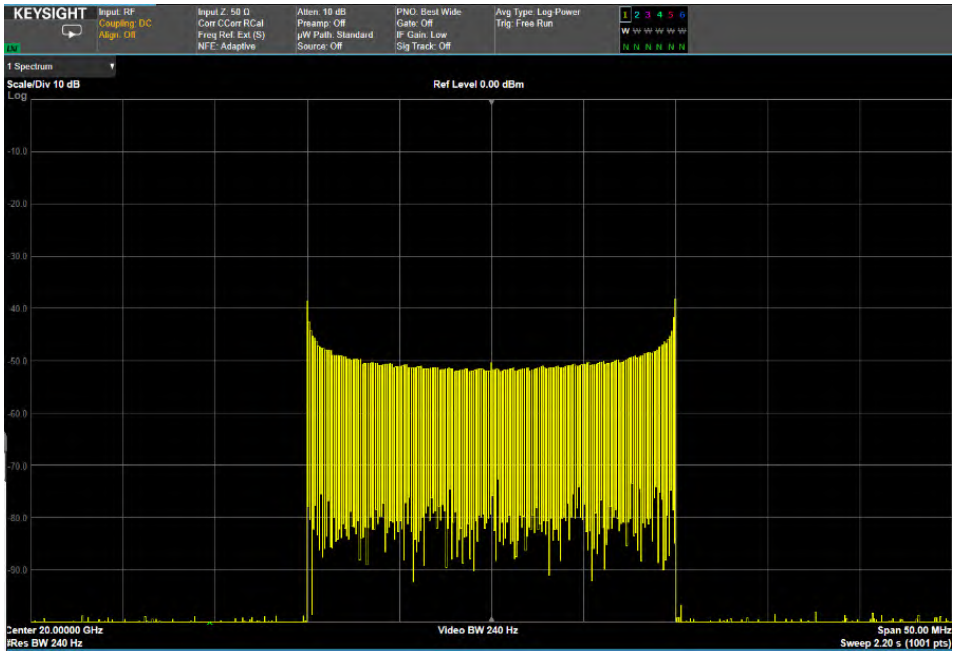


7. Set RF Out to **On** by selecting the numbered channel indicator switch.

   This enables the RF Out for the indicated channels, in this case Channel 1 if using a multi-channel VXG.

On the X-Series Signal Analyzer:

1. Select **Mode Preset** to set Spectrum Analyzer mode to a known state.
2. Select **Frequency** and set Center Frequency to **20 GHz** and Span to **50 MHz**.
3. Select **BW** and set Res BW to **240 Hz**.



Generating Phase Modulation from the Output Modulation Block

NOTE The MXG must have Option UNT, installed.

NOTE Ensure the equipment and triggers are properly configured. Refer to "Equipment Setup" on page 13 and "Setting Up Triggers on the X-Series Signal Analyzer" on page 15.

- **Signal block PM** is inside the signal generation chain, before the RF upconversion. It applies true phase modulation to the carrier as part of the signal definition. The instantaneous carrier phase is mathematically modulated by the PM waveform. See"Generating Phase Modulation from the Signal Block" on page 37.
- **Output Modulation block PM** is in the RF output stage, after upconversion and signal generation. It dynamically pulls the RF carrier frequency around its nominal value and does not modify the waveform or baseband signal.

Key Characteristics:

- Represents hardware phase instability, not signal modulation
- Closely related to LO phase perturbations
- Can stress DUT carrier recovery, PLLs, and phase tracking loops
- Limited by RF output dynamics and synthesizer behavior

Typical Use:

- Simulating LO phase noise or vibration-induced phase error
- Carrier recovery and PLL robustness testing
- Stressing coherent receivers without altering waveform content

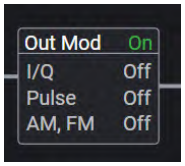
Use Output Modulation block when PM represents "transmitter phase behavior."

Follow the steps below for setting up PM, which is applied at the RF output stage, by directly varying the RF output power.

On the MXG:

1. Select **Preset** > **Preset** to set the instrument to a known state.
2. Set Frequency to **6 GHz** and Amplitude to **0 dBm**.

3. Select the **Out Mod** block.



4. Select the arrow button for **5-8**, then the right arrow for **PM1**.

   In the MXG Output Modulation block, PM1 and PM2 are not different kinds of phase modulation. They are simply two independent, cascaded phase-modulation stages applied at the RF output stage. PM1 is the first RF output phase-modulation path and PM2 is the second RF phase-modulated path, applied after PM1.

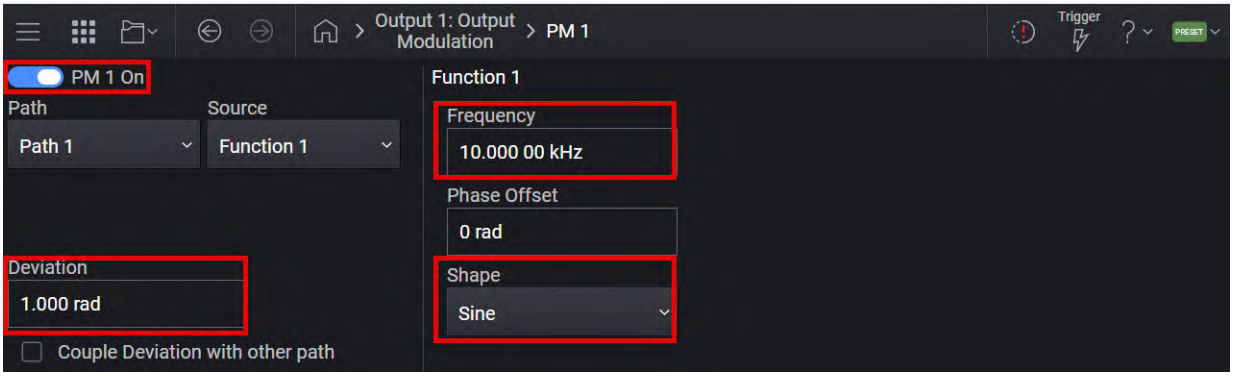
5. In the PM1 Setup:

   a. Set Shape to **Sine**.

   b. Set Frequency to **10 kHz**.

   c. Set Deviation to **1 rad**.

   d. Set PM1 **On**.



6. Set RF Out to **On** by selecting the numbered channel indicator switch.

This enables the RF Out for the indicated channels, in this case Channel 1 if using a multi-channel MXG.

On the X-Series Signal Analyzer:

1. From the Menu Panel (on the top right of the display), select **Mode/Meas** > **Analog Demod** mode.

NOTE

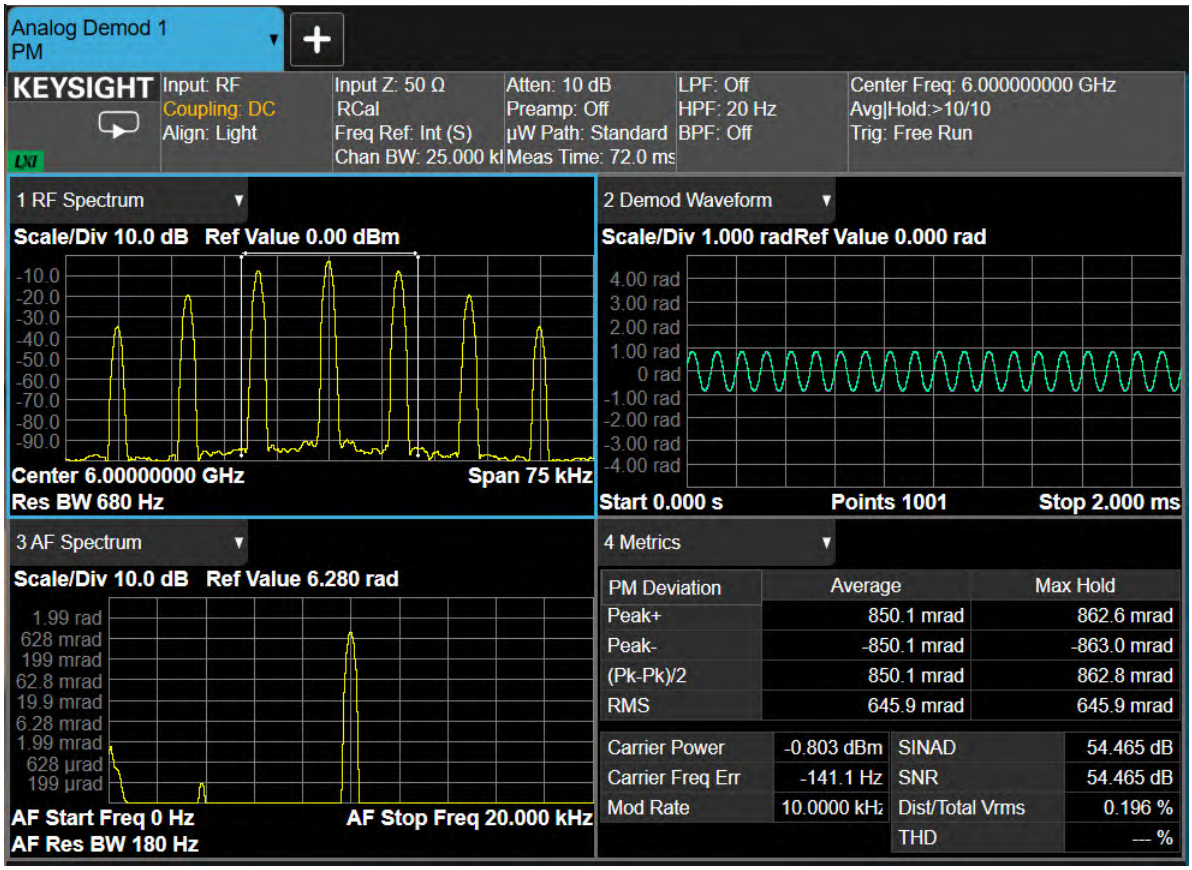
If accessing the instrument via a Remote Desktop connection, select the Screen tab (at the top of the display) to open the Mode/Measurement/View Selector window.

2. Select **Mode Preset** to set Spectrum Analyzer mode to a known state.
3. From the Menu Panel (on the top right of the display), select **Mode/Meas** > **Analog Demod** mode > **PM** Measurement > **Quad** View.

NOTE

If accessing the instrument via a Remote Desktop connection, select the Screen tab (at the top of the display) to open the Mode/Measurement/View Selector window.

4. Select **Frequency** and set Center Frequency to **6 GHz**.


Analog Demod 1
PM
KEYSIGHT
Input: RF
Coupling: DC
Align: Light
Input Z: 50 Ω
RCal
Freq Ref: Int (S)
Chan BW: 25.000 kl
Atten: 10 dB
Preamp: Off
µW Path: Standard
Meas Time: 72.0 ms
LPF: Off
HPF: 20 Hz
BPF: Off
Center Freq: 6.000000000 GHz
Avg|Hold:>10/10
Trig: Free Run
1 RF Spectrum
Scale/Div 10.0 dB Ref Value 0.00 dBm
-10.0
-20.0
-30.0
-40.0
-50.0
-60.0
-70.0
-80.0
-90.0
Center 6.00000000 GHz
Res BW 680 Hz
Span 75 kHz
2 Demod Waveform
Scale/Div 1.000 radRef Value 0.000 rad
4.00 rad
3.00 rad
2.00 rad
1.00 rad
0 rad
-1.00 rad
-2.00 rad
-3.00 rad
-4.00 rad
Start 0.000 s
Points 1001
Stop 2.000 ms
3 AF Spectrum
Scale/Div 10.0 dB Ref Value 6.280 rad
1.99 rad
628 mrad
199 mrad
62.8 mrad
19.9 mrad
6.28 mrad
1.99 mrad
628 µrad
199 µrad
AF Start Freq 0 Hz
AF Res BW 180 Hz
AF Stop Freq 20.000 kHz
4 Metrics
PM Deviation
Average
Max Hold
Peak+
850.1 mrad
862.6 mrad
Peak-
-850.1 mrad
-863.0 mrad
(Pk-Pk)/2
850.1 mrad
862.8 mrad
RMS
645.9 mrad
645.9 mrad
Carrier Power
-0.803 dBm
SINAD
54.465 dB
Carrier Freq Err
-141.1 Hz
SNR
54.465 dB
Mod Rate
10.0000 kHz
Dist/Total Vrms
0.196 %
THD
--- %

Generating Pulse Modulation from the Output Modulation Block

NOTE

The VXG must have Options UNT, ST6, and 600 installed.

NOTE

Ensure the equipment and triggers are properly configured. Refer to "Equipment Setup" on page 13 and "Setting Up Triggers on the X-Series Signal Analyzer" on page 15.

- **Signal block PM** is inside the signal generation chain, before the RF upconversion. It applies true phase modulation to the carrier as part of the signal definition. The instantaneous carrier phase is mathematically modulated by the PM waveform. See"Generating Phase Modulation from the Signal Block" on page 37.
- **Output Modulation block PM** is in the RF output stage, after upconversion and signal generation. It dynamically pulls the RF carrier frequency around its nominal value and does not modify the waveform or baseband signal.

Key Characteristics:

- Represents hardware phase instability, not signal modulation
- Closely related to LO phase perturbations
- Can stress DUT carrier recovery, PLLs, and phase tracking loops
- Limited by RF output dynamics and synthesizer behavior

Typical Use:

- Simulating LO phase noise or vibration-induced phase error
- Carrier recovery and PLL robustness testing
- Stressing coherent receivers without altering waveform content

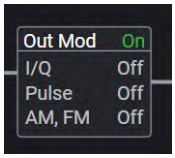
Use Output Modulation block when PM represents "transmitter phase behavior."

Follow the steps below for setting up PM, which is applied at the RF output stage, by directly varying the RF output power.

On the VXG:

1. Select **Preset** > **Preset** to set the instrument to a known state.
2. Set Frequency to **20 GHz** and Amplitude to **0 dBm**.

3. Select the **Out Mod** block.



4. Select the arrow button for **5-8**, then the right arrow for **PM1**.

   In the VXG Output Modulation block, PM1 and PM2 are not different kinds of phase modulation. They are simply two independent, cascaded phase-modulation stages applied at the RF output stage. PM1 is the first RF output phase-modulation path and PM2 is the second RF phase-modulated path, applied after PM1.

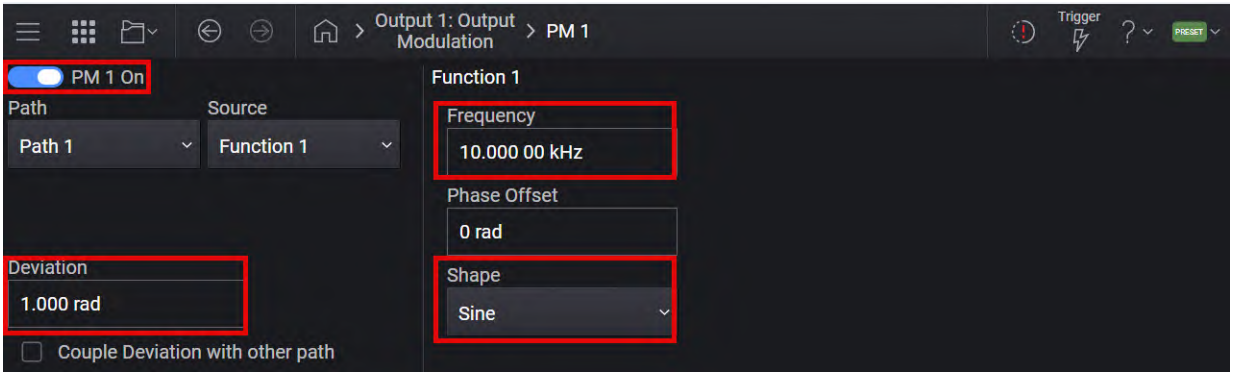
5. In the PM1 Setup:

   a. Set Shape to **Sine**.

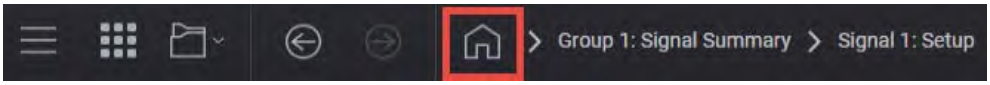
   b. Set Frequency to **10 kHz**.

   c. Set Deviation to **1 rad**.

   d. Set PM1 **On**.



6. Close the Vector Modulation Signal Setup by selecting the **Home** icon at the top of the display.



7. Set RF Out to **On** by selecting the numbered channel indicator switch.

   This enables the RF Out for the indicated channels, in this case Channel 1 if using a multi-channel VXG.

On the X-Series Signal Analyzer:

1. From the Menu Panel (on the top right of the display), select **Mode/Meas** > **Analog Demod** mode.

NOTE

If accessing the instrument via a Remote Desktop connection, select the Screen tab (at the top of the display) to open the Mode/Measurement/View Selector window.

2. Select **Mode Preset** to set Spectrum Analyzer mode to a known state.
3. From the Menu Panel (on the top right of the display), select **Mode/Meas** > **Analog Demod** mode > **PM** Measurement > **Quad** View.

NOTE

If accessing the instrument via a Remote Desktop connection, select the Screen tab (at the top of the display) to open the Mode/Measurement/View Selector window.

4. Select **Frequency** and set Center Frequency to **20 GHz**.

Analog Demod 1
PM
KEYSIGHT
Input: RF
Coupling: DC
Align: Off
Input Z: 50 Ω
RCal
Freq Ref: Ext (S)
Chan BW: 25.000 kHz
Atten: 10 dB
Preamp: Off
µW Path: Standard
Meas Time: 72.0 ms
LPF: Off
HPF: 20 Hz
BPF: Off
Center Freq: 20.000000000 GHz
Avg|Hold:>10/10
Trig: Free Run
1 RF Spectrum
Scale/Div 10.0 dB
Ref Value 0.00 dBm
Log
-10.0
-20.0
-30.0
-40.0
-50.0
-60.0
-70.0
-80.0
-90.0
Center 20.00000000 GHz
Span 75 kHz
Res BW 680 Hz
2 Demod Waveform
Scale/Div 1.000 rad
Ref Value 0.000 rad
Lin
4.00 rad
3.00 rad
2.00 rad
1.00 rad
0 rad
-1.00 rad
-2.00 rad
-3.00 rad
-4.00 rad
Start 0.000 s
Points 1001
Stop 2.000 ms
3 AF Spectrum
Scale/Div 10.0 dB
Ref Value 6.280 rad
Lin
1.99 rad
628 mrad
199 mrad
62.8 mrad
19.9 mrad
6.28 mrad
1.99 mrad
628 µrad
199 µrad
AF Start Freq 0 Hz
AF Stop Freq 20.000 kHz
AF Res BW 180 Hz
4 Metrics
PM Deviation
Average
Max Hold
Peak+
849.9 mrad
902.7 mrad
Peak-
-850.4 mrad
-907.3 mrad
(Pk-Pk)/2
850.1 mrad
905.0 mrad
RMS
646.3 mrad
646.5 mrad
Carrier Power
-8.331 dBm
SINAD
30.486 dB
Carrier Freq Err
2.068 mHz
SNR
30.486 dB
Mod Rate
10.0000 kHz
Dist/Total Vrms
3.110 %
THD
--- %

Generating a Pulse Train from the Output Modulation Block

Oscilloscope requirements:

- Minimum bandwidth of 2 GHz
- 50 Ω and 1 MΩ input impedance

NOTE Pulse Train requires Option 320 and Option PME/PMR.

A pulse train is a repeating sequence of RF pulses defined by timing parameters such as pulse width, repetition interval, and optionally varying pulse characteristics (amplitude, frequency, phase).

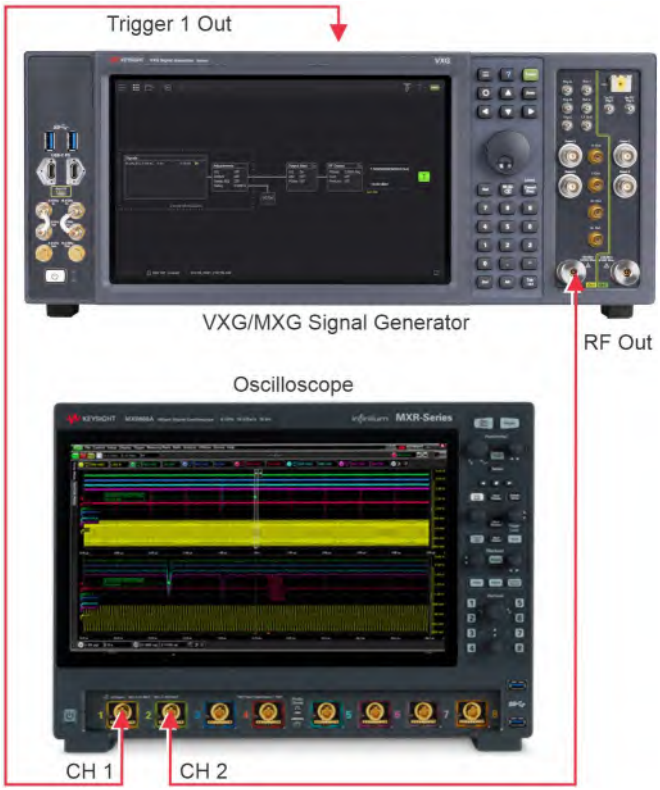
For this example, we will create an Identification Friend or Foe (IFF) pulse train. The IFF is an interrogation between aircraft involves a "challenge-response" mechanism where the interrogating aircraft transmits a specific pulse chain pattern (coded interrogation) on 1030 MHz, prompting the target aircraft's transponder to reply on 1090 MHz. This process allows for identification, bearing, and range determination, particularly crucial in combat scenarios.

Interrogation Signals (1030 MHz): The interrogator transmits a pair of pulses, designated \(P_{1}\) and \(P_{3}\), with a specific time spacing (spacing defines the interrogation mode, such as Mode 1, 2, 3/A, or C).

Pulse Space (Mode 1-3):

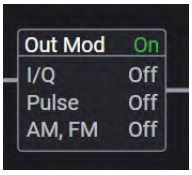
- **Mode1:** Two pulses 3 μs apart (±0.2 us)
- **Mode 2:** Two pulses 5 μs apart
- **Mode 3/A:** Two pulses 8 μs apart
- **Mode C:** Two pulses 21 μs apart.

Setup Diagram



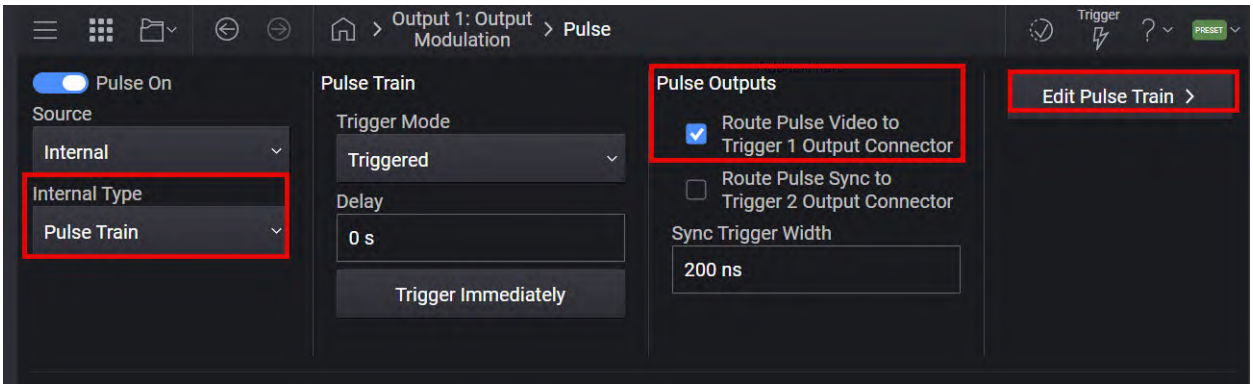
On the VXG:

1. Select **Preset** > **Preset** to set the instrument to a known state.
2. Set Frequency to **1.03 GHz** and Amplitude to **0 dBm**.
3. Select the **Out Mod** block.



4. Select the right arrow for **Pulse**.
5. In the Pulse Setup:
   a. Set Internal Type to **Pulse Train**.
   b. Select **Route Pulse Video to Trigger 1 Output Connector**.

c. Select **Edit Pulse Train**.



6. In the Pulse Train Editor, set:

   – Pulse 1 On Time to **3.0 µs** > Off Time to **1.2 µs** > Repeat to **1**, then add another Row.

   – Pulse 2 On Time to **3.0 µs** > Off Time to **3.2 µs** > Repeat to **1**, then add another Row.

   – Pulse 3 On Time to **5.0 µs** > Off Time to **1.2 µs** > Repeat to **1**, then add another Row.

   – Pulse 4 On Time to **5.0 µs** > Off Time to **3.2 µs** > Repeat to **1**, then add another Row.

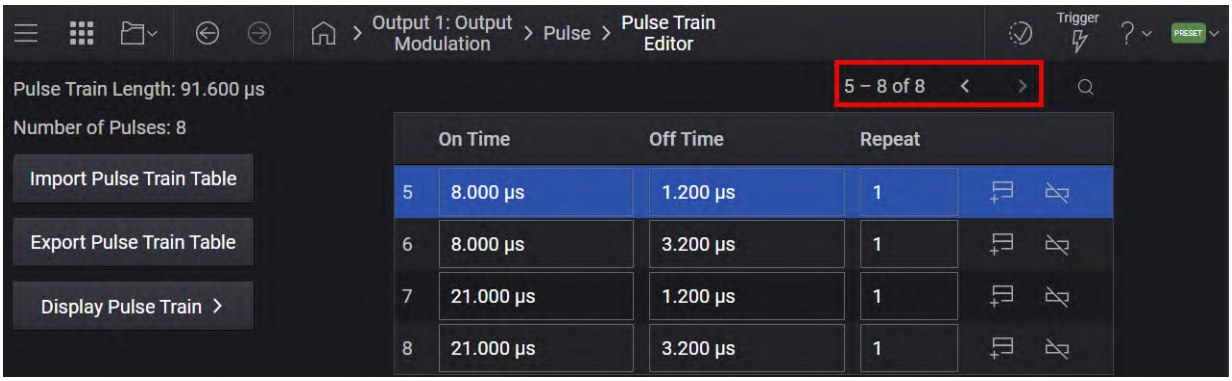
   Use the right arrow above to view rows 5 through 8.

   – Pulse 5 On Time to **8.0 µs** > Off Time to **1.2 µs** > Repeat to **1**, then add another Row.

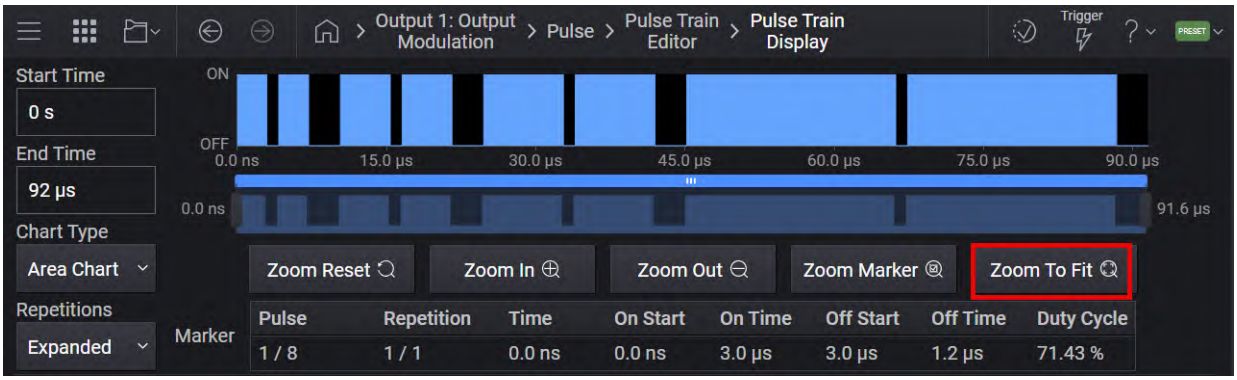
   – Pulse 6 On Time to **8.0 µs** > Off Time to **3.2 µs** > Repeat to **1**, then add another Row.

   – Pulse 7 On Time to **21.0 µs** > Off Time to **1.2 µs** > Repeat to **1**, then add another Row.

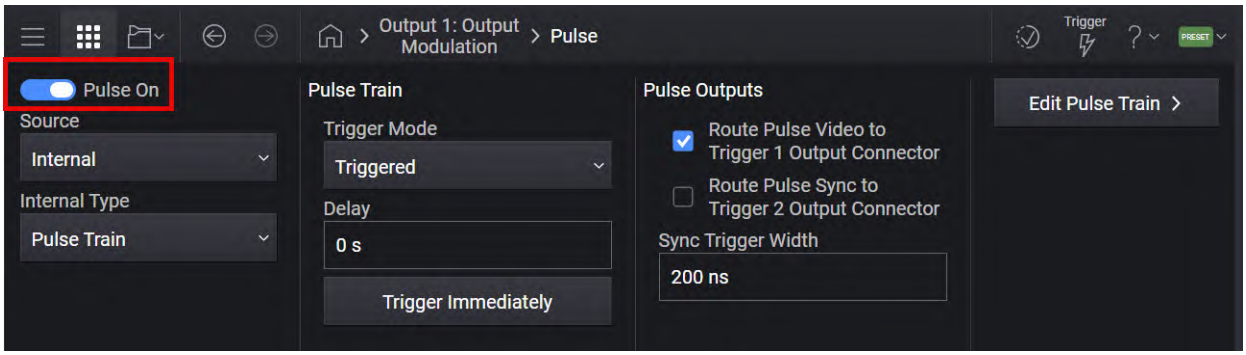
– Pulse 8 On Time to **21.0 µs** > Off Time to **3.2 µs** > Repeat to 1.



7. Select **Display Pulse Train** > **Zoom to Fit** to see the pulse train you created.



8. Press the **Back Arrow** twice to return to the main Pulse setup and set **Pulse On**.

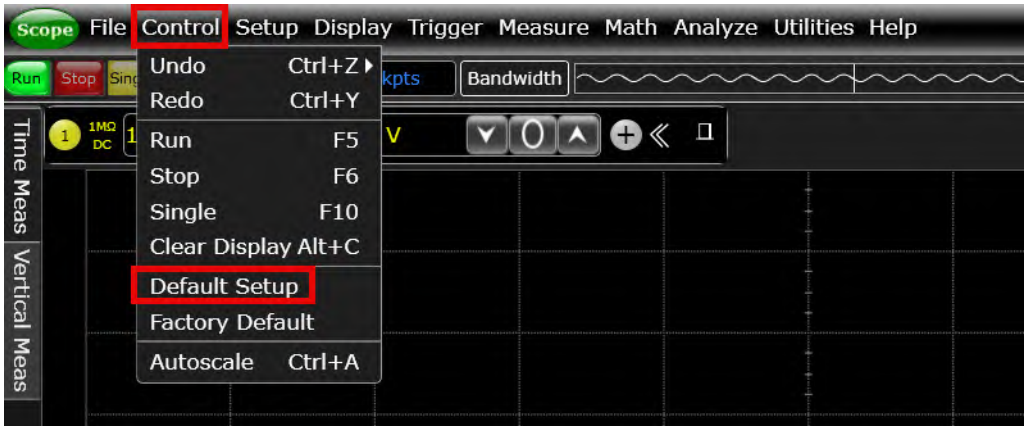


9. Set RF Out to **On** by selecting the numbered channel indicator switch.

This enables the RF Out for the indicated channels, in this case Channel 1 if using a multi-channel VXG.

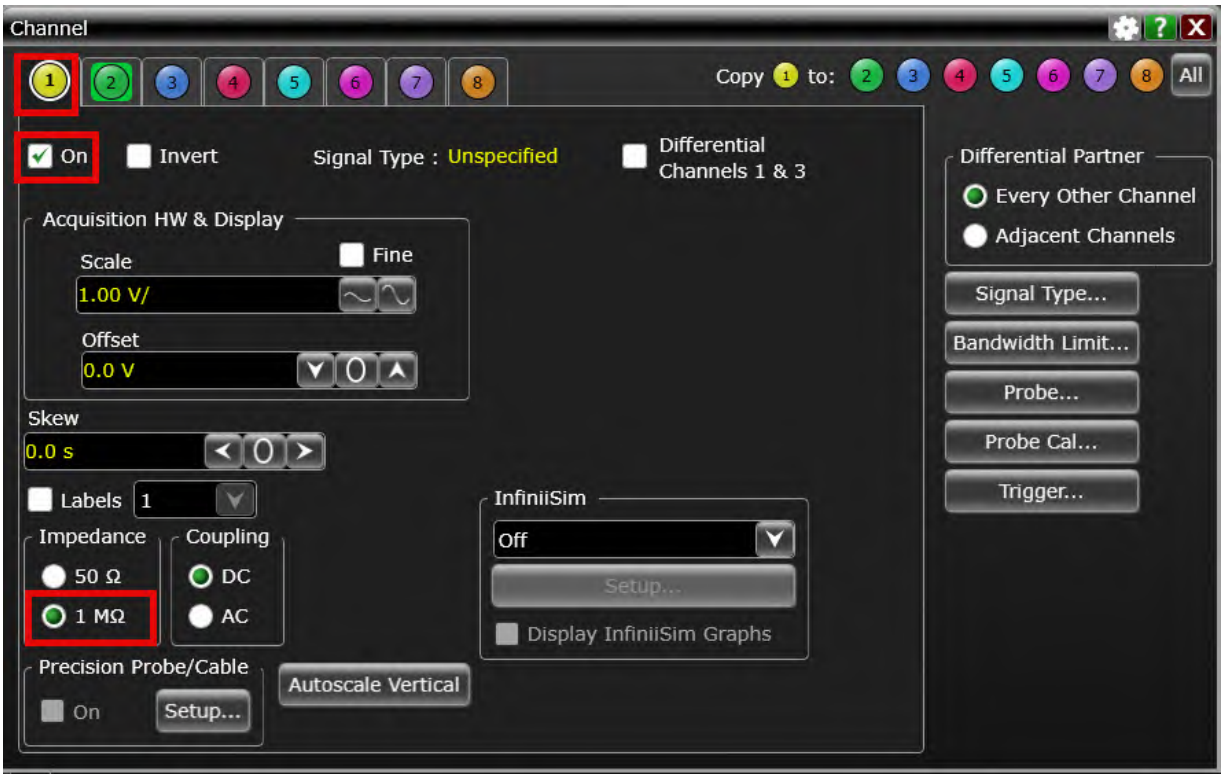
On the MXR Series Oscilloscope:

1. Select **Control** > **Default Setup** to set the scope to a known state.



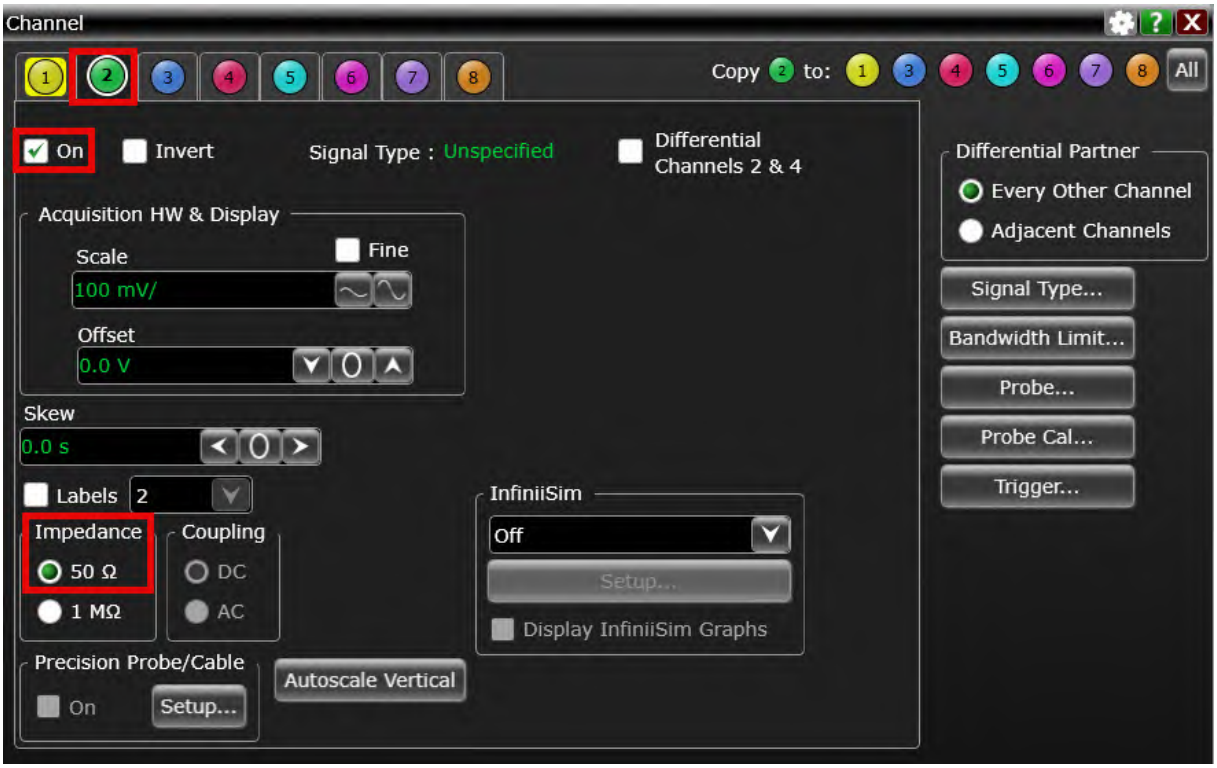
2. Select **Setup** > **Channel 1** and verify:
   - Impedance is set to **1M Ω**
   - Scale is set to **1V**
   - Offset is set to **0 V**.

– Channel 1 is on by default

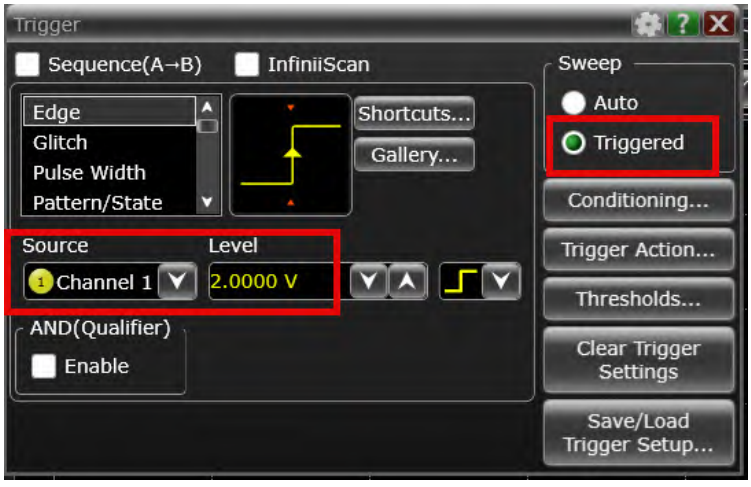


3. Select **Channel 2** and set:
   – Impedance to **50 Ω**
   – Scale to **100 mV**
   – Offset to **O V**
   – Turn **On** Channel 2

– **Close** the Setup dialog box.

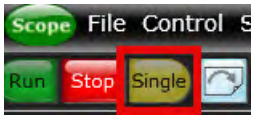


4. Select **Trigger** > **Setup** and set Sweep to **Triggered** > Level to **2 V** and Close the dialog box.

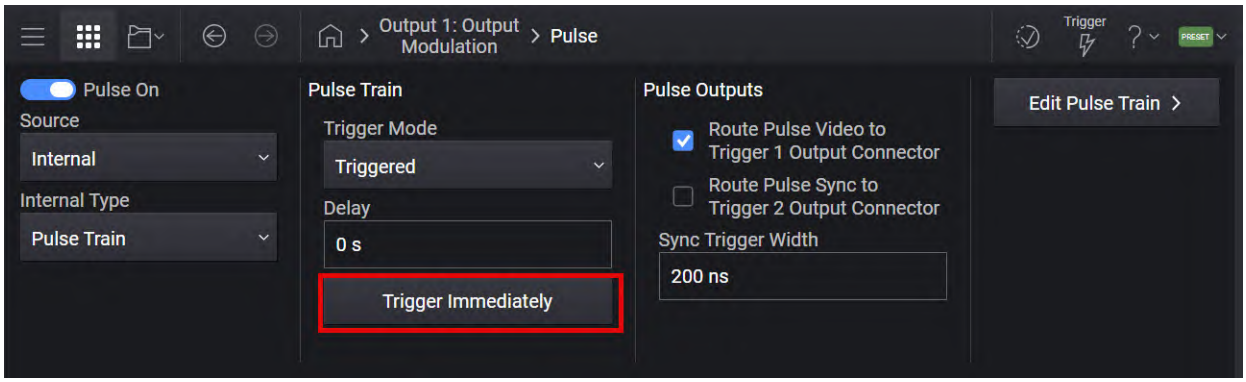


5. Set the Horizontal Scale to **20 µs/** and Horizontal Position to **40 µs**.

6. Select **Single Sweep**.



7. On the VXG, select T**rigger Immediately** and then view the results on the MXR.



Using Step Sweep Mode.

NOTE

Ensure the equipment and triggers are properly configured. Refer to "Equipment Setup" on page 13 and "Setting Up Triggers on the X-Series Signal Analyzer" on page 15.

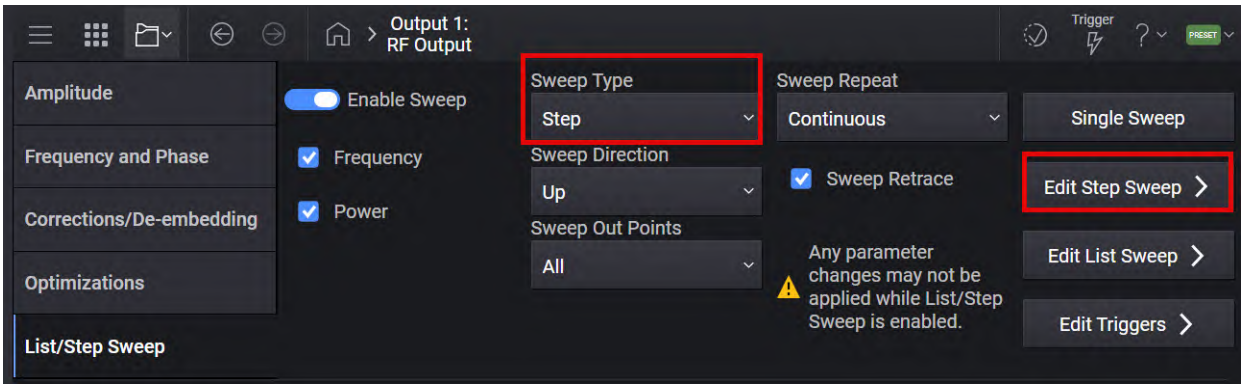
Step Sweep is a structured sweep derived from start/stop parameters. The VXG calculates the intermediate points automatically. Typical uses are:

- Frequency response scans
- Flatness testing
- Quick characterization where spacing is uniform
- Analyzer-synchronized sweeps with predictable timing

On the VXG:

1. Select **Preset** > **Preset** to set the instrument to a known state.
2. Select the **RF Output** block > **List/Step Sweep** tab.

3. Set the Sweep Type to **Step** and select **Edit Step Sweep**.



4. In the Step Sweep set:

a. Frequency Start to **27.995 GHz**

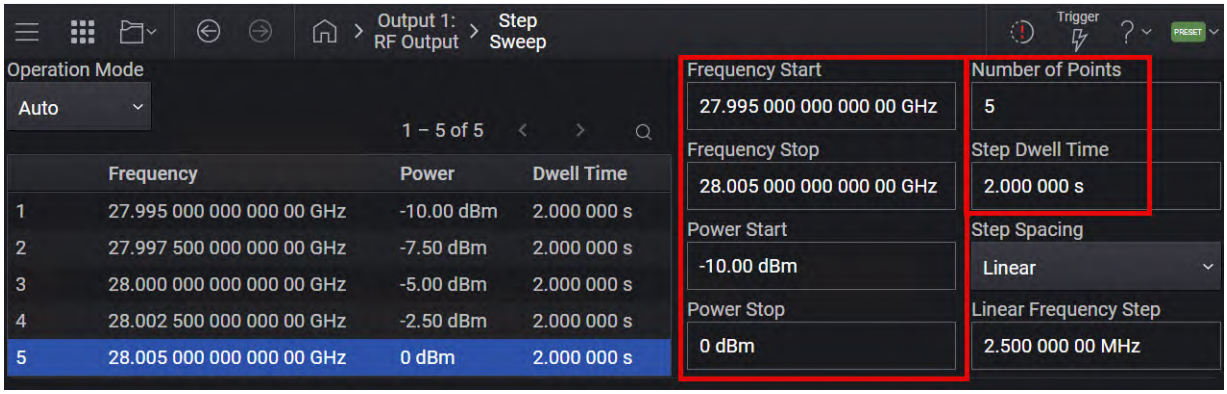
b. Frequency Stop to **28.005 GHz**

c. Start Power to **-10 dBm**

d. Stop Power to **0 dBm**.

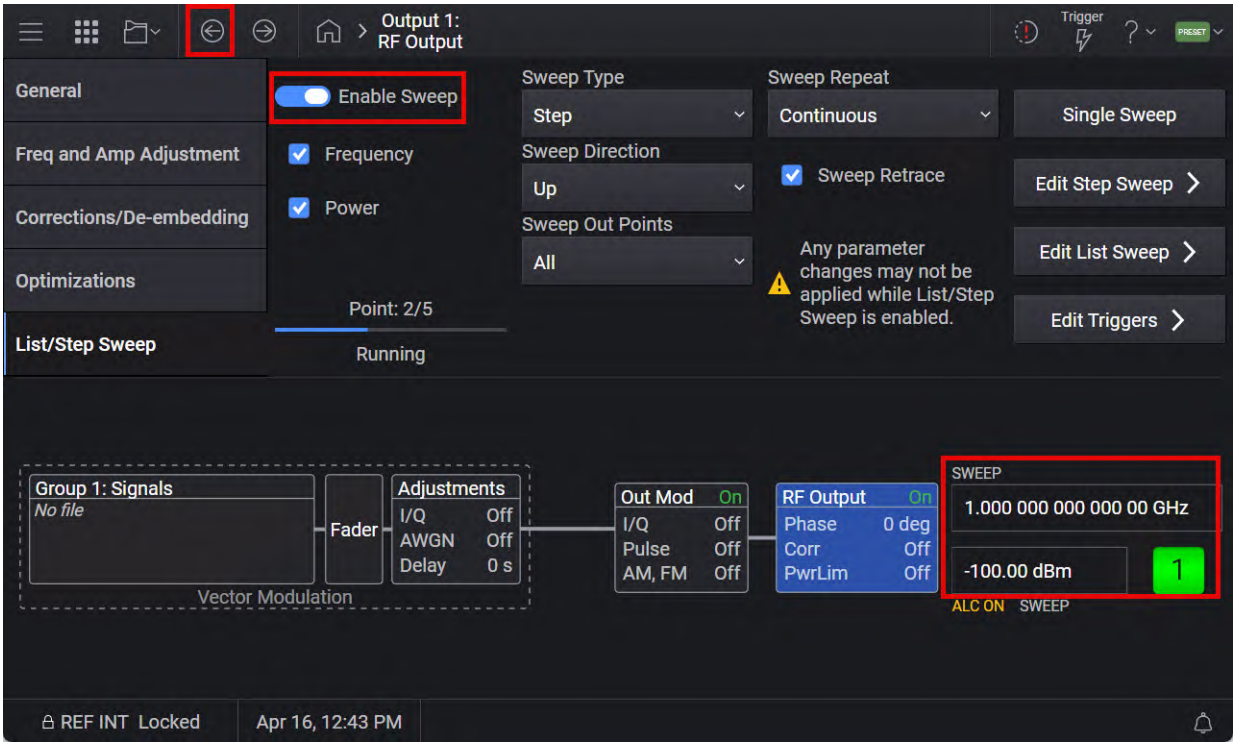
e. Number of points to **5**

f. Step Dwell Time to 2



5. Select the **Back** arrow at the top of the display and select **Enable Sweep**.
6. Set RF Out to **On**.

Notice that the bottom right block now says Sweep instead of Frequency. This is because we are in List/Step Sweep mode.



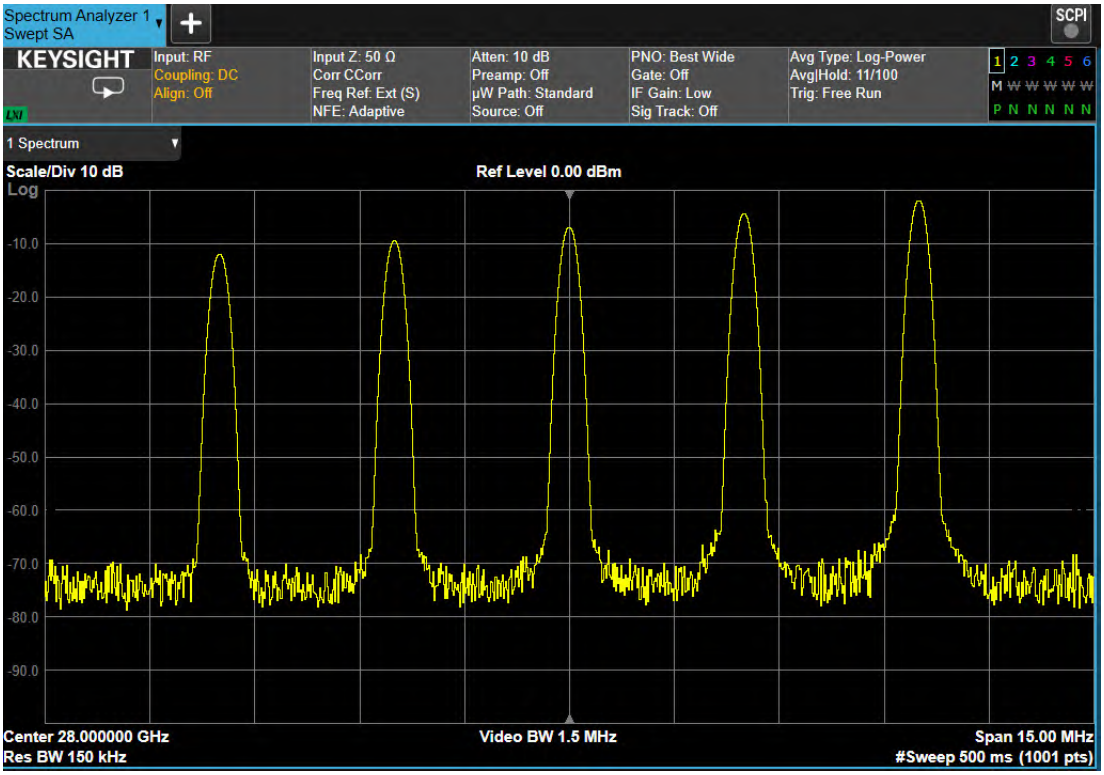
On the X-Series Signal Analyzer:

1. From the Menu Panel (on the top right of the display), select **Mode/Meas** > **Spectrum Analyzer** mode.

NOTE

If accessing the instrument via a Remote Desktop connection, select the Screen tab (at the top of the display) to open the Mode/Measurement/View Selector window.

2. Select **Mode Preset** to set Spectrum Analyzer mode to a known state.
3. Select **Frequency** and set Center Frequency to **28 GHz** and Span to **15 MHz**.
4. Select **Trace** > **Max Hold** to view the results of the signals at different frequencies and power levels.


Spectrum Analyzer 1
Swept SA
SCPI
KEYSIGHT
Input: RF
Coupling: DC
Align: Off
Input Z: 50 Ω
Corr CCorr
Freq Ref: Ext (S)
NFE: Adaptive
Atten: 10 dB
Preamp: Off
µW Path: Standard
Source: Off
PNO: Best Wide
Gate: Off
IF Gain: Low
Sig Track: Off
Avg Type: Log-Power
Avg|Hold: 11/100
Trig: Free Run
1 2 3 4 5 6
M W W W W W
P N N N N N
1 Spectrum
Scale/Div 10 dB
Ref Level 0.00 dBm
Log
-10.0
-20.0
-30.0
-40.0
-50.0
-60.0
-70.0
-80.0
-90.0
Center 28.000000 GHz
Res BW 150 kHz
Video BW 1.5 MHz
Span 15.00 MHz
#Sweep 500 ms (1001 pts)

Using List Sweep Mode.

NOTE

Ensure the equipment and triggers are properly configured. Refer to "Equipment Setup" on page 13 and "Setting Up Triggers on the X-Series Signal Analyzer" on page 15.

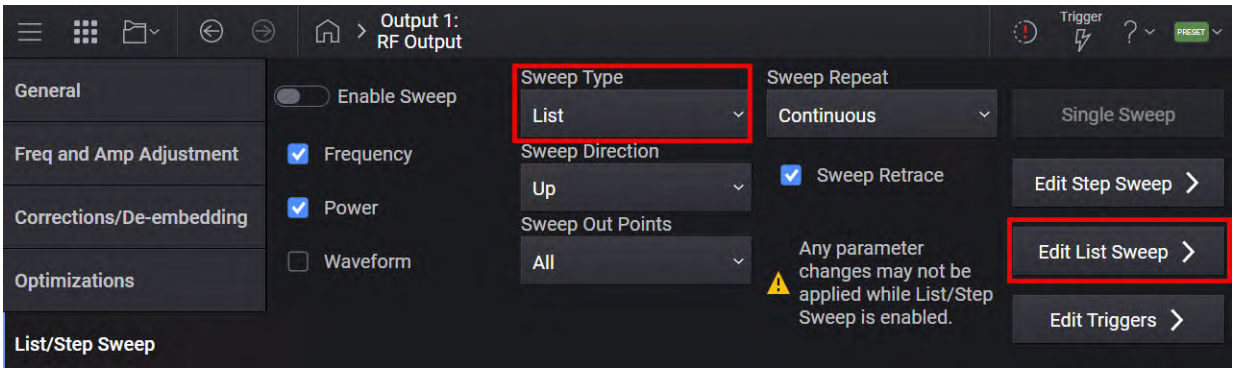
A list sweep lets you step through explicit frequency and/or power points (not linearly spaced) with independent dwell time per point. This is different from a start/stop step sweep. Typical uses are:

- Calibration at discrete frequencies
- PA characterization at selected operating points
- Testing band edges only
- Matching legacy test sequences

On the VXG:

1. Select **Preset** > **Preset** to set the instrument to a known state.
2. Select the **RF Output** block > **List/Step Sweep** tab.

3. Set the Sweep Type to **List** and select **Edit List Sweep**.



4. In the List/Sweep for setup for Row 1

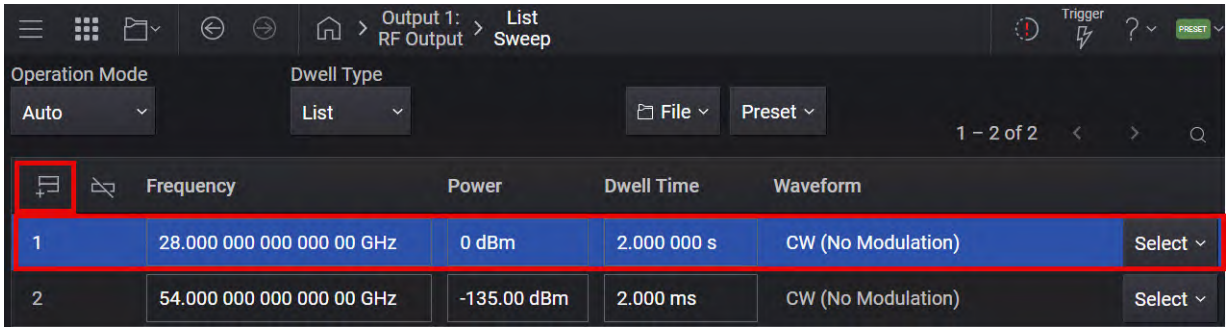
a. Set Frequency to **28 GHz**.

b. Set Power to **0 dBm**.

c. Set Dwell time to **2 seconds**, so we can see the effects of Sweep mode on the analyzer.

d. Choose the **Select** dropdown and choose **CW**.

e. Select the **Add** row icon to add another row.



5. In the List/Sweep for setup for Row 2

a. Set Frequency to **28 GHz**.

b. Set Power to **0 dBm**.

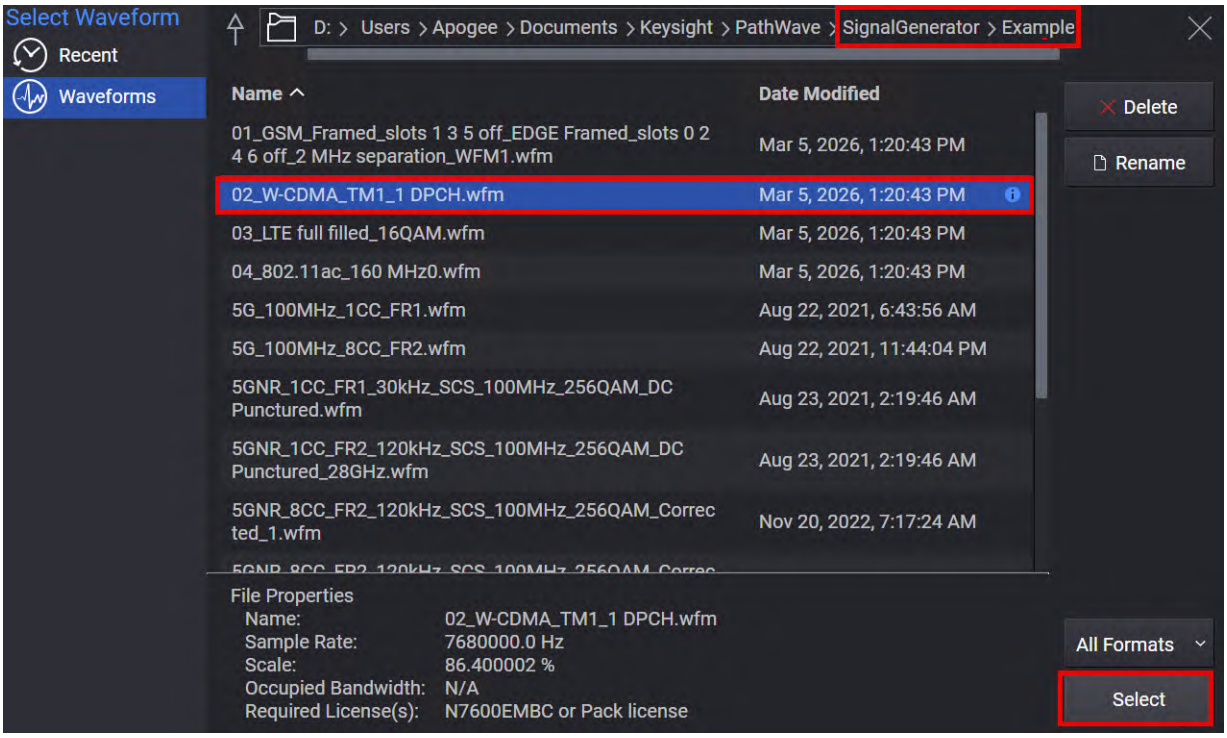
c. Set Dwell time to **2 seconds**,.

d. Choose the **Select** dropdown and choose **Waveform**.

e. Navigate to:

```
D:\Users\Instrument\Documents\Keysight\PathWave
\SignalGenerator\Examples
```

and choose
`02_W-CDMA_TM1_1 DPCH.wfm` > **Select**.



f. Select the **Add** row icon to add another row.

6. In the List/Sweep for setup for Row 3

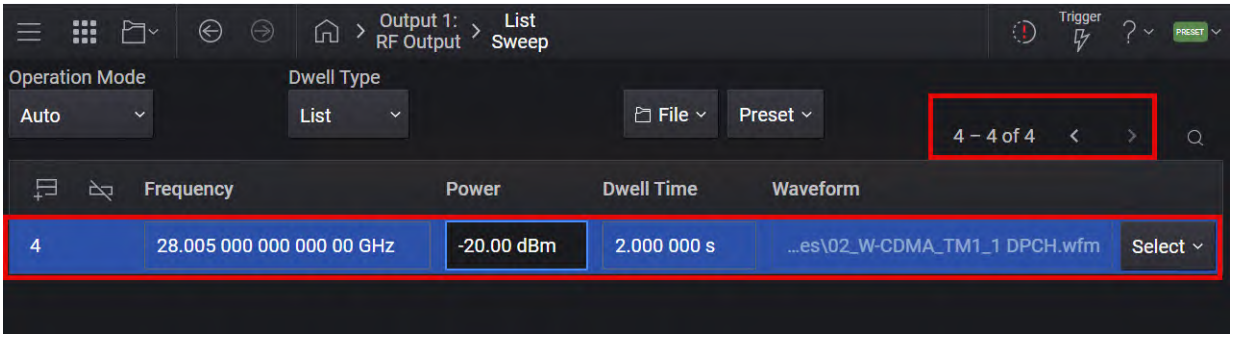
a. Set Frequency to **28.005 GHz**.

b. Set Power to **-20 dBm**.

c. Set Dwell time to **2 seconds**.

d. Choose the **Select** dropdown and choose **CW**.

e. Select the **Add** row icon to add another row. Note that rows 4 and greater will be added to a second page.



7. In the List/Sweep for setup for Row 4

   a. Set Frequency to **28.005 GHz**.

   b. Set Power to **-20 dBm**.

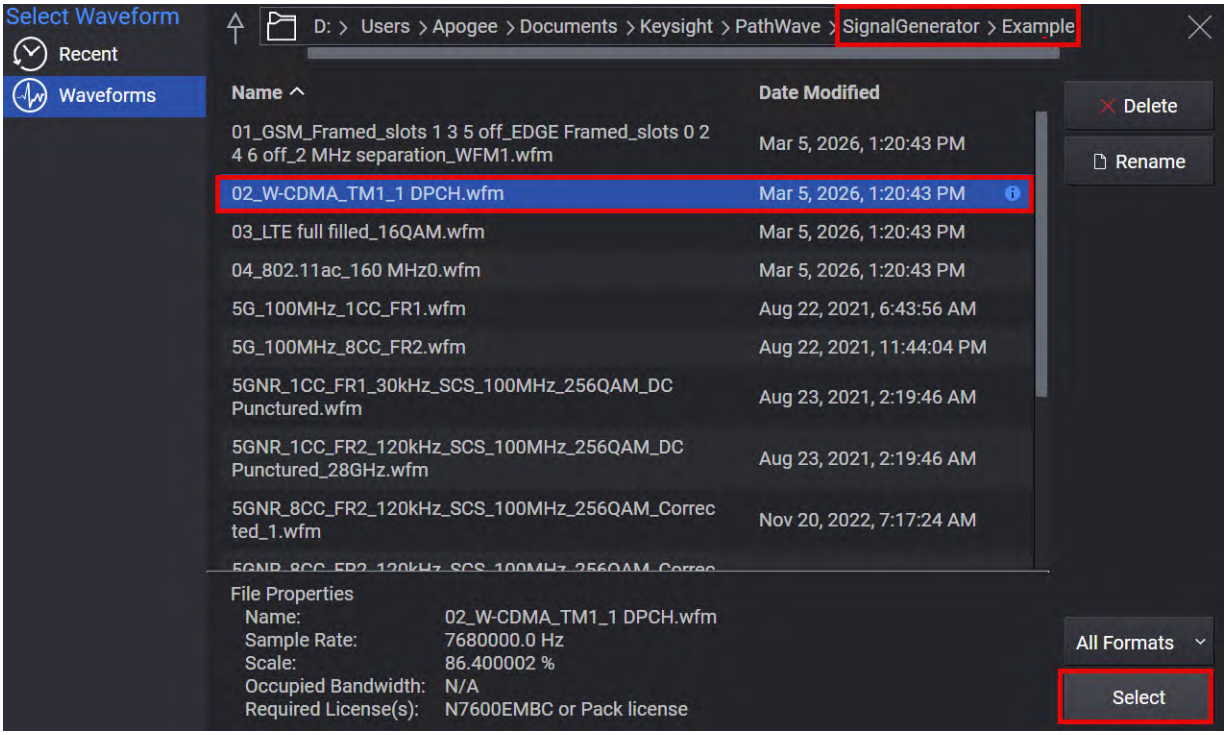
   c. Set Dwell time to **2 seconds**.

   d. Choose the **Select** dropdown and choose **Waveform**.

   e. Navigate to:

      `D:\Users\Instrument\Documents\Keysight\PathWave\SignalGenerator\Examples`

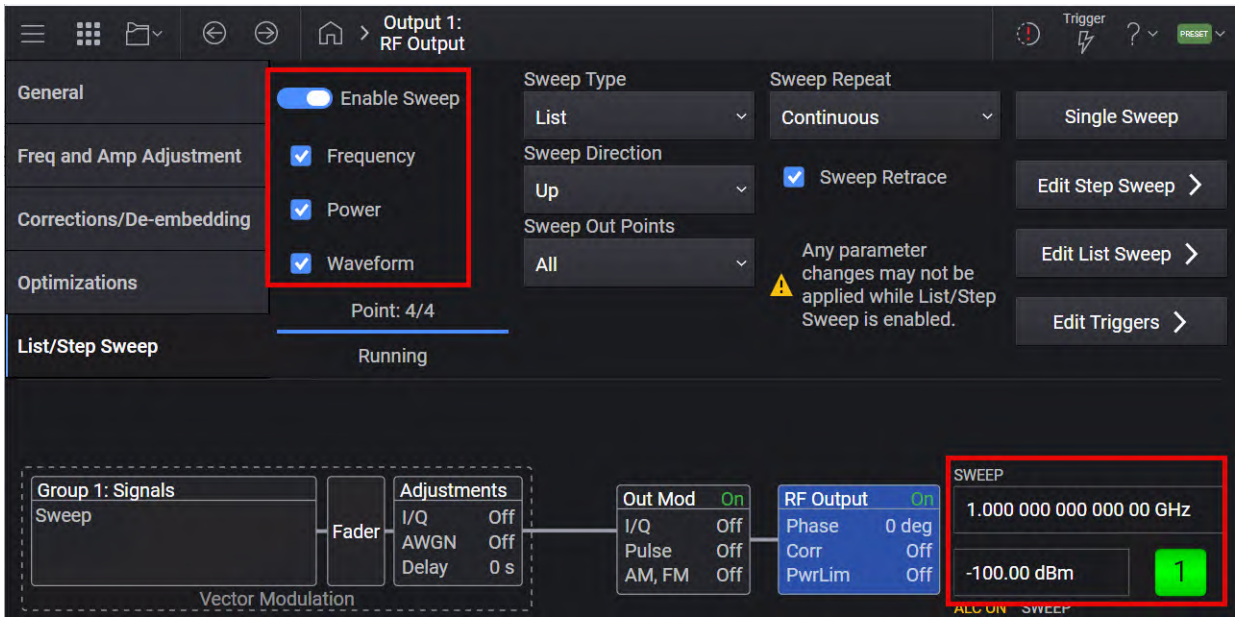
      and choose `02_W-CDMA_TM1_1 DPCH.wfm` > **Select**.



   f. Select the **Back** arrow at the top of the display and select **Waveform** and **Enable Sweep**.

   g. Set RF Out to **On**.

Notice that the bottom right block now says Sweep instead of Frequency. This is because we are in List/Step Sweep mode.



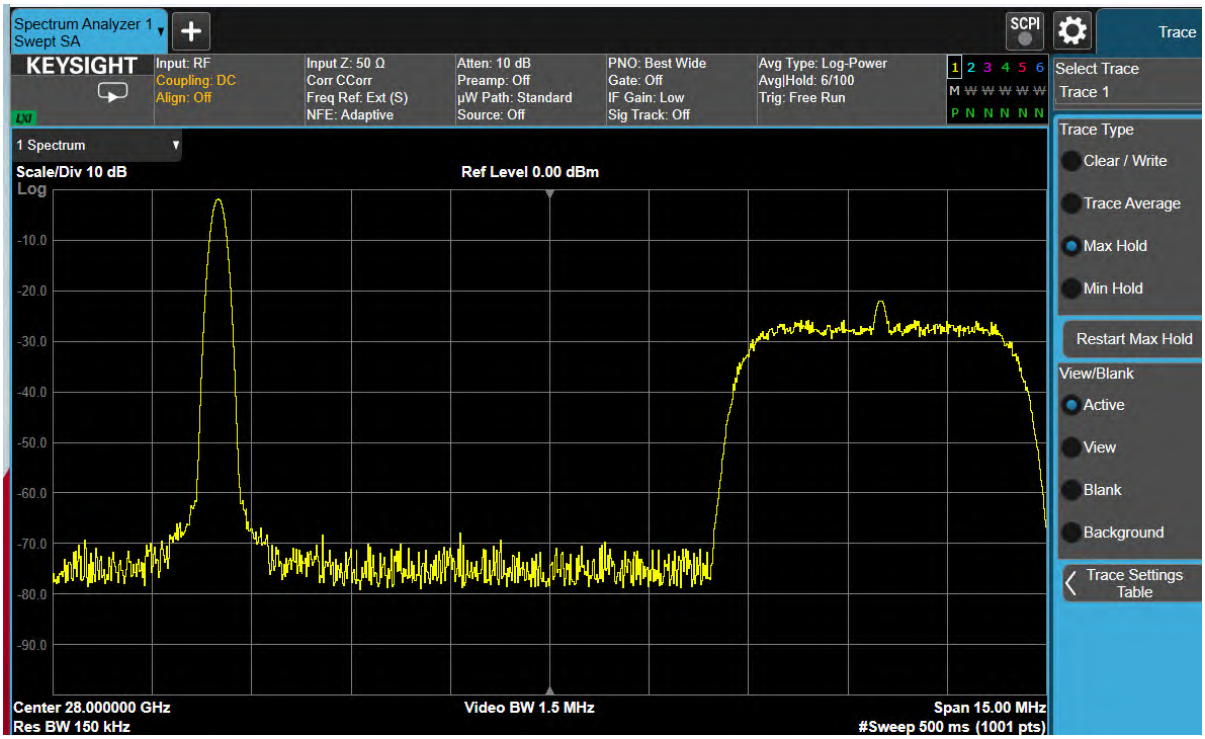
On the X-Series Signal Analyzer:

1. From the Menu Panel (on the top right of the display), select **Mode/Meas** > **Spectrum Analyzer** mode.

NOTE

If accessing the instrument via a Remote Desktop connection, select the Screen tab (at the top of the display) to open the Mode/Measurement/View Selector window.

2. Select **Mode Preset** to set Spectrum Analyzer mode to a known state.
3. Select **Frequency** and set Center Frequency to **28 GHz** and Span to **15 MHz**.
4. Select Sweep > Sweep Time to **500 ms**.
5. Select **Trace** > **Max Hold** to view the results of the two signals at different frequencies and power levels.


Spectrum Analyzer 1
Swept SA
KEYSIGHT
Input: RF
Coupling: DC
Align: Off
Input Z: 50 Ω
Corr CCorr
Freq Ref: Ext (S)
NFE: Adaptive
Atten: 10 dB
Preamp: Off
µW Path: Standard
Source: Off
PNO: Best Wide
Gate: Off
IF Gain: Low
Sig Track: Off
Avg Type: Log-Power
Avg|Hold: 6/100
Trig: Free Run
SCPI
Trace
Select Trace
Trace 1
Trace Type
Clear / Write
Trace Average
Max Hold
Min Hold
Restart Max Hold
View/Blank
Active
View
Blank
Background
Trace Settings Table
1 Spectrum
Scale/Div 10 dB
Log
Ref Level 0.00 dBm
-10.0
-20.0
-30.0
-40.0
-50.0
-60.0
-70.0
-80.0
-90.0
Center 28.000000 GHz
Res BW 150 kHz
Video BW 1.5 MHz
Span 15.00 MHz
#Sweep 500 ms (1001 pts)

Using the M9484C as a Multifunction Generator

Introduction

A function generator is a type of signal generator that is used to generate various waveforms with adjustable parameters to generate complex signals. Keysight's Multifunction Generator option 303 and UNT, first seen on the N5171B, includes a slew of function generators which can output complex signals. These signals can either be output directly to the LF port or routed to the modulators, ultimately output through the RF port. When using the function generators for modulation, the adjustable parameters include modulation type, frequency, amplitude, and phase. Often used in the fields of prototyping, development, testing, and calibration especially in the context of communication systems. An accurate function generator can help engineers accelerate the research and design process to bring high performance designs to the market quicker.

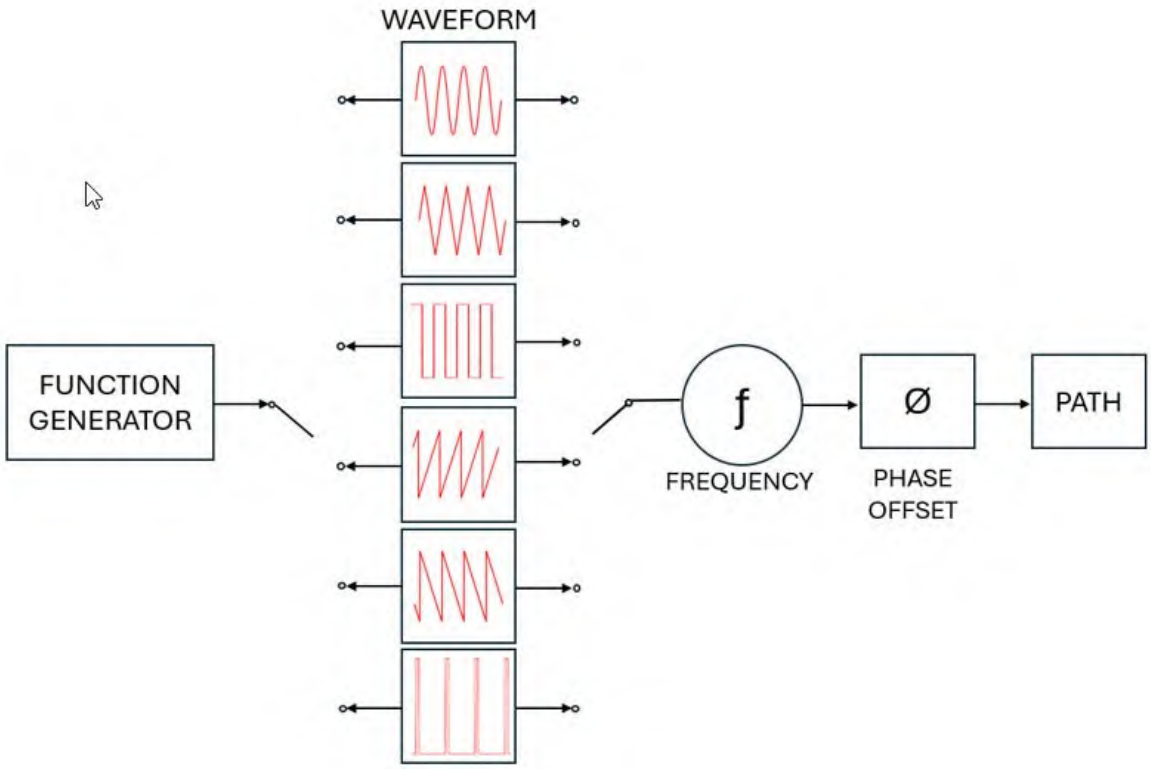
Supported Signal Generators with Option 303 and UNT

- N5171B
- N5172B
- N5173B
- N5181B
- N5182B
- N5183B
- N5185A
- N5186A
- M9484C

Theory of Operation

Multifunction generator N5186A Option 303 is a collection of six function generators that can be used independently or in parallel (up to five), ultimately being applied to the RF signal of choice or routed to the Low Frequency (LF) Output port. The LF Output port is designed to output low-frequency signals, making it ideal for directly using the function generators as well as measuring them. Ignoring the external function generator inputs (Ext 1 and Ext 2), there are six selectable function generators: Function Generator 1 (Func Gen 1), Function Generator 2 (Func Gen 2), Dual Function Generator (Dual Func Gen), Swept Function Generator (Swept Func Gen), Noise Generator 1 (Noise Gen 1), and Noise Generator 2 (Noise Gen 2).

Figure 1-1 Multifunction Generator Block Diagram

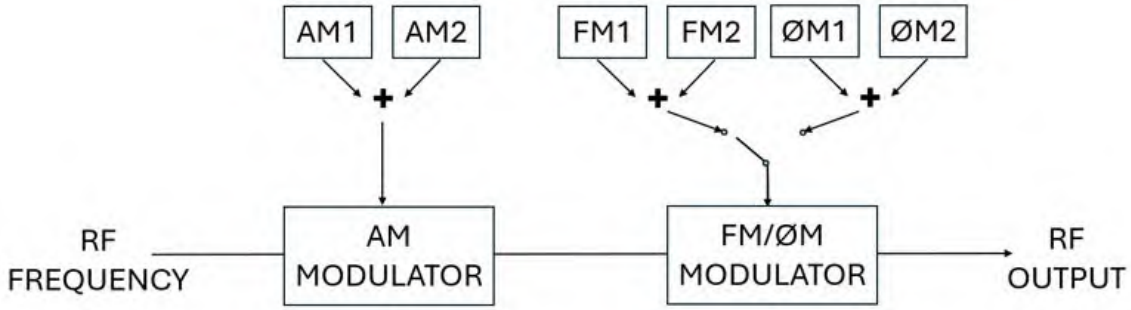


In Figure 1-1, each function generator (except Noise Gen 1 and Noise Gen 2) can access six periodic waveforms (sine, triangle, square, positive ramp, negative ramp, and pulse), with controllable frequency and phase offset.

The six periodic waveforms are considered the "Basic Waveforms". The Basic Waveforms can be made more complex by combining them using the Dual Function, demonstrated in “Dual Function” on page 85. The Basic Waveforms can also be swept using the Swept Function, demonstrated in “Swept Function” on page 91.

The output of the function generators can be routed to the Amplitude Modulator and the Frequency/Phase modulator. A function generator can be assigned to each of the paths (AM1, AM2, FM1, FM2, ØM1, ØM2) which are summed and applied to the AM and FM/ØM modulators. The modulators are then applied to the RF output signal.

Figure 1-2 Function Generator Paths



Using Multifunction Generator Option 303

Functions can be combined by summation and applied to modulations on the RF output. Modulation formats include AM, FM, and ØM. The LF port can be used to monitor the behavior of a single function generator that is being used by a modulator.

The multifunction generator's flexibility enables a wide range of configurations, catering to numerous testing applications. This adaptability ensures efficient and effective simulation of real-world scenarios for testing RF and LF designs. Ultimately, the multifunction generator enhances testing capabilities, allowing quicker problem solving.

Figure 1-3 Modulation Types

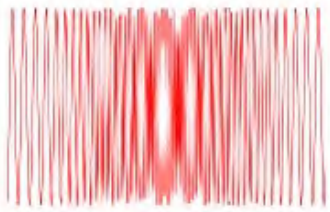
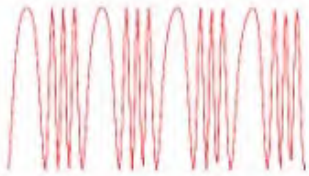
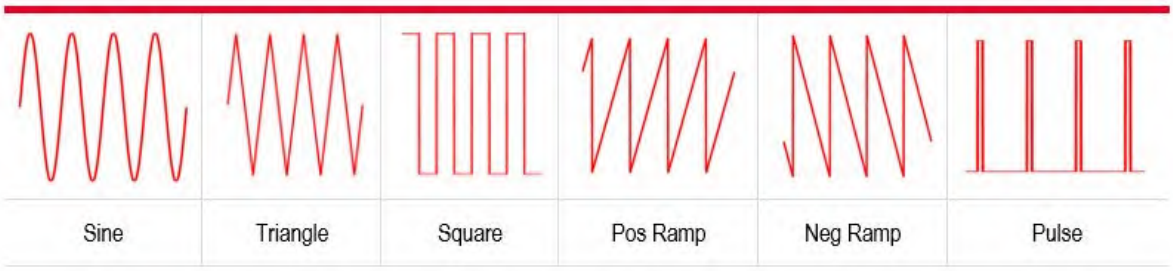
Modulation	Parameters	Description
AM:	• AM Type • AM Path • AM Depth • AM Source • AM Waveform • AM Rate • AM Phase Offset	The AM modulator allows the user to select a carrier frequency, message frequency, message shape, depth of modulation, modulation rate, and phase offset. Giving the user full freedom to generate the desired amplitude modulated signal.
FM: 	• FM Path • FM Dev • FM Source • FM Waveform • FM Rate • FM Phase Offset	The FM modulator allows the user to select a carrier frequency, deviation from carrier frequency, the modulation rate, the modulation waveform, and phase offset. Giving the user full freedom to generate their desired frequency modulated signal.
ØM: 	• ØM Path • ØM Dev • ØM Source • ØM Waveform • ØM Rate • ØM Phase Offset	The ØM modulator allows the user to select a carrier frequency, phase deviation, modulation rate, modulation waveform, and phase offset. Giving the user full freedom to generate their desired phase modulated signal.

Figure 1-4 LF Output

Function	Parameters	Description
DC:	• DC Offset	The DC function allows the user to choose a DC offset voltage between -1 V to 5 V in 1 mV increments for MXG and -5 V to 5 V for VXG
Monitor:	• Source	This Monitor function allows the user to monitor the behavior of a single function generator. This is for validating the behavior of the function generator when it is being used for modulation.
Function 1 & 2:	• Frequency • Phase Offset • Shape	These functions can take on any of the basic waveforms with a selectable frequency and phase offset.
Dual Function:	• Amp Percent 1 • Amp Percent 2 • Frequency 1 • Frequency 2 • Shape 1 • Shape 2 • Phase Offset 2	The Dual Function generator is useful for applications where two function generators with a known phase offset are needed.
Noise 1 & 2:	• Type	The noise source functions allow the user to select either Uniform or Gaussian Noise.
Swept Function:	• Sweep Waveform Shape • Sweep Start to Stop Time • Start Frequency • Stop Frequency • Sweep Shape	The Sweep function allows the user to configure a sawtooth or triangle sweeping signal, with selectable start and stop frequencies and sweep time.

Figure 1-5 Basic Waveforms

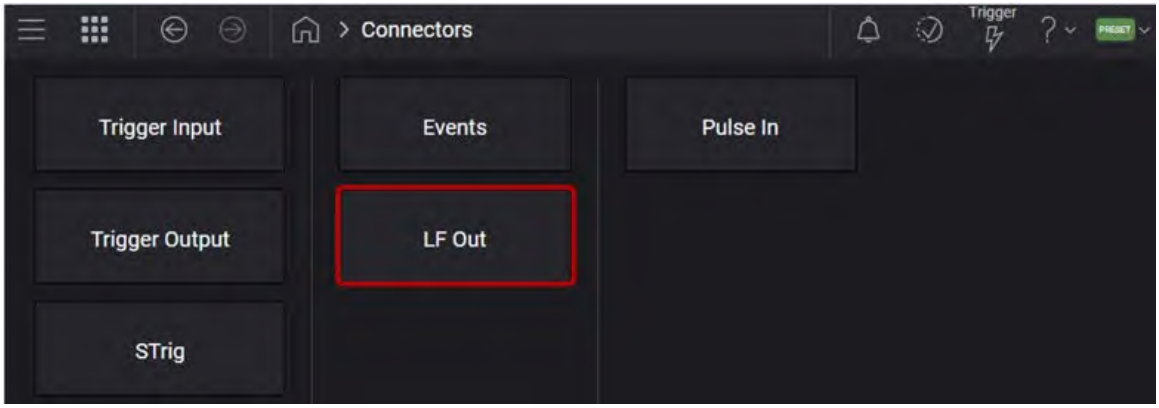


Multifunction Generator Demonstrations

An easy way to visualize the behavior of the internal function generators in the time domain is to use the LF Out port. By routing the function generator of choice to the LF Out, an Oscilloscope can capture its behavior.

1. Select **Preset** > **Preset** to set the instrument to a known state.
2. Select the **System** menu > **Connectors**.

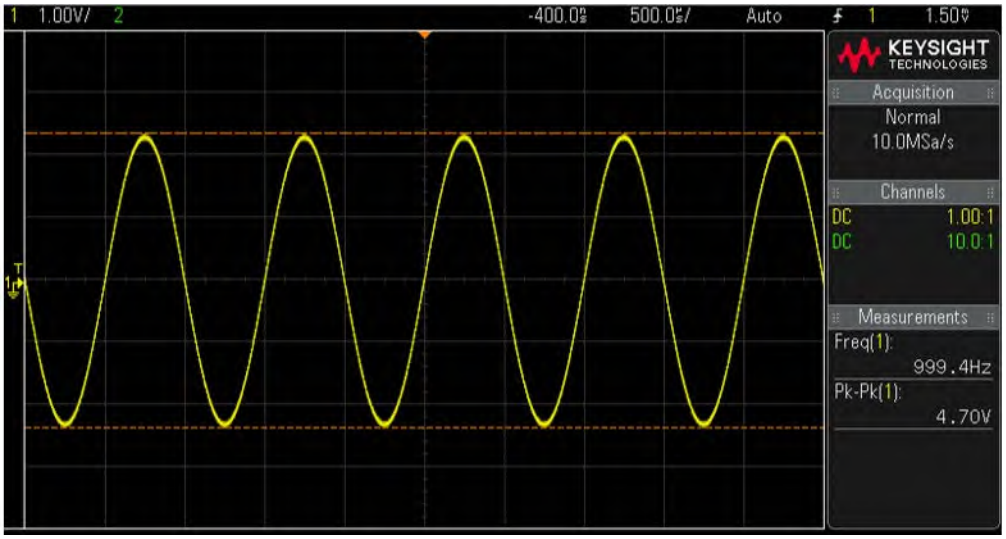
3. In the Connectors window, select **LF Out**.



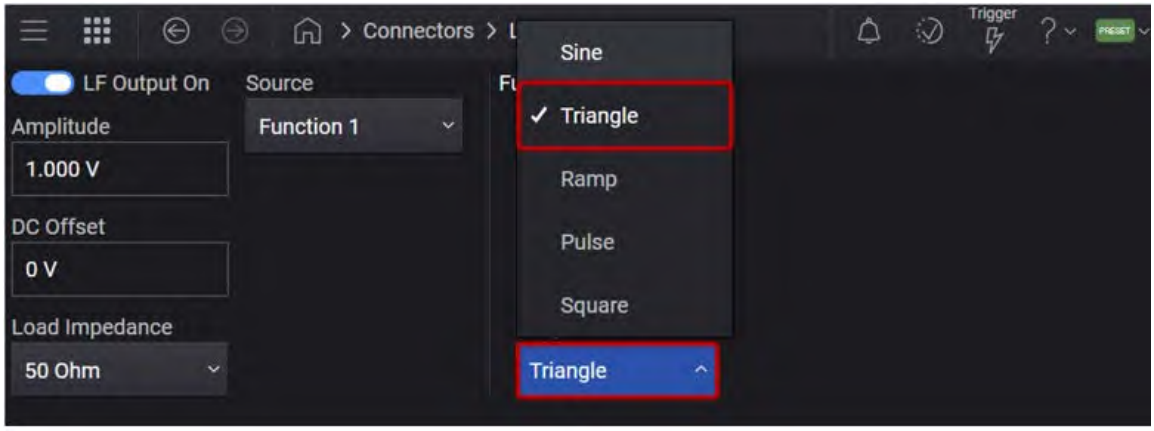
4. In the LF Out window, set:
   - Source to **Function 1**
   - Shape to **Sine**
   - Frequency to **1 kHz**
   - Amplitude to **1 V**

– LF Output **On**

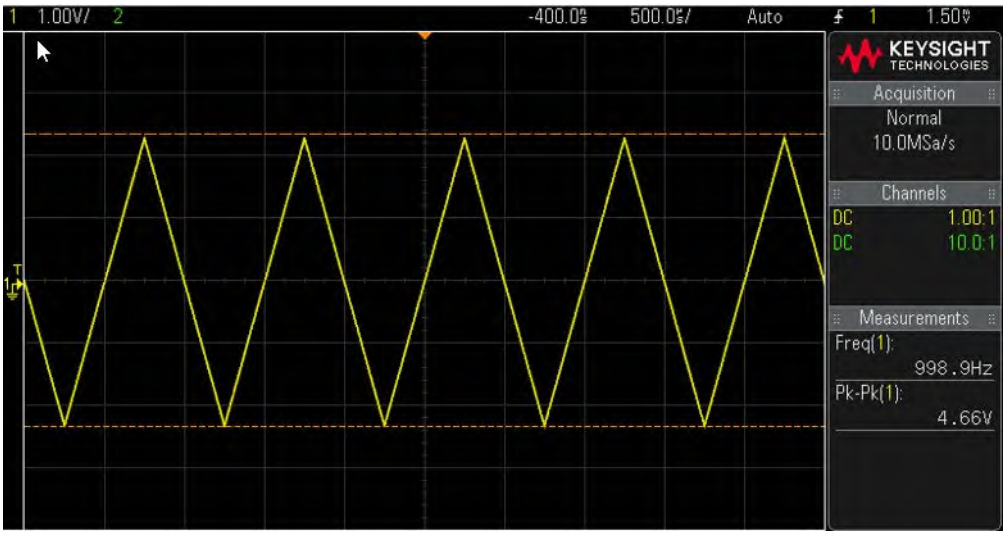
5. Capture the signal on the oscilloscope.



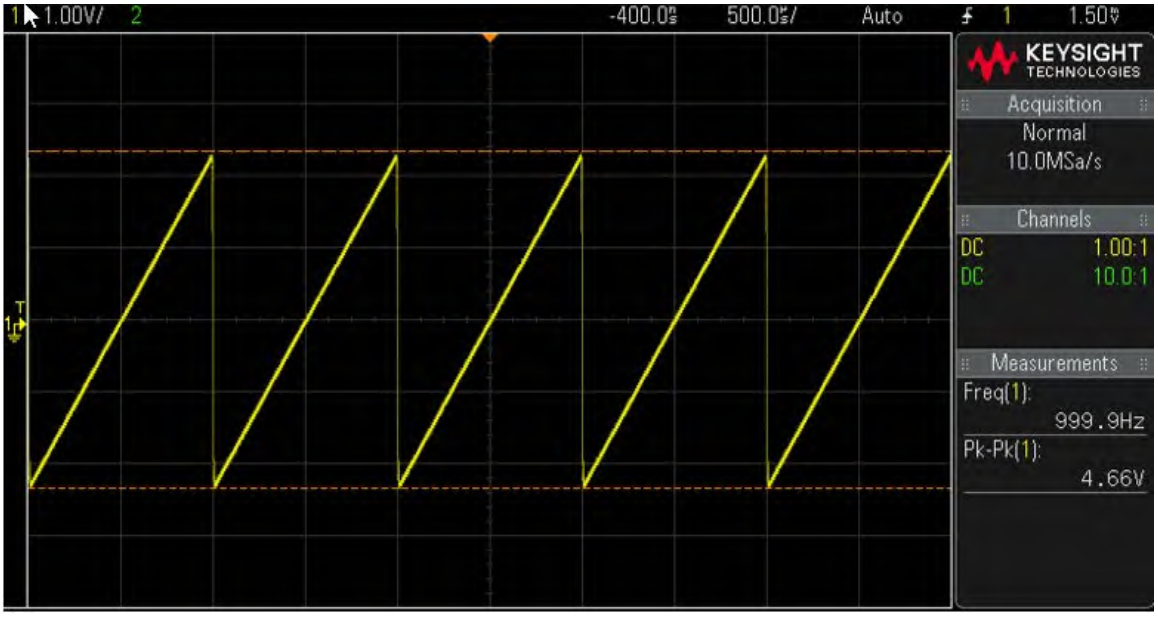
6. On the VXG, change the shape to **Triangle**.



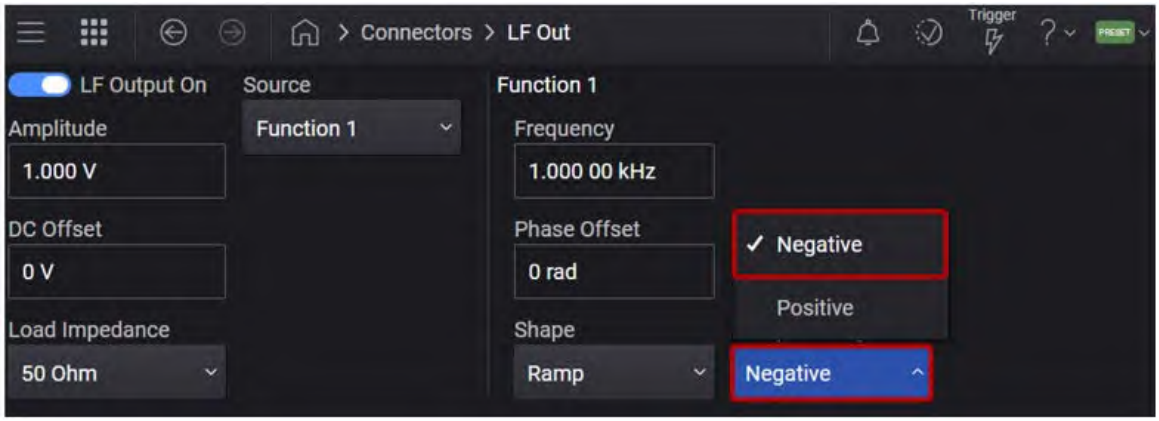
7. Capture the signal on the oscilloscope.



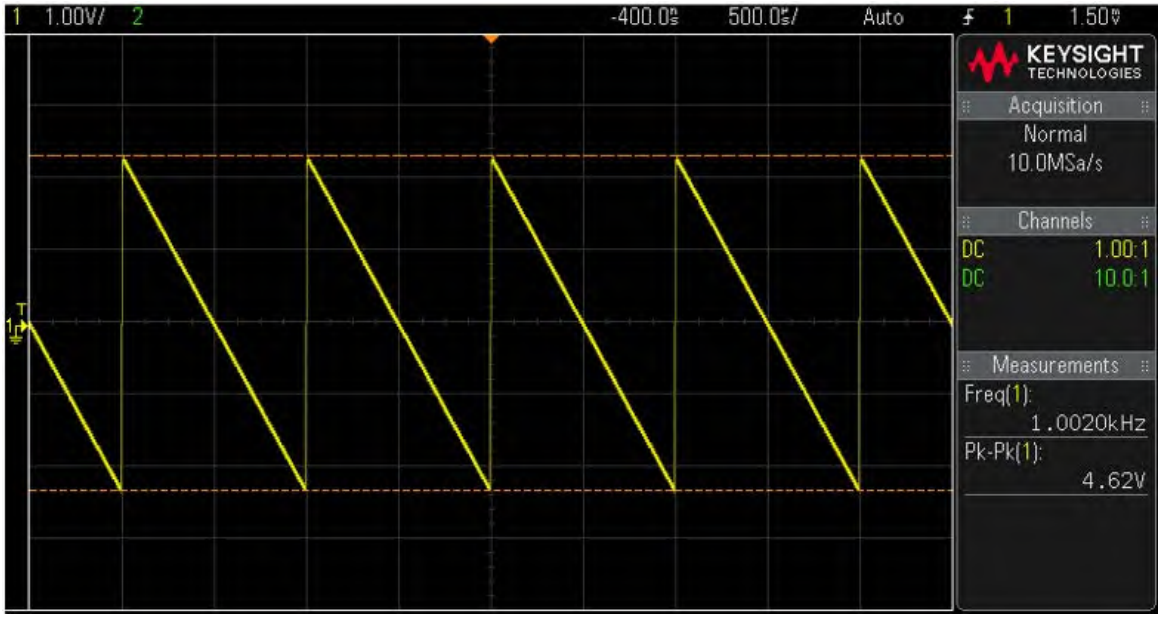
8. On the VXG, change the shape to **Ramp**.
9. Capture the signal on the oscilloscope.



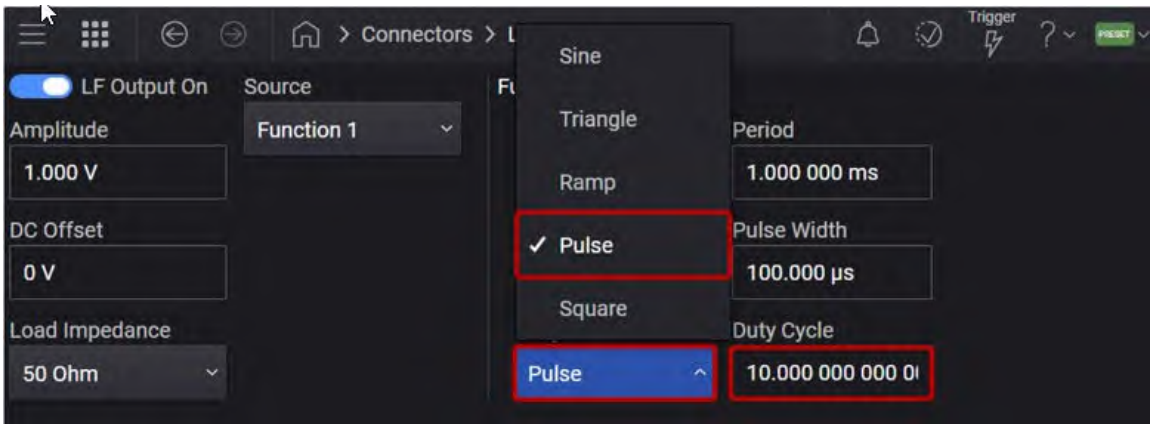
10. Set the Ramp Polarity to **Negative**.



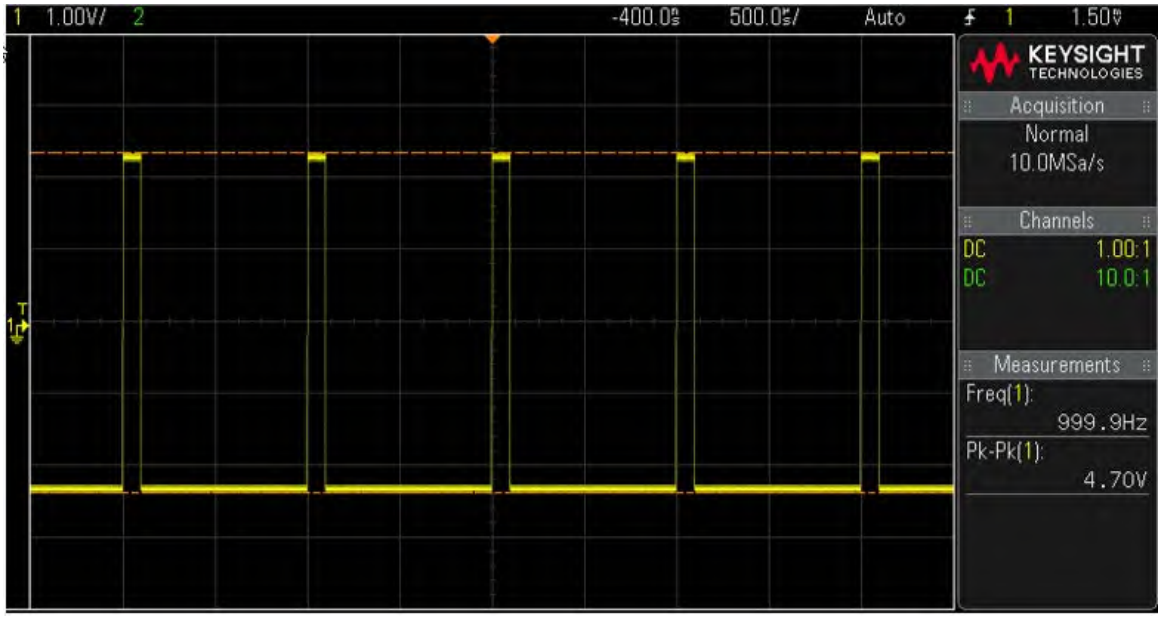
11. Capture the signal on the oscilloscope.



12. Set Shape to **Pulse** and set Duty Cycle to **10%**.

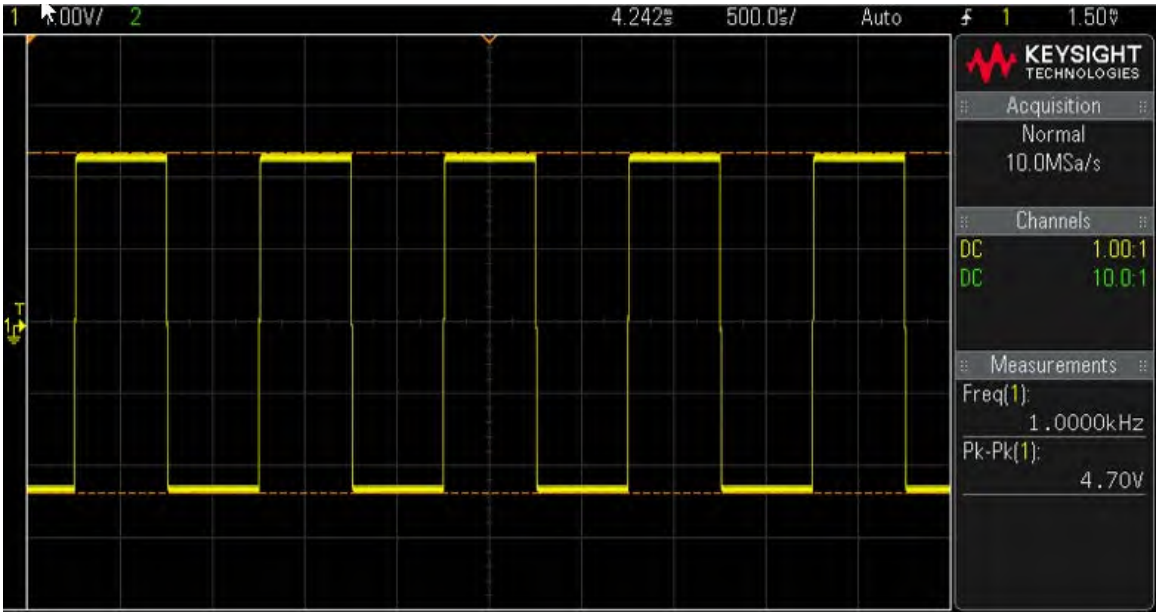


13. Capture the signal on the oscilloscope.



14. Set the Shape to **Square**.

15. Capture the signal on the oscilloscope.



Special Function Generators

The list of available function generators that are included with Option 303 are:

- Function 1
- Function 2
- Dual Function
- Noise 1
- Noise 2
- Swept function

In the “Multifunction Generator Demonstrations” on page 80, the characteristics of function generators 1 and 2 are captured. Function generators such as Dual Function, Noise 1 & 2, and Swept Function have unique functions that can also be visualized using an oscilloscope and signal analyzer.

Dual Function

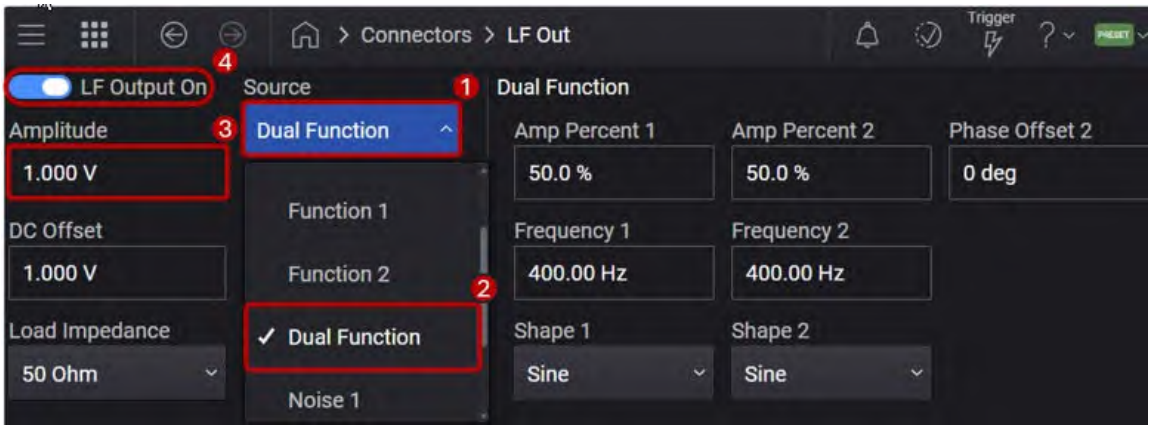
The Dual Function generator is similar to having both Function 1 and Function 2 operating at the same time, but with an adjustable phase offset. Providing precise control of phase difference between two functions.

An easy way to visualize the behavior of a dual function generator is by using an oscilloscope.

1. Select **Preset** > **Preset** to set the instrument to a known state.

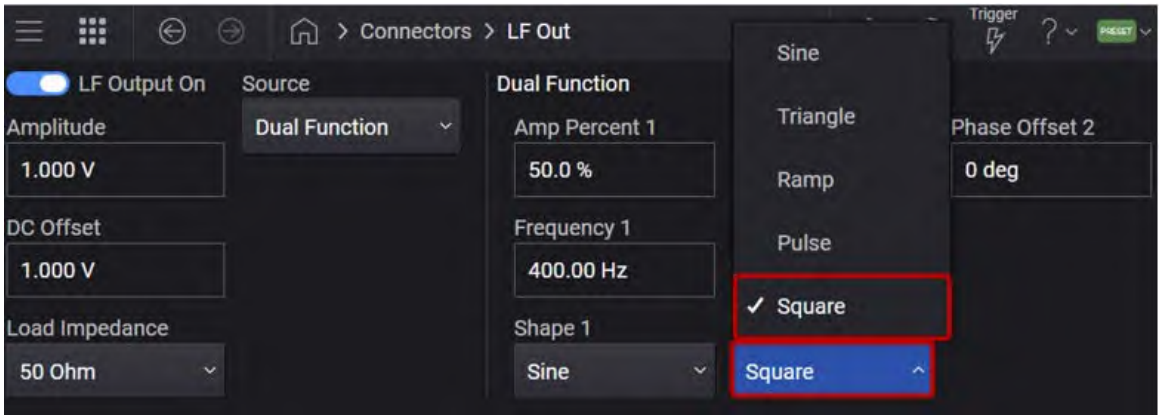
2. Select the **System** menu > **Connectors**.

3. In the Connectors window, select **LF Out**.
4. In the LF Out window, set:
   - Source to **Dual Function**
   - Amplitude to **1 V**
   - Toggle LF Output **On**



5. Set Shape 2 to **Square**.

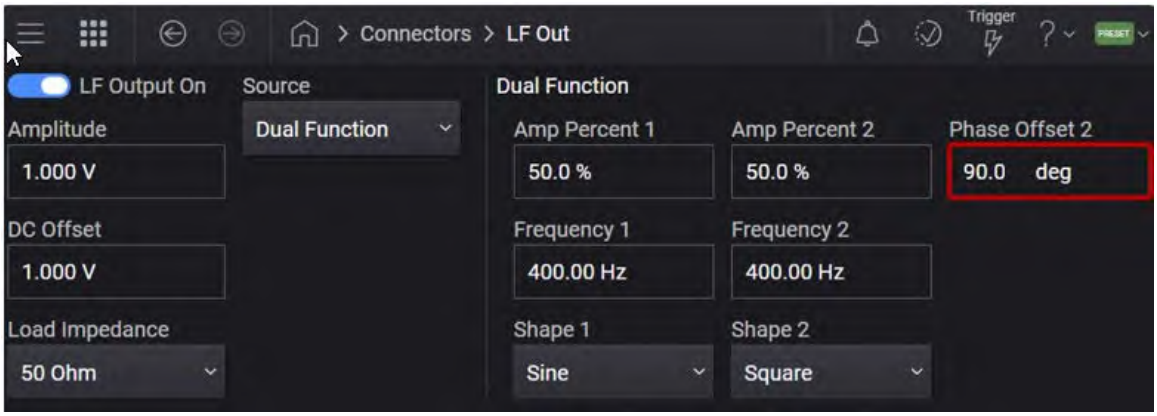
A simply way to demonstrate the phase relationship between functions is to set one to square and adjust the phase and monitor the output.



6. Capture the signal on an oscilloscope.

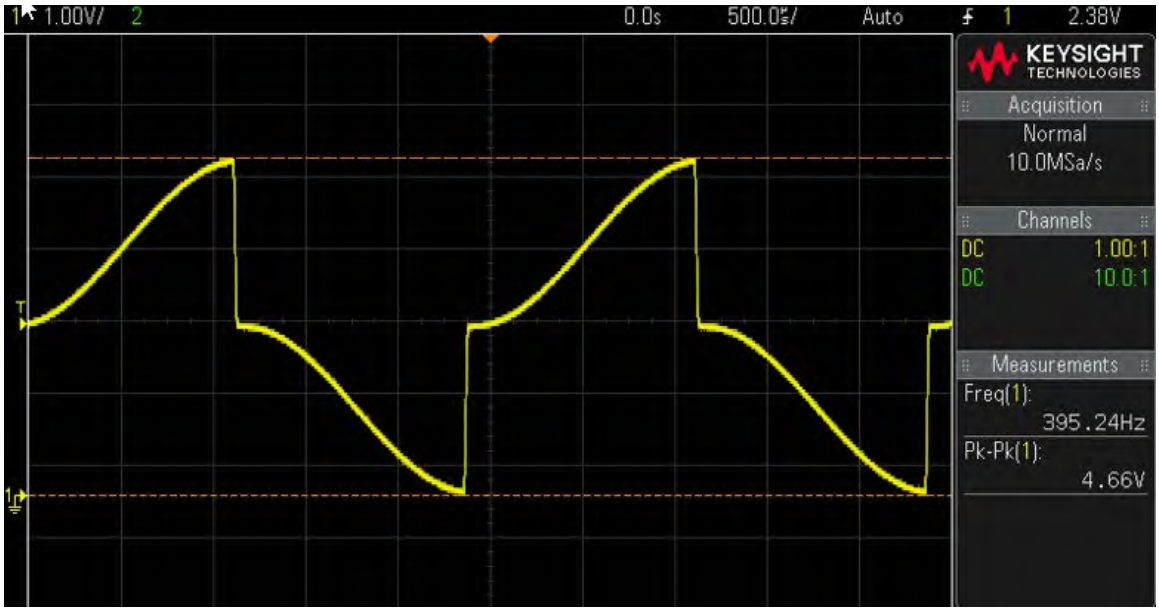
   You can see the Sine and Square waves are in phase.

7. In the LF Out window, change the Phase Offset 2 to **90 degrees**.

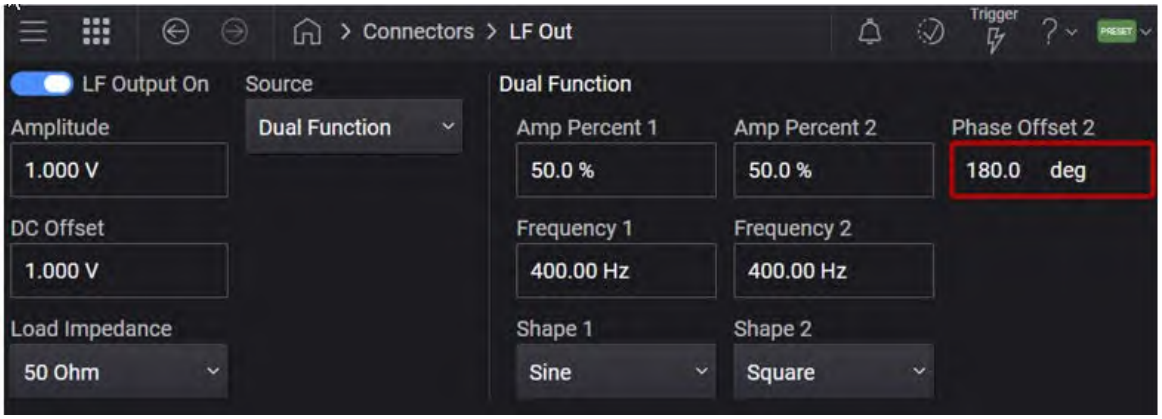


8. Capture the signal on the oscilloscope.

   You can see the Sine and Square waves are 90 degrees out of phase.

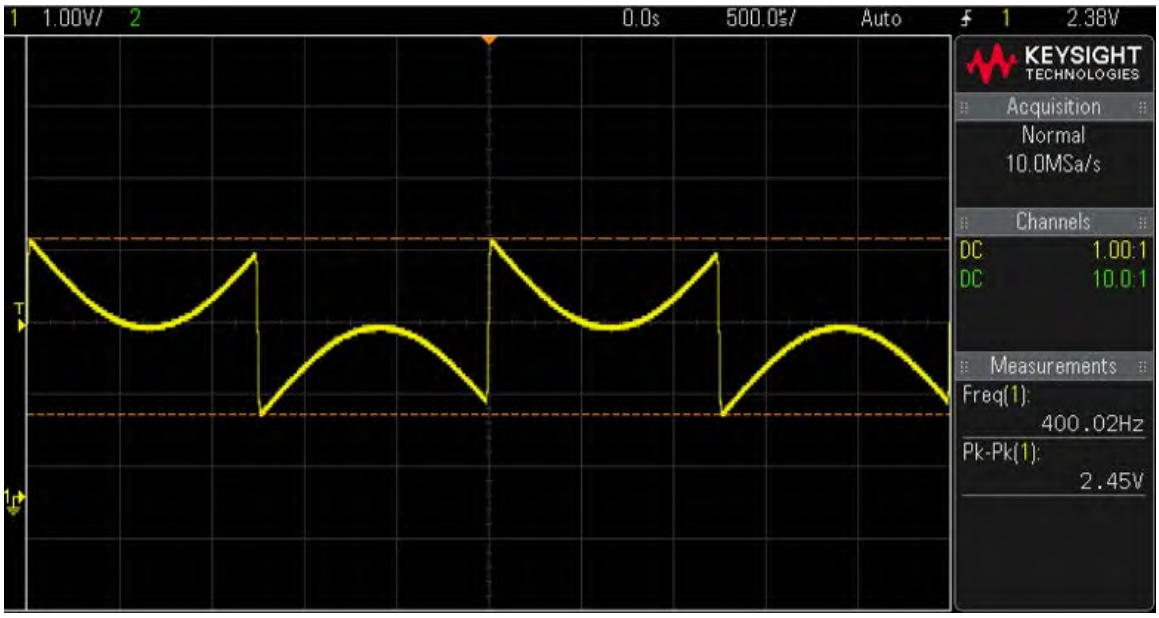


9. Change the Phase Offset 2 to **180 degrees**.



10. Capture the signal on the oscilloscope.

The Sine and Square waves are 180 degrees out of phase.



Noise Generator

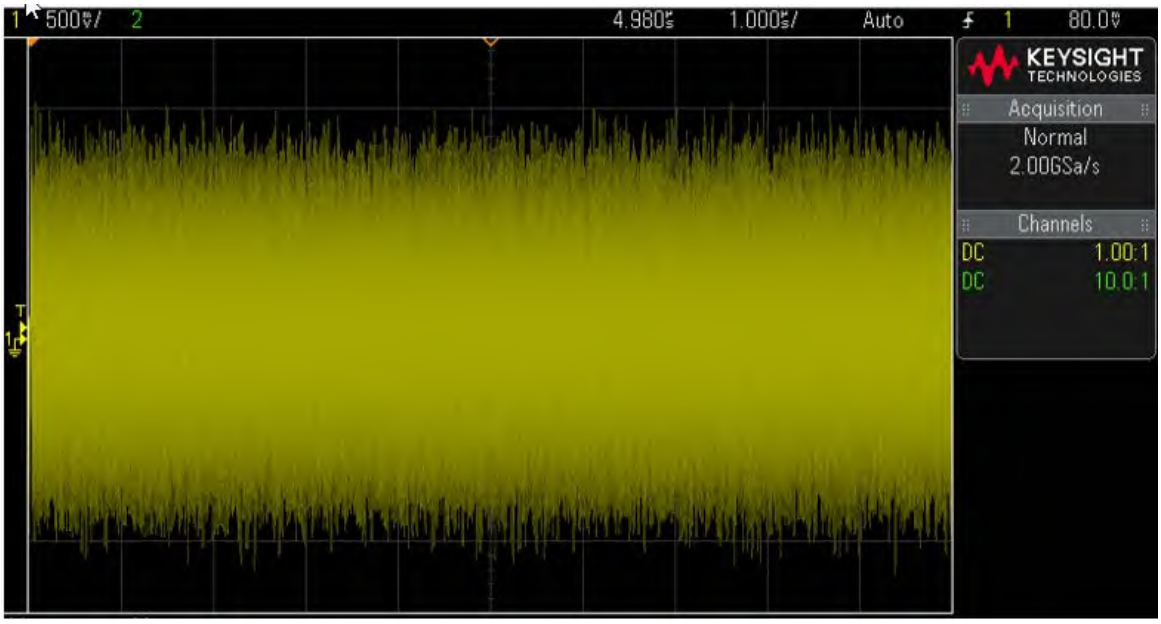
There are two noise generators (Noise 1 and Noise 2) that both provide the same functionality. The noise generators have selectable noise type (Uniform or Gaussian). Noise generators play a critical role in RF immunity testing, fault isolation testing, and many more. The noise generators can be routed to the LF port to observe their behavior.

1. Select **Preset** > **Preset** to set the instrument to a known state.

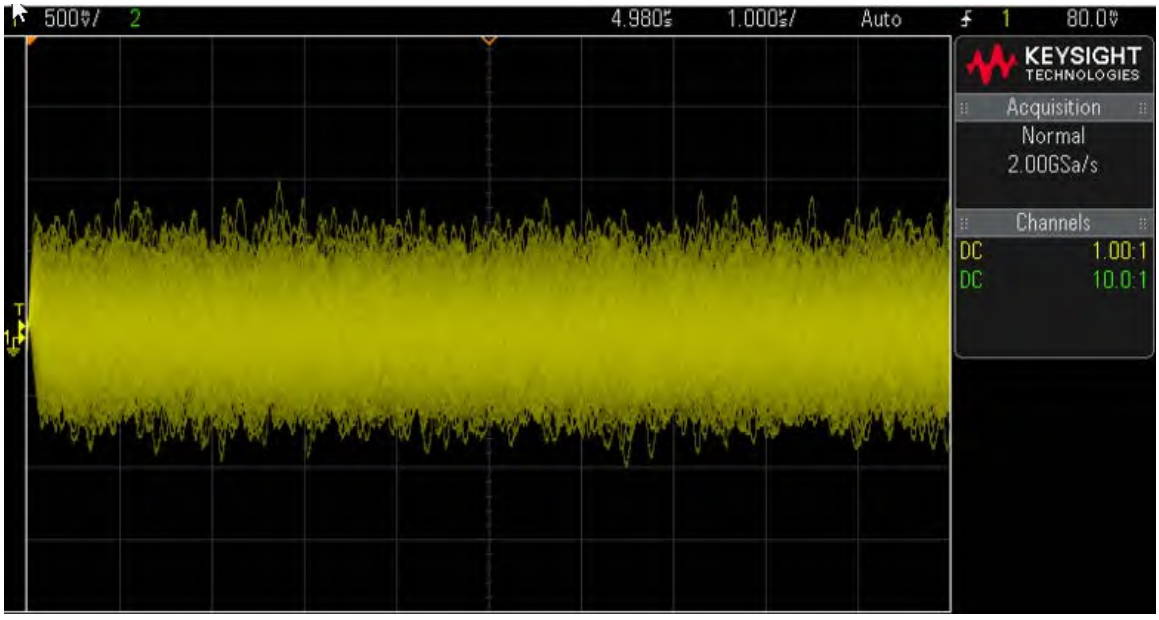
2. Select the **System** menu > **Connectors**.

3. In the Connectors window, select **LF Out**.
4. In the LF Out window, set:
   - Source to **Noise 1**
   - Type to **Uniform**
   - Amplitude to **500 mV**
   - DC Offset to **0**
   - Toggle LF Output **On**

5. Capture the signal on the oscilloscope



6. Set Type to **Gaussian**.
7. Capture the signal on the oscilloscope



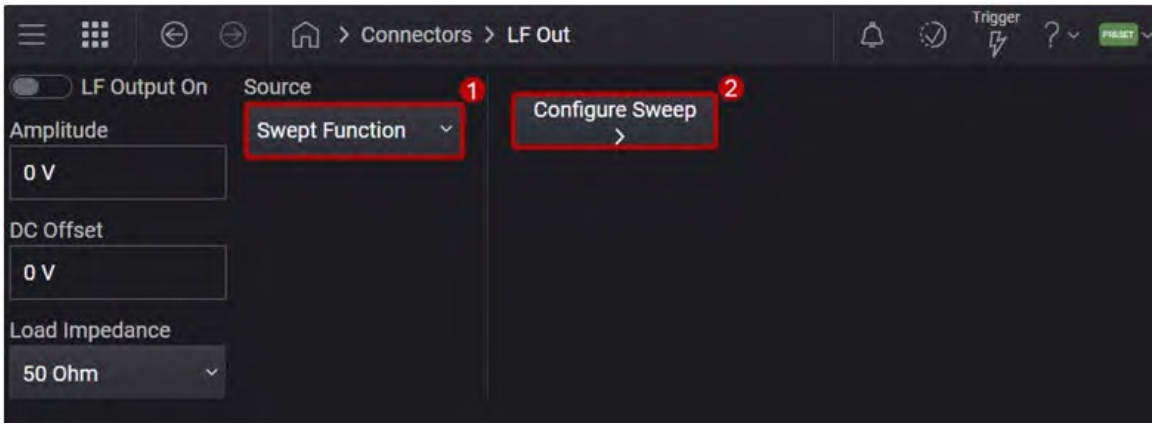
Swept Function

The Swept Function generator has a selectable start frequency, stop frequency, and sweep start to stop time as well as selectable sweep shape, sweep waveform shape, and trigger source. With a frequency range of 10 mHz

to 10 MHz and a start-to-stop time range of 20 ns to 89 s. The Swept Function Generator is key in conducting frequency response testing for filters, amplifiers, and various other RF components. The Swept Function Generator can be routed to the LF port to observe it's behavior.

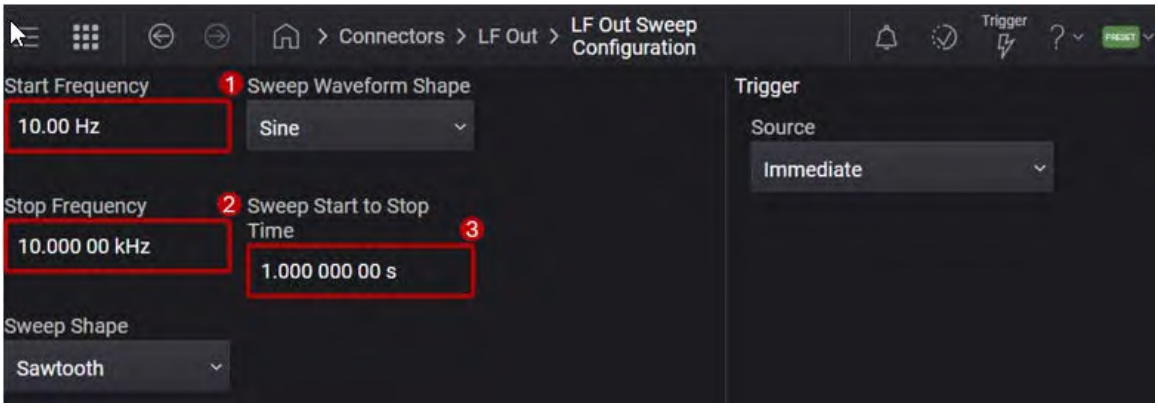
1. Select **Preset** > **Preset** to set the instrument to a known state.
2. Select the **System** menu > **Connectors**.

3. In the Connectors window, select **LF Out**.
4. In the LF Out window, set:
   - Source to **Sweep Function**
   - **Configure Sweep**

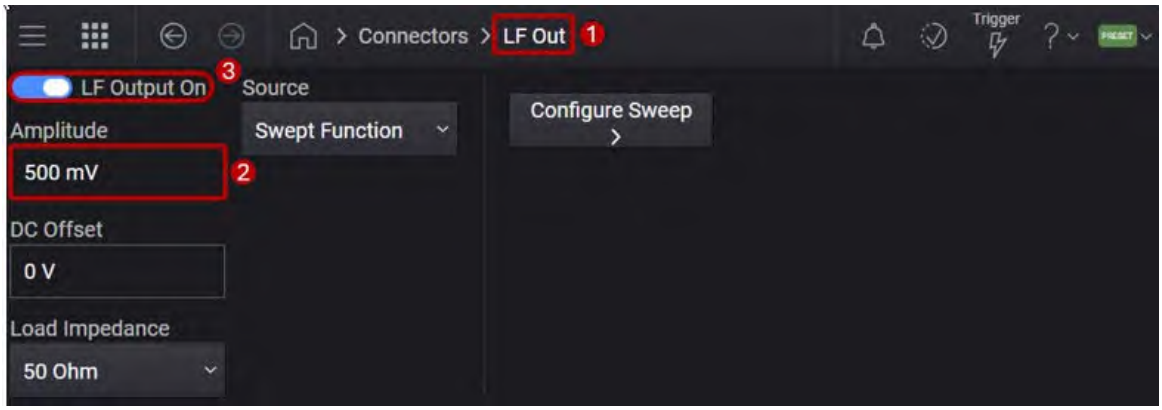


5. Set:
   - Start Frequency to **10 Hz**
   - Stop Frequency to **10 kHz**

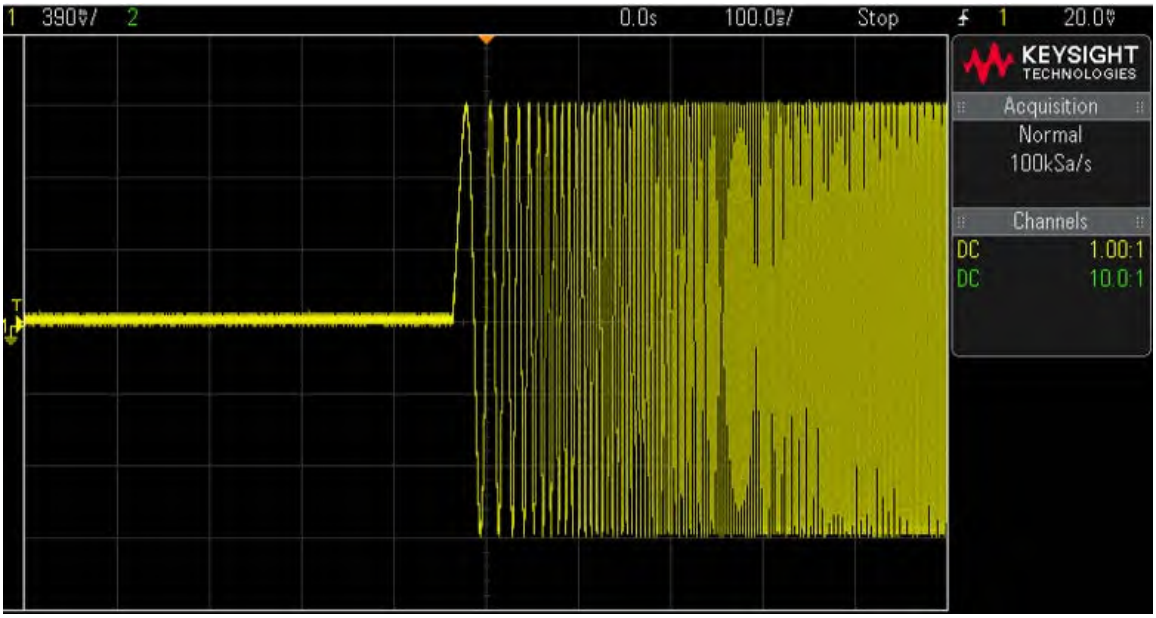
– Sweep Start to Stop Time to **1 s**



6. Select the **Back** Arrow and select:
   – **LF Out**
   – Amplitude to **500 mV**
   – Toggle LF Output **On**



7. Capture the signal on the oscilloscope



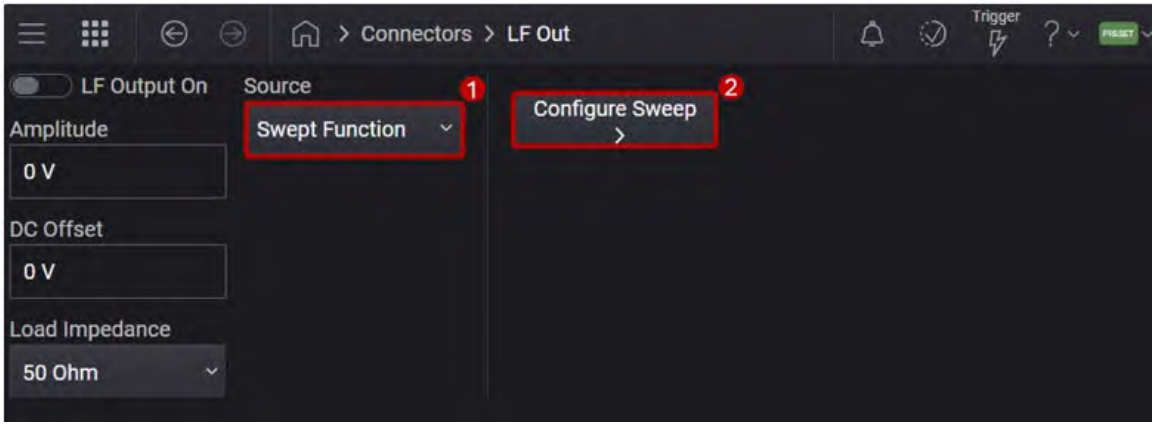
Sawtooth and Triangle Sweep Shapes

The Swept function generator has two selectable sweep shapes (Sawtooth and Triangle). The Sawtooth sweep shape is a linear sweep from start frequency to stop frequency. When the stop frequency is reached the function generator jumps back to the start frequency and repeats again. The Triangle waveform sweep shape starts at the start frequency then sweeps to the stop frequency, but when the stop frequency is reached the function generator sweeps back down in frequency until reaching the start frequency. This behavior is best seen on a spectrogram measurement.

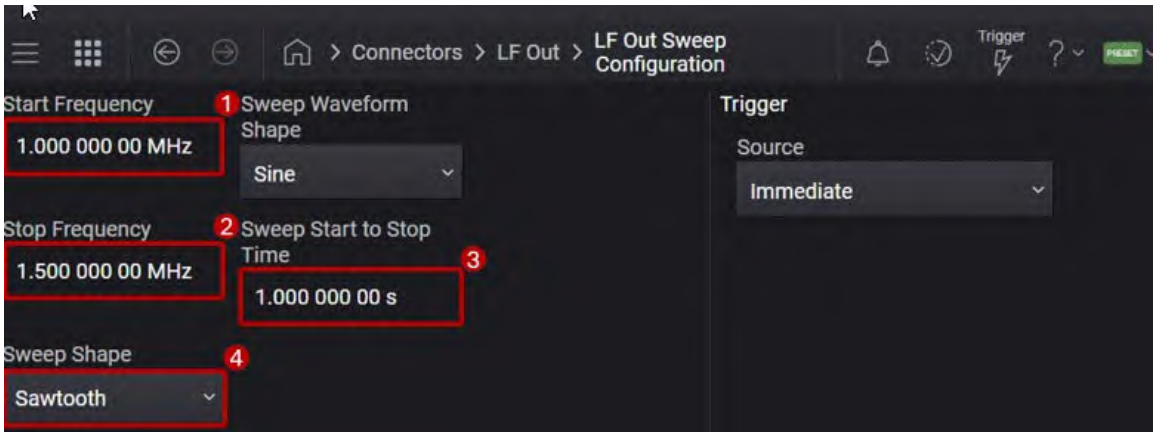
1. Select **Preset** > **Preset** to set the instrument to a known state.
2. Select the **System** menu > **Connectors**.

3. In the Connectors window, select **LF Out**.

4. In the LF Out window, set:
   - Source to **Sweep Function**
   - **Configure Sweep**

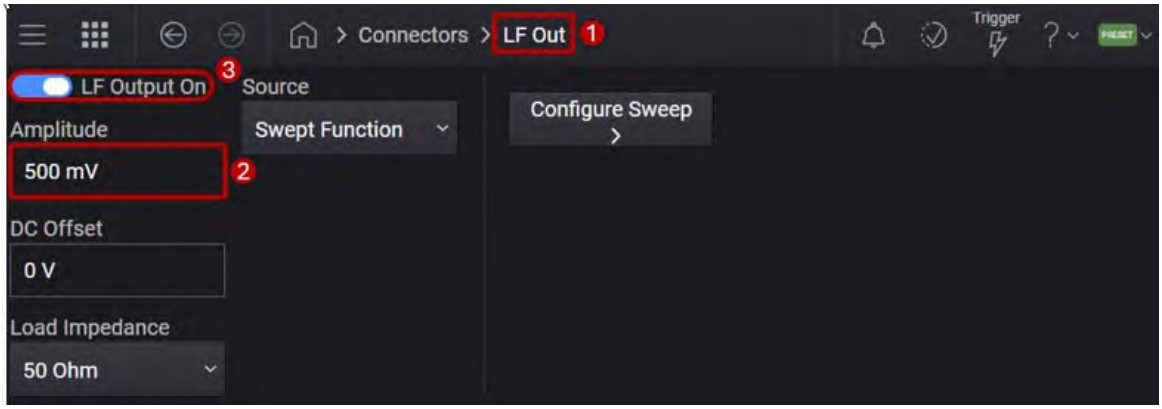


5. Set:
   - Start Frequency to **1 MHz**
   - Stop Frequency to **1.5 MHz**
   - Sweep Start to Stop Time to **1 s**
   - Sweep Shape to **Sawtooth**

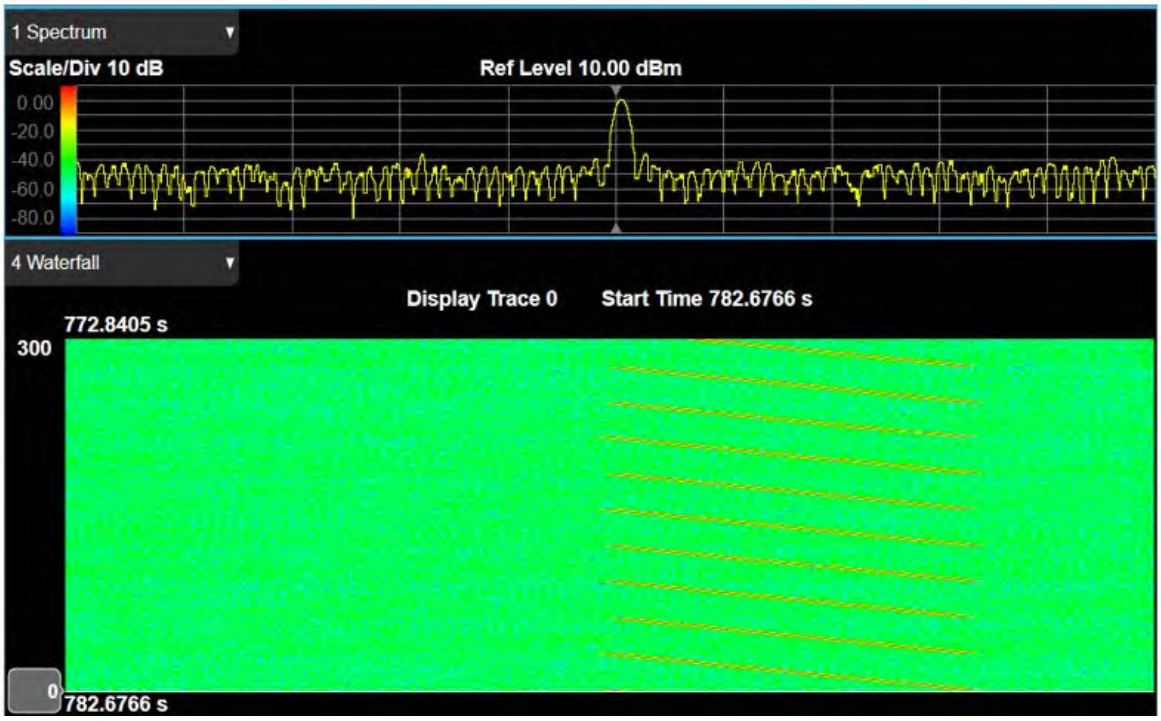


6. Select the **Back** Arrow and select:
   - Amplitude to **500 mV**

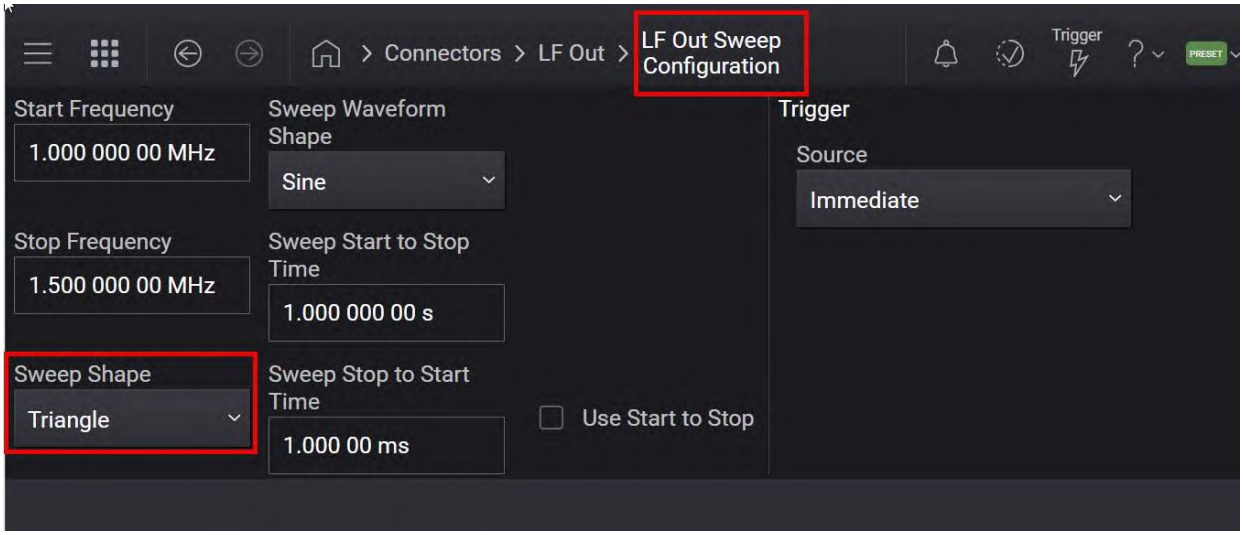
– Toggle LF Output **On**



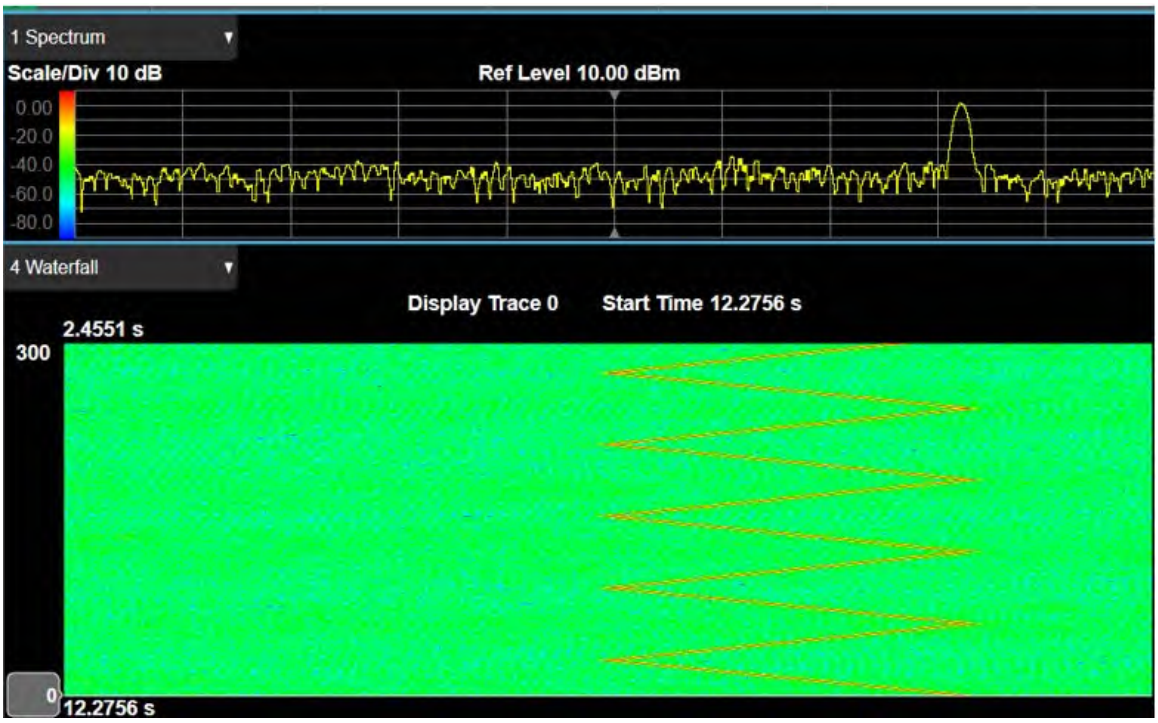
7. Capture the signal on a Spectrogram.



8. Select **Configure Sweep** > **Sweep Shape Triangle**.

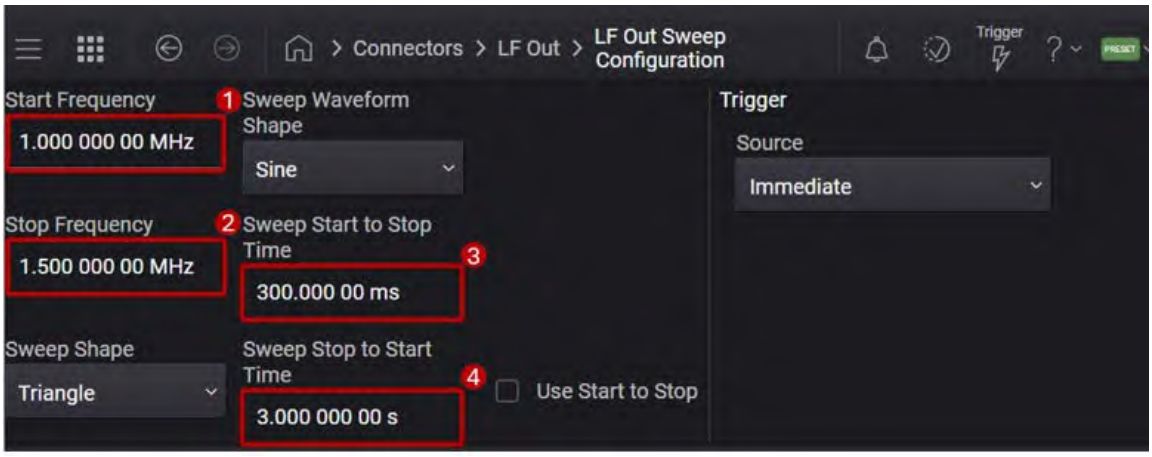


9. Select the **Back** Arrow and Toggle LF Output **On**.
10. Capture the signal on the Spectrogram.



The Triangle Sweep shape has two Sweep Start to Stop Time inputs, one for sweep up and one for sweep down. By changing the sweep times to be unequal, the sweep shape that is seen on the spectrogram will become nonuniform.

11. In the LF Output Sweep Configuration window, set Sweep Start to Stop Time to **300 ms** and Sweep Start to Stop Time to **3s**.



12. Capture the signal on the Spectrogram.

Notice that the signal sweeps from a low to high frequency at a faster rate than from a high to low frequency.

Radio Frequency Modulation Option UNT

In the sections above the function generators were routed to the LF port to observe their operation individually. This section will demonstrate how Option 303 effects the RF signal at the RF port. The function generators are used to

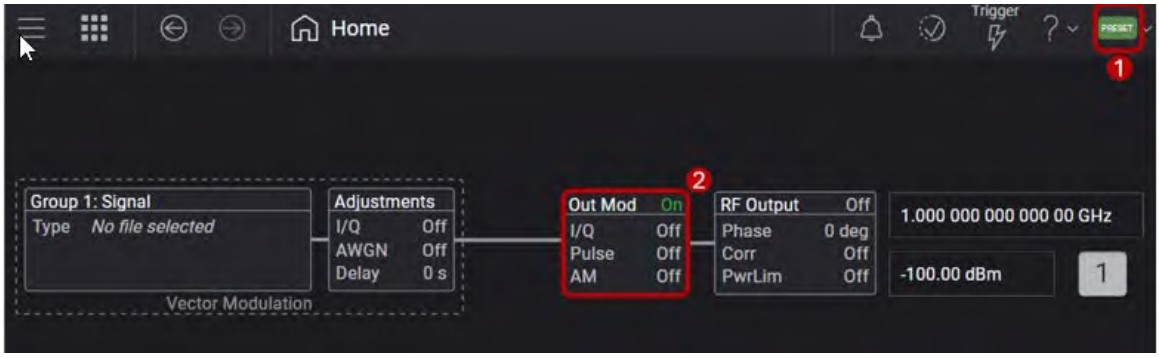
modulate the RF signal through amplitude and frequency/phase modulators (see Figure 1-2 on page 77). We can see how the modulators affect the original RF signal in both time and frequency domain using a signal analyzer and oscilloscope.

Amplitude Modulation

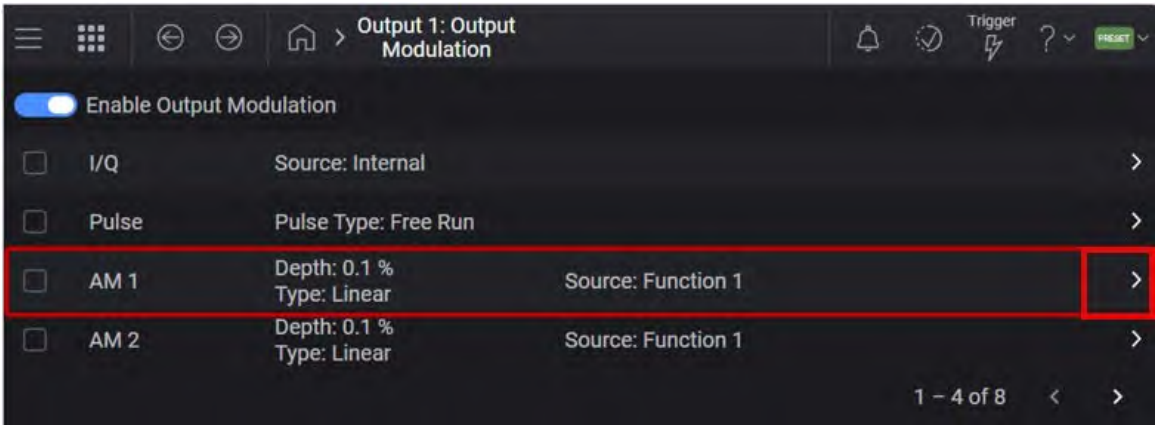
Using an oscilloscope, amplitude modulation can be observed in the time domain. The function generators produce the modulating signal, creating the envelope, while the RF signal serves as the carrier wave.

To apply amplitude modulation:

1. Select **Preset** > **Preset** to set the instrument to a known state and select the **Out Mod** block.

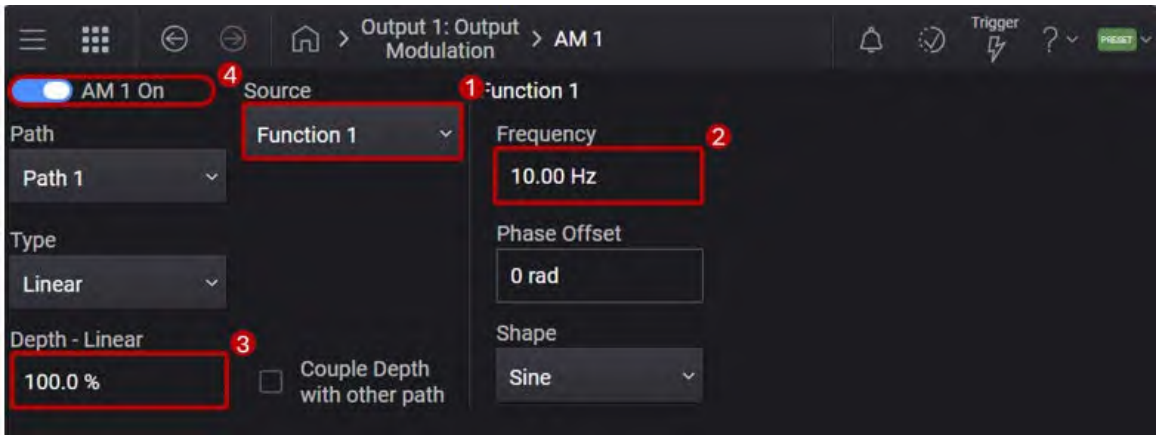


2. In the Output Mode window, select **AM1** arrow at the far left side of the window.

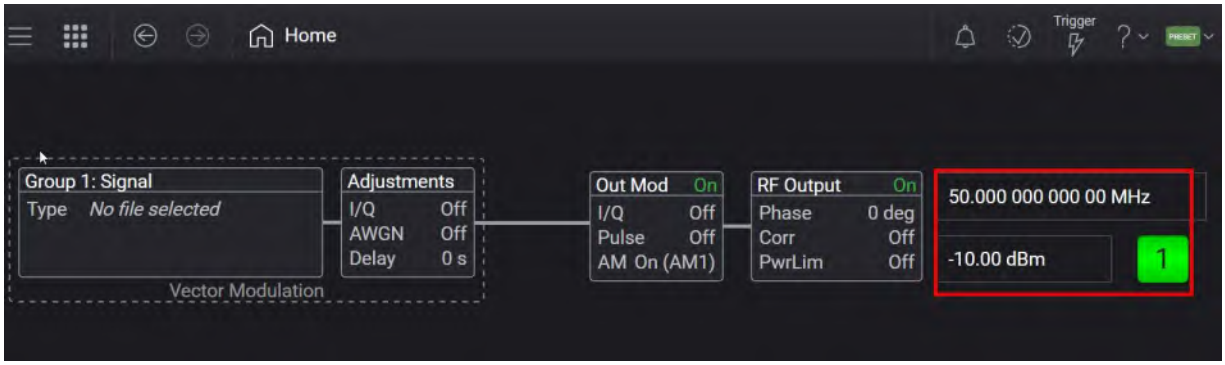


3. In the AM1 window, set:
   - Source to **Function 1**
   - Frequency to **10 Hz**
   - Depth - Linear to **100%**

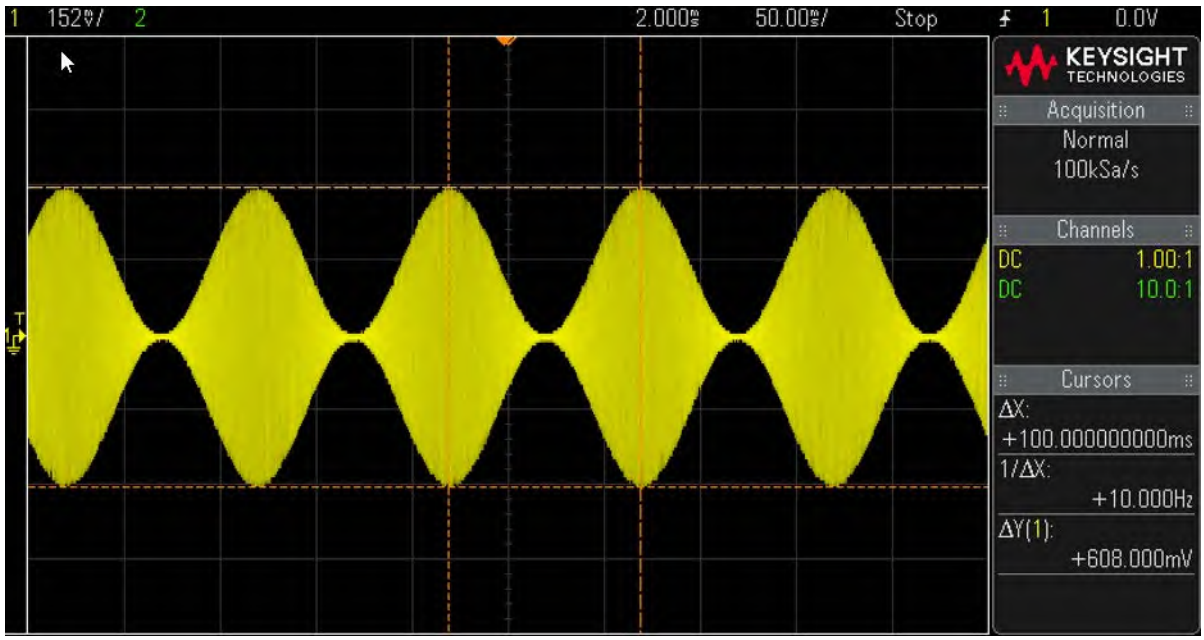
– Toggle AM 1 **On**



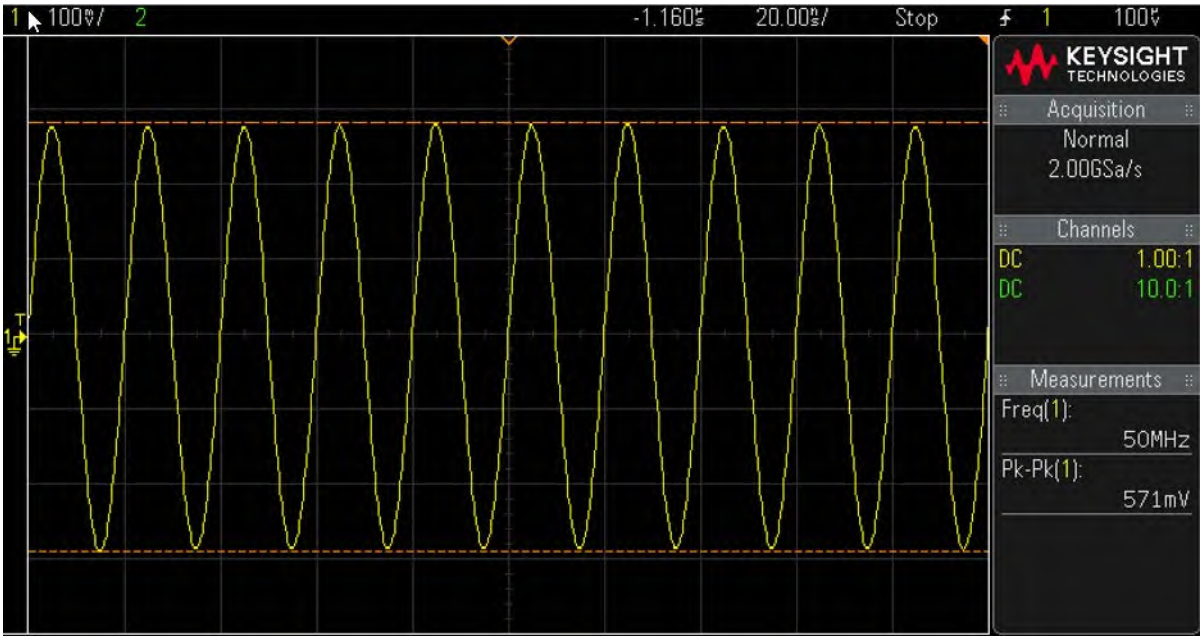
4. Select the **Home** icon and set:
   – Frequency to **50 MHz**
   – Power to **-10 dBm**
   – Toggle RF output **On**



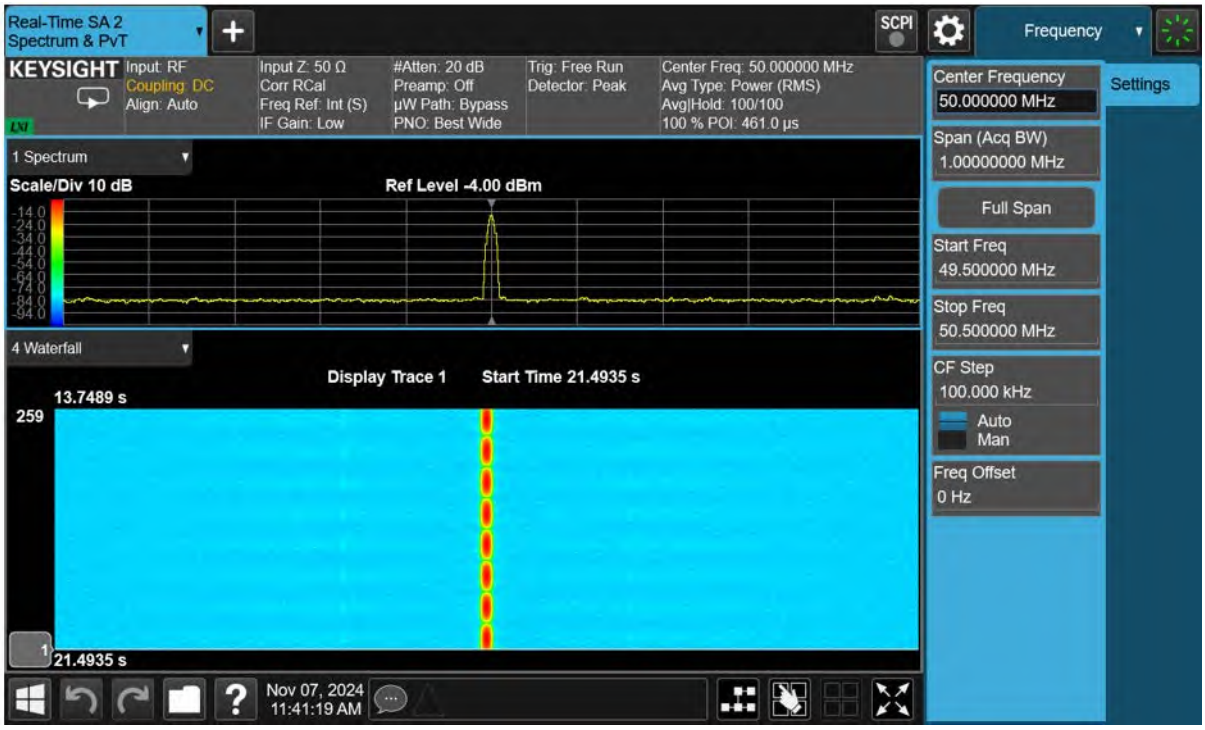
5. Capture the envelop on the oscilloscope. The shape of the function corresponds to the shape of the envelope. (In this example, the function shape is set to sine.)



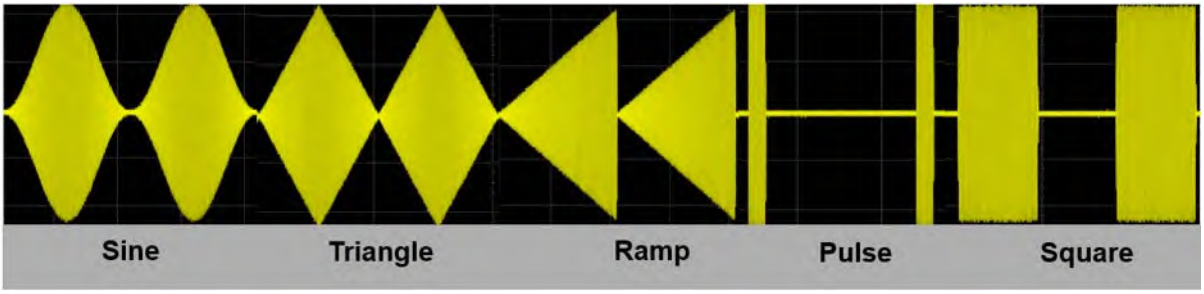
6. Capture the carrier on the oscilloscope.



You can also view the modulation on a spectrogram. The modulation can be visualized as varying intensity of the carrier frequency over time.



The shape of the function generator corresponds to the shape of the envelope when generating AM signals.

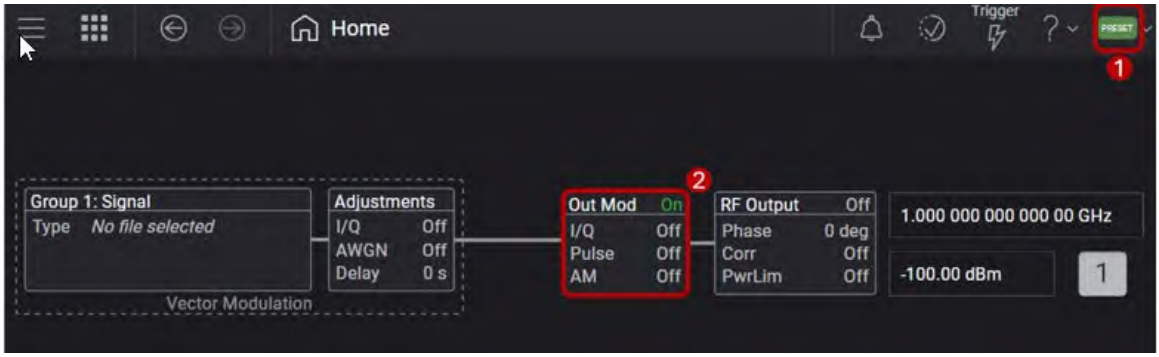


Frequency Modulation

Using a signal analyzer, frequency modulation can be observed using a spectrogram. The function generator produces the modulating signal, which varies the frequency of the RF signal.

To apply frequency modulation:

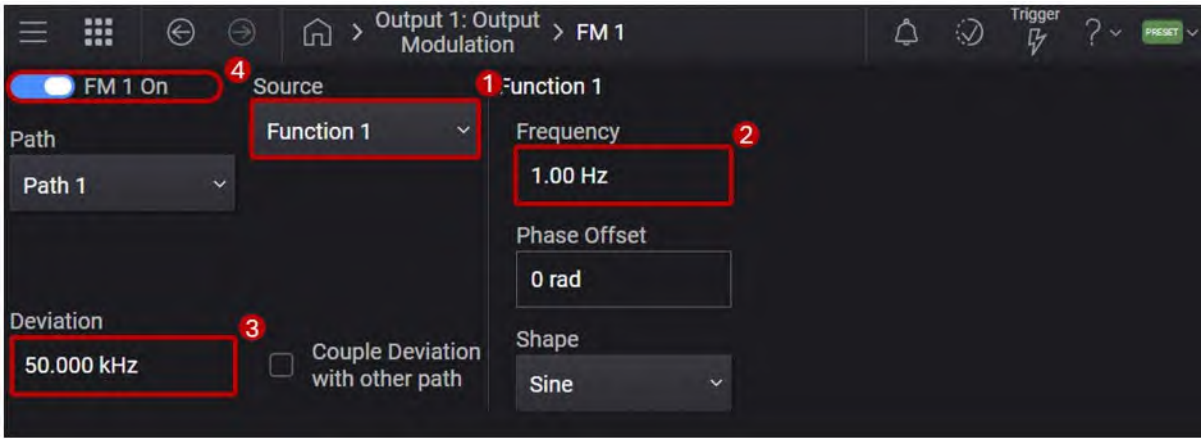
1. Select **Preset** > **Preset** to set the instrument to a known state and select the **Out Mod** block.



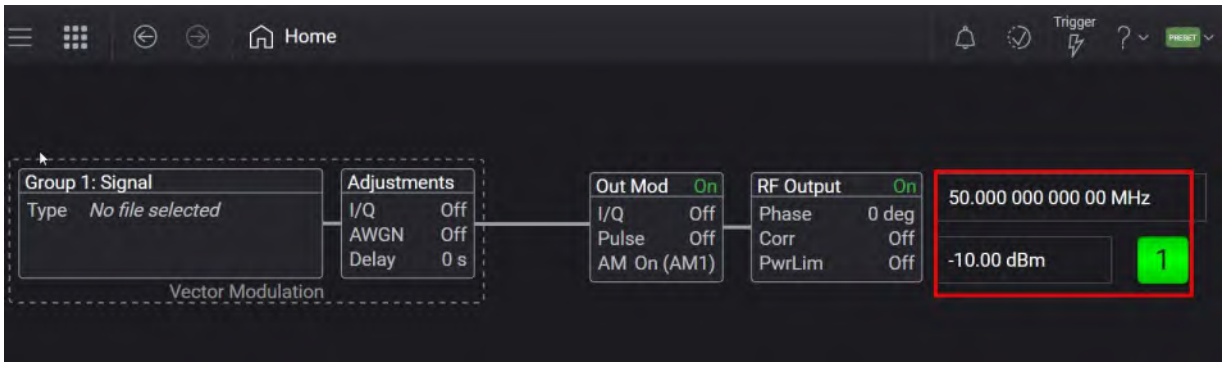
2. In the Output Mode window, select the right arrow at the bottom of the display to select choices 5-8, then select the **FM1** arrow at the far left side of the window.

3. In the FM1 window, set:
   - Source to **Function 1**
   - Frequency to **1 Hz**
   - Deviation to **50 kHz**

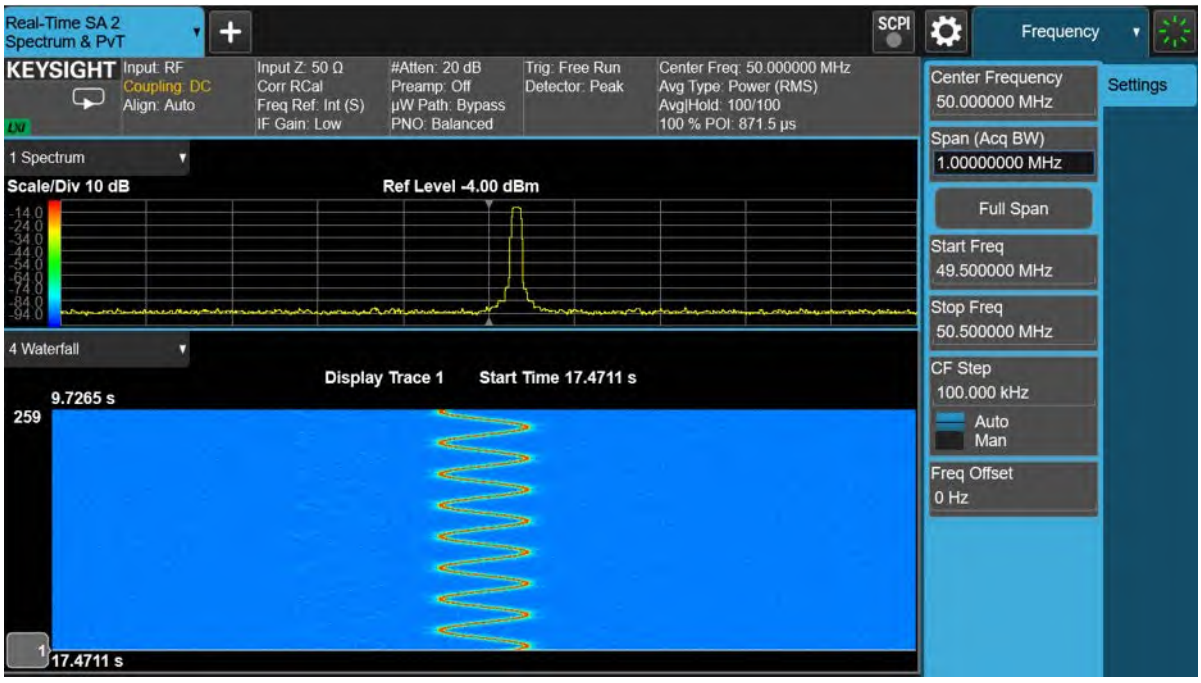
– Toggle FM 1 **On**



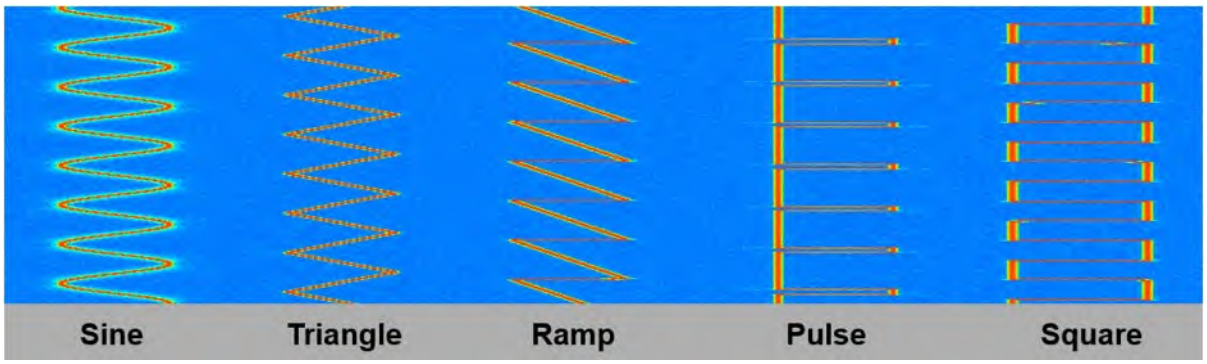
4. Select the **Home** icon and set:
   – Frequency to **50 MHz**
   – Power to **-10 dBm**
   – Toggle RF output **On**



5. Capture the signal on the Spectrogram.



The shape of the function generator corresponds to the shape of the frequency modulation.

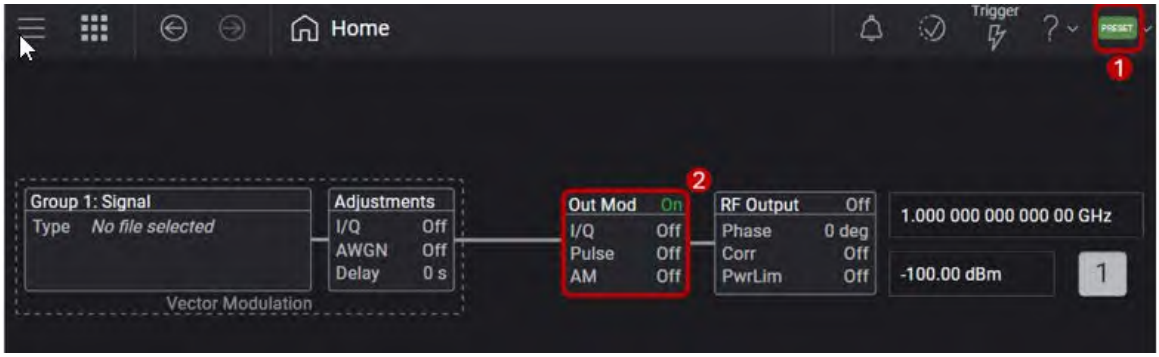


Phase Modulation

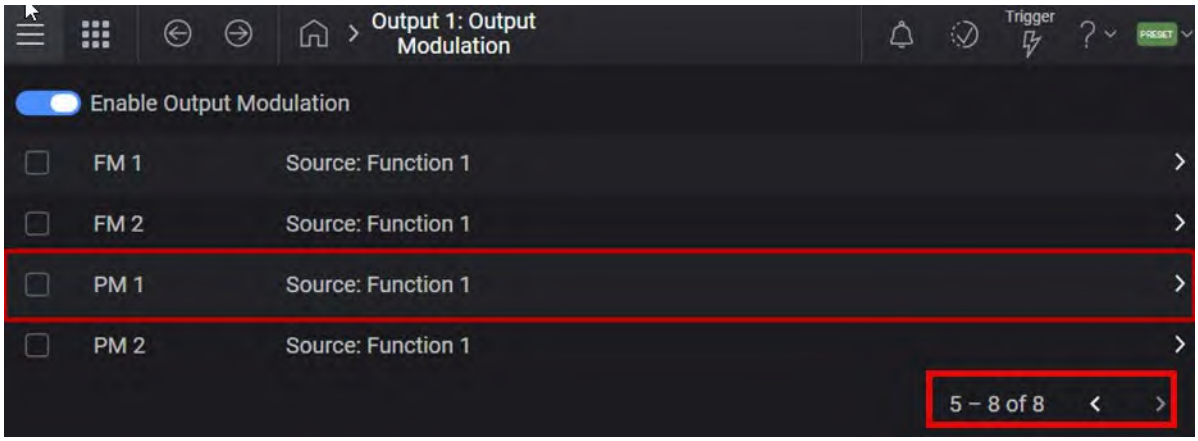
Phase Modulation can be visualized using an Oscilloscope in the time domain. The function generator produces the modulating signal, which varies the phase of the RF signal.

To apply phase modulation:

1. Select **Preset** > **Preset** to set the instrument to a known state and select the **Out Mod** block.

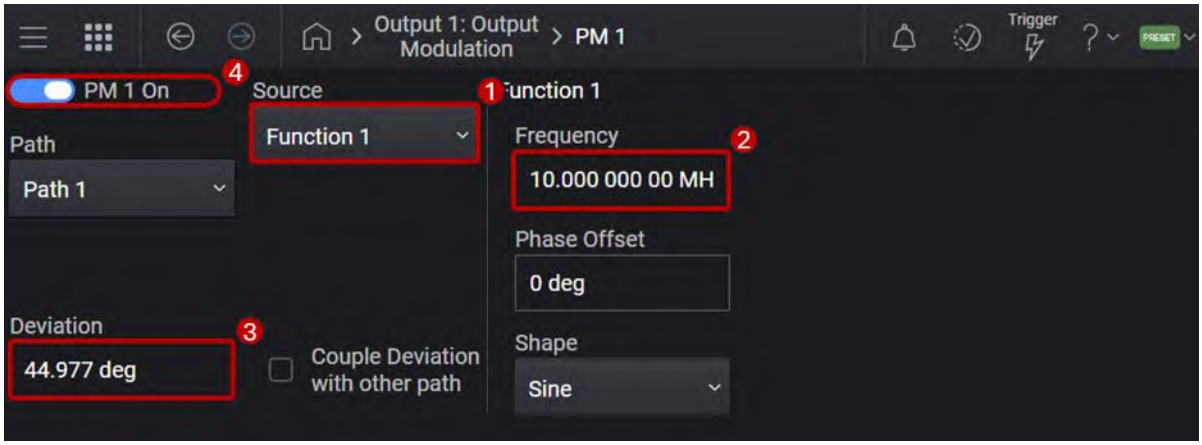


2. In the Output Mode window, select the right arrow at the bottom of the display to select choices 5-8, then select the **PM1** arrow at the far left side of the window.

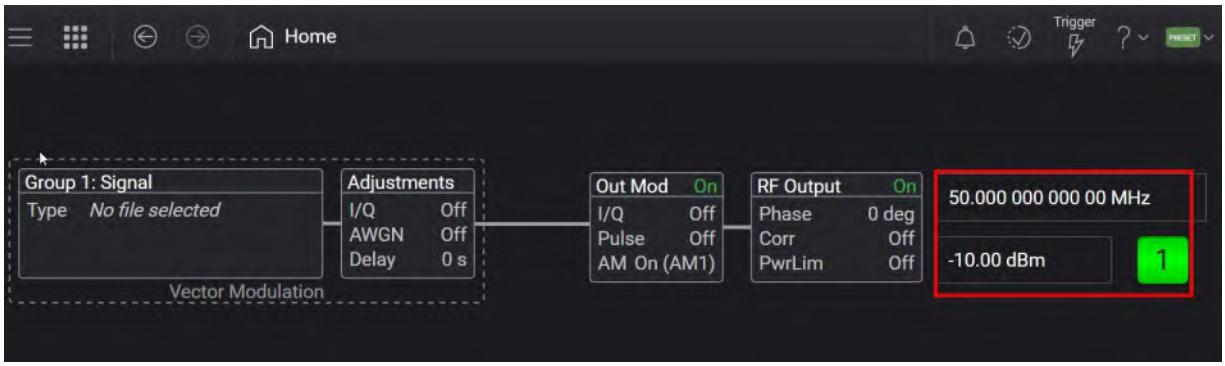


3. In the PM1 window, set:
   - Source to **Function 1**
   - Frequency to **10 MHz**
   - Deviation to **45 deg**

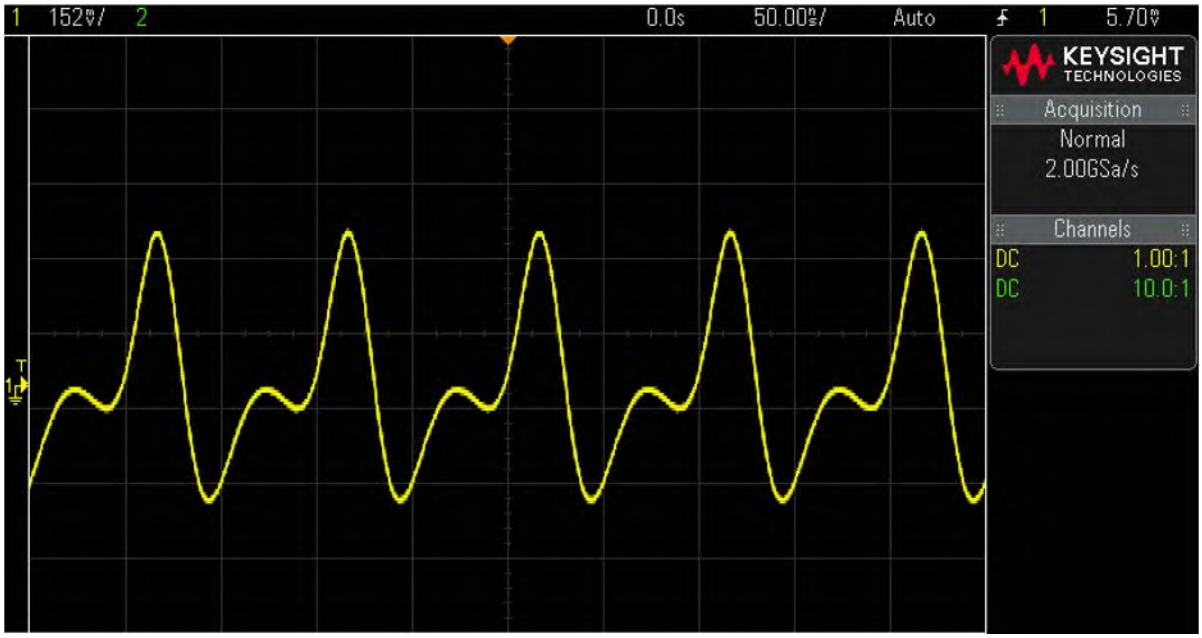
– Toggle PM 1 **On**



4. Select the **Home** icon and set:
   – Frequency to **50 MHz**
   – Power to **-10 dBm**
   – Toggle RF output **On**



5. Capture the signal on the Oscilloscope.



Applications

In many scenarios a RF communication system is subject to dynamic environments which can alter the characteristics of an ideal signal. In an ideal environment, the signal being generated and transmitted is the exact signal being received. Physically we know that this is not possible, the signal will be subject to some change. Therefore, a good communication system is resilient and can operate effectively within some range of impairments.

Consider an environment where signal strength fluctuates over time due to varying physical obstructions between the source and receiver. It would be inefficient and inconsistent to physically implement a RF environment to test your communication system. A more effective and repeatable test would be to simulate the conditions of such an environment. Using AM, we can simulate the impairment of varying signal strength. With selectable frequency, shape, and depth the AM modulator is capable of a wide range of simulated environments to support signal strength testing.

One of the design challenges of mobile communications systems comes from Doppler shift. As a transmitter and receiver move towards or away from one another, the signal will compress or stretch, effectively changing the perceived frequency. An effective way to simulate such an environment is by using the FM modulator. Using FM, we can simulate the impairment of varying frequency caused by Doppler shift. With selectable frequency deviation and deviation rate, the FM modulator can effectively simulate Doppler shift environments that mobile communication systems often encounter.

When in an environment that causes reflections, a wireless communication system will be subject to multipathing. Consider a communication system that operates in a major city. Large buildings made of concrete and glass windows create many reflections of the signal being sent. These reflections cause variation in phase with respect to the original signal. Using PM, we can simulate the impairment of phase variation caused by multipathing. With selectable phase deviation, deviation rate, and offset the PM modulator can effectively simulate a simple multipath environment caused by reflective surfaces.

Multifunction Generator Option 303 allows you to apply multiple impairments simultaneously. If needed, the function generators can be configured such that five impairments are applied at the same time. It is important to note that there are two paths per modulator, which means that no more than two functions can be applied to one modulation type. The FM/PM modulator can only do one modulation type at a time, meaning that PM and FM cannot run simultaneously. AM and FM or AM and PM can be summed, however.

Conclusion

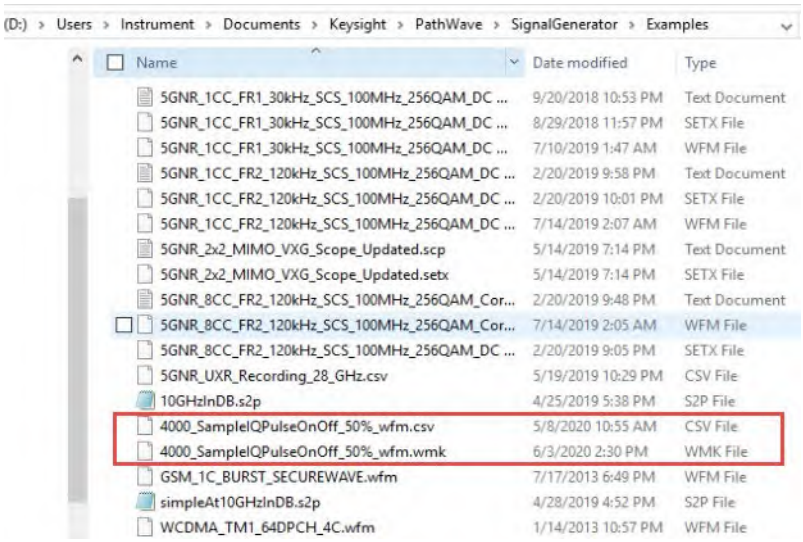
Simulating RF environments using amplitude, frequency, and phase modulation provides a robust and repeatable method for testing communication systems. By leveraging these modulation techniques, engineers can effectively simulate real-world conditions such as fluctuating signal strength, Doppler shifts, and multipath reflections. This approach not only enhances the reliability of testing but also ensures that communication systems are resilient and capable of operating under various impairments. The Multifunction Generator Option 303 further extends this capability by allowing multiple impairments to be applied simultaneously, offering a comprehensive solution for thorough and efficient receiver testing. Through these simulation techniques, we can better prepare and design communication systems to perform reliably in dynamic and challenging environments.

Generating Synchronized Pulse (IQ + Analog) Modulation

NOTE

Ensure the equipment and triggers are properly configured. Refer to "Equipment Setup" on page 13 and "Setting Up Triggers on the X-Series Signal Analyzer" on page 15.

You will need to create your own ASCII text marker file and a binary file with the exact same name. Both files must be stored in the same file folder. For this example, we will use example files that have been stored on the VXG.



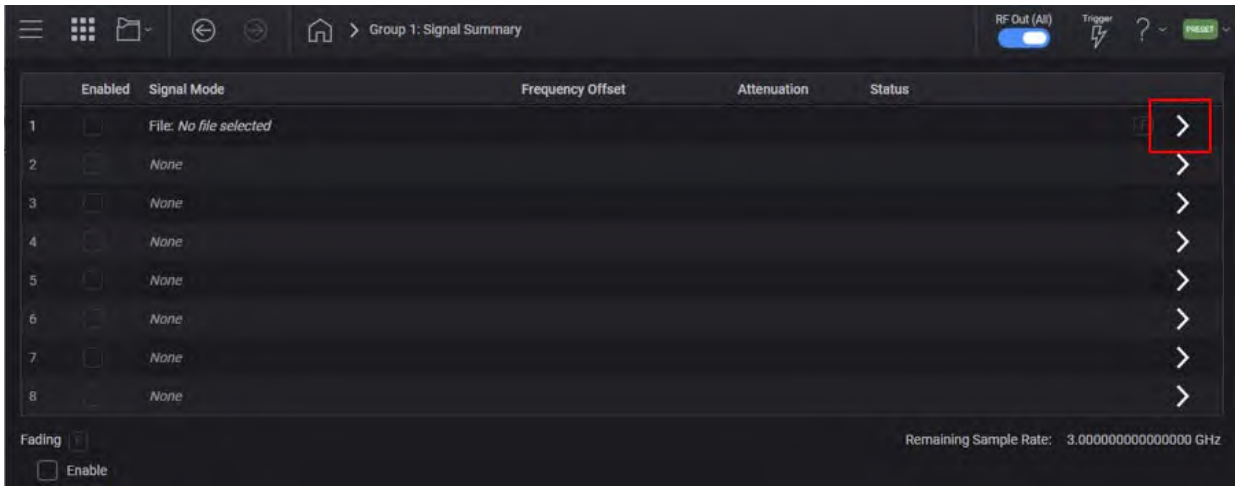
On the VXG:

1. Select **Preset** > **Preset** to set the instrument to a known state.
2. Set the Frequency to **1 GHz** and Power to **0.0 dBm**.
3. Select the **Signal** block to open.

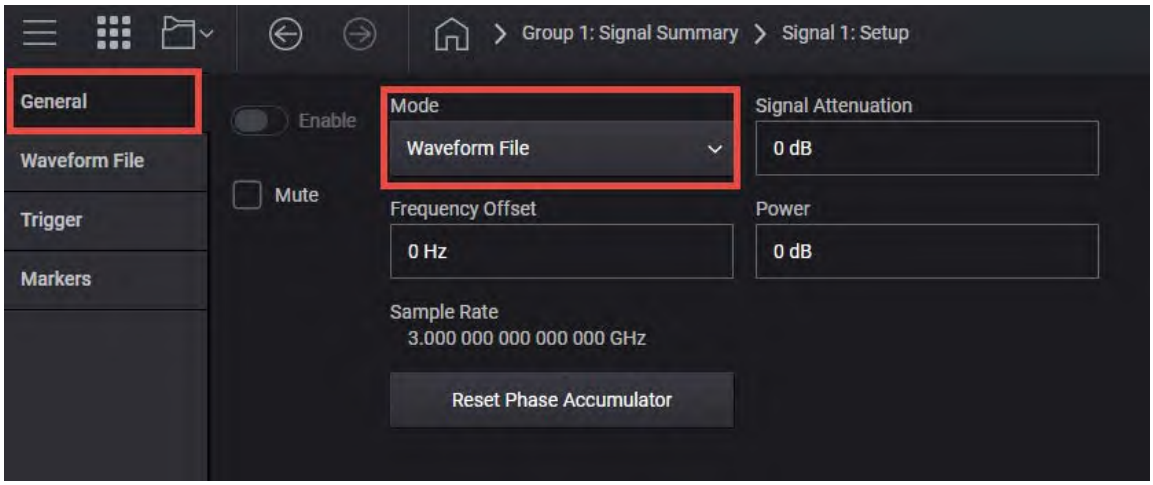
4. Select the arrow for Signal 1 to open the Signal Setup window.

NOTE

This screen is only accessible if Option M9484C-8SG (8 virtual signal generators) is installed. For all other option configurations, continue to the next step.



5. Select the **Mode** dropdown and select **Waveform File**.



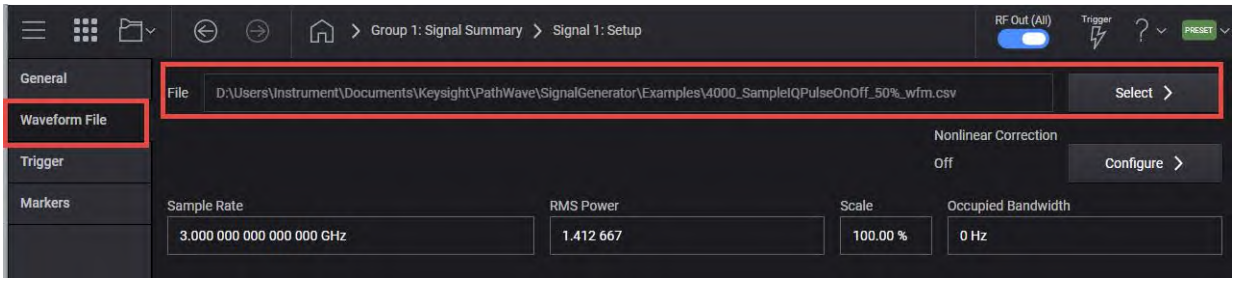
6. Select the **Waveform File** tab (left pane), and then in the Waveform Setup area, use File **Select** to navigate to:

   `D:\Users\Instrument\Documents\Keysight\PathWave\SignalGenerator\Examples`

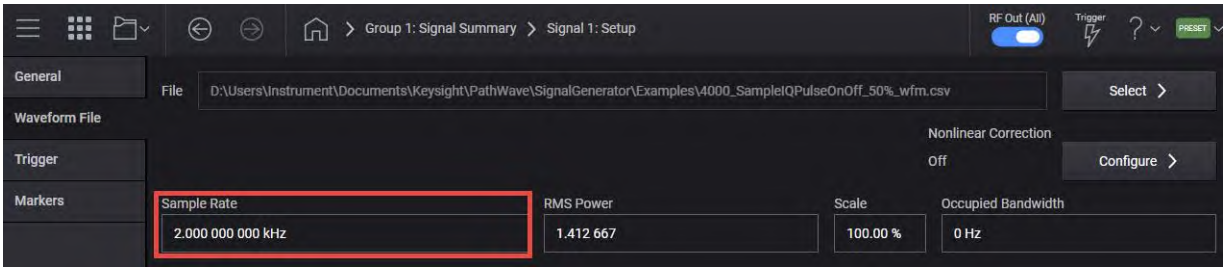
   and choose

`4000_SampleQPulseOnOff_50%_wfm.csv`

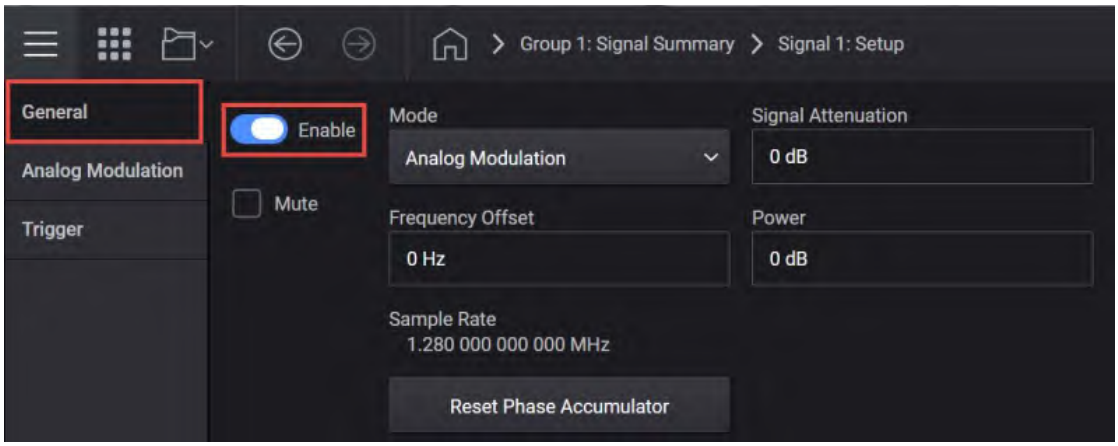
then **Select**.



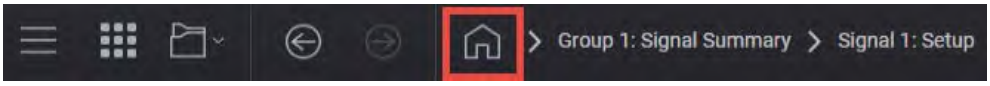
7. Set the Sample Rate to **2 kHz**.



8. Select the **General** tab > **Enable**.



9. Close the Vector Modulation Signal Setup by selecting the **Home** icon at the top of the display.



10. Set RF Out to **On** by selecting the numbered channel indicator switch.

This enables the RF Out for the indicated channels, in this case Channel 1 if using a multi-channel VXG.

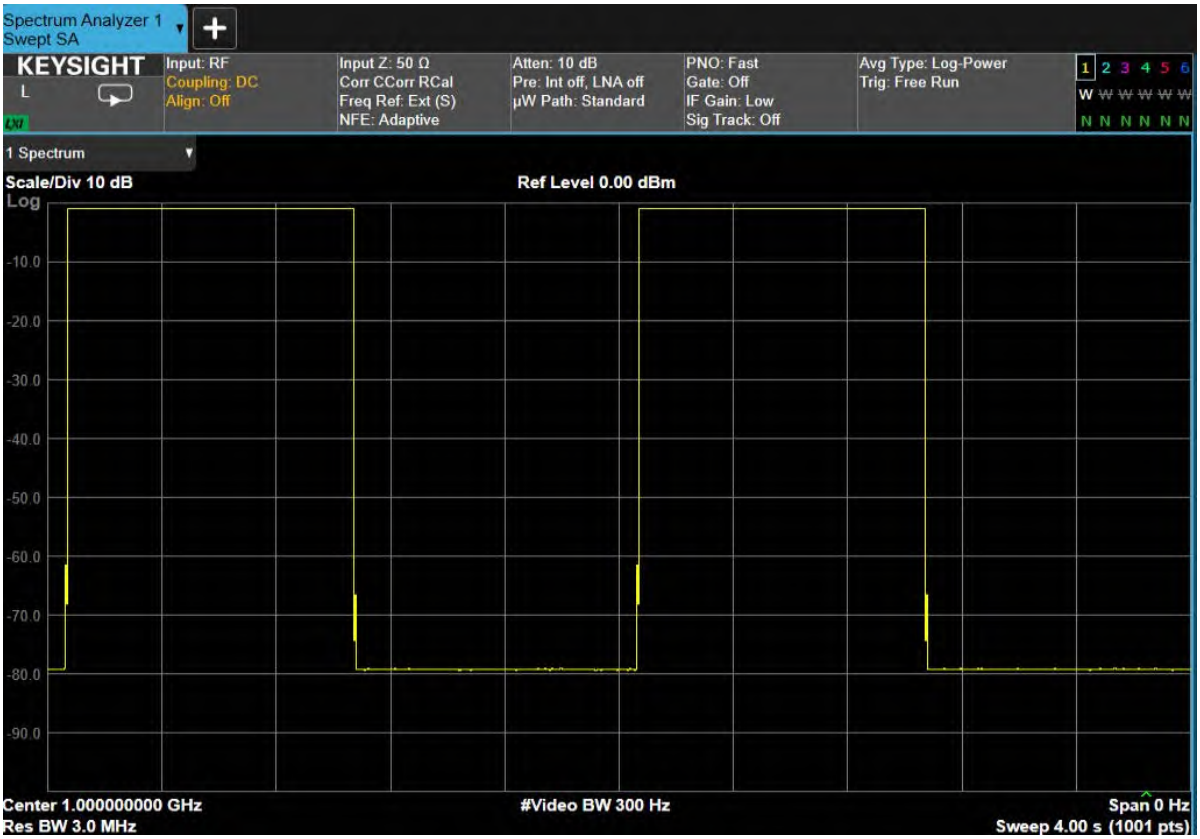
On the X-Series Signal Analyzer:

1. From the Menu Panel (on the top right of the display), select **Mode/Meas** > **Spectrum Analyzer** mode > **Swept SA** Measurement > **Normal** View.

NOTE

If accessing the instrument via a Remote Desktop connection, select the Screen tab (at the top of the display) to open the Mode/Measurement/View Selector window.

2. Select **Mode Preset** to set Spectrum Analyzer mode to a known state.
3. Select **Frequency** and set Center Frequency to **1.0 GHz** and Span to **0 Hz**.
4. Select **BW** and set the Video BW to **300 Hz**.
5. Select **Sweep** and set the Sweep Time to **4.0 s**.



Generating a Multitone Signal

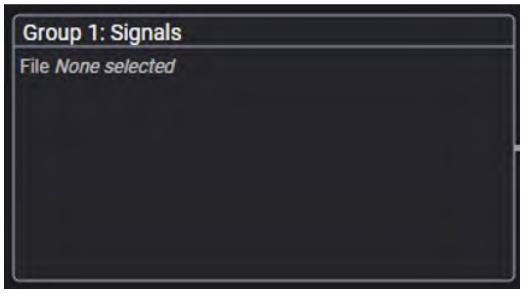
This example shows you how to create a multitone signal, which allows you to separate the usable frequency band into multiple channels. This can make a signal that is difficult to characterize in the time domain more readable.

NOTE The VXG must have the N7621APPC PathWave Signal Generation for Multitone license installed.

NOTE Ensure the equipment and triggers are properly configured. Refer to "Equipment Setup" on page 13 and "Setting Up Triggers on the X-Series Signal Analyzer" on page 15.

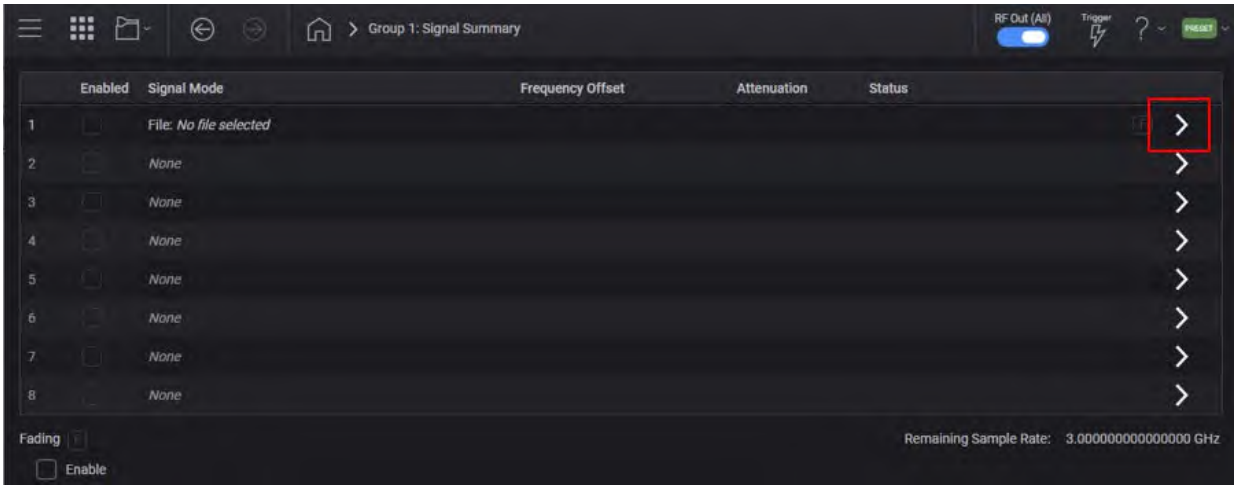
On the VXG:

1. Select **Preset** > **Preset** to set the instrument to a known state.
2. Set Frequency to **20 GHz** and Amplitude to **-10 dBm**.
3. Select the **Signal** block to open.

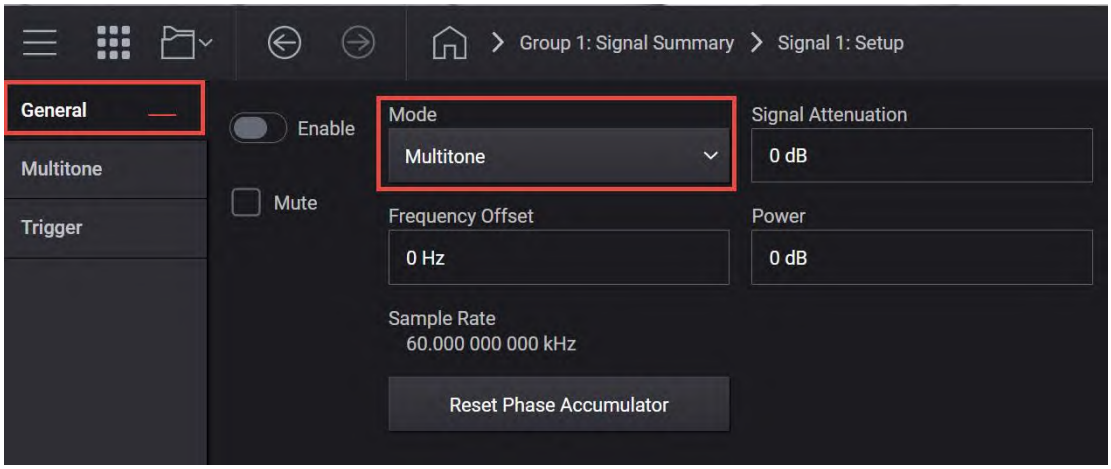


4. Select the arrow for Signal 1 to open the Signal Setup window.

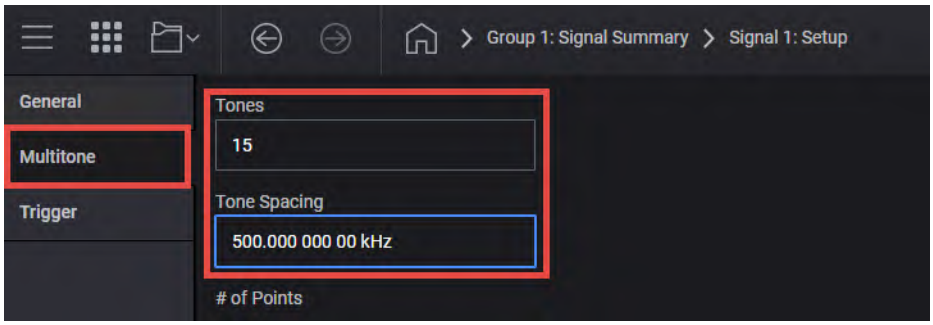
NOTE This screen is only accessible if Option M9484C-8SG (8 virtual signal generators) is installed. For all other option configurations, continue to the next step.



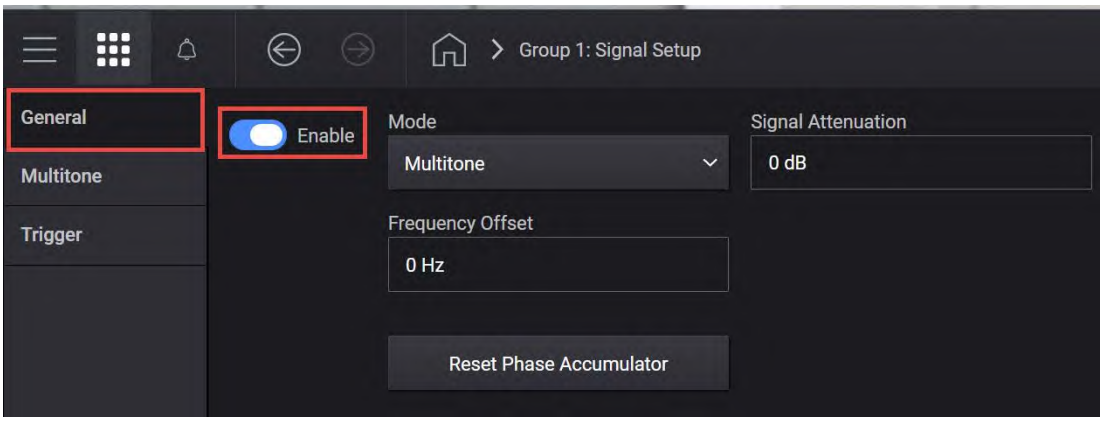
5. Select the **Mode** dropdown and select **Multitone**.



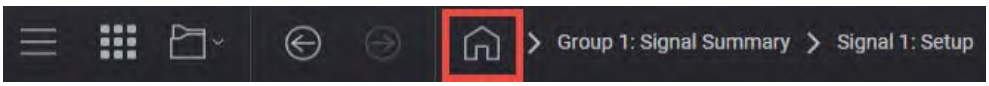
6. Select the **Multitone** tab and set:
   a. Set Tones to **15**.
   b. Set Tone Spacing to **500 kHz**.



7. Select the **General** tab > **Enable**.



8. Close the Vector Modulation Signal Setup by selecting the **Home** icon at the top of the display.



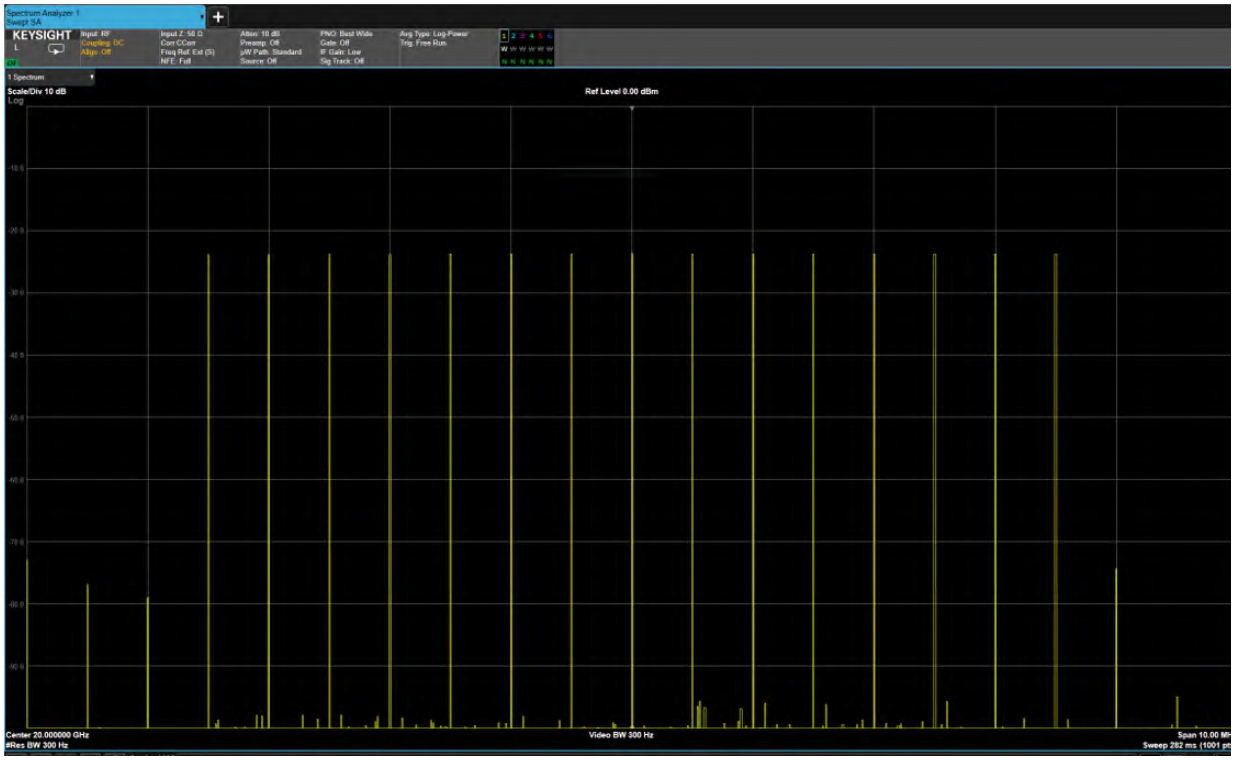
9. Set RF Out to **On** by selecting the numbered channel indicator switch.

   This enables the RF Out for the indicated channels, in this case Channel 1 if using a multi-channel VXG.

On the X-Series Signal Analyzer:

1. Select **Mode Preset** to set Spectrum Analyzer mode to a known state.
2. Select **Frequency** and set Center Frequency to **20 GHz** and Span to **10 MHz**.
3. Select **BW** and set Res BW to **300 Hz**.

   Observe the 15 tones.


KEYSIGHT
Scale/Div 10 dB
Log
Ref Level 0.00 dBm
Center 20.000000 GHz
#Res BW 300 Hz
Video BW 300 Hz

Using the equivalent SCPI commands

On the VXG:

```
SYSTem:PRESet
RF1:FREQuency:CW 20GHZ
RF1:POWer:AMPLitude -10dBm
SIGNal1:MODE MTONes
SIGNal1:MTONe:ARB:NTON 15
SIGNal1:MTONe:ARB:FSP 500KHZ
SIGNal1 ON
RF1:OUTPut:STATe ON
```

For multi-channel instruments, set RF Out (all) to On.

```
RFALl:OUTPut ON
```

On the X-Series Signal Analyzer:

Change to Spectrum Analyzer mode, Swept SA measurement.

```
INSTrument:CONFigure:SA:SAN
DISPlay:VIEW:ADVanced:SELect "NORMAL"
FREQuency:CENTer 20GHZ
FREQuency:SPAN 10MHZ
```

Setting Up Waveform File Vector Modulation

In this section, we will load a GSM and a LTE waveform into the VXG to demonstrate the accuracy of the VXG's vector modulation using error vector magnitude (EVM) measurement applications available on the X-Series Signal Analyzer.

The VXG supports all ARB waveforms that are provided on the X-Series sources. This section will use a few of the ARB files that come with the X-Series sources.

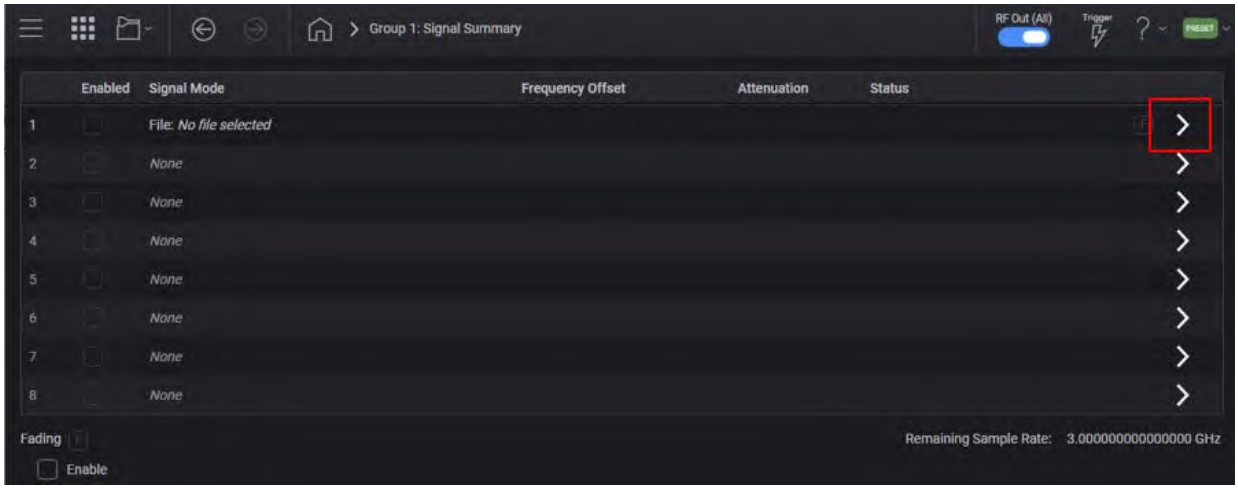
NOTE Ensure the equipment and triggers are properly configured. Refer to "Equipment Setup" on page 13 and "Setting Up Triggers on the X-Series Signal Analyzer" on page 15.

On the VXG:

1. Select **Preset** > **Preset** to set the instrument to a known state.
2. Set Frequency to **20 GHz** and Amplitude to **0 dBm**.
3. Select the **Group 1: Signals** block to open.

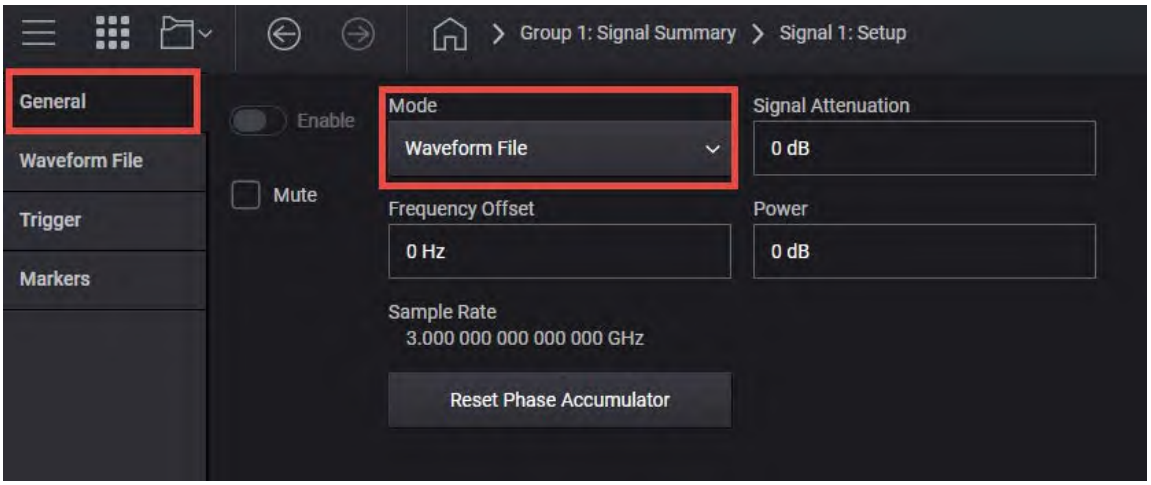
4. Select the arrow for Signal 1 to open the Signal Setup window.

NOTE This screen is only accessible if Option M9484C-8SG (8 virtual signal generators) is installed. For all other option configurations, continue to the next step.



5. In the Vector Modulation Signal Setup:

   a. Select the **Mode** dropdown and set to **Waveform File**.

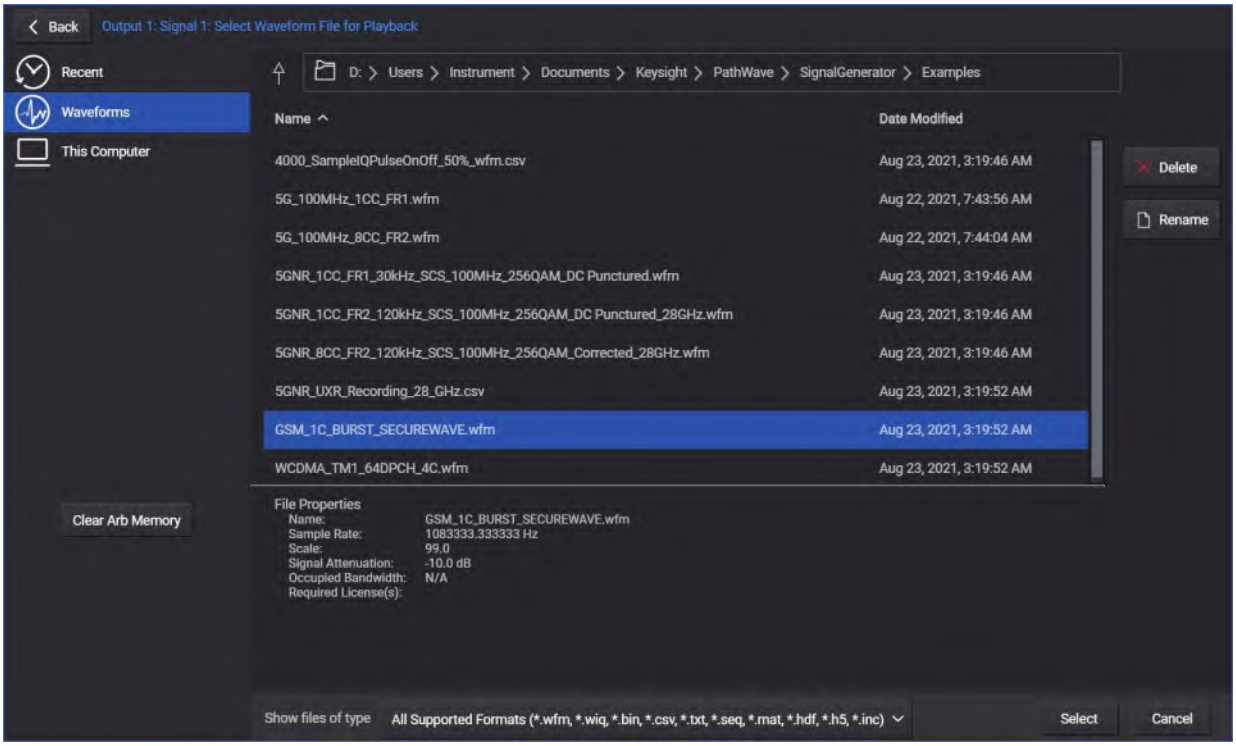


   b. In the left pane, select the **Waveform File** tab.

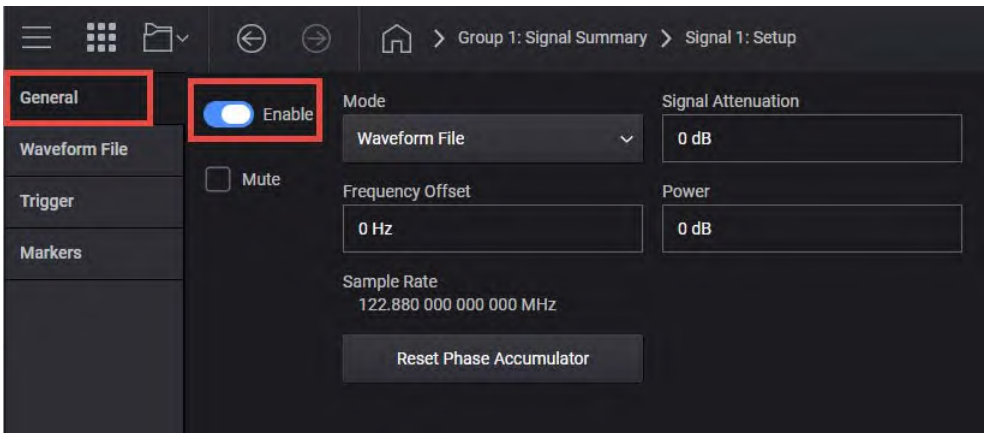
   c. Use File **Select** to navigate to:

      ```
      D:\Users\Instrument\Documents\Keysight\PathWave
      \SignalGenerator\Examples
      ```

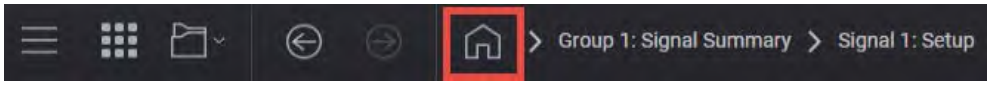
d. Highlight `GSM_1C_BURST_SECUREWAVE.wfm`, then **Select**.



e. Select the **General** tab > **Enable**.



6. Close the Vector Modulation Signal Setup by selecting the **Home** icon at the top of the display.

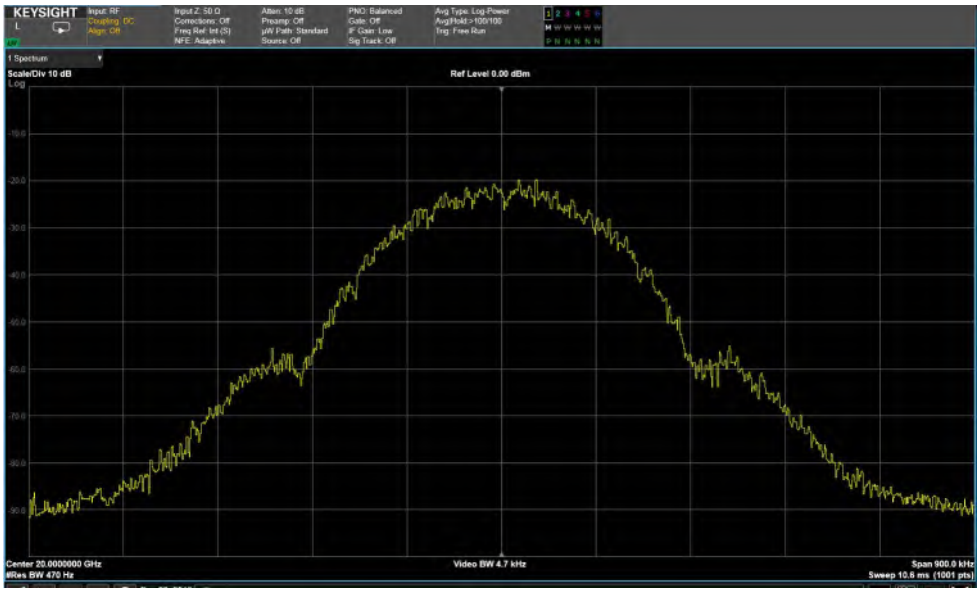


7. Set RF Out to **On** by selecting the numbered channel indicator switch.

This enables the RF Out for the indicated channels, in this case Channel 1 if using a multi-channel VXG.

On the X-Series Signal Analyzer:

1. Select **Mode Preset** to set Spectrum Analyzer mode to a known state.
2. Select **Frequency** and set Center Frequency to **20 GHz** and Span to **900 kHz**.
3. Select **BW** and set Res BW to **470 Hz**.
4. Select **Trace** and set Trace Type to **Max Hold**.
5. Observe the GSM signal.



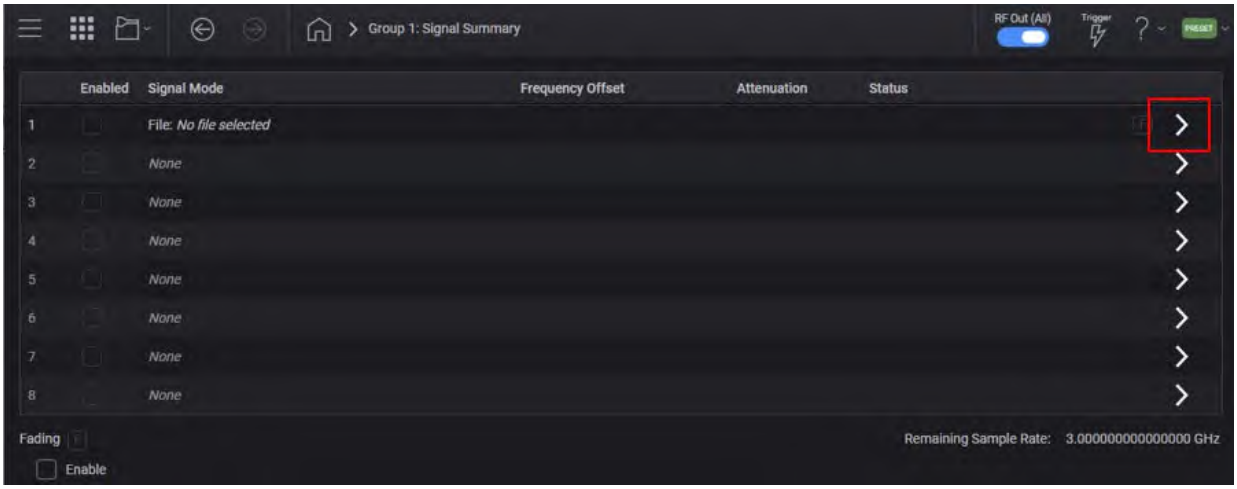
On the VXG:

1. Select the **Group 1: Signals** block to open.

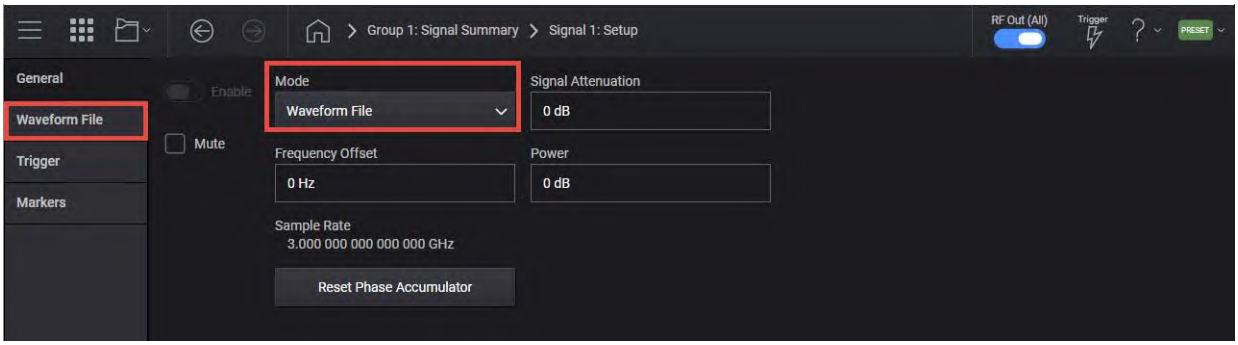
2. Select the arrow for Signal 1 to open the Signal Setup window.

NOTE

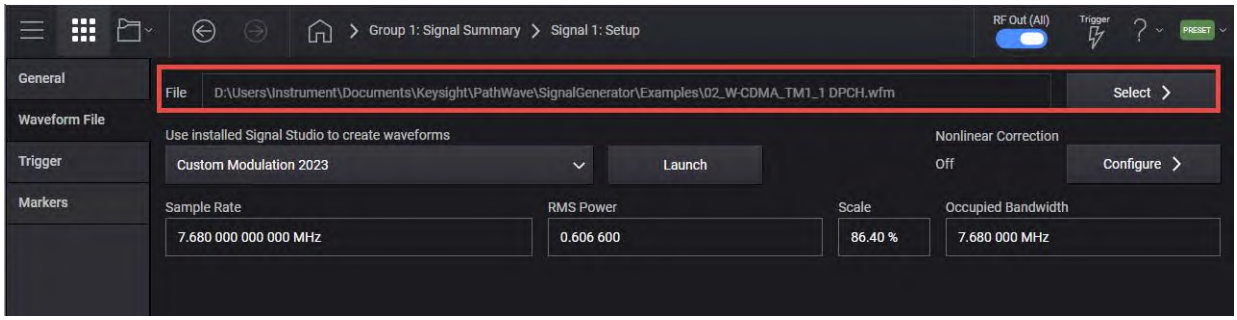
This screen is only accessible if Option M9484C-8SG (8 virtual signal generators) is installed. For all other option configurations, continue to the next step.



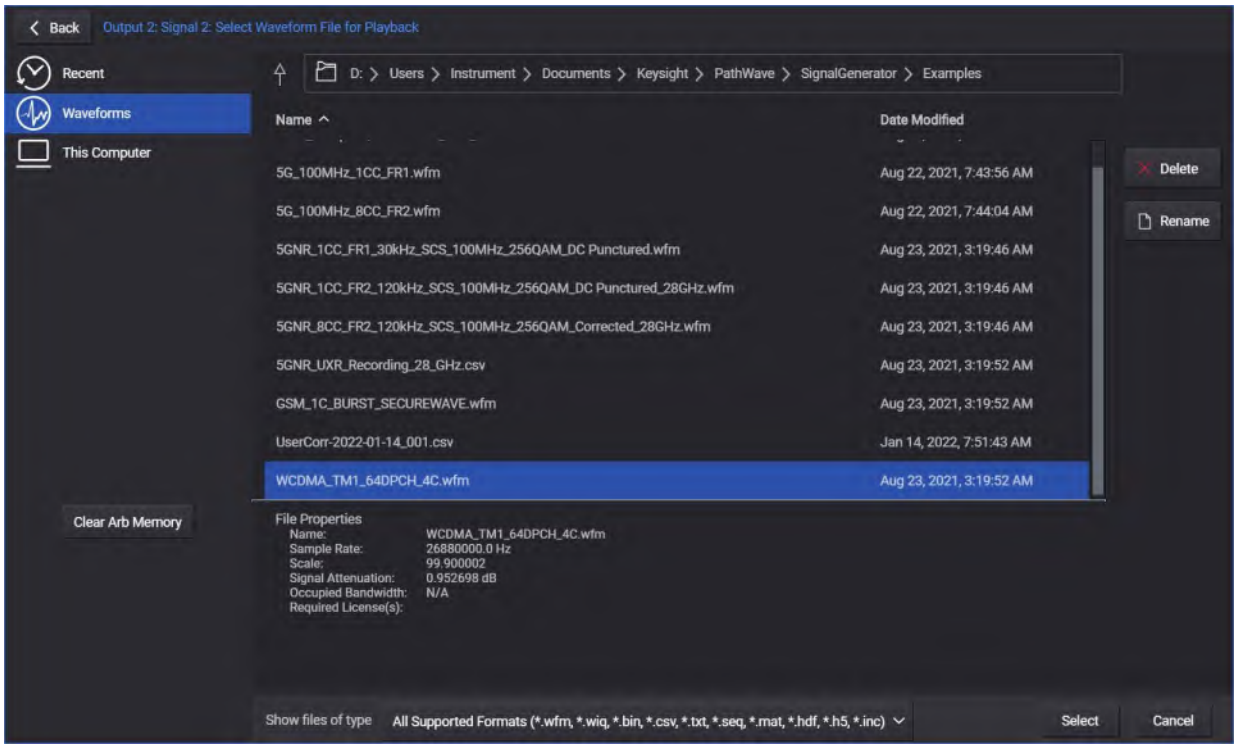
3. Select the **Mode** drop down menu and select **Waveform File** and then select the **Waveform File** tab on the left side.



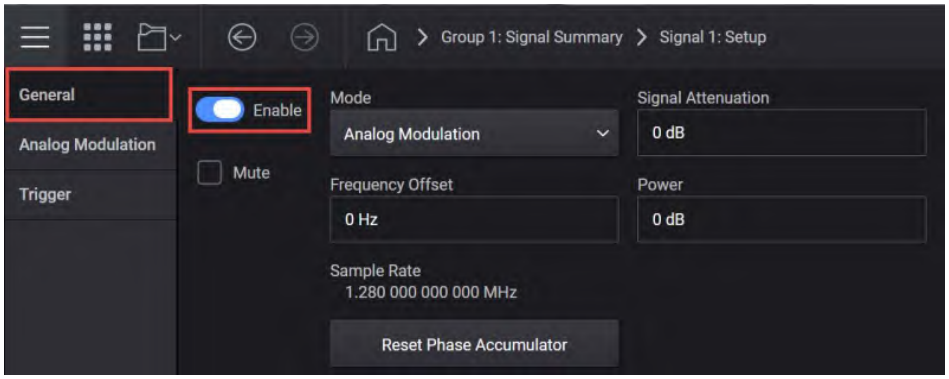
4. Use File **Select** to navigate to:
   `D:\Users\Instrument\Documents\Keysight\PathWave\SignalGenerator\Examples`



5. Set the file extension to All Files (*.*), highlight `WDCMA_TM1_64DPCH_4C.wfm`, then **Select**.



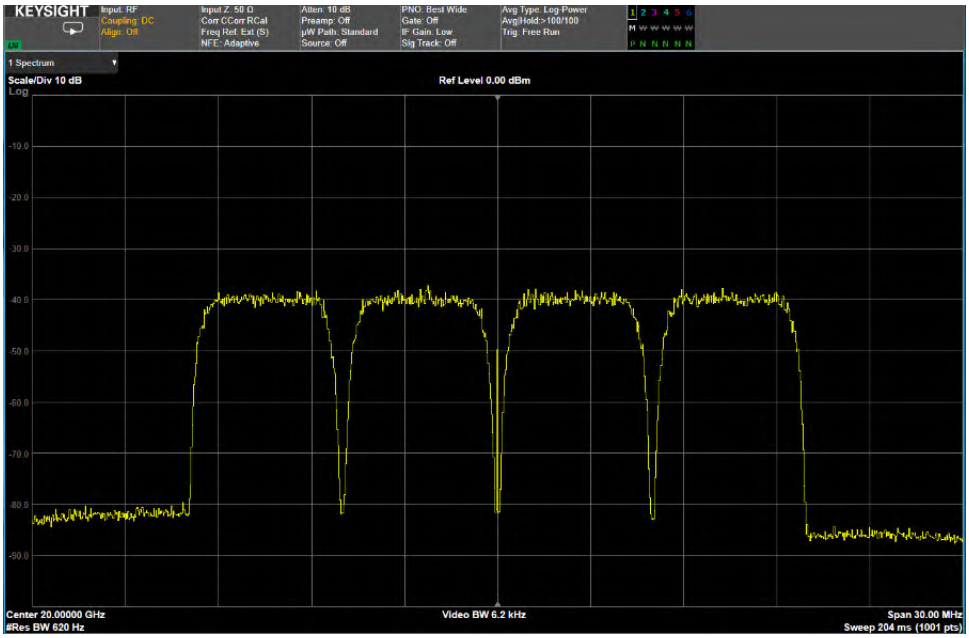
6. Select the **General** tab > **Enable**.



7. Ensure that **RF Out** is on.

On the X-Series Signal Analyzer:

1. Select **BW** and set Res BW to **620 Hz**.
2. Select **Frequency** and set Span to **30 MHz**.
3. Observe the WCDMA signal.



Using the equivalent SCPI commands

On the VXG:

```
SYSTem:PRESet

RF1:FREQuency:CW 20GHZ

RF1:POWer:AMPLitude 0dBm

SIGNal1:MODE WAVeform

SIGN1:WAV " D:\Users\Instrument\Documents\Demo
Waveforms\GSM_1C_Burst.wfm"

SIGNal1 ON

RF1:OUTPut:STATe ON
```

For multi-channel instruments, set RF Out (all) to On.

```
RFALl:OUTPut ON
```

On the X-Series Signal Analyzer:

```
FREQuency:CENTer 20GHZ

FREQuency:SPAN 900KHZ

ACPower:BANDwidth 470 Hz

DISPlay:TXPower:WINDow1:TRACe:MAXHold ON
```

On the VXG:

```
SIGN1:WAV " D:\Users\Instrument\Documents\Demo
Waveforms\WDCMA_TM1_64DPCH_4C.wfm"

SIGNal1 ON

RF1:OUTPut:STATe ON
```

On the X-Series Signal Analyzer:

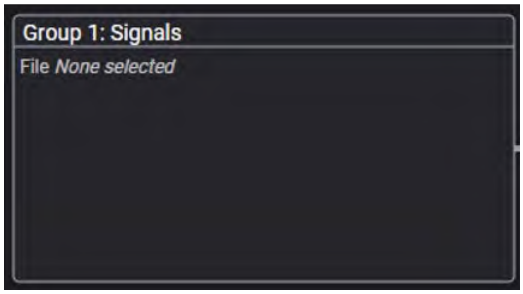
```
ACPower:BANDwidth 620 Hz

FREQuency:SPAN 30MHZ
```

Creating a Waveform Sequence

A waveform sequence (.seq file) is an ordered list of waveform files that are played one after another, with control over repetition, order, and markers. The VXG then steps through the sequence during playback.

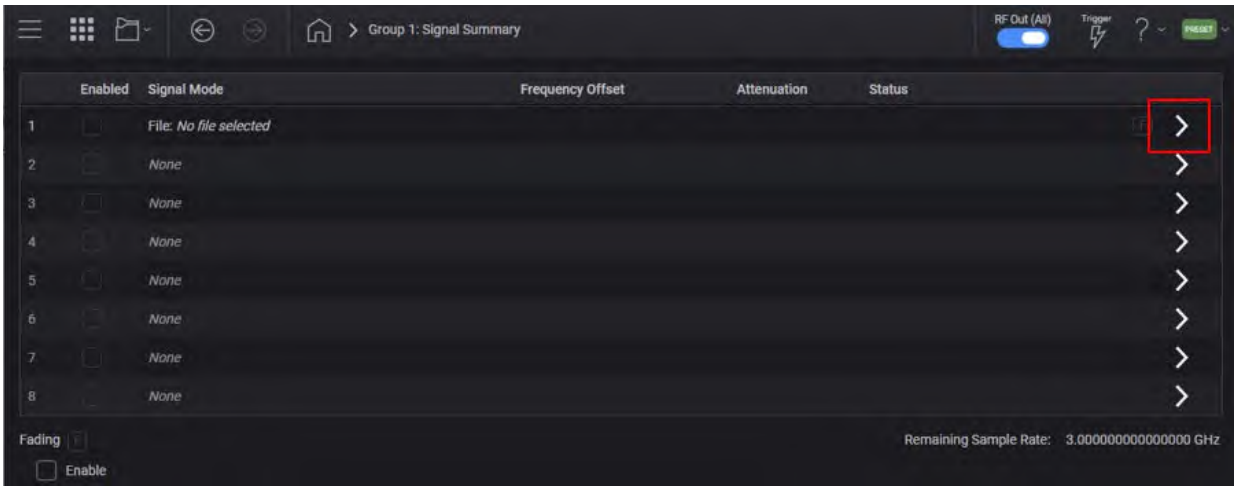
On the VXG:

1. Select **Preset** > **Preset** to set the VXG to a known state.
2. Select the **Group 1: Signals** block to open.

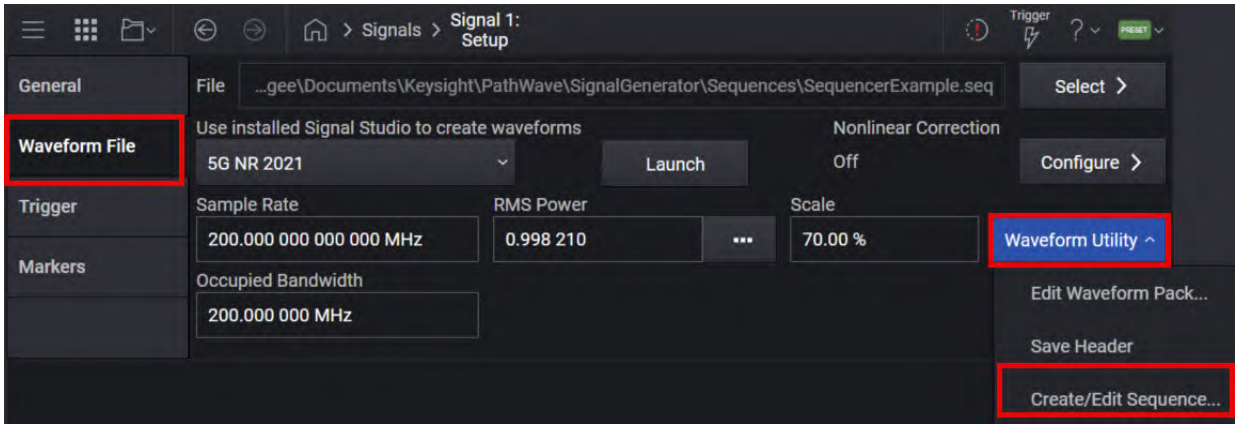


3. Select the arrow for Signal 1 to open the Signal Setup window.

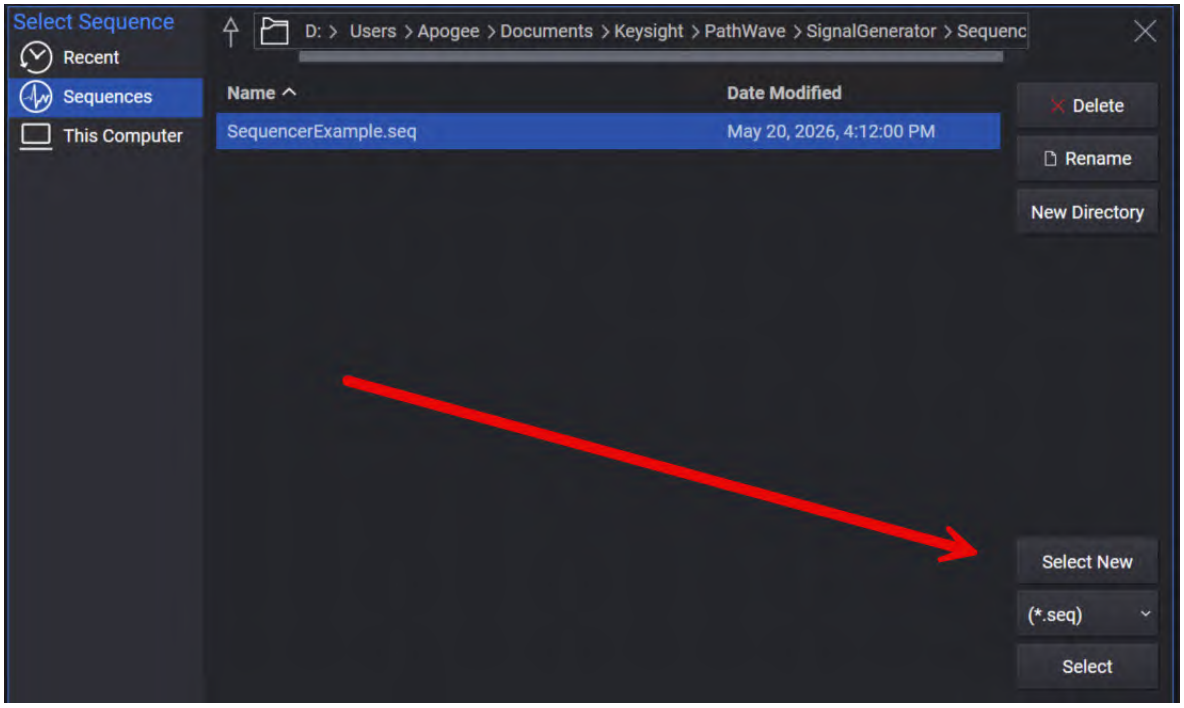
NOTE This screen is only accessible if Option M9484C-8SG (8 virtual signal generators) is installed. For all other option configurations, continue to the next step.



4. Select the **Mode** drop down menu and select **Waveform File** and then select the **Waveform File** tab > **Waveform Utility** > **Create/Edit Sequence**.



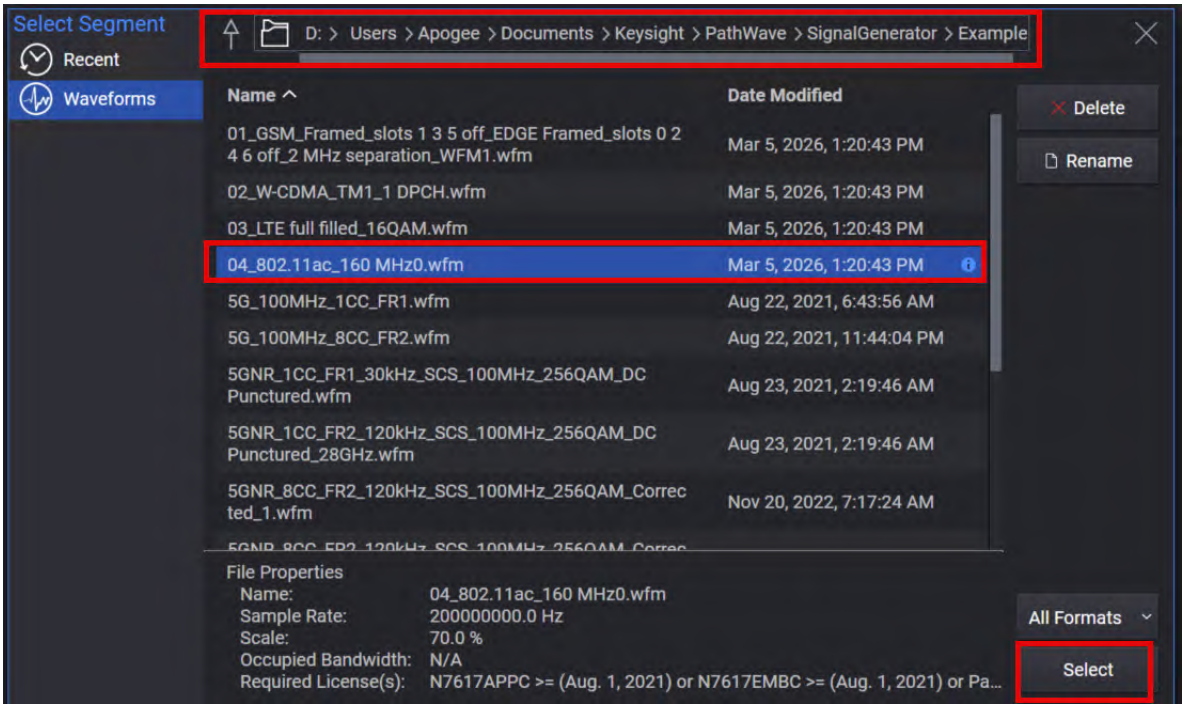
5. Chose Sequence File **Select** > **Select New** to name the new sequence.



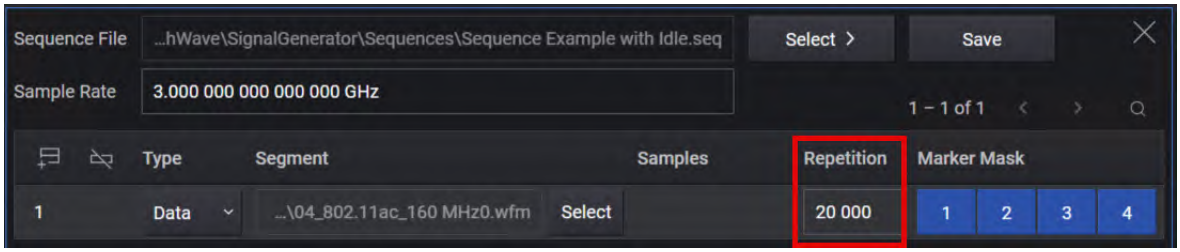
6. In Row 1, set Type to Data > Select.

   For this measurement example, we will choose waveform files from the Examples folders. `D:\Users\Instrument\Documents\Keysight\PathWave\SignalGenerator\Examples`.

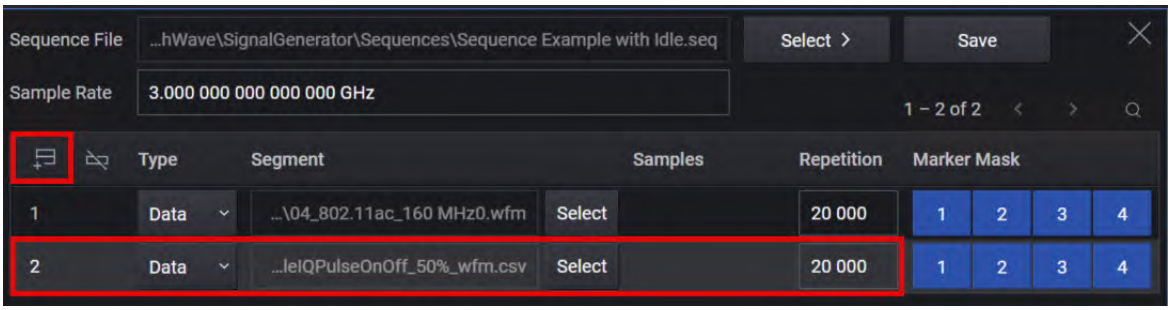
Scroll down and highlight `04_902.11ac_160 MHz0.wfm` > **Select**.



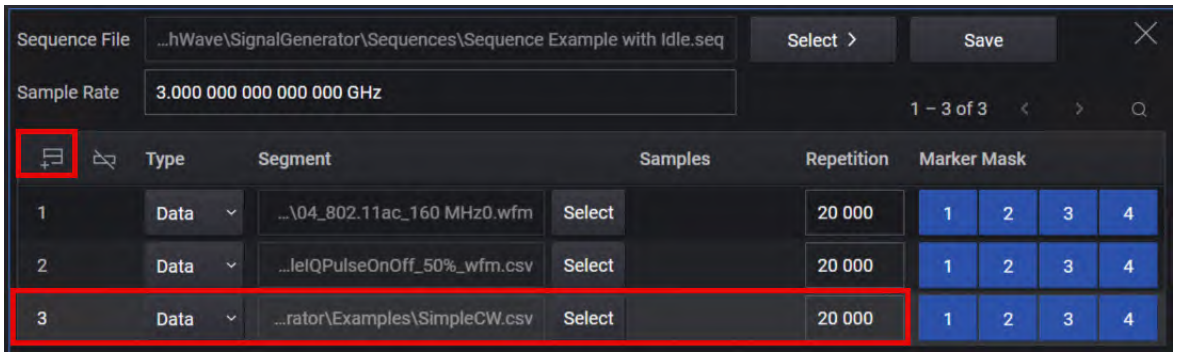
7. Set Repetition to **20 000**.



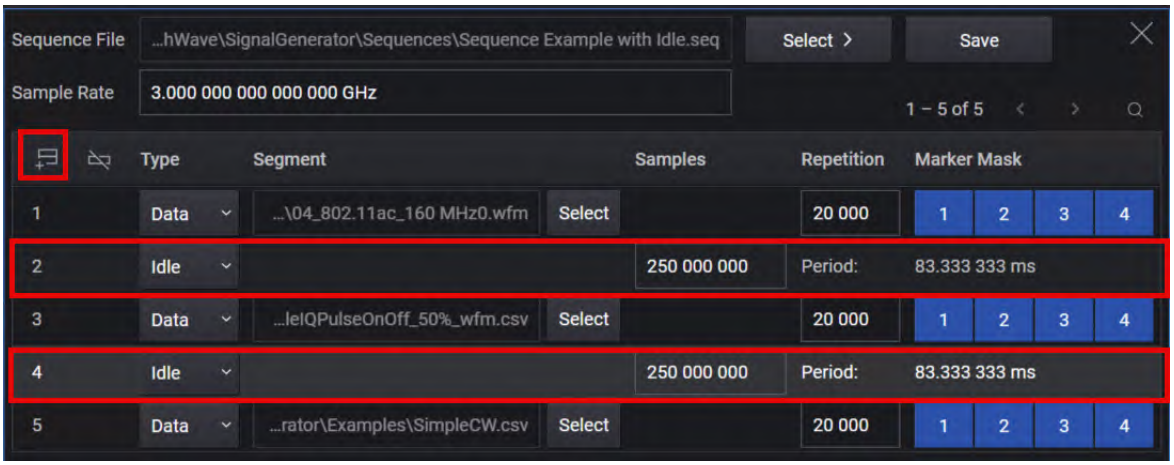
8. Add a new row and **Select** `4000_SampleIQPulseOnOff_50%_wfm.csv` and set Repetition to **20 000**.



9. Add a new row and **Select** `SimpleCW.csv` and set Repetition to **20 000**.



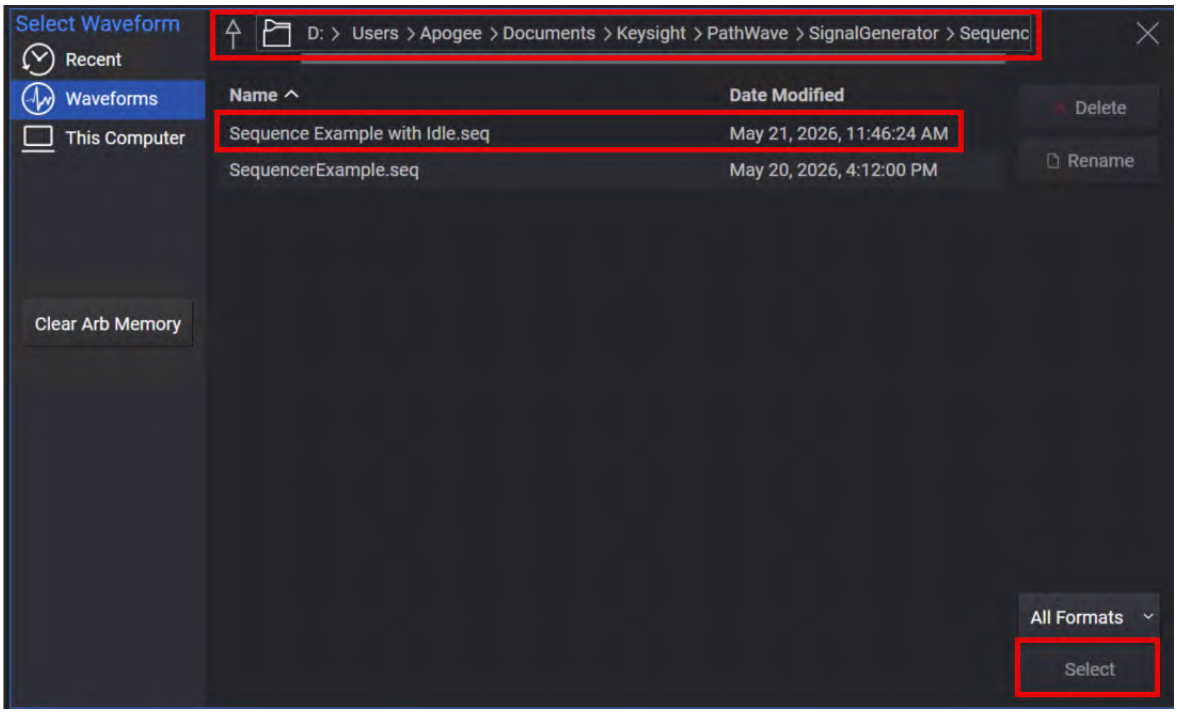
10. Now we will add some Idle time between each Waveform. Highlight **Row 2** and **Add** another row. In Row 2, change Type to **Idle** and enter **250 000 000** Samples which calculates to a Period of 83.333.333 ms. Repeat settings and add a row between rows 3 and 4.



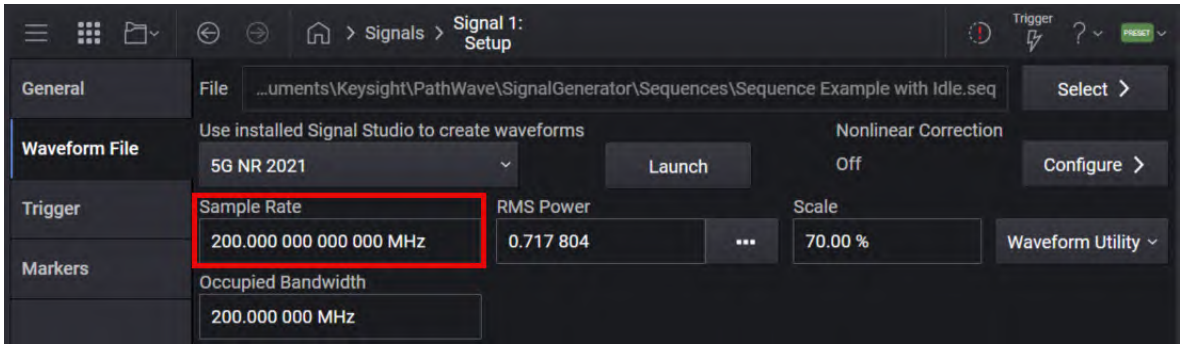
11. Set the Sample Rate to **200 MHz** > **Save**.

    This value must be changed for the correct sample rate for the Waveforms to be sequenced.

12. In the Signal Setup dialog box, use File **Select** to navigate to the Sequences Files and select `Sequence Example with Idle.seq`.

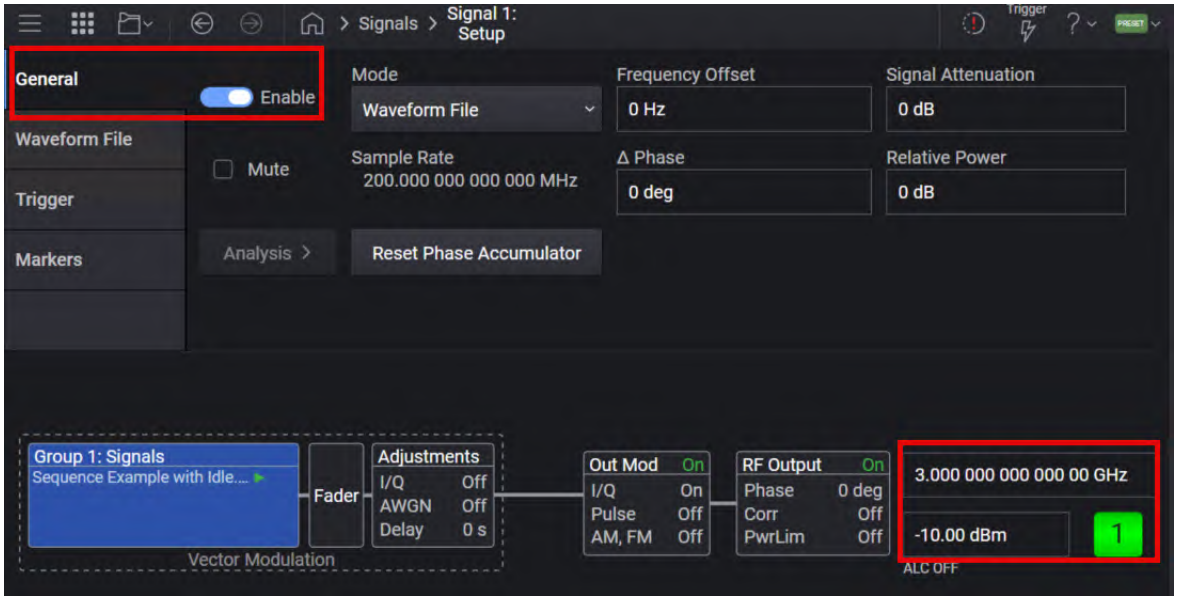


Notice that the Sample Rate in the Signal 1 Setup has changed to 200 MHz, the value we set in the Sequence Editor.



13. Select the **General** tab and **Enable** the waveform play back.

14. Set Frequency to **3 GHz** > RF Output to **-10 dBm** > turn on Channel **1**.



On the X-Series Signal Analyzer:

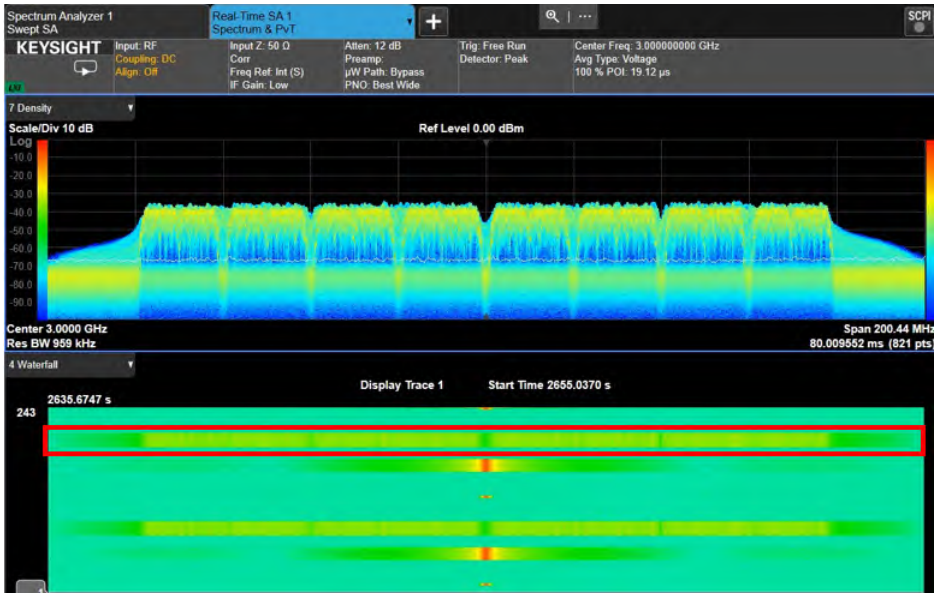
1. Select **Mode/Meas** > **Real Time Spectrum Analyzer Mode** > **Spectrum & PVT** Measurement > **Density Spectrum** View.

NOTE

If accessing the instrument via a Remote Desktop connection, select the Screen tab (at the top of the display) to open the Mode/Measurement/View Selector window.

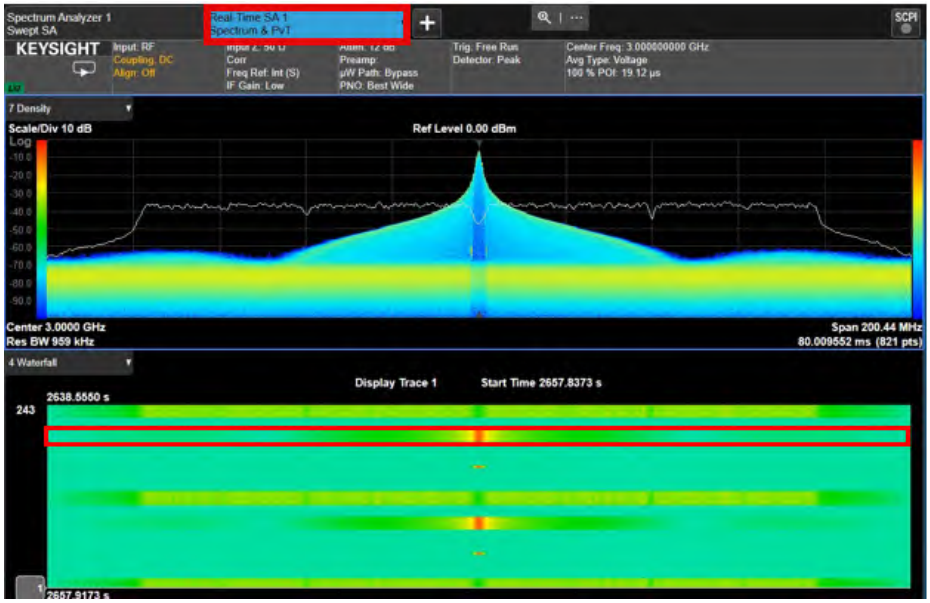
2. Select **Frequency** and set Center Frequency to **3.0 GHz** and view the results of the waveforms playing in a sequence.

Figure 1-6 Results of 04_802.11ac_160 MHz0.wfm



The Density view on top is showing the 8 160 MHz carriers. The Highlighted portion of the Waterfall view shows the portion related to the 160 MHz carriers.

Figure 1-7 Results of 4000_SampleIQPulseOnOff_50%_wfm.csv

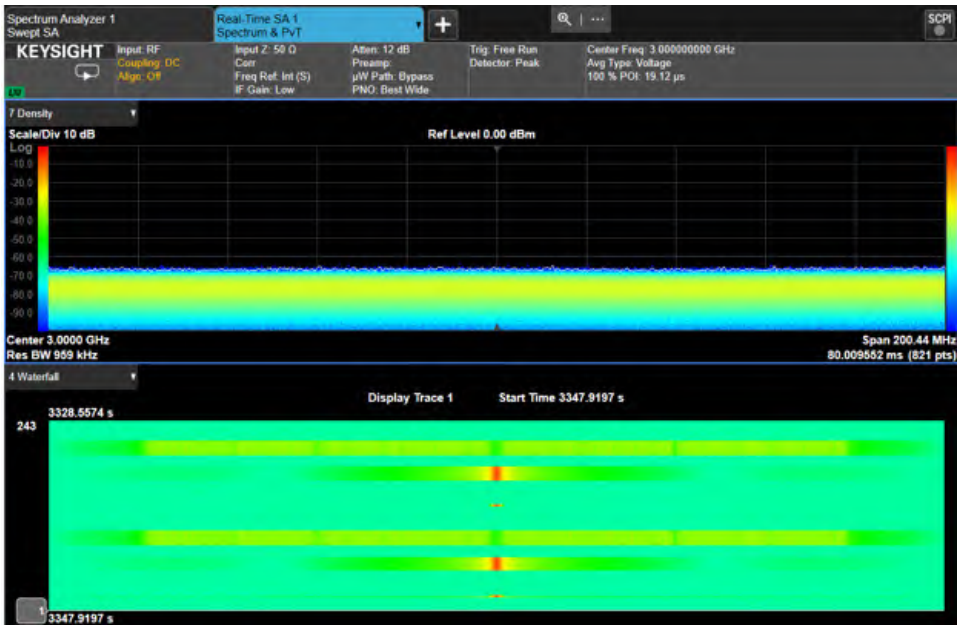


The Density view on top shows the Pulse. The Highlighted portion of the Waterfall view shows the portion related to Pulse.

Figure 1-8 Results of SimpleCW.csv

The Density view on top shows the CW signal, it is a very faint yellow signal in the center of the display. The Highlighted portion of the Waterfall view shows the portion related to the CW signal.

Figure 1-9 Idle Times added to the sequencer file



Idle times are display are solid green between each waveform in the Waterfall view.

2 Corrections

- "Corrections/De-embedding Using PathWave N7653APPC Software" on page 136
  - "Using a Spectrum Analyzer to Make the Corrections Measurement" on page 142
  - "Using a Power Meter to Make the Corrections Measurement" on page 149
  - "Adding Fixture Blocks using s2p Files" on page 136
- "Instrument Nonlinear Correction" on page 155
  - "Setting Up an INC 1CC 4 GHz, 10 dBm EVM Measurement" on page 155
  - "Setting Up an INC 8CC 28 GHz, 10 dBm EVM Measurement" on page 167
- "Phase Coherency and Alignment Between Channels" on page 176

Corrections/De-embedding Using PathWave N7653APPC Software

De-embedding is used to remove the effects of the test fixtures and cabling from the measurement results. De-embedding uses a model of the test fixture and mathematically removes the fixture characteristics (cables, connectors and other passive components) between the source and the device under test (DUT). Once the desired topology has been characterized, its effects can be removed from the output signal, moving the effective reference plane to the point at which the power sensor was connected.

Blocks can be added from supported file formats (.s2p, .csv, .uflat) or by direct measurement, using one of the supported power sensors (power meter, spectrum analyzer, or a network analyzer).

NOTE The VXG must have the N7653APPC PathWave Automatic Channel Response Correction and S-parameter De-embedding license installed.

Adding Fixture Blocks using s2p Files

Amplitude and phase can be corrected by adding multiple s2p files as Fixture Blocks.

An s2p file (also known as a Touchstone file) is an ASCII text file used for documenting the n-port network parameter data and noise data of linear active devices, passive filters, passive devices, or interconnect networks. Each record contains 1 stimulus value and 4 S-parameters (total of 9 values)

The first line in the figure below (# GHz DB R 50) designates:

- (GHz) designates the frequency in Hz, kHz, MHz, or GHz
- (S) the measurements are in S parameters (rather than Y or Z)
- (DB) the values are given in decibel/angle. Instead of DB, you can have RI (real/imaginary) or MA (magnitude/angle)
- (50) the characteristic impedance is 50 ohms

```
!S2P File: Measurements: S11,S21,S12,S22:
# GHz S DB R 50
   9.0 -100 -100  10.0 -100  10.0 -100 -100 -100
   9.5 -100 -100   5.0 -100   5.0 -100 -100 -100
  10.0 -100 -100   0.0 -100   0.0 -100 -100 -100
  10.5 -100 -100  -5.0 -100  -5.0 -100 -100 -100
  11.0 -100 -100 -10.0 -100 -10.0 -100 -100 -100
```

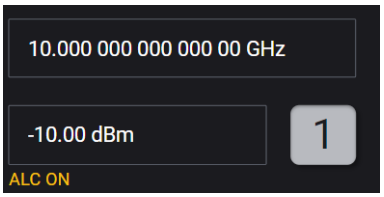
NOTE If there is not a first line header, the default format is GHz, S-parameters, and magnitude/angle.

1. Create s2p files in Notepad in the format shown above.

TIP An s2p example file is included on the VXG. Go to

`D:\Users\Instrument\Documents\Keysight\PathWave\SignalGenerator\Examples\simpleAt10GHzInDB.s2p`

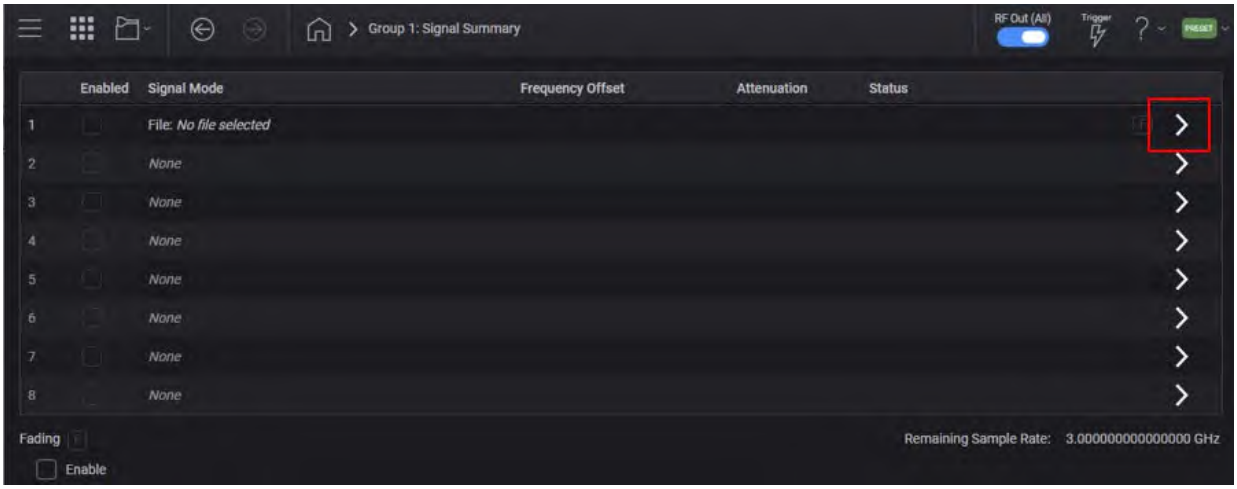
2. Select **Preset** > **Preset** to set the instrument to a known state.
3. In the VXG Output area, set the Frequency to **10 GHz** and **Power** to –10 dBm.



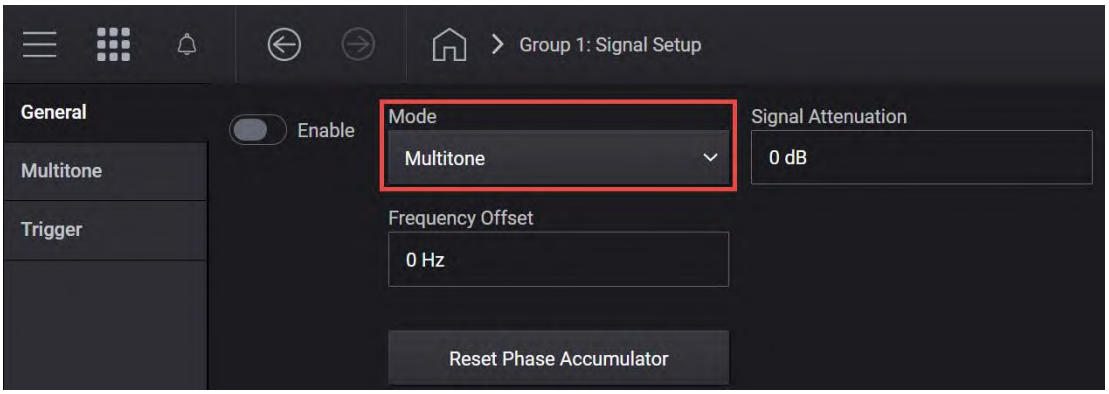
4. Select the **Group 1: Signals** block to open.

5. Select the arrow for Signal 1 to open the Signal Setup window.

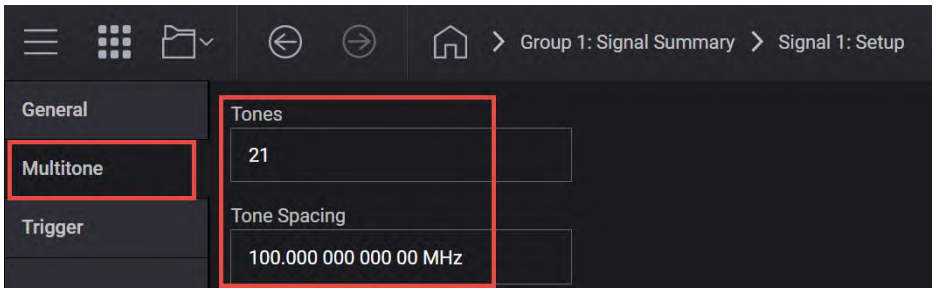
NOTE This screen is only accessible if Option M9484C-8SG (8 virtual signal generators) is installed. For all other option configurations, continue to the next step.



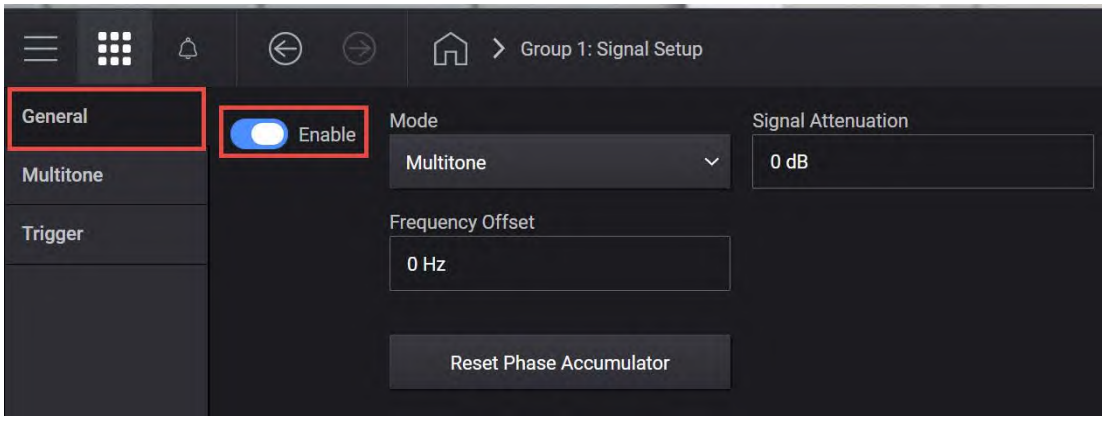
6. Select the **Mode** dropdown and select **Multitone.**



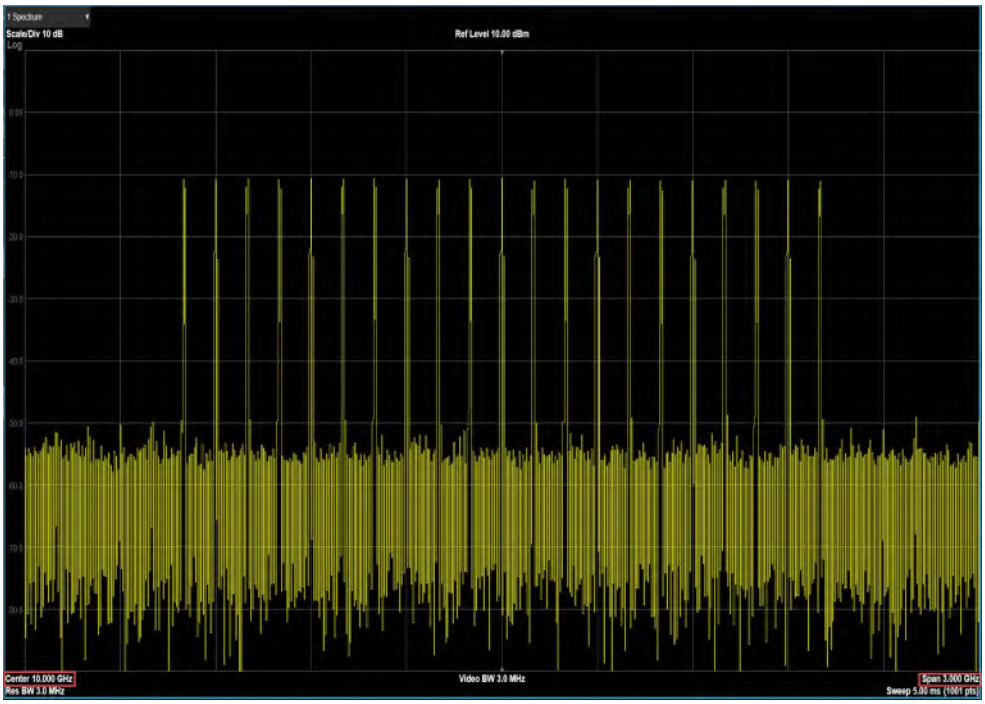
7. From the left pane, select **Multitone** and set:
   - Tones to **21**
   - Tone Spacing to **100 MHz**



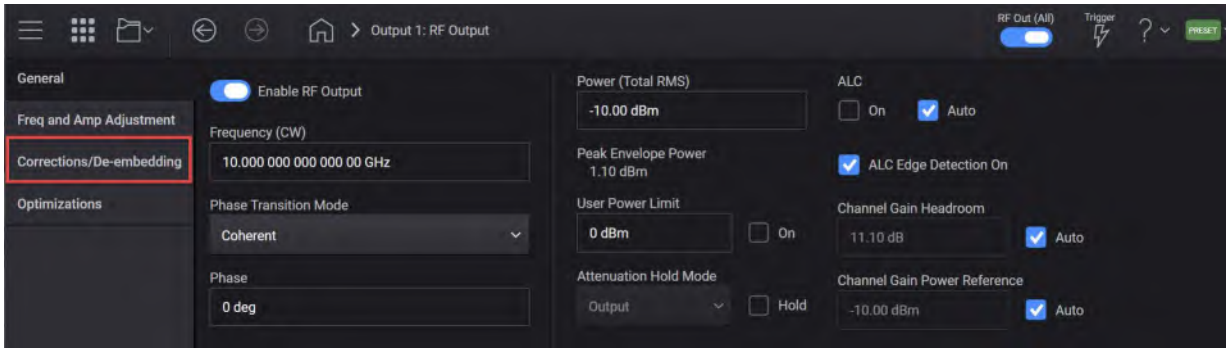
8. Select the **General** tab > **Enable**.



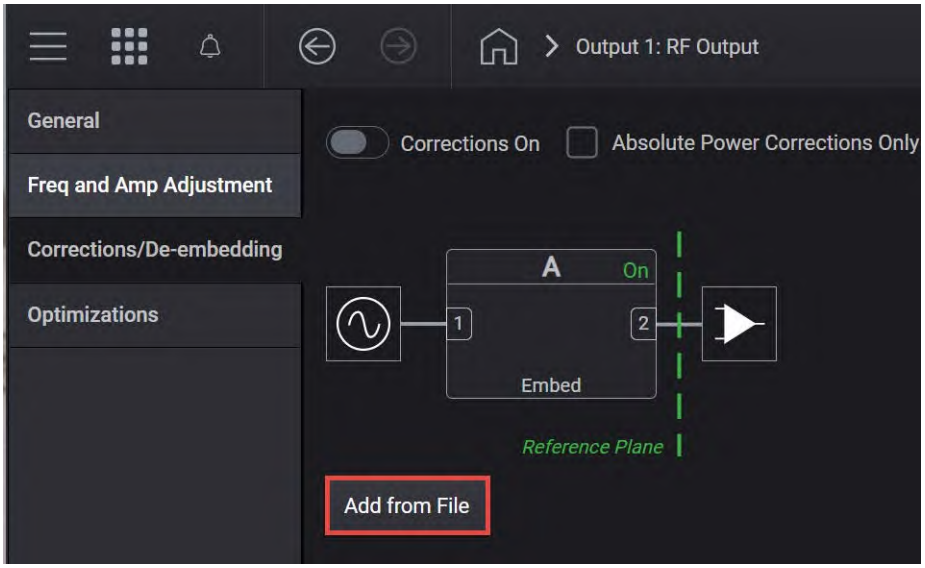
9. Close the Signal block by selecting the **Home** icon, and then select **RF Out** to turn on.
10. On the X-Series Signal Analyzer spectrum analyzer in Spectrum Analyzer Mode:
    - Select **Mode Preset** to set Spectrum Analyzer mode to a known state.
    - Set the Center to **10 GHz**
    - Set Span to **3 GHz**



11. On the VXG, select the **RF Output** block > **Corrections/De-embedding**.



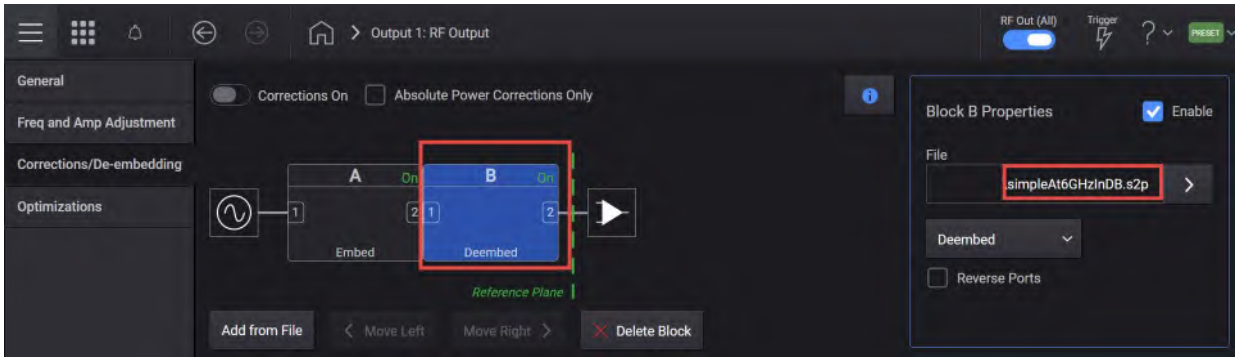
12. In the Corrections Setup dialog, select **Add from File**.



13. Navigate and select the s2p file and then **Select**.

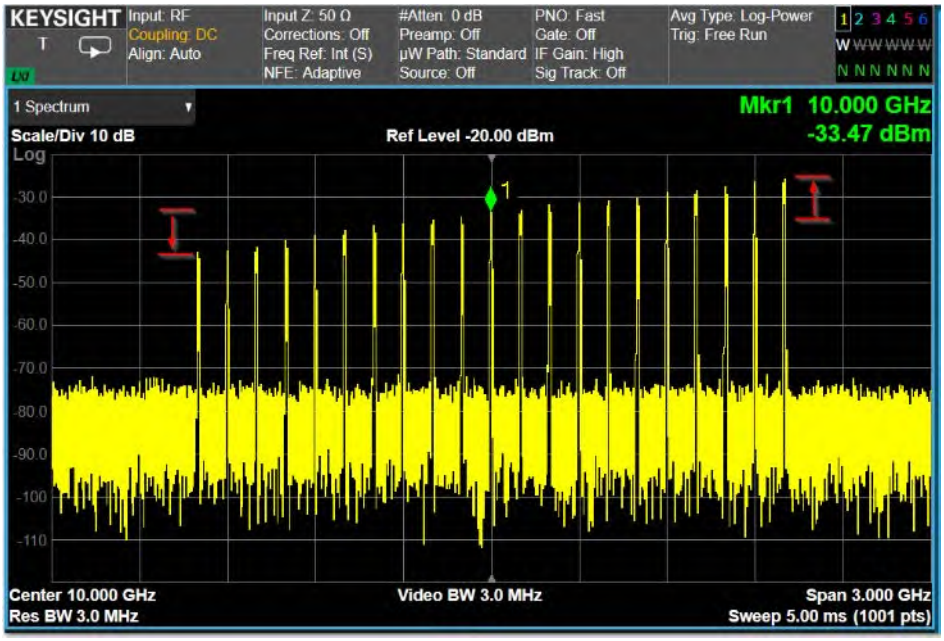
```
D:\Users\Instrument\Documents\Keysight\PathWave
\SignalGenerator\Examples\simpleAt10GHzInDB.s2p
```

Notice that Block B is added in the Correction Setup diagram.

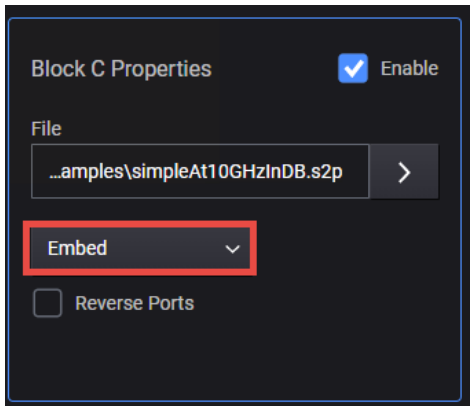


14. Turn **Corrections On**.

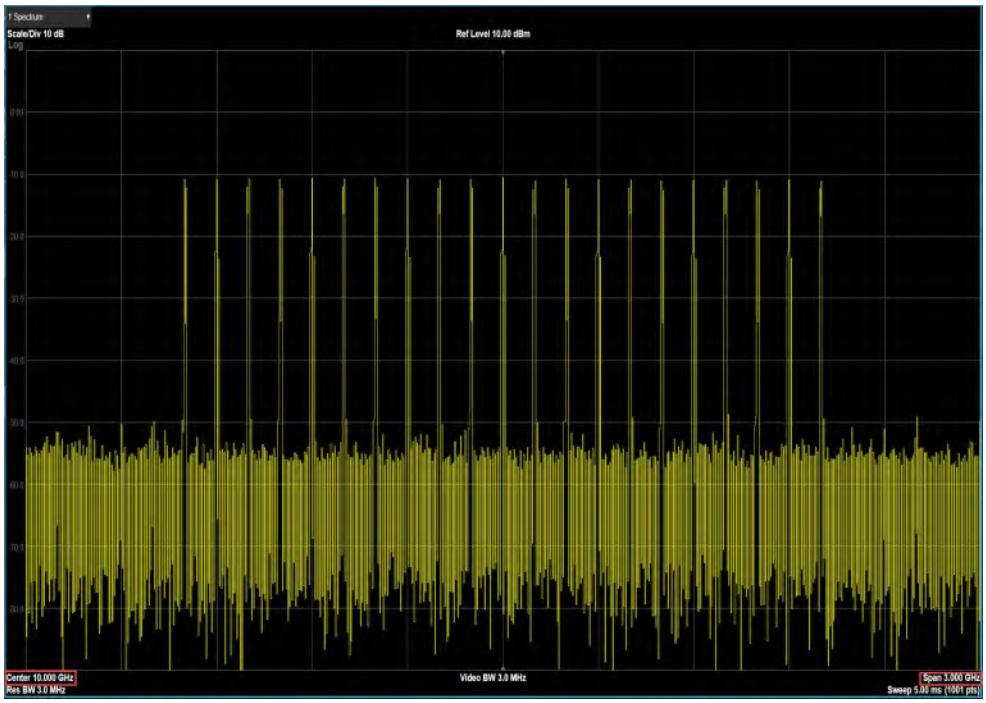
View the results on the signal analyzer. Observe how the .s2p file has impacted the signal.



15. Add a third block using the same file name as shown in the steps above. Under Block C Properties, set to **Embed**.



View the results on the X-Series spectrum analyzer. Notice that the corrections are no longer shown. This is because the de-embedded corrections applied in Block B cancel the embedded corrections applied in Block C.



Using a Spectrum Analyzer to Make the Corrections Measurement

When using a spectrum analyzer, it must be locked to the VXG Frequency Reference. This is important because the power measurement can be inaccurate due to a narrow resolution bandwidth (RBW) used in the spectrum analyzer. Supported Keysight X-Series signal analyzers are:

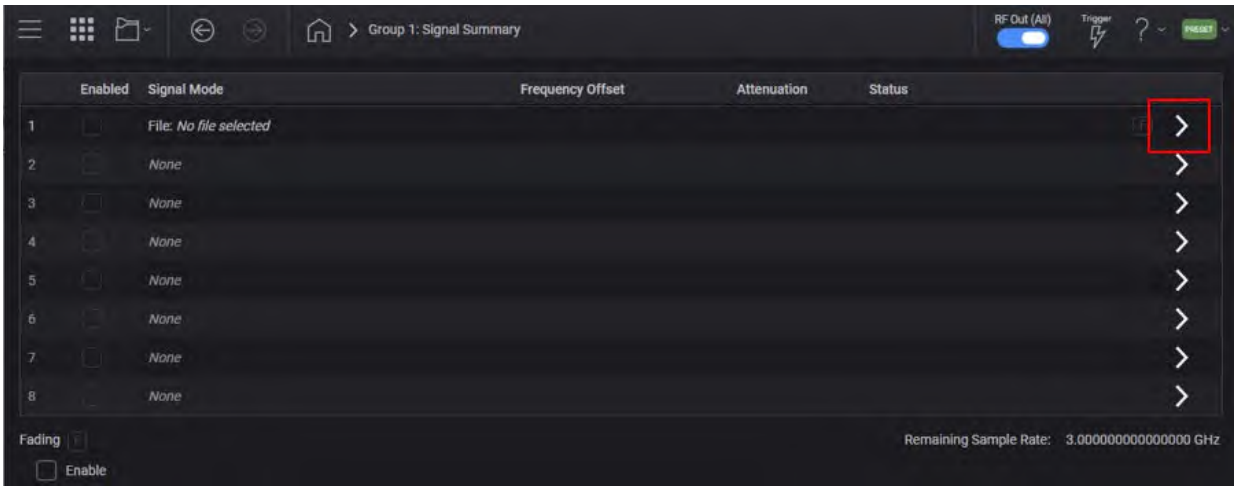
– N9000A/B CXA

– N9010A/B EXA
– N9020A/B MXA
– N9030A/B PXA
– N9040B and N9041B UXA

On the VXG:

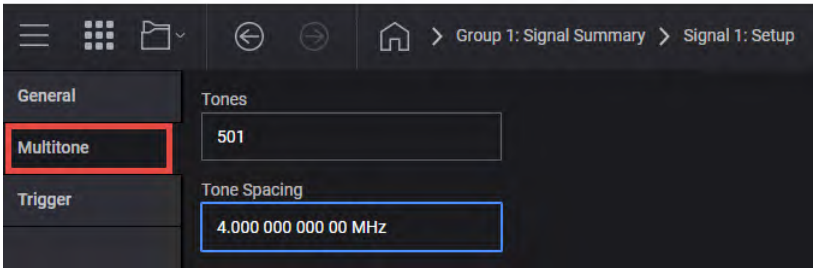
1. Connect the VXG 10 MHz Out to the N90x0A/B Ext Reference In.
2. Connect cable or DUT between the VXG RF Out and the signal analyzer RF in.
3. Select **Preset** > **Preset** to set the instrument to a known state.
4. Set the Frequency to **12 GHz** and Amplitude to **–10 dBm**.
5. Select the **Signal** block to open.
6. Select the arrow for Signal 1 to open the Signal Setup window.

NOTE This screen is only accessible if Option M9484C-8SG (8 virtual signal generators) is installed. For all other option configurations, continue to the next step.



7. Select the **Mode** dropdown and select **Multitone**.

8. Select the **Multitone** tab from the left pane and then, configure the signal to have **501 Tones** with **4 MHz Tone Spacing**, then select the **General** tab > **Enable**.



9. Select the **Home** icon and set RF Out to **On** by selecting the numbered channel indicator switch.

   This enables the RF Out for the indicated channels, in this case Channel 1 if using a multi-channel VXG.

On the Signal Analyzer:

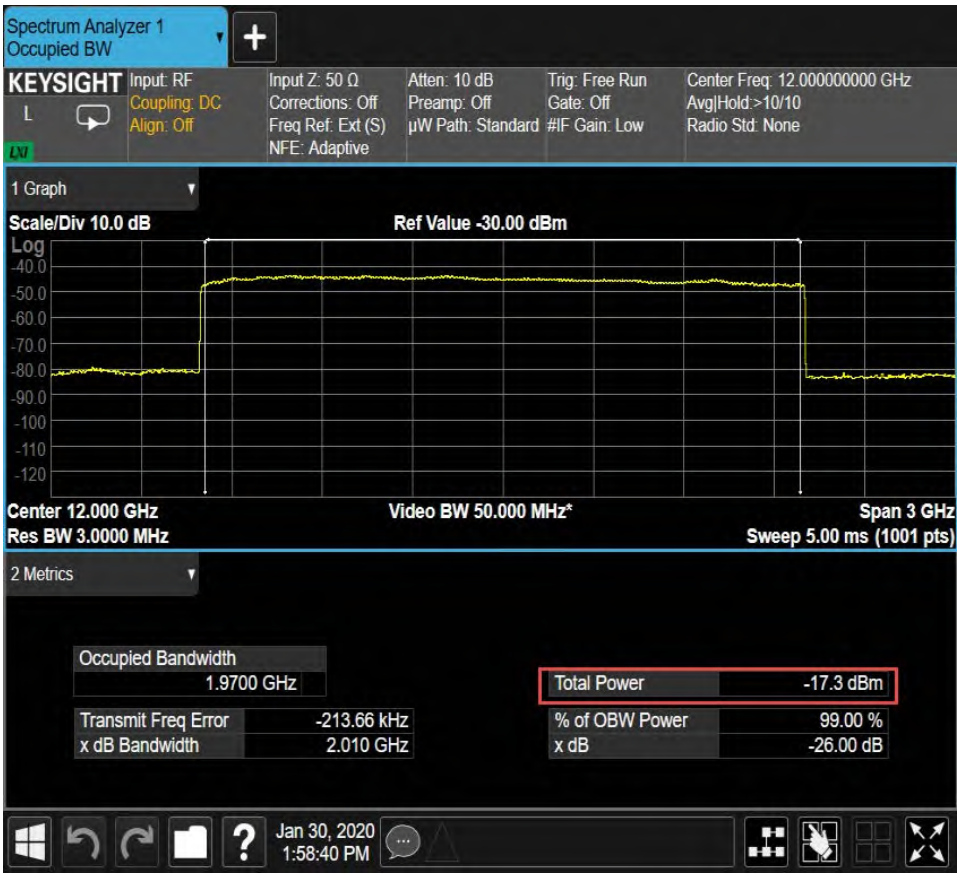
1. From the Menu Panel (on the top right of the display), select **Mode/Meas** > **Spectrum Analyzer** mode > **Occupied BW** Measurement > **OBW Results** View.

NOTE

If accessing the instrument via a Remote Desktop connection, select the Screen tab (at the top of the display) to open the Mode/Measurement/View Selector window.

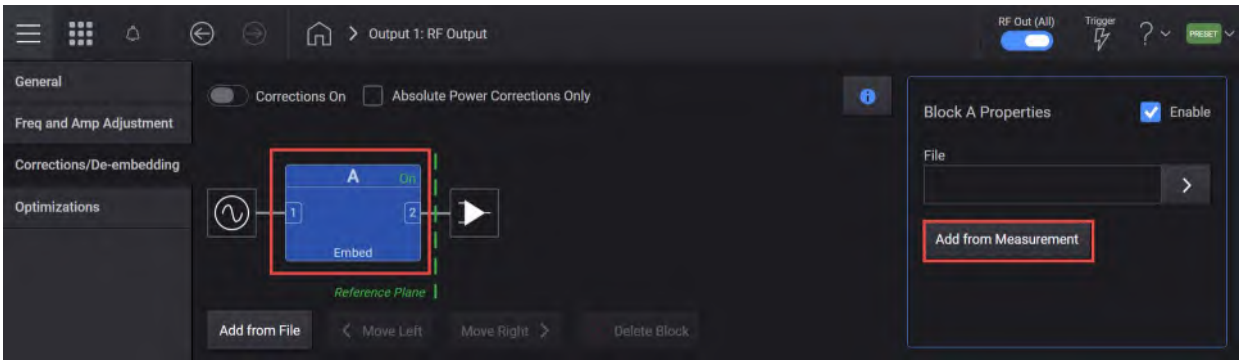
2. Set the Center Frequency to **12 GHz** and the Span to **3 GHz**.

   Observe how the fixturing is impacting this signal, including the flatness and the total power.

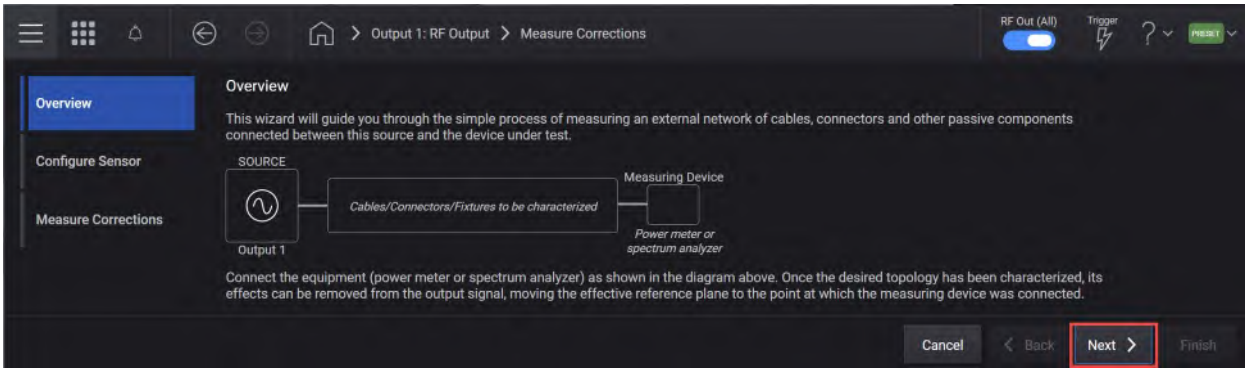


On the VXG:

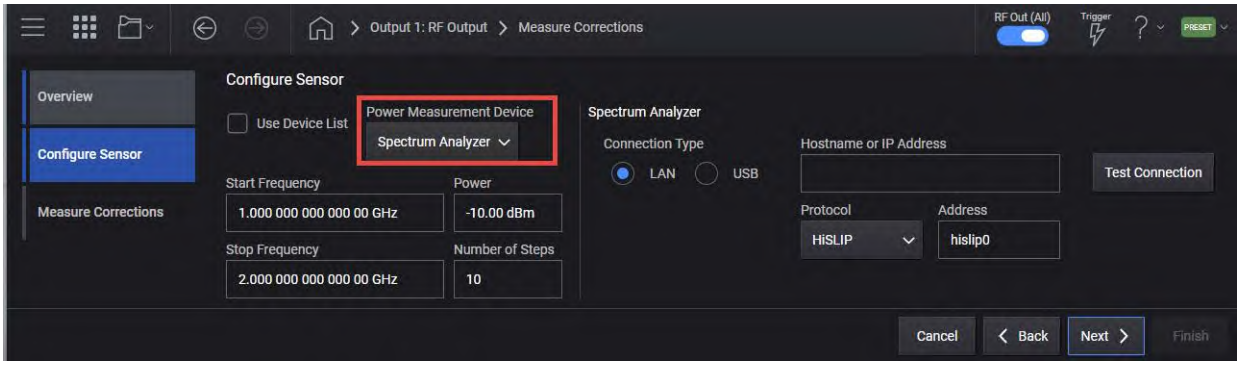
1. Select the **RF Output** block > **Corrections/De-embedding** tab to open the Correction Setup.
2. Select **Block A** to highlight > **Add from Measurement** to open the Measure Corrections Block Wizard.



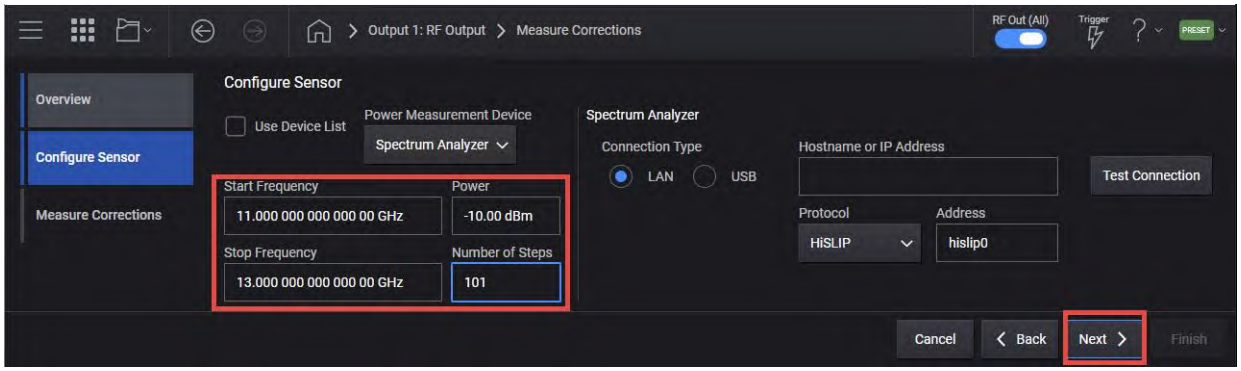
3. Connect the power sensor (in this case, the X-Series signal analyzer) as shown in the diagram below. After reading the overview, select **Next** to move to the Configure Sensor setup.



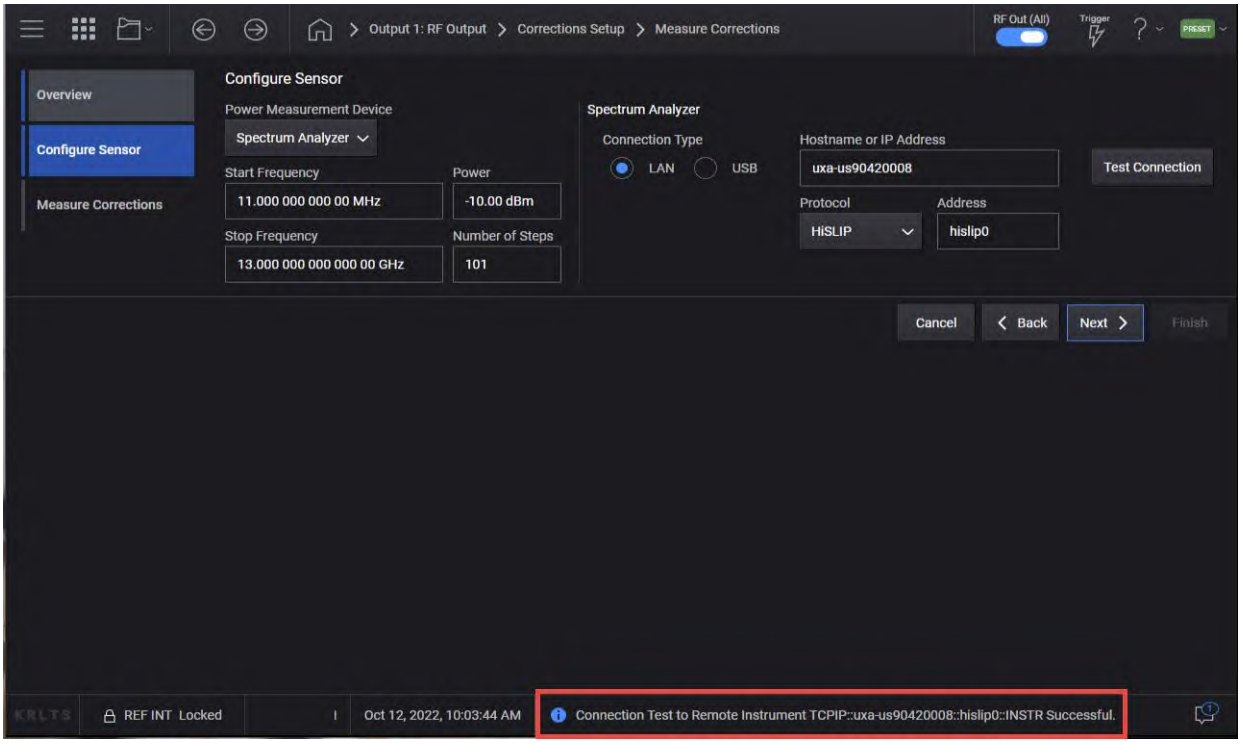
4. In the Configure Sensor setup, select the **Power Measurement Device** dropdown and select **Spectrum Analyzer**.



5. Set the Start and Stop Freq, Amplitude, and the Num Steps. For this example Start Freq **11 GHz**, Stop Freq **13 GHz**, Amplitude to the highest power used in your measurement (For this example, **-10 dBm**, which we already set in the main window. If you change the value here, it will update the value in the main measurement window.) and Number of Steps to **101**.

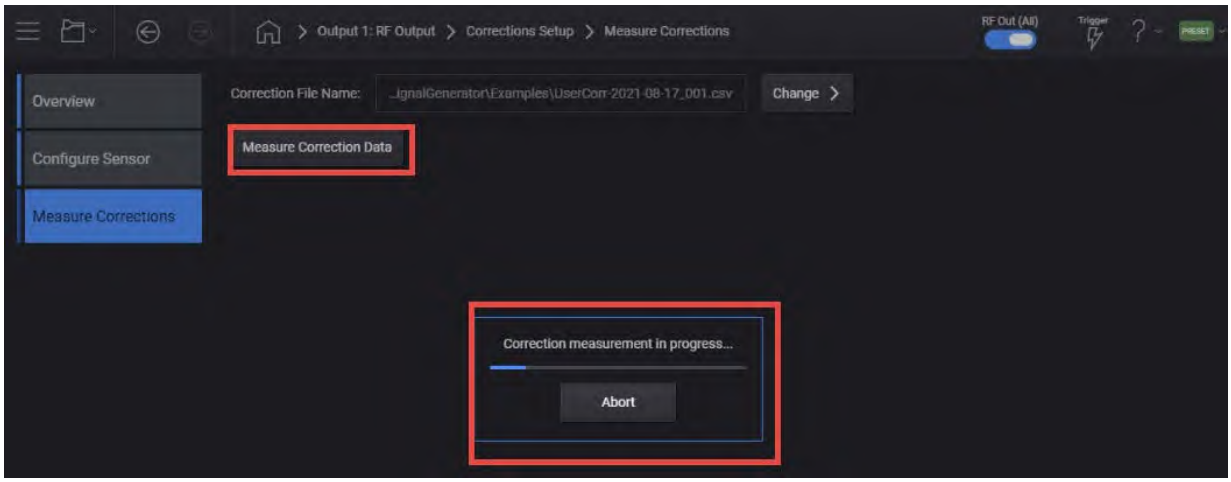


6. Set Connection Type to **LAN**, enter the LAN Address and set the Protocol parameters to **HiSLIP,** and then select **Test Connection**.



7. Once you are successfully connected, select **Next** to move to the Measure Corrections step.
8. Select **Measure Correction Data**.

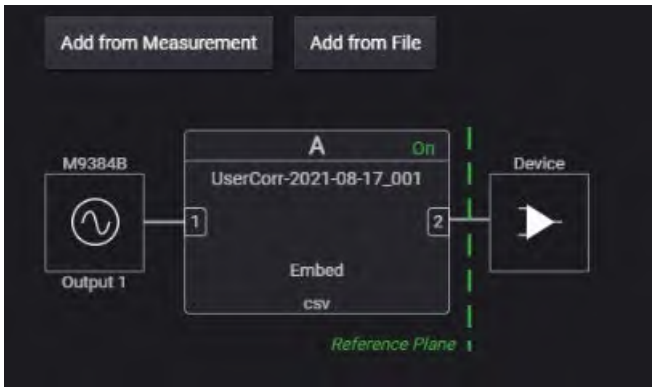
During the measurement, the VXG outputs a CW between the Start and Stop Frequencies for the specified number of steps and output power. It will take some time to measure all 101 points, and the progress is indicated by the blue bar. You can watch the signal analyzer as it steps through this process.



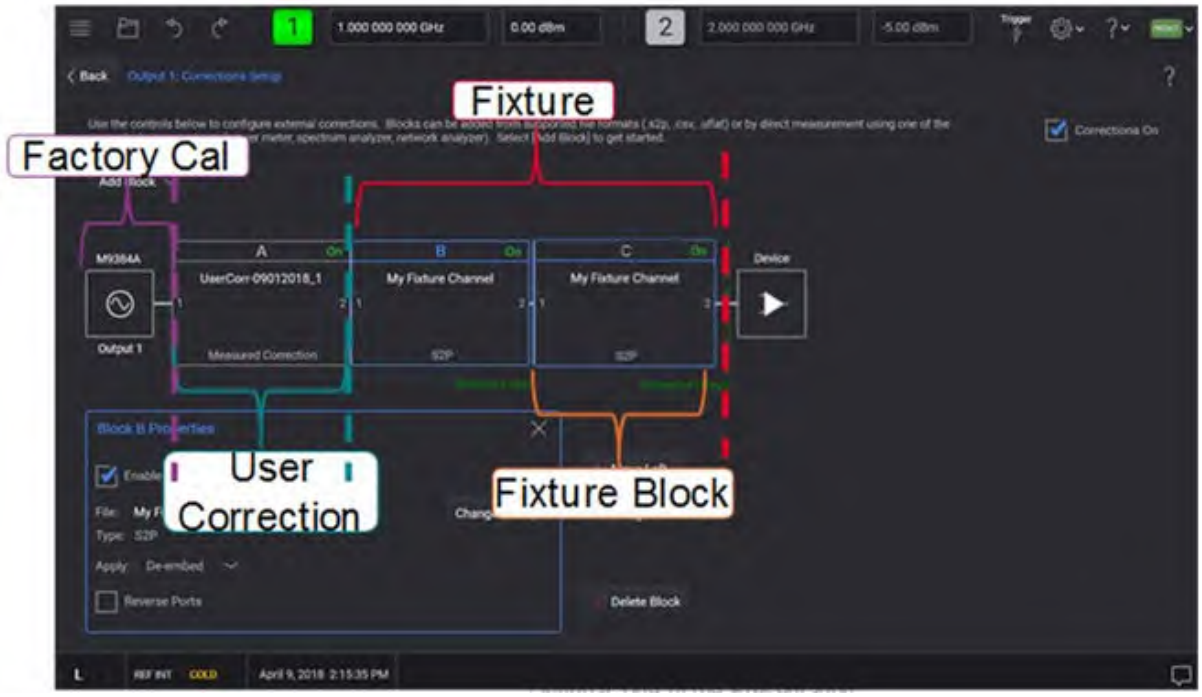
The measurement results are saved to a csv file using an automatically generated file name.

9. When the measurement is complete, select **Finish**, then select **Corrections On**.

The output csv file is set to Block A.



Block A is dedicated for User Correction. The image below shows how blocks are assigned in the User Correction and Fixture block.



10. On the signal analyzer, **Restart** (on the left corner of the user interface) the measurement (because it is applying averaging). Observe how the measured corrections impacted the signal. You can easily toggle Corrections on and off on the VXG to see the difference.

Using a Power Meter to Make the Corrections Measurement

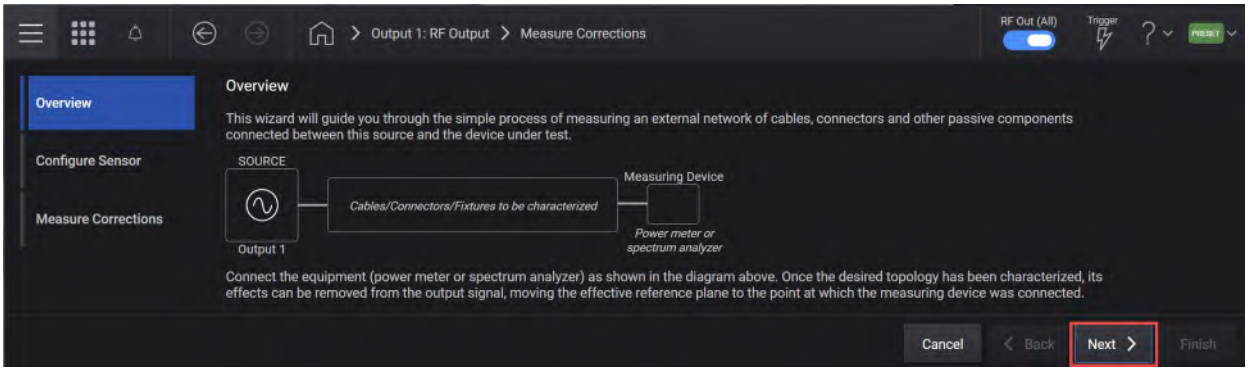
The following USB power sensors can be used for the power measurement.

- U8487A-CFG007
- U8485A-CFG006
- U2000A
- U2001A
- U2002A
- U2004A
- U2000B
- U2001B
- U2000H
- U2001H
- U2002H

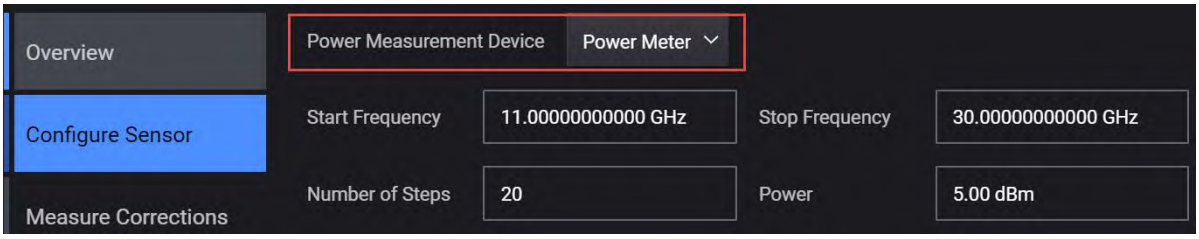
On the VXG:

1. Connect the VXG 10 MHz Out to the N90x0A/B Ext Reference In.
2. Select the **RF Output** block > **Corrections/De-embedding** block to open the Correction Setup.
3. Select the **A Block** (to highlight) then **Add from Measurement** to open the Measure Corrections Block Wizard.

4. Connect the power sensor as shown in the diagram below then select **Next**.



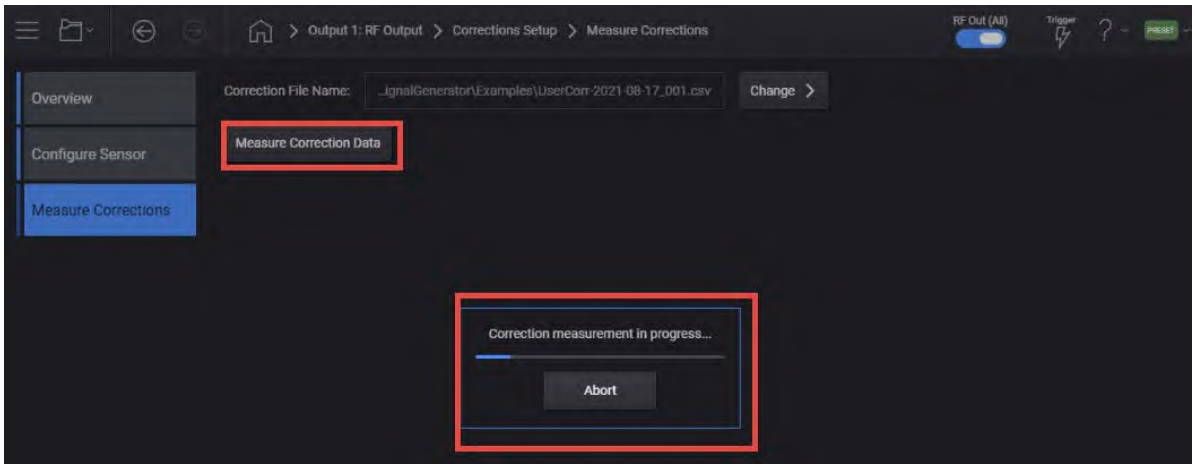
5. Select **Next** to go to Configure Sensor dialog and select the **Power Measurement Device** dropdown > **Power Meter**.



6. Set the Start and Stop Freq, Amplitude, and the Num Steps. For this example Start Freq **26 GHz**, Stop Freq **30 GHz**, Amplitude to the highest power used in your measurement **5 dBm**, and Num Steps to **20**.

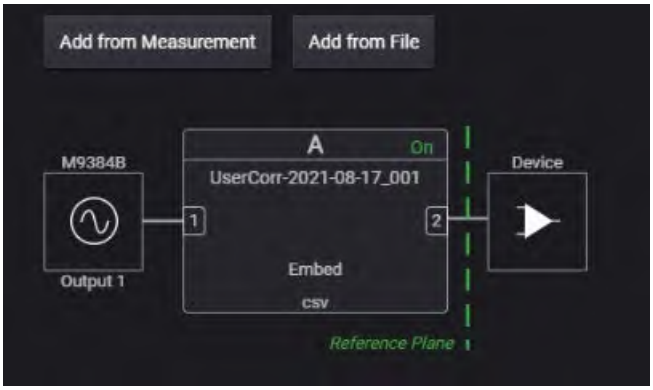
7. Select the Connection Type to **USB**, and then specify the Device and VISA Address.
8. Select **Test Connection** to verify connectivity, and then select **OK** then **Next** to continue.

   You can also calibrate and zero out the power sensor before measuring corrections.
9. Select **Next** to go to the Measure Corrections dialog and select **Measure Correction Data**.

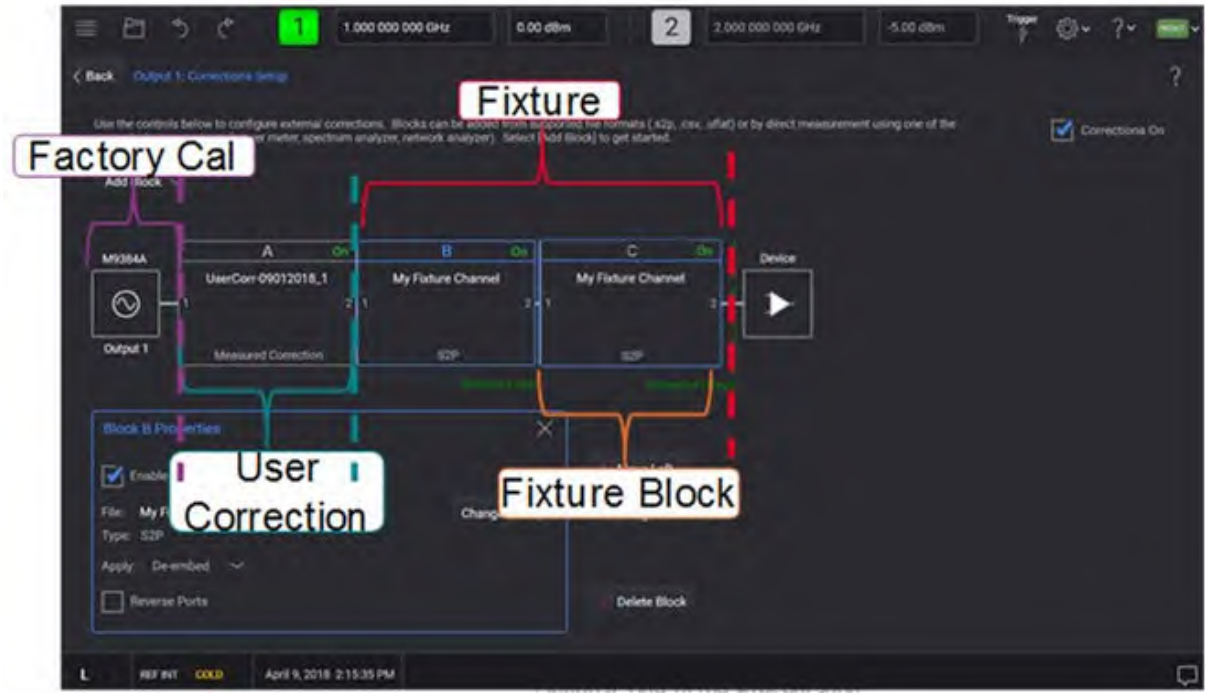


During the measurement, the VXG outputs a CW between the Start and Stop Frequencies for the specified number of steps and output power. The measurement results are saved to a csv file using an automatically generated file name.

10. Select **Finish**. The output csv file is set to Block A.



Block A is dedicated for User Correction. The image below shows how blocks are assigned in the User Correction and Fixture block.



11. Select **Corrections On** to apply.

Using the Equivalent SCPI Commands

Using a spectrum analyzer to make the corrections measurement

On the VXG:

```
SYSTem:PRESet
```

Set the power level to the highest level used in your measurement.

```
RF1:POWer:AMPLitude 5dBm
CORRection:PMDevice SANalyzer
CORRection:FLATness:STEP:STARt 26GHZ
CORRection:FLATness:STEP:STOP 30GHZ
CORRection:FLATness:STEP:POINts 20
CORRection:SANalyzer:COMMunicate:TYPE SOCKets
```

Set the LAN address and protocol parameters for your spectrum analyzer.

```
CORRection:SANalyzer:COMMunicate:LAN:IP "192.168.1.5"
CORRection:SANalyzer:COMMunicate:LAN:PORT 5025
CORRection:FLATness:CALibrate
CORRection ON
```

Using a power meter to make the corrections measurement.

On the VXG:

```
SYSTem:PRESet
RF1:POWer:AMPLitude 5dBm
CORRection:PMDevice PMETer
CORRection:FLATness:STEP:STARt 26GHZ
CORRection:FLATness:STEP:STOP 30GHZ
CORRection:FLATness:STEP:POINts 20
CORRection:SANalyzer:COMMunicate:TYPE USB
```

NOTE Use query `CORRection:PMETer:COMMunicate:USB:LIST?` for a list of all connected USB devices.

```
CORRection:PMETer:COMMunicate:USB:DEVice "instr0"
```

[Optional] `CORRection:PMETer:CALibrate`

[Optional] `CORRection:PMETer:ZERO`

```
CORRection:FLATness:CALibrate
```

```
CORRection ON
```

Adding fixture blocks using s2P files

On the VXG:

```
SYSTem:PRESet
RF1:FREQuency:CW 10GHZ
RF1:POWer:AMPLitude -20dBm
SIGNal1:MODE MTONe
SIGNal1:MTONe:ARB:NTON 21
SIGNal1:MTONe:ARB:FSP 100MHZ
SIGNal1 ON
RF1:OUTPut:STATe ON
```

For multi-channel instruments, set RF Out (all) to On.

```
RFALl:OUTPut ON
```

On the X-Series Signal Analyzer:

```
SYSTem:PRESet
FREQuency:CENTer 10GHZ
FREQuency:SPAN 3GHZ
```

On the VXG:

Set the path to the s2p data as block 2 (you can use 1 through 4).

```
CORRection:BLOCk2:FILE "FixtureChannel2"
CORRection:BLOCk2 ON
```

Add block C with the same file.

```
CORRection:BLOCk3:FILE "FixtureChannel2"
```

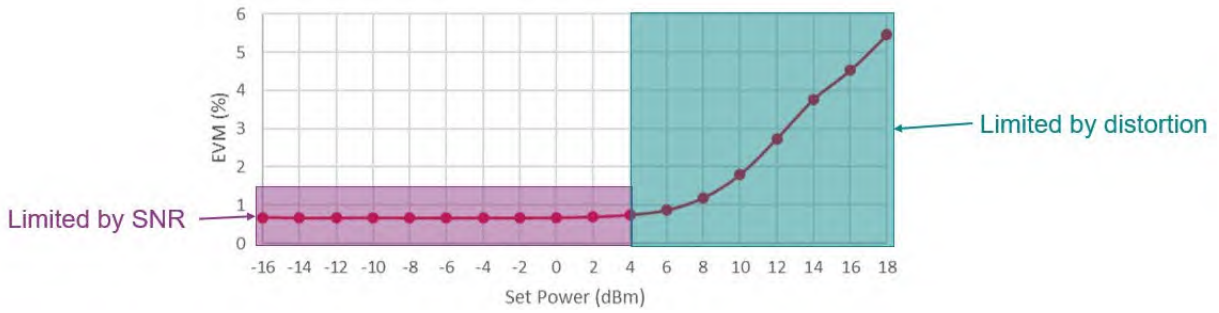
Change Block C to Embed.

```
CORRection:BLOCk3:APPLy EMBedding
CORRection:BLOCk3 ON
```

Instrument Nonlinear Correction

Instrument Nonlinear Correction (INC) is a digital pre-distortion (DPD) based correction using a Keysight signal analyzer (N9042B, N9040B, or N9030B) to compensate for nonlinearities in the VXG. INC is useful in reducing EVM (and other metrics impacted distortion) at high power levels and extending the range of the power with linear output of the VXG. INC is a point correction valid for a given frequency/power/waveform combination and can be used for EVM and ACP measurements at high power levels (>5 dB). The VXG must have the N7653APPC PathWave automatic channel response correction and S-parameter de-embedding license installed.

Instrument nonlinear corrections compensate for nonlinearities in the VXG. Improvement to EVM will only be seen at power levels where distortion is the limiting factor. INC will not improve EVM in the area where signal to noise ratio (SNR) is the limiting factor.

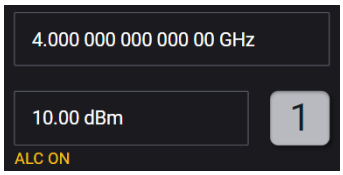


Setting Up an INC 1CC 4 GHz, 10 dBm EVM Measurement

On the VXG:

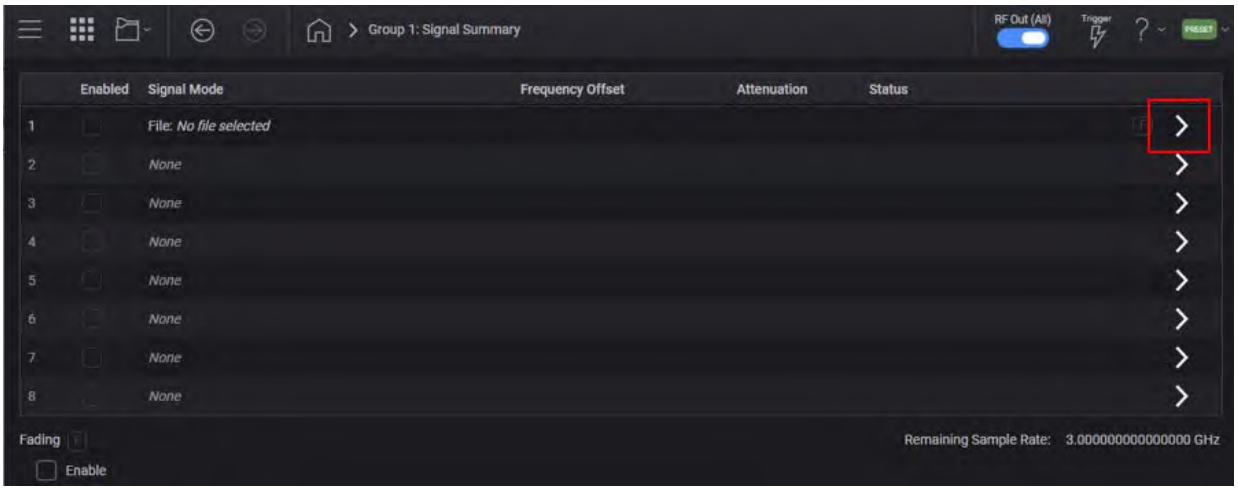
In order to compare before and after correction results, we will start by making an EVM measurement without applying corrections.

1. Select **Preset** > **Preset** to set the instrument to a know state.
2. In the Output area, set Frequency to **4 GHz** and Power to **10 dBm**.



3. Select the **Signal** block to open the Vector Modulation Signal Setup panel.

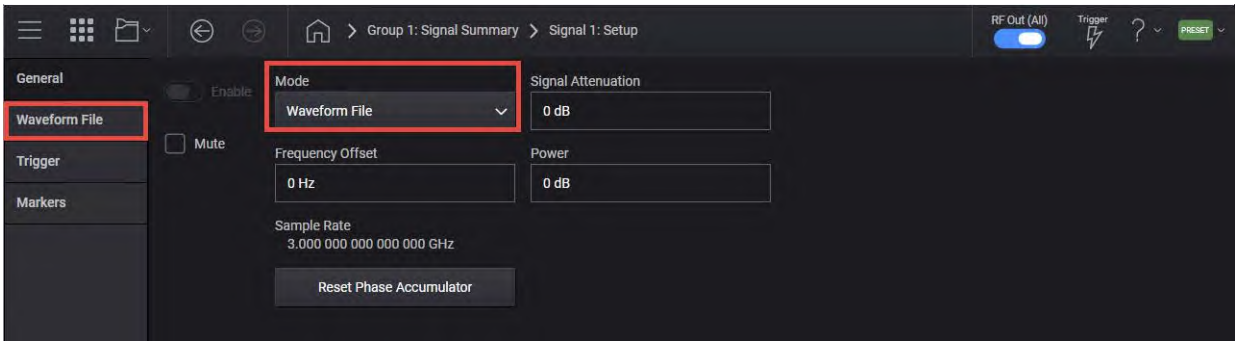
4. Select the arrow for Signal 1 to open the Signal Setup window.



NOTE

This screen is only accessible if Option M9484C-8SG (8 virtual signal generators) is installed. For all other option configurations, continue to the next step.

5. Set Mode to **Waveform File**, then select the **Waveform File** tab in the left pane.

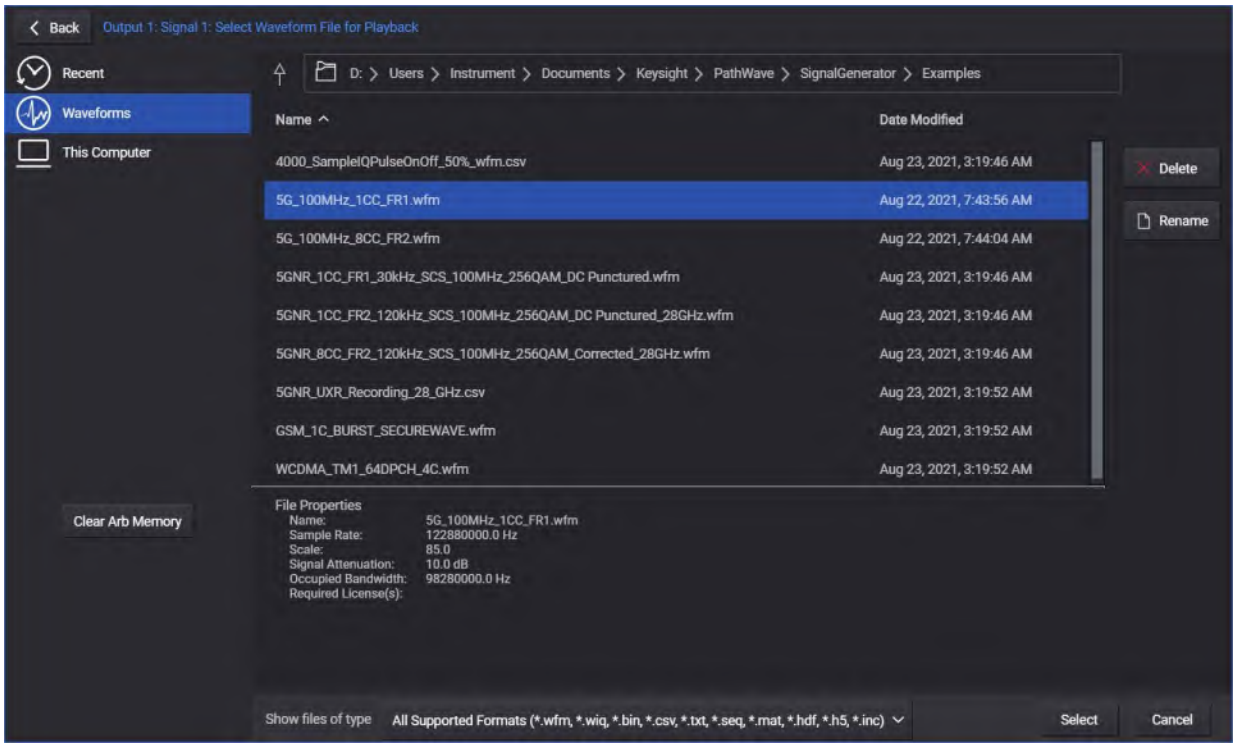


6. In the Waveform Playback Setup area, use File **Select** to navigate to:

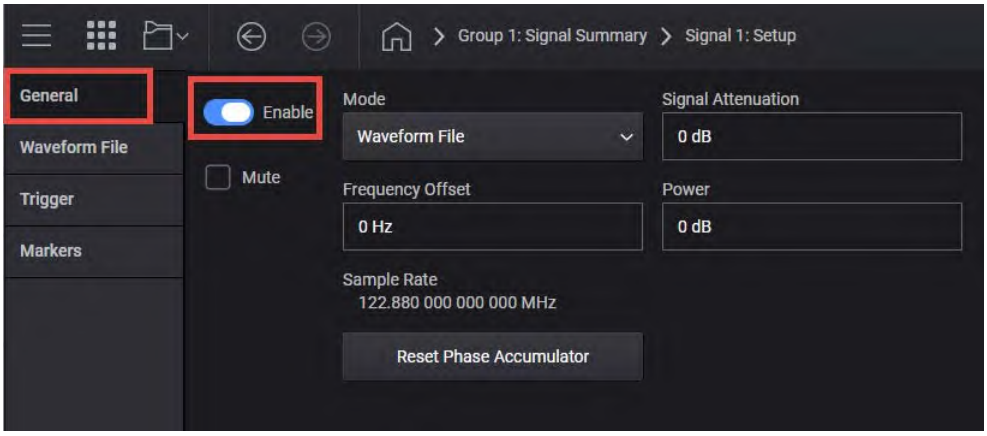
   `D:\Users\Instrument\Documents\Keysight\PathWave\SignalGenerator\Examples`

   and choose
   `5G_100MHz_1CC_FR1.wfm`

   then **Select**.



7. Select the **General** tab > **Enable**.



8. Select the **Home** icon and set RF Out to **On**.

On the UXA:

1. From the X-Series Signal Analyzer Menu Panel (on the top right of the display), select **Mode/Meas** > **5G NR & V2X** Mode > **OK**.

NOTE

If accessing the instrument via a Remote Desktop connection, select the Screen tab (at the top of the display) to open the Mode/Measurement/View Selector window.

2. Select **Mode Preset** to set Spectrum Analyzer 5G NR & V2X mode to a known state.
3. From the X-Series Signal Analyzer Menu Panel (on the top right of the display), select **Mode/Meas** > **5GNR & V2X** Mode > **Modulation Analysis** Measurement > **OK**.

NOTE

All example waveforms and setup files are located on the VXG at:

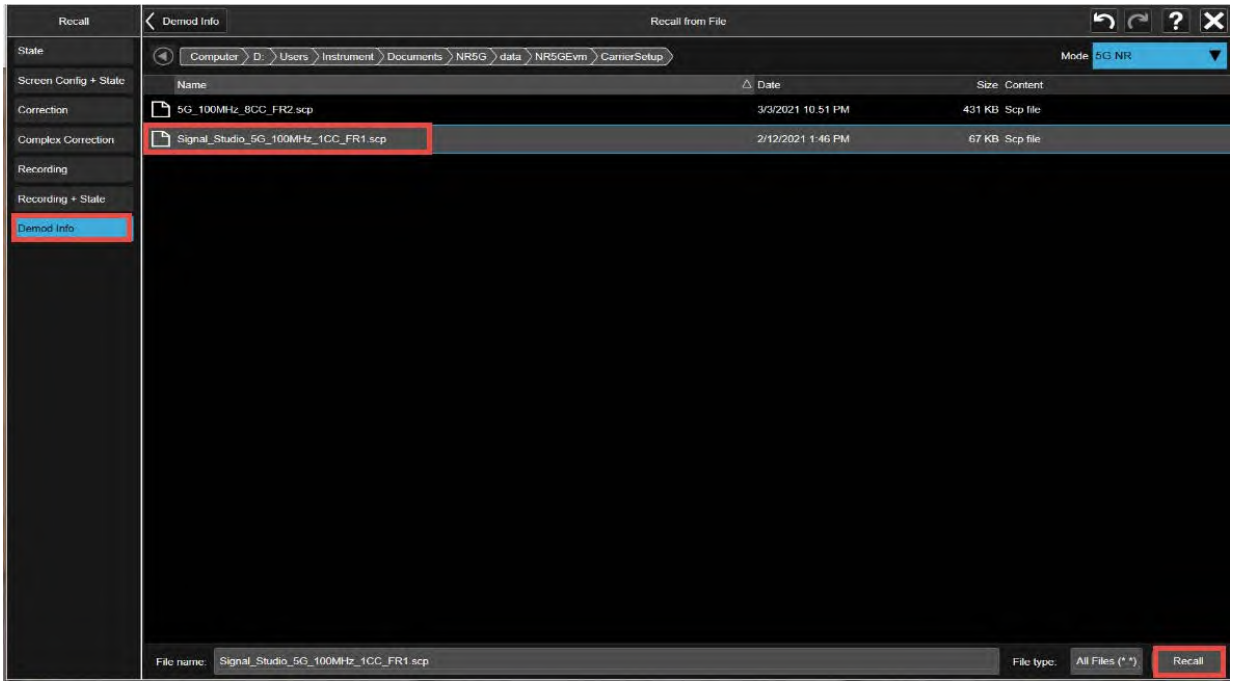
D:\Users\Instrument\Documents\Keysight\PathWave \SignalGenerator\Examples

You will need to copy over the setup files to the X-Series Signal Analyzer or the PC running the VSA application.

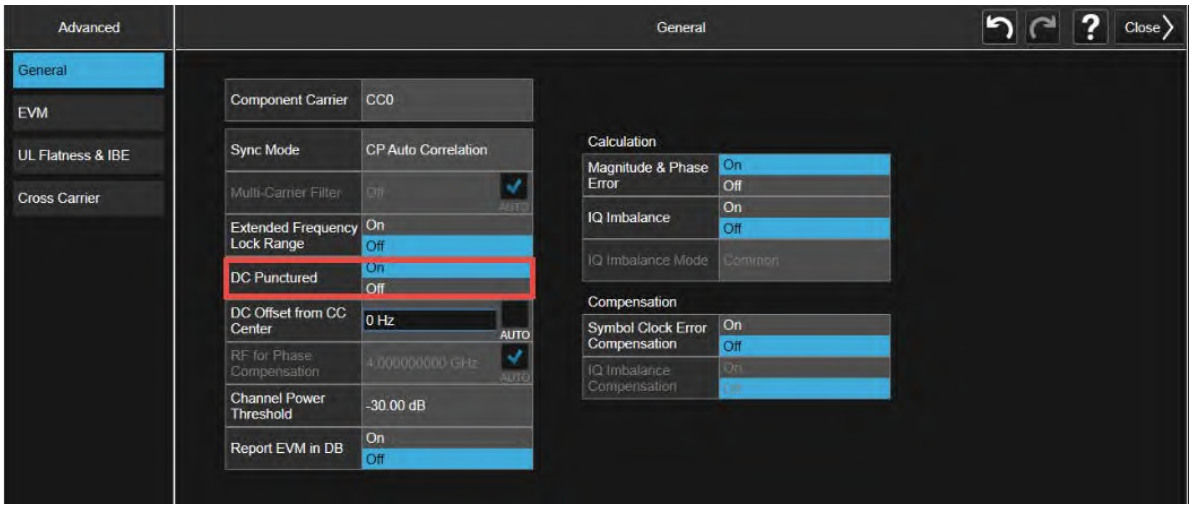
4. Select **Recall** (If accessing the X-Series Signal Analyzer remotely, select the Folder icon at the bottom of the display) **Demod Info** > Data Type set to **CC Setup** > **Recall From** >

`Signal Studio_5G_100 MHz_1CC_FR1.scp`

Recall

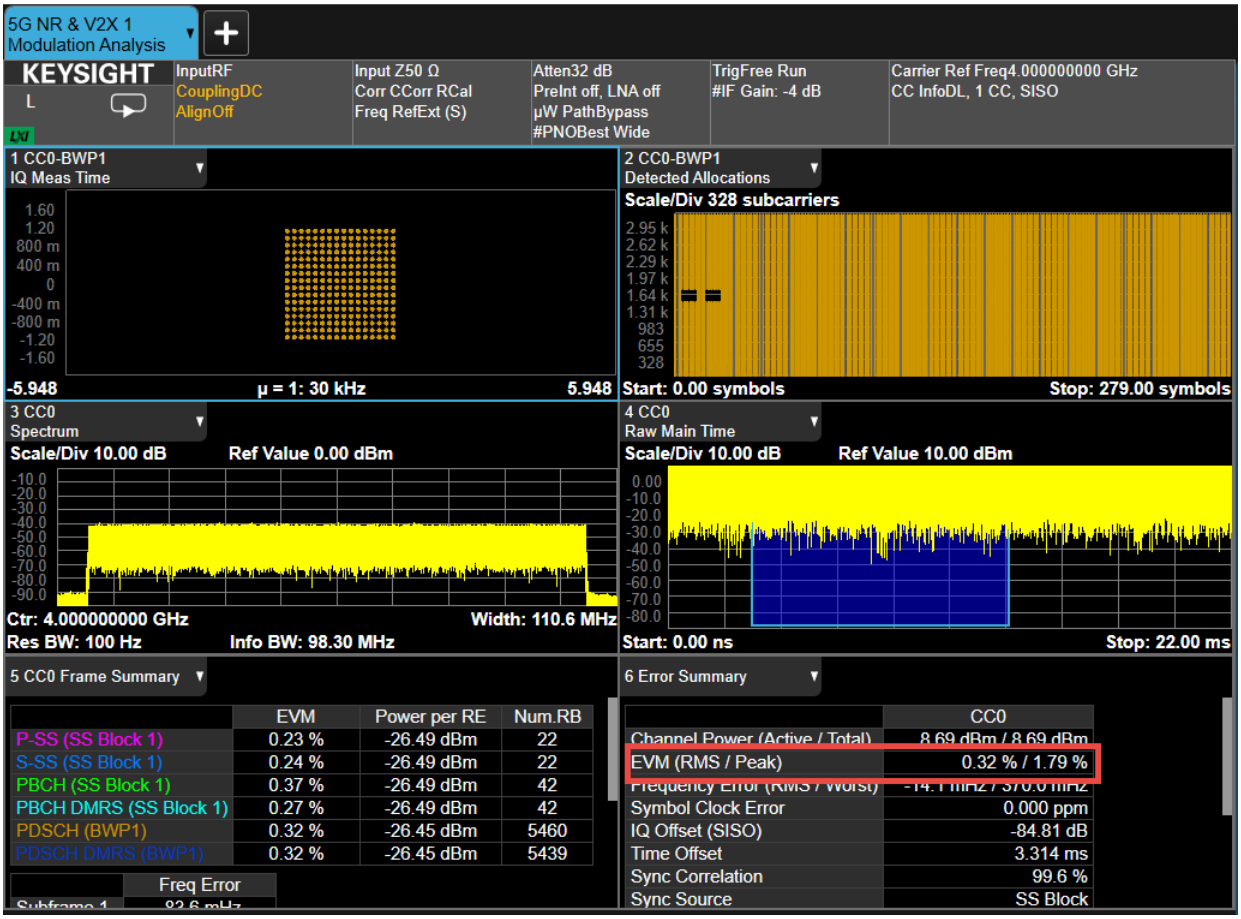


5. The Signal Stuidio_5G_100 MHz_1CC_FR1.scp file automatically sets the Carrier Frequency to 4 GHz.
6. From the Main menu, select **Meas Setup** > **Advanced** tab > **Advanced Demod Setup** and set DC Punctured to **On**, then **Close** the Advanced Demod Setup window.



7. In the Meas Setup window, select the **Settings** tab > **Optimize EVM**.

   Record the results. For this example EVM is 0.32% and EVM Peak is 1.79%


5G NR & V2X 1
Modulation Analysis
KEYSIGHT
InputRF
CouplingDC
AlignOff
Input Z50 Ω
Corr CCorr RCal
Freq RefExt (S)
Atten32 dB
PreInt off, LNA off
µW PathBypass
#PNOBest Wide
TrigFree Run
#IF Gain: -4 dB
Carrier Ref Freq4.000000000 GHz
CC InfoDL, 1 CC, SISO
1 CC0-BWP1
IQ Meas Time
-5.948
µ = 1: 30 kHz
5.948
2 CC0-BWP1
Detected Allocations
Scale/Div 328 subcarriers
Start: 0.00 symbols
Stop: 279.00 symbols
3 CC0
Spectrum
Scale/Div 10.00 dB
Ref Value 0.00 dBm
Ctr: 4.000000000 GHz
Width: 110.6 MHz
Res BW: 100 Hz
Info BW: 98.30 MHz
4 CC0
Raw Main Time
Scale/Div 10.00 dB
Ref Value 10.00 dBm
Start: 0.00 ns
Stop: 22.00 ms
5 CC0 Frame Summary
EVM
Power per RE
Num.RB
P-SS (SS Block 1) 0.23 % -26.49 dBm 22
S-SS (SS Block 1) 0.24 % -26.49 dBm 22
PBCH (SS Block 1) 0.37 % -26.49 dBm 42
PBCH DMRS (SS Block 1) 0.27 % -26.49 dBm 42
PDSCH (BWP1) 0.32 % -26.45 dBm 5460
PDSCH DMRS (BWP1) 0.32 % -26.45 dBm 5439
Freq Error
6 Error Summary
CC0
EVM (RMS / Peak) 0.32 % / 1.79 %
Symbol Clock Error 0.000 ppm
IQ Offset (SISO) -84.81 dB
Time Offset 3.314 ms
Sync Correlation 99.6 %
Sync Source SS Block

To Measure ACP:

a. Select **Mode/Meas** > **5GNR & V2X** Mode > **ACP** > **OK**.

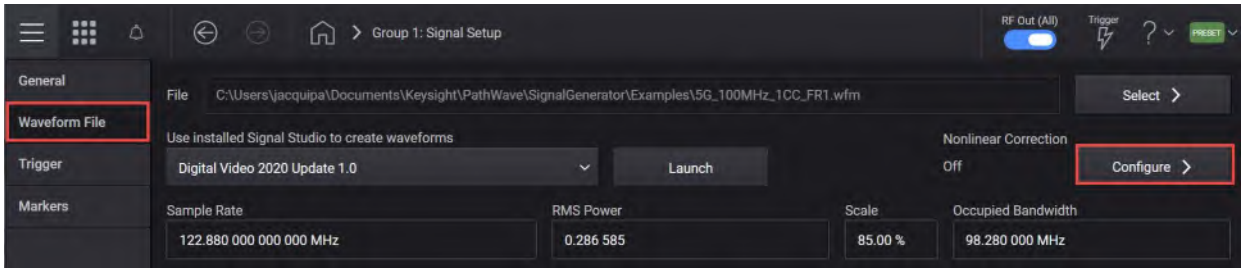
b. Select **Amplitude** > **Signal Path** tab > and set µW Path Control to **Low Noise Path Enable** (if Option LNP is available on your analyzer).

Record the results for the Lower and Upper ACP results. For this example -50.9 dBc (lower) and -50.9(upper).

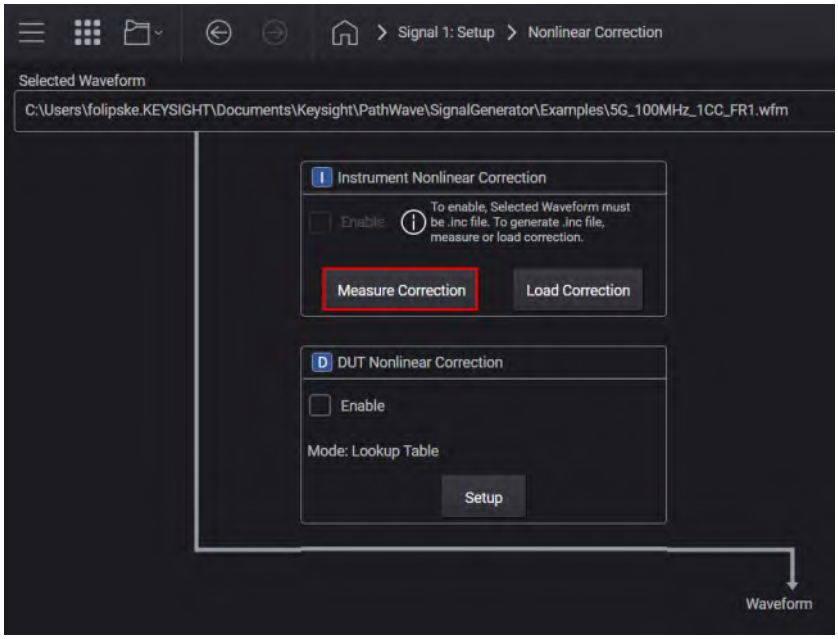
c. Select **Mode/Meas** > **5GNR & V2X** > **Modulation Analysis** > **OK**.

On the VXG:

1. Open the Signal block and select the **Waveform File** tab > **Configure** to open the Nonlinear Correction Setup.

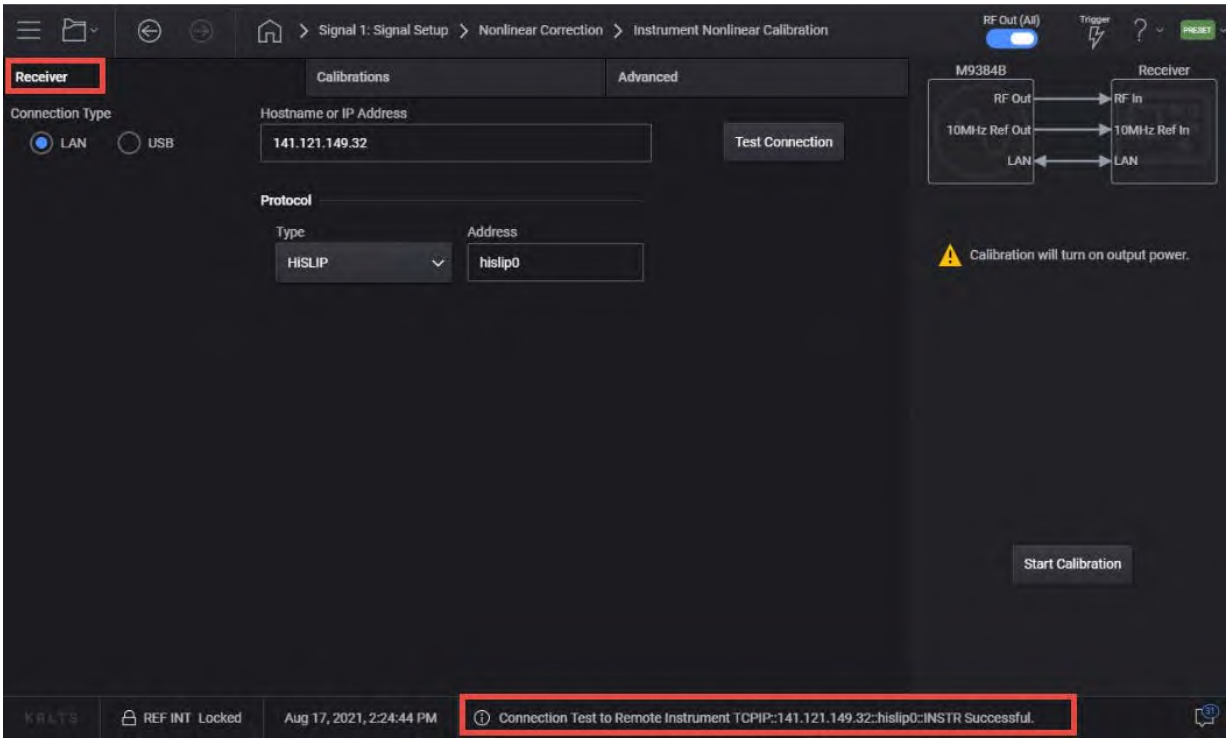


2. In the Instrument Nonlinear Correction area, select **Measure Correction** to open the Instrument Nonlinear Calibration screen.

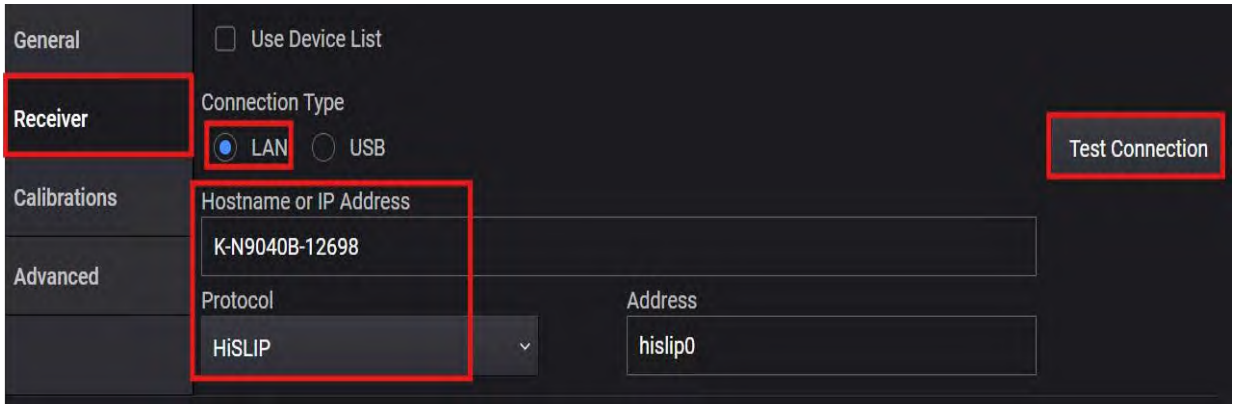


3. Under the **Receiver** tab, enter your connection information. For this example:
   - select **LAN** as the Connection Type
   - enter the IP address for you signal analyzer
   - set the Protocol to **HiSLIP**
   - **Test Connection**

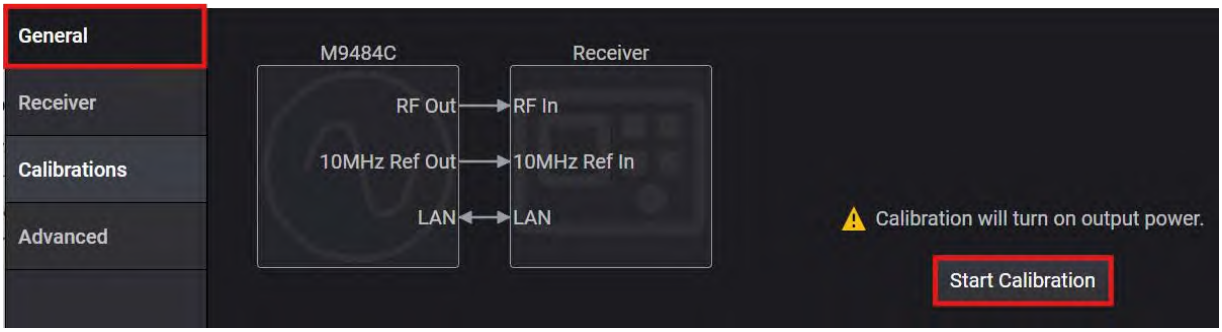
The Connection status will be displayed in the Notifications area at the bottom of the main window.



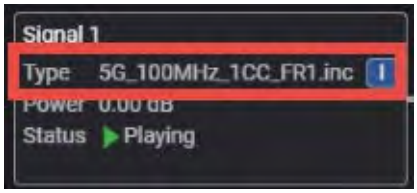
4. Select the **Calibrations** tab and select **EVM** and **ACP** for the Calibration types.



5. Select **General** tab > **Start Calibration.**



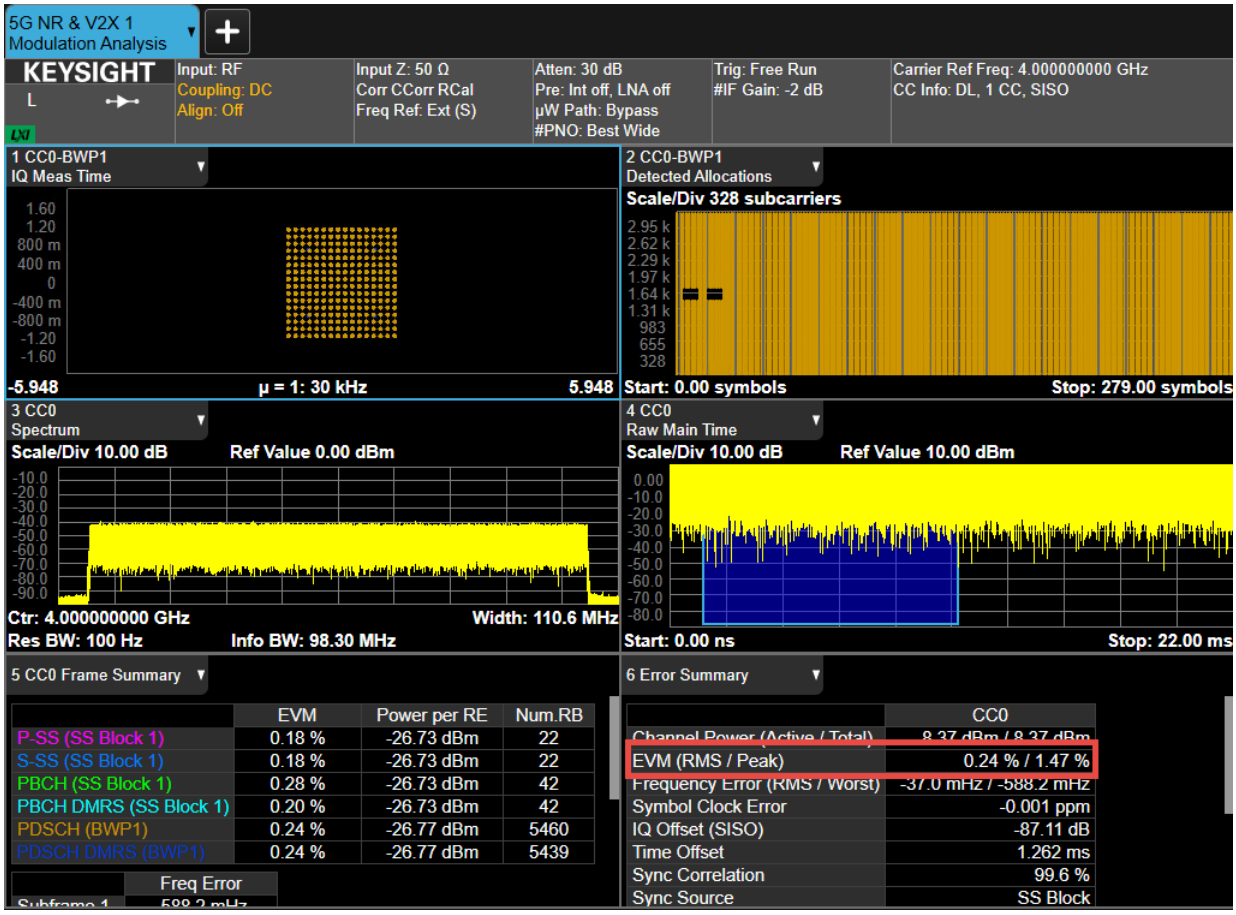
The calibration will take a few minutes to complete. Once done, the INC file will be created and it will automatically be used in place of the .wfm file originally loaded. An "I" will be displayed in the Signal block indicating this status.



On the UXA:

1. Select **Optimize EVM**.

Note the changes to EVM RMS and EVM Peak values. For this example EVM RMS is 0.24% (before 0.32%) and EVM Peak is 1.47% (before 1.79%).

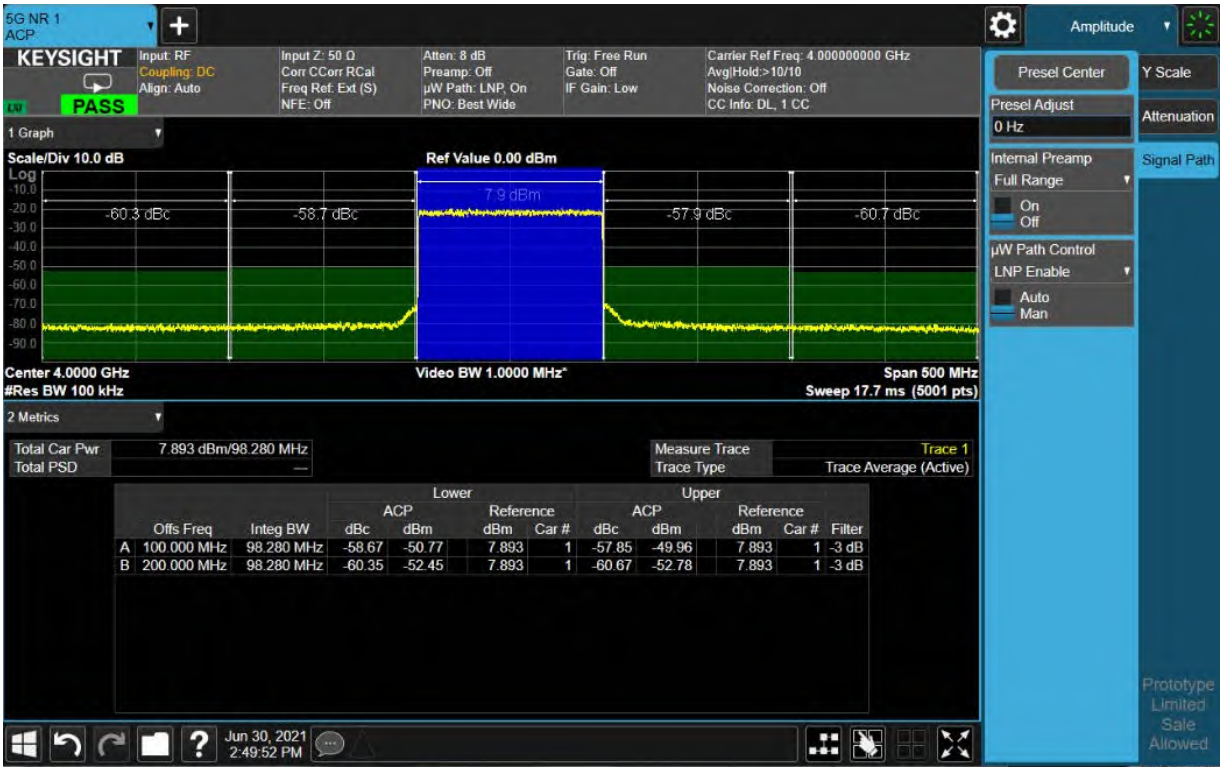


To Measure ACP:

a. Select **Mode/Meas** > **5GNR & V2X** Mode > **ACP** > **OK**.

b. Select **Amplitude** > **Signal Path** tab > and set µW Path Control to **Low Noise Path Enable**.

   Note the changes to the Lower and Upper ACP values. For this example Lower -58.67 dBc (before -50.9 dBc) and Upper -57.85 dBc (before -50.9).


5G NR 1
ACP
KEYSIGHT
PASS
Input: RF
Coupling: DC
Align: Auto
Input Z: 50 Ω
Corr CCorr RCal
Freq Ref: Ext (S)
NFE: Off
Atten: 8 dB
Preamp: Off
µW Path: LNP, On
PNO: Best Wide
Trig: Free Run
Gate: Off
IF Gain: Low
Carrier Ref Freq: 4.000000000 GHz
Avg|Hold:>10/10
Noise Correction: Off
CC Info: DL, 1 CC
Amplitude
Presel Center
Y Scale
Presel Adjust
0 Hz
Attenuation
Internal Preamp
Full Range
On
Off
Signal Path
µW Path Control
LNP Enable
Auto
Man
1 Graph
Scale/Div 10.0 dB
Ref Value 0.00 dBm
Log
-10.0
-20.0
-30.0
-40.0
-50.0
-60.0
-70.0
-80.0
-90.0
-60.3 dBc
-58.7 dBc
7.9 dBm
-57.9 dBc
-60.7 dBc
Center 4.0000 GHz
#Res BW 100 kHz
Video BW 1.0000 MHz*
Span 500 MHz
Sweep 17.7 ms (5001 pts)
2 Metrics
Total Car Pwr
7.893 dBm/98.280 MHz
Total PSD
—
Measure Trace
Trace 1
Trace Type
Trace Average (Active)
Lower
Upper
ACP
Reference
Offs Freq
Integ BW
dBc
dBm
dBm
Car #
Filter
A 100.000 MHz 98.280 MHz -58.67 -50.77 7.893 1 -57.85 -49.96 7.893 1 -3 dB
B 200.000 MHz 98.280 MHz -60.35 -52.45 7.893 1 -60.67 -52.78 7.893 1 -3 dB
Jun 30, 2021
2:49:52 PM
Prototype
Limited
Sale
Allowed

Setting Up an INC 8CC 28 GHz, 10 dBm EVM Measurement

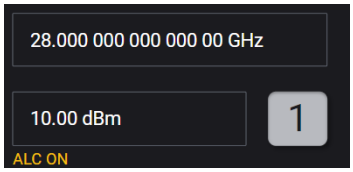
NOTE

This example requires the M9484C to have a minimum RF Bandwidth of 1 GHz (Option M9484C-R10 or R25).

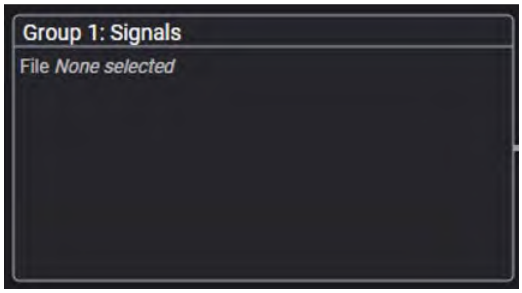
On the VXG:

In order to compare before and after correction results, we will start by making an EVM measurement without applying corrections.

1. Select **Preset** > **Preset** to set the instrument to a know state.
2. In the Output area, set Frequency to **28 GHz** and Power to **10 dBm**.



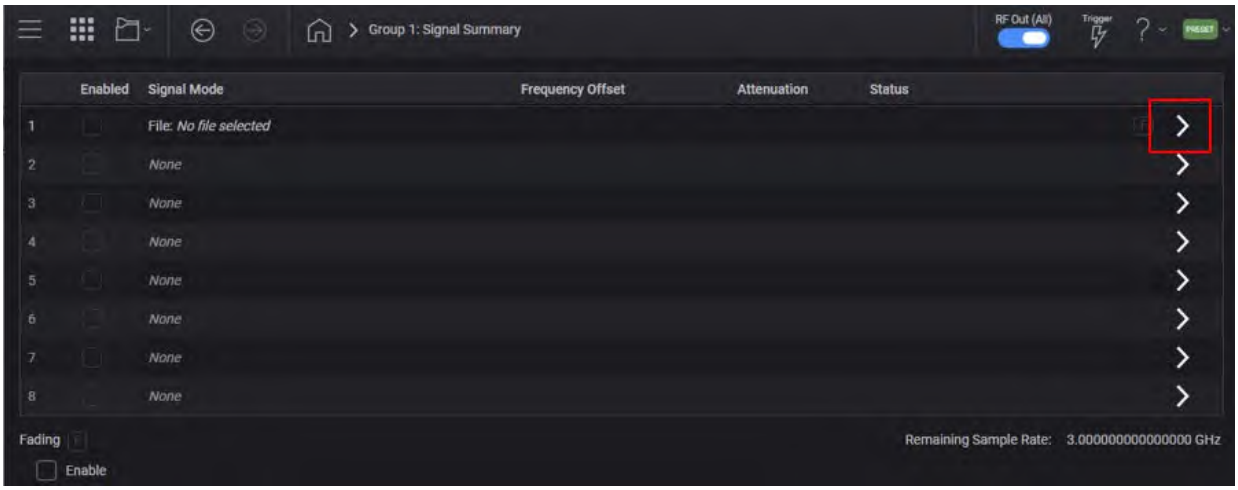
3. Select the **Group 1: Signals** block to open.



4. Select the arrow for Signal 1 to open the Signal Setup window.

NOTE

This screen is only accessible if Option M9484C-8SG (8 virtual signal generators) is installed. For all other option configurations, continue to the next step.

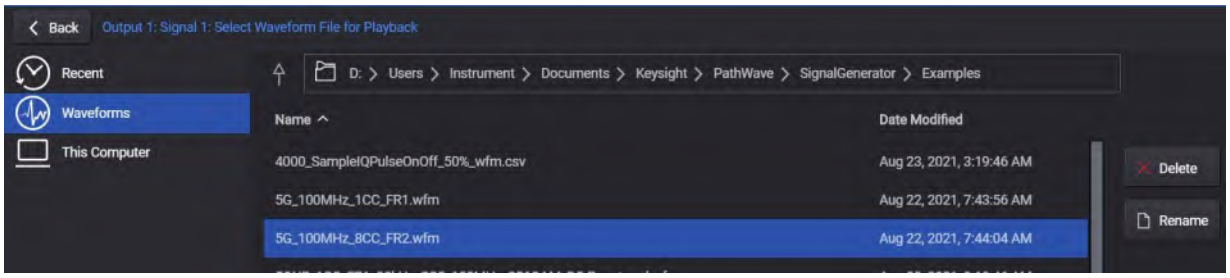


5. Set Mode to **Waveform File** and then select the **Waveform File** tab in the left pane.
6. In the Waveform Playback Setup area, use File **Select** to navigate to:

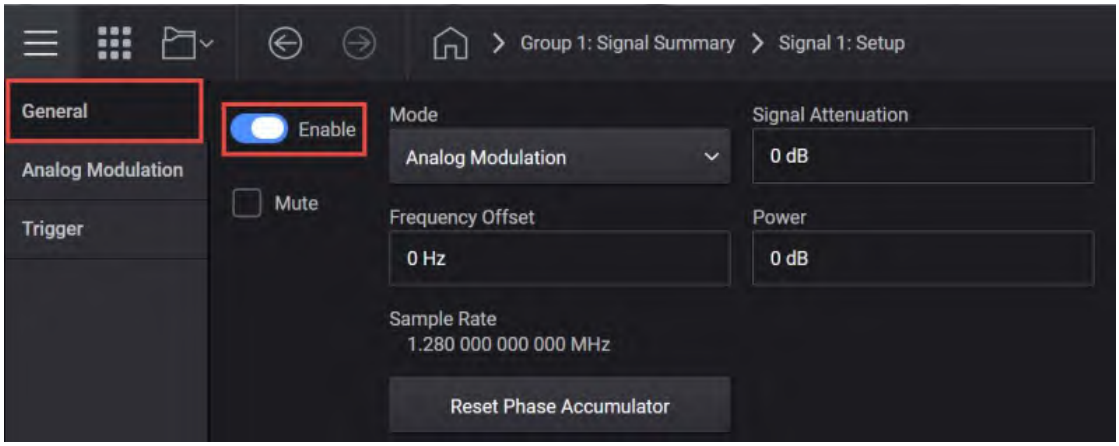
   ```
   D:\Users\Instrument\Documents\Keysight\PathWave
   \SignalGenerator\Examples
   ```

   ```
   5G_100MHz_8cc_FR2.wfm
   ```

   then **Select**.



7. Select the **General** tab > **Enable**.



8. Select the **Home** icon and set RF Out to **On**.

On the UXA:

1. From the X-Series Signal Analyzer Menu Panel (on the top right of the display), select **Mode/Meas** > **5GNR & V2X** Mode > **OK**.

NOTE

If accessing the instrument via a Remote Desktop connection, select the Screen tab (at the top of the display) to open the Mode/Measurement/View Selector window.

2. Select **Mode Preset** to set Spectrum Analyzer 5GNR mode to a known state.
3. From the X-Series Signal Analyzer Menu Panel (on the top right of the display), select **Mode/Meas** > **5GNR & V2X** Mode > **Modulation Analysis** Measurement > **OK**.

NOTE

All example waveforms and setup files are located on the VXG at:

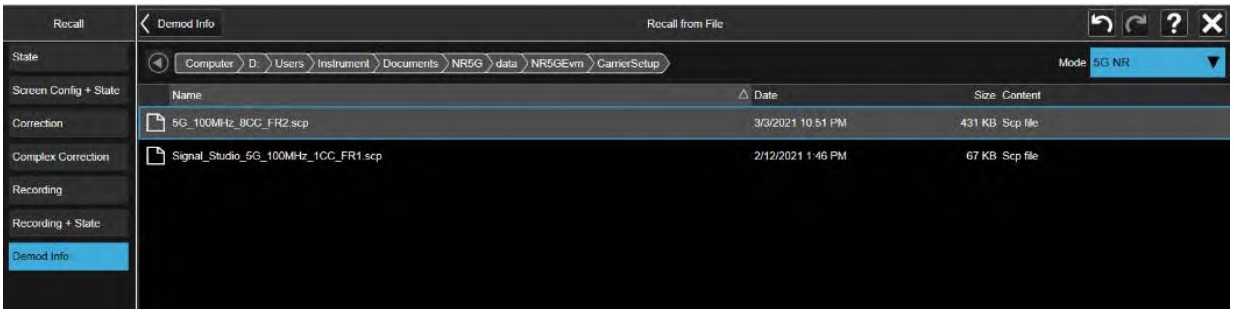
D:\Users\Instrument\Documents\Keysight\PathWave \SignalGenerator\Examples

You will need to copy over the setup files to the X-Series Signal Analyzer or the PC running the VSA application.

4. Select **Recall** (If accessing the X-Series Signal Analyzer remotely, select the Folder icon at the bottom of the display) **Demod Info** > Set Data Type to **CC Setup** > **Recall From** >

`5G_100 MHz_8CC_FR2.scp`

Recall



5. The .scp file automatically sets the Carrier Frequency to 28 GHz.
6. Select **Meas Setup** > **Advanced** tab > **Advanced Demod Setup** and set DC Punctured to **On**, then **Close** the Advanced Demod Setup window.
7. Select the **Sweep** > **Single Sweep**.
8. Select **Meas Setup** > **Settings** tab > **Optimize EVM**.

   It will take a few minutes for the UXA to measure all 8 carriers.

   Record the results.

For this example CC0 EVM RMS is 6.21%, EVM Peak is 27.33% and for CC7 EVM RMS is 4.51% and EVM Peak is 15.75%

To Measure ACP:

a. Select **Mode/Meas** > **5GNR & V2X** Mode > **ACP** > **OK**.

b. Select **Amplitude** > **Signal Path** tab > and set µW Path Control to **Low Noise Path Enable** (if your analyzer has Option LNP).

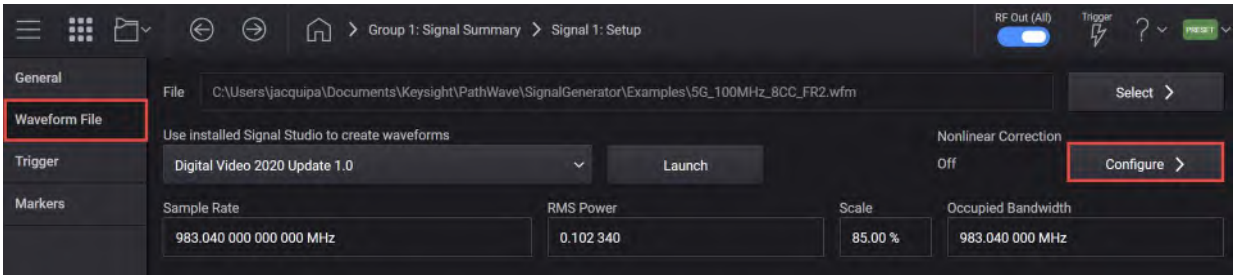
c. Select **Sweep** > **Restart** to take a new sweep.

Note the values of the Lower and Upper ACP. For this example Lower, at 100 MHz offset is -26.7 dBc and Upper -27.06 dBc.

d. Select **Mode/Meas** > **5GNR & V2X** > **Modulation Analysis** > **OK**.

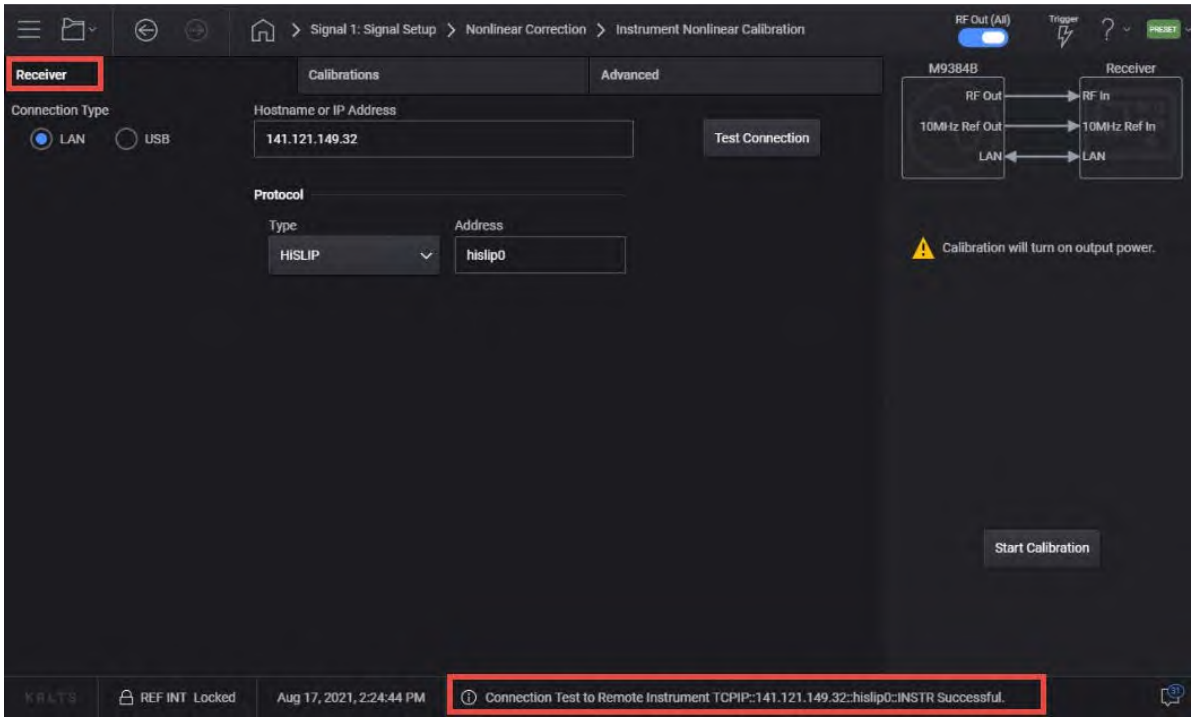
On the VXG:

1. In the Signal block, select **Waveform File** Mode > **Waveform File** tab > **Configure** to open the Nonlinear Correction Setup.

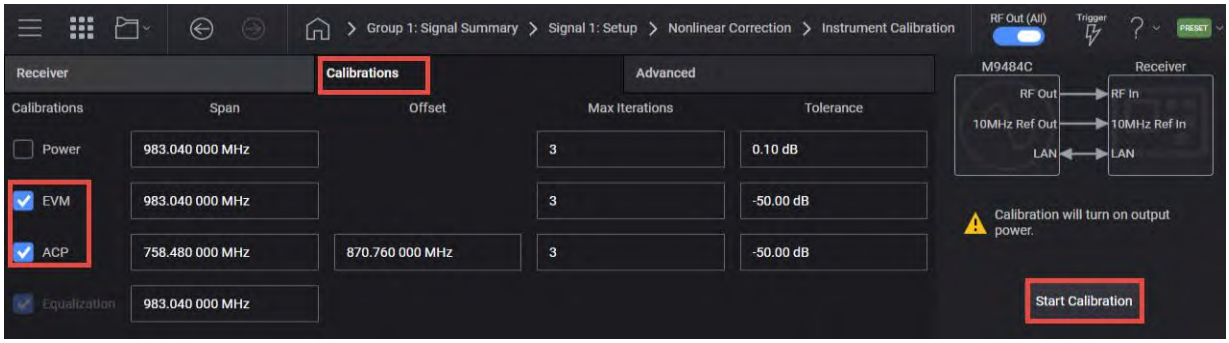


2. In the Instrument Nonlinear Correction area, select **Measure Correction** to open the Instrument Nonlinear Calibration screen.
3. Under the **Receiver** tab, enter your connection information and test the connection. For this example, select **LAN** as the Connection Type > Enter the IP address for you signal analyzer (for this example, 141.121.149.32) as the Hostname > **Test Connection**.

The Connection status will be displayed in the Notifications area at the bottom of the main window.



4. Select the **Calibrations** tab and select **EVM** and **ACP** for the Calibrations. Select the **General** tab > **Start Calibration**. The calibration will take a few minutes to complete.



On the UXA:

1. Select the **Sweep** > **Restart**.
2. Select **Optimize EVM**.

   Note the changes to EVM RMS and EVM Peak values.

For this example CC0 EVM RMS went from 6.21% to 3.13%, EVM Peak went from 27.33% to 11.68% and for CC7 EVM RMS went from 4.51% to 2.87% and EVM Peak went from 15.75% to 10.55%.

To Measure ACP:

a. Select **Mode/Meas** > **5GNR & V2X** Mode > **ACP** > **OK**.

b. Select **Amplitude** > **Signal Path** tab > and set µW Path Control to **Low Noise Path Enable**.

c. Select **Sweep** > **Restart** to take a new sweep.

   Note the changes to the Lower and Upper ACP values. For this example Lower, at 100 MHz offset is -32.27 (before -26.7 dBc) and Upper -31.15 (before -27.06 dBc).

5G NR 1
ACP
KEYSIGHT
PASS
Input: RF
Coupling: DC
Align: Auto
Input Z: 50 Ω
Corr CCorr RCal
Freq Ref: Ext (S)
NFE: Off
Atten: 28 dB
Preamp: Off
µW Path: LNP, On
PNO: Best Wide
Trig: Free Run
Gate: Off
IF Gain: Low
Carrier Ref Freq: 28.000000000 GHz
Avg|Hold: 10/10
Noise Correction: Off
CC Info: DL, 8 CCs
Sweep
Sweep Time
42.4 ms
Auto
Man
Sweep / Measure
Continuous
Single
Restart
Pause
Sweep/ Control
Sweep Config
X Scale
1 Graph
Scale/Div 10.0 dB
Ref Value 0.00 dBm
Log
-10.0
-20.0
-30.0
-40.0
-50.0
-60.0
-70.0
-80.0
-90.0
-31.9 dBc
-32.3 dBc
-4.8 dBm
-5.2 dBm
-5.0 dBm
-5.2 dBm
-5.1 dBm
-5.1 dBm
-5.1 dBm
-4.9 dBm
-33.1 dBc
-33.3 dBc
Center 28.0000 GHz
#Res BW 100 kHz
Video BW 1.0000 MHz^
Span 1.198 GHz
Sweep 42.4 ms (12001 pts)
2 Metrics
Total Car Pwr
3.988 dBm/760.320 MHz
Total PSD
Measure Trace
Trace 1
Trace Type
Trace Average (Active)
Lower
Upper
ACP
Reference
Offs Freq
Integ BW
dBc
dBm
dBm
Car #
Filter
A 100.000 MHz 98.280 MHz -32.27 -37.04 -4.775 1 -33.15 -37.92 -4.775 1 -3 dB
B 200.000 MHz 98.280 MHz -31.85 -36.63 -4.775 1 -33.32 -38.10 -4.775 1 -3 dB
Prototype
Limited
Sale
Allowed

Phase Coherency and Alignment Between Channels

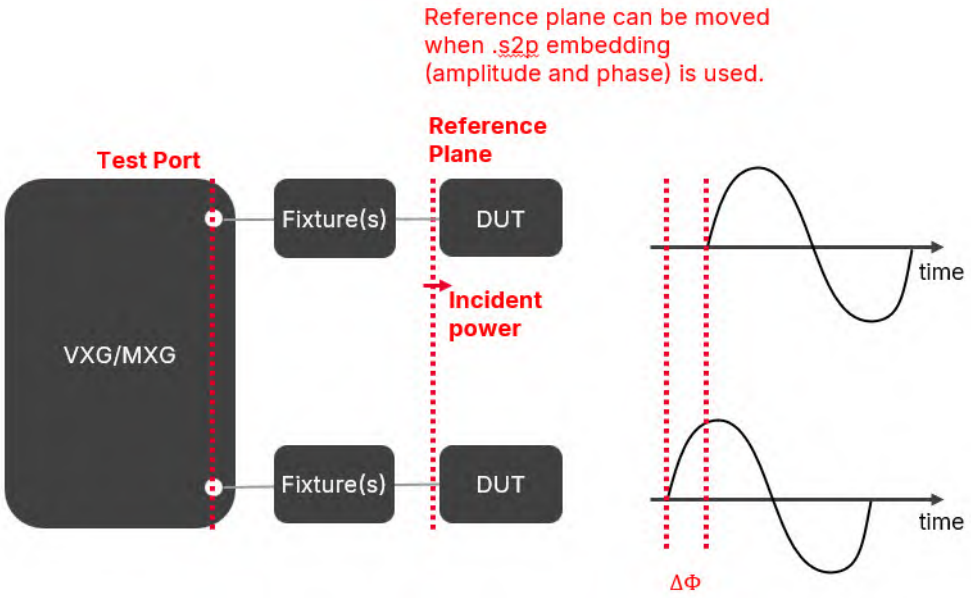
Phase coherency and alignment are critical aspects of modern RF and microwave test systems, particularly in applications requiring precise multi-channel signal generation and measurement.

Phase Coherency: Refers the condition where two or more signals having a constant relative phase over time/temp (which also means they are at the same frequency).

Phase Aligned: Refers the condition where phase delta of each channel is zero in all frequencies of modulated signal bandwidth.

Alignments are required for phase aligned channels, even in phase coherent signal generators.

Refer to the Phase Coherency and Alignment application note for more information.
https://www.keysight.com/us/en/assets/3125-1283/application-notes/Phase-Coherency-and-Alignment.pdf



Required Options

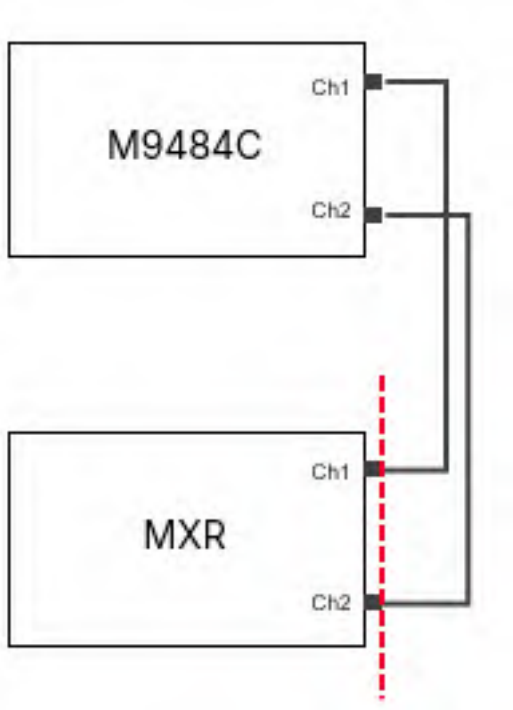
- M9484C/N5186A Option PCH: Phase coherent, and phase relationship stays repeatable even with frequency change. Firmware version A:19.01.00 or later
- N7653APPC: PathWave Signal Generation for Corrections/De-embedding, Embedded Application

– Oscilloscope (MXR, UXR, V-Series) with the PathWave 89600 VSA

NOTE

Note that the MXR, UXR, V-Series have all been tested for this measurement, but others may work as well. In this example, an MXR with Firmware Version 11.71.00035 and VSA Firmware Version 2026 30.00.306 was used.

Phase Alignment for Modulated Signal Using an MXR

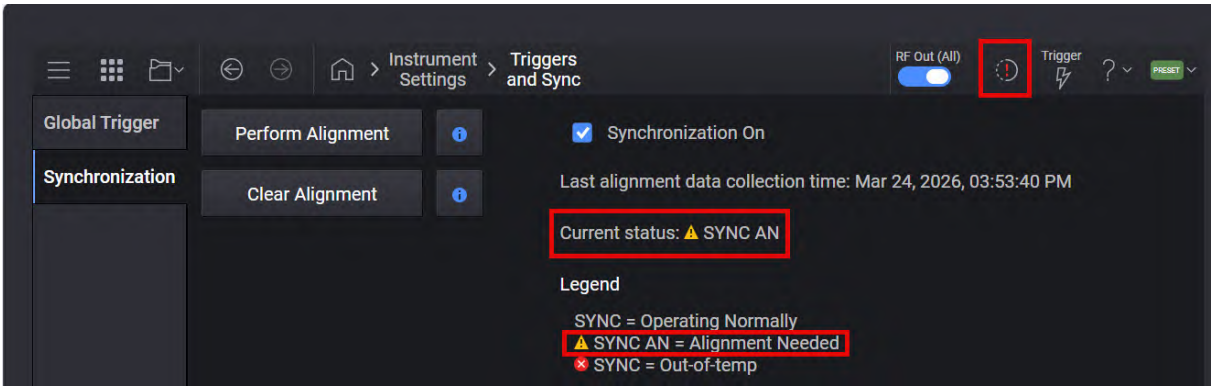


Setting up a Single CW Tone on the VXG

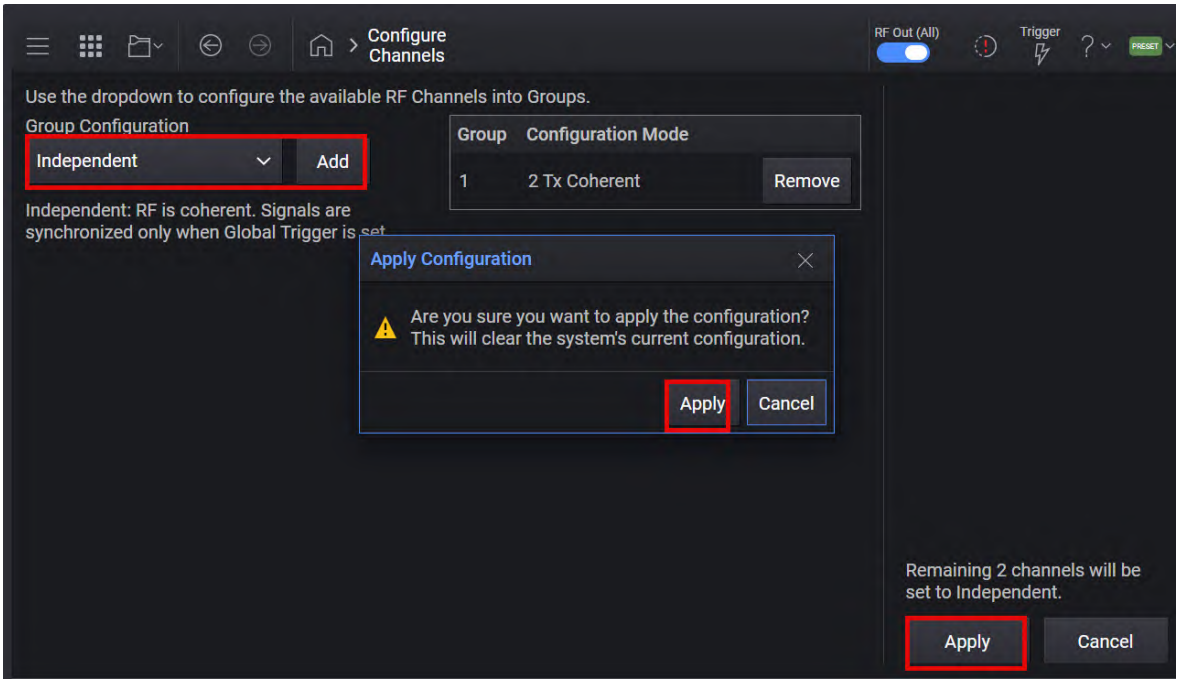
Generate a simple single CW tone on both channels and view them on the oscilloscope in the time domain. You will see that the two waveforms are offset - they're phase coherent (same frequency, stable relationship), but not phase aligned.

NOTE

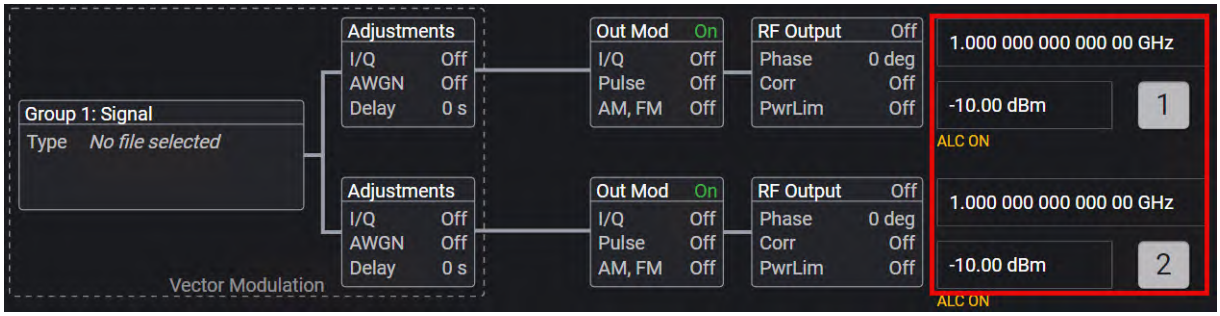
The VXG Internal Sync Alignment must be validated prior to running the Channel-to-Channel Alignment. The internal alignment procedure aligns the instrument's circuitry for the current operating temperature. The instrument should be turned on for a minimum of 1 hour prior to running. As long as the operating temperature remains stable, this alignment should only needed once. To verify if the Synchronization Alignment needs to be run, select the **Status** icon on the top left side of the display, then run if necessary.



1. Connect the equipment as shown above.
2. Select **Preset** > **Preset** to set the VXG to a known state.
3. Select the **System** menu (triple bar on top left) > **Configure Channels** > **Group Configuration** dropdown and select **2 Tx Coherent** > **Add** > **Apply** > **Apply**.

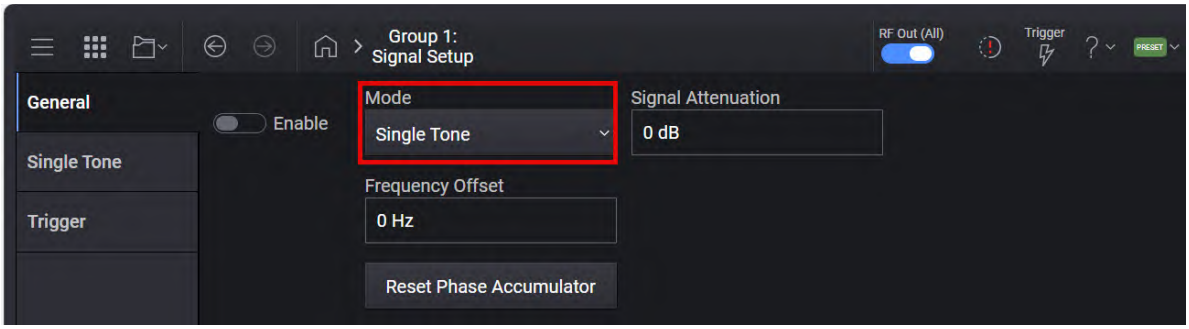


4. Set Channel 1 and 2 Frequency to **1 GHz** and Power to **-10 dBm**.



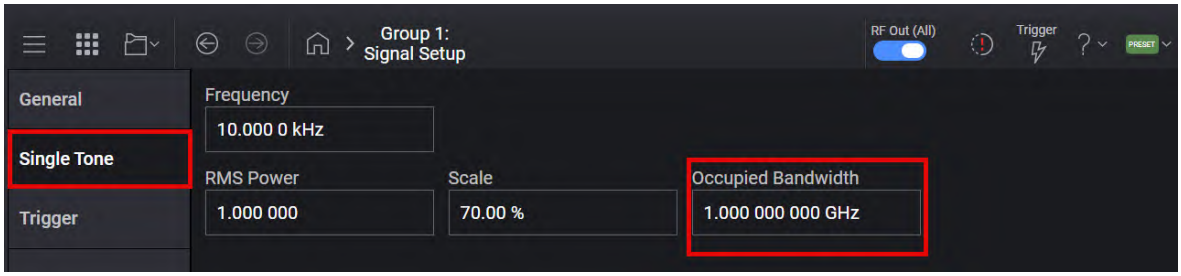
5. Select **Group 1: Signals** block to open.

6. From the Mode dropdown, select **Single Tone**.



7. Select the **Signal Tone** tab and change the Occupied Bandwidth to **1 GHz** for the MXG or **1.28 GHz** for the VXG.

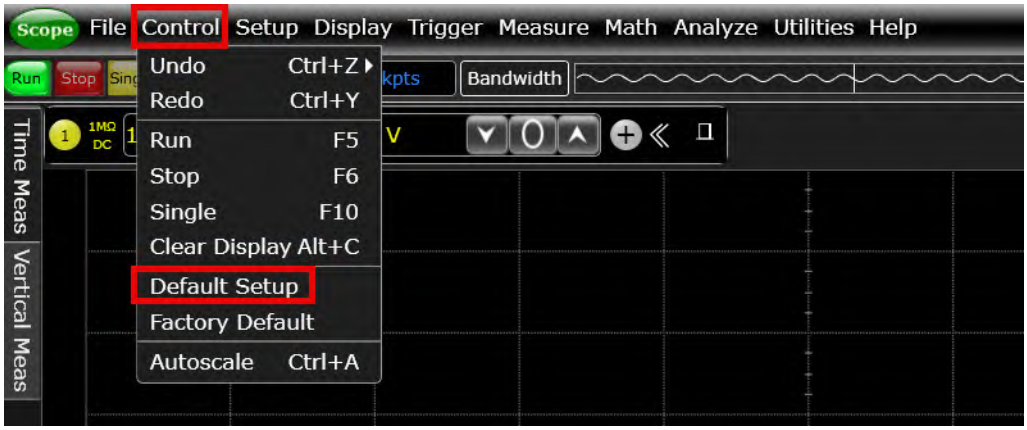
   For the best result, use consistent occupied bandwidth between calibration and measurement.



8. Select the **General** tab > **Enable**.
9. Set RF Out to **On** by selecting the numbered channel indicator next to both **Channel 1 and Channel 2**. switch.

Setting up the MXR Series Oscilloscope

1. Select **Control** > **Default Setup** to set the MXR to a known state.

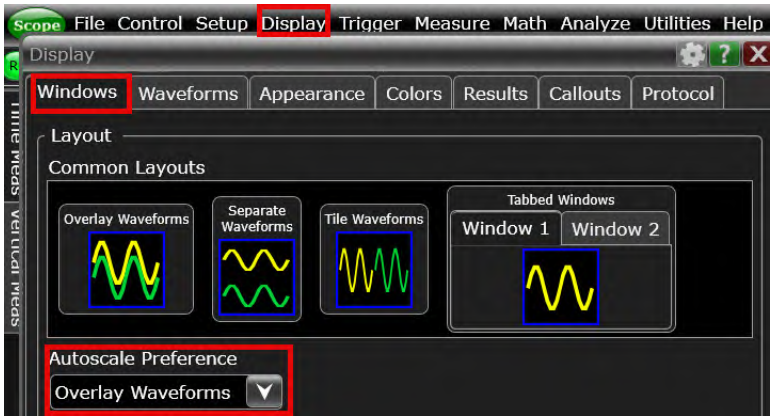


2. On the MXR, select **Setup** > **Channel 1** and set Impedance to **50 Ω** and then **Copy 1 to 2** to set Channel 2 to the same settings. Close the Setup dialog box.

Tip: There is no impedance setting for the V-Series oscilloscope.

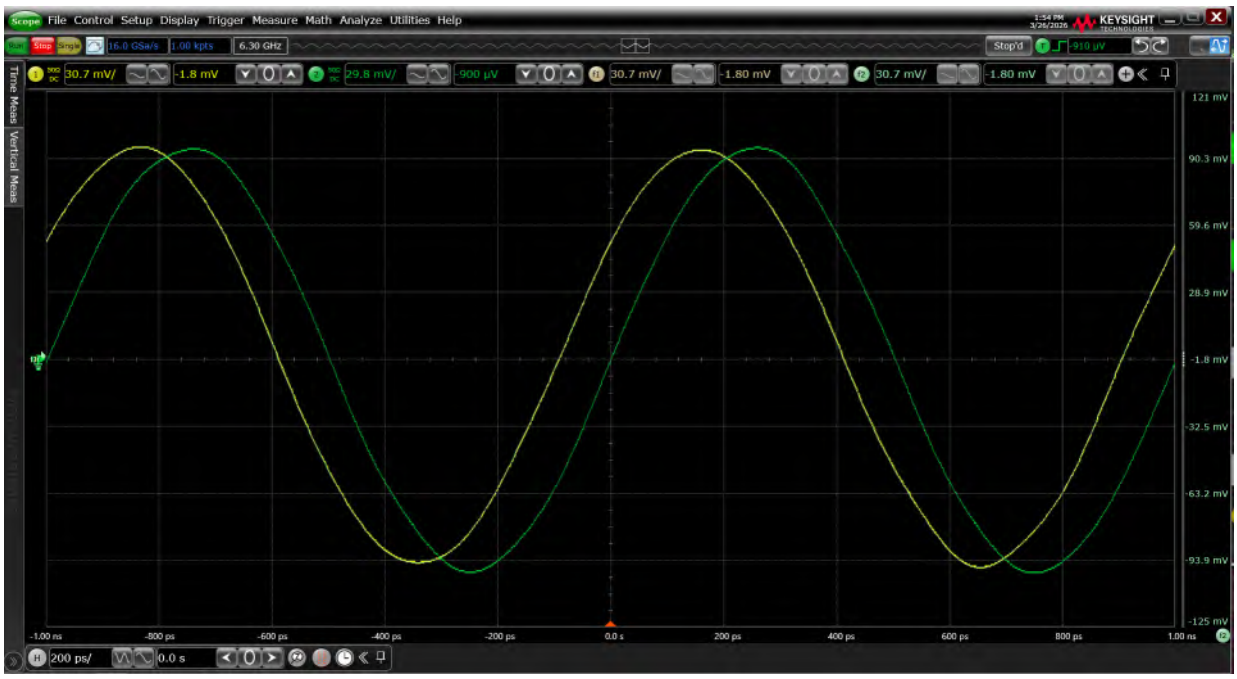
3. Select **Display** > **Setup** > **Windows** tab > and set **Autoscale Preference** to **Overlay Waveforms. Close** the dialog box.

   Tip: On the V-Series scope, set the Grid Layout to **Stack Grids Vertically**.



4. Select **Control** > **Autoscale**.

5. Select **Single** Sweep. Single Sweep and notice that the two signals are out of phase.



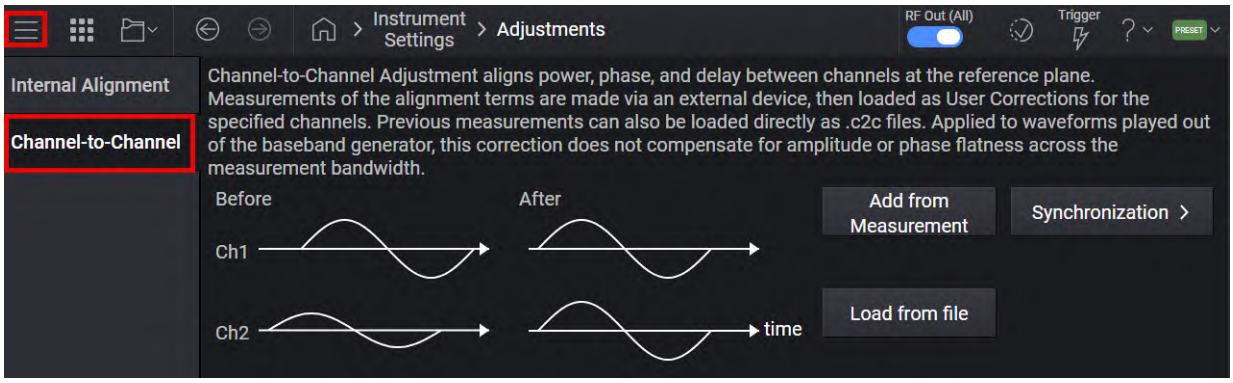
Channel-to-Channel Adjustment Wizard

Simplified workflow: Wizard style automated process for channel alignment across multiple frequencies.

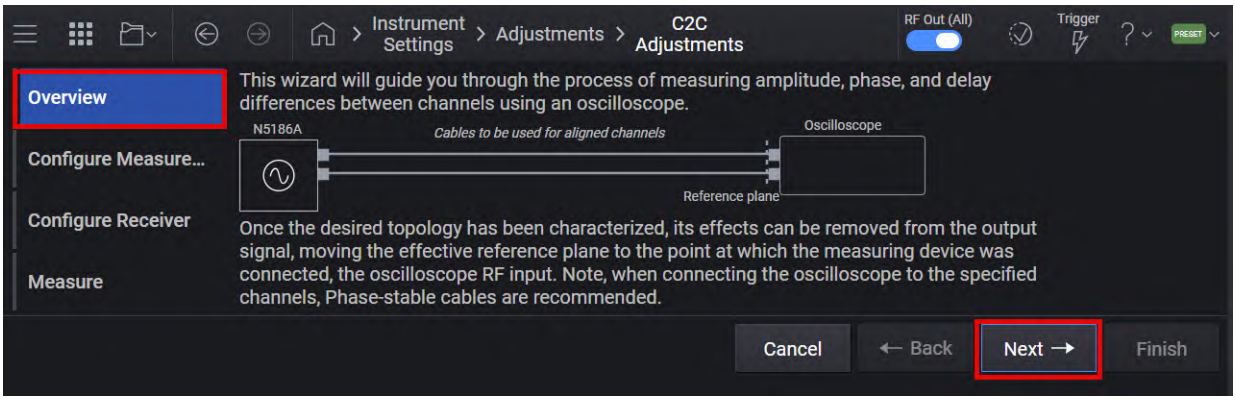
NOTE **The VSA must be running in order for the Channel-to-Channel adjustment to run.**

The VXG Adjustment Wizard automatically processes for channel alignment across multiple frequencies. The alignment (skew) is transfered from the oscilloscope to the source.

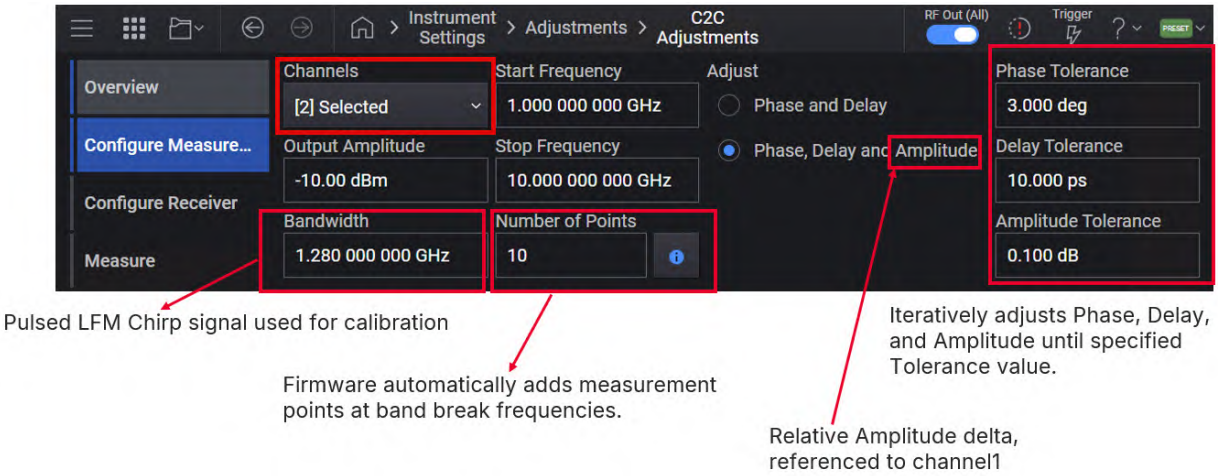
1. On the oscilloscope, open the VSA software by selecting **Analyze** > **Launch VSA**. This may take a few minutes to open.
2. On the **VXG**, select the **System** menu > **Settings** > **Adjustments** > **Channel to Channel**.



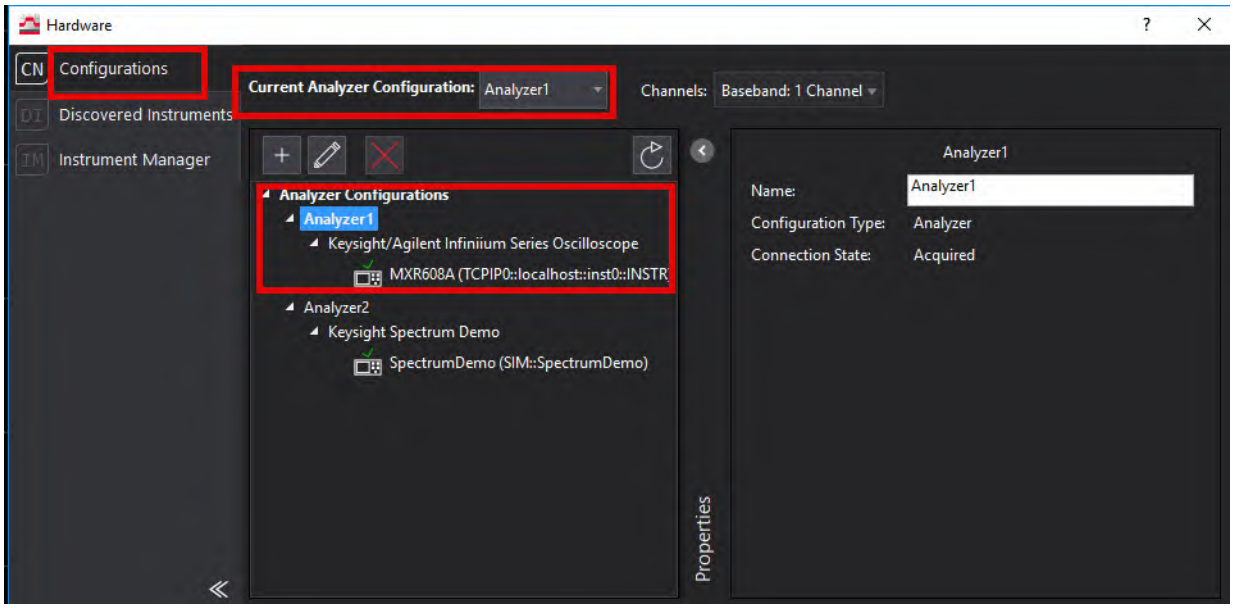
3. In the **Adjustments** window, review the information and then select **Add from Measurement**.
4. In the **Overview** window, review the information and then select **Next**.



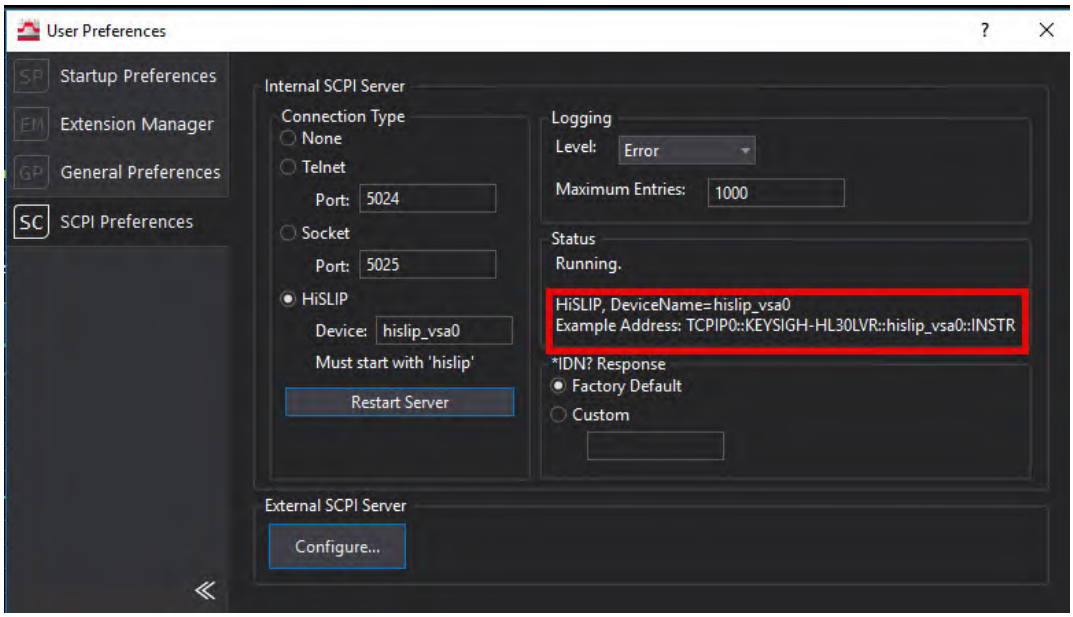
5. In the **Configure Measure** tab, select the **Channels** dropdown and select **Channels 1 and 2**. Customize the other settings as needed. Select **Next** to continue.



6. In the VSA, select **Utilities** > **Hardware** > **Configurations** and select the configuration for the MXR and close the dialog box.



7. Select **Utilities** > **SCPI Preferences** and copy the `Example Address` and note the HiSLIP Device name for the VSA. For this example, **vsa0**.

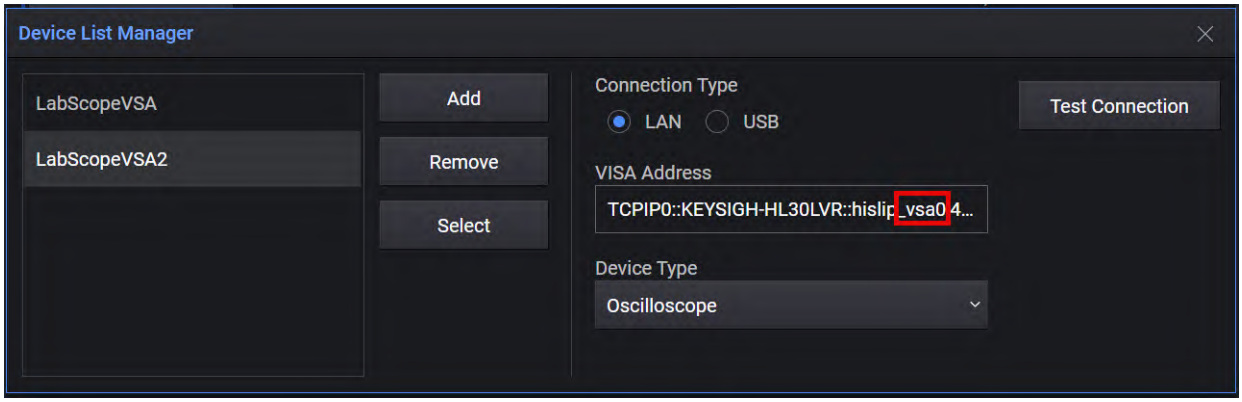


8. Go back to the **VXG**. In the **Configure Receiver** tab, select **Configure Device**.

   a. In the Device Manager window select **Add** to specify a name for the VSA connected to the oscilloscope.

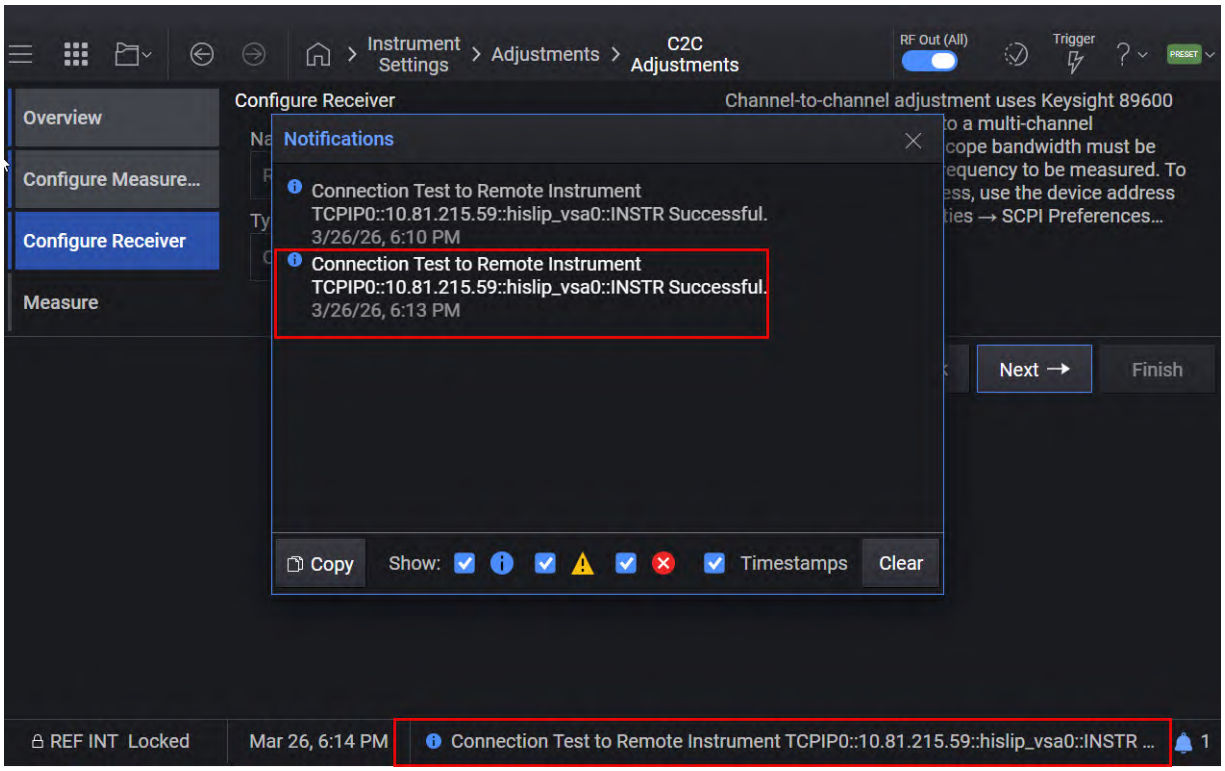
   b. Enter the VISA address for the MXR by pasting the VISA address into the text editor. Make sure to add the HiSLIP Device name for the VSA. For this example, vsa:0.

   c. Set Device Type to **Oscilloscope**.

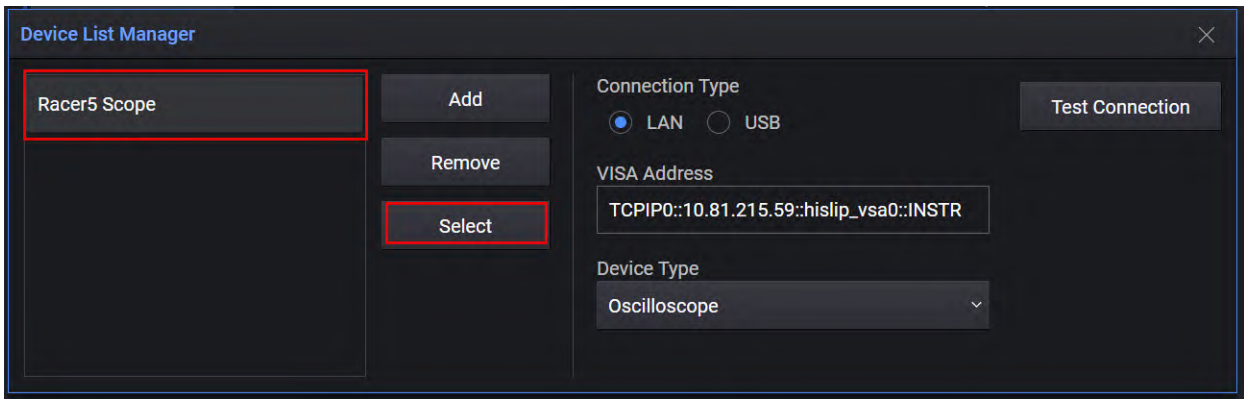


9. Select **Test Connection** to verify.

Then Connection results appear in at the bottom of the window. You may have to temporarily close the Device List Manager dialog box in order to read the full message at the bottom of the display.

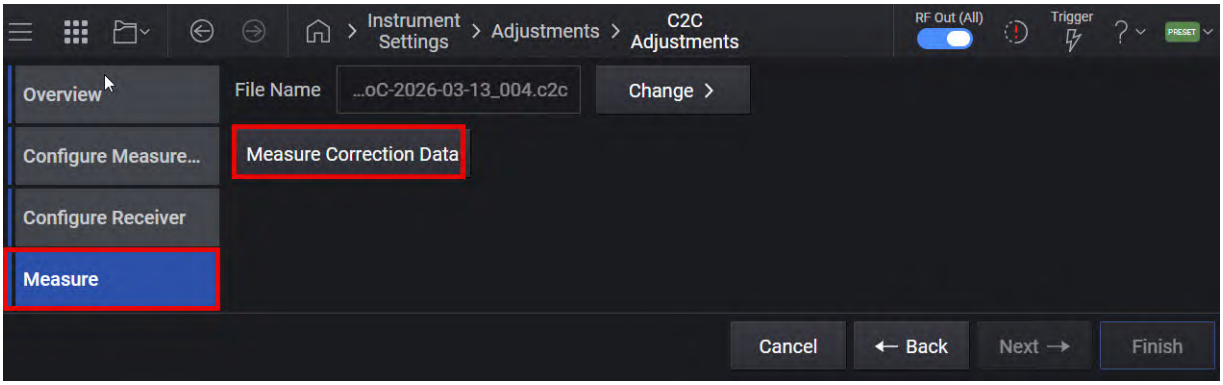


10. After confirming the connection to the VSA running on the MXR, select **Configure Device** > highlight the **Device name** > **Select** > **Next**.

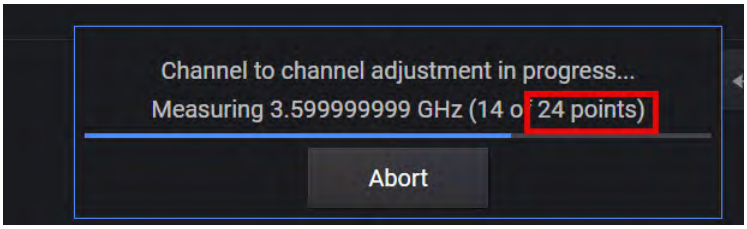


11. On the VXG **Measure** tab, select **Measure Correction Data** > **Measure**.

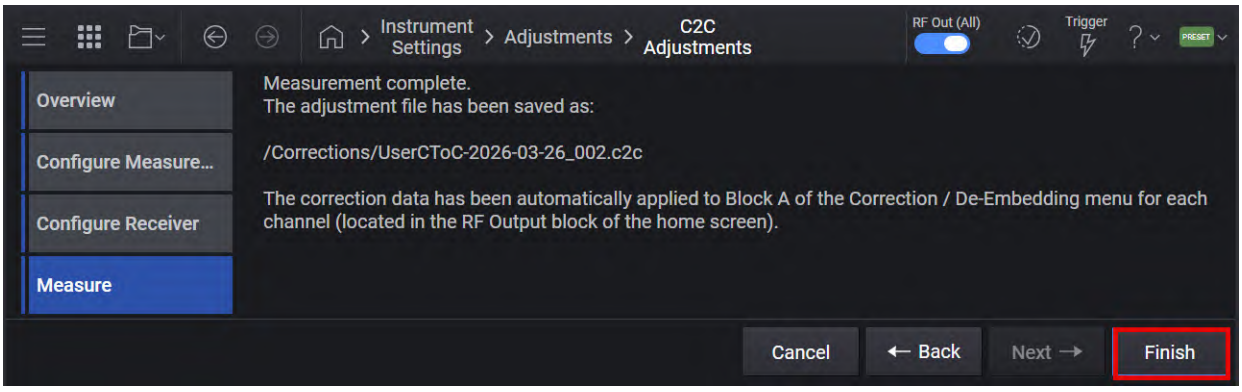
You can specify a filename and location for the correction data, or just use the default name and location.



Notice that the status message shows the adjustments measuring more points that what you specified in the Configure Measurement tab. This is because the frequency range of the VXG is divided into bands, and measurement points are added at the start and end of every break.



12. Select **Finish** when the adjustment is complete.



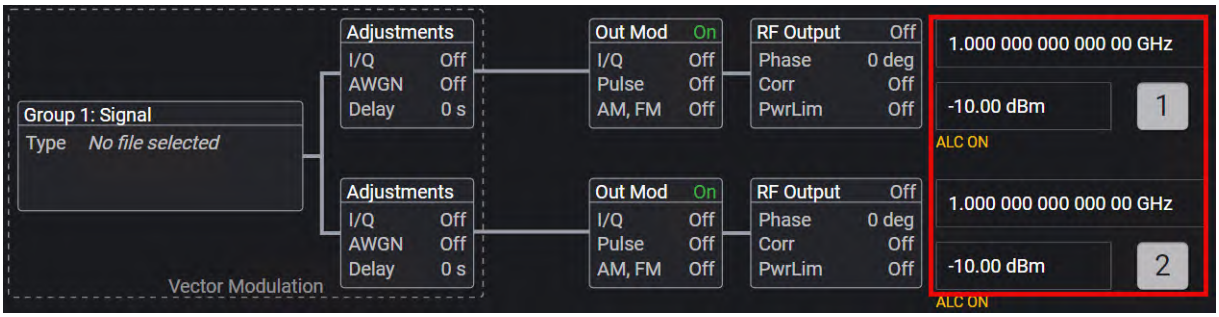
13. Close the VSA application. In the MXR, select **Control** > **Autoscale** and view the results of the channel-to-channel calibration.

Put the MXR back into **Single** Sweep to see the corrected traces.

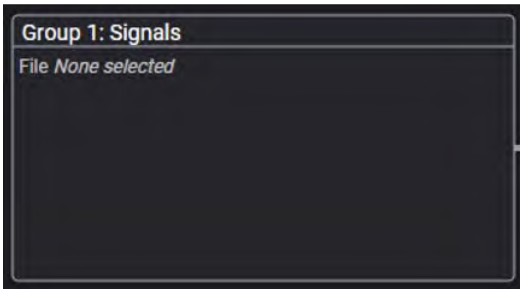
Setting up a Pulse LFM Chirp Waveform on the VXG

Now we will setup a measurement with the pulsed LFM chirp to show that phase alignment holds across the modulated signal bandwidth, not just at the carrier.

1. Select **Preset** > **Preset** to set the VXG to a known state.
2. Select the **System** menu > **Configure Channels** > **Group Configuration** dropdown and select **2 Tx Coherent** > **Add** > **Apply** > **Apply**.
3. Set Channel 1 and 2 Frequency to **1 GHz** and Power to **-10 dBm**.



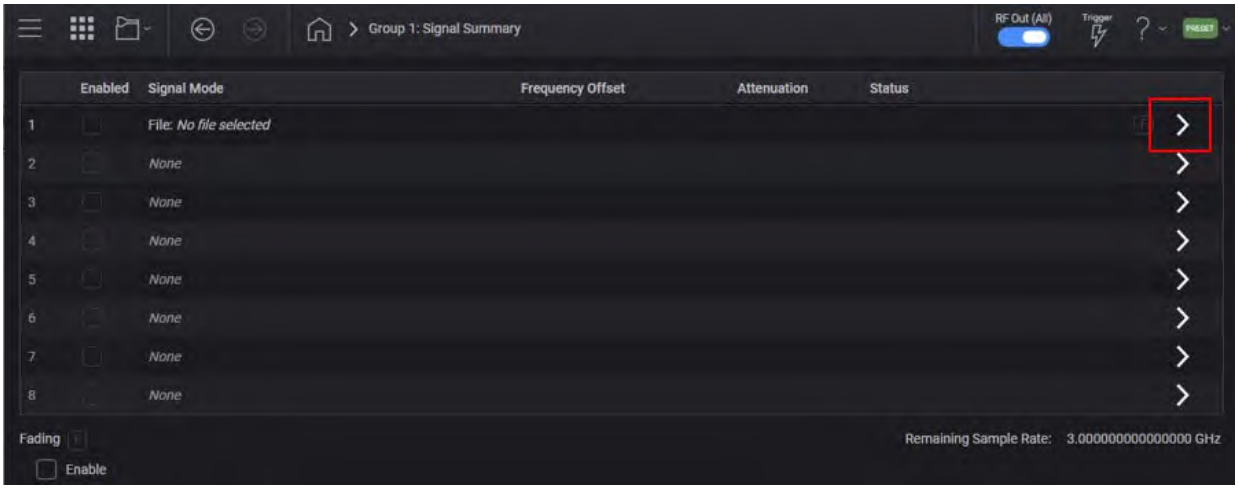
4. Select **Group 1: Signals** block to open.



5. Select the arrow for Signal 1 to open the Signal Setup window.

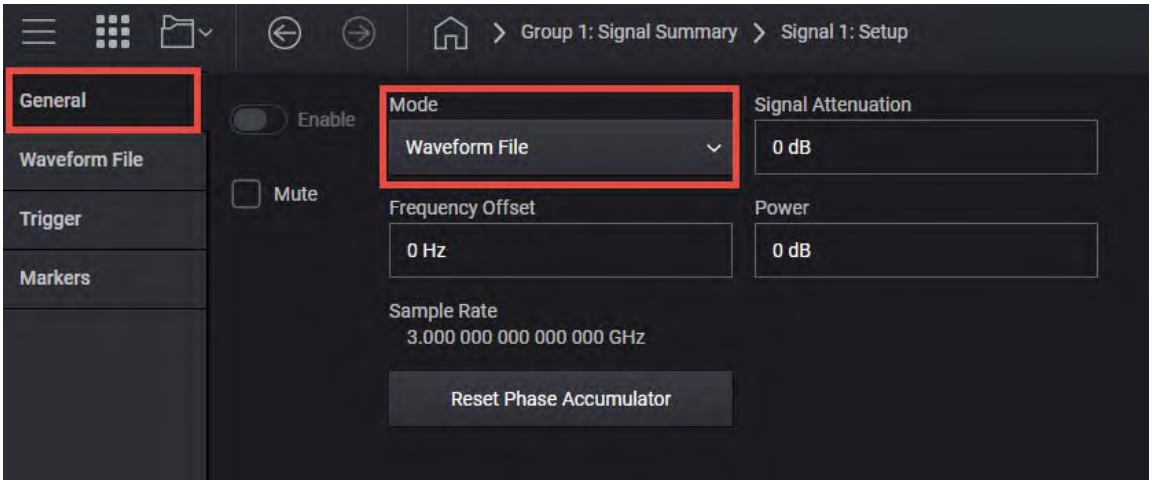
NOTE

This screen is only accessible if Option M9484C-8SG (8 virtual signal generators) is installed. For all other option configurations, continue to the next step.



6. In the Vector Modulation Signal Setup:

a. Select the **Mode** dropdown and set to **Waveform File**.

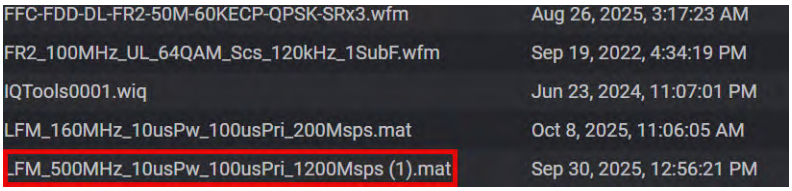


b. In the left pane, select the **Waveform File** tab.

c. Use **File Select** to navigate to:

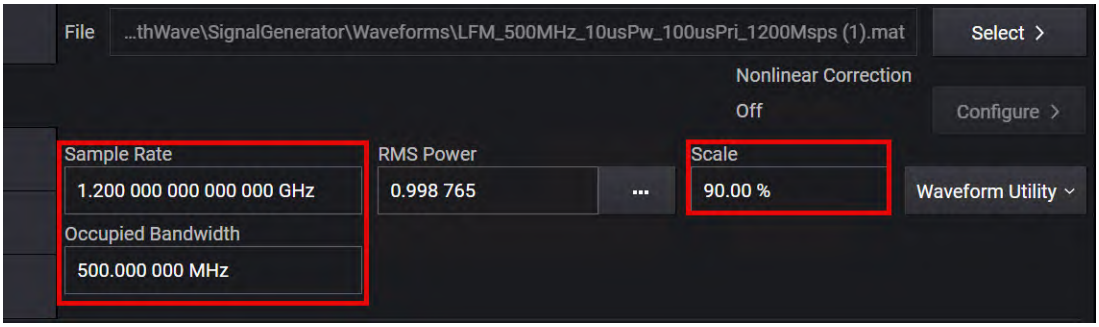
`D:\Users\Instrument\Documents\Keysight\PathWave\SignalGenerator\Examples`

and choose `LMF_500MHz_10usPw_100usPri_1200Msps.mat` › **Select**.

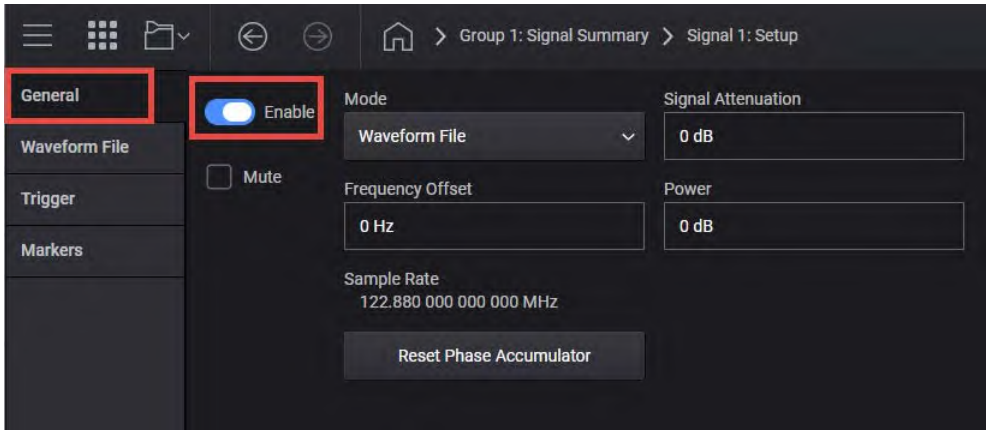


d. Set:

- Sample Rate to **1.2 GHz**
- Scale to **90%**
- Occupied Bandwidth to **1.28 GHz**



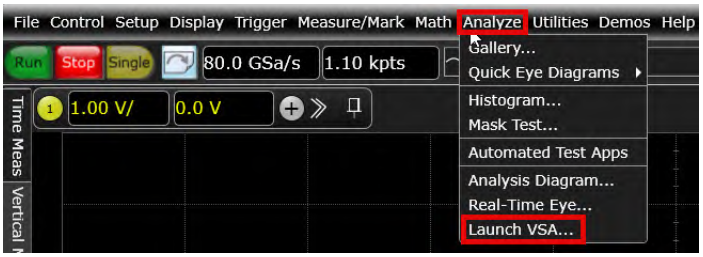
e. Select the **General** tab > **Enable**.



7. Set RF Out to **On** by selecting the numbered channel indicator next to both **Channel 1 and Channel 2**. switch.

Setting Up the VSA to Measure the Chirp

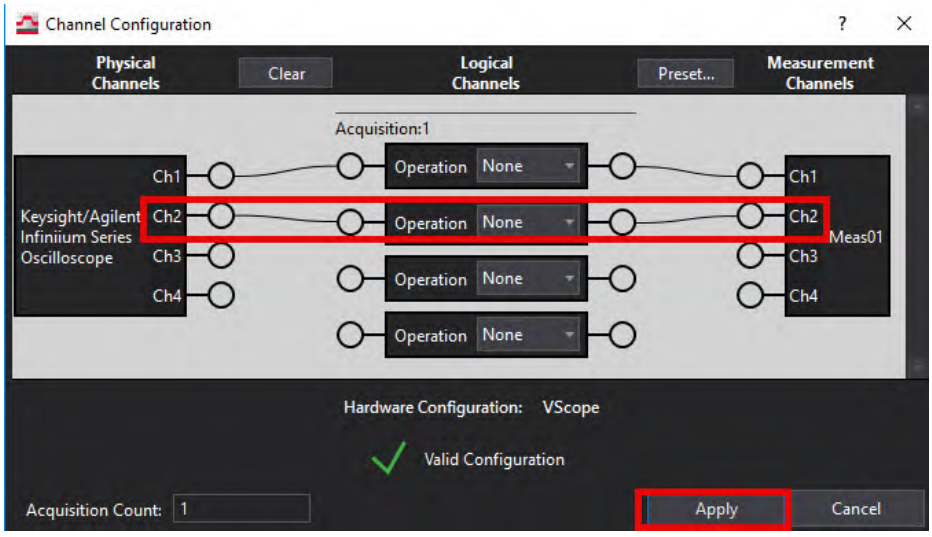
1. On the **MXR**, select **Analyze** > **Launch VSA** to open the Vector Signal Analysis application.



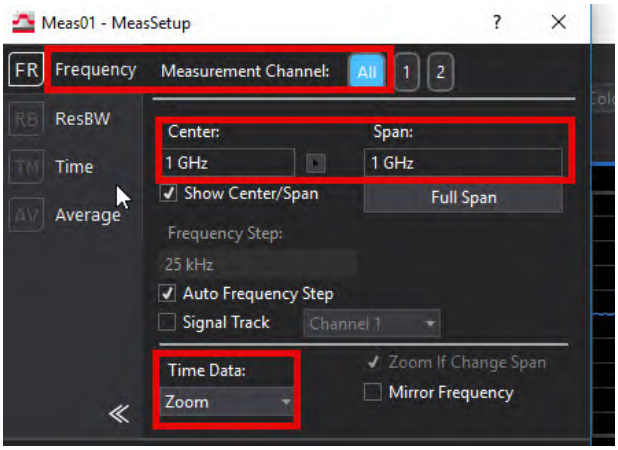
It may take a couple minutes for the VSA to open. You may notice the following message on the MXR display. This is just alerting you that any changes to the MXR will not take affect while the VSA is running.

2. From the VSA menu bar, select **File** > **Preset** > **Applications and Traces** to set the VSA to a known state.
3. Select **Meas Setup** > **Measurement Type** > **Vector.**
4. Select **Input** > **Channels** > **Custom** to open Channel Configuration.

5. Click and drag the Oscilloscope Ch2 node to the second Operation node, then click and drag the right side node to the Ch2 node as shown below. Select **Apply** and close the dialog box.

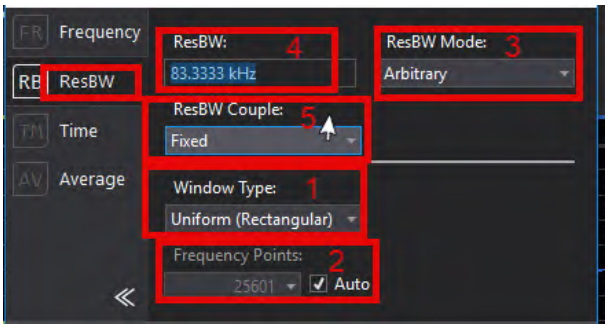


6. Select **MeasSetup** > **Frequency** tab and set Center to **1 GHz** > Span **1 GHz**.

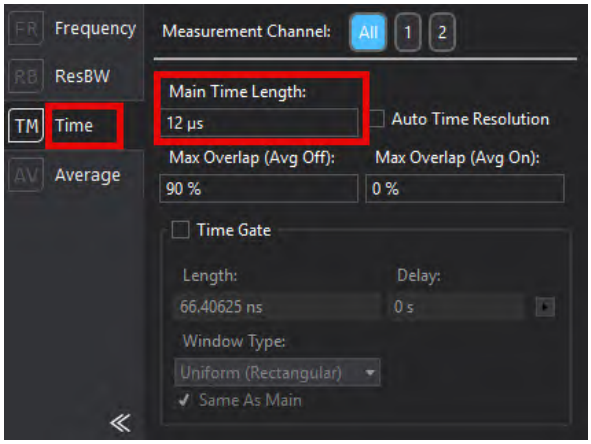


7. From the MeasSetup dialog box select **ResBW** tab. Set the following parameters in this order:
   a. Window Type to **Uniform**
   b. Frequency Points to **Auto**
   c. ResBW Mode to **Arbitary**
   d. ResBw to **83.3333 kHz**

e. ResBW Couple to **Fixed**



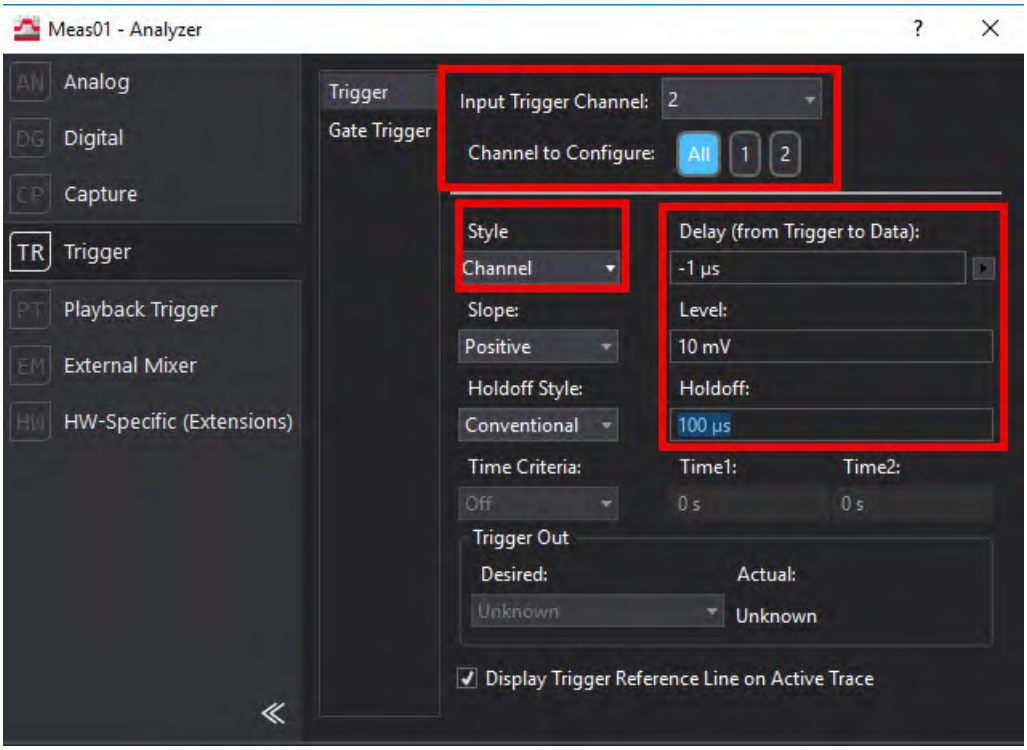
8. From the MeasSetup dialog box select **Time** tab and set Main Time Length to **12 µs** for All channels. Close the dialog box.



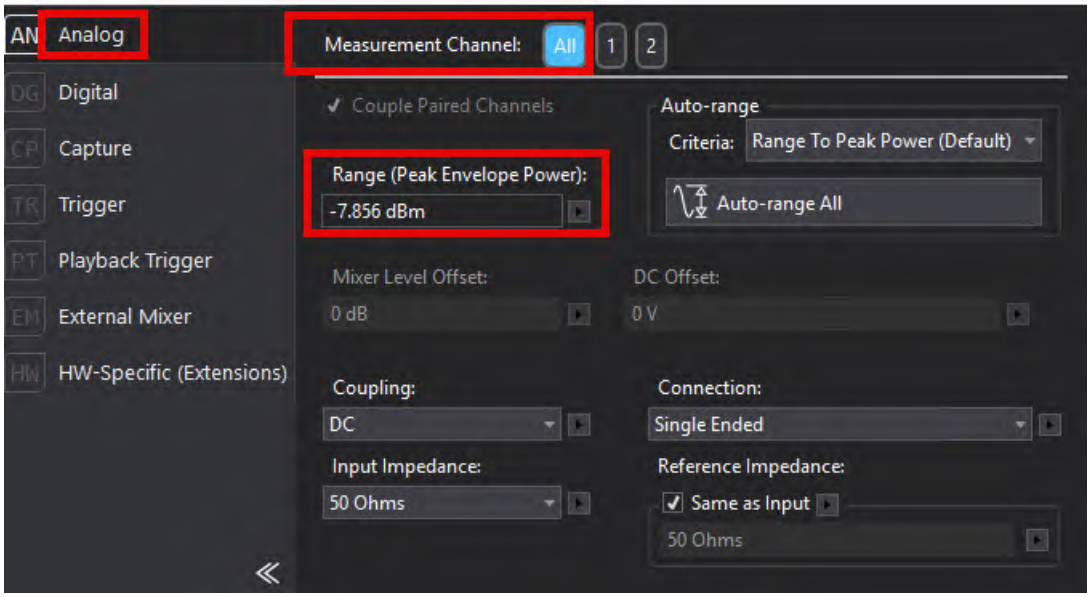
9. Select **Input** > **Trigger** tab and set:
   - Input Trigger to **2**
   - Style to **IF Mag**

     Tip: When using a V-Series oscilloscope, set Style to **Channel**
   - Delay (from Trigger to Data) to **-1µs**
   - Level to **10 mV**

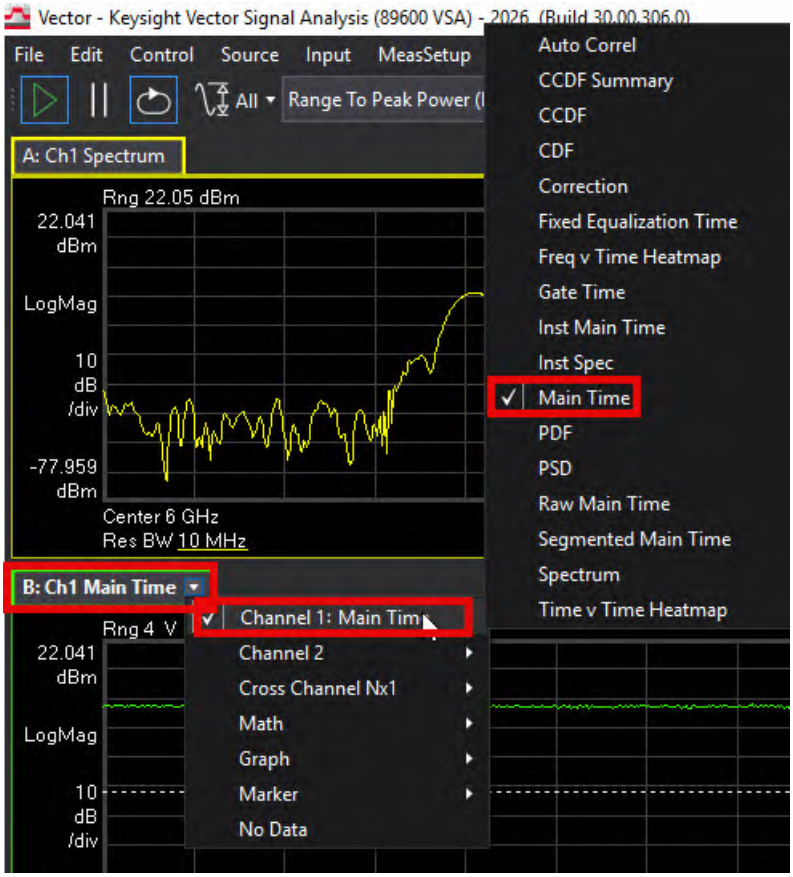
– Holdoff Style Conventional to**100 µs**



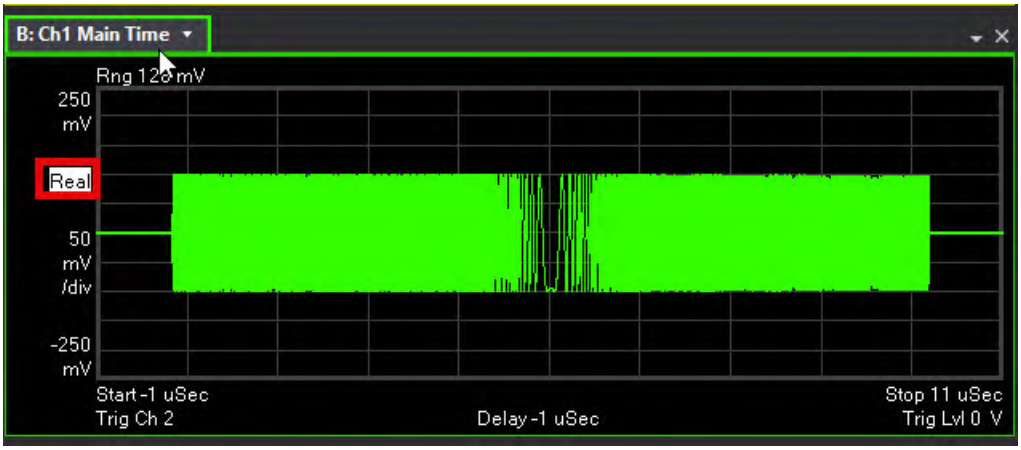
10. From the **Input** menu > **Analog** tab and set Range (Peak Envelope Power) to **-7.85 dBm** for **All** Measurement Channels. These settings will center the waveforms on the windows. Close the dialog box.



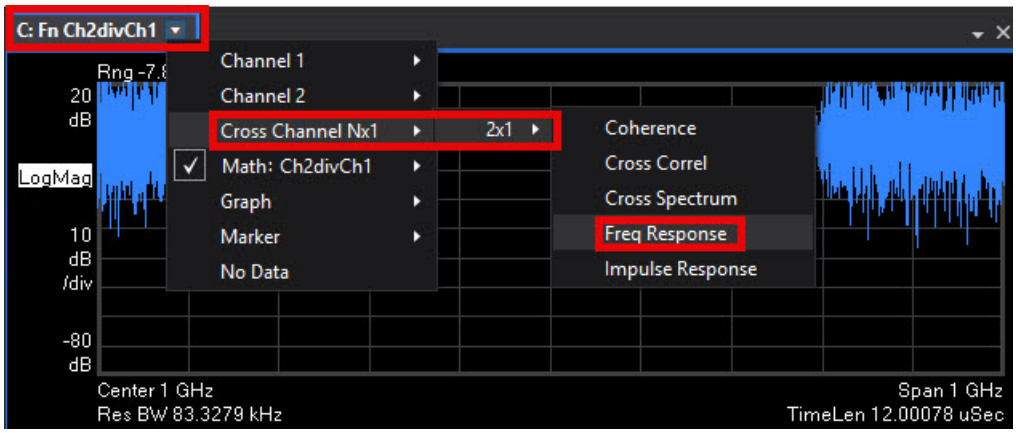
11. Select **Window B** dropdown menu and select **Channel 1 Main Time**.



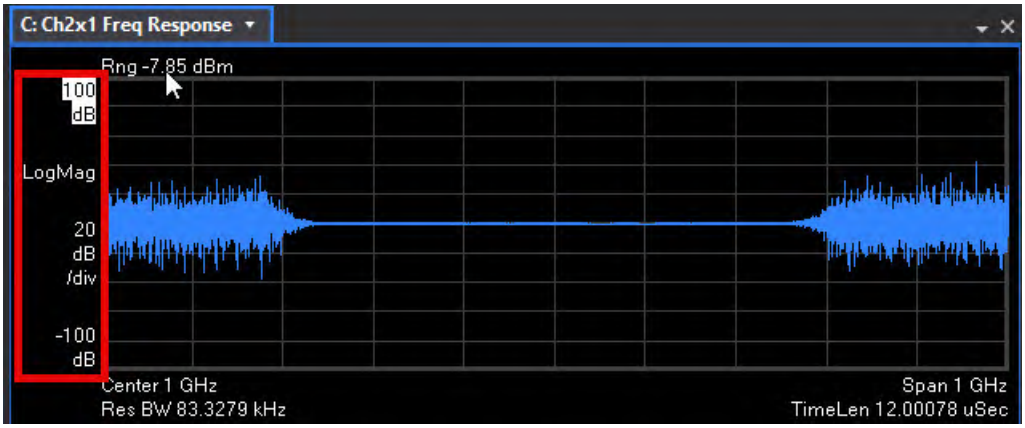
12. In the Channel B Window, select **LogMag** on the left side and change to **Real**.



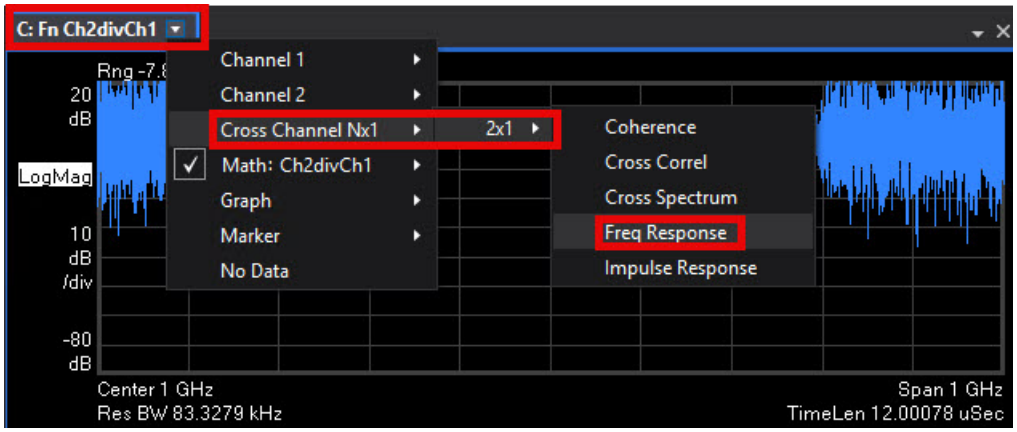
13. Select **Window C** dropdown menu and select **Cross Channel Nx1** > **Frequency Response**.



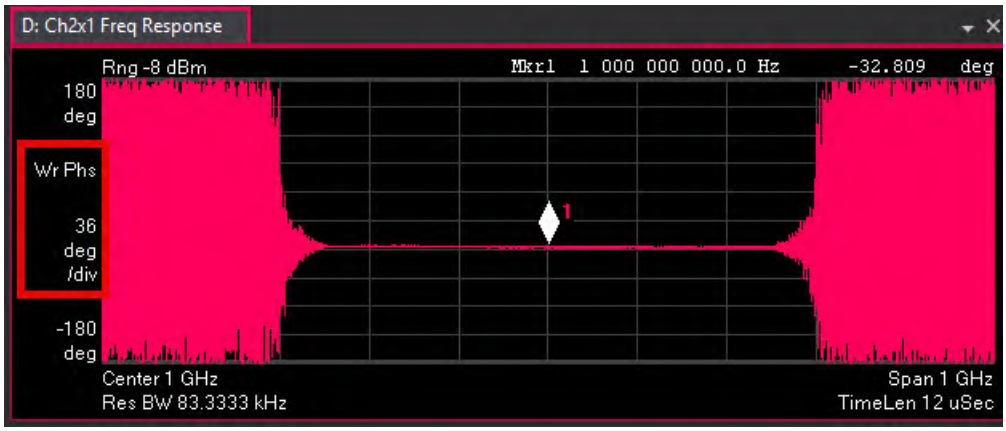
14. Change db/div to **20 dB** and the range from **100 dB to -100 dB**.



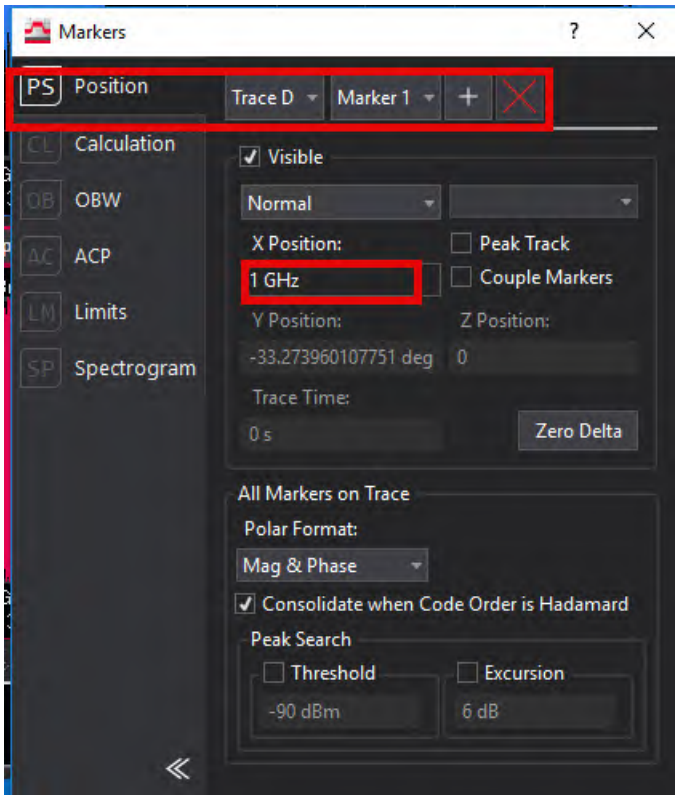
15. Select **Window D** dropdown menu and select **Cross Channel Nx1** > **Frequency Response**.



16. In the Channel D Window, select **LogMag** on the left side and change to **Wrap Phase**.

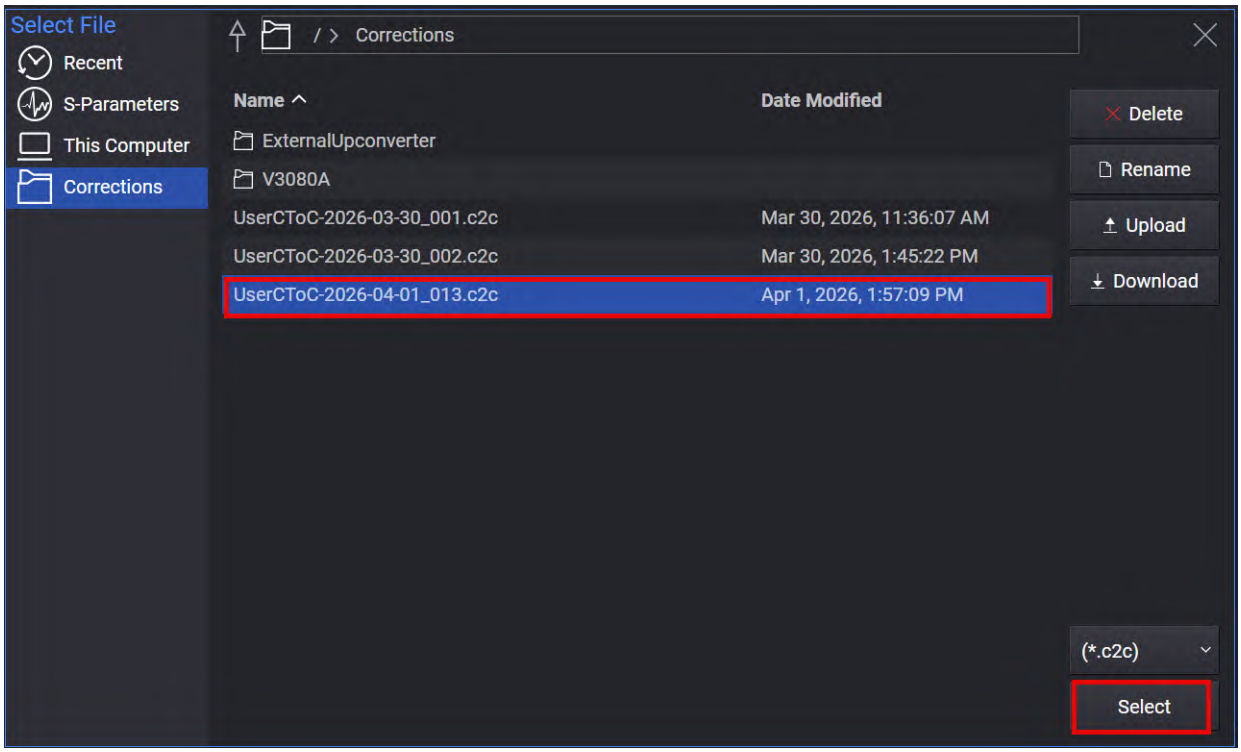


17. Select **Markers** > Marker1 **On**. Set Marker 1 **X Position** to **1 GHz** on **Trace D**. Note the degrees of offset.

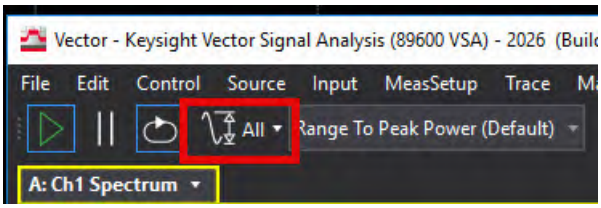


Now we need to turn the corrections on for this waveform.

18. From the VXG, select the **System** menu icon > **Settings** > **Adjustments** > **Channel to Channel** > **Load from File**. Scroll down and fine the most recent UserCToC file and **Select**.



19. From the **VSA** menu bar, select **Auto Range** and view the calibrated response.



The following screen capture was taken before Channel-to-Channel Adjustment.

The following screen capture was taken after Channel-to-Channel Adjustment. The adjustment went from 133 degrees of offset to <1 degree of offset.

3 Advanced Measurements

- "Channel Bonding" on page 202
  - "Generating a Multi-Carrier DVB-S2X Signal for Satellite TV" on page 203
  - "Generating a Wide Band QPSK" on page 217
  - "Generating a 5G NR Waveform" on page 226
- "Setting Up 8 Virtual Signal Generators" on page 236
- "Using the VSA Flex Frame to Create a DVB-S2X Signal" on page 247
- "Creating & Analyzing a 5G NR DL 2x2 MIMO Measurement Using Two UXAs" on page 287
- "Creating and Analyzing a 4x4 MIMO using a UXR Running VSA Software" on page 310
- "Setting up a 1x8 MIMO and PUCCH Fading Example" on page 324
- "Setting a 1x4 MIMO and Fading HARQ Example" on page 339
- "Setting a 1x4 SYNC and Fading (5GNR & Custom) Example" on page 357
- "Setting up Envelope Tracking" on page 383

Channel Bonding

Data requirements are pushing bandwidths and modulation complexity across many global industries.

- Global data consumption is growing
- IoT drives new data consumption
- Autonomous vehicles require data links
- Satellite networks need to provide 5G backbone in 5G Non-Terrestrial Networks (NTN)
- New applications generate new sources of data intelligence

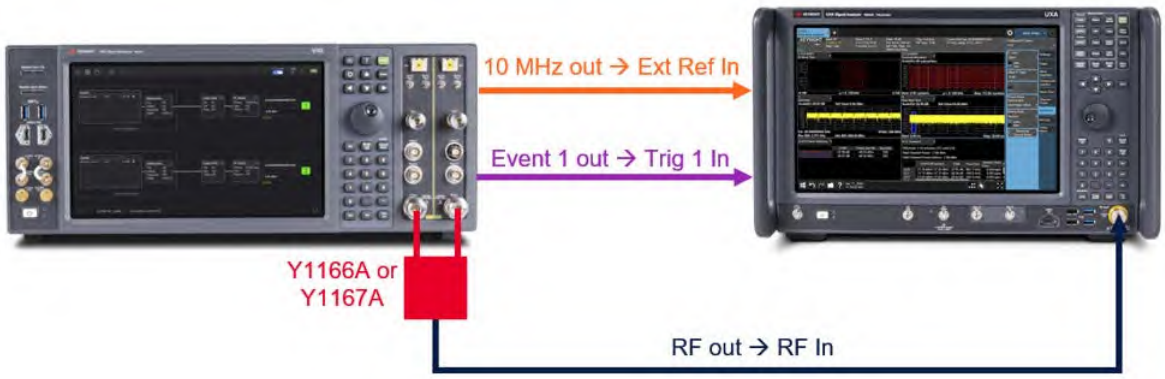
The following three examples are showing

- "Generating a Multi-Carrier DVB-S2X Signal for Satellite TV" on page 203
- "Generating a Wide Band QPSK" on page 217
- "Generating a 5G NR Waveform" on page 226

For this measurement example, we are using the following equipment:

- M9484C with options
  - Two channels (M9484C-001 and 002)
  - RF bandwidth of 2.5 GHz per channel (M9484C-R25)
  - Channel bonding, 5 GHz (M9484C-CB5 and M9484C-PCH)
  - PathWave Signal Generation for Custom Modulation (N7608APPC)
  - PathWave Automatic Channel Response Correction and S-parameter De-embedding (N7653APPC)
- Accessories
  - Channel Bonding Combiner kit (Y1166A or Y1167A, frequency dependent)
- N9042B UXA
  - Analysis bandwidth of 4 GHz (N9042B-R40)
  - 89600 PathWave Vector Signal Analysis software, VSA

Figure 3-1 Channel Bonding Setup

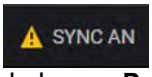


Generating a Multi-Carrier DVB-S2X Signal for Satellite TV

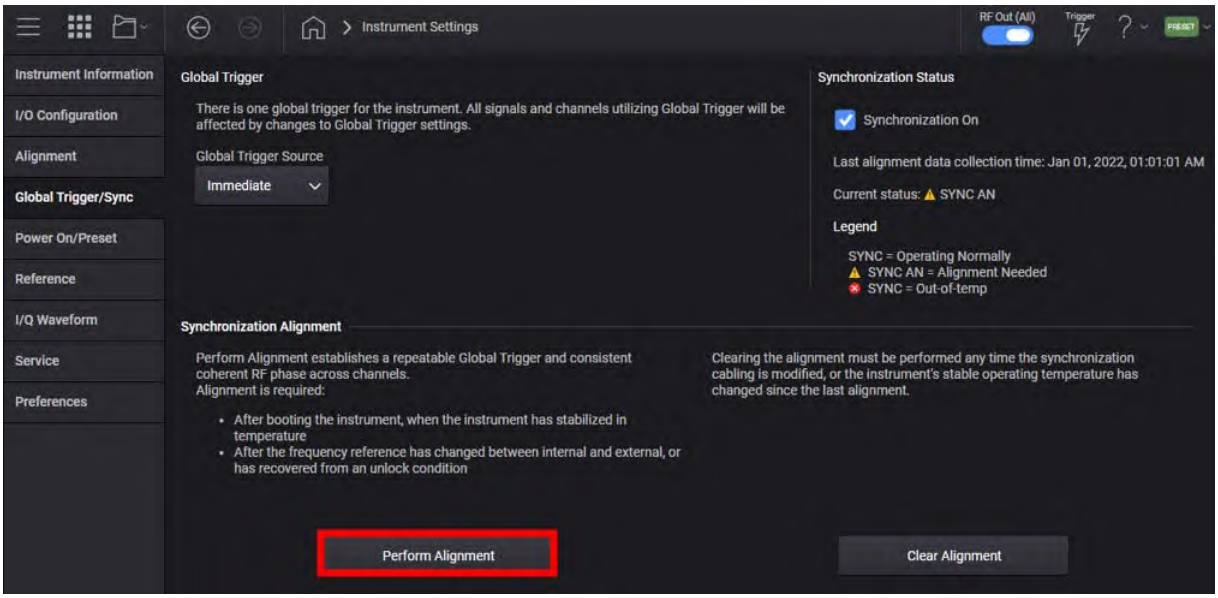
DVB-S2X is a digital satellite television broadcast standard. Channel bonding can be used for UHDTV services where statistical multiplexing is used to allow many television channels can use the same bandwidth. Other applications include Direct to Home (DTH), Digital Satellite News Gathering (DSNG) and professional VSAT.

On the VXG:

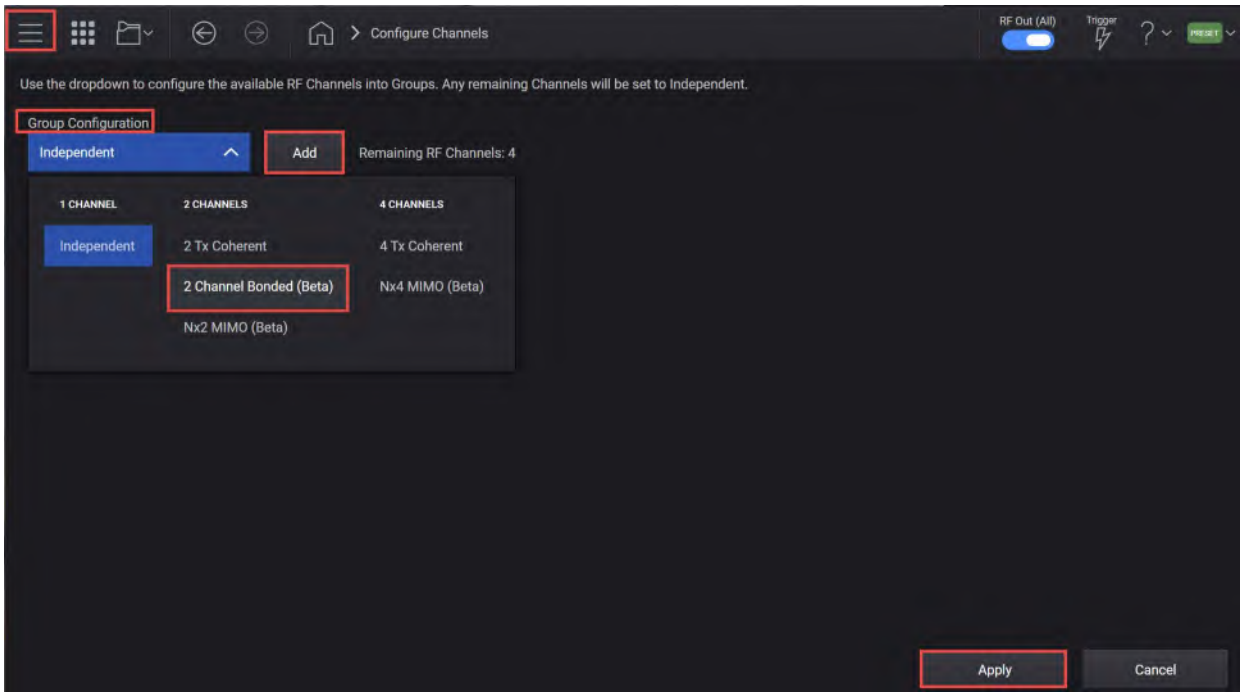
1. Connect the equipment as shown above.
2. Select **Preset** > **Preset** to set the VXG to a known state.

If a Synchronization Alignment is required, indicated by 

in the bottom left corner, tap or click the warning message and choose **Perform Alignment** before proceeding.

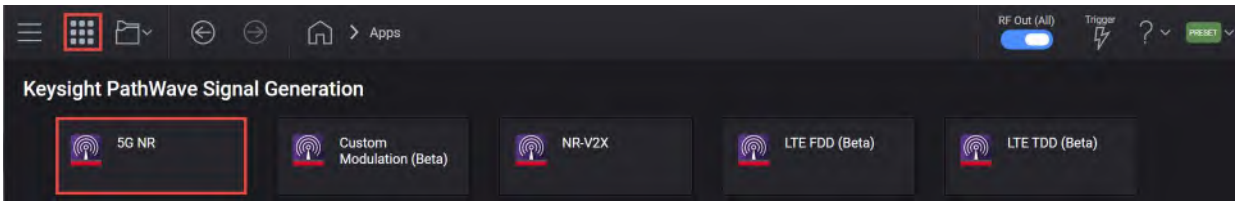


3. Select the System menu (triple bar icon at the top left of the window), and then select **Configure Channels** > **2 Channel Bonded** > **Add** (for 4 channel instruments only) > **Apply** > **Apply**.

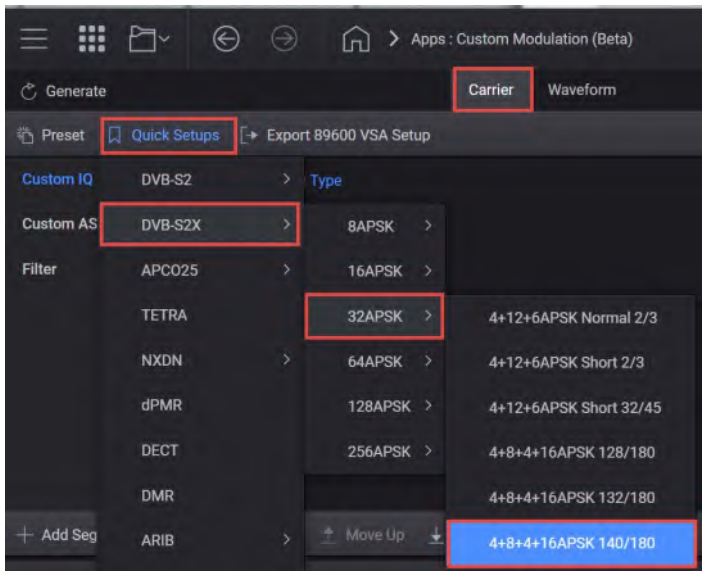


4. Set the Center Frequency to **12 GHz** and Power to **0 dBm**.

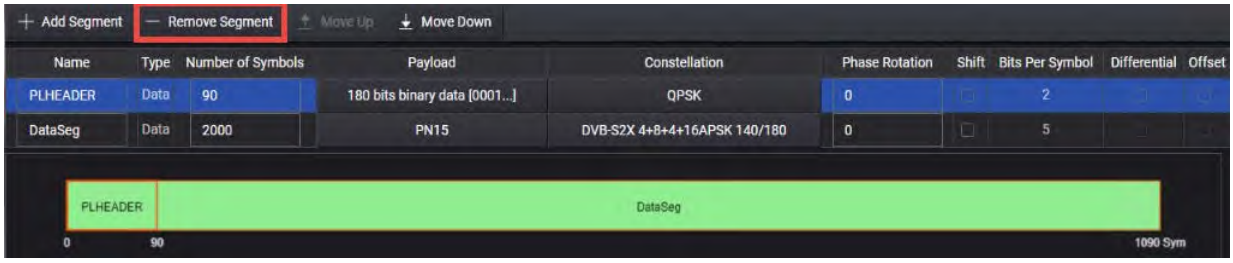
5. Select the **Apps** block to open, then select **Custom Modulation**.



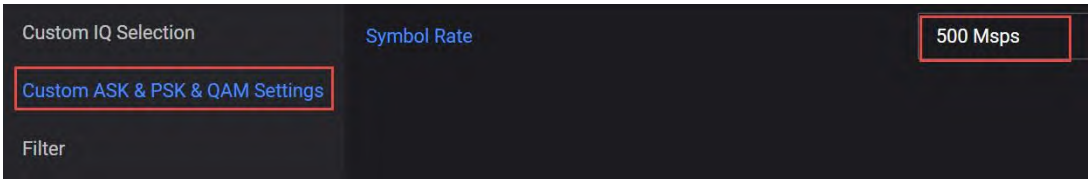
6. In the Custom Modulation setup, select the **Carrier** tab > **Quick Setups** > **DVB-S2X** > **32APSK** > **4+8+4+16APSK 140/180**.



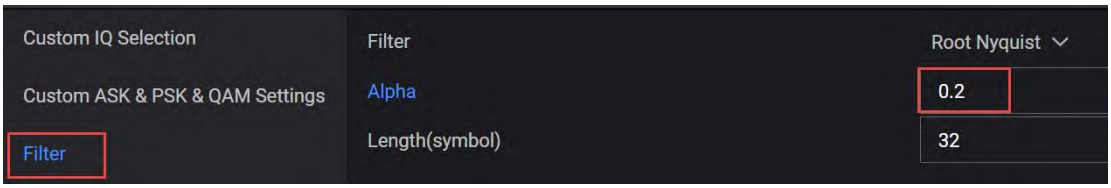
7. From the bottom pane, select **Remove Segment** to remove the Header. This will simplify the demonstration.



8. From the left pane, select **Custom ASK & PSK & QAM Settings** and set the Symbol Rate to **500 Msps** per channel.

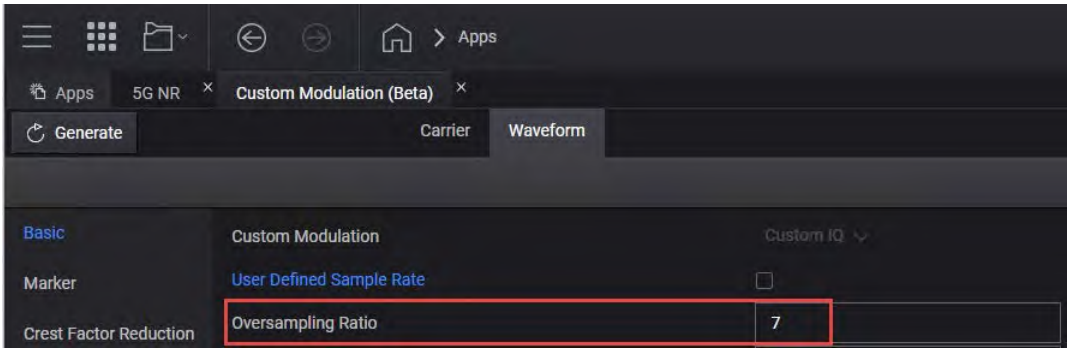


9. Select **Filter** > Alpha to **0.2**.



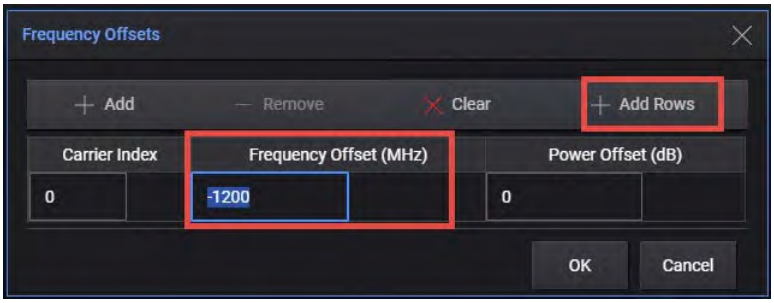
10. Select the **Waveform** tab > **Basic** > Oversampling Ratio to **7** x multiplier which provides a span of 3.5 GHz.

    Each channel is 2.56 GHz for a combined sample rate of 5.12 GHz. This could be set by entering 5.12 GHz as the User Defined Sample Rate however; for Multi-carrier it is best to set the Oversampling Ratio.

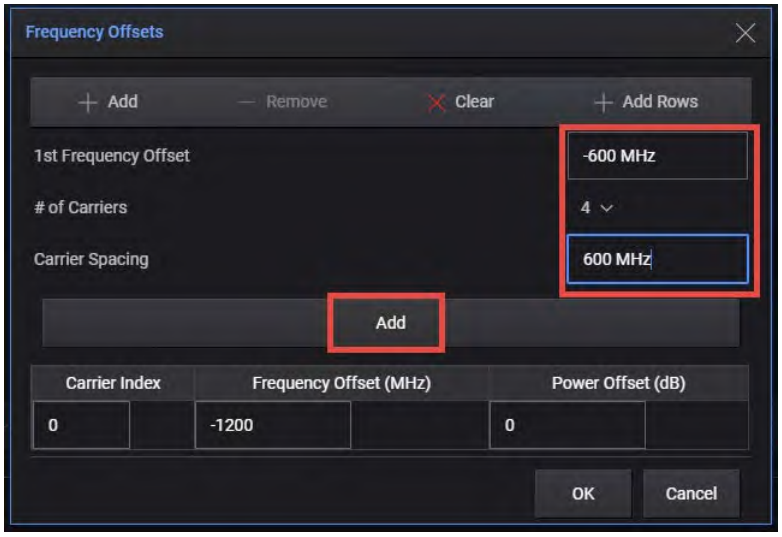


11. From the left pane, select **Multi Carrier** and select **Multicarrier Enabled**.

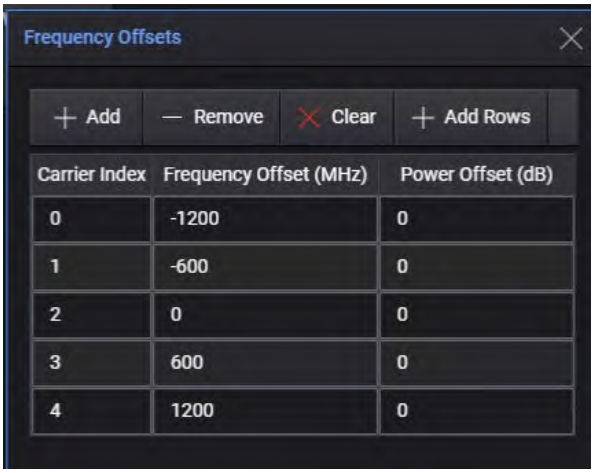
12. Open the **Frequency Offsets** setup screen. For Carrier Index 0, set the Frequency Offset to **-1200 MHz**, then select **Add Rows**.



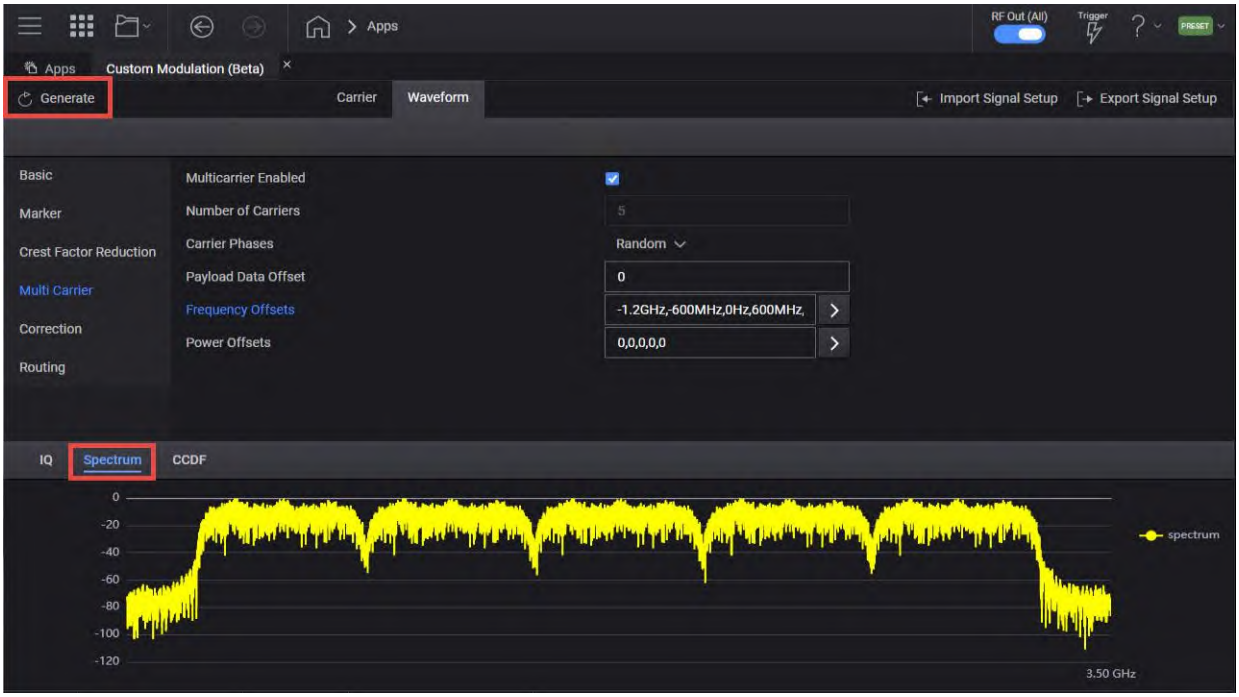
13. Enter the 1st Frequency Offset of **-600 MHz**, # of Carriers **4**, and Carrier Spacing **600 MHz**. Select **Add**.



All five carriers will be setup. Select **OK** to close the Frequency Offsets dialog.

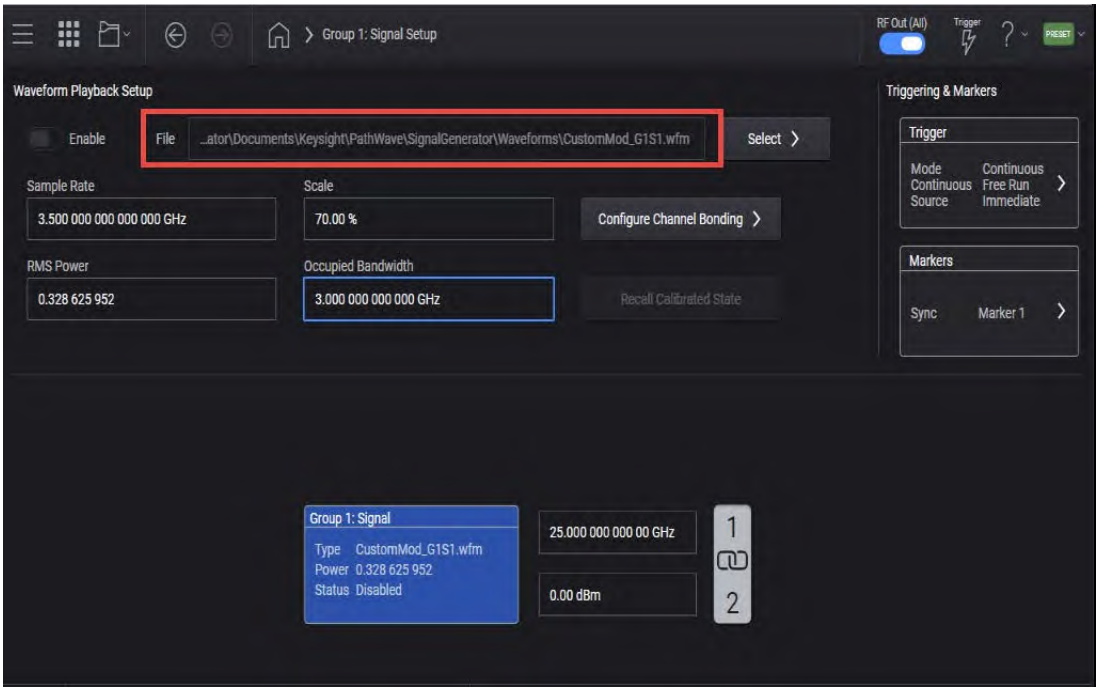


14. **Generate** the waveform and select the **Spectrum** tab from the bottom pane to ensure the resulting waveform is correct.

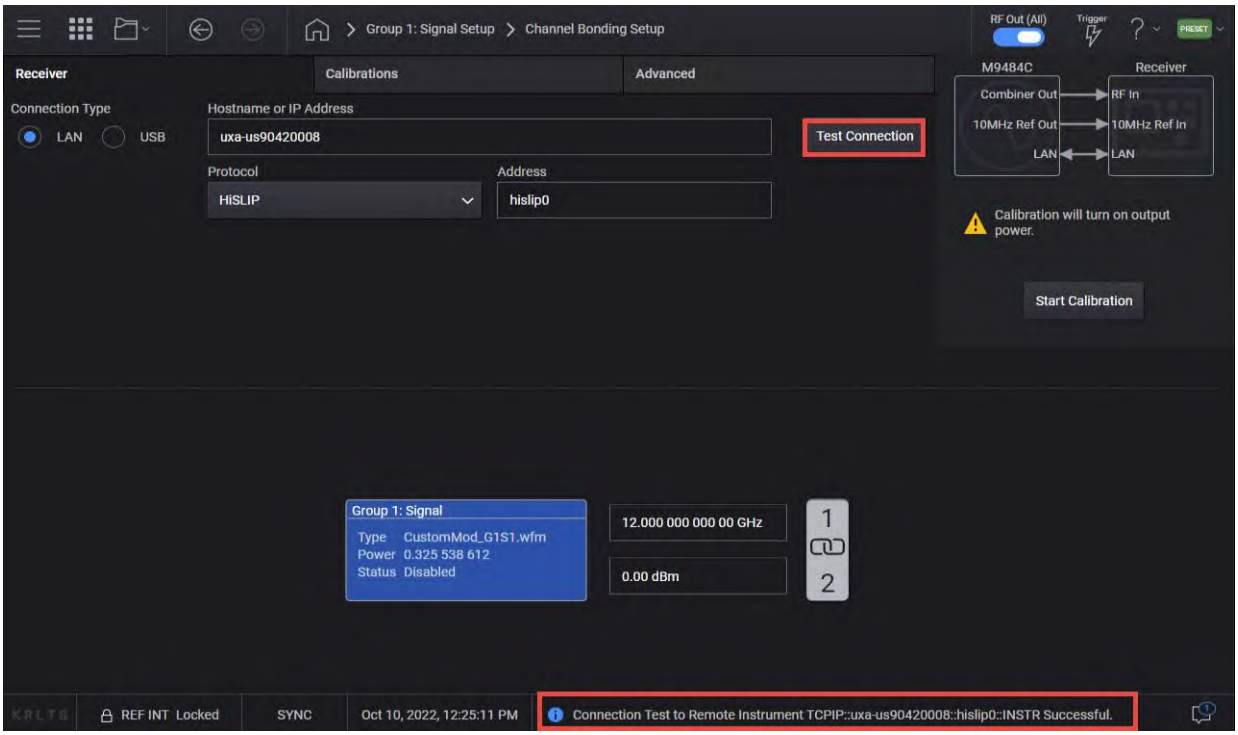


15. Select the **Home** icon to return to the main window and then select the **Signal Block** to open.

16. In the Signal Setup dialog box, notice that new custom modulation file is already selected (`CustomMod_G1S1.wfm`). Set the Occupied Bandwidth to **3 GHz**.



17. Select **Configure Channel Bonding** and configure the Hostname or IP address for the UXA, then **Test Connection**. Ensure that you have successfully connected to the UXA, a message will be displayed in the Status area at the bottom of the display.

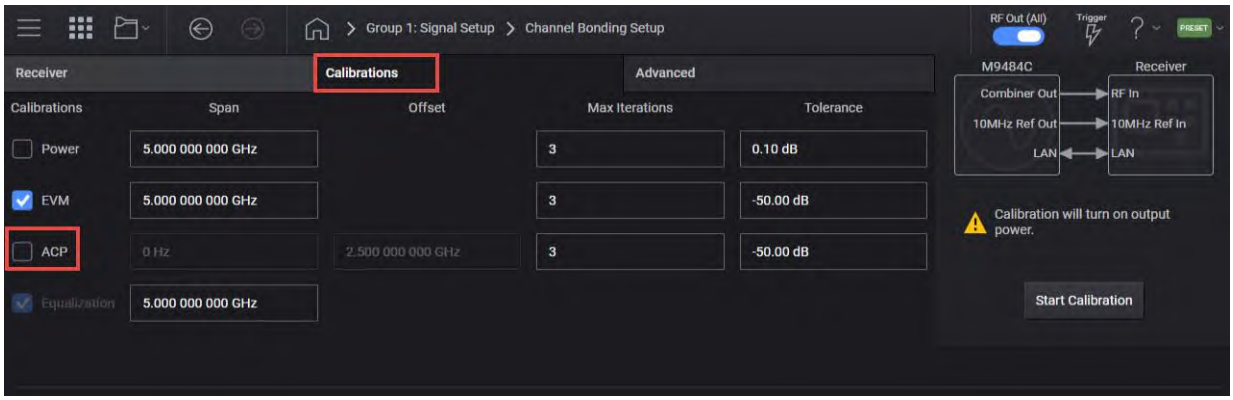


18. Select the **Calibrations** tab and clear the **ACP** calibration.

    In some cases it is not possible (or necessary) to improve the ACP of the bonded waveform. In these cases, it is best to disable it.

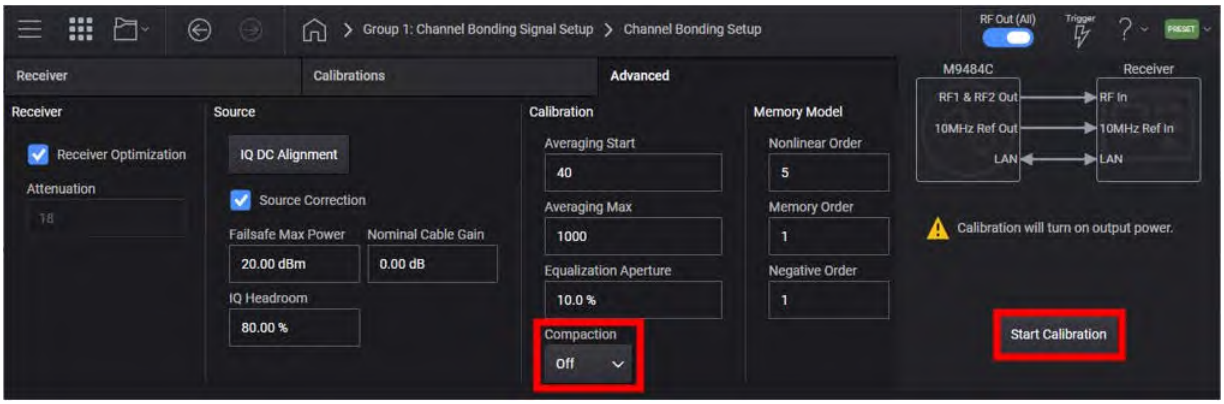
    - If the ACP cannot be improved, in some cases, the highest fidelity bonded waveform will result from Bonding with ACP disabled.
    - ACP increases the time it takes to perform Bonding, and if the receiver does not have sufficient analysis bandwidth to cover the ACP regions + modulated bandwidth of the waveform, then frequency-stitching will be performed and this will further increase the Bonding time.
    - With ACP enabled, the resulting signal will typically have significantly increased bandwidth, sample rate, and number of samples (as compared to a Bonded signal with ACP disabled).

– Note that the available ACP region bandwidth is limited by the bandwidth of the source, so if the available source bonded-bandwidth is 5 GHz, and the bonded waveform is 5 GHz, then there is no available BW for performing an ACP calibration.



19. Select the **Advanced** tab and set **Compaction Off**, then **Start Calibration**.

When the waveform length is short enough, compaction is not needed. However, for 5G signals (which tend to be relatively long and have a lot of IQ samples), the calibration time can be long (without compaction). So for signals with lots of IQ samples, the suggestion is to use compaction as this will reduce the calibration time. The calibration takes a few minutes to complete. However, the highest fidelity bonded waveform may result without compaction. How much compaction to use can be optimized (none, automatic, or a specified compaction level) by experimenting with the different settings.



The calibration takes a few minutes to complete.

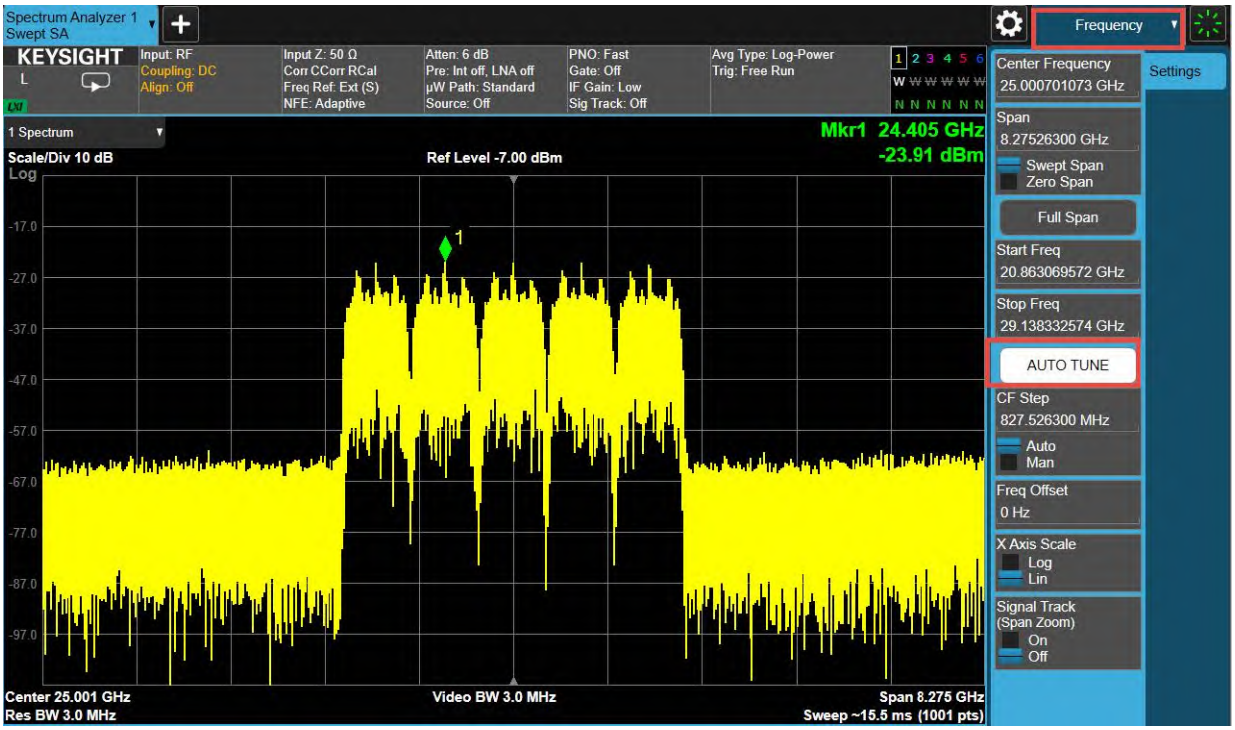
Notice that the CustomMod_G1S1.bnd waveform is now playing.

On the Signal Analyzer:

1. Select **Frequency** > **Auto Tune** to observe the spectrum.

NOTE

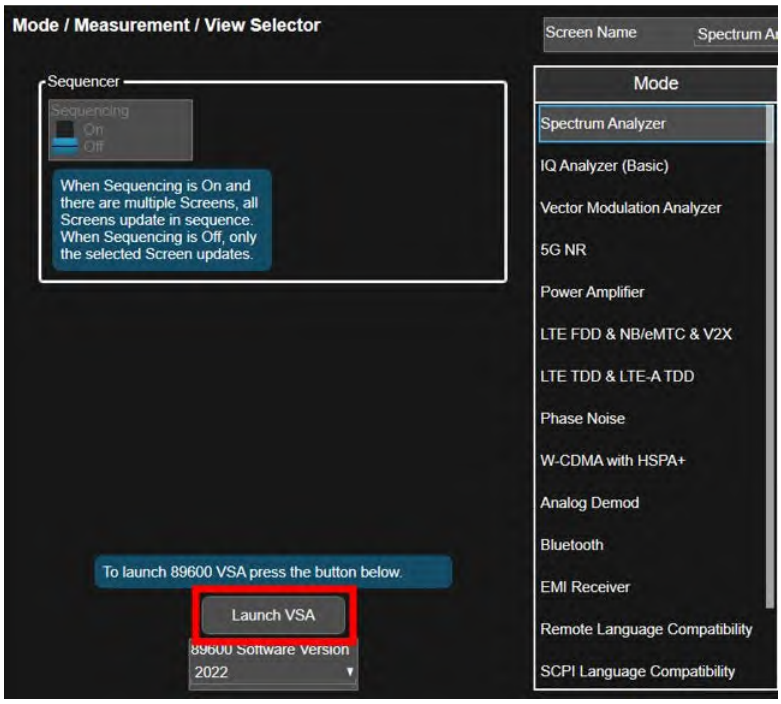
Ensure that signal analyzer is set to Spectrum Analyzer mode > Swept SA Measurement > View Normal.



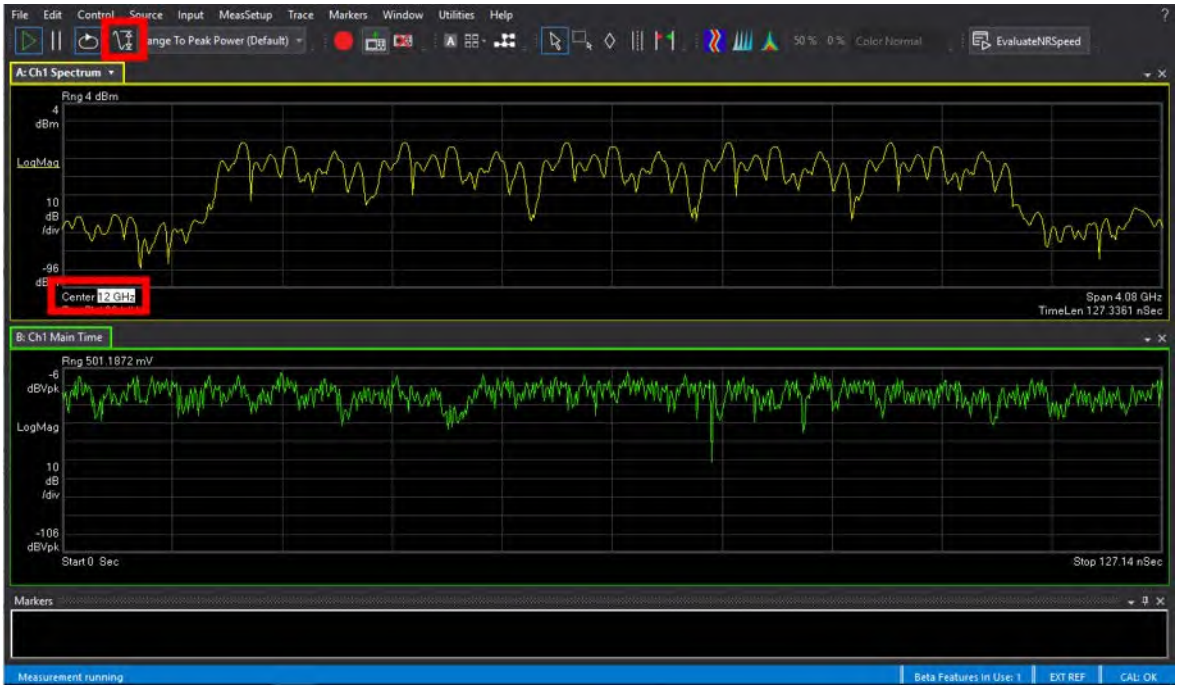
2. Open the VSA software by selecting **Mode Meas** > **Launch VSA**.

NOTE

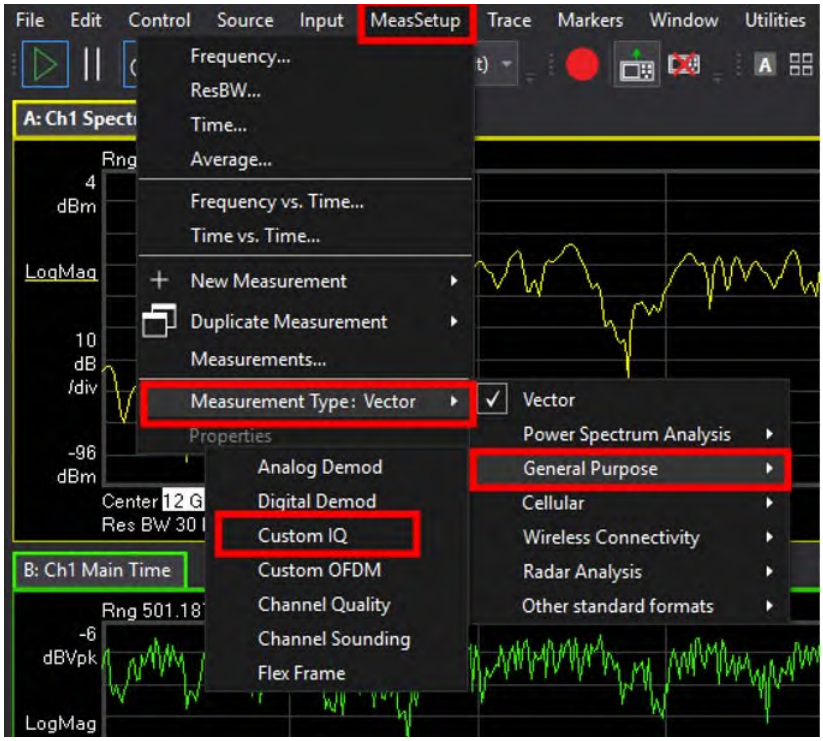
If accessing the instrument via a Remote Desktop connection, select the Screen tab (at the top left of the display) to open the Mode/Measurement/View Selector window.



3. In the Spectrum window, select **Center frequency** and set to **12 GHz**, then select **Auto-range**.

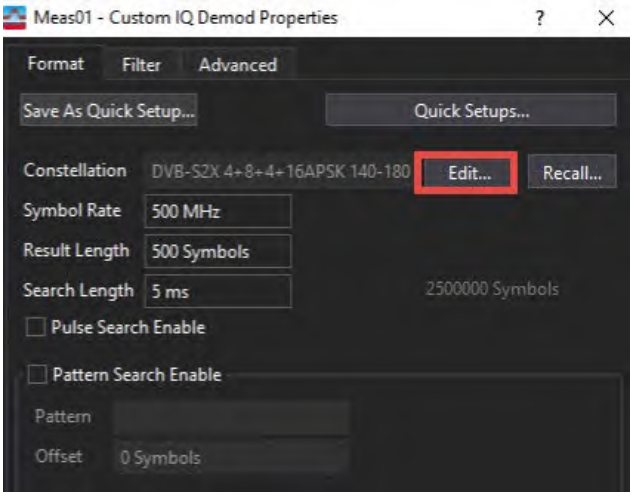


4. From the menu bars, select **MeasSetup** > **Measurement Type: Vector** > **General Purpose** > **Custom IQ**.



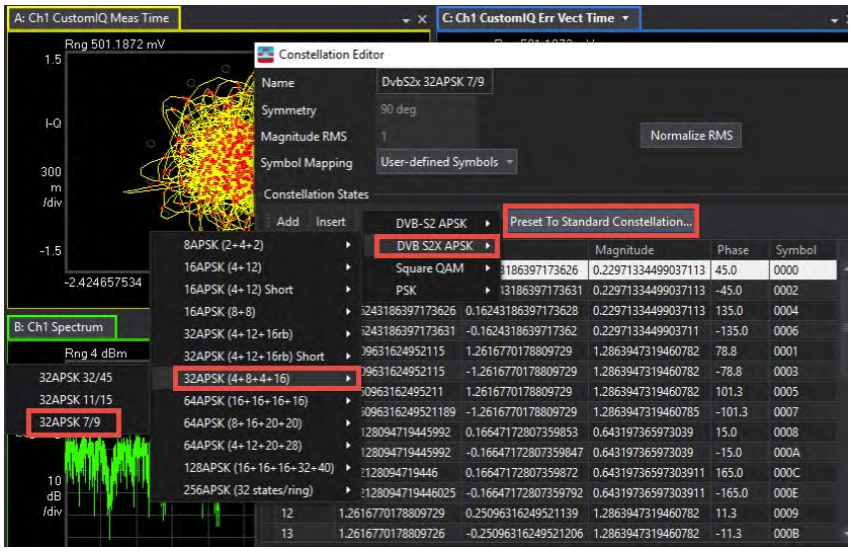
5. Select **MeasSetup** > **Custom IQ Demod Properties**.

   a. In the **Format** tab, select **Constellation Edit**.



   b. Select **Preset to Standard Constellation** > **DVB S2X APSK** > **32APSK (4+8+4+16)** > **32APSK 7/9**. This is equivalent to the 4+8+4+16APSK 140/180 setting we chose in the Custom Modulation setup.

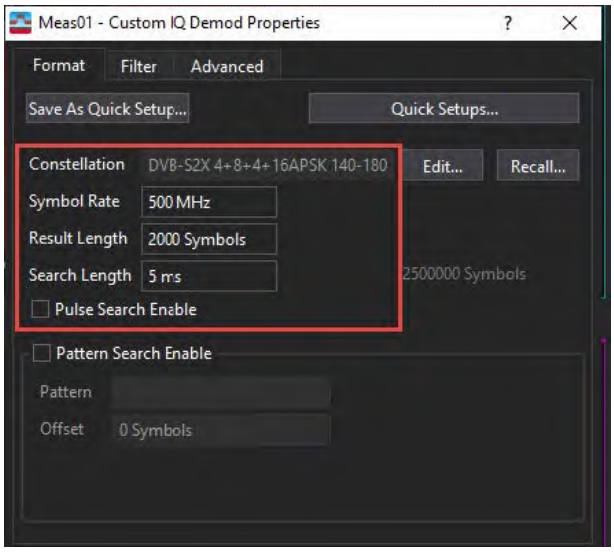
c. Select **OK** to close the Constellation Edit dialog.



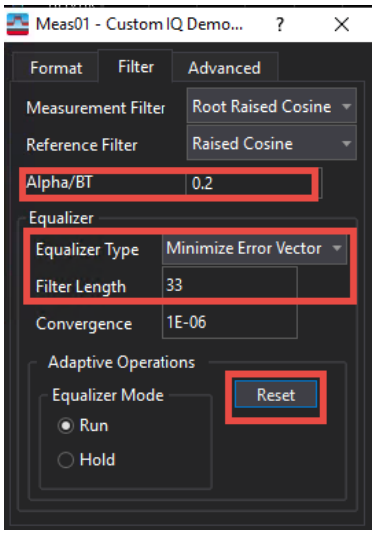
d. Turn off **Pulse Search Enable** (checkbox cleared).

e. Set the Result Length to **2000 Symbols**.

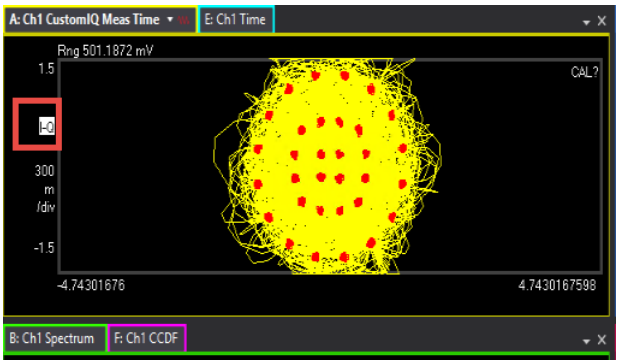
f. Set the Symbol Rate to **500 MHz**. The Result Length and Symbol Rate will adjust the resolution bandwidth to better define the waveform.



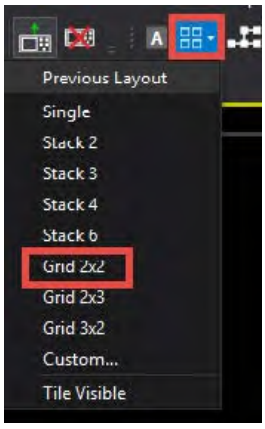
g. Select the **Filter** tab and set Alpha/BT to **0.2**, Filter Length to **33**, and Equalizer Type to **Minimize EVM**. Close the Custom IQ Demod Properties window and notice the center carrier is now being demodulated.



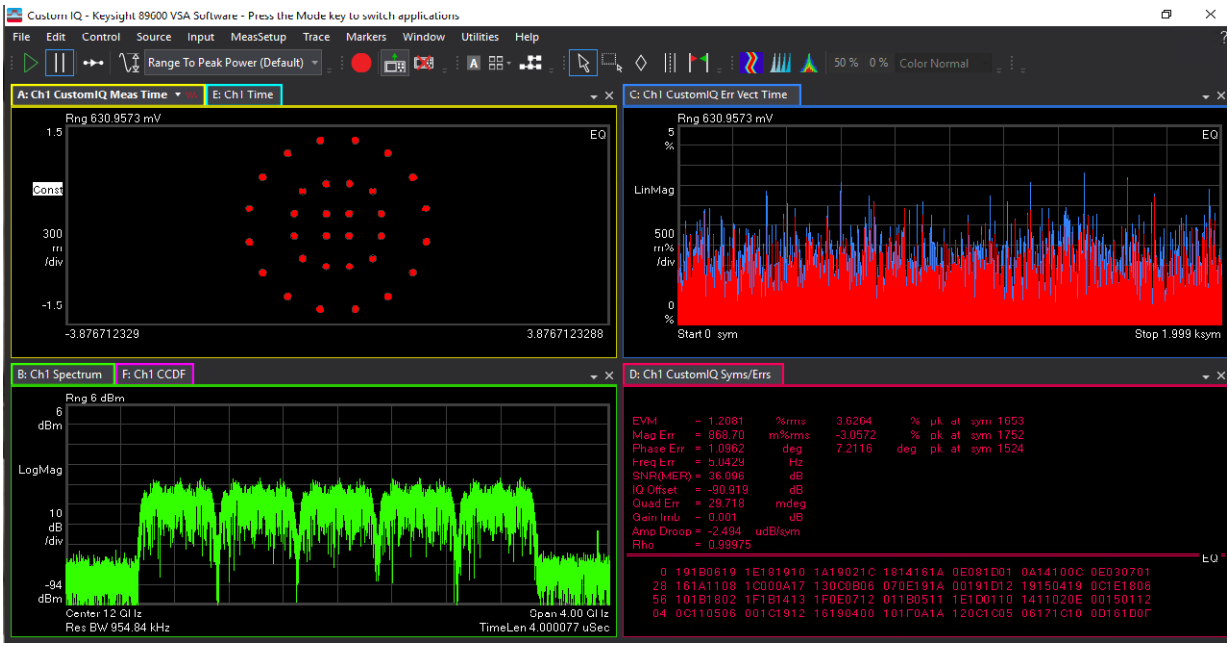
6. In the CustomIQMeas Time window, click on **I-Q** and change to **Constellation**, if desired.



7. Change the window layout to Grid 2x2.



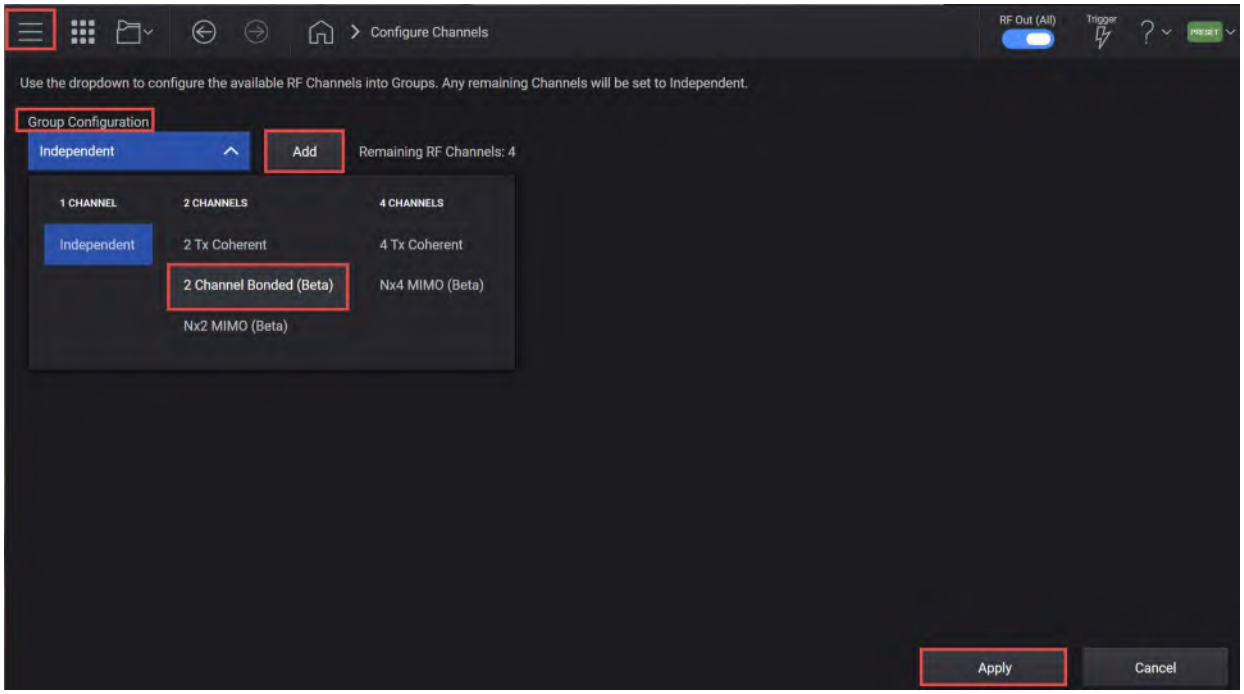
8. Note the measured EVM value in the CustomIQSyms/Errs window. For this example, 1.2%rms.



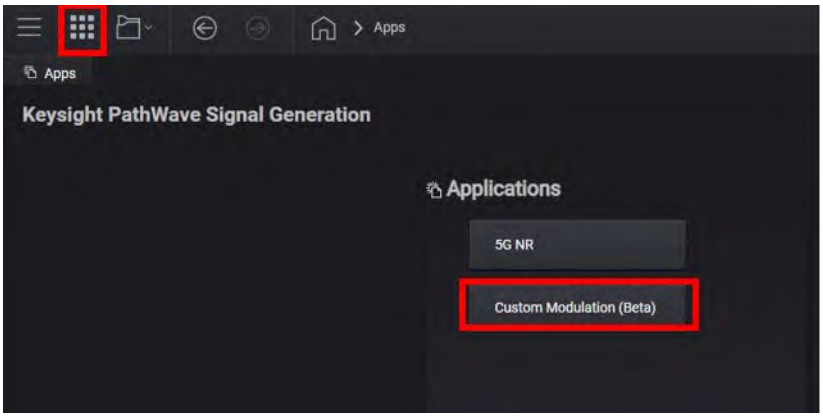
Generating a Wide Band QPSK

On the VXG:

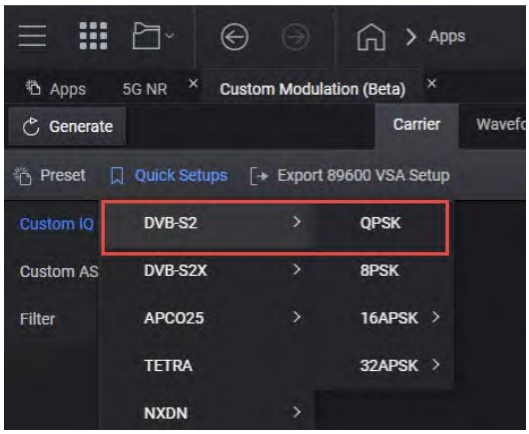
1. Connect the equipment as shown in Figure 3-1 on page 203.
2. Select **Preset** > **Preset** to set the VXG to a known state.
3. Select the System menu (triple bar tab at the top left of the window), and then select **Configure Channels** > **Group 1:2 Channel Bonded** > **Add** (for 4 channel instruments only) > **Apply**.



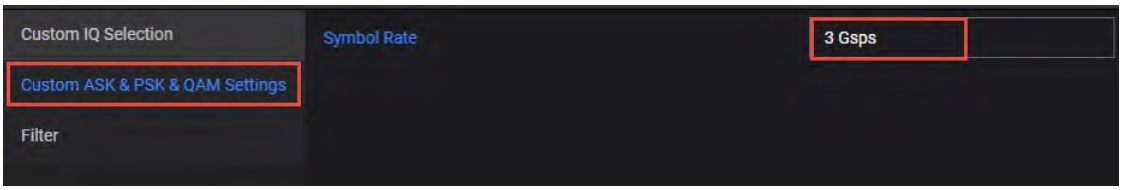
4. Set the Center Frequency to **12 GHz** and Power to **0 dBm**.
5. Select the **Apps** block to open, then select **Custom Modulation**.



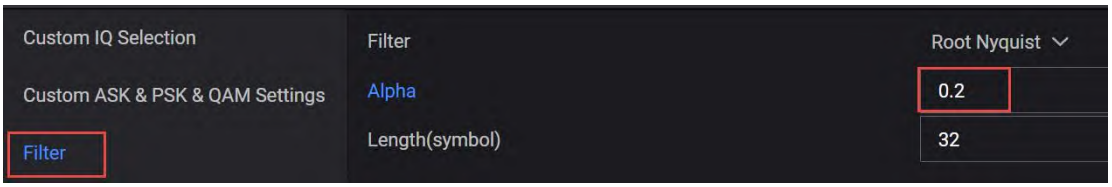
6. In the Custom Modulation setup, select the **Carrier** tab > **Quick Setups** > **DVB-S2** > **QPSK**.



7. From the bottom pane, select **Remove Segment** to remove the Header. This will simplify the demonstration.
8. From the left pane, select **Custom ASK & PSK & QAM Settings** and set the Symbol Rate to **3 Gsps** per channel.

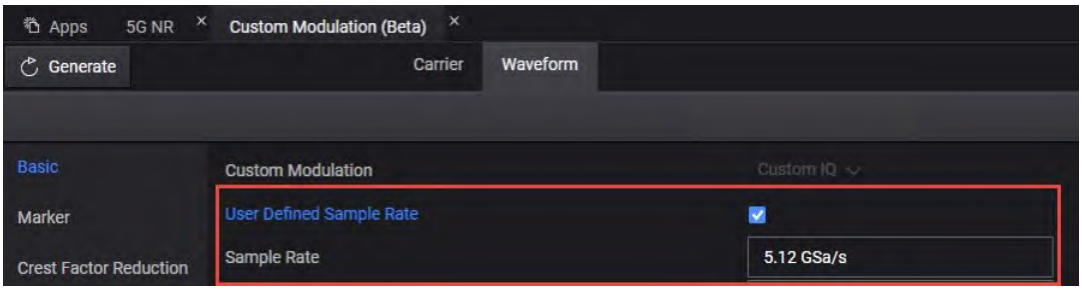


9. Select **Filter** > Alpha to **0.2**.



10. Select the **Waveform** tab > **Basic** > Select **User Defined Sample Rate** and set Sample Rate to **5.12 GSa/s**.

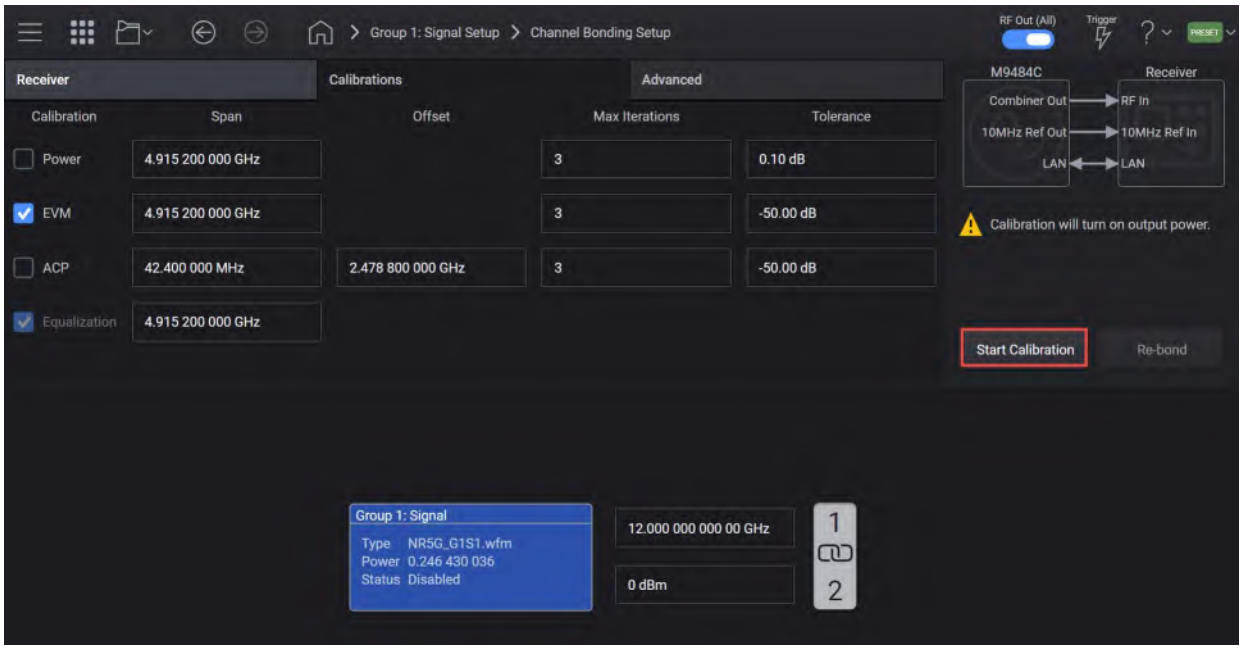
    Each channel is 2.56 GHz for a combined sample rate of 5.12 GHz.



11. **Generate** the waveform and select the **Spectrum** tab from the bottom pane.

12. Select the **Home** icon to return to the main window and then select the **Signal Block** to open.
13. In the Signal Setup dialog box, notice that new custom modulation file is already selected (`CustomMod_G1S1.wfm`).
14. Set the Occupied Bandwidth to **3.6 GHz** (or 1.2 x 3 GHz) and the Sample Rate to **5.12 GHz** Samples per second.
15. Select **Configure Channel Bonding** and configure the Hostname or IP address for the UXA, then **Test Connection**. Ensure that you have successfully connected to the UXA.

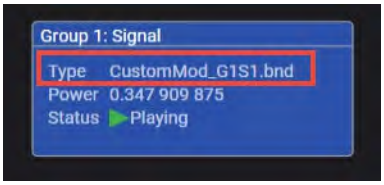
16. Select the **Calibrations** tab and clear the **ACP** calibration.



17. Select **Start Calibration**.

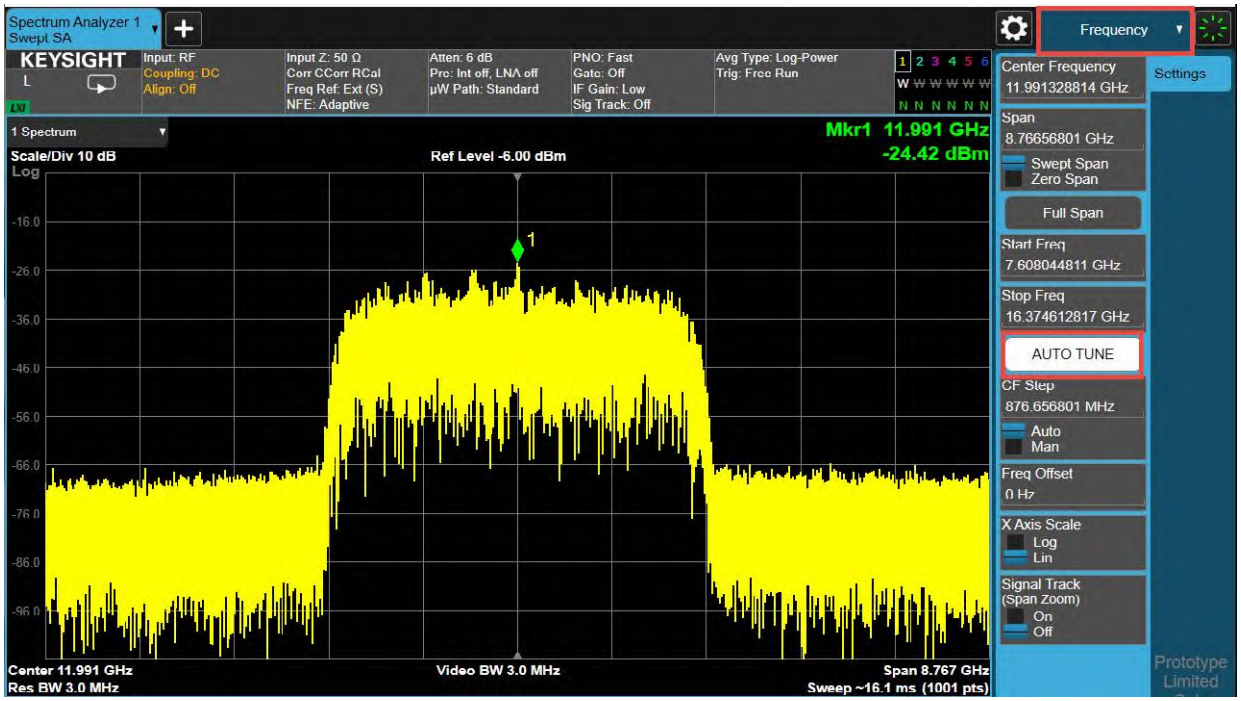
The calibration takes a few minutes to complete. Note that the VXG displays an `Invalid file selection` message. This is because the VXG no longer sees the .wfm waveform instead the newly calibrated waveform has a .bnd file extension.

Notice that the CustomMod_G1S1.bnd waveform is now playing.



On the Signal Analyzer:

1. Select **Frequency** > **Auto Tune** to observe the spectrum.

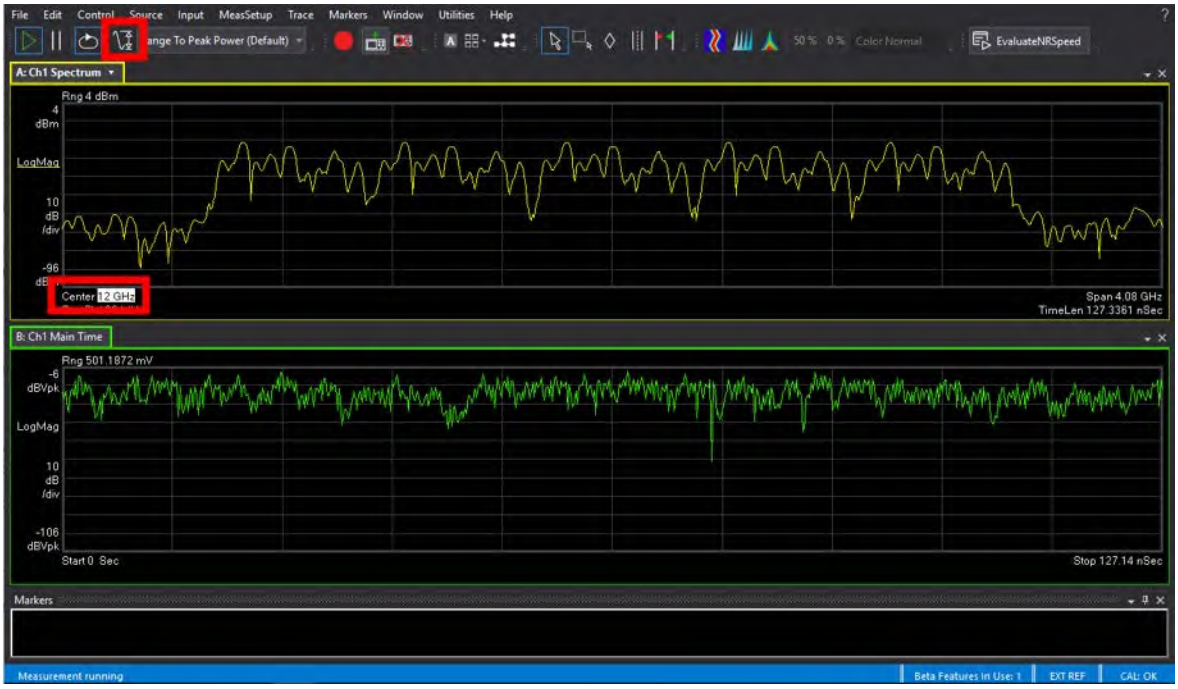


2. Open the VSA software by selecting **Mode Meas** > **Launch VSA**.

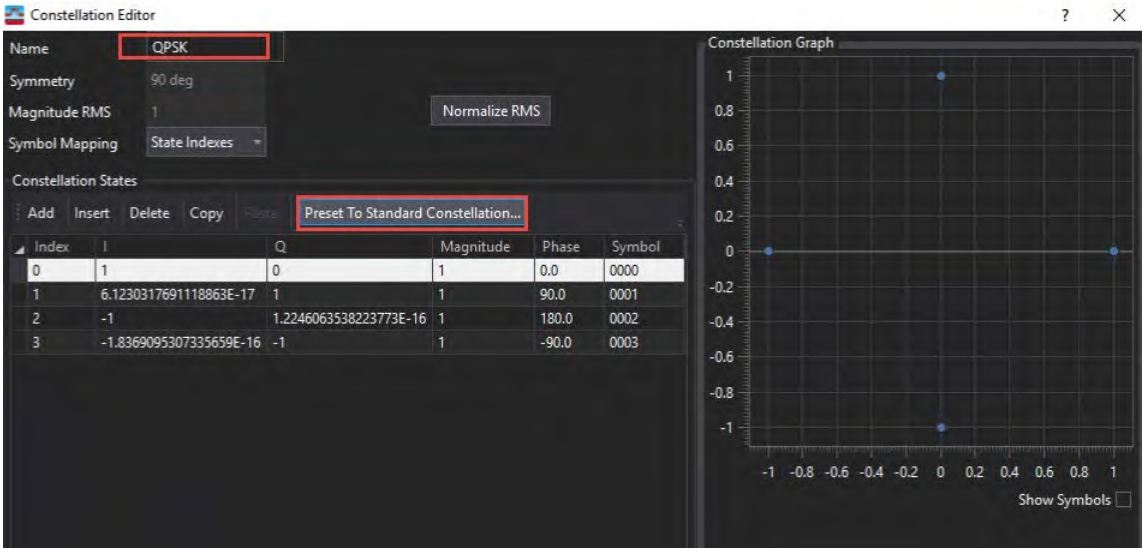
NOTE

If accessing the instrument via a Remote Desktop connection, select the Screen tab (at the top left of the display) to open the Mode/Measurement/View Selector window.

3. In the Spectrum window, set the Center Frequency to **12 GHz** and **Auto-range**.

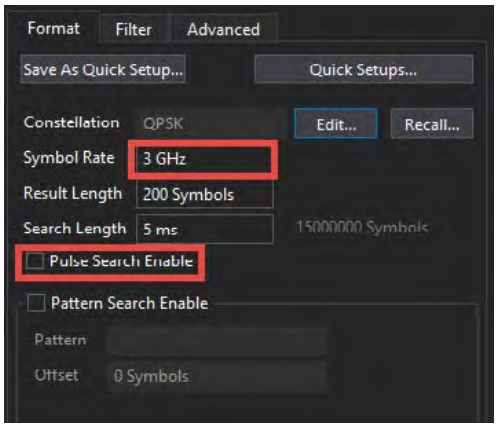


4. From the menu bar, select **MeasSetup** > **Measurement Type** > **General Purpose** > **Custom IQ**.
5. From the menu bar, select **MeasSetup** > **Custom IQ Demod Properties**.
   a. Select the **Format** tab and select Constellation **Edit**.
   b. Select **Preset to Standard Constellation** > **PSK** > **QPSK**.
   c. Select **OK** to close the Constellation Edit dialog.

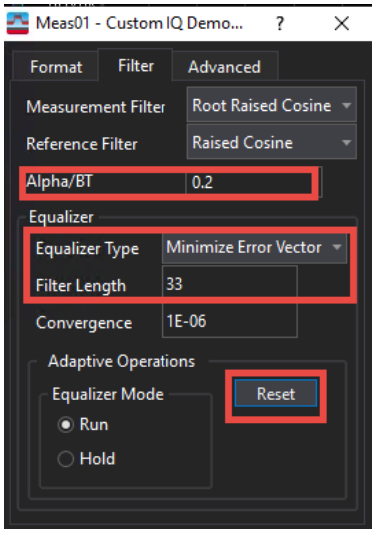


d. Turn off **Pulse Search Enable** (checkbox cleared).

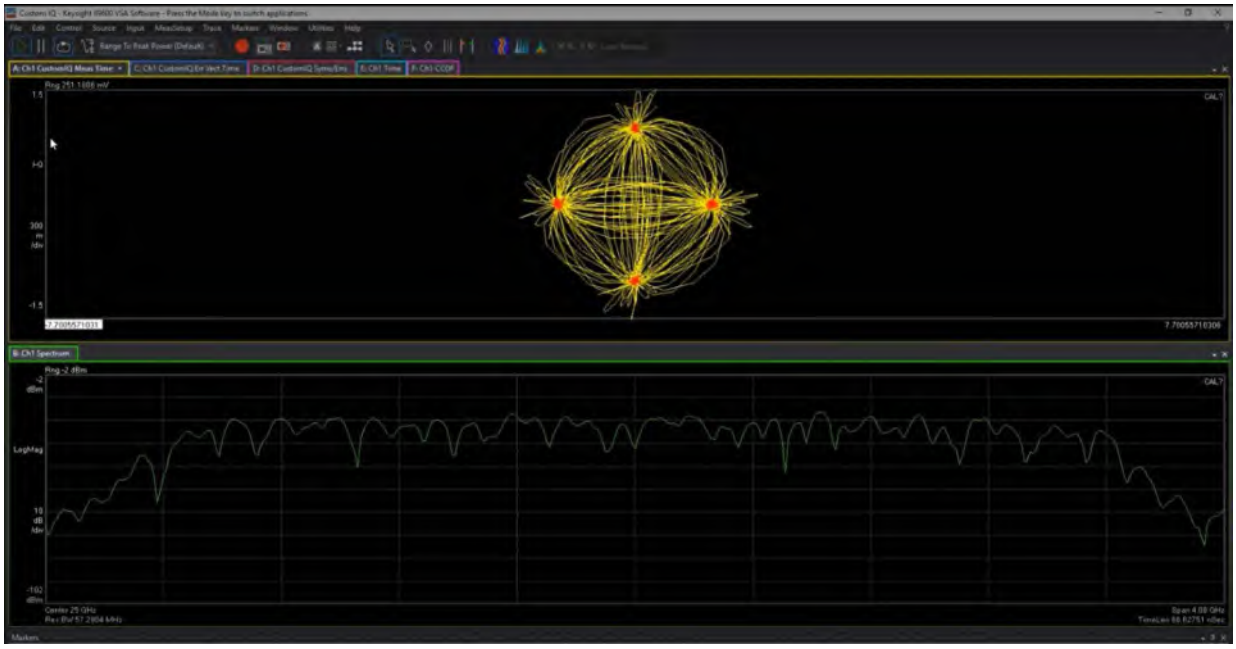
e. Set the Symbol Rate to **3 GHz**.



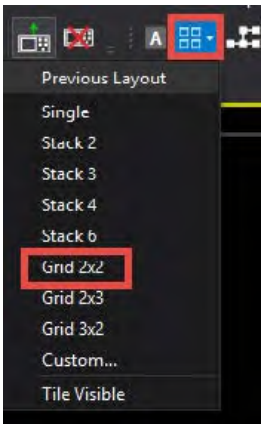
f. Select the **Filter** tab and set Alpha/BT to **0.2**, Filter Length to **33**, Equalizer Type to **Minimize Error Vector**, and **Reset** the Filter.



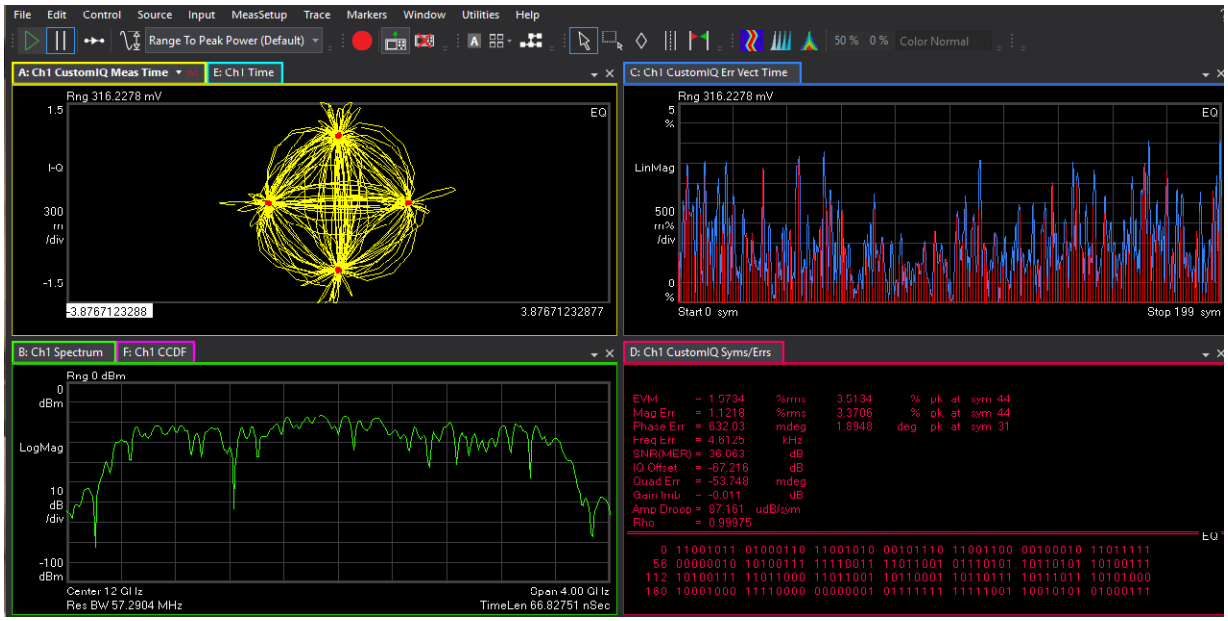
6. View the results.



7. In the menu bar, select the **Trace Layout** icon and select the **Grid 2 x 2** format.



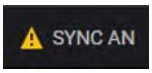
This allows you to view more measurement details like the EVM results, which for this example is ~1.6% rms.


File Edit Control Source Input MeasSetup Trace Markers Window Utilities Help
Range To Peak Power (Default)
A: Ch1 CustomIQ Meas Time
E: Ch1 Time
Rng 316.2278 mV
EQ
1.5
I-Q
300
/div
-1.5
-3.8767123288
3.8767232877
C: Ch1 CustomIQ Err Vect Time
Rng 316.2278 mV
EQ
5
%
LinMag
500
/div
0
%
Start 0 sym
Stop 199 sym
B: Ch1 Spectrum
F: Ch1 CCDF
Rng 0 dBm
0
dBm
LogMag
10
dB
/div
-100
dBm
Center 12 GHz
Res BW 57.2904 MHz
Span 4.00 GHz
TimeLen 66.82751 nSec
D: Ch1 CustomIQ Syms/Errs
EVM = 1.5734 %rms 3.5134 % pk at sym 44
Mag Err = 1.1218 %rms 3.3706 % pk at sym 44
Phase Err = 632.03 mdeg 1.8948 deg pk at sym 31
Freq Err = 4.6125 kHz
SNR(MER) = 36.063 dB
IQ Offset = -67.216 dB
Quad Err = -53.748 mdeg
Gain Imb = -0.011 dB
Amp Droop = 87.161 udB/sym
Rho = 0.99975
EQ
0 11001011 01000110 11001010 00101110 11001100 00100010 11011111
56 00000010 10100111 11110011 11011001 01110101 10110101 10100111
112 10100111 11011000 11011001 10110001 10110111 10111011 10101000
160 10001000 11110000 00000001 01111111 11111001 10010101 01000111

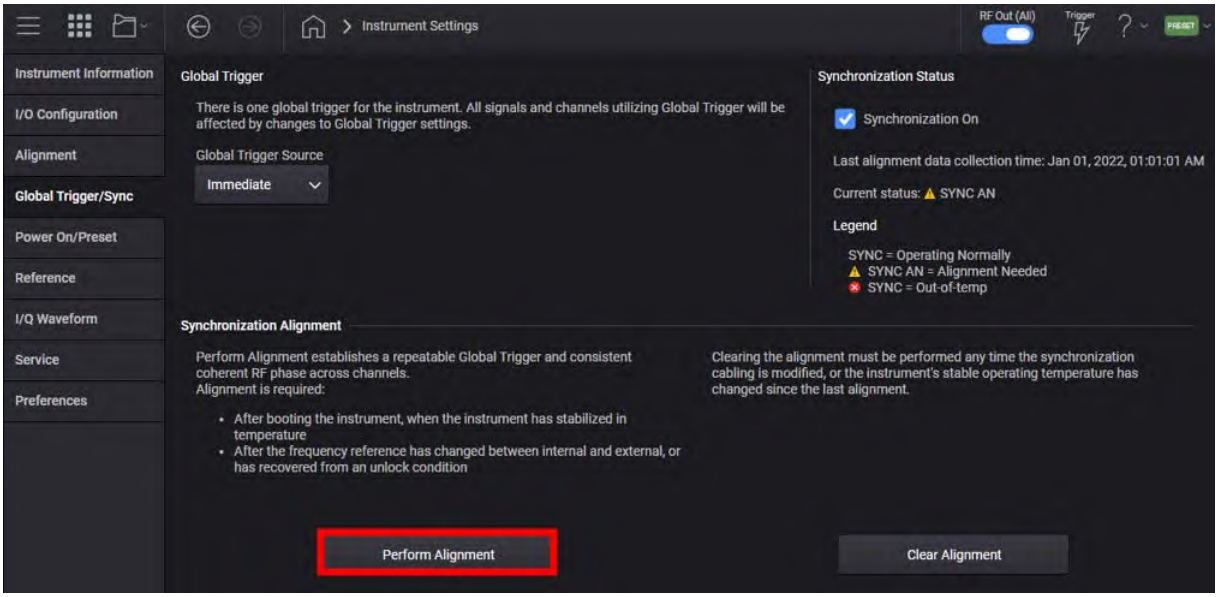
Generating a 5G NR Waveform

On the VXG:

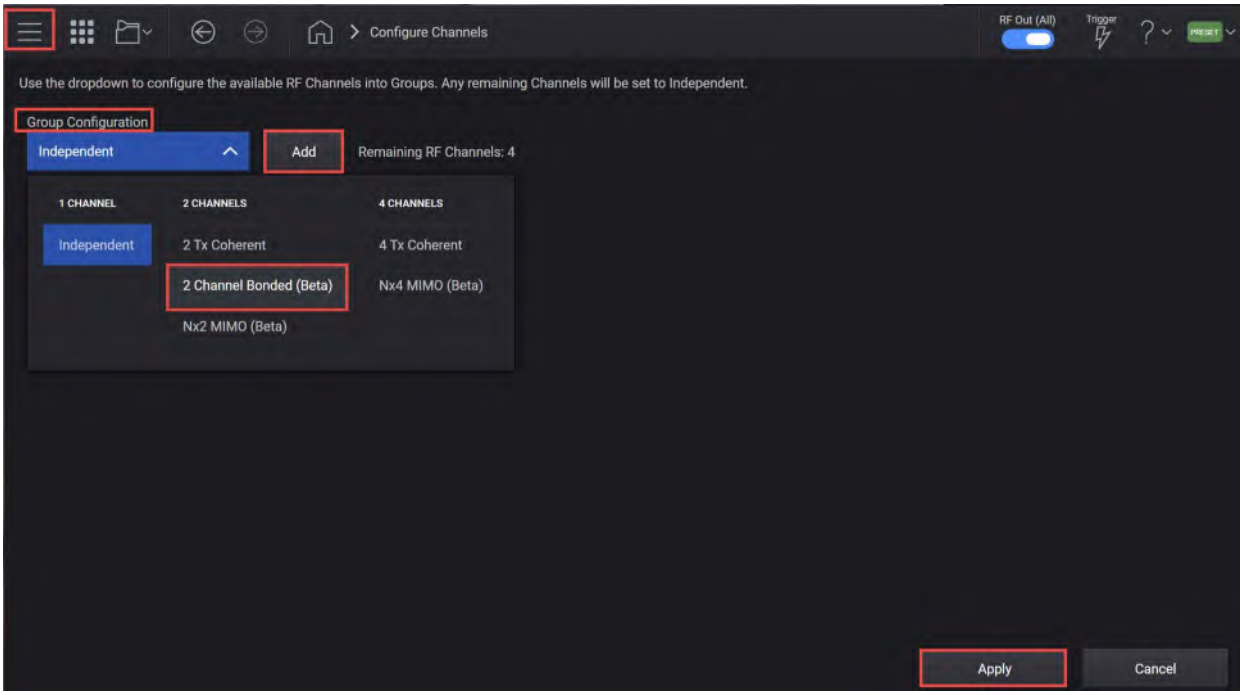
1. Connect the equipment as shown in Figure 3-1 on page 203.
2. Select **Preset** > **Preset** to set the VXG to a known state.

   If a Synchronization Alignment is required, indicated by 

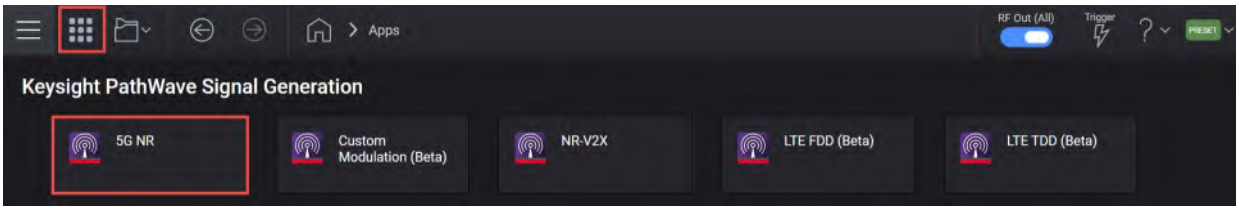
in the bottom left corner, tap or click the warning message and choose **Perform Alignment** before proceeding.



3. Select the System menu (triple bar tab at the top left of the window), and then select **Configure Channels** > **Group 1:2 Channel Bonded** > **Add** (for 4 channel instruments only) > **Apply** > **Apply**.

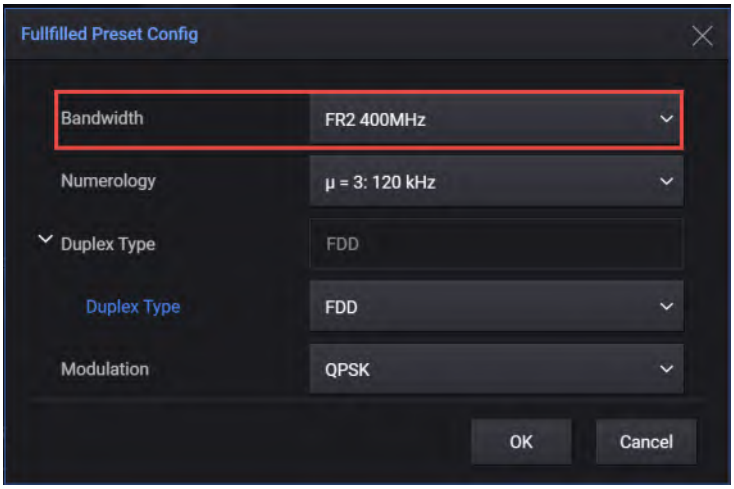


4. Set the Center Frequency to **12 GHz** and Power to **0 dBm**.
5. Select the **Apps** block to open, then select **5G NR**.



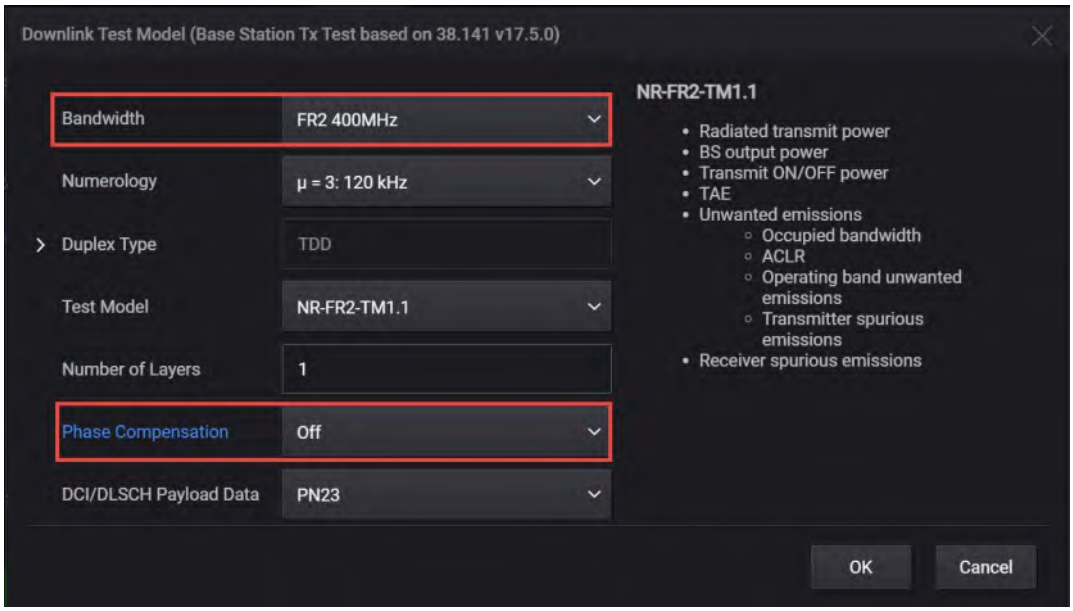
6. Select the **Carrier** tab.

7. Select **Full-Filled Config** and set Bandwidth to **FR2 400 MHz**, and then select **OK**.

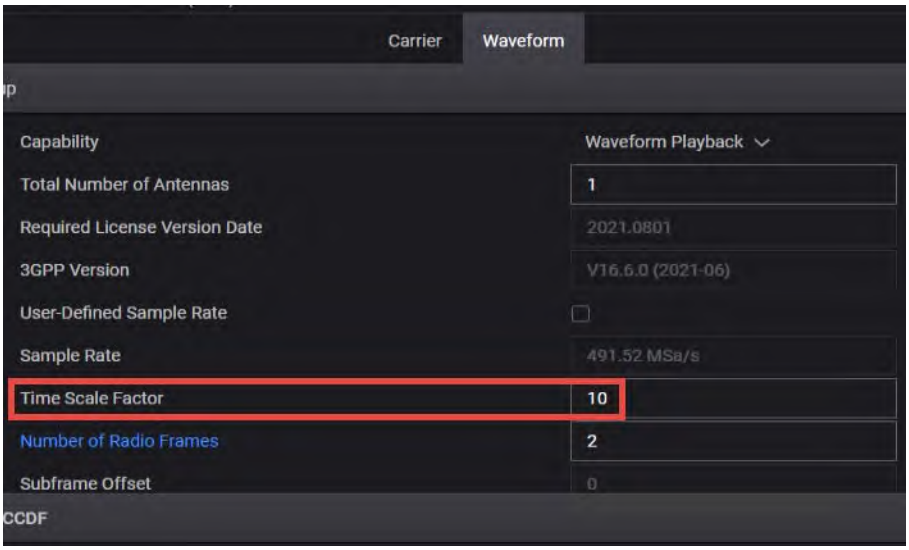


Full Filled Configurations is a great place to start creating your waveform. You can select FR, bandwidth, and modulation format with a fully allocated PDSCH data channel. This would represent a spectrally correct signal that is typically used in power amplifier measurements. You can modify the RB allocations and Slot allocations if fully allocated data channels are not desired.

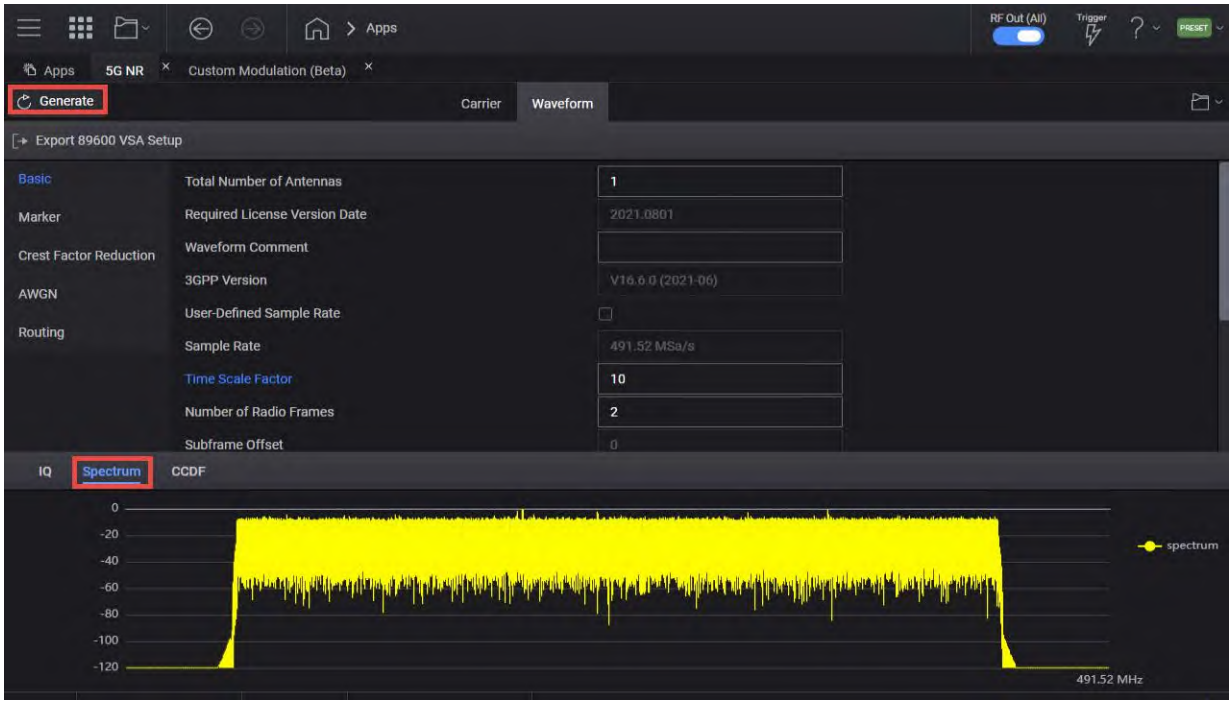
8. Select **DL Test Model** to open. set Bandwidth to **FR2 400 MHz** and set Phase Compensation to **Off**, and then select **OK**.



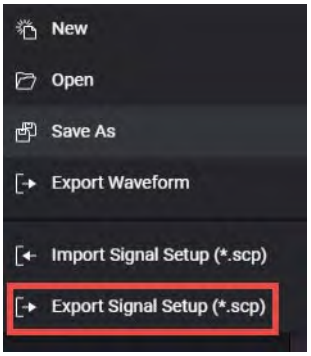
9. Select the **Waveform** Tab, and set the Time Scale Factor to **10**.



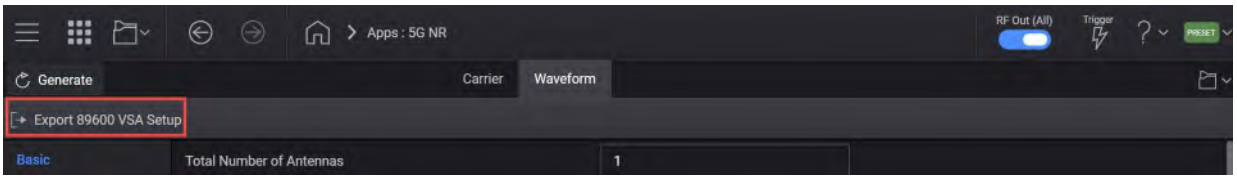
10. **Generate** the waveform and select the **Spectrum** tab from the bottom pane. The waveform will take a moment to finish generating.



11. Select the folder icon in the right corner of the display, and select **Export Signal Setup (*.scp)** file. In the File Name field, create a name for the setup file, then select **Enter**. Note the location where you saved this file as you will need to copy it to the UXA.



12. Select **Export 89600 VSA Setup**. In the File Name field, create a name for the setup file, then select **Enter**. Note the location where you saved this file as you will need to copy it to the UXA.



To copy the two setup files to the UXA:

   a. Select the **System** menu icon (triple bar), then **Switch to Windows**.

   b. Select the **Windows** icon (bottom left corner), open **File Explorer** and navigate to the setup files and **Copy** both files.

   c. On the UXA, select the **Windows** icon, open **File Explorer** and **Save** the two setup files.

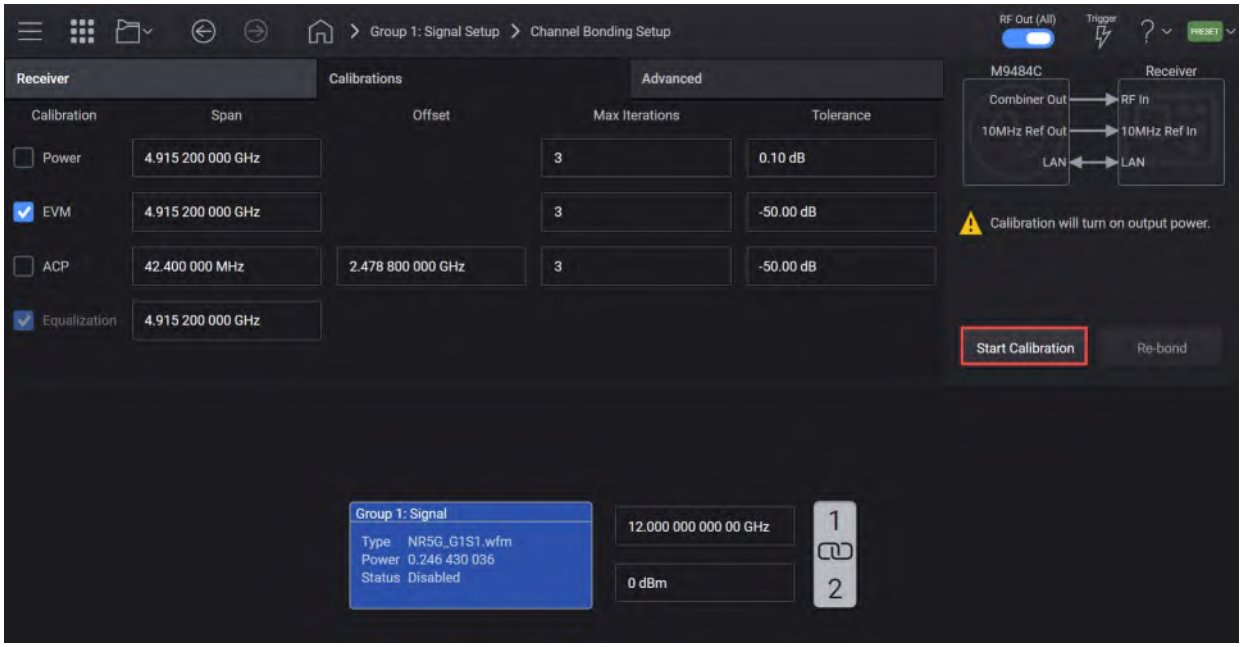
   d. On the VXG, select the PathWave SG icon in the Desktop to restore the PathWave application.

13. Select the **Home** icon to return to the main window and then select the **Signal Block** to open.

14. In the Signal Setup dialog box, notice that new custom modulation file is already selected (`NR5G_G1S1.wfm`).

15. Select **Configure Channel Bonding** and configure the Hostname or IP address for the UXA, then **Test Connection**. Ensure that you have successfully connected to the UXA.

16. Select the **Calibrations** tab and clear the **ACP** calibration, then select **Start Calibration**.



On the Signal Analyzer:

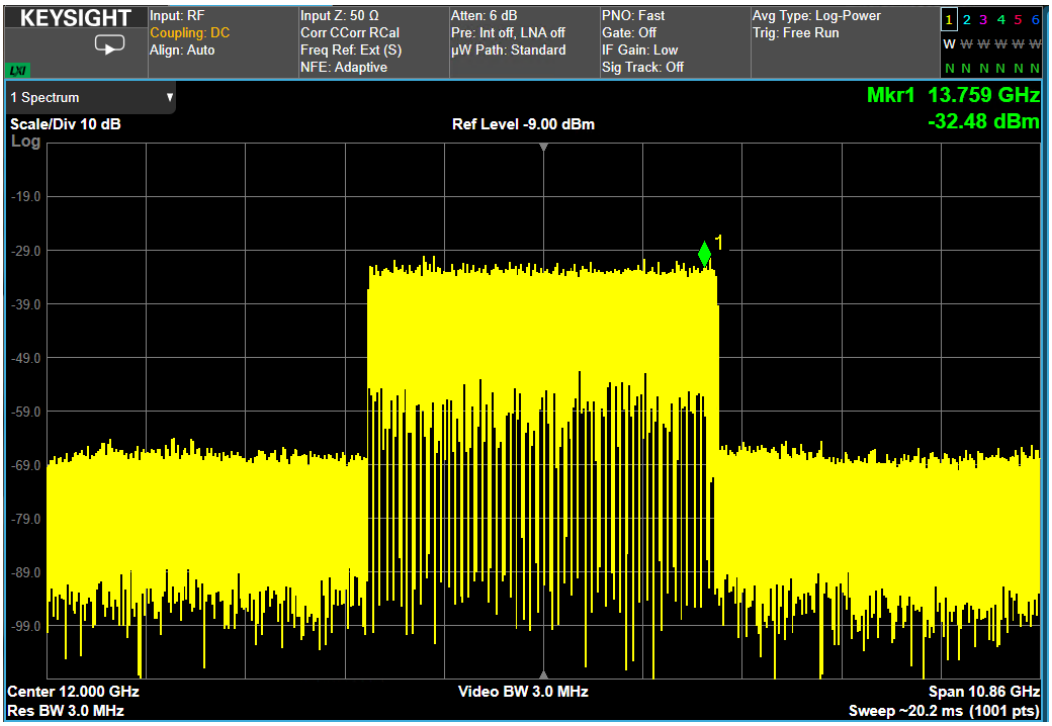
Below are two methods for viewing the results of the 5G NR wideband waveform.

– Using X-Apps 5G NR & V2X Modulation mode Modulation Analysis measurement

– Using 89600 VSA

Using X-Apps 5G NR & V2X 1 Mode

1. Select **Frequency** > **Auto Tune** to observe the channel bonded signal.



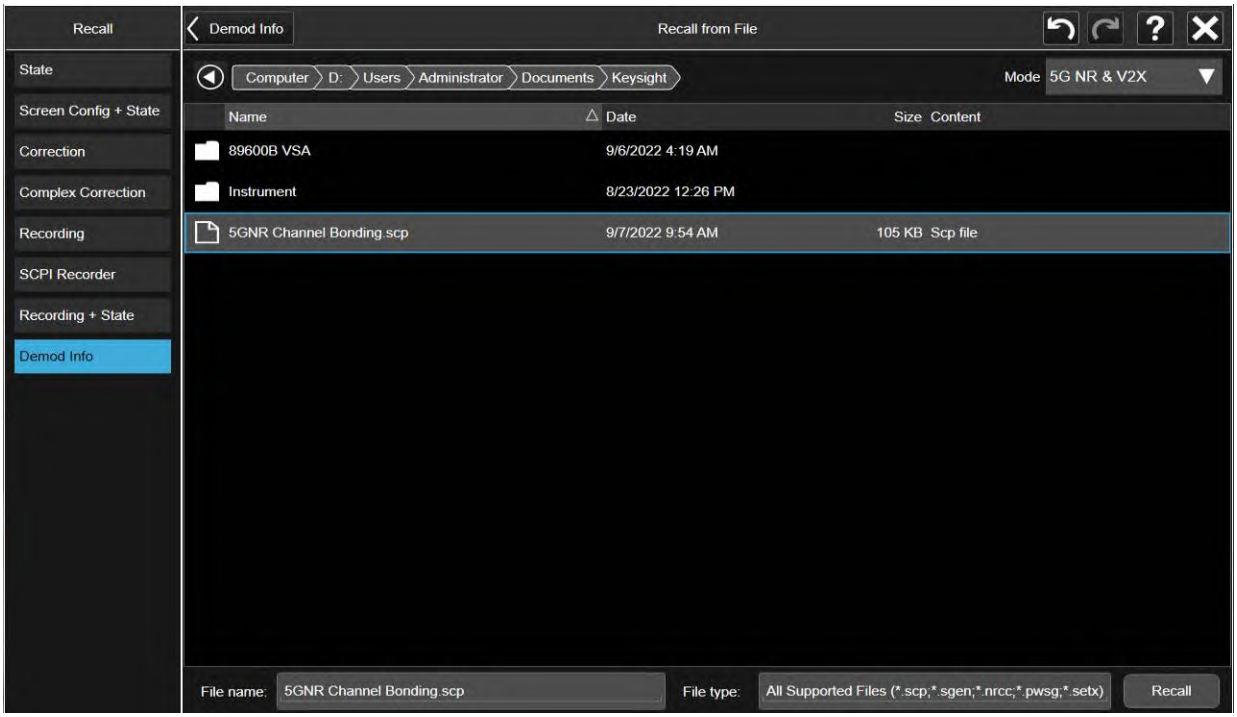
2. From the N9042B select **Mode/Meas** > **5G NR &V2X** Mode > **OK**.

NOTE

If accessing the instrument via a Remote Desktop connection, select the Screen tab (at the top left of the display) to open the Mode/Measurement/View Selector window.

3. Select **Mode Preset** to set Spectrum Analyzer 5G NR mode to a known state.
4. From the N9042B Menu Panel (or the Screen tab), select **Mode/Meas** > **5G NR & V2X** Mode > **Modulation Analysis** Measurement > **OK**.
5. Select **Recall** (If accessing the N9042B remotely, select the Folder icon at the bottom of the display) **Demod Info** > Data Type set to **CC Setup** > **Recall From**.

6. Navigate to the .scp file you created in step 11 above and select **Recall**.



7. From the Menu Panel, select **Meas Setup** > **Settings** tab > **Optimize EVM**. For this example, the EVM RMS is ~1.9%.

Using 89600 VSA

1. Open the VSA software by selecting **Mode Meas** > **Launch VSA**.

NOTE

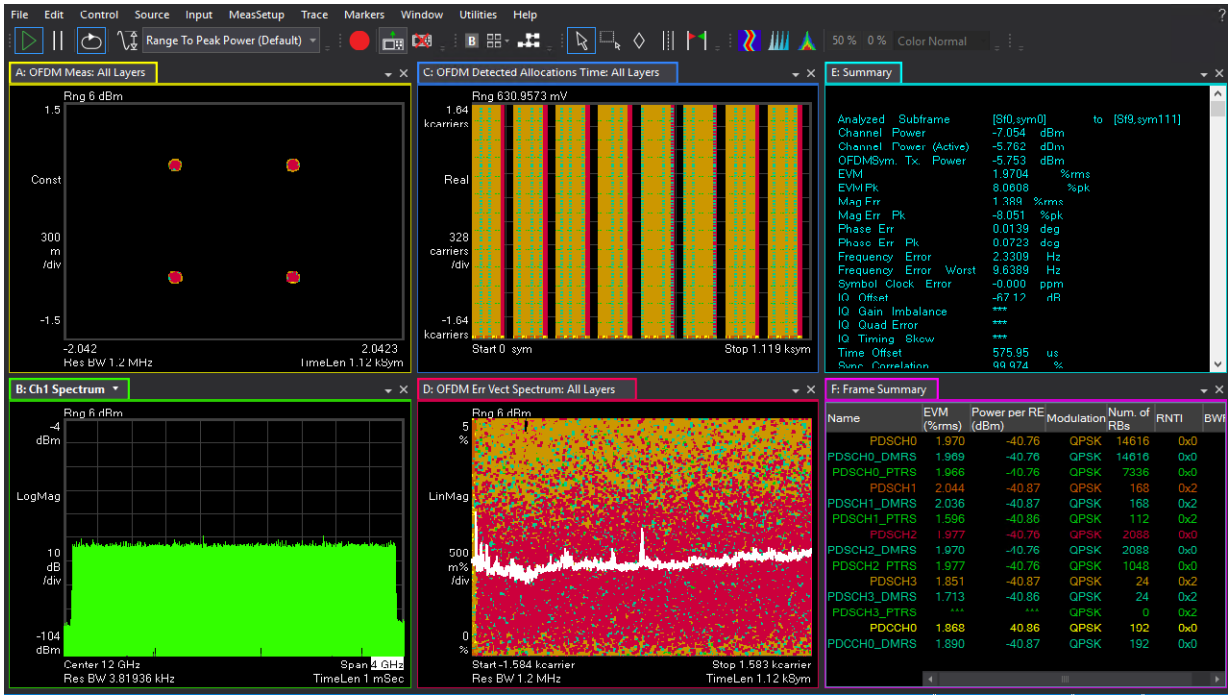
If accessing the instrument via a Remote Desktop connection, select the Screen tab (at the top left of the display) to open the Mode/Measurement/View Selector window.

2. In the Spectrum window, set the Center Frequency to **12 GHz** and **Auto-range**.

3. From the menu bar, select **File** > **Recall Setup** > **Recall** and select the .setx file you copied over from the VXG.

4. From the menu bar, select **MeasSetup** > **5G NR Demod Properties** > **Time** tab and set Time Scale Factor to **10**. Close the Demod Properties windows and observe the results.

5. In the Spectrum window, set the Span to **4 GHz**, then select the Auto-range icon at the top of the display.



In this example, the EVM is ~2%

Setting Up 8 Virtual Signal Generators

The M9484C Option 8SG provides eight virtual signal generators (multiple IQ paths to RF) allowing you to emulate up to 8 signals simultaneously with one channel. This feature is most beneficial in receiver design and test applications in which the current implementation often involves RF combining of multiple signal generators due to dynamic range limitations. 8SG can simplify device characterization where multi-carrier or multi-radio access technology signals are required.

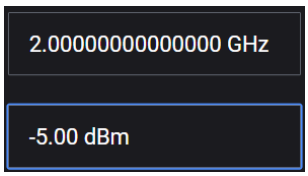
NOTE **The VXG must have Option M9484C-8SG (8 virtual signal generators) installed.**

For this example we will use the following equipment:

- M9484C VXG
  - RF bandwidth of 2.5 GHz per channel (M9484C-R25)
  - 8 Virtual Signal Generators (M9484C-8SG)
  - N7631APPC PathWave Signal Generation for 5G NR measurement application
- X-Series Signal Analyzer
  - N9085EMOE 5G NR Measurement application

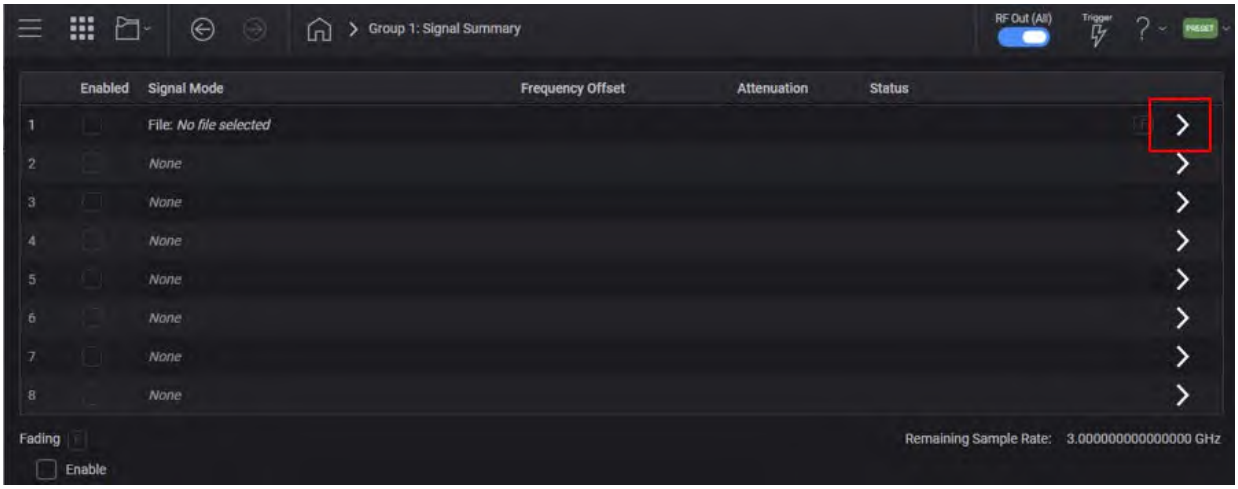
On the VXG:

1. Select **Preset** > **Preset** to set the instrument to a known state.
2. In the Output area for Channel 1, set Frequency to **2 GHz** and Power to **-5 dBm**.

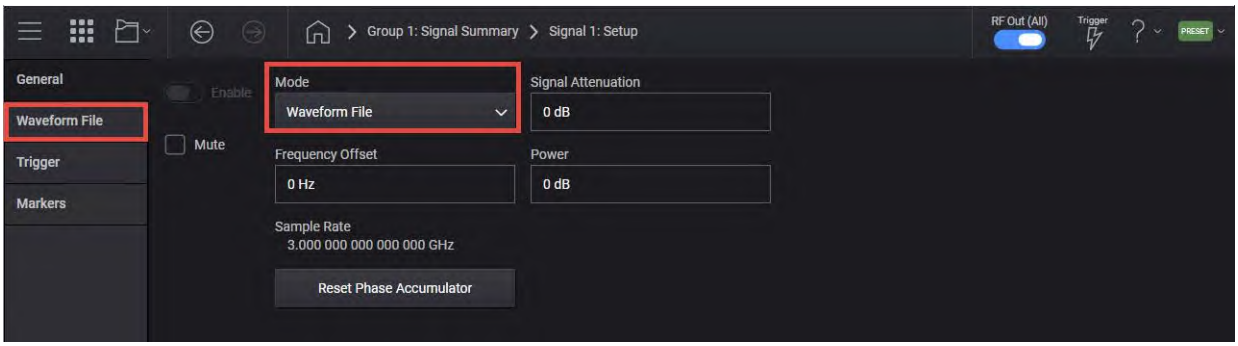


3. Select **Group 1 Signal** block to open.

4. Select the arrow for Signal 1 to open the Signal Setup window.



5. Select the **Mode** drop down menu and select **Waveform File** and then select the **Waveform File** tab on the left side.

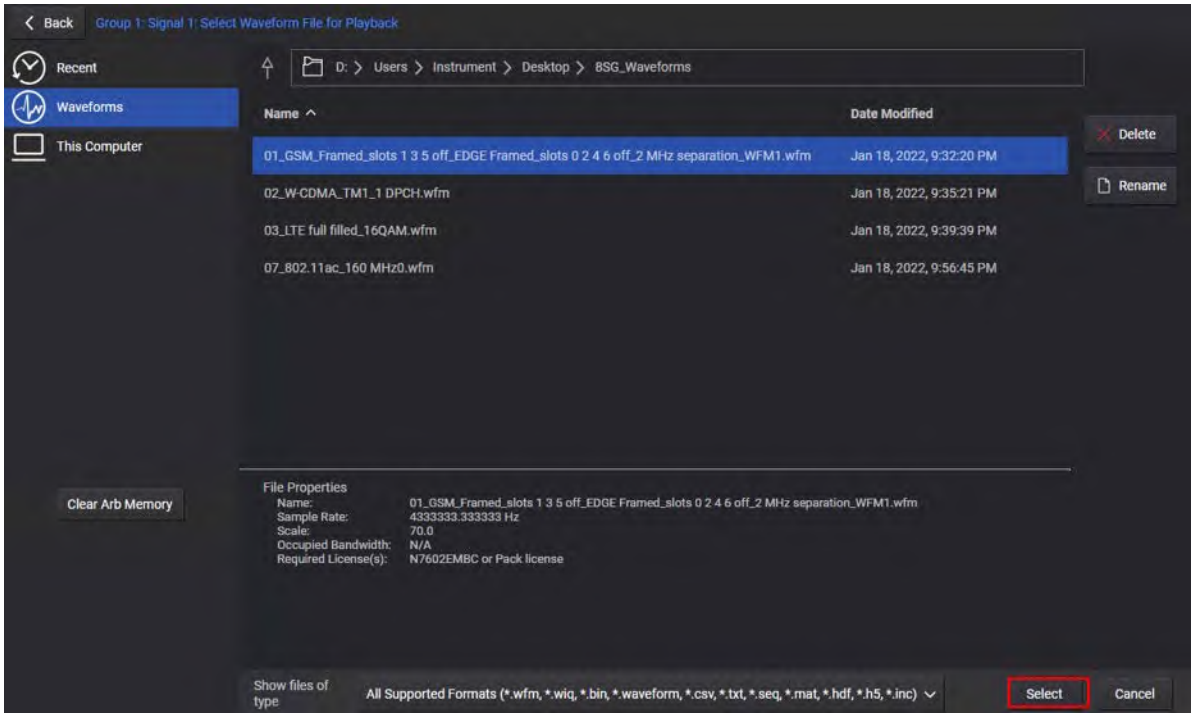


6. In the Waveform Playback Setup area, use File **Select** to navigate to:

```
D:\Users\Instrument\Documents\Keysight\PathWave
\SignalGenerator\Examples
```

```
and choose
01_GSM_Framed_slots 1 3 5 off_EDGE Framed_slots 0 2 4 6 off_2MHz
separation_WFM1.wfm
```

then **Select**.



NOTE

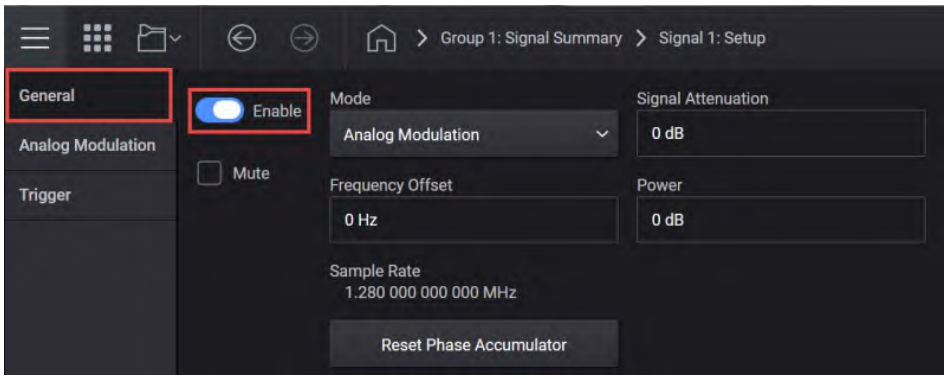
Whenever you enable a particular waveform file that is not currently enabled in another IQ path, it uploads prior to playback. If the file is already enabled in one or more other IQ paths, an upload does not occur and the waveform plays directly.

If you update the IQ data for a particular waveform file, you must first disable the IQ paths where the old version of the same file is being used so that it uploads, replacing the old data.

7. Select the **General** tab and set the Frequency Offset to the desired value. For this example **–1.2 GHz**.

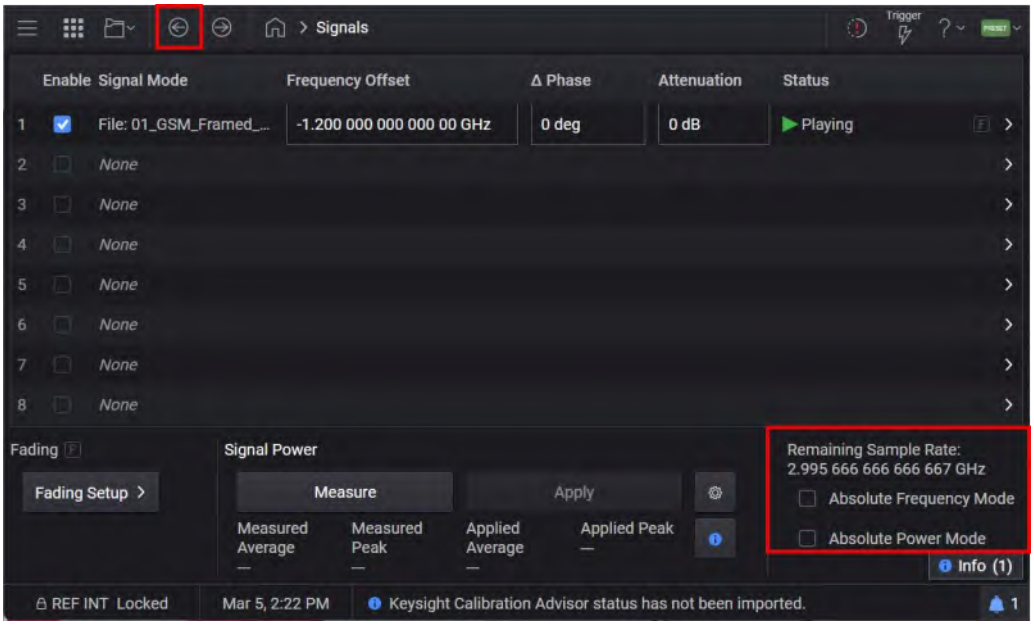
   The Frequency Offset is the distance from the channel's RF center frequency (set in step 2) to the center frequency of the signal. You can adjust the signal attenuation for each signal to allow scaling the signal down without disrupting the output.

8. Select **Enable**.



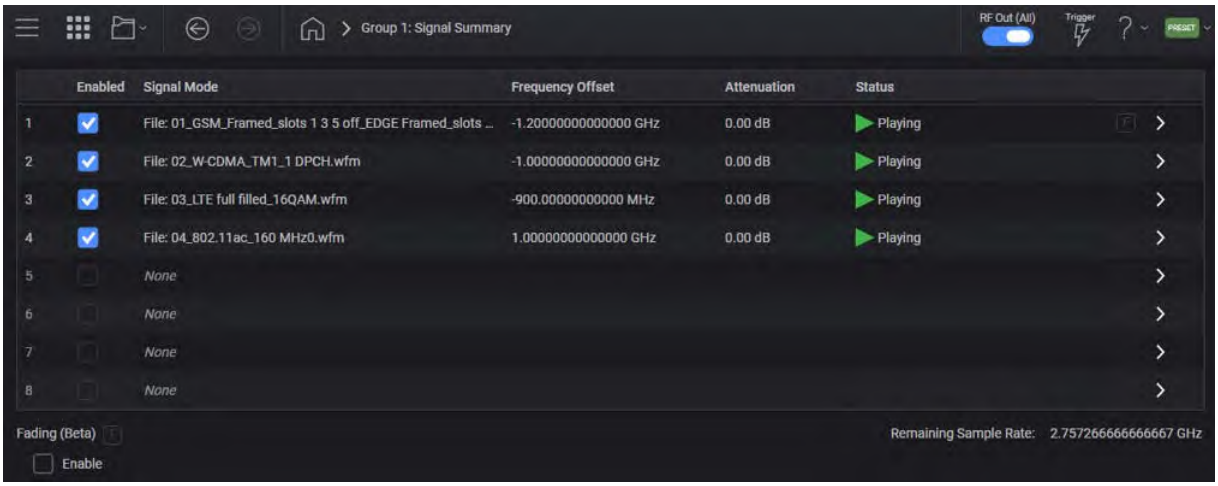
9. Select the Back arrow and view the **Group 1: Signal Summary**.

Notice the Remaining Sample Rate has decreased from 3.0 GHz to 2.9956 GHz now that Signal 1 is consuming 4.333 MHz.

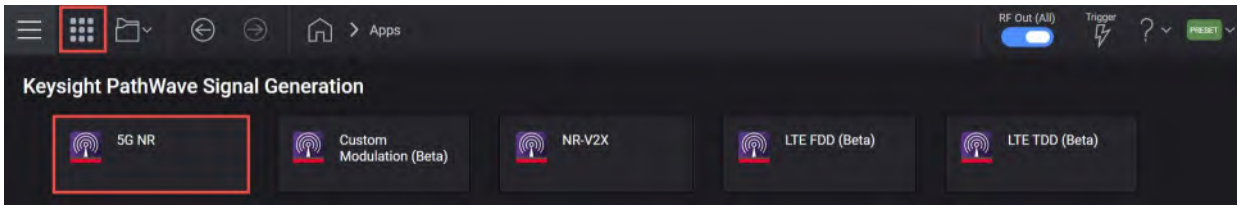


10. Repeat step step 5 through step 9 for Signal 2, Signal 3, and Signal 4 with the following changes.
    - Signal 2
        - Waveform: `D:\Users\Instrument\Documents\Keysight\PathWave\SignalGenerator\Examples\02_W-CDMA_TM1_1 DPCH.wfm`
        - Frequency Offset: **–1.0 GHz**
    - Signal 3
        - Waveform: `D:\Users\Instrument\Documents\Keysight\PathWave\SignalGenerator\Examples\03_LTE full filled_16QAM.wfm`
        - Frequency Offset: **–900 MHz**
    - Signal 4
        - Waveform: `D:\Users\Instrument\Documents\Keysight\PathWave\SignalGenerator\Examples\04_802.11ac_160 MHz0.wfm`

– Frequency Offset: **1.0 GHz**



11. Signals 5 -7 will be configured using the embedded PathWave Signal Generation for 5G NR (N7631APPC). Select the **Radio Apps** icon in the top left of the window, and then select **5G NR/5G-Advanced**.

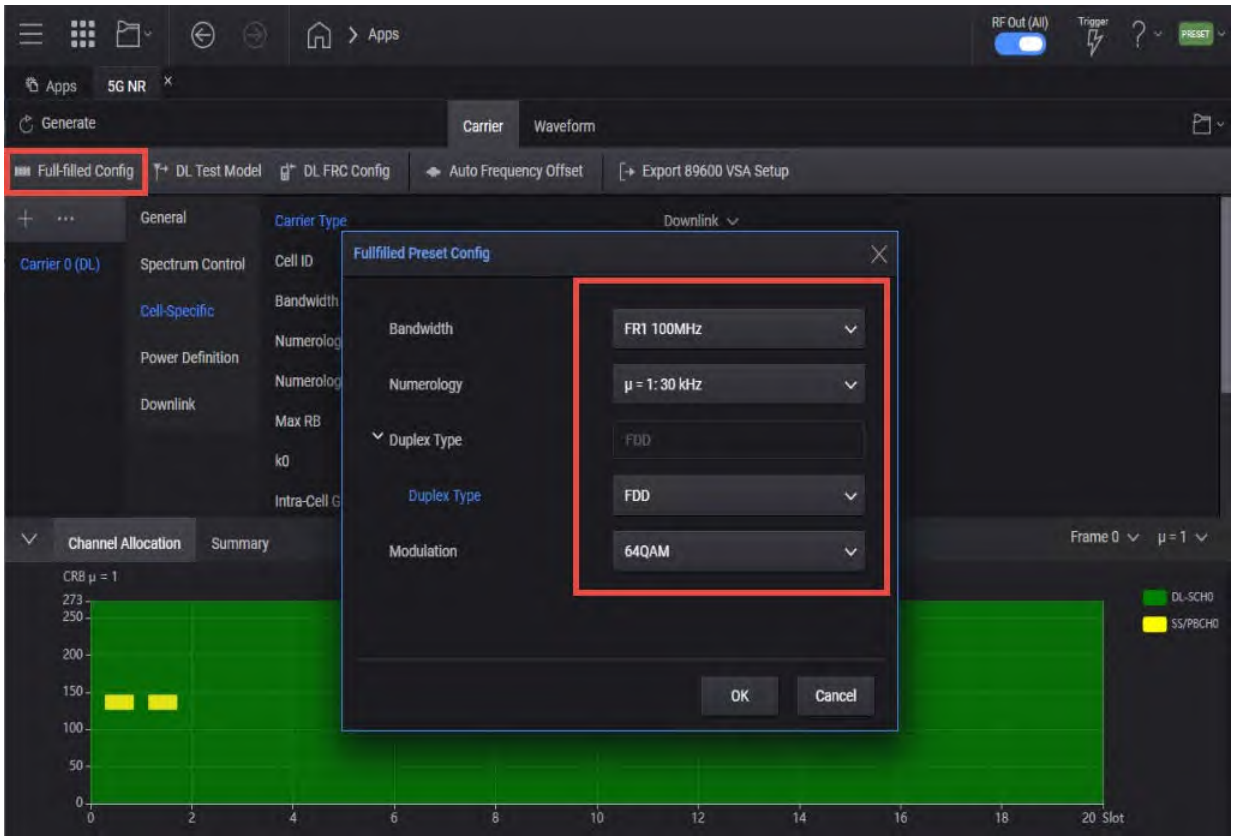


12. To create signal 5, select the **Carrier** tab > **Full-filled** Config and set

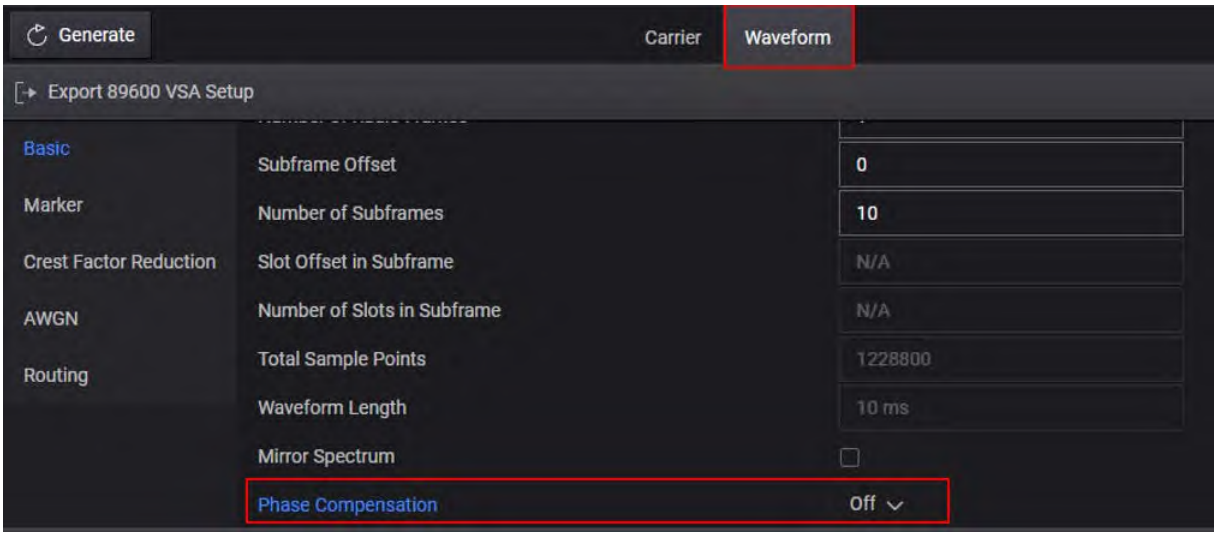
– Bandwidth to **FR1 100 MHz**

– Duplex Type to **FDD**

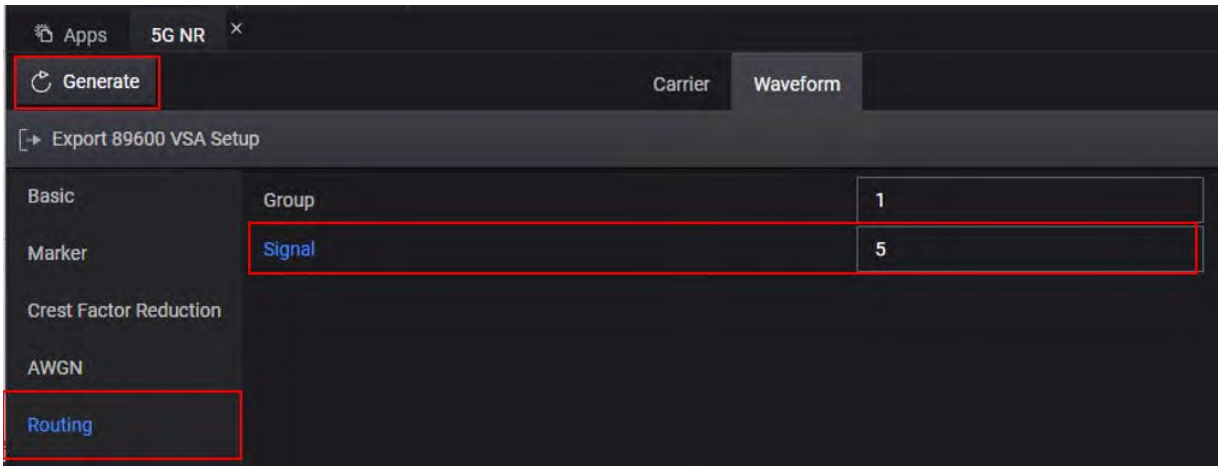
– Modulation to **64QAM**, then select **OK**



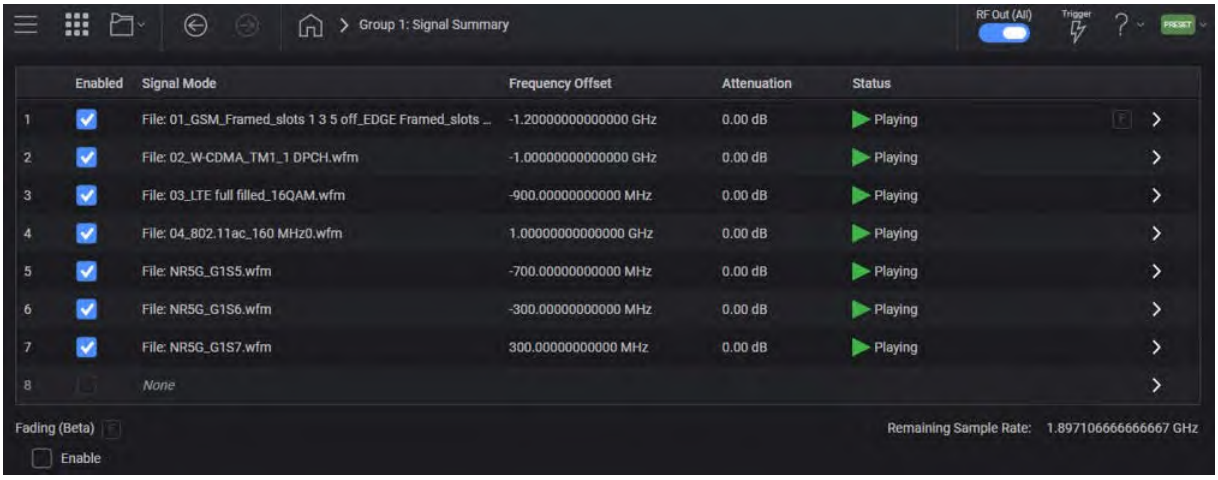
13. Select the **Waveform** tab and scroll down to turn Phase Compensation **Off**. This will allow the waveform to be offset in frequency without needing to be re-generated.



14. From the left pane, select the **Routing** tab, set the signal number to **5**, and then **Generate**.

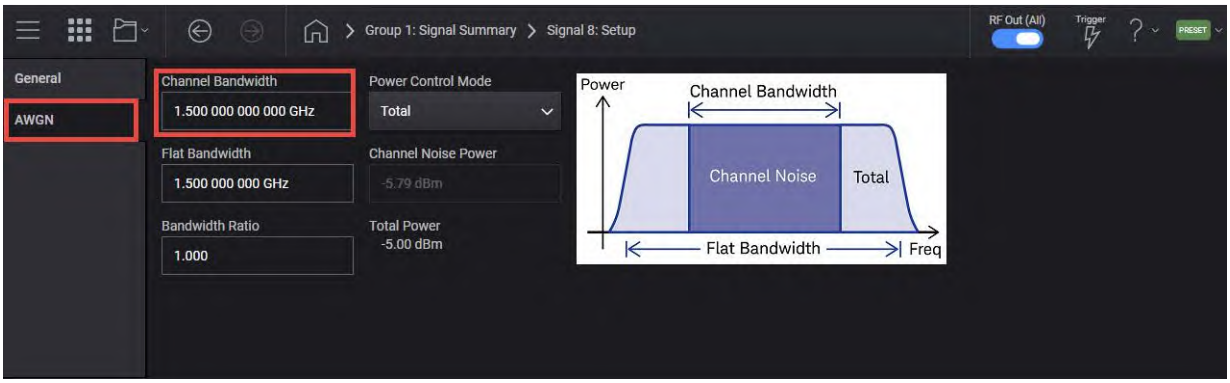


15. To create signal 6, select the **Carrier** tab > **Full-filled Config** and set Bandwidth to **FR2 200 MHz**.
16. Select the **Waveform** tab, then from the left pane, select the **Routing** tab, set the signal number to **6**, and then **Generate**.
17. To create signal 7, select the **Carrier** tab > **Full-filled Config** and set Bandwidth to **FR2 400 MHz**.
18. Select the **Waveform** tab, then from the left pane, select the **Routing** tab, set the signal number to **7**, and then **Generate**.
19. Return to Group 1: Signal Summary by selecting the **Home** icon > **Group1: Signals** and set the Frequency Offset for each of the newly generated 5G NR signals as follows:
    - Signal 5 to –700 MHz
    - Signal 6 to –300 MHz
    - Signal 7 to 300 MHz

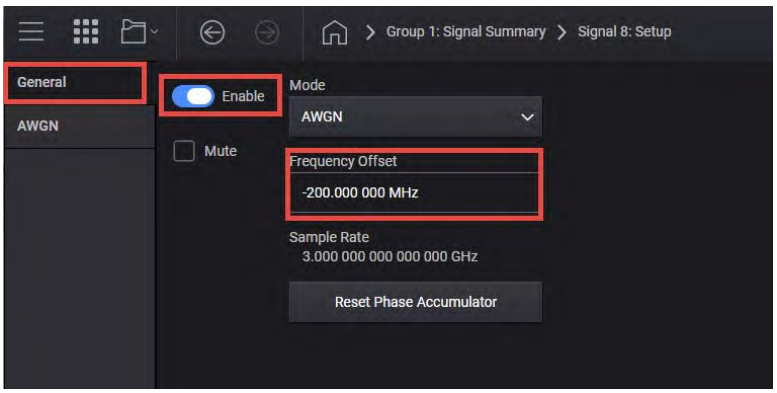


20. Select the arrow for Signal 8.

21. Select the **Mode** drop down and select **AWGN**. Select the AWGN tab and set Channel Bandwidth to **1.5 GHz.**

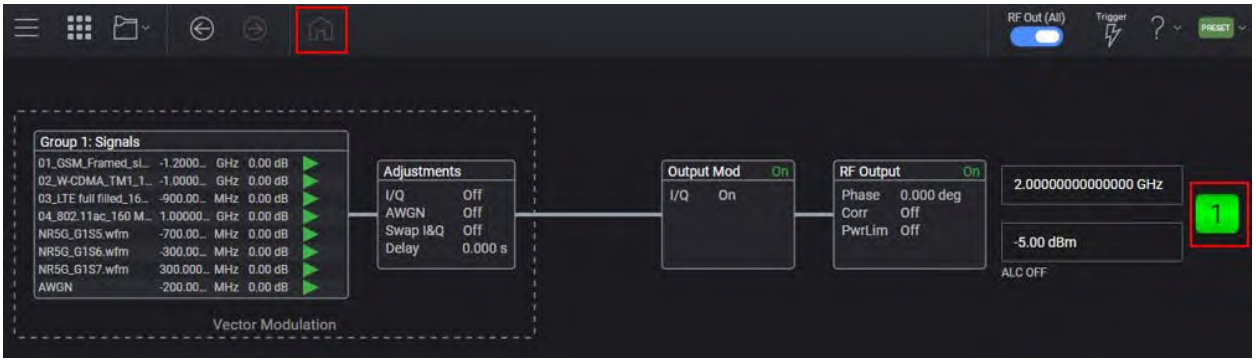


22. Select the **General** tab and set Frequency Offset to **-200 MHz** and **Enable** the AWGN signal.

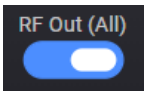


23. Select the **Home** icon and set RF Out to **On** by selecting the numbered channel indicator switch.

This enables the RF Out for the indicated channels, in this case Channel 1 if using a multi-channel VXG.



24. **For multi-channel instruments only:** In the top right corner of the display, set RF Out (All) to **On** by selecting the switch.



NOTE

In order to turn on RF for any channel, both the RF Out for the specific channel (for example, Channel 1 or Channel 2), and RF Out All must be turned on.

On the Signal Analyzer:

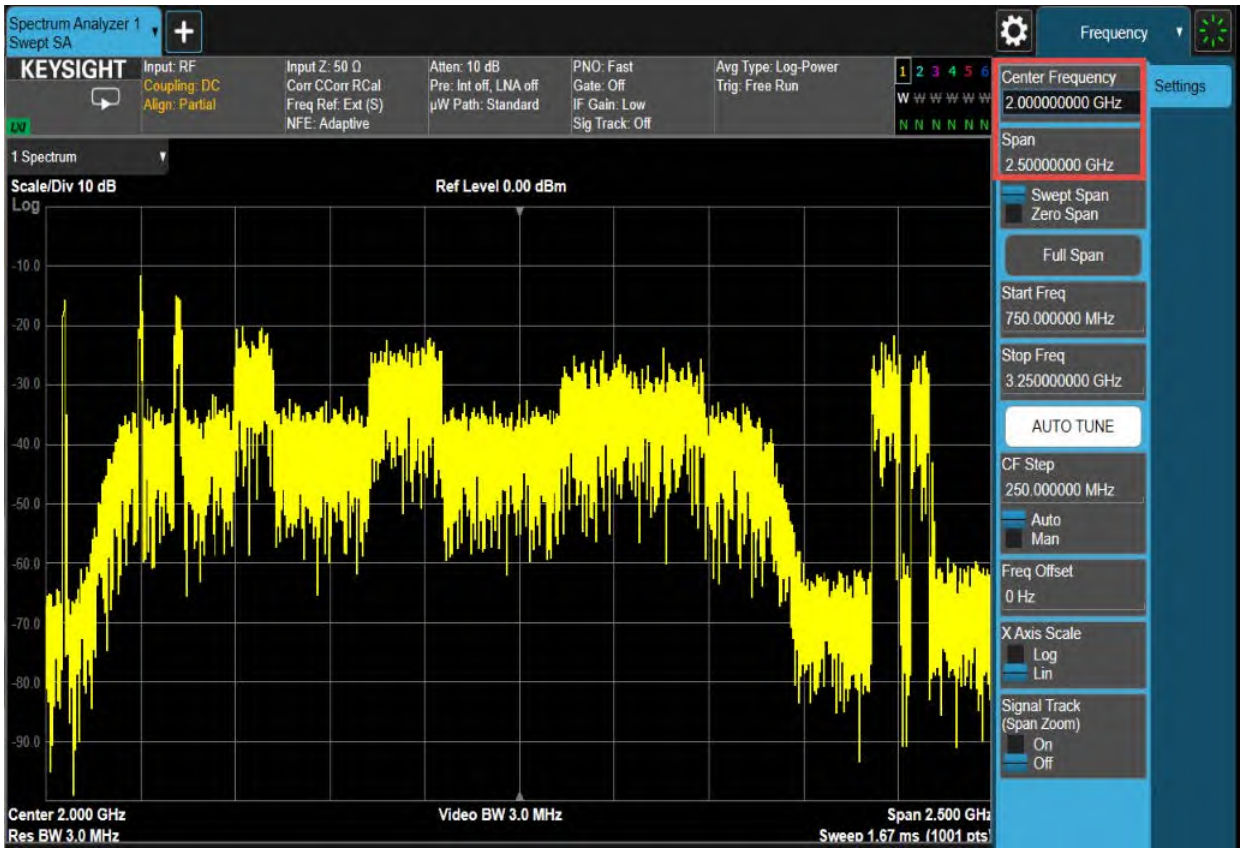
1. Select **Mode/Meas** > **Spectrum Analyzer** Mode > **Swept SA** Measurement.

NOTE

If accessing the instrument via a Remote Desktop connection, select the Screen tab (at the top of the display) to open the Mode/Measurement/View Selector window.

2. Select **Mode Preset** to set the spectrum analyzer to a known state.

3. Adjust the center frequency and span to view all of the signals around the RF center frequency. For this example, set the Center Frequency to **2 GHz** and Span to **2.5 GHz**.



4. Adjust the Resolution Bandwidth and Video Bandwidth as needed.
5. Now that 8 waveforms are playing, try some of the following experiments to see how different settings impact the signals.
   - Toggle the Enable checkbox for various signals.
   - Adjust the Frequency Offset of various signals. Note that signals can overlap.
   - Set up a Channel Power measurement on the Signal Analyzer. Observe how changing different signal settings (relative power, attenuation, mute, enable, etc.) either impact or do not impact Total Channel Power.
   - Use the Auto Configure Analyzer feature to demodulate one of the 5G NR signals. Note, the center frequency will need to be manually adjusted on the Signal Analyzer to match the Frequency Offset of the signal set on the VXG.
   - Change Trigger and/or Marker settings.

Using the VSA Flex Frame to Create a DVB-S2X Signal

The Flex Frame extension (89601AYAC) for the 89600 VSA software provides a means of creating complex frames to demodulate and analyze advanced custom IQ signals.

A frame can contain a combination of flexible elements, or allocations. Each allocation can be a Preamble, Pilot, Data or an Idle period. Allocations have configurable payload PN sequence and modulation type.

As you build the flexible frame in the Frame Definition tab, an Allocation Summary panel displays the order of each assembled frame part and allocation, and shows how the total number of symbols are distributed across the frame.

There are two parts to this example.

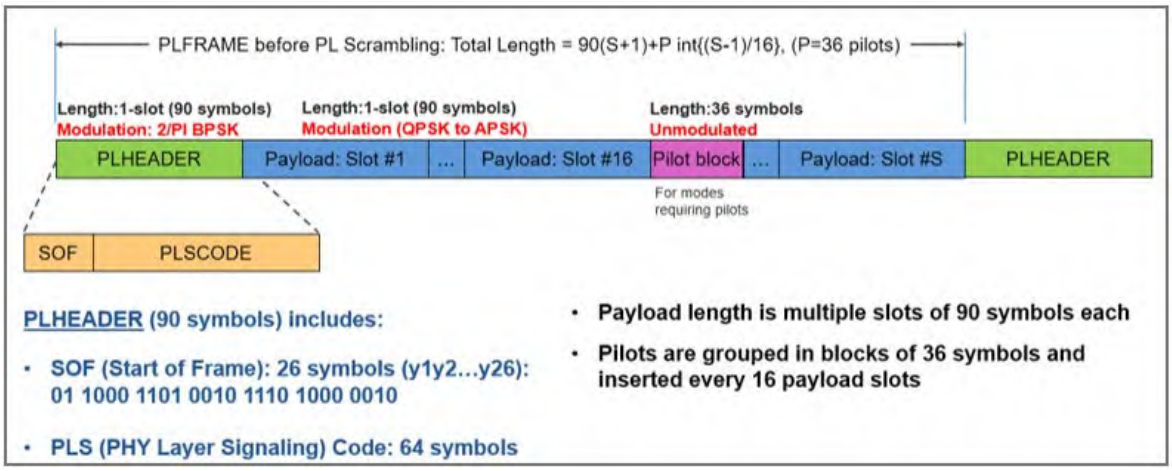
The first example uses N7623C Signal Studio for Digital Video. The N7623C generates a fully coded, full-standard compliant DVB-S2X waveform. Then, we will use VSA Flex Frame to demod the signal.

The second example uses PathWave N7608C Custom Modulation. Since the N7623C cannot provide an uncoded signal, we will use the N7608C to uncoded the signal to perform a Bit Error Rate (BER) measurement.

For this example, we will use the following equipment:

- N7623C Signal Studio for Digital Video (Software release 2020 Update 1.0 or higher
- N7608C Signal Studio for Custom Modulation (Software release 2021 or higher)
- 89600 VSA Digital Demodulation Analysis, Flex Frame Measurement (Software release VSA 2022 or higher)

DVB-S2X PHY Layer Framing



256 APSK modulation is used in this demo guide. The coding rate is represented differently between Signal Studio for Digital Video (N7623C), Signal Studio for Custom modulation (N7608C), and VSA Flex Frame (89601AYAC). N7608C and 89601AYAC use “Implementation MODCOD” name and N7623C uses “Canonical MODCOD” name.

In this example, we will use a coding rate of 32/45 in the N7623C which corresponds to 128/180 in the N7608C and VSA.

N7623C Signal Studio for Digital Modulation	N7608C Signal Studio for Custom Modulation	89601AYAC Flex Frame
256APSK 29/45	256APSK 116/180	256APSK 116/180
256APSK 2/3	256APSK 20/30	Not supported
256APSK 31/45	256APSK 124/180	256APSK 124/180
256APSK 32/45	256APSK 128/180	256APSK 128/180
256APSK 11/15	256APSK 22/30	Not supported
256APSK 3/4	256APSK 135/180	256APSK 135/180

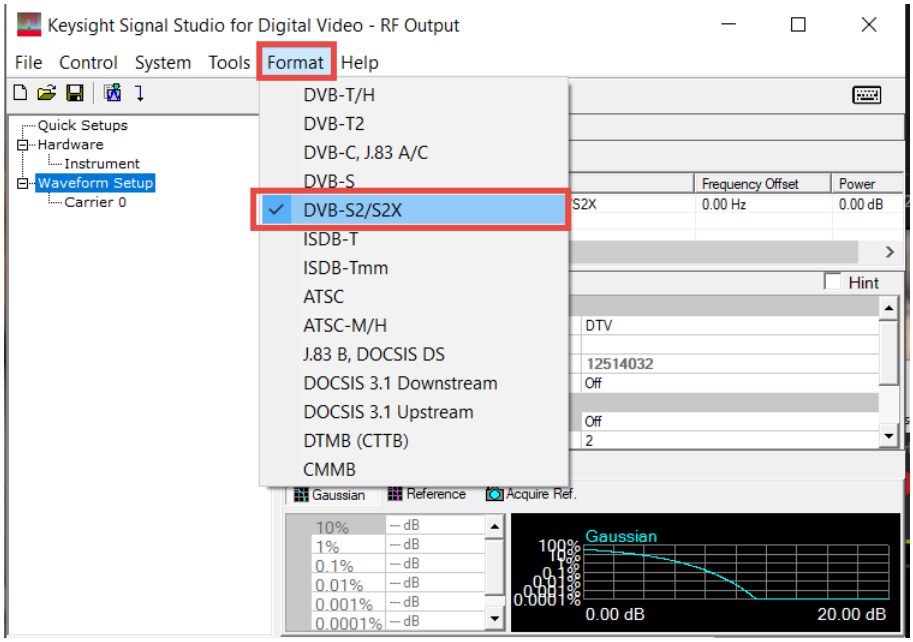
Part 1: Generating a DVB-S2X Waveform Using N7623C Signal Studio for Digital Modulation

NOTE

DVB-S2 and DVB-S2X operate in Ku band (10-14 GHz) and Ka band (18-30 GHz). This example uses a sub-6 GHz hardware thus using a center frequency of 4.5 GHz. Modify the frequency based on the hardware you are using.

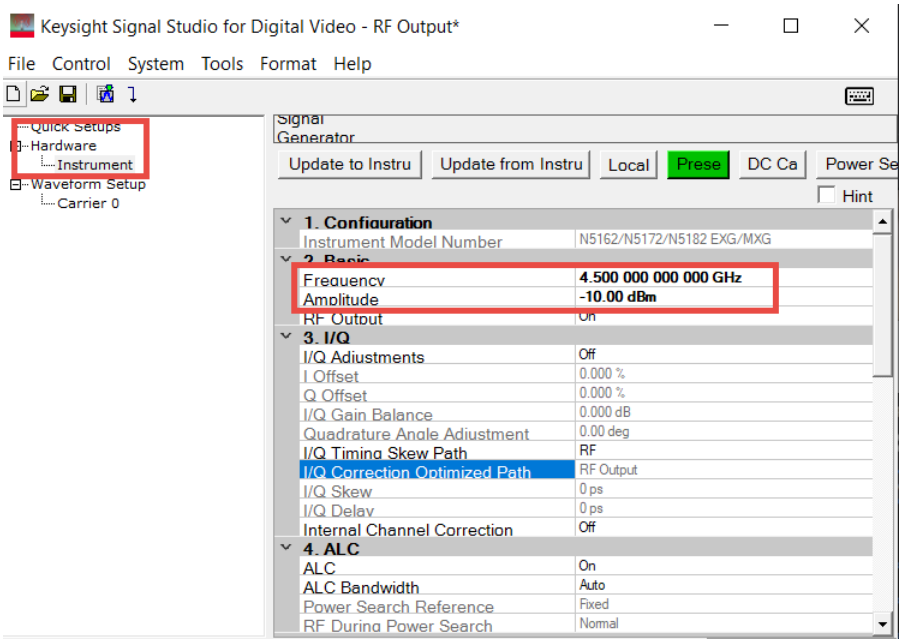
Setting Up the Waveform in Signal Studio for Digital Video

1. From the main menu, select **Format** > **DVB-S2/S2X**.



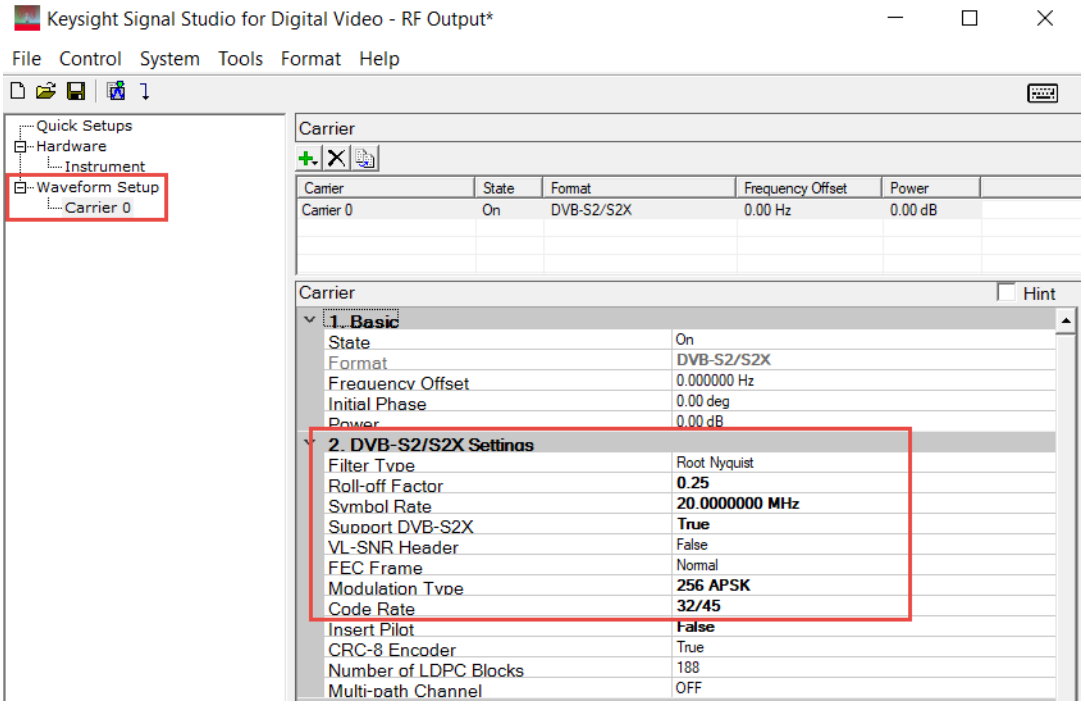
2. From the left panel, select **Quick Setups** > **Hardware** > **Instrument**.

3. Set the Frequency to **4.5 GHz** and the Amplitude to **-10 dBm**.



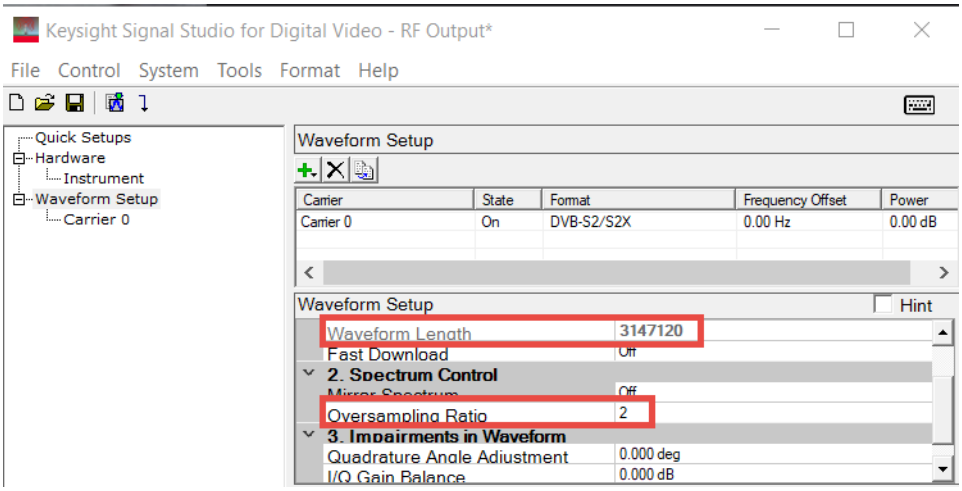
4. Under **Waveform** Setup, select **Carrier** and configure the DVB-S2/S2X signal to the following parameters.
   - Roll of Factor to **0.25**
   - Symbol Rate to **20 MHz**
   - Support DVB-S2X to **True**
   - Modulation Type to **256 APSK**
   - Code Rate to **32/45**

This is equivalent to MODCOD 256APSK 128/180 in the VSA.



5. In the **Waveform Setup** window, set the Oversampling Ratio to **2**.

   This results in a Waveform Length of 3147120. We will use this length when configuring the signal in VSA Flex Frame.



6. From the **Control** menu, select **Generate**.
7. Select **Control** menu > **Download** to download it to your Signal Generator.

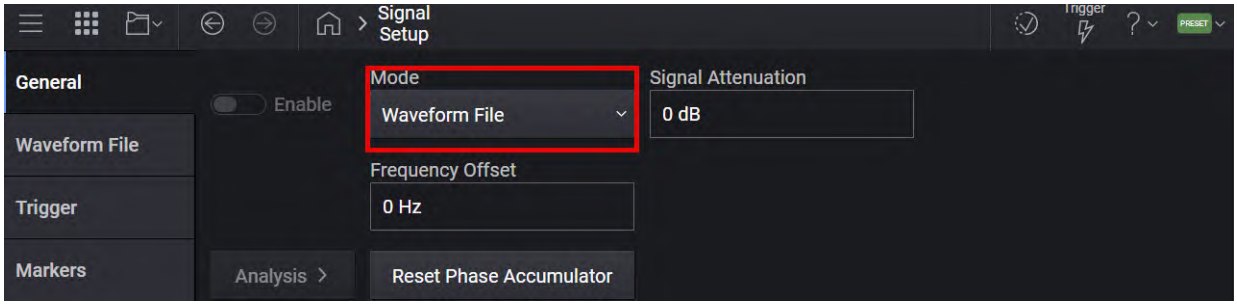
To Export and Load a Waveform File from the PC to the MXG

NOTE

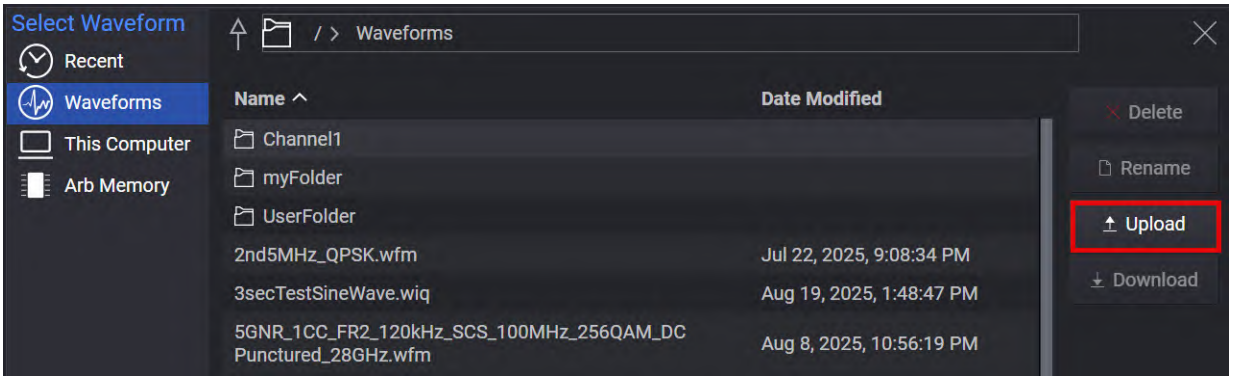
Depending on the signal generator you are using, the steps for downloading the waveform may be different. If this is the case, go to File > Export Waveform Data and Save the file. Then load it onto the your signal generator.

For this example we will export the Signal Studio wave we just created using the Edit Waveform Pack License feature in PathWave. This feature allows you to play Signal Studio waveform files without purchasing Signal Studio software packages. You can access this feature using the Edit Waveform Pack option in the Waveform Utility menu of the Waveform File tab.

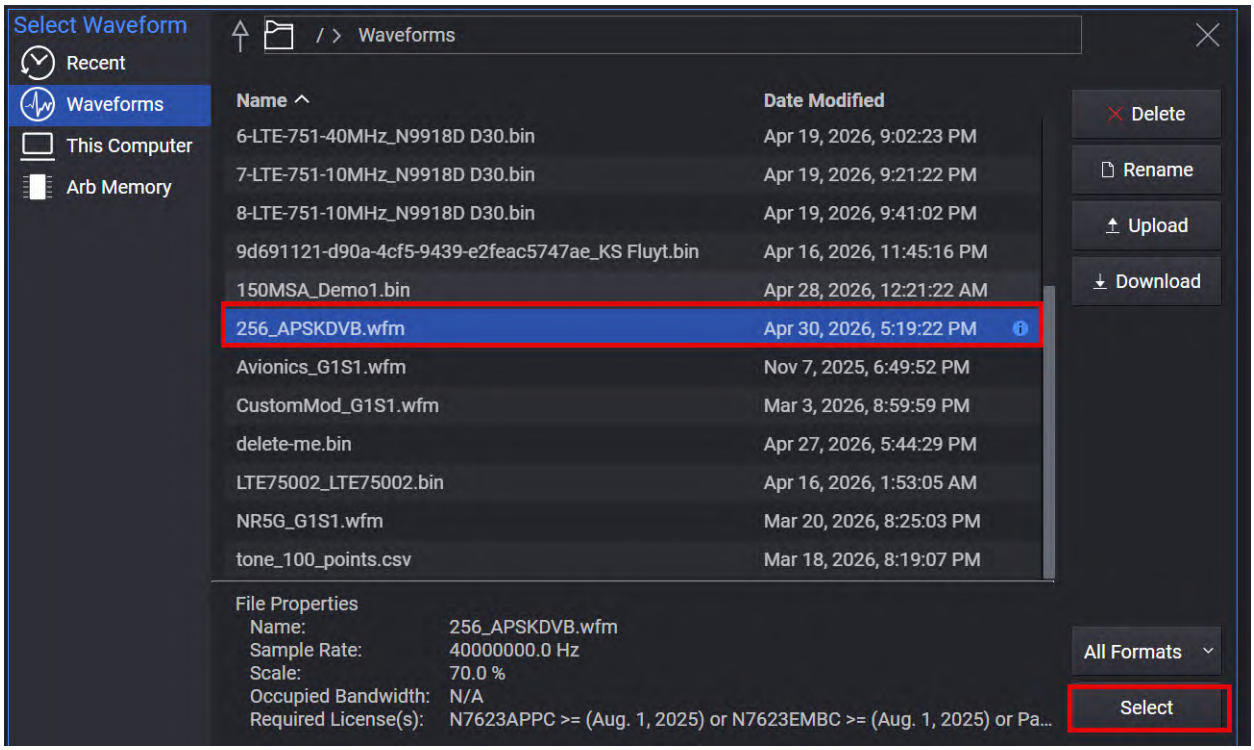
1. In Signal Studio, select **File** > **Export** and save the file to your PC.
2. On the MXG, select the Signal Block > set Mode to **Waveform File**.



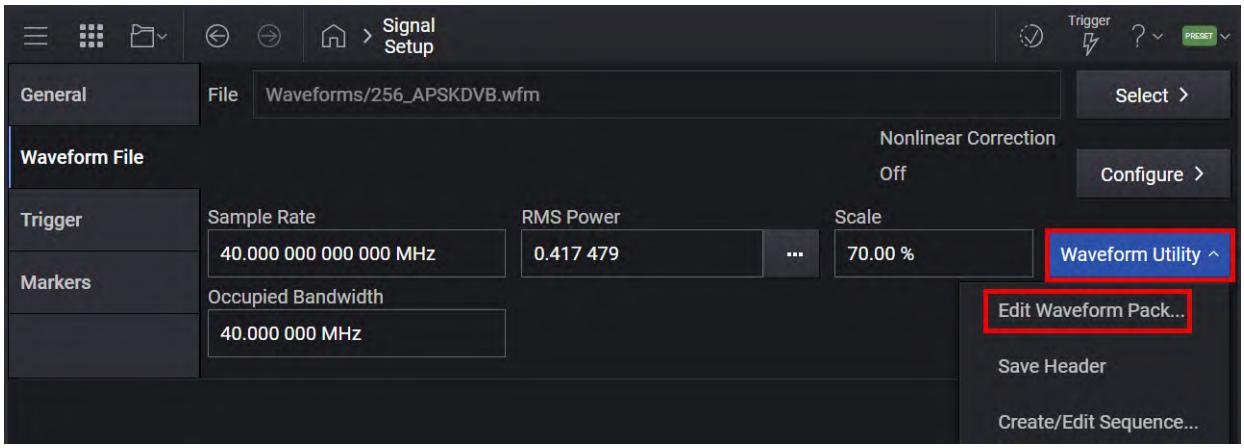
3. Select the **Waveform File** tab > **Select** > **Upload** >then scroll down to find the waveform file and **Open**.



4. **Close** the Upload Complete dialog, scroll down and **Select** the waveform file.

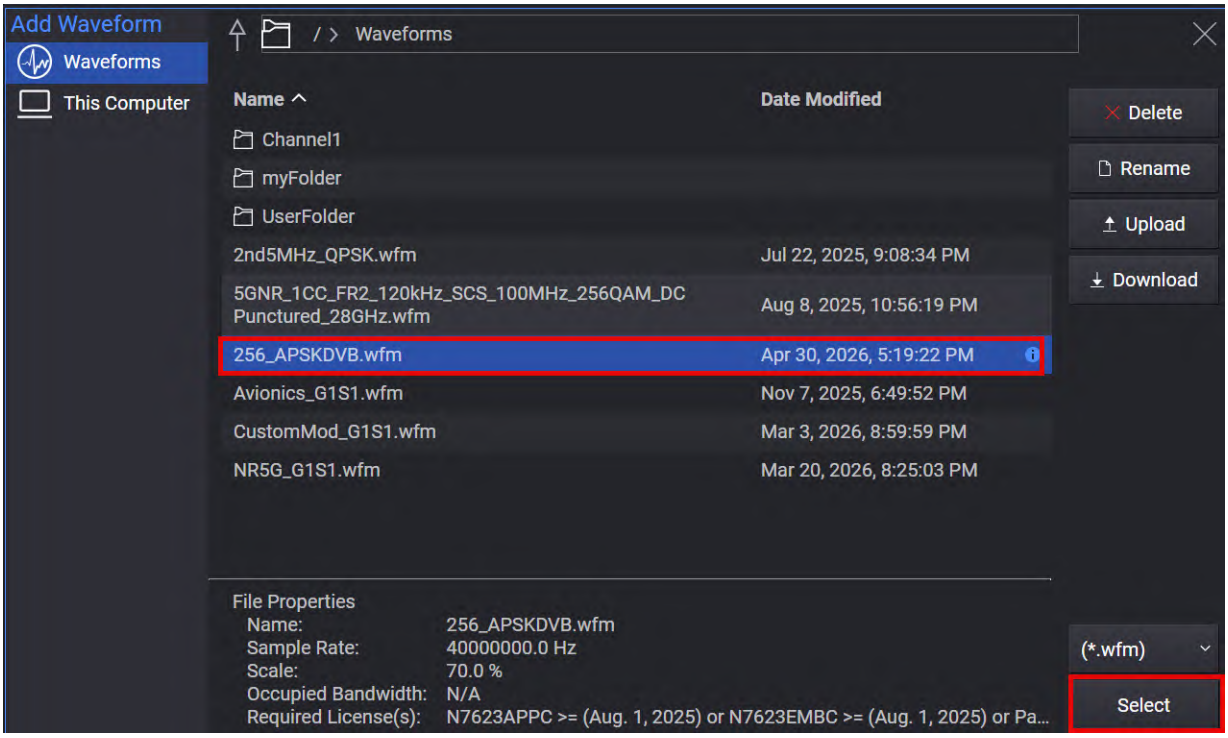


5. Select **Waveform Utility** > **Edit Waveform Pack**.

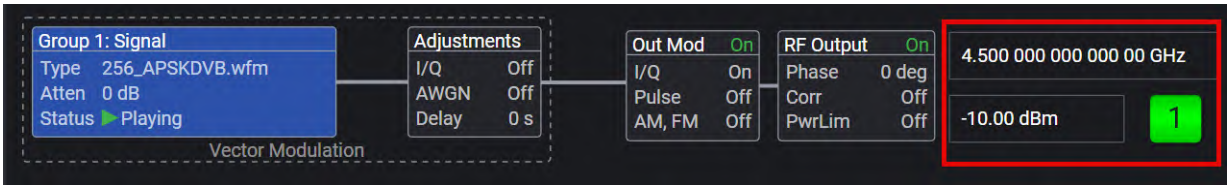


6. Select **Add Waveform** and scroll down and **Select** the .wfm file.

The Signal Studio generated waveform is now available for use.



7. Select the **General** tab and **Enable**.
8. Set the RF out to **4.5 GHz** > Power to **-10 dB**m > Turn RF Out **On**.



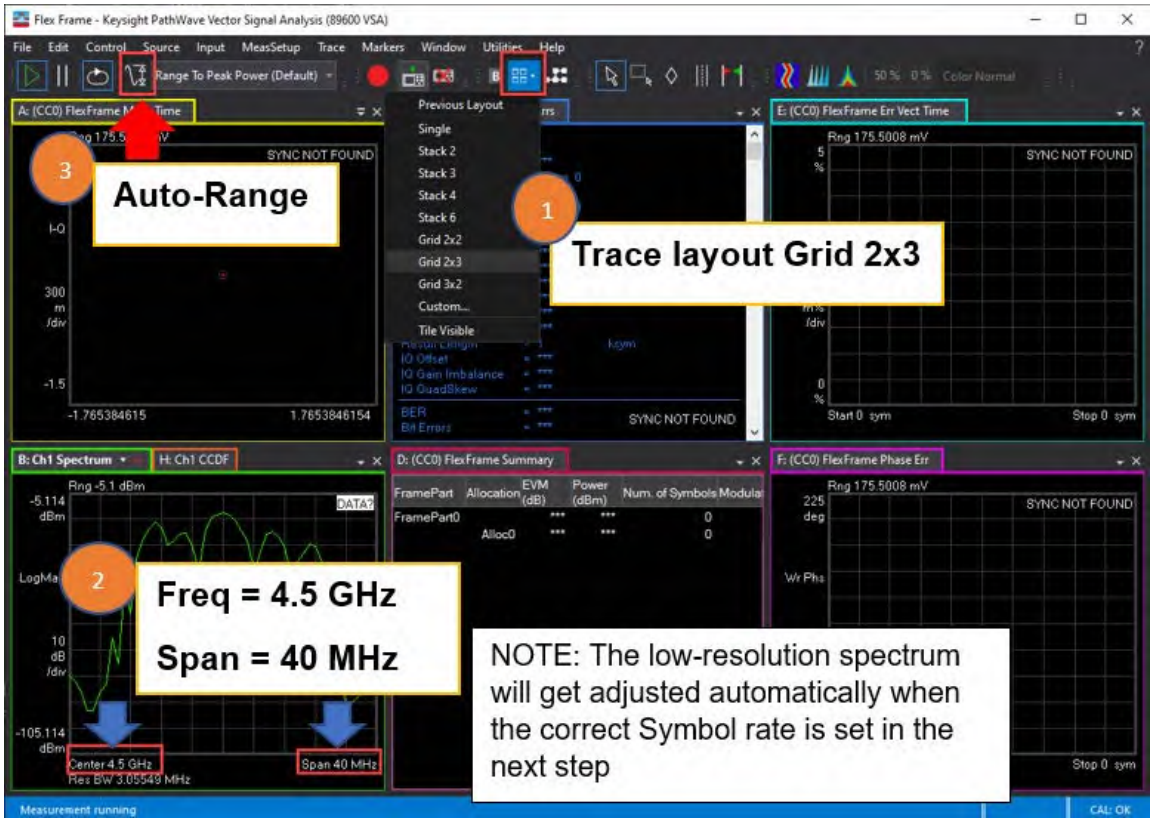
Setting up and Analyzing the Signal Using VSA Flex Frame

NOTE

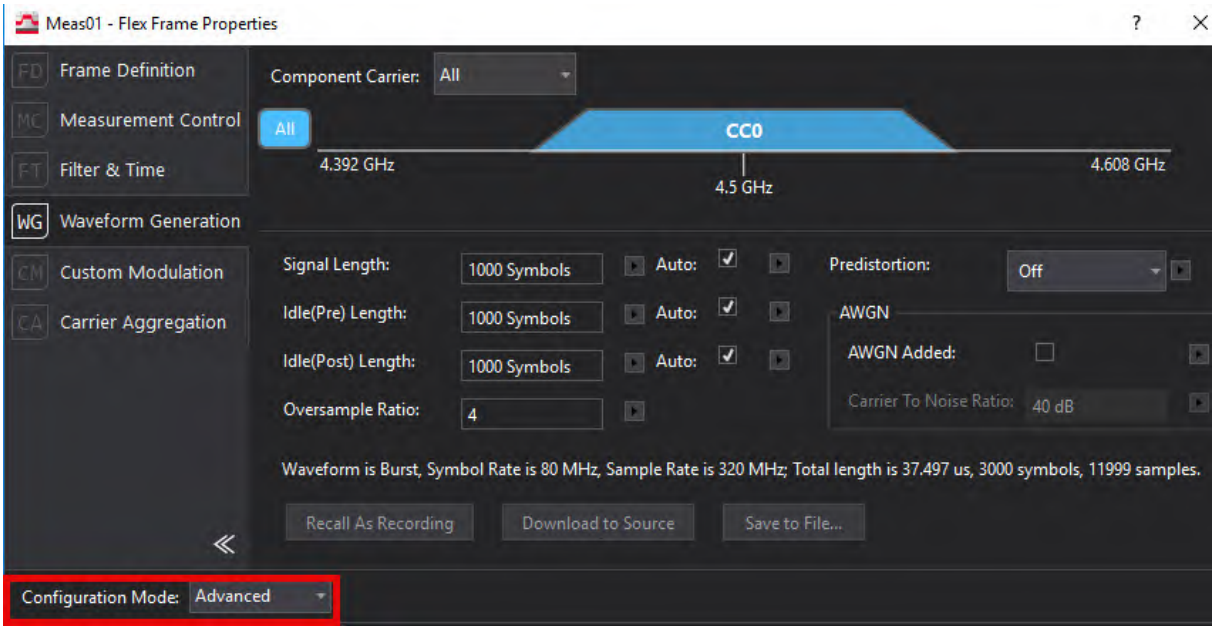
Changing settings on the VSA can take longer to take effect than expected. Wait to change the next setting until the previous setting has been updated.

1. From the VSA menu bar, select **MeasSetup** > **Measurement Type** > **General Purpose** > **Flex Frame**.
2. Set the correct frequency, span, and input range. For this example, set:
   - Change the Trace Layout Grid to 2 x 3.
   - Change Trace B to **Ch1 Spectrum**, and set Center Frequency to **4.5 GHz**

- Span to **40 MHz**
- Autorange



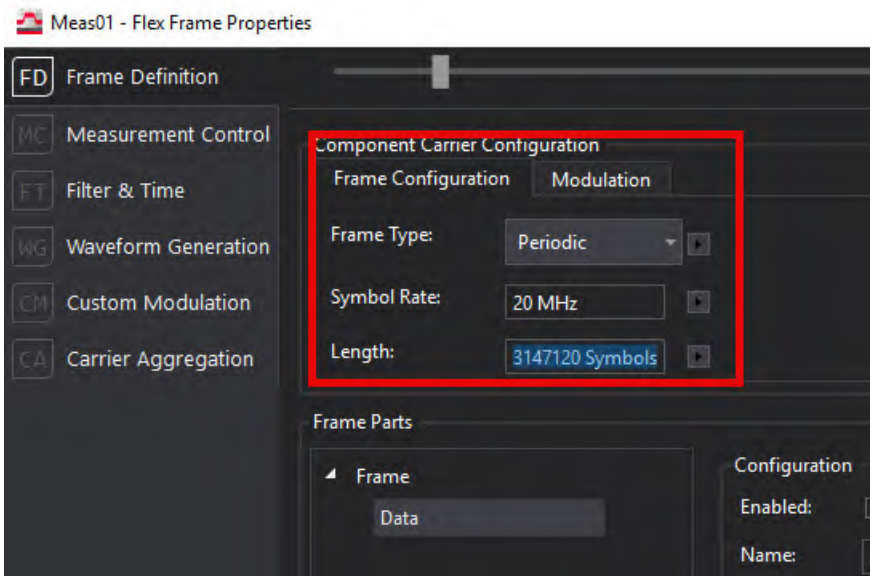
3. Select **MeasSetup** > **Flex Frame Properties** > **Configuration Mode to Advanced.**



4. Select the **Frame Definition** tab and set:
   - Frame Type to **Periodic**
   - Symbol Rate to **20 MHz**
   - Length to **3147120 Symbols**

     To match the waveform length in Signal Studio.

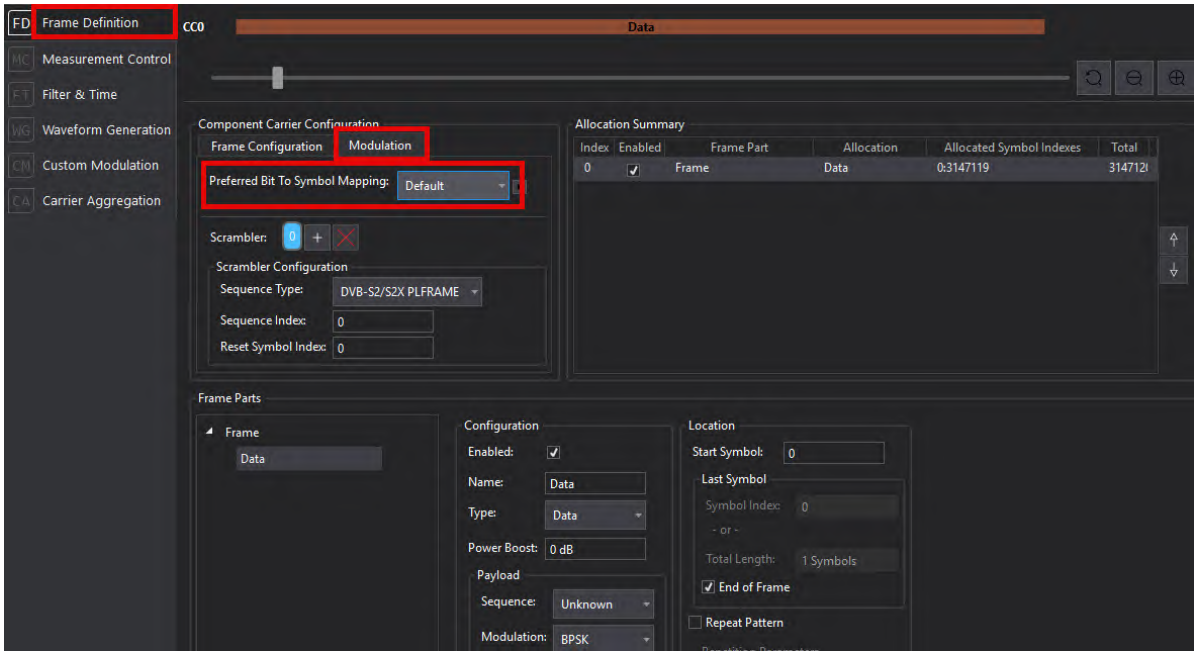
For a bursted signal, Frame Type is set to Burst, and the frame length can be automatically set based on the detected burst length.



5. In the Component Carrier Configuration area, select the **Modulation** tab and set Preferred Bit to Symbol Mapping to **Default**.

   When testing a DUT, if the bit mapping into constellation point is as defined by DVB-S2X, DVB-RCS2 or 802.11ad/ay standard, selecting the correct standard under "Bit To Symbol Mapping" is required for accurate Tx BER measurement. When using Signal Studio, leave it as Default.

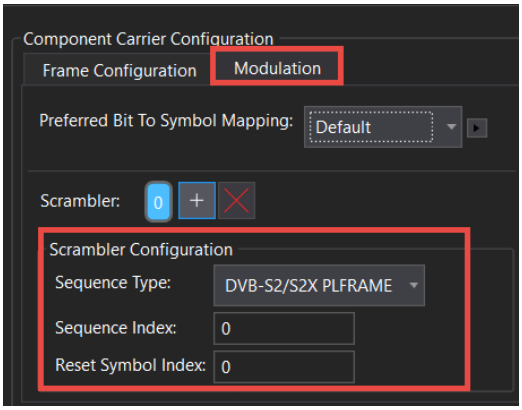
   Constellation mapping definition between VSA Flex Frame and N7608C Signal Studio is not aligned. VSA comes with a set of bit mappings (.txt file) to recall into the N7608C and this is required for successful demodulation. You can find it under: C:\Program Files\Keysight\<version>\89600 VSA Software\Help\Signals\FlexFrame.



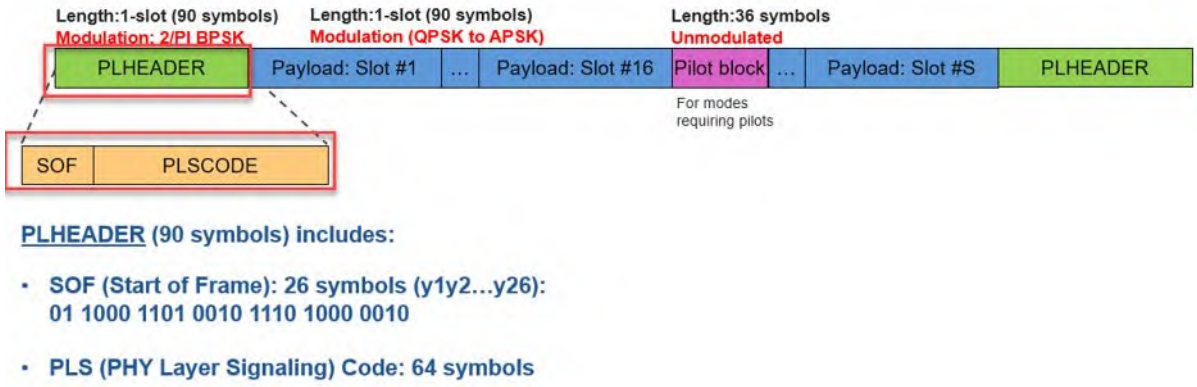
6. Set the Sequence Type to **DVB-S2/S2X PLFRAME**.

   This selects the scrambling method as described in section 5.5.4 of EN 302.301-1.

7. Both Sequence Index and Reset Symbol are set to **0**.

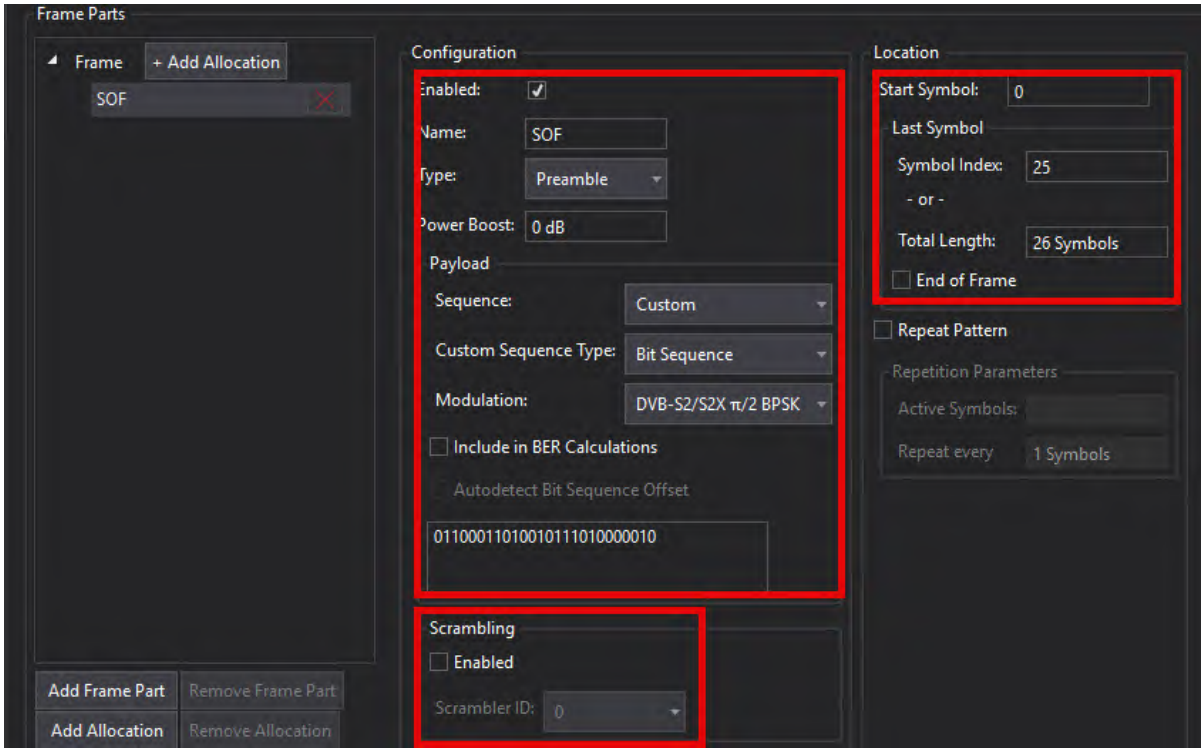


Configure the Preamble (PLHEADER)



8. Scroll down to the **Frame Parts** area and configure the Start of Frame Index (SOF), 26 symbols long:
   - In the Frame Parts > Configuration area, set
     - Name to **SOF**
     - Type to **Preamble**
     - Sequence to **Custom**
     - Modulation to **DVB-S2/S2X π/2BPSK**
     - Enter the 26 symbols **01100011010010111010000010**
     - (011 000 110 100 101 110 100 000 10)
   - In the Location area,
     - Clear the End of Frame checkbox
     - Start Symbol to **0**
     - Total Length to **26**

Note that Scrambling is not applied to Preamble.



Configure the PLS (PHY Layer Signaling) Code 64 symbols long

9. In the Frame Parts area:

- Select **Add Allocation**

10. In the Configuration area, set:

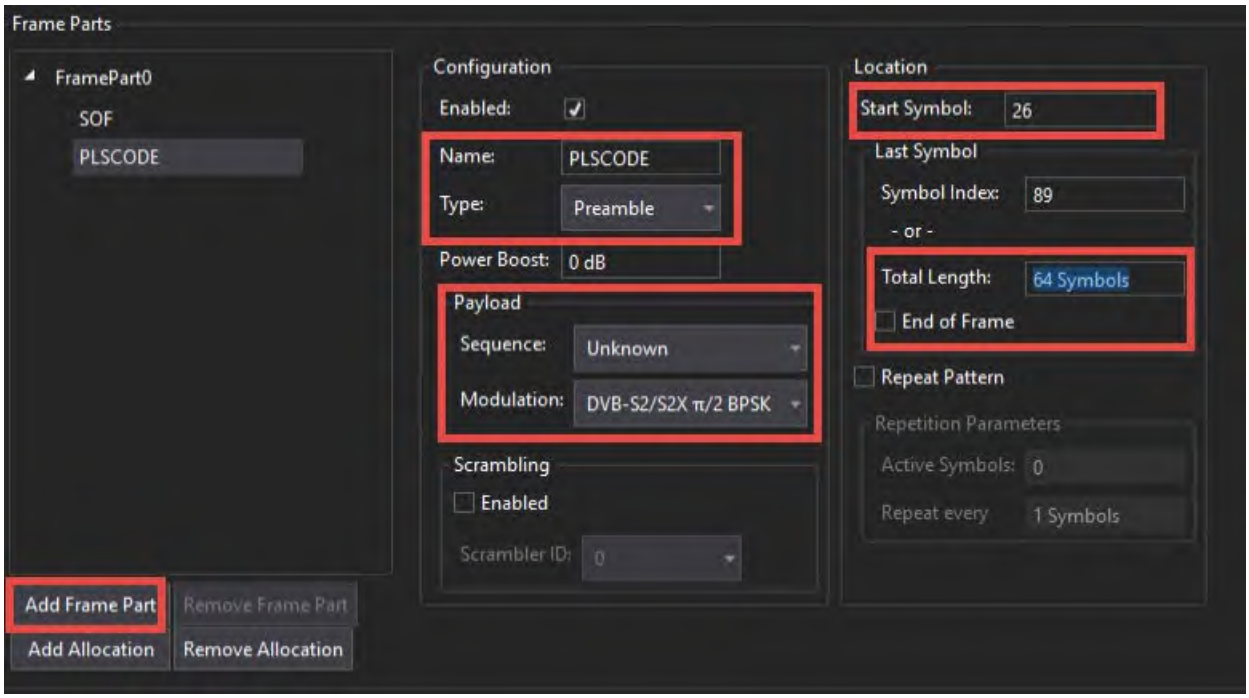
- Name to **PLSCODE**
- Type to **Preamble**
- Sequence to **Unknown**
- Modulation to **DVB-S2/S2Xπ/2 BPSK**

  Notice that Scrambling is not applied to Preamble.

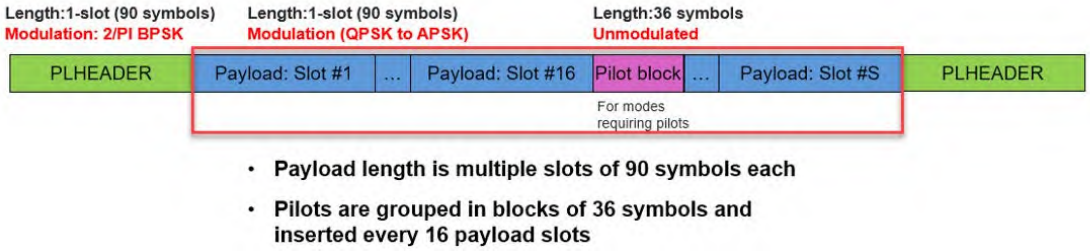
11. In the Location area:

- Clear the End of Frame checkbox
- Start Symbol to **26**

– Total Length to **64 Symbols**



Configure the Payload



- **Payload length is multiple slots of 90 symbols each**
- **Pilots are grouped in blocks of 36 symbols and inserted every 16 payload slots**

12. In the Frame Part area, select **Add Allocation**.

13. In the Configuration area:

– Name to **Data**

– Type to **Data**

– Sequence to **Unknown**

– Modulation to DVB-S2/S2X APSK

– MODCOD to **256APSK Normal 128/180**

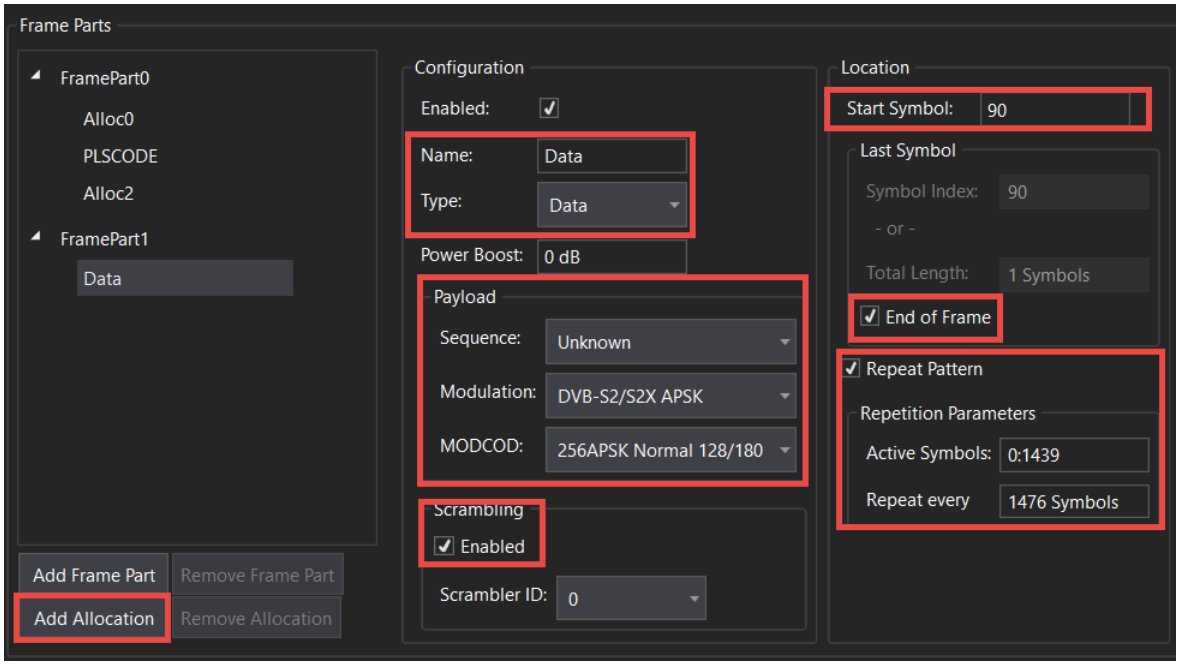
– Scrambling to **Enabled**

14. In the Location area, set:

- Start Symbol to **90**
- Last Symbol to **End of Frame**
- Select **Repeat Pattern**
- Set Active Symbols to **0:1439**

  Before the Pilot block, there are 16 payload slots, each 90 symbols long resulting in 1440 active symbols.
- Repeat every to **1476 Symbols**

  The 1440 payload symbols are followed by a 36-symbols Pilot block resulting in a total of 1476 symbols. This will repeat until the end of the frame length.



Configure the Pilot

15. In the Frame Part area, select **Add Allocation**.

16. In the Configuration area:

- Name to **Pilot**
- Type to **Pilot**
- Sequence to **Unknown**
- Modulation to **QPSK**

  Pilots are unmodulated, but they are identified by

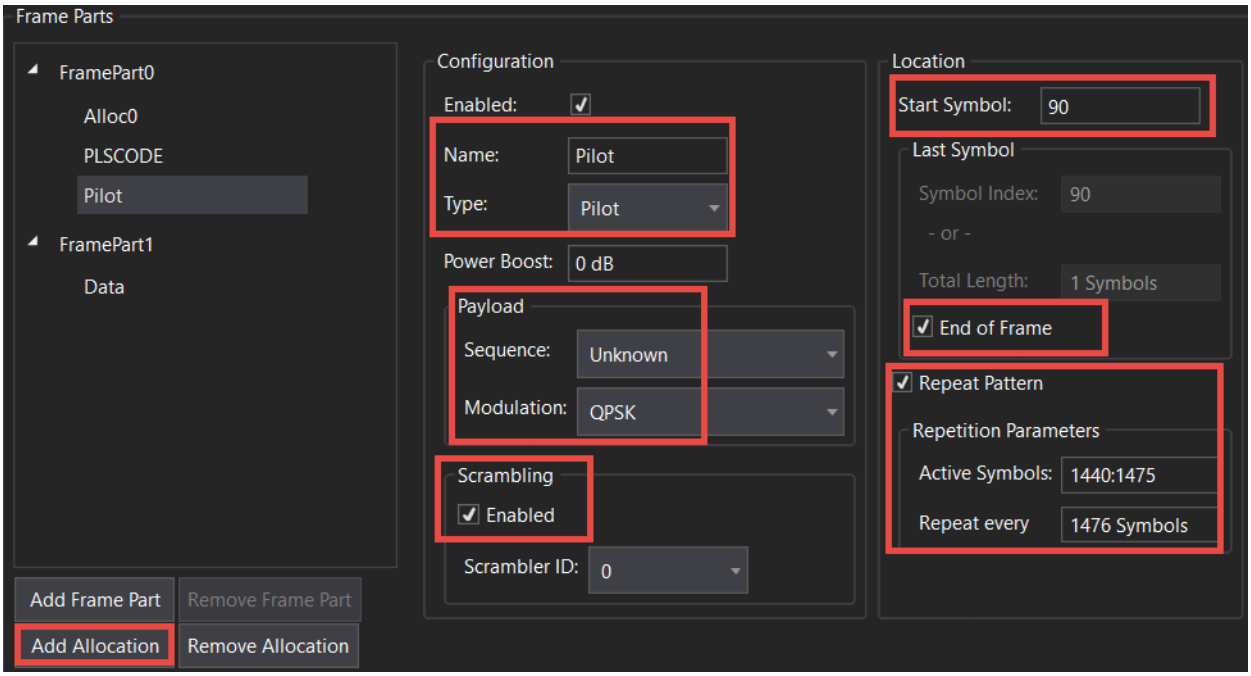
  $I=Q=1/\sqrt{2}$
- Scrambling to **Enabled**

17. In the Location area, set:

- Start Symbol to **90**
- Last Symbol, select **End of Frame**
- Select **Repeat Pattern**
- Set Active Symbols to **1440:1475**

  Pilot is 36 symbols long following a 1440 symbols long payload.
- Repeat every to **1476 Symbols**

  The 1440 payload symbols are followed by a 36-symbols Pilot block resulting in a total of 1476 symbols. This will repeat until the end of the frame length.



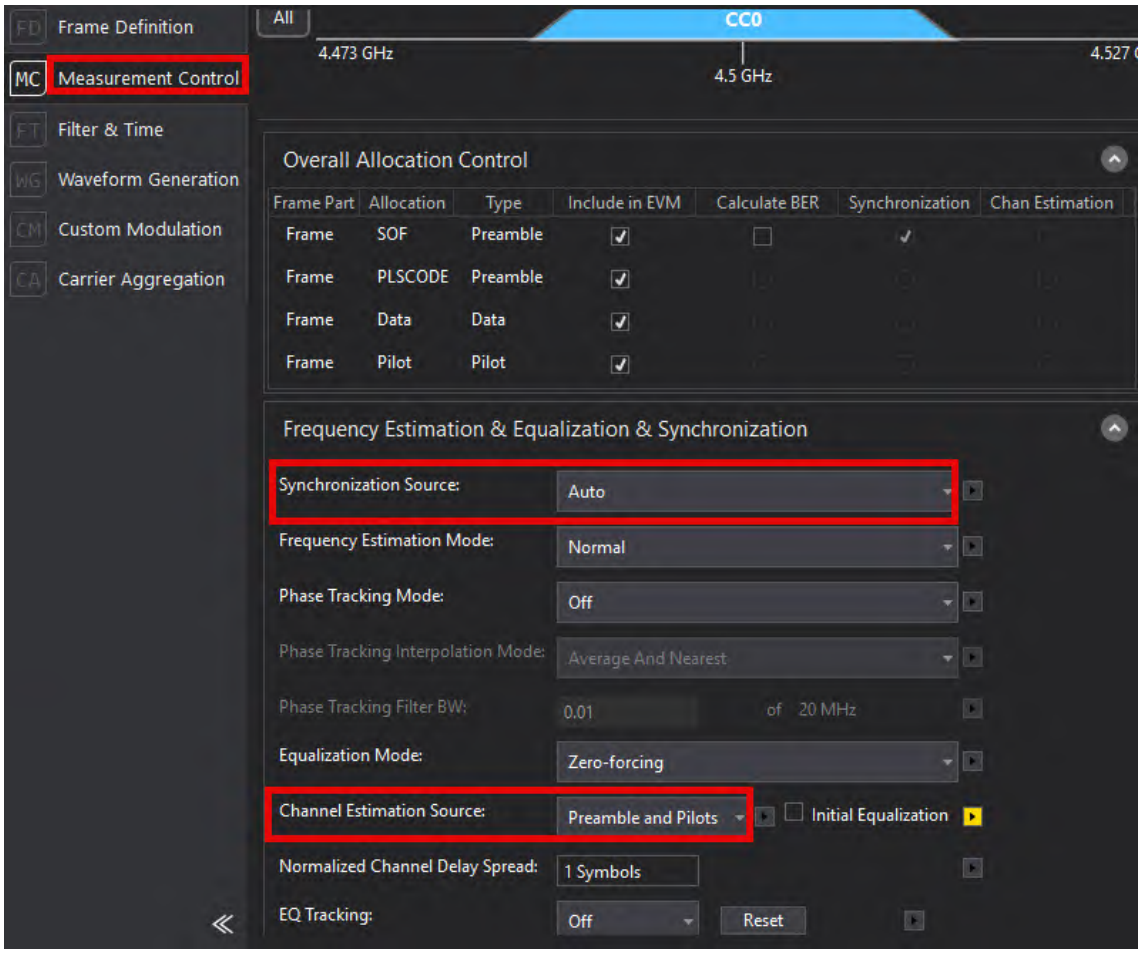
Configure Synchronization and Equalization

18. Select the **Measurement Control** tab, and set:

- Synchronization Source to **Auto**

  By default, SOF is used for synchronization. This can be changed to "Customized" to add additional frame parts for synchronization.
- Channel Estimation Source to **Preamble and Pilots**

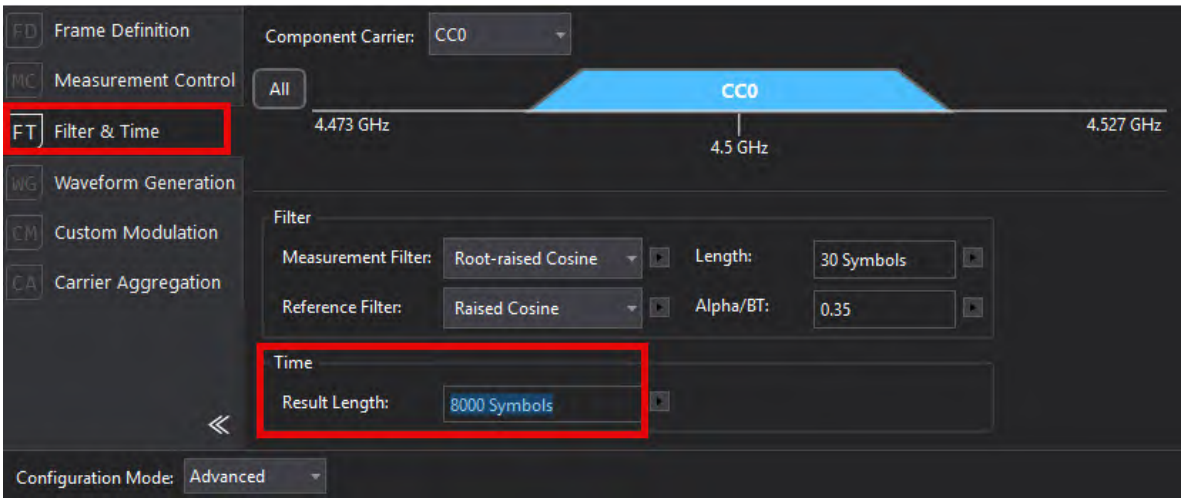
Equalization is off by default. You can choose the different frame parts for channel estimation and equalization.



Configure the Analysis Region

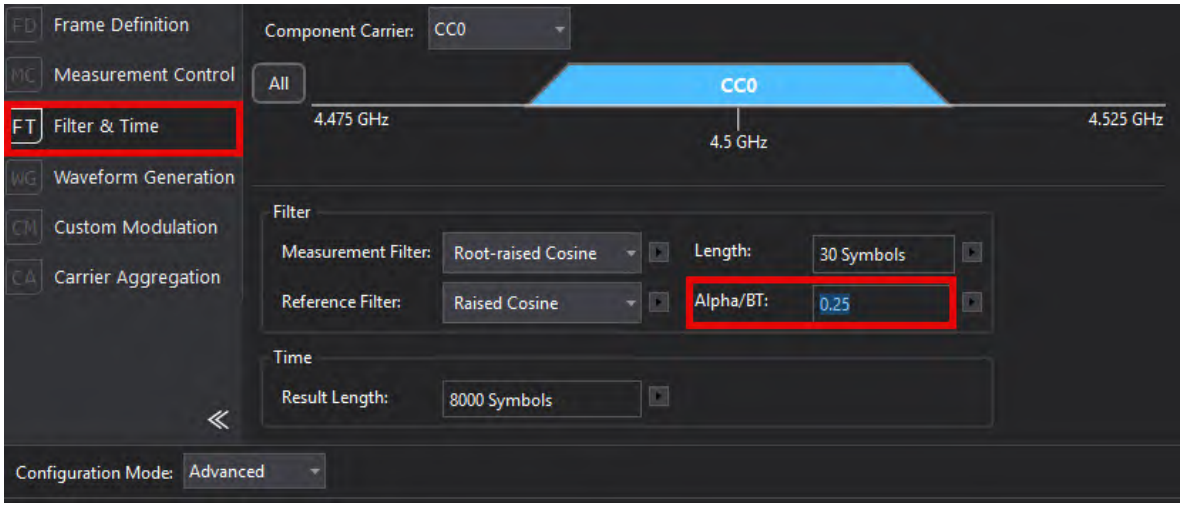
19. Select the **Filter & Time** tab, set the Result Length to **8000 Symbols**.

Setting the result length to ≤ the waveform length when oversampling ratio of 1, provides the best performance. In Signal Studio, when oversampling ratio is set to 1, the waveform length is 8370. Using a result length of ≤ 8370 symbols provides the best EVM performance.


Frame Definition
Measurement Control
Filter & Time
Waveform Generation
Custom Modulation
Carrier Aggregation
Component Carrier:
CC0
All
CC0
4.473 GHz
4.5 GHz
4.527 GHz
Filter
Measurement Filter:
Root-raised Cosine
Length:
30 Symbols
Reference Filter:
Raised Cosine
Alpha/BT:
0.35
Time
Result Length:
8000 Symbols
Configuration Mode:
Advanced

Configure the RRC filter

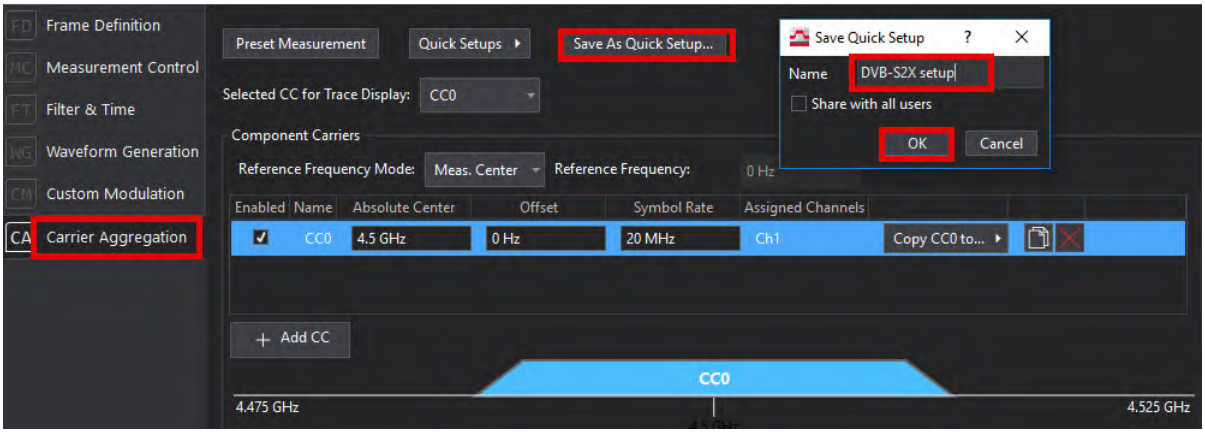
20. Set Alpha/BT: to 0.25.



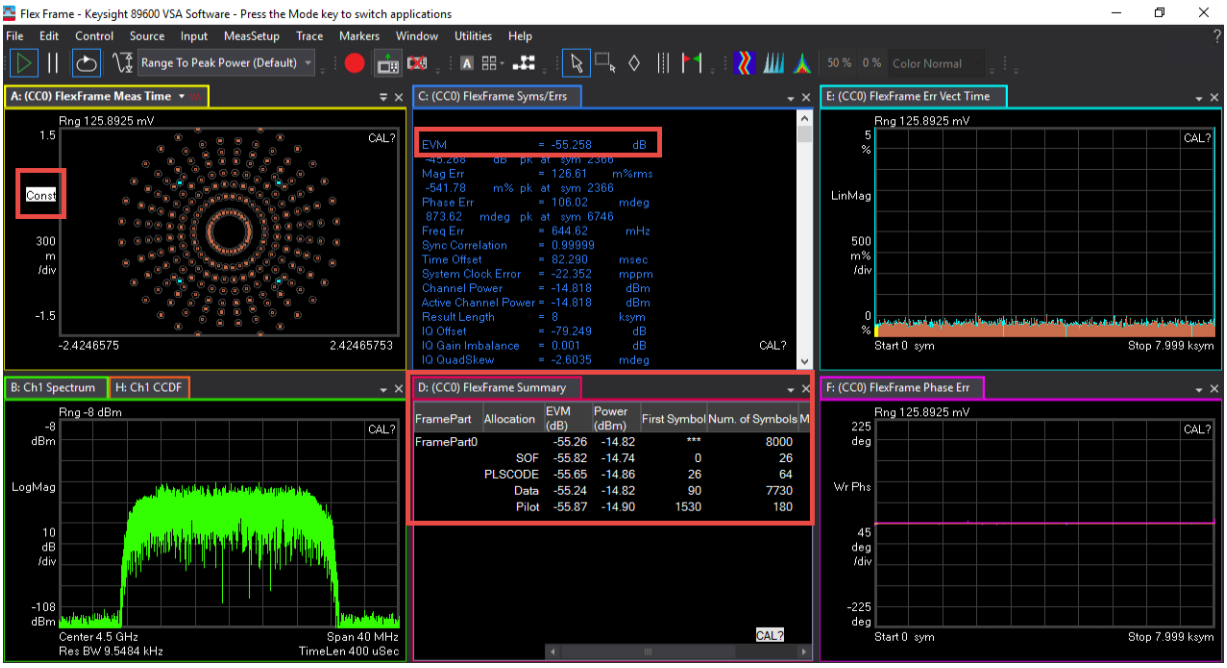
Save the Configuration as a Quick Setup

21. In the **Carrier Aggregation** tab, select **Save As Quick Setup**.

This saves the measurement settings for future use. Select **Share with all users** if you want to share with other users on the PC running the VSA software.



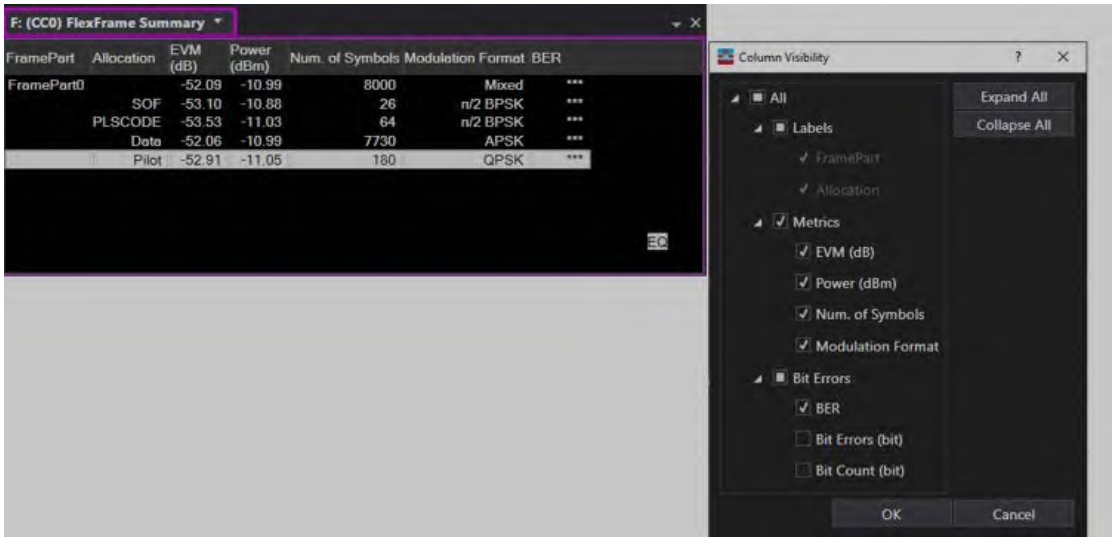
22. Close the Properties window and change Trace A to a Constellation Diagram, then view the results.



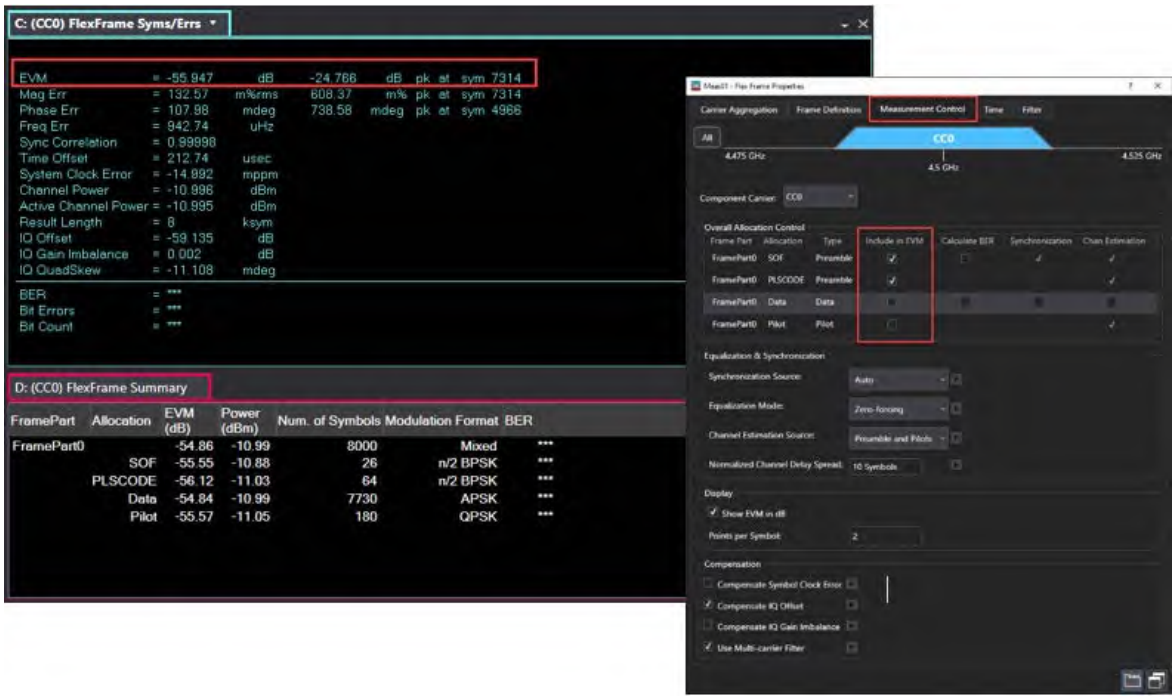
You should see a successful demodulation with very low EVM. Multiple traces and tables are available including Error Vector Spectrum, Error Vector Time, demod, and BER Bits.

The FlexFrame Summary table is a type of Matrix Table that provides sortable rows by column, selectable column visibility, and copy/paste and export functionality to share rows of content or complete tables to applications like email, text editing or spreadsheet programs.

Right click the column header to display the menu.



FlexFrame Summary always displays the EVM of each active segment. Flex Frame Syms/Errs table shows composite EVM of segment(s) selected under Measurement Control -> Include EVM. By default, it shows all segments.



Part 2: Generating a DVB-S2X Waveform Using the N7608APPC Signal Studio for Custom Modulation

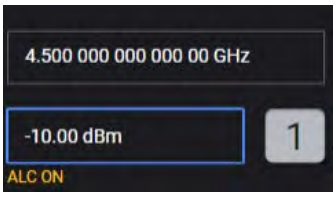
For this example, we will use the embedded N7608APPC application on the N5186A MXG.

NOTE

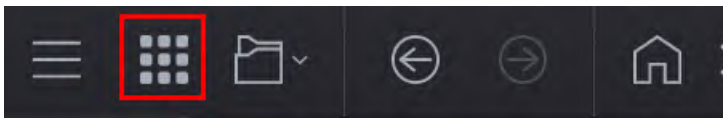
The MXG must have the N7608APPC Signal Generation for Custom Modulation license installed. You can also use N7608C Signal Studio for Custom Modulation

Setting Up the Waveform in PathWave Custom Modulation

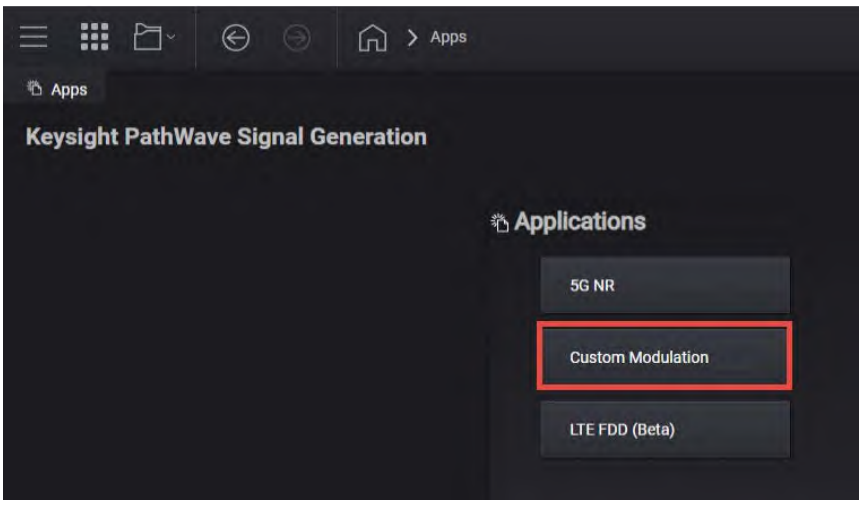
1. On the MXG, select **Preset** > **Preset** to set the MXG to a known state.
2. In the Output area, set Frequency to **4.5 GHz** and Power to **-10 dBm**.



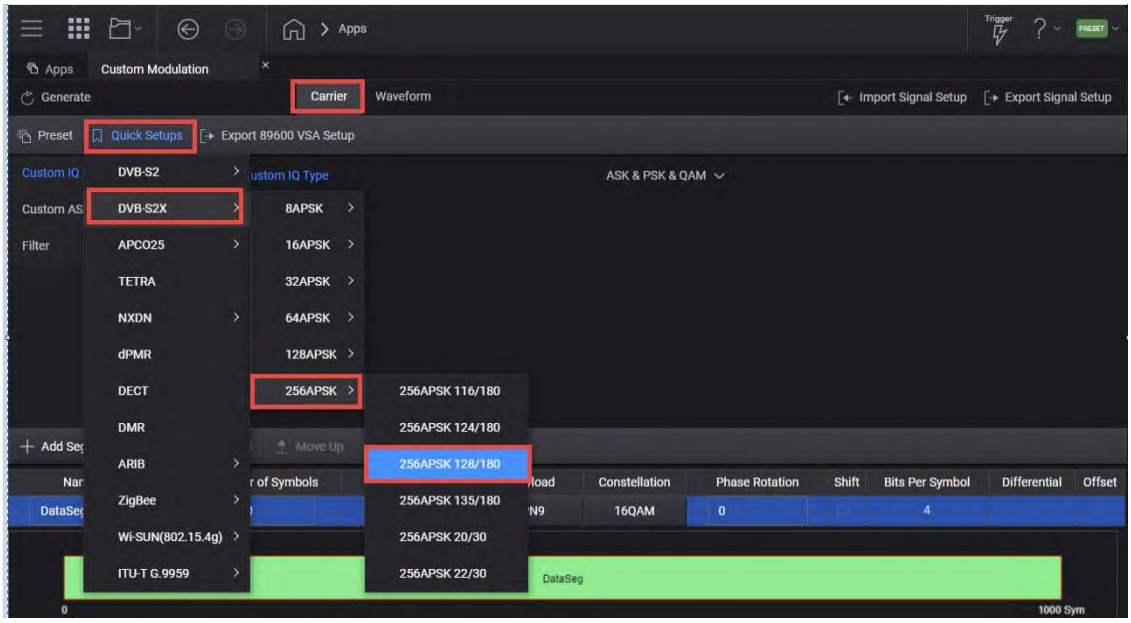
3. Select the **Apps** block to open the Vector Modulation Signal Setup panel.



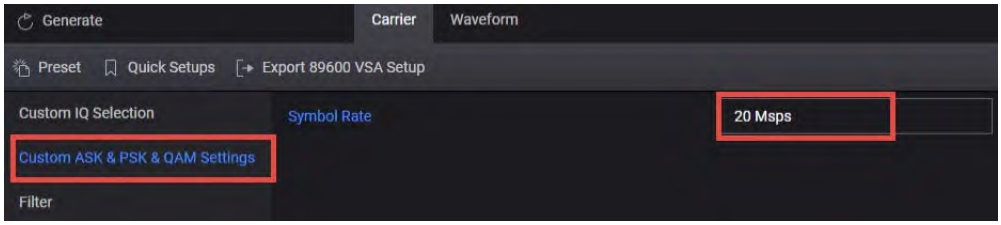
4. Select **Custom Modulation** to enter Custom Modulation Signal Mode.



5. In the Custom Modulation setup, select the **Carrier** tab > **Quick Setups** > **DVB-S2X** > **256APSK** > **256APSK 128/180**.



6. From the left pane, select **Custom ASK & PSK & QAM Settings** and set the Symbol Rate to **20 Msps** per channel.



7. Select **Filter** > Alpha to **0.25**.

   Filter length is 32 symbols, which we will set this value in the VSA.

Configure the PLHEADER (Preamble)

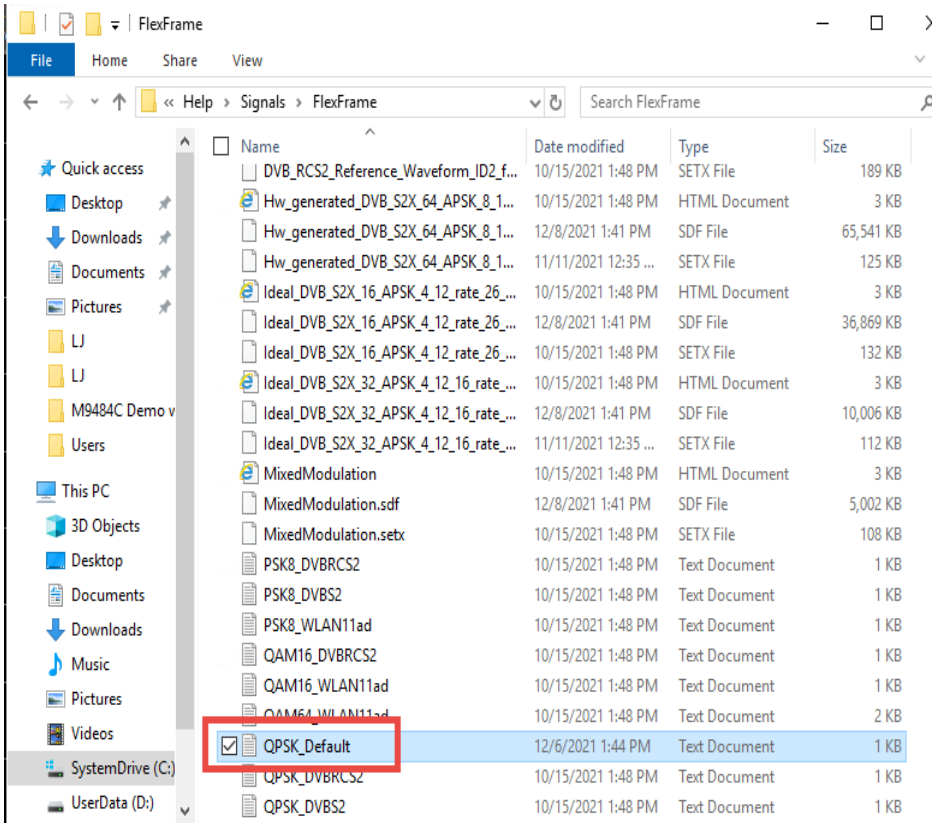
In Part 1, the PLHEADER using π/2-BPSK modulation is split into two: SOF (Start of Frame) of 26 symbols and PLS (PHY Layer Signaling) of 64 symbols for a total of 90 symbols. The N7608C by default uses a single segment for PLHEADER with 90 symbols using QPSK modulation and provides a payload of

180 symbols. For the purpose of this example, we will merge the two. You can split the PLHEADER into two by adding a segment and provide a payload of 52 symbols for the SOF. Since N7608C doesn't support π/2-BPSK modulation, it uses QPSK so it would require a payload of 52 symbols for the SOF, instead of the 26-symbols used in Part 1.

NOTE

Default constellation symbol mappings of VSA Flex Frame and N7608C/APPC do not match. VSA provides .txt files to load into the N7608C/APPC to match the mapping. These .txt files are in the VSA installer under:

C:\Program Files\Keysight\<version>\89600 VSA Software\Help\Signals\FlexFrame.



NOTE

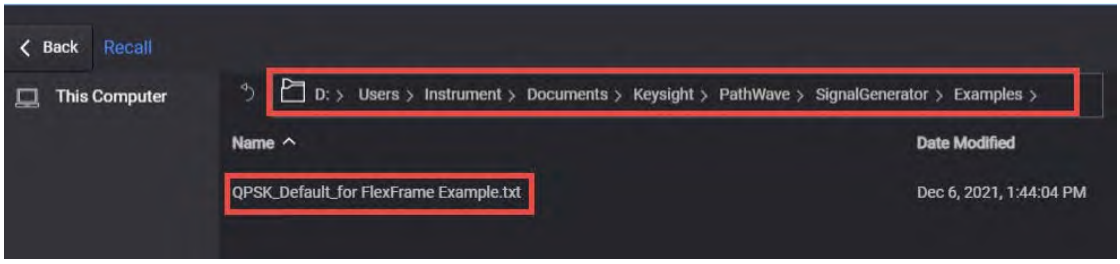
For the convenience of this demonstration, the QPSK text file has already been copied onto the MXG/VXG.

`D:\Users\Instrument\Documents\Keysight\PathWave \SignalGenerator\Examples\QPSK_Default_for FlexFrame Example`

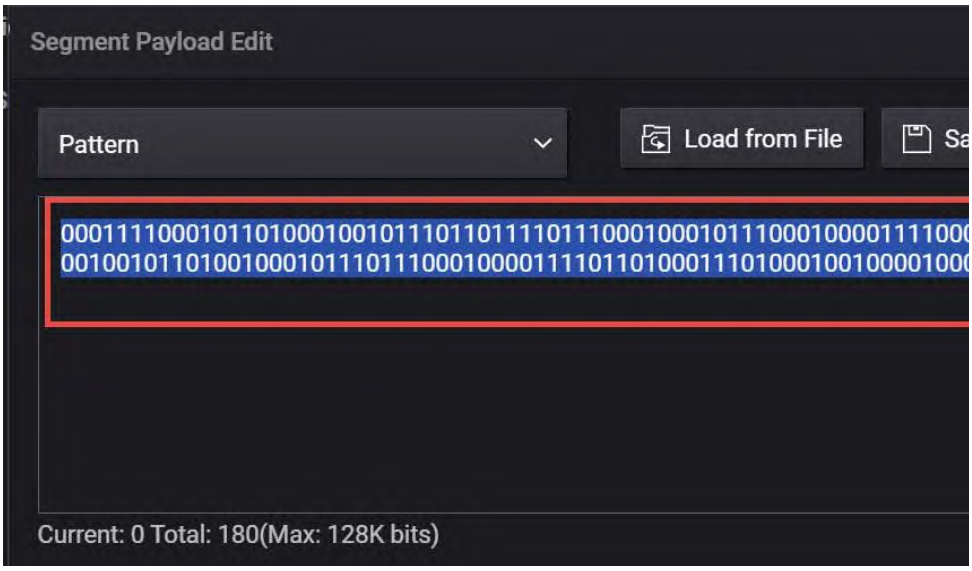
8. Select the **Segments** tab.

9. In the Constellation Edit dialog, select **Recall**.

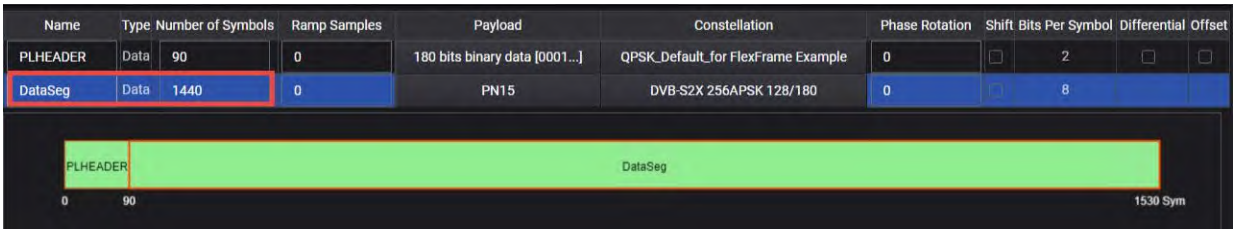
10. Navigate to the text file and choose **Select**. Select **OK** to Close the Constellation Edit dialog.



11. Set Payload Data to **180 bits binary data [0001...]** to open the Segment Payload Edit dialog. Select and copy the 180 symbols. These will be pasted in to the VSA for synchronization.



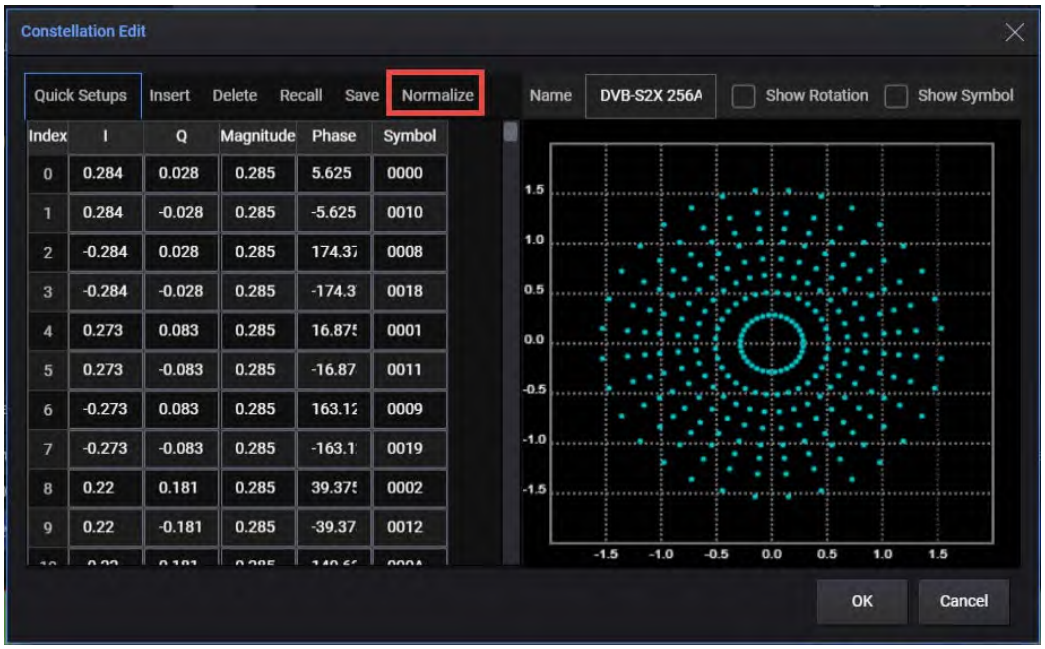
12. To configure the Payload, set the Number of Symbols to **1440**.



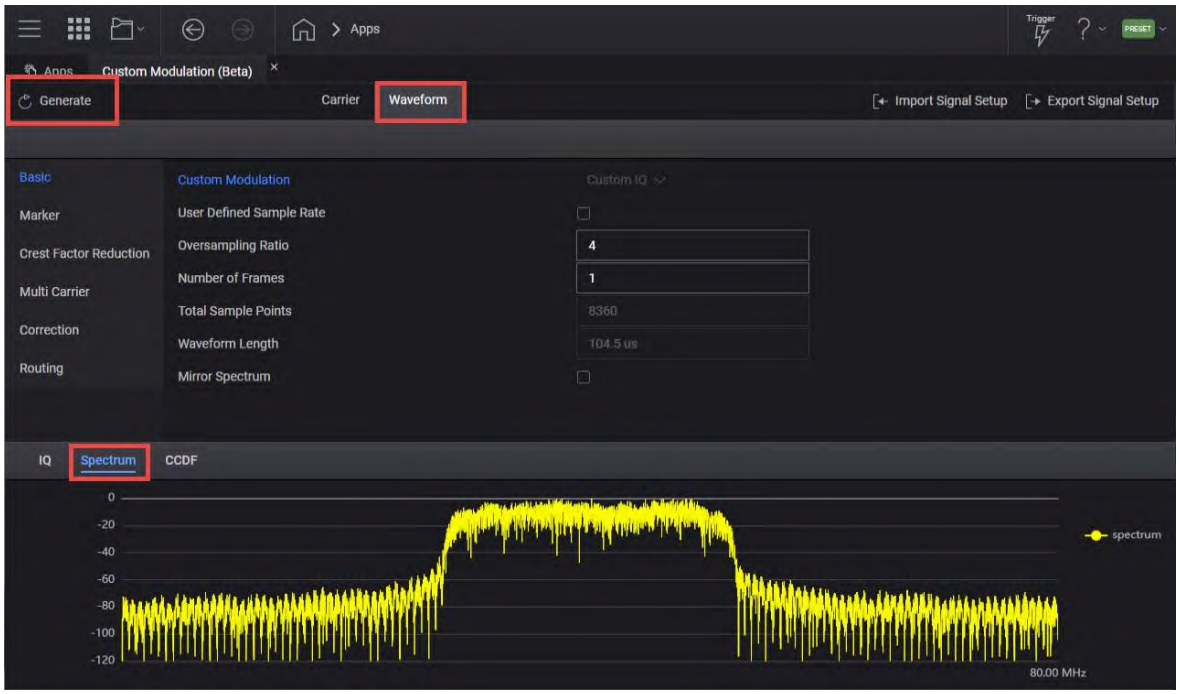
Use 16 payload slots (90 symbols/slot) for the radio frame. These are the slots prior to Pilot. Pilots are optional and not generated (See part 1 of this demo guide for a frame with pilot segment).

13. Select Segment 1. In the DataSeg row, under the Constellation area, select **DVB-S2X256APSK 128/180** to open the Constellation editor, and then select **Normalize**.

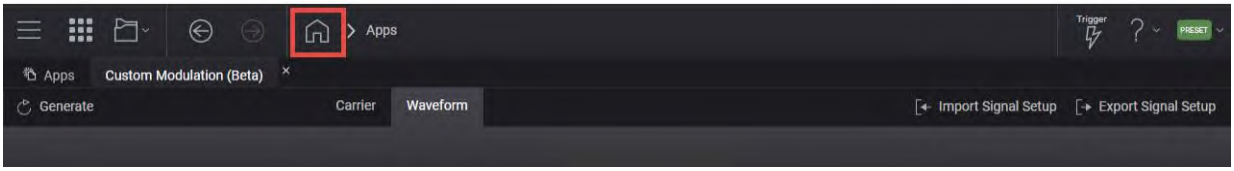
It is a good idea to normalize the power. Difference power levels between data and preamble can result in high EVM.



14. Close the Constellation editor and select **Generate** to generate the waveform. To view the Generated waveform, select the **Waveform** tab and select **Spectrum**.



15. Select **Home** to exit the setup panel.



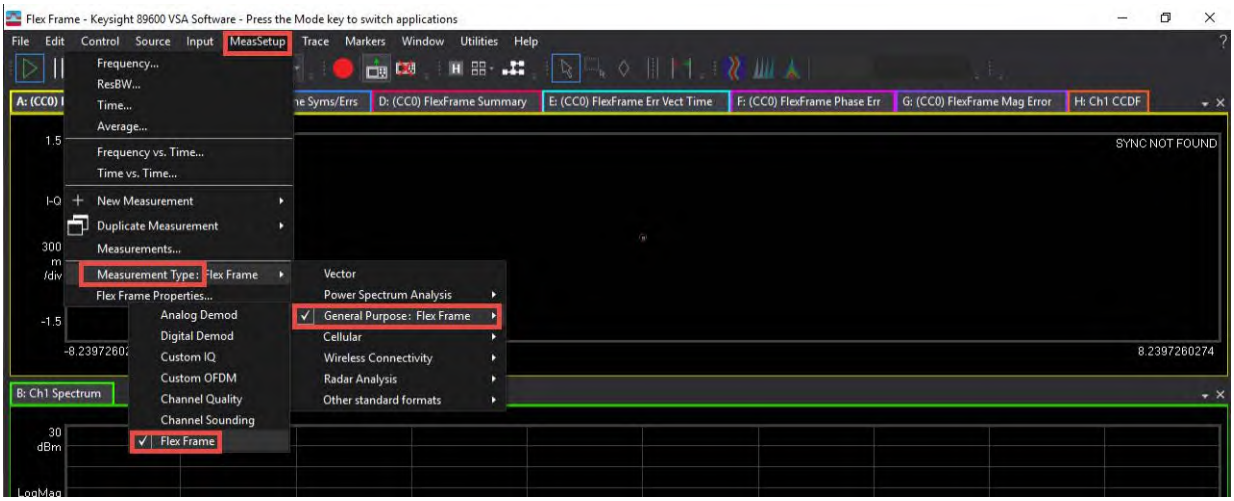
16. Set RF Out to **On** by selecting the numbered channel indicator switch.

This enables the RF Out for the indicated channels, in this case Channel 1 if using a multi-channel MXG.

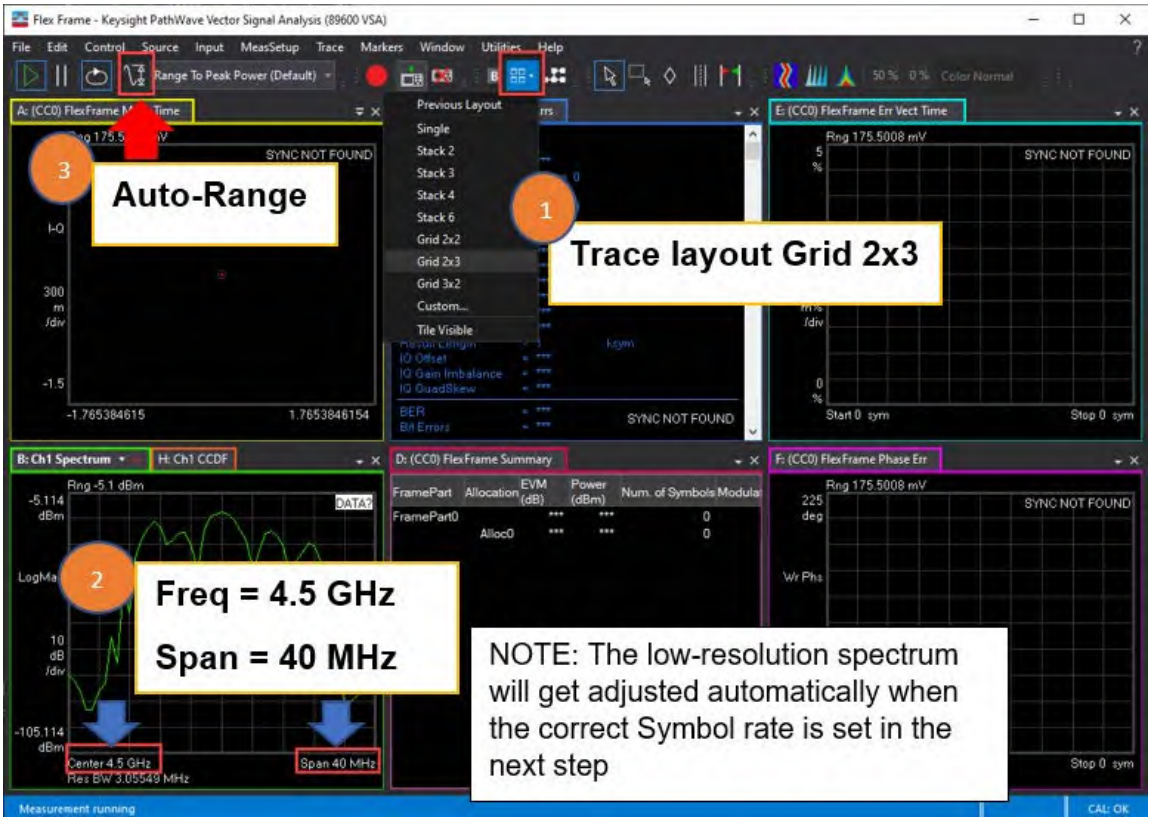
Setting up and Analyzing the Signal Using VSA Flex Frame

NOTE Changing settings on the VSA can take longer to take affect than expected. Wait to change the next setting until the previous setting has been updated.

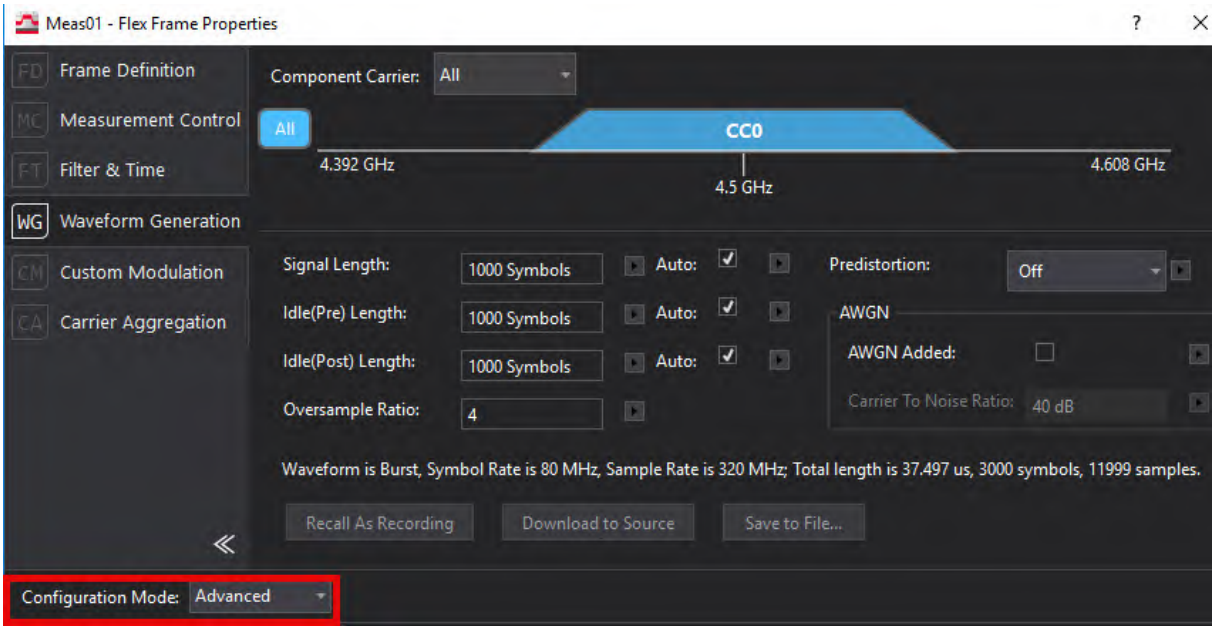
1. From the VSA menu bar, select **MeasSetup** > **General Purpose** > **Flex Frame**.



2. Set the correct frequency, span, and input range. For this example, set:
   - Change the Trace Layout Grid to 2 x 3.
   - Change Trace B to **Ch1 Spectrum**, and set Center Frequency to **4.5 GHz**
   - Span to **40 MHz**
   - Autorange



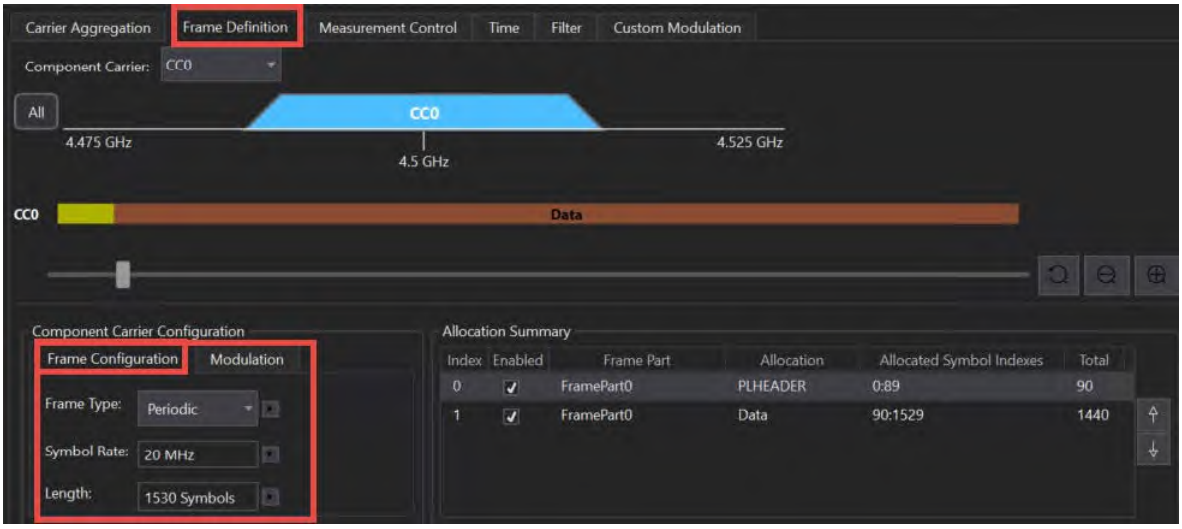
3. Select **MeasSetup** > **Flex Frame Properties** > **Configuration Mode to Advanced.**



4. In the Flex Frame Properties window, select the **Frame Definition** tab and set:
   - Frame Type to **Periodic**
   - Symbol Rate to **20 MHz**
   - Length to **1530 Symbols**

     This is 90 symbols of PLHEADER plus 1430 symbols of data.

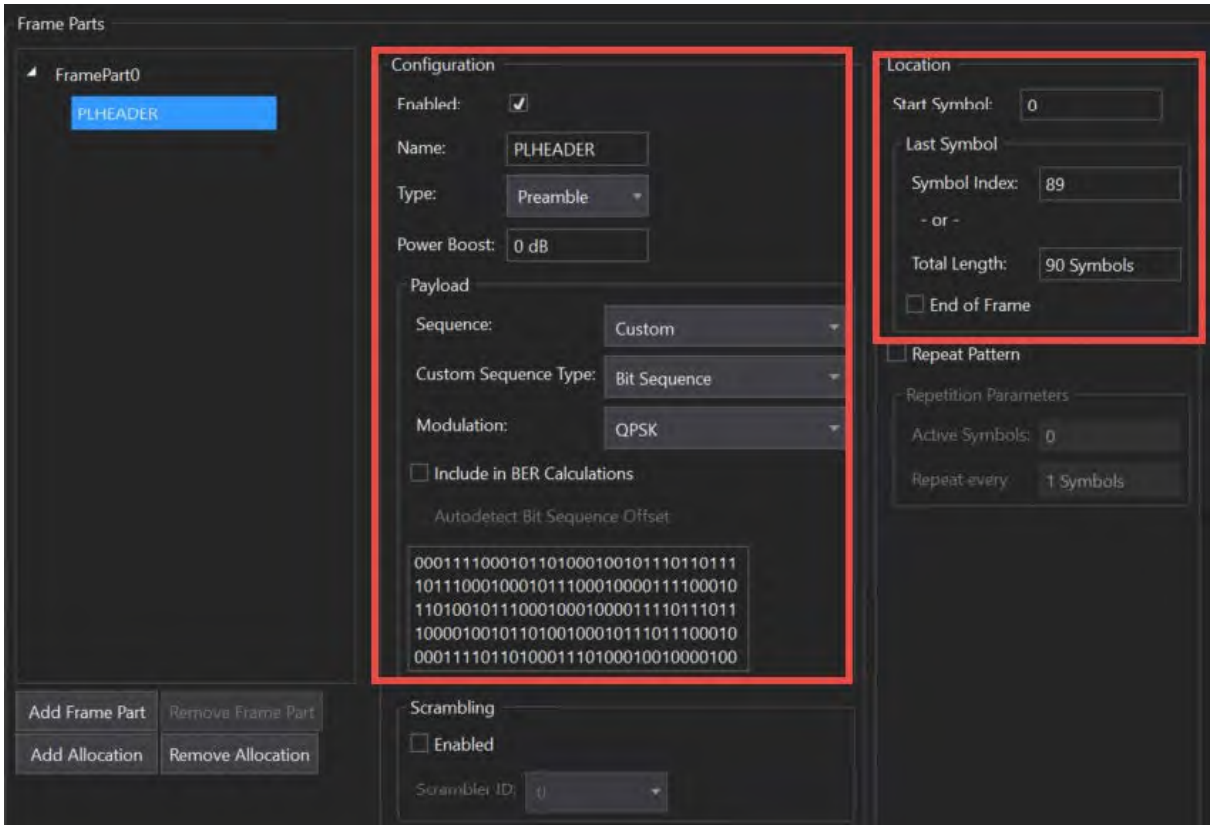
For a bursted signal, Frame Type is set to Burst, and the frame length can be automatically set based on the detected burst length.



Configure the Preamble (PLHEADER)

5. Configure the Start of Frame Index (Preamble), 90 symbols long:
   - In the Configuration area, set
     - Name to **PLHEADER**
     - Type to **Preamble**
     - Sequence to **Custom**
     - Modulation to **QPSK**
     - Enter the 180 symbols you copied over from the Custom Modulation PLHEADER Payload setup in step 11.
   - In the Location area,
     - Clear the End of Frame checkbox
     - Start Symbol to **0**

– Total Length to **90**

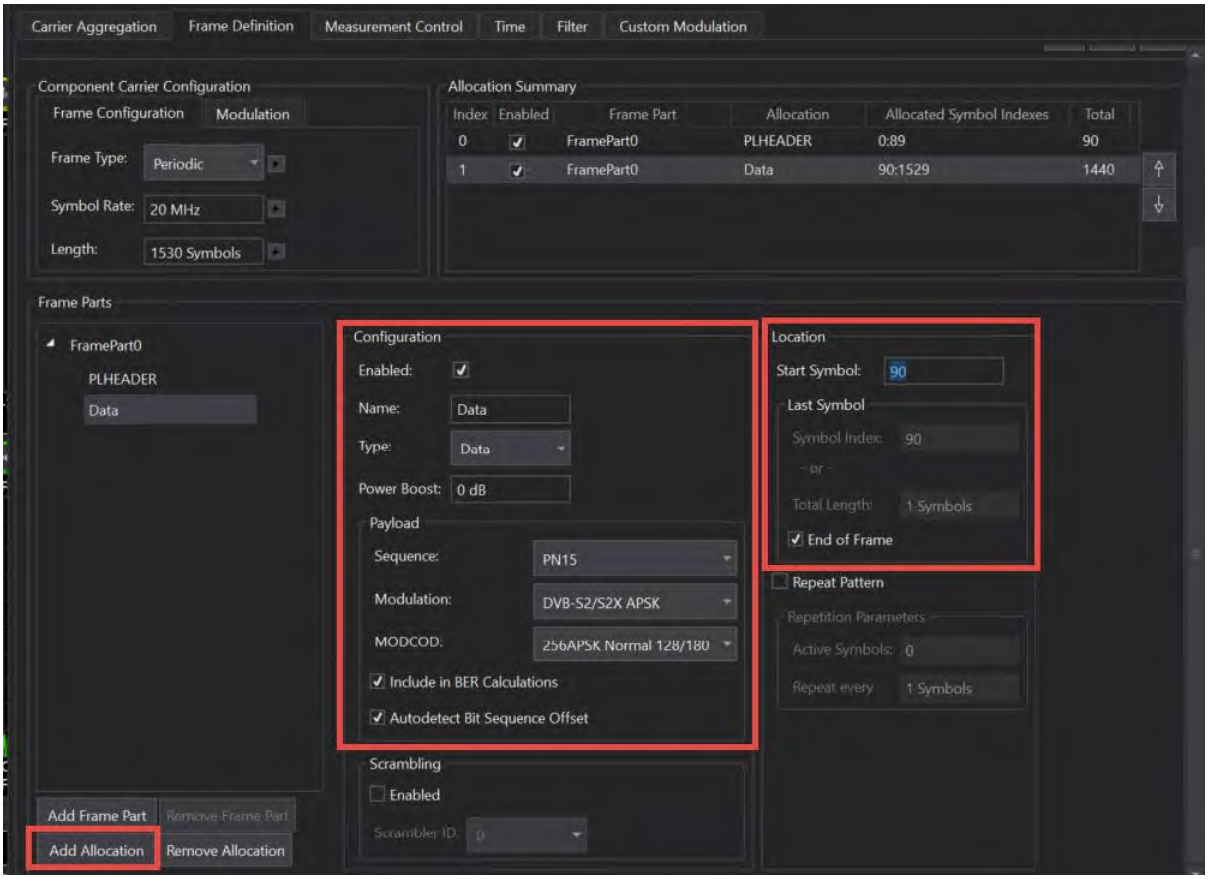


6. Configure the Payload 1440 symbols long:
   – In the Frame Parts area:
     – Select **Add Allocation**
7. In the Configuration area, set:
   – Name to **Data**
   – Type to **Data**
   – Sequence to **PN15**
   – Modulation to **DVB-S2/S2X APSK**
   – MODCOD to **256APSK Normal 128/180**
   – Select **Include in BER Calculations**
   – Select **Autodetect Bit Sequence Offset**

TIP **Include in BER Calculations**: BER is supported if the allocation's payload sequence is either a PN type or a Custom bit sequence.

Autodetect Bit Sequence Offset: Supported when the segment(s) is not used for synchronization.
If not selected (unchecked), demodulator assumes the bit sequence starts at the beginning of the allocation.
If selected, demodulator finds the best match to the bit sequence.

8. In the Location area, set:
   - Start Symbol to **90**
   - Last Symbol to **End of Frame**

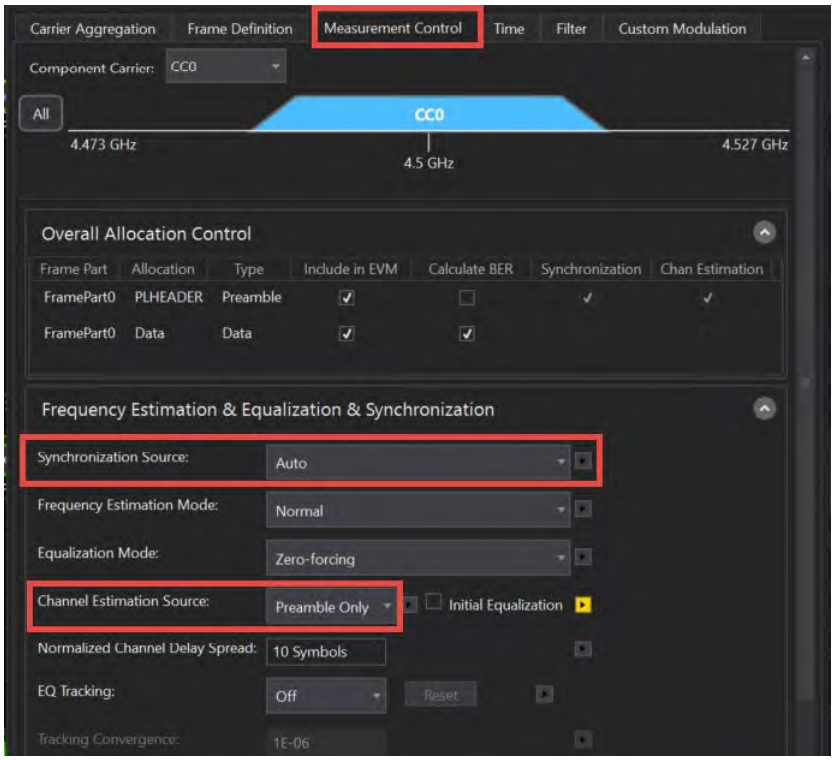


Configure Synchronization and Equalization

9. Select the **Measurement Control** tab, and set:
   - Synchronization Source to **Auto**

     By default, PLHEADER is used for synchronization. This can be changed to "Customized" to also include the Data segment for synchronization. Calculated BER must be cleared (unchecked) from Data before selecting if for Synchronization.
   - Channel Estimation Source to **Preamble Only**

Equalization is off by default. You can choose the different frame parts for channel estimation and equalization.



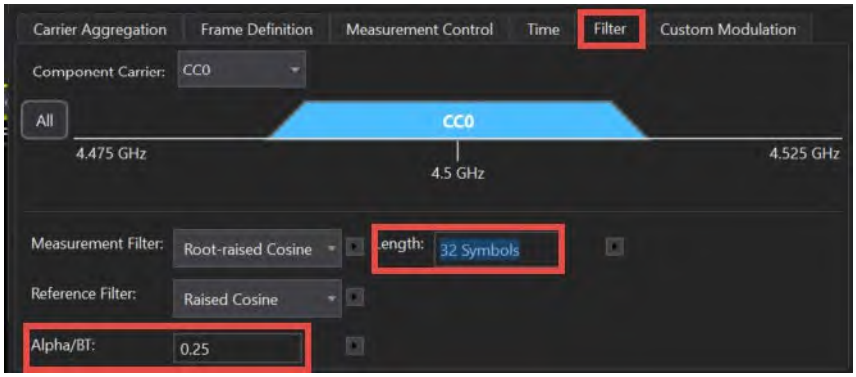
Configure the Analysis Region

10. Select the **Filter and Time** tab, set the Result Length to **1530 Symbols**.

90 for the preamble plus 1440 for the data.

Configure the RRC filter

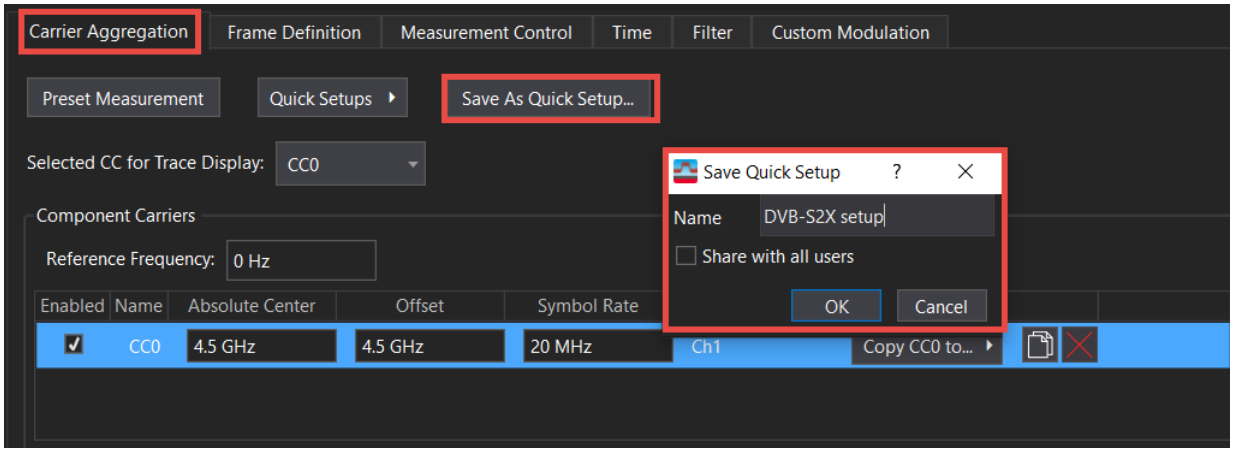
11. In the **Filter and Time** tab, set Alpha/BT: to **0.25** and Length to **32 Symbols**.



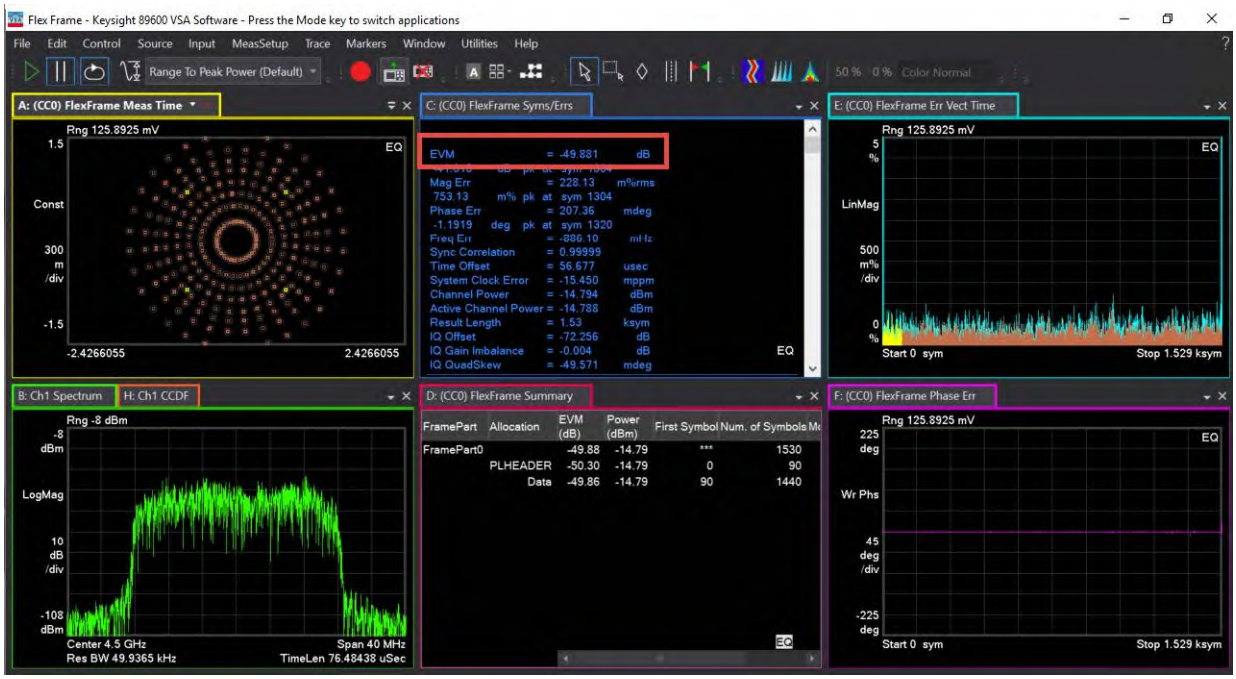
Save the Configuration as a Quick Setup

12. In the **Carrier Aggregation** tab, select **Save As Quick Setup**.

This saves the measurement settings for future use. Select **Share with all users** if you want to share with other users on the PC running the VSA software.

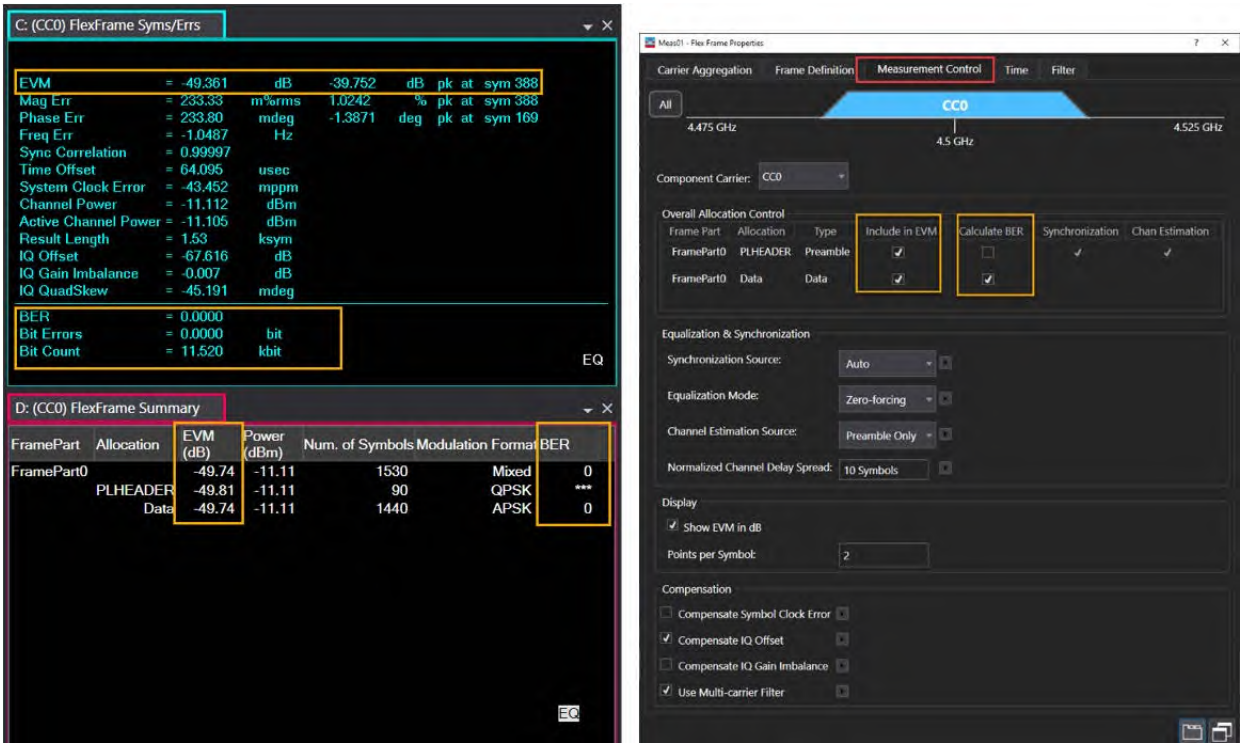


13. Close the Properties window and change Trace A to a Constellation Diagram, then view the results.



You should see a successful demodulation with very low EVM. Multiple traces and tables are available including Error Vector Spectrum, Error Vector Time, demod, and BER Bits.

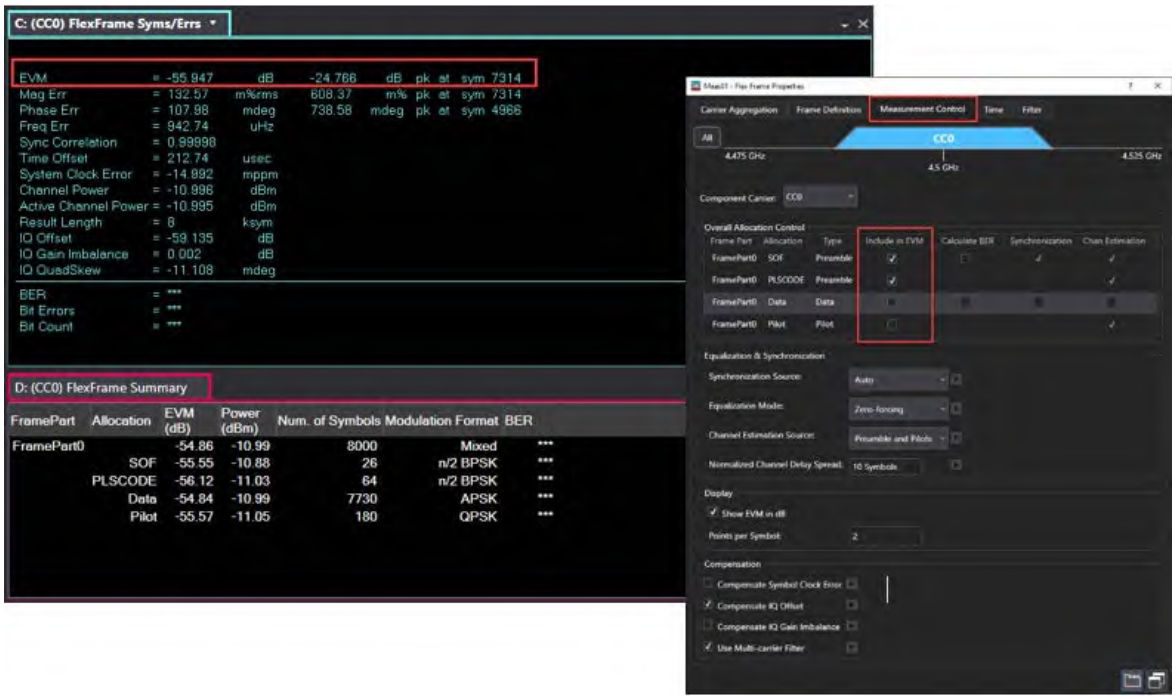
The FlexFrame Summary table always displays EVM of each active segment. However, "Flex Frame Syms/Errs" Table shows composite EVM of segment(s) selected under Measurement Control -> Include EVM. By default, it is all segments.



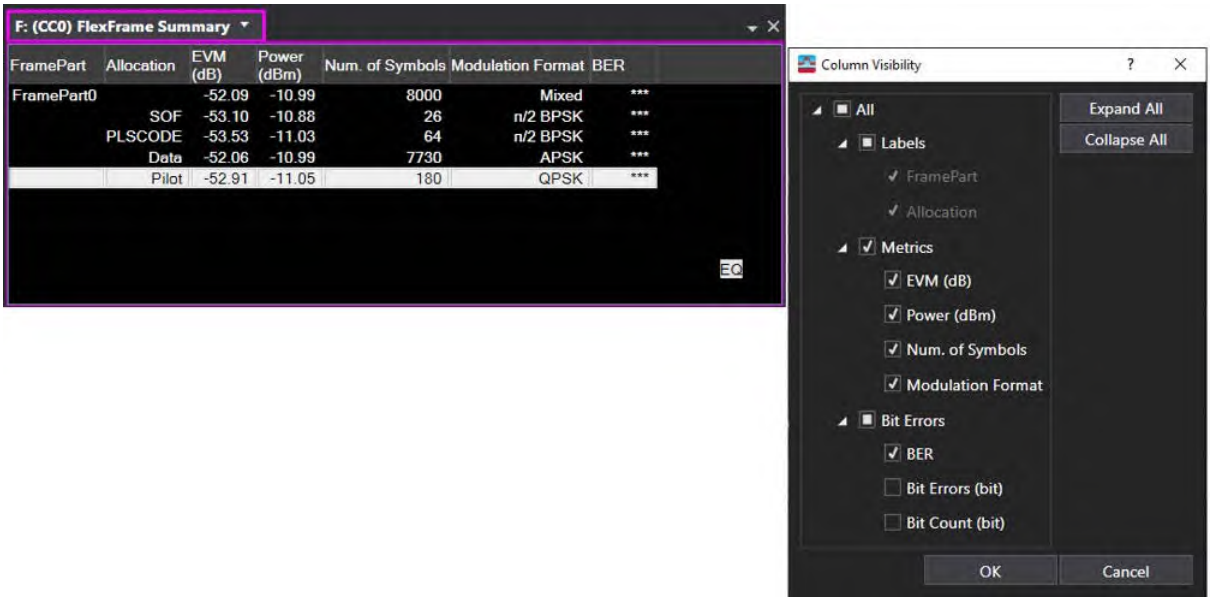
TxBER/RawBER is returned for the Data segment. NOTE: BER is supported if the allocation's payload sequence is either a PN type or a Custom bit sequence. Reference and measured bits are returned.

- BER is expressed as a ratio and calculated as Bit Errors/Bit Count.
- Turn On averaging to accumulate bit count.
- Recording with 0% overlap gives gap free BER data (limited by memory).

FlexFrame Summary always displays the EVM of each active segment. Flex Frame Syms/Errs table shows composite EVM of segment(s) selected under Measurement Control -> Include EVM. By default, it shows all segments.



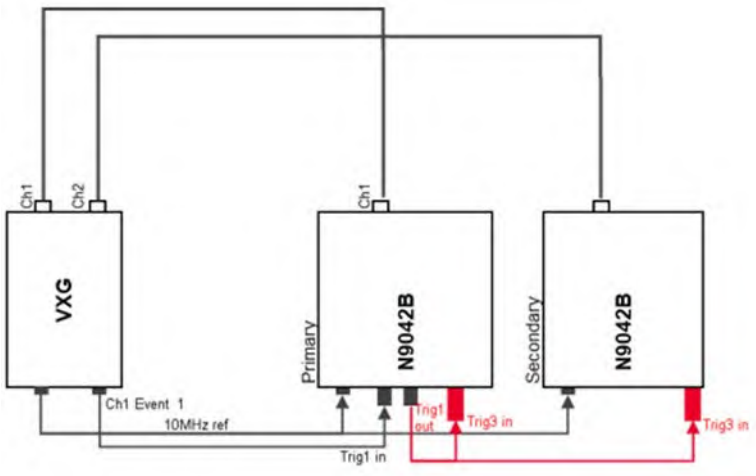
FlexFrame Summary table is a type of Matrix Table that provides sortable rows by column, selectable column visibility, and copy/paste and export functionality to share rows of content or complete tables to applications like email, text editing or spreadsheet programs. Right click the column header to display the menu.



Creating & Analyzing a 5G NR DL 2x2 MIMO Measurement Using Two UXAs

We will create a 5G NR MIMO signal using a VXG with PathWave Signal Generation, and then analyze it with using both the X-Series 5G NR measurement application and the 89600 VSA software.

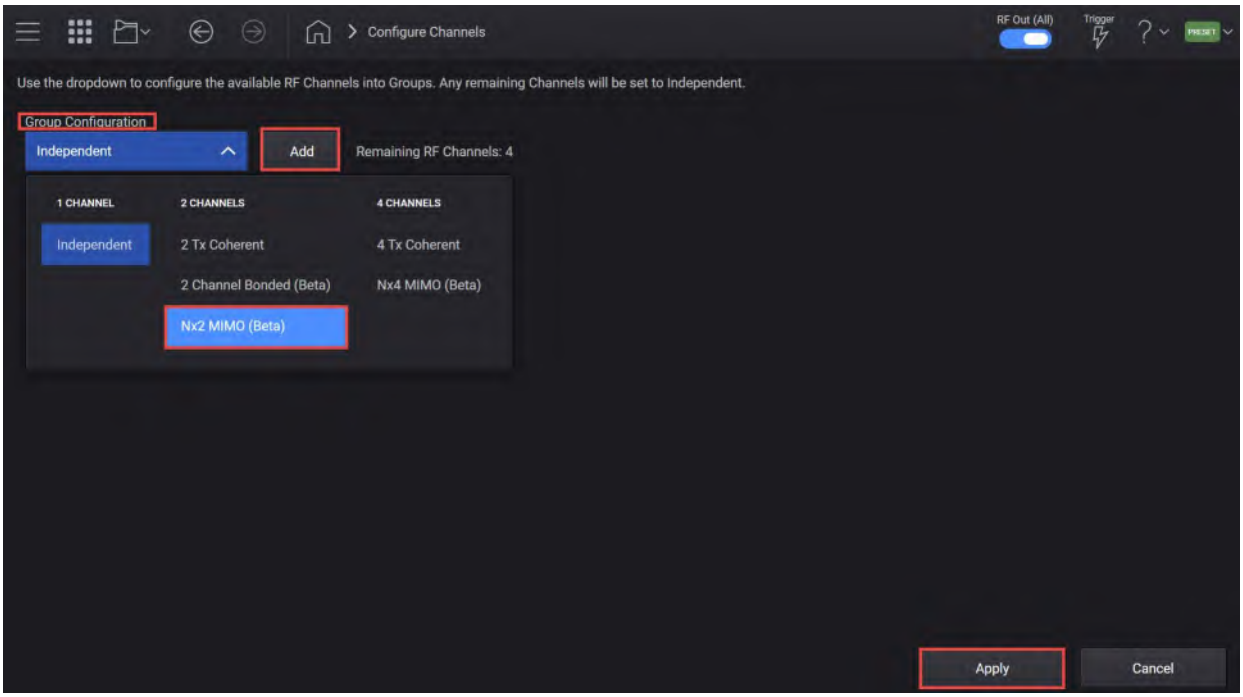
The hardware setup includes a 2-channel VXG and two N9042B UXAs.



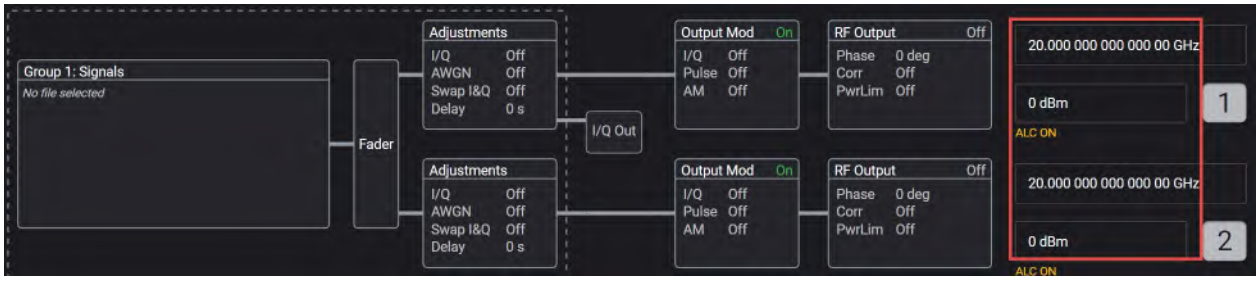
M9484C VXG	N9042B UXA
Required Options – RF bandwidth of 160 MHz or above – Two-channel configuration	Required Options – UXA 1 N9042MMOB Multi-box Synchronization for MIMO, Analysis Bandwidth of 160 MHz or above – UXA 2 Analysis Bandwidth of 160 MHz or above
Required Software – M9484C Firmware version A.11.01 or higher – 7631APPC PathWave Signal Generation for 5G NR	Recommended Software – N9042B UXA Firmware version 2023 (XA34) or higher – N9085EMOE 5G NR Measurement application – 89600 VSA 89601200C for VSA Base Core 89601BHNC for VSA 5G NR

On the VXG

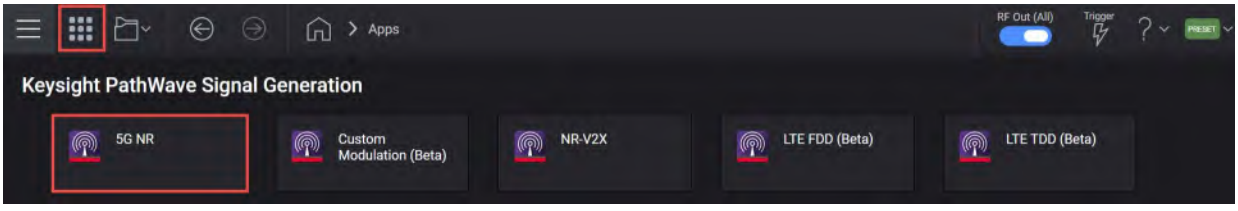
1. Connect the equipment as shown above.
2. Select **Preset** > **Preset** to set the VXG to a known state.
3. Select the System menu (triple bar icon at the top left of the window), and then select **Configure Channels** > **Nx2 <MIMO** > **Add** > **Apply** > **Apply**.



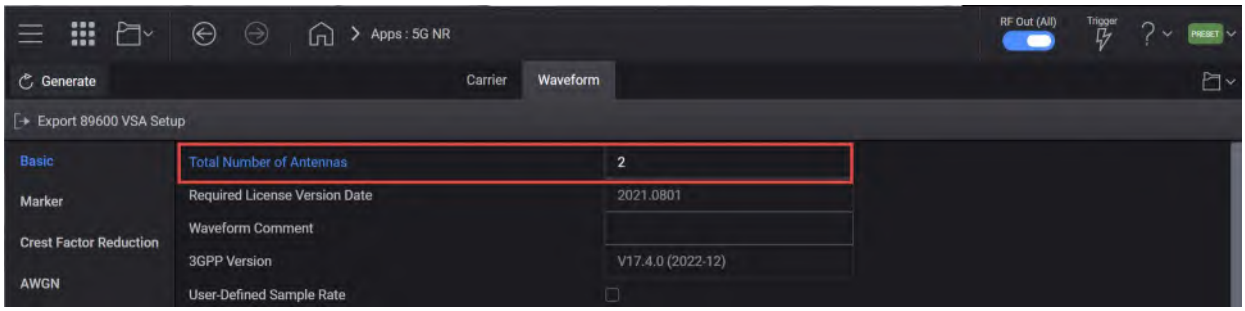
4. Set the Frequency to **20 GHz** and Power to **0 dBm**.



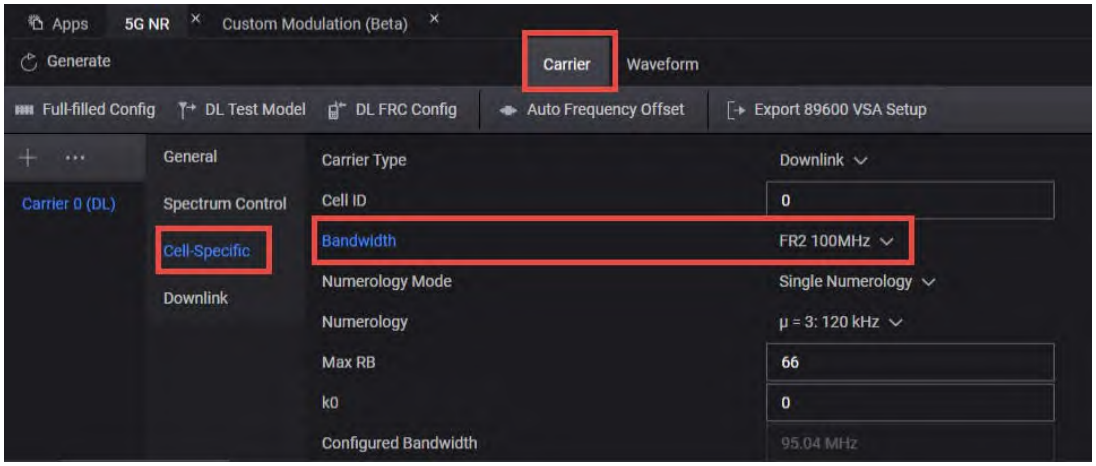
5. Select the **Apps** block to open, then select **5G NR**.



6. Select the **Waveform** tab and set the Total Number of Antennas **2**. This means that two antenna port signals will be generated.

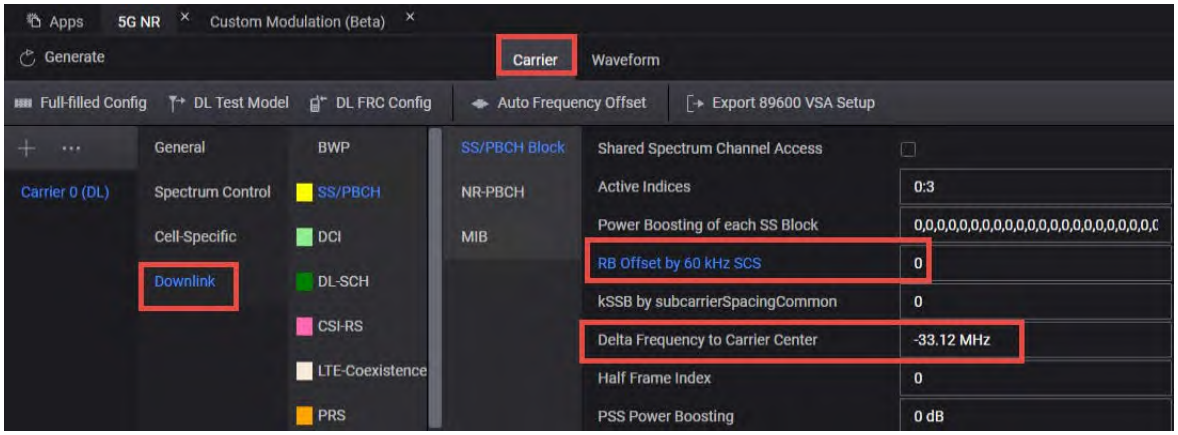


7. Select the **Carriers** tab > **Cell Specific** node, and set Bandwidth to **FR2 100 MHz**. Keep the rest of the default settings.



8. Select the **Downlink** node > **SS/PBCH** > **RB Offset by 60 kHz SCS** and set to **0**.

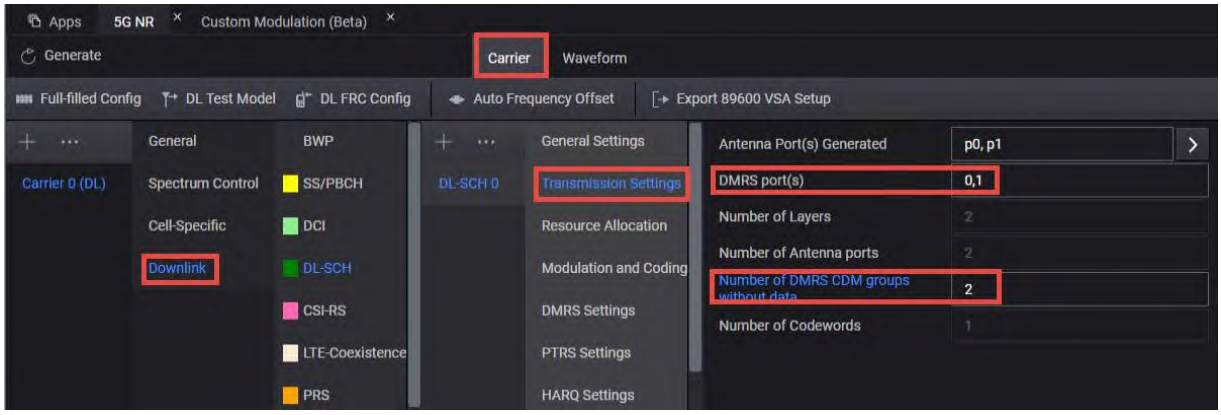
This automatically changes the Delta Frequency to Carrier Center to -33.12 MHz. Using 0 RB Offset puts the SSB at the lower edge of the carrier.



9. Select the **Downlink** node > **DL-SCH**> **Transmission Settings** and confirm DMRS ports is set to **0,1**. Set Number of DMRS CDM groups without data to **2**.

   Number of DMRS CDM groups without data is used to determine how many resource elements are reserved for DMRS. When this parameter is changed, the DMRS power boosting is automatically coupled based on table 4.1-1 of 38.214.

   You will see the layer number is updated to 2 and each layer is assigned with a particular DMRS port. You'll also see that Antenna Ports Generated is automatically set to P0,P1, which will map the multiple antenna port signals to different antennas (instruments).



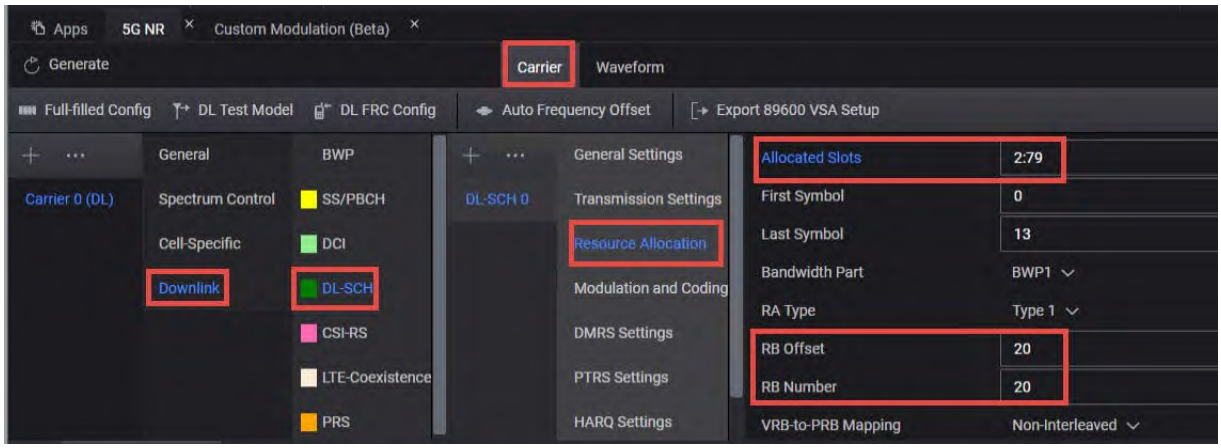
10. Select the **Resource Allocation** node and set:

    – Allocated Slots to **2:79**

    – RB Offset to **20**

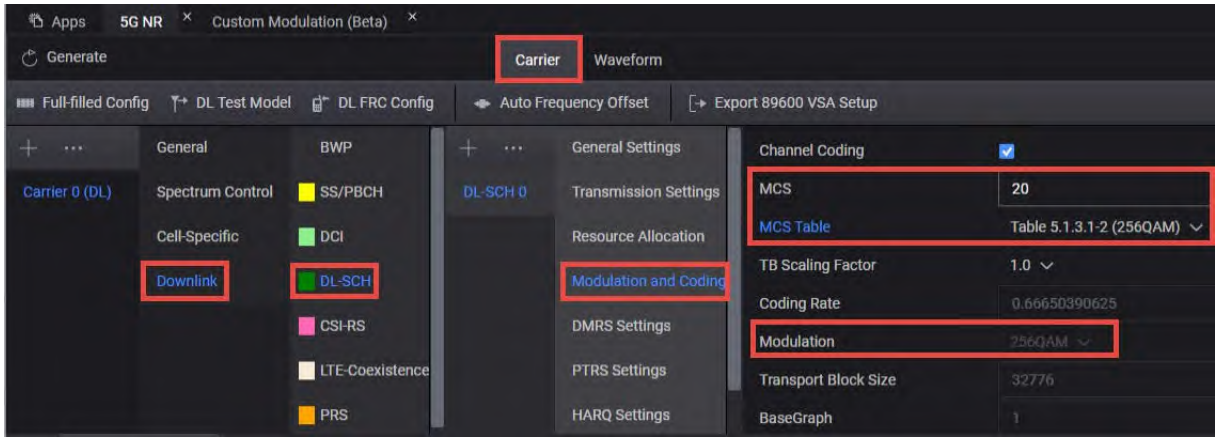
    – RB Number to **20**

DL-SCH0 will occupy the RBs 20-40 and DL-SCH1 will occupy the rest of the resources.



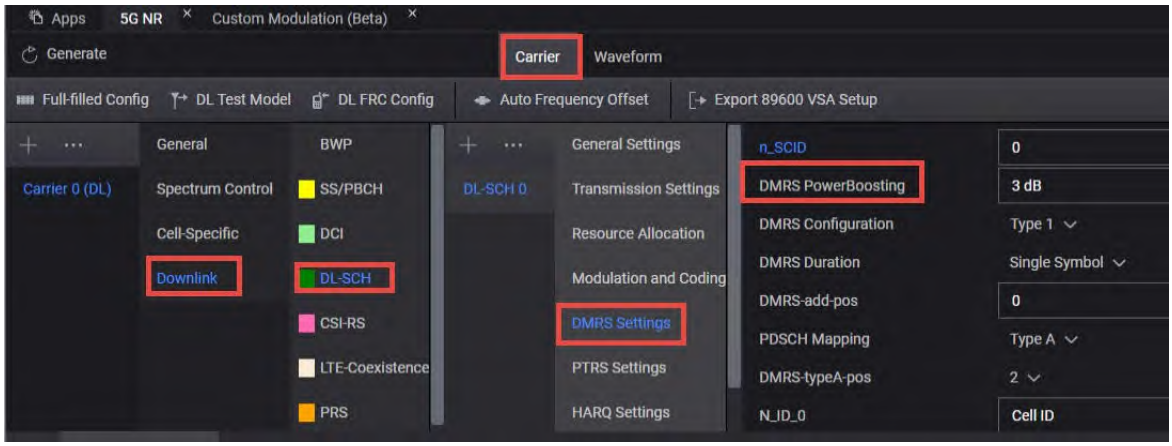
11. Select the **Modulation and Coding** node and set:
    - MCS to **20**
    - MCS Table to **5.1.3.1-2 (256 QAM)**

Notice that modulation is updated to 256 QAM.

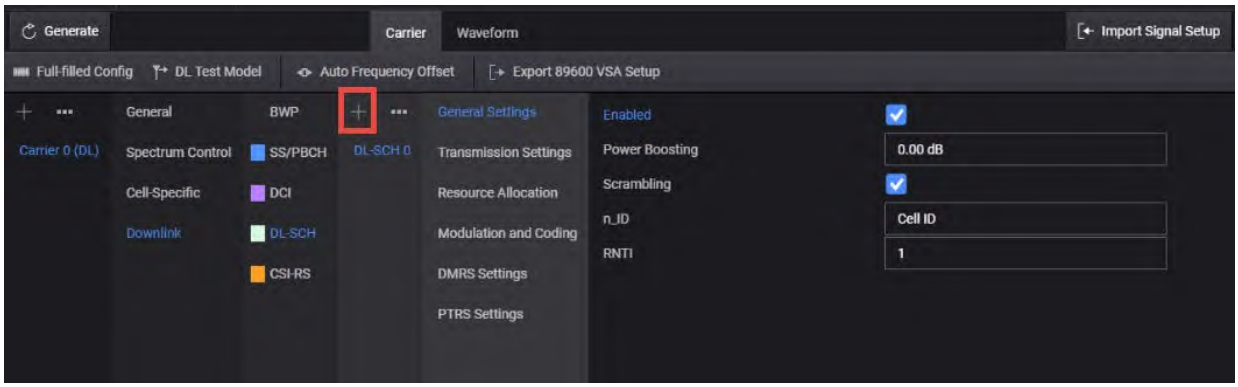


12. Select the **DMRS Settings** node and set DMRS Power Boosting to **3 dB**.

When CDM Group Number is 2, DMRS power boosting becomes 3 dB per 3GPP.

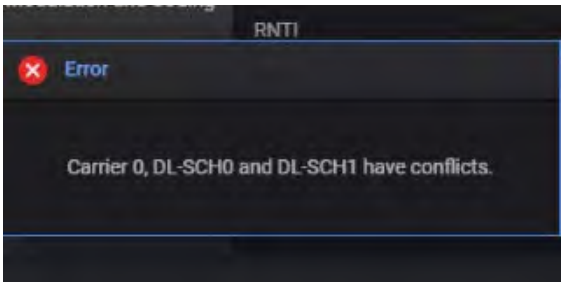


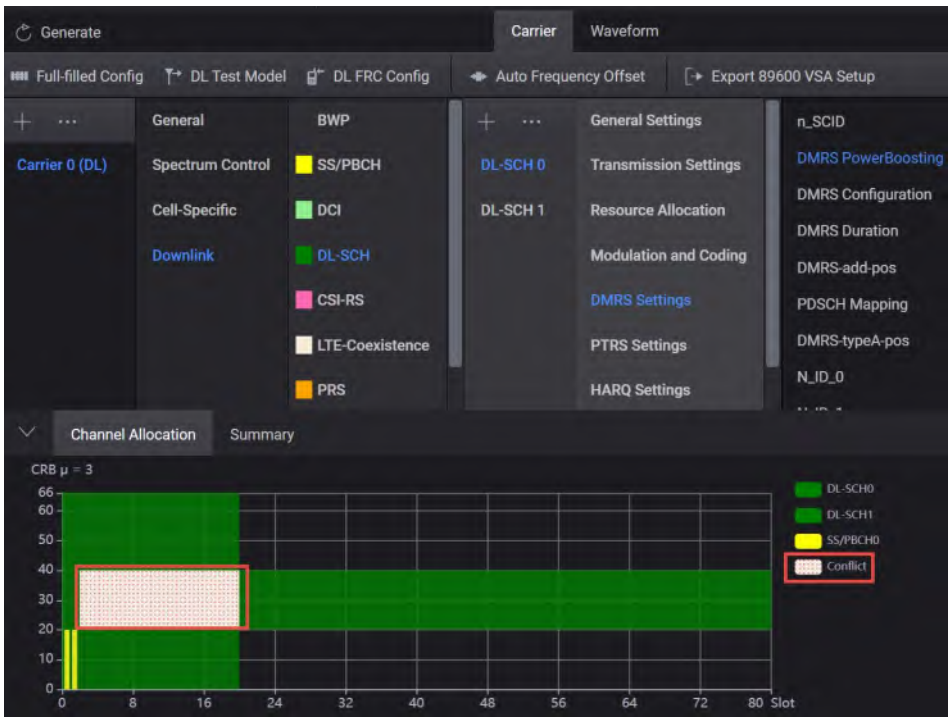
13. Under the DL-SCH 0 node, select the + icon to add a new **+DL-SCH** (DL-SCH1).



NOTE

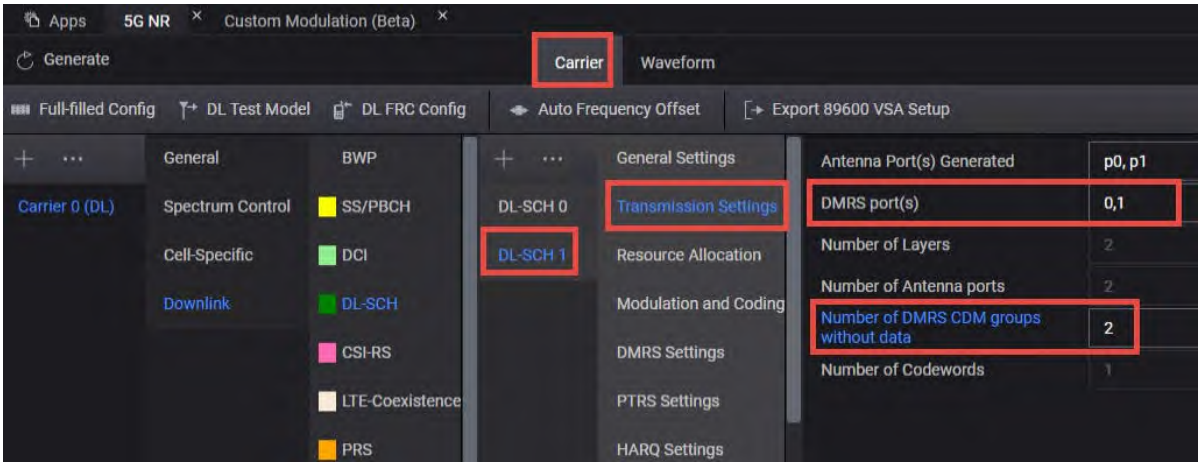
This will create a conflict between DL-SCH0 and DL-SCH1. This will be resolved in a later step.





14. From the Channels node, select **DL-SCH** > **DL-SCH1** > **Transmission Settings** and set DMRS port(s) to **0,1.** Change the Number of DMRS CDM groups without data to **2**.

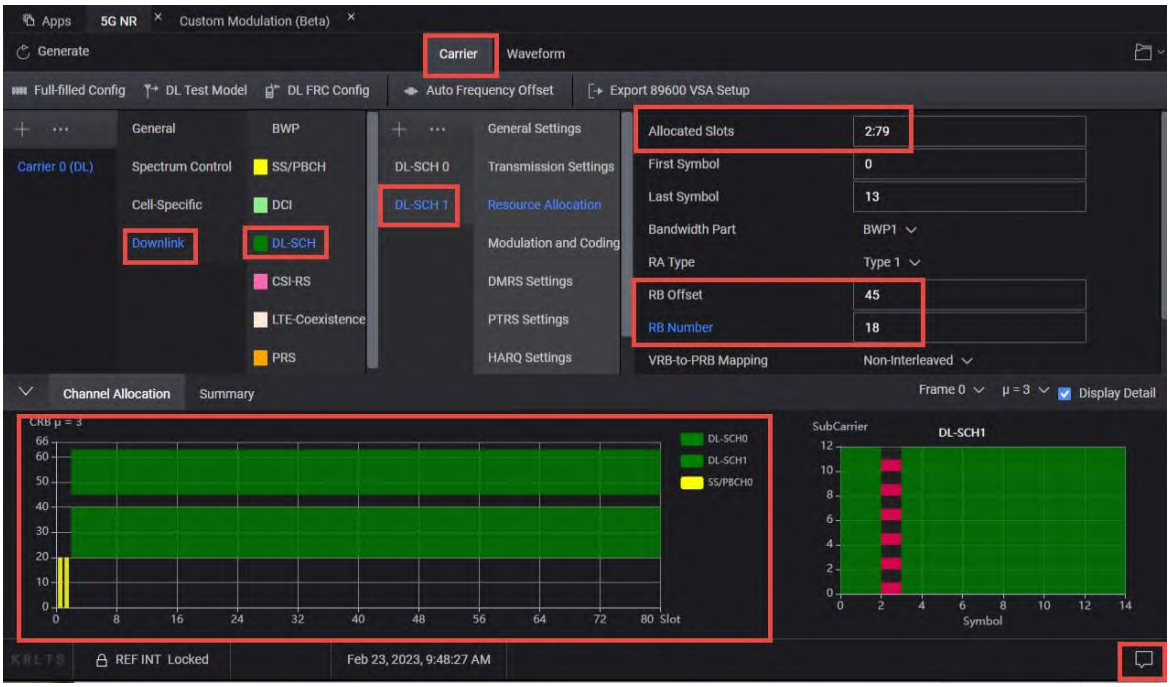
Notice that Number of Layers is updated to 2 and each layer is assigned with a particular DMRS port. Antenna Ports Generated is automatically assigned P0,P1, which will map generated multiple antenna port signals to different antennas (instruments).



15. Select the **Resource Allocation** node and set:

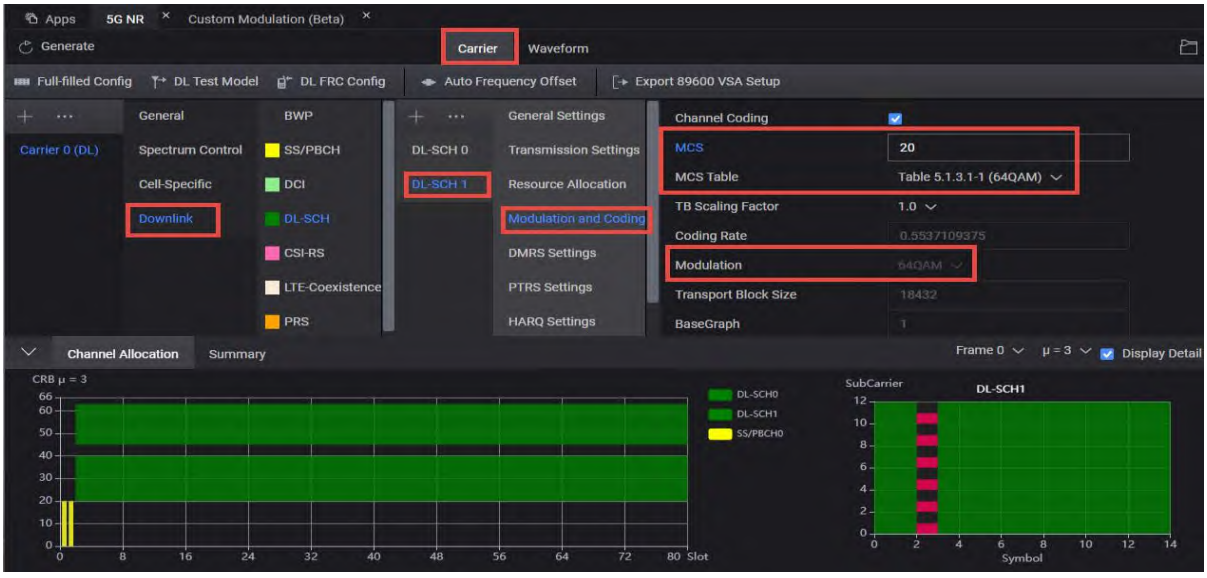
- Allocated Slots to **2:79**
- RB Offset to **45**
- RB Number to **18**

Notice that this resolves the conflict between DL-SCH channels. (The pink shading is removed from the Channel Allocation graph. To clear the conflict error message at the bottom of the display, select the Message icon (bottom, right corner) and select **Clear**.



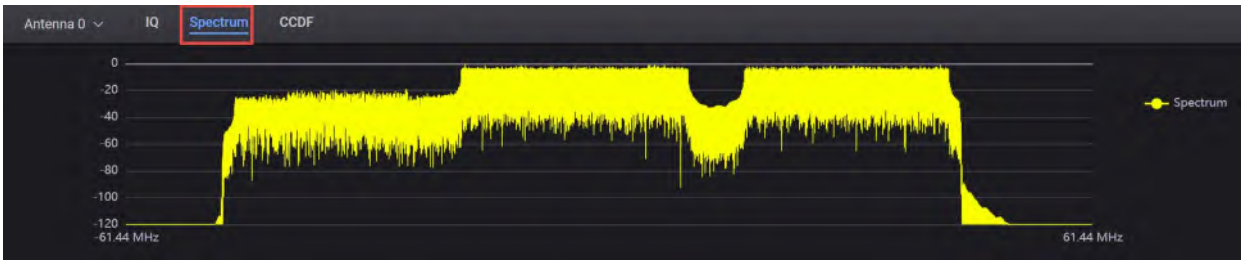
16. Select the **Modulation and Coding** node, and set MCS to **20**.

Use the default MCS Table to Table 5.1.3.1-1 (64QAM). You will see the modulation is updated to 64QAM.



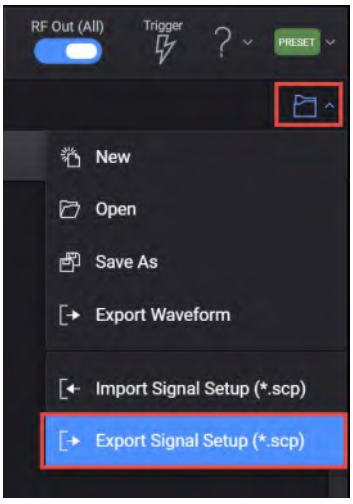
17. Select the **Waveform** tab > Select **Generate**.

    View the waveform by selecting Spectrum on the bottom display.



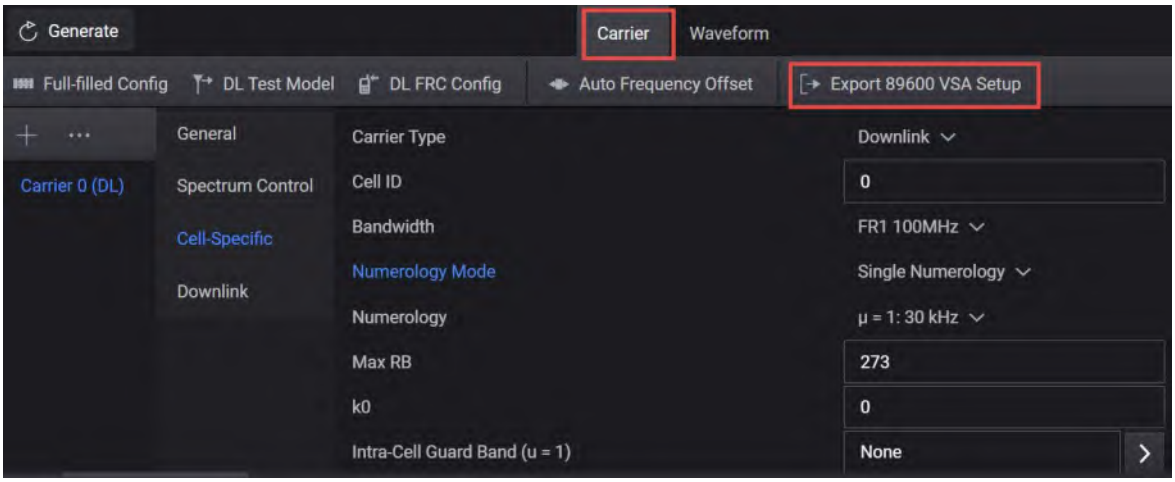
18. Select the **File** icon on the top right and select Export Signal Setup (*.scp), and save the file.

    You will need to copy this setup file to the UXA to speed up demodulation on the UXA using X-Apps.

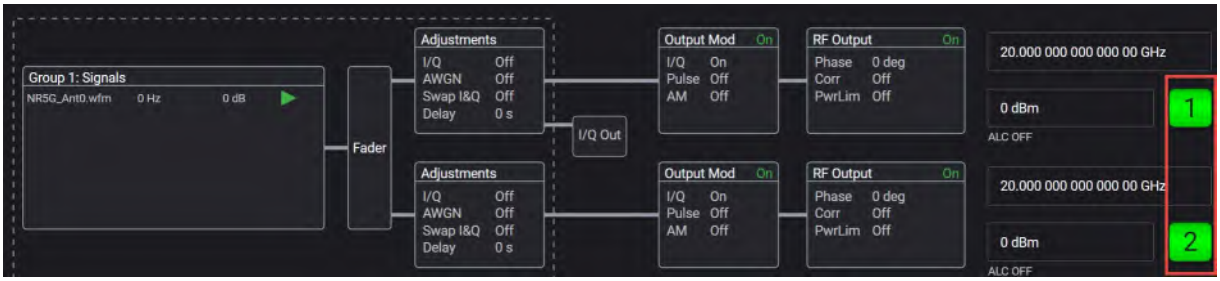


19. Select the **Carrier** tab and select **Export 89600 VSA Setup** and save the file.

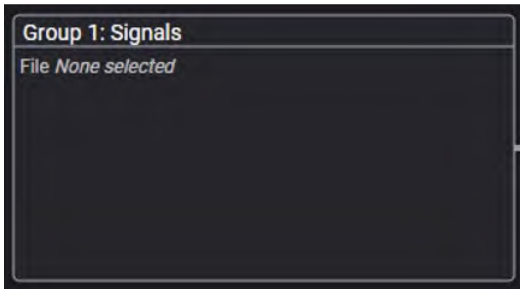
You will need to copy this setup file to the UXA to speed up demodulation on the UXA using the 89600 VSA software.



20. Select the Home icon and set RF Out to **On** by selecting the numbered channel indicator switches, and then select **RF Out (All)** master control switch to turn the RF output On for both channels or off for both channels.

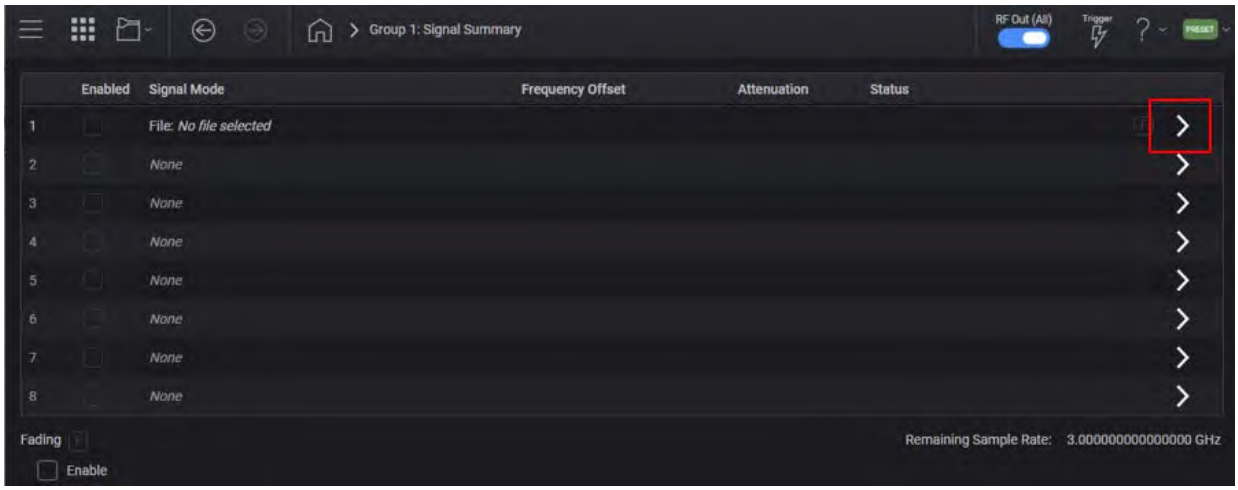


21. Select the **Signal** block to open.

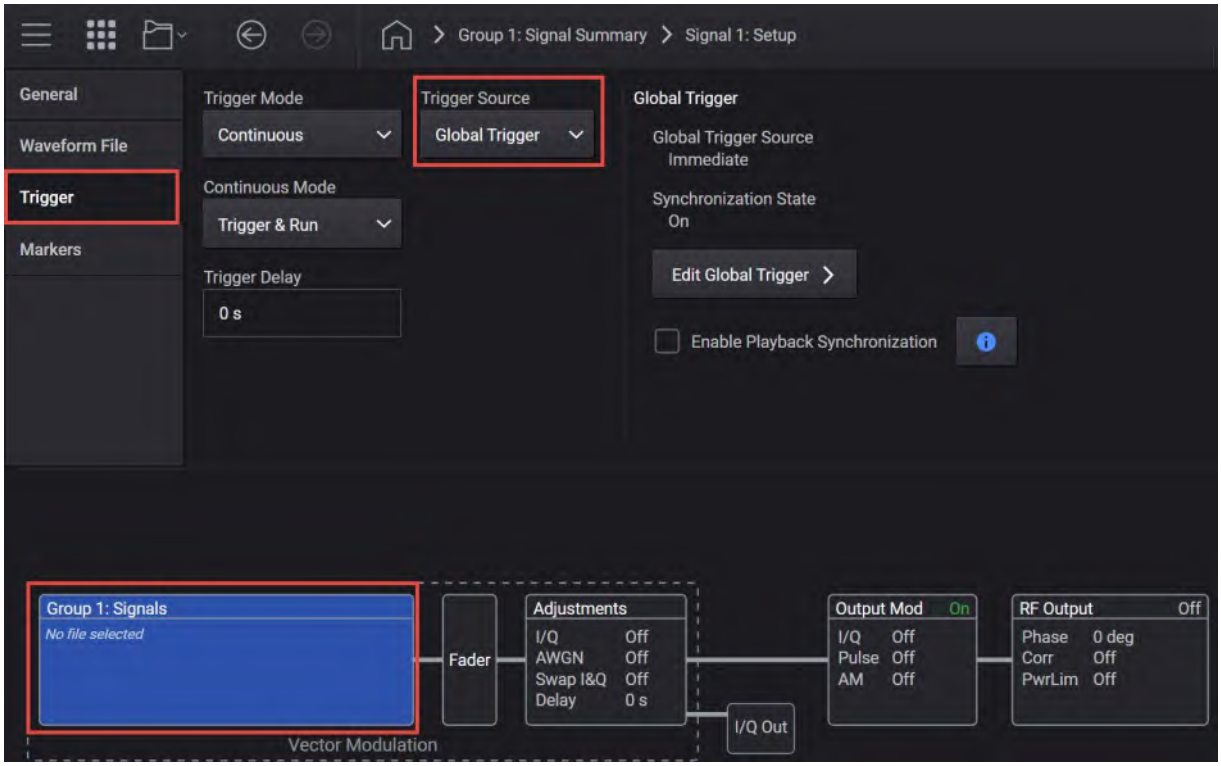


22. Select the arrow for Signal 1 to open the Signal Setup window.

NOTE This screen is only accessible if Option M9484C-8SG (8 virtual signal generators) is installed. For all other option configurations, continue to the next step.



23. Select the **Trigger** tab and set Trigger Source to **Global Trigger**.



Demodulate the 5G NR 2x2 MIMO in X-Apps

NOTE Changing settings on the VSA can take longer to take affect than expected. Wait to change the next setting until the previous setting has been updated.

On the Primary UXA:

1. From the N9042B select **Mode/Meas** > **5G NR &V2X** Mode > **OK**.

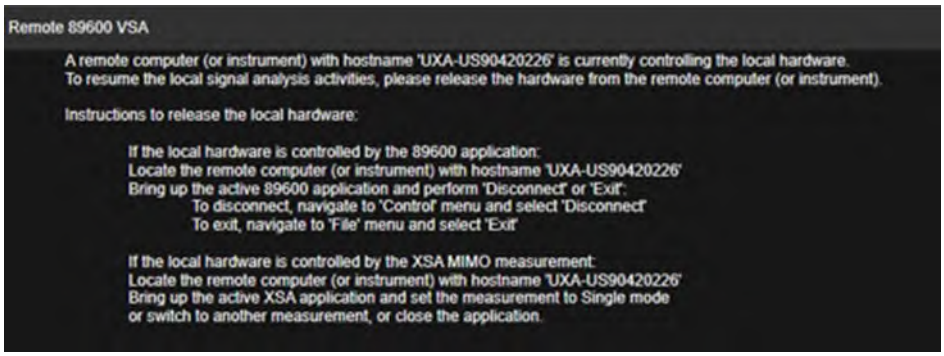
NOTE If accessing the instrument via a Remote Desktop connection, select the Screen tab (at the top left of the display) to open the Mode/Measurement/View Selector window.

2. Select **Mode Preset** to set Spectrum Analyzer 5G NR mode to a known state.
3. From the N9042B Menu Panel (or the Screen tab), select **Mode/Meas** > **5G NR & V2X** Mode > **Modulation Analysis** Measurement > **OK**.
4. From the Menu Panel, select **Meas Setup** > **Radio** and turn **MIMO On**.

5. Select **Multi Channel Config** to open the Configuration dialog box.
   - Set Channel 1 IP Address to **Local**
   - Enter the IP address for Channel 2

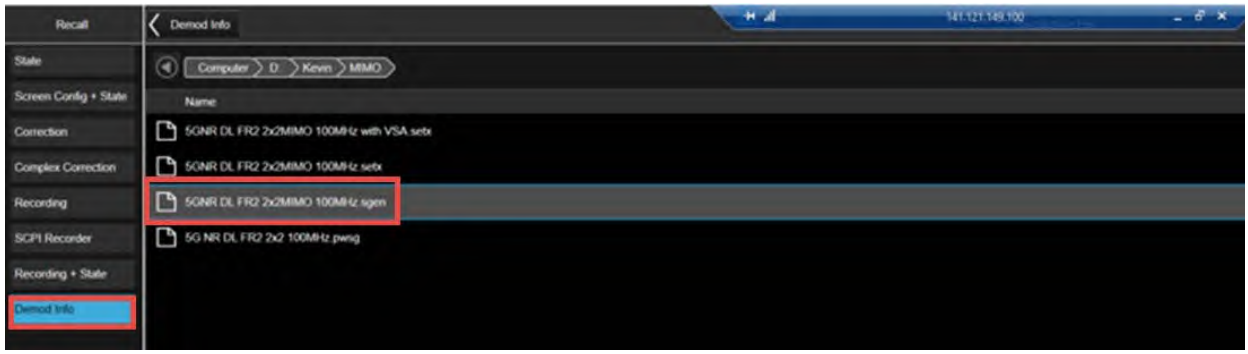
– Select **Locked** for both Channel 1 and Channel 2

Once MIMO mode is ON and both channels are locked, the secondary N9042B display changes as shown below.

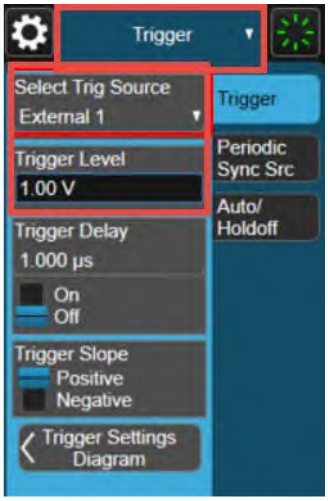


6. Select **Recall** (If accessing the signal analyzer remotely, select the Folder icon at the bottom of the display) > **Demod Info** > set Data Type to **CC Setup** > **Recall From** > and **Recall** the setup file.

   You must be in the correct analysis mode to recall the .scp file.



7. Select **Trigger** > **Trigger** tab and set Select Trigger Source to **External 1** and Trigger Level to **1 V**.



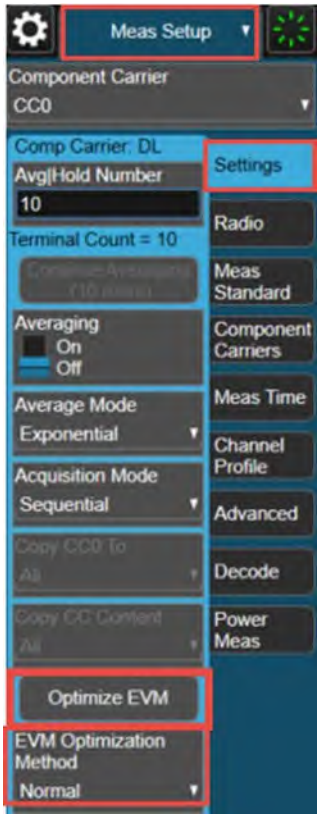
8. Select **Meas Setup** > **Settings** tab> **Optimize EVM**.

   Optimize EVM is an immediate action to adjust the hardware settings to minimize EVM.

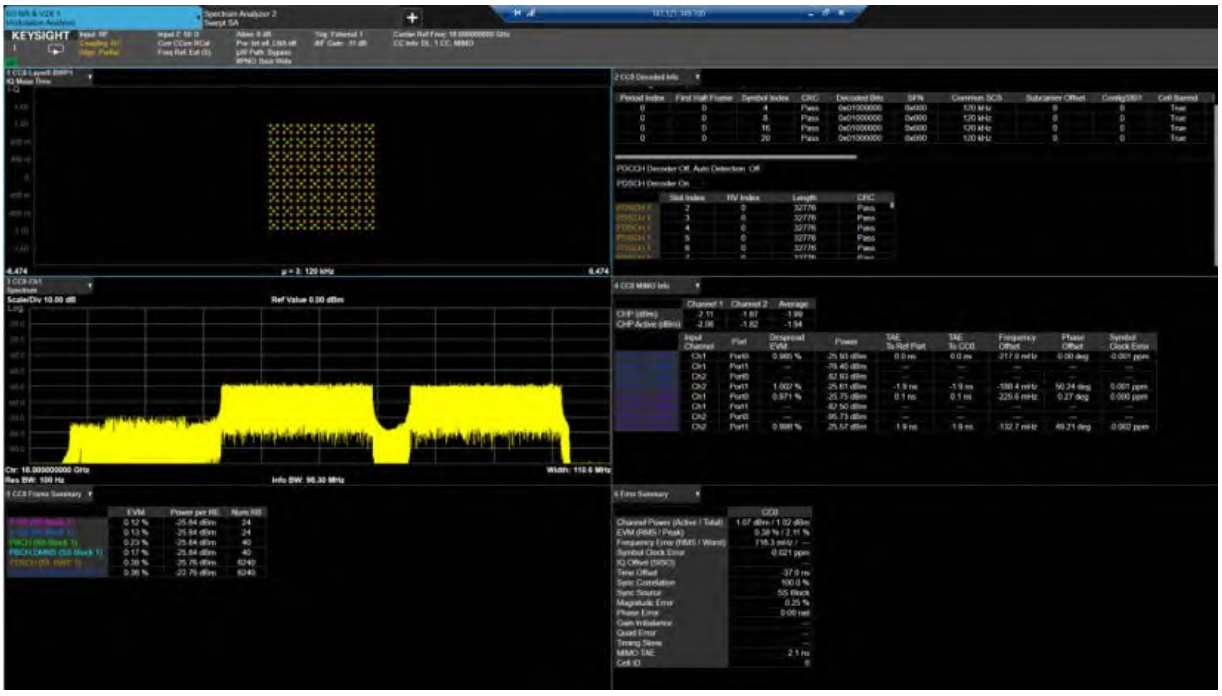
   – Normal method will capture input signal, measure its peak power then the algorithm will find proper setting combination for attenuation, preamp and IF Gain.

   – Iterative method will capture input signal multiple times in a iteration process, demodulate the signal and calculate EVM for each iteration and find setting combination for attenuation, preamp and IF Gain with minimum EVM.

For 5G NR signals, this method is much slower, than normal, but with achieve better EVM results. You can also manually adjust IF Gain, Frequency Extender Attenuation (use when measurement setup includes a V3050A frequency extender).



View the results.



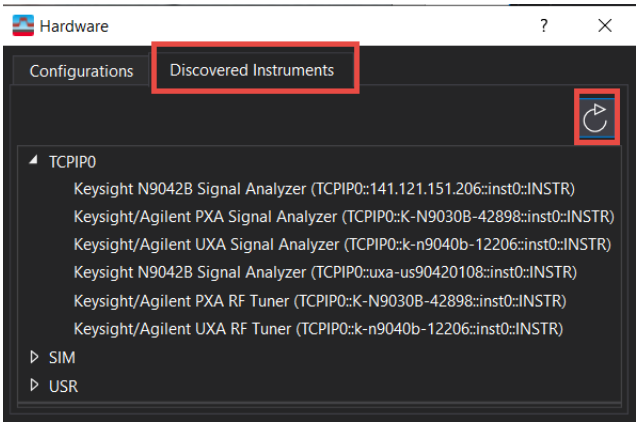
Demodulate the 5G NR 2x2 MIMO Using VSA:

1. Open the VSA software by selecting **Mode Meas** > **Launch VSA**.

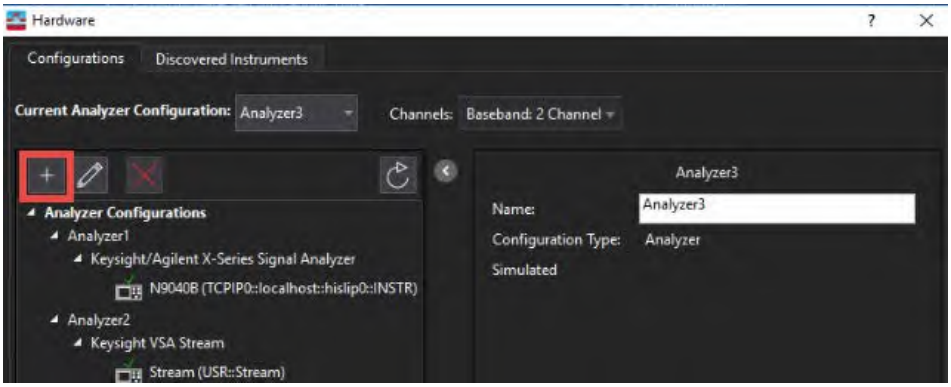
NOTE

If accessing the instrument via a Remote Desktop connection, select the Screen tab (at the top left of the display) to open the Mode/Measurement/View Selector window.

2. To configure the hardware, go to **Utilities** > **Hardware** > **Configurations**.
3. Open the **Discovered Instruments** tab and verify that the two UXAs are listed, if not, select the **Rediscover Instruments** icon.



4. In the **Configuration** tab, select the **+ icon**.

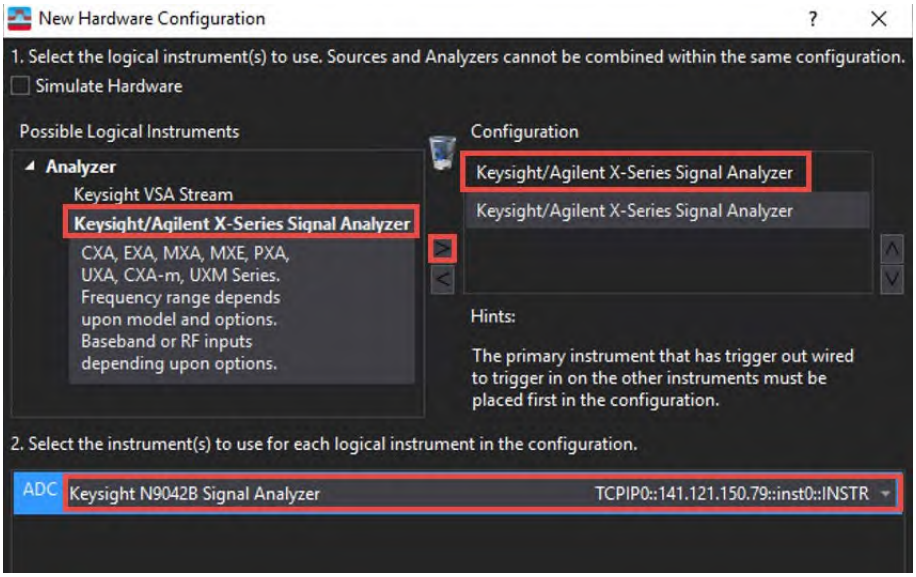


5. Scroll down the Possible Logical Instruments and select **Keysight/Agilent X-Series Signal Analyzer** twice.

   a. Select the first Keysight/Agilent X-Series Signal Analyzer and then press the right arrow to move it under the Configuration area.

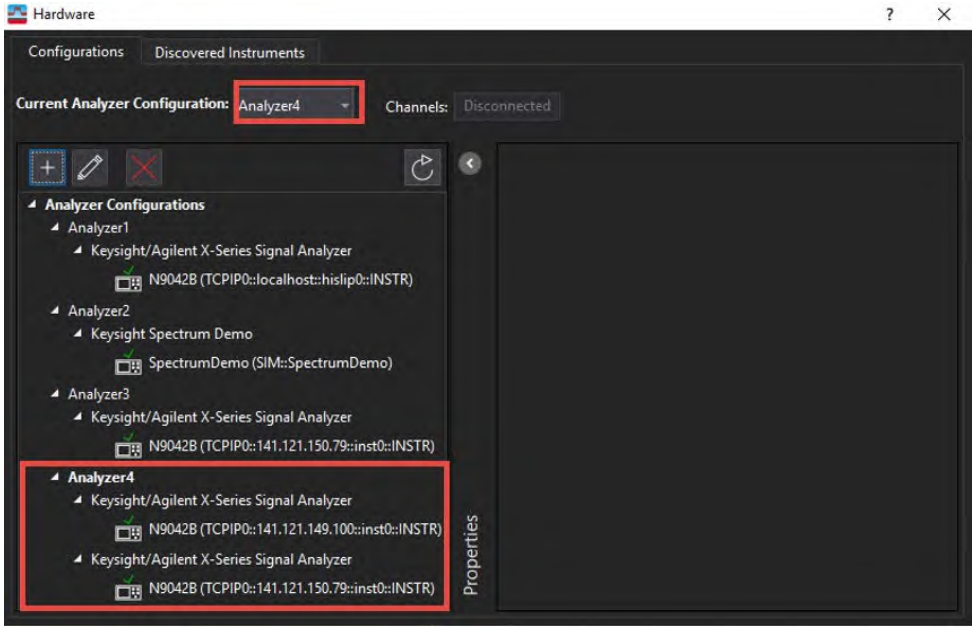
   b. From the middle of the dialog box, select the down arrow next to the Analyzer and select the IP address of the Primary UXA.

   c. Repeat the two steps above for the second signal analyzer but select the IP address of the Secondary UXA.

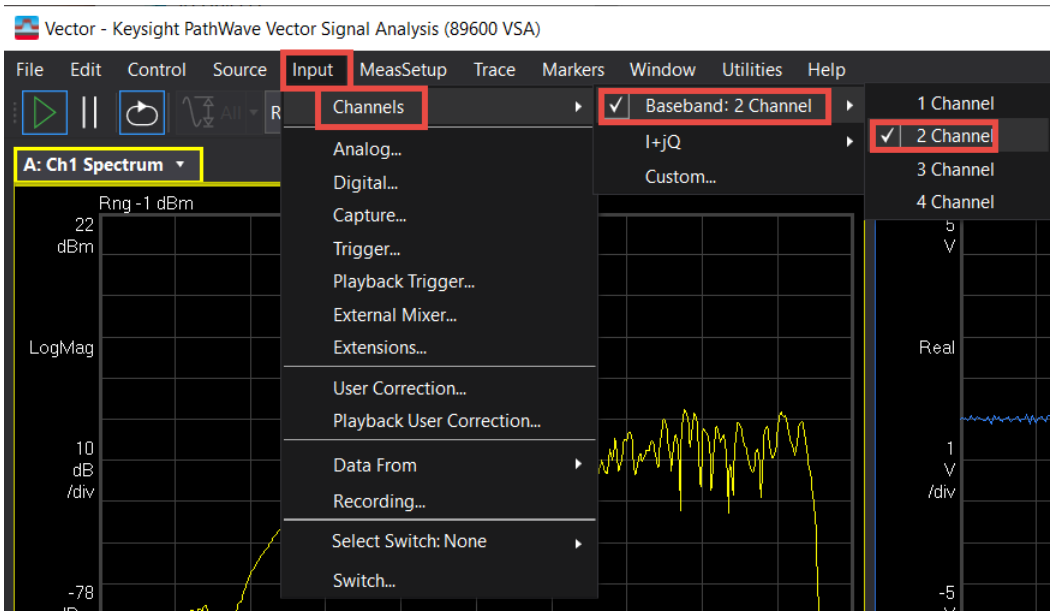


6. Select **OK** to create the new configuration.

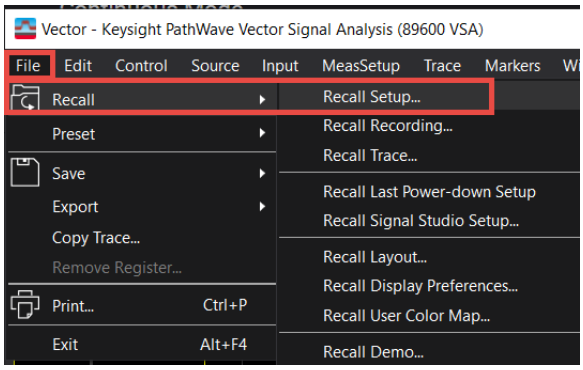
7. In the **Configurations** tab, set the Current Analyzer Configuration, select the Analyzer number for the new configuration. In this example **Analyzer 3**.



8. From the menu bar, select **Input** > **Channels** > **RF** > **2 Channel**.



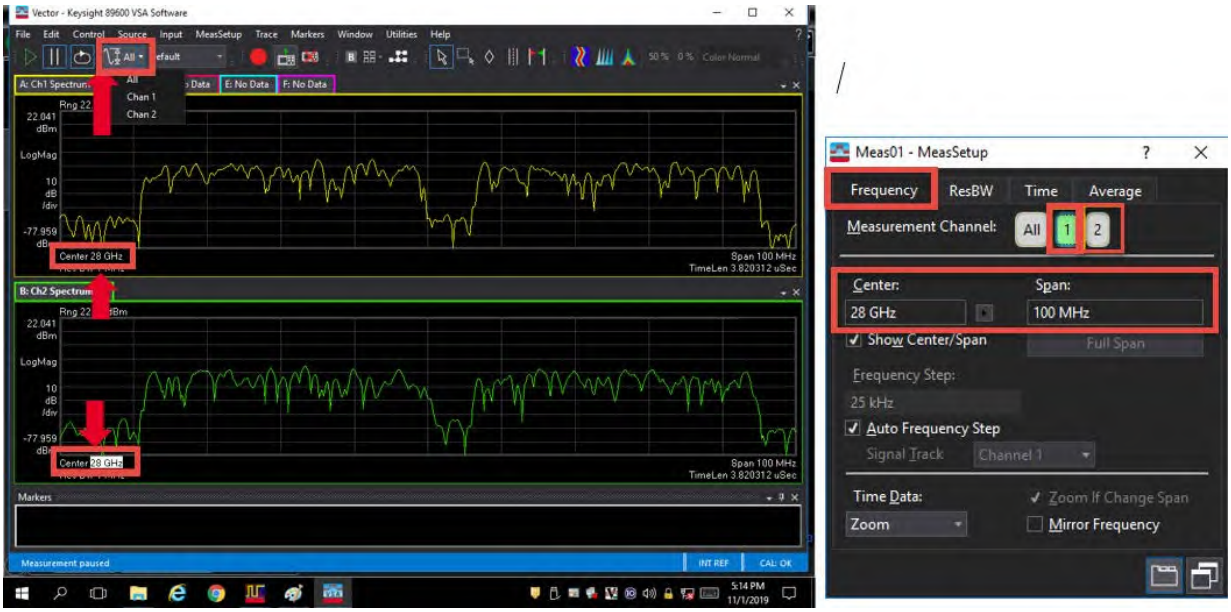
9. From the menu bar, select **File** > **Recall** > **Recall Setup** and navigate to the .setx file you save during the VXG waveform generation.



You can also set the frequency and span of both channels under Meas Setup > Frequency.

If you are using a UXR, Select Autorange .

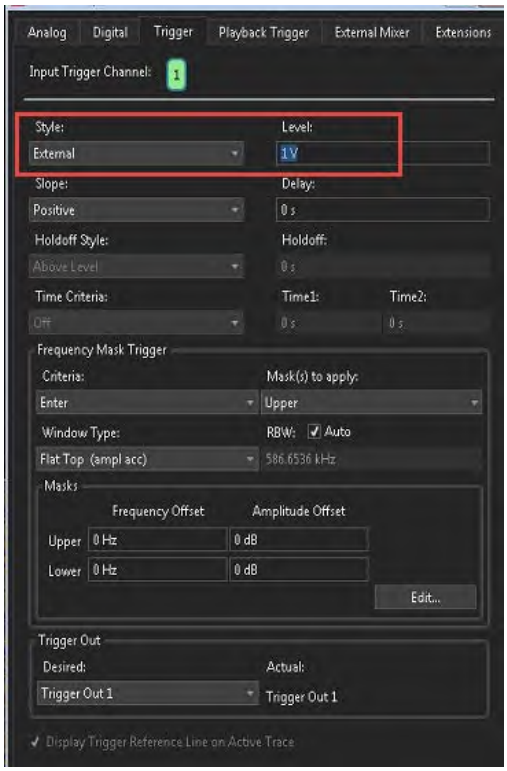
Auto Range samples the current input signal and then sets the full scale input range to the minimum range that includes the peak voltage sample of the input signal.



10. From the menu bar, select **Input** > **Trigger** and set:

    – Style to **External**

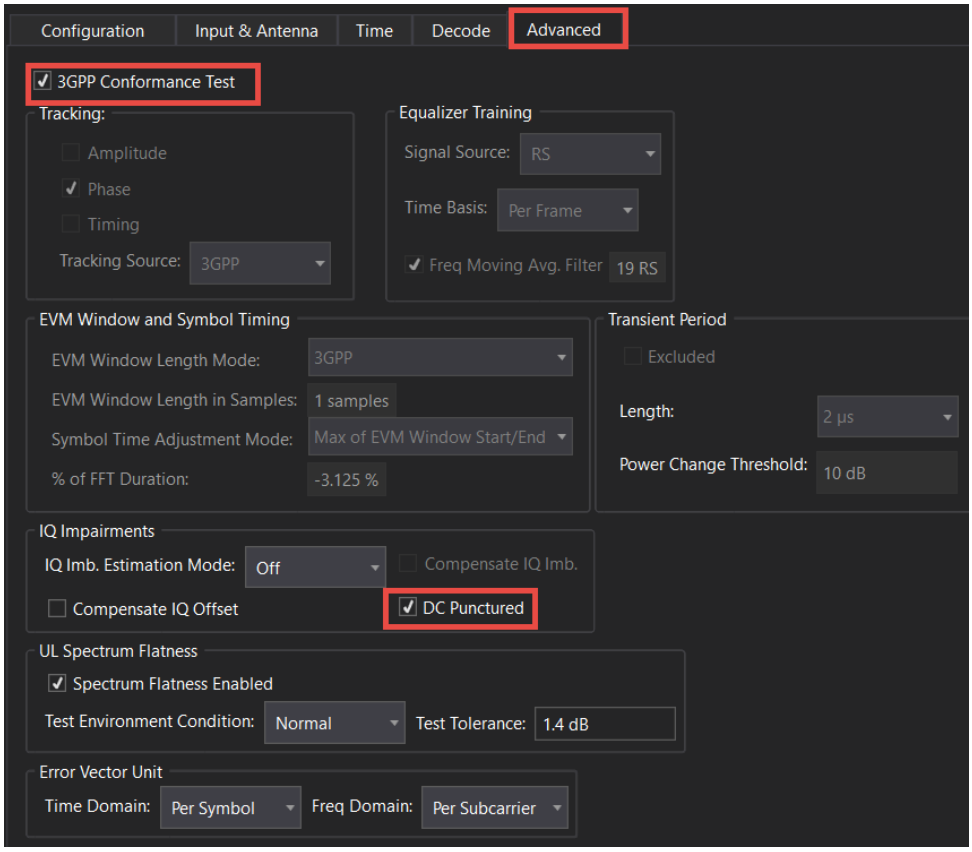
– Level to **1 V**



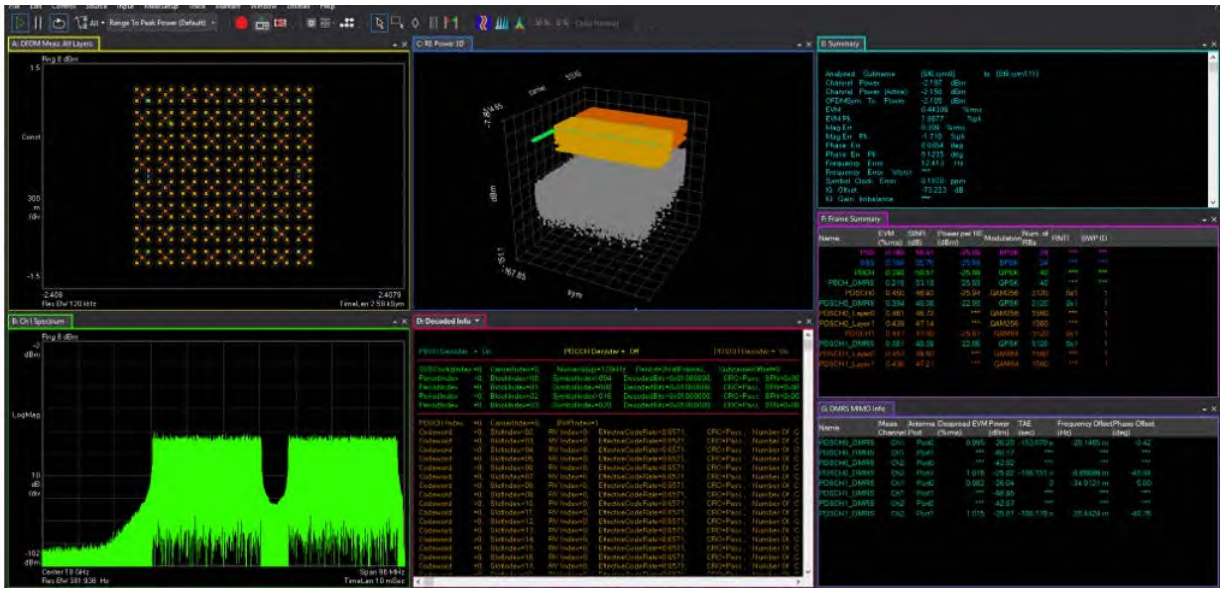
11. From the menu bar, select **MeasSetup** > **5G NR Demod Properties**> **Advanced** tab and select the **DC Punctured** check box.

    As part of conformance test, 3GPP has defined different equalizer training and tracking for EVM measurements. 3GPP Conformance Test is enabled by default where Tracking, Equalizer Training and EVM Window, and Symbol Timing is applied per 3GPP conformance test requirement. For FR1, no tracking is applied. For FR2, Phase Tracking using PTRS is applied.

5G NR counts the DC subcarrier as a valid subcarrier for rate-matching purposes. The network decides if the DC subcarrier is modulated or not. High LO feedthrough will impact demodulation and EVM performance of the input signal. This is OFF by default.

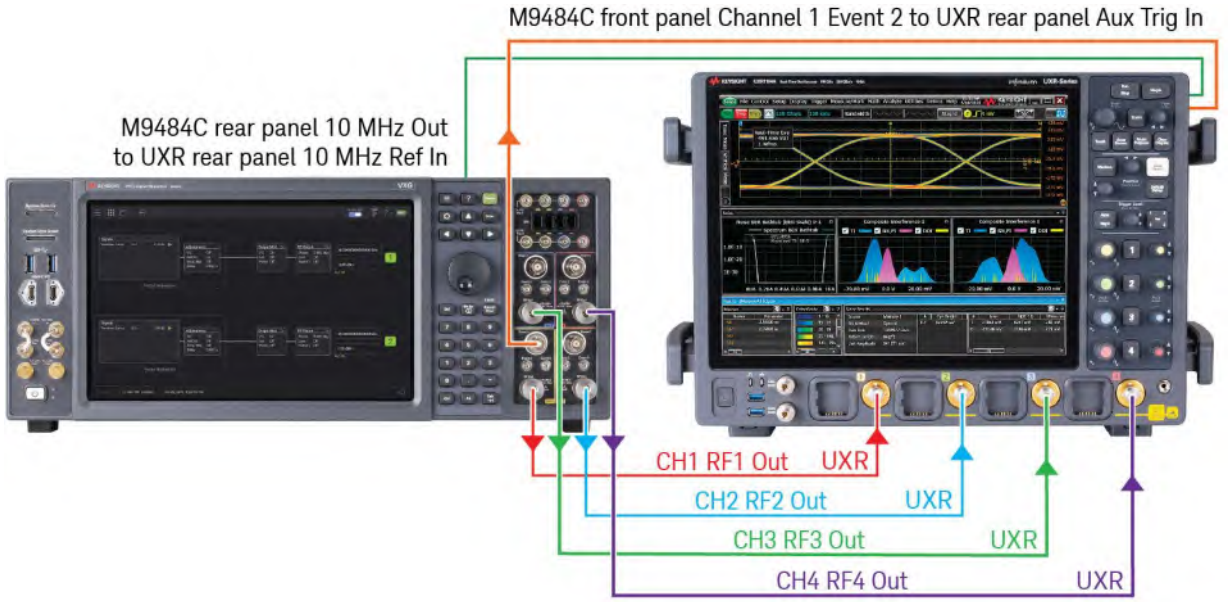


12. Close the 5G NR Properties dialog.
13. From the toolbar, select the **Auto-Range** dropdown and select **EVM-Table or Algorithm Based**.
    - EVM - Table or Algorithm Based performs EVM optimization based on prescribed table/algorithm instead of using EVM in the feedback loop. This method of EVM auto-range is normally faster than Meas Based Iteration, but may not achieve the most optimal setup for minimized EVM).
    - EVM - Meas Based Iteration has a feedback loop around the entire measurement, uses the measured EVM in the feedback loop, and tries to adjust hardware parameters to minimize the EVM. It is the slowest EVM auto-range method, but it should achieve the most optimal setup for good EVM.



Creating and Analyzing a 4x4 MIMO using a UXR Running VSA Software

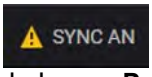
Equipment Setup



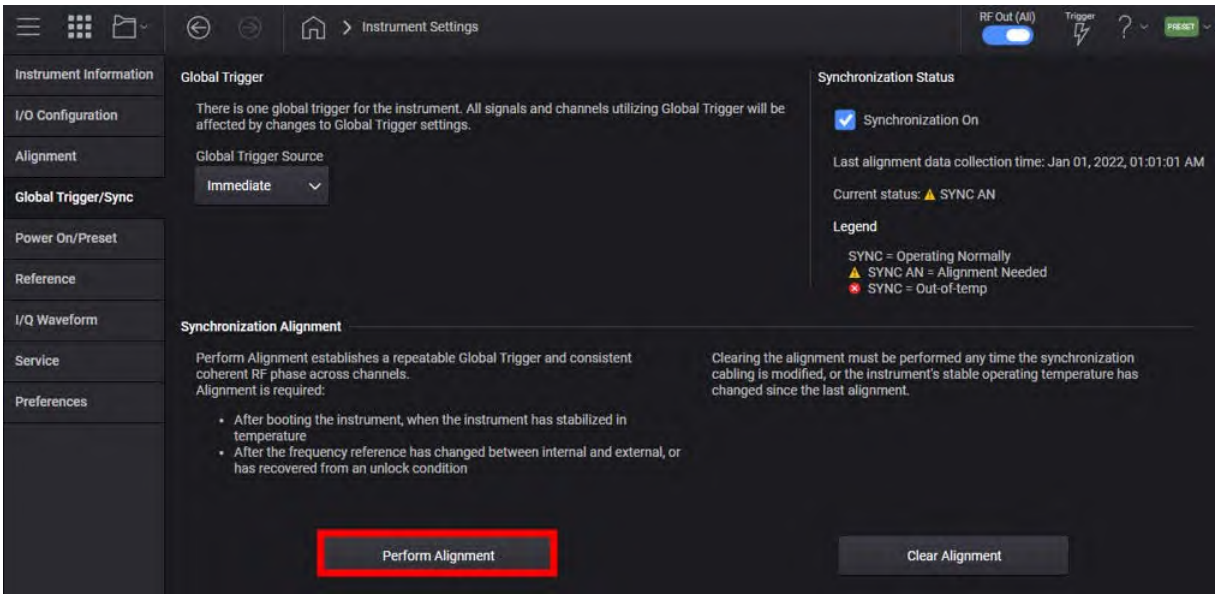
- M9484C
  - Four channels (M9484C-004)
  - N7631APPC PathWave Signal Generation for 5G NR
- UXR0334
  - UXR000-602: DDC option with 2 GHz BW
  - 89600 PathWave Vector Signal Analysis (VSA)

On the VXG:

1. Connect the equipment as shown above.
2. Select **Preset** > **Preset** to set the VXG to a known state.

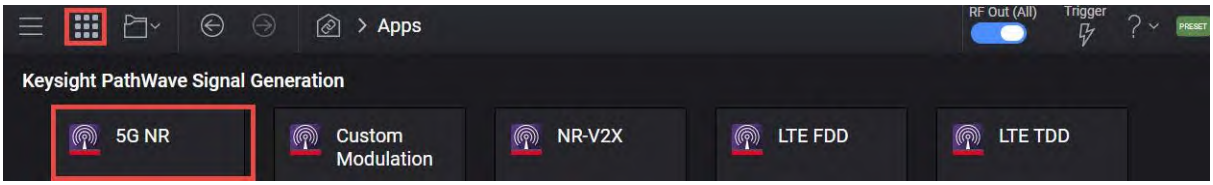
If a Synchronization Alignment is required, indicated by 

in the bottom left corner, tap or click the warning message and choose **Perform Alignment** before proceeding.

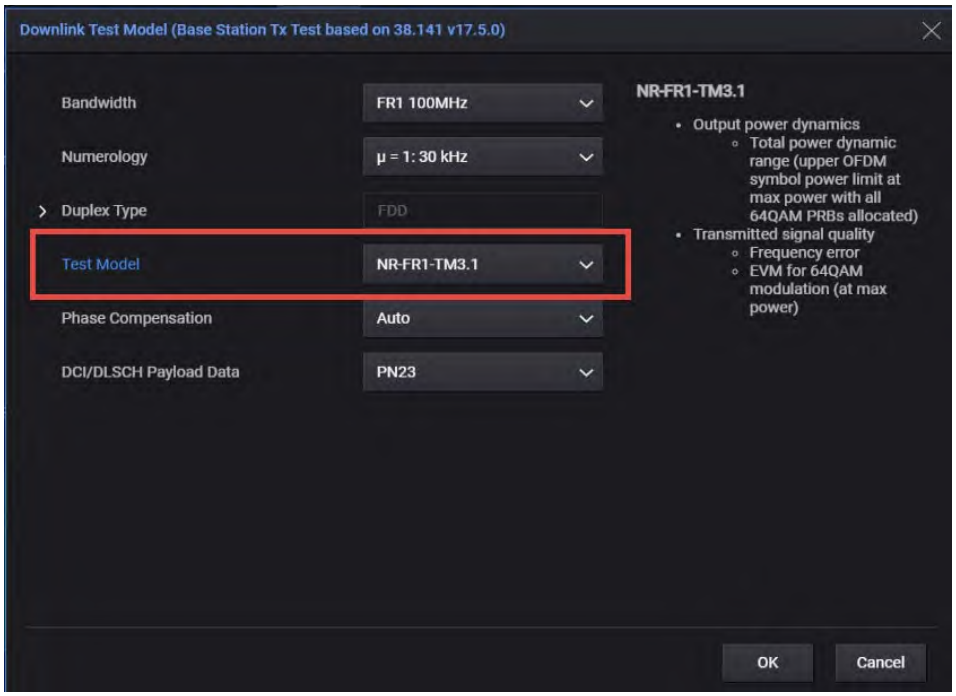


3. In the Output area for all four channels, set the Frequency to **5 GHz** and Power to **–10 dBm**.

4. Select the **Apps** block to open, then select **5G NR**.

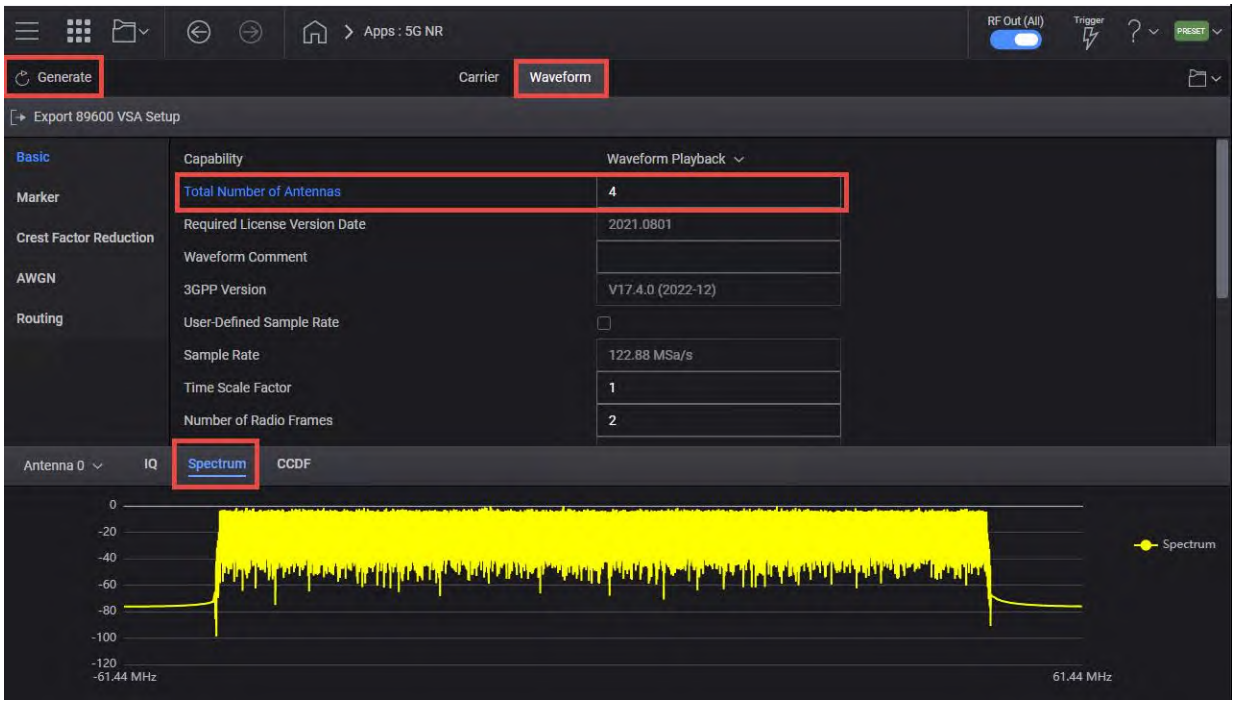


5. Select the **Carrier** tab > **DL Test Model** and set the Test Model to **NR-FR1-TM3.1.** Leave other settings as default and select **OK**.

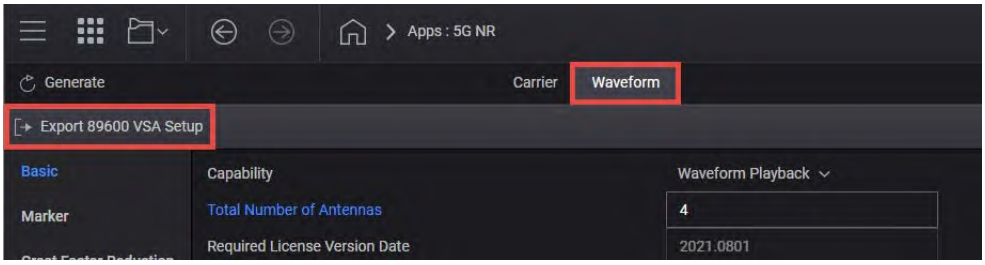


6. Select the **Waveform** tab and set the Total Number of Antennas to **4**. This indicates that two antenna port signals will be generated.

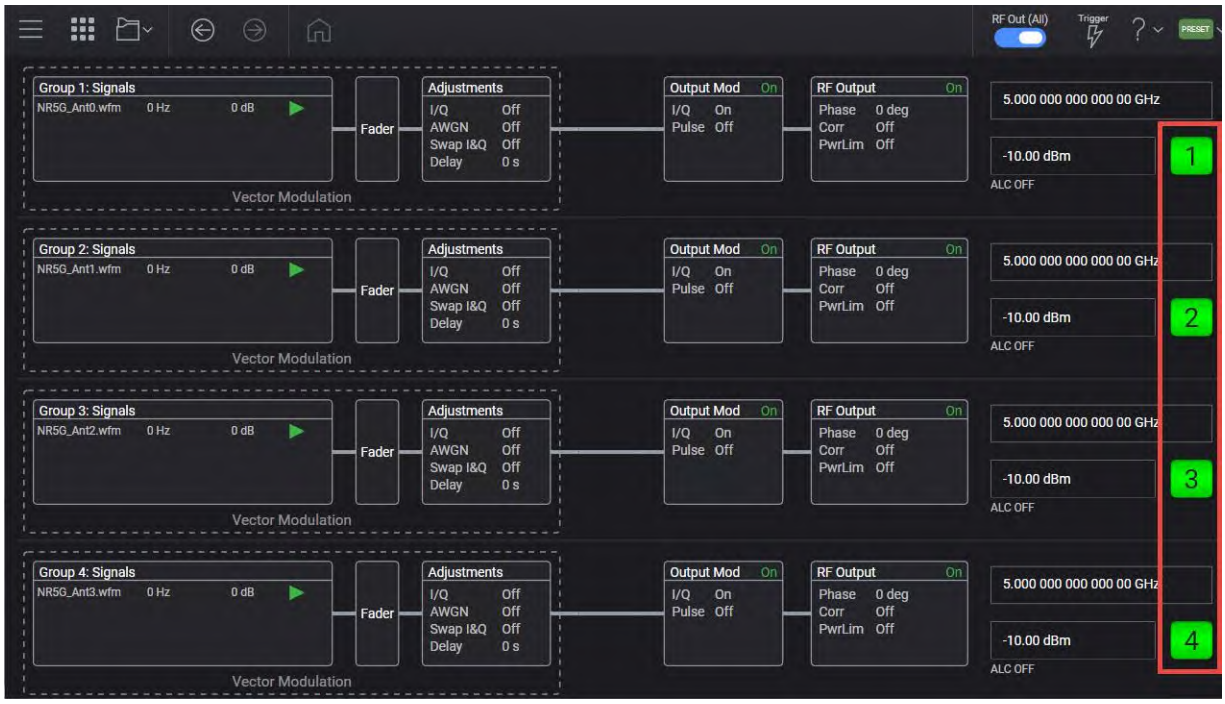
7. Select **Generate** to apply the signals to each channel. The expected spectrum can be observed in the bottom window.



8. To simplify analysis, select **Export 89600 VSA Setup** and save the file.



9. Select the **Home** icon to return to the main block diagram, and then select the gray **1**, **2**, **3**, and **4** to turn on each channel.



On the UXR

1. Open the VSA software.

   To access the VSA software, go to the Windows Start menu and find Keysight 89600 Software (latest installed version) folder and run the software.

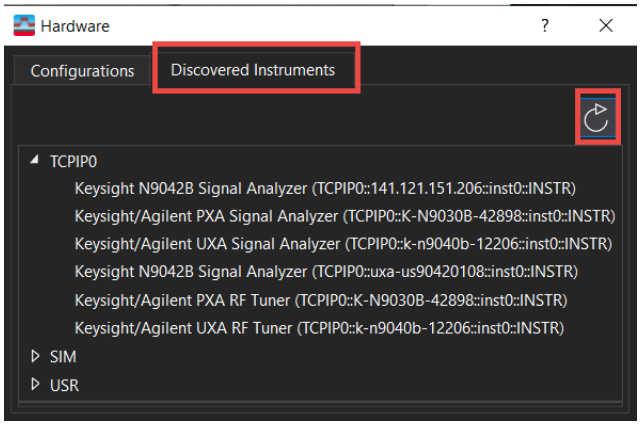
NOTE

If accessing the instrument via a Remote Desktop connection, select the Screen tab (at the top of the display) to open the Mode/Measurement/View Selector window.

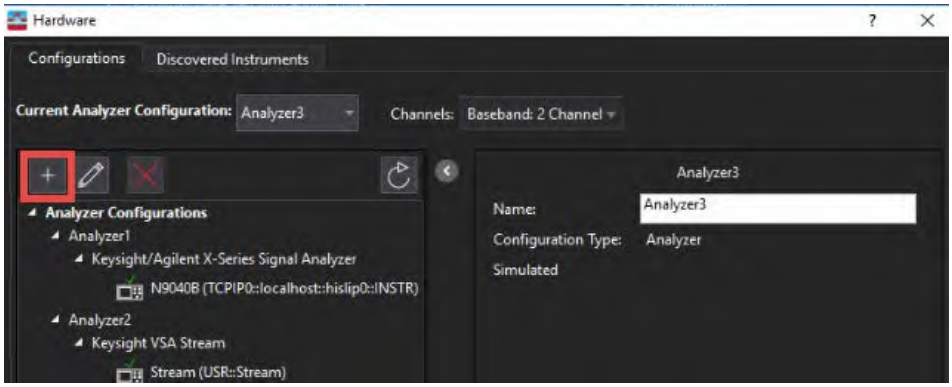
2. From the VSA menu bar, select **File** > **Preset** > **All** to set the VSA to a known state.
3. To configure the hardware, go to **Utilities** > **Hardware** > **Configurations**.

   If it is already configured, go to step 10.

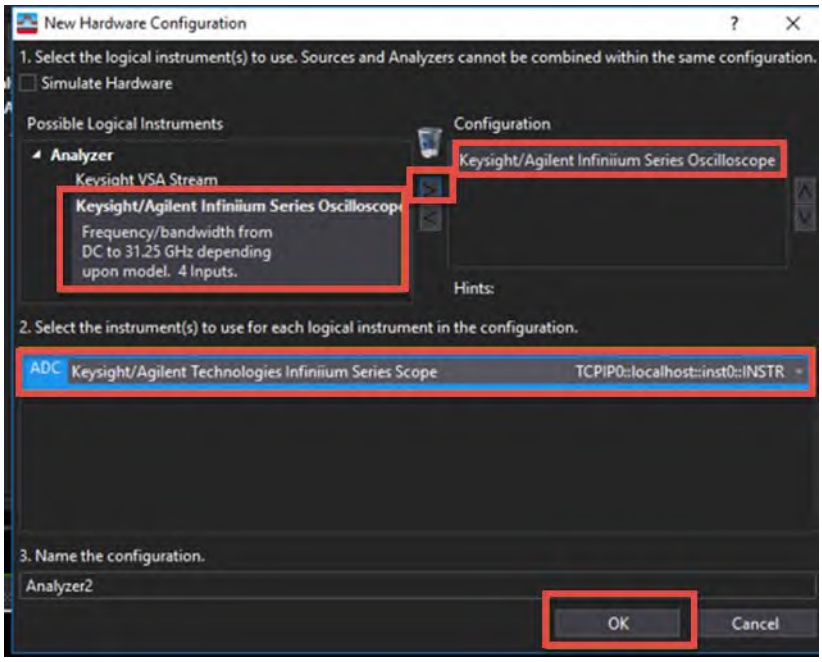
4. Open the **Discovered Instruments** tab and verify that the UXR is listed, if not, select the **Rediscover Instruments** icon.



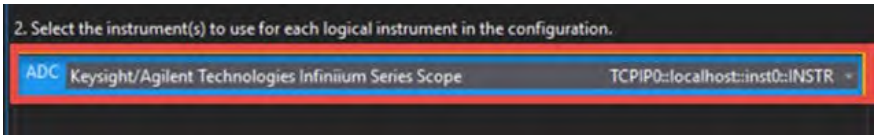
5. In the **Configuration** tab, select the **+ icon**.



6. Scroll down the Possible Logical Instruments and select **Keysight/Agilent Infiniium Series Oscilloscope**, and then select the right arrow to move it under Configuration.

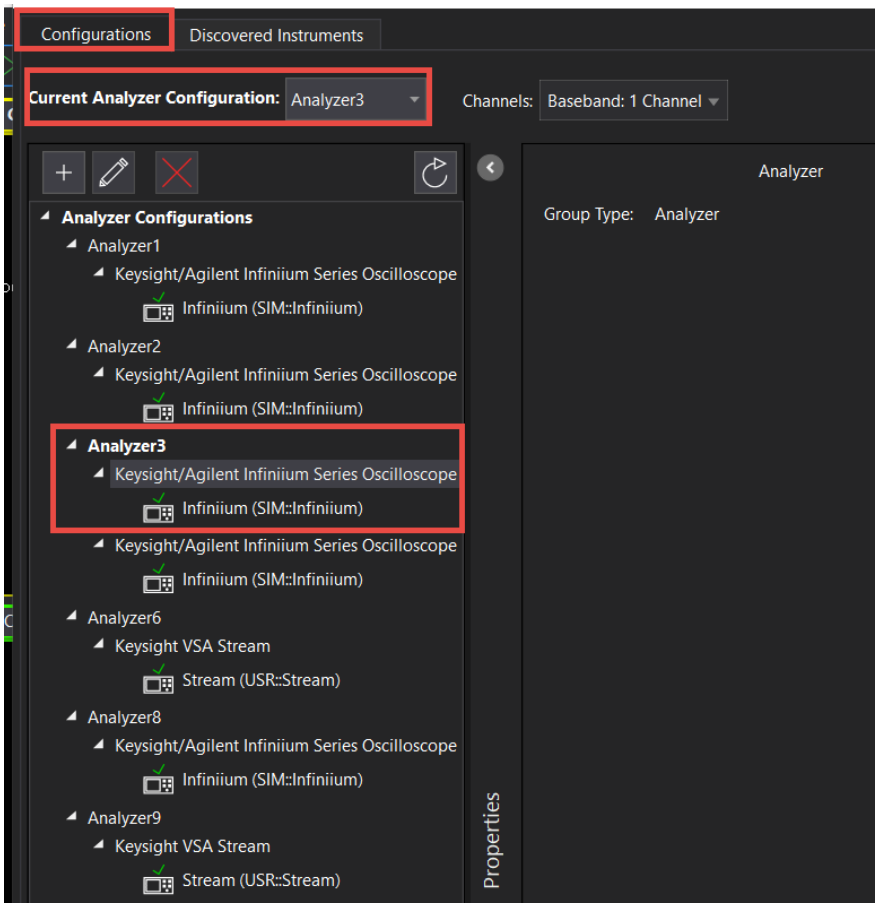


7. From the middle of the dialog box, select the UXR to be used.

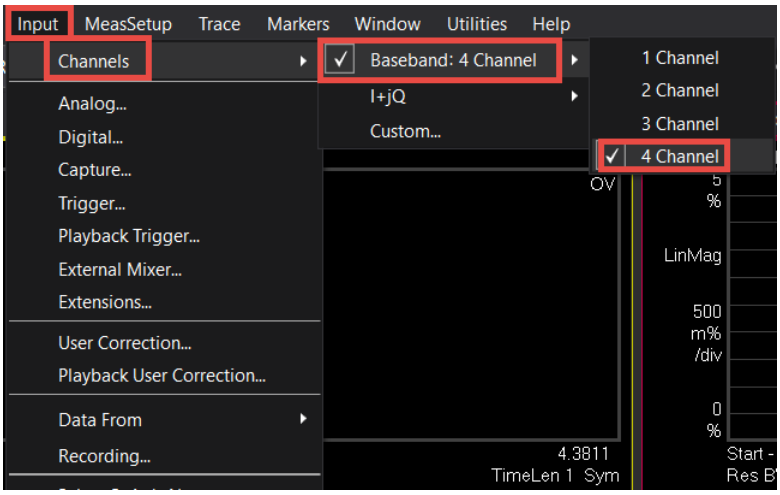


8. Select **OK** to create the UXR configuration.

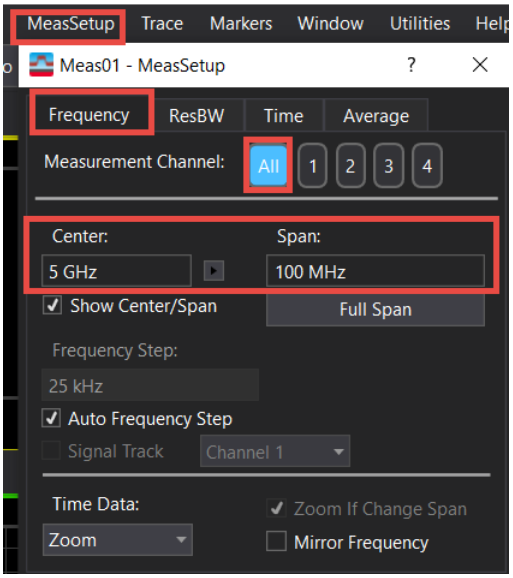
9. In the **Configurations** tab, set the Current Analyzer Configuration by selecting the Analyzer number for the new configuration. In this example **Analyzer 3**.



10. From the menu bar, select **Input** > **Channels** > **Baseband** > **4 Channels**.



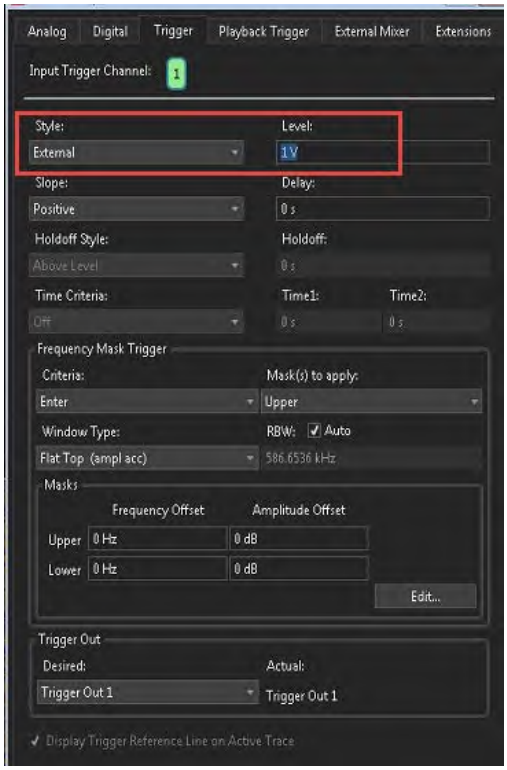
11. From the menu bar, select **MeasSetup** > **Frequency** and set the Center Frequency to **5 GHz** and Span to **100 MHz** on all four channels.



12. If External Trigger is available, it is recommended to use it to speed up the demodulation of the measurement. From the menu bar, select **Input** > **Trigger** and set:

    – Style to **External**

    – Level to **1 V**



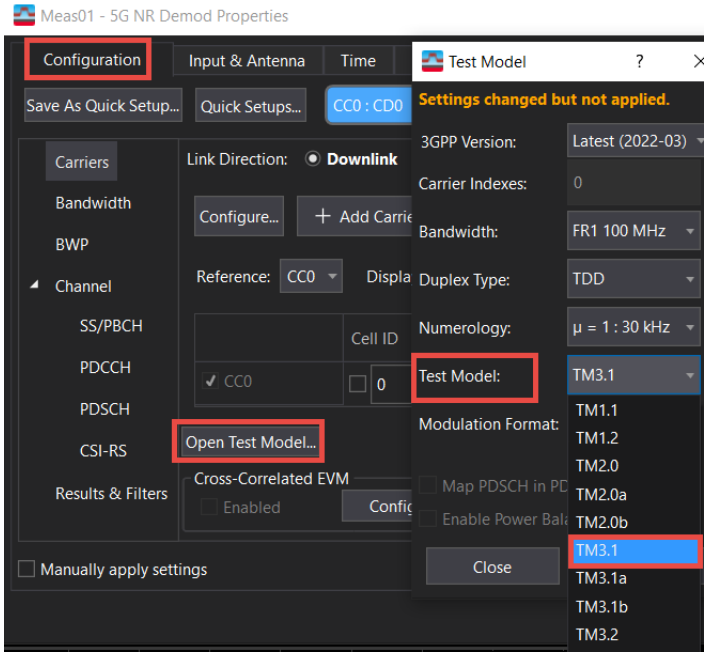
13. Select **File** > **Recall** > **Recall Setup** and navigate to the setup file you saved during the VXG 5G NR setup.

NOTE

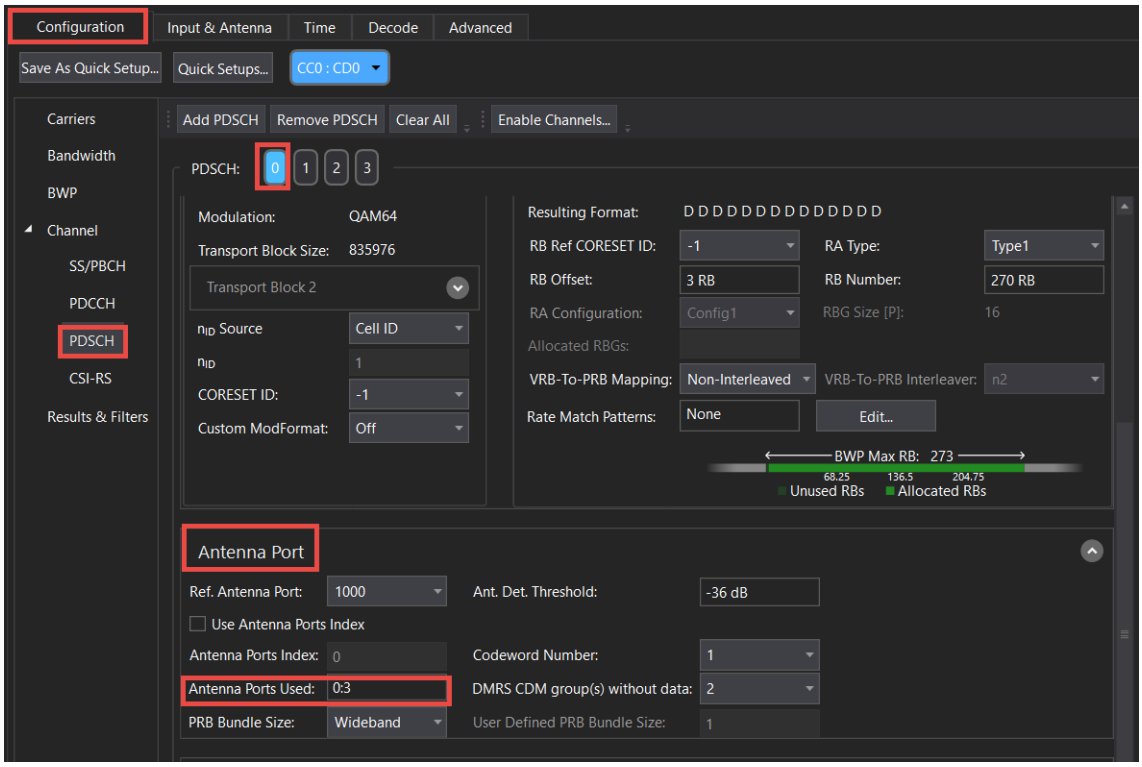
You will need to copy over the setup files to the X-Series Signal Analyzer or the PC running the VSA application, or continue with the settings below to setup the VSA manually.

14. From the menu bar, select **MeasSetup** > **Measurement Type** > **Cellular** > **5G NR** > **5G NR Modulation Analysis**.

15. To configure the Analysis region, select **MeasSetup** > **5G NR Demod Properties** > **Configuration** tab and select **Open Test Model** > **TM3.1** > **Apply** > **Close**.



16. Select the **PDSCH** tab, scroll down, open the **Antenna Port** tab and set Antenna Ports Used to **0:3**. Repeat for the remaining PDSCH channels by selecting the corresponding number at the top.



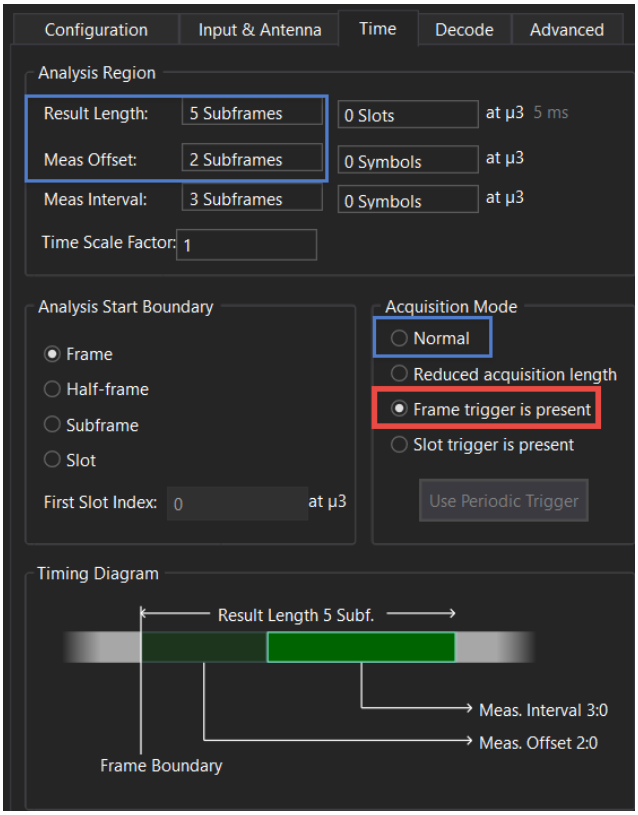
17. To reduce measurement time, select **MeasSetup** > **5G NR Demod Properties** >**Time** and set:

    – Result Length to **5 Subframes**

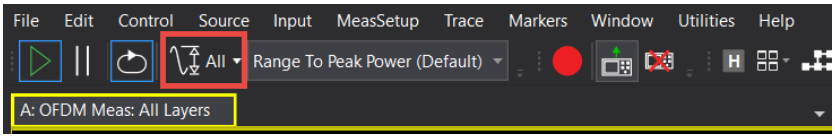
    – Meas Interval to **2 Subframes**

    – Enable **Frame Trigger is present**.

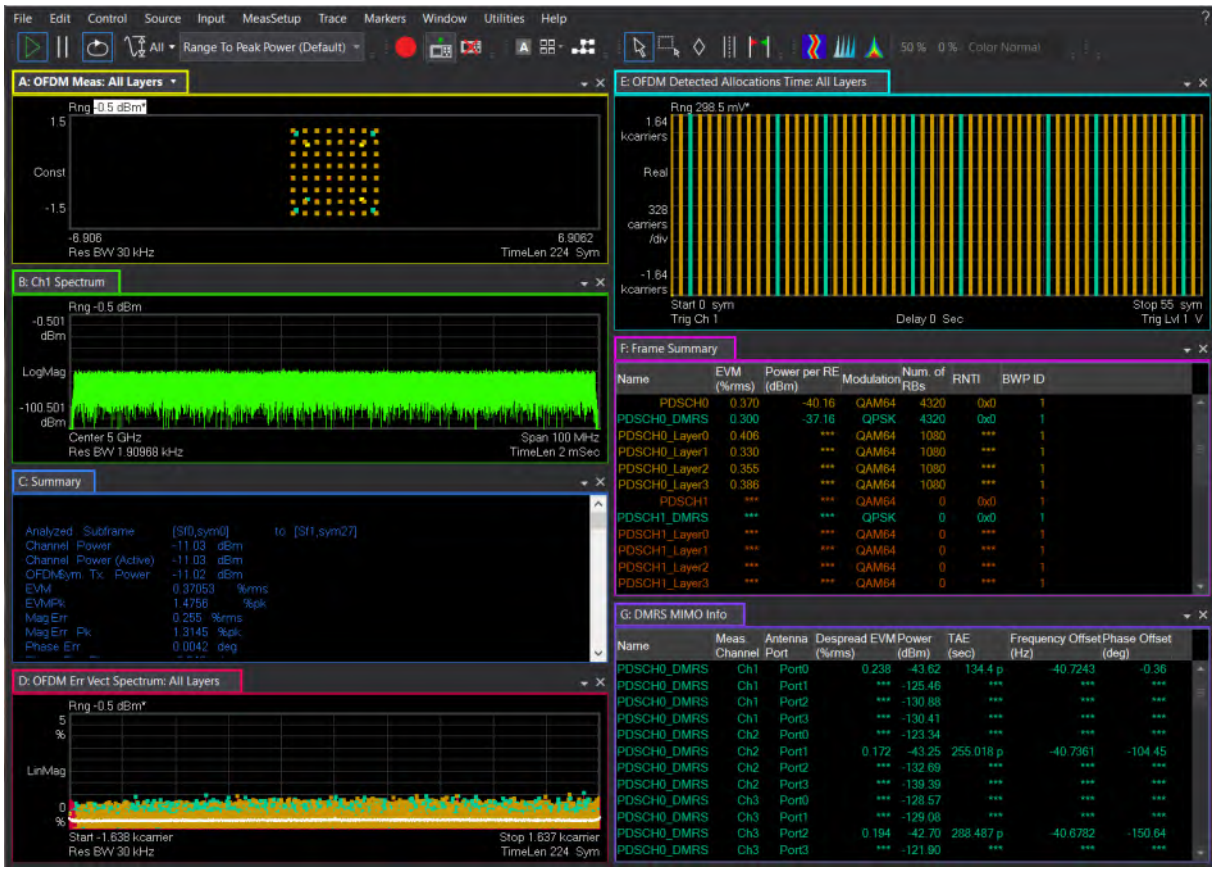
We are using an external trigger for this example so selecting Frame Trigger is Present will use the external trigger and will speed up the measurement significantly. If not using an external trigger, set to Normal.



18. At the top of the display, select **Autorange**.



19. Observe the demodulation results.



- Trace A: Composite constellation diagram showing different modulation formats for the different channels/signals present in the transmitted signal. See Trace F (Frame Summary) for the list of channels and modulation formats.
- Trace C: Summary trace showing composite error metrics.
- Trace D: Error Vector Spectrum showing EVM versus subcarrier and symbol.
- Trace E: Detected Allocations Time showing the detected allocations of all channels/signals within the measurement interval.
- Trace F: Frame Summary. EVM, per Layer EVM, Power per RE, Mod Format, Number of RB, RNTI, and BWP ID of the individual channels/signals.
- Trace G: MIMO Info table shows EVM, power, and time, frequency and phase offset for each antenna port.

MIMO Info is a type of Matrix Table that provides sortable rows by column, selectable column visibility, and copy/paste and export functionality to share rows of content or complete tables to applications like email, text editing or spreadsheet programs.

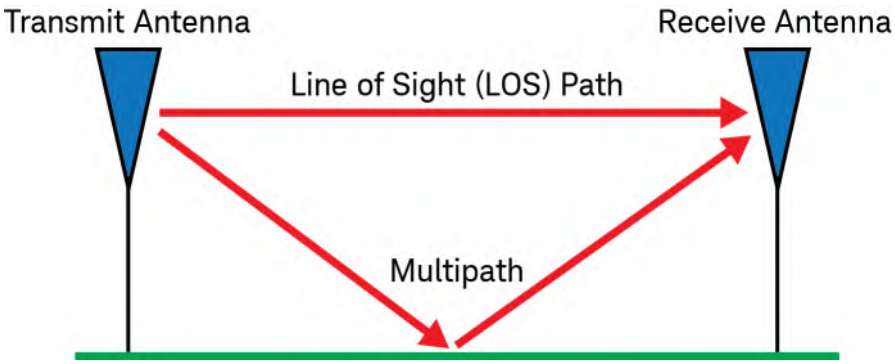
Setting up a 1x8 MIMO and PUCCH Fading Example

NOTE

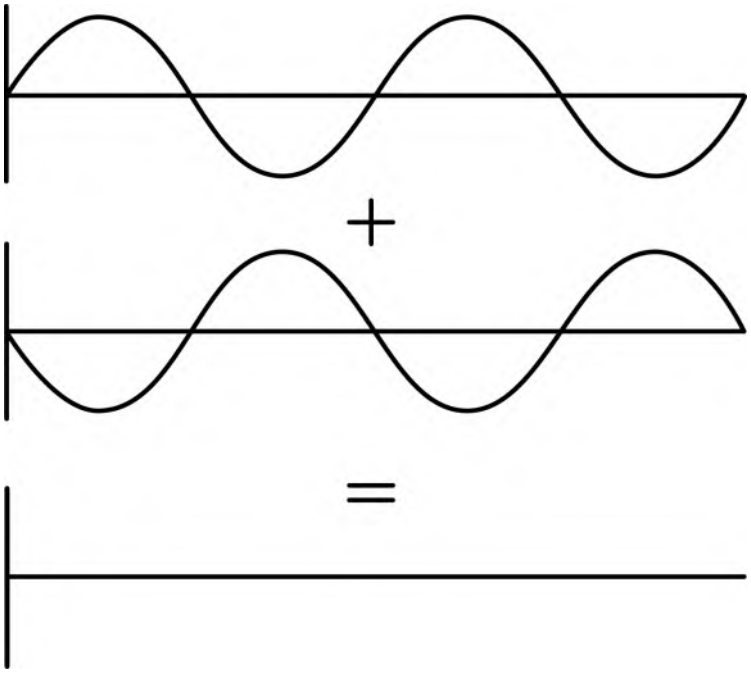
The VXG must have the N7605AP0C 3GPP MIMO Fading license installed.

Fading is the change in the attenuation of a signal due to many variables which include geographical position, time, and frequency. Fading, also known as Channel Simulation, allows the user to apply degradations to signals to mirror real-world conditions. Some examples of real-world fading causes are multipath propagation (where the signal reflects off different obstacles in the signal path), multi-path fading, weather (such as rain) and showing from obstacles affecting the signal propagation. The existence of obstacles surrounding the transmitter and receiver create multiple paths for the transmitted signal to travel. The receiver sees a summation of multiple copies of the transmitted signal that have traveled different paths. Each received signal will arrive with differences in attenuation, delay and phase shift on the journey from source to receiver. This may result in constructive or destructive interference.

The figure below demonstrates signal transmission with multipath fading.



The figure below shows line of sight and multipath reflection summed together to create destructive interference results.



Strong destructive interference is typically referred to as a deep fade and causes temporary failure of the received signal as the channel signal to noise ratio may drop quite drastically. A common real-world example of a deep fade is when a driver stops at a traffic light and the FM radio station signal that was strong and clear becomes static, but the signal is successfully received if the car moves a very short distance. The signal suddenly degrades in one point because the car stopped at a point where the signal had severe destructive interference. Cellular phone communications can have similar issues.

The VXG supports several different channel models, levels of correlations between channels, and selectable Doppler shifts. These parameters combine to determine a fading profile. Not all permutations of all settings are available and installed on the instrument. The expected use model is to select the required parameters in this order: standard, channel model, correlation matrix, and then Doppler shift. As each is selected, the remaining settings will be updated to reflect the possibilities available.

The fading profile is selectable for each signal path - meaning one per signal on each RF output present in the current instrument group. For example, in 1x2 MIMO, the signal's fader is individually configurable on RF output 1 and 2.

This example is based on the 8.3.1 PUCCH test from the 3GPP standard. First we will configure the waveform, then add AWGN and finally configure Fading. See the tables below for the specific settings that we will configure.

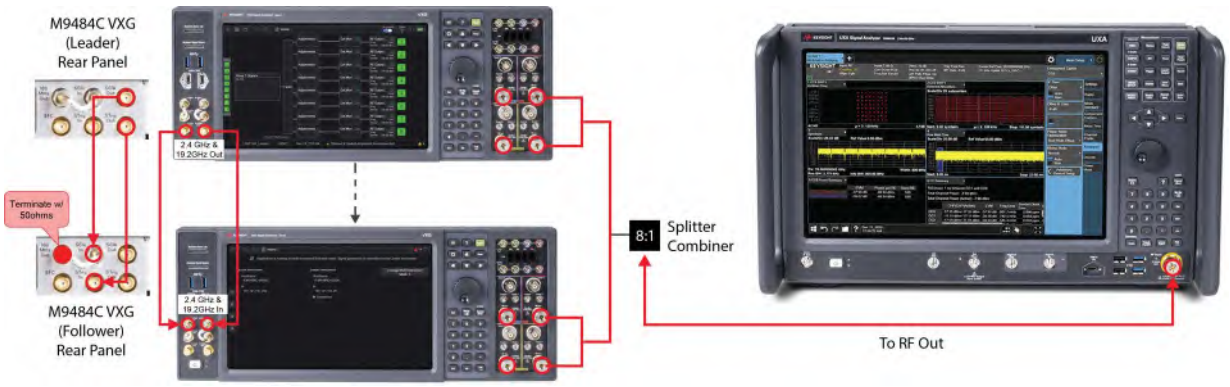
Subcarrier spacing (kHz)	Channel bandwidth (MHz)	AWGN power level
15	5	-83.5 dBm / 4.5 MHz
	10	-80.3 dBm / 9.36 MHz
	20	-77.2 dBm / 19.08 MHz
30	10	-80.6 dBm / 8.64 MHz
	20	-77.4 dBm / 18.36 MHz
	40	-74.2 dBm / 38.16 MHz
	100	-70.1 dBm / 98.28 MHz

PUCCH only occupies 1 resource block (RB), so the carrier bandwidth is 30 kHz Subcarrier Space (SCS) x 12 Resource Elements (RE) x 1 RB = 360 kHz.

The Channel mode and Signal to Noise Ratio (SNR) is as follows:

Number of TX antennas	Number of RX antennas	Propagation conditions and correlation matrix (annex G)	Number of OFDM symbols	Channel bandwidth / SNR (dB)			
				10 MHz	20 MHz	40 MHz	100 MHz
1	2	TDLC-300-100 Low	1	10.4	10.4	10.1	9.8
			2	4.8	4.2	4.4	4.1
1	4	TDLC-300-100 Low	1	4.0	4.0	3.6	3.9
			2	0.3	0.2	0.1	-0.2
1	8	TDLC-300-100 Low	1	-0.4	-0.4	-0.5	-0.4
			2	-3.1	-3.2	-3.4	-3.3

Equipment Setup



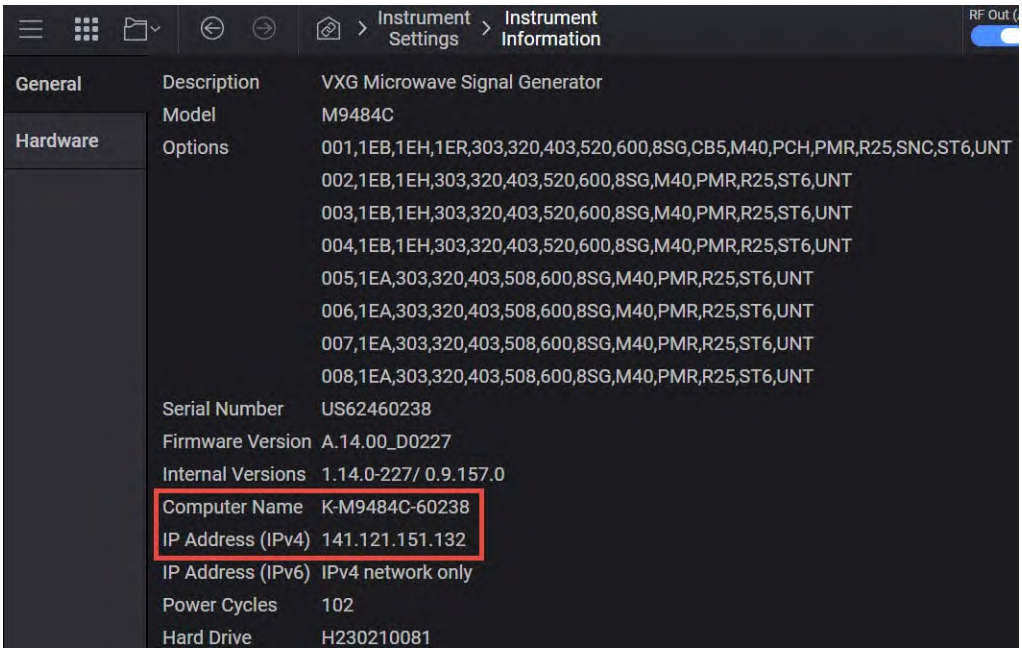
- M9484C
  - Four channels (M9484C-004) with options PCH and SNC
  - N7631APPC PathWave Signal Generation for 5G NR
  - N7605AP0C PathWave Signal Generation for 3GPP Real-time Fading
- N9042B UXA

  N9042B UXA Firmware version 2023 (XA34) or higher

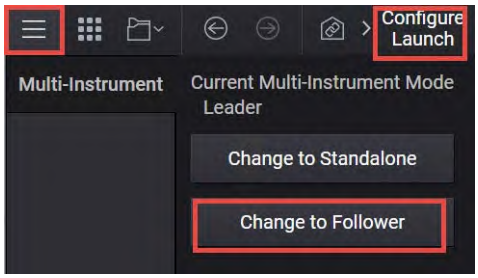
  - N9085EMOE 5G NR Measurement application
- 8:1 Power Splitter/Combiner

Configuring 1x8 MIMO

1. Connect the equipment as shown above.
2. **Restart** both VXGs and wait for both VXGs to fully boot and PathWave to load.
3. On the Follower, select the **System** icon > Instrument Settings, and note the **IP Address or the Computer Name**. You will need this in a later step.



4. One the Follower, select the **System** icon > **Configure Launch** > **Change to Follower**

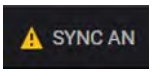


5. **Restart** the Follower.
6. Once the Follower is fully booted and PathWave has installed to the point that it shows a message "Waiting for Leader" is displayed, select the Leader's **System** icon > **Configure Launch** > **Change to Leader**.

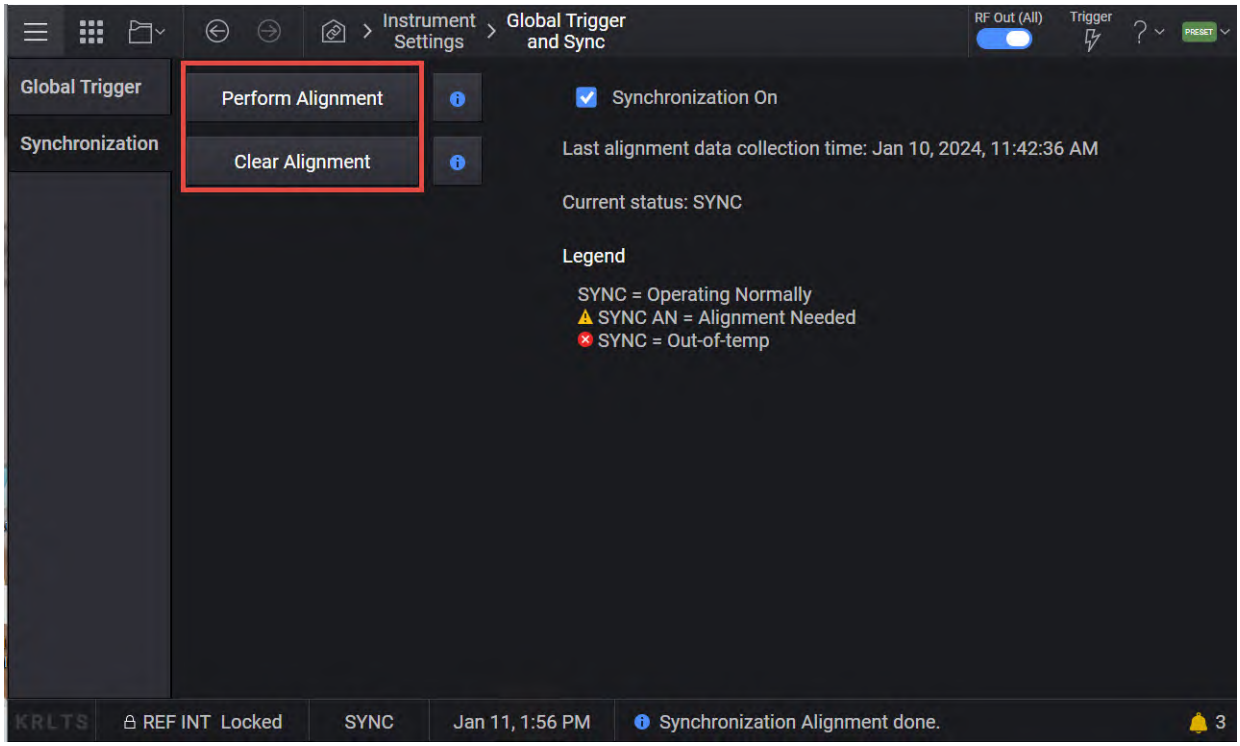
7. Enter the IP address of the Follower into the **Leader** and then select **Apply**.

   Wait for the Leader has PathWave running.

8. On the **Leader**, select **Preset** > **Preset** to set the VXG to a known state.

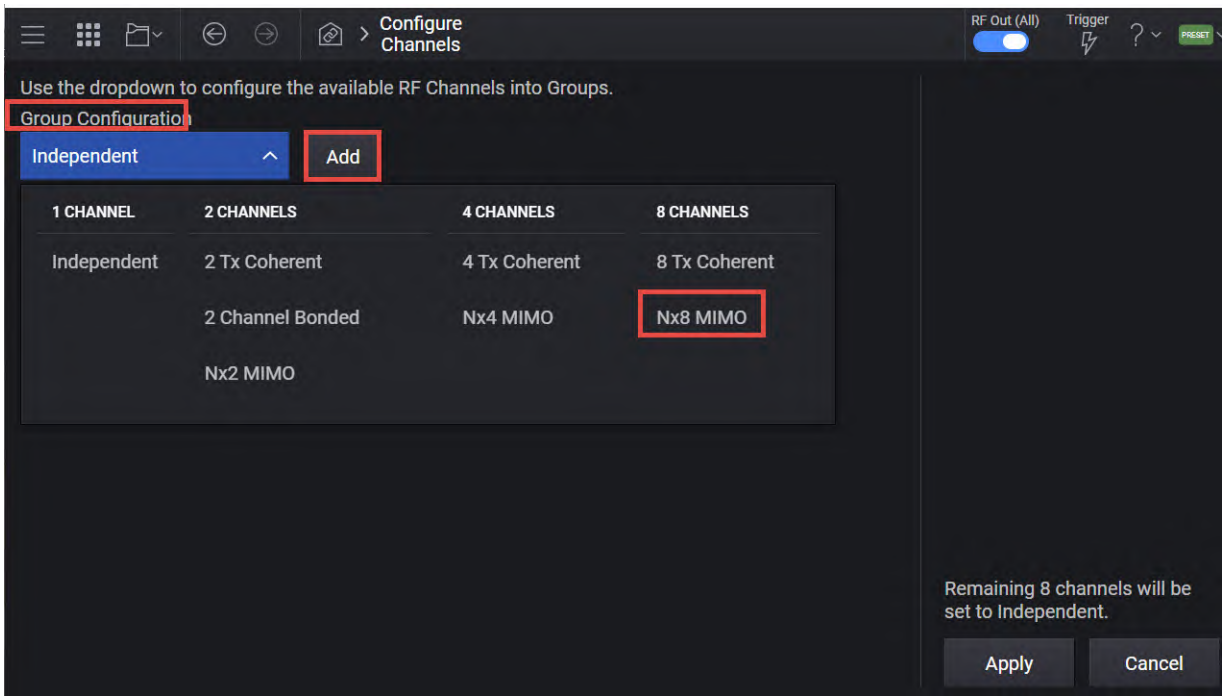
   If a Synchronization Alignment is required, indicated by 

in the bottom left corner, select the warning message and choose **Clear Alignment** and then **Perform Alignment**.



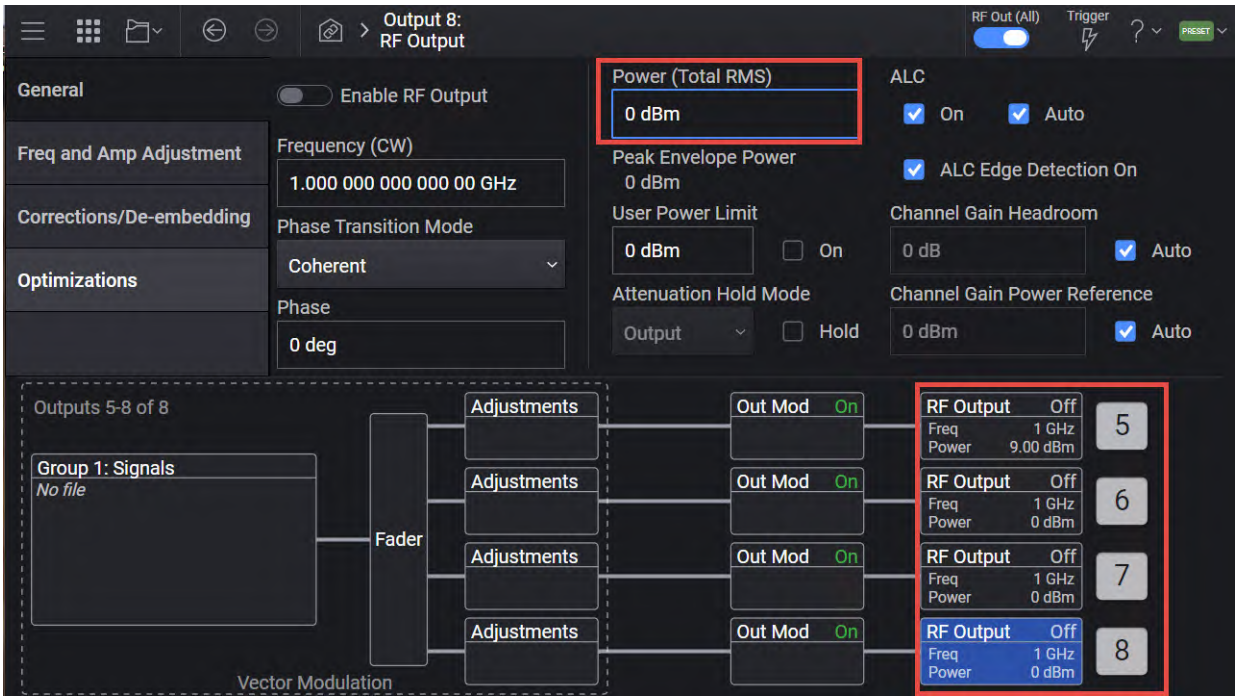
9. Select the **System** icon (triple bar on top left) > **Configure Channels**.

10. Select the Group Configuration dropdown and select **Nx8 MIMO** > **Add** > **Apply**.

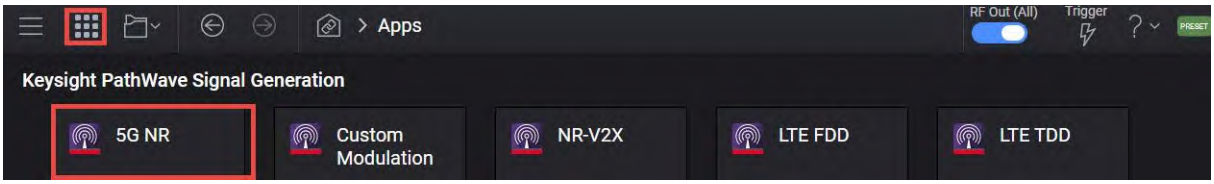


11. In the Output area for all four channels, set the Power to **0 dBm** and use the default frequency value of 1 GHz.

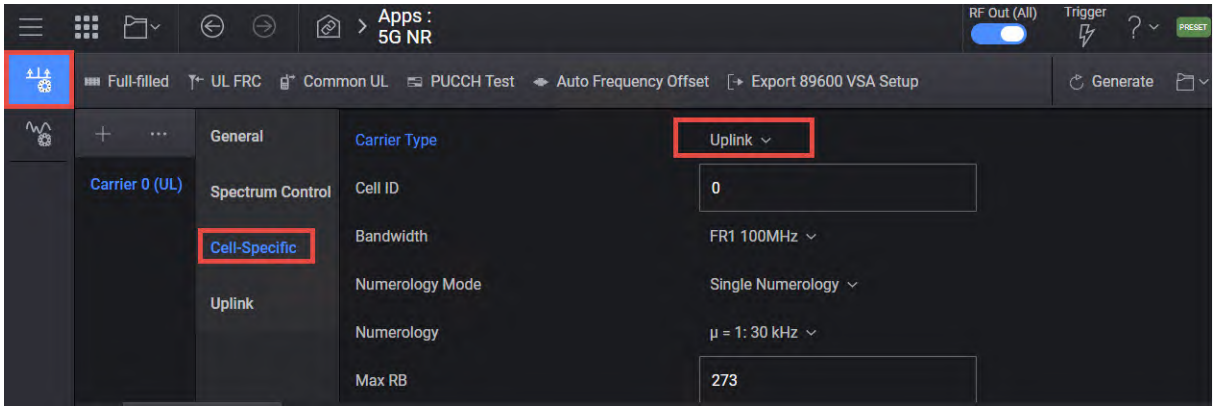
    After setting up the first 4 channels, select the **Home** icon to access channels 5 - 8 and set Power to **0 dBm**.



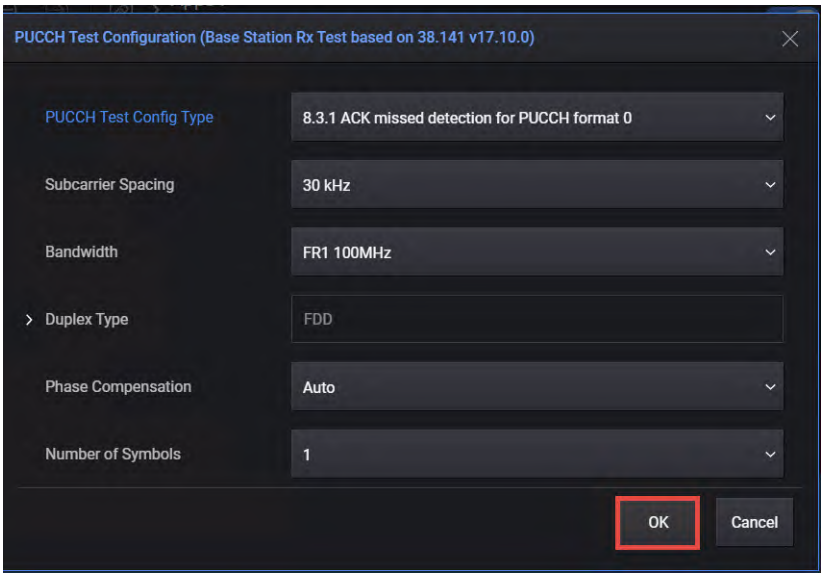
12. Select the **Apps** block to open, then select **5G NR**.



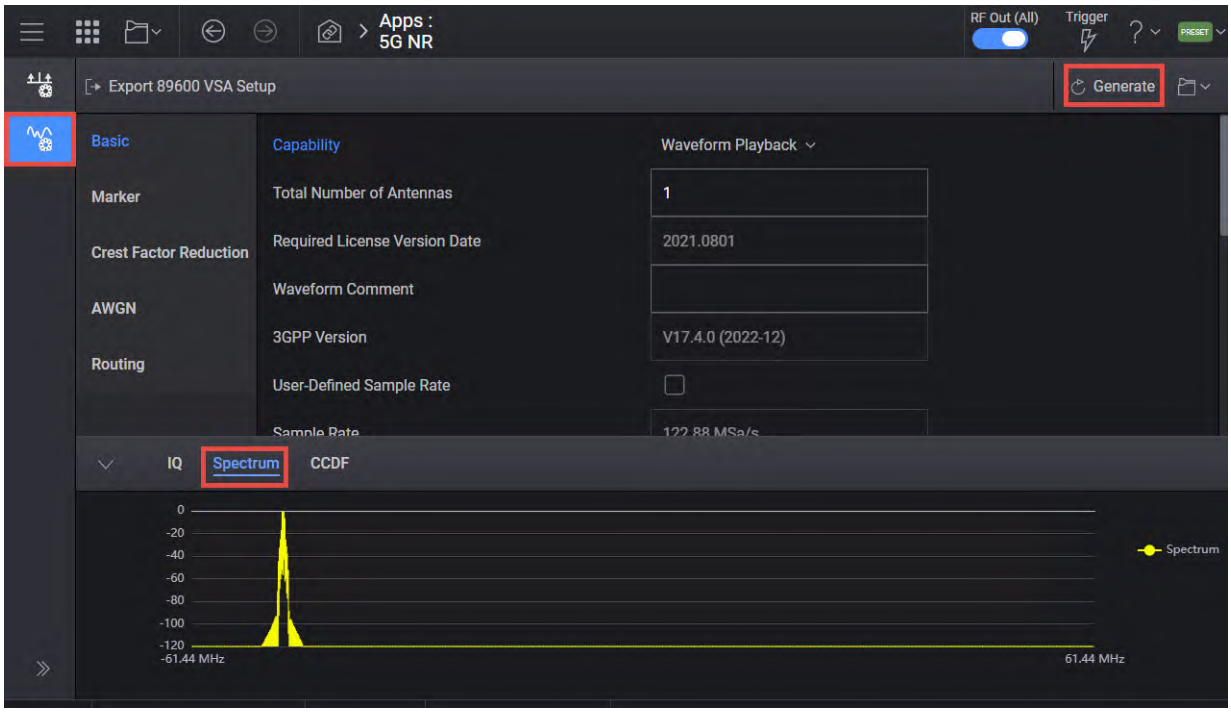
13. Select the **Carrier** tab > **Cell-Specific** and set Carrier Type to **Uplink**.



14. Select **PUCCH Test** and note the settings available for Base Station Rx Test based on 38.141 v17.10.0. For this example, we will leave them set to the default settings based on 8.3.1 PUCCH test of FR1 Bandwidth set to 100 MHz and Subcarrier Spacing of 30 kHz. Select **OK**.

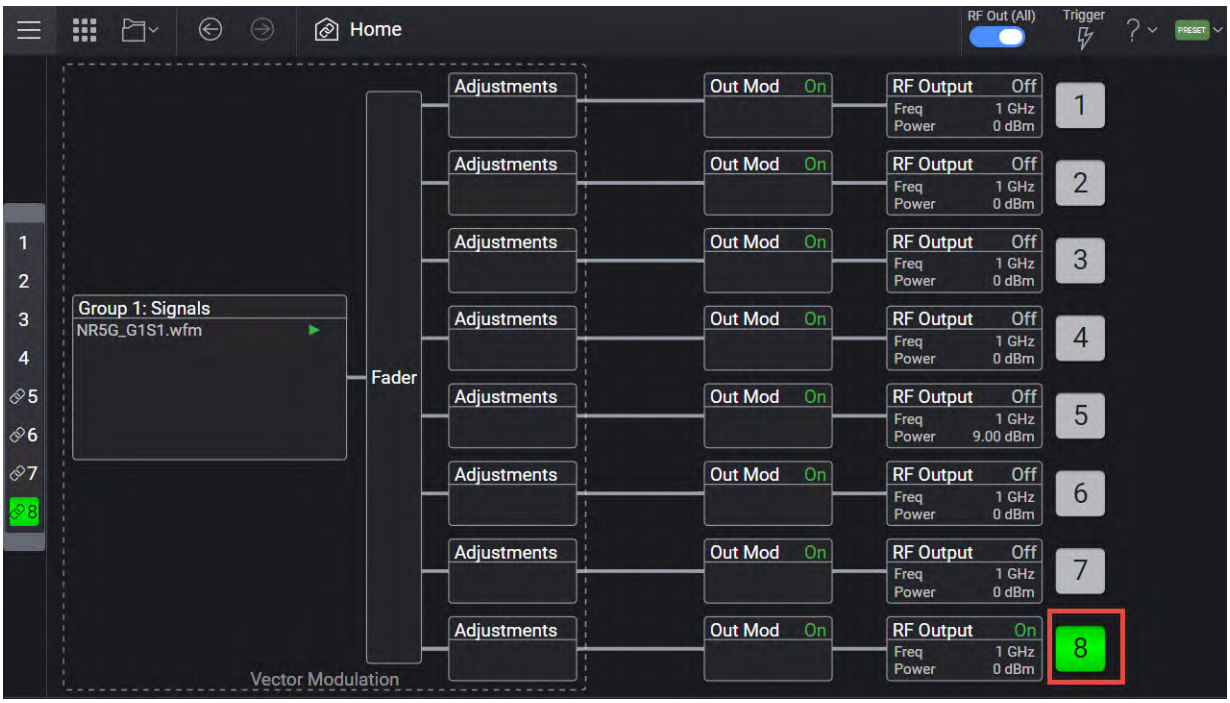


15. Select the **Waveform** tab, then **Generate** to apply the signal to each channel. The expected Spectrum can be observed in the bottom window.

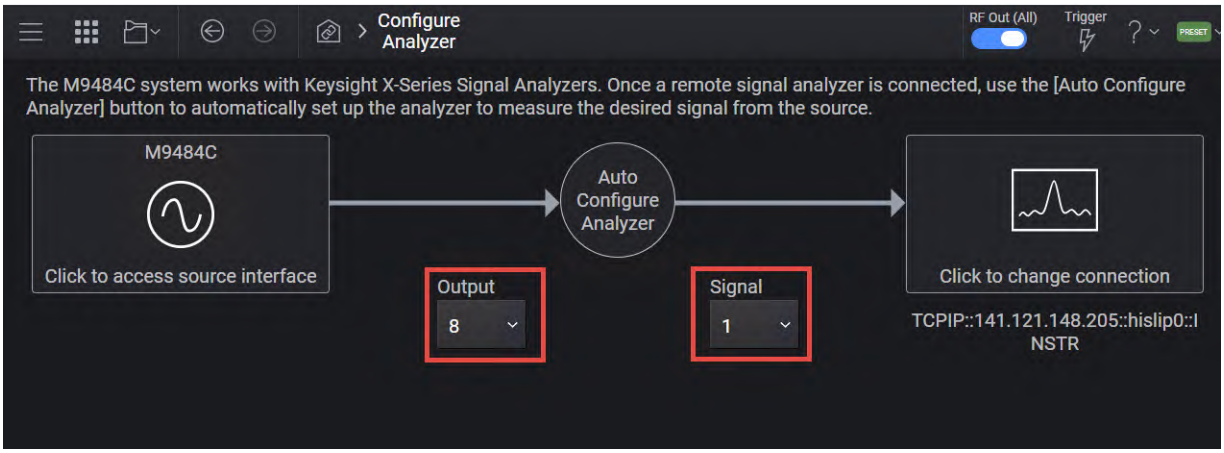


16. Select the **Home** icon and turn on one of the channels.

For this example, we will turn on **Channel 8**.



17. Select the System menu and select **Configure Analyzer** and set Output to **8** and Signal to **1**.



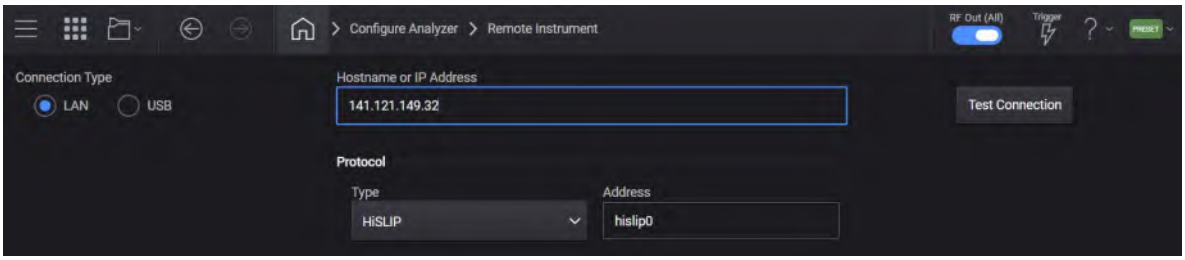
18. In the System Configuration setup, click on the Remote Signal Analyzer block (on far right) to setup the communication channel to the Keysight X-Series Signal Analyzer.

19. Set Connection Type to **LAN**.

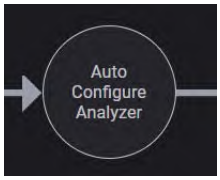
20. In the Connect Remote X-Series Signal Analyzer setup, enter the Hostname or IP Address for the signal analyzer.

21. Under Set Protocol, select **HiSLIP** and use the Remote name **hislip0**.

22. Select **Test Connection** to verify, then **Back**.



23. Select **Auto Configure Analyzer** to send the VXG settings to the analyzer.



24. Verify the configuration setup on the UXA.

You can verify the waveform for all 8 channels by turning them off and on one at a time.

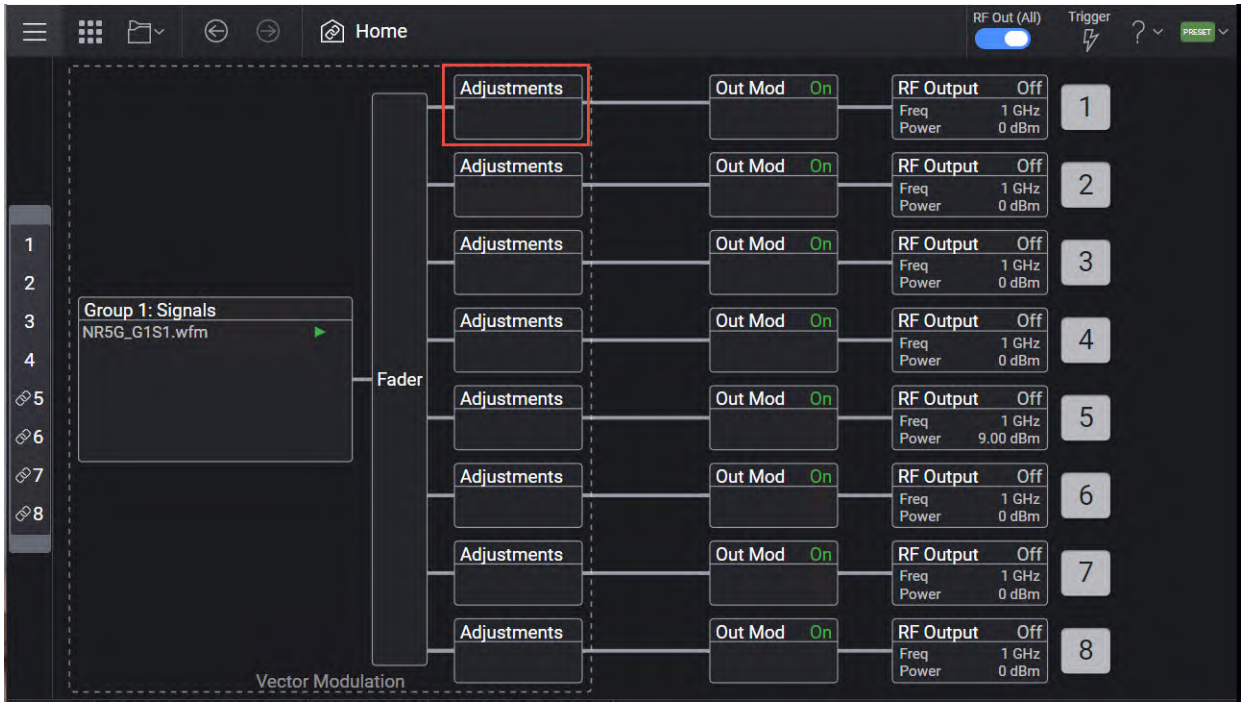
Setting Up Fading for PUCCH

Continue on from the MIMO setup above to setup this fading example.

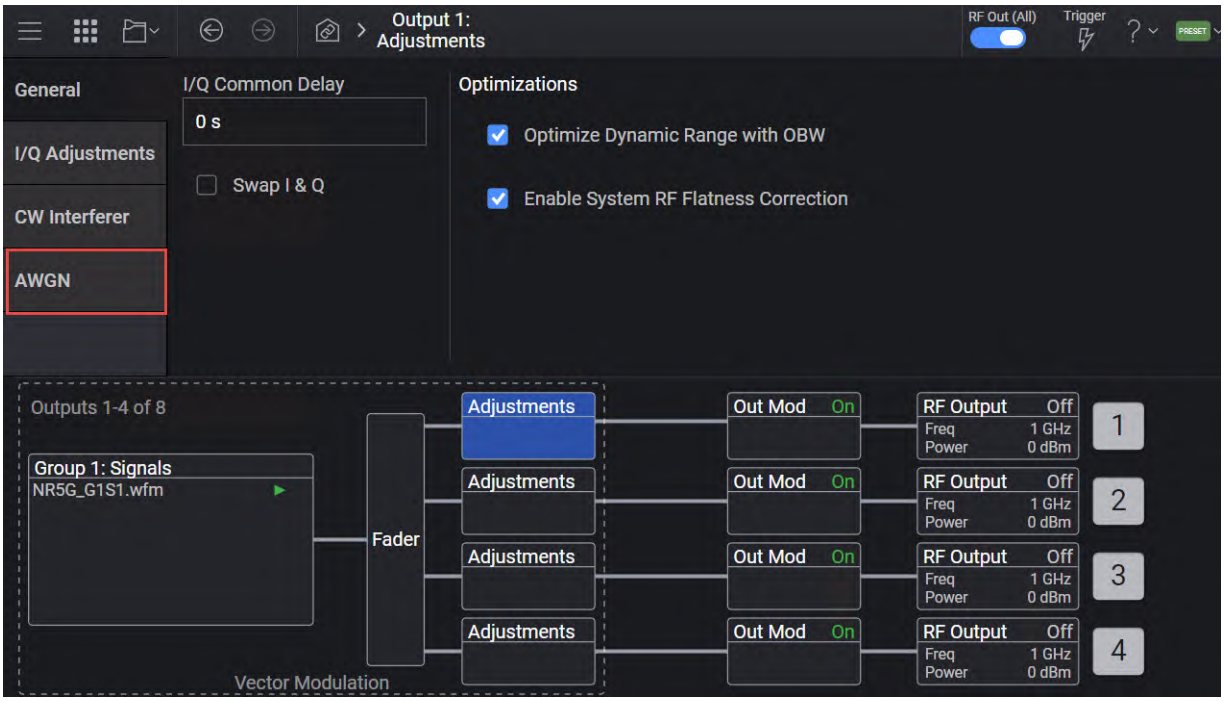
1. Select the **Fader** block and configure all 8 channels as follows:
   - Set the Channel Model to **TDLC300**
   - Select **Fade** to make it the active function
   - After setting up all 8 channels, select the **Home** icon to return to the main display

The MIMO Order should already be set to 1x8 and the Standard set to NR5G. Do not Enable Fading, this will be done in a later step.

2. Select the Channel 1 **Adjustments** block to open.

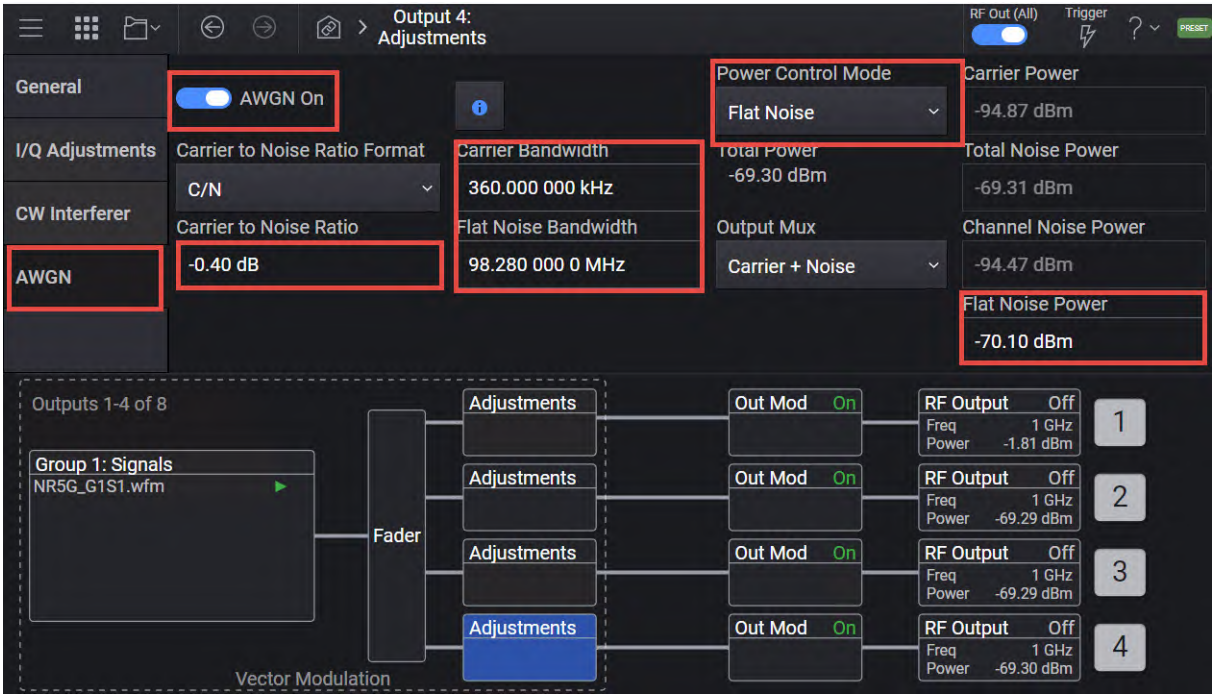


3. In the left pane, select the **AWGN** tab.



4. Set up the AWGN for **all 8 channels** as follows:
   - Power Control Mode to **Flat Noise**
   - Carrier to Noise Ration to **-0.40 dB**
   - Carrier Bandwidth to **360 kHz**
   - Flat Noise Bandwidth to **98.280 MHz**
   - Flat Noise Power to **-70.1 dBm**

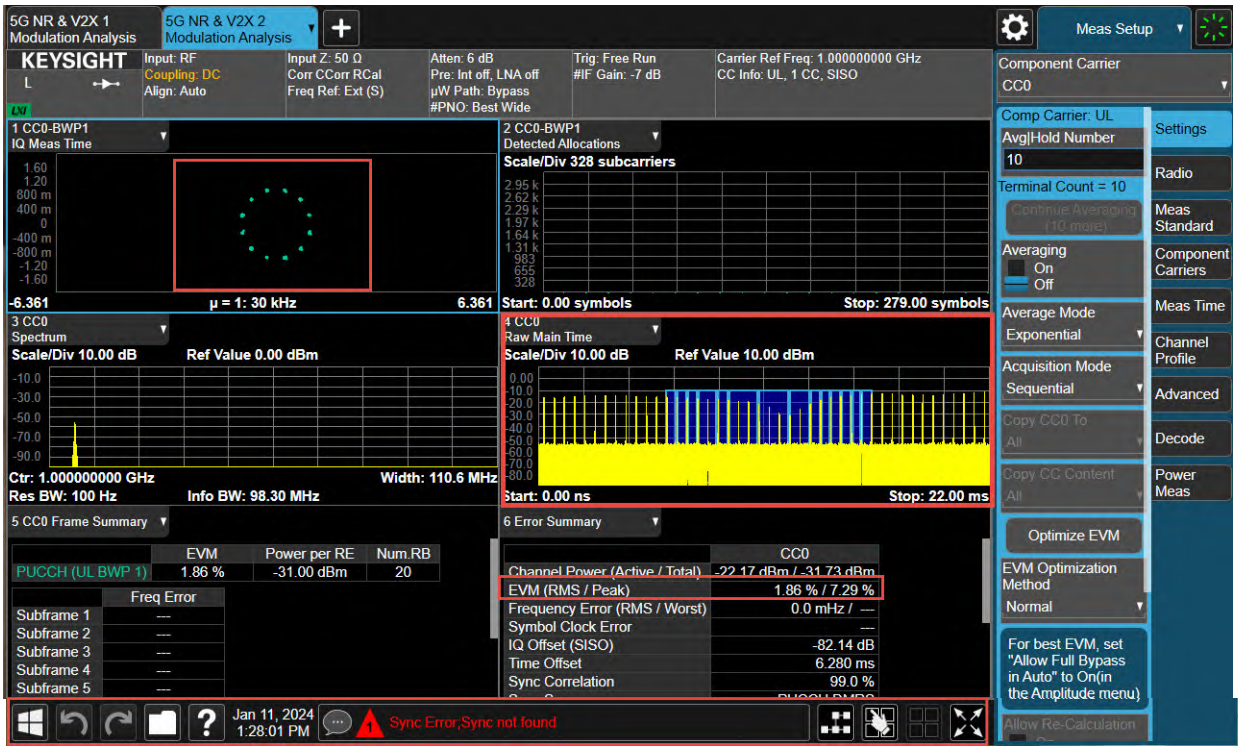
– Set **AWGN On**



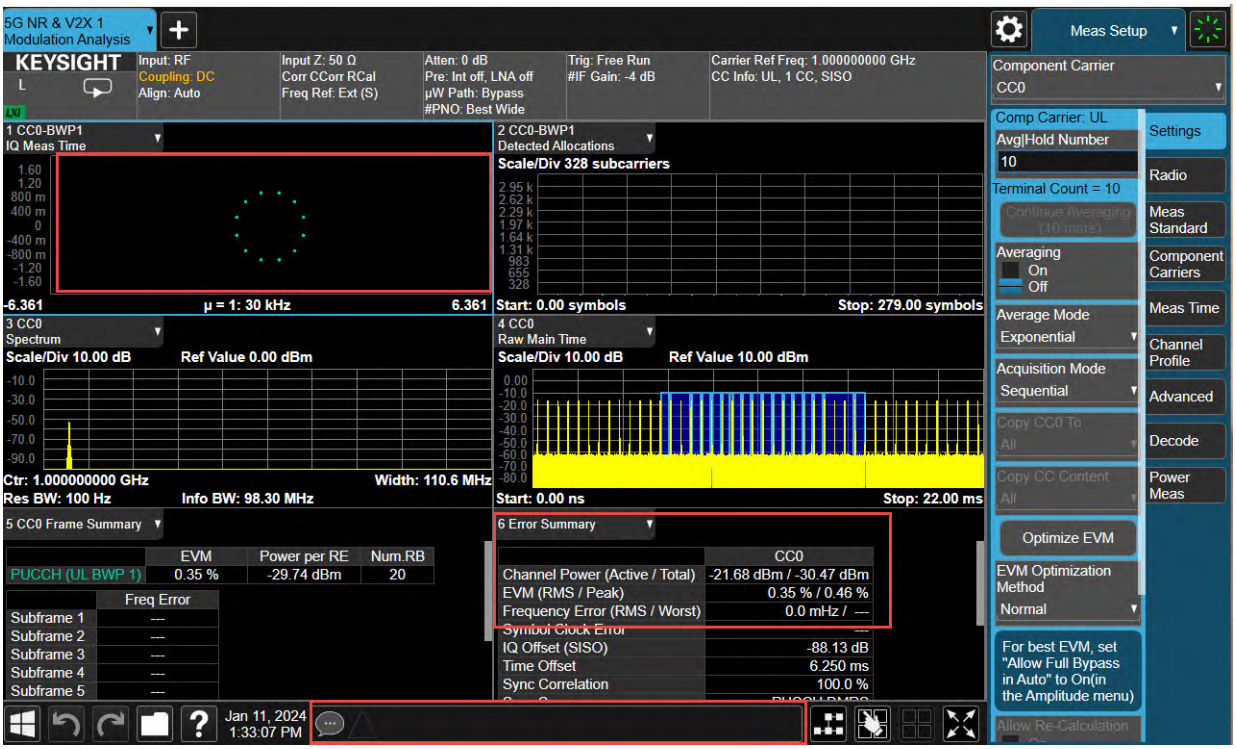
5. Select Channel 1 **RF Output** block and set Output Phase Transition to **Continuous**. Repeat for Channels 2 - 8.
6. Select the **Fader** block and **Enable Fading**.

   This completes the Fading setup. You will not see any signal on the UXA as the signal level is too low.
7. In order to see the affects of **Fading**:
   – Set **RF Output** on Channel 1 to **-10 dBm**.
   – Select Channel 1 **Adjustments** block > **AWGN** > and turn off **AWGN**.

You may notice that the Raw Main Time data varies in amplitude, the Constellation shift in the IQ Meas Time display, the EVM value increases (For this example, it went from .35% RMS to 1.86% RMS), or the error message "Sync Error; Sync not found" is displayed at the bottom of the screen.



8. Select the **Fader** block and turn **Fading off**.


5G NR & V2X 1
Modulation Analysis
KEYSIGHT
Input: RF
Coupling: DC
Align: Auto
Input Z: 50 Ω
Corr CCorr RCal
Freq Ref: Ext (S)
Atten: 0 dB
Pre: Int off, LNA off
µW Path: Bypass
#PNO: Best Wide
Trig: Free Run
#IF Gain: -4 dB
Carrier Ref Freq: 1.000000000 GHz
CC Info: UL, 1 CC, SISO
Meas Setup
Component Carrier
CC0
1 CC0-BWP1
IQ Meas Time
-6.361
µ = 1: 30 kHz
6.361
2 CC0-BWP1
Detected Allocations
Scale/Div 328 subcarriers
Start: 0.00 symbols
Stop: 279.00 symbols
3 CC0
Spectrum
Scale/Div 10.00 dB
Ref Value 0.00 dBm
Ctr: 1.000000000 GHz
Width: 110.6 MHz
Res BW: 100 Hz
Info BW: 98.30 MHz
4 CC0
Raw Main Time
Scale/Div 10.00 dB
Ref Value 10.00 dBm
Start: 0.00 ns
Stop: 22.00 ms
5 CC0 Frame Summary
EVM
Power per RE
Num.RB
PUCCH (UL BWP 1)
0.35 %
-29.74 dBm
20
Freq Error
Subframe 1
Subframe 2
Subframe 3
Subframe 4
Subframe 5
6 Error Summary
CC0
Channel Power (Active / Total)
-21.68 dBm / -30.47 dBm
EVM (RMS / Peak)
0.35 % / 0.46 %
Frequency Error (RMS / Worst)
0.0 mHz /
IQ Offset (SISO)
-88.13 dB
Time Offset
6.250 ms
Sync Correlation
100.0 %
Comp Carrier: UL
Avg|Hold Number
10
Terminal Count = 10
Continue Averaging (10 more)
Averaging
On
Off
Average Mode
Exponential
Acquisition Mode
Sequential
Copy CC0 To
All
Copy CC Content
All
Optimize EVM
EVM Optimization Method
Normal
For best EVM, set "Allow Full Bypass in Auto" to On(in the Amplitude menu)
Allow Re-Calculation
Settings
Radio
Meas Standard
Component Carriers
Meas Time
Channel Profile
Advanced
Decode
Power Meas
Jan 11, 2024
1:33:07 PM

Setting a 1x4 MIMO and Fading HARQ Example

This example shows how to do the Chapter 8 Base Station test. We will use an N5182B MXG to simulate a base station. The MXG will send the HARQ feedback signal out of the BB TRG1port. The Event 1 port will provide the 10 ms frame pulse. We will use the Chapter 8.2.1 PUSCH test with Transform Precoding disabled. It requires a MIMO Fading configuration with HARQ feedback.

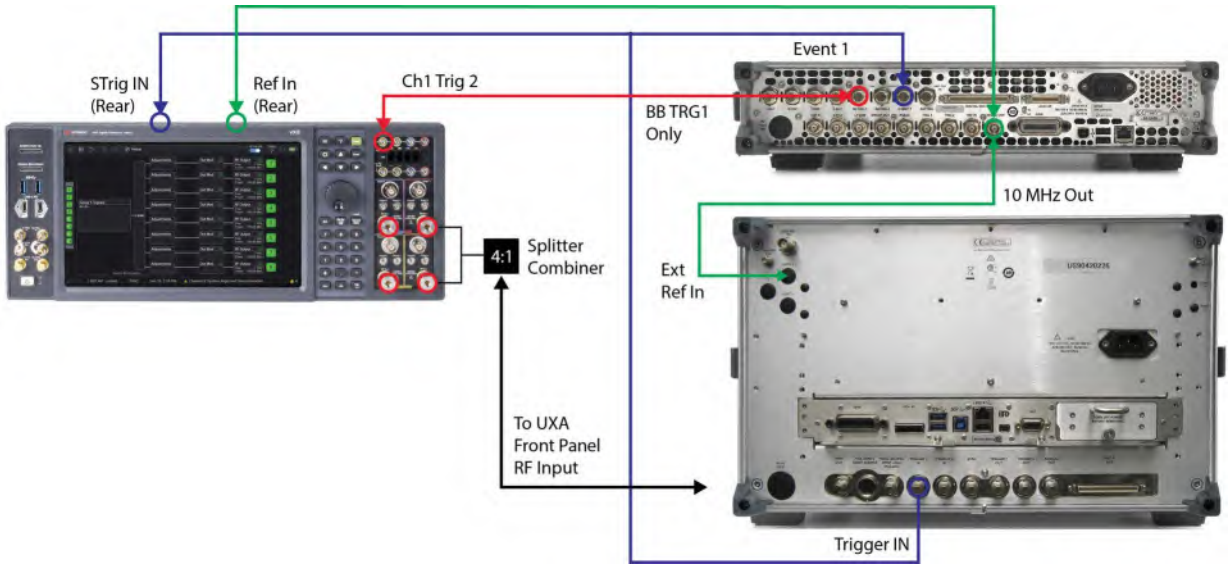
Test Purpose

Receiver's ability to achieve throughput under multipath fading propagation condition for a given Signal to Noise Ratio (SNR).

Conditions

- Assumes HARQ retransmission
- Transform Precoding is disabled
- RB (Resource Block) assignment uses full applicable bandwidth

Equipment Setup



- M9484C/N5186A
  - Four channels (M9484C-004) with options PCH and SNC
  - N7631APPC PathWave Signal Generation for 5G NR
  - N7605AP0C PathWave Signal Generation for 3GPP Real-time Fading
- N9042B UXA

  N9042B UXA Firmware version 2023 (XA34) or higher

  - N9085EMOE 5G NR Measurement application
- 4:1 Power Splitter/Combiner

Basic Steps

1. Connect wanted signal, fading simulator, AWGN
2. Configure AWGN according to table 8.2.1.4.2-1
3. Configure wanted signal FRC and adjust carrier power to have specified SNR.
   Carrier Power = AWGN power = SNR
4. Configure channel model for fading simulator
5. Measure throughput on gNB

Table 8.2.1.4.2-1: AWGN power level at the BS input

Sub-carrier spacing (kHz)	Channel bandwidth (MHz)	AWGN power level
15 kHz	5	-86.5 dBm / 4.5MHz
	10	-83.3 dBm / 9.36MHz
	20	-80.2 dBm / 19.08MHz
30 kHz	10	-83.6 dBm / 8.64MHz
	20	-80.4 dBm / 18.36MHz
	40	-77.2 dBm / 38.16MHz
	100	-73.1 dBm / 98.28MHz

Table 8.2.1.5-7: Test requirements for PUSCH with 70% of maximum throughput, Type A, 100 MHz channel bandwidth, 30 kHz SCS

Tx Ant	Rx Ant	Cyclic prefix	Channel Model	Max Throughput	FRC (annex A)	DMRS AddPos	SNR (dB)
1	2	Normal	TDLB100-400 Low	70 %	G-FR1-A3-14	pos1	-2.2
		Normal	TDLC300-100 Low	70 %	G-FR1-A4-14	pos1	10.8
		Normal	TDLA30-10 Low	70 %	G-FR1-A5-14	pos1	13.6
	4	Normal	TDLB100-400 Low	70 %	G-FR1-A3-14	pos1	-5.2
		Normal	TDLC300-100 Low	70 %	G-FR1-A4-14	pos1	7.1
		Normal	TDLA30-10 Low	70 %	G-FR1-A5-14	pos1	9.6
	8	Normal	TDLB100-400 Low	70 %	G-FR1-A3-14	pos1	-8.1
		Normal	TDLC300-100 Low	70 %	G-FR1-A4-14	pos1	3.8
		Normal	TDLA30-10 Low	70 %	G-FR1-A5-14	pos1	6.4
2	2	Normal	TDLB100-400 Low	70 %	G-FR1-A3-28	pos1	2.2
		Normal	TDLC300-100 Low	70 %	G-FR1-A4-28	pos1	20.0
	4	Normal	TDLB100-400 Low	70 %	G-FR1-A3-28	pos1	-1.4
		Normal	TDLC300-100 Low	70 %	G-FR1-A4-28	pos1	12.4
	8	Normal	TDLB100-400 Low	70 %	G-FR1-A3-28	pos1	-4.4
		Normal	TDLC300-100 Low	70 %	G-FR1-A4-28	pos1	7.9

In this example we will use 30 kHz sub carrier spacing, 100 MHz channel bandwidth. We will use a 1x4 MIMO configuration, the TDLB100-400 channel model, FRC G-FR1-A3-14, and SNR of -5.2 dB.

Setting up the MXG

Two setup files has been included to have the MXG mimic the characteristics of a Base Station. Follow the steps below to load the setup file on the MXG.

TIP You must download these to your PC to run this demonstration. Go to

`D:\Users\Instrument\Documents\Keysight\PathWave \SignalGenerator\Examples\HARQ playback for BaseStation Fading measurement_AuxPattern0_N7622C.scp and AuxPattern16_R16_1p92M_RV0_1_2_3_per500us_10ns_BaseStation Fading Measurement.tx.` Download these files to your PC.

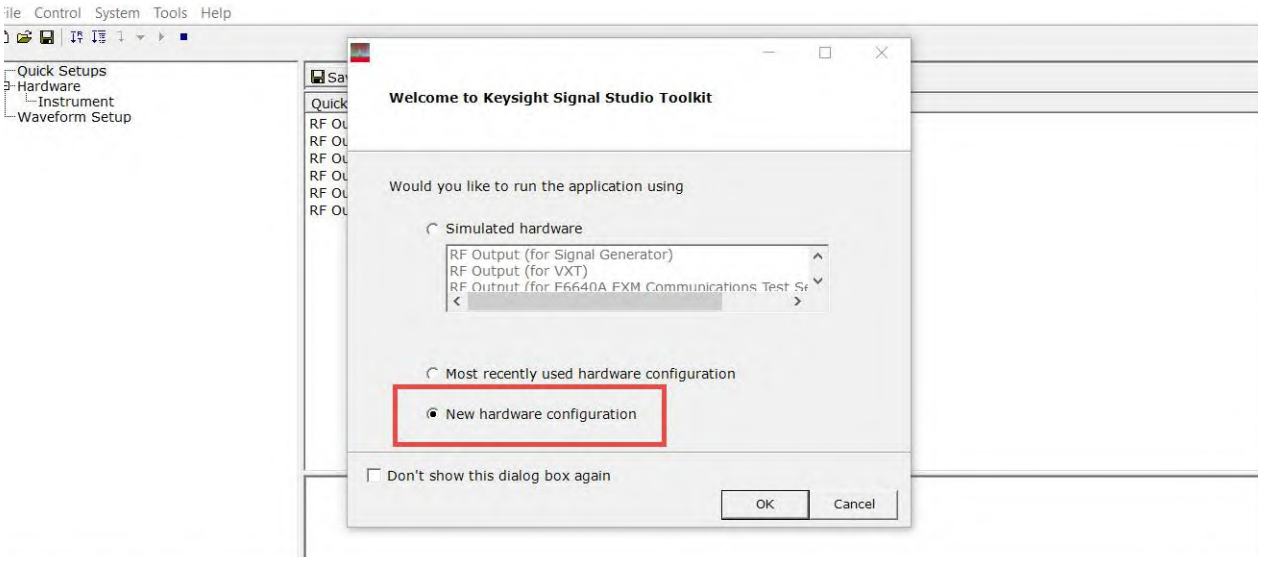
1. Go to:

   https://www.keysight.com/us/en/lib/software-detail/instrument-firmware-software/n7622c-signal-studio-toolkit-software-2207782.html

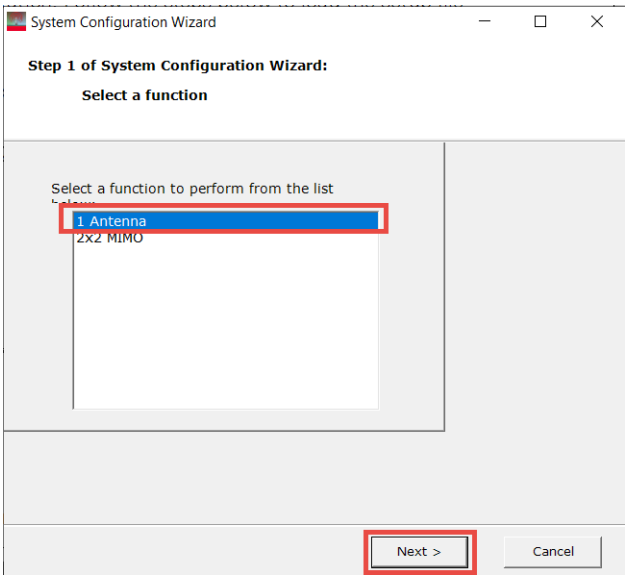
   and download the N7622C signal Studio Tool Kit to your PC.

2. Go to the Windows Start menu and select **Toolkit 2022**.

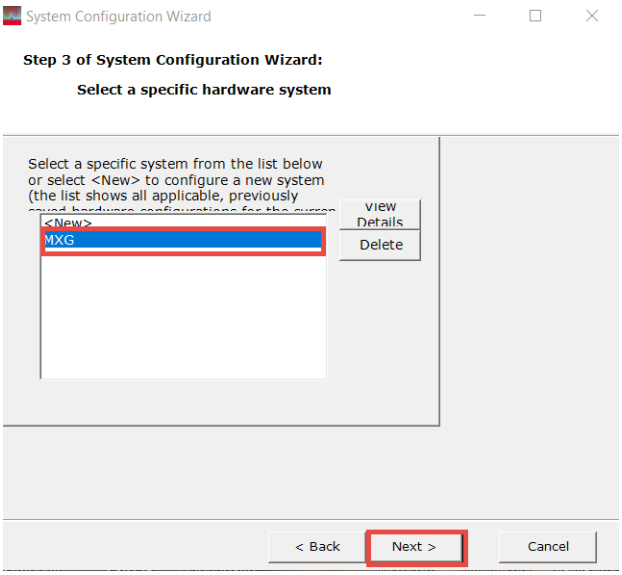
3. In the Keysight Signal Studio Toolkit Welcome screen, select New **Hardware Configuration**.



4. Select **1 Antenna** > **Next**.

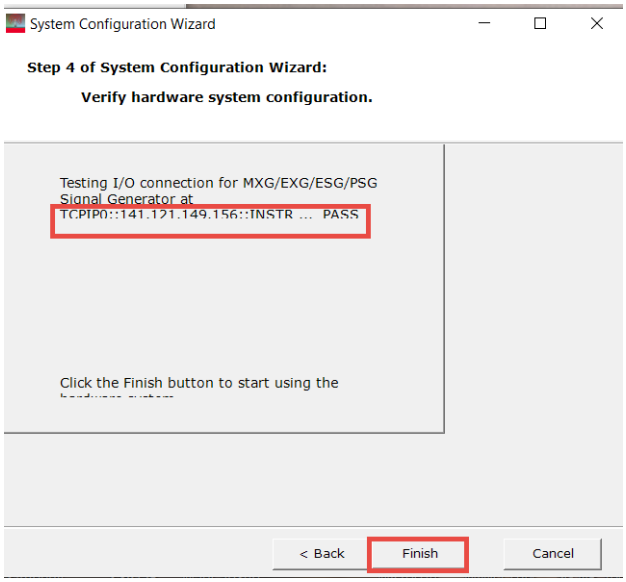


5. Select **MXG** > **Next**.



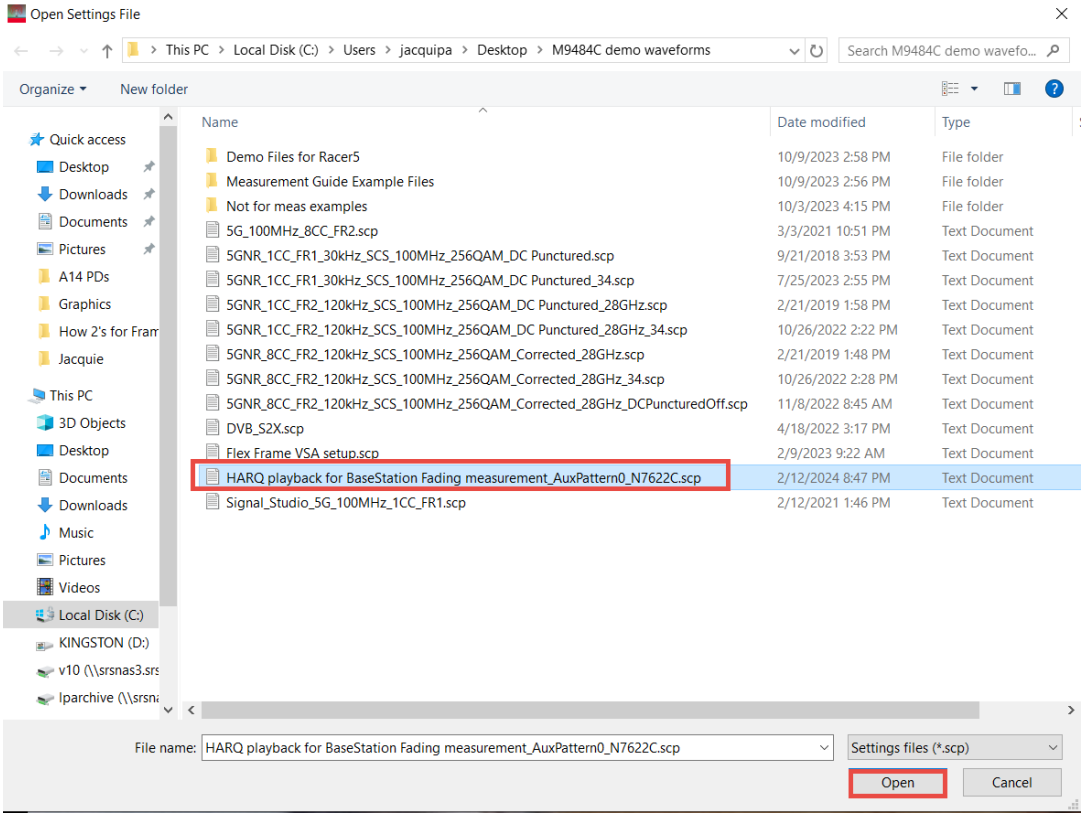
6. Enter the IP address of the MXG, then select **Next**.

7. Test the connection to the MXG and then select **Finish**.



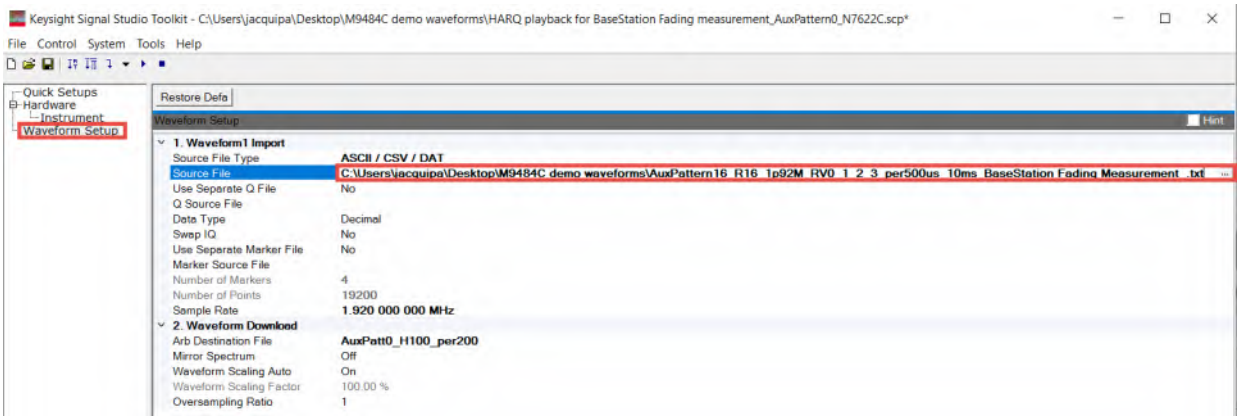
8. In the Signal Studio Toolkit, select **File** > **Open Settings File** and navigate to the settings file that is included in the Examples folder on the VXG.

   `HARQ playback for BaseStation Fading measurement_AuxPattern0_N7622C.scp.`

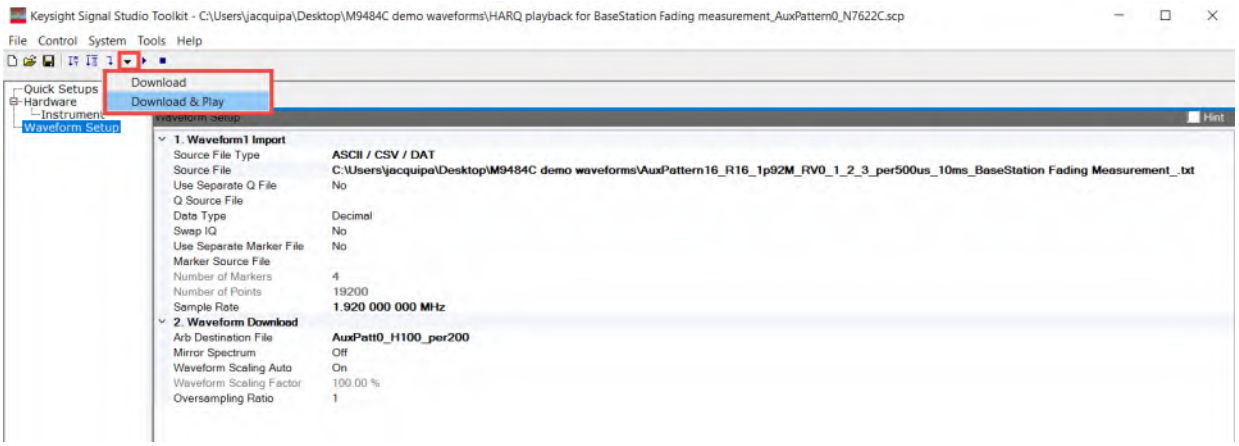


9. In the Signal Studio Toolkit, select the **Waveform Setup** tab in the left pane. In the Waveform Setup area, select the ellipsis (...) on the far right and select the text file that is included in the Examples folder on the VXG.

   `AuxPattern16_R16_1p92M_RV0_1_2_3_per500us_10ns_BaseStation Fading Measurement.txt.`

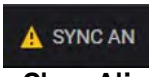


10. At the top of the display select the down arrow and select **Download & Play**.

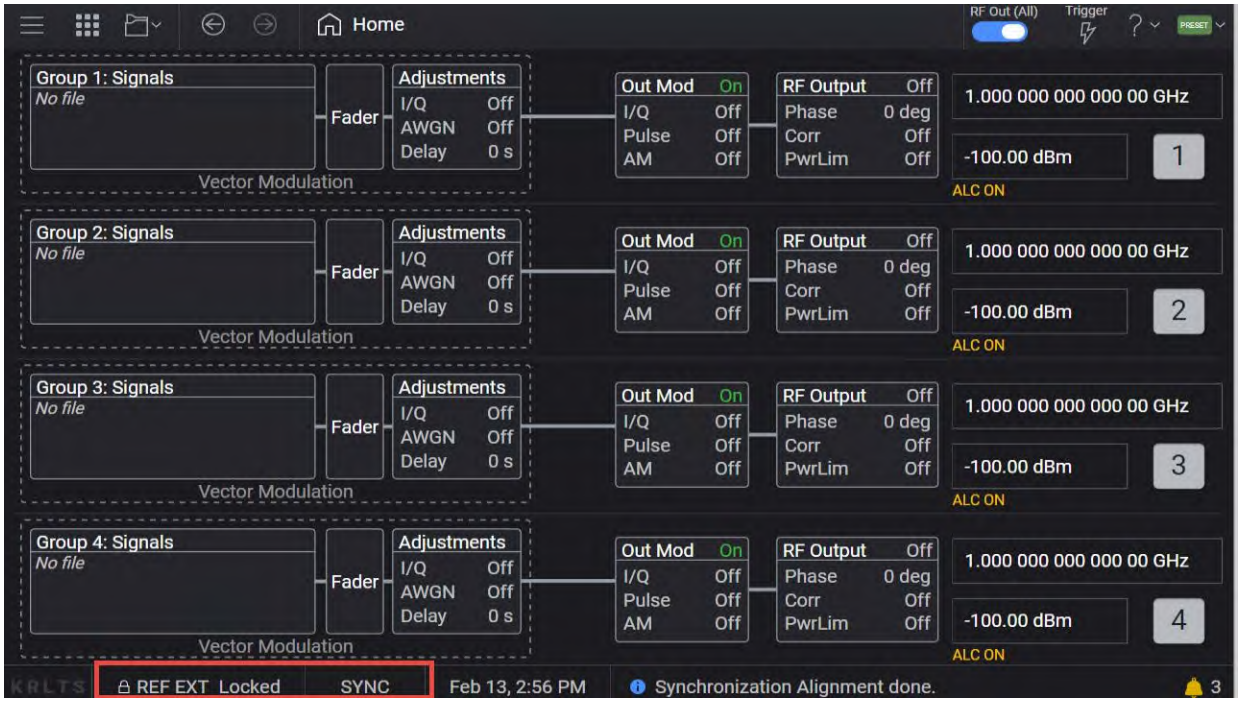


Setting Up the MXG/VXG

1. Select **Preset** > **Preset** to set the instrument to a known state.
2. Select the **System** menu (triple bar icon) > **Settings** > **Reference** > and set Source to **External**.

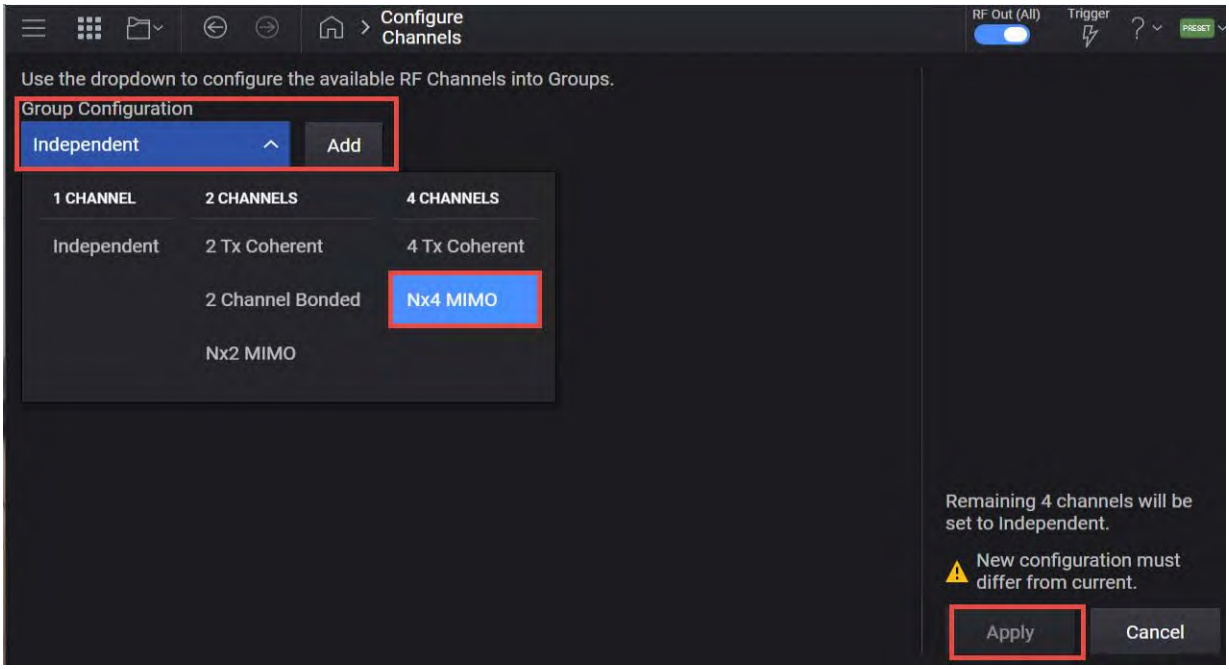
3. If a Synchronization Alignment is required, indicated by 

in the bottom left side, select the warning message and choose **Clear Alignment** and then **Perform Alignment**.



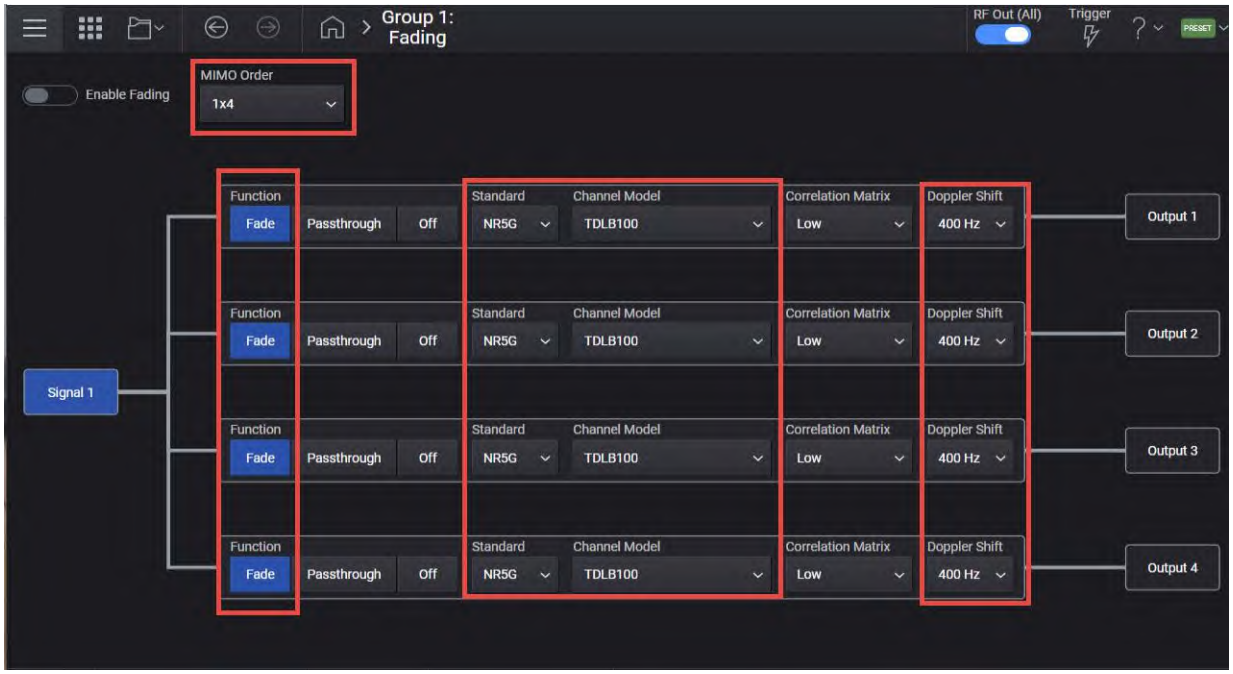
4. Select the **System** icon (triple bar on top left) > **Configure Channels**.

5. Select the Group Configuration dropdown and select **Nx4 MIMO** > **Add** > **Apply**.

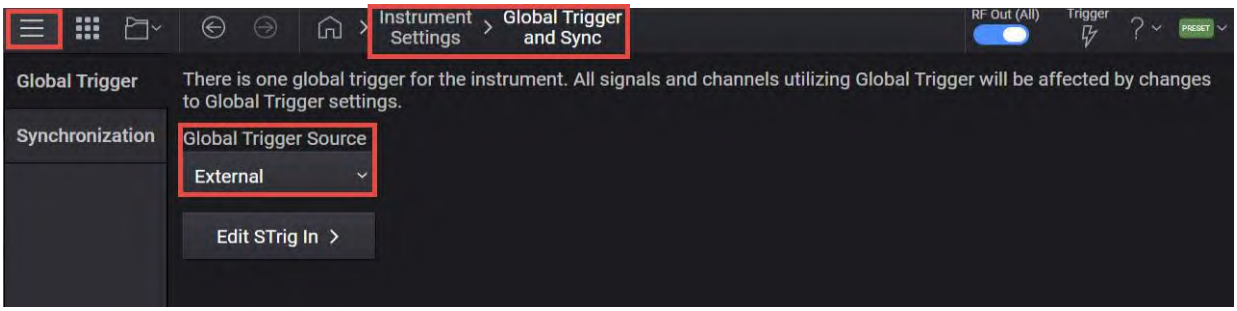


6. Select the **Fader** block > **MIMO Order** dropdown and select **1x4.**
7. Configure all 4 channels as follows:
   - Ensure the Standard is set to **NR5G**
   - Set the Channel Model to **TDLB100** which will automatically set the Doppler Shift to 400 Hz.
   - Select **Fade** to make it the active function.

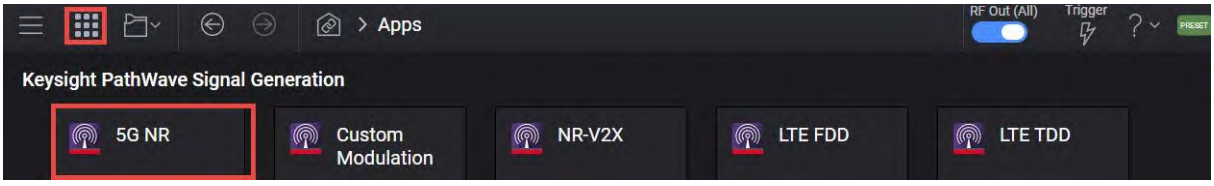
Do not Enable Fading, this will be done in a later step. Later, if you are doing some troubleshooting you can switch the Function from Fade to Passthrough.



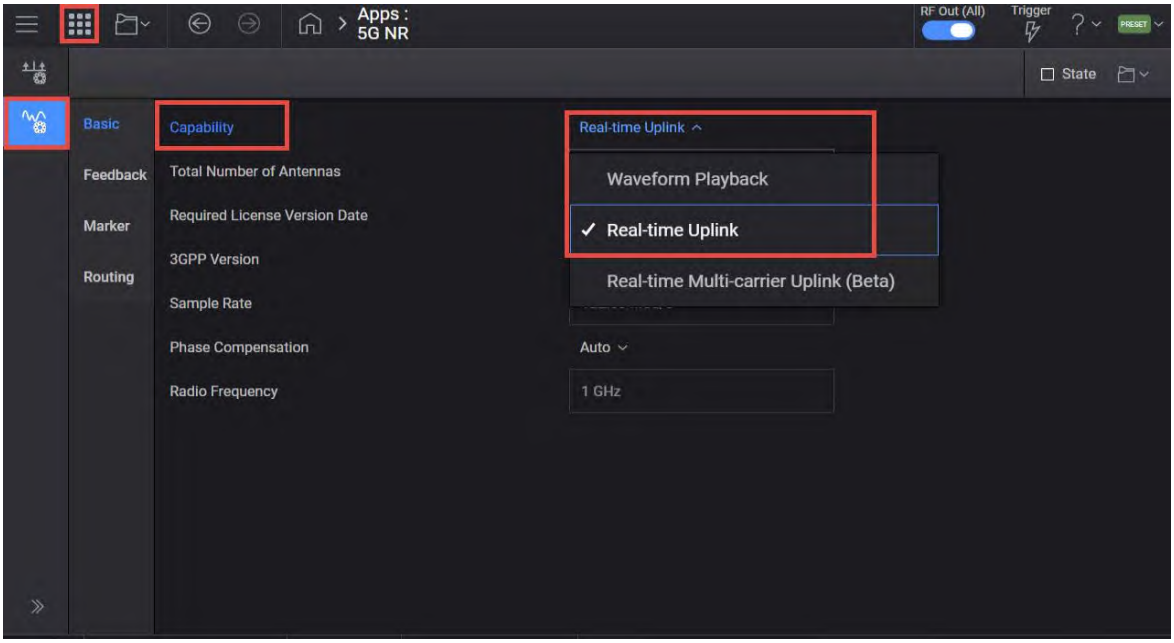
8. Select the **System** icon > **Settings** > **Global Trigger and Sync** and set Global Trigger Source to **External** to synchronize to the Base Station.



9. To configure the FR3 annex G-FR1-A3-14, select the **Apps** block to open, then select **5G NR**.

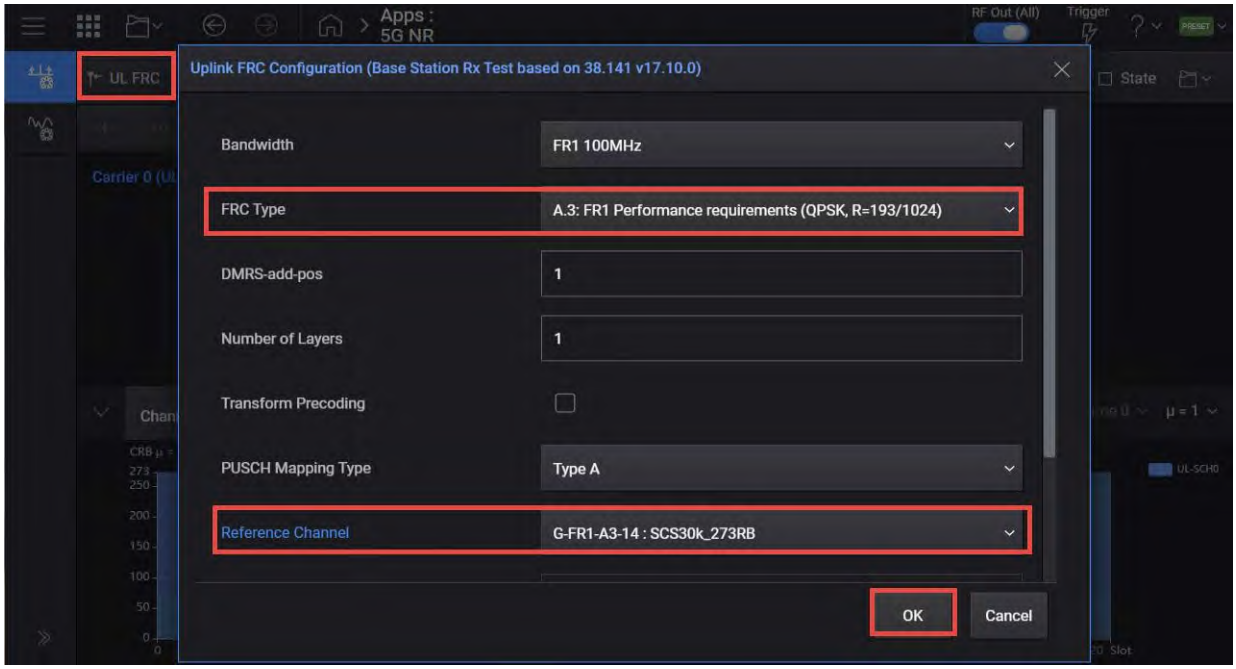


10. To setup the HARQ test, select the **Waveform** tab in the left pane then select the **Capability** dropdown and set to **Real-time Uplink**.

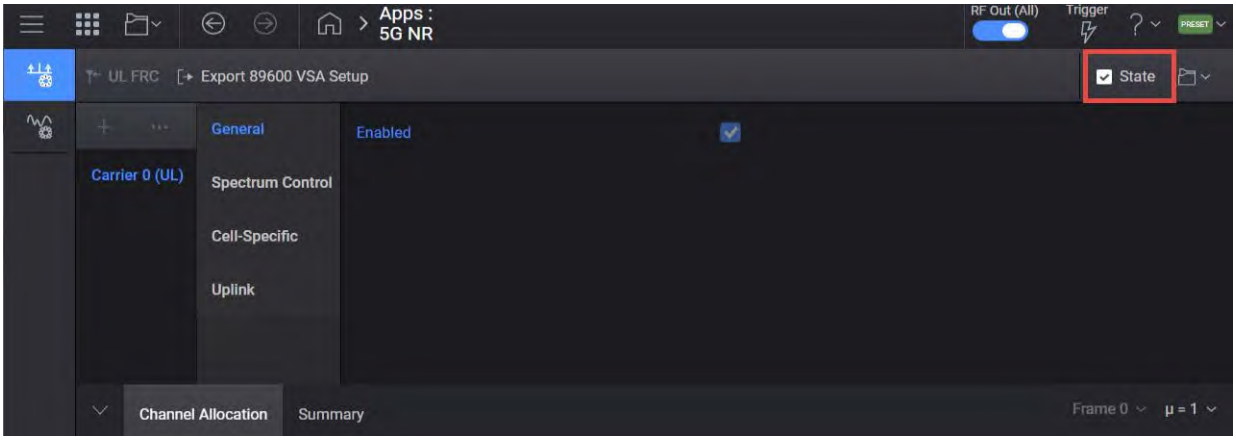


11. Select the **Carrier** tab > **UL FRC** dropdown and set:

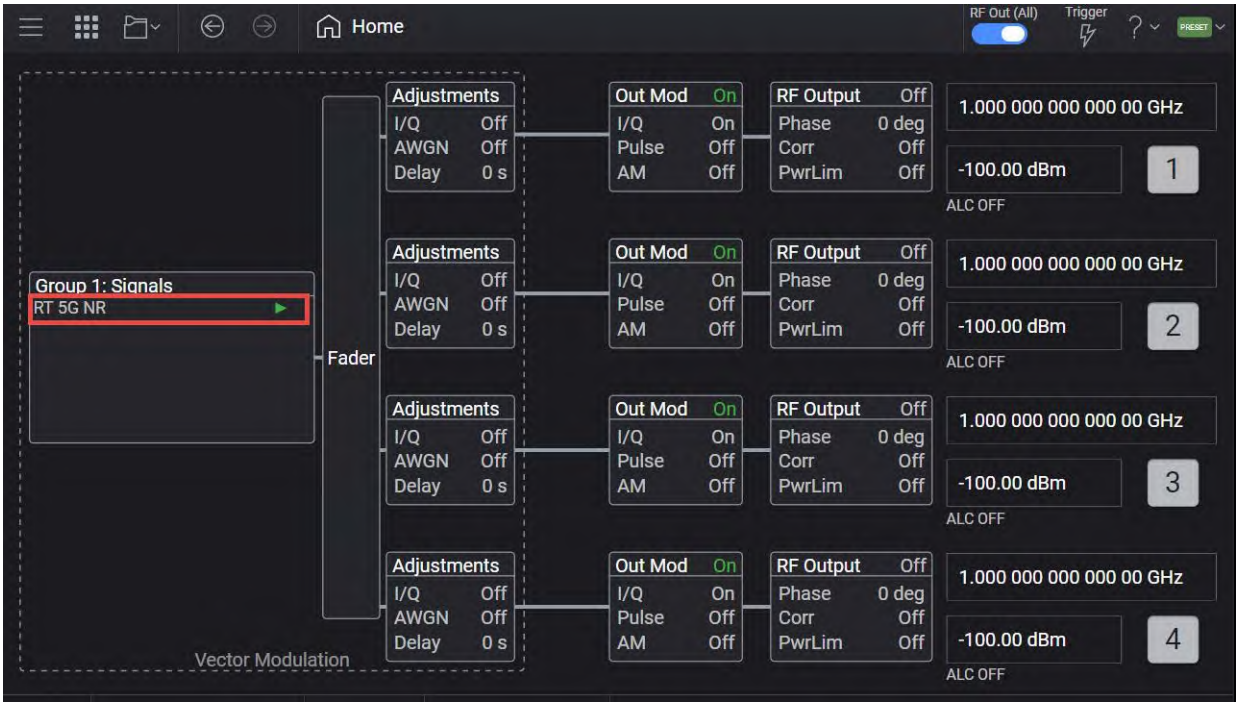
- FRC Type to **A.3:FR1 Performance requirements (QPSK, R=193/1024)**
- Reference Channel to **G-FR1-A3-14:SCS30k_273RB**
- select **OK** to configure



12. At the top right corner, select the **State** checkbox to Enable. Note: It will take a couple of minutes for the checkbox to update.



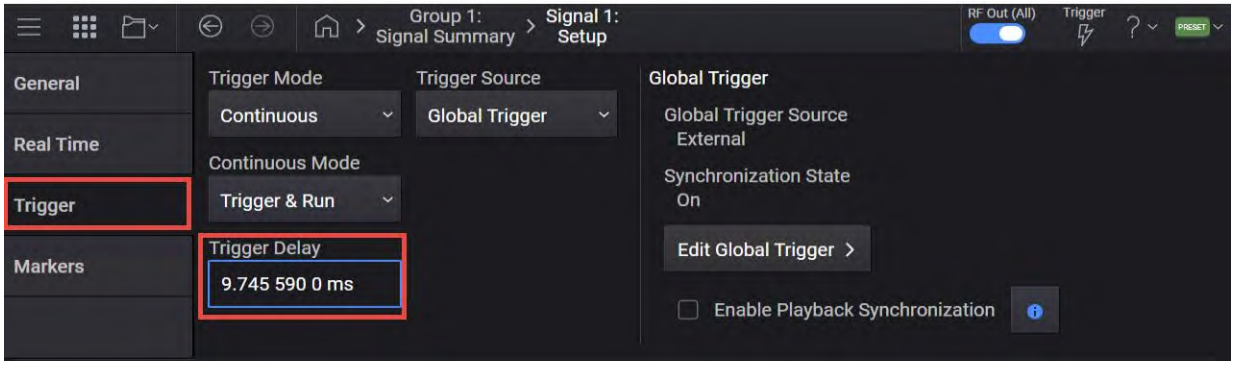
13. Select the Home icon. Notice the **RT 5G NR** signal is already playing.



14. To setup the trigger delay, select the **Signals** block and clear the checkbox next to **RT:5G NR** and select the right arrow to open the Signal Setup.

15. Select the **Trigger** tab and set Trigger Delay to **9.745590 ms**.

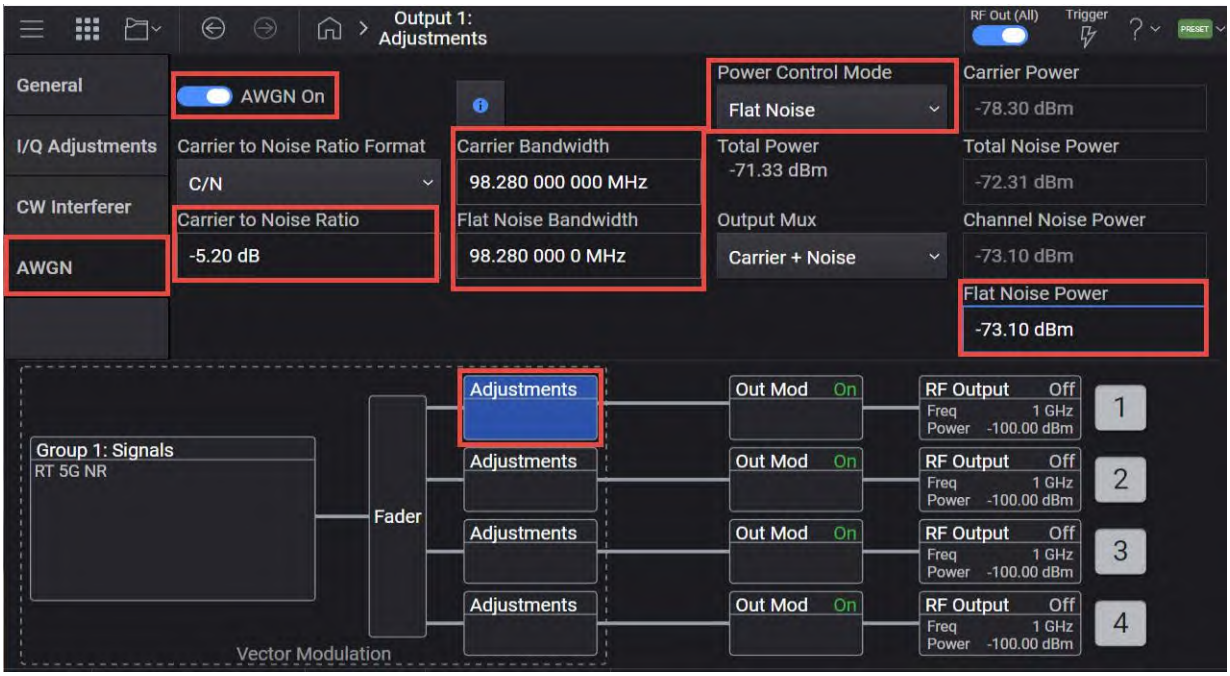
    This is to ensure that Frame timing is aligned with the VXG signal output as there is a latency with the VXG so we want to align the VXG with the Base Station with the next time frame.



16. Select the Channel 1 **Adjustments** block > **AWGN** tab and setup the AWGN on **all 4 channels** as follows:

    - Power Control Mode to **Flat Noise**
    - Carrier to Noise Ration to **-5.20 dB**
    - Carrier Bandwidth to **98.280 MHz**
    - Flat Noise Bandwidth to **98.280 MHz**
    - Flat Noise Power to **-73.1 dBm**

– Set **AWGN On**

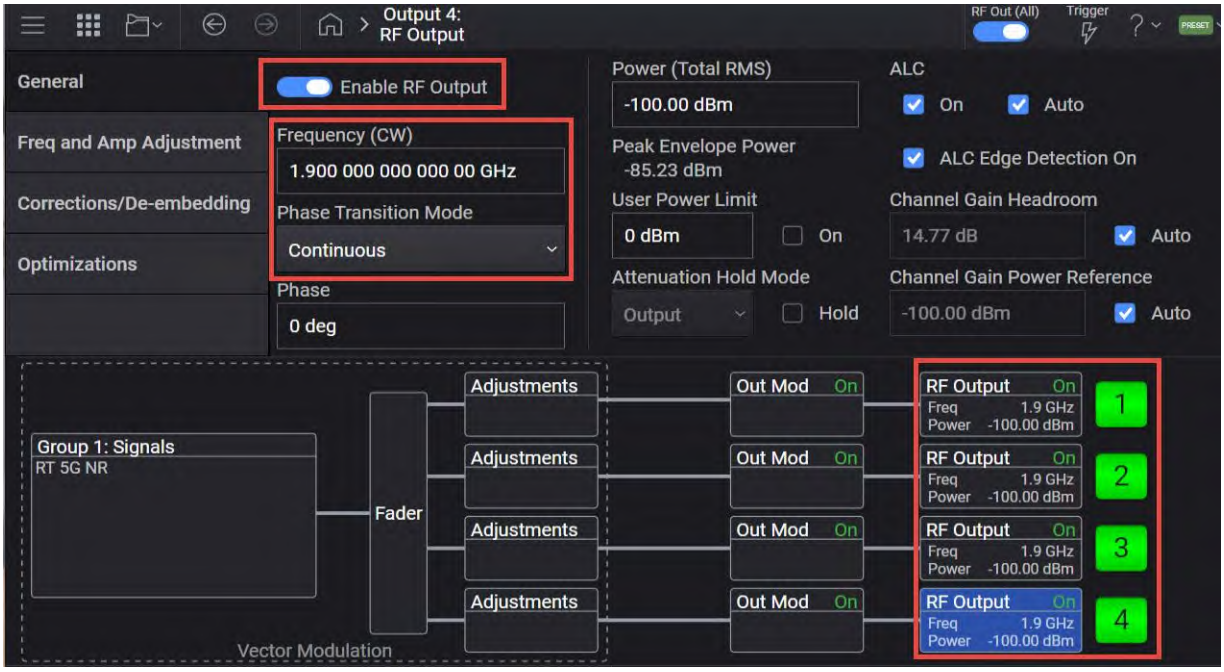


17. To configure the RF Output, select the **RF Output** block and set all **4 channels** as follows:

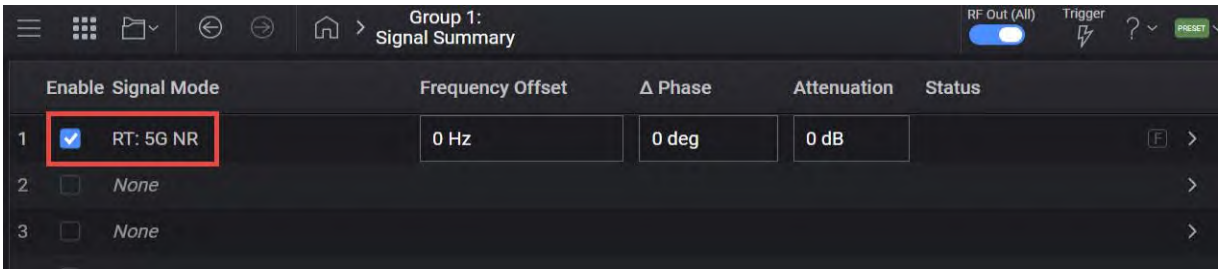
– Frequency (CW) to **1.9 GHz**

– Phase Transition Mode to **Continuous**

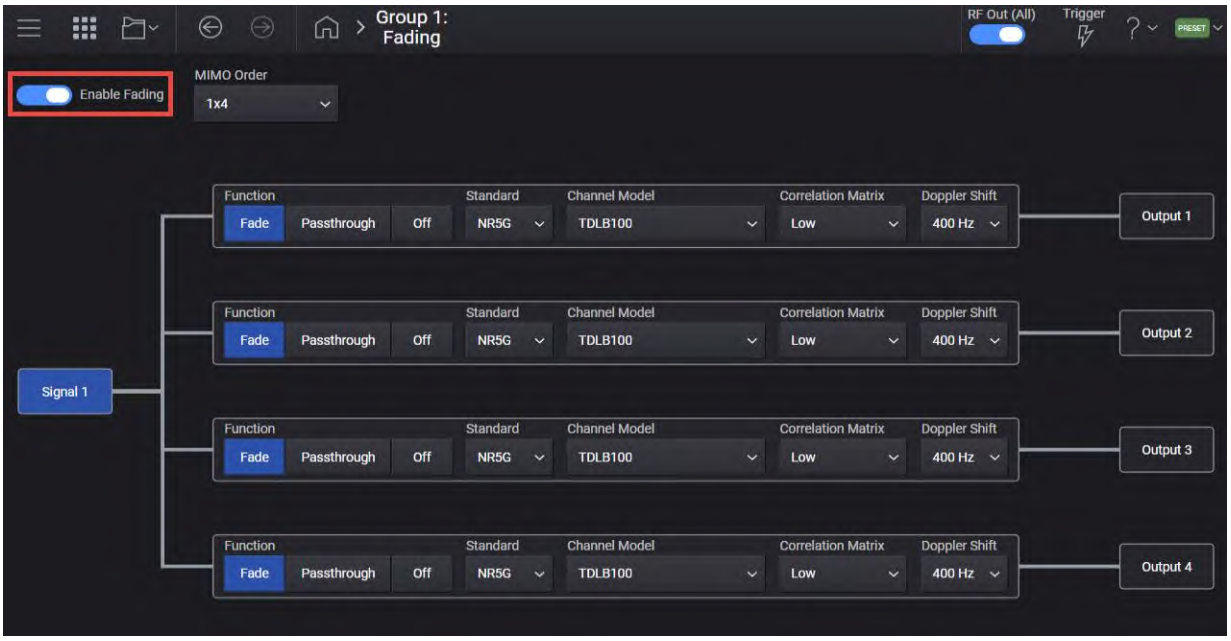
– **Enable RF Output**



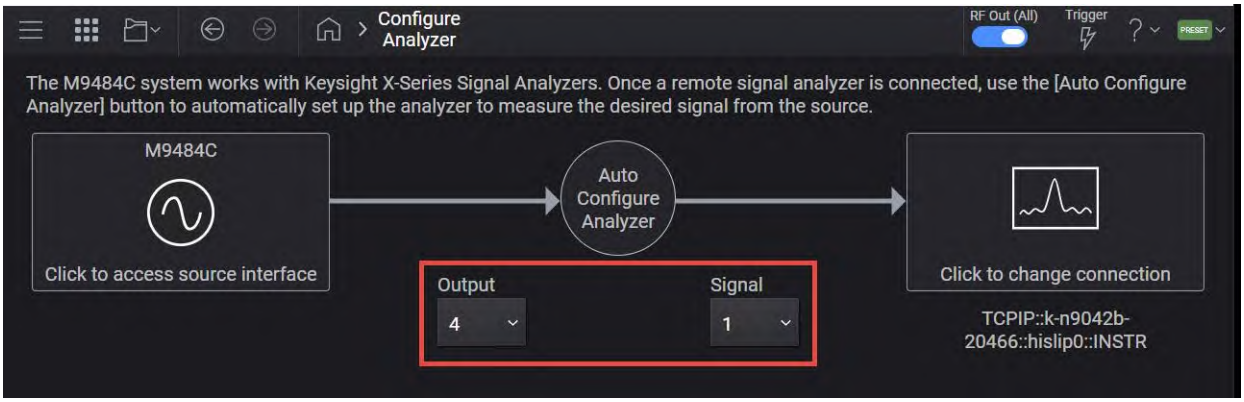
18. Select the **Home** icon > **Signal Block** > **Enable** the signal.



19. Select the **Fader** block and **Enable Fading**.

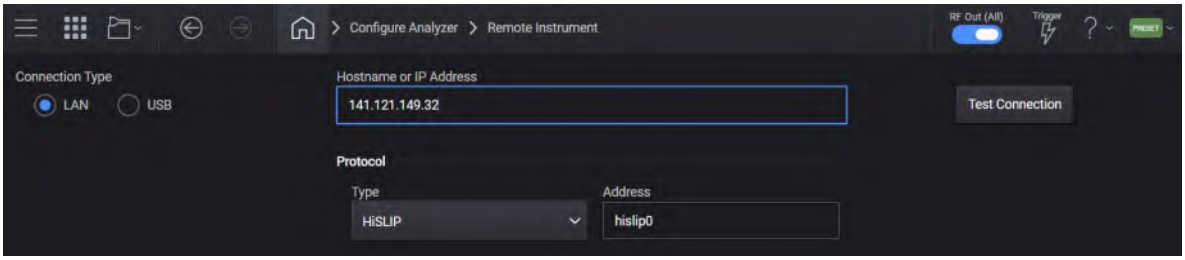


20. Select the **System** menu and select **Configure Analyzer** and set Output to **4** and Signal to **1**.

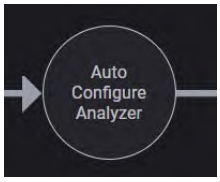


21. In the System Configuration setup, click on the Remote Signal Analyzer block (on far right) to setup the communication channel to the Keysight X-Series Signal Analyzer.

22. Set Connection Type to **LAN**.
23. In the Connect Remote X-Series Signal Analyzer setup, enter the Hostname or IP Address for the signal analyzer.
24. Under Set Protocol, select **HiSLIP** and use the Remote name **hislip0**.
25. Select **Test Connection** to verify, then **Back**.



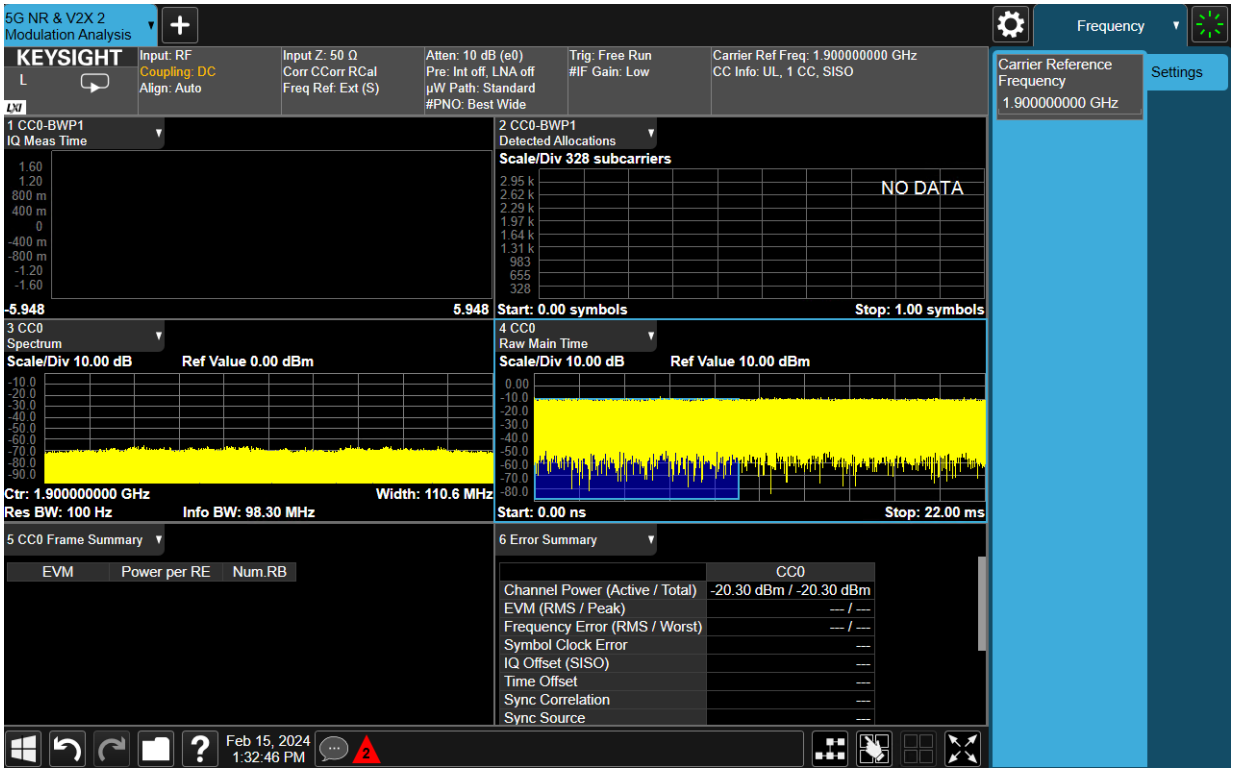
26. Select **Auto Configure Analyzer** to send the VXG settings to the analyzer.



27. Select the **Fader** block and **Enable Fading**.

    This completes the Fading setup. You will not see any signal on the UXA as the signal level is too low.

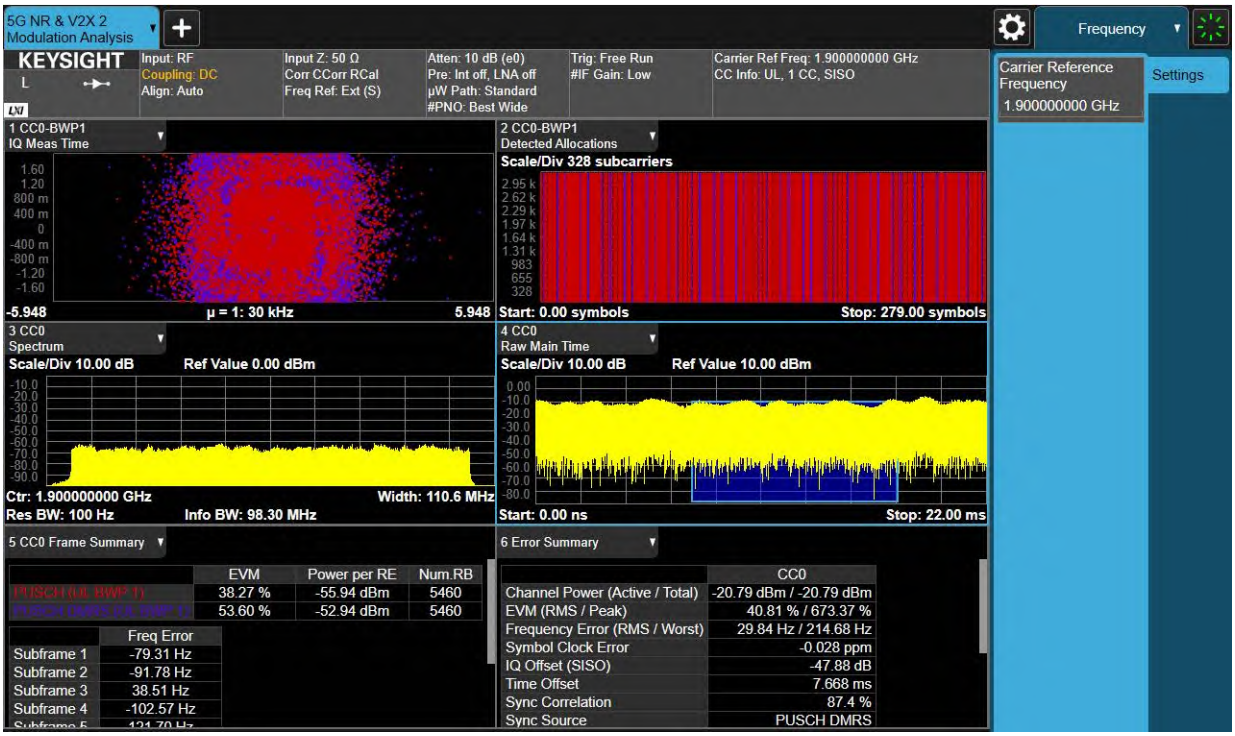
28. Verify the configuration setup on the UXA. Notice that You will not see any signal on the UXA as the signal level is too low.



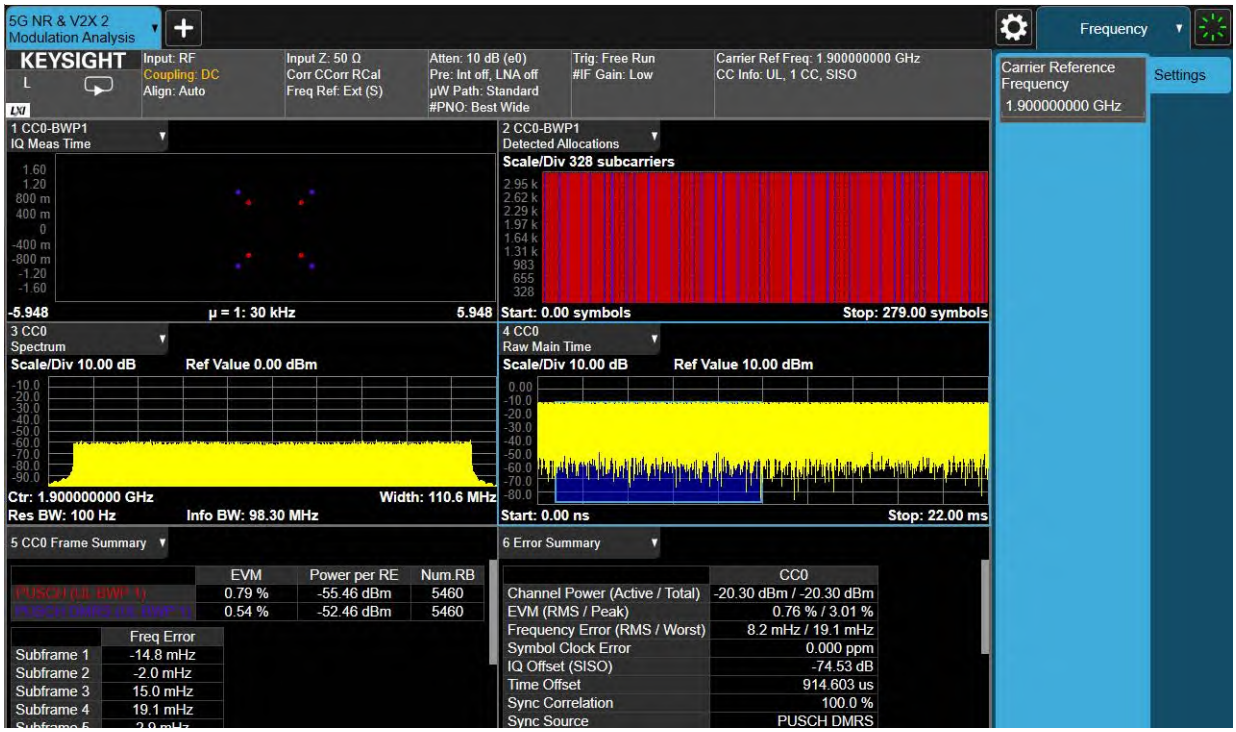
29. In order to see the affects of **Fading**:

– Set **RF Output** on Channel 1 to **-10 dBm**.

– Select Channel 1 **Adjustments** block > **AWGN** > and turn off **AWGN**.



30. Select the **Fader** block and turn **Fading off**.



Setting a 1x4 SYNC and Fading (5GNR & Custom) Example

Overview

This example will show the following:

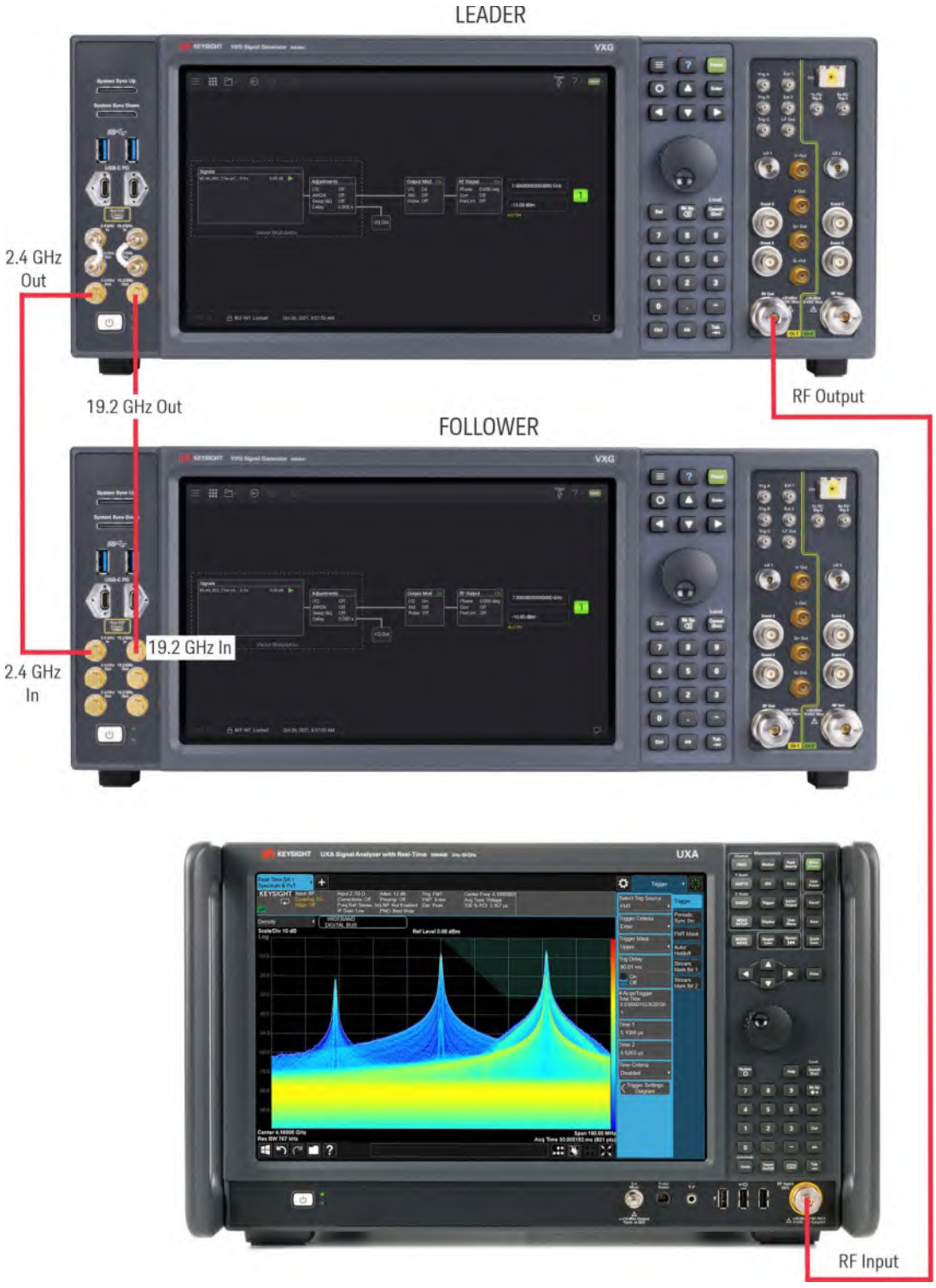
- Configure Nx4 MIMO using 2 VXG-C
- 5GNR MIMO configuration using Test Case Manager 2
- Custom Fading using Channel Studio TDL tool

System Requirements

- M9484C
  - Two, 2-channel (M9484C-002) with options PCH and SNC, with same version of instrument software installed on both instruments.
  - Options 8SG and 403 on each channel of the Leader and Follower.
  - (For 5GNR example) N7631APPC PathWave Signal Generation for 5G NR on Leader
  - (For both 5GNR and Custom Fading example) N7605AP0C PathWave Signal Generation for 3GPP Real-time Fading, on the Leader only.
  - (For Custom Fading) Embedded Application and F9860000A Channel Studio, on the Leader only.
- Additional Software
  - 3GPP Fading Profile (v1.0.5) for both the Leader and the Follower.
  - (For 5GNR example) Test Case Manger 2 (v.0.8.8) on the Leader
  - (For Custom Fading example) Channel Studio TDL Tool (v1.1) on the Leader
- X-Series Signal Analyzer
- Cables
  - 4x SMA cables (2 for front panel and 2 for rear panel)
  - RF cable to connect to the signal analyzer

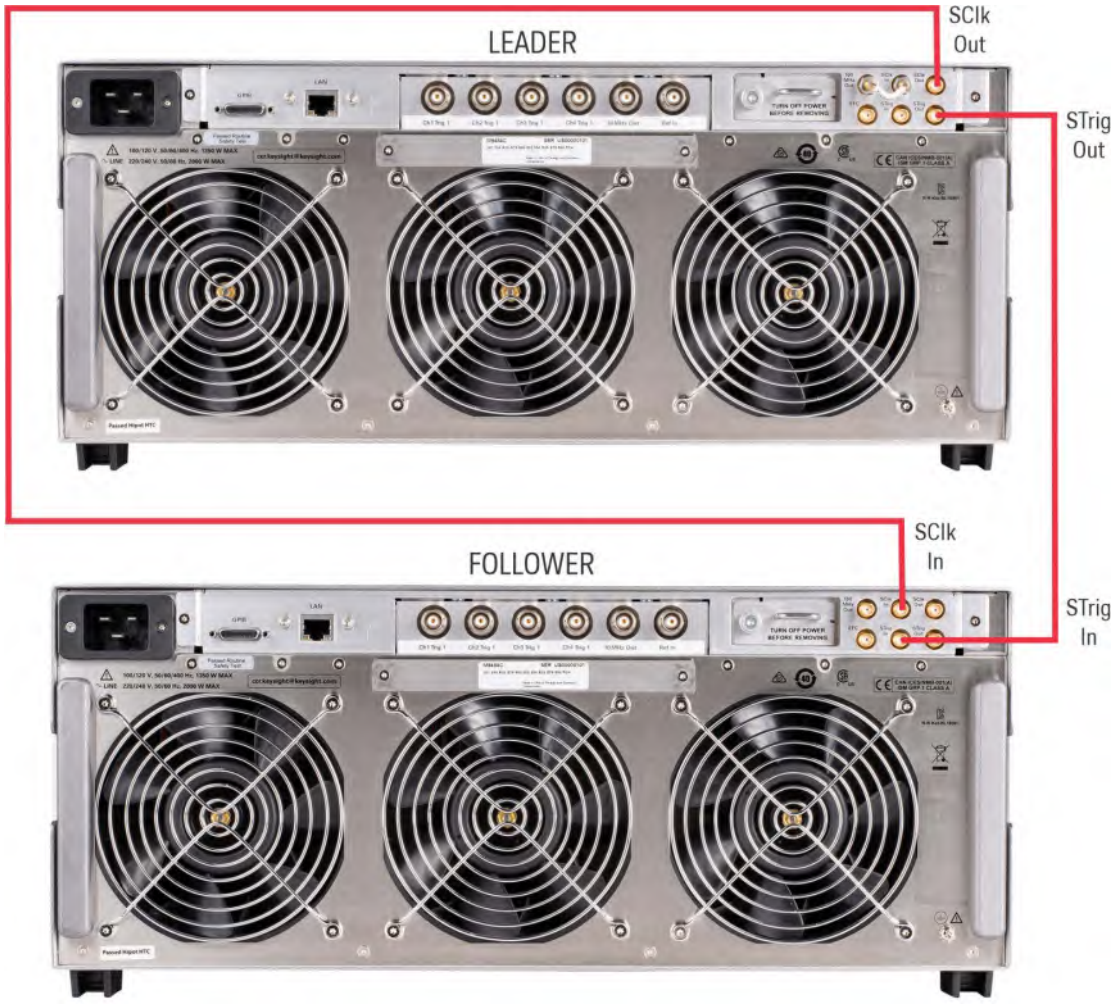
System Setup

Front Panel



1. Remove the jumper between the 19.2 GHz ports on the Follower.
2. (For VXG configurations that have 2.4 GHz ports) Remove the jumper from the 2.4 GHz ports on the Follower.
3. Connect an SMA cable from the 19.2 GHz Out on the Leader to the 19.2 GHz In on the Follower.
4. (For VXG configurations that have 2.4 GHz ports) Connect an SMA cable from the 2.4 GHz Out on the Leader to the 2.4 GHz In on the Follower.
5. Connect the RF cable between the VXG RF Out to the UXA RF Input.

Rear Panel

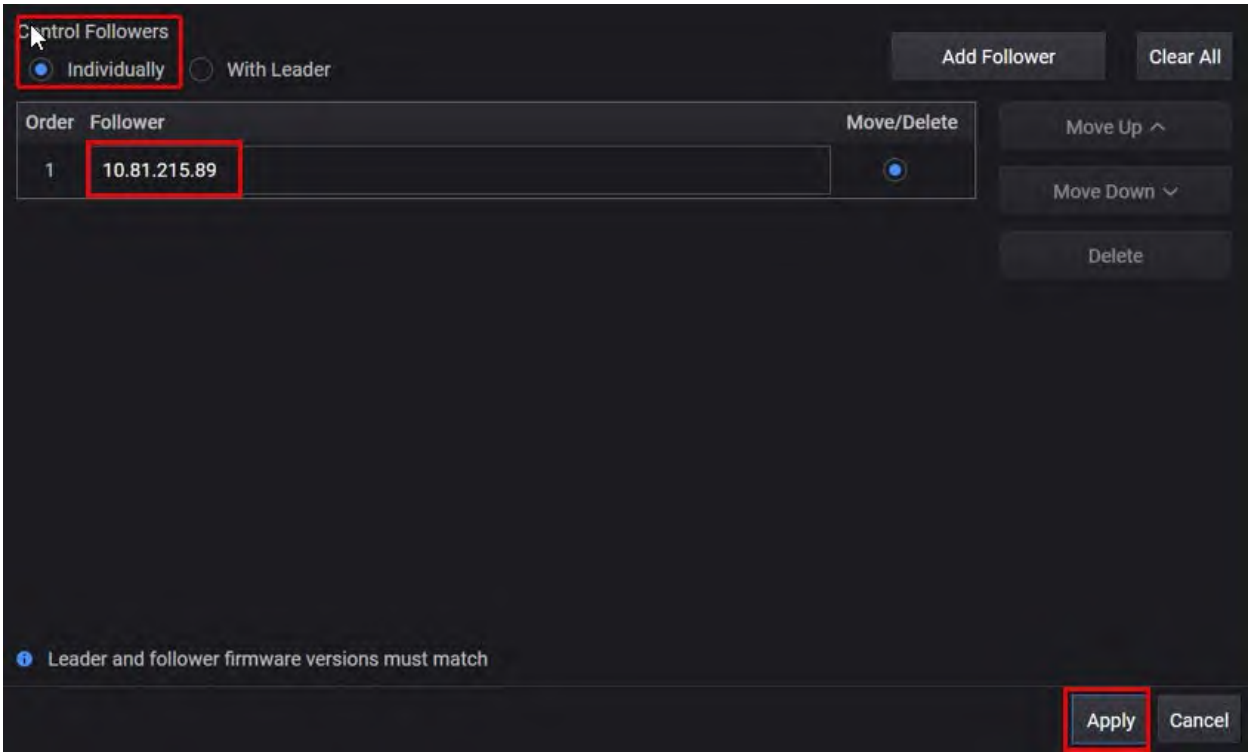


1. On the Follower, disconnect the jumper between the 100 MHz and SClk In.
2. Connect an SMA cable from SClk Out on the Leader to the SClk In on the Follower.

3. Connect an SMA cable from STrig Out on the Leader to the STrig In on the Follower.

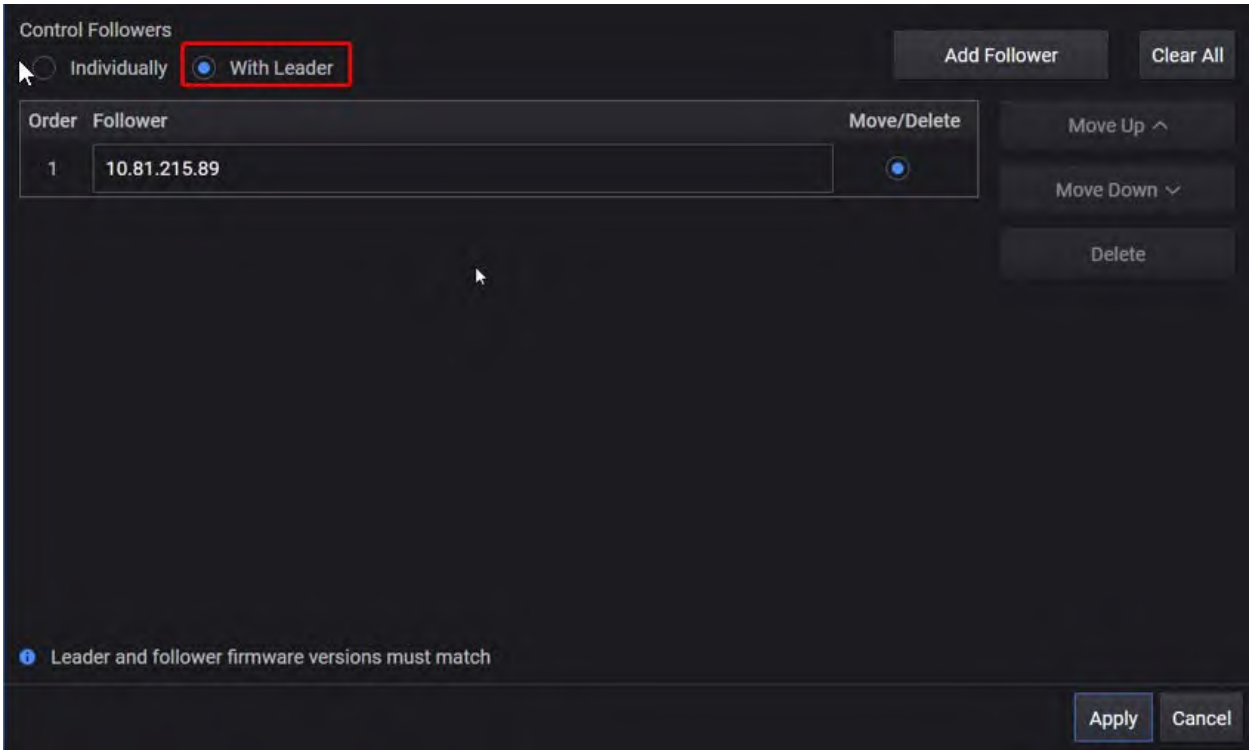
Setting Up the VXG for Multi-Instrument Synchronization

1. Start the Leader PathWave application as normal.
2. To Configure the Follower:
   a. Select the **System** icon (triple bar on top left) > **Exit M9484C**.
   b. Wait a minute to allow the firmware to completely shutdown.
3. Start the Follower using the **Launch as Follower** icon on the Desktop.
4. Wait for the connection to be established. (Note: this can take several minutes depending on the hardware configuration.)
5. To Configure the Leader:
   a. Select the **System** menu (triple bar icon) > **Configure Launch**.
6. Select the **Multi-instrument** tab and select **Leader**.
7. Select **Individually** and enter the IP address of the **Follower** > **Apply** > **OK**.



8. Once both the Leader and Follower are started, select the **System** menu > **Configure Launch**.

9. Select **With Leader** > **Apply** > **OK**.

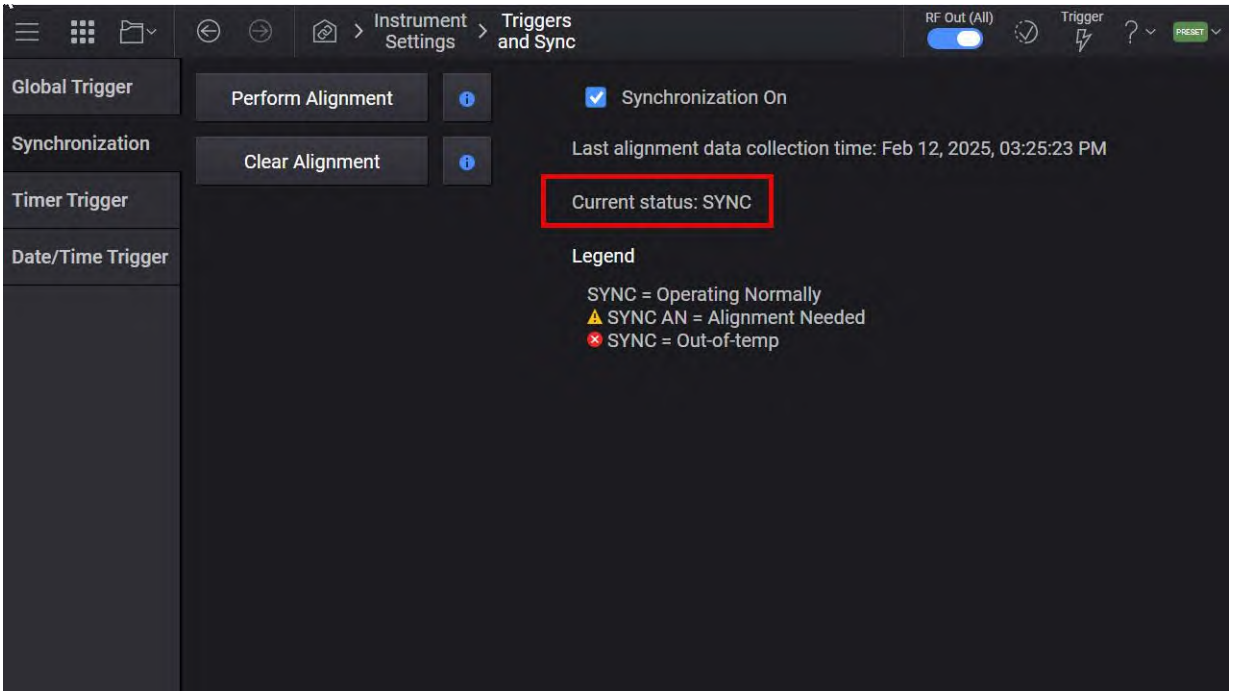


10. Wait for main window to appear on the Leader. You will see four channels. Channels 3 and 4 display a Chain icon which indicates they are on the Follower VXG-C.

11. Select the "circle" icon in the top bar and then select **SYNC**.

12. Select **Clear Alignment** and then **Perform Alignment**.

Once the Synchronization is finished, confirm the Current Status is `SYNC`.



Configuring 5G NR Chapter 8 Receiver Test

For this example, we will configure 5G NR gNB Chapter 8 Receiver Test with the following configuration from TS38.141-1 8.2.1 PUSCH with transform precoding disabled.

Table 8.2.1.5-7: Test requirements for PUSCH with 70% of maximum throughput, Type A, 100 MHz channel bandwidth, 30 kHz SCS

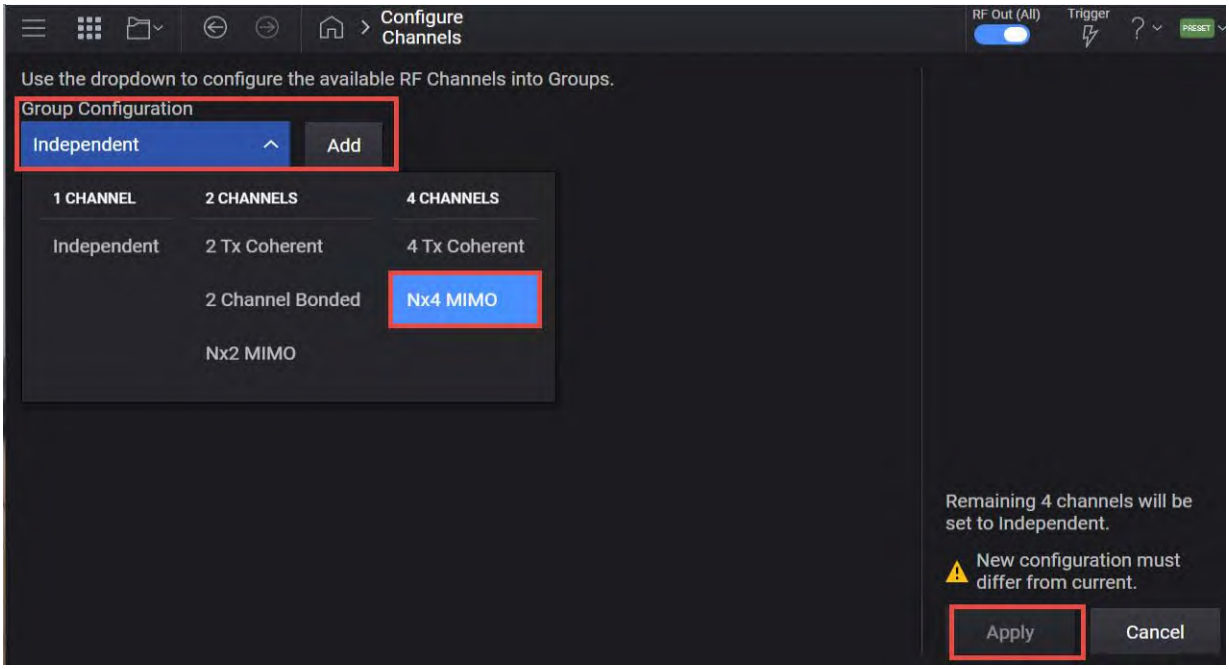
Number of TX antennas	Number of RX antennas	Cyclic prefix	Propagation conditions and correlation matrix (annex G)	Fraction of maximum throughput	FRC (annex A)	Additional DM-RS position	SNR (dB)
1	2	Normal	TDLB100-400 Low	70 %	G-FR1-A3-14	pos1	-2.2
		Normal	TDLC300-100 Low	70 %	G-FR1-A4-14	pos1	10.8
		Normal	TDLA30-10 Low	70 %	G-FR1-A5-14	pos1	13.6
		Normal	TDLA30-10 Low	70%	G-FR1-A8-5	pos1	21.7
	4	Normal	TDLB100-400 Low	70 %	G-FR1-A3-14	pos1	-5.2
		Normal	TDLC300-100 Low	70 %	G-FR1-A4-14	pos1	7.1
		Normal	TDLA30-10 Low	70 %	G-FR1-A5-14	pos1	9.6
		Normal	TDLA30-10 Low	70%	G-FR1-A8-5	pos1	17.3
	8	Normal	TDLB100-400 Low	70 %	G-FR1-A3-14	pos1	-8.1
		Normal	TDLC300-100 Low	70 %	G-FR1-A4-14	pos1	3.8
		Normal	TDLA30-10 Low	70 %	G-FR1-A5-14	pos1	6.4
		Normal	TDLA30-10 Low	70%	G-FR1-A8-5	pos1	13.7
2	2	Normal	TDLB100-400 Low	70 %	G-FR1-A3-28	pos1	2.2
		Normal	TDLC300-100 Low	70 %	G-FR1-A4-28	pos1	20.0
	4	Normal	TDLB100-400 Low	70 %	G-FR1-A3-28	pos1	-1.4
		Normal	TDLC300-100 Low	70 %	G-FR1-A4-28	pos1	12.4
	8	Normal	TDLB100-400 Low	70 %	G-FR1-A3-28	pos1	-4.4
		Normal	TDLC300-100 Low	70 %	G-FR1-A4-28	pos1	7.9
4	4	Normal	TDLB100-400 Low	70 %	G-FR1-A3-42	pos1	3.0
		Normal	TDLC300-100 Low	70 %	G-FR1-A9-4	pos1	16.8
		Normal	TDLA30-10 Low	70 %	G-FR1-A12-6	pos1	21.6
	8	Normal	TDLB100-400 Low	70 %	G-FR1-A3-42	pos1	-0.4
		Normal	TDLC300-100 Low	70 %	G-FR1-A9-4	pos1	9.5
		Normal	TDLA30-10 Low	70 %	G-FR1-A12-6	pos1	13.2

Figure 3-2

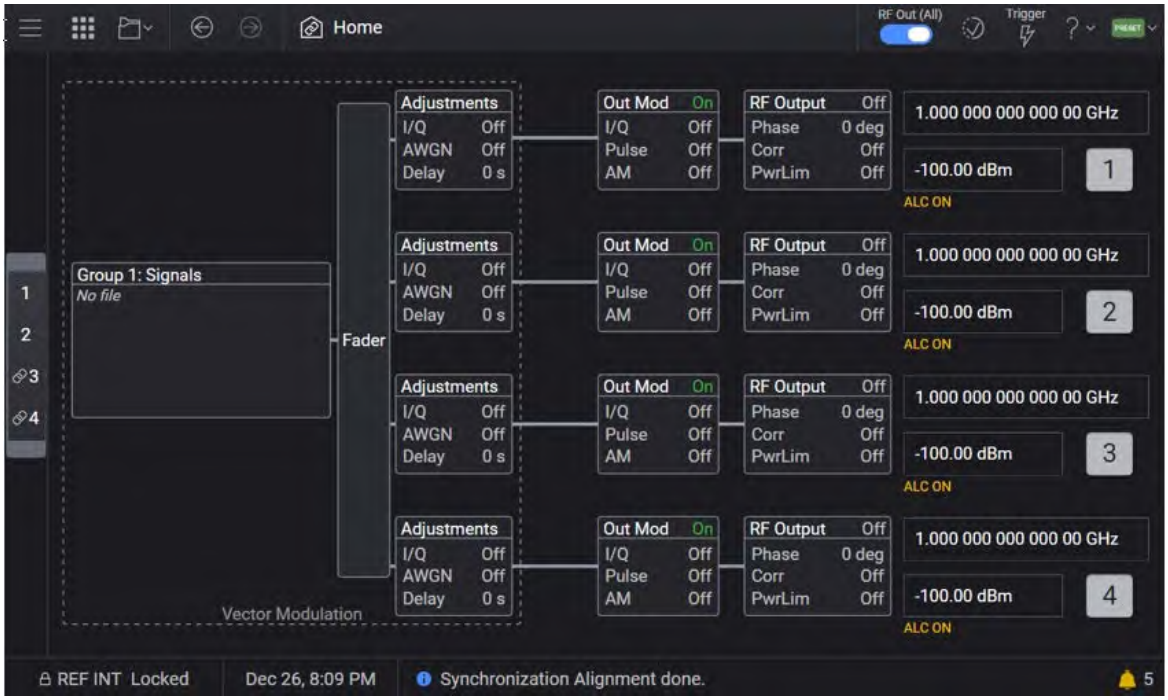
Frequency	1950MHz
Duplex	FDD
SCS	30kHz
System BW	100MHz
# of TX ANT	1
# of RX ANT	4
Channel Model	TDLA30-10 Low
FRC	G-FR1-A5-14
PUSCH Mapping Type	Type A
Transform Precoding	OFF
Noise BW	98.28MHz
Noise Power	-73.1dBm
SNR	9.6 dB

1. Select the **System** menu > **Configure Channels**.

2. Select the Group Configuration dropdown and select **Nx4 MIMO** > **Add** > **Apply** > **Apply**.



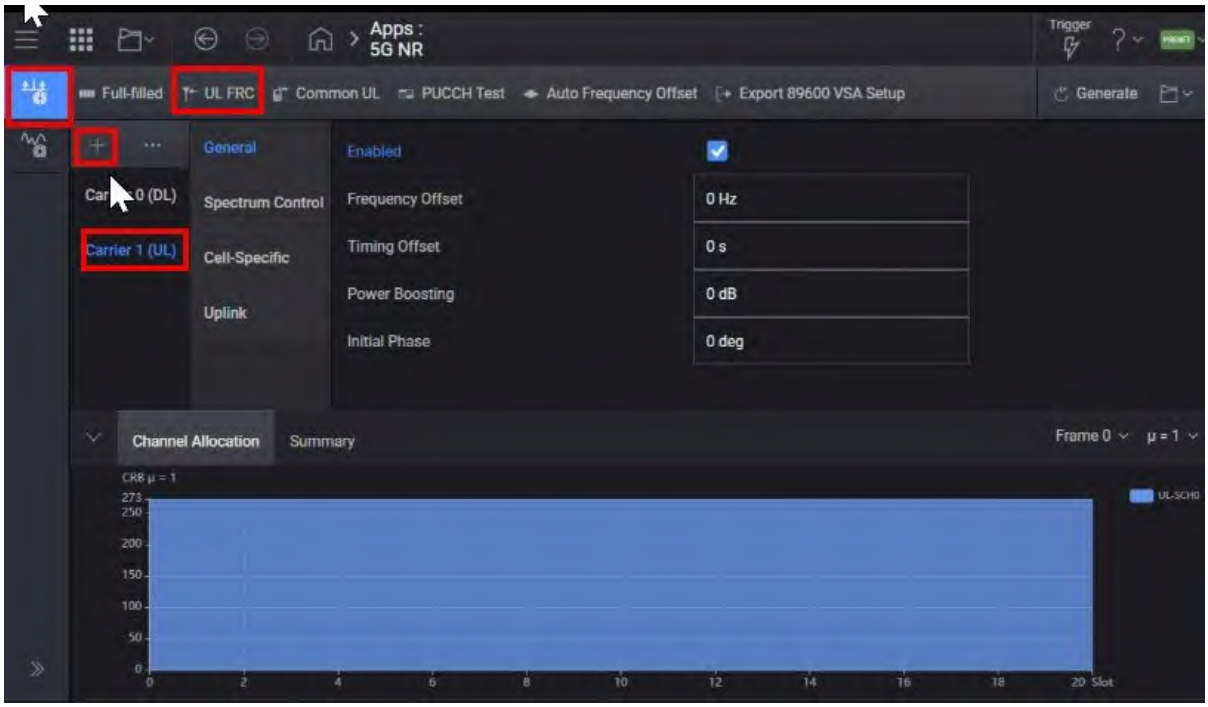
Now you can see that Nx4 MIMO is configured. Two channels from the Leader and two channels from the Follower.



3. Set Frequency on all 4 channels to **1. 95 GHz**.
4. To configure the 5GNR Uplink signal, select the **Apps** block to open.

a. Select **5G NR/5G-Advanced**

b. Select the **Carrier** tab and select the "+" plus icon to add an uplink carrier.

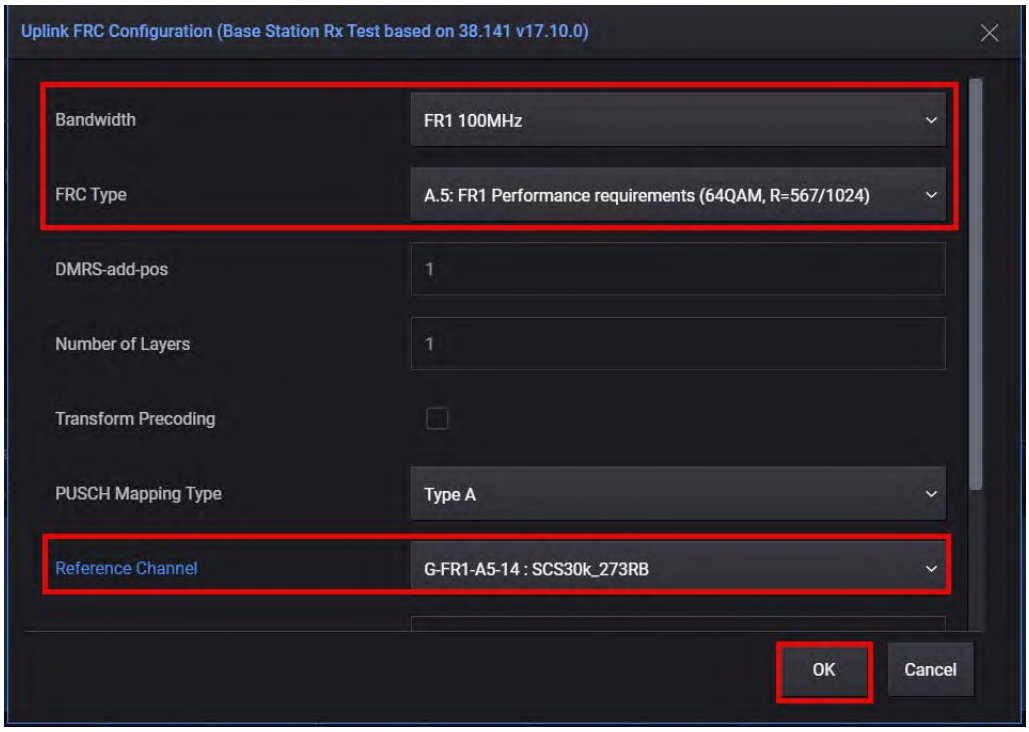


c. From the top of the display, select **UL:-FRC**.

d. Set the Uplink FRC Configuration to:

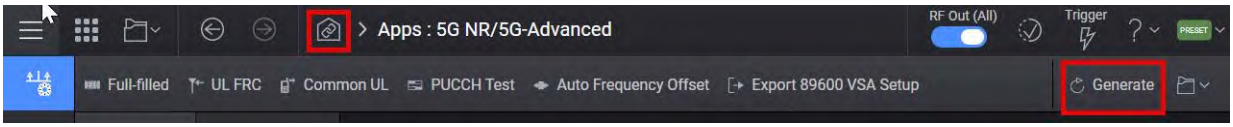
- Bandwidth > **FR1 100 MHz**
- FRC Type > **A.5: FR1 Performance requirements (64QAM, R=567/1024)**
- Reference Channel > **G-FR1-A5-14:SCS30k_273RB**

– **OK**



This FRC occupies 273 RB with 30 kHz SCS. The occupied BW is 30 kHz x 12 x 273 =98.28 MHz.

5. Select **Generate** (top right corner) to apply the signals to each channel and then select the **Home** icon to return to the main window.



6. To configure Fading, select the **Fader** block > **MIMO Order** dropdown and select **1x4.**
7. Configure all 4 channels as follows:
   – Ensure the Standard is set to **NR5G**.

     You can select different radio standards 5GNR, LTE, or Custom Fading.

   – Channel Model to **TDLB30** which will automatically set the Doppler Shift to 400 Hz.

     Choices are: TDLA30, TDLB100, or TDLC300. The selection will vary depending on the MIMO Order. For 1x4, these three models are available.

   – Correction Matrix to **Low**.

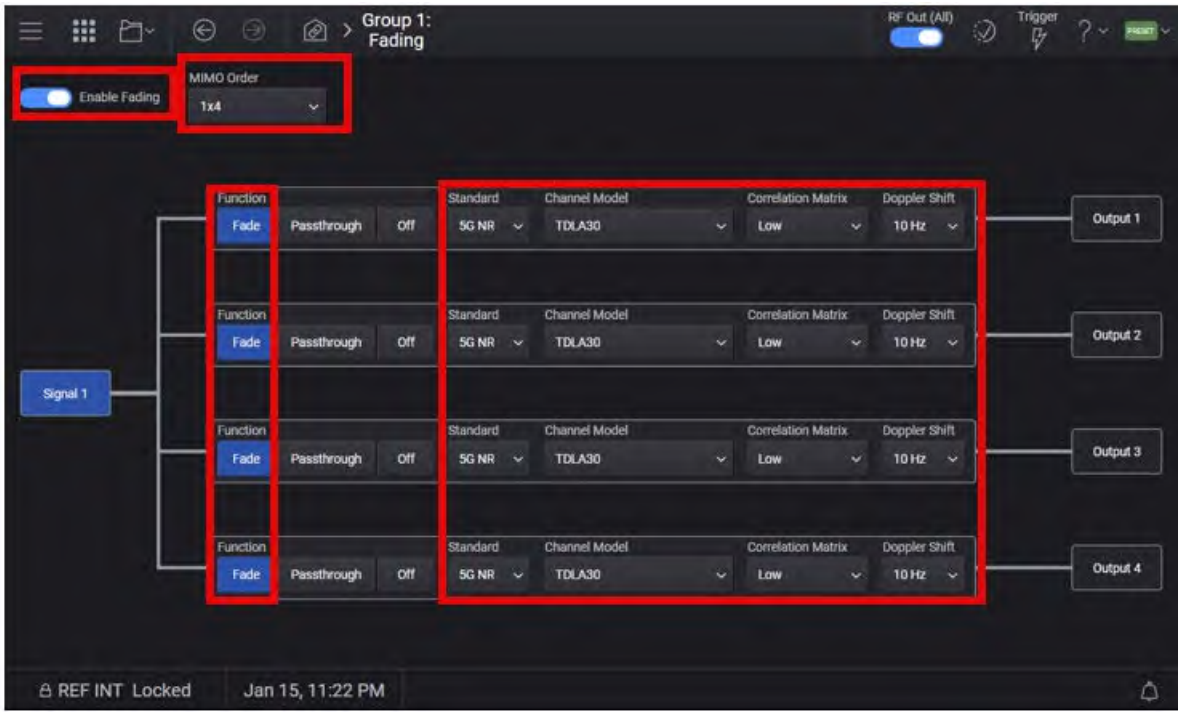
Choices are: Low, Medium, Medium A, or High. Low is commonly used by 3GPP.

– Doppler Shift to **10 Hz**.

  Choices are: 5 Hz, 10 Hz, 35 Hz, 75 Hz, 300 Hz. The selection depends on the selected MIMO order and Channel Model.

– Select **Fade** (highlight to Enable) for all nodes to make it the active function.

– Select **Enable Fading**.



8. Select the **Adjustments** block for Channel 1.

   – Select the **AWGN** tab and set:

   – Power Control Mode to **Flat Noise**

   – Carrier Bandwidth to **98.28 MHz**

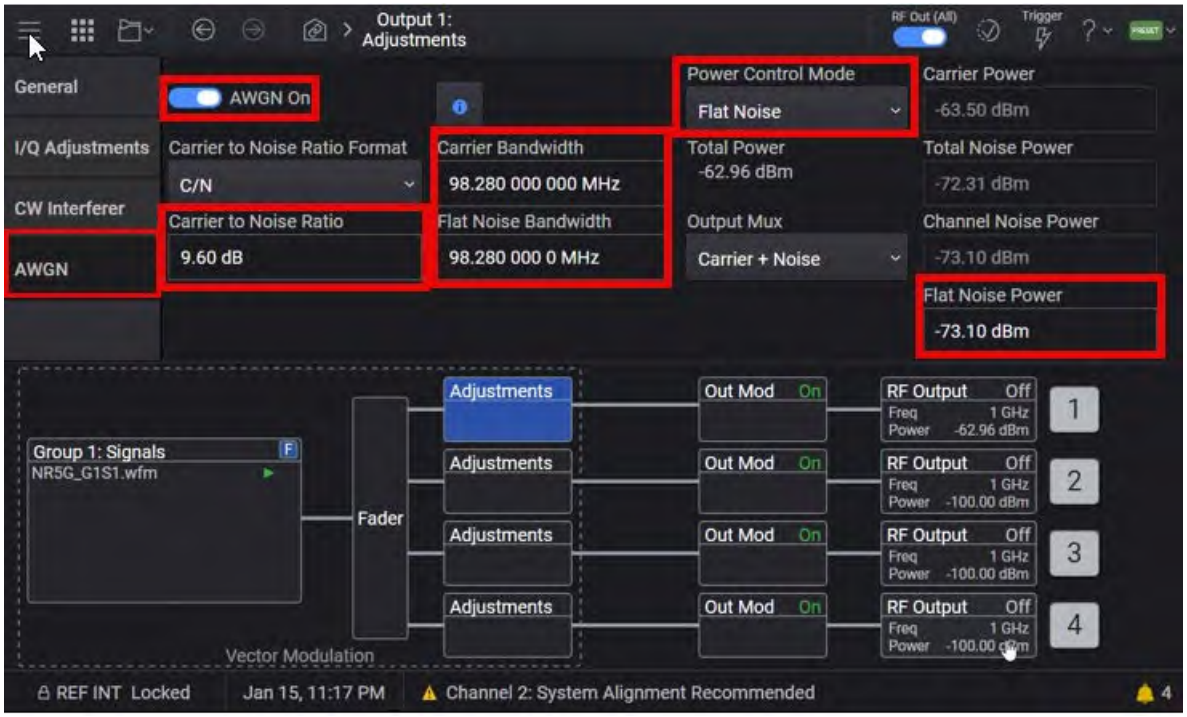
   – Flat Noise Bandwidth to **98.28 MHz**

   – Flat Noise Power to **-73.1 dBm**.

   – Carrier to Noise Ratio (SNR) to **9.6 dB**

   – Toggle AWGN On.

– Repeat these steps for the other three channels.



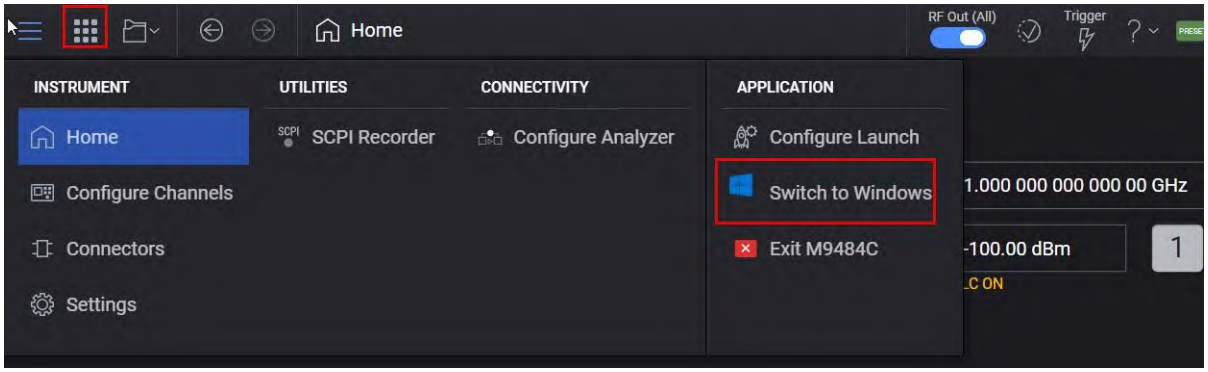
9. Turn **RF Output On** for all Channels.

Configure Chapter 8 Test using Test Case Manager 2

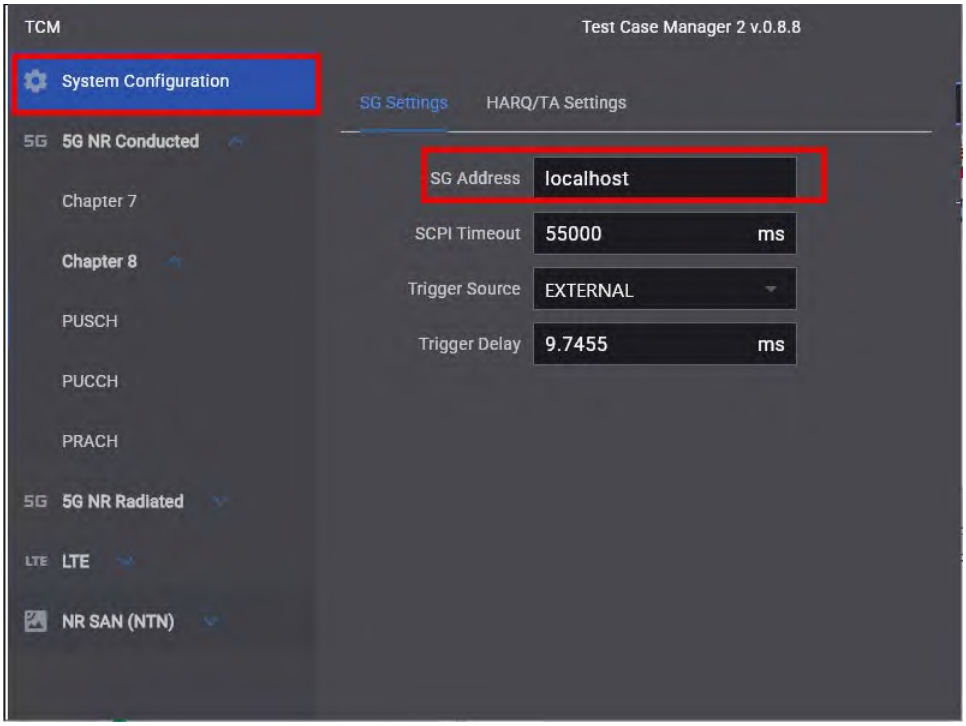
Test Case Manager 2 (TCM2) is a utility that simplifies chapter 7 and 8 test configurations for 5G NR, LTE and NTN. We will use TCM2 to setup the same configuration as above example.

1. Select **Preset** > **Preset** to set the PathWave Signal Generator (PWSG) to a known state.

2. Select the **System** icon > **Switch to Windows**.

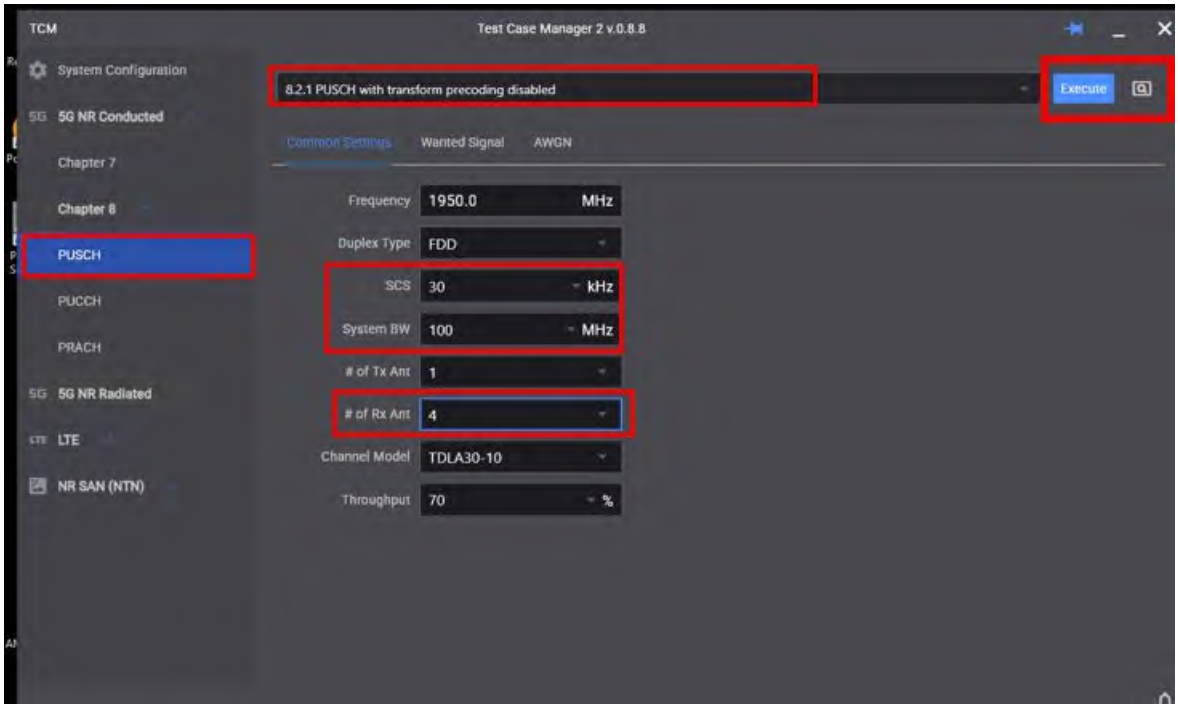


3. From the Windows **Start** menu, select **Keysight Test Case Manager 2.**
4. In the TCM2 main window, select **System Configuration** and set SG Address to **localhost**.



5. In the left pane, select **5G NR Conducted** > **Chapter 8** > **PUSCH**.
6. With 8.2.1 PUSCH with transform precoding disabled, select the Common Settings dropdown menu and set:
    a. SCS to **30 kHz**
    b. System BW to **100 MHz**

c. # of Rx ANT to **4**

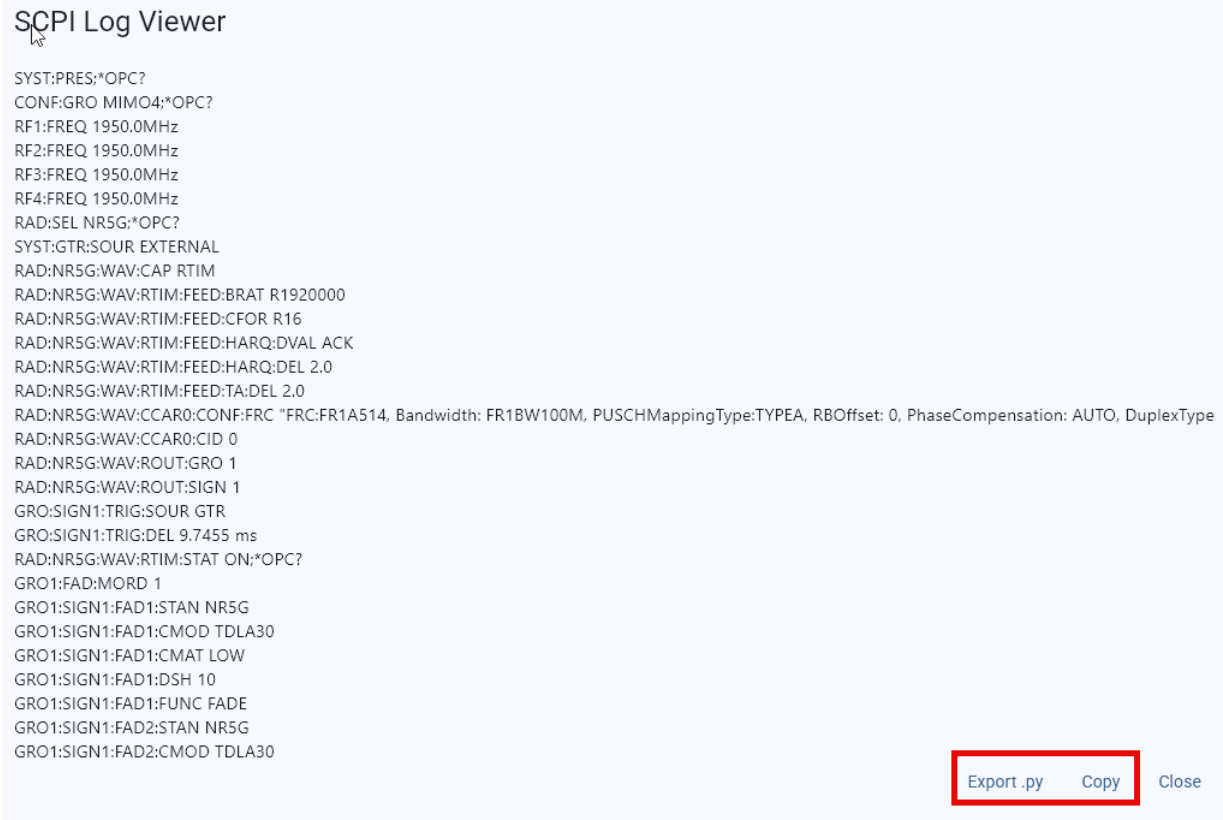


d. Select the **AWGN** dropdown menu and review that Noise Power, Noise BW and SNR are the same as in Figure 3-2.

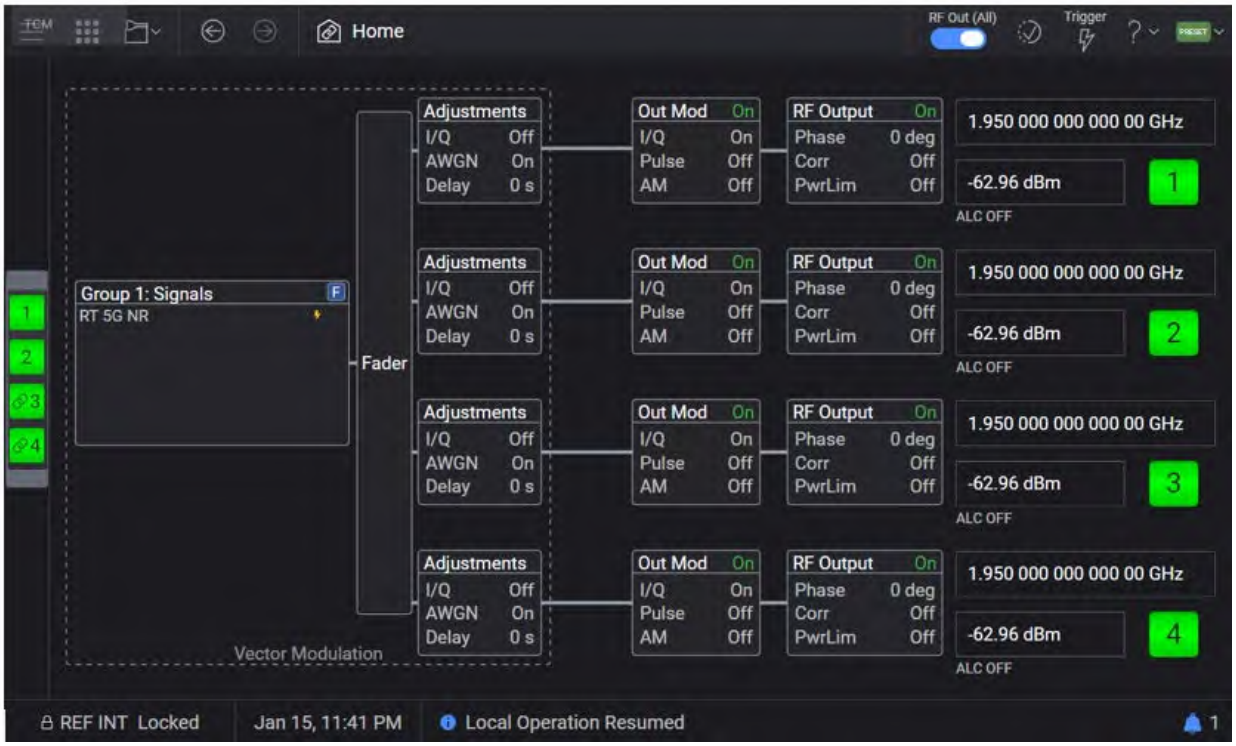
e. Select **Execute**.

A blue progress bar appears at the bottom of the screen.

f. Once completed, select the **magnifying glass icon** (next to the Execute icon) to review what SCPI commands are sent. You can copy them to the Clipboard, or you can also save a Python script for automation.



g. **Minimize** the TCM2 window and go back to the VXG-C PWSG application.



You can see TCM2 can significantly simplify the steps to configure the conformance test.

Configure Custom Fading

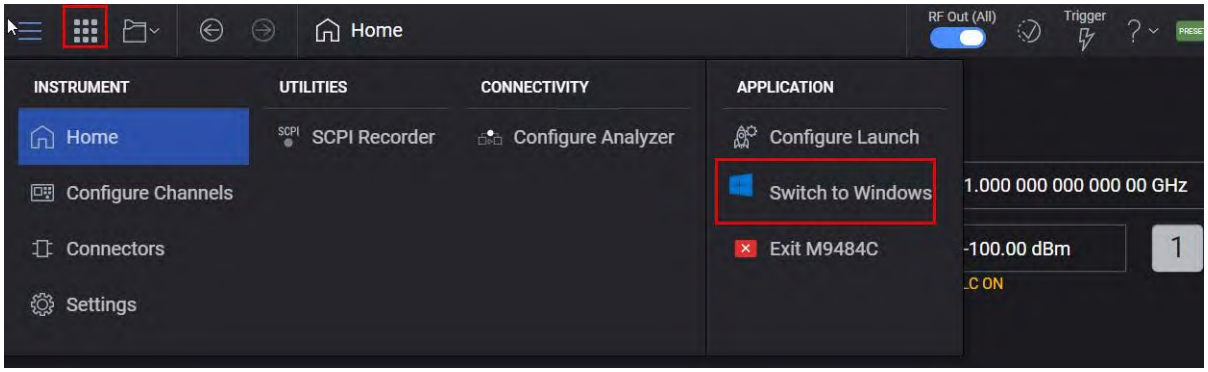
In this example, we will configure sample custom fading profile as shown below.

# of Tx Ant	2
# of Rx Ant	2
# of Paths	3
Path 1: Power	-5 dB
Path 1: Delay	100 ns
Path 1: Amplitude distribution	Rayleigh
Path 1: Doppler Spectrum	Jakes
Path 1: Max Doppler:	500 Hz
Path 1: F Start	-500Hz
Path 1: F Stop	500Hz
Path 1: MIMO Correlation	Low
Path 2: Power	-10 dB
Path 2: Delay	200 ns
Path 2: Amplitude distribution	Rayleigh
Path 2: Doppler Spectrum	Jakes
Path 2: Max Doppler:	500 Hz
Path 2: F Start	-500Hz
Path 2: F Stop	500Hz
Path 2: MIMO Correlation	Low
Path 3: Power	-15 dB
Path 3: Delay	300 ns
Path 3: Amplitude distribution	Rayleigh
Path 3: Doppler Spectrum	Jakes
Path 3: Max Doppler:	500 Hz
Path 3: F Start	-500Hz
Path 3: F Stop	500Hz
Path 3: MIMO Correlation	Low

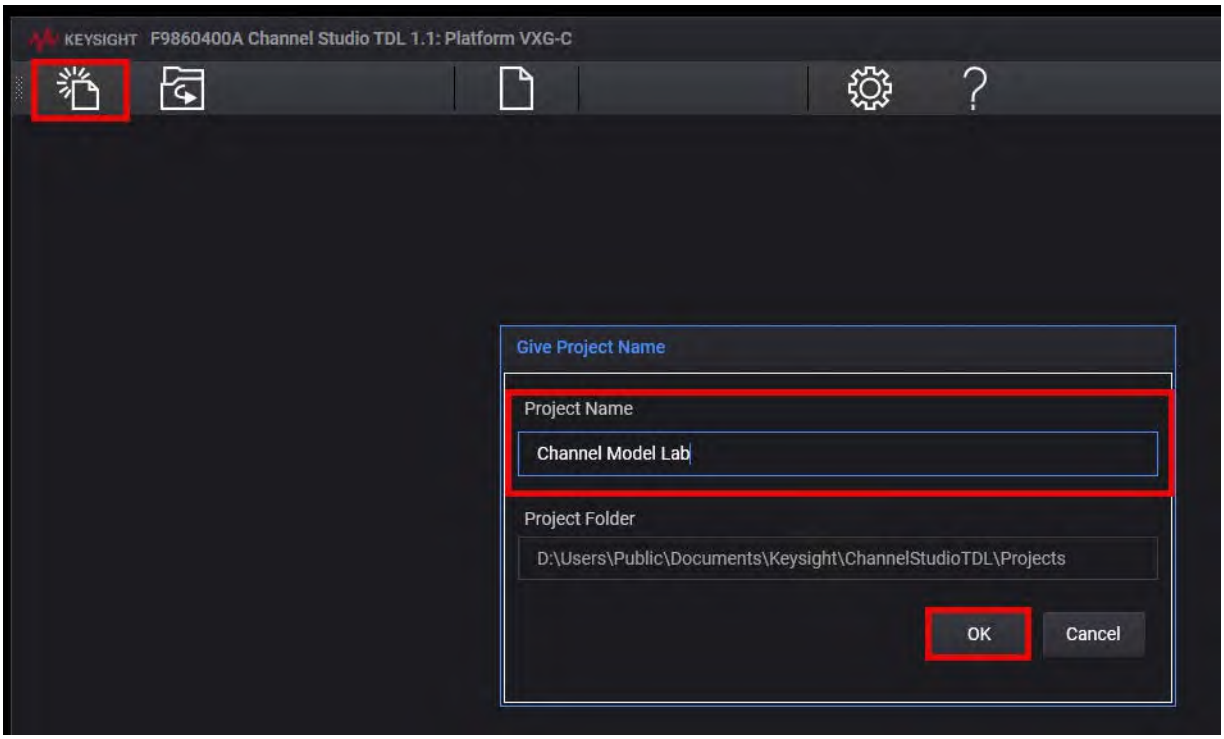
Configure Channel Model using the Channel Studio TDL Tool

1. Select **Preset** > **Preset** to set the PWSG to a known state.

2. Select the **System** icon > **Switch to Windows**.

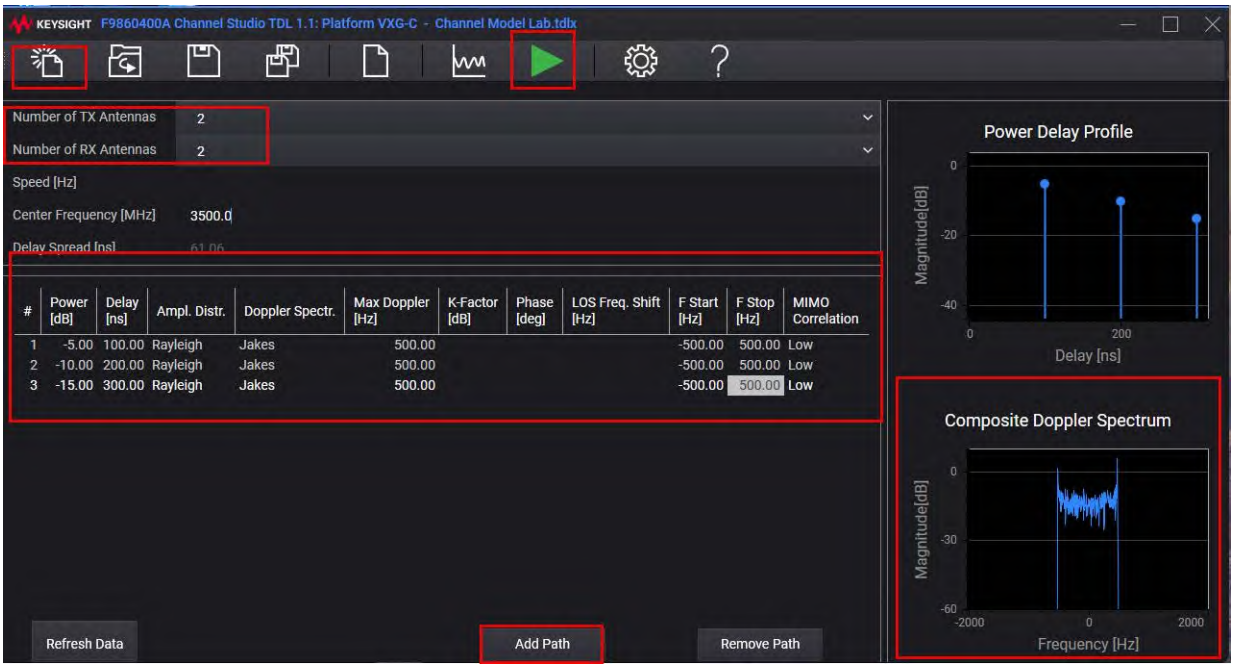


3. Select the Windows **Start** menu and select > **Keysight Channel Studio TDL F9860400A**.
4. Select the **Create New Project** icon on the top bar (left most icon).



5. Create a unique name for your project and select **OK**.
6. Fill out the table according to the settings listed above.
7. Select the **Green Triangle** to Generate a profile.

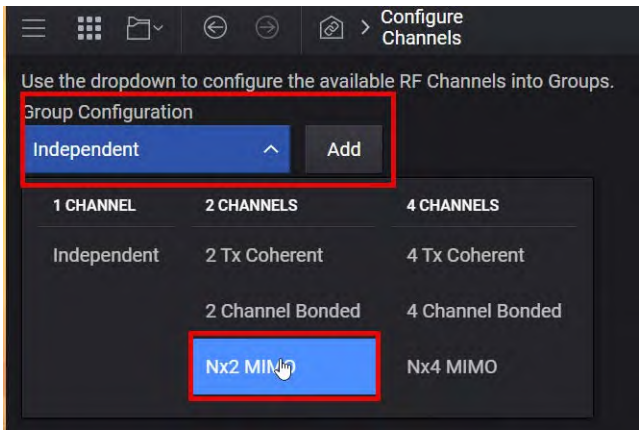
Once generated, you'll see the Doppler spectrum. The expected Doppler spread is 2x of Doppler shift, which is 1 kHz.



8. **Minimize** the Channel Studio window and open **PWSG**.

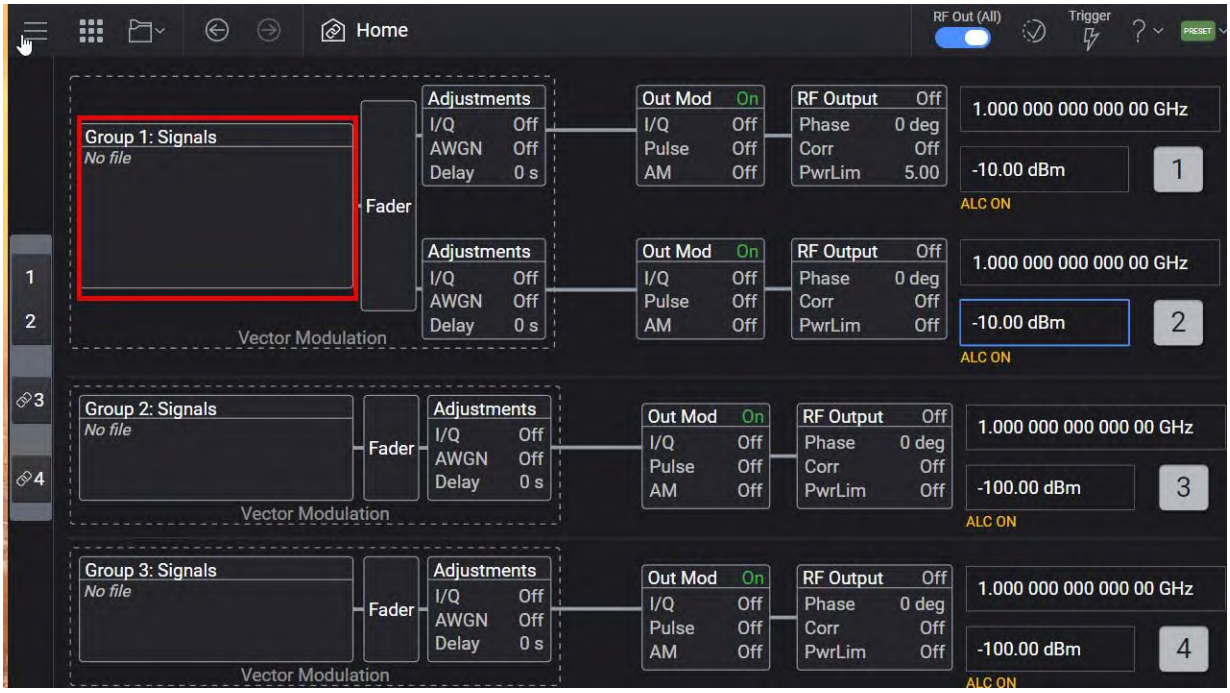
Configure MIMO and Signals

1. Select the **System** menu > **Configure Channels**.
2. Select the Group Configuration dropdown and select **Nx2 MIMO** > **Add** > **Apply** > **Apply**.

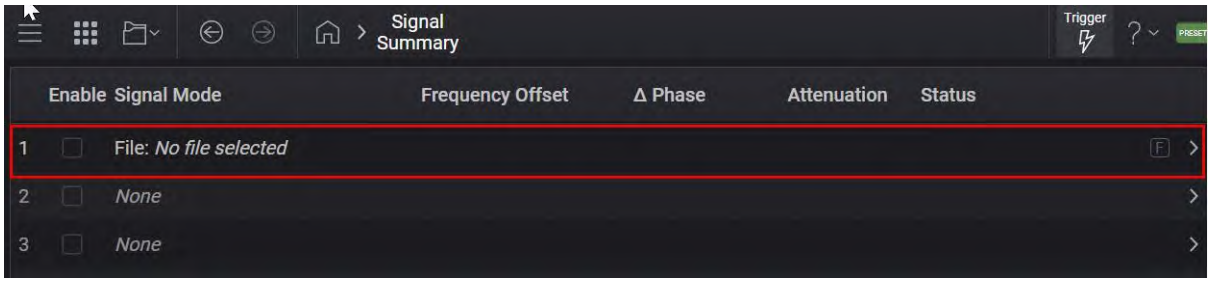


3. For both Channel 1 and 2, set Frequency to **1 GHz**, and Output Power to **-10 dBm.**

4. Select the **Group 1: Signals** block to open.



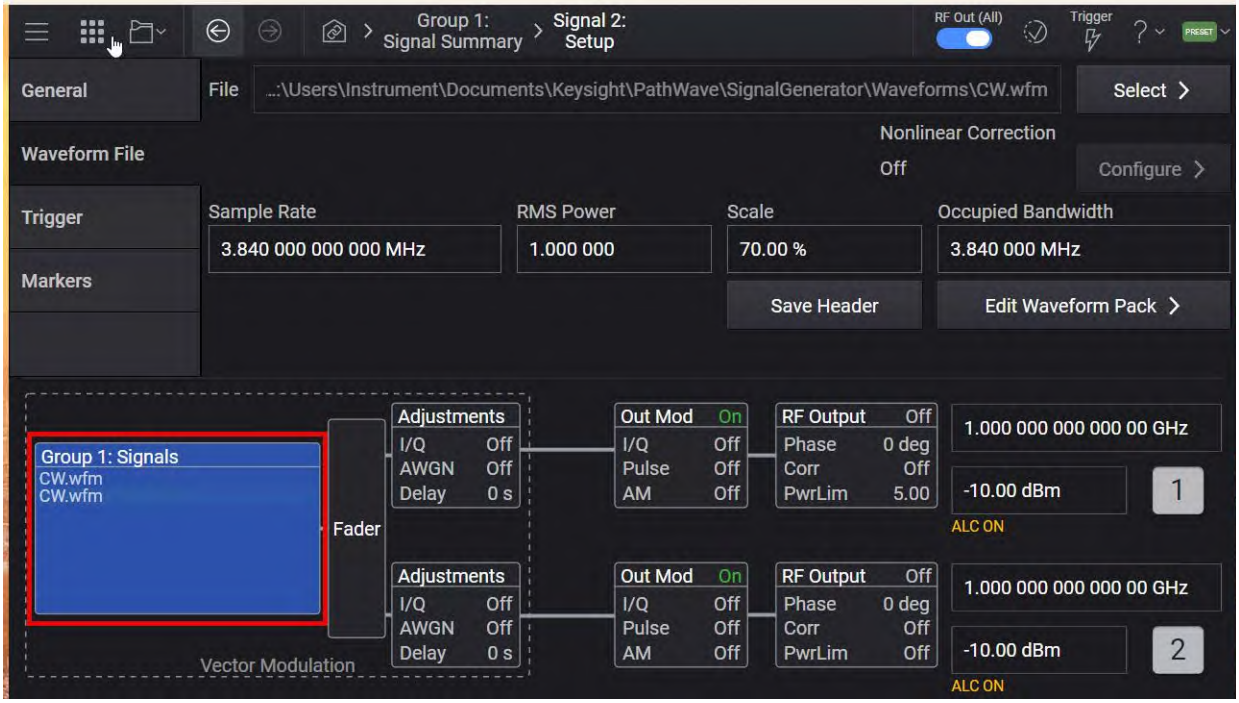
5. In the Signal Summary window, select **File: No file selected arrow** to open the Signal 1 Setup.



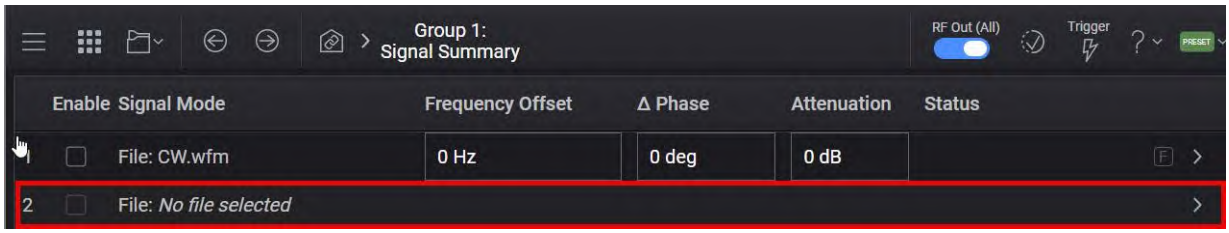
6. In the left pane select the **Waveform File** tab.
7. Use **File Select** to navigate to:

   `D:\Users\Instrument\Documents\Keysight\PathWave\SignalGenerator\Examples`

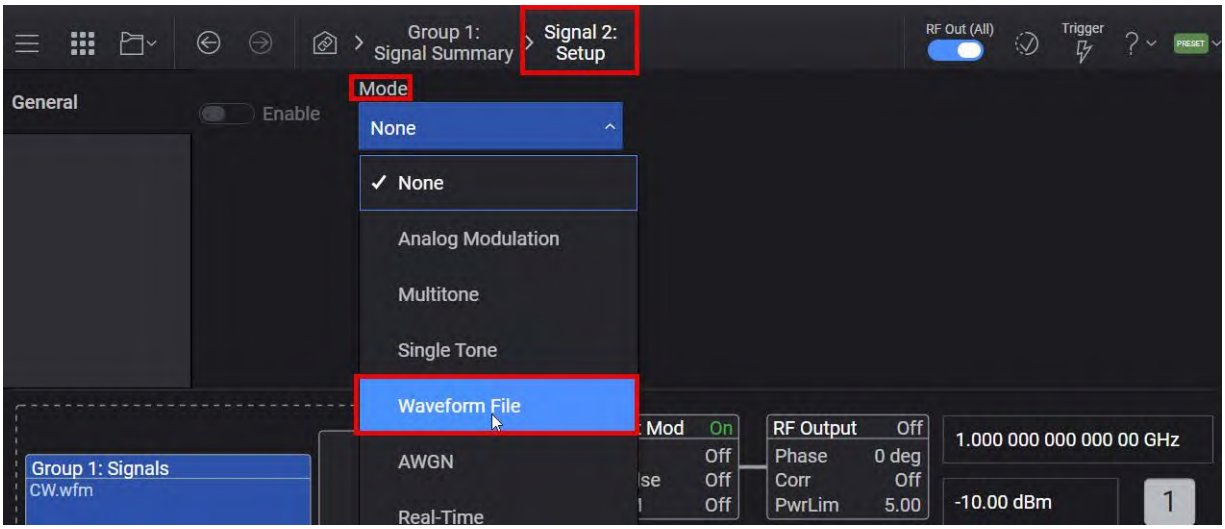
and select **CW.wfm**



8. Select the **Back** icon. In the Signal Summary window, select **Signal 2** setup file.



9. In Signal 2 Setup, set the Mode to **Waveform**.

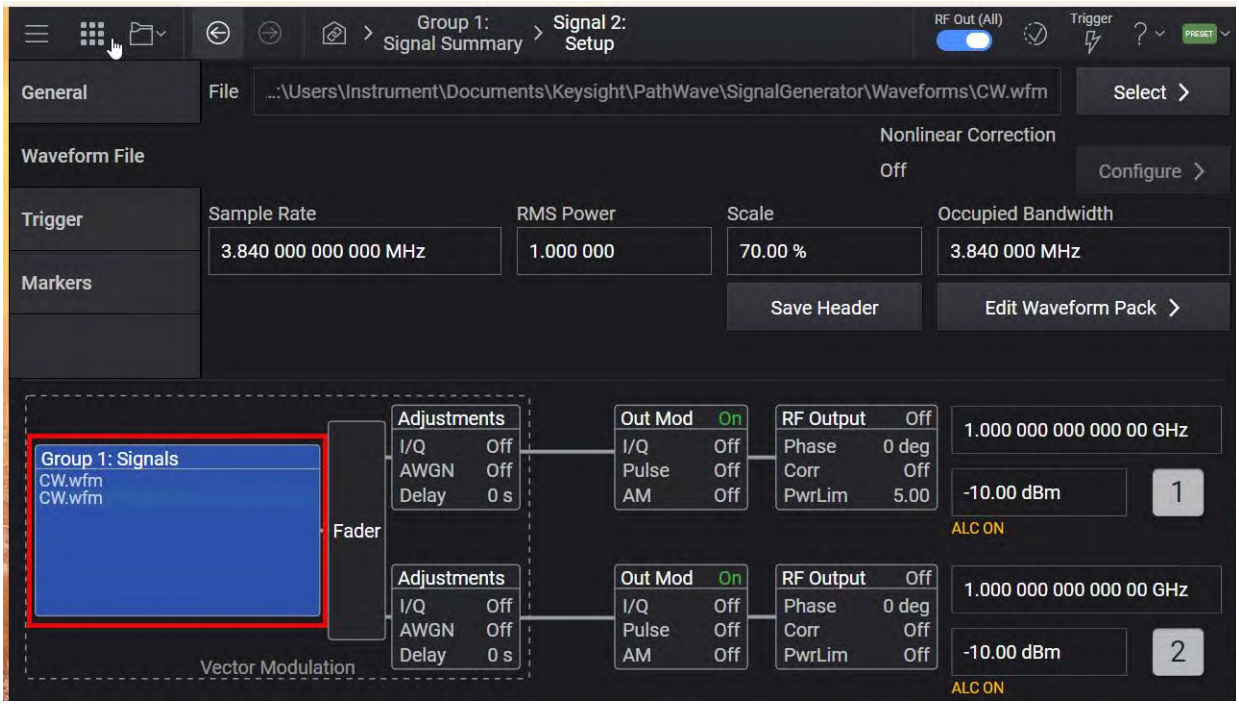


10. In the left pane select the **Waveform File** tab.

11. Use **File Select** to navigate to:

    `D:\Users\Instrument\Documents\Keysight\PathWave\SignalGenerator\Examples`

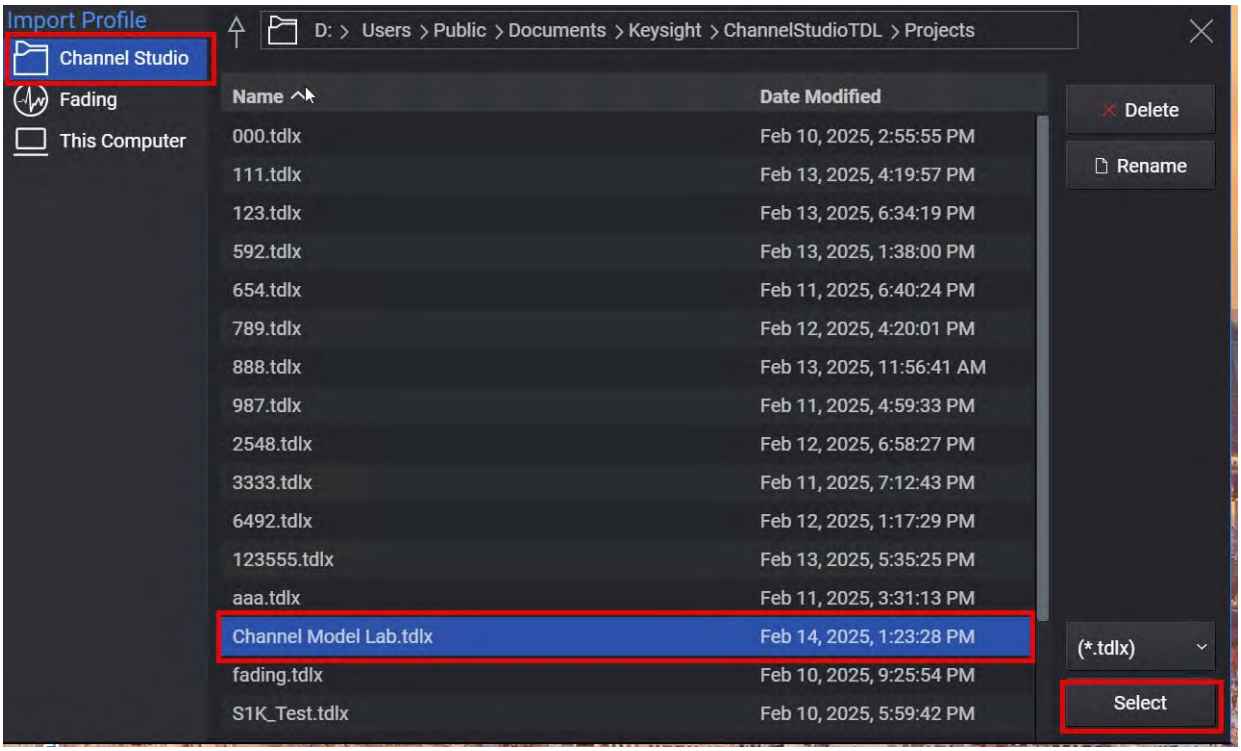
    and select **CW.wfm**



Import Custom Fading Profile

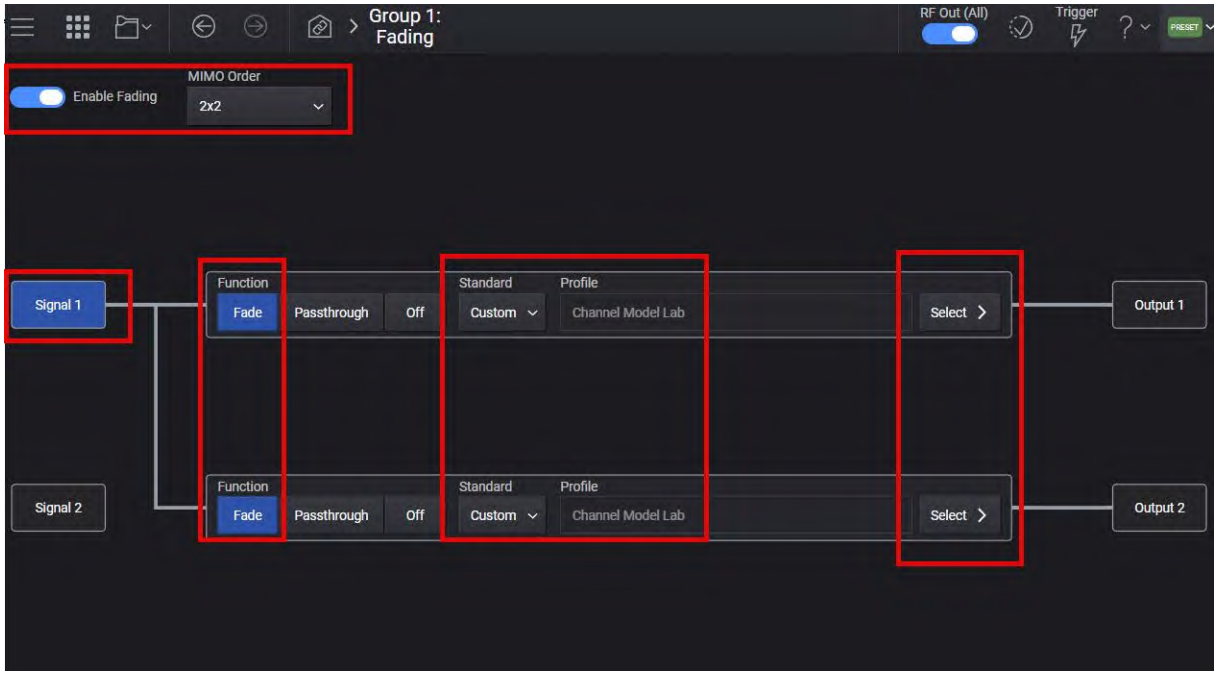
1. On the VXG, select the **Fader** block.
2. Set MIMO Order to **2x2**.
3. For the first path, set Standard to **Custom** > **Select**.
4. Select **Import Profile** > select the project you created > **Select**.

   You should see the imported profile listed in the table. If you do not see the profile, try changing MIMO Filter to All x All, and make sure the profile matches with selected MIMO order.

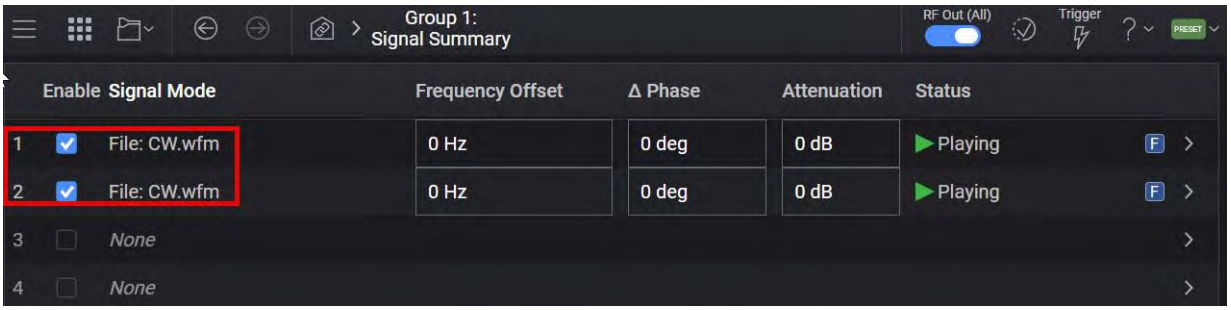


5. For the second path, set Standard to **Custom** > **Select**.
6. Select **Import Profile** > select the project you created > **Select**.

7. Set the Function to **Fade** (highlight to Enable) for both paths and then **Enable Fading**.



8. Select the **Home** icon > **Signal** block > **Enable Signal Mode** to play the CW waveform (CW.wfm)

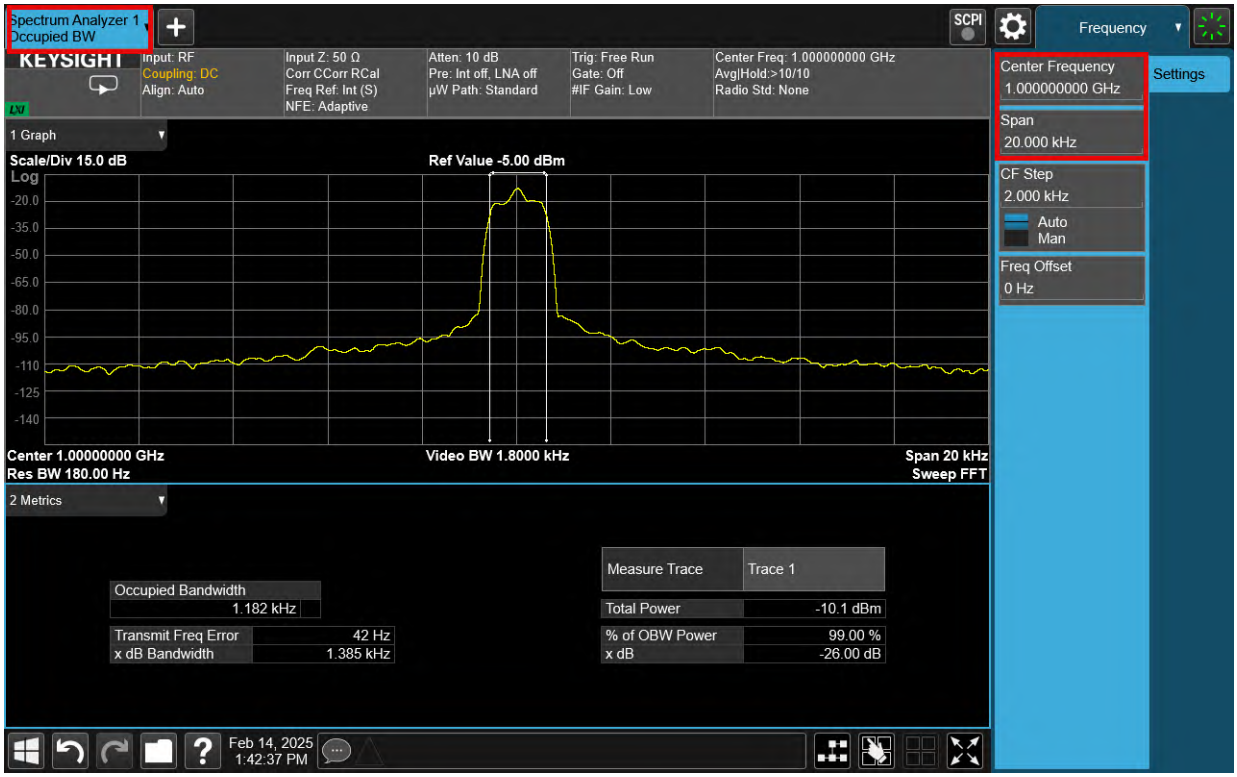


9. Play the CW waveform (CW.wfm), and increase the Output Power to **0 dBm**.

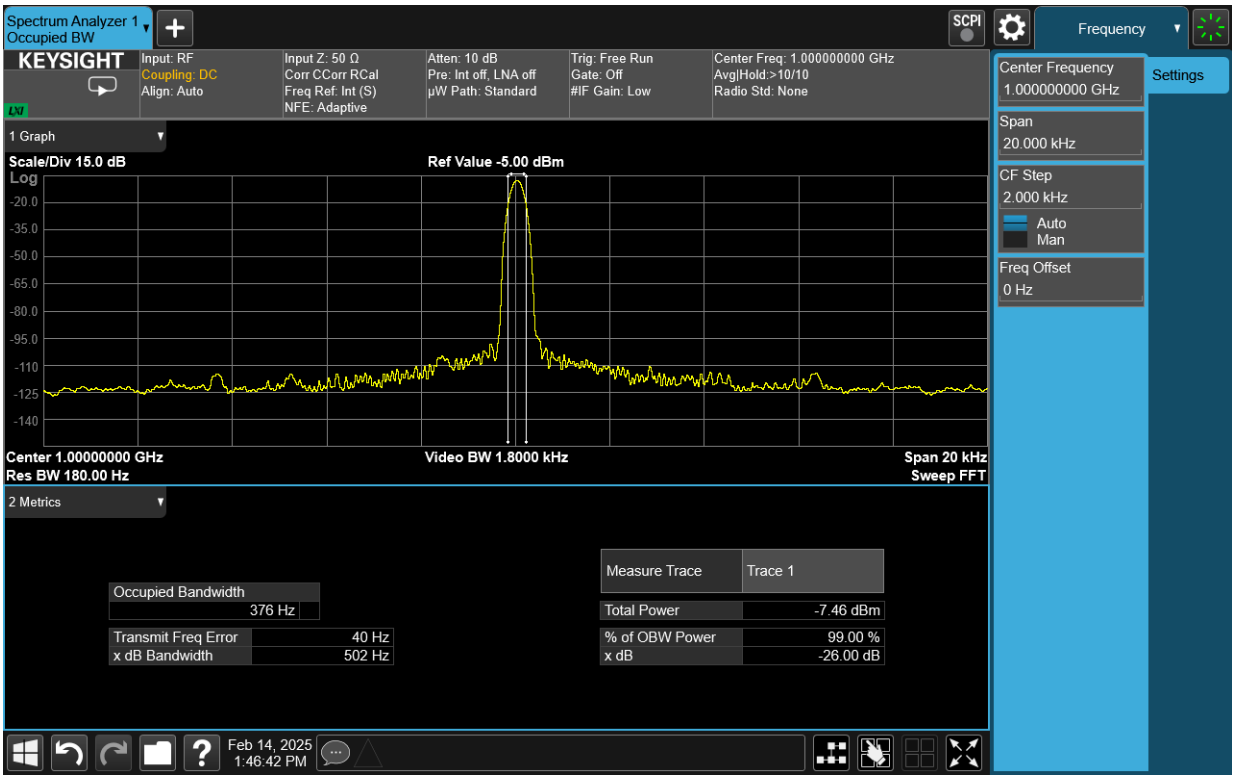
10. Measure Occupied Bandwidth on the X-Series Signal Analyzer to observe the waveform has the expected Doppler spread. (1 kHz OBW).

    – Select **Mode/Meas** > **Spectrum Analyzer** Mode > **Occupied BW** Measurement > **OBW Results** View.

– Select **Frequency** > Center Frequency to **1 GHz** > Span to **20 kHz**.



11. On the VXG, turn Fading **Off** to see the effects of fading.



Setting up Envelope Tracking

Envelope Tracking (ET) improves PA efficiency by dynamically adjusting PA supply voltage (Vcc) according to the instantaneous RF envelope. This ensures the PA receives only the required voltage at each moment, reducing power consumption while maintaining performance.

Key Benefits

- Vcc is reduced during low-power regions, extending battery life (especially in mobile devices).
- PA can be overridden with higher Vcc in high power regions to enhance linearity and output power.

How ET Works

- The RF waveform is processed to generate an envelope signal using a shaping table.
- The envelope signal drives the Envelope Tracking Power Supply (ETPS), which modulates Vcc in real time.
- ETPS must support wide modulation bandwidth to accurately follow the RF envelope.
- Precise and repeatable timing alignment between the RF and envelope path is critical for correct ET operation.

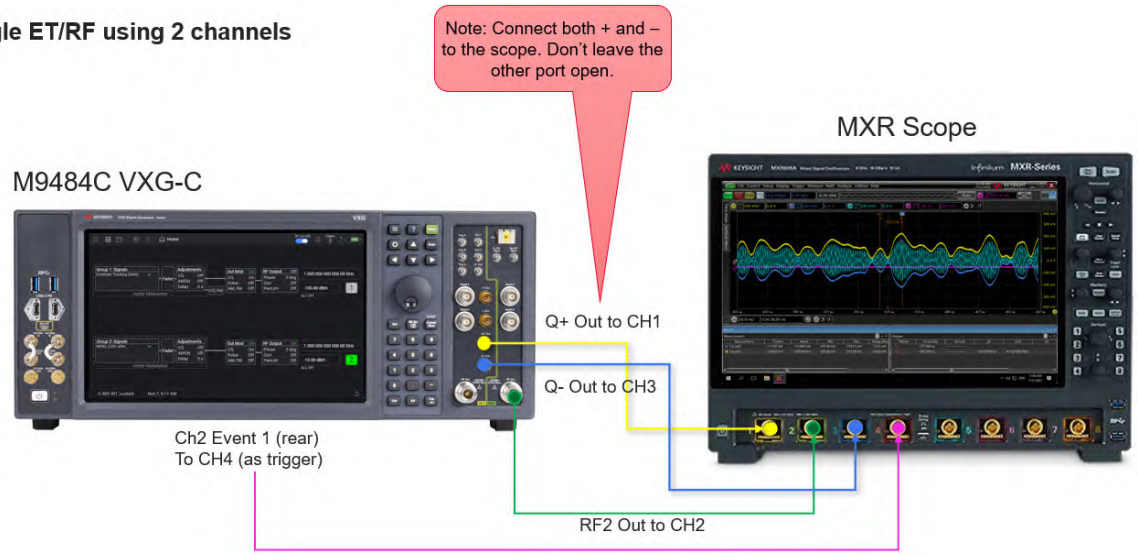
System Requirements

- VXG Firmware Version A.20.00 or higher
- For 2-channel operation - M9484C-PCH, AN1 and DIQ
- For 1-channel operation or dual ET operation -M9484C-PCH, 8SG, AN1 DIQ
- N7655APPC PathWave Signal Generation for Envelope Tracking

NOTE **AN1/DIQ is only available for 1 or 2-channel configurations of the VXG-C.**

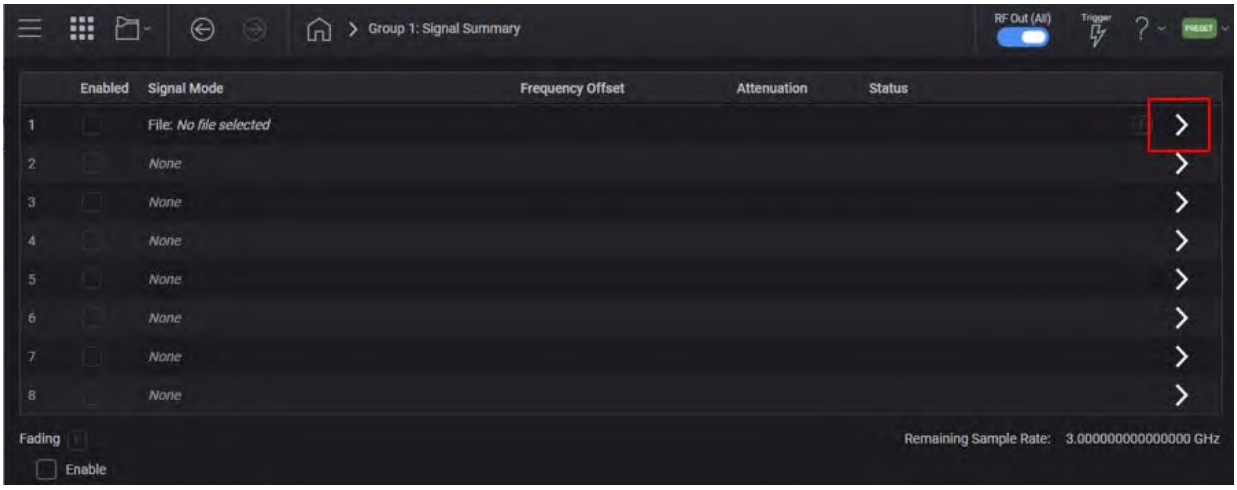
Setup

Single ET/RF using 2 channels

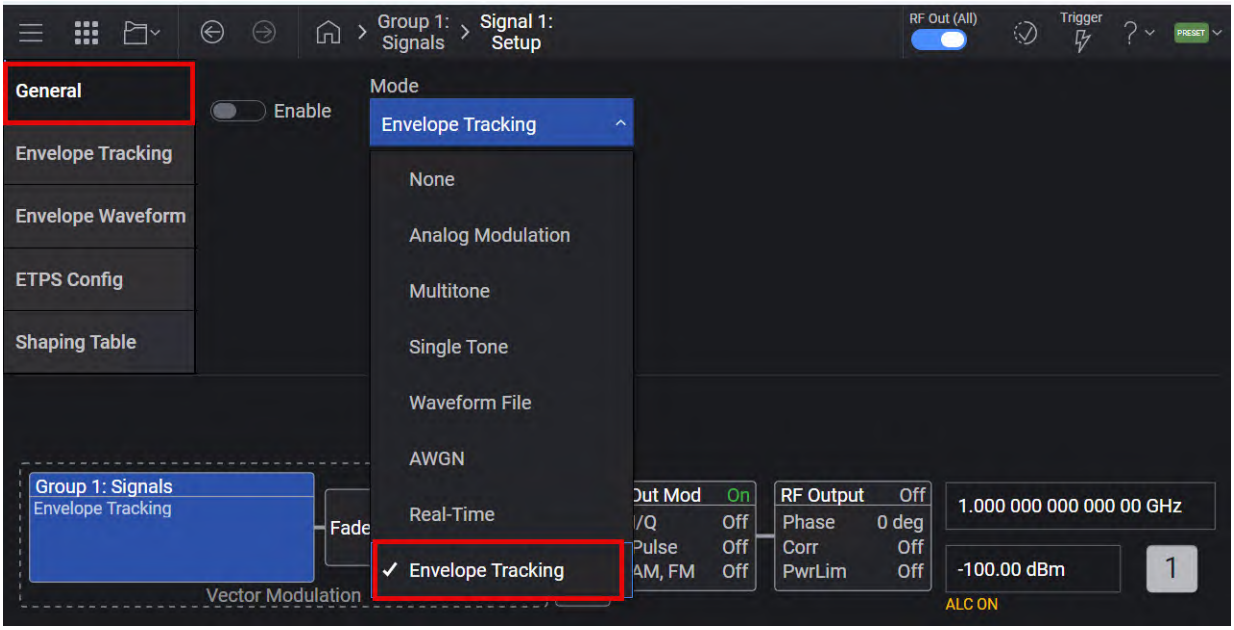


On the VXG

1. Connect the equipment as shown above.
2. Select **Preset** > **Preset** to set the VXG to a known state.
3. Select **Group1: Signals** block.
4. Select the arrow for Signal 1 to open the Signal Setup window.



5. In the **General** tab select Mode dropdown and choose **Envelope Tracking**.



6. Select the **Envelope Tracking** tab and set **Envelope Output to Q (RF2)**

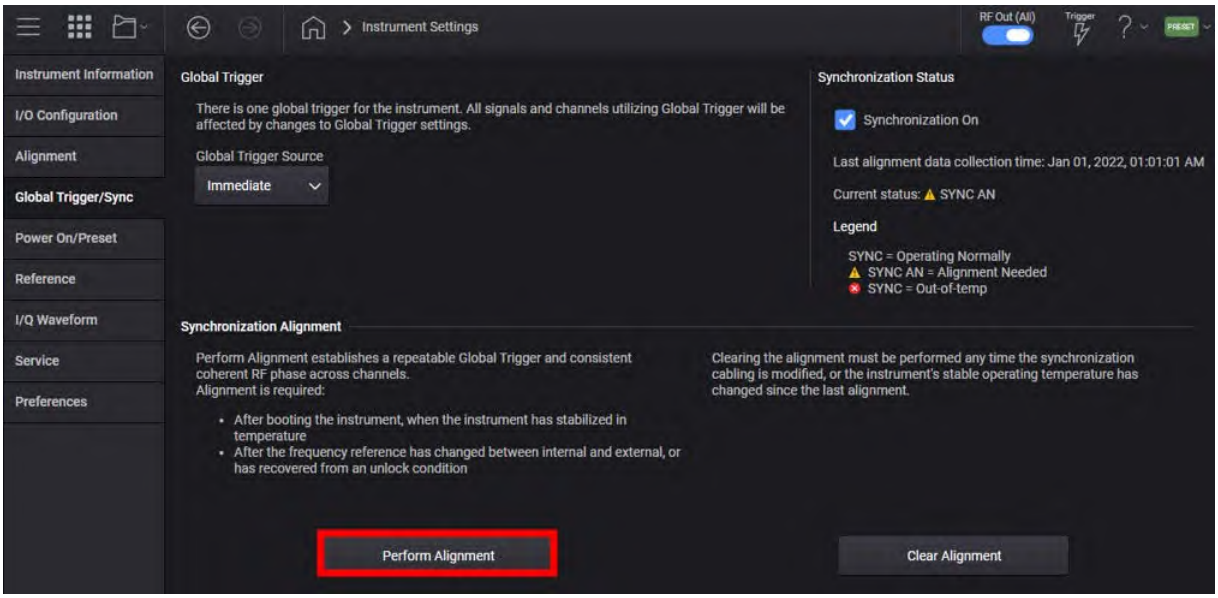
   Note: If Sync Align hasn't been performed, the Synchronization dialog box will be displayed.

   – Select **Perform Alignment**. When completed, select the back arrow at the top of the display.

   Selecting Envelope Tracking mode automatically configures the designated RF signal with the following couplings.

   – Trigger is set to Global Trigger

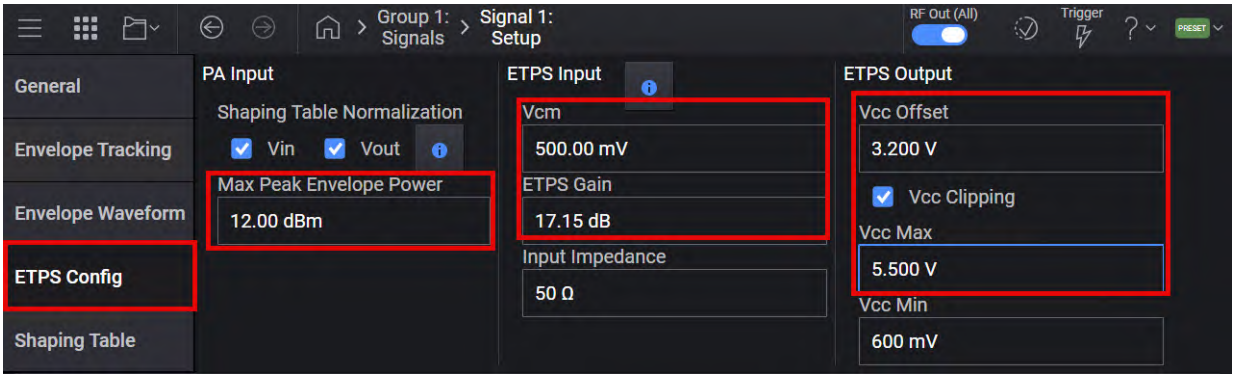
– Group 2 Signals Trigger delay is set to 10 µs (to allow for negative envelope delay)



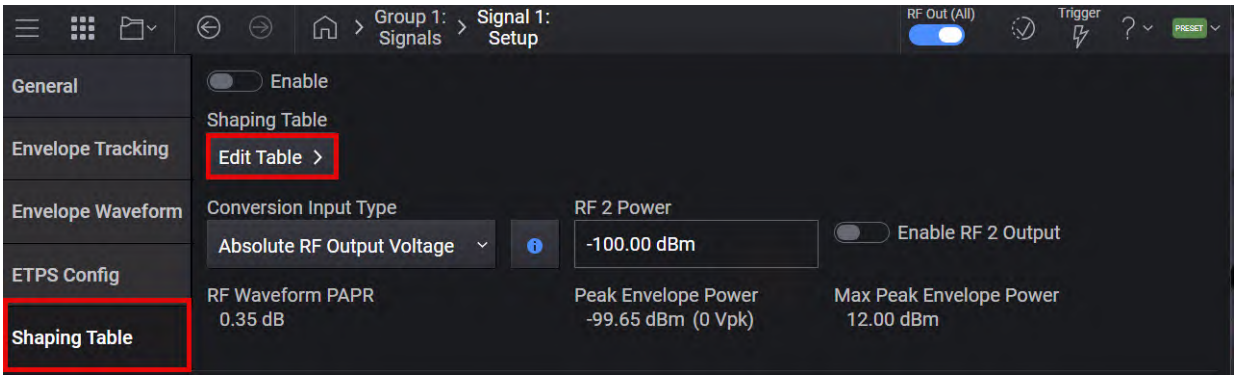
Notice that the Envelope will be Generated automatically by default.

7. Select the **ETPS Config** tab and set:
   – Max Peak Envelope Power to **12 dBm**
   – Vcm to **500 mV**
   – ETPS Gain to **17.15 dB**
   – Vcc Offset to **3.2V**
   – Select **Vcc Clipping** to turn on

– Vcc Max to **5.5 V**



8. Select the **Shaping Table** tab > **Edit Table**.

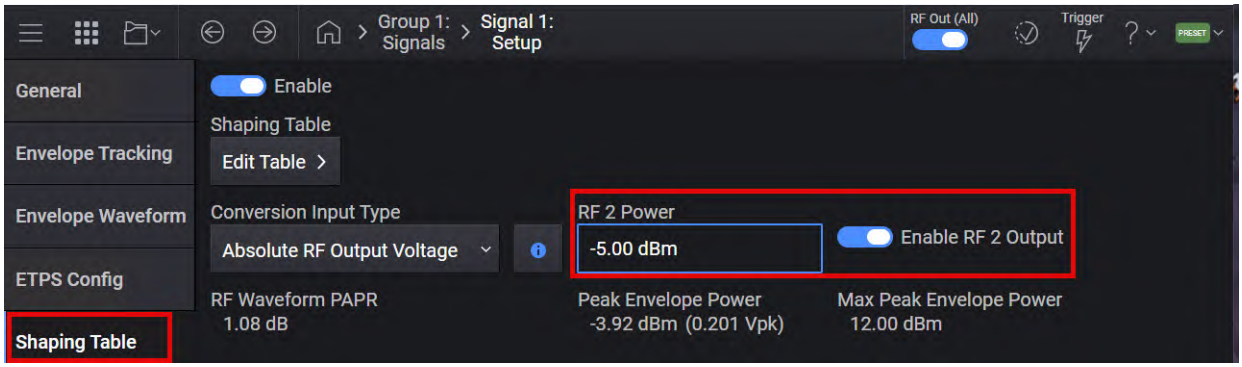


9. Set:

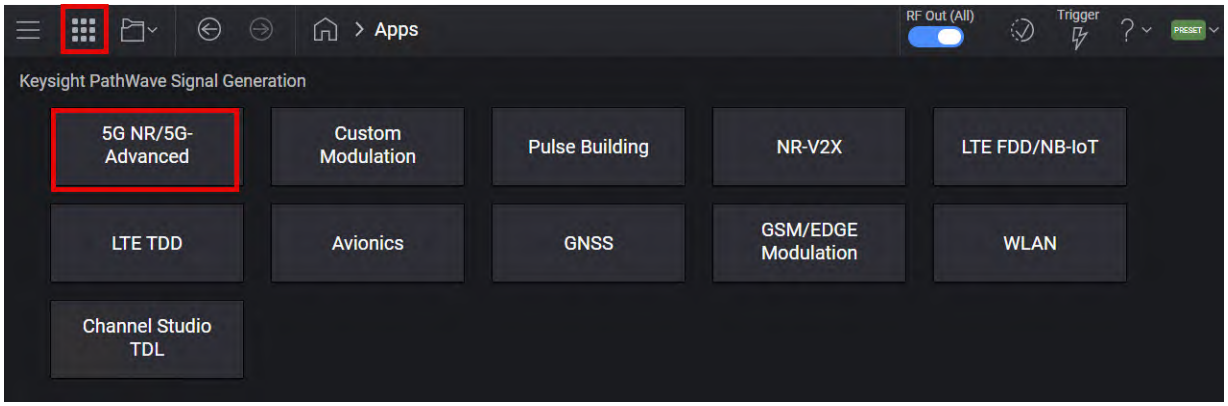
– Row 1 Vin to 0 - Vout to 0.050
– Row 2 Vin to 0.050 - Vout to 0.050
– Add a row
– Row 3 Vin to 0.950 - Vout to 0.950
– Add a row

– Row 4 Vin to 1.0 - Vout to 0.950

10. Select the **Back** arrow button at the top of the display and set **RF 2 Power** to **-5 dBm** > Enable **RF 2 Output** > Enable **On**.

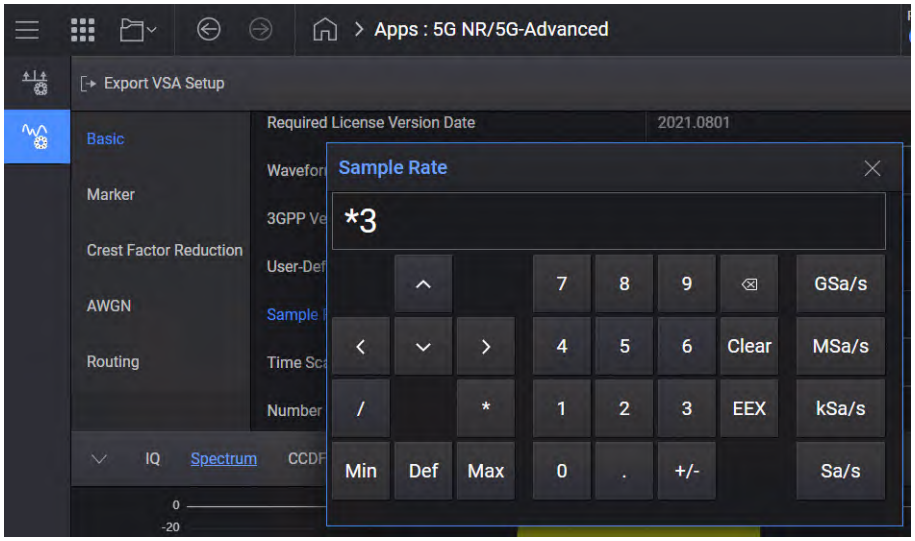


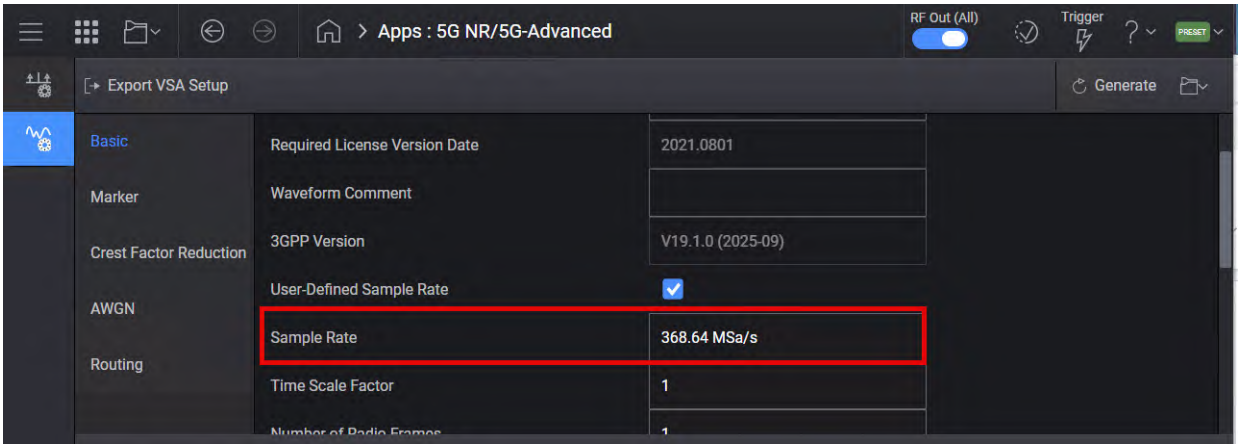
11. From the top of the display, select the **Apps** block icon > **5G NR/5G-Advanced**.



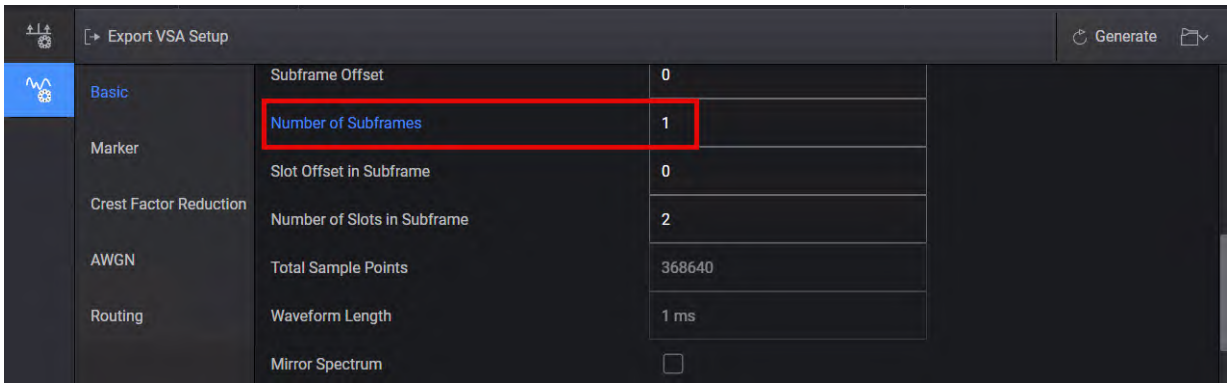
12. In the **Carrier** tab, set Carrier Type to **Uplink**.

13. Select the **Waveform** tab, select **User-Defined Sample Rate**, and then enter Sample Rate as ***3** to set 3x oversampling > **Enter**.





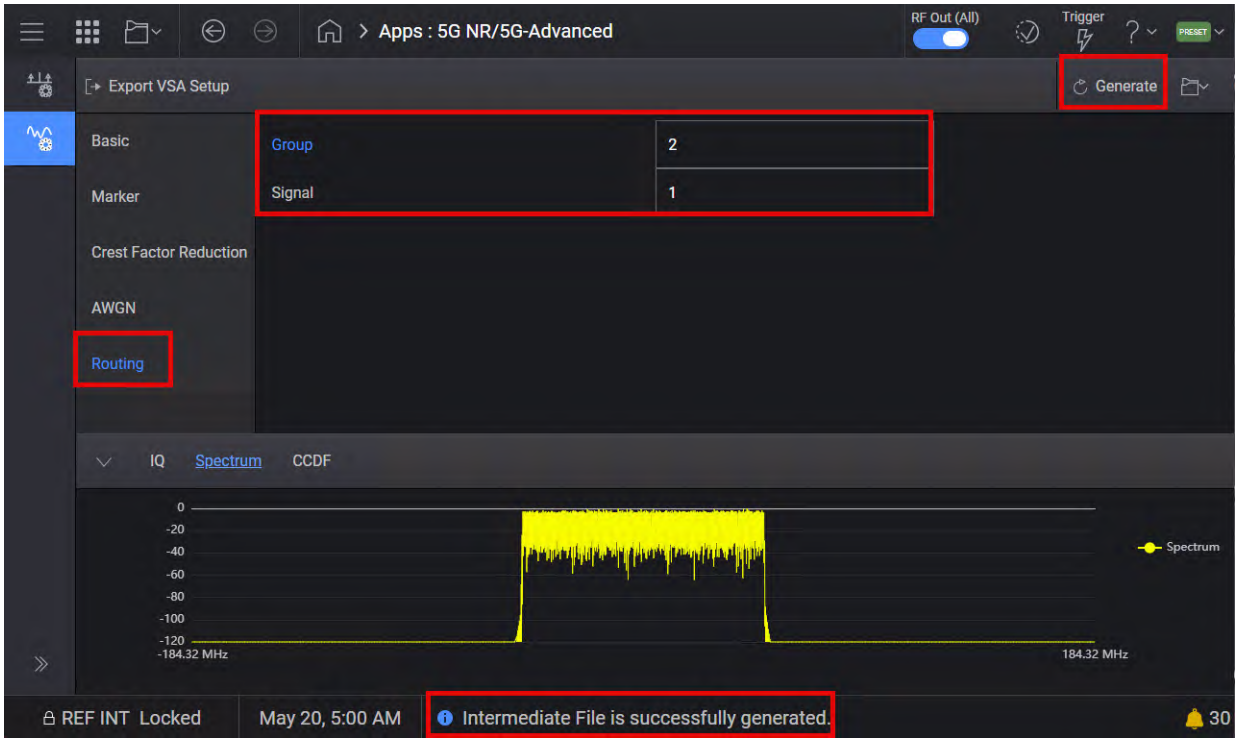
14. Scroll down and set the Number of Subframes to **1**.



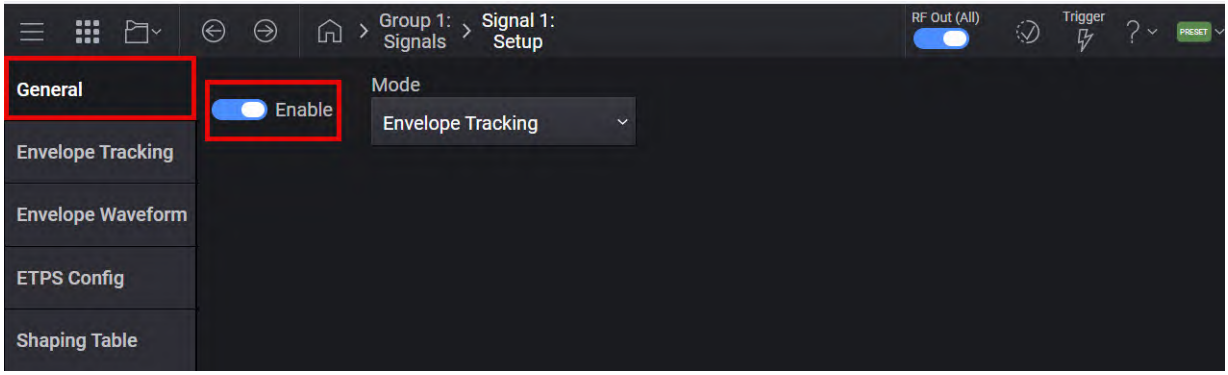
15. Select the Routing tab and set:

- Group to **2**
- Signal to **1**
- **Generate** the waveform

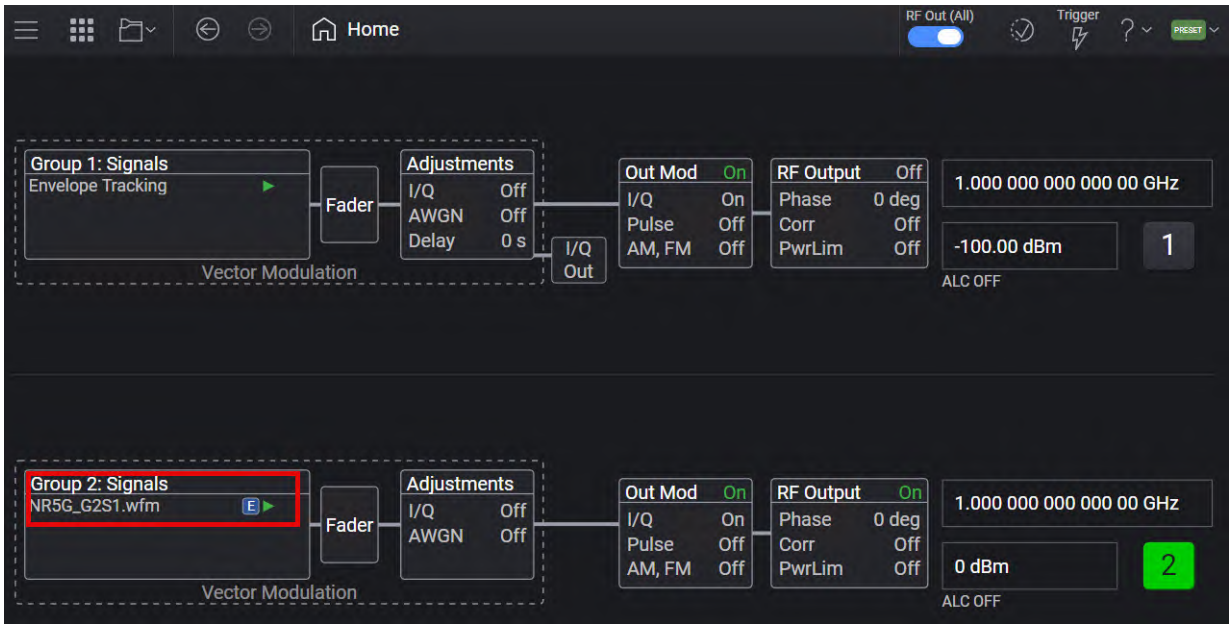
The Auto Generate checkbox is selected by default. Thus, the Intermediate Envelope waveform will be generated automatically when a new RF waveform is loaded.



16. Select the **General** tab > **Enable**.



17. Select the **Home** icon and notice "E" is shown on the Group 2 Signals view indicating it is coupled with Envelope Tracking.

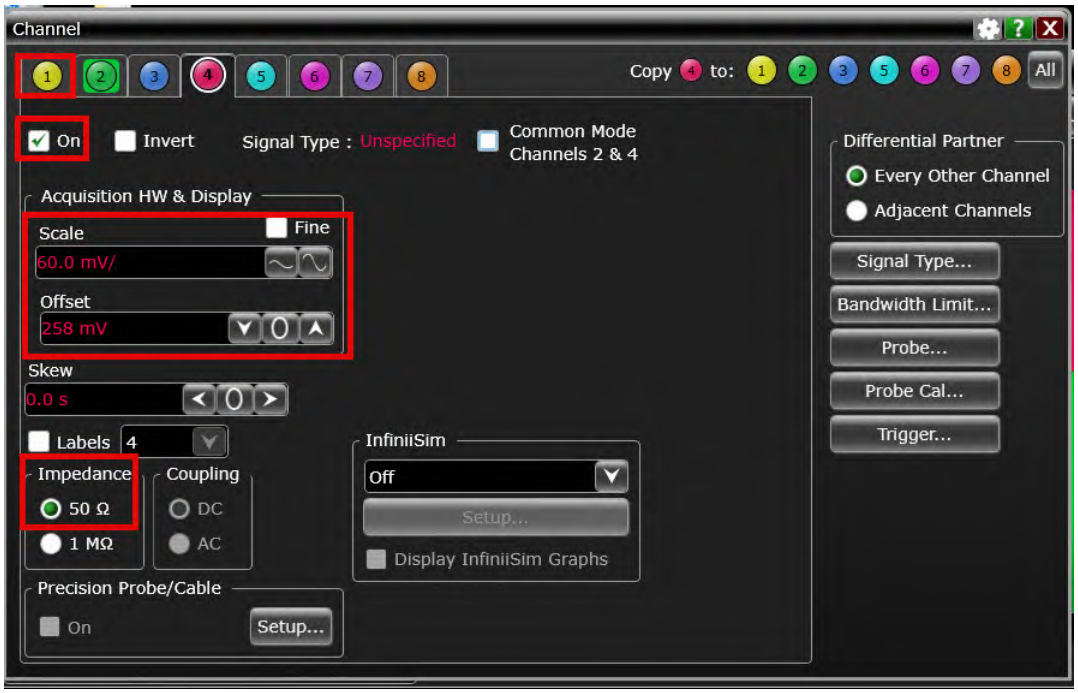


On the MXR

1. Select **Control** > **Default Setup** to set the MXR to a known state.

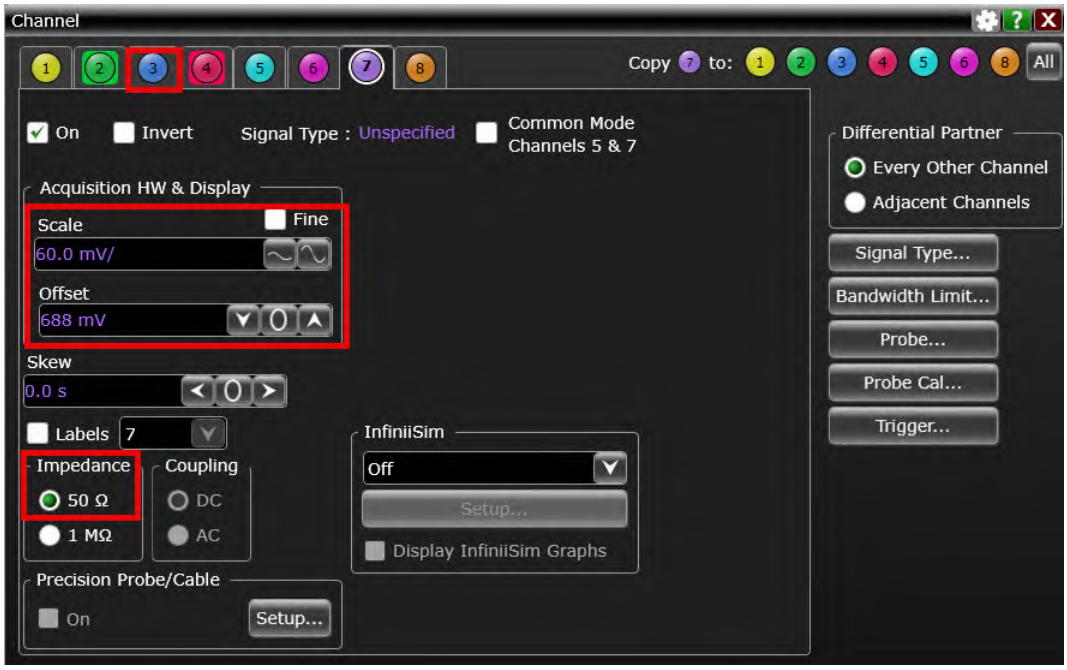
   Adjust oscilloscope display settings to measure RF and envelope. Follow the steps below to set up the scaling to match this waveform.

2. Select > **Setup** > Channel 1, VXG Q+ Out (channel 4 with current setup) and set to **50 Ω**. Change Scale to **60 mV** > Offset to **258 mV** > turn **On** Channel 1.

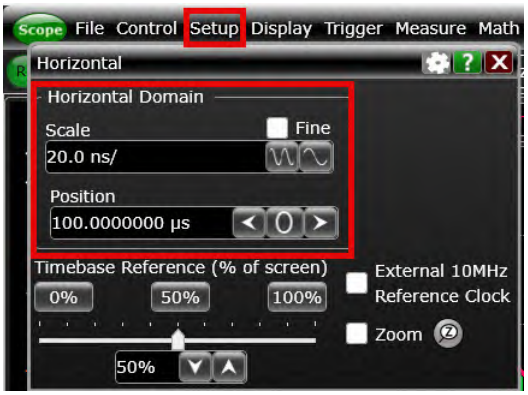


3. Select Channel 2 and set to **50 Ω**. Change Scale to **200 mV** > Offset to **-34 mV** > turn **On** Channel 2.

4. Select Channel 3, VXG Q1 Out (channel 7 for current setup) and set to **50 Ω**. Change Scale to **60 mV** > Offset to **688 mV** > turn **On** Channel 3. **Close** the Channel Setup dialog box.

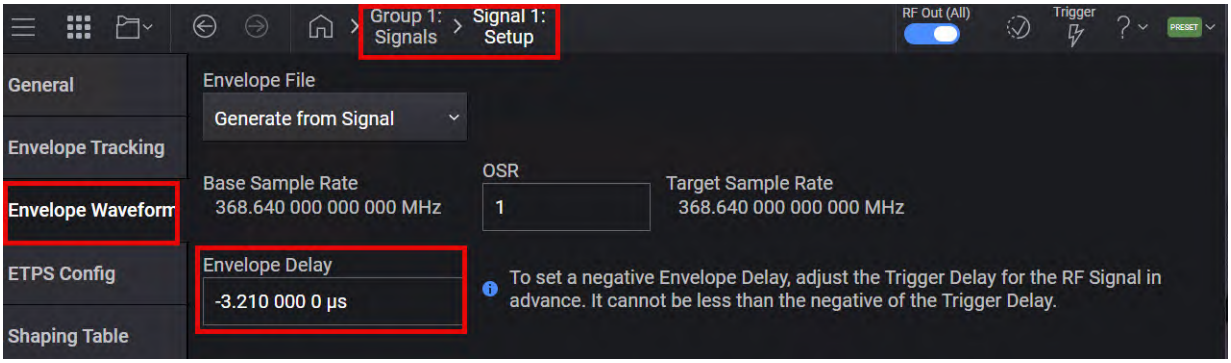


5. Select **Setup** > **Horizontal** and set Scale to **20 ns** and Horizontal Position to **100 us**.



6. On the **VXG**, select the **Signal Block 1 Signal 1** and select the **Envelope Waveform** tab. Set Delay to **0 µs**.

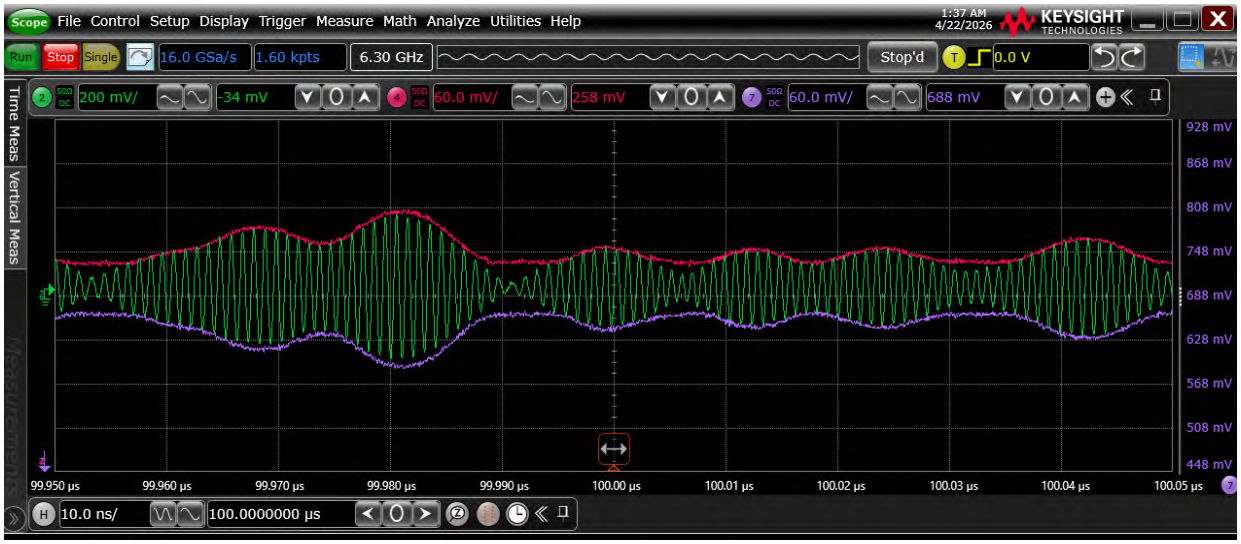
Note that the initial envelope delay is compensated by default. RF and Envelope are roughly aligned with Envelope Delay is 0 µs.



7. On the MXR, select **Single** Sweep.
8. This will roughly align RF and Envelope. Adjust the Delay to further align timing between RF and Envelope by looking at the MXR display.

   Increase delay to move the envelope towards the right.

   If you see DAC over range with higher power, reduce Scale under the VXG Signal 1 setup > Envelope Tracking tab.



4 Other X-Series Signal Analyzer Measurements

- "Setting Up a Group Delay Measurement" on page 398
- "Setting Up Millimeter-Wave Measurements" on page 409
  - "Setting up a 5G NR Millimeter-Wave Measurement" on page 409
  - "Setting-up a Millimeter-Wave DVB-S2X 64APSK Measurement" on page 418
  - "Setting-up a Millimeter-Wave 5G NR, 2 GHz BW FR2 Measurement" on page 440
- "Using the X-Series Analyzer's SCPI Recorder Function" on page 464
- "Using the X-Series Analyzer's Preload/Unload Function" on page 477
- "Using Real-time Spectrogram Streaming (Option RRT) for N9032B and N9042B" on page 479

Setting Up a Group Delay Measurement

Group delay is a measure of phase distortion. Group delay is the actual transit time of a signal through a device under test as a function of frequency. When specifying group delay, it is important to specify the aperture used for the measurement.

In a group delay measurement:

- The linear phase shift component is converted to a constant value (representing the average delay).
- The higher order phase shift component is transformed into deviations from constant group delay (or group delay ripple).
- The deviations in group delay cause signal distortion, just as deviations from linear phase cause distortion.
- The measurement trace depicts the amount of time it takes for each frequency to travel through the device under test.

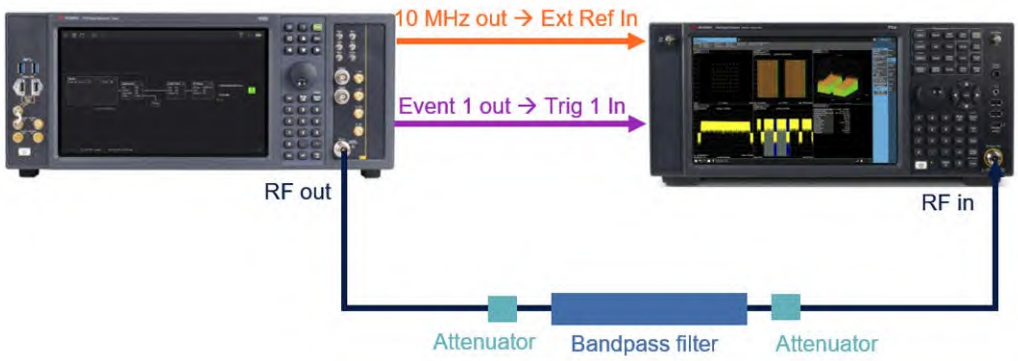
In this measurement example, we will use the M9484C VXG to generate a wideband modulated signal, drive this through a filter and the N9042B UXA will analyze the output signal and display the filter characteristics (gain, phase, and group-delay). Also, all settings to the VXG will be made via the Connection Management setup in the N9042B/N9056EM0E application.

We will start by running a calibration to remove the fixture characteristics (cables, connectors, and other passive components) between the source and the device under test (DUT), in this case the filter. The filter in this example is a 500 MHz bandpass filter, centered at 10.27 GHz. Settings my need to be adjusted for your filter.

For this example we are using the follow equipment:

- M9484C
    - Frequency range to 13.5 GHz (M9484C-514, M9484C-520, M9484C-532, or M9484C-544, or M9484C-554)
- N9042B (you can also use a N9040B, N9033B, N9030B, N9020B, or N9021B)
    - N9056EMOE Channel Quality/Group Delay Measurement application
- Accessories
    - Two 3 dB attenuators
    - Baseband filter

1. Connect the two 3 dB attenuators, cables and adapters between the VXG and UXA (omitting the filter).



   After running a calibration to remove the effects of the fixturing, we will insert the filter between the two attenuators.

2. From the N9042B, select **Mode/Meas** > **Channel Quality/Group Delay** Mode > **Group Delay** Measurement > **Quad** display View.

NOTE If accessing the instrument via a Remote Desktop connection, select the Screen tab (at the top of the display) to open the Mode/Measurement/View Selector window.

   The Quad display shows the Spectrum, Gain Phase, Gain Magnitude, and Group Delay measurements.

3. In the **Frequency** menu panel, set the Center Frequency to **10.27 GHz** and Span to **1 GHz**.

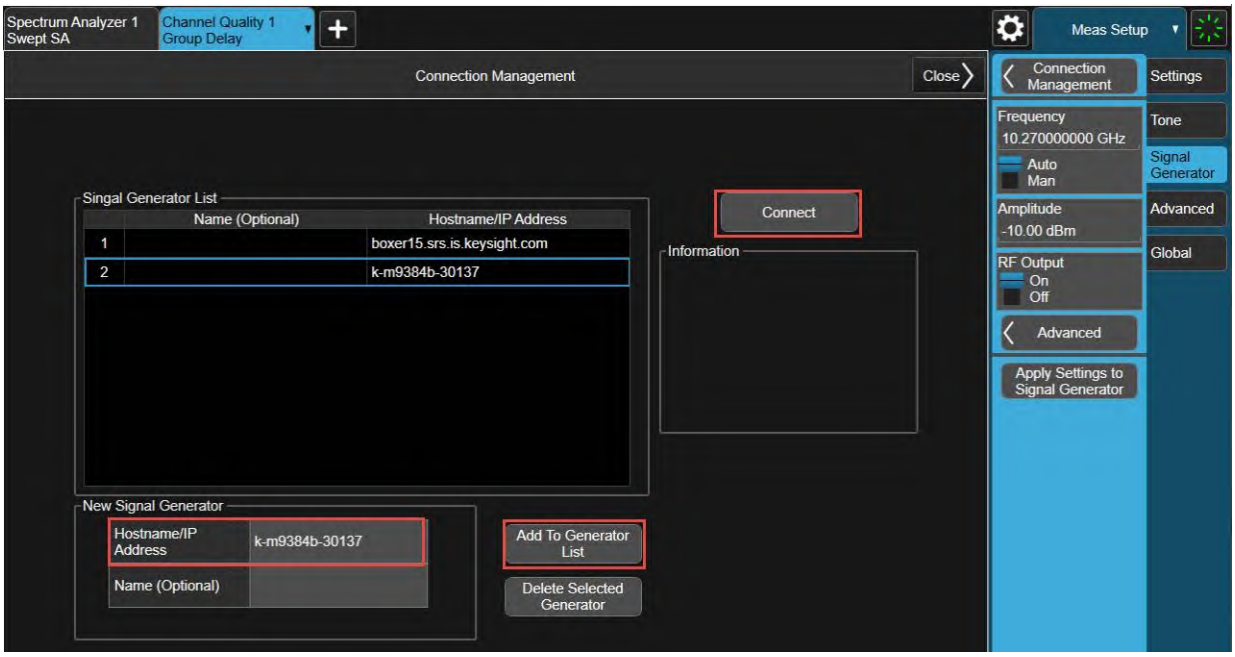
4. In the **Meas Setup** menu panel, select the **Signal Generator** tab and confirm the Center Frequency is set to **10.27 GHz** (the frequency of the filter) and set the Amplitude to **+10 dBm**.

   The power is set to +10 dBm to compensate for the affects of the attenuators on the edges of the trace.This is because the SNR has been reduced by the attenuators and without increasing the power you would be essentially measuring the "noise" at the edges.

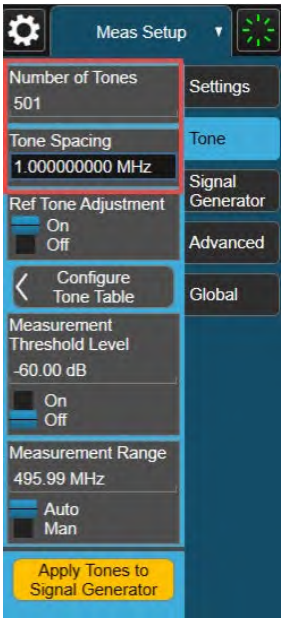
5. Select **Connection Management**.

6. In the New Signal Generator area, specify the **Hostname or IP address** of the VXG, then select **Add to Generator List** > **Connect**.

This allows us to setup the VXG directly from the UXA.



7. Select **Apply Settings to Signal Generator** to send these settings directly to the VXG, then **Close** the Connection Management window.
8. In the **Meas Setup** menu panel select the **Tone** tab and set the Number of Tones to **501** with a **Tone Spacing of 1 MHz** to provide a 500 MHz wideband signal.



Notice that the span has automatically changed to 626.3 MHz to accommodate the Tone settings.

9. Select **Configure Tone Table**. There are lots of settings to choose from, but we will use these default settings.
   - Fill Type to **Random Phase**
   - Random Seed Type to **Fixed**
   - Random Seed to **0**
10. Select **Apply Fill** and all of the phase points get filled in for the 501 points. (Originally there were 64.)
11. Select **Apply Tones to Signal Generator** to setup the multitone signal.

    You will notice there is some ripple in the traces.

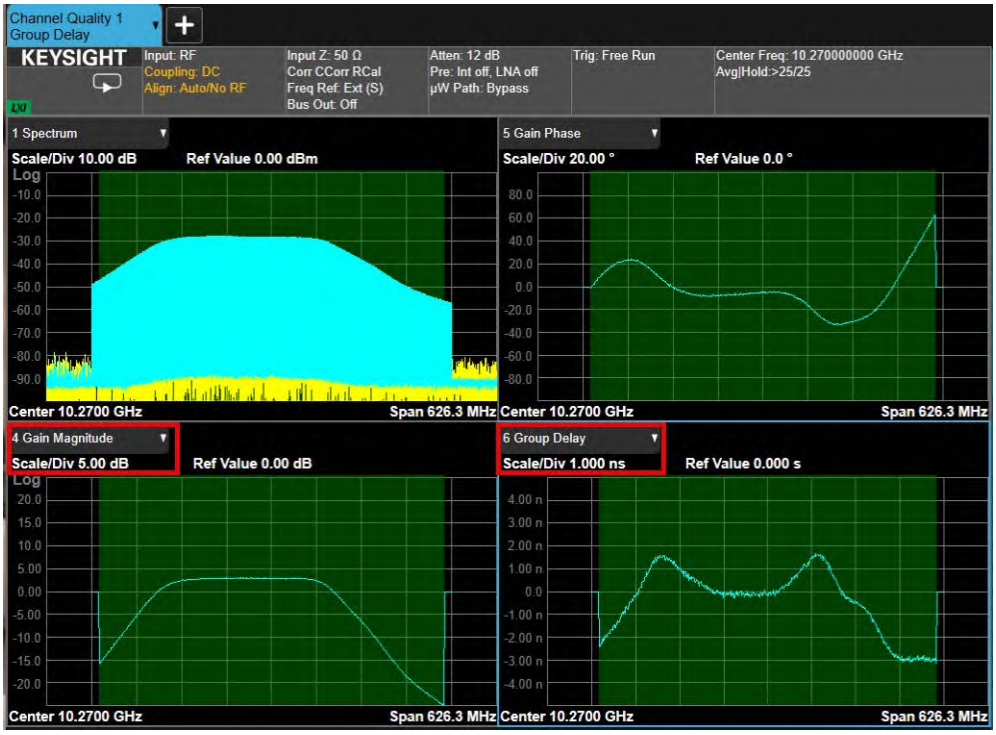
NOTE

If there is an Input Overload in the UXA, go to **Amplitude** > **Attenuation** tab and increase the **Mech Atten 2 dB** at a time until the overload condition is resolved.

12. **Close** the Configure Tone table.
13. Select the **Calibration** tab > **Calibrate Tones**.

    The calibration will calculate the difference between tone definition and average input trace (magnitude and phase).
14. Once the calibration is complete, connect the filter between the two 3 dB attenuators to see the response of the filter.

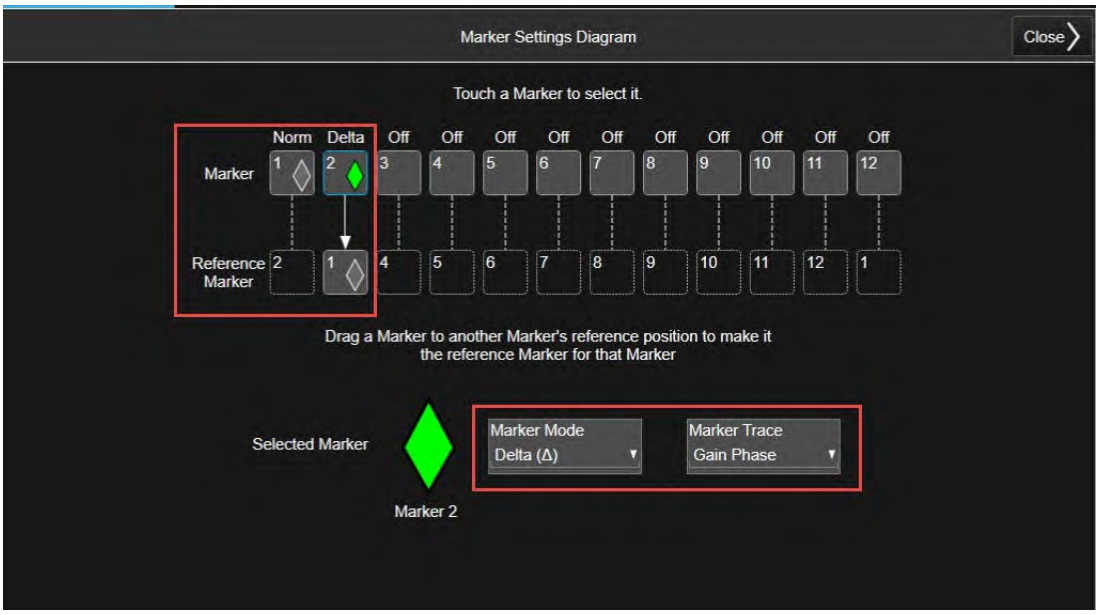
15. For this example, set the **Gain Magnitude Scale/Div to 5 dB** and the **Group Delay Scale/Div to 1 ns**. For your specific measurement adjust the screen settings to a value that allows you to get the best signal resolution on your measurement displays.



16. From the menu panel, select **Marker** > **Settings** > **Marker Settings Diagram**.

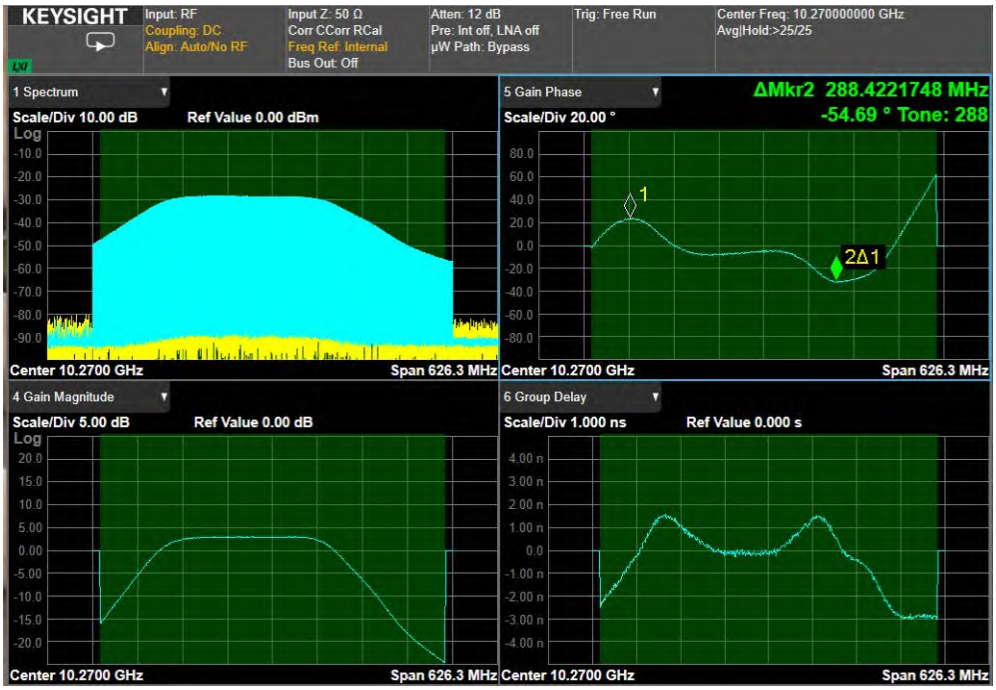
17. Activate Marker 1 by setting Marker Mode to **Normal** and set Marker Trace to **Gain Phase**. Click and drag Marker 1 to the Reference Marker 2 position. Activate Marker 2 by setting Marker Mode to **Delta**, then set Marker Trace to **Gain Phase** to put the marker on the Gain Phase measurement, then **Close** the window.

The Gain Phase is really input versus output instead of absolute phase.

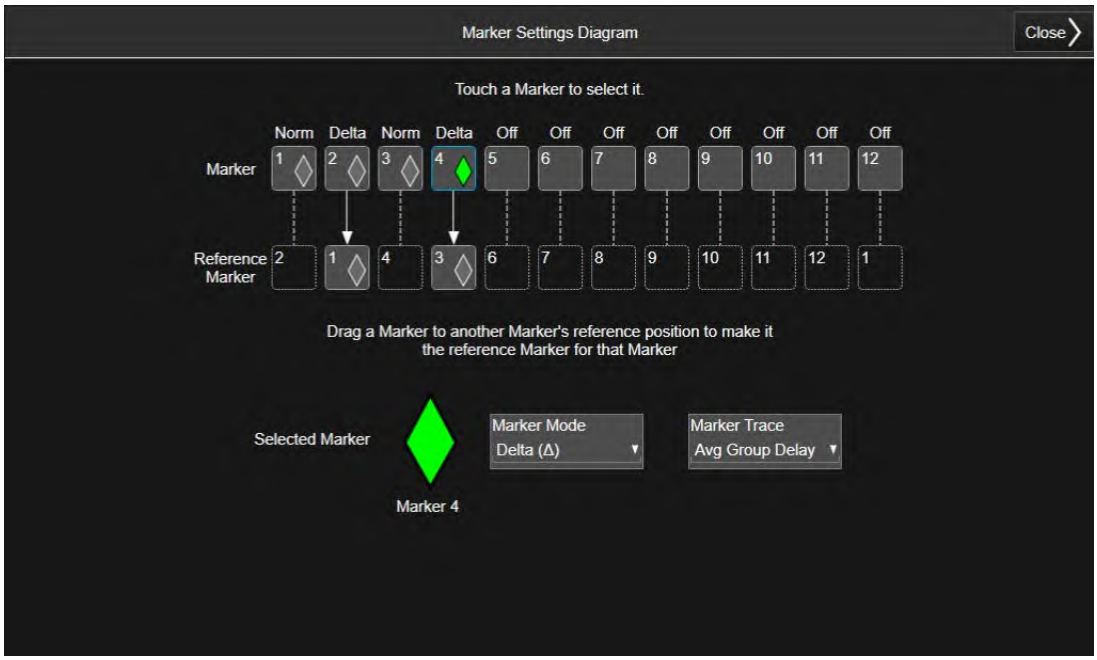


18. Move Marker 1 to the Peak of the Signal within the bandwidth range of the signal and Marker 2 to the minimum.

    This provides the peak-to-peak deviation across this portion of the bandwidth of the filter. For this example, Peak to Peak Deviation is 54.69 °.

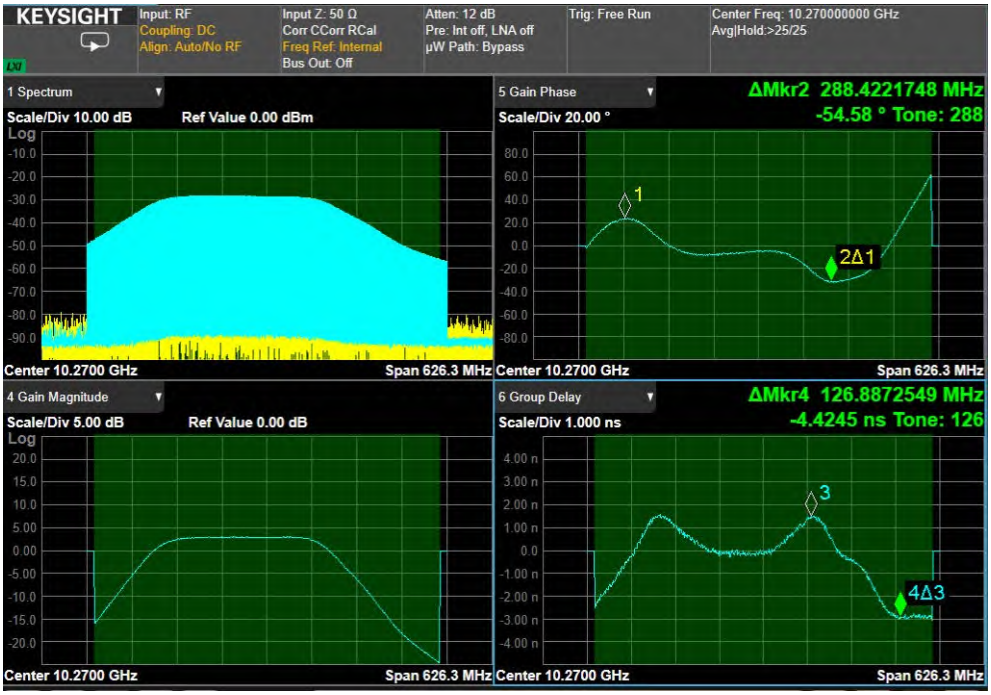


19. Go back into the **Marker** Menu and turn on Marker 3 and set Marker Mode to Normal and Marker Trace to **Avg Group Delay**. Drag Marker 3 to Marker 4's Reference Position. Turn on Marker 4 and set Marker Mode to **Delta** and Marker Trace to **Avg Group Delay**. **Close** the Marker Settings Diagram.

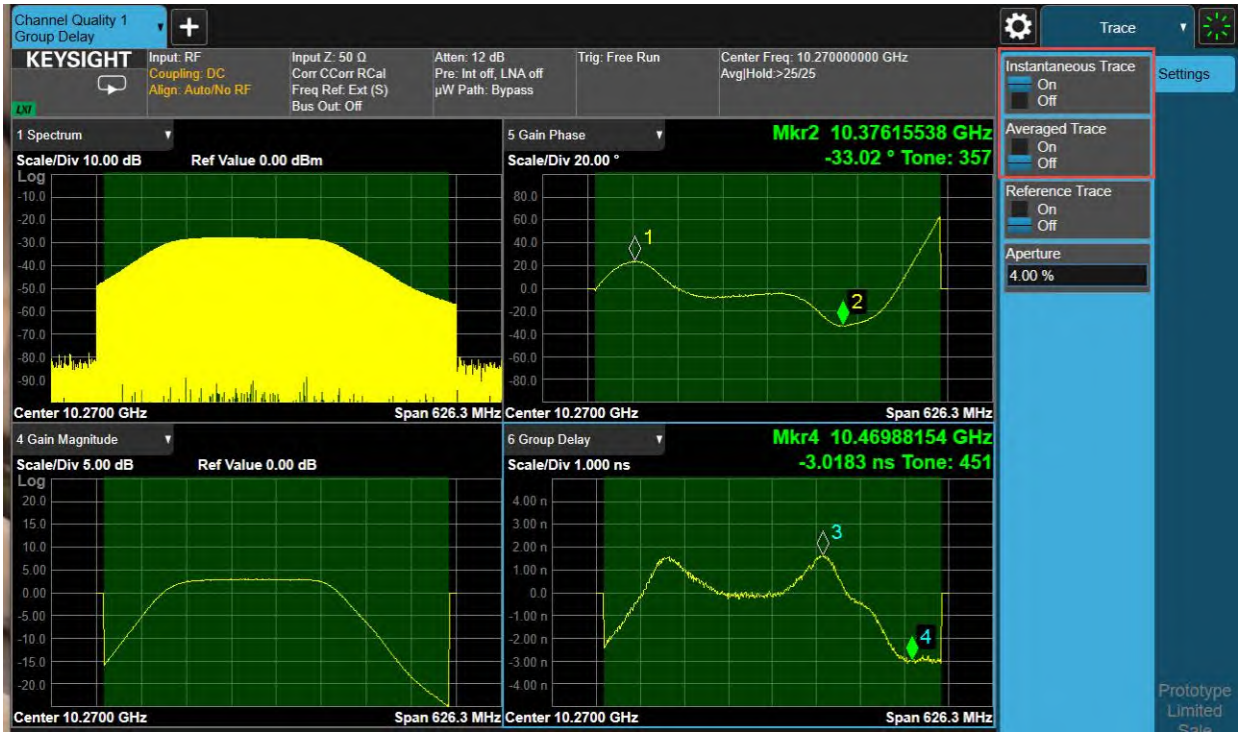


20. Move Marker 3 to the peak of the trace and Marker 4 to the minimum.

For this example, peak to peak Average Group Delay is 4.42 ns.

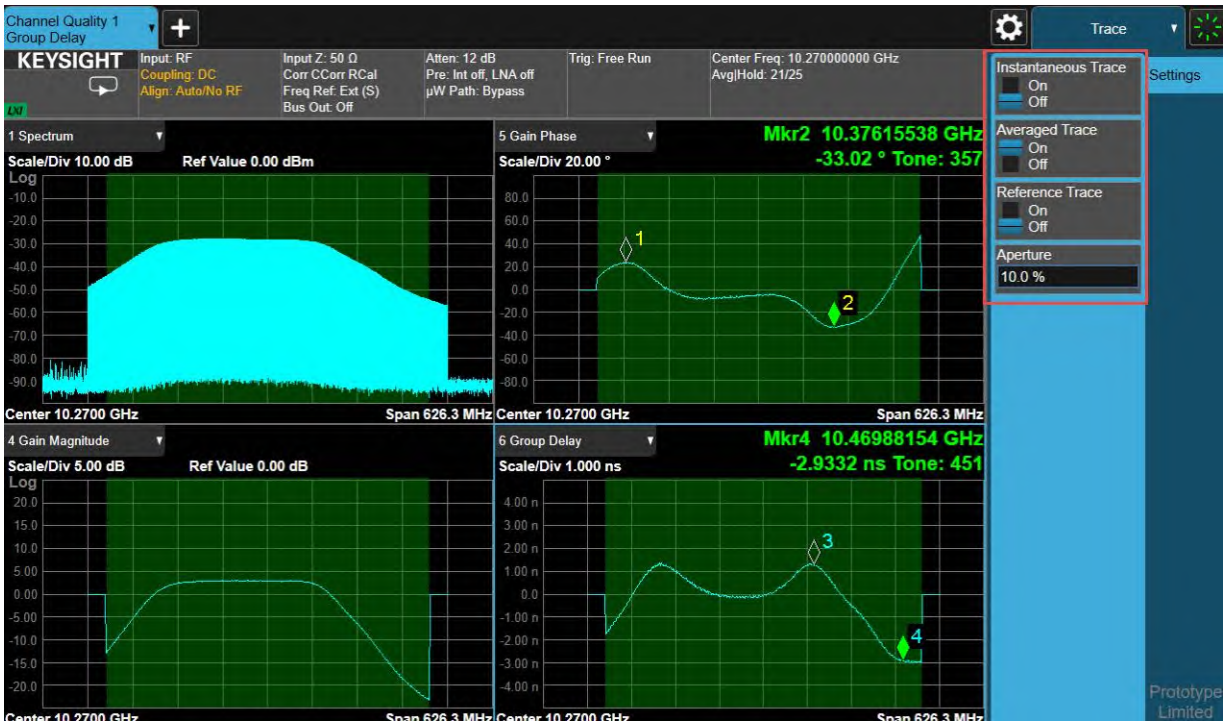


21. From the menu panel, select **Trace** > Averaged Trace **Off**. With Averaged Trace Off, you can capture smalls spurs that might occur on the trace.

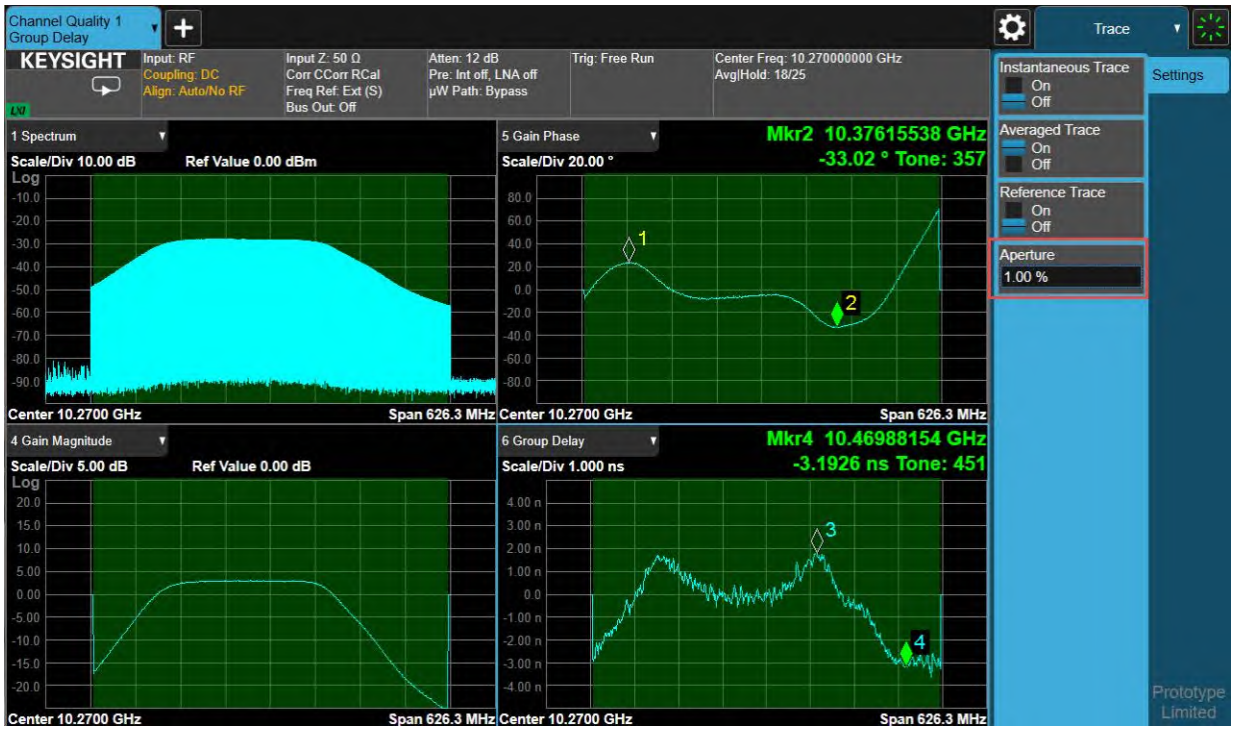


22. Turn Instantaneous Trace **Off** and Averaged Trace **On** then set the Aperture to **10%** to get a smoother result, but lose a little bit more at the beginning and the end of the 500 MHz band.

The Group Delay Aperture is the span over which the math of the delta phase over delta frequency is calculated.



23. Now set Aperture to a very narrow aperture of **1%**. Any deviation or noise in the phase trace will show up a lot more. However as it averages down, the noise will average away.



24. From the N9042B, select **Mode/Meas** > **Channel Quality/Group Delay** Mode > **Group Delay** Measurement > **3x2** View.

The 3x2 View will add the RF Envelope (Instantaneous Envelope) and Metrics. Other interesting Views to look are the IQ Waveform that is similar to the RF Envelope, or add the Phase View. Phase only is the absolute phase of the tone.

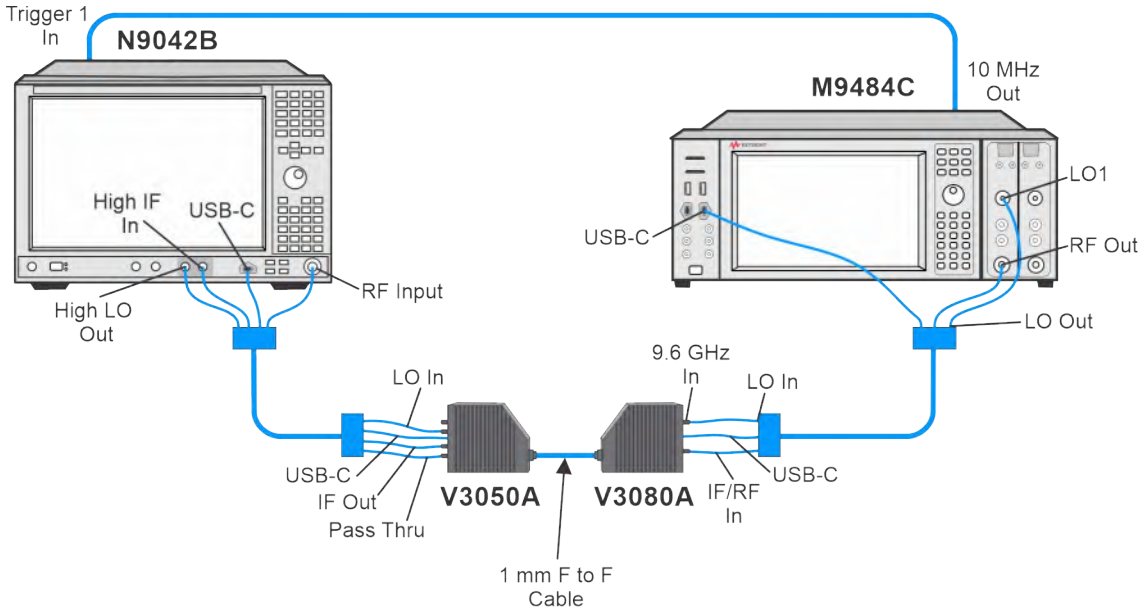
Channel Quality 1
Group Delay
KEYSIGHT
Input: RF
Coupling: DC
Align: Auto/No RF
Input Z: 50 Ω
Corr CCorr RCal
Freq Ref: Ext (S)
Bus Out: Off
Atten: 12 dB
Pre: Int off, LNA off
µW Path: Bypass
Trig: Free Run
Center Freq: 10.270000000 GHz
Avg|Hold:>25/25
1 Spectrum
Scale/Div 10.00 dB
Ref Value 0.00 dBm
Center 10.2700 GHz
Span 626.3 MHz
5 Gain Phase
Mkr2 10.37615538 GHz
-33.03 ° Tone: 357
Scale/Div 20.00 °
Center 10.2700 GHz
Span 626.3 MHz
4 Gain Magnitude
Scale/Div 5.00 dB
Ref Value 0.00 dB
Center 10.2700 GHz
Span 626.3 MHz
6 Group Delay
Mkr4 10.46988154 GHz
-3.1936 ns Tone: 451
Scale/Div 1.000 ns
Center 10.2700 GHz
Span 626.3 MHz
3 RF Envelope
Scale/Div 10.00 dB
Ref Value 0.00 dBm
0.000 s
30.166 us
Meas Time 30.166 µs (23615 pts)
9 Metrics
Basic Sample Averaged
Signal Power -4.17 dBm -4.18 dBm
Frequency Error 0 Hz 0 Hz
Clock Error 0.0001 ppm 0.0009 ppm
Time Offset 152.3146 ns 595.5620 ns
Group Delay Sample Averaged Max Hold
RMS 67.1852 ps 67.1886 ps 67.3144 p

Setting Up Millimeter-Wave Measurements

The V3080A Vector Signal Generator Frequency Extenders are designed to extend the frequency range of Keysight's M9484C for millimeter-wave applications up to 110 GHz.

The V3050A Signal Analyzer Frequency Extenders are unbranded, preselected devices designed to extend the frequency range of Keysight's N9042B UXA for millimeter-wave applications up to 110 GHz.

We will connect the output of the M9848C VXG to the V3080A, then connect the output of the V3080A to the input of the V3050A, and finally connect the V3050A to the UXA to measure frequencies at 60 GHz and above. For complete setup instructions, refer to the V3080A Startup Guide.

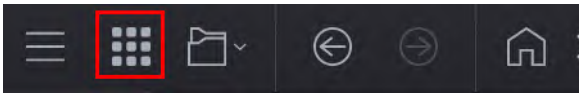


Setting up a 5G NR Millimeter-Wave Measurement

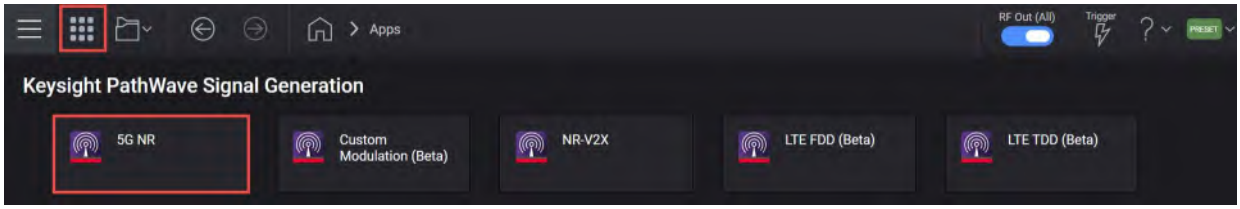
NOTE The VXG must have the N7631APPC Signal Generation for 5G NR license installed.

On the VXG:

1. Select **Preset** > **Preset** to set the VXG to a known state.
2. In the Output area, set Frequency to **60 GHz** and Power to **-10 dBm**.
3. Select the **Radio Apps** block to open the mode selection panel.

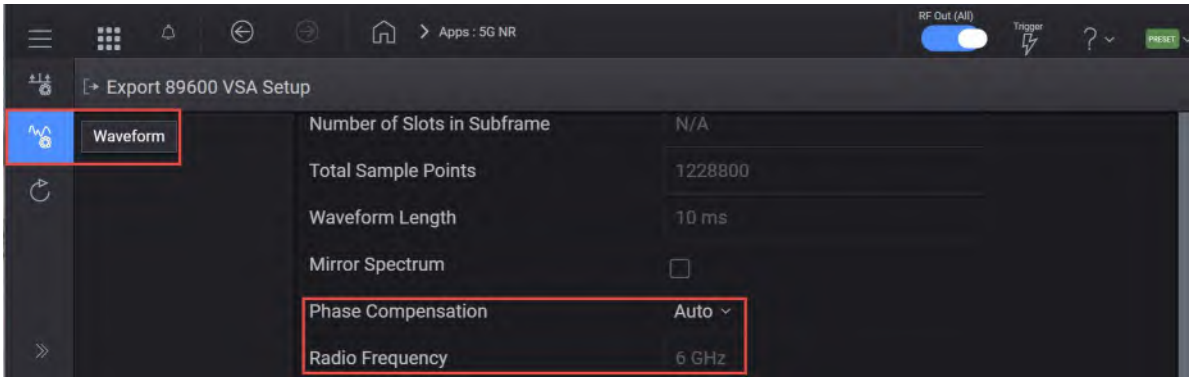


4. Select **5G NR/5G-Advanced** to enter the 5G NR signal mode.



5. In the 5G NR setup, select the **Waveform** tab and observe the Phase Compensation and Radio Frequency settings.

Phase Compensation is applied by default at baseband for RF up-conversion. This means it depends on the carrier frequency, and the waveform should only be played at the carrier frequency it was generated for, unlike conventional Signal Studio waveforms, which are independent from carrier frequency. Phase Compensation is set to Auto by default.

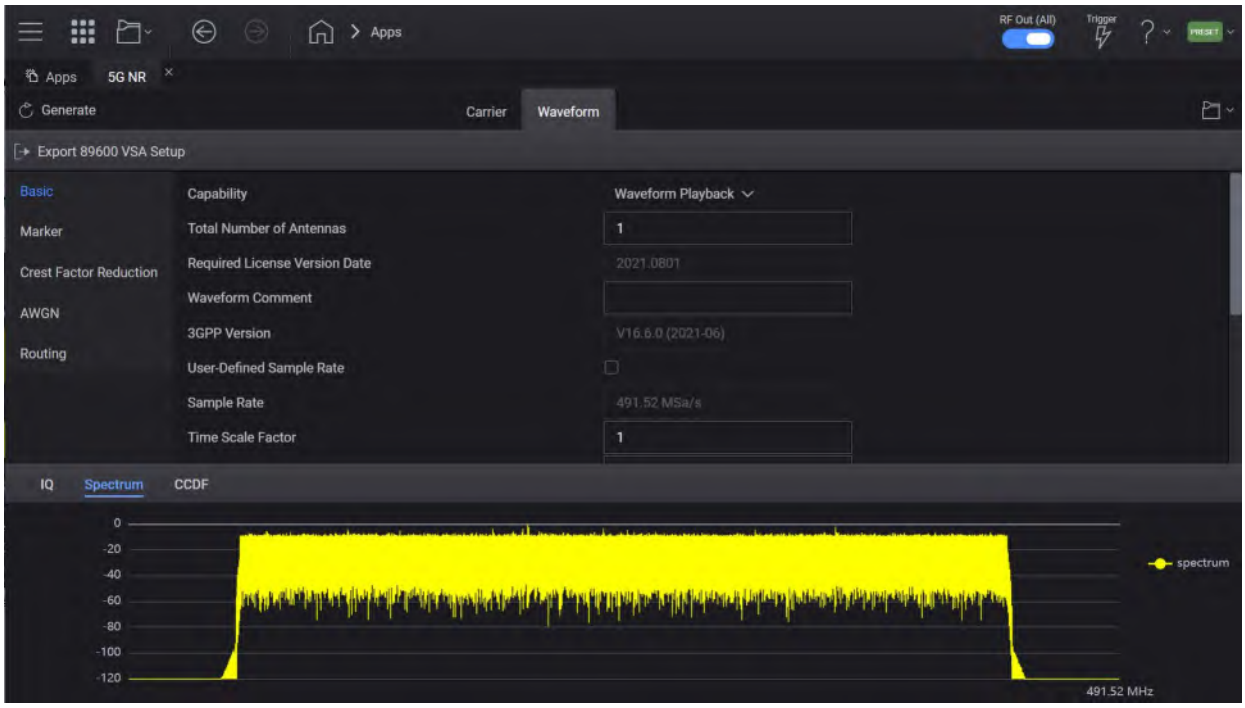


TIP Some applications, like power amplifier measurements use the same waveform at different frequencies and can be time consuming to generate a separate waveform for each frequency. In this case, you would turn Phase Compensation Off, so the waveform becomes independent from the carrier frequency. You need to do the same on the analysis side, or the demodulation will fail.

6. Select the **Carrier** tab.
7. Select **Test Model** and set Bandwidth to **FR2 400 MHz** and **Test Model to NR-FR2-TM3.1a**.

Notice that the description of the currently selected test model appears to the right. These test models are defined in the section 3GPP TS 4.9.2 of 38.141.

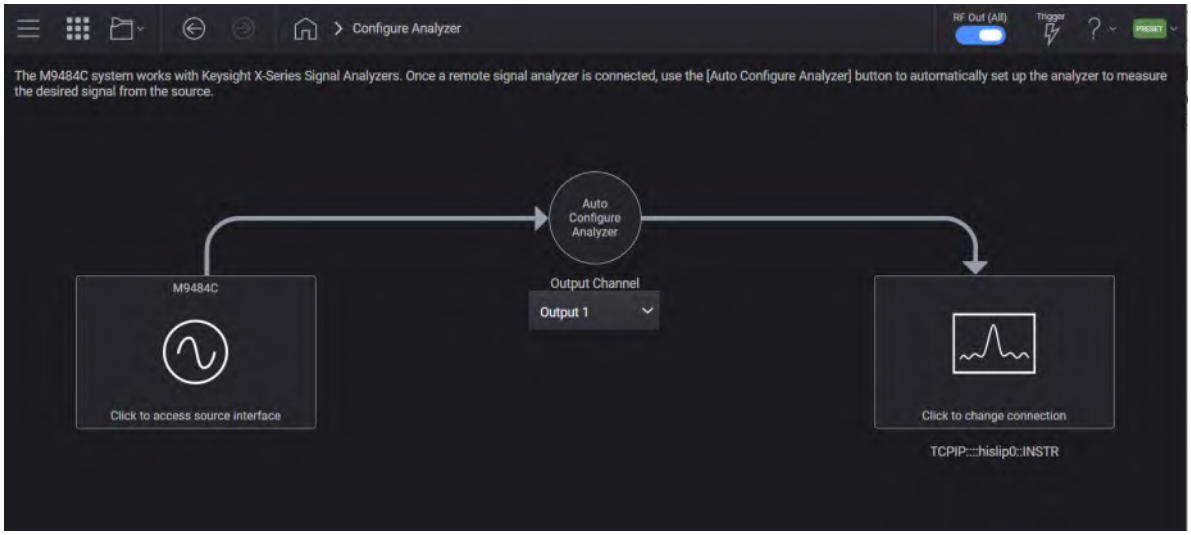
8. Select **OK** to exit the Test Model setup.
9. Select **Generate** to generate the Waveform, and then select **Home** to exit the setup panel.



10. Set RF Out to **On** by selecting the numbered channel indicator switch.

This enables the RF Out for the indicated channels, in this case Channel 1 if using a multi-channel VXG.

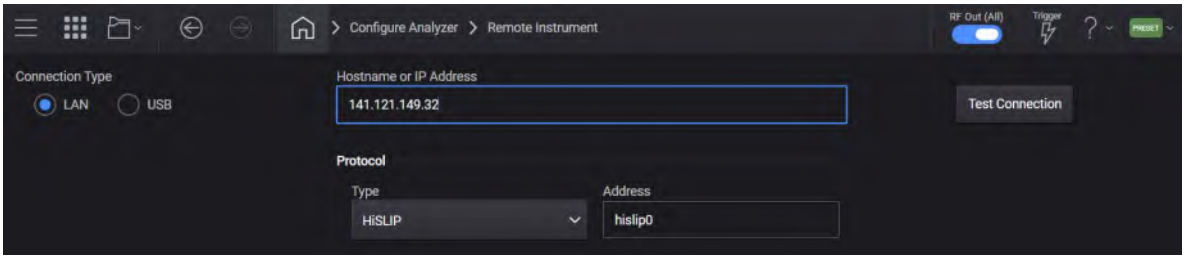
11. Select the System menu (triple bar tab at the top left of the window) to open the System Menu and then select **Configure Analyzer**.



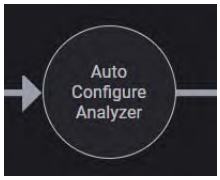
12. In the System Configuration setup, click on the Remote Signal Analyzer block (on far right) to setup the communication channel to the Keysight X-Series Signal Analyzer.

13. Set Connection Type to **LAN**.
14. In the Connect Remote X-Series Signal Analyzer setup, enter the Hostname or IP Address for the signal analyzer.
15. Under Set Protocol, select **HiSLIP** and use the Remote name **hislip0**.

16. Select **Test Connection** to verify, then **Back**.



17. Select **Auto Configure Analyzer** to send the VXG settings to the analyzer.



18. On the N9042B, select **Meas Setup** > **Optimize EVM**.

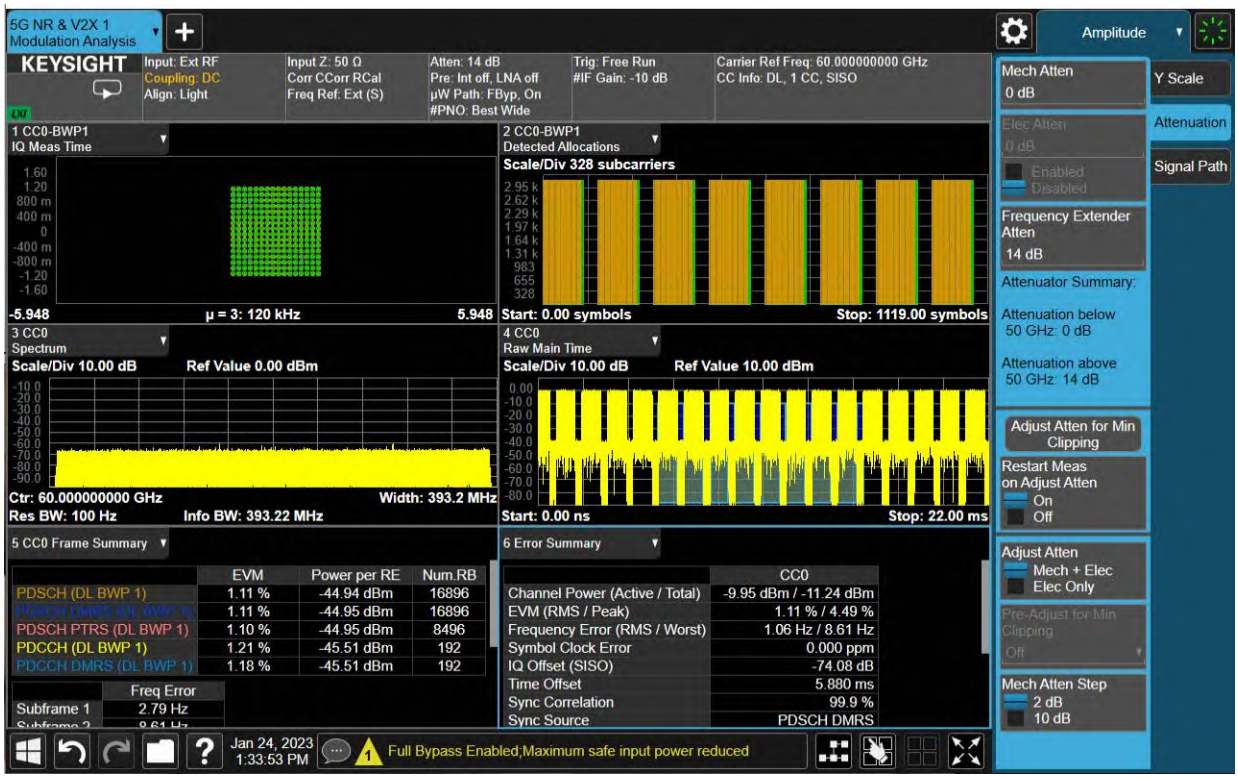
Optimize EVM is an immediate action to adjust the hardware settings to minimize EVM.

- Normal method will capture input signal, measure its peak power then the algorithm will find proper setting combination for attenuation, preamp and IF Gain.

– Iterative method will capture input signal multiple times in a iteration process, demodulate the signal and calculate EVM for each iteration and find setting combination for attenuation, preamp and IF Gain with minimum EVM.

  For 5G NR signals, this method is much slower, than normal, but with achieve better EVM results. You can also manually adjust IF Gain, Frequency Extender Attenuation (use when measurement setup includes a V3050A frequency extender).

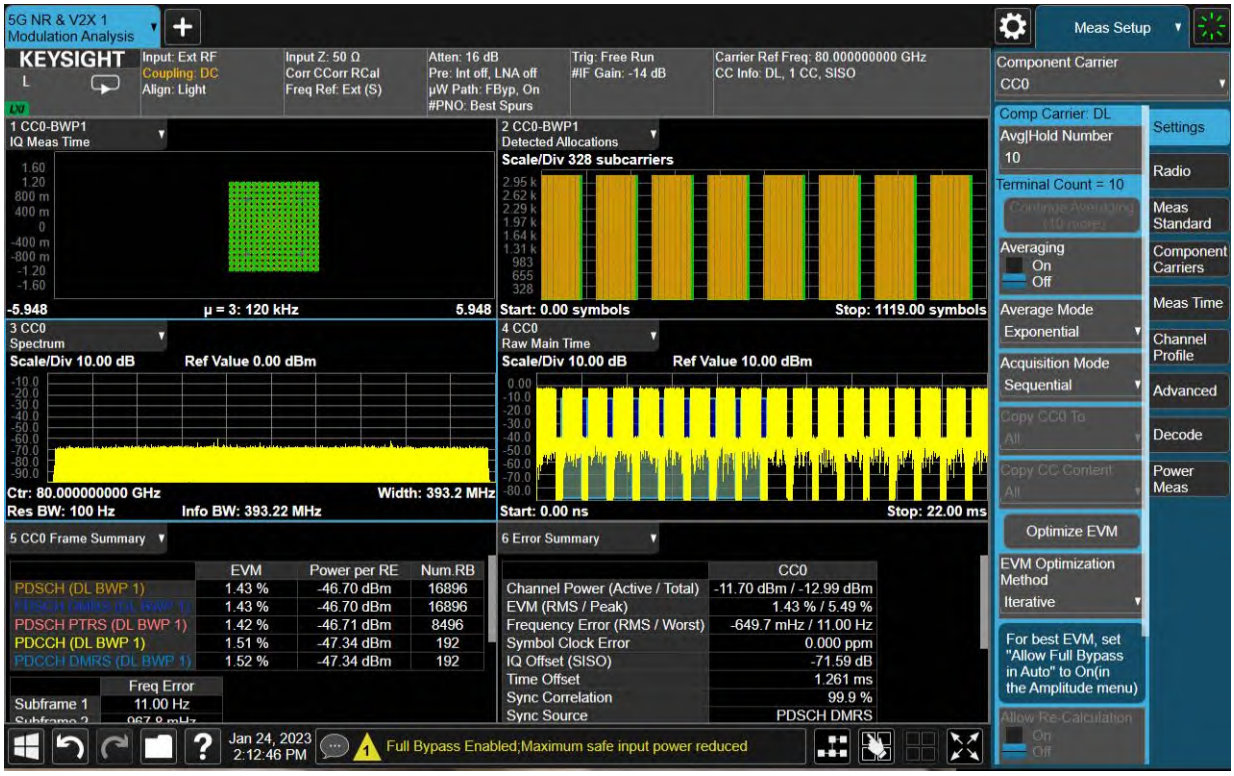
19. View the results on the signal analyzer.



20. Change the frequency and power level of the M9484C and frequency of the N9042A to view other results. Remember to Optimize EVM after each change.

This measurement was done at 70 GHz, - 10 dBm.

This measurement was done at 80 GHz, - 10 dBm.



This measurement was done at 85 GHz, - 10 dBm.

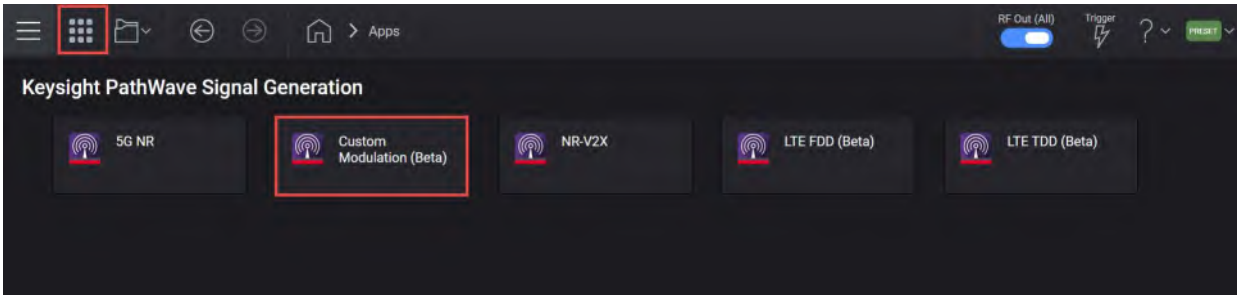
Setting-up a Millimeter-Wave DVB-S2X 64APSK Measurement

NOTE The VXG must have the N7608C Signal Studio for Custom Modulation license installed.

Setting up the VXG

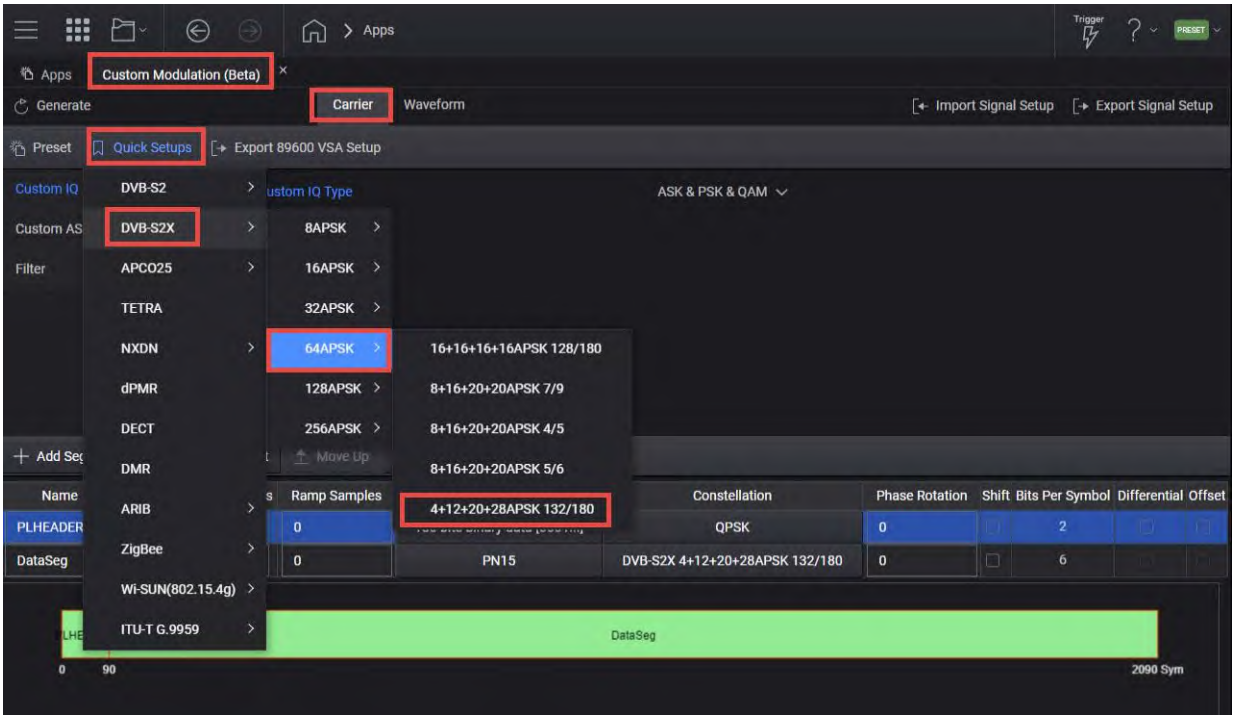
1. Select **Preset** > **Preset** to set the VXG to a known state.
2. In the Output area, set Frequency to **81 GHz** and Power to **-10 dBm**.
3. Select the **Radio Apps** block to open the mode selection panel.

4. Select **Custom Modulation** to enter Custom Modulation Signal Mode.

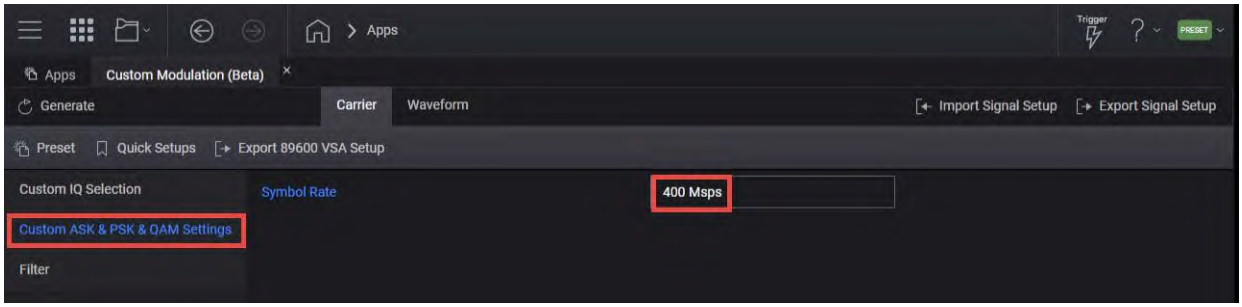


5. Select the **Carrier** tab.

6. In the Custom Modulation setup, select the **Carrier** tab > **Quick Setups** > **DVB-S2X** > **64APSK** > **4+12+20+28APSK 132/180**.

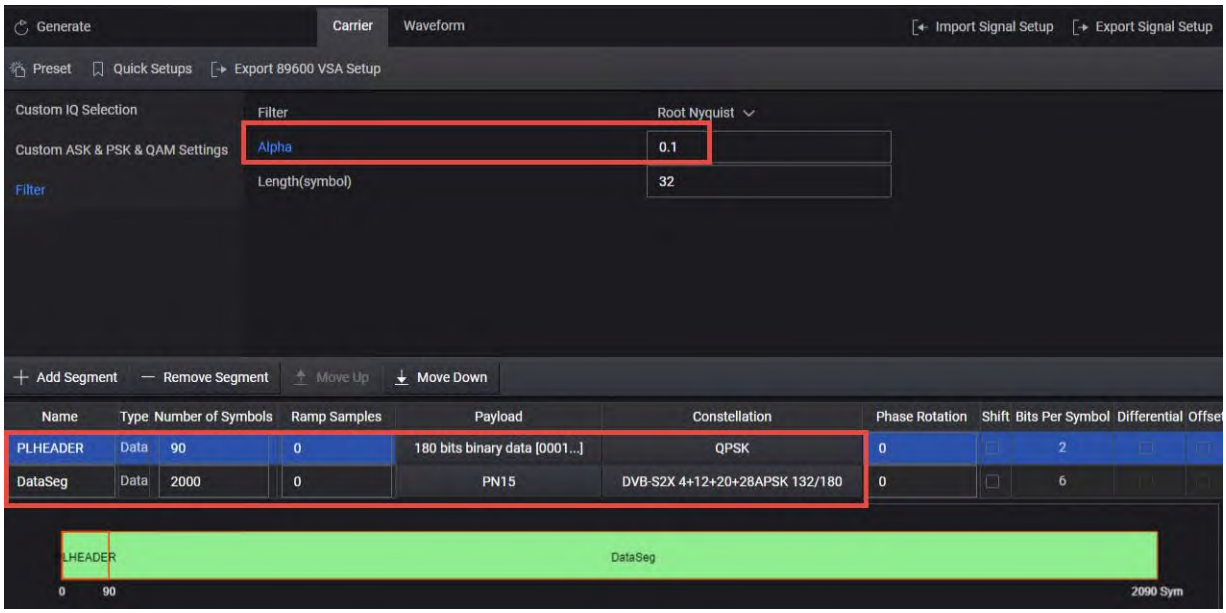


7. From the left pane, select **Custom ASK & PSK & QAM Settings** and set the Symbol Rate to **400 Msps** per channel.

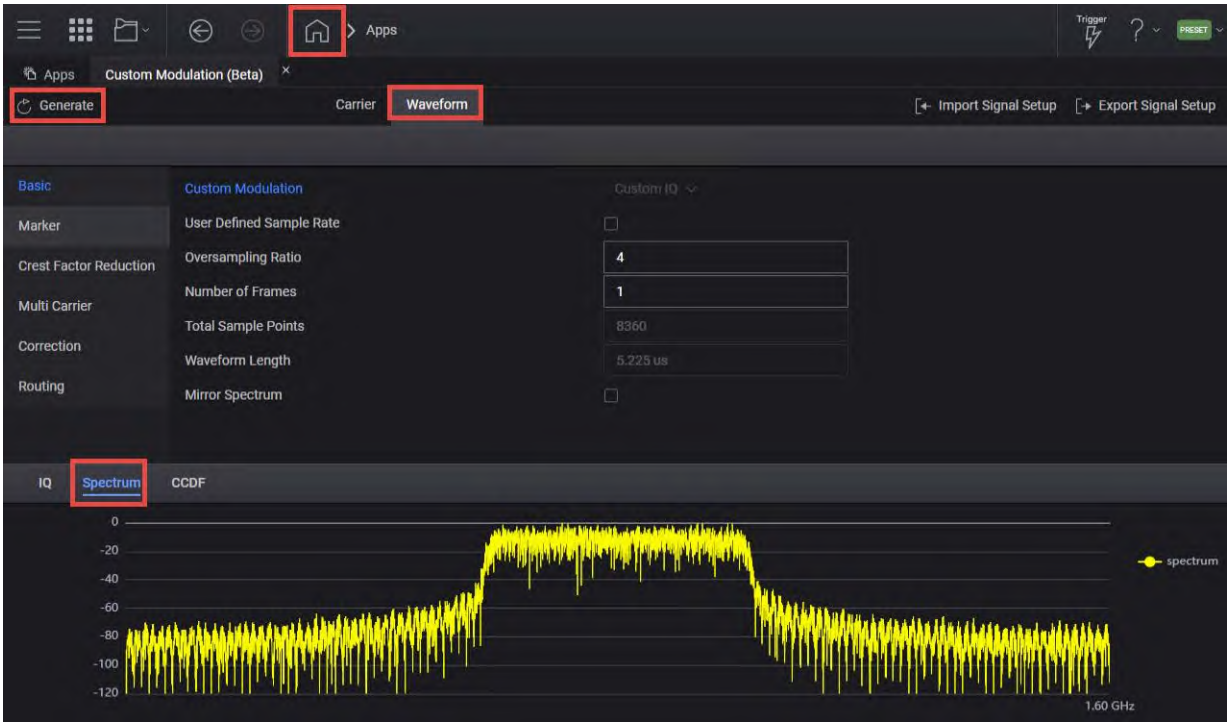


8. Select **Filter** > Alpha to **0.1**.

We will match these settings in the UXA setup.



9. Select the Waveform tab, then **Generate** to generate the Waveform, and then select **Home** to exit the setup panel.

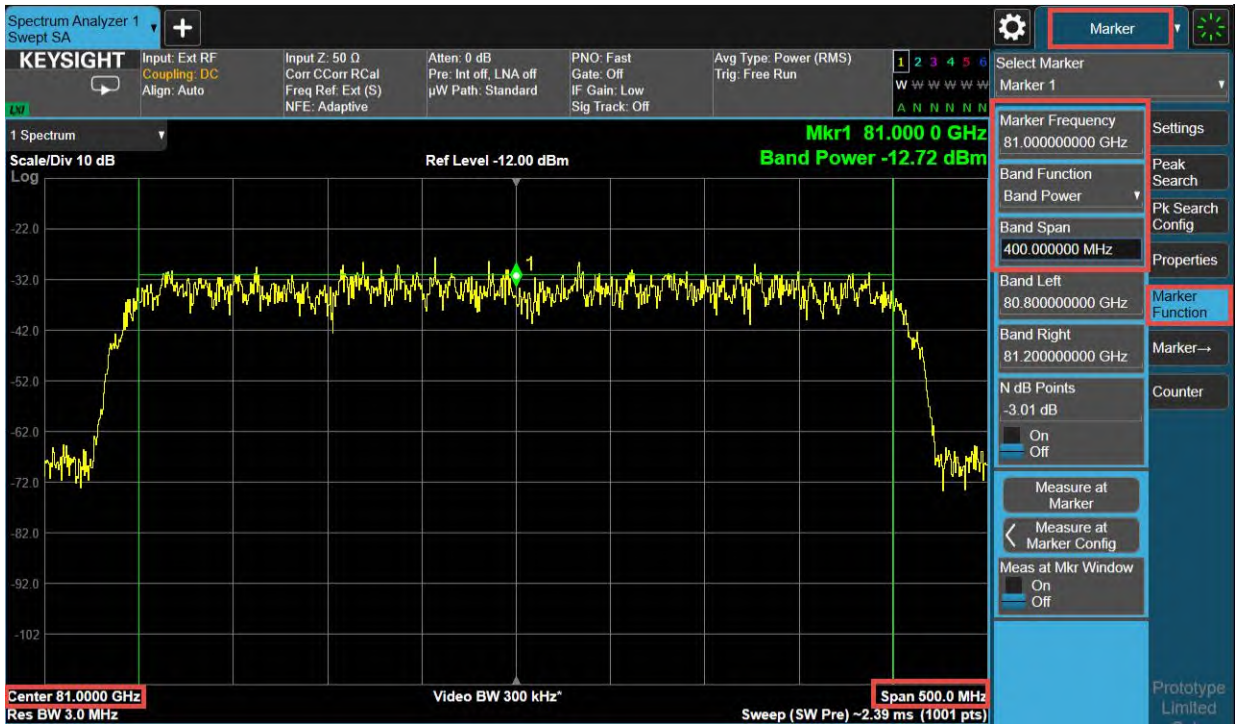


10. Set RF Out to **On** by selecting the numbered channel indicator switch.

This enables the RF Out for the indicated channels, in this case Channel 1 if using a multi-channel VXG.

Setting Up the UXA Using X-Apps

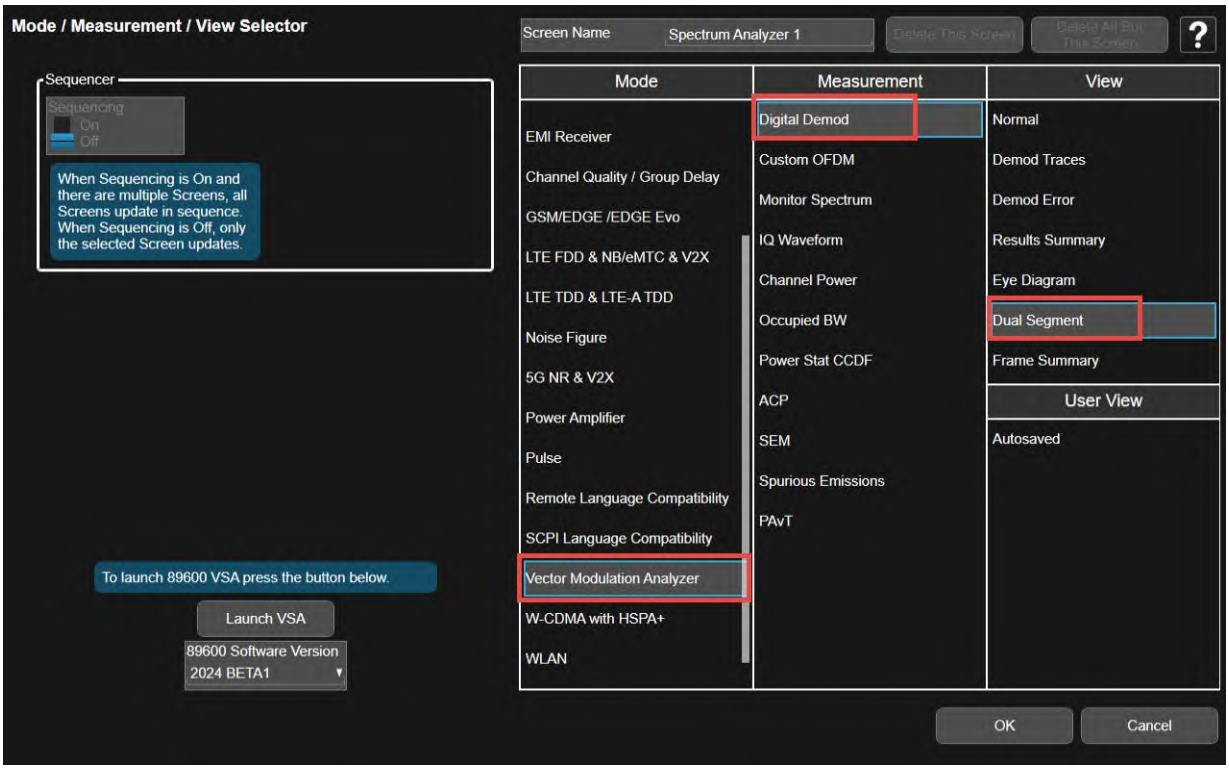
1. Using Spectrum Analyzer Swept SA mode, set Frequency to **81 GHz** and Span to **500 MHz**.
2. Select **Marker** > **Marker Function** tab > Band Function and select to **Band Power**.
3. Set Marker Frequency to **81 GHz.**
4. Set Band Span to **400 MHz**.



5. Select **Mode/Meas** > **Vector Modulation Analyzer** Mode > **Digital Demod** Measurement > **Dual Segment** View.

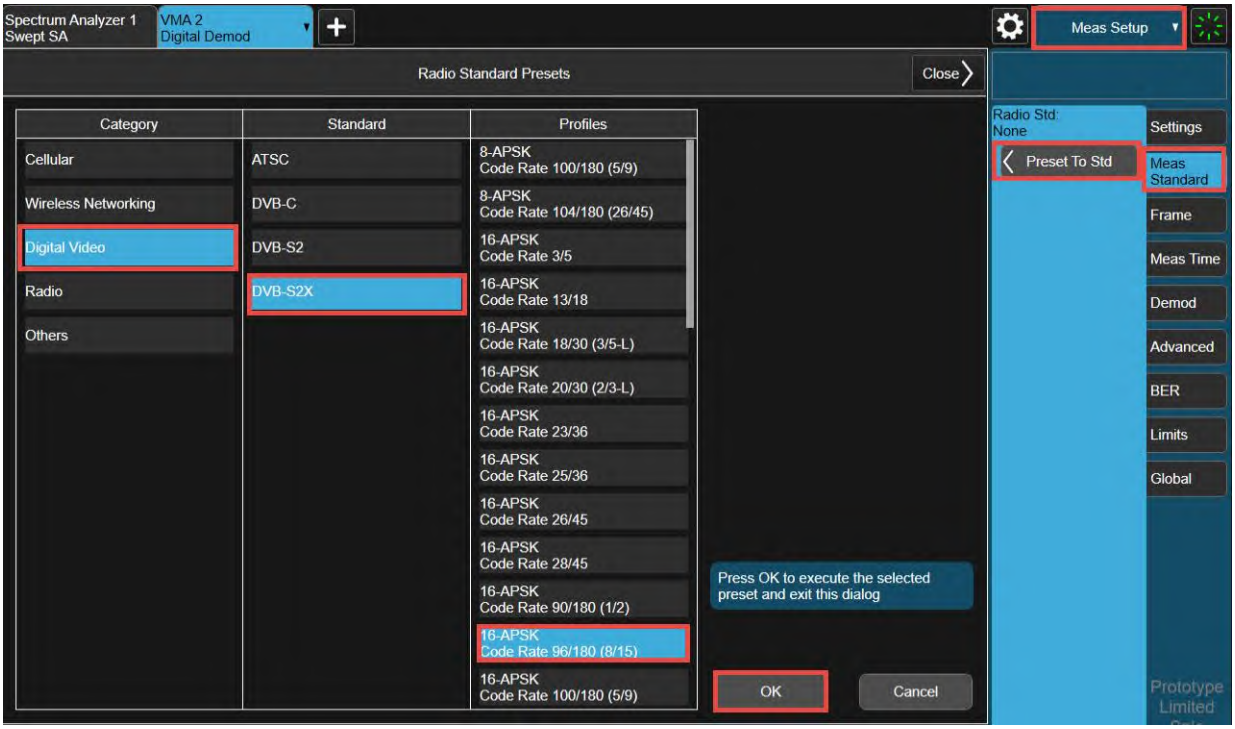
NOTE

If accessing the instrument via a Remote Desktop connection, select the Screen tab (at the top of the display) to open the Mode/Measurement/View Selector window.

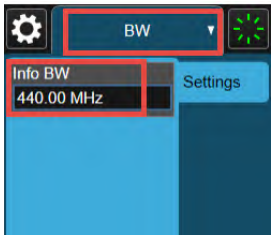


6. Select **Frequency** and set to **81 GHz**.

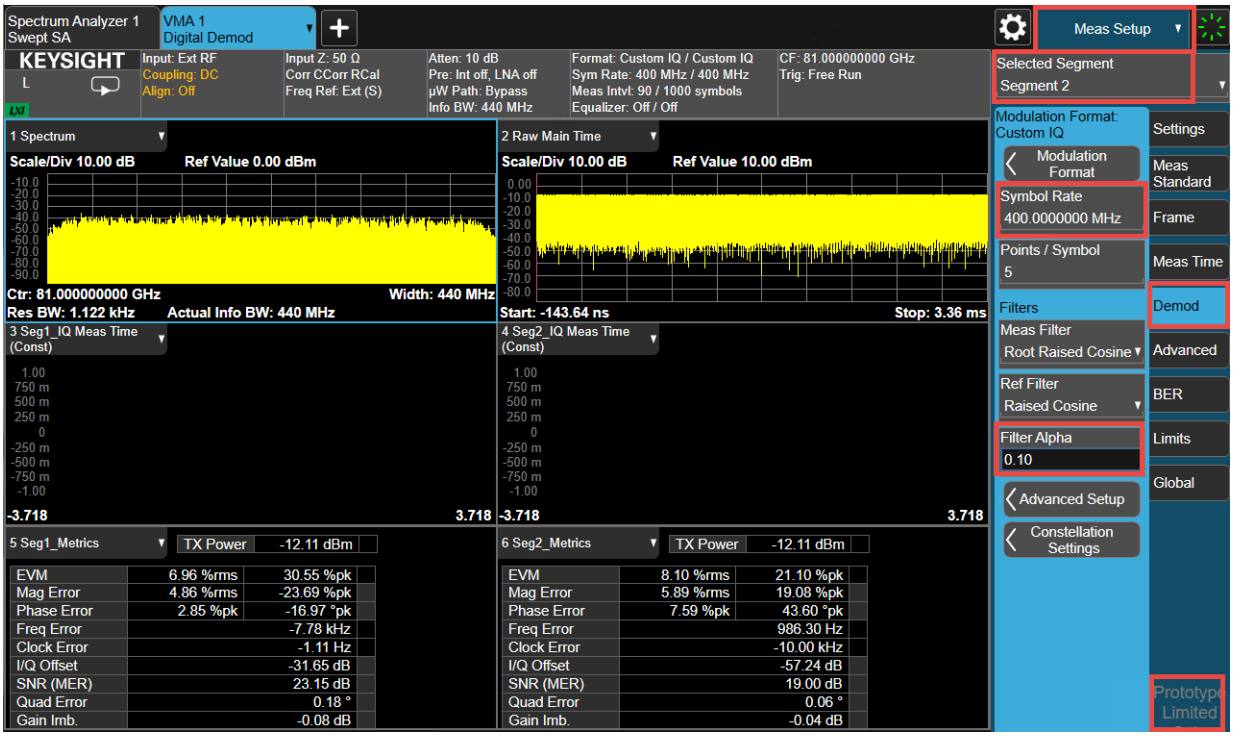
7. Select **Meas Setup** >**Meas Standard** tab > **Presel to Std**, and then select **Digital Video** > **DVB-S2X** > **64-APSK Code Rate 132/180 (11/15)**, then **OK** to close the window.



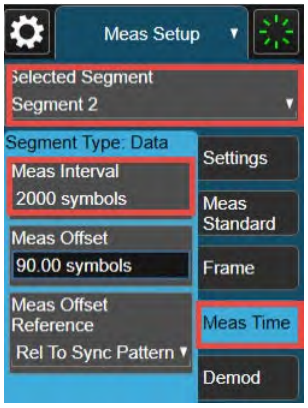
8. Select **BW** and set to Info BW to **440 MHz**.



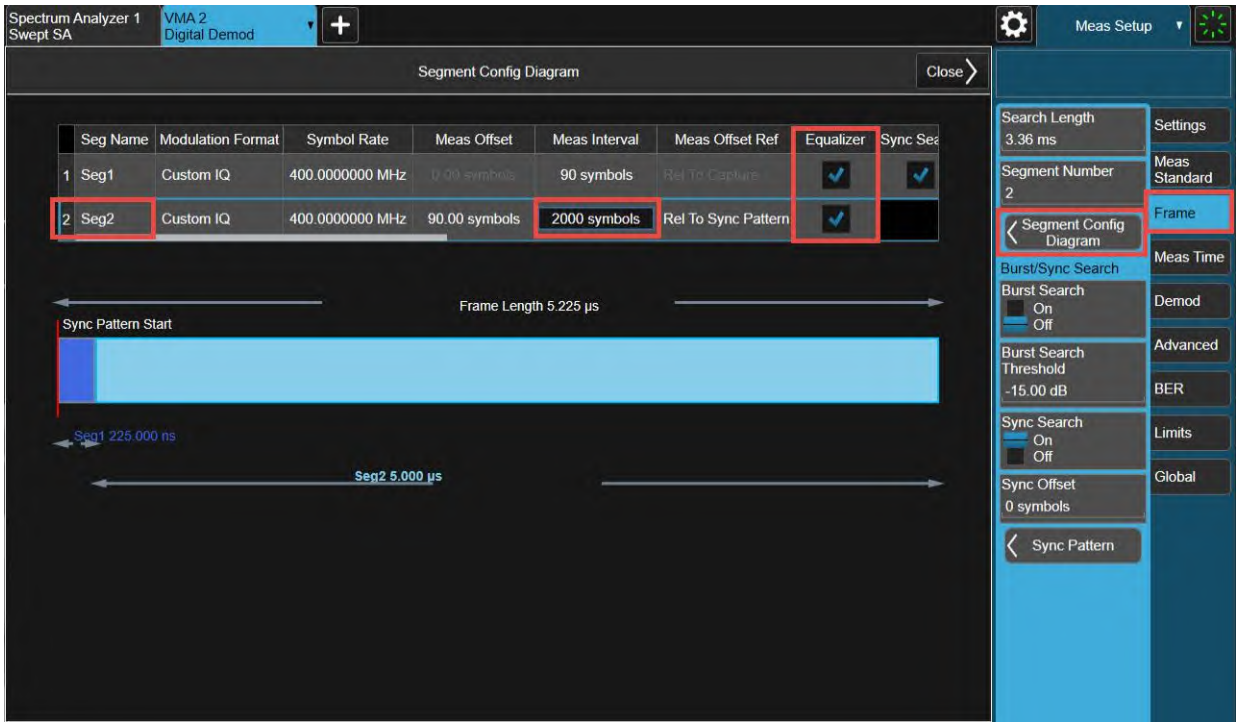
9. Select **Meas Setup** > **Demod** tab and set the Symbol Rate to **400 MHz** and Filter Alpha to **0.1** for both **Segment 1** and **Segment 2** to match the settings in the Custom Modulation Setup.



10. Select the **Meas Time** tab and note that Meas Interval for Segment 1 is set to 90 symbols, the same as the Header setting in the Custom Modulation setup. Select **Segment 2** and set Meas Interval to **2000** symbols to match the Data Segment in the Custom Modulation setup.

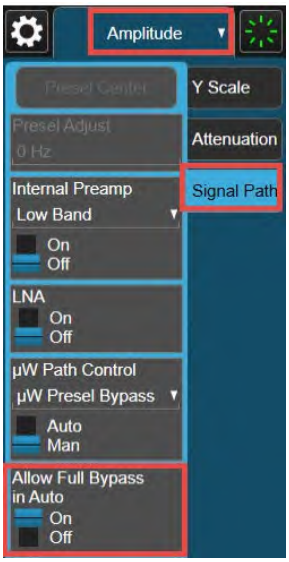


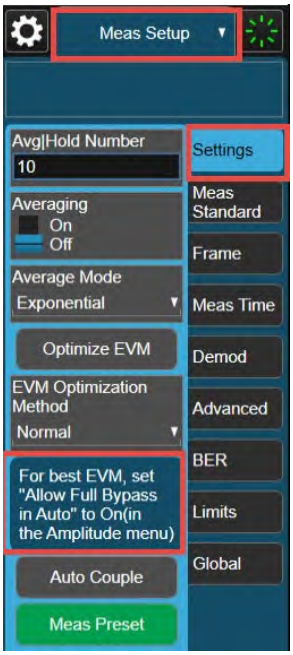
11. Select the **Frame** tab > **Segment Config Diagram** and set **Equalizer** to turn on for both **Segment 1** and **Segment 2**, then **Close** the window.



12. Select **Amplitude** > **Signal Path** tab > select Allow Full Bypass in Auto to turn **On**.

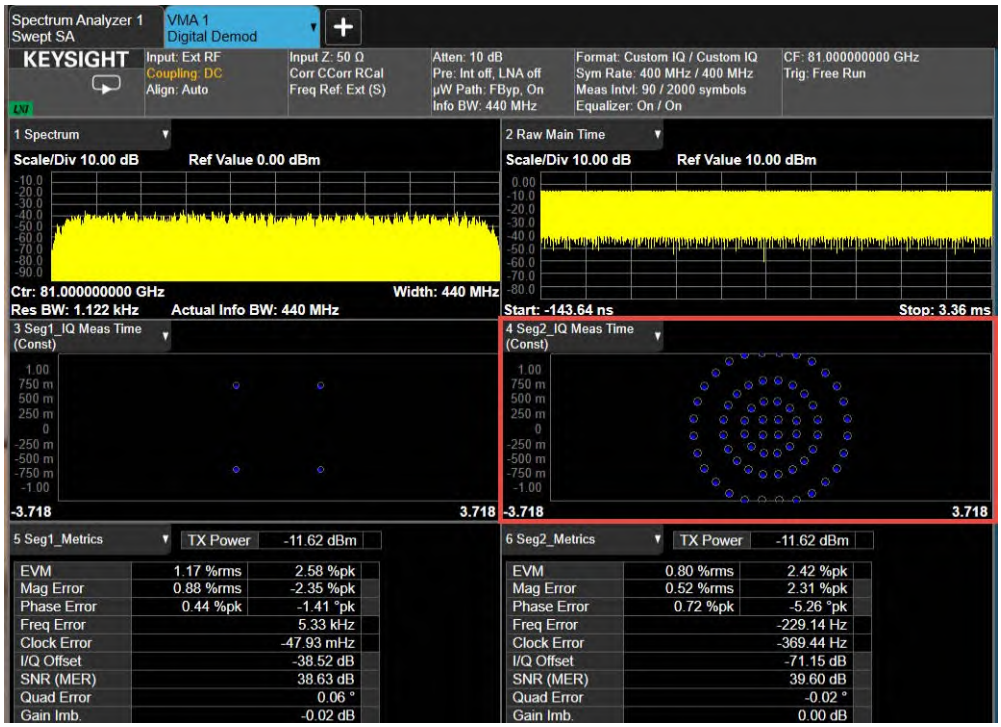
    Notice that is recommended in the Meas Setup > Settings tab to turn this on for best EVM results.

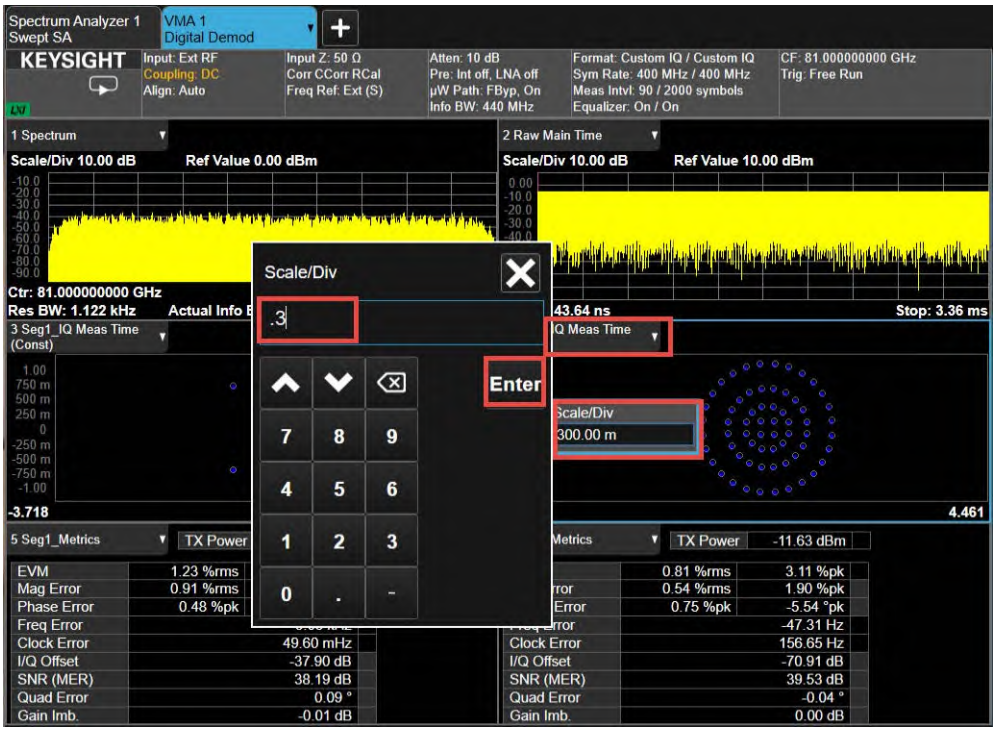




13. At the top of the display, select **Equalizer On/Off**. From the dropdown, select **Equalizer Reset**.

14. In the **Seg2_IQ Meas Time** display, notice that the constellation diagram goes off the display. To adjust the scaling, select the left side (click on thr scale number) of the window and set the **Scale/Div** to **.3** (300.00 m per division) > **Enter**.



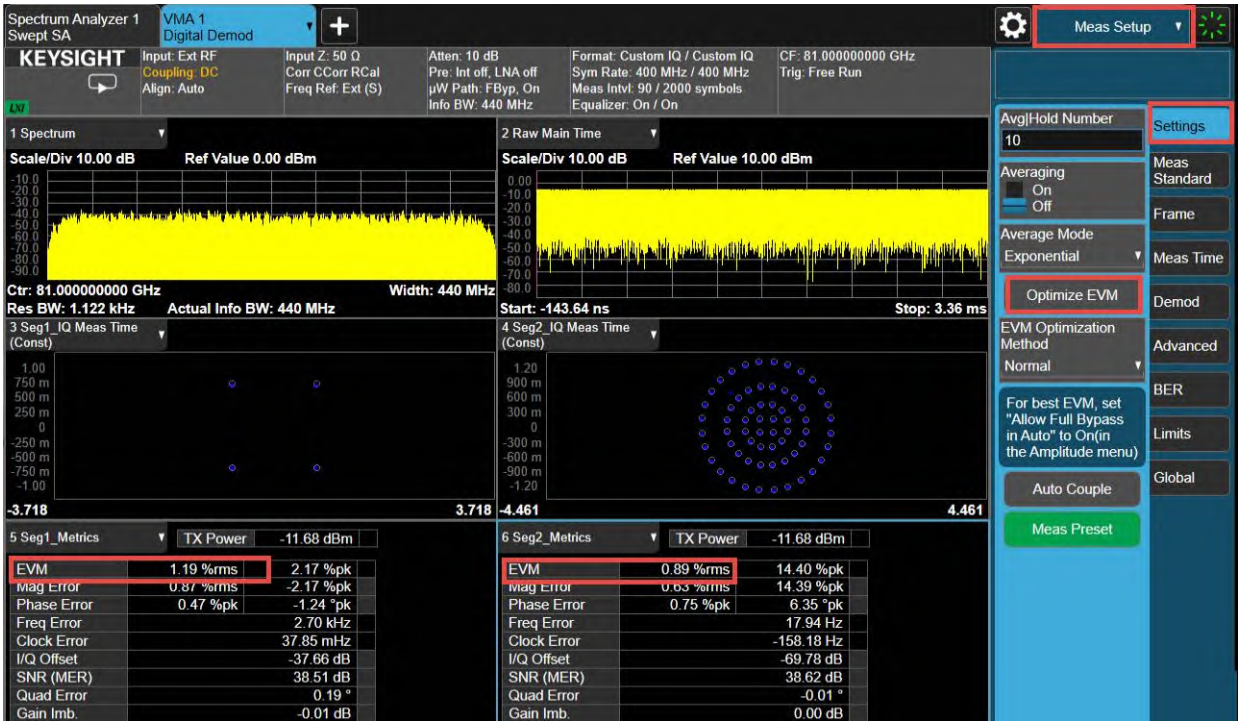


NOTE

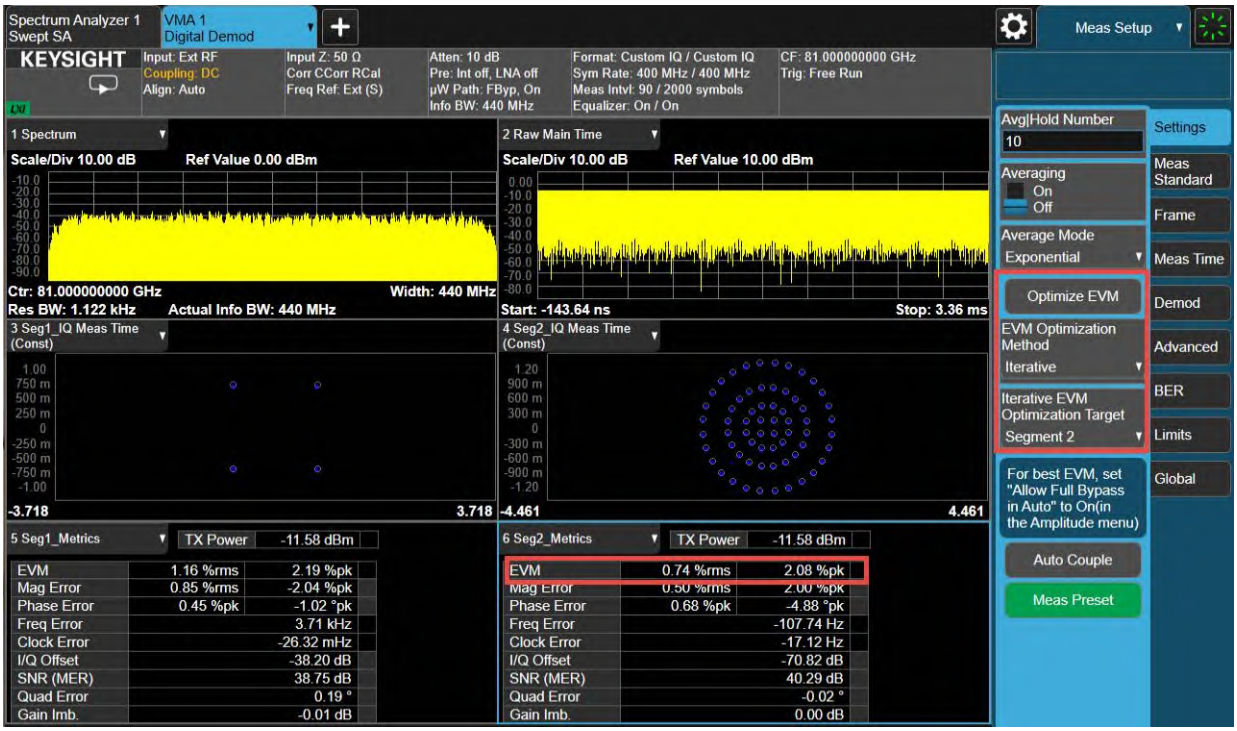
You must set this value to 300 instead of .3, the scaling gets changed to 250 k per division instead of 250 m per division.

15. Select **Meas Setup** > **Settings** tab > **Optimize EVM**.

The default setting uses the Normal method for Optimize EVM. This is a quick way to improve your measurement results. The EVM will continue to reduce as the UXA continues to sweep. This is because the Equalizer is correcting for linear distortions in the measurement setup.



You can also set EVM Optimization Method Iterative mode, then select either segment for optimization. This method takes a few minutes to complete. For the example below, we set the Optimization Target to Segment 2. Notice how the EVM went from .89% to .74%.



TIP You can also manually adjust to optimize EVM.

- Select Amplitude menu > Attenuation > Frequency Extender Atten. Use the step keys to increase and decrease the attenuation by 2 dB per step. After each increment, wait for the update to determine where the lowest EVM point is achieved.
- Select Meas Setup > Advanced > IF Gain Other. Use the step keys to increase and decrease the attenuation by 1 dB per step. After each increment, wait for the display to updated to determine where the lowest EVM point is achieved.

In this example, we found the optimum settings were 10 dB of Mechanical Attenuation and -8 dB of IF Gain. We will use these settings in the 89600 VSA setup.

Setting Up the UXA Using 89600 VSA

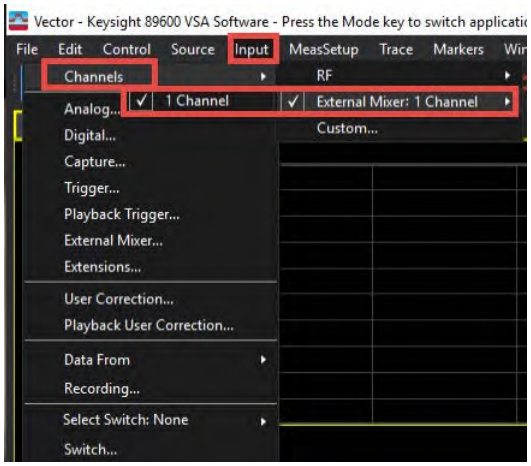
NOTE Changing settings on the VSA can take longer to take affect than expected. Pausing the measurement until all settings have been made will help speed up the time. Wait to change the next setting until the previous setting has been updated.

1. Open the VSA software.

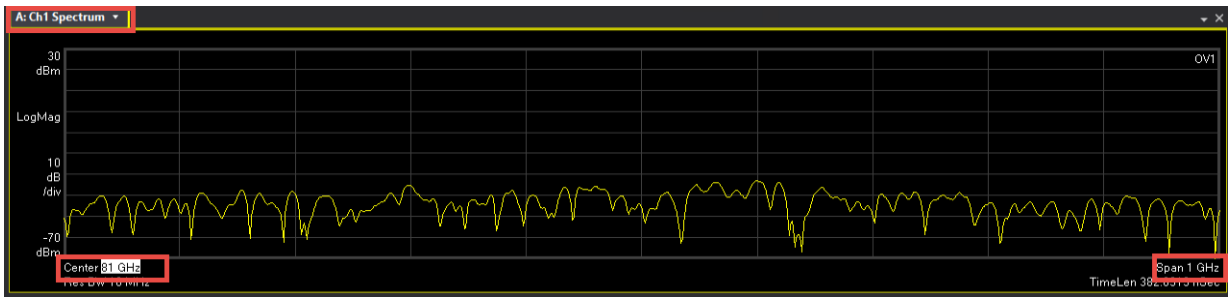
   To access the VSA software, go to the Windows Start menu and find Keysight 89600 Software (latest installed version) folder and run the software.

NOTE If accessing the instrument via a Remote Desktop connection, select the Screen tab (at the top of the display) to open the Mode/Measurement/View Selector window.

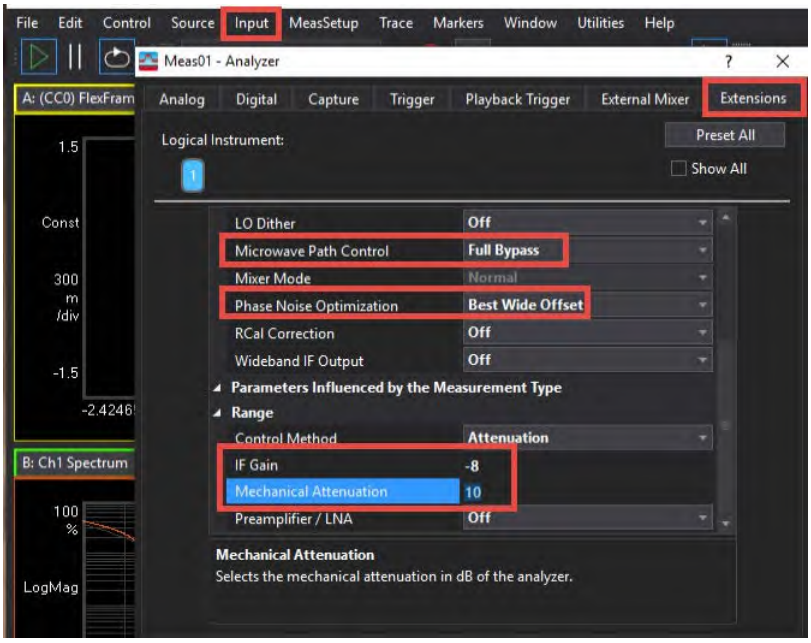
2. From the VSA menu bar, select **File** > **Preset** > **All** to set the VSA to a known state.
3. From the menu bar, select **Input** > **Channels** > **External Mixer** > **1 Channel**.



4. In the **Spectrum** window, select **Center** and set the Center Frequency to **81 GHz**. and Span to **1 GHz**.

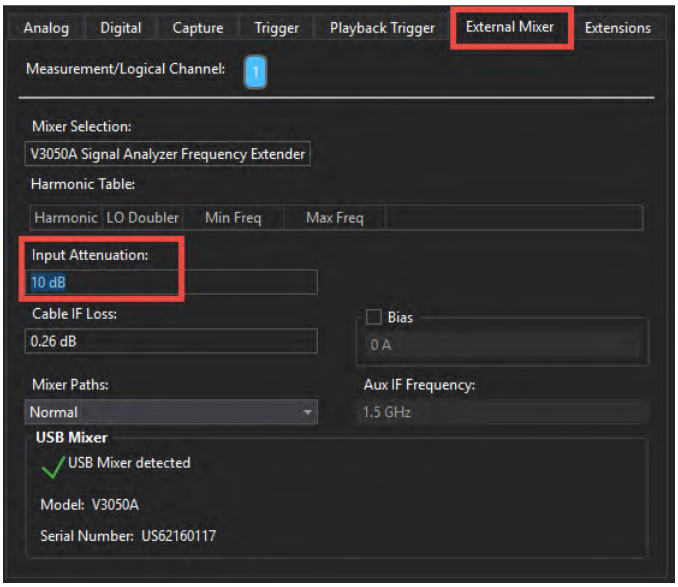


5. From the menu bar, select **Input** > **Extensions** and set:
   - Microwave Path Control to **Full Bypass**
   - Phase Noise Optimization to **Best Wide Offset**

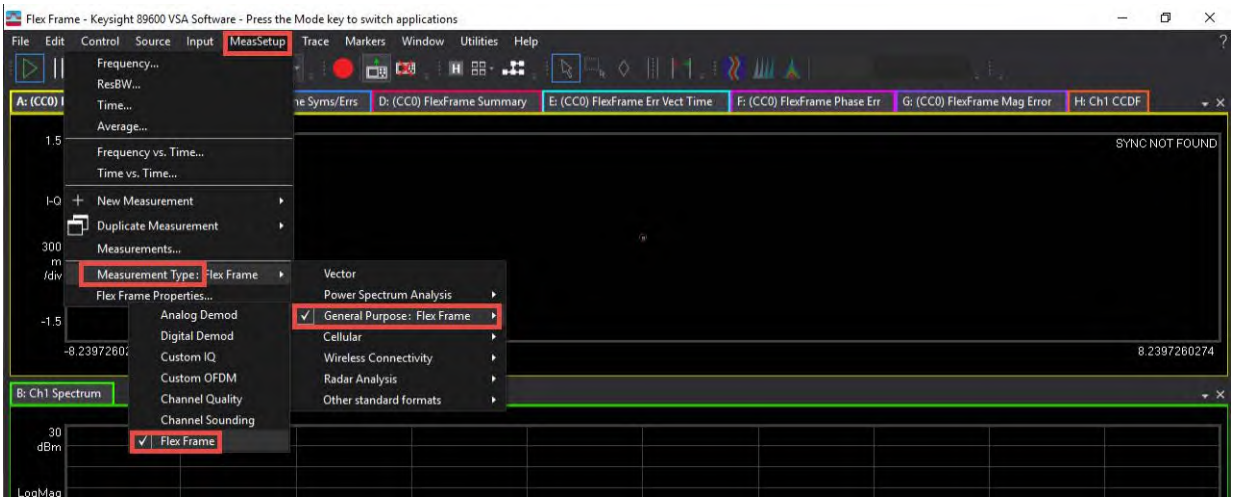


   - IF Gain to **-8 dB**
   - Select the External Mixer tab and set Input Attenuation to **10 dB**

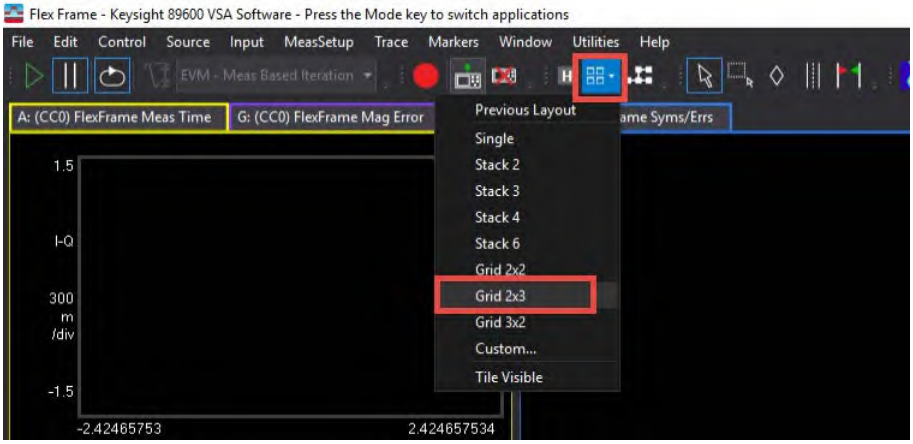
     These are the optimized settings we used in the X-Apps measurement.



6. From the menu bar, select **MeasSetup** > **General Purpose** > **Flex Frame**.

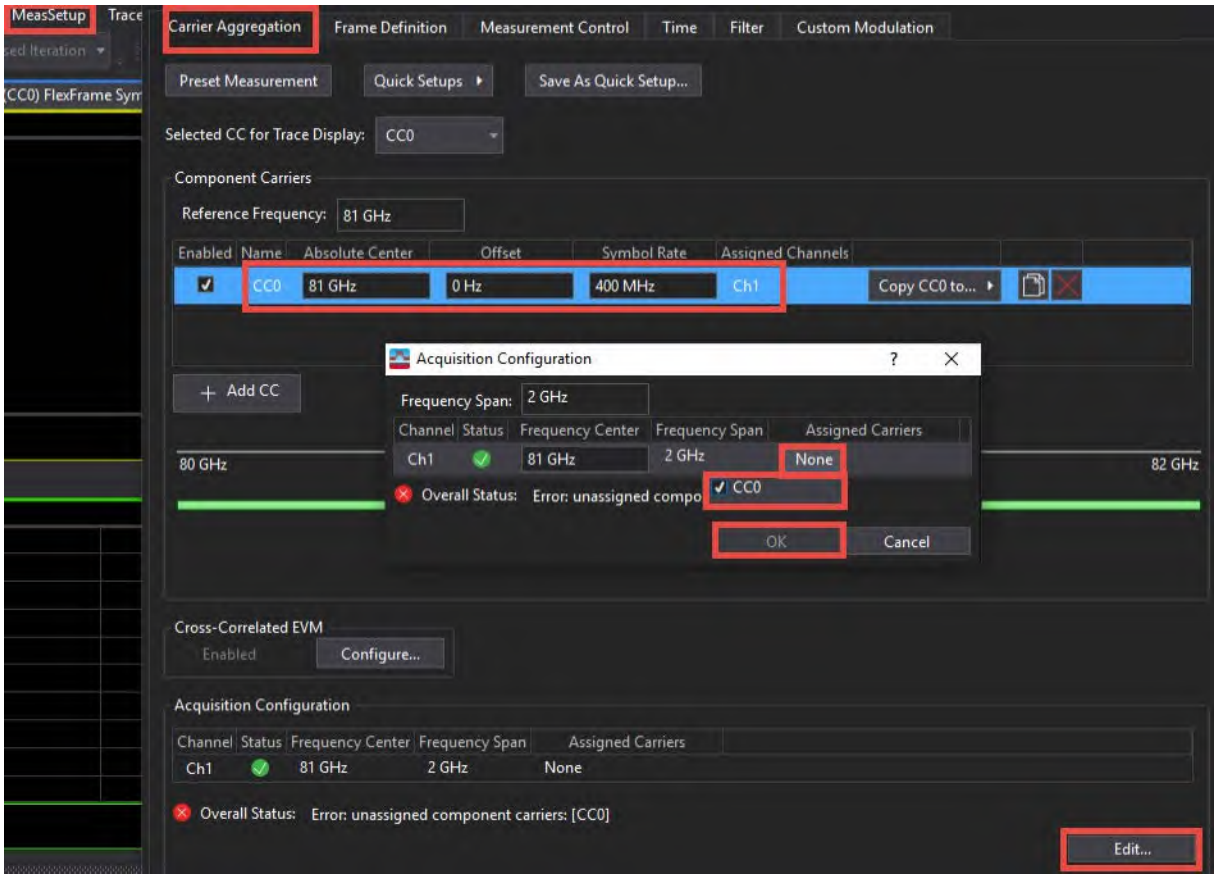


7. Change the Trace Layout Grid to **2 x 3**.

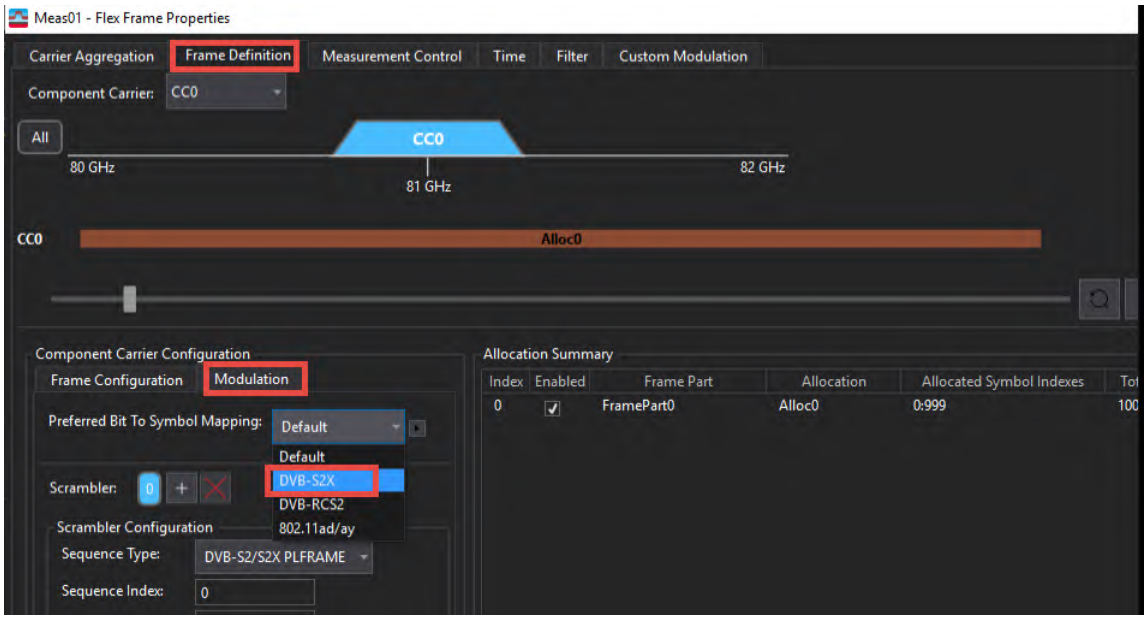


8. Select **MeasSetup** > **Flex Frame Properties** > **Carrier Aggregation** tab, and set:
   - CCO Absolute Center frequency to **81 GHz**
   - Symbol Rate to **400 MHz**.

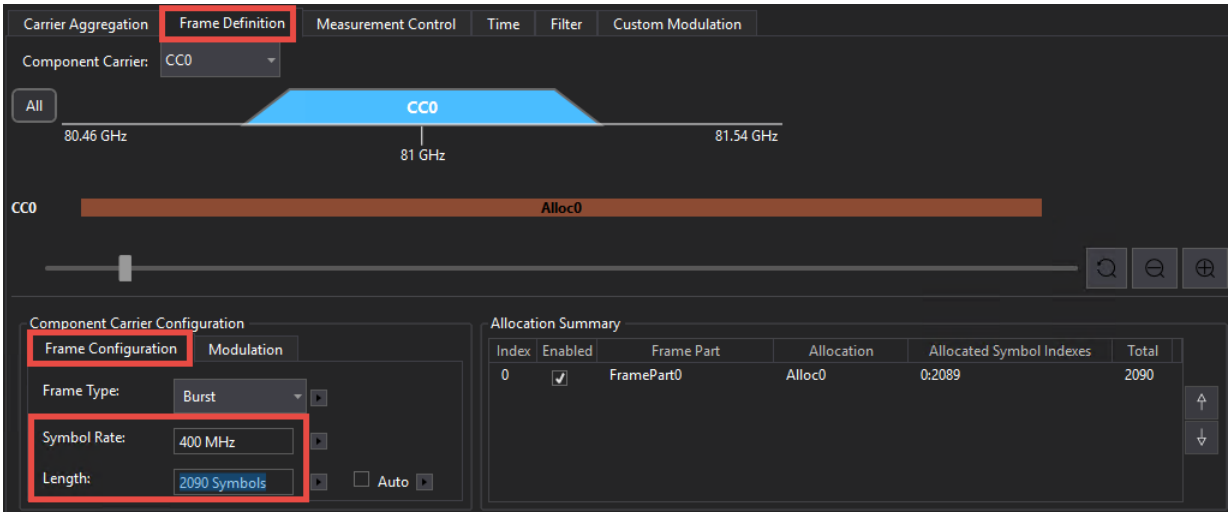
– **Edit** dropdown and set Assigned Carriers to **CC0**.



9. In the Flex Frame Properties window, select the **Frame Definition** tab. Under Component Carrier Configuration, select the **Modulation** tab, and then set Preferred Bit to Symbol Mapping to **DVB-S2X**.

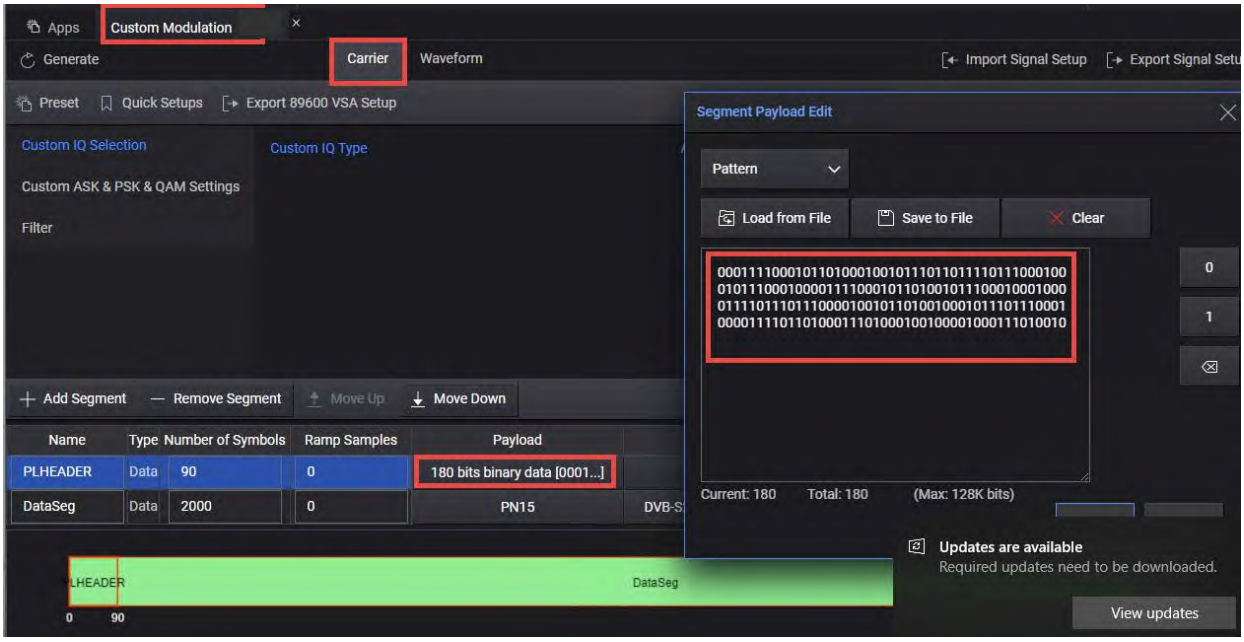


10. In the Frame Definition tab, under Component Carrier Configuration, select the **Frame Configuration** tab and set Symbol Rate to **400 MHz** and Length to **2090 Symbols**.



11. To configure the Pilot:
    - In the Frame Parts area, select **Alloc0**
    - In the Configuration area, set
        - Name to **PLHEADER**
        - Type to **Pilot**
        - Sequence to **Custom**
        - Modulation to **QPSK**
        - Select **Include BER Calculations**

– Go back into the VXG Custom Modulation setup **Carrier** tab and select PLHEADER **Payload** to open and copy the symbols.



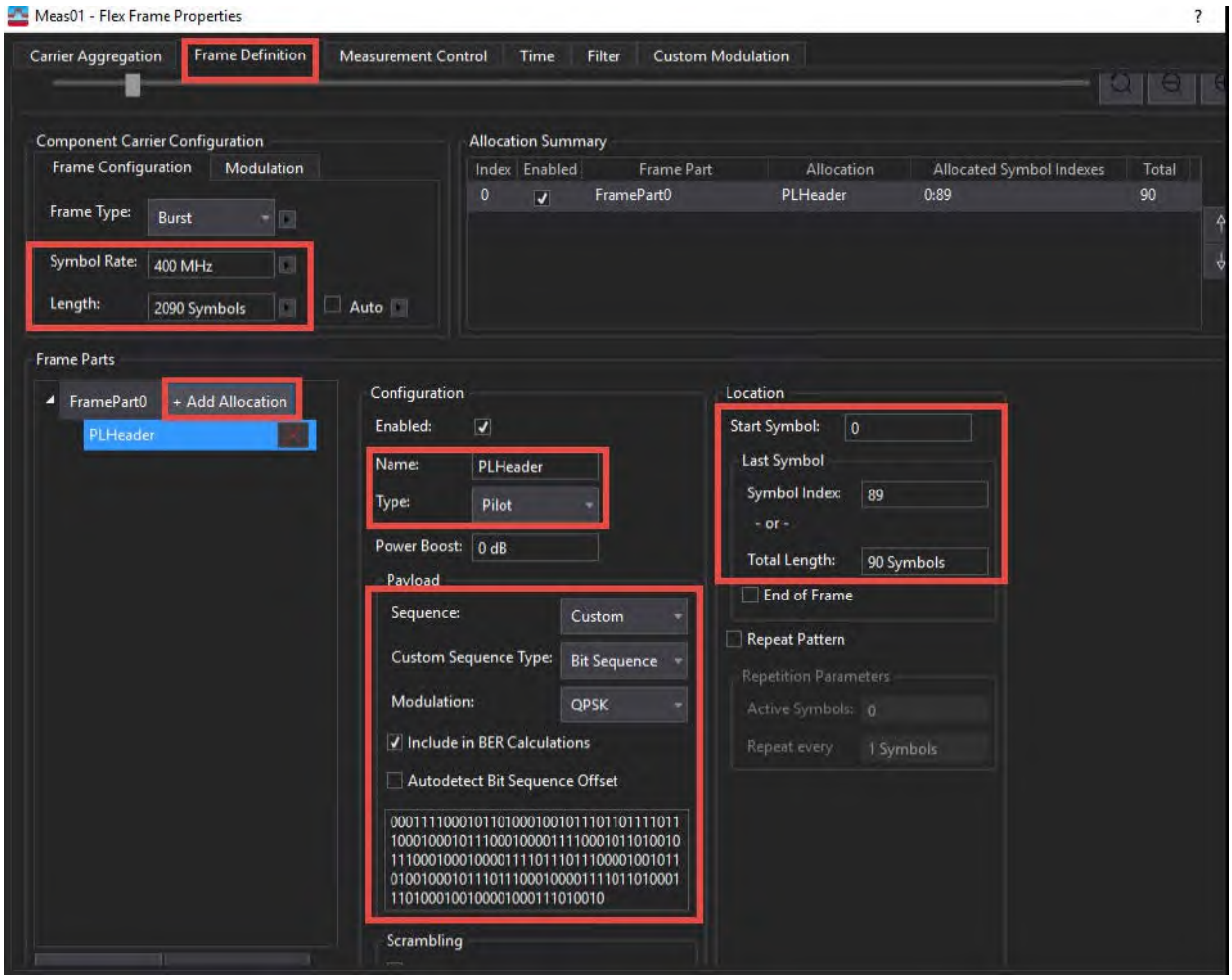
– Paste the symbols you copied over from the Custom Modulation PLHEADER Payload setup.

– In the Location area,

– Clear the End of Frame checkbox

– Start Symbol to **0**

– Total Length to **90**



12. In the Frame Parts area, select **Add Allocation** and set:

– Name to **Data**

– Type to **Data**

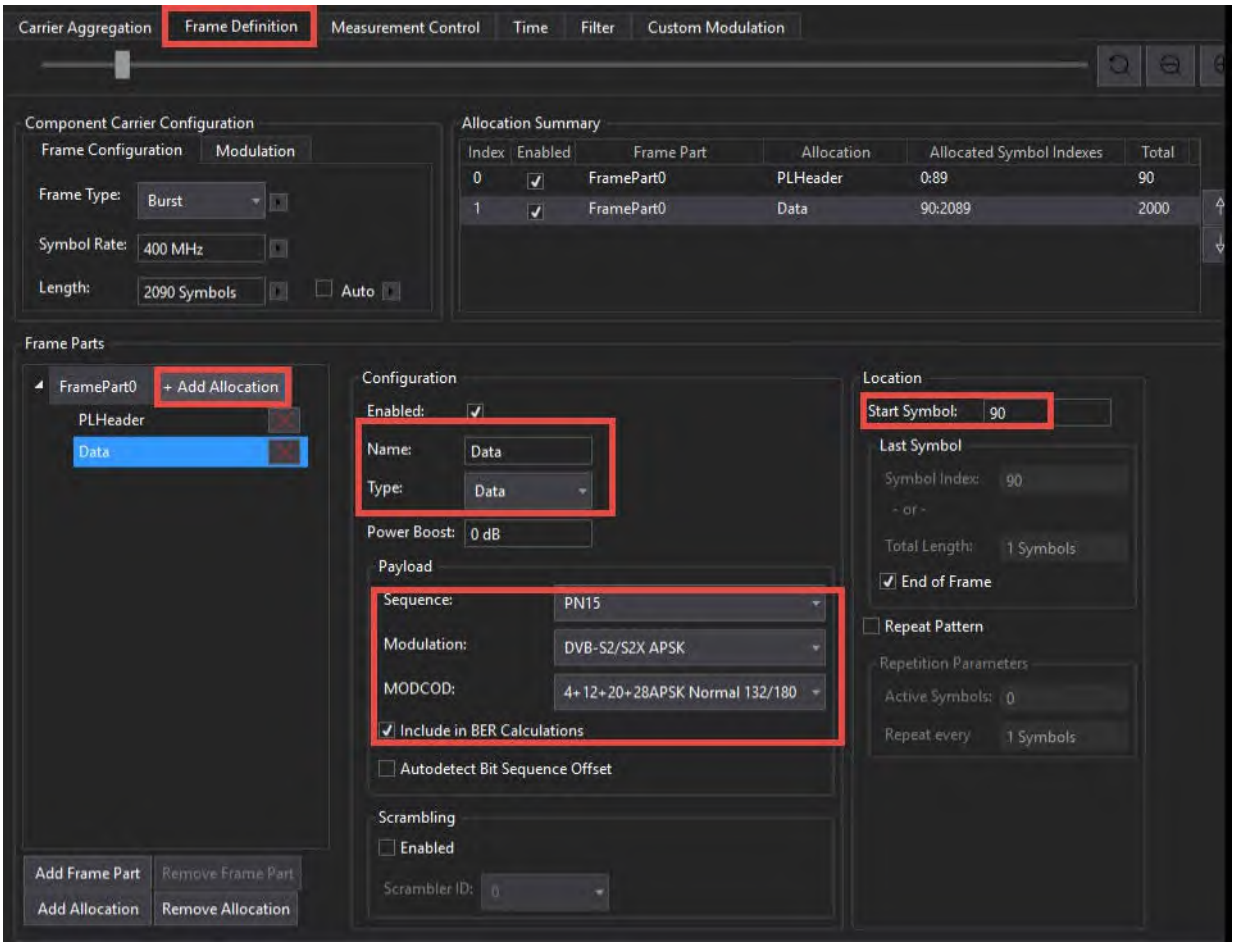
– Sequence to **PN15**

– Modulation to **DVB S2/S2X APSK**

– MODCOD to **4+12+20+ 28APSK Normal 132/180**

– Select **Include in BER Calculations**

– In the Location area, set Start Symbol to **90**.



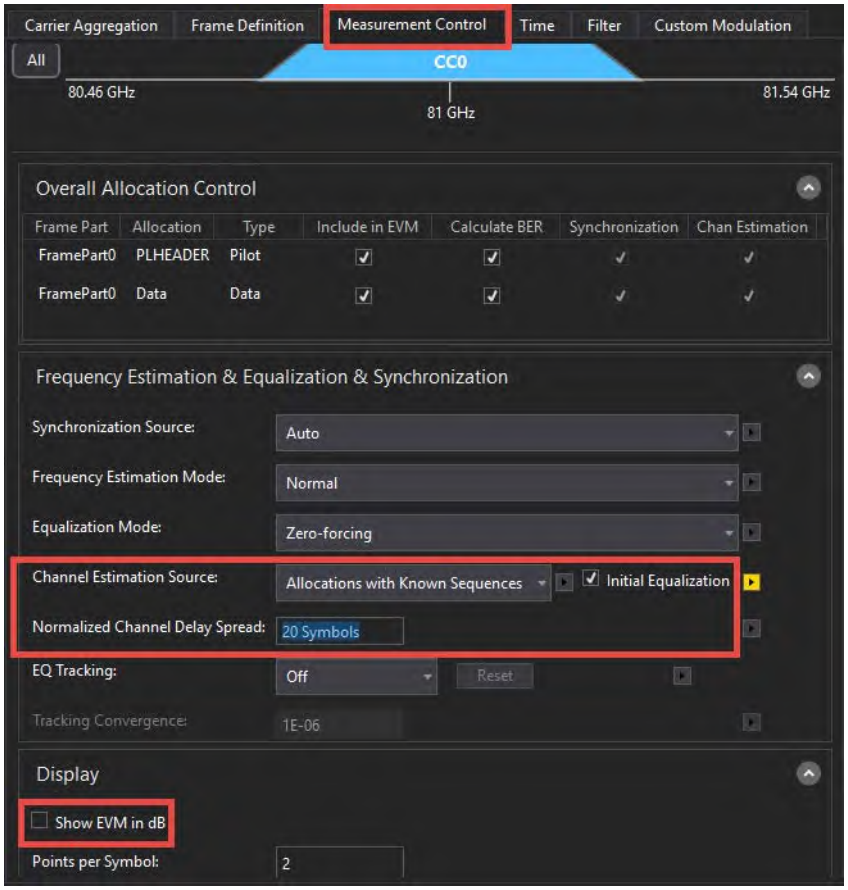
13. In the **Measurement Control** tab, and set:

– Channel Estimation Source to **Allocations with Known Sequences**

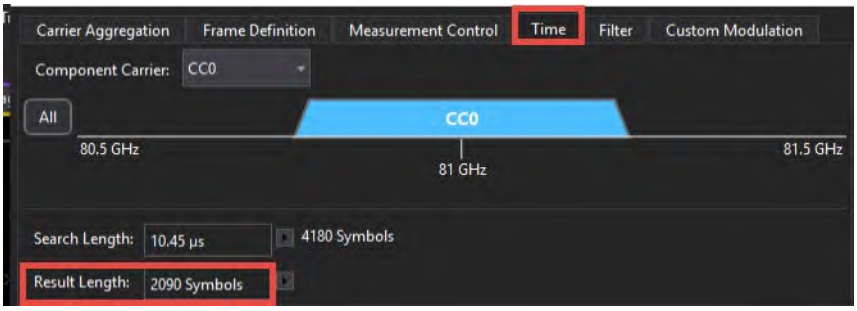
– Select **Initial Equalization**

– Normalized Channel Delay Spread to **20 Symbols**

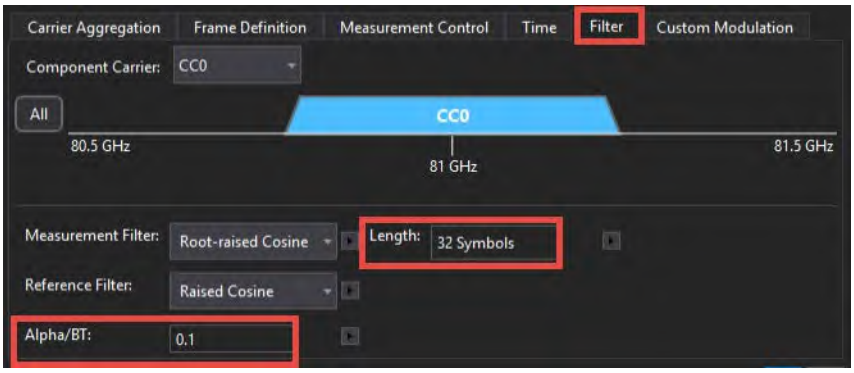
– Clear the **Show EVM in dB** checkbox



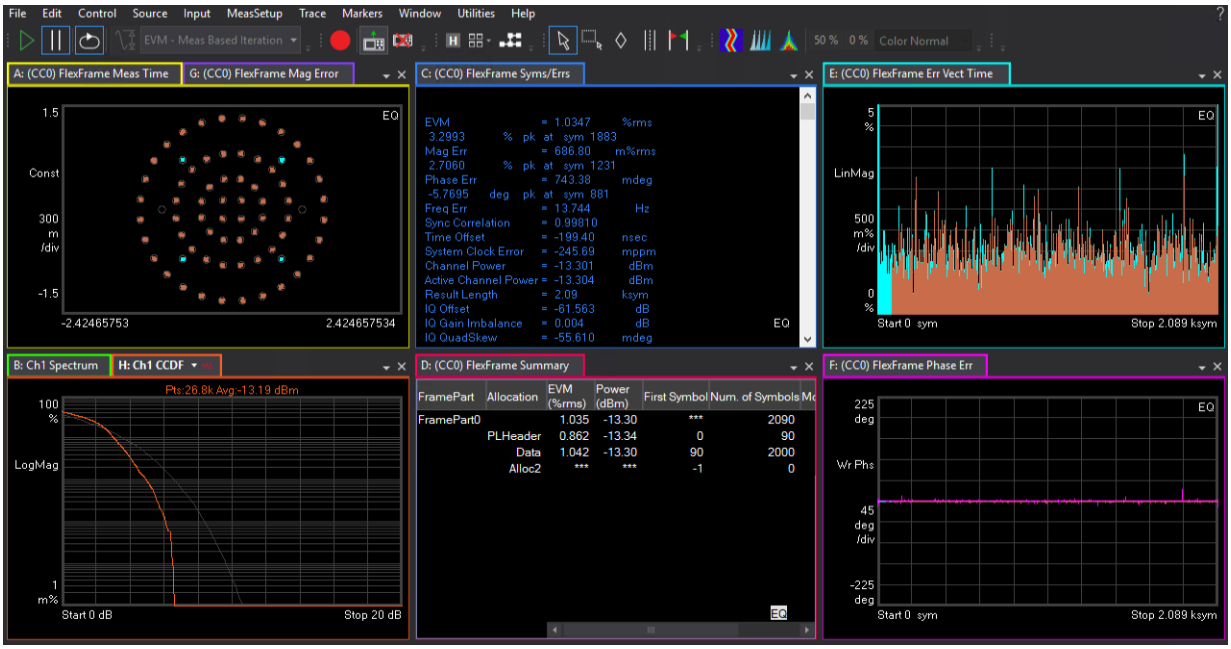
14. In the **Time** tab, set Result Length to **2090 Symbols**.



15. In the **Filter** tab, set Alpha BT to **0.1** and Length to **32 Symbols** and then close the Flex Frame Properties window.



16. View the results of the measurement. Try to improve EVM by adjusting the IF Gain and Mechanical Attenuation settings in the **Input** >**Extensions** and the Input > External Mixer windows.



Setting-up a Millimeter-Wave 5G NR, 2 GHz BW FR2 Measurement

This measurement example will show you how to configure and analyze a 2000 MHz 5G NR, FR2, signal. 3GPP Rel-17 introduced a new frequency band 52.6 - 71 GHz, with new numerology (480 kHz, 960 kHz) and new bandwidth (800 MHz, 1600 MHz, 2000 MHz). Support for these new numerologies/bandwidths was added in VSA2023 Update 2.0 release and will be part of PWSG 2023 Update 1.0.

Software Requirements

PathWave Signal Generation for 5G NR	N7631APPC	2023U1 or later
89600 VSA software	89601BHNC	VSA2023U2 or later

Using the graphical user interface

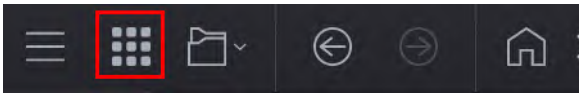
On the VXG:

1. Select **Preset** > **Preset** to set the VXG to a known state.
2. In the Output area, set Frequency to **70 GHz** and Power to **-10 dBm**.

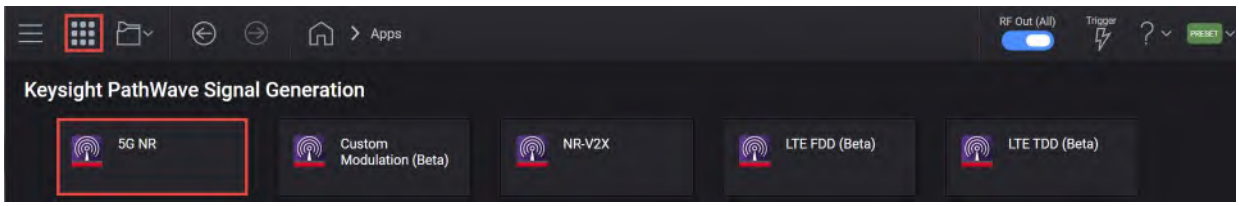
NOTE

Waveform generation and analysis speed will be very slow since we are generating and analyzing a 2 GHz wide NR signal. (That is, 5 times more data compared to a 400 MHz 5G NR signal.) To improve some demodulation speed, use VSA2024, which has some speed improvement and also use an external trigger for demodulation. (N9042B rear panel Trigger 1 In to M9484C rear panel 10 MHz Ref Out.) Furthermore, to improve VSA setup speed Stop the measurement until all parameters are correctly configured, then run the measurement (Control menu > Stop).

3. Select the **Radio Apps** block to open the mode selection panel.

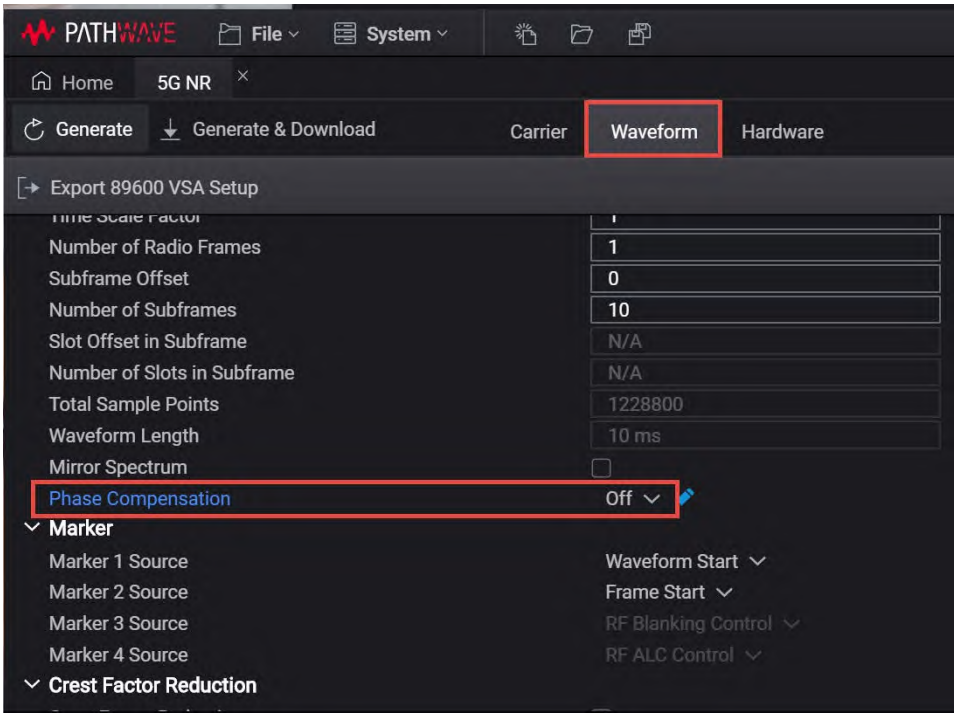


4. Select **5G NR**.



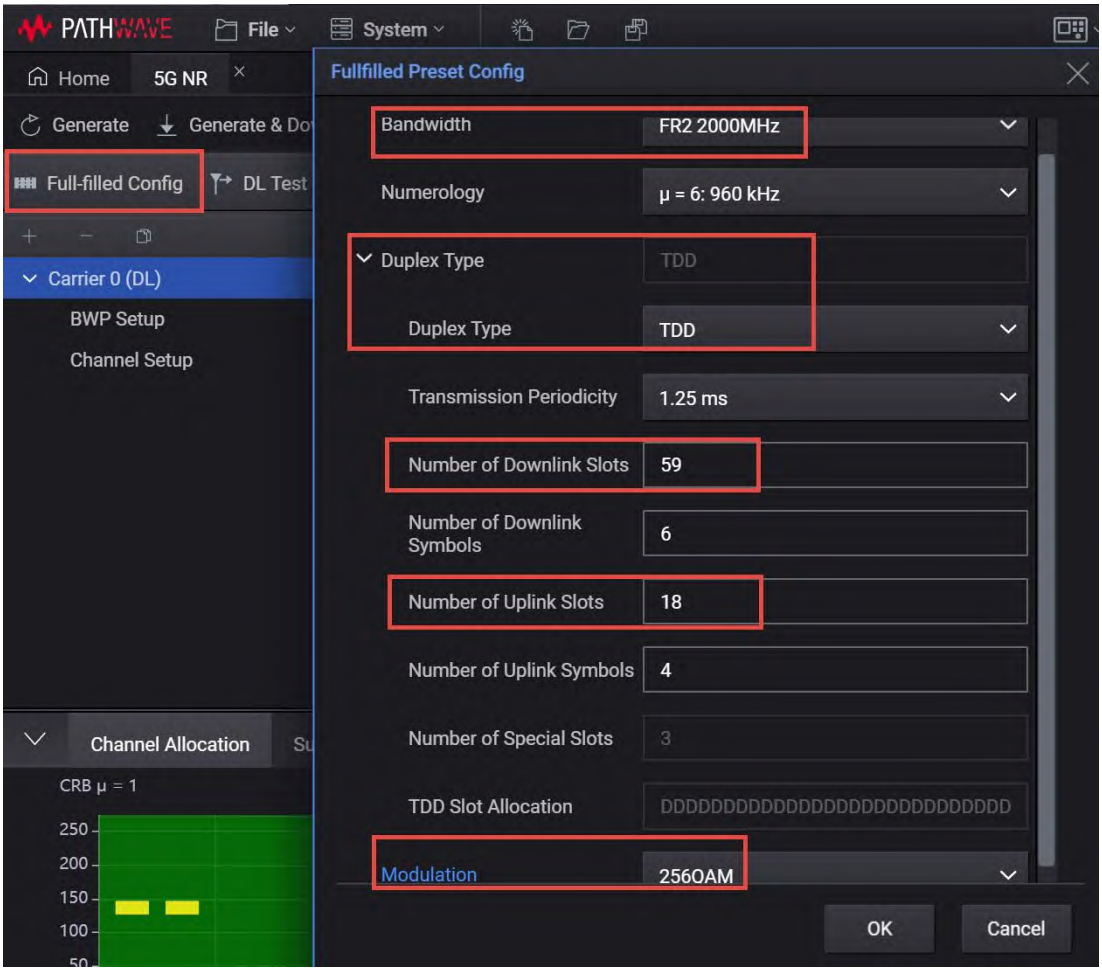
5. Select the **Waveform** tab and turn off **Phase Compensation** so the same waveform can be used at different frequencies.

Phase compensation is applied by default at baseband for RF up-conversion. This means, it depends on carrier frequency, so a waveform needs to be generated per carrier frequency unlike conventional waveforms which are independent from carrier frequency. Phase Compensation default is set to AUTO.



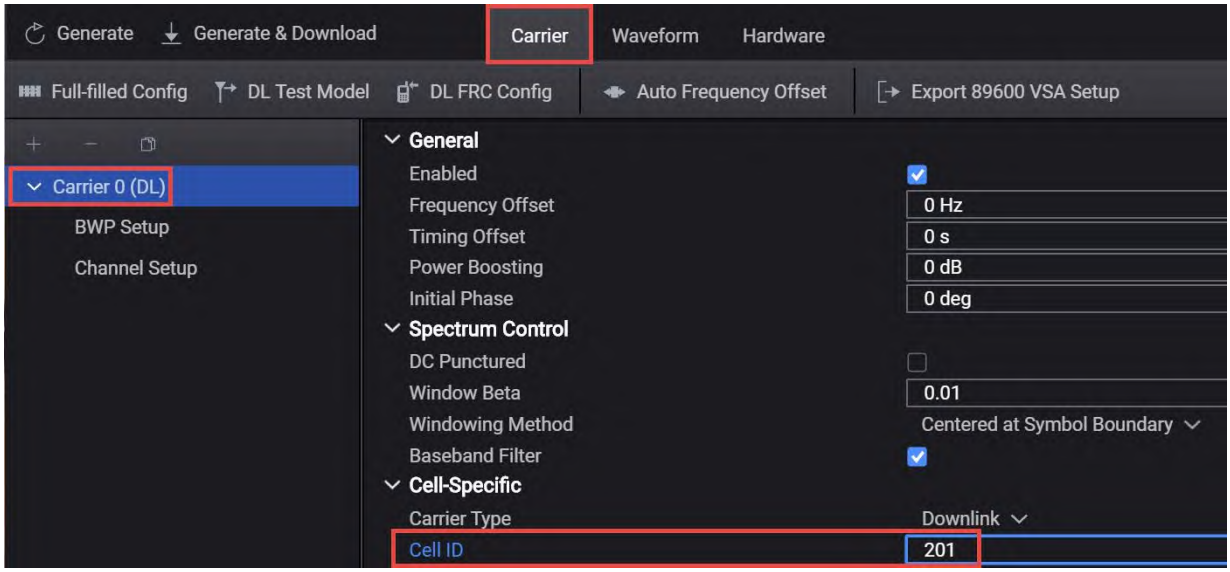
6. To configure the Downlink carrier, select the **Carrier** tab > **Full-filled Config** and set:
   - Bandwidth to **FR2 2000 MHz**
   - Duplex Type to **TDD**
   - Number of Downlink slots to **59**
   - Number of Uplink Slots to **18**
   - Modulation to **256 QAM**

– Select **OK** to close the window.

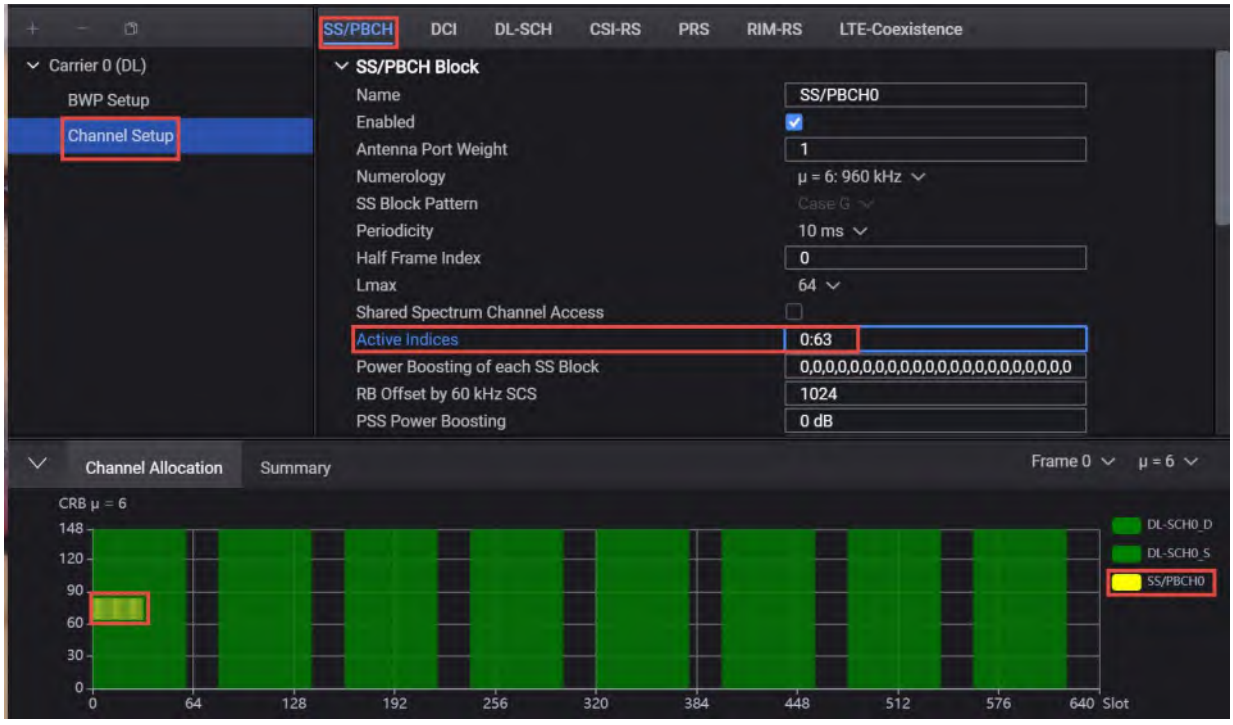


These parameters are based on 3GPP endorsed Test Model for 2000 MHz bandwidth.

7. Under the Carrier (DL) side tab, scroll down and set Cell ID to **201**.



8. Select **Channel Setup** > **SS/PBCH** and set Active Indices to **0:63**.

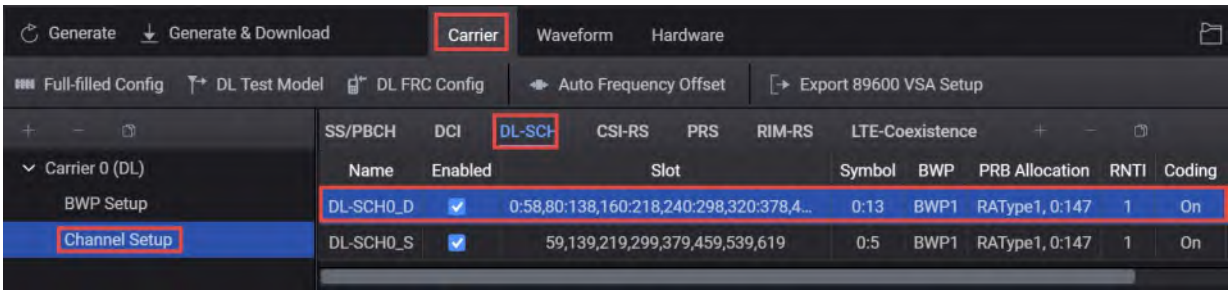


With RB Offset by 60 kHz SCS set to 1024 and kSSB by subcarrierSpacingCommon set to 0, the SSB is set to the center of the carrier. This is shown in the Channel Allocation graph at the bottom of the window. If you set it to 0, the SSB will be set in the lower band of the carrier.

Unlike LTE, there is no fixed frequency location for SS/PBCH in NR. By default, PWSG transmits SS/PBCH (20 RB or 240 subcarriers wide) centered at the carrier bandwidth (2000 MHz channel BW in this example). This is a very important parameter since it shows were the SS/PBCH is located within the channel bandwidth in the frequency domain and it is relative to common resource block 0 (CRB0).

9. Select **DL-SCH** > **DL-SCH0_D**.

   You should see DL-SCH0_D and DL-SCH0-S. These are configured from the Full-filled Config settings above. We will make a couple edits to the Reference Signal configurations.



   – Confirm Allocated Slots is set to **0:58,80:138,160:218,240:298,320:378,400:458,480:538,560:618**

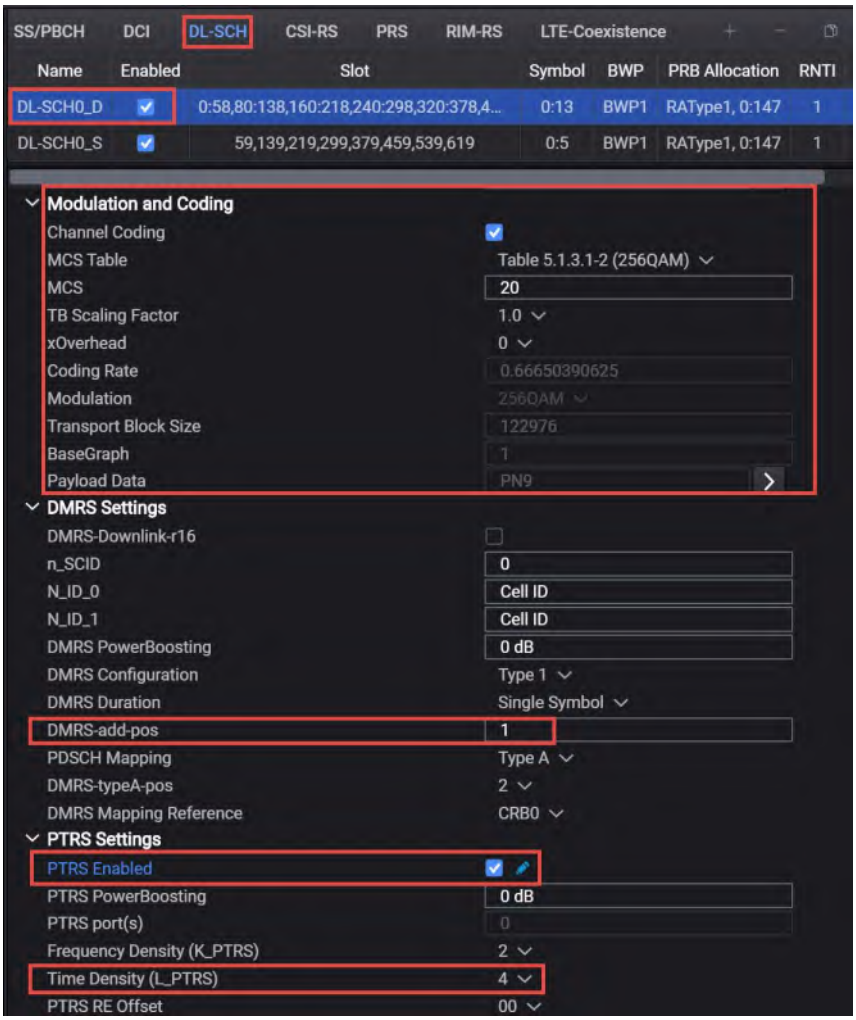
     These values will be used in the VSA/X-App. You can copy these values for easier setup later.

   – Confirm that MCS Table is set to **256QAM** and MCS to **20**. MCS table, MCS value, scaling factor, etc. must match with the VSA (or X-App) for successful decoding (CRC pass/fail).

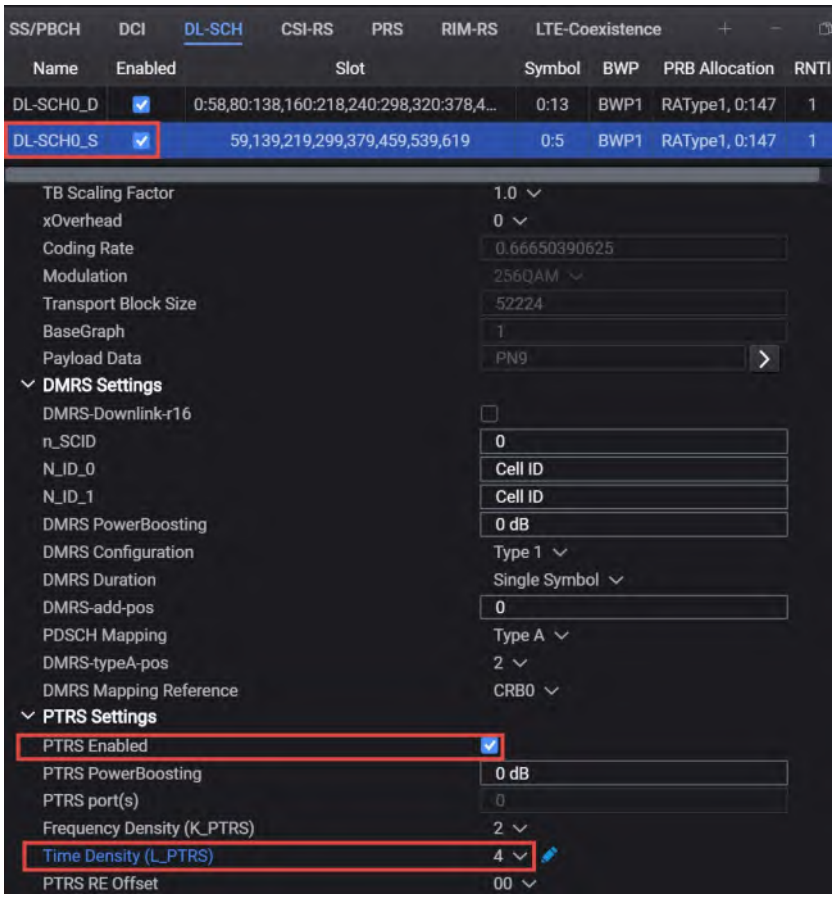
   – Set DMRS-add-pos to **1**. Additional DMRS is transmitted on a symbol before the last symbol.

   – Select **PTRS Enabled**. For FR2, PTRS is used for phase tracking.

– Set Time Density to **4**.

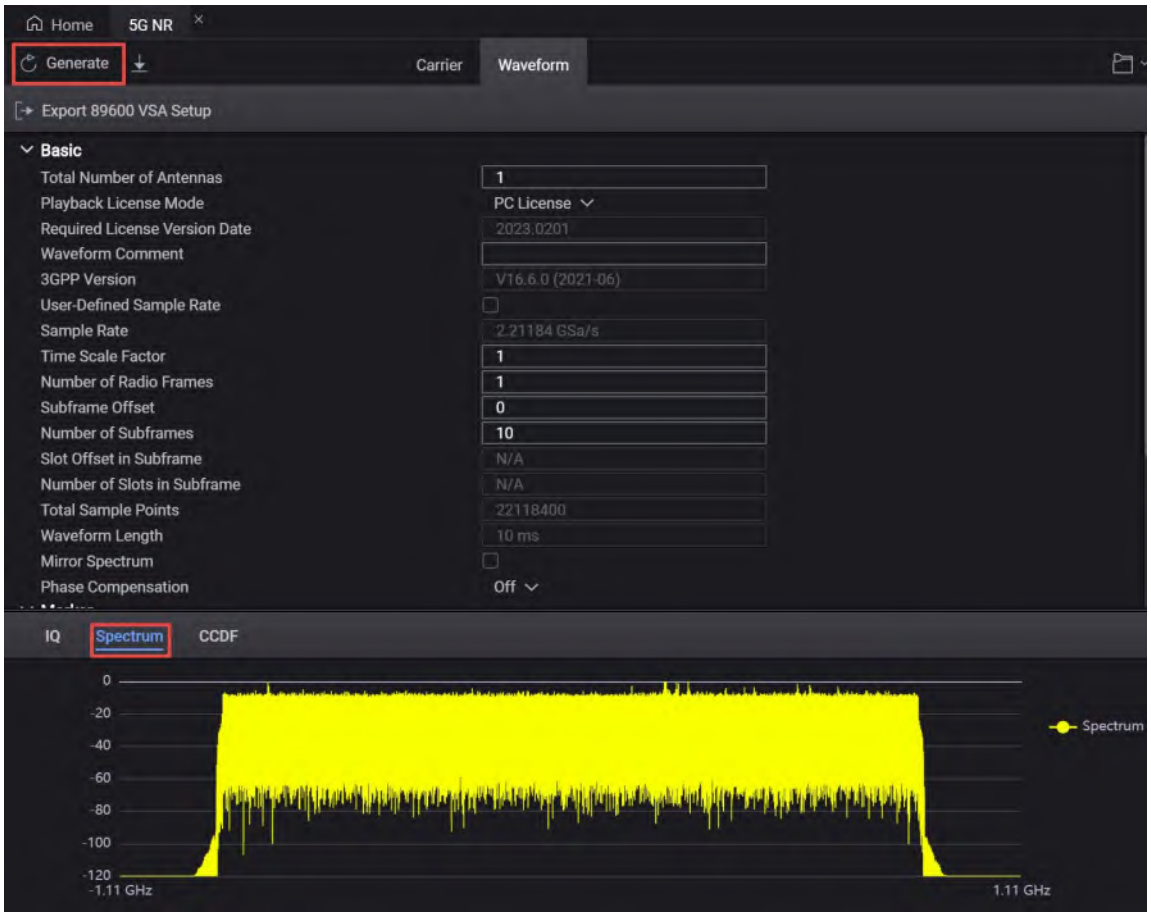


10. Select **DL-SCH0_S**. Set **PTRS Enabled** and Time Density to **4.** Confirm that Allocated Slots is set to **59,139,219,299,379,459,539,619**.



11. Select the Waveform tab, then **Generate** to generate the Waveform, and then select **Home** to exit the setup panel.

TIP When using 89600 VSA software for demodulation, you can export the VSA setup file from PathWave to demodulate the waveform quickly and easily or export the setup file into X-Apps.



12. Set RF Out to **On** by selecting the numbered channel indicator switch.

This enables the RF Out for the indicated channels, in this case Channel 1 if using a multi-channel VXG.

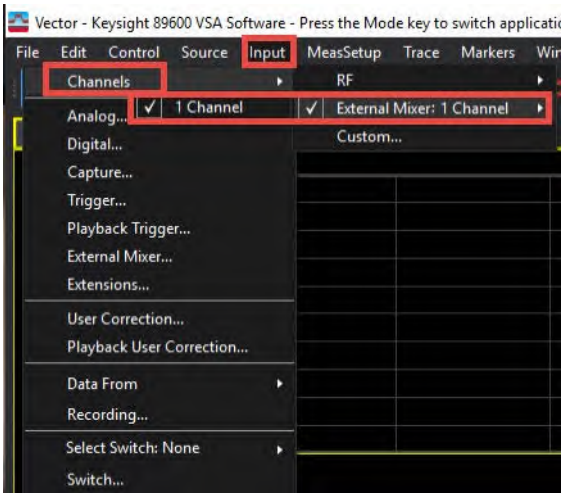
Setting Up the UXA Using 89600 VSA

NOTE Changing settings on the VSA can take longer to take affect than expected. Stopping the measurement (Control menu > Stop) until all settings have been made will help speed up the setup time.

1. Open the VSA software.

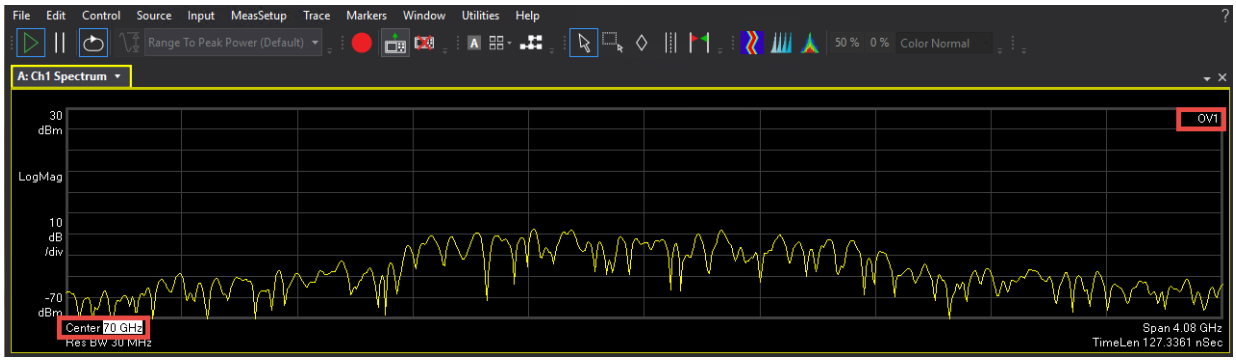
NOTE If accessing the instrument via a Remote Desktop connection, select the Screen tab (at the top of the display) to open the Mode/Measurement/View Selector window, then select **Launch VSA**.

2. From the VSA menu bar, select **File** > **Preset** > **All** to set the VSA to a known state.
3. From the menu bar, select **Input** > **Channels** > **External Mixer** > **1 Channel**.



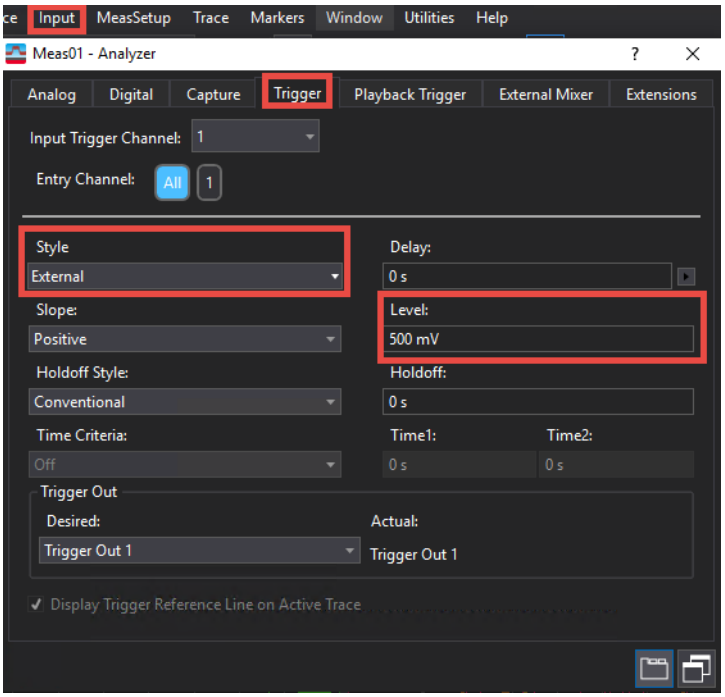
4. In the **Spectrum** window, select **Center** and set the Center Frequency to **70 GHz**.

   Notice there is an **OV1** error indicator. The OVx indicator appears when the trace was created from data that contained an ADC overload. We will resolve this in step 7.

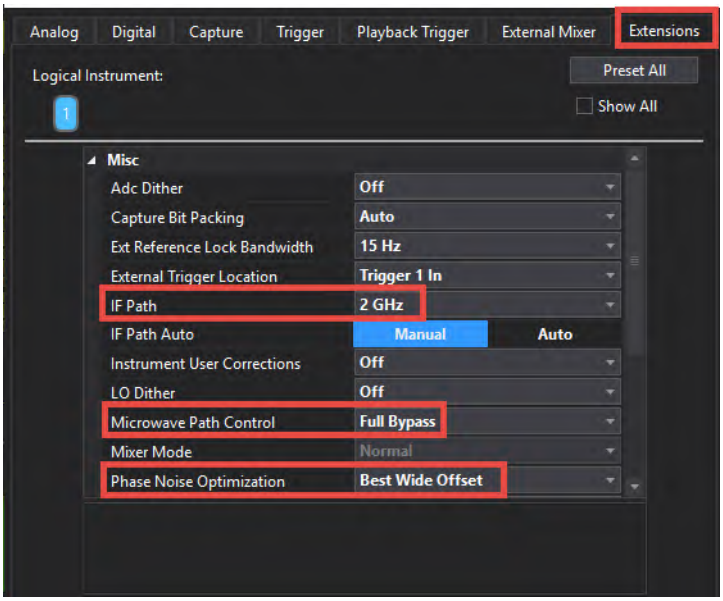


5. From the menu bar, select **Input** > **Trigger** and set Style to **External** and Level to **500 mV**.

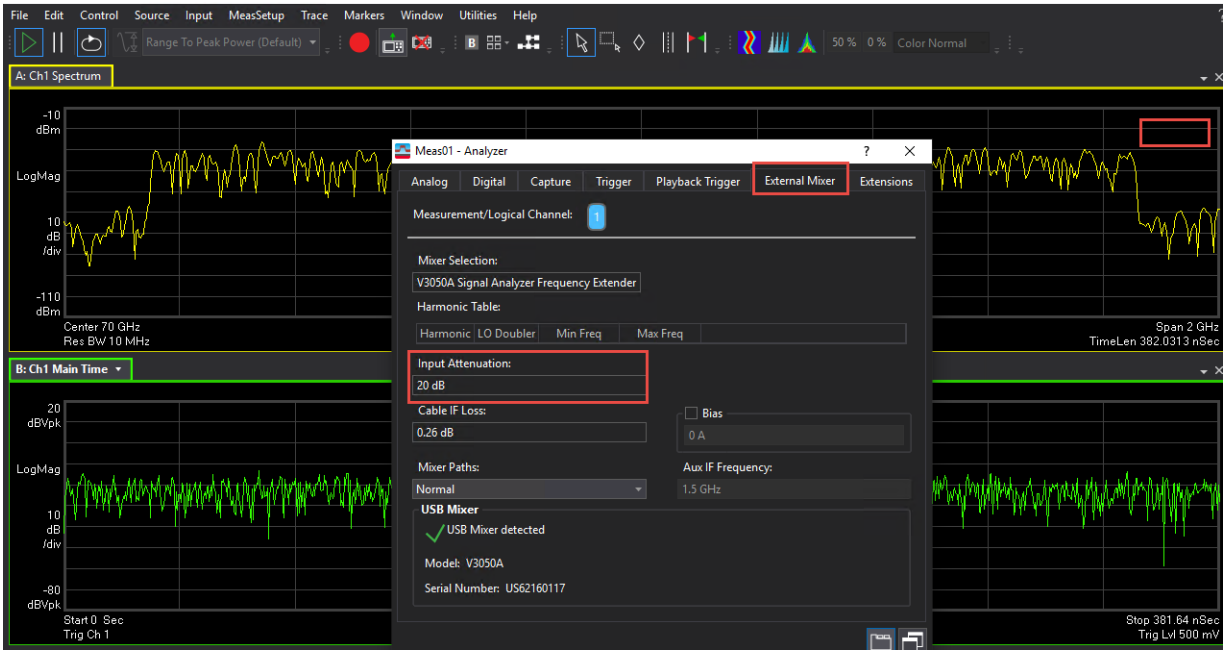
   Using the External Trigger will help to improve the demodulation speed.



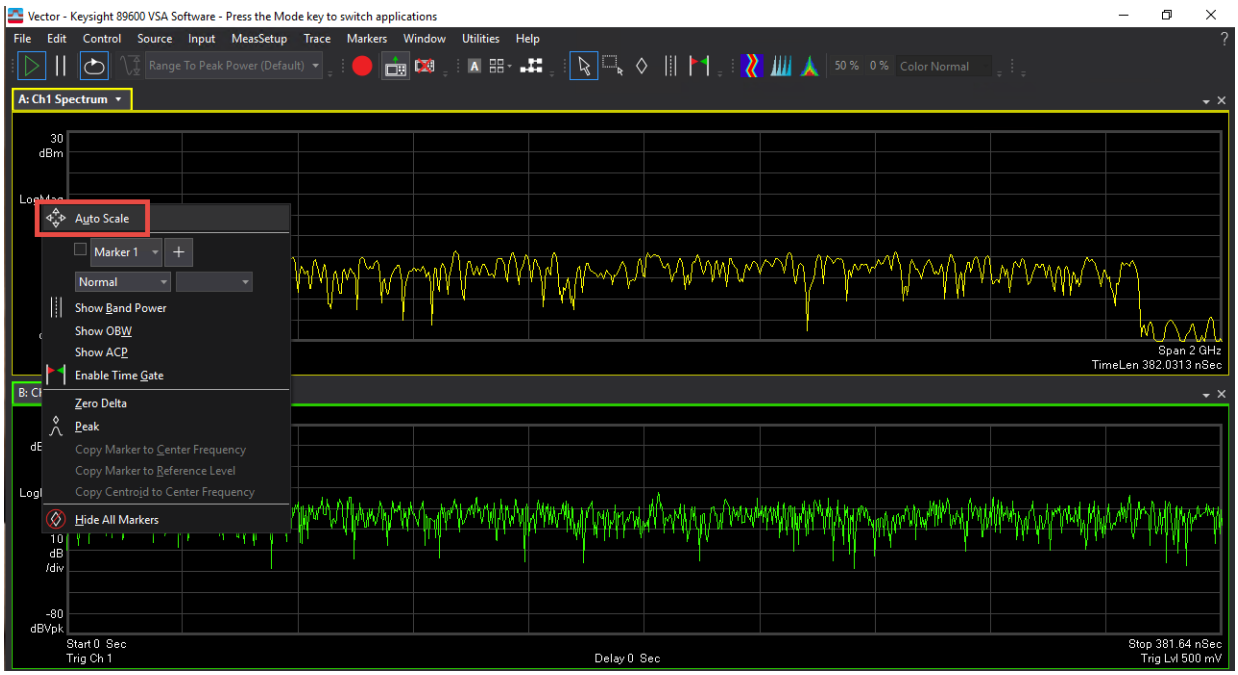
6. Select the **Extensions** tab and set:
   - IF Path to **2 GHz**
   - Microwave Path Control to **Full Bypass**
   - Phase Noise Optimization to **Best Wide Offset**



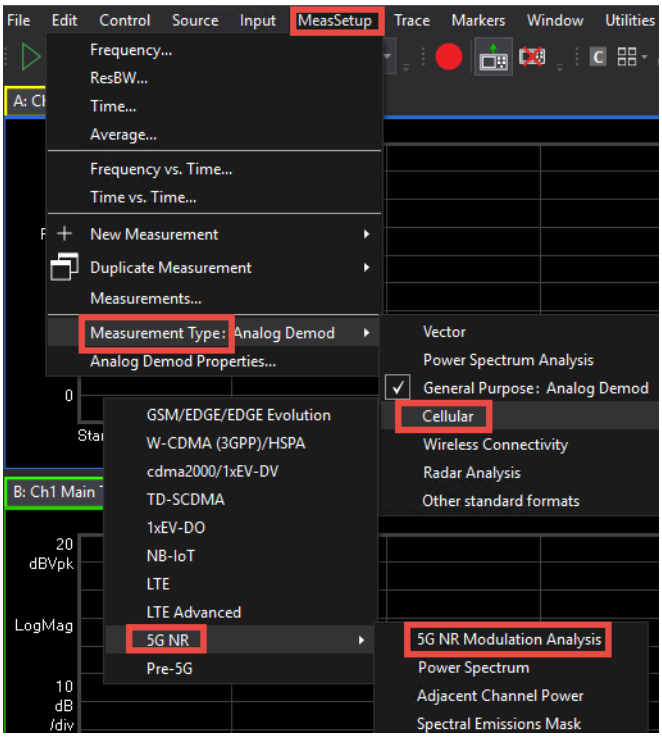
7. Select the **External Mixer** tab and set Input Attenuation to approximately **20 dB**, or a level where the overload flag disappears. This is to bring the waveform up on the spectrum display.



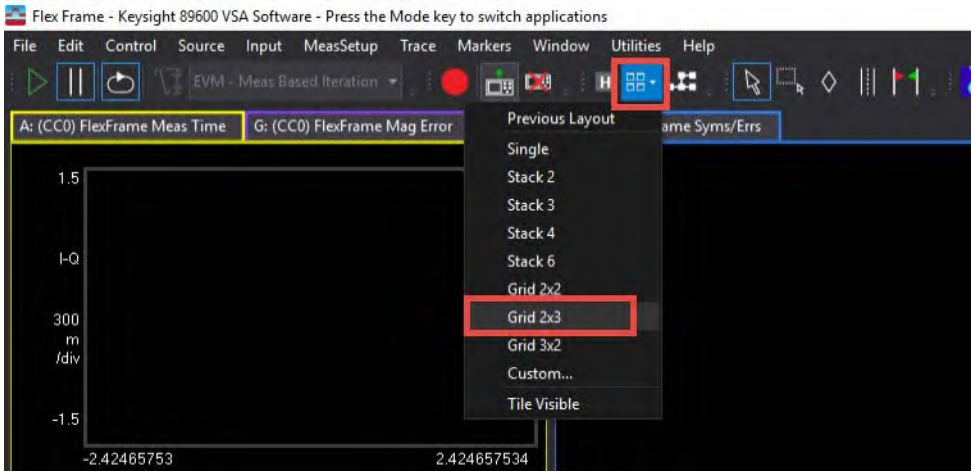
8. In the **Spectrum** window, right click on the Spectrum display and select **Auto Scale**.



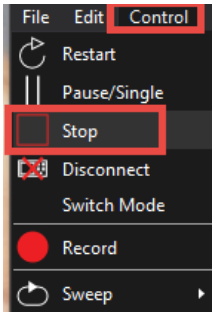
9. From the menu bar, select **MeasSetup** > **Cellular** > **5G NR** > **5G NR Modulation Analysis**.



10. Change the Trace Layout Grid to **2 x 3**. Right click on the Spectrum window and select **Auto Scale** to bring the waveform back up on the display.



11. Select the **Control** menu > **Stop** to stop the measurement until all parameters have configured. This will greatly improve setup speed.



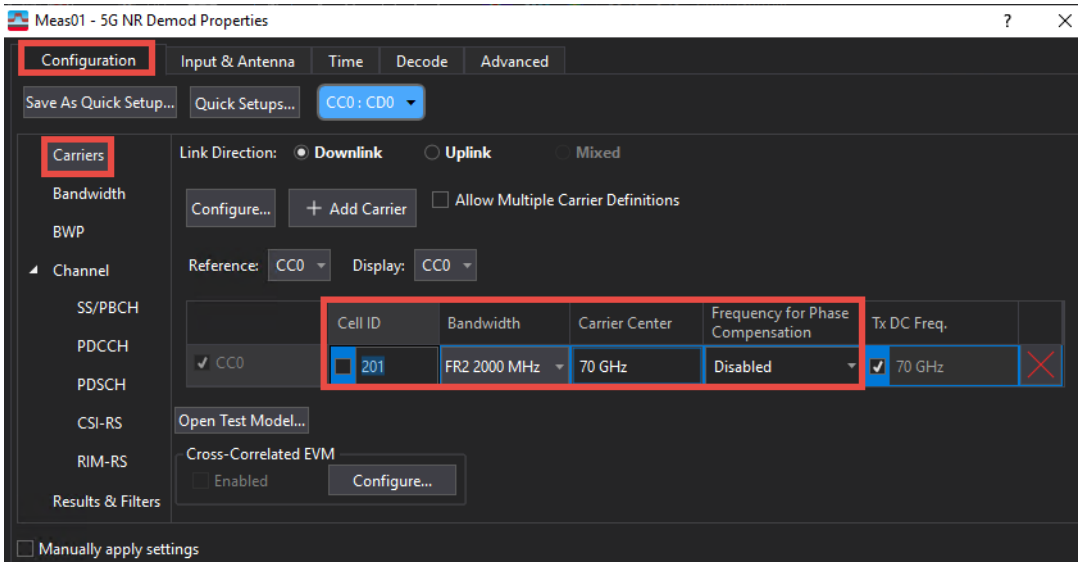
12. Select **MeasSetup** > **5G NR Demod Properties** > **Configuration** tab > **Carriers** panel and set:

    – Cell ID to **201** (or select the box to the right to Auto Detect the cell ID)

    – Bandwidth to **FR2 2000 MHz**

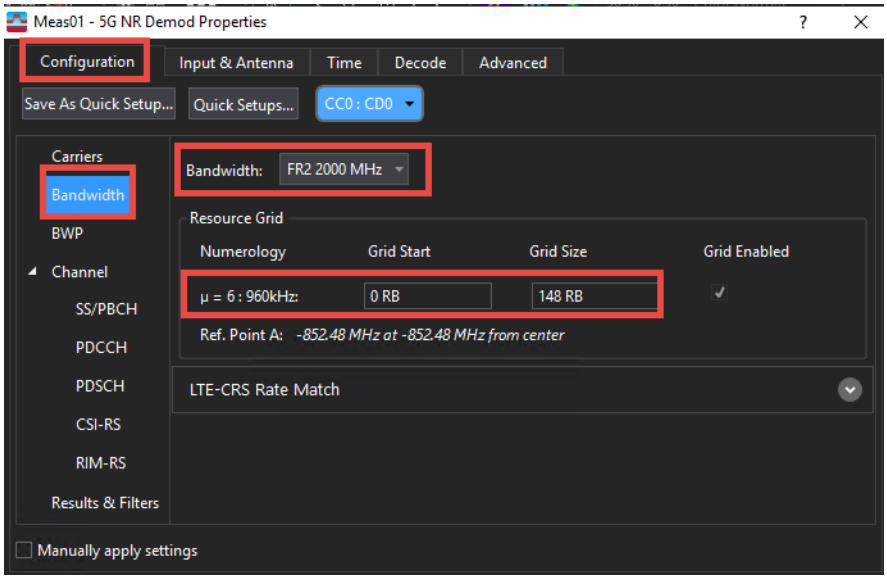
    – Frequency for Phase Compensation to **Disabled**.

    Cell ID is carried on PSS and SSS so the SS/PBCH must be transmitted and enabled for Auto Cell ID to work and must be a Downlink since SS/PBCH is only transmitted in the DL.



13. Select **MeasSetup** > **5G NR Demod Properties** > **Configuration** tab > **Bandwidth** panel. This section of the UI is to configure Resource Grid for each Numerology. Numerology is set to $\mu = 6$:960 kHz.

For FR2 2000 MHz, only the 960 kHz subcarrier spacing is available. This is why the other numerologies are not shown. The Max RB is 148 RB. We will use this value when we configure BWP, SS/PBCH, and PDSCH parameters.

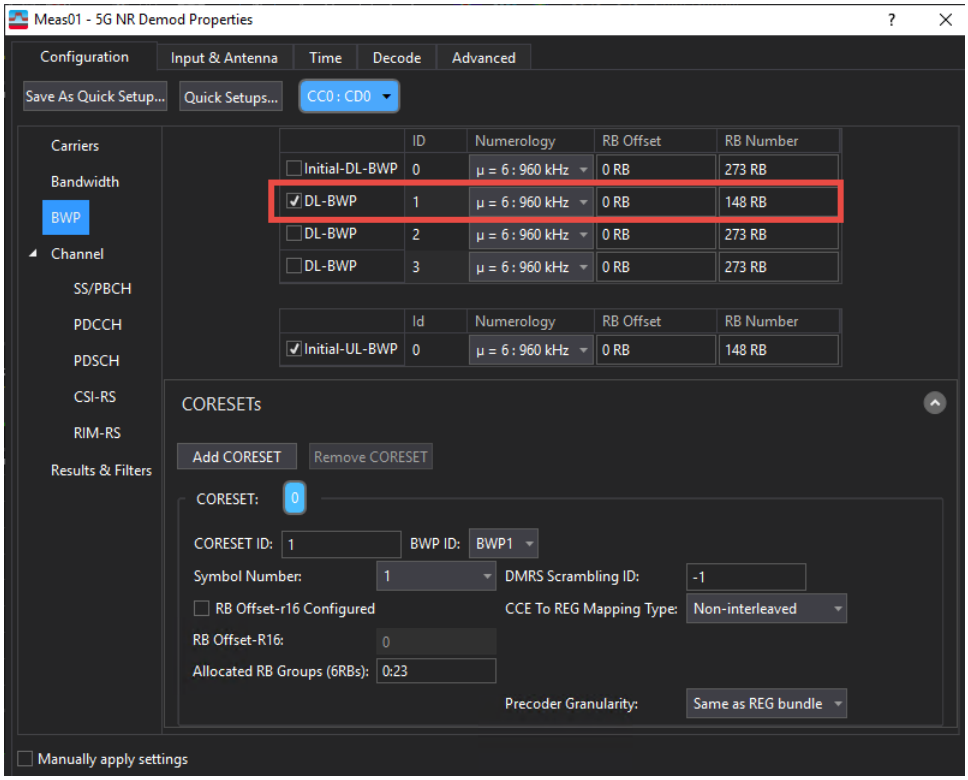


14. Select **MeasSetup** > **5G NR Demod Properties** > **Configuration** tab > **BWP** panel. We will use the default settings for DL-BWP 1.

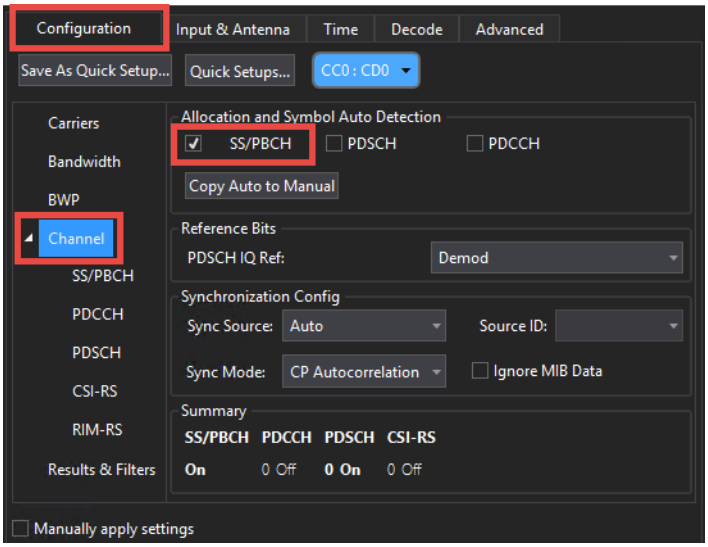
    This section of the UI is to configure the DL BWP allocation within the component carrier and the control resource set (CORESET). CORESET is used for PDCCH configuration and there is no PDCCH in this measurement example, so we don't need to make changes.

    A CC with 2000 MHz at FR2 has a max RB of 148 for 960 kHz numerology. In this example, the DL-BWP uses the full carrier bandwidth of 148 RBs

When using multiple BWPs within a single component carrier, you will spread the total 148 RBs across the multiple BWPs.

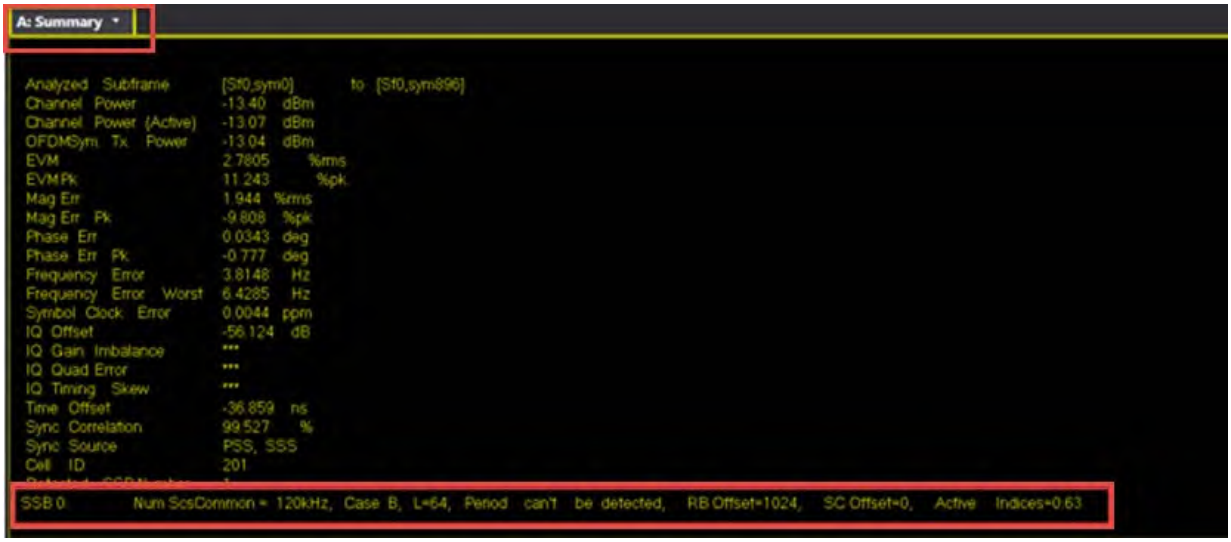


15. Select **MeasSetup** > **5G NR Demod Properties** > **Configuration** tab > **Channel** > **SS/PBCH**.



The auto detected parameters are returned in the Summary table trace. Once you see what has been auto detected and are happy with what you see, you can do "Copy Auto to Manual" to copy the detected parameters into the SS/PBCH settings. See image below.

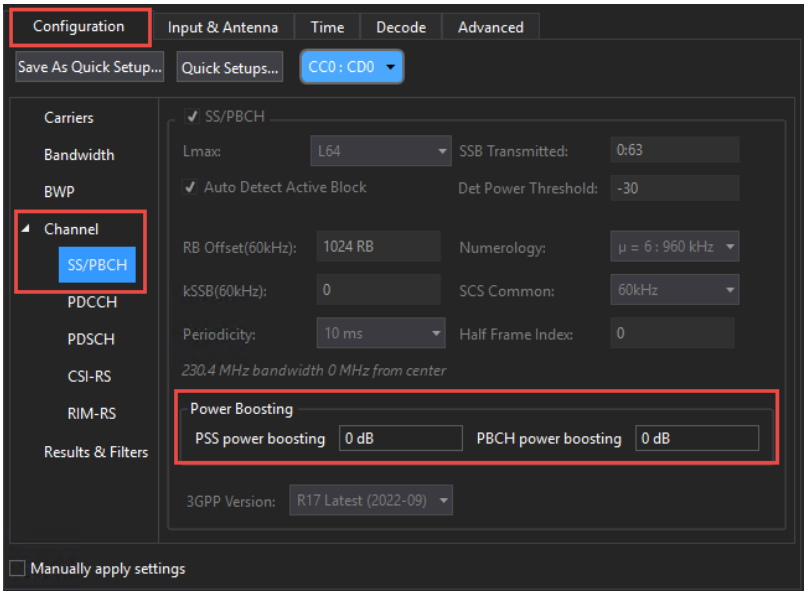
The auto detected SSB parameters are returned in the Summary table. "Copy Auto to Manual" will copy these detected parameters into the SS/PBCH settings.



16. Select **MeasSetup** > **5G NR Demod Properties** > **Configuration** tab > **Channel** > **SS/PBCH**.

    PSS and PBCH Power boosting cannot be auto detected, you must enter power boosting values under SS/PBCH. We will use the default values. Notice that the remaining values are grayed out as we are in Auto Detection mode.

    Note that the frequency location of a SSBlock is not fixed. In our software, the default value of 1024RB and kSSB of 0 subcarriers places it in the center of the carrier bandwidth. If you set RB Offset to 0 dB, it places the SSB on the lower edge of the carrier bandwidth



17. Select **MeasSetup** > **5G NR Demod Properties** > **Configuration** tab > **Channel** > **PDSCH** and set:

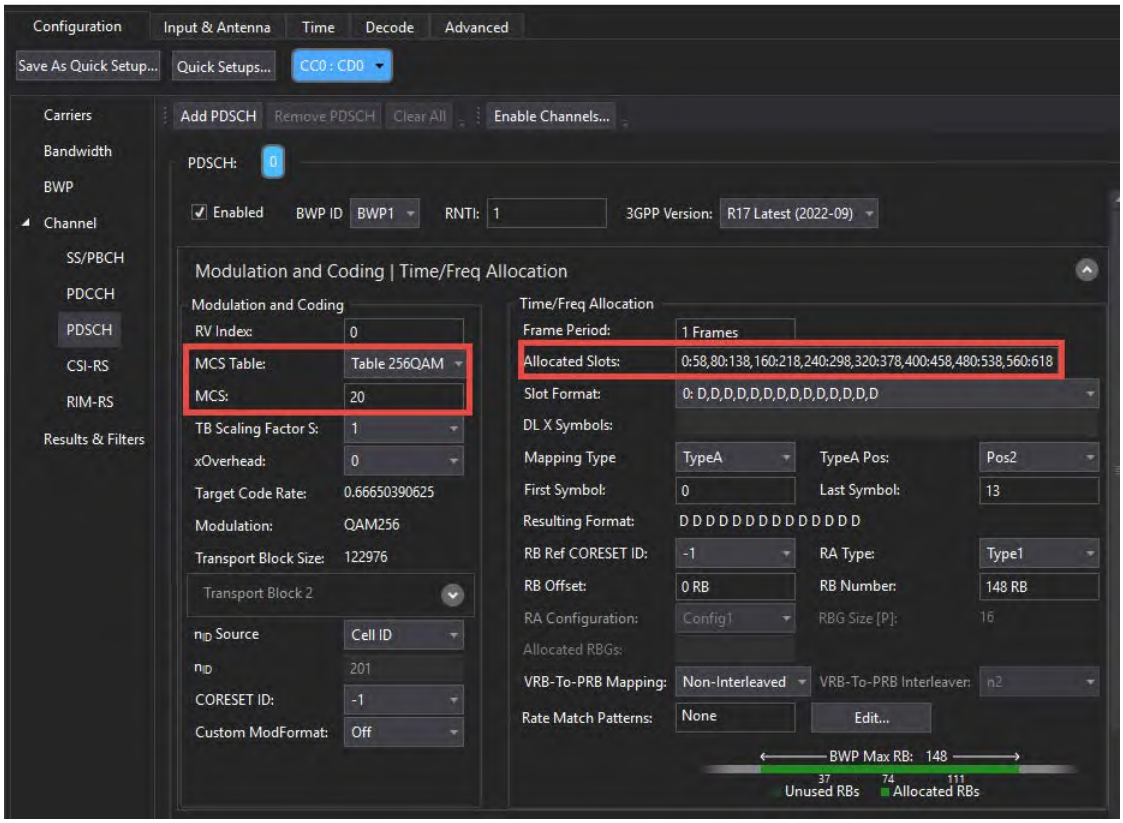
    – MCS Table to **Table 256QAM**

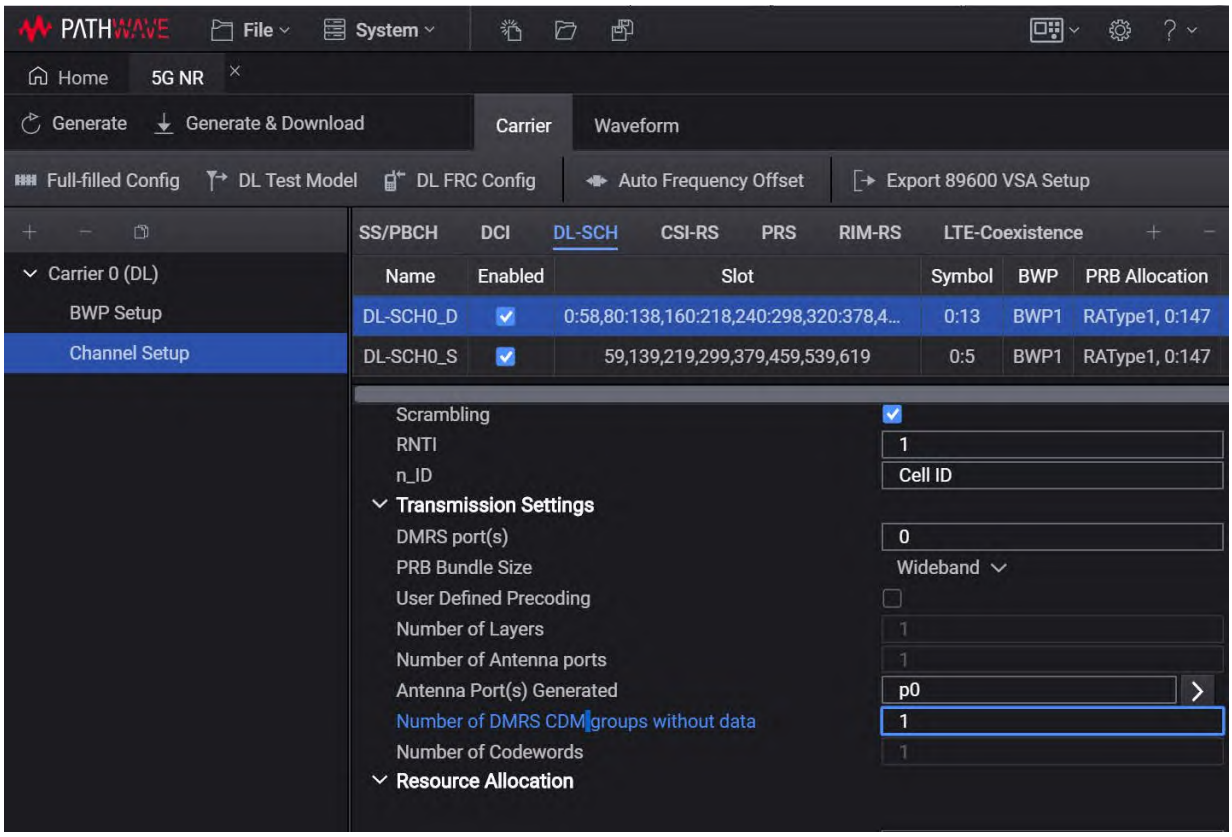
    – MCS to **20**

      Transport Block Size value must match the PathWave 5G NR settings for successful decoding (CRC result).

    – Allocated Slots to **0:58,80:138,160:218,240:298,320:378,400:458,480:538,560:618**

TIP You can copy/paste these values from **PathWave 5G NR Signal Generation**.

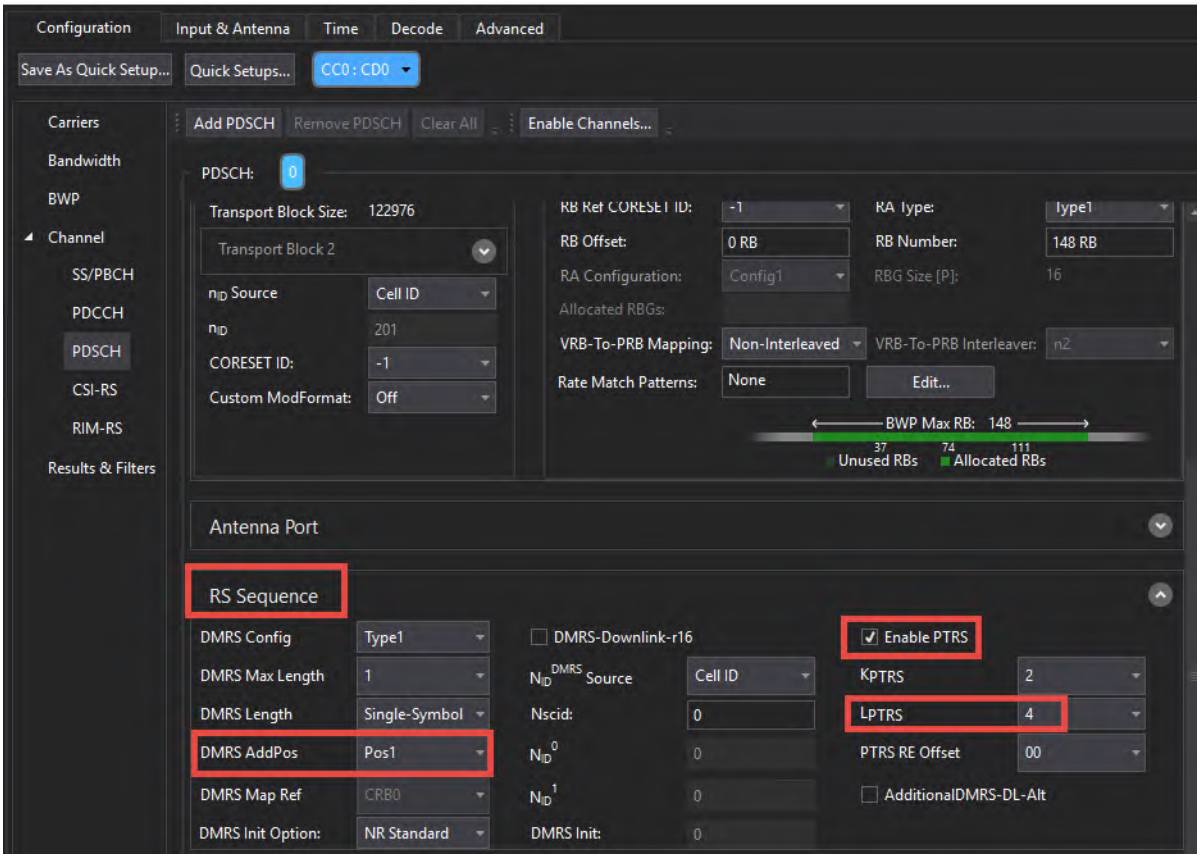




18. Scroll Down to **RS Sequence** and set:

- DMRS AddPos to **Pos1**
- Select **Enable PTRS**

– LPTRS to **4**



19. Select **Add PDSCH** and then select **PDSCH 1**. Set:

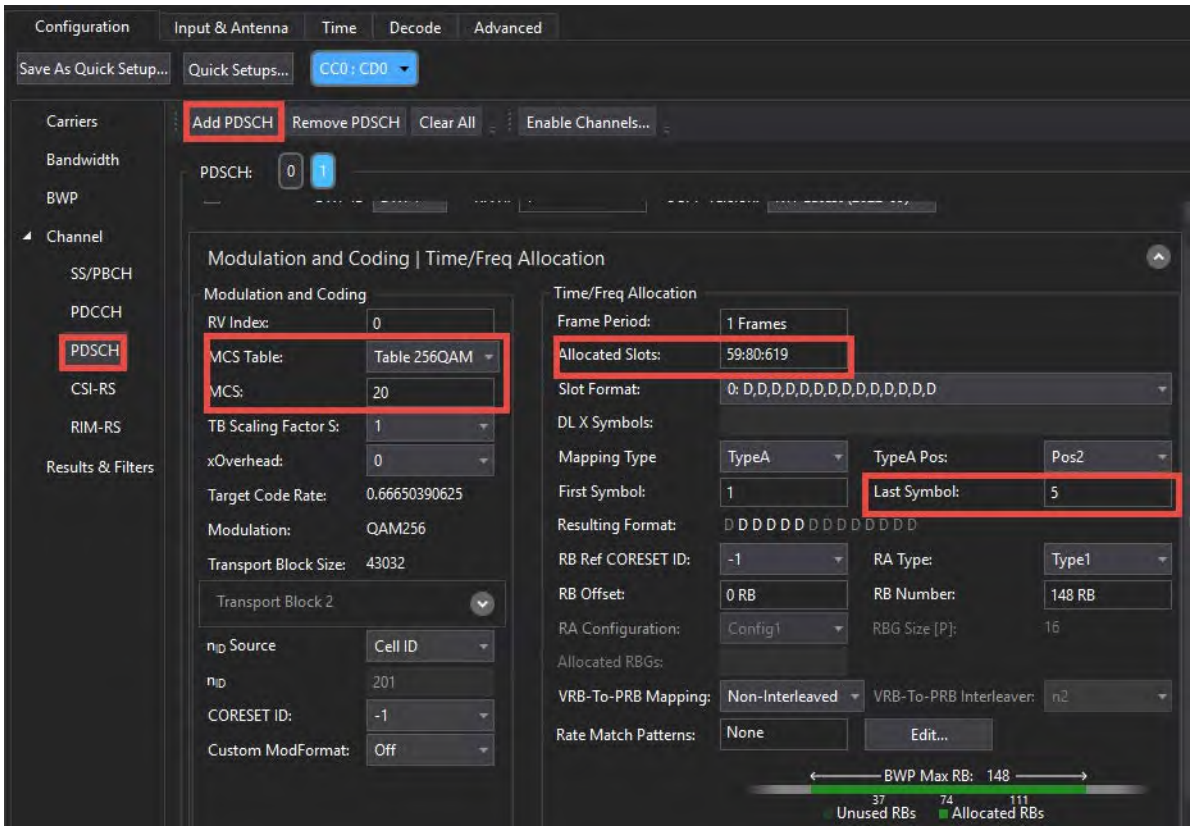
– MCS Table to **Table 256 QAM**

– MCS to **20**

Transport Block Size value must match the PathWave 5G NR settings for successful decoding (CRC result). The Value depends on parameters in this section and the RNTI value of the PDSCH much also match. This will not affect EVM.

– Allocated Slots to **59:80:619**

– Last Symbol to **5**

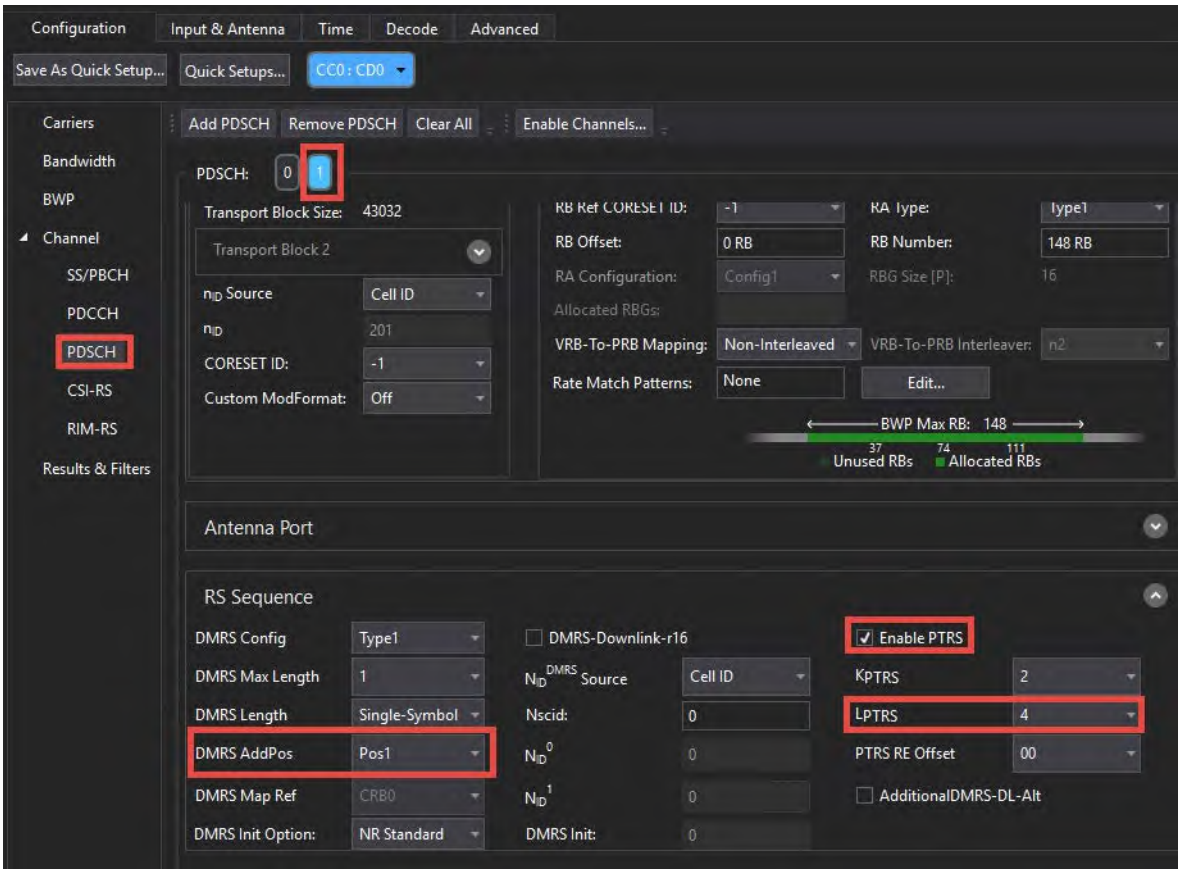


20. Scroll Down to **RS Sequence** and set:

– DMRS AddPos to **Pos1**

– Select **Enable PTRS**

– LPTRS to **4**



21. In the **Time** tab set:

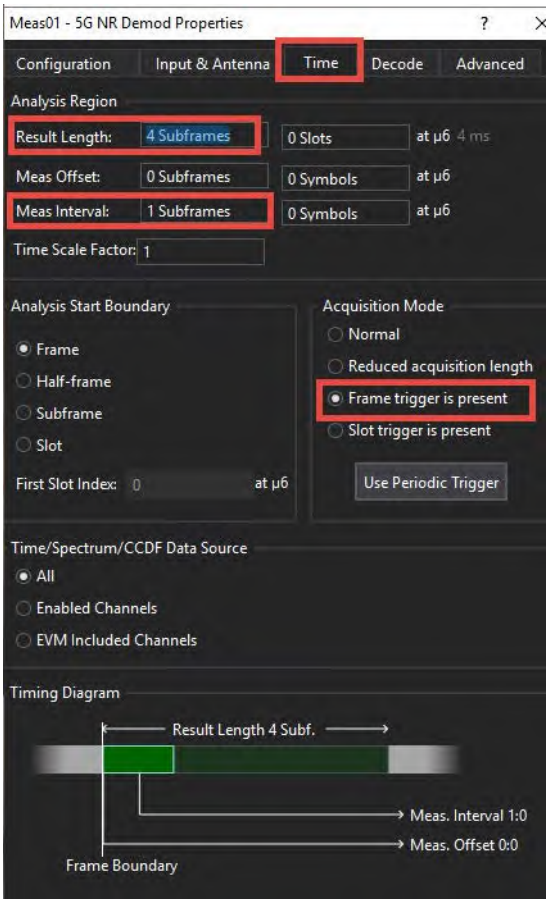
– Result Length to **4 Subframes**

– Meas Interval to **1 Subframes**

We are reducing the analysis region since a 2000 MHz carrier has a large amount of data resulting in very slow demodulation speed. (That is, 5x more data than 400 MHz carrier.)

– Acquisition Mode to **Frame Trigger is Present**

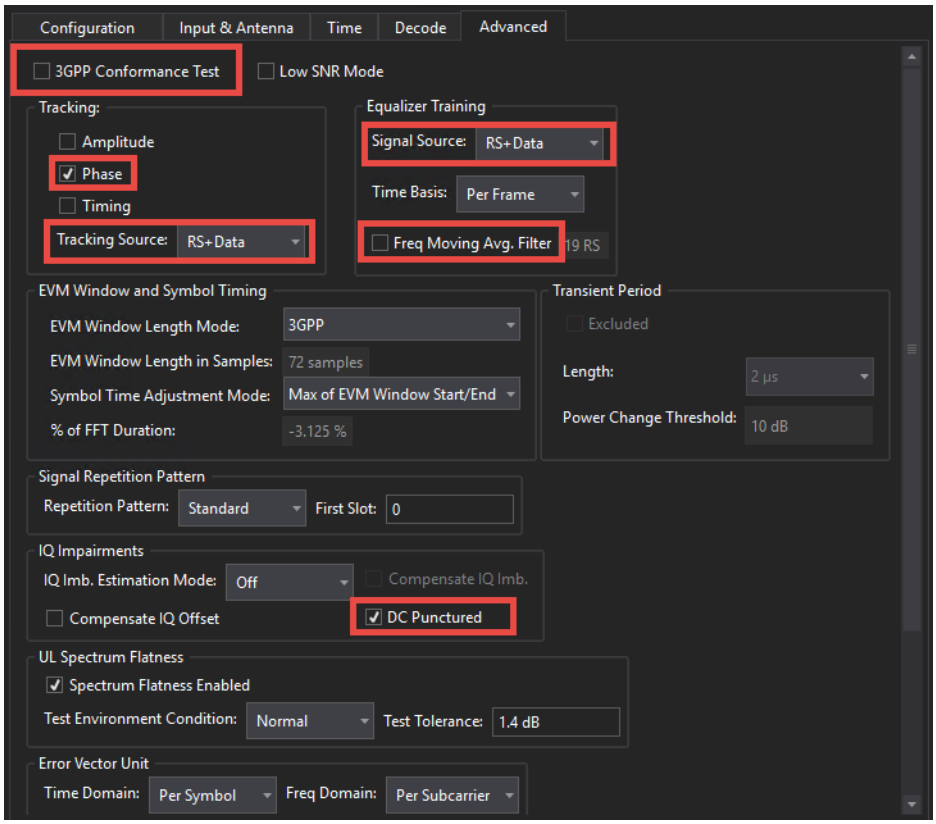
We are using an external trigger for this example, so selecting "Frame Trigger is Present" will use the external trigger and will speed up the measurement significantly.



22. Select the **Advanced** tab:

- Clear the **3GPP Conformance** checkbox
- In the Tracking area, set Tracking Source to **RS + Data** and select **Phase**
- In the Equalizer Training area, set Signal Source to **RS + Data** and clear the **Freq Moving Avg. Filter** checkbox.
- Turn on **DC Punctured**.

5G NR counts the DC subcarrier as a valid subcarrier for rate-matching purposes. High LO feedthrough will impact demodulation and EVM performance of the input.



23. Close the 5G NR Demod Properties window and select Control > Restart to restart the measurement.

24. View the results of the measurement. Try to improve EVM by adjusting the Mechanical Attenuation setting until the OV1 indicator goes away in the IF Gain and Mechanical Attenuation settings in the **Input** >**Extensions** and the **Input** > **External Mixer windows**.

For this example, the optimum settings were Input Attenuation set to 8 dB and IF gain set to -12 dB.

File Edit Control Source Input MeasSetup Trace Markers Window Utilities Help
A: OFDM Meas: All Layers
Const
-2.425
Res BW 960 kHz
2.4247
TimeLen 1.018 kSym
C: Summary
Analyzed Subframe [Sf0,sym0] to [Sf0,sym896]
Channel Power -13.12 dBm
Channel Power (Active) -12.8 dBm
OFDMSym. Tx. Power -12.76 dBm
EVM 2.0643 %rms
EVM Pk 12.213 %pk
Mag Err 1.460 %rms
Mag Err Pk 11.578 %pk
Phase Err 0.0261 deg
Phase Err Pk 0.6238 deg
Frequency Error -4.4758 Hz
Frequency Error Worst ***
Symbol Clock Error 0.0000 ppm
IQ Offset -65.702 dB
IQ Gain Imbalance ***
E: OFDM Detected Allocations Time: All Layers
Real
Start 0 sym
Trig Ch 1
Delay 0 Sec
Stop 895 sym
Trig Lvl 500 mV
B: Ch1 Spectrum
LogMag
Center 70 GHz
Res BW 3.81936 kHz
Span 2 GHz
TimeLen 1 mSec
D: OFDM Err Vect Spectrum: All Layers
LinMag
Start -888 carrier
Res BW 960 kHz
Stop 887 carrier
TimeLen 1.018 kSym
F: Frame Summary
Name EVM (%rms) SINR (dB) Power per RE (dBm) Modulation Num. of RBs RNT
PSS 1.791 34.94 -45.48 BPSK 384
SSS 1.855 34.63 -45.49 BPSK 384
PBCH 1.767 35.06 -45.41 QPSK 640
PBCH_DMRS 1.764 35.07 -45.41 QPSK 640
PDSCH0 2.074 33.66 -45.25 QAM256 8732
PDSCH0_DMRS 2.050 33.77 -45.25 QPSK 8092
PDSCH0_PTRS 2.092 33.59 -45.26 QPSK 4366
PDSCH1 2.227 33.05 -45.23 QAM256 148
PDSCH1_DMRS 2.252 32.95 -45.25 QPSK 148
PDSCH1_PTRS 2.331 32.65 -45.24 QPSK 74

Using the X-Series Analyzer's SCPI Recorder Function

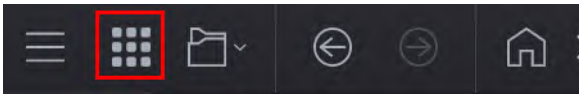
The SCPI Recorder feature allows you to view active recording content, and edit the content. Right-click or touch and hold on any UI control to display a menu allowing you to record the SCPI associated with the control. You can also record a series of commands. These commands can be viewed and edited directly, or you can also play, save/recall for future use.

For this example, we will generate a simple 5G NR 100 MHz signal.

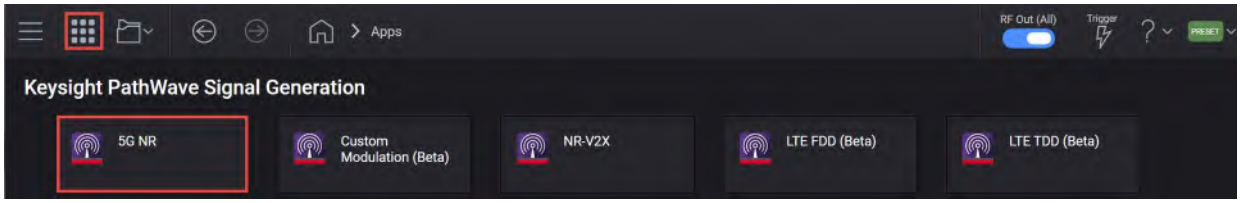
Using the graphical user interface

On the VXG:

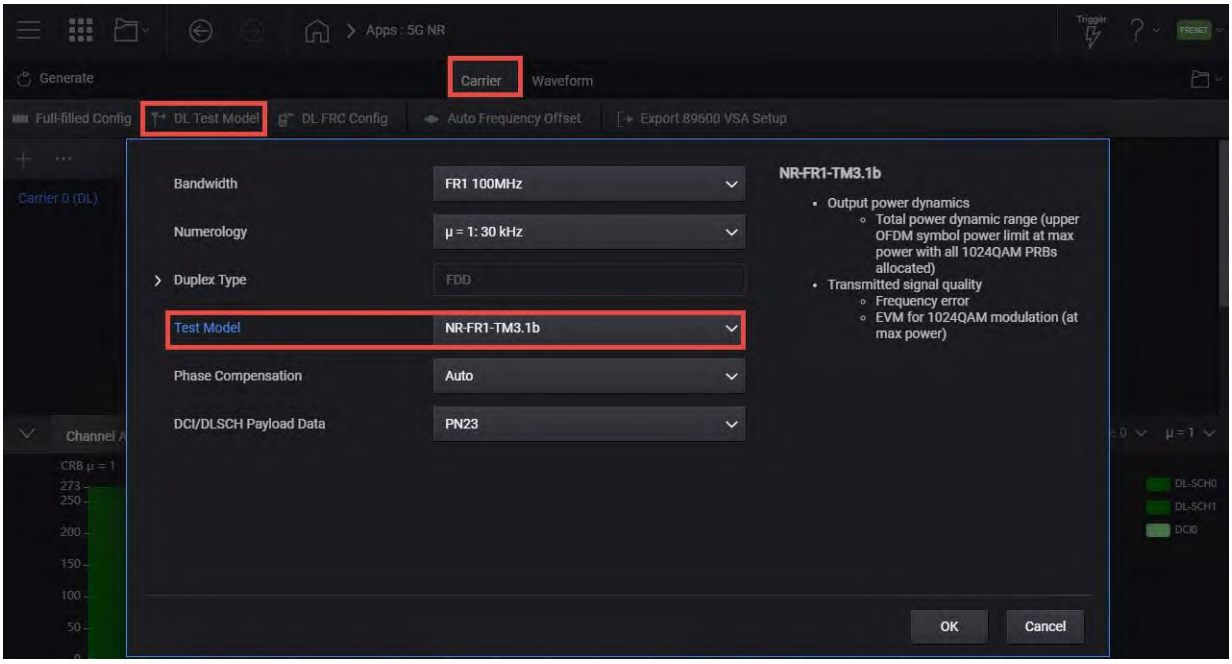
1. Select **Preset** > **Preset** to set the VXG to a known state.
2. In the Output area, set Frequency to **3.5 GHz** and Power to **0 dBm**.
3. Select the **Radio Apps** block to open the mode selection panel.



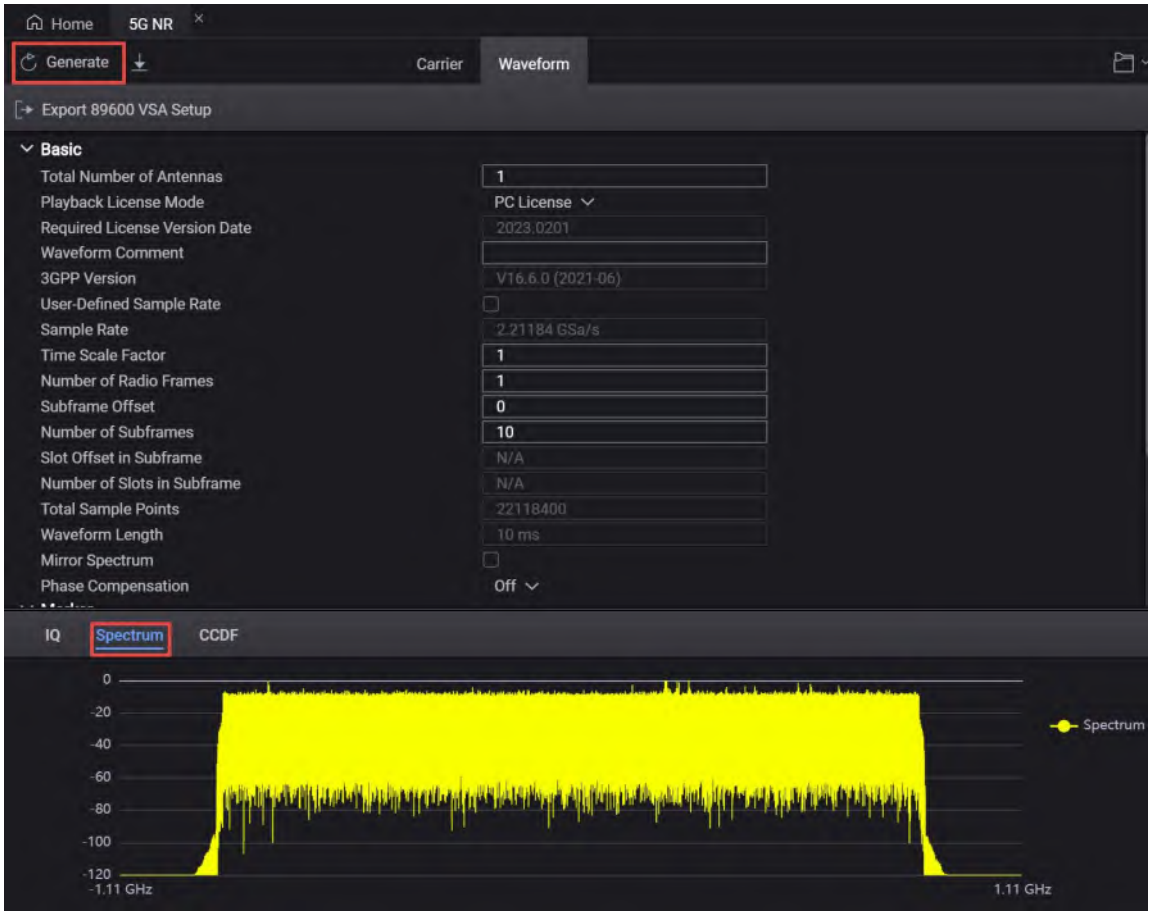
4. Select **5G NR**.



5. Select the **Carrier** tab > **DL Test Model** and set Test Model to **NR-FR1-TM3.1b**, leave the remain settings at their default values, and select **OK**.



6. Select the **Waveform** tab, then **Generate** to generate the Waveform, and then select **Home** to exit the setup panel.



7. Set RF Out to **On** by selecting the numbered channel indicator switch.

   This enables the RF Out for the indicated channels, in this case Channel 1 if using a multi-channel VXG.

On the X-Series Signal Analyzer:

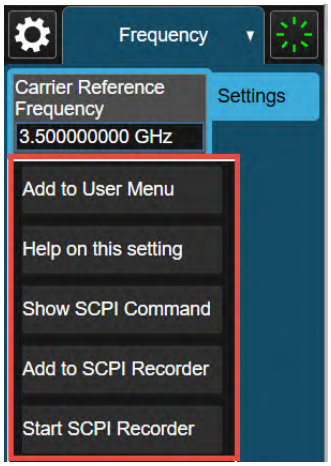
1. Select **Mode/Meas** > **5G NR &V2X** Mode > **OK**.

NOTE

If accessing the instrument via a Remote Desktop connection, select the Screen tab (at the top left of the display) to open the Mode/Measurement/View Selector window.

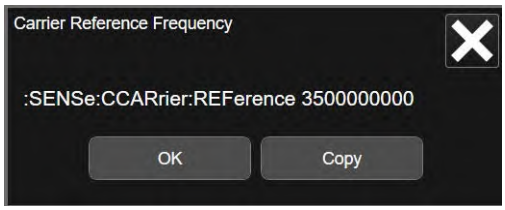
2. Select **Mode Preset** to set Spectrum Analyzer 5G NR mode to a known state.

3. From the N9042B Menu Panel (or the Screen tab), select **Mode/Meas** > **5G NR & V2X** Mode > **Modulation Analysis** Measurement > **OK**.
4. Select **Frequency** and set Carrier Reference Frequency to **3.5 GHz**.
   - Touch and hold the **Carrier Reference Frequency** function for at least one second on the touch screen or if using a mouse pointer and right click to open the menu.



   - Select **Show SCPI Command**.

     The SCPI command for this function is displayed in the dialog box.

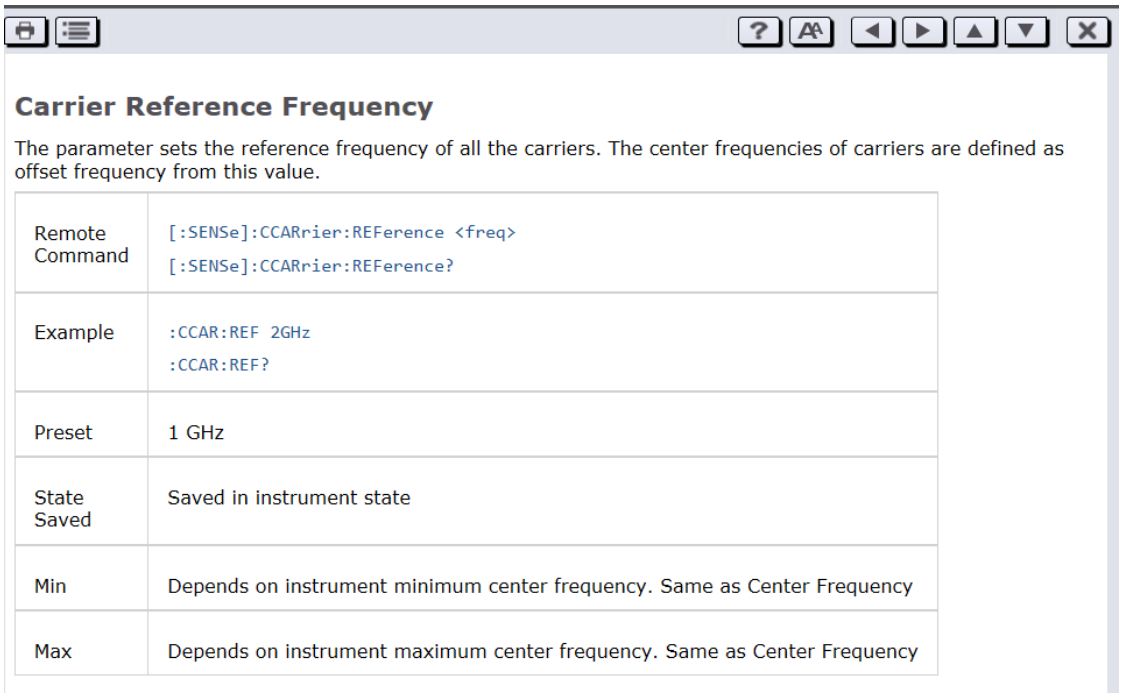


     Select Copy to copy the current SCPI command string for further use.

     If you use Windows Remote Desktop or a VNC viewer with clipboard synchronization enabled, you can copy the SCPI command into the clipboard. This makes SCPI commands available on the fly via a simple copy/paste operation.

   - Select **Help on this setting** to open the SCPI syntax document page.

You will see the background information for that setting including allowed values, SCPI command and examples, and other useful information.



Carrier Reference Frequency

The parameter sets the reference frequency of all the carriers. The center frequencies of carriers are defined as offset frequency from this value.

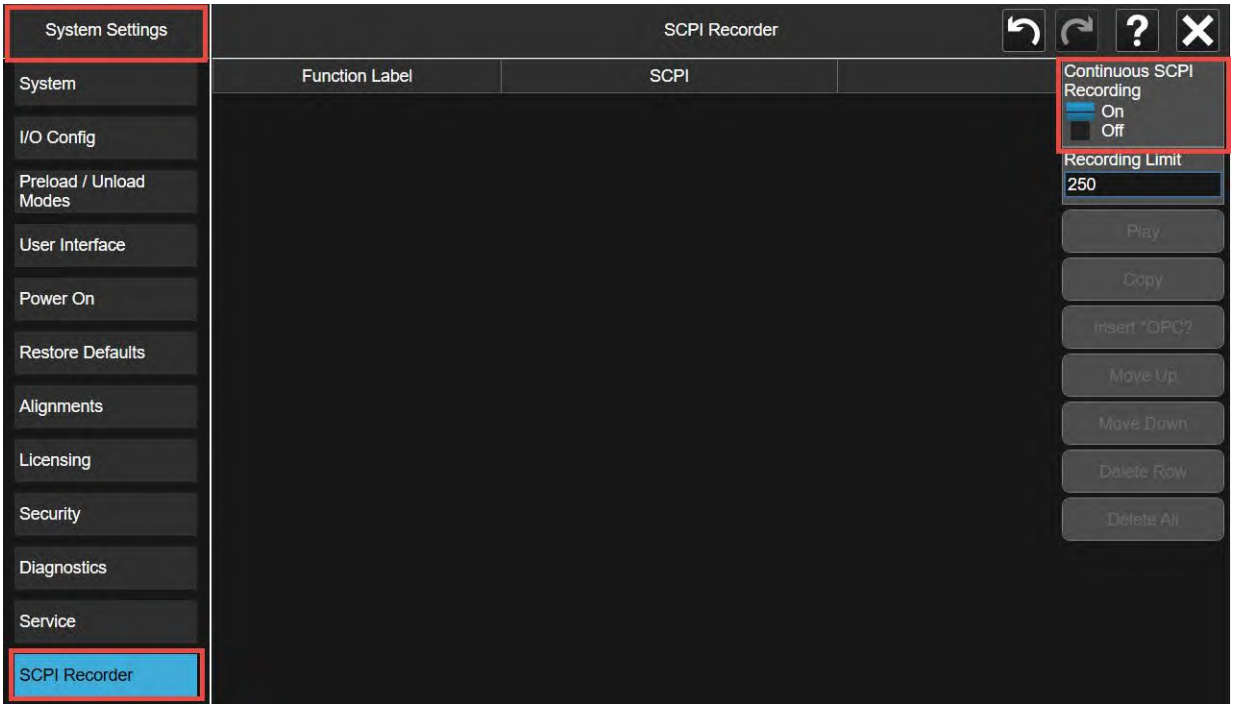
Remote Command	[:SENSe]:CCARrier:REFerence <freq> [:SENSe]:CCARrier:REFerence?
Example	:CCAR:REF 2GHz :CCAR:REF?
Preset	1 GHz
State Saved	Saved in instrument state
Min	Depends on instrument minimum center frequency. Same as Center Frequency
Max	Depends on instrument maximum center frequency. Same as Center Frequency

To Start Continuous SCPI Recording:

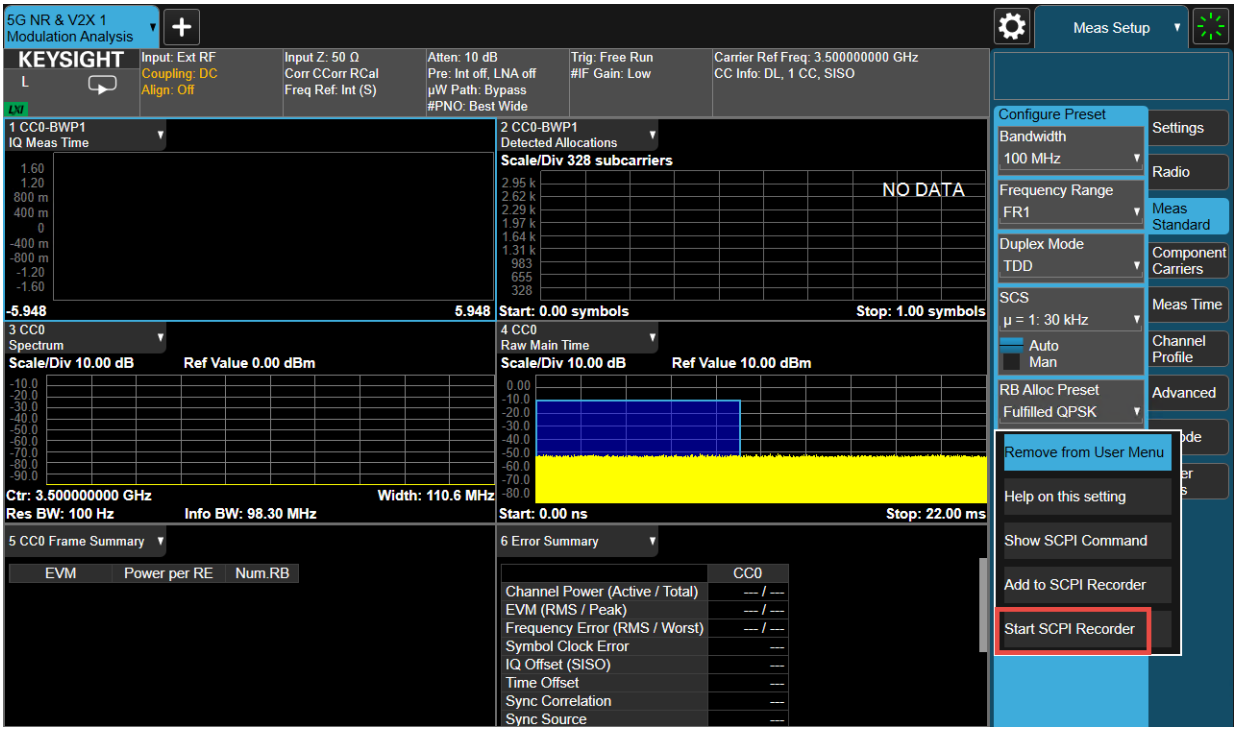
The SCPI Recorder supports an automatic mode of operation. In this mode, a series of manual operations can be added to the list automatically without any further user activity. You can start the recording process by enabling the Continuous SCPI Recording, then disable it to stop recording after completing all configuration steps. This is normally the best choice for fast and convenient SCPI list recording.

There are two ways to start continuous SCPI recording:

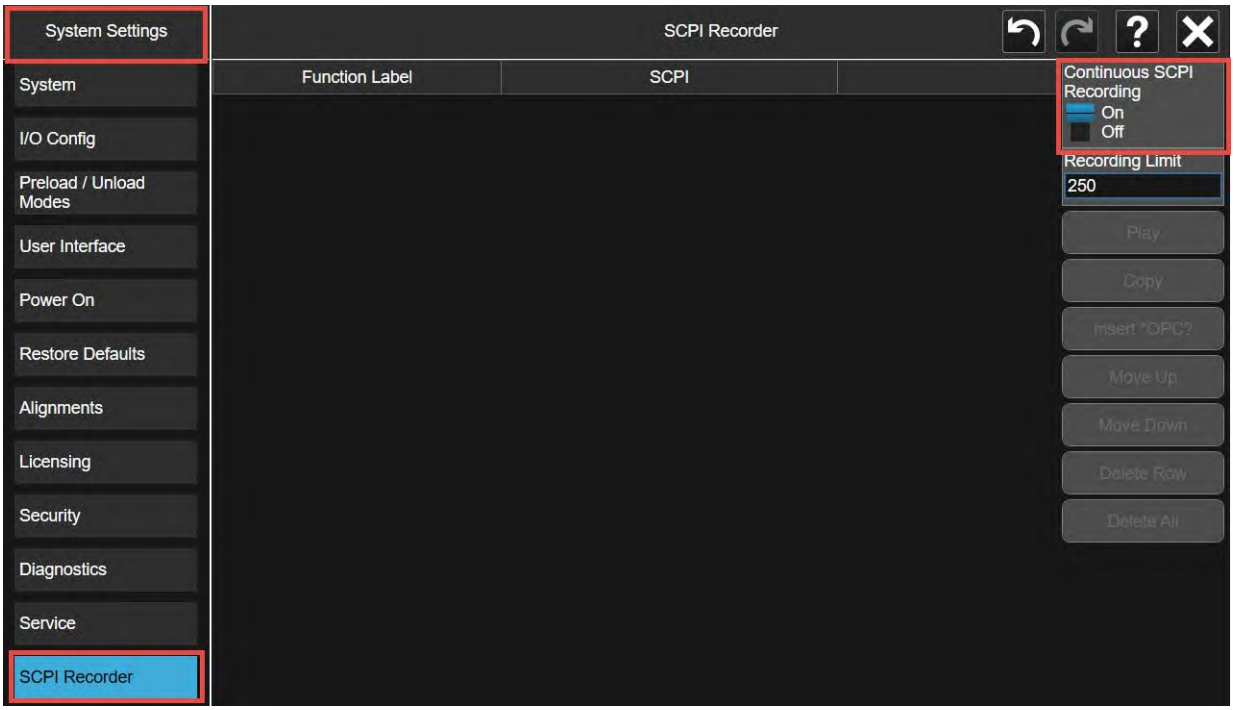
– From the System Settings > SCPI Recorder.



– From the context sensitive menu of the function key.



1. Select **Mode Preset** to set 5G NR V2X mode to a known state.
2. For this example, select the **System Settings** icon (Gear wheel on top right of the display) > **SCPI Recorder** > **Continuous SCPI Recording On**. Close the System Settings window by selecting the "X" at the top of the window.



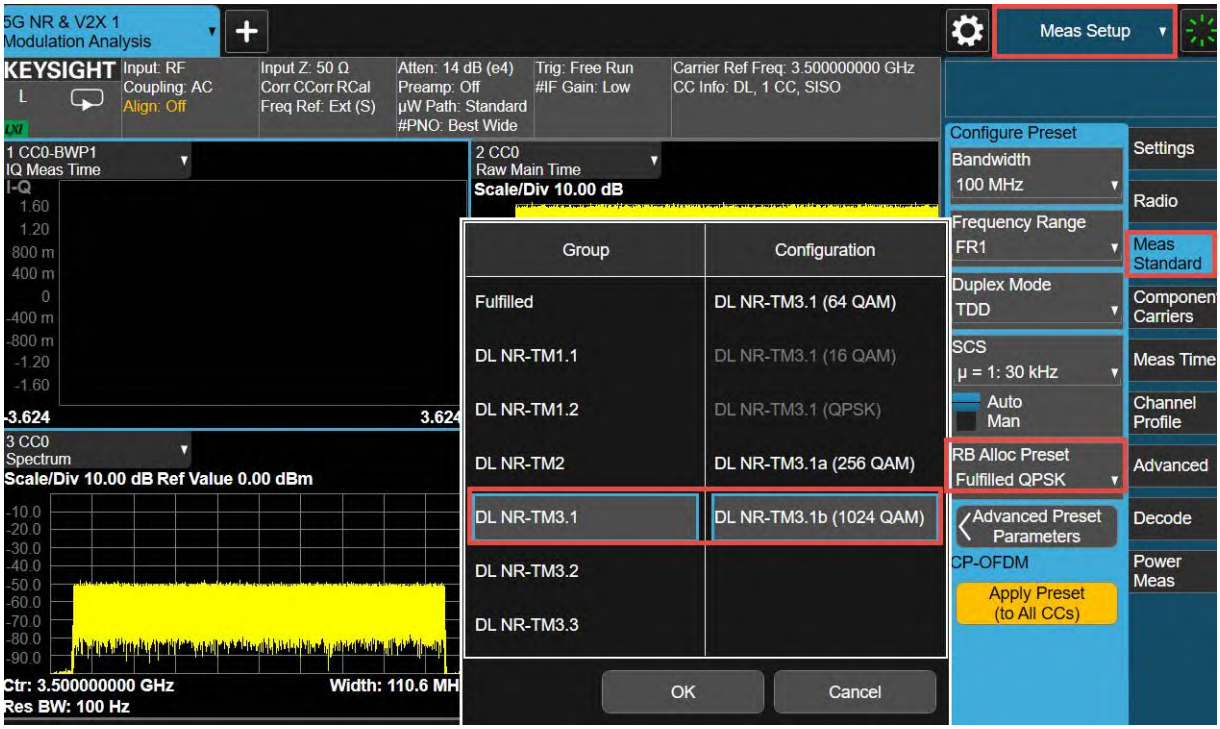
3. Select **Mode/Meas** > **5G NR &V2X** Mode > **Modulation Analysis** Measurement > **Normal** View > **OK**.

NOTE

If accessing the instrument via a Remote Desktop connection, select the Screen tab (at the top left of the display) to open the Mode/Measurement/View Selector window.

4. Select **Frequency** > **Carrier Reference Frequency** and set to **3.5 GHz**.

5. Select **Meas Setup** > **Meas Standard** > **RB Alloc Preset** > **DL NR-TM3.1** > **DL NR-TM3.1b (1024QAM)** > **OK**.

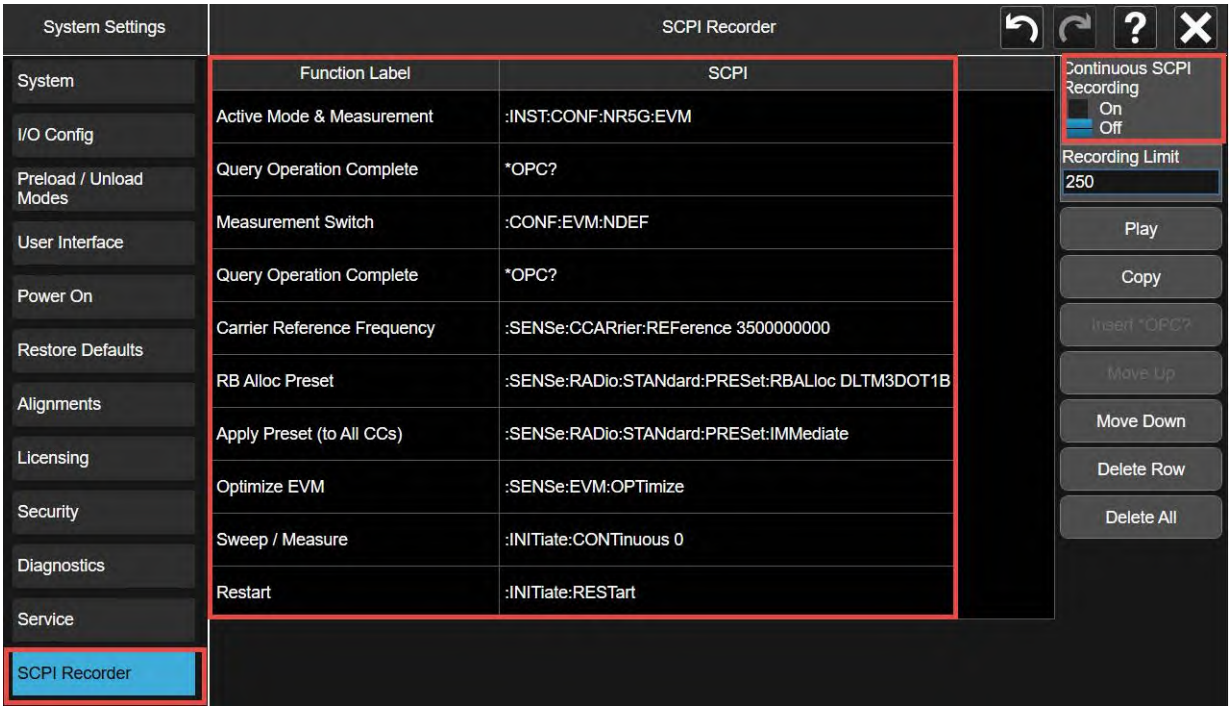


6. Select **Apply Preset (to All CCs)**.

7. Select the **Settings** side tab and select **Optimize EVM**.

8. Select **Sweep** > **Sweep/Meas to Single** > **Restart**.

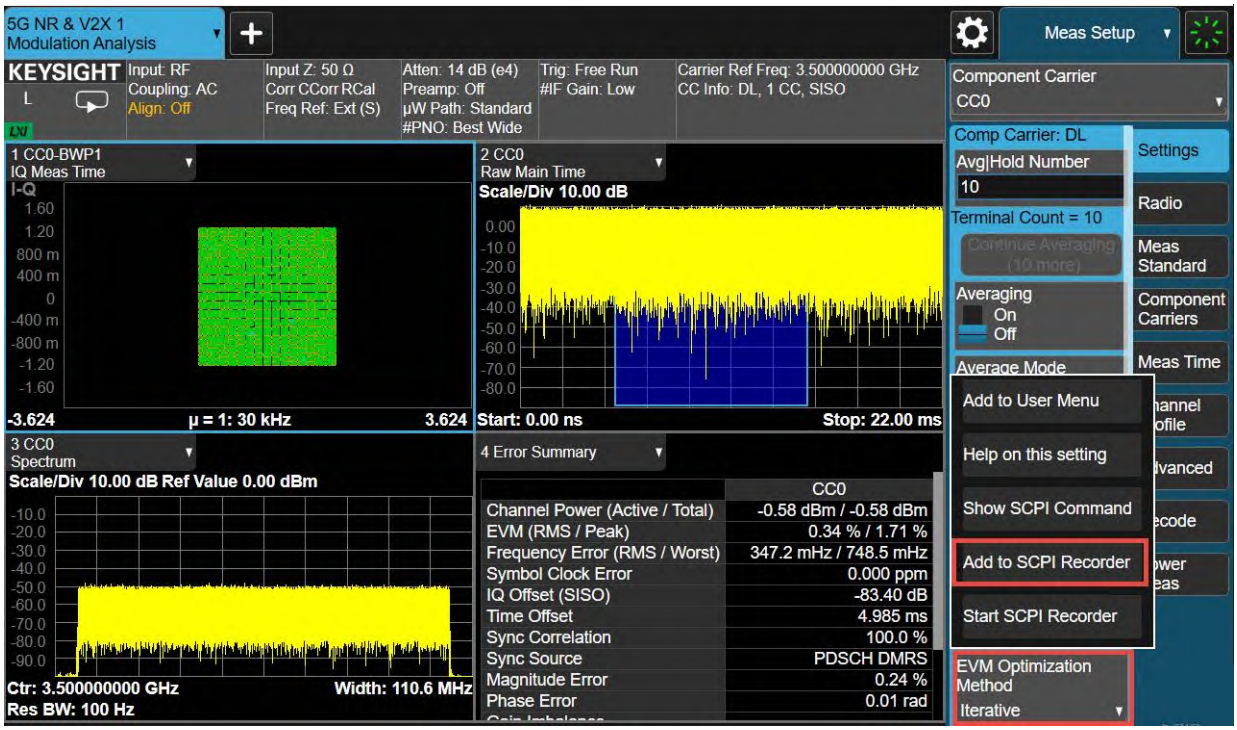
9. Select the **System Settings** icon (Gear wheel on top right of the display) > **SCPI Recorder** > **Continuous SCPI Recording Off**. Close the System Settings window by selecting the "X".



Manual Recording, Edit, Save and Play Functions

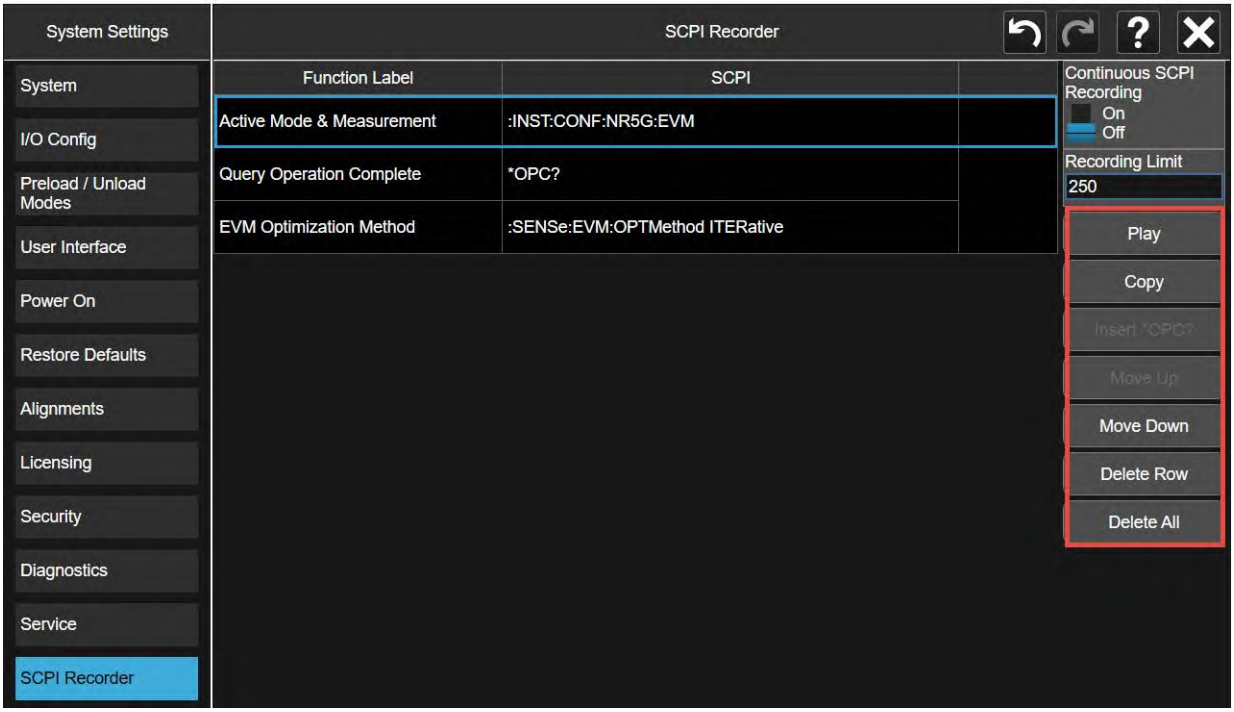
The SCPI Recorder supports manual recording of a list consisting of SCPI commands comprising any number of parameter variations when Continuous SCPI Recording is off. In this mode, the user can decide which SCPI commands should be added to the SCPI list. This mode of recording is helpful if a certain configuration must be figured out, and you only want to record the final, correct settings and not every variation or keystroke.

1. Touch and hold the any function for at least one second on the touch screen or if using a mouse pointer and right click to open the menu and select **Add to SCPI Recorder**. For this is example we will change EVM Optimization Method to Iterative.

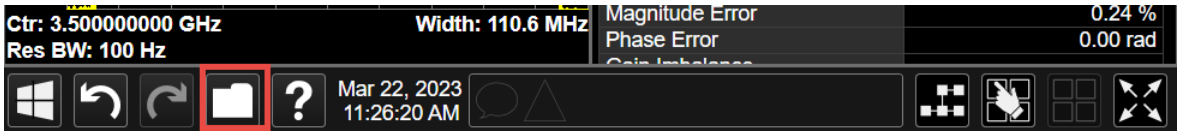


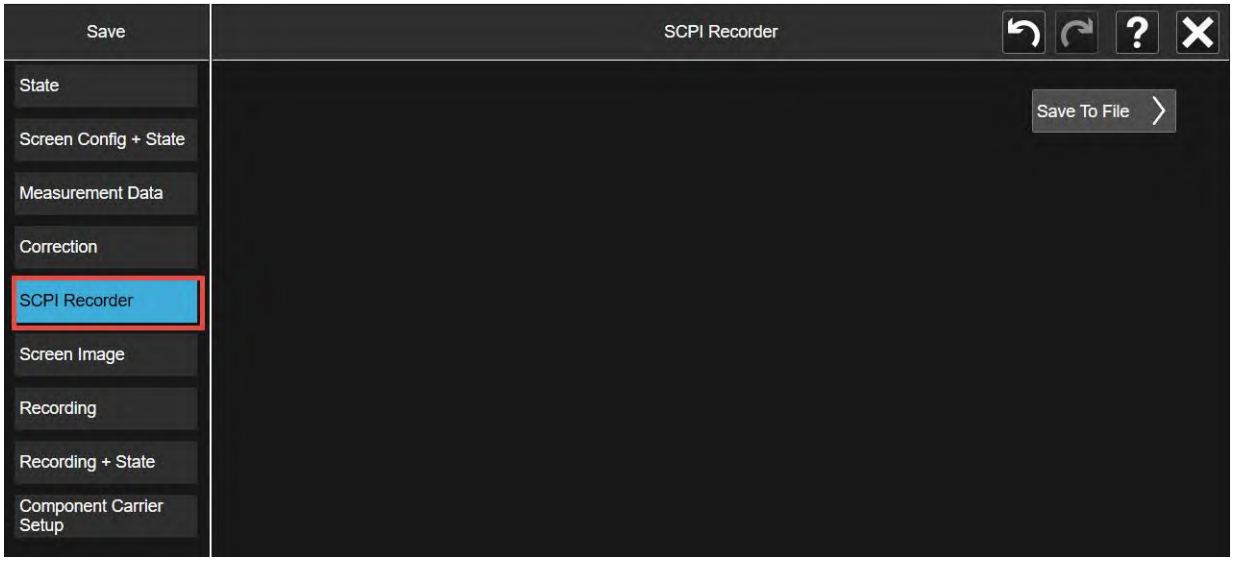
2. To view the newly added command, go to **System Settings** icon (Gear wheel on top right of the display) > **SCPI Recorder**.

From the SCPI Recorder provides several operations to edit the SCPI list you created. You can Play, Copy, Move Up or Down the selected SCPI command, Delete the command, or Delete All the commands.



The recorded SCPI list can be exported as a script file in .TXT format via the Save/Recall function at the bottom of the main window.

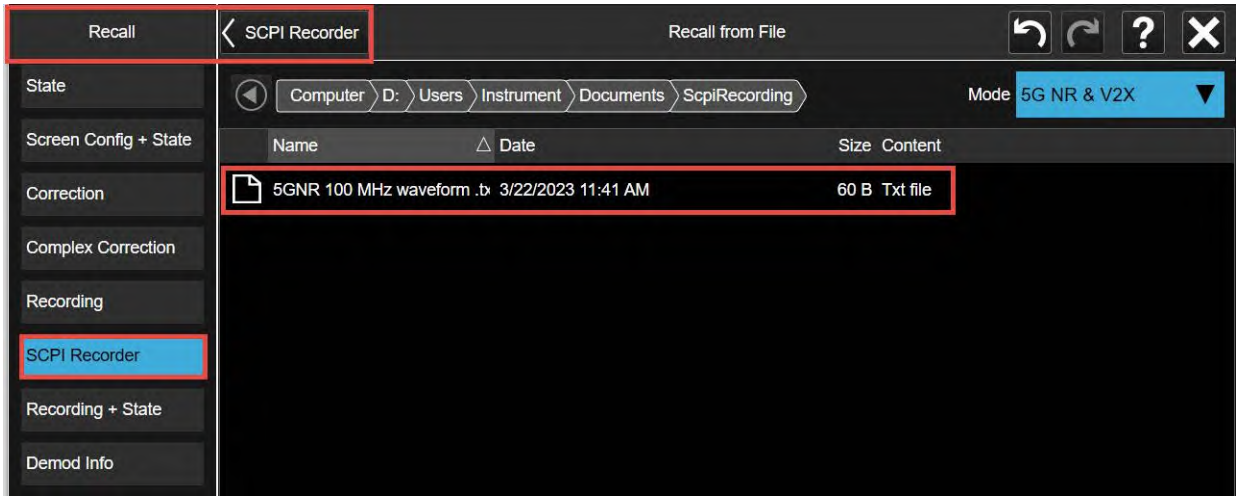




There are two ways to playback SCPI commands. One is recalling from a script file. And another one is pressing the 'Play' button in 'SCPI Recorder' tab view.

Recalling SCPI script file allows you to play back a series of operations in the same or another instrument, which helps you setup complex measurement scenarios quickly and easily.

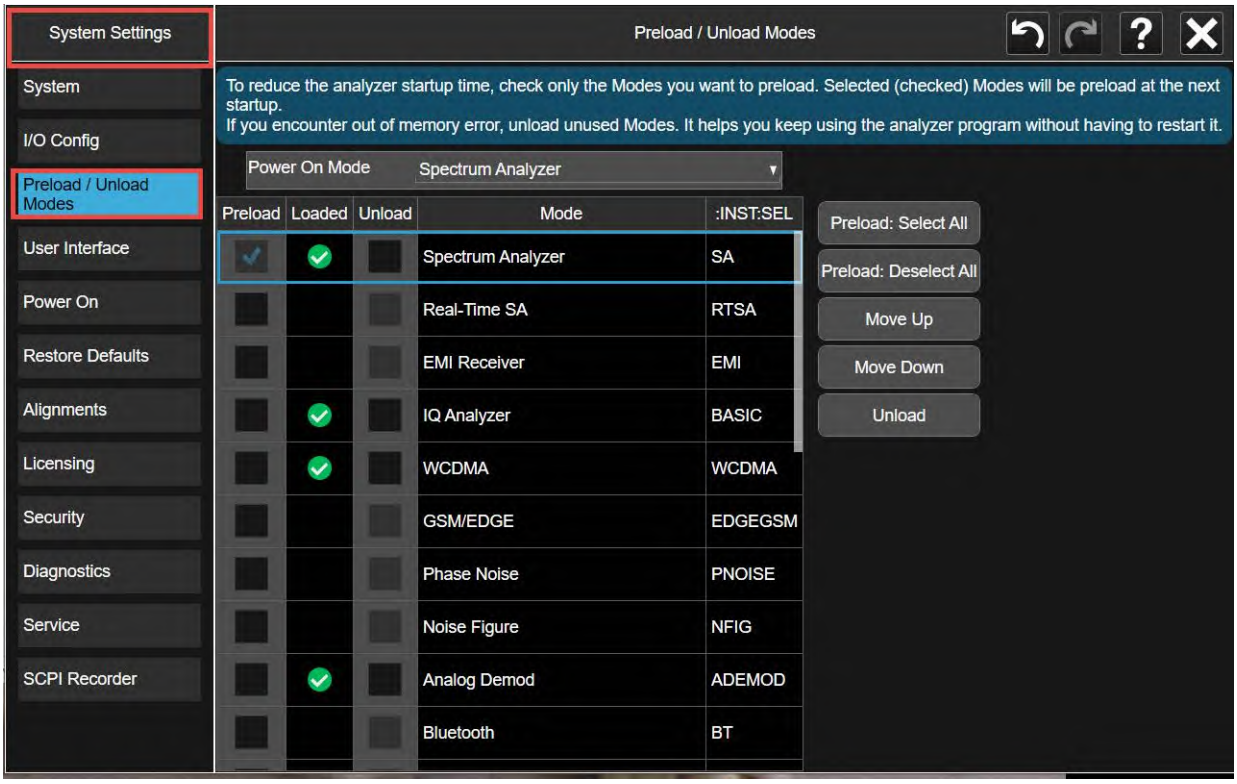
To recall the SCPI recording, go to Save/Recall and navigate to the saved file.



Using the X-Series Analyzer's Preload/Unload Function

The Preload/Unload function allows you to flexibly configure which applications should be preloaded when the software starts up, or also unload any previously loaded applications. This can help to save startup time or system memory as needed.

1. Select the **System Settings** icon (Gear wheel on top right of the display) > **Preload/Unload**.



- **Power On Mode** displays a list of licensed Applications. Use this control to change the factory default Power-On Mode. The instrument will execute the selected Application after power up. Selecting the Power-On Mode here automatically enables that Mode for preloading.
- **Select All, Deselect All** toggles the Preload checkbox state for all applications listed, except for the Power-On application.
- **Move Up, Move Down** allows you to reorder the listed applications in the table. This is the order in which they are displayed in the Mode/Measurement/View Selector dialog.
- **Unload Mode** unloads the specified mode.
- **Loaded Modes** returns a list of loaded modes.

2. Set RF Out to **On** by selecting the numbered channel indicator switch.

   This enables the RF Out for the indicated channels, in this case Channel 1 if using a multi-channel instrument.

Using Real-time Spectrogram Streaming (Option RRT) for N9032B and N9042B

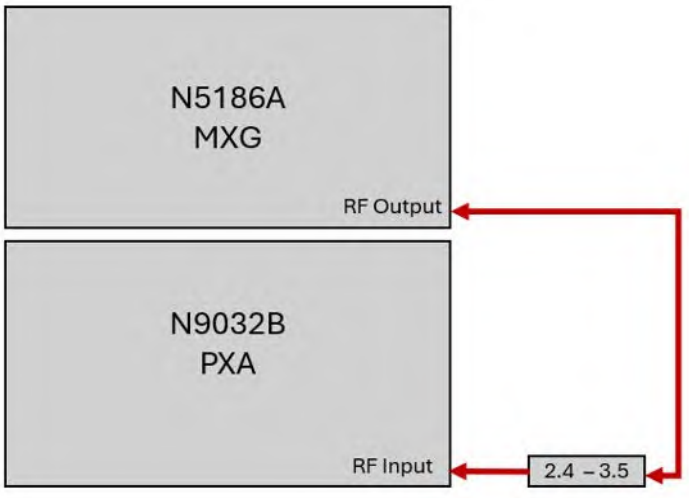
Overview

Real-time Spectrogram Streaming combines the high-speed capture capabilities of RTSA equipped analyzers with the convenient flexibility of post-processing, allowing you to set their desired measurement parameters such as: center frequency, span, and acquisition time and then stream the resulting gapless spectrogram data to binary files. This improves the storage capacity of captured slices by 8x. The saved binary files can be used for customized post-processing. A MATLAB program (RTSA viewer) is provided to introduce users to post process analysis of the saved binary files. In this procedure you will learn how to set up Real-time Spectrogram Streaming and how to use the Rtsa viewer to view your captured spectrogram slices.

Equipment:

- N5186A MXG or M9484C VXG
- N9032B PXA or N9042B UXA with Option RRT
- 3.5 mm female to female cable
- For a mmWave signal analyzer, 2.4 mm female to 3.5 mm male adaptor

  For a μWave signal analyzer, Type N male to 3.5 mm female



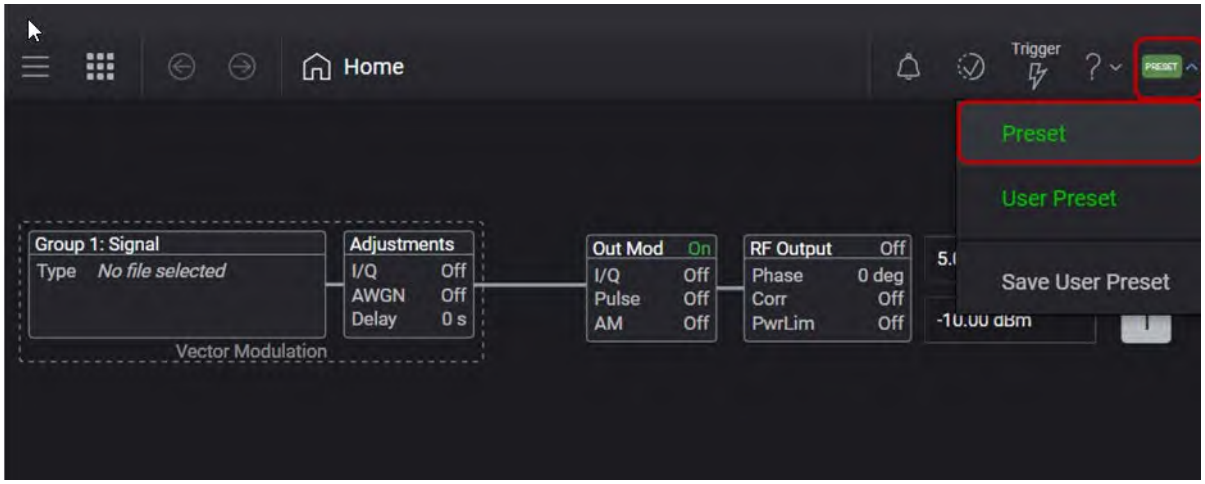
Using the 3.5 mm cable and the 3.5 mm to 2.4 mm (or Type N) adaptor, connect the MXG/VXG RF output to the RF input of the PXA/UXA.

On the MXG/VXG Signal Generator:

Start by generating a sweeping frequency modulated signal for its distinctive formation in the spectrogram screen. Note: This is purely an example for simplicity and demonstrative purposes.

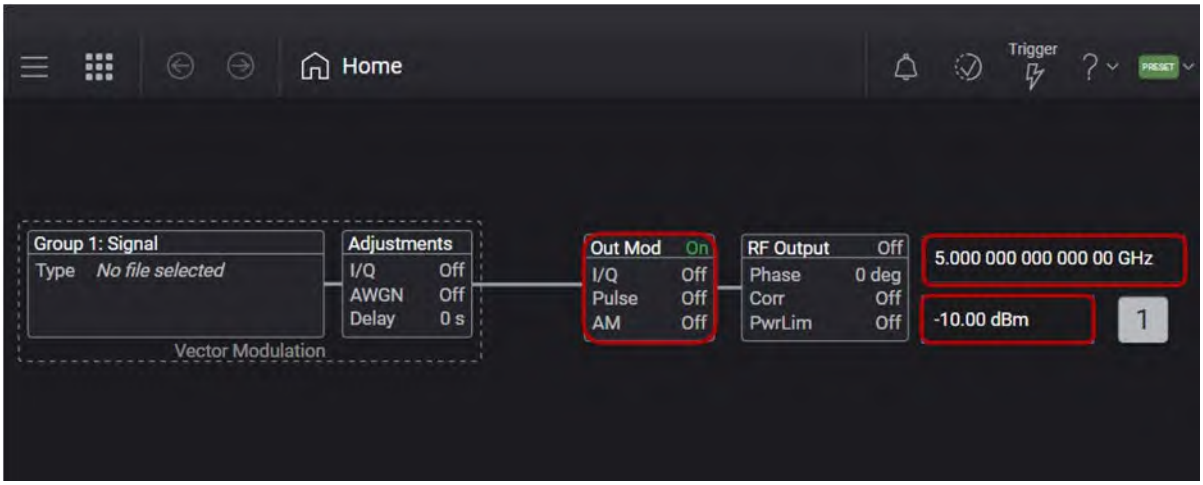
NOTE This is purely an example for simplicity and demonstrative purposes. This process can be used for other signals such as 5G NR and WLAN.

1. Select **Preset** > **Preset** to set the instrument to a known state.



2. Set Frequency to **5 GHz** and Power to **-10 dBm**.

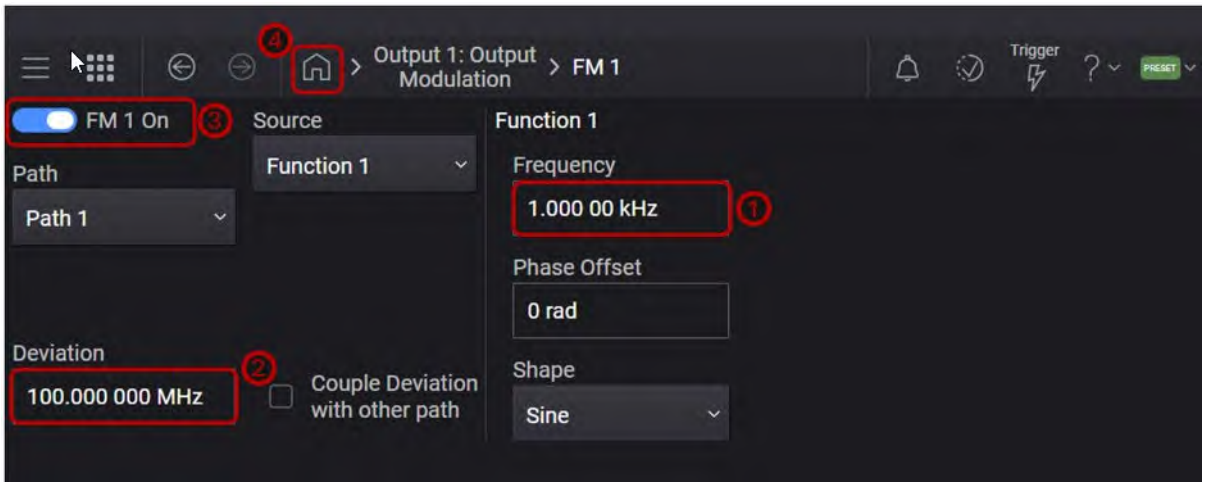
   These values are coupled to CW Frequency and Total Power (RMS) in the corresponding RF Output Block.



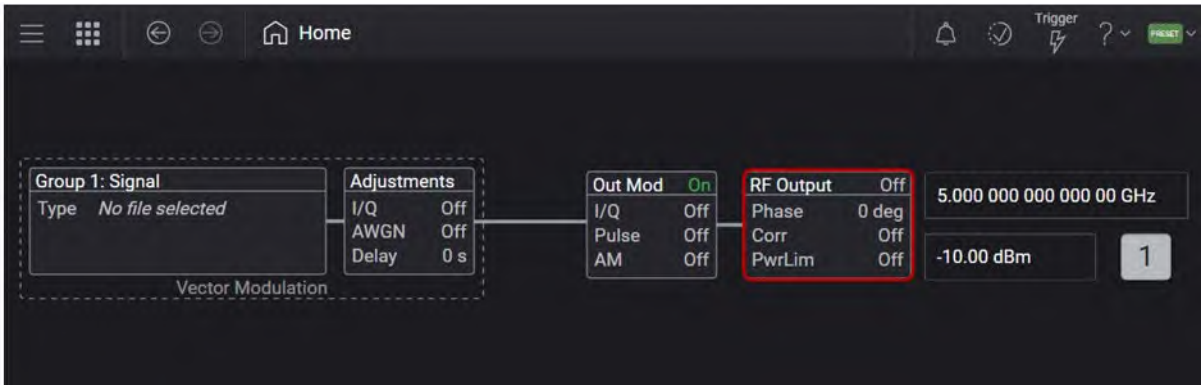
3. Select the **Out Mod** block to open.

4. In the Output Mode window, select the right arrow at the bottom of the display to select choices **5-8**, then select the **FM 1** arrow at the far left side of the window.

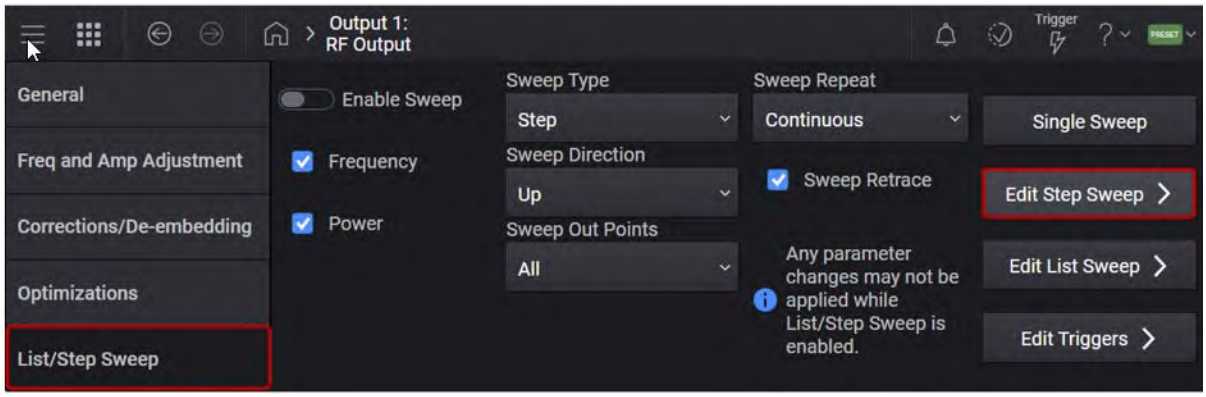
5. In the Output Modulation FM 1 window, set:
   - Frequency to **1 kHz**
   - Deviation to **100 MHz**
   - Toggle **FM 1 on**
   - select the **Home** icon at the top of the screen



6. Select the **RF Output** block.

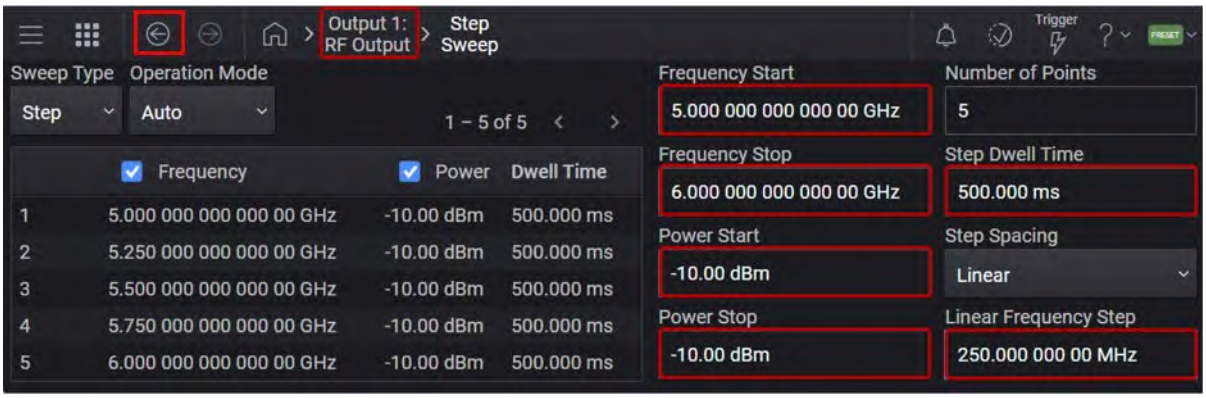


7. In the RF Output block, set:
   - **List/Step Sweep** tab
   - **Edit Step Sweep**

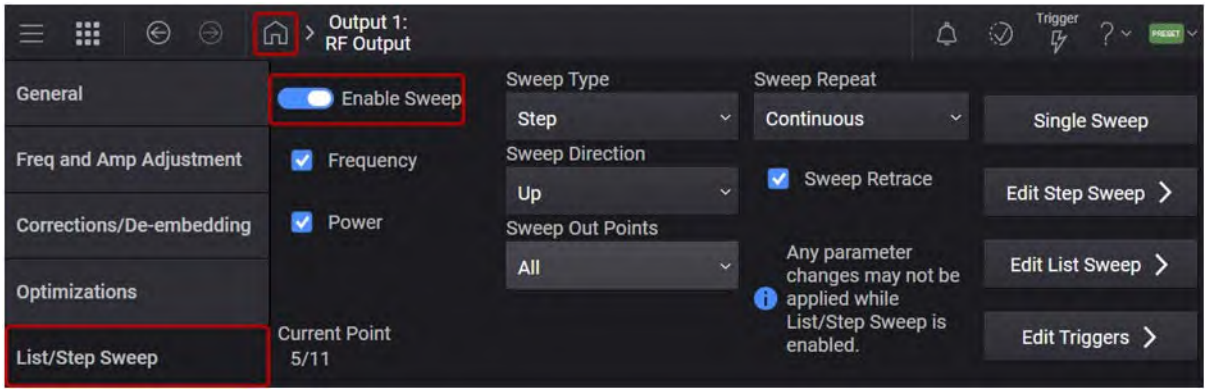


8. In the Step Sweep window, set:
   - Frequency Start to **5 GHz**
   - Frequency Stop to **6 GHz**
   - Power Start to **-10 dBm**
   - Step Dwell Time to **500 ms**
   - Linear Frequency Step to **250 MHz**

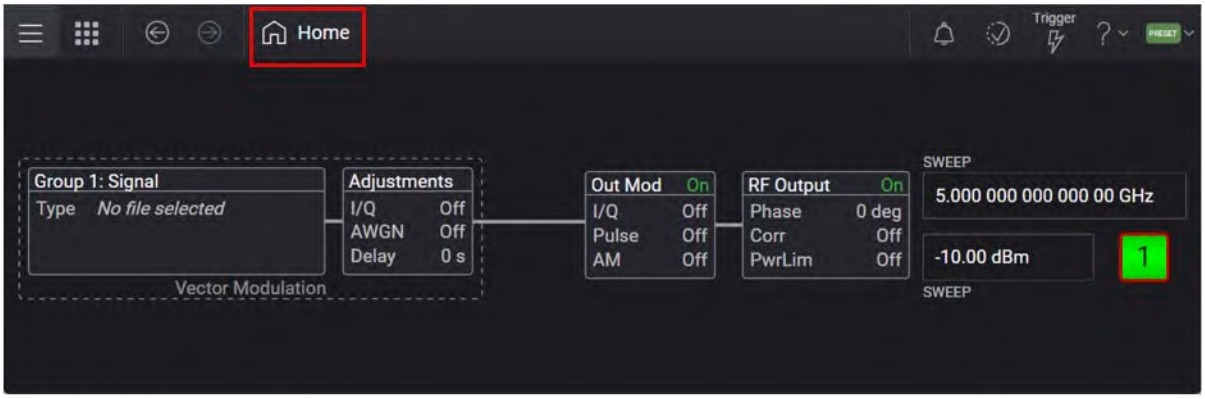
– select the **Back** arrow to return to Output 1: RF Output.



9. Select **List/Step Sweep** >toggle **Enable Sweep** to turn on.



10. Select the **Home** Icon and turn **RF Output On**.



On the PXA or UXA Signal Analyzer:

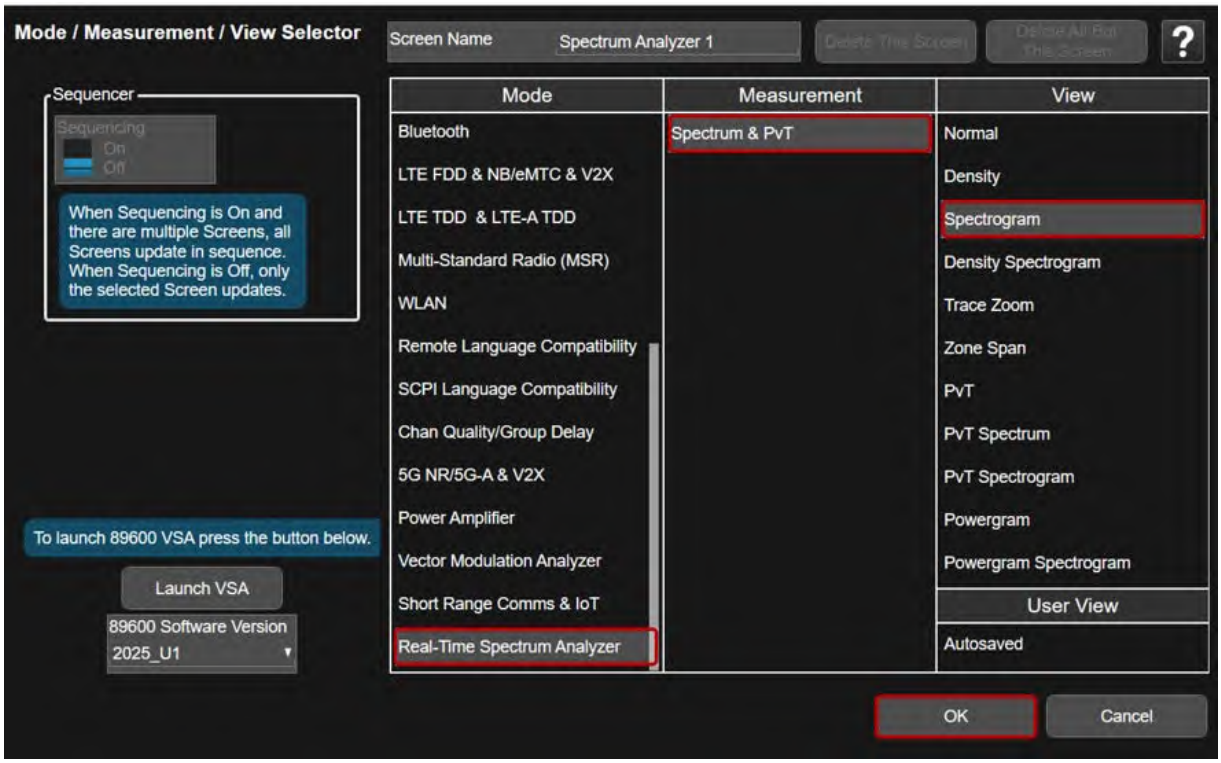
Now that the signal has been created, we can use RTSA with Option RRT to capture the signal into bin files.

1. Select **Mode/Meas** > **Real-Time Spectrum Analyzer** Mode > **Spectrum & PvT** Measurement.

NOTE

If accessing the instrument via a Remote Desktop connection, select the Screen tab (at the top of the display) to open the Mode/Measurement/View Selector window.

2. Select **Mode Preset** to set Real-time Spectrum Analyzer mode to a known state.
3. Select **Mode/Meas** > **Real-Time Spectrum Analyzer** Mode > **Spectrum & PvT** Measurement > **Spectrogram** View.

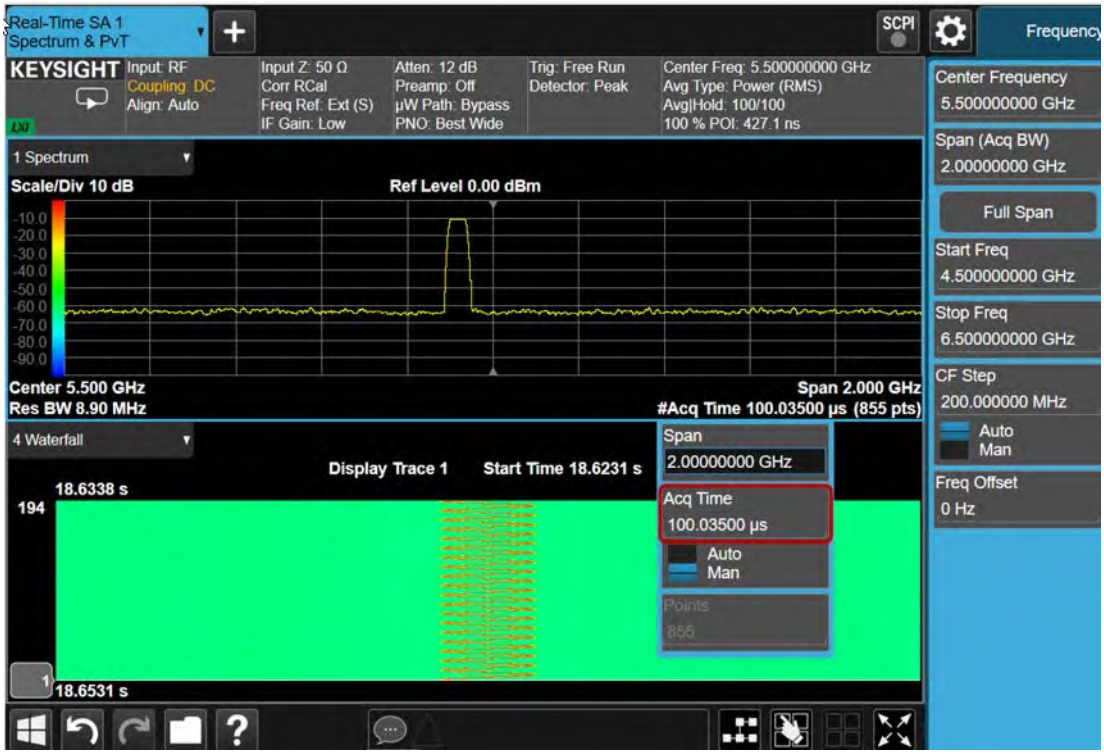


4. Select **Frequency** and set Center Frequency to **5.5 GHz** and Span to **2 GHz**.

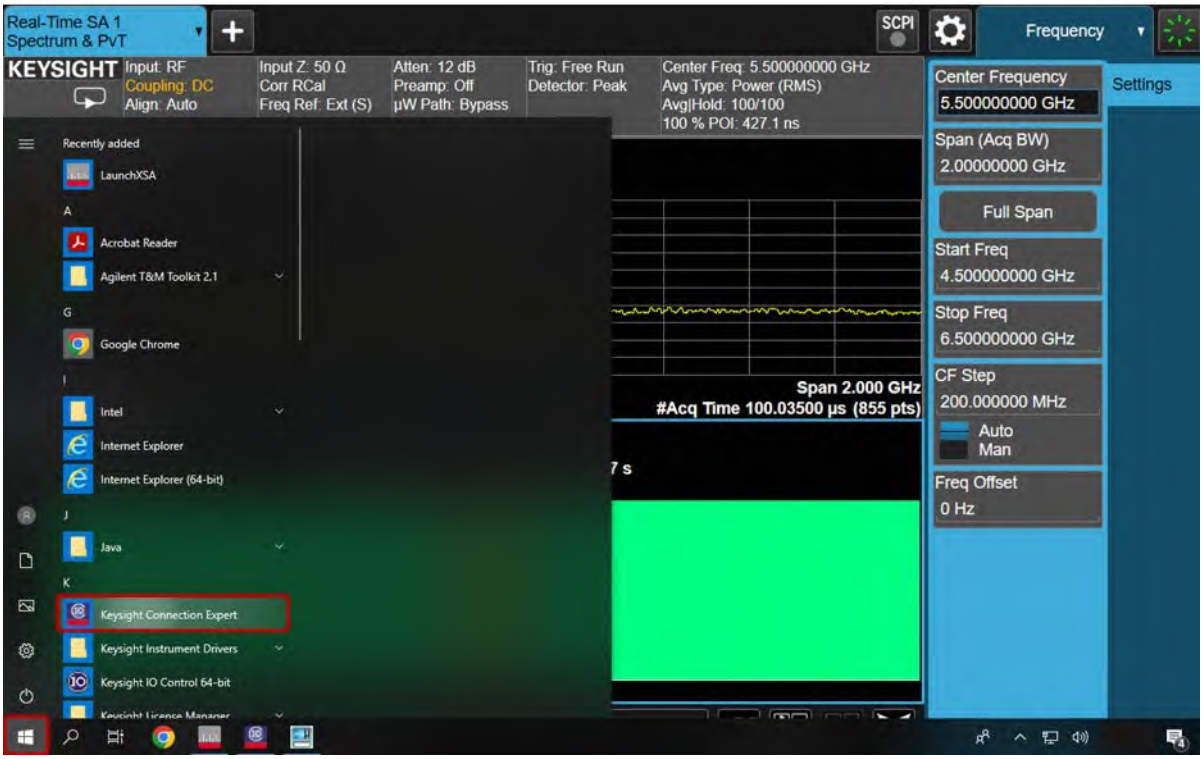
Notice the acquisition time is preset to 30 ms and how changing the acquisition time affects the spectrogram.

5. On the lower right area of the Spectrum window, select **Acq Time** and change to **100µs**.

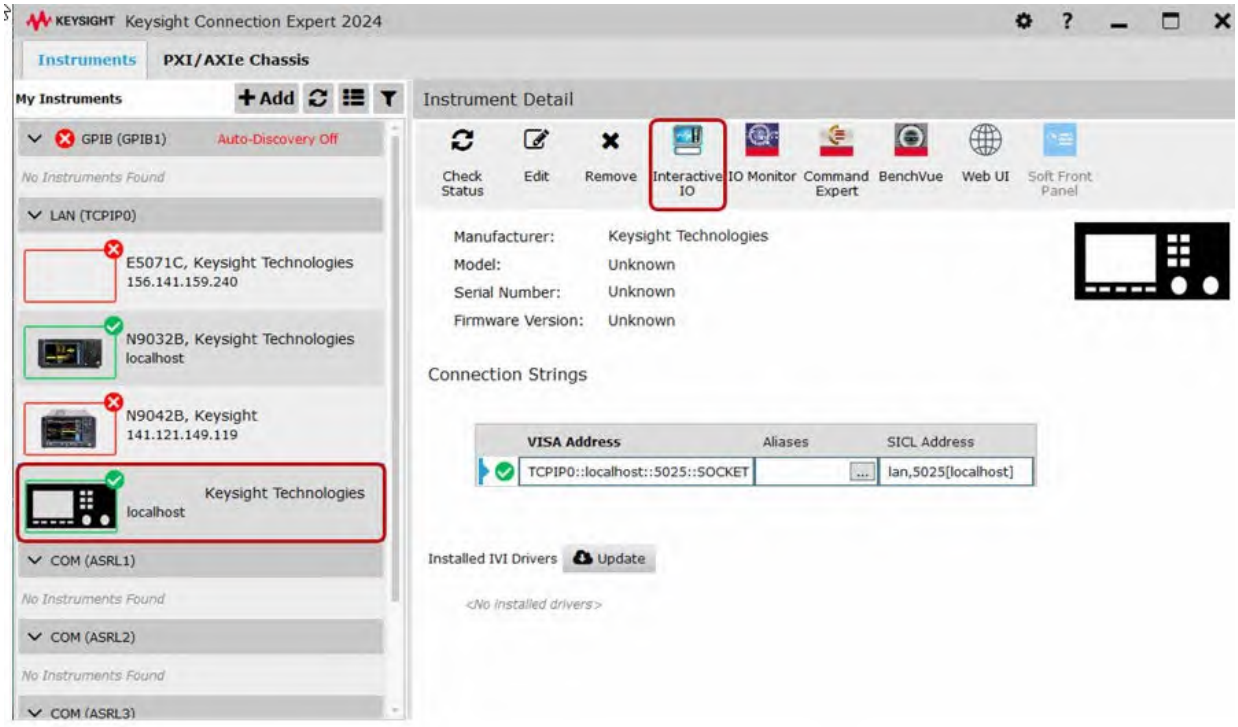
Notice how the FM signal is now resolved.



6. At the bottom of the display, select the **Windows** icon > **Keysight Connection Expert**.



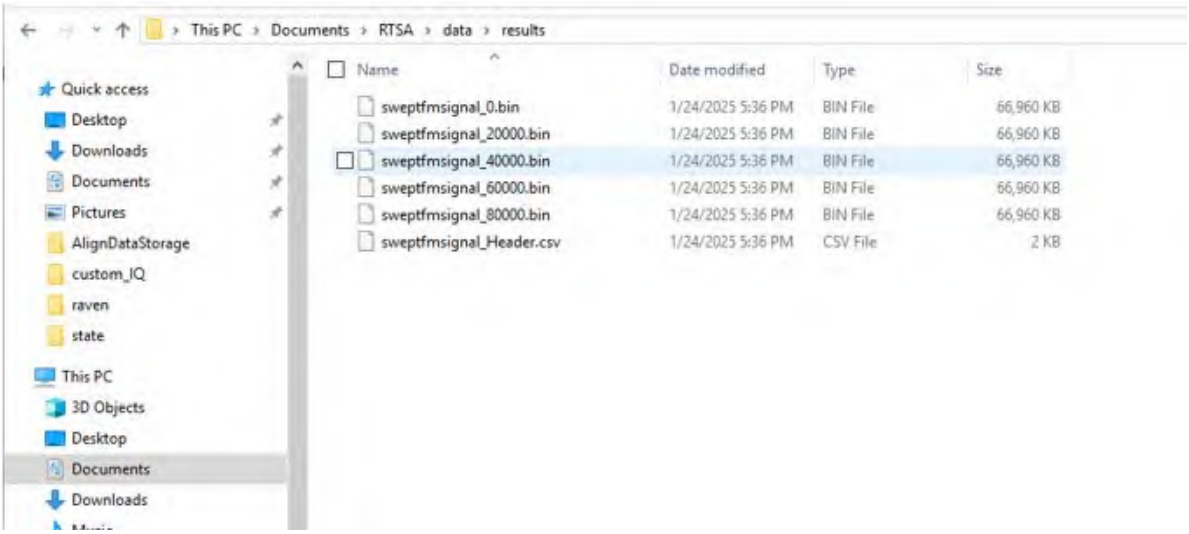
7. In the Keysight Connection Expert window's left pane, select **Keysight Technologies localhost** and then **Interactive IO**.



8. In the **Command** area, type in the following commands:
   - `STR:TYPE FINite` > **Send Command**
   - `STR:TYPE?` > **Send Command**
   - `STR:LENGth 100,000` > **Send Command**
   - `STR:LENGth?` > **Send Command**
   - `STR:FNAMe "sweptfmsignal"` > **Send Command**
   - `STR:FNAMe?` > **Send Command**
   - `STR:STARt` > **Send Command**

   Once the Start command is sent, the analyzer will execute the number of iterations and store the resulting bin file in Documents > RTSA > data > results.

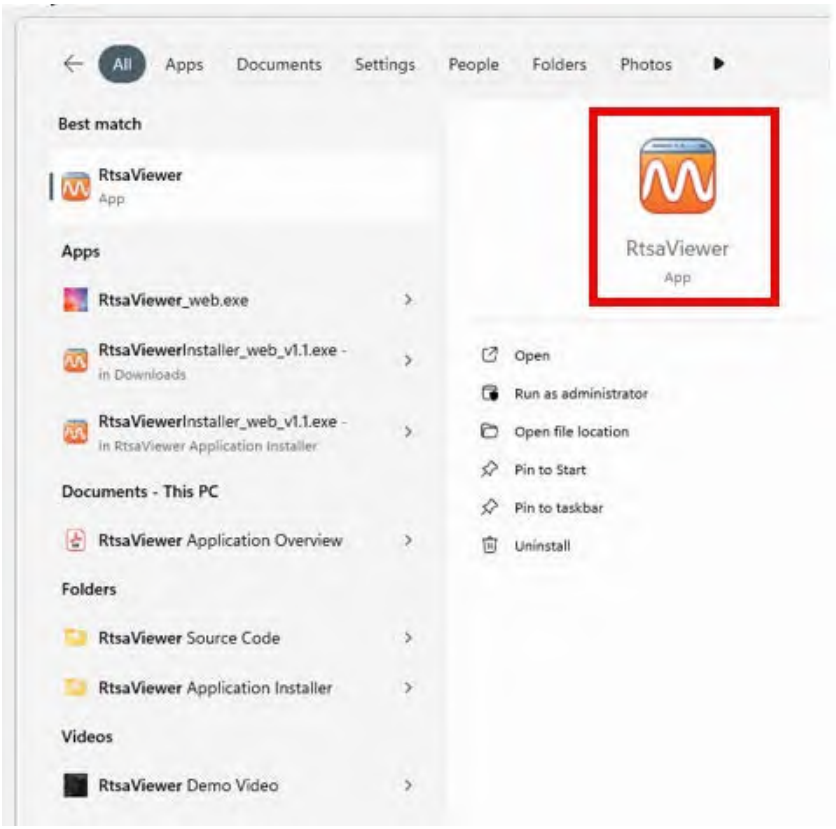
9. From the bottom of the X-series app, select **Windows File Explorer** and navigate to **Documents** > **RTSA** > **data** >**results**.



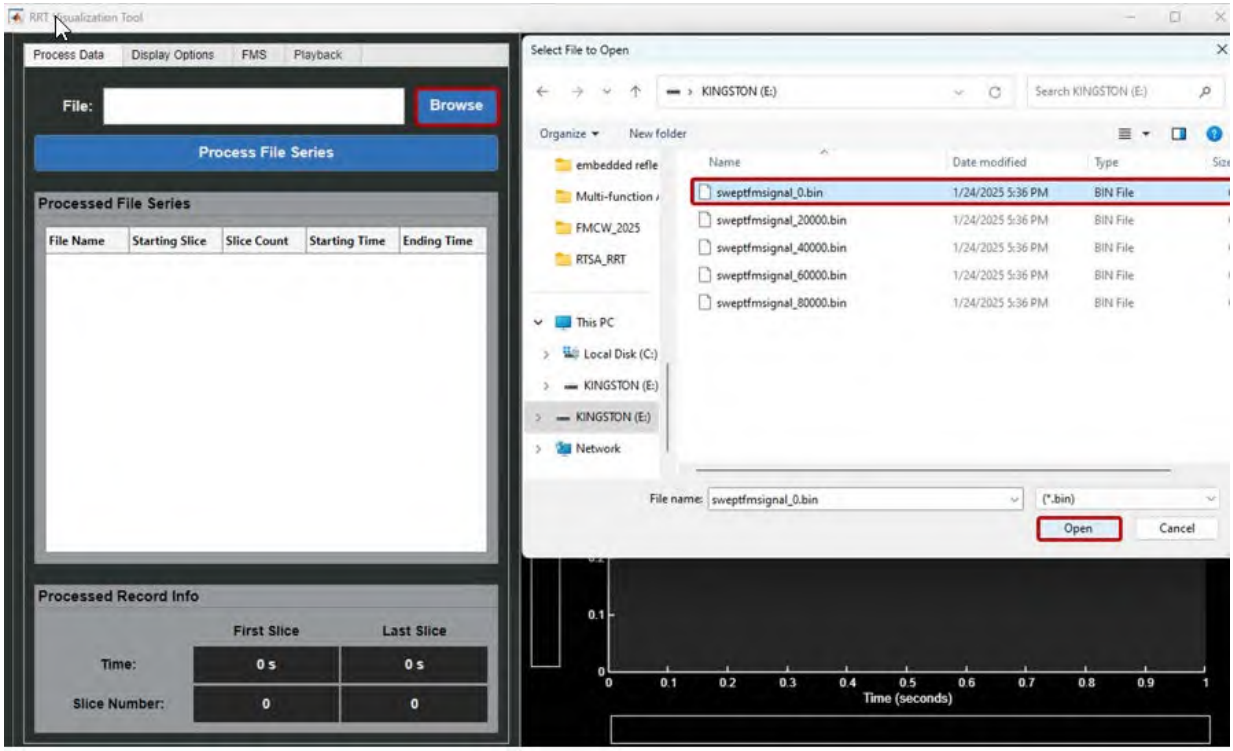
These bin files can now be uploaded to the RtsaViewer for processing and analysis. You can open RtsaViewer locally on the analyzer or transfer the files to another PC equipped with the RtsaViewer.

Using the Rtsa Viewer

1. In the Windows search bar, type `RtsaViewer` and **Open** the application.



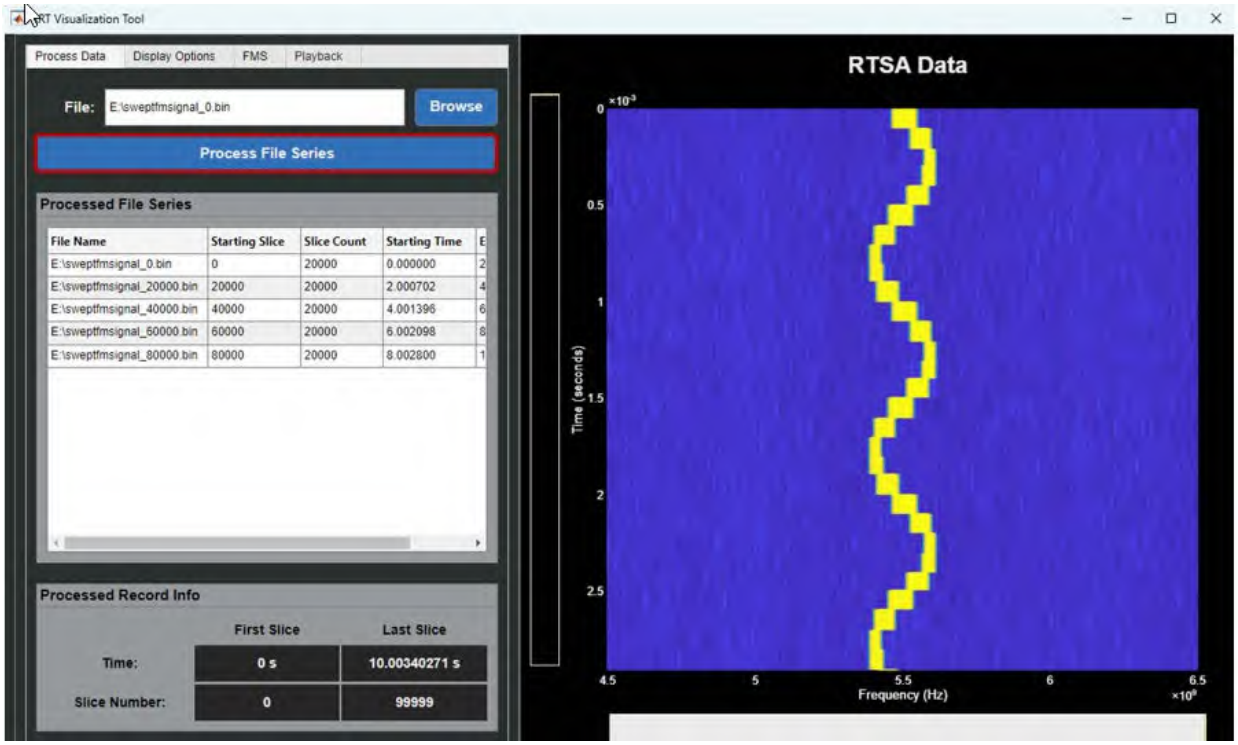
2. In the RtsaViewer, select **Browse** and navigate to your bin files. (Documents > RTSA > data >results). Select `sweptfmsignal_0.bin` > **Open**.



NOTE

The RRT Visualization Tool window may be hidden by the X-apps display. If you cannot see the visualization tool, select **Alt** > **Tab** to position it in front of the X-apps display.

3. Select **Process File Series** to generate the file table.



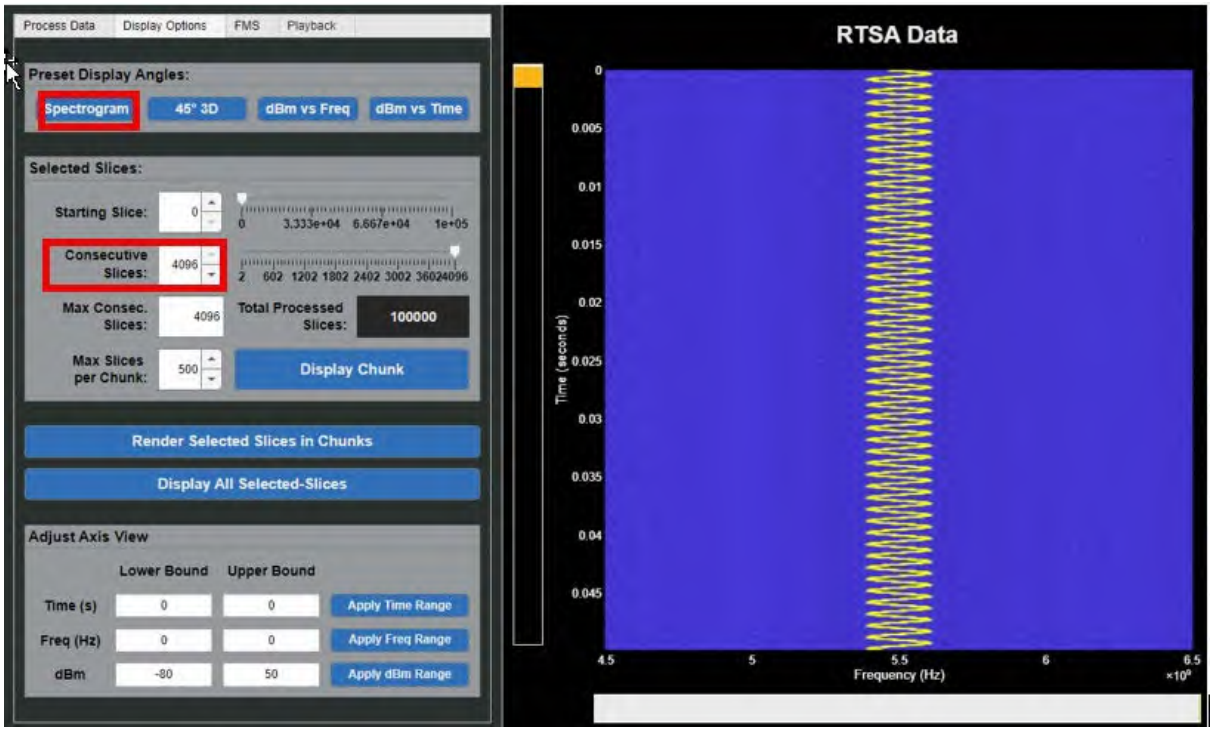
4. Select **Display Options** and set Consecutive Slices to **4096** to view the sames displays as shown in the pictures below.

Display Option Settings

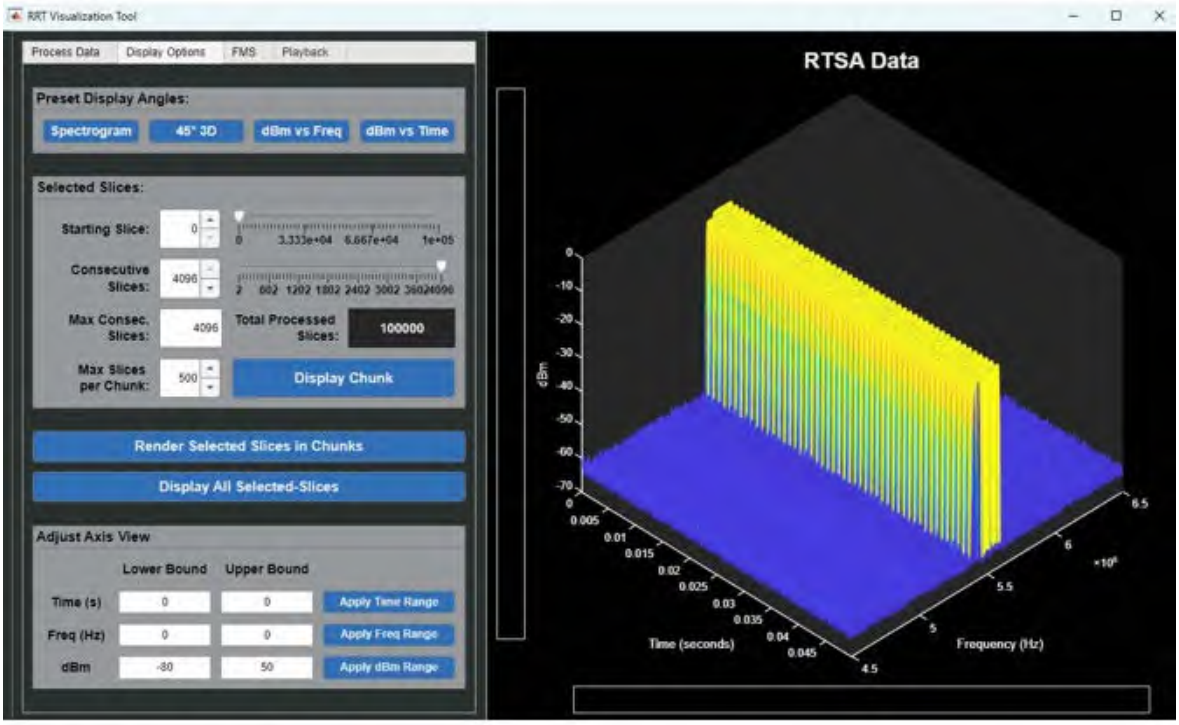
The **Display Options** tab contains settings to customize the display.

Preset Display Angles Area

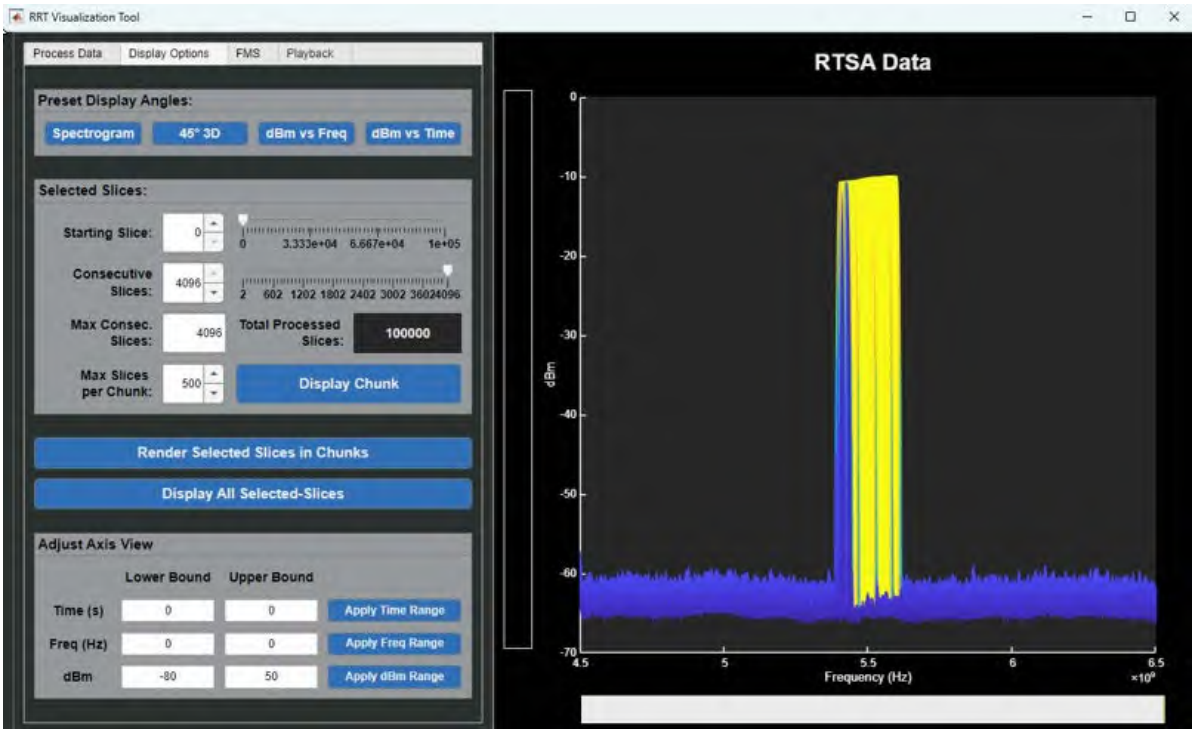
– **Spectrogram View** - plots the selected slices with axes of time in seconds and frequency in Hz.



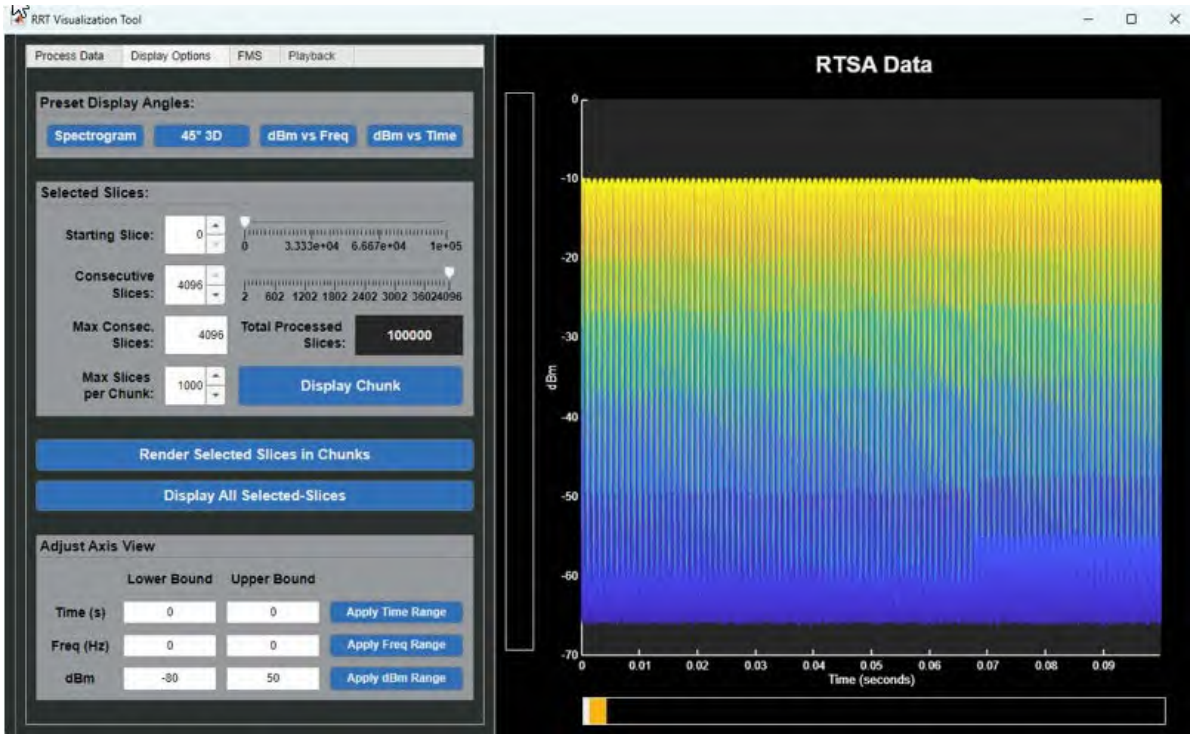
– **45° 3D View** - plots the selected slices on a 45-degree rotated, three-dimensional plot with axes of power in dBm, time in seconds, and frequency in Hz.



– **dBm vs Freq View** - plots the selected slices with axes of power in dBm and frequency in Hz, showing the power distributed over the spectrum.

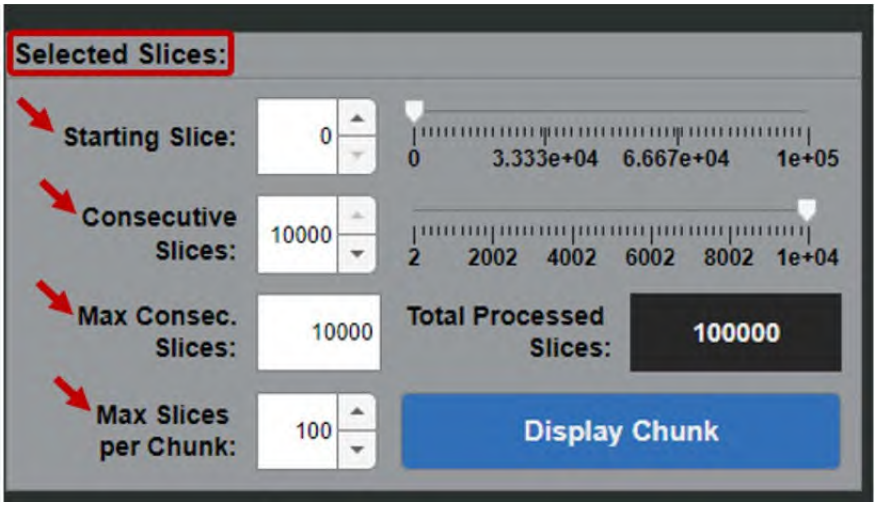


– **dBm vs Time View** - plots power in dBm over time in seconds.



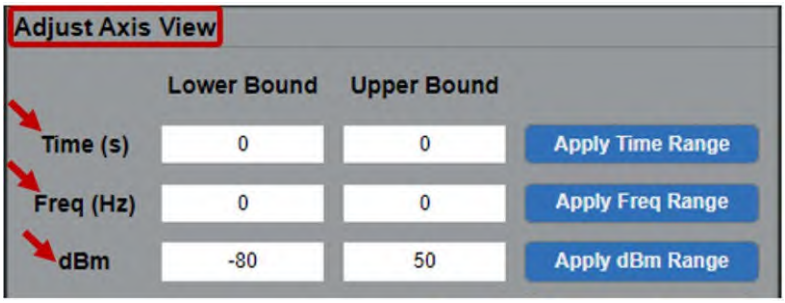
Selected Slices Area

The Selected Slices menu has adjustable settings including Starting Slice, Consecutive Slices, Max Consecutive Slices, and Max Slices per Chunk. Adjusting these settings allows for complete control of the signal slices you want to view.



Adjust Axis View

The Adjust Axis View can be used to alter the Axis of the viewable plot in time, frequency, and power without affecting the Selectable Slices settings.



FMS Tab

The FMS tab allows you to isolate signals by specifying corner bounds of an upper mask, then selecting **Run FMS**. Any signal considered inside the mask will be shown in the **Frequency Mask "Search" hits** table. While the upper mask includes a default of 4 bounds, mask bounds may be added or removed by right clicking the Upper Mask Settings table.

To render and display specific events, highlight the FMS hit you wish to see plotted. The example program's display will update to render the capture portion including the signal that triggered the mask (not the isolated signal passing over the mask alone).

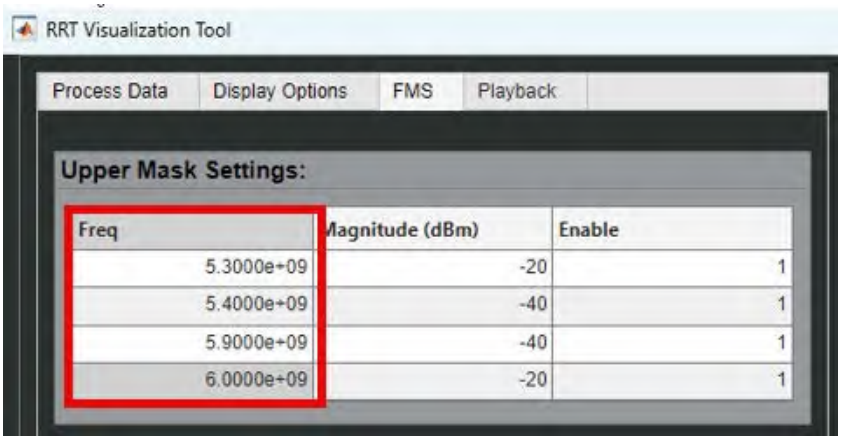
Additionally, at the bottom of the FMS tab, the Display of the Selected Event can be customized between three display options:

– **Beginning** shows the moment the signal entered the mask.

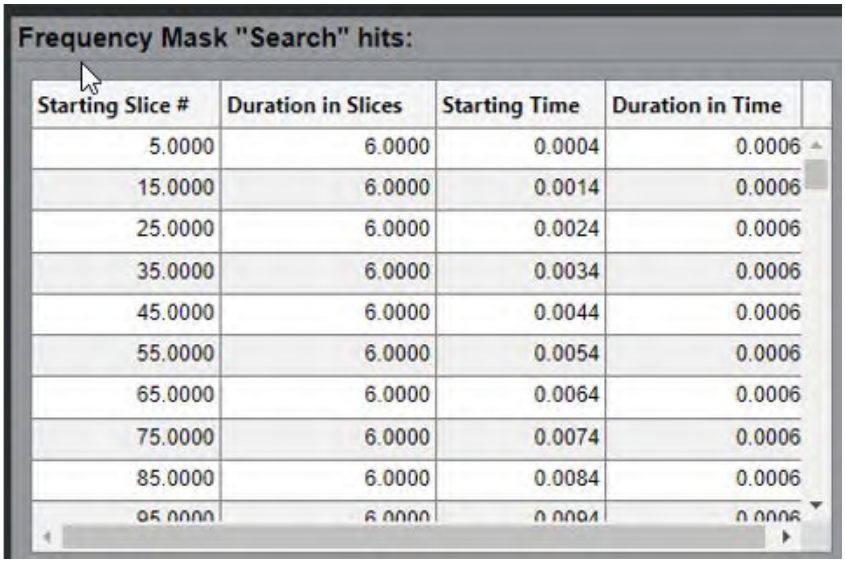
– **Ending** shows the moment the signal left the mask.

– **Entire** shows the full duration between when the signal entered and exited the upper mask

1. Set the Frequencies to **5.3 GHz**, **5.4 GHz**, **5.9 GHz**, and **6.0 GHz**.



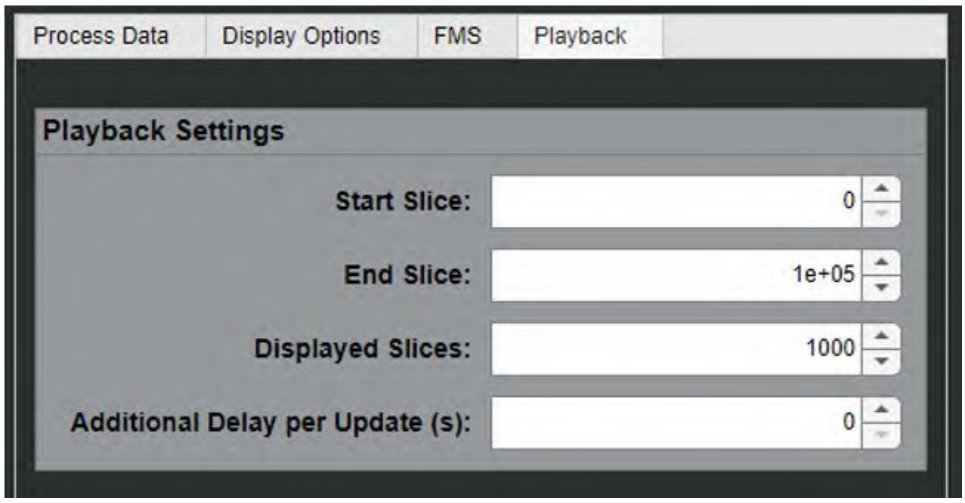
2. Check for signals that are found using the mask.



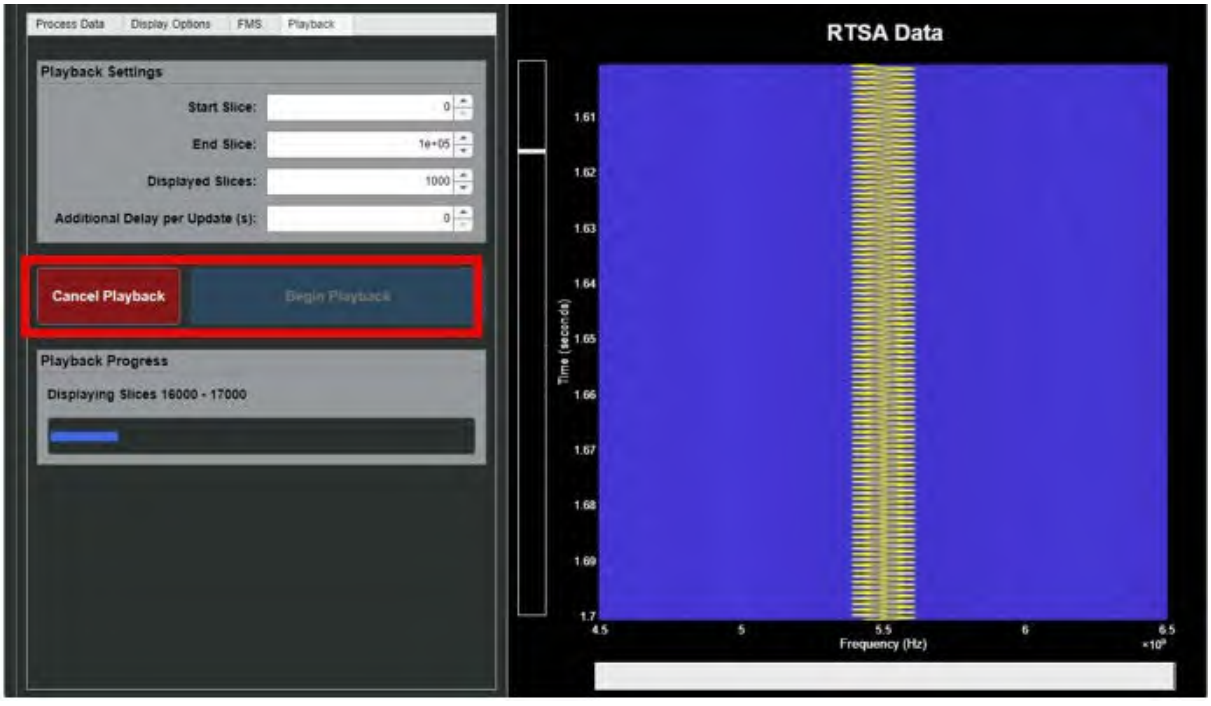
Playback Tab

The Playback tab provides you with a quicker method for visually parsing data. Given a specified start and end slice within the uploaded capture, displayed slices (at a time), and desired additional delay per update in seconds, the playback feature will scroll through the provided range, rendering and

displaying Displayed Slices on the axes at a given time. This screen will then show for some latency + the desired additional delay before updating to the next screen; unlike rendered displays generated through Display Options, rendering is cleared the moment it leaves the display.



To begin Playback, select the **Begin Playback**. Playback will continue until completion, regardless of the tab you may be on. If for any reason you wish to stop the playback, select **Cancel Playback**, which is only available during an active playback. You may monitor the playback's progress relative to your uploaded capture through the loading bar.



Try using the tabs and settings to get a feel for the RTSA Viewer application.

Conclusion

Option RRT allows you to record RTSA spectrogram slices for post capture processing. The option comes with an example program (RTSA viewer) that extracts the spectrogram data from the bin files that are created from recording. You can create your own applications or build off of RTSA viewer to perform in depth signal processing.

Additional Resources

Application note: Real-Time-Spectrogram-Streaming.pdf

5 Custom Modulation Measurements

This section includes the following topics:

– "Using PathWave E7608APPC to Create a Waveform File then Automatically Configure the Analyzer to View the Results" on page 500

Using PathWave E7608APPC to Create a Waveform File then Automatically Configure the Analyzer to View the Results

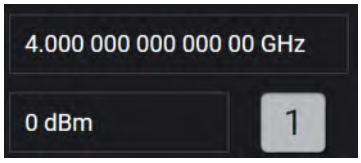
PathWave software tools can be used to create, download, and playback waveforms through the MXG.

This example shows you how to create and analyze a 16QAM using the embedded PathWave software and then automatically configure the analyzer to make the measurement.

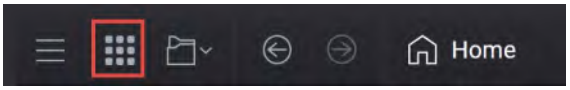
NOTE The MXG must have the E7608APPC Custom Modulation license installed and Signal Analyzer Version 2023 A.36.xx or later.

On the MXG:

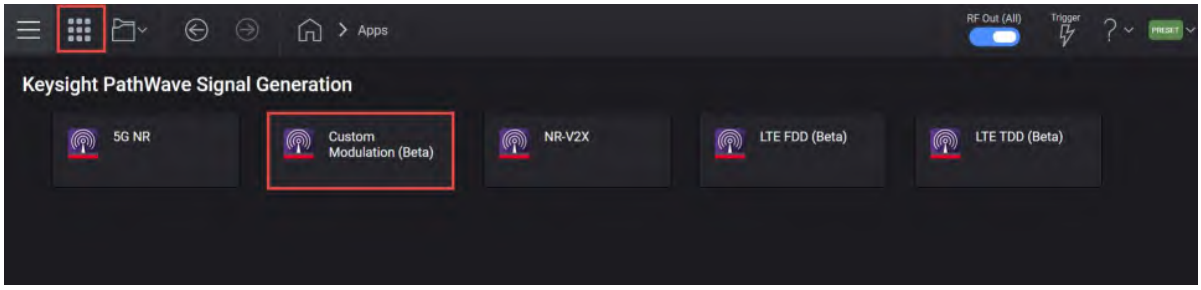
1. Select **Preset** > **Preset** to set the MXG to a known state.
2. In the Output area, set Frequency to **4 GHz** and Power to **0 dBm**.



3. Select the **Apps** block to open the mode selection panel.

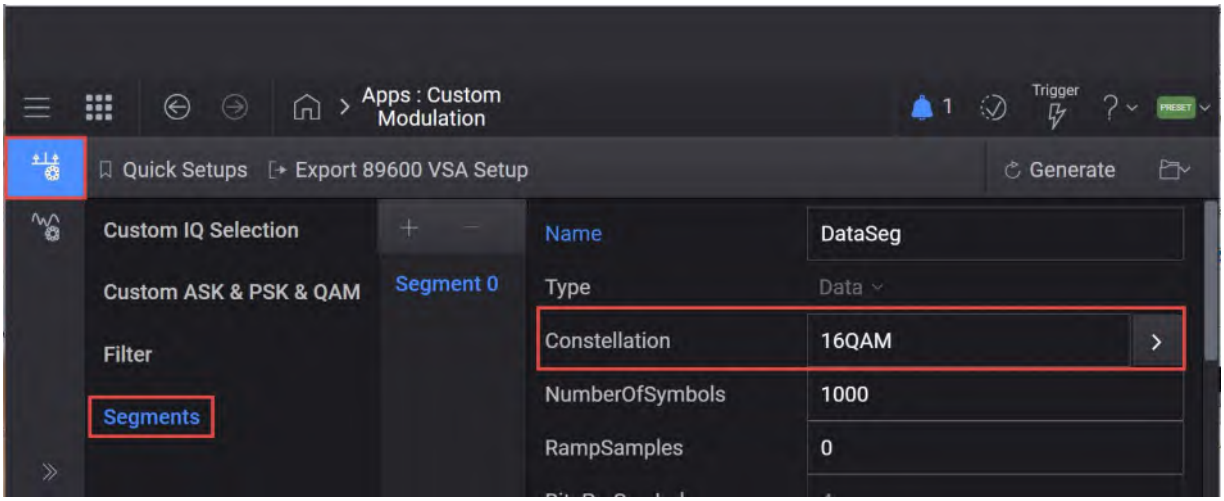


4. Select **Custom Modulation** to enter the custom modulation application.

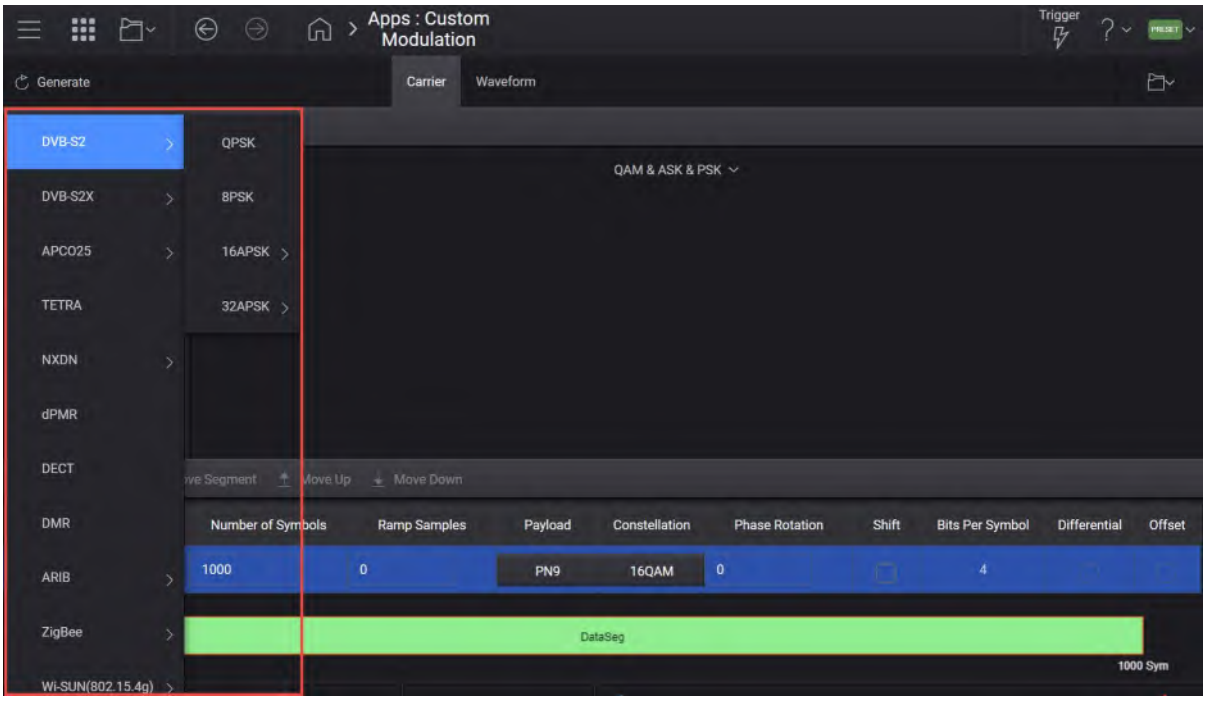


5. In the Custom Modulation setup, select the **Carrier** tab.

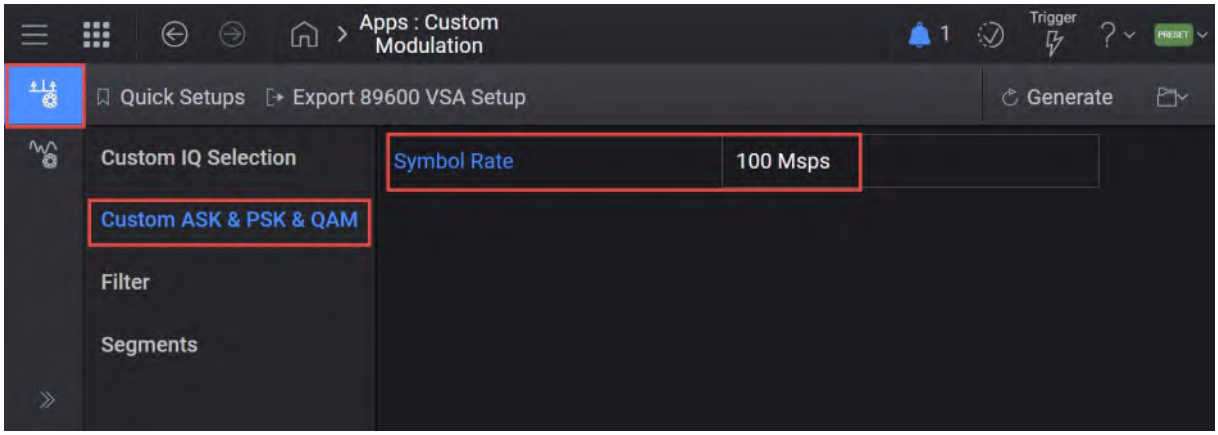
We will use the default 16QAM constellation for this example.



TIP Use Quick Setups for convenient saving and loading of common configurations. Quick Setups are factory supplied configurations and cannot be deleted by users.



6. In the left pane, select **Custom ASK & PSK & QAM** and set Symbol Rate to **100 Msps**.



7. Select the **Waveform** tab > **Generate**.

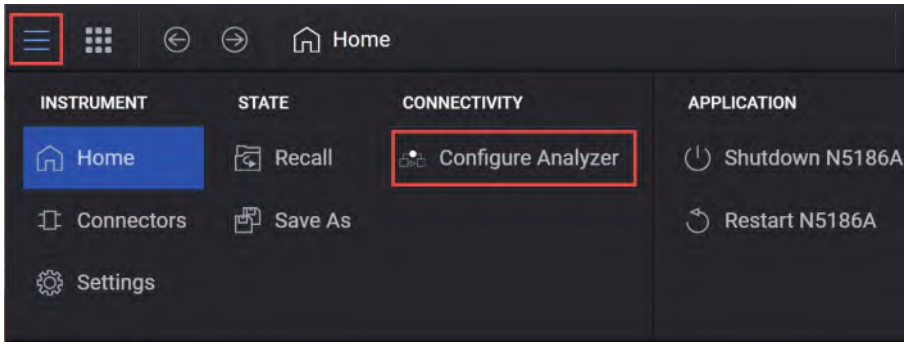
   Scroll down to the bottom half of the display and select **Spectrum** to view the Waveform.

8. Select the **Home** icon and set RF Out to **On** by selecting the numbered channel indicator switch.

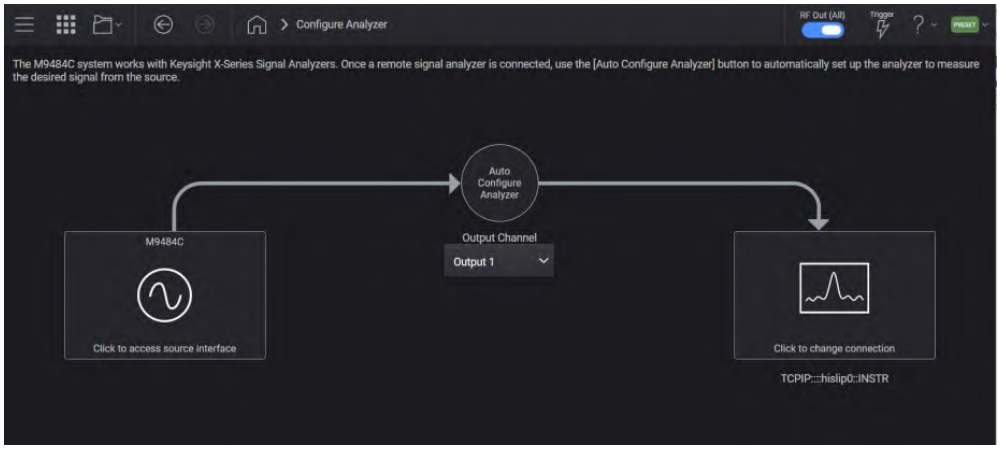
   This enables the RF Out for the indicated channels, in this case Channel 1 if using a multi-channel MXG.

Setup Auto Configure Analyzer

1. Select the **System** menu (triple bar tab at the top left of the window) and select **Configure Analyzer**.

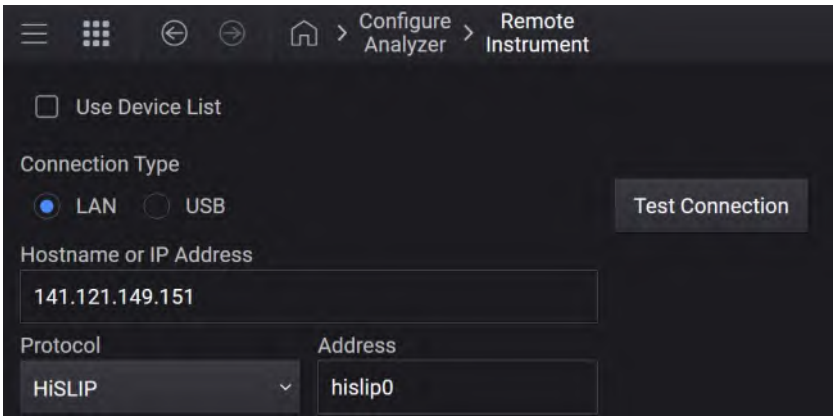


2. In the Configure Analyzer setup, click on the **Remote Signal Analyzer** block (on far right) to setup the communication channel to the Keysight X-Series Signal Analyzer.



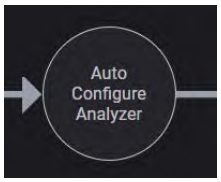
3. In the Remote Instrument window, **Clear** the Use Device List checkbox and enter the device information:
   - Set Connection Type to **LAN**
   - Enter the VISA Address of the analyzer.
   - Test Connection

– Select the Info icon (Bell shaped icon at top right of the display) to view the connection status.

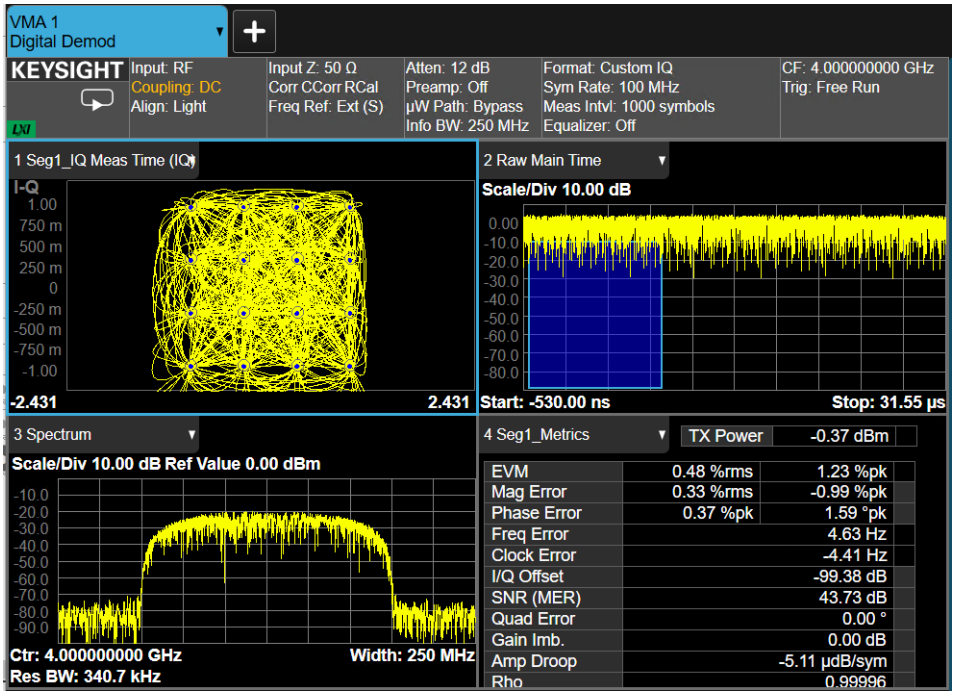


The Use Device List function allows you to set up more than one Signal Analyzer to be used for a channel's Auto Configuration.

4. Select the Back Arrow and select **Auto Configure Analyzer** to send the MXG settings to the analyzer.



5. View the results on the signal analyzer.



6 5G New Radio (NR) Measurements using X-Apps

This section includes the following topics:

- "5G Waveform, EVM, and ACP Analysis Using X-Applications" on page 508
  - "Setting Up Triggers on the Signal Analyzer using 5G NR Mode" on page 508
  - "Setting Up a 1 CC 28 GHz EVM Measurement" on page 510
  - "Setting up an 8 CC 28 GHz EVM Measurement" on page 517
  - "Setting Up a 1 CC 3.5 GHz ACP Measurement" on page 527
  - "Using PathWave N7631APPC to Create a Waveform File then Automatically Configure the Analyzer to View the Results" on page 532

5G Waveform, EVM, and ACP Analysis Using X-Applications

The VXG enables 5G testing with a low error vector magnitude (EVM) at high frequencies. The VXG has extremely good EVM at high power levels. However, not all signal analyzers can capture this low value. We will use the X-Series signal analyzer with the 5G NR X-Series application to observe EVM and adjacent channel power (ACP).

Setting Up Triggers on the Signal Analyzer using 5G NR Mode

NOTE Refer to "Equipment Setup" on page 13 for connecting the instruments and accessing the VXG SFP.

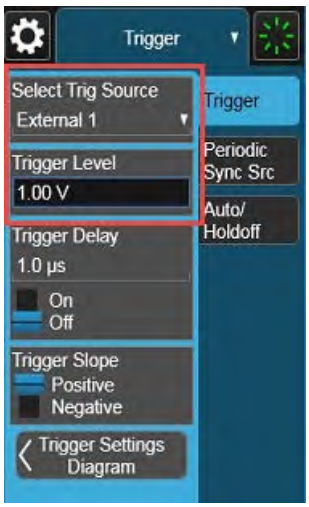
Setting Up Triggers on the X-Series Signal Analyzer

Using the graphical user interface

1. On the signal analyzer, select **Mode/Meas** > **5G NR & V2X** mode.

NOTE If accessing the instrument via a Remote Desktop connection, select the Screen tab (at the top of the display) to open the Mode/Measurement/View Selector window.

2. Select **Mode Preset** to set 5G NR mode to a known state.
3. Select **Mode/Meas** > **5G NR V2X** Mode > **Modulation Analysis** Measurement.
4. From the Menu Panel (on the top right of the display), select **Trigger** and set Select Trigger Source to **External 1** and Trigger Level to **1 V**.



Using the equivalent SCPI commands

On the X-Series Signal Analyzer:

```
INSTrument:CONFigure:NR5G

SYSTem:PRESet
```

Change the current window to 5G NR Modulation Analysis Measurement Mode

```
INSTrument:CONFigure:NR5G:EVM

TRIGger:EVM:SOURce EXTernal1

TRIGger:EXTernal1:LEVel 1V
```

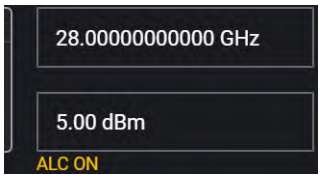
Setting Up a 1 CC 28 GHz EVM Measurement

NOTE

Ensure the equipment and triggers are properly configured. Refer to "Equipment Setup" on page 13 and "Setting Up Triggers on the Signal Analyzer using 5G NR Mode" on page 508.

On the VXG:

1. Select **Preset** > **Preset** to set the instrument to a known state.
2. In the Output 1 area, set Frequency to **28 GHz** and Power to **5 dBm**.

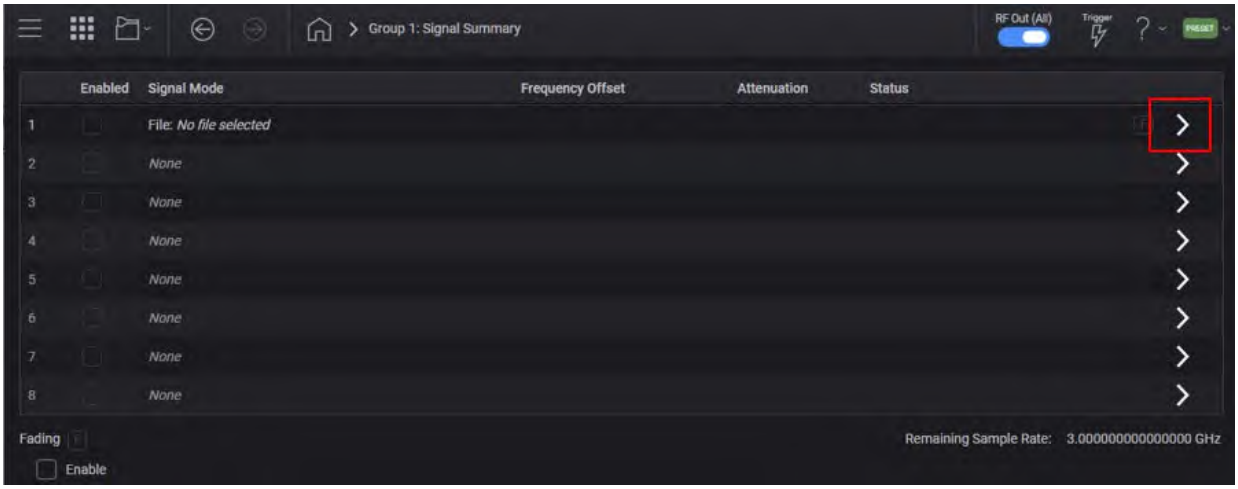


3. Select **Group 1: Signals** block to open.

4. Select the arrow for Signal 1 to open the Signal Setup window.

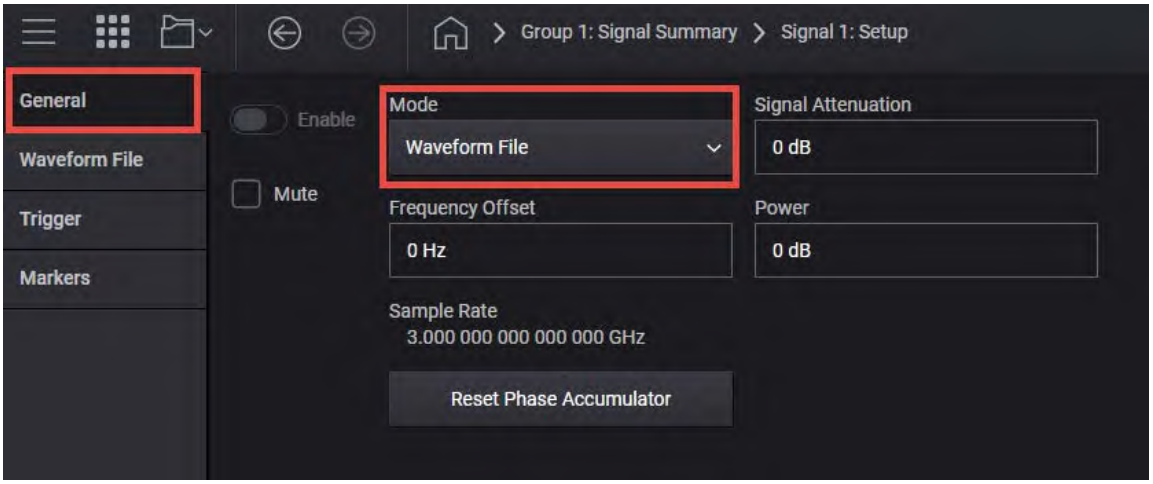
NOTE

This screen is only accessible if Option M9484C-8SG (8 virtual signal generators) is installed. For all other option configurations, continue to the next step.



5. In the Vector Modulation Signal Setup:

   a. Select the **Mode** dropdown and set to **Waveform File**.

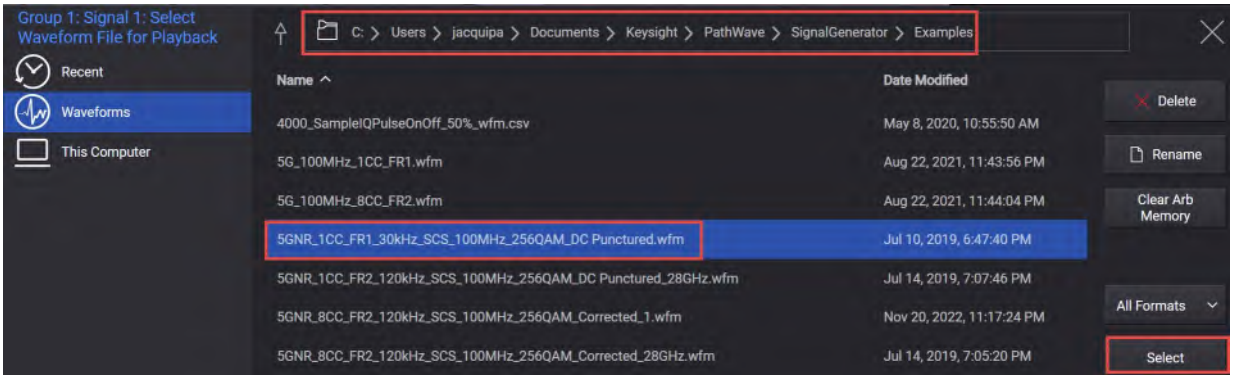


   b. In the left pane, select the **Waveform File** tab.

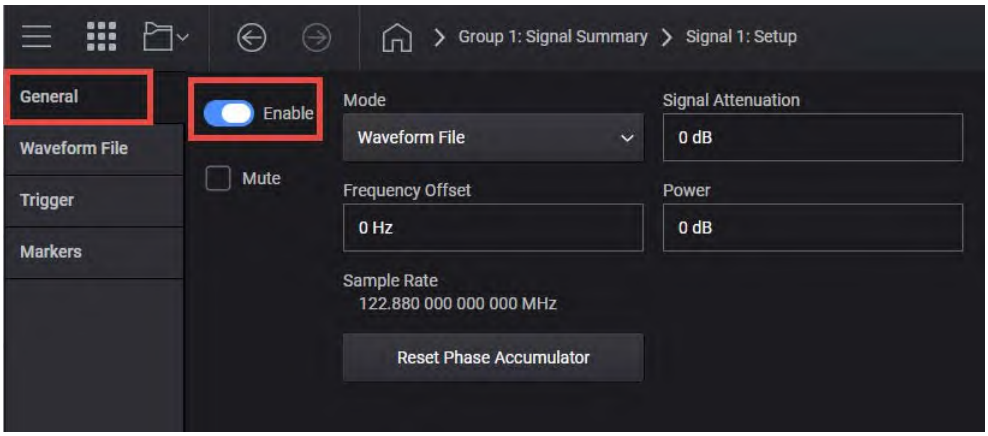
   c. Use **File Select** to navigate to:

      `D:\Users\Instrument\Documents\Keysight\PathWave\SignalGenerator\Examples`

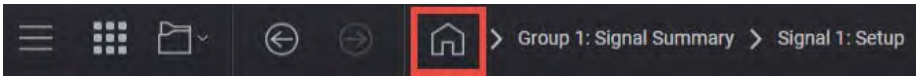
and choose `5GNR_1CC_FR2_120kHz_SCS_100MHz_256QAM_DCPunctured_28GHz.wfm` > **Select**.



d. Select the **General** tab > **Enable**.



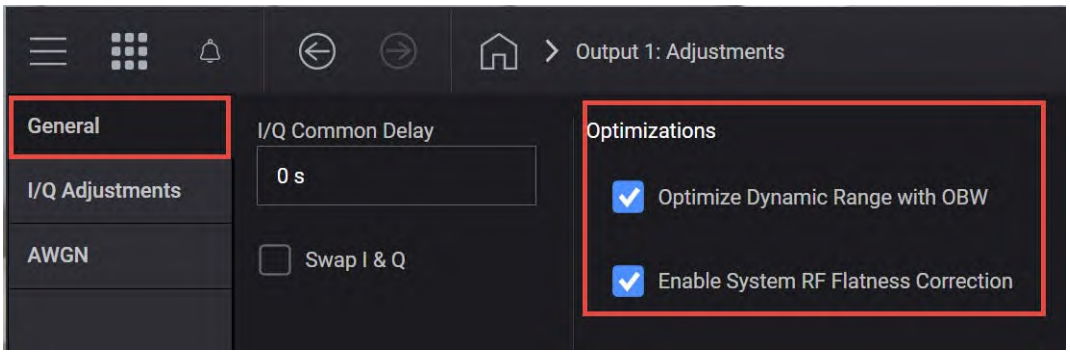
6. Close the Vector Modulation Signal Setup by selecting the **Home** icon at the top of the display.



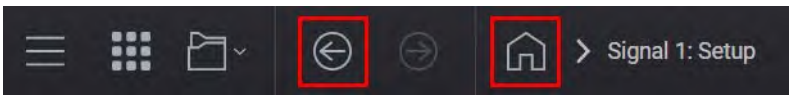
7. Select the **Adjustments** block and select **Optimize Dynamic Range with OBW** and **Enable System RF Flatness Correction**.

Optimize Dynamic Range with OBW This setting filters the system RF flatness correction coefficients over the instantaneous bandwidth indicated in the waveform header (or in the “Occupied Bandwidth” settings area under the Signal block > Occupied Bandwidth setting). This has the potential to improve EVM performance by not having to correct for flatness errors outside the requested bandwidth. For example, if RF flatness correction was done at 2 GHz, but you are only interested in an 800 MHz section, then applying the correction flatness to that portion only can improve signal to noise ratio, and therefore EVM when there is a lot of hardware roll off.

Enable System RF Flatness Correction - Disabling this function disables the factory calibrated RF channel flatness equalizer. Depending on the hardware channel response, this may hurt or improve the EVM. This is due to the dynamic range implications as it relates to signal to noise ratio. The greater the RF hardware variations in flatness, the greater the amount of correction is required, the greater the correction effectively reduces the number of resolution DAC bits that can be used, which degrades the signal to noise ratio and therefore potentially EVM. The trade-off is to balance between flatness and signal to noise ratio.



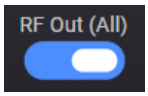
8. Close the Adjustments Setup by either selecting the **Back** or the **Home** icon at the top of the display.



9. Set RF Out to **On** by selecting the numbered channel indicator switch.

   This enables the RF Out for the indicated channels, in this case Channel 1 if using a multi-channel VXG.

10. **For two channel instruments only:** In the top right corner of the display, set RF Out (All) to **On** by selecting the switch.



NOTE

In order to turn on RF for any channel, both the RF Out for the specific channel (for example, Channel 1 or Channel 2), and RF Out All must be turned on.

On the X-Series Signal Analyzer:

NOTE

All example waveforms and setup files are located on the VXG at:

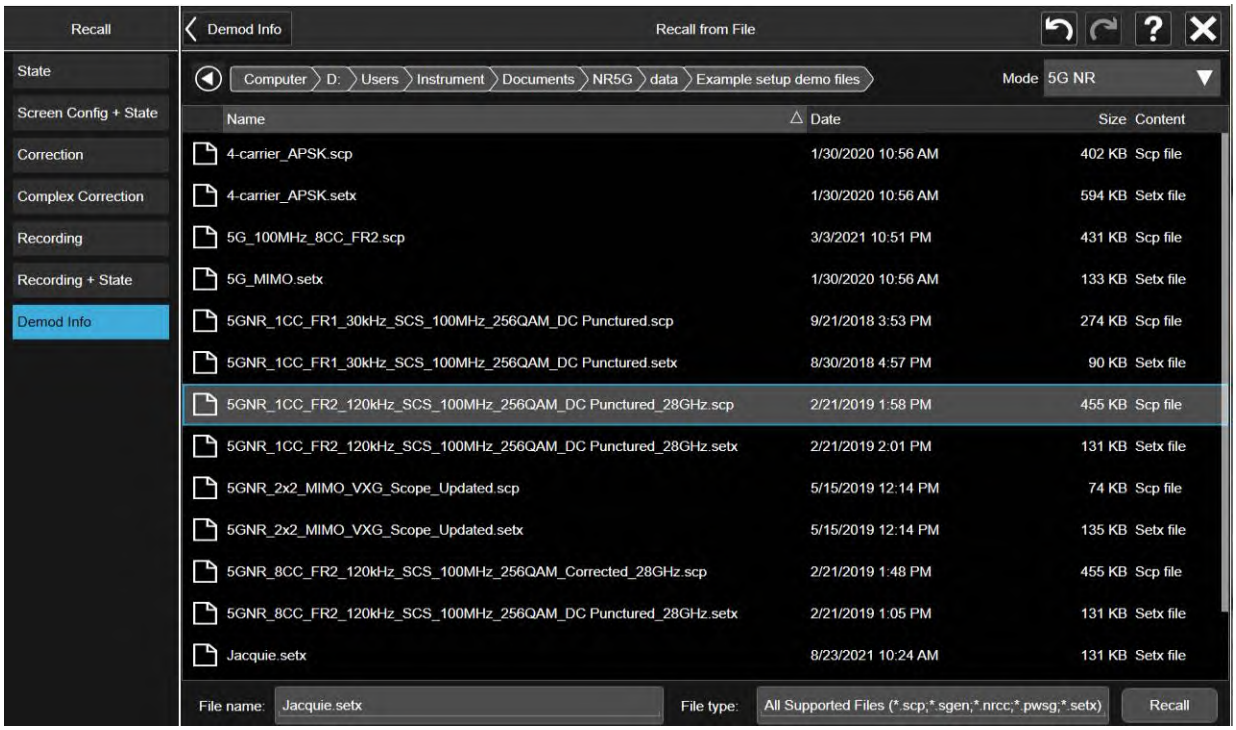
```
D:\Users\Instrument\Documents\Keysight\PathWave
\SignalGenerator\Examples
```

You will need to copy over the setup files to the X-Series Signal Analyzer or the PC running the VSA application.

1. Select **Recall** (If accessing the signal analyzer remotely, select the Folder icon at the bottom of the display) **Demod Info** > Set Data Type to **CC Setup** > **Recall From** >

   `5GNR_1CC_FR2_120kHz_SCS_100MHz_256QAM_DC punctured_28GHz_34.scp`

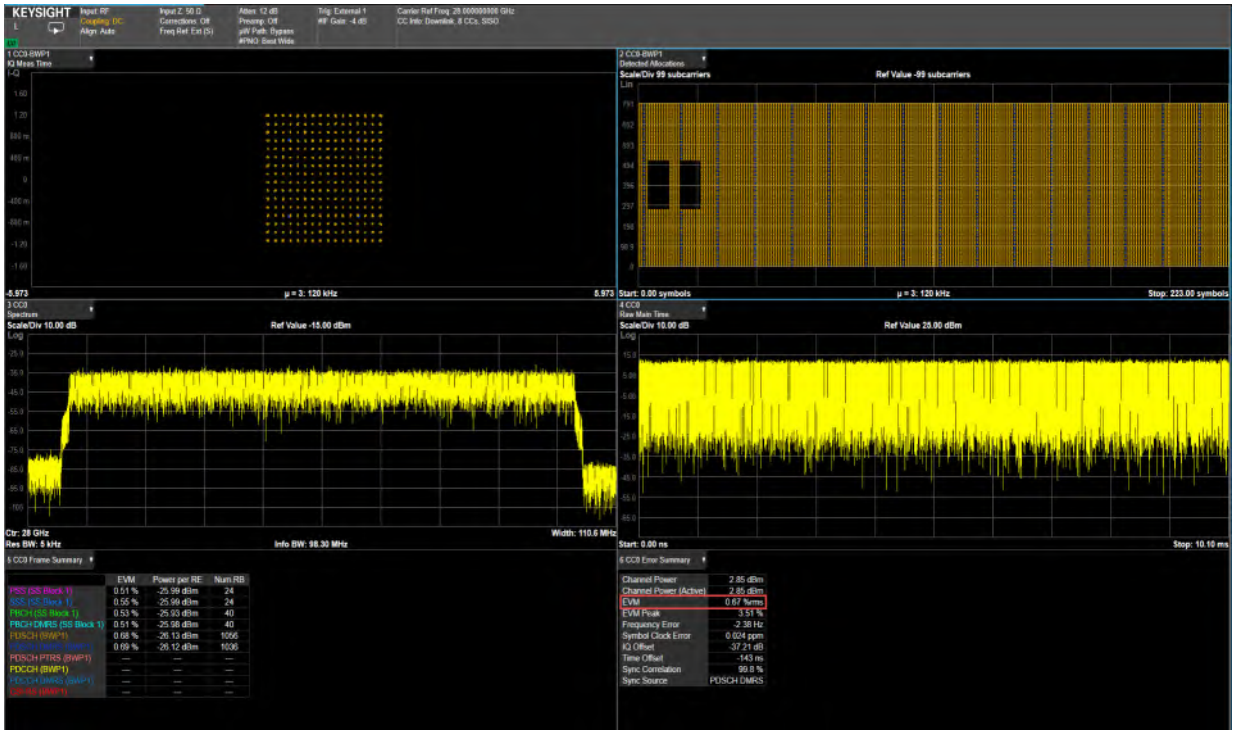
   Recall



2. Select **Meas Setup** > **Meas Time** and set:
   - Search Length to **10 ms**
   - Result Length to **2 Sub Frame**
   - Frame Trigger to **On**
3. Select the **Settings** tab > **Optimize EVM**.

The Optimize EVM function automatically sets the combination of preamplification, mechanical and electronic attenuation, and IF gain based on the measured signal peak level.

EVM should be less than 1%.



Using the equivalent SCPI commands

On the VXG:

```
SYSTem:PRESet

RF1:FREQuency:CW 28GHZ

RF1:POWer:AMPLitude 5dBm

SIGNal1:MODE WAVeform
```

Navigate to the desired waveform file.

```
SIGNal1:WAVeform "D:\Users\Instrument\Documents\demo
waveforms
5GNR_1CC_FR2_120kHz_SCS_100MHz_256QAM_DC punctured_
28GHz_34.scp"

SIGNal1 ON

IQO:CORR:OPT:DYN:RANG:OBW ON

RF1:OUTPut ON
```

For multi-channel instruments, set RF Out (all) to On.

```
RFALl:OUTPut ON
```

On the X-Series Signal Analyzer:

NOTE

All example waveforms and setup files are located on the VXG at:

```
D:\Users\Instrument\Documents\Keysight\PathWave
\SignalGenerator\Examples
```

You will need to copy over the setup files to the X-Series Signal AnalyzerX-Series Signal Analyzer or the PC running the VSA application.

```
MMEMory:LOAD:EVM:SETup CC0,
"D:\Users\Instrument\Documents\NR5G\data\NR5GEvm\CarrierSetu
p\5GNR_1CC_FR2_120kHz
SCS_100MHz_256QAM_DCpunctured_28GHz.scp"

EVM:CCARrier0:TIME:LENGth:SEARch 10ms

EVM:CCARrier0:TIME:LENGth:RESult 2

EVM:CCARrier0:FRAMe:TRIGger ON

EVM:CCARrier0:DC:PUNCture ON

EVM:CCARrier0:PHASe:COMPensation:AUTO OFF

EVM:CCARrier0:PHASe:COMPensation:FREQuency 0 Hz

EVM:OPTimize
```

Setting up an 8 CC 28 GHz EVM Measurement

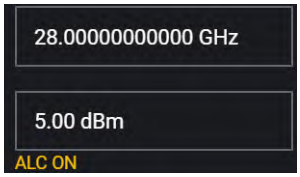
This procedure shows you how to use the X-Series signal analyzer with the 5G NR & V2X X-Series application to observe EVM on an eight-carrier waveform.

NOTE

Ensure the equipment and triggers are properly configured. Refer to "Equipment Setup" on page 13 and "Setting Up Triggers on the Signal Analyzer using 5G NR Mode" on page 508.

On the VXG:

1. Select **Preset** > **Preset** to set the VXG to a known state.
2. Set Frequency to **28 GHz** and Amplitude to **5 dBm**.

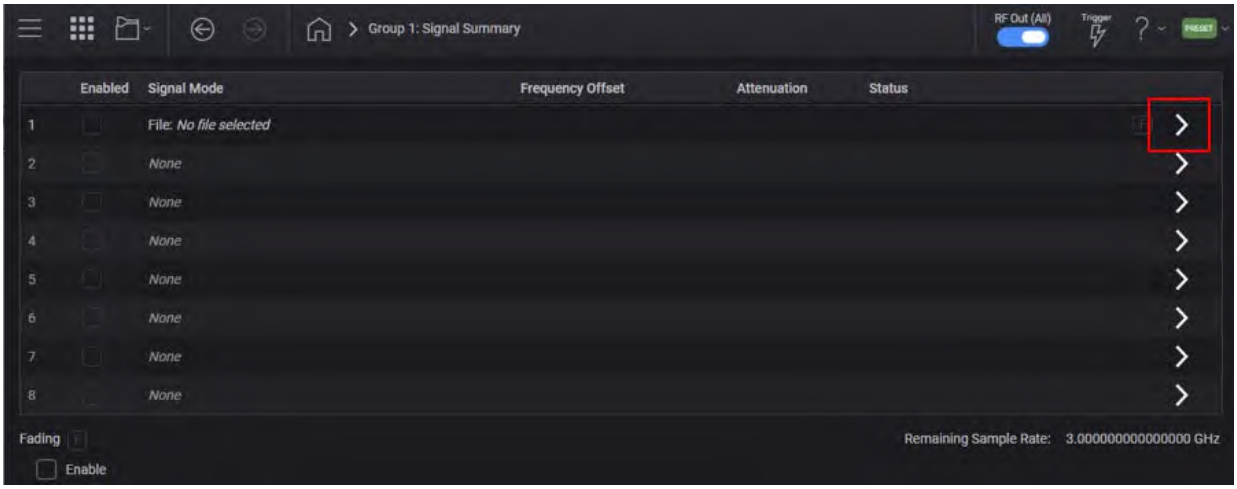


3. Select **Group 1: Signals** block to open.

4. Select the arrow for Signal 1 to open the Signal Setup window.

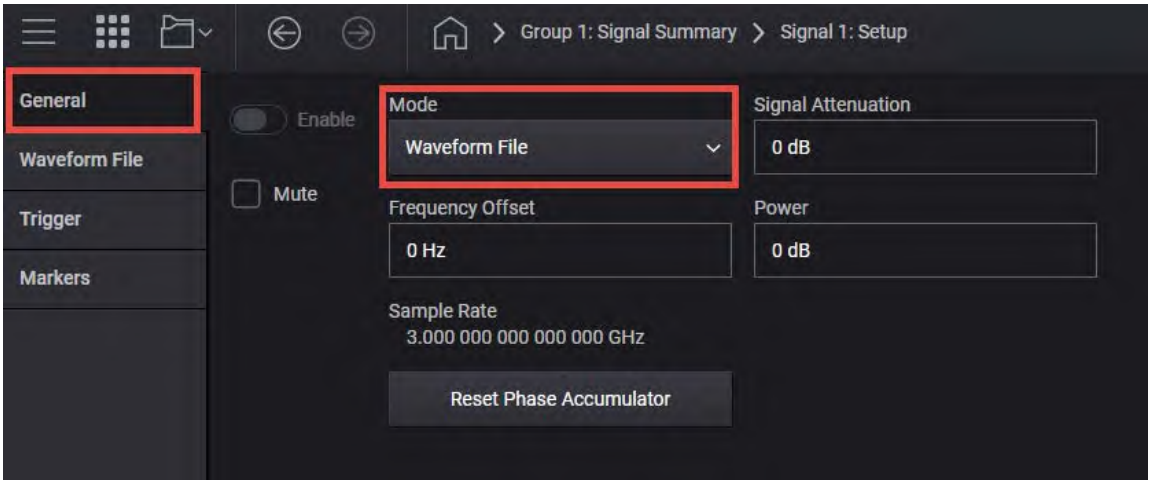
NOTE

This screen is only accessible if Option M9484C-8SG (8 virtual signal generators) is installed. For all other option configurations, continue to the next step.



5. In the Vector Modulation Signal Setup:

   a. Select the **Mode** dropdown and set to **Waveform File**.

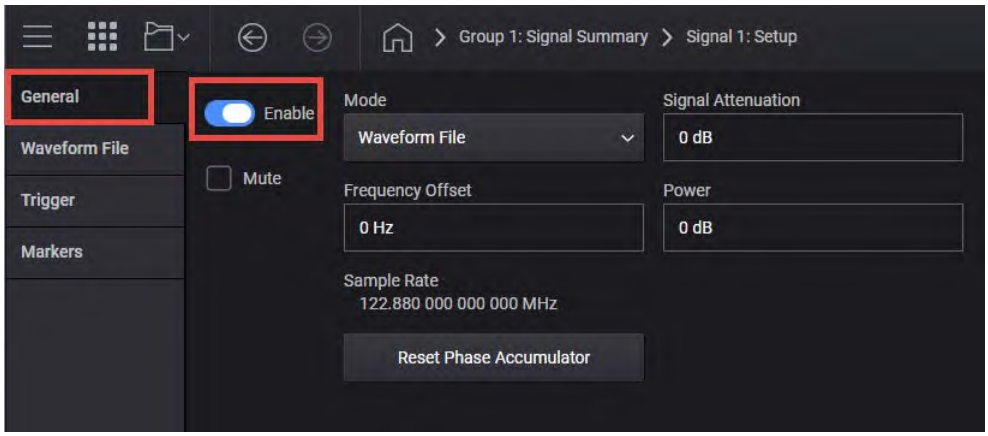


   b. In the left pane, select the **Waveform File** tab.

   c. Use **File Select** to navigate to:

      `D:\Users\Instrument\Documents\Keysight\PathWave\SignalGenerator\Examples`
      and choose
      `5GNR_8CC_FR2_120kHz_SCS_100MHz_256QAM_Corrected_28GHz.wfm`
      then **Select**.

d. Select the **General** tab > **Enable**.



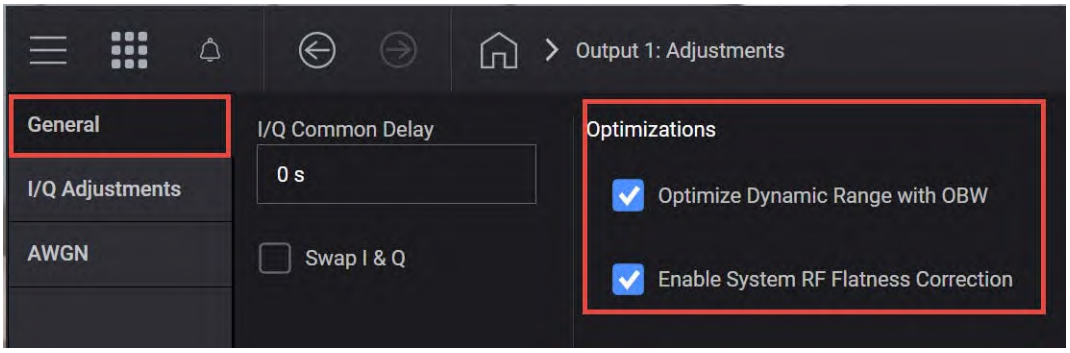
6. Close the Signal Setup by selecting the **Home** icon at the top of the display.

7. Select the **Adjustments** block and select **Optimize Dynamic Range with OBW** and **Enable System RF Flatness Correction**.

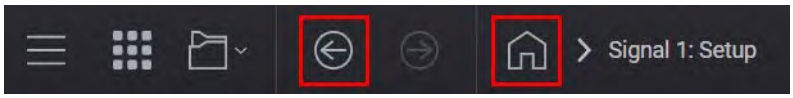
Optimize Dynamic Range with OBW This setting filters the system RF flatness correction coefficients over the instantaneous bandwidth indicated in the waveform header (or in the "Occupied Bandwidth" settings area under the Signal block > Occupied Bandwidth setting). This has the potential to improve EVM performance by not having to correct for flatness errors outside the requested bandwidth. For example, if RF flatness correction was done at 2 GHz, but you are only interested in an 800 MHz section, then applying the correction flatness to that portion only can improve signal to noise ratio, and therefore EVM when there is a lot of hardware roll off.

Enable System RF Flatness Correction - Disabling this function disables the factory calibrated RF channel flatness equalizer. Depending on the hardware channel response, this may hurt or improve the EVM. This is due to the dynamic range implications as it relates to signal to noise ratio. The greater the RF hardware variations in flatness, the greater the amount of correction is required, the greater the correction effectively reduces the number of

resolution DAC bits that can be used, which degrades the signal to noise ratio and therefore potentially EVM. The trade-off is to balance between flatness and signal to noise ratio.



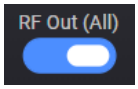
8. Close the Adjustments Setup by either selecting the **Back** or the **Home** icon at the top of the display.



9. Set RF Out to **On** by selecting the numbered channel indicator switch.

   This enables the RF Out for the indicated channels, in this case Channel 1 if using a multi-channel VXG.

10. **For multi-channel instruments only:** In the top right corner of the display, set RF Out (All) to **On** by selecting the switch.



NOTE

In order to turn on RF for any channel, both the RF Out for the specific channel (for example, Channel 1 or Channel 2), and RF Out All must be turned on.

On the X-Series Signal Analyzer:

NOTE

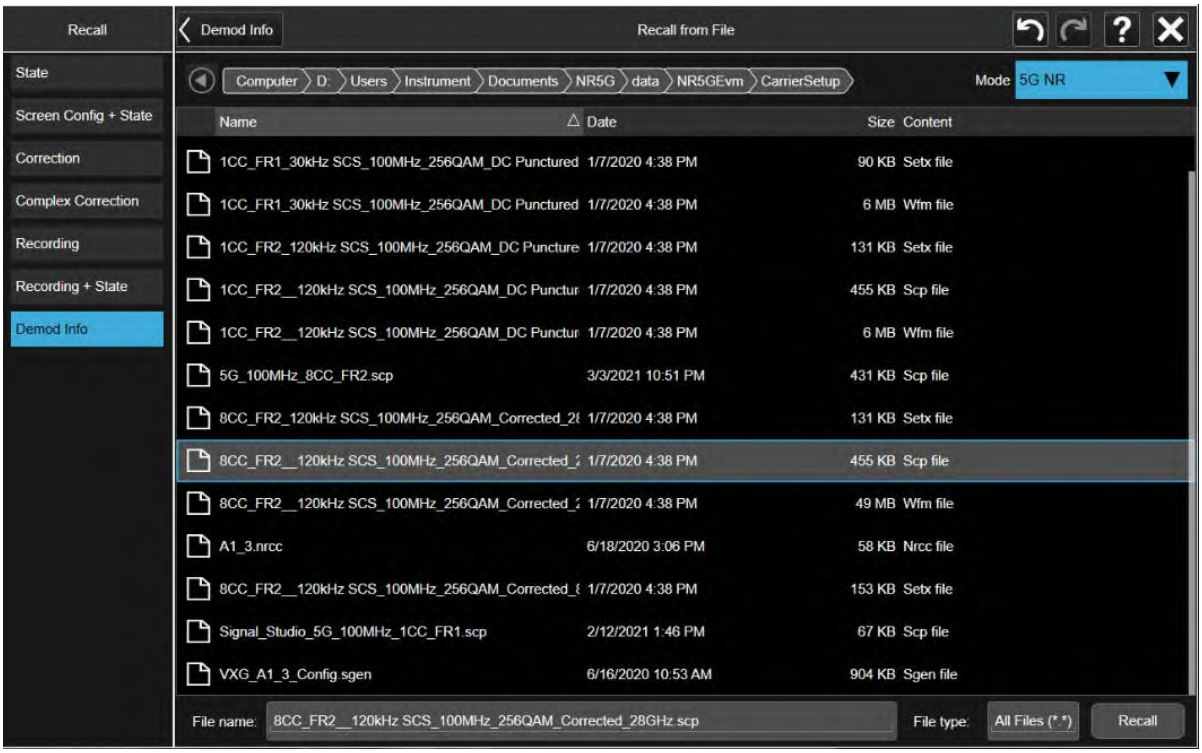
All example waveforms and setup files are located on the VXG at:

`D:\Users\Instrument\Documents\Keysight\PathWave\SignalGenerator\Examples`

You will need to copy over the setup files to the signal analyzer or the PC running the VSA application.

1. Select **Recall** (If accessing the signal analyzer remotely, select the Folder icon at the bottom of the display) > **Demod Info** > set Data Type to **CC Setup** > **Recall From** > `5GNR_8CC_FR2_120kHz_SCS_100MHz_256QAM_Corrected_28GHz_34.scp` > **Recall**.

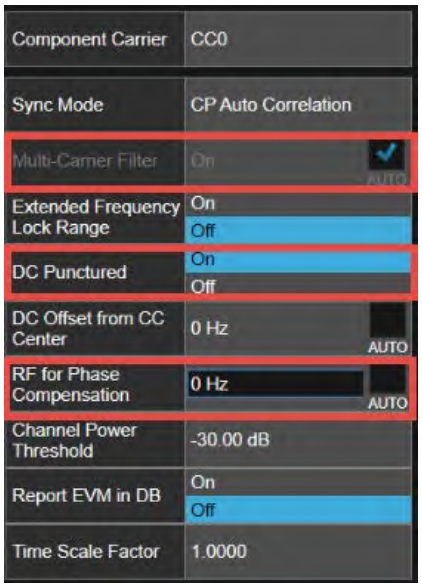
   You must be in the correct analysis mode (in this case, 5G NR & V2X1 >Modulation Analysis mode) to recall the .scp file.



2. Select **Meas Setup** > **Meas Time** tab and set:
   - Search Length to **10 ms**
   - Result Length to **2 Sub Frame**
   - Frame Trigger to **On**
3. Select **Meas Setup** > **Advanced** tab > **Advanced Demod Setup** and verify that both **Multi-Carrier Filter** and **DC Punctured** are on.

The multi-carrier filter is used to filter out the unwanted carriers and minimize leakage into the component carrier of interest.

4. Ensure that RF for Phase Compensation is *not* selected and the value is **0 Hz**, then **Close** the Advanced Settings table.



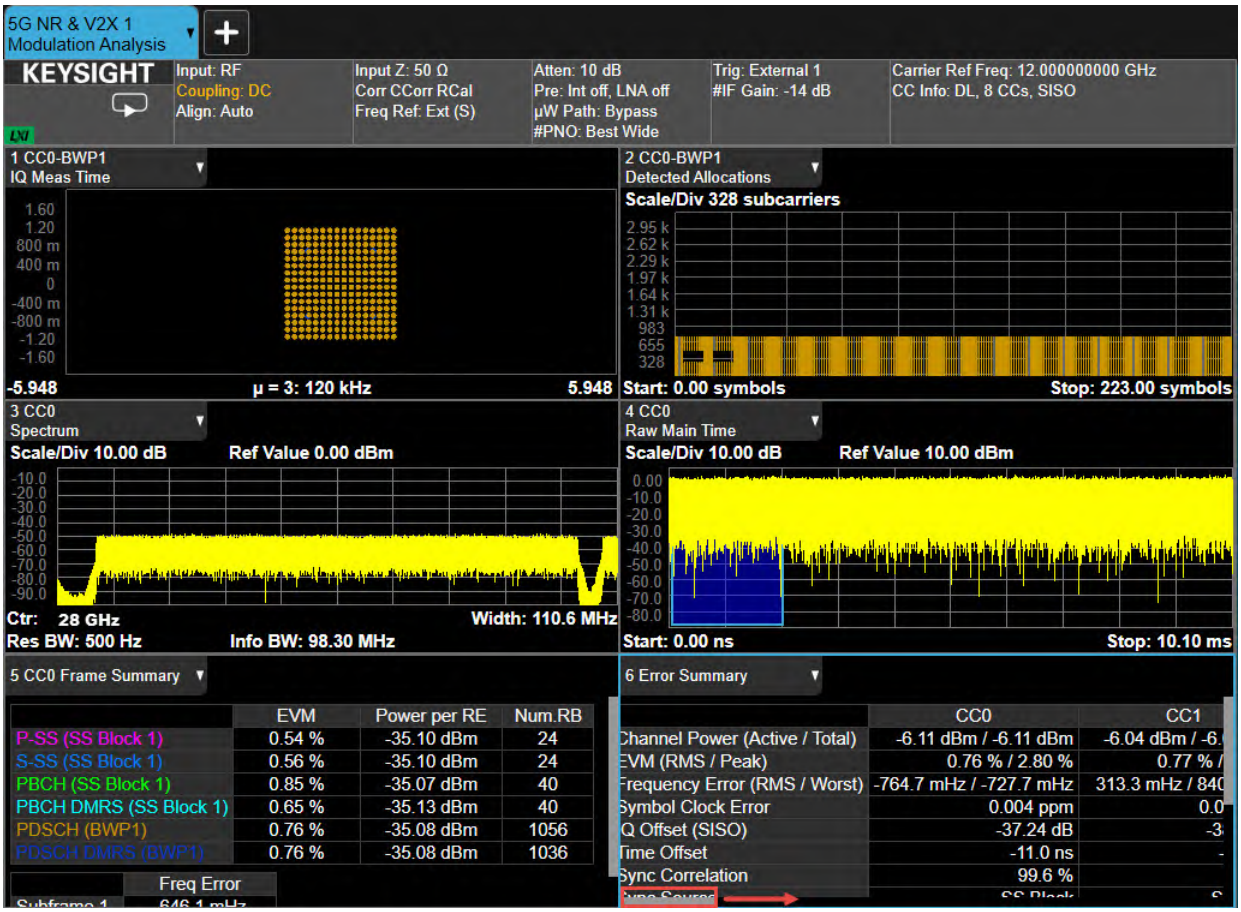
5. Select **Meas Setup** > **Settings** tab > **Optimize EVM**.

   It will take a couple of minutes for the UXA to measure all 8 carriers.

NOTE

If you are getting a "Sync Error; Sync not found" or "Input Overload - Detected; ADC over range" message), go to Amplitude > Attenuation > and increase the Mechanical Attenuation by 2 dB until the Sync Error is resold.

Notice that the Error Summary measurement has a scroll bar allowing you to view the results of all 8 component carriers.


5G NR & V2X 1
Modulation Analysis
KEYSIGHT
Input: RF
Coupling: DC
Align: Auto
Input Z: 50 Ω
Corr CCorr RCal
Freq Ref: Ext (S)
Atten: 10 dB
Pre: Int off, LNA off
µW Path: Bypass
#PNO: Best Wide
Trig: External 1
#IF Gain: -14 dB
Carrier Ref Freq: 12.000000000 GHz
CC Info: DL, 8 CCs, SISO
1 CC0-BWP1
IQ Meas Time
-5.948
µ = 3: 120 kHz
5.948
2 CC0-BWP1
Detected Allocations
Scale/Div 328 subcarriers
Start: 0.00 symbols
Stop: 223.00 symbols
3 CC0
Spectrum
Scale/Div 10.00 dB
Ref Value 0.00 dBm
Ctr: 28 GHz
Width: 110.6 MHz
Res BW: 500 Hz
Info BW: 98.30 MHz
4 CC0
Raw Main Time
Scale/Div 10.00 dB
Ref Value 10.00 dBm
Start: 0.00 ns
Stop: 10.10 ms
5 CC0 Frame Summary
EVM
Power per RE
Num.RB
P-SS (SS Block 1) 0.54 % -35.10 dBm 24
S-SS (SS Block 1) 0.56 % -35.10 dBm 24
PBCH (SS Block 1) 0.85 % -35.07 dBm 40
PBCH DMRS (SS Block 1) 0.65 % -35.13 dBm 40
PDSCH (BWP1) 0.76 % -35.08 dBm 1056
PDSCH DMRS (BWP1) 0.76 % -35.08 dBm 1036
Freq Error
6 Error Summary
CC0
CC1
Channel Power (Active / Total) -6.11 dBm / -6.11 dBm
EVM (RMS / Peak) 0.76 % / 2.80 %
Frequency Error (RMS / Worst) -764.7 mHz / -727.7 mHz
Symbol Clock Error 0.004 ppm
Q Offset (SISO) -37.24 dB
Time Offset -11.0 ns
Sync Correlation 99.6 %

Using the equivalent SCPI commands

On the VXG:

```
SYSTem:PRESet

RF1:FREQuency:CW 28GHZ

RF1:POWer:AMPLitude 5dBm

SIGNal1:MODE WAVeform
```

Navigate to the desired waveform file.

```
SIGNal1:WAVeform "D:\Users\Instrument\Documents\demo
waveforms\5GNR_8CC_FR2_120kHz_SCS_100MHz_Corrected_28GHz.wfm
"

SIGNal1 ON

IQO:CORR:OPT:DYN:RANG:OBW ON

IQO:CORR:CHANNel:FLATness ON

RF1:OUTPut ON
```

For multi-channel instruments, set RF Out (all) to On.

```
RFALl:OUTPut ON
```

On the X-Series Signal Analyzer:

NOTE

All example waveforms and setup files are located on the VXG at:

```
D:\Users\Instrument\Documents\Keysight\PathWave
\SignalGenerator\Examples
```

You will need to copy over the setup files to the X-Series Signal Analyzer or the PC running the VSA application.

Navigate to the desired waveform file.

```
MMEMory:LOAD:EVM:SETup CC0,
"D:\Users\Instrument\Documents\NR5G\data\NR5GEvm\CarrierSetu
p\8CC_FR2_120kHz_SCS_100MHz_256QAM_Corrected_28GHz_34.scp"

EVM:CCARrier0:TIME:LENGth:SEARch 10ms

EVM:CCARrier0:TIME:LENGth:RESult 2

EVM:CCARrier0:FRAMe:TRIGger ON

EVM:CCARrier0:MCFilter ON

EVM:CCARrier0:DC:PUNCture ON

EVM:CCARrier0:PHASe:COMPensation:AUTO OFF

EVM:CCARrier0:PHASe:COMPensation:FREQuency 0 Hz
```

To Optimize EVM for Multi-Carrier Waveforms

```
POWer:ATTenuation 0dB

[POWer:ATTenuation 2dB], …

EVM:IF:GAIN:LEVel 0dB
```

To View the Results of Two Carriers

Set the top right window (Window 2) to display IQ Meas Time

```
DISPlay:EVM:WINDow2:DATA MTIM
```

Set the top right window to display Component Carrier 1

```
DISPlay:EVM:WINDow2:CCARrier CC1
```

Set the middle right window to display the FFT of the Raw Main Time waveform

```
DISPlay:EVM:WINDow4:DATA SPEC
```

```
DISPlay:EVM:WINDow4:CCARrier CC1
```

Set the bottom right window to display a table of general measurement numeric results

```
DISPlay:EVM:WINDow6:DATA DRES
```

```
DISPlay:EVM:WINDow6:CCARrier CC1
```

Set the bottom left window to display the error summary for CC0

```
DISPlay:EVM:WINDow5:DATA DRES
```

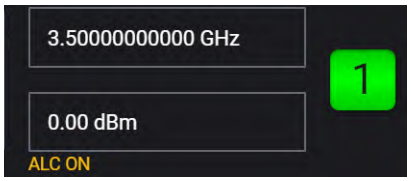
Setting Up a 1 CC 3.5 GHz ACP Measurement

NOTE

Ensure the equipment and triggers are properly configured. Refer to "Equipment Setup" on page 13 and "Setting Up Triggers on the Signal Analyzer using 5G NR Mode" on page 508.

On the VXG:

1. Select **Preset** > **Preset** to set the VXG to a known state.
2. Set Frequency to **3.5 GHz** and Amplitude to **0 dBm**.

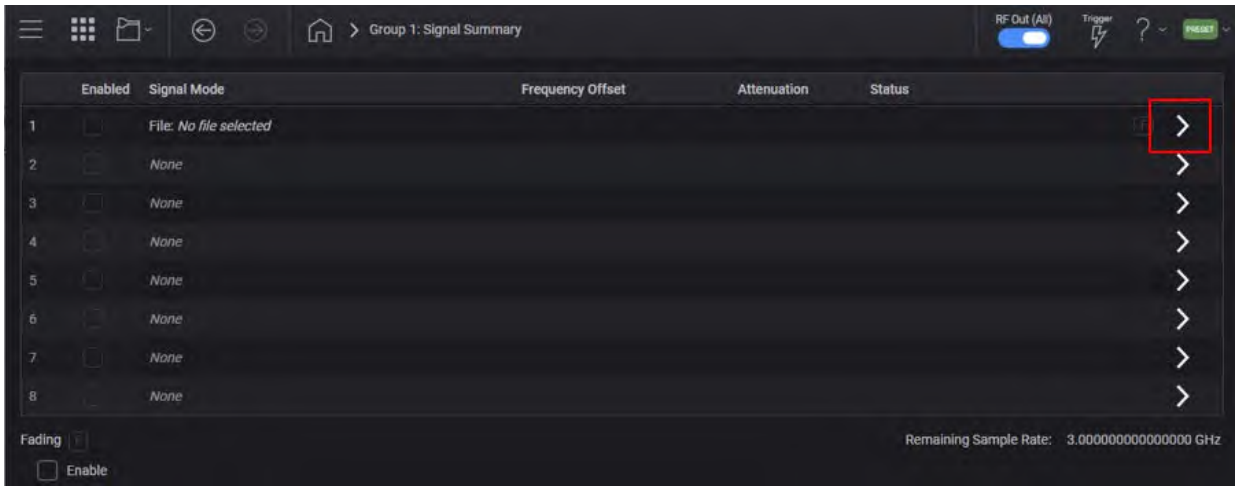


3. Select **Group 1: Signals** block to open.

4. Select the arrow for Signal 1 to open the Signal Setup window.

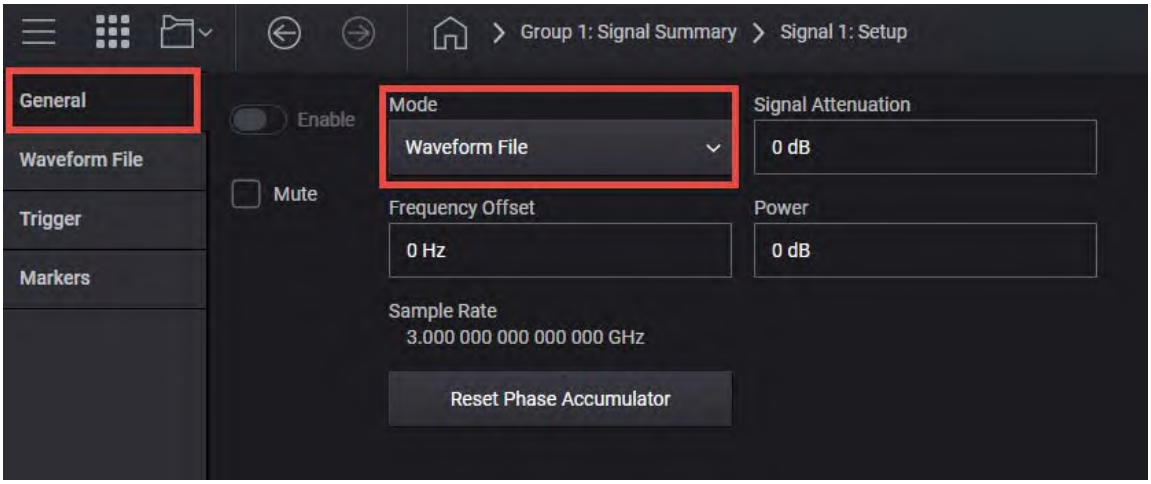
NOTE

This screen is only accessible if Option M9484C-8SG (8 virtual signal generators) is installed. For all other option configurations, continue to the next step.



5. In the Vector Modulation Signal Setup:

   a. Select the **Mode** dropdown and set to **Waveform File**.

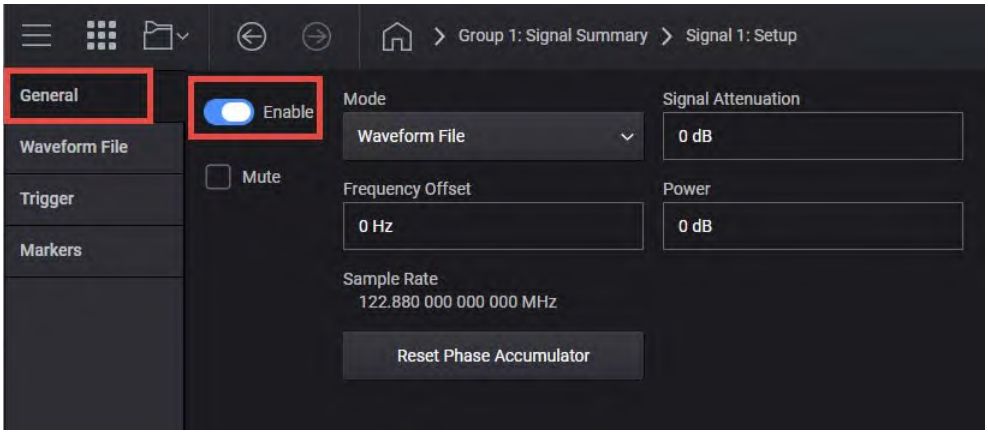


   b. In the left pane, select the **Waveform File** tab.

   c. Use **File Select** to navigate to:

      `D:\Users\Instrument\Documents\Keysight\PathWave\SignalGenerator\Examples`
      and choose
      `5GNR_1CC_FR1_30kHz_SCS_100MHz_256QAM_DCPunctured.wfm`
      then **Select**.

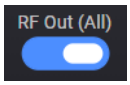
d. Select the **General** tab > **Enable**.



6. Set RF Out to **On** by selecting the numbered channel indicator switch.

   This enables the RF Out for the indicated channels, in this case Channel 1 if using a multi-channel VXG.

7. **For multi-channel instruments only:** In the top right corner of the display, set RF Out (All) to **On** by selecting the switch.



NOTE

In order to turn on RF for any channel, both the RF Out for the specific channel (for example, Channel 1 or Channel 2), and RF Out All must be turned on.

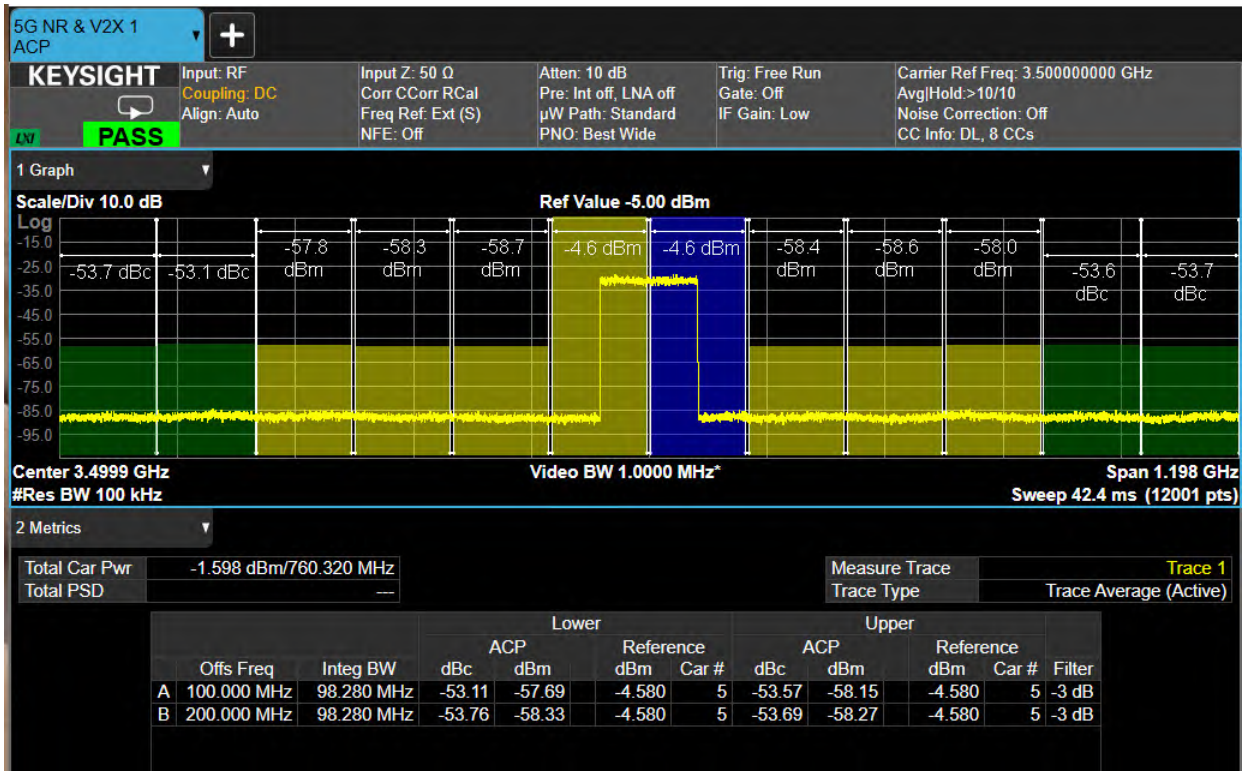
On the X-Series Signal Analyzer:

1. Select **Mode Preset** to set 5G NR V2X mode to a known state.
2. Select **Mode/Meas** > **5G NR & V2X** Mode > **ACP** Measurement.

NOTE

If accessing the instrument via a Remote Desktop connection, select the Screen tab (at the top of the display) to open the Mode/Measurement/View Selector window.

3. Select **Frequency** and set Carrier Reference Frequency to **3.5 GHz**.


5G NR & V2X 1
ACP
KEYSIGHT
PASS
Input: RF
Coupling: DC
Align: Auto
Input Z: 50 Ω
Corr CCorr RCal
Freq Ref: Ext (S)
NFE: Off
Atten: 10 dB
Pre: Int off, LNA off
µW Path: Standard
PNO: Best Wide
Trig: Free Run
Gate: Off
IF Gain: Low
Carrier Ref Freq: 3.500000000 GHz
Avg|Hold:>10/10
Noise Correction: Off
CC Info: DL, 8 CCs
1 Graph
Scale/Div 10.0 dB
Ref Value -5.00 dBm
Log
-15.0
-25.0
-35.0
-45.0
-55.0
-65.0
-75.0
-85.0
-95.0
-53.7 dBc
-53.1 dBc
-57.8 dBm
-58.3 dBm
-58.7 dBm
-4.6 dBm
-4.6 dBm
-58.4 dBm
-58.6 dBm
-58.0 dBm
-53.6 dBc
-53.7 dBc
Center 3.4999 GHz
#Res BW 100 kHz
Video BW 1.0000 MHz*
Span 1.198 GHz
Sweep 42.4 ms (12001 pts)
2 Metrics
Total Car Pwr -1.598 dBm/760.320 MHz
Total PSD ---
Measure Trace Trace 1
Trace Type Trace Average (Active)
Lower
Upper
ACP
Reference
Offs Freq
Integ BW
dBc
dBm
dBm
Car #
Filter
A 100.000 MHz 98.280 MHz -53.11 -57.69 -4.580 5 -53.57 -58.15 -4.580 5 -3 dB
B 200.000 MHz 98.280 MHz -53.76 -58.33 -4.580 5 -53.69 -58.27 -4.580 5 -3 dB

Using the equivalent SCPI commands

On the VXG:

```
SYSTem:PRESet

RF1:FREQuency:CW 3.5GHZ

RF1:POWer:AMPLitude 0dBm

SIGNal1:MODE WAVeform

SIGNal1:WAVeform "D:\Users\Instrument\Documents\MCS demo
waveforms\5GNR_1CC_FR1_30kHz_SCS_100MHz_256QAM_DCPunctured.w
fm"

SIGNal1 ON

DM:OPTimization:CHANnel ACP

RF1:OUTPut ON
```

For multi-channel instruments, set RF Out (all) to On.

```
RFALl:OUTPut ON
```

On the X-Series Signal Analyzer:

```
SYSTem:PRESet

INSTrument:CONFigure: NR5G:ACP

CCARrier:REFerence 3.5GHZ

ACPower:CORRection:NOISe ON
```

Using PathWave N7631APPC to Create a Waveform File then Automatically Configure the Analyzer to View the Results

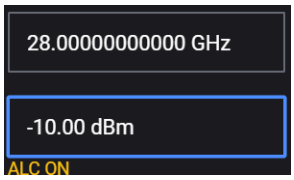
PathWave software tools can be used to create, download, and playback waveforms through the VXG.

This example shows you how to create and analyze a 5G NR signal using the embedded PathWave software and then automatically configure the analyzer to make the measurement.

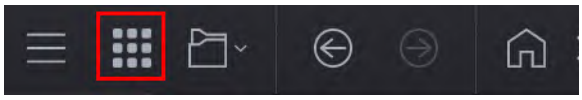
NOTE The VXG must have the N7631APPC Signal Generation for 5G NR license installed.

On the VXG:

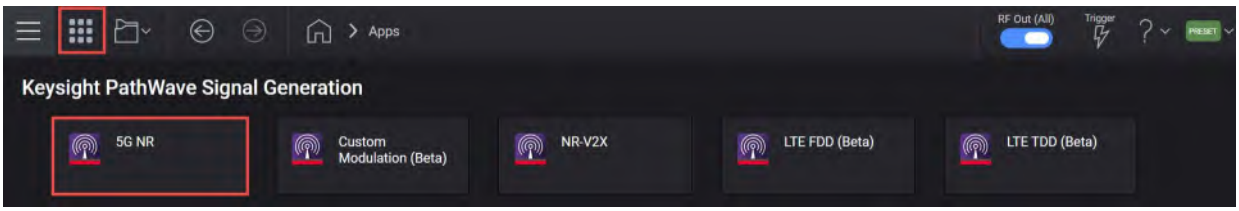
1. Select **Preset** > **Preset** to set the VXG to a known state.
2. In the Output area, set Frequency to **28 GHz** and Power to **-10 dBm**.



3. Select the **Radio Apps** block to open the mode selection panel.



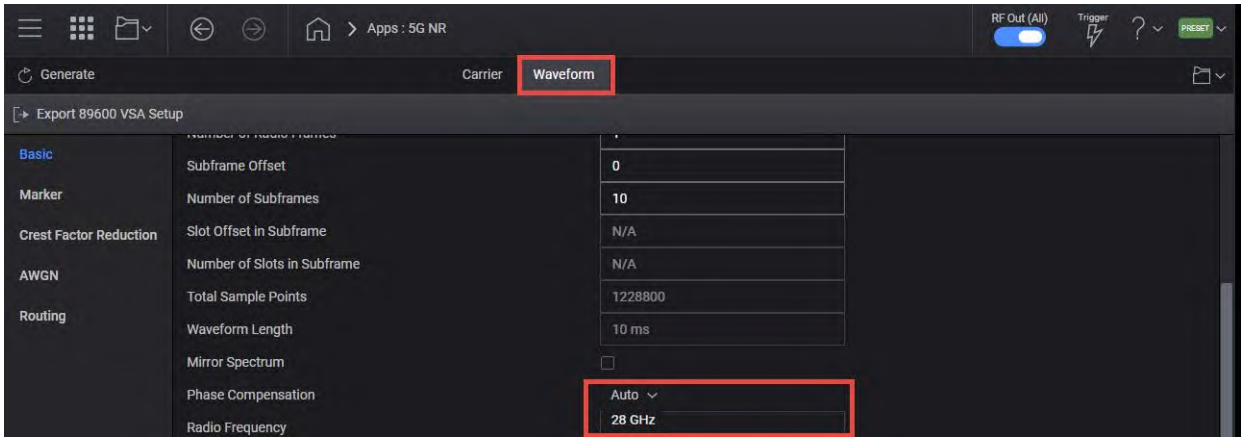
4. Select **5G NR/5G-Advanced** to enter the 5G NR signal mode.



5. In the 5G NR setup, select the **Waveform** tab and observe the Phase Compensation and Radio Frequency settings.

   Phase Compensation is applied by default at baseband for RF up-conversion. This means it depends on the carrier frequency, and the waveform should only be played at the carrier frequency it was generated for, unlike conventional Signal Studio waveforms, which are independent

from carrier frequency. Phase Compensation is set to Auto by default. The frequency is coupled with the VXG frequency setting. You can select Manual to specify a different frequency, or turn it off.



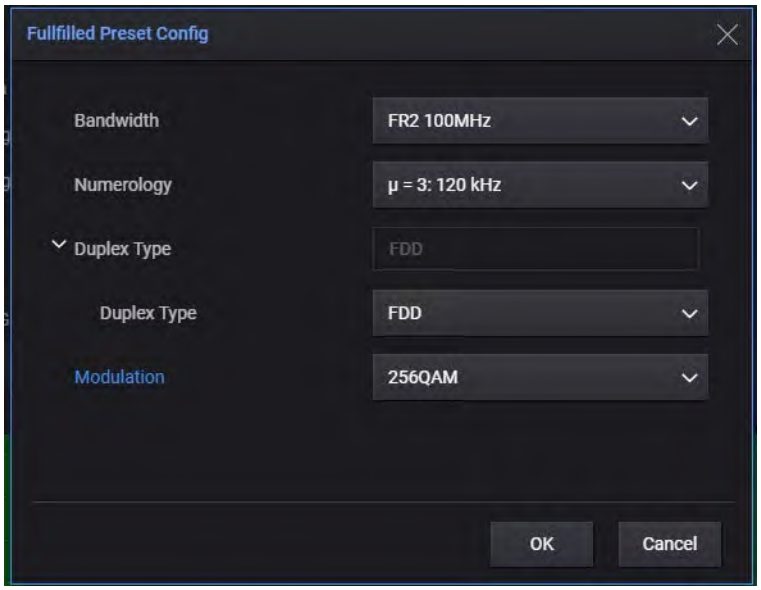
TIP Some applications, like power amplifier measurements use the same waveform at different frequencies and can be time consuming to generate a separate waveform for each frequency. In this case, you would turn Phase Compensation Off, so the waveform becomes independent from the carrier frequency. You need to do the same on the analysis side, or the demodulation will fail.

6. Select the **Carrier** tab.
7. Select **Full-Filled Config** and set Bandwidth to **FR2 100 MHz**, Numerology to **μ= 3:120 kHz**, and Modulation to **256 QAM**.

NOTE If you have a signal analyzer with demodulation bandwidth that is wide enough to cover the other FR2 bandwidths, 200 MHz or 400 MHz, you can choose to use a wider bandwidth.

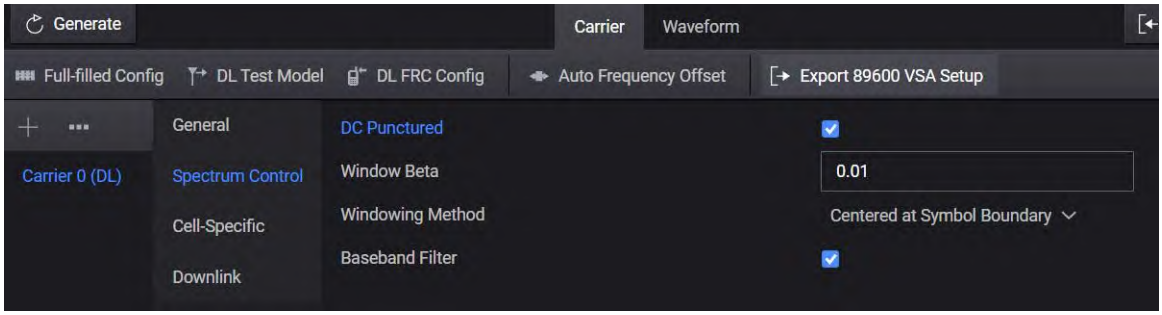
Full Filled Configurations is a great place to start creating your waveform. You can select FR, bandwidth, and modulation format with a fully allocated PDSCH data channel. This would represent a spectrally correct

signal that is typically used in power amplifier measurements. You can modify the RB allocations and Slot allocations if fully allocated data channels are not desired.



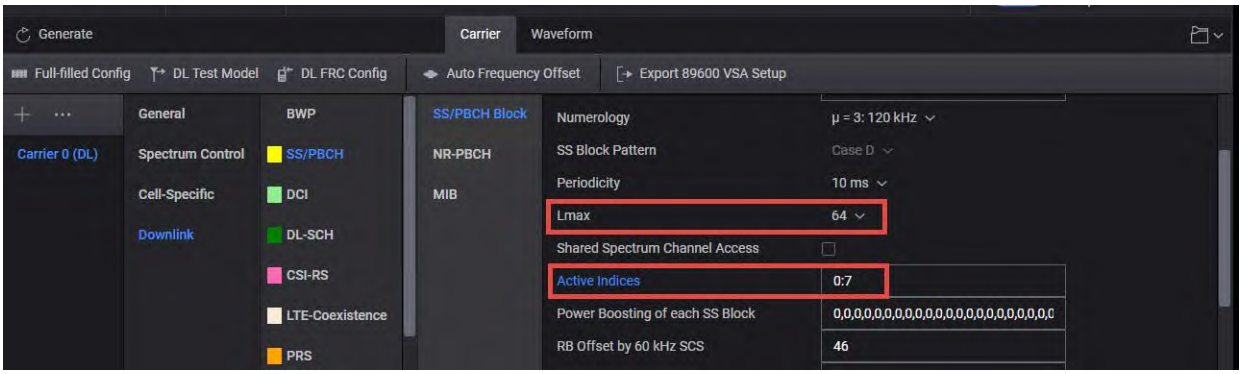
8. Select **OK** to exit the Full Filled Config setup.
9. From the Carrier panel (in the left pane), select **Spectrum Control** > **DC Punctured On**.

   When DC Punctured is enabled, the DC subcarrier is excluded from the measurement results. This is often helpful to avoid obscuring measurement results with artifacts of LO feedthrough.



10. From the Carrier panel (in the left pane), select **Downlink** > **SS PBCH** > **SS PBCH Block** and set Lmax to **64** and Active Indices to = **0:7**.

Instead of transmitting all 64 beams, we enable only 8 of them by setting Active Indices to 0:7.

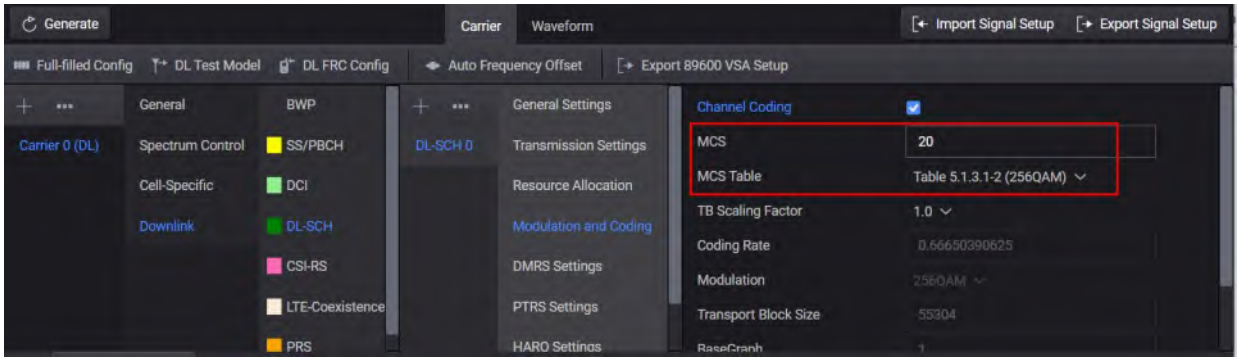


NOTE

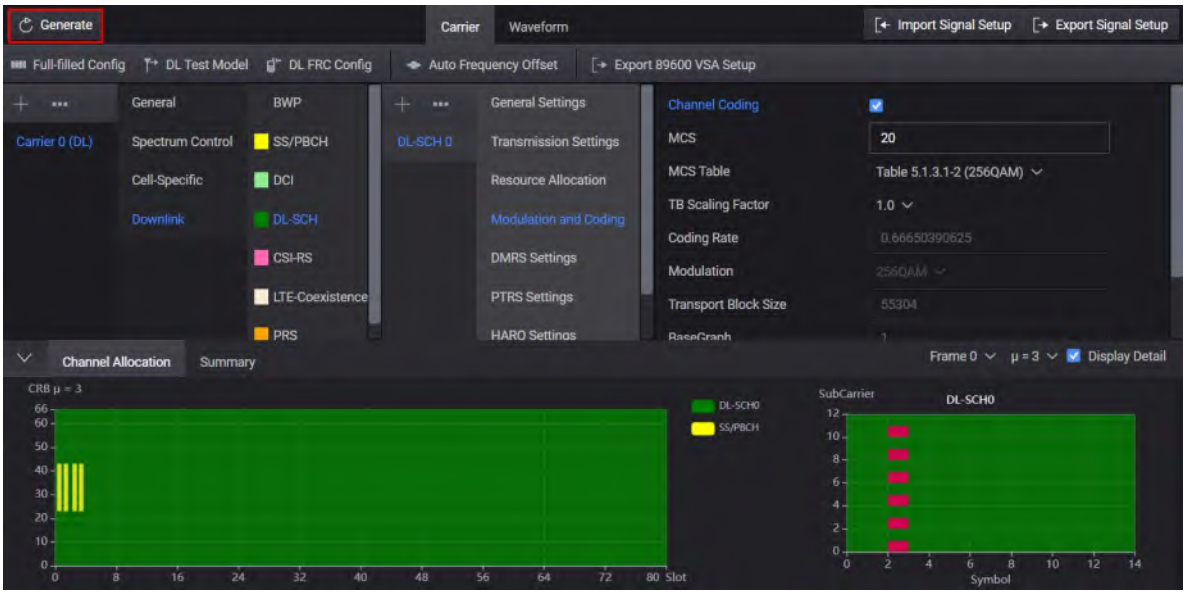
Lmax sets the maximum number of possible SS/PBCH blocks in a transmission opportunity. This value is fixed for a given carrier bandwidth. Typically, higher frequency and bandwidth carriers use a larger set of SS/PBCH opportunities to support a larger set of antenna beams.Lmax is either 4 or 8 for FR1 and 64 for FR2.

11. From the **Carrier** panel (in the left pane), select **Downlink** > **DL-SCH** > **DL-SCH0** > **Modulation and Coding** and set MCS Table to **Table 5.1.3.1-2 (256QAM)** and MCS to **20**.

    The Modulation Coding Scheme (MCS) specifies the modulation, coding and overall spectral efficiency of the PDSCH as specified in 3GPP Table 5.1.3.1.



12. Select **Generate** to generate the Waveform, and then select **Home** to exit the setup panel.

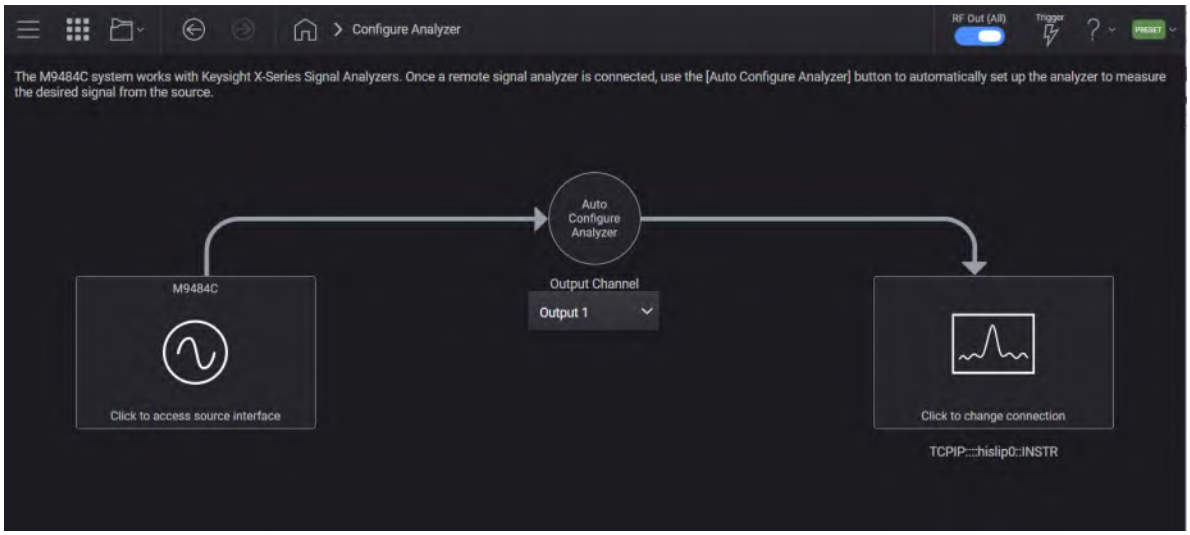


13. Set RF Out to **On** by selecting the numbered channel indicator switch.

    This enables the RF Out for the indicated channels, in this case Channel 1 if using a multi-channel VXG.

Setup Auto Configure Analyzer

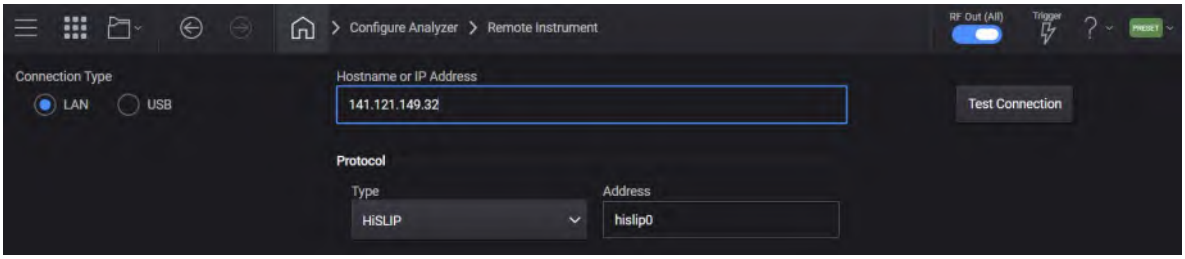
1. Select the **System** menu (triple bar tab at the top left of the window) to open the System Menu and then select **Configure Analyzer**.



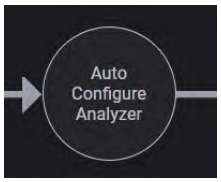
2. In the System Configuration setup, select the Remote Signal Analyzer block (on far right) to setup the communication channel to the Keysight X-Series Signal Analyzer.

3. Set Connection Type to **LAN**.
4. In the Connect Remote X-Series Signal Analyzer setup, enter the Hostname or IP Address for the signal analyzer.
5. Under Set Protocol, select **HiSLIP** and use the Remote name **hislip0**.

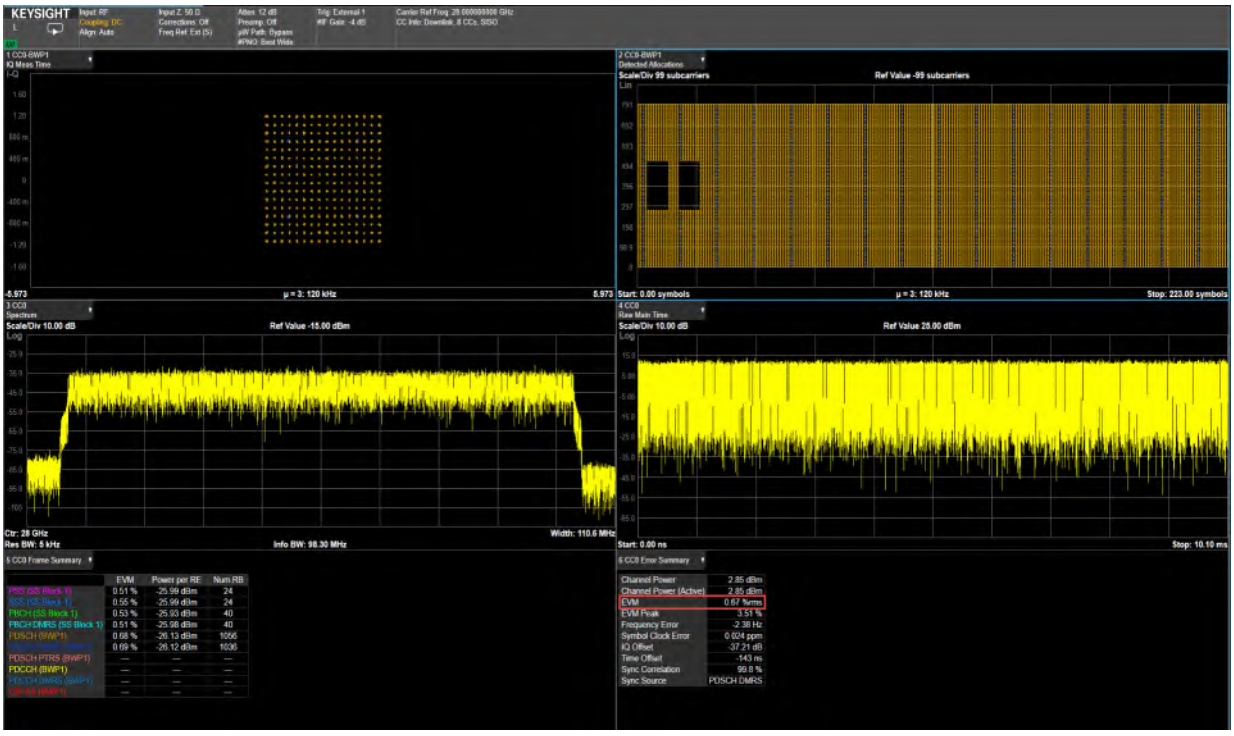
6. Select **Test Connection** to verify, then the **Back** icon to go back to the main Configure Analyzer display.



7. Select **Auto Configure Analyzer** to send the VXG settings to the analyzer.



8. View the results on the signal analyzer.



The System Configuration selection brings up a screen that lets you perform a Generation-to-Analysis work flow. The “Auto Configure Analyzer” button will automatically transfer the setup from the source to a Keysight X-Series signal analyzer, in order to measure the desired signal from the source. If you are generating a 3GPP 5G New Radio signal, and the 5G New Radio application is licensed on the analyzer, the analyzer will perform demodulation of the signal. Otherwise, the analyzer will auto-tune on the signal being generated. The Key-

sight X-Series Signal Analyzer requires firmware version x.24.00 or greater.

More About the 5G NR Traces

- Trace 1: shows the composite constellation diagram showing different modulation formats for the different channels/signals present in the transmitted signal. See Trace 6 (Frame Summary) for the list of channels. Again, when enabling BWP, SS Block is grayed out. When enabling SS Block, channels withing BWP are grayed out.
- Trace 2: shows the envelope of the captured raw data. This data is unprocessed and includes additional points acquired for settling of the filters involved in subsequent processing, such as the demodulation filtering.
- Trace 3: shows the FTT of the Raw Main Time waveform.
- Trace 4: shows the general measurement numeric results.
- Trace 5: shows the detected allocations of all channels/signals within the measurement interval (2 Subframes in this example).
- Trace 6: shows EVM, Power, and Number of RB of the individual channels/signals.

TIP The Frame Summary table shows Num. RB for PDSCH of 1056 and 1016 for PDSCH-DMRS. The signal configuration has only 66 RBs. The RB result in the Frame Summary Table is per Slot. In this example procedure, we changed the measurement interval to 2 Subframes, which is 16 slots for 120 kHz numerology (8 slots/subframe). 66 * 16 = 1056. For PDSCH-DMRS, the value is 1016 because two of the 16 slots overlap with the SS Block which occupies 20 RBs resulting in 20 RBs being punctured from DMRS in these two slots. Therefore, the total RB Number for DMRS is 1056 - 20 * 2 = 1016.

Using the equivalent SCPI commands

On the VXG:

```
SYSTem:PRESet

RF1:FREQuency:CW 28GHZ

RF1:POWer:AMPLitude -10dBm

RADio:SELect NR5G

RADio:NR5G:WAVeform:CCARrier:BWIDth FR2BW100M
```

Select μ = 3: 120kHz:

```
RADio:NR5G:WAVeform:CCARrier:SNUMerology MU3

RADio:NR5G:WAVeform:CCARrier:DLINk:SSBLock:LMAX 64

RADio:NR5G:WAVeform:CCARrier:DLINk:SSBLock:ACTive:INDices
"0:7"

RADio:NR5G:WAVeform:CCARrier:DLINk:SCH0:MCS 20

RADio:NR5G:WAVeform:CCARrier:DLINk:SCH0:MCS:TABLe TABLe52

RADio:NR5G:WAVeform:GENerate

GROup:SIGNal1 ON

RF1:OUTPut ON
```

For multi-channel instruments, set RF Out (all) to On.

```
RFALl:OUTPut ON
```

7 5G NR Measurements Using the VSA Software

This section includes the following topics:

- "5G Waveform and EVM Analysis Using VSA Software" on page 542
  - "Setting Up a 1 CC 28 GHz EVM Measurement" on page 542
  - "Setting Up an 8 CC 28 GHz EVM Measurement" on page 553
  - "Creating a Basic 5G NR Signal Using PathWave N7631APPC Embedded Software" on page 563
  - "Creating a DL MIMO Signal Using PathWave N7631APPC Signal Generation" on page 583

5G Waveform and EVM Analysis Using VSA Software

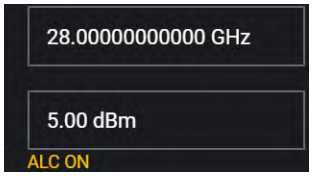
The VXG enables 5G testing with a low error vector magnitude (EVM) at high frequencies. The VXG has extremely good EVM at high power levels. However, not all signal analyzers can capture this low value. We will use the X-Series Signal Analyzer with the Vector Signal Analysis (VSA) software to observe EVM.

Setting Up a 1 CC 28 GHz EVM Measurement

NOTE Ensure the equipment and triggers are properly configured. Refer to "Equipment Setup" on page 13.

On the VXG:

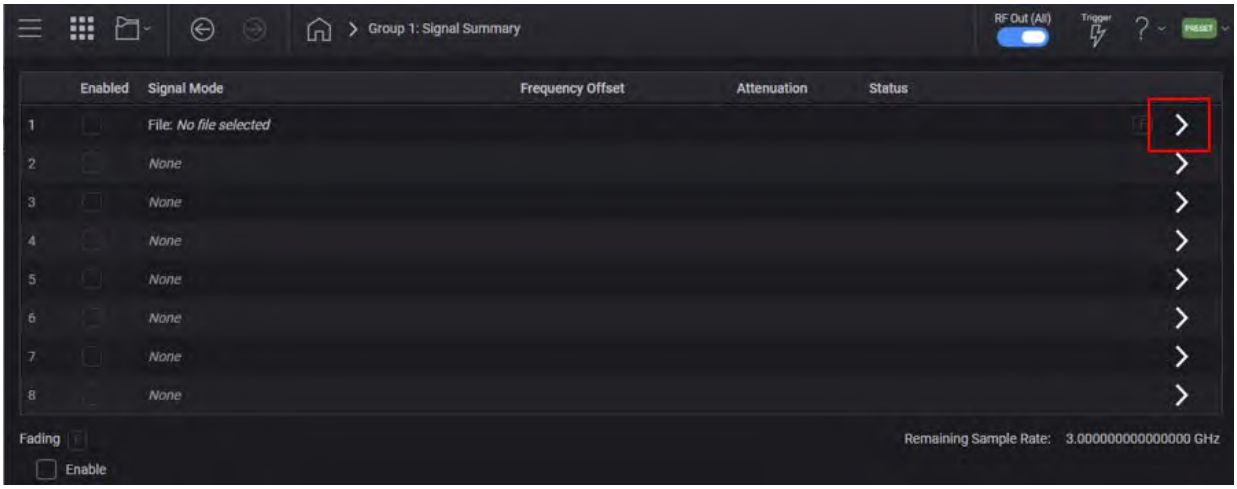
1. Select **Preset** > **Preset** to set the instrument to a known state.
2. In the Output 1 area, set Frequency to **28 GHz** and Power to **5 dBm**.



3. Select **Group 1: Signals** block to open.

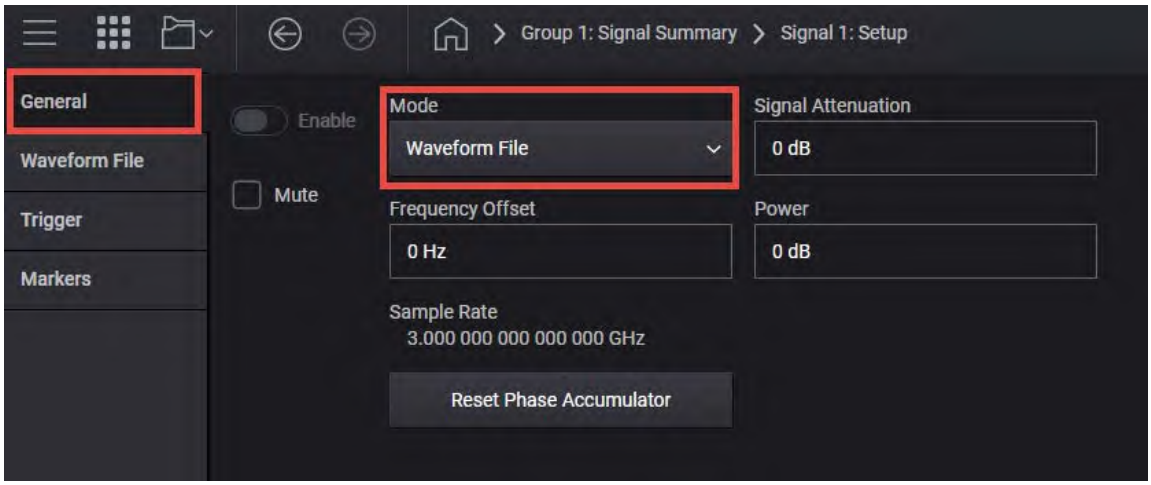
4. Select the arrow for Signal 1 to open the Signal Setup window.

NOTE This screen is only accessible if Option M9484C-8SG (8 virtual signal generators) is installed. For all other option configurations, continue to the next step.



5. In the Vector Modulation Signal Setup:

   a. Select the **Mode** dropdown and set to **Waveform File**.



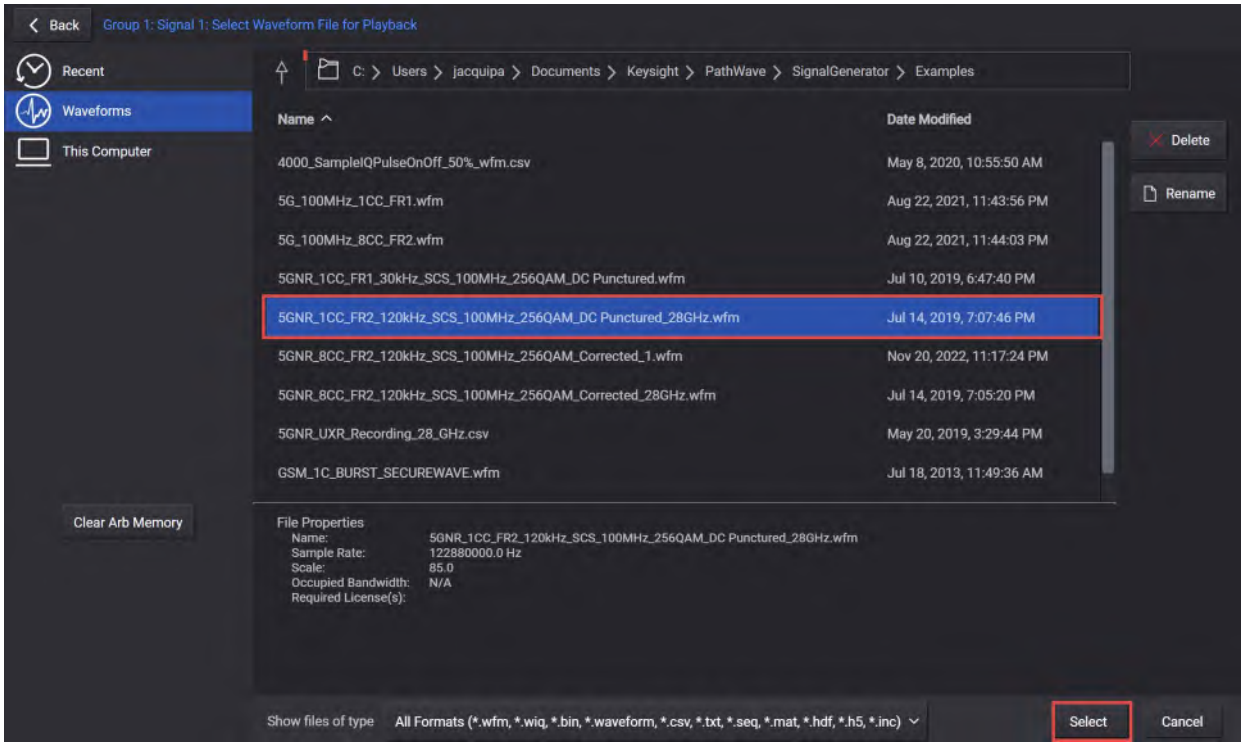
   b. In the left pane, select the **Waveform File** tab.

   c. Use **File Select** to navigate to:

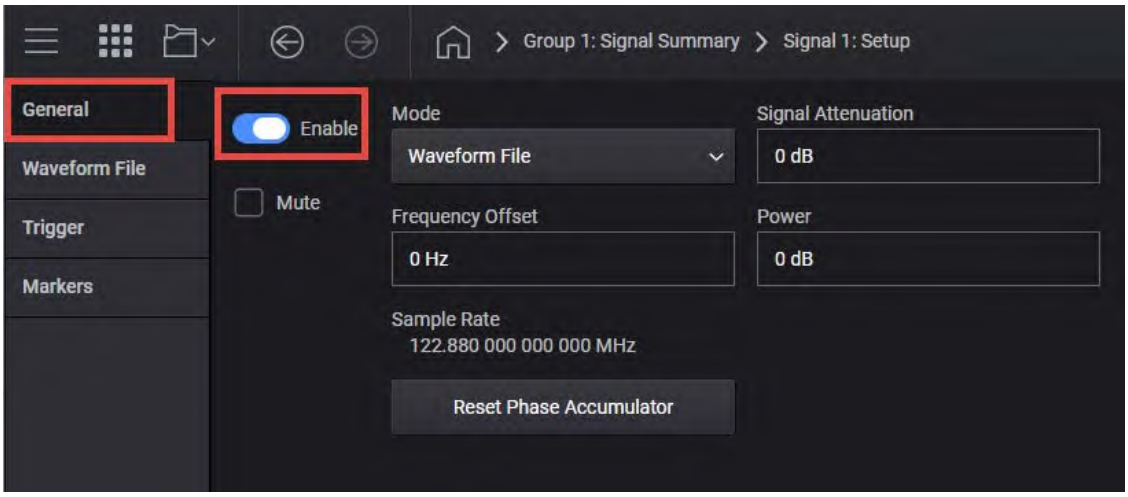
      `D:\Users\Instrument\Documents\Keysight\PathWave\SignalGenerator\Examples`

      and choose

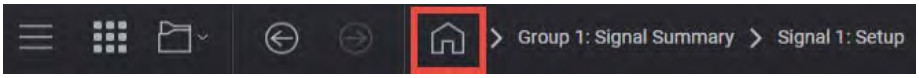
`5GNR_1CC_FR2_120kHz_SCS_100MHz_256QAM_DCPunctured_28GHz.wfm`
then **Select**.



d. Select the **General** tab > **Enable**.



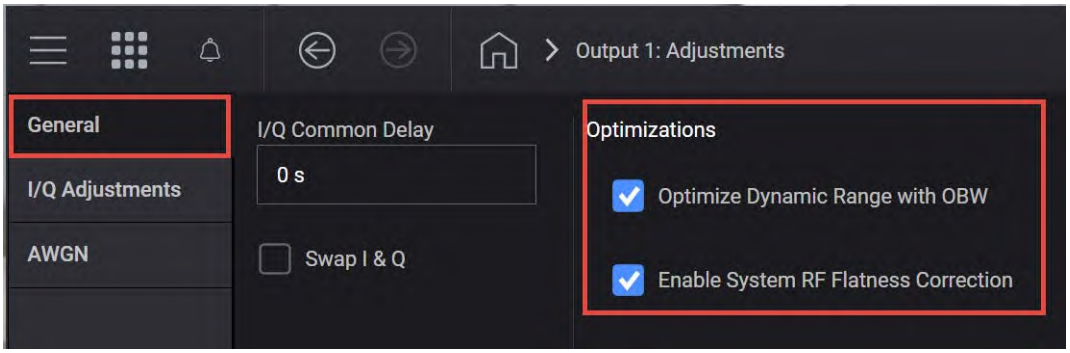
6. Close the Vector Modulation Signal Setup by selecting the **Home** icon at the top of the display.



7. Select the **Adjustments** block and select **Optimize Dynamic Range with OBW** and **Enable System RF Flatness Correction**.

   Optimize Dynamic Range with OBW This setting filters the system RF flatness correction coefficients over the instantaneous bandwidth indicated in the waveform header (or in the “Occupied Bandwidth” settings area under the Signal block > Occupied Bandwidth setting). This has the potential to improve EVM performance by not having to correct for flatness errors outside the requested bandwidth. For example, if RF flatness correction was done at 2 GHz, but you are only interested in an 800 MHz section, then applying the correction flatness to that portion only can improve signal to noise ratio, and therefore EVM when there is a lot of hardware roll off.

   Enable System RF Flatness Correction - Disabling this function disables the factory calibrated RF channel flatness equalizer. Depending on the hardware channel response, this may hurt or improve the EVM. This is due to the dynamic range implications as it relates to signal to noise ratio. The greater the RF hardware variations in flatness, the greater the amount of correction is required, the greater the correction effectively reduces the number of resolution DAC bits that can be used, which degrades the signal to noise ratio and therefore potentially EVM. The trade-off is to balance between flatness and signal to noise ratio.

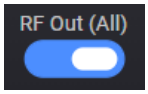


8. Close the Adjustments Setup by either selecting the **Back** or the **Home** icon at the top of the display.

9. Set RF Out to **On** by selecting the numbered channel indicator switch.

   This enables the RF Out for the indicated channels, in this case Channel 1 if using a multi-channel VXG.

10. **For two channel instruments only:** In the top right corner of the display, set RF Out (All) to **On** by selecting the switch.



NOTE

In order to turn on RF for any channel, both the RF Out for the specific channel (for example, Channel 1 or Channel 2), and RF Out All must be turned on.

On the X-Series Signal Analyzer:

– Open the VSA software by selecting **Mode Meas** > **Launch VSA**.

NOTE

If accessing the instrument via a Remote Desktop connection, select the Screen tab (at the top of the display) to open the Mode/Measurement/View Selector window.

In the VSA software:

1. From the menu bar, select **File** > Preset > **Applications and Traces** to set the VSA to a known state.

NOTE

All example waveforms and setup files are located on the VXG at:

```
D:\Users\Instrument\Documents\Keysight\PathWave
\SignalGenerator\Examples
```

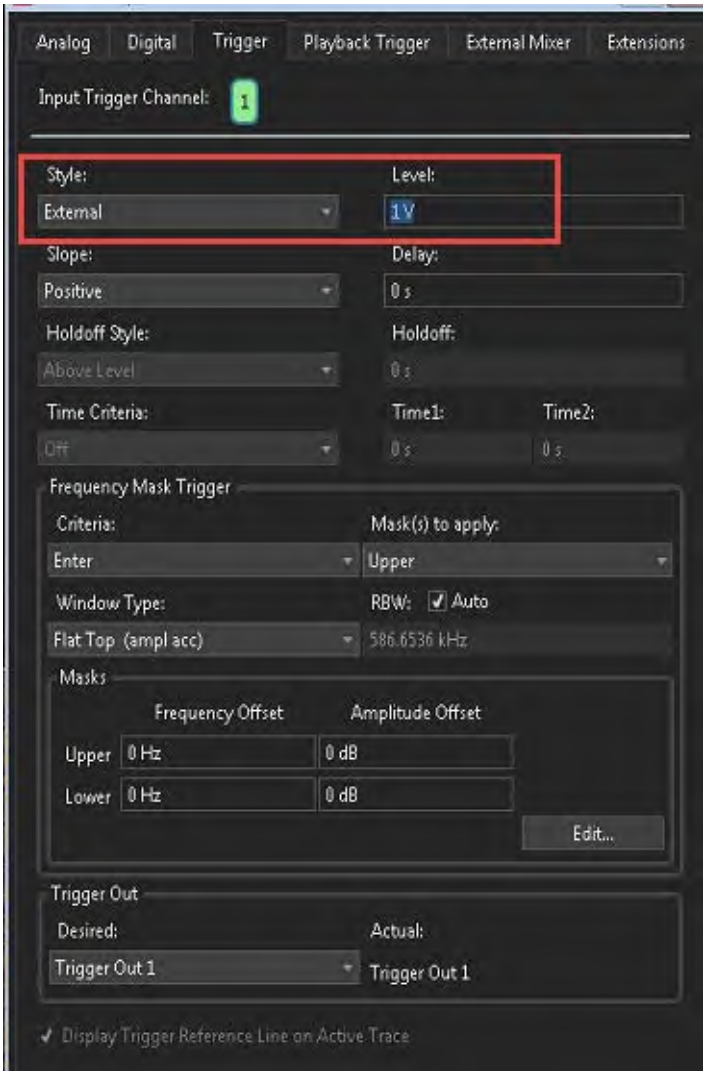
You will need to copy over the setup files to the X-Series Signal Analyzer or the PC running the VSA application.

2. Select **File** > **Recall** > **Recall Setup** and navigate to `5GNR_1CC_FR2_120kHz SCS_100MHz_256QAM_DC Punctured_28GHz.setx`

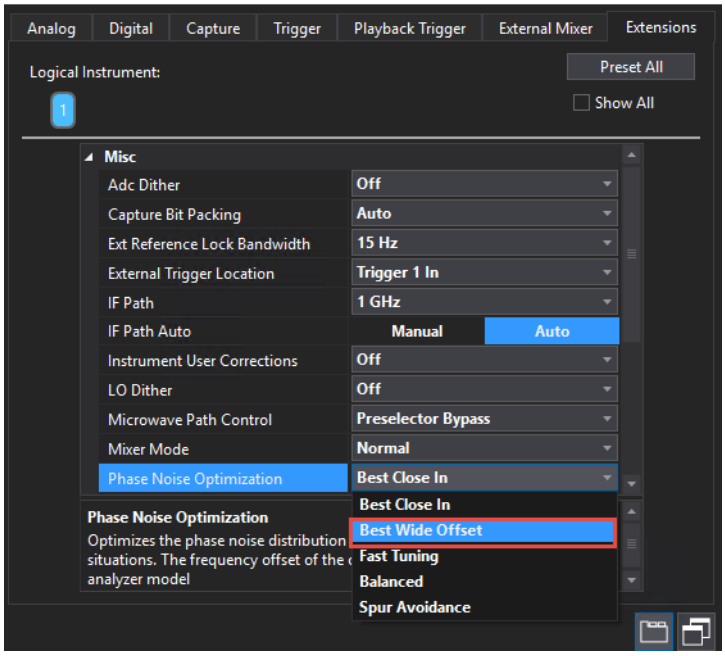
3. From the toolbar, select the **Pause** Icon.

   Pausing the measurement will help to speed up the setup time.

4. From the menu bar, select **Input** > **Trigger** and set Style to **External** and Level to **1.0 V**.

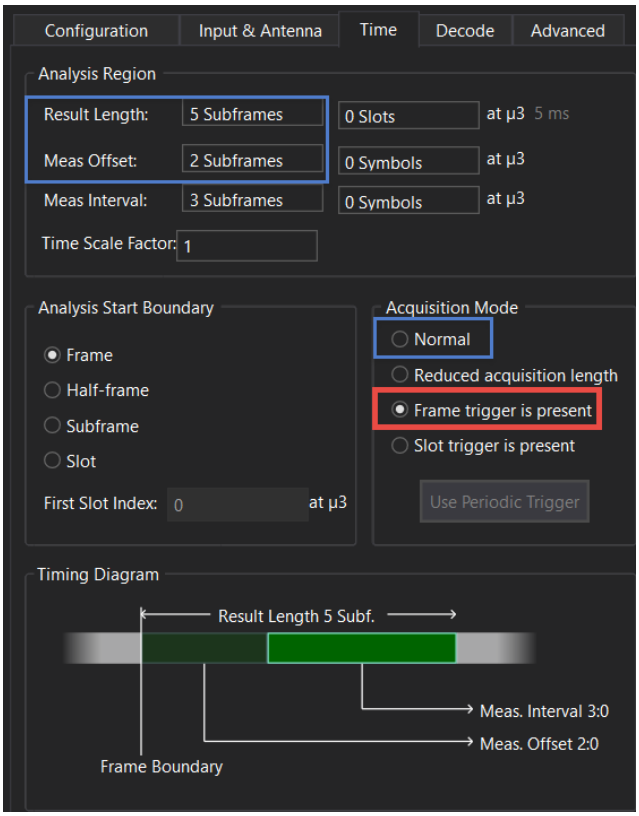


5. Select the **HW-Specific (Extensions)** tab and change the External Trigger location to **Trigger 1 In** (if using an N9040B with Option H1G, select Trigger 3) and Phase Noise Optimization to **Best Wide Offset**.



6. From the menu bar, select **MeasSetup** > **5G NR Demod Properties** > **Time** tab.
   - Set Result Length to **10 Subframes**.
   - Set Meas Interval to **2 Subframes**.

– Select **Frame Trigger is Present**.

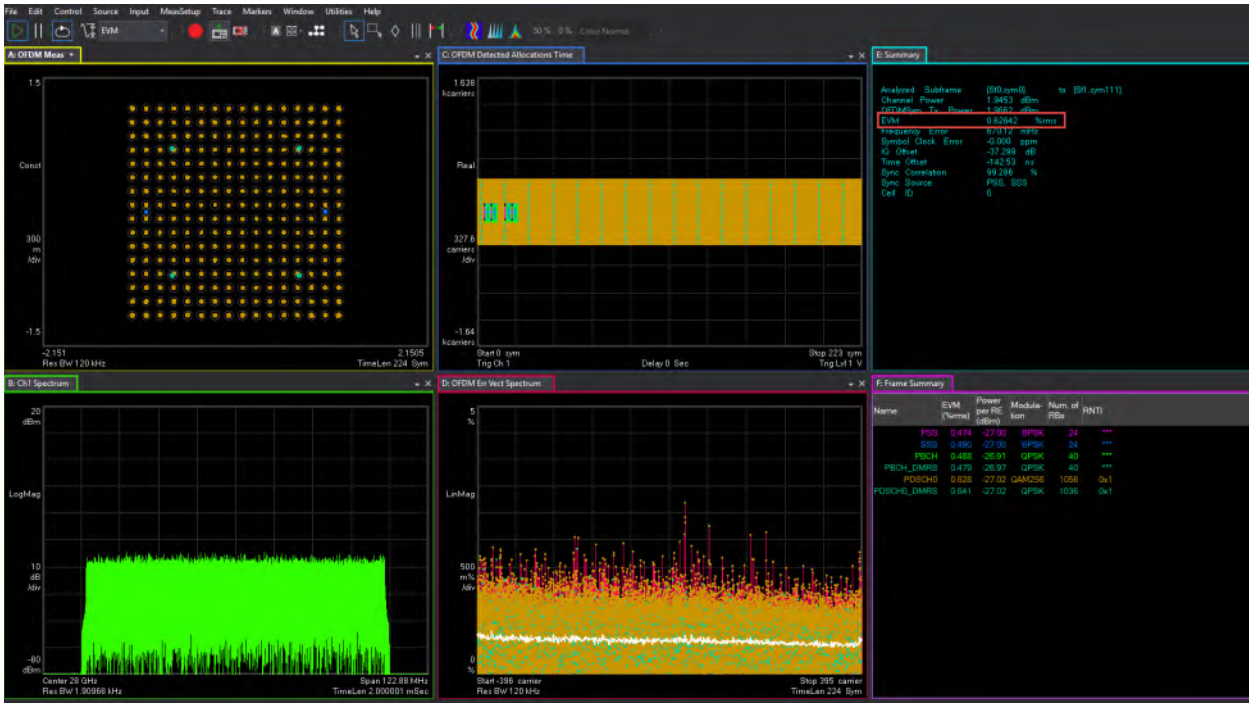


7. From the toolbar, select the **Auto-Range** dropdown and select **EVM-Table or Algorithm Based**.

   – EVM - Table or Algorithm Based performs EVM optimization based on prescribed table/algorithm instead of using EVM in the feedback loop. This method of EVM auto-range is normally faster than Meas Based Iteration, but may not achieve the most optimal setup for minimized EVM).

   – EVM - Meas Based Iteration has a feedback loop around the entire measurement, uses the measured EVM in the feedback loop, and tries to adjust hardware parameters to minimize the EVM. It is the slowest EVM auto-range method, but it should achieve the most optimal setup for good EVM.

8. Select the **Auto-Range** icon to run the measurement for EVM optimization. This may take a few minutes to complete.


File Edit Control Source Input MeasSetup Trace Markers Window Utilities Help
EVM
A: OFDM Meas
C: OFDM Detected Allocations Time
E: Summary
Analyzed Subframe
Channel Power
EVM
Frequency Error
Symbol Clock Error
IQ Offset
Time Offset
Sync Correlation
Sync Source
Cell ID
B: Ch1 Spectrum
D: OFDM Err Vect Spectrum
F: Frame Summary
Name
EVM (%rms)
Power per RE (dBm)
Modulation
Num. of RBs
RNTI
PSS
SSS
PBCH
PBCH_DMRS
PDSCH0
PDSCH0_DMRS

Using the equivalent SCPI commands

On the VXG:

```
SYSTem:PRESet

RF1:FREQuency:CW 28GHZ

RF1:POWer:AMPLitude 5dBm

SIGNal1:MODE WAVeform

SIGNal1:WAVeform "D:\Users\Instrument\Documents\MCS demo
waveforms
5GNR_1CC_FR2_120kHz_SCS_100MHz_256QAM_DCPunctured_28GHz.wfm"

SIGNal1 ON

IQO:CORR:OPT:DYN:RANG:OBW ON

IQO:CORR:CHAN:FLAT ON

RF1:OUTPut ON
```

For multi-channel instruments, set RF Out (all) to On.

```
RFALl:OUTPut ON
```

On the X-Series Signal Analyzer:

```
INSTrument:SELect VSA89601
```

NOTE

All example waveforms and setup files are located on the VXG at:

`D:\Users\Instrument\Documents\Keysight\PathWave \SignalGenerator\Examples`

You will need to copy over the setup files to the X-Series Signal Analyzer or the PC running the VSA application.

```
SYSTem:PRESet

MMEMory:LOAD
"D:Users\Instrument\Documentts\NR5G\data\NR5GEvm\Carrier
Setup\5GNR_1CC_FR2_120kHz
SCS_100MHz_256QAM_DCPunctured_28GHz.setx"

INITiate:PAUSe

INPut:TRIGger:STYle "EXTERNAL"

INPut:TRIGger:LEVel:EXTernal 1V

INPut:EXTension:PARameters:SET "ExtTriggerLoc", 2

INPut:EXTension:PARameters:SET "PhaseNoiseOptDualLoop", 1
```

Set the 5G NR Demod Result Length to 10 Subframes:

```
NR5G:RLENgth 10
NR5G:SUBFrame:INTerval 2
```

Set the Acquisition Mode to “Frame Trigger is Present”

```
nr5g:FRAMe:TRIGger:ENABled 1
```

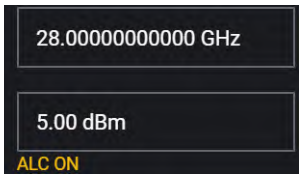
Setting Up an 8 CC 28 GHz EVM Measurement

NOTE Ensure the equipment and triggers are properly configured. Refer to "Equipment Setup" on page 13.

Using the graphical user interface

On the VXG:

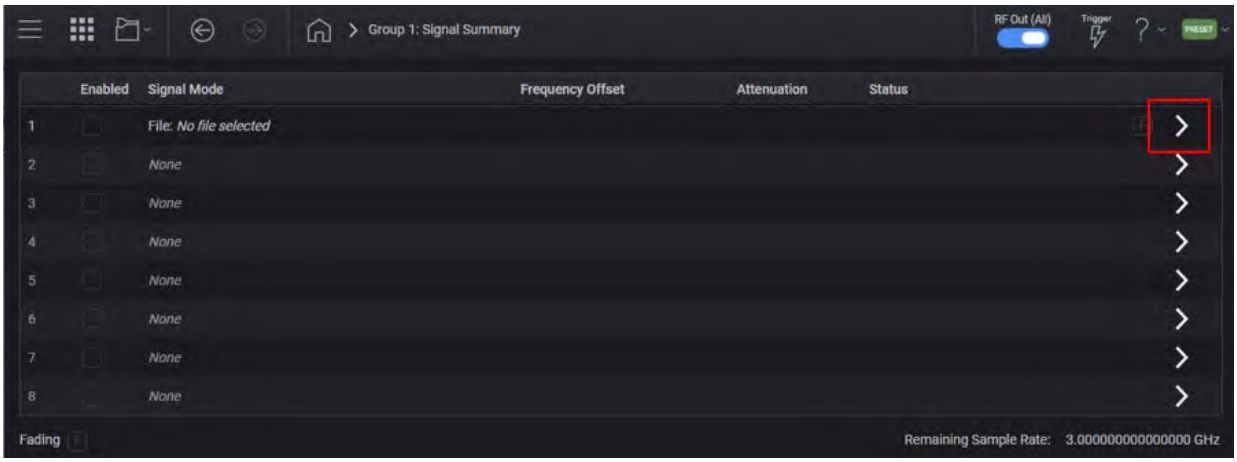
1. Select **Preset** > **Preset** to set the VXG to a known state.
2. Set Frequency to **28 GHz** and Amplitude to **5 dBm**.



3. Select **Group 1: Signals** block to open.

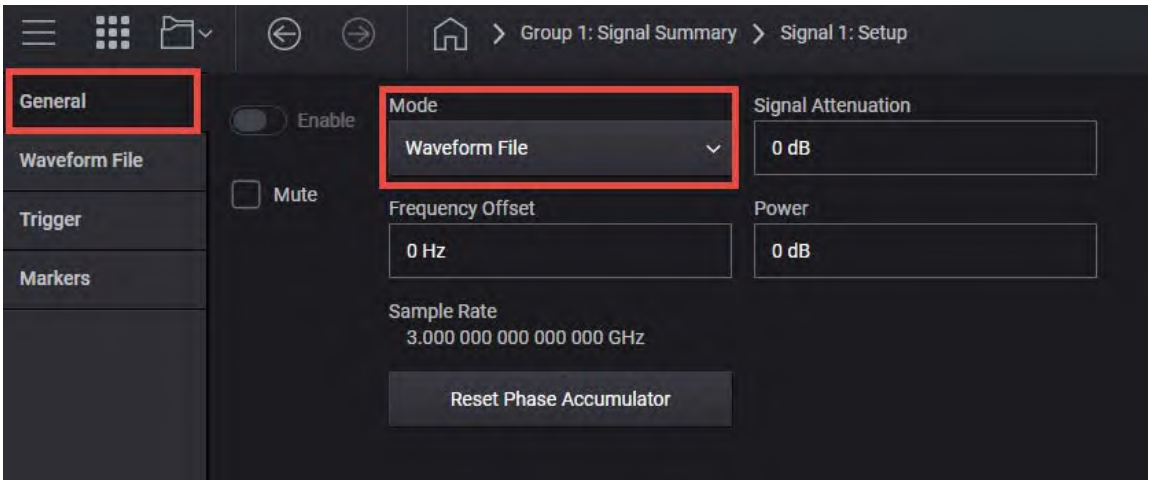
4. Select the arrow for Signal 1 to open the Signal Setup window.

NOTE This screen is only accessible if Option M9484C-8SG (8 virtual signal generators) is installed. For all other option configurations, continue to the next step.



5. In the Vector Modulation Signal Setup:

   a. Select the **Mode** dropdown and set to **Waveform File**.

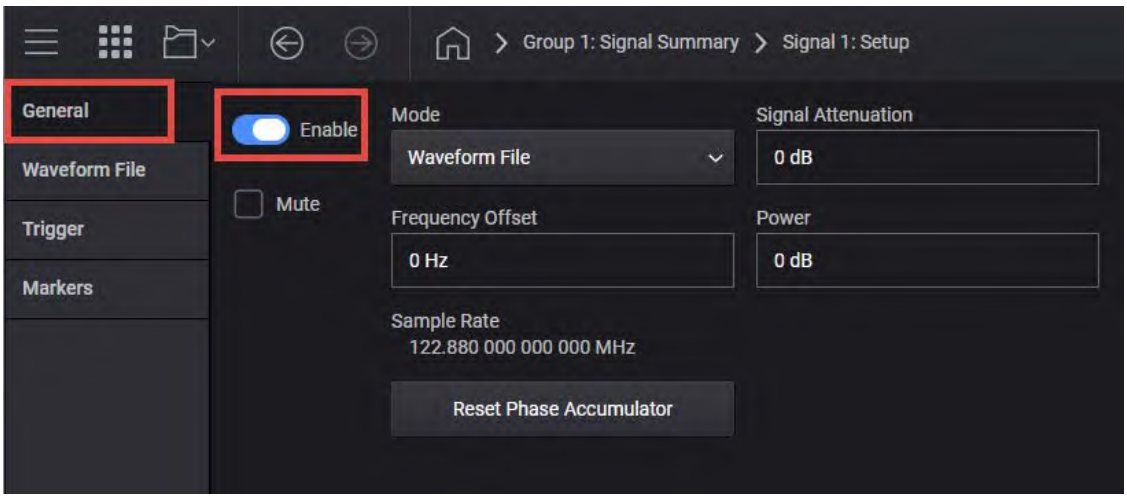


   b. In the left pane, select the **Waveform File** tab.

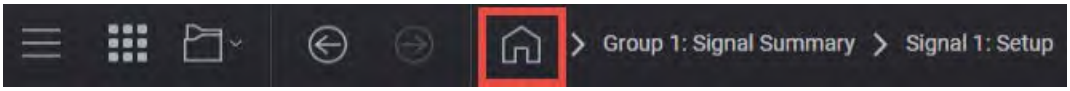
   c. Use **File Select** to navigate to:

      `D:\Users\Instrument\Documents\Keysight\PathWave\SignalGenerator\Examples`
      and choose
      `5GNR_8CC_FR2_120kHz_SCS_100MHz_256QAM_Corrected_28GHz.wfm`
      then **Select**.

   d. Select the **General** tab > **Enable**.



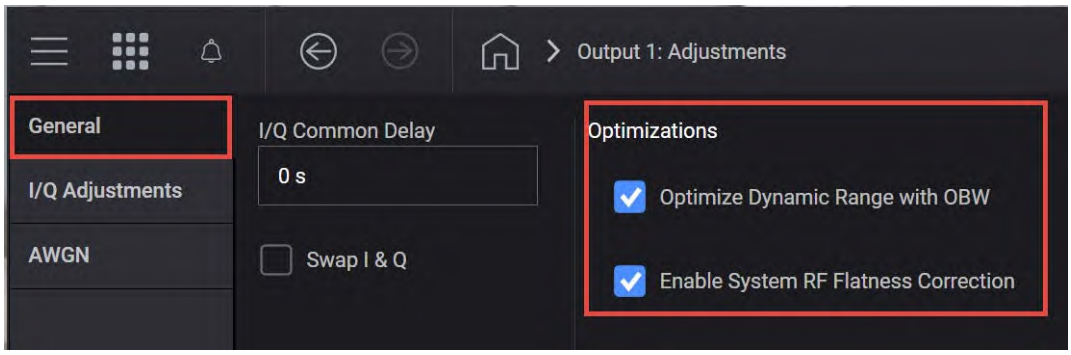
6. Close the Signal Setup by selecting the **Home** icon at the top of the display.



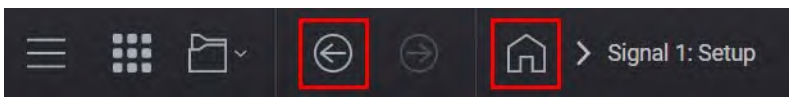
7. Select the **Adjustments** block and select **Optimize Dynamic Range with OBW** and **Enable System RF Flatness Correction**.

Optimize Dynamic Range with OBW This setting filters the system RF flatness correction coefficients over the instantaneous bandwidth indicated in the waveform header (or in the "Occupied Bandwidth" settings area under the Signal block > Occupied Bandwidth setting). This has the potential to improve EVM performance by not having to correct for flatness errors outside the requested bandwidth. For example, if RF flatness correction was done at 2 GHz, but you are only interested in an 800 MHz section, then applying the correction flatness to that portion only can improve signal to noise ratio, and therefore EVM when there is a lot of hardware roll off.

Enable System RF Flatness Correction - Disabling this function disables the factory calibrated RF channel flatness equalizer. Depending on the hardware channel response, this may hurt or improve the EVM. This is due to the dynamic range implications as it relates to signal to noise ratio. The greater the RF hardware variations in flatness, the greater the amount of correction is required, the greater the correction effectively reduces the number of resolution DAC bits that can be used, which degrades the signal to noise ratio and therefore potentially EVM. The trade-off is to balance between flatness and signal to noise ratio.



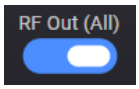
8. Close the Adjustments Setup by either selecting the **Back** or the **Home** icon at the top of the display.



9. Set RF Out to **On** by selecting the numbered channel indicator switch.

This enables the RF Out for the indicated channels, in this case Channel 1 if using a multi-channel VXG.

10. **For multi-channel instruments only:** In the top right corner of the display, set RF Out (All) to **On** by selecting the switch.



NOTE

In order to turn on RF for any channel, both the RF Out for the specific channel (for example, Channel 1 or Channel 2), and RF Out All must be turned on.

On the X-Series Signal Analyzer:

– Open the VSA software by selecting **Mode Meas** > **Launch VSA**.

NOTE

If accessing the instrument via a Remote Desktop connection, select the Screen tab (at the top of the display) to open the Mode/Measurement/View Selector window.

In the VSA software:

1. From the menu bar, select **File** > **Preset** > **Applications and Traces** to set the VSA to a known state.

NOTE

All example waveforms and setup files are located on the VXG at:

`D:\Users\Instrument\Documents\Keysight\PathWave \SignalGenerator\Examples`

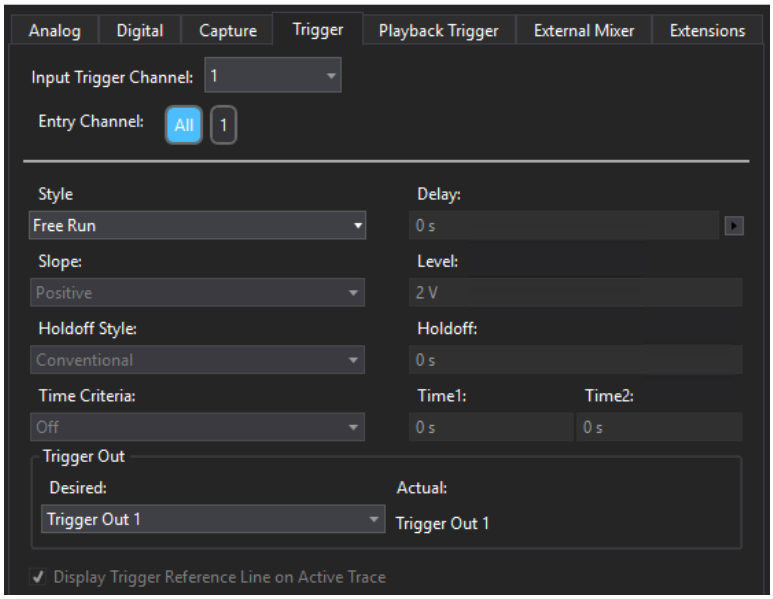
You will need to copy over the setup files to the X-Series Signal Analyzer or the PC running the VSA application.

2. Select **File** > **Recall** > **Recall Setup** and navigate to D:Users\Instrument\Documents\NR5G\data\NR5GEvm\Carrier Setup, then open `5GNR_8CC_FR2_120kHz_SCS_100MHz_256QAM_Corrected_28GHZ_2026.setx`

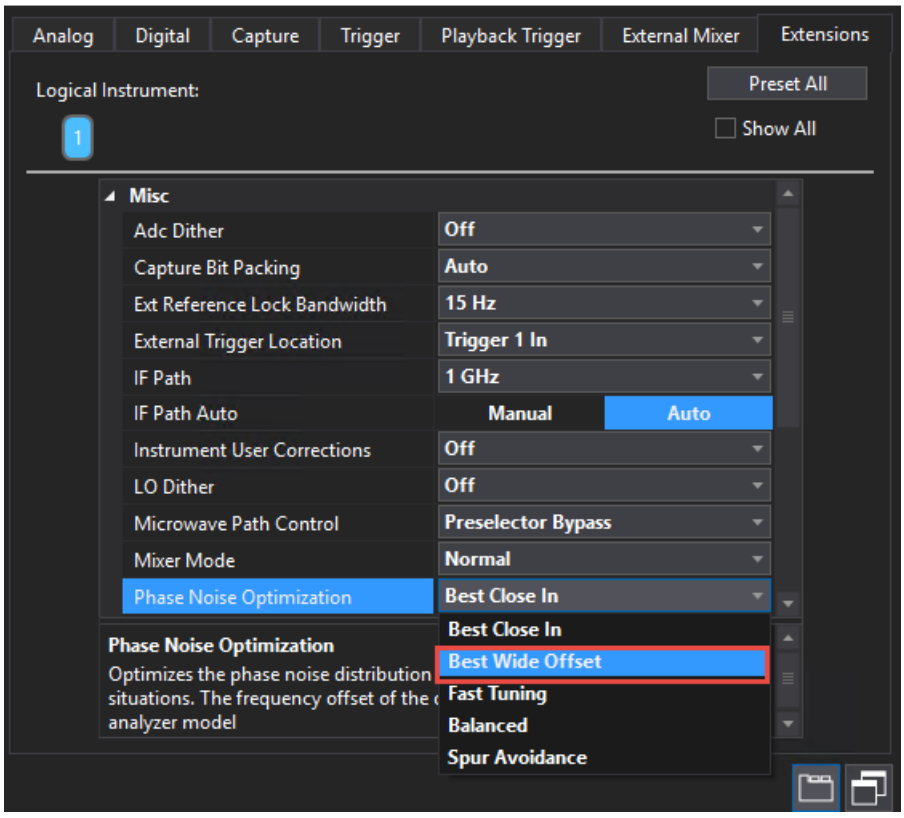
3. From the toolbar, select the **Pause** Icon.

   Pausing the measurement will help to speed up the setup time.

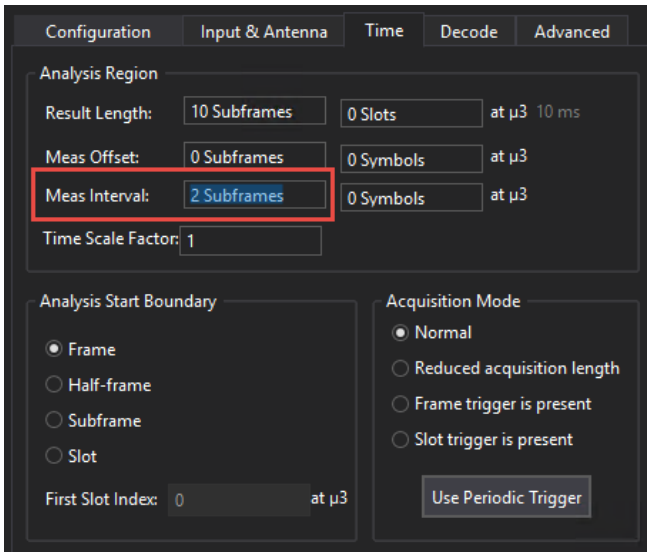
4. From the menu bar, select **Input** > **Trigger** and set Style to **Free Run**.



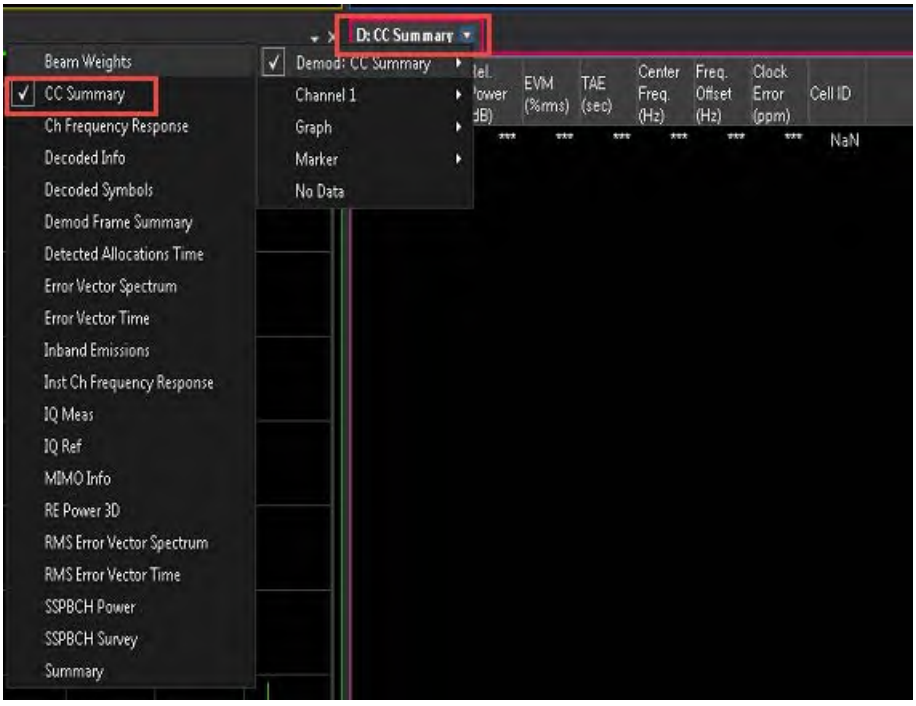
5. Select the **HW-Specific (Extensions)** tab and change Phase Noise Optimization to **Best Wide Offset**. (For an N9040B only, set External Trigger Location to T**rigger 3 In**.)



6. From the menu bar, select **MeasSetup** > **5G NR/5G Demod Properties** > **Time** tab.
   - Set Result Length to **10 Subframes**.
   - Set Meas Interval to **2 Subframes**.



7. In the bottom center trace window, select the Trace Data menu and select **Demod:** > **CC Summary**.

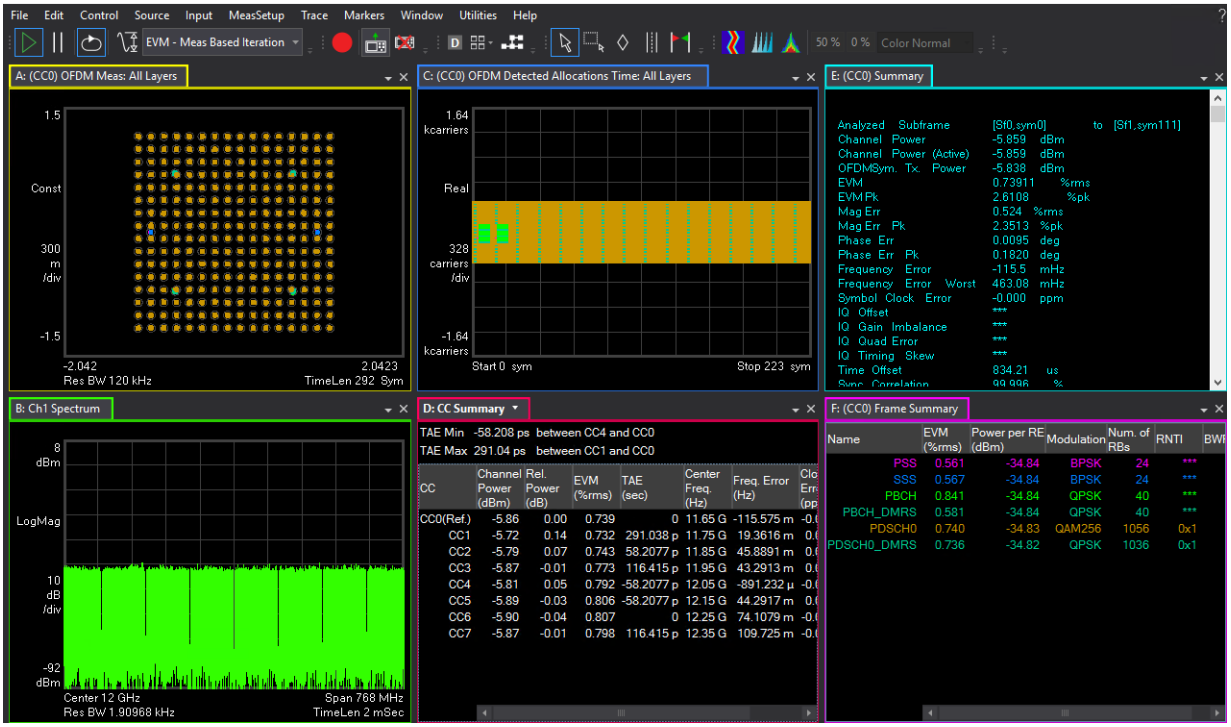


8. From the toolbar, select the **Autorange** dropdown and select **EVM-Table or Algorithm Based**.

– EVM - Table or Algorithm Based performs EVM optimization based on prescribed table/algorithm instead of using EVM in the feedback loop. This method of EVM auto-range is normally faster than Meas Based Iteration, but may not achieve the most optimal setup for minimized EVM).

– EVM - Meas Based Iteration has a feedback loop around the entire measurement, uses the measured EVM in the feedback loop, and tries to adjust hardware parameters to minimize the EVM. It is the slowest EVM auto-range method, but it should achieve the most optimal setup for good EVM.

9. Select the **Autorange** icon to run the measurement for EVM optimization on all eight channels. This may take a few minutes to complete.



CC	Channel Power (dBm)	Rel. Power (dB)	EVM (%rms)	TAE (sec)	Center Freq. (Hz)	Freq. Error (Hz)	Clo Err (pp
CC0(Ref.)	-5.86	0.00	0.739	0	11.65 G	-115.575 m	-0.(
CC1	-5.72	0.14	0.732	291.038 p	11.75 G	19.3616 m	0.(
CC2	-5.79	0.07	0.743	58.2077 p	11.85 G	45.8891 m	0.(
CC3	-5.87	-0.01	0.773	116.415 p	11.95 G	43.2913 m	0.(
CC4	-5.81	0.05	0.792	-58.2077 p	12.05 G	-891.232 µ	-0.(
CC5	-5.89	-0.03	0.806	-58.2077 p	12.15 G	44.2917 m	0.(
CC6	-5.90	-0.04	0.807	0	12.25 G	74.1079 m	-0.(
CC7	-5.87	-0.01	0.798	116.415 p	12.35 G	109.725 m	-0.(

Using the equivalent SCPI commands

On the VXG:

```
SYSTem:PRESet

RF1:FREQuency:CW 28GHZ

RF1:POWer:AMPLitude 5dBm

SIGNal1:MODE WAVeform

SIGNal1:WAVeform "D:\Users\Instrument\Documents\MCS demo
waveforms\5GNR_8CC_FR2_120kHz_SCS_100MHz_Corrected_28GHz.wfm
"

SIGNal1 ON


IQO:CORR:OPT:DYN:RANG:OBW ON

IQO:CORR:CHAN:FLAT ON

RF1:OUTPut ON
```

For multi-channel instruments, set RF Out (all) to On.

```
RFALl:OUTPut ON
```

On the X-Series Signal Analyzer

```
INSTrument:SELect VSA89601

MMEMory:LOAD:DEMO
"D:\Users\Instrument\Documents\NR5G\data\NR5GEvm\CarrierSetu
p\1CC_FR2_120kHz SCS_100MHz_256QAM_DC Punctured_28GHz.setx"

INITiate:PAUSe

INPut:TRIGger:STYLe "External"

INPut:TRIGger:LEVel:EXTernal 1V

INPut:EXTension:PARameters:SET "ExtTriggerLoc", 2

INPut:EXTension:PARameters:SET "PhaseNoiseOptDualLoop", 1

NR5G:RLENgth 10

NR5G:SUBFrame:INTerval 2

NR5G:FRAMe:TRIGger:ENABled 1

NR5G:CAGGregation:CONFigure "Contiguous8CC"

NR5G:DC:PUNCtured 1

NR5G:MCFilter:ENABled 1
```

```
FREQuency:CENTer 28 GHz

INPut:ANALog:CRITeria:RANGe:AUTO "EVM", -1
```

NOTE

All example waveforms and setup files are located on the VXG at:

```
D:\Users\Instrument\Documents\Keysight\PathWave
\SignalGenerator\Examples
```

You will need to copy over the setup files to the X-Series Signal Analyzer or the PC running the VSA application.

Creating a Basic 5G NR Signal Using PathWave N7631APPC Embedded Software

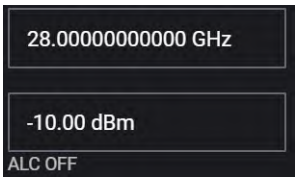
PathWave software tools can be used to create, download, and playback waveforms through the VXG.

This example shows you how to create and analyze a 5G NR signal using the embedded PathWave software. Your VXG must include the N7631APPC license.

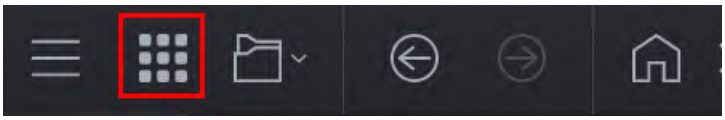
NOTE The VXG must have the N7631APPC Signal Generation for 5G NR license installed.

On the VXG:

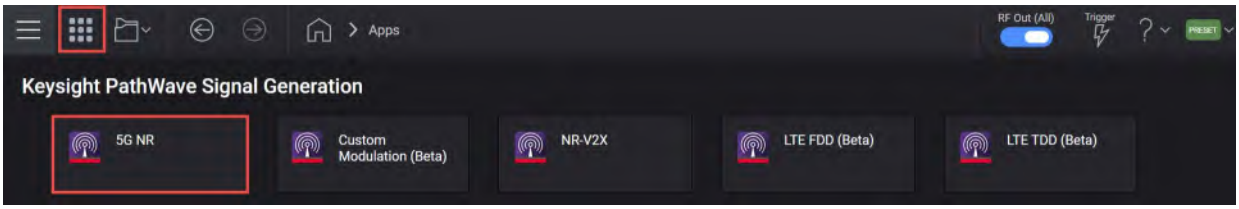
1. Select **Preset** > **Preset** to set the VXG to a known state.
2. In the Output area, set Frequency to **28 GHz** and Power to **-10 dBm**.



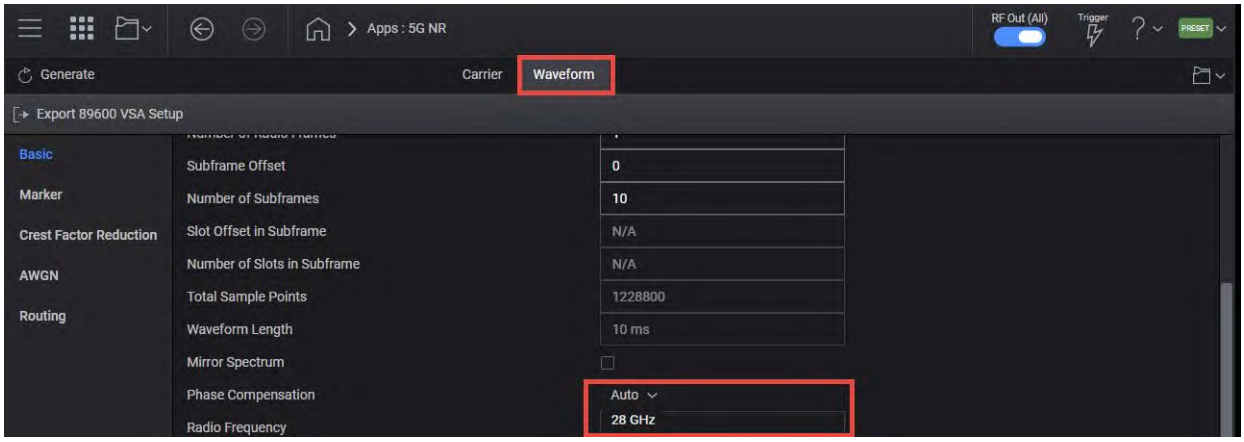
3. Select the **Radio Apps** block to open the Vector Modulation Signal Setup panel.



4. Select **5G NR/5G-Advanced** to enter 5GNR Signal Mode.



5. In the 5G NR setup, select the **Waveform** tab and observe the Phase Compensation and Radio Frequency settings.



TIP

Phase Compensation is applied by default at baseband for RF up-conversion. This means the waveform generation is per the carrier frequency, unlike conventional Signal Studio waveforms, which are independent from carrier frequency. Phase Compensation is set to Auto by default.

Some applications, like power amplifier measurements use the same waveform at different frequencies and can be time consuming to generate a separate waveform for each frequency. In this case, you would turn Phase Compensation Off, so the waveform becomes independent from the carrier frequency. You need to do the same on the analysis side, or the demodulation will fail.

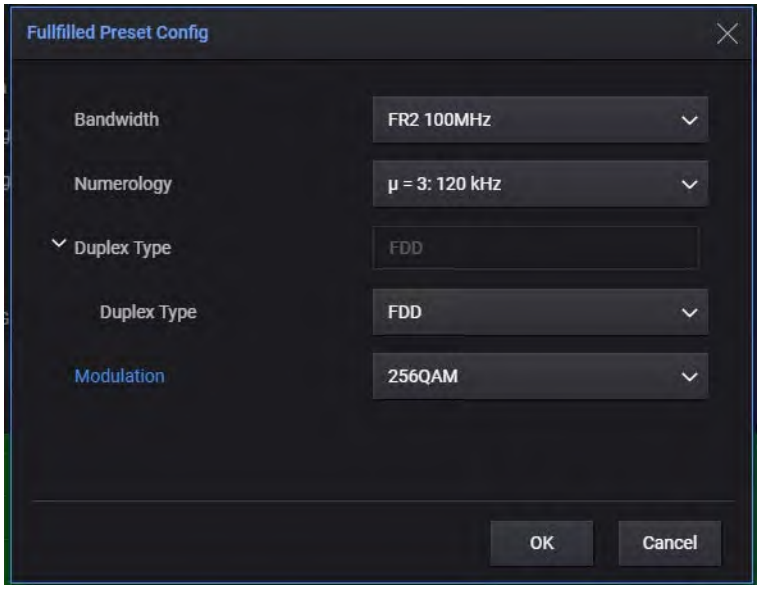
6. Select the **Carriers** tab.
7. Select the **Full-Filled Config** tab and set Bandwidth to **FR2 100 MHz**, Numerology to **μ = 3: 120 kHz**, and Modulation to **256QAM**.

NOTE

If you have a signal analyzer with demodulation bandwidth that is wide enough to cover the other FR2 bandwidths, 200 MHz or 400 MHz, you can choose to use a wider bandwidth.

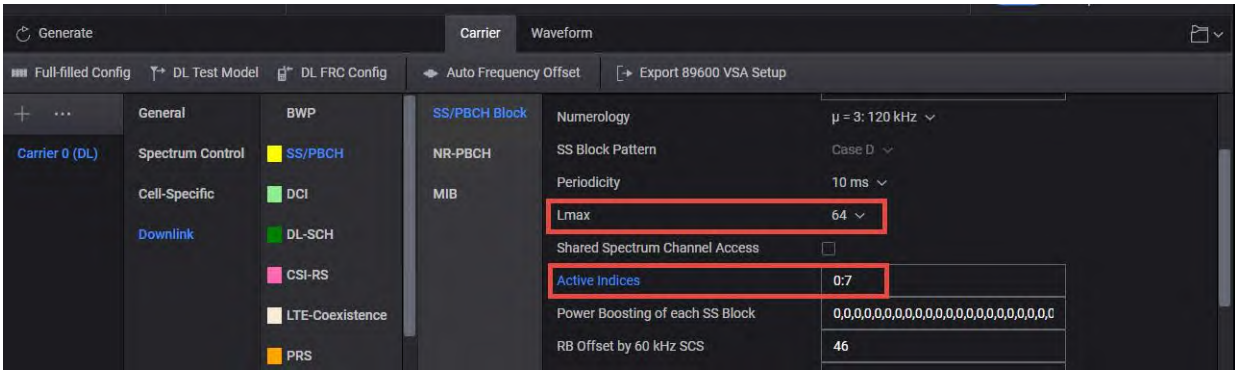
Full Filled Configurations is a great place to start creating your waveform. You can select FR, bandwidth, and modulation format with a fully allocated PDSCH data channel. This would represent a spectrally correct

signal that is typically used in power amplifier measurements. You can modify the RB allocations and Slot allocations if fully allocated data channels are not desired.



8. Select **OK** to exit the Full Filled Config setup.
9. From the Carrier panel (in the left pane), select **Downlink** > **SS PBCH** > **SS PBCH Block** and change Lmax to **64** and Active Indices to = **0:7**.

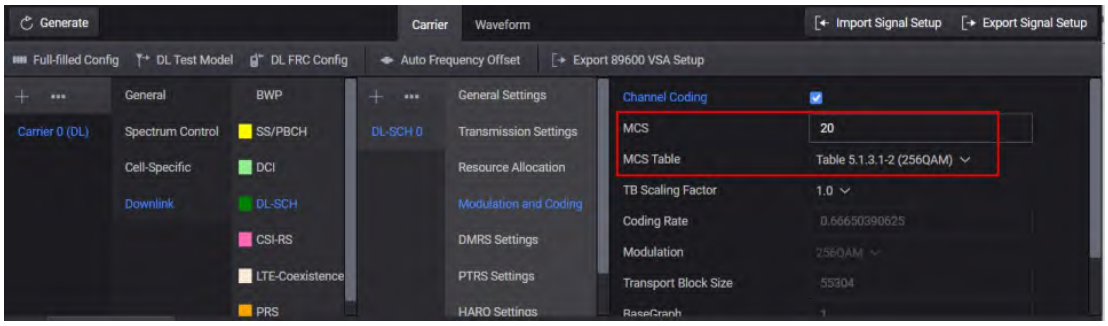
   Instead of transmitting all 64 beams, we enable only 8 of them be setting Active Indices to 0:7.



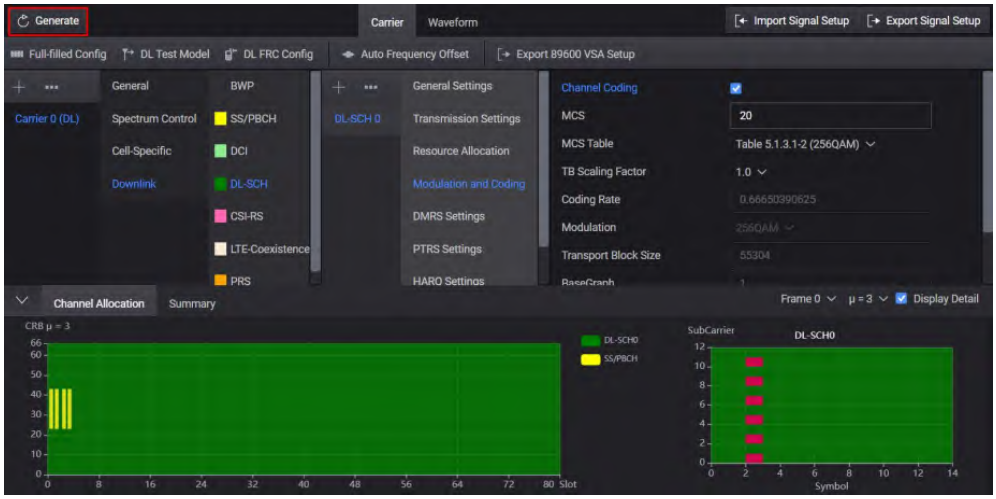
NOTE

Lmax sets the maximum number of possible SS/PBCH blocks in a transmission opportunity. This value is fixed for a given carrier bandwidth. Typically, higher frequency and bandwidth carriers use a larger set of SS/PBCH opportunities to support a larger set of antenna beams.Lmax is either 4 or 8 for FR1 and 64 for FR2.

10. From the Carrier panel (in the left pane), select **Downlink** > **DL-SCHs** > **DL-SCH0** > **Modulation and Coding** and verify that MCS Table is set to **Table 5.1.3.1-2 (256QAM)** and MCS to **20**.



11. Select **Generate** to generate the Waveform.



12. Select the Home icon to return to the main window and set RF Out to **On** by selecting the numbered channel indicator switch.

    This enables the RF Out for the indicated channels, in this case Channel 1 if using a multi-channel VXG.

On the X-Series Signal Analyzer:

– Open the VSA software by selecting **Mode Meas** > **Launch VSA**.

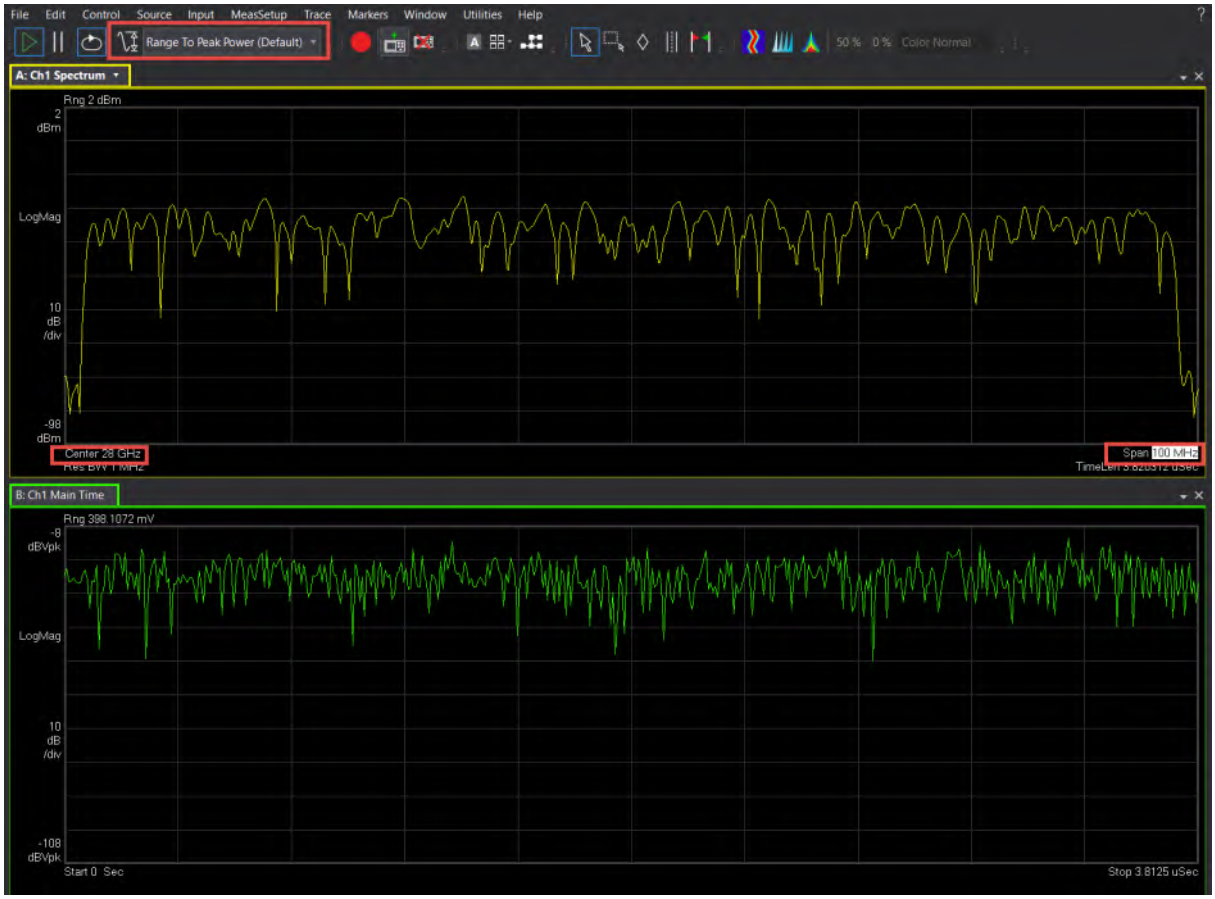
NOTE

If accessing the instrument via a Remote Desktop connection, select the Screen tab (at the top of the display) to open the Mode/Measurement/View Selector window.

In the VSA software:

1. From the menu bar, select **File** > **Preset** > **Applications and Traces** to set the VSA to a known state.
2. Set the Center Frequency to **28 GHz** > Span to **100 MHz**, then select **Auto Range**.

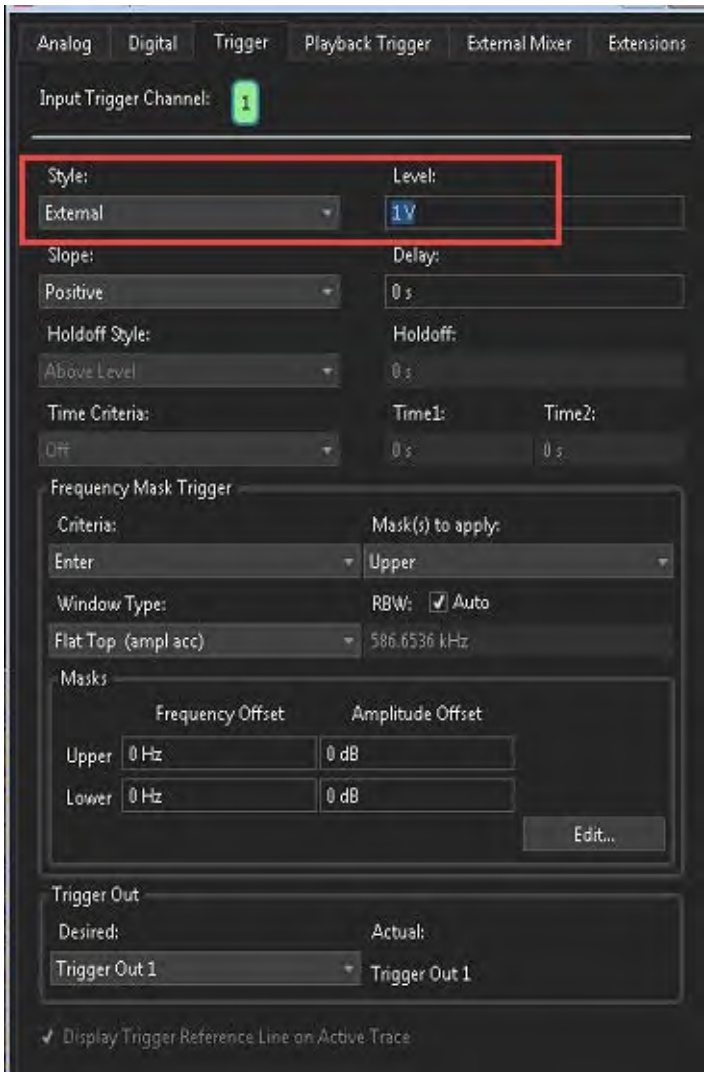
   Auto Range samples the current input signal and then sets the full scale input range to the minimum range that includes the peak voltage sample of the input signal.



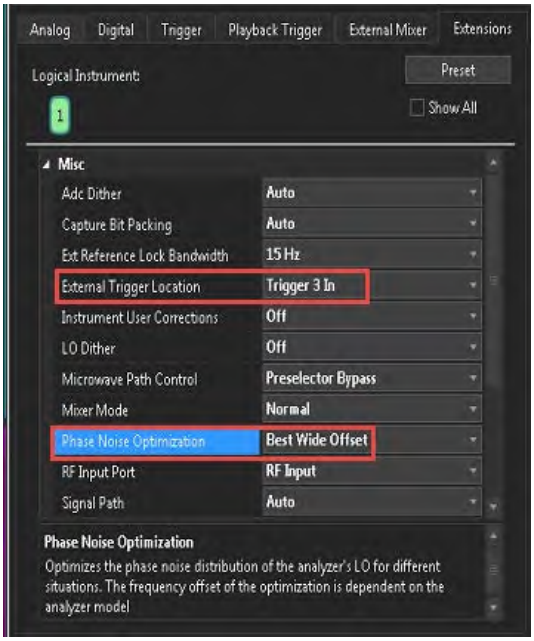
3. From the toolbar, select the **Pause** Icon.

TIP Pausing the measurement will help to speed up the setup time.

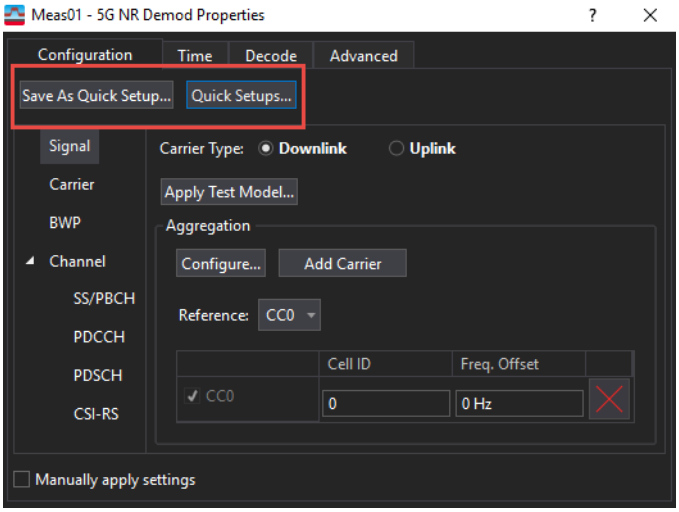
4. From the menu bar, select **Input** > **Trigger** and set Style to **External** and Level to **1.0 V**.



5. Select the **HW-Specific Extensions** tab and change the External Trigger location to **Trigger 1 In** or **Trigger 3 In** (Use Trigger 3 if your N9040B has Option H1G, 1 GHz BW) and set Phase Noise Optimization to **Best Wide Offset**.



6. From the menu bar, select **MeasSetup** > **Measure Type Cellular** > **5G NR** > **5G NR Modulation Analysis**.
7. From the menu bar, select **MeasSetup** > **5G NR Demod Properties** > **Configuration** tab.

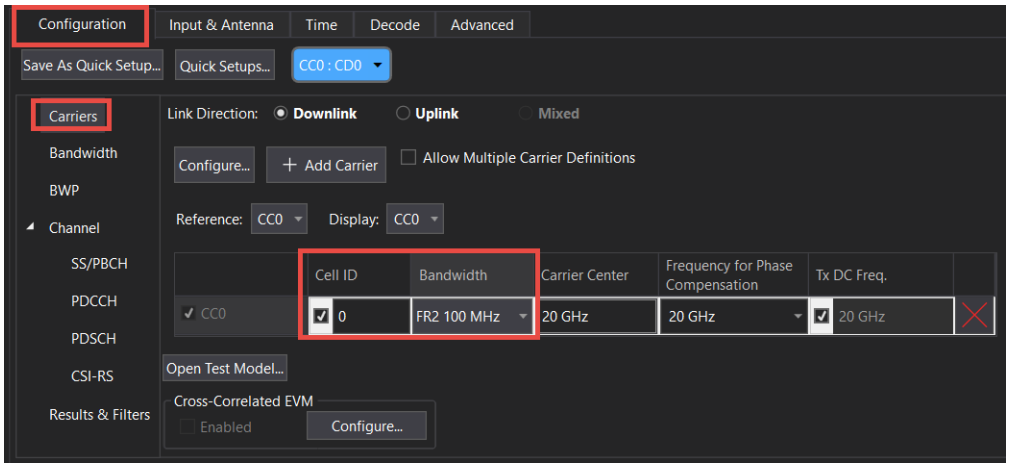


TIP Use Quick Setups for convenient saving and loading of common configurations. Quick Setups are factory supplied configurations and cannot be deleted by users. (This includes Signal Studio Downlink and Uplink defaults.)

Save As Quick Setup (saved by the user) can be specific to the current user or can be made available to others users. These setups can also be deleted.

8. Select the **Carriers** panel.

   The Carrier dialog is used to configure component carrier level parameters: Cell ID, Bandwidth, and Resource Grid for each Numerology.



   – Set Cell ID to **0**

     When the checkbox is selected, Cell ID auto-detection is enabled for the carrier.

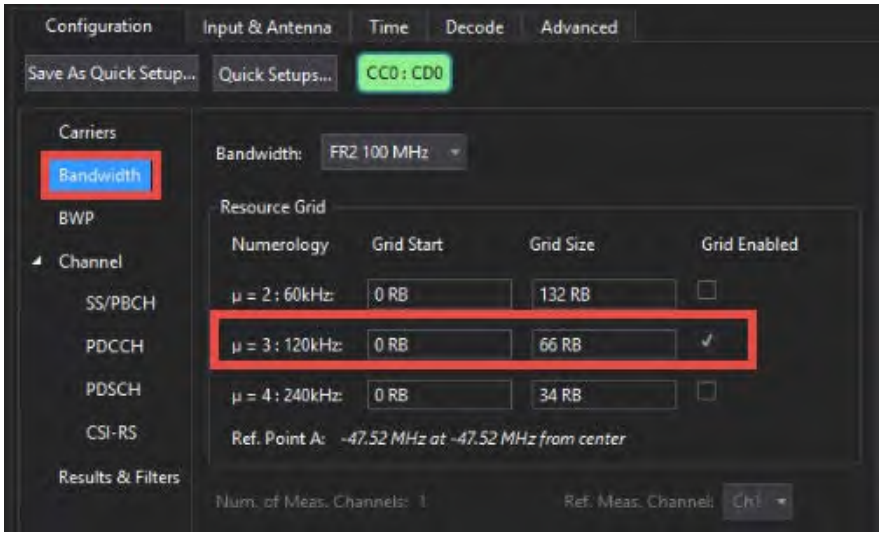
NOTE Cell ID is carried on PSS and SSS so the SS/PBCH must be transmitted and enabled for Auto Cell ID to work. It must also be a downlink since SS/PBCH is only transmitted on a downlink.

   – Set Bandwidth to **FR2 100 MHz**

9. Select the **Bandwidth** panel and set Numerology to **$\mu = 3$: 120 kHz**

   For FR2 100 MHz, the Max RB for 120 kHz numerology is 66 RB. This value will be used when we configure BWP, SS/PBCH, and PDSCH.

FR2 only uses 60, 120, or 240 kHz numerologies. This is why 15 and 30 kHz numerologies do not show under the Resource Grid.

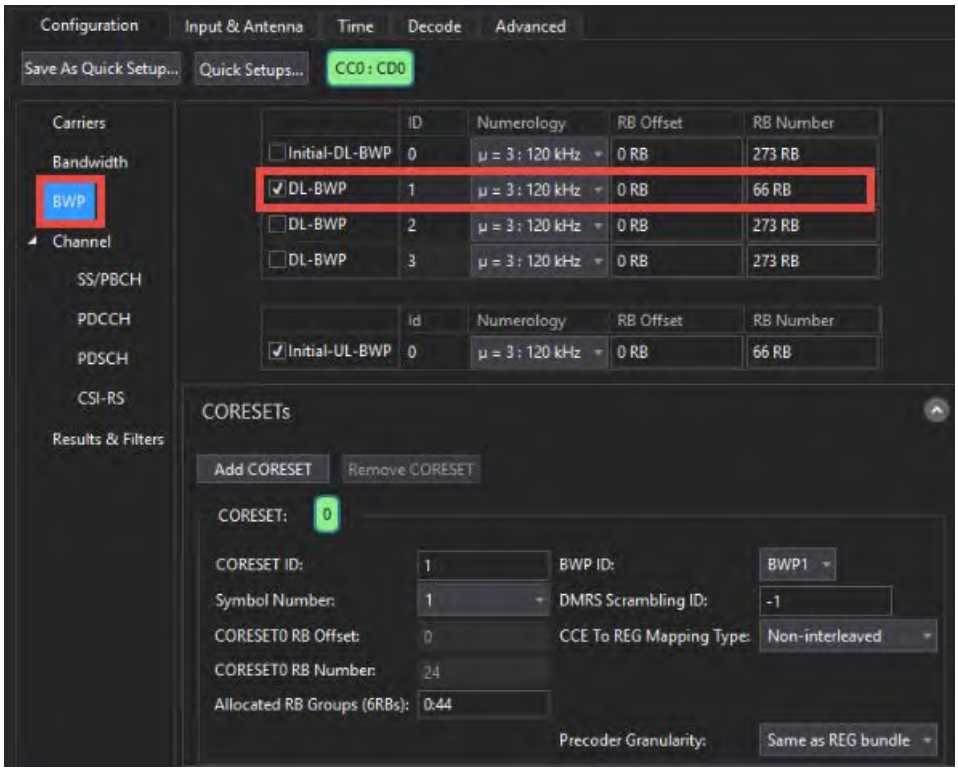


10. Select the **BWP** pane, then enable **DL-BWP**.

This section is used to configure the Downlink BWP allocation within the component carrier and the control resource set (CORESET). CORESET is used for PDCCH configuration and will not be used in this example. We will use the default values of:

- Numerology **μ= 3: 120 kHz**
- RB offset **0 RB**
- RB Number **66 RB**

A component carrier with 100 MHz at FR2 has a maximum RB of 66 for 120 kHz numerology. In this example, DL-BWP uses the full carrier bandwidth of 66 RBs as shown below.



TIP VSA supports a single BWP within a component carrier (CC). Signal Studio and X-Apps support multiple BWPs within a CC. You can use VSA multi-measurement to analyze multiple BWPs simultaneously.

11. Select the **Channel** pane > **SS/PBCH**, enable **SS/PBCH** and set the following parameters:

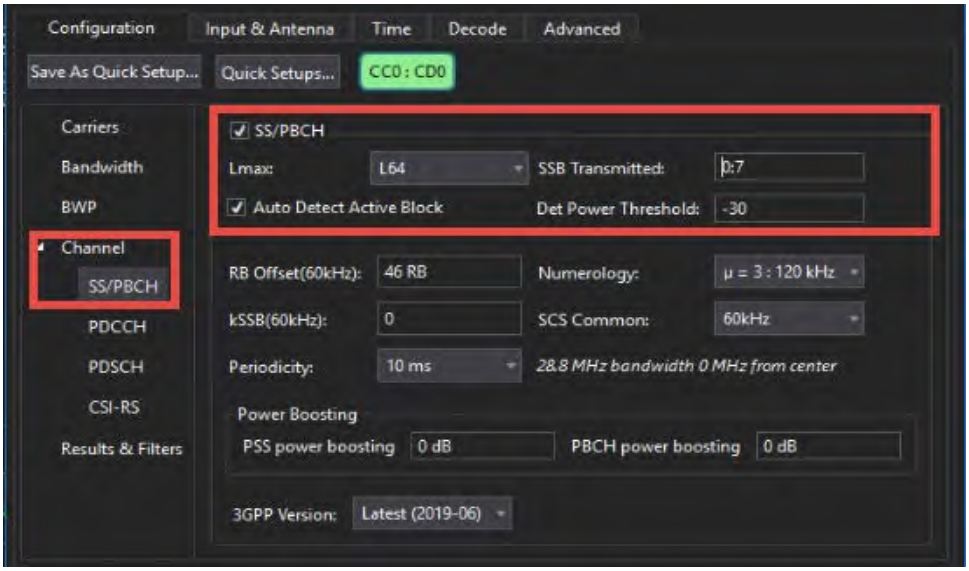
    – Lmax to **L64**

      Lmax specifies the maximum number of SS/PBCH Blocks (that is, beams) in an SS Block period.

      – FR1 up to 3 GHz, Lmax = 4
      – FR1 3 to 6 GHz, Lmax = 8
      – FR2 6 to 52.6 GHz, Lmax = 64

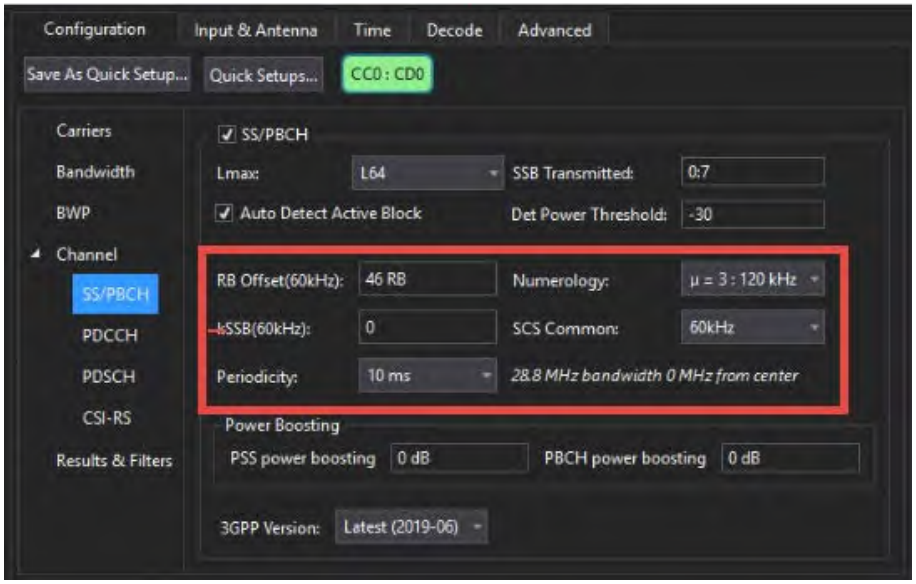
    – SSB Transmitted to **0:7**

For this example, we will not use all 64 SS Blocks; we will activate only 8 of them. You can also select the **Auto Detect Active Block** and set the **Det Power Threshold** to auto detect the active SS Blocks.



These following settings use the default values.

- RB Offset(60kHz) = **46 RB**
- kSSB(60kHz) = **0**
- Periodicity = **10 ms**
- Numerology = **μ= 3 = 120 kHz**



The frequency location of an SS Block is not fixed. The default value of 46 RB and kSSB of 0 subcarriers places it in the center of the carrier bandwidth. This value is calculated based on the 60 kHz numerology as defined by 3GPP (15 and 30 kHz sub-carrier spacing uses 15 kHz numerology, 120 and 240 kHz spacing uses 60 kHz numerology.)

For example, to calculate the 120 kHz sub-carrier spacing, the value would be

66RB/2 = 33 RB

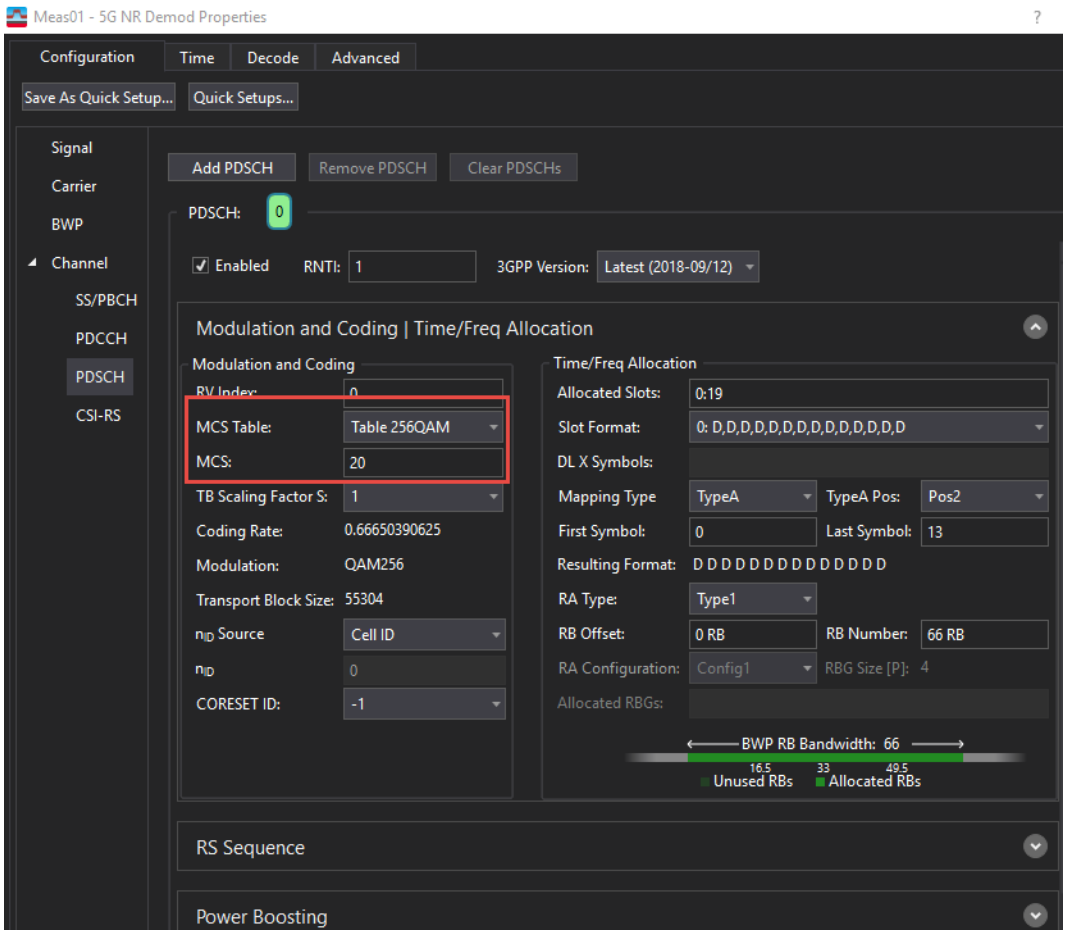
This is the center of the carrier bandwidth. However, the SS Block is 20 RB wide, so the start of the SS Block is 23 RB.

23 RB in 120 kHz is 23 * 2 = 46 RB in 60 kHz

This is the default value used in the VSA software for 120 kHz numerology.

12. From the **Channel** pane, select **PDSCH** and then set the following parameters:

    – Under Modulation and Config section, set MCS Table to **Table 256QAM** and MCS to **20**.



TIP

3GPP has different Tables for PDSCH MCS.

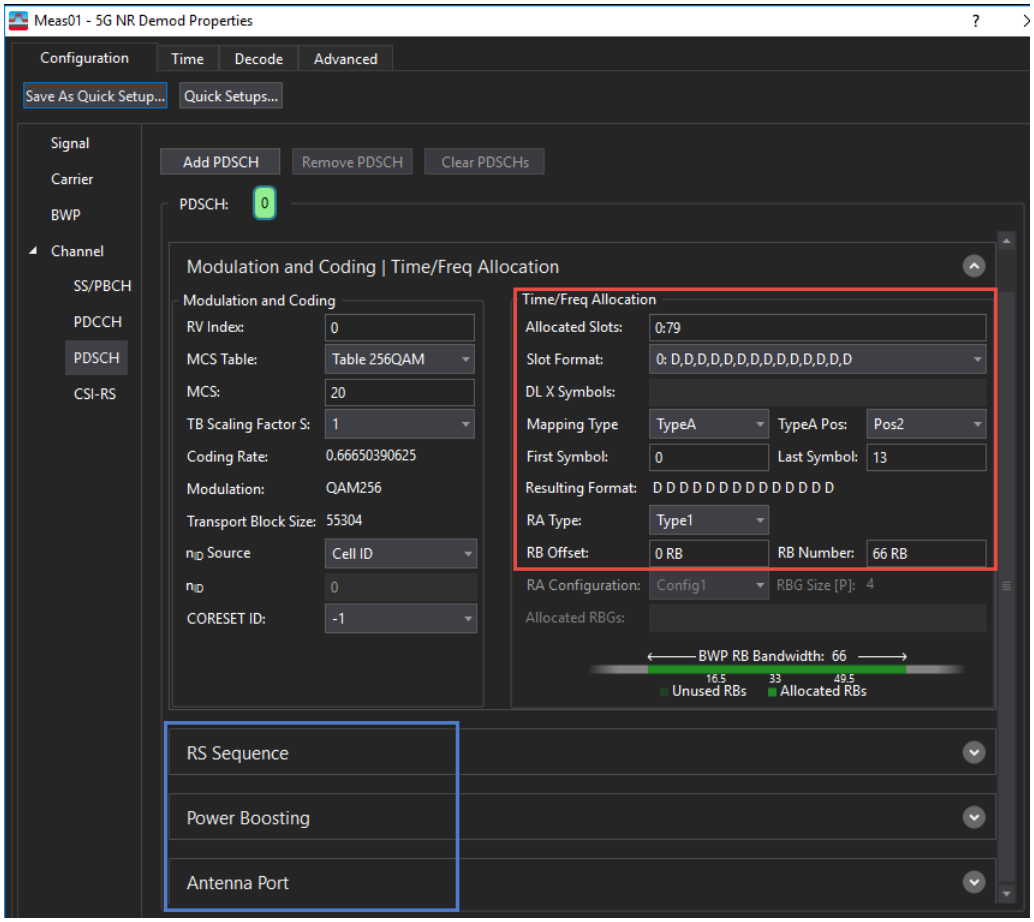
Table 5.1.3.1-2 has 256 QAM as maximum modulation

Table 5.1.3.1-3 has 64 QAM as maximum modulation

Table 5.1.3.1-1 has 64 QAM as maximum modulation, and is for low spectrum efficiency (LowSE).

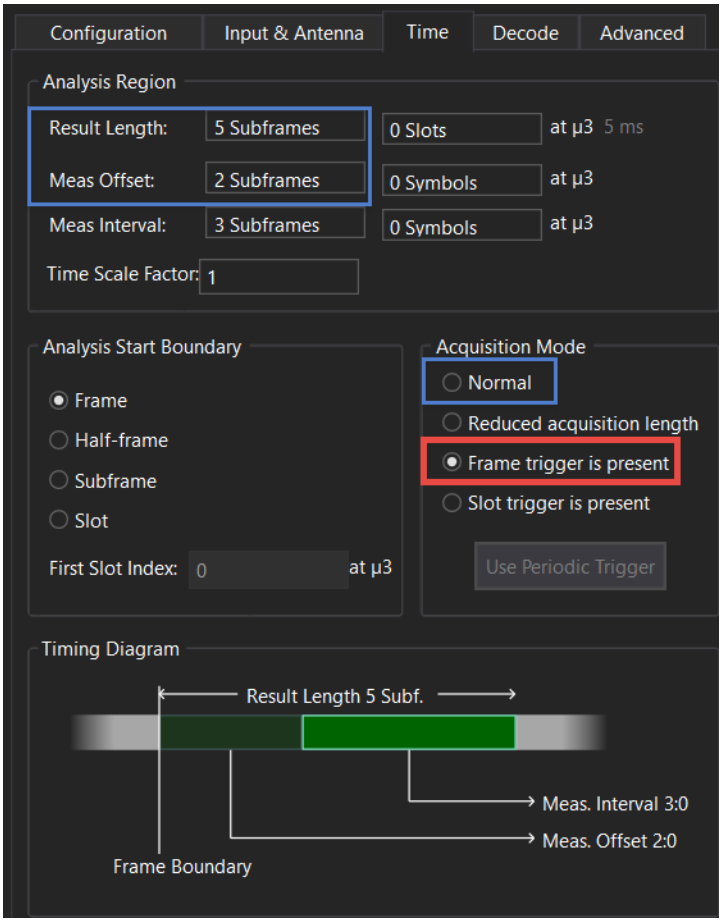
See 3GPP TS38.214 for more information.

- Under the Time/Freq Allocation section, set
  - Allocated Slots to **0:79**
  - Slot Format to **0**
  - First Symbol to **0**
  - Last Symbol to **13**
  - RB Offset to **0 RB**
  - RB Number to **66 RB**



We will use the default values for RS Sequence, Power Boosting, and Antenna Port. Power s would not change the DMRS sequences, but they might be interested in modifying Power Boosting.

13. Select the **Time** tab and set:
    - Result Length to **10 Subframes**.
    - Meas Interval to **2 Subframes**.
    - Select **Frame Trigger is Present**.



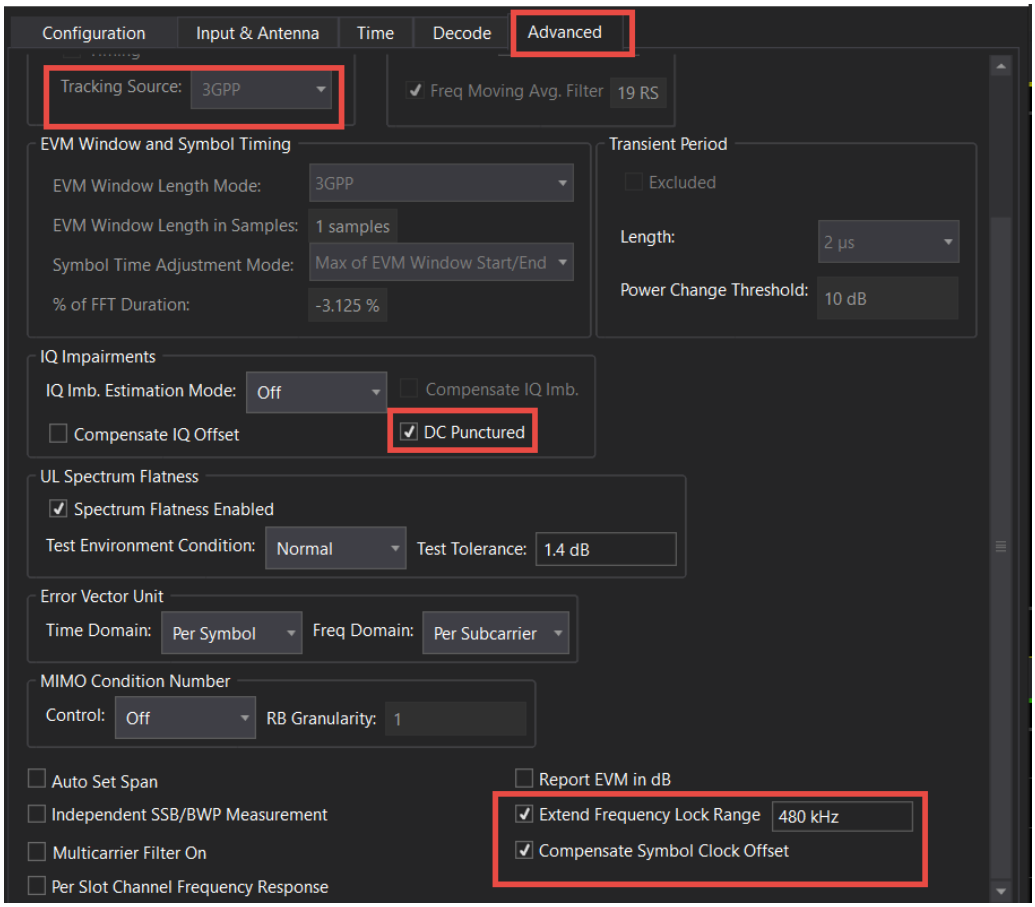
14. Select the **Advanced Settings** tab and select **DC Punctured**.

    5G NR counts the DC subcarrier as a valid subcarrier for rate-matching purposes. The network decides if the DC subcarrier is modulated or not. High LO feedthrough will impact demodulation and EVM performance of the input signal. This setting is off by default.

    Use **Extend Frequency Lock Range** if you cannot lock to the input signal.

    Compensate Symbol Clock Offset is used along with timing track to compensate for clock error in the input signal. The Symbol Clock Error result is reported in the Summary trace.

3GPP Conformance Test > **Phase** is enabled by default. It is specified in the 3GPP requirement and is used to compensate for phase differences between symbols caused by up conversion or down conversion. Getting this setting wrong will cause demodulation issues.

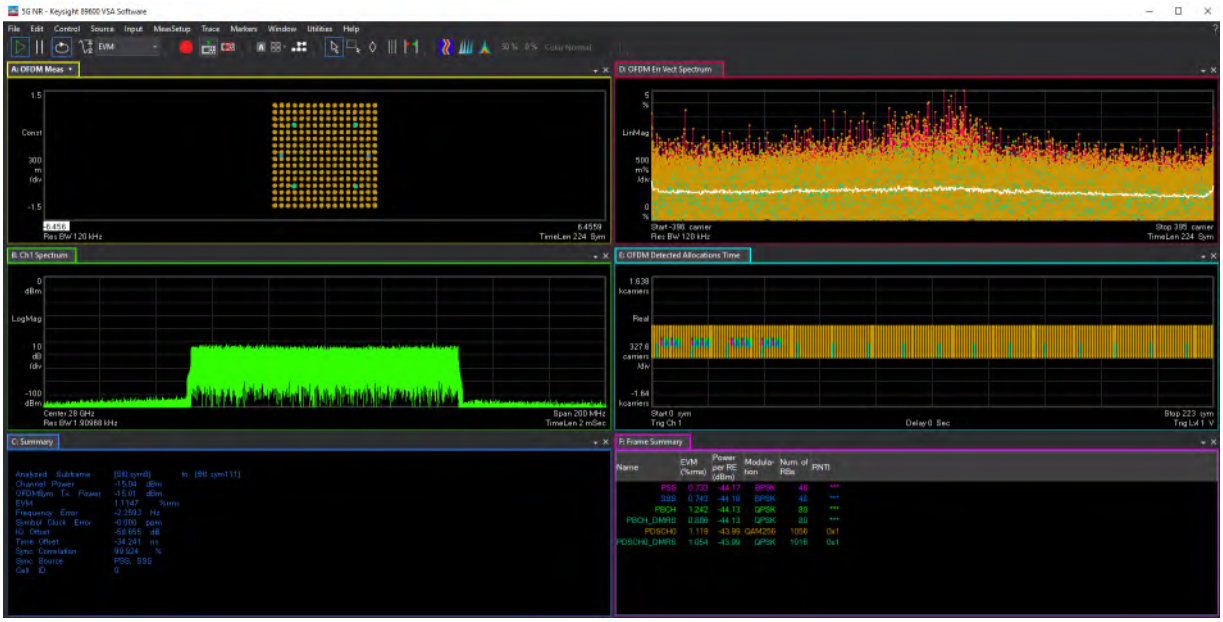


15. From the menu bar, select **Window** > **Trace Layout** > **Tile Visible**.

    This will display all six 5G NR traces on the display.

16. Select the **Auto-Range** icon to run the measurement for EVM optimization. This may take a few minutes to complete.

TIP The VSA has color coding based on channel type, marker coupling to pinpoint error, symbol level analysis (by changing the measurement interval, and measurement offset into a single symbol).



More About the 5G NR Traces

- Trace A: shows the composite constellation diagram showing different modulation formats for the different channels/signals present in the transmitted signal.
- Trace B: shows the frequency spectrum.
- Trace C: shows the composite error metrics. You can also see the automatically detected Cell ID value and the Channel Power.
- Trace D: shows the EVM versus subcarrier and symbol.
- Trace E: shows the detected allocations of all channels/signals within the measurement interval. (For this example, 2 Subframes) You can see the 8 active SS Blocks in the first four slots.
- Trace F: shows the frame summary: EVM Power per RE, Modulation format, Number of RB and RTNI of the individual channels/signals.

For more information on these traces, see the Online help.

Trace Filtering

The VSA has per-trace filtering to filter by subsegments and channels. Subsegment refers to Layers, (in case of MIMO), BWP, and SS Block. In the downlink, Layers in DL-BWP and SS Block are presented as subsegments. In the uplink, Layers in UL-BWP are presented as subsegments.

The following traces can be filtered by subsegments and channel types.

- OFDM Meas
- OFDM Ref
- Error Vector Spectrum
- Error Vector Time
- Detected Allocations Time trace can be filtered per channel only

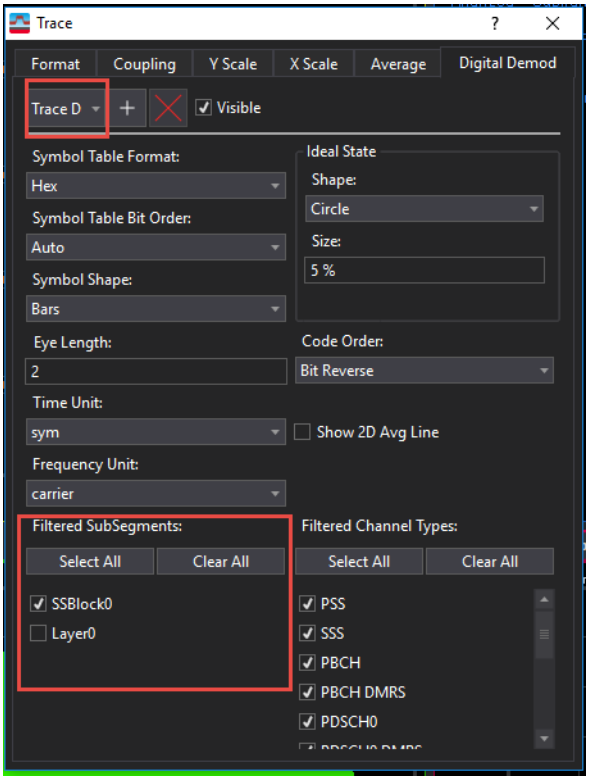
The rest of the traces do not support per-trace filtering.

A Simple Example

We will apply per-trace filtering to Error Vector Spectrum (Trace D) and Detected Allocations Time (Trace E) traces.

1. From the menu bar, select **Trace** > **Digital Demod**.
2. In the Trace Dialog, select **Trace D** (OFDM Error Vector Spectrum) from the dropdown.
3. In the Filtered SubSegments area, clear the **Layer0** check box.

This allows you to take a look at SS Block. You can also filter with SS Block Channels/Signals you want to see by using the Filtered Channel Type.



4. Select Trace E (OFDM Detected Allocations Time) from the Trace dialog.
5. In the Filtered SubSegments area, clear the **SS Block0** check box.

   This alloys you to look at Layer 0 information, which is the DL-BWP information.
6. In the Filter Channel Types area, **Clear All** selections, then select just **PDSCH-DMRS**.

   You now see a gap in two of the DMRS symbols. Why is this?

   Notice that the Frame Summary table shows Num. of RBs for PDSCH of 1056 and 1016 for PDSCH-DMRS. The signal configuration only had 66 RBs. Why the change in the measurement result?

   The RB result in the Frame Summary table is per Slot. In this example, we changed the measurement interval to 2 Subframes, which is 16 slots for 120 kHz numerology (8slots/subframe). 66*16=1056. For PDSCH-DMRS, the value is 1016 because of the 16 slots overlap with the SS Block, which occupies 20 RBs. This results in 20 RBs being punctured from DMRS in these two slots, so the total RB Number for DMRS is 1056-20*2 = 1016.

Using the equivalent SCPI commands

On the VXG:

```
SYSTem:PRESet

RF1:FREQuency:CW 28GHZ

RF1:POWer:AMPLitude -10dBm

RADio:SELect NR5G

RADio:NR5G:WAVeform:CCARrier:BWIDth FR2BW100M

RADio:NR5G:WAVeform:CCARrier:SNUMerology MU3

RADio:NR5G:WAVeform:CCARrier:DLINk:SSBLock:LMAX 64

RADio:NR5G:WAVeform:CCARrier:DLINk:SSBLock:ACTive:INDices
"0:7"

RADio:NR5G:WAVeform:CCARrier:DLINk:SCH0:MCS 20

RADio:NR5G:WAVeform:CCARrier:DLINk:SCH0:MCS:TABLe TABLe52

RADio:NR5G:WAVeform:GENerate

GROup:SIGNal1 ON

RF1:OUTPut ON
```

For multi-channel instruments, set RF Out (all) to On.

```
RFALl:OUTPut ON
```

On the X-Series Signal Analyzer:

```
INSTrument:SELect VSA89601

SYSTem:PRESet

FREQuency:CENTer 28 GHz

INPut:ANALog:RANGe:AUTO

INITiate:PAUSe

INPut:TRIGger:STYLe "External"

INPut:TRIGger:LEVel:EXTernal 1V

INPut:EXTension:PARameters:SET "ExtTriggerLoc", 2 (This will
set it to Trigger 3)

NR5G:FRAMe:TRIGger:ENABled 1

INPut:EXTension:PARameters:SET "PhaseNoiseOptDualLoop", 1

MEASure:CONFigure NR5G
```

```
NR5G:CCARrier:CIDentity:AUTO 1 |OR| NR5G:CCARrier:CIDentity
0

NR5G:CCARrier:TBANdwidth "FRTwo100MHz"

NR5G:DBWP:ENABled 1

NR5G:DBWP:NUMerology "Mu3"

NR5G:DBWP:ROFFset 0

NR5G:DBWP:RNUMber 66

NR5G:SSBLock:ENABled 1

NR5G:SSBLock:LMAX "L64"

NR5G:SSBLock:AINDexes "0:7"

NR5G:DBWP:PDSCh1:MCS:TABLe "Table2"

NR5G:DBWP:PDSCh1:MCS 20

NR5G:DBWP:PDSCh1:SLOT:ALLocated "0:79"

NR5G:CCAR:PDSCH1:SFI 0

NR5G:CCAR:PDSCH1:SINDex:FIRSt 0

NR5G:CCAR:PDSCH1:SINDex:LAST 13

NR5G:CCAR:PDSCH1:ROFFset 0

NR5G:CCAR:PDSCH1:RNUMber 66

NR5G:RLENgth 10

NR5G:SUBFrame:INTerval 2

NR5G:FRAMe:TRIGger:ENABled 1

NR5G:DC:PUNCtured 1

NR5G:MCFilter:ENABled 1

NR5G:FREQuency:LOCK:EXTended 1

NR5G:COMPensate:SYMBol:CLOCk:OFFset 1

DISPlay:LAYout 3,2

INPut:ANALog:CRITeria:RANGe:AUTO "EVM", -1
```

Creating a DL MIMO Signal Using PathWave N7631APPC Signal Generation

NOTE The VXG must have the N7631APPC Signal Generation for 5G NR license installed.

This procedure will show you how to configure and analyze a downlink (DL) multiple input, multiple output (MIMO) signal using PathWave Signal Generation and the 89600 VSA software. The hardware setup below includes a 2-channel VXG signal generator and a 2+ channel, 33 GHz or higher Infiniium UXR-Series Oscilloscope.

TIP If you do not have access to a Keysight Infiniium UXR Real-Time Oscilloscope, a UXR recording waveform file has been included in the Example waveform file folder:

```
D:\Users\Instrument\Documents\Keysight\PathWave
\SignalGenerator\Examples
```

You will need to copy the **MIMO_UXR_Recording.sdf** file to the X-Series Signal Analyzer or the PC running the VSA application. Steps have been included below to successfully run the recording.

Hardware Requirement

- 2 channel M9484C VXG
- UXR with DDC option:
  - UXR000-601: 160 MHz BW or
  - UXR000-602: 2 GHz BW

  Hardware accelerated DDC decimates oscilloscope captures in real-time enabling deep captures and fast measurement speed.

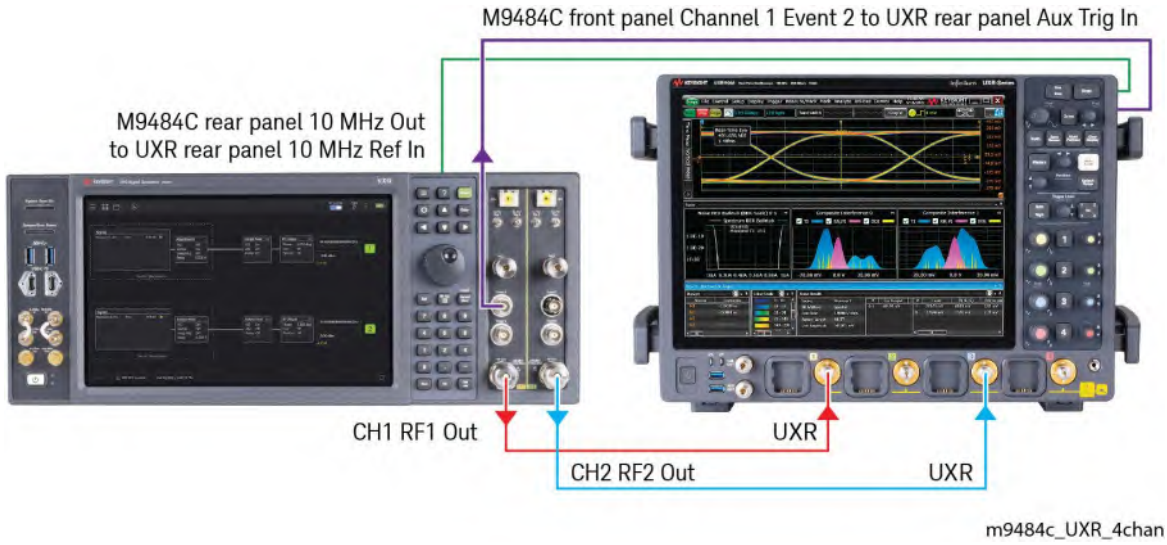
Software Requirements

PathWave Signal Generation for 5G NR	N7631APPC	Use latest release from Keysight.com
89600 VSA software	89601BHNC	VSA2023U2 or later

Equipment Setup for the M9484C

- M9484B CH1 front panel RF Out to UXR front panel CH1
- M9344B CH2 front panel RF Out to UXR front panel CH3
- M9484C front panel Channel 1 Event 2 to UXR rear panel Aux Trig In

– M9484C rear panel 10 MHz Out to UXR rear panel 10 MHz Ref In.

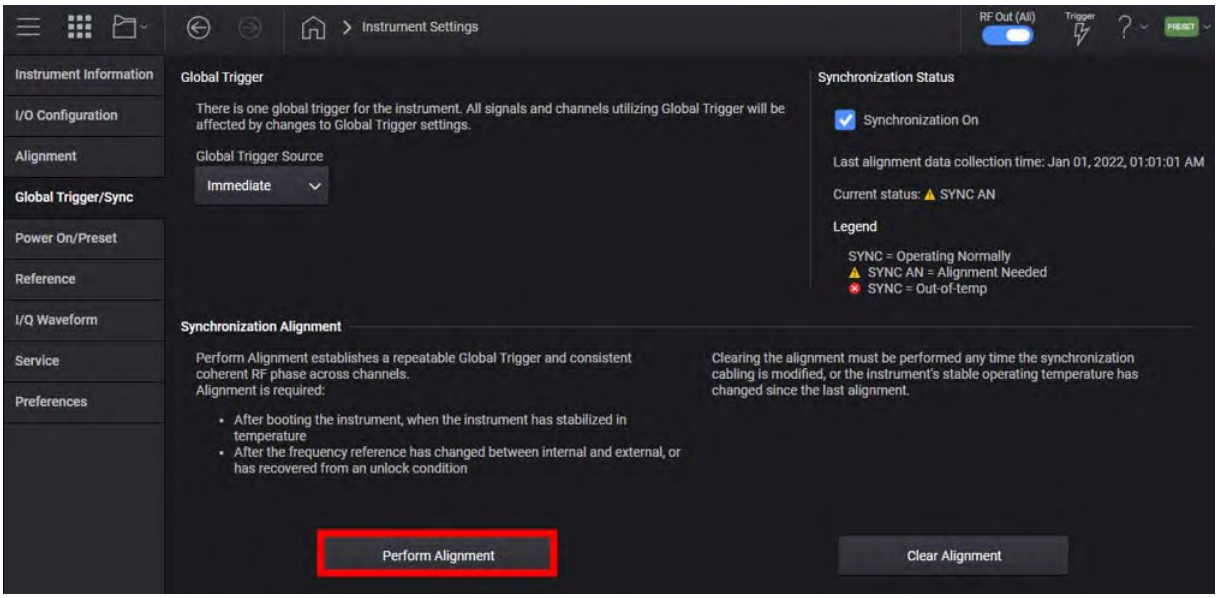


On the VXG:

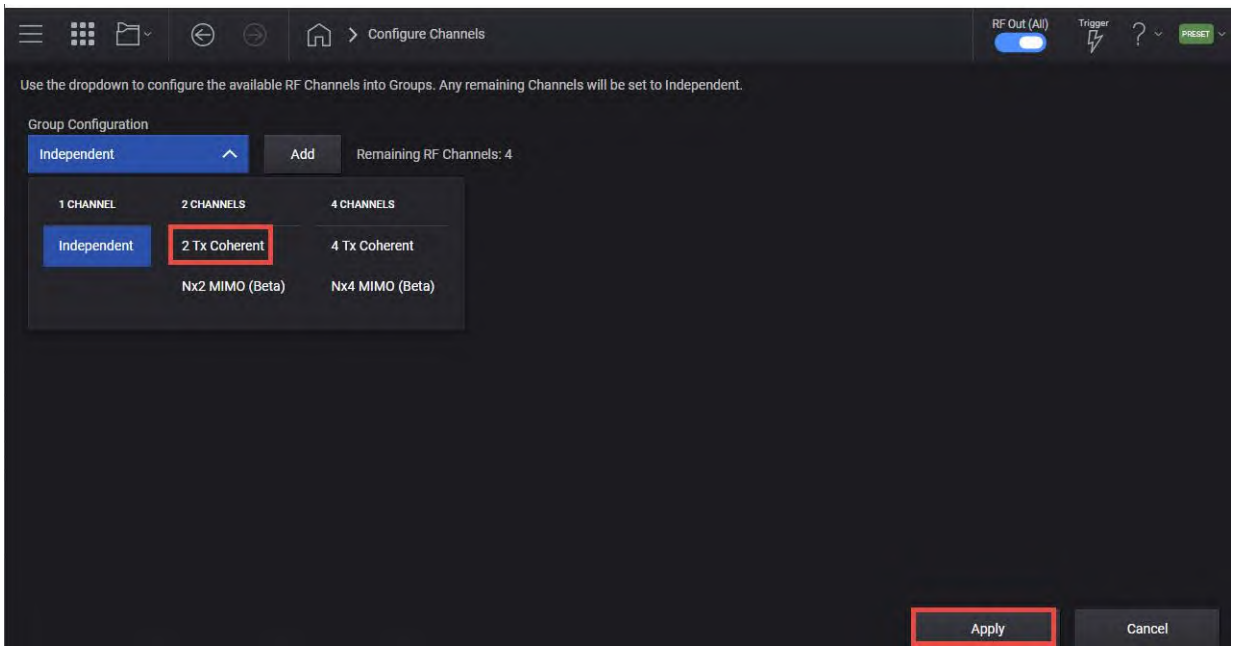
1. Select **Preset** > **Preset** to set the VXG to a known state.

   If a Synchronization Alignment is required, indicated by

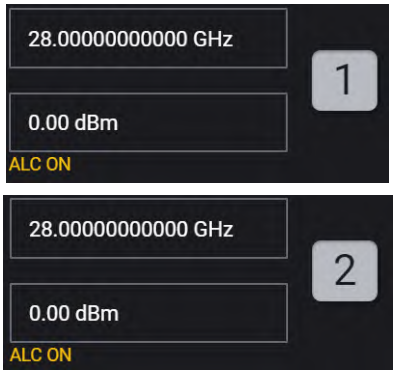
in the bottom left corner, tap or click the warning message and choose **Perform Alignment** before proceeding.



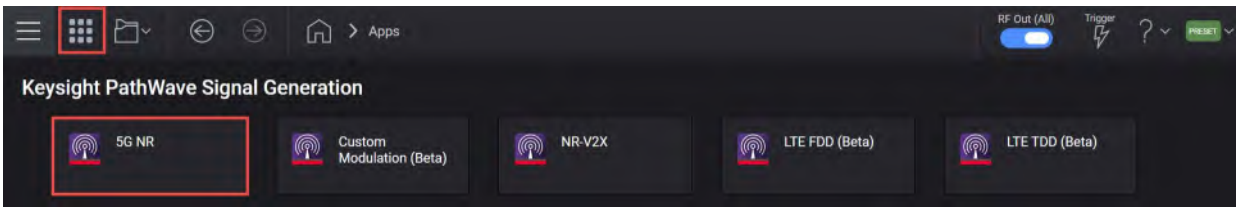
2. In the Menu/Tool Bar, select the Triple Bar icon (top left corner of the display, and then select **Configure Channels**. Change the signal configuration from Independent to **2 Tx Coherent** > **Add** > **Apply** > **Apply**.



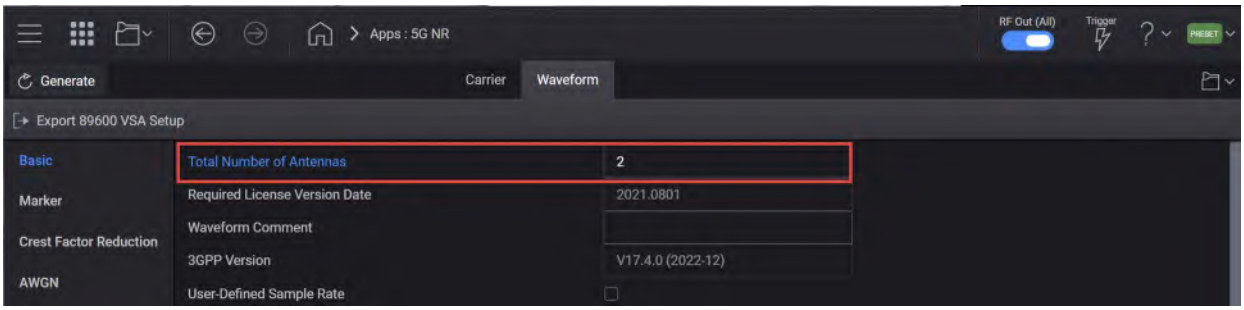
3. In the Output area for both Channel 1 and Channel 2, set Frequency to **28 GHz**, or the frequency you are using, and Power to **0 dBm**.



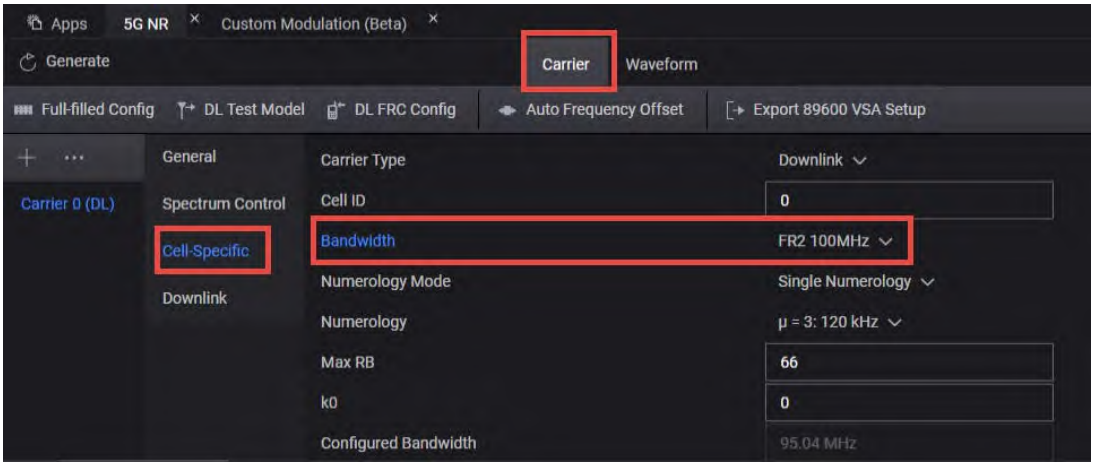
4. Select the **Apps** block to open, then select **5G NR/5G-Advanced**.



5. Select the **Waveform** tab and set the Total Number of Antennas **2**. This means that two antenna port signals will be generated.

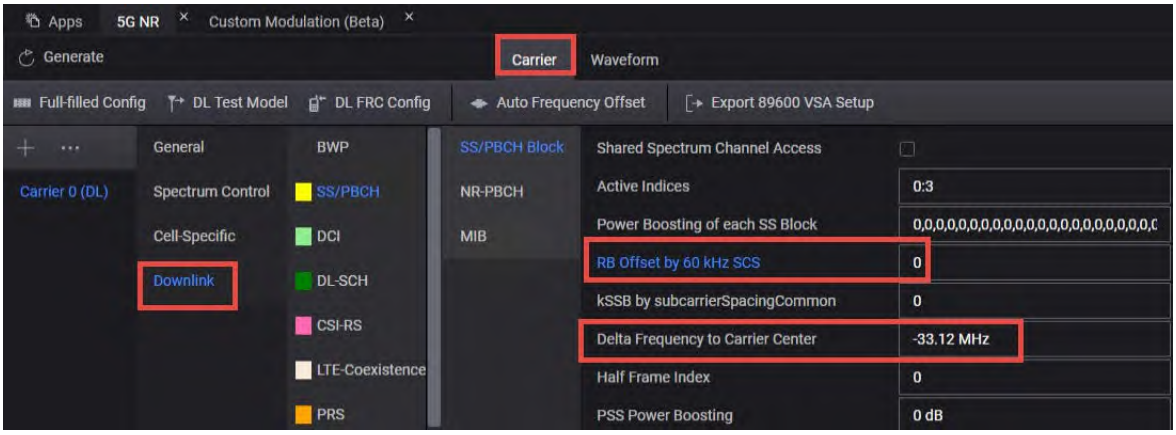


6. Select the **Carriers** tab > **Cell Specific** node, and set Bandwidth to **FR2 100 MHz**. Keep the rest of the default settings.



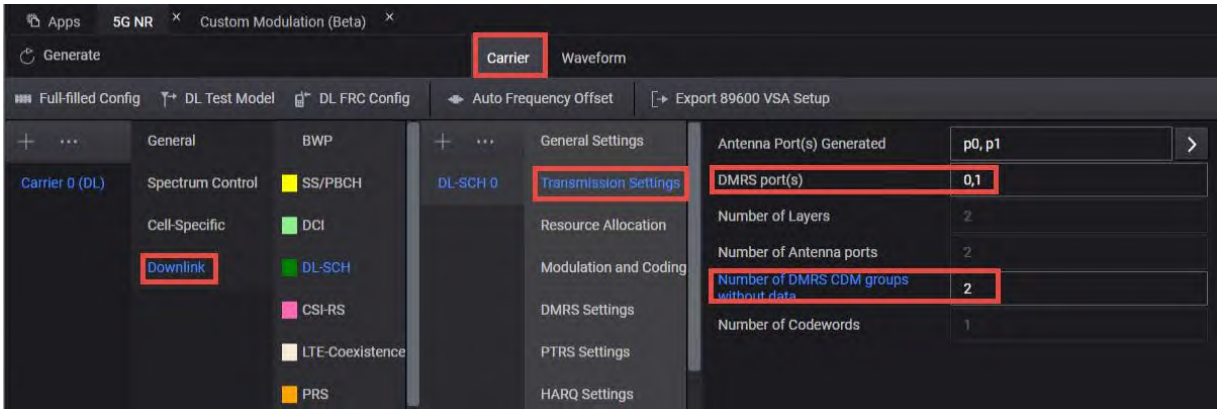
7. Select the **Downlink** node > **SS/PBCH** > **RB Offset by 60 kHz SCS** and set to **0**.

This automatically changes the Delta Frequency to Carrier Center to -33.12 MHz. Using 0 RB Offset puts the SSB at the lower edge of the carrier.



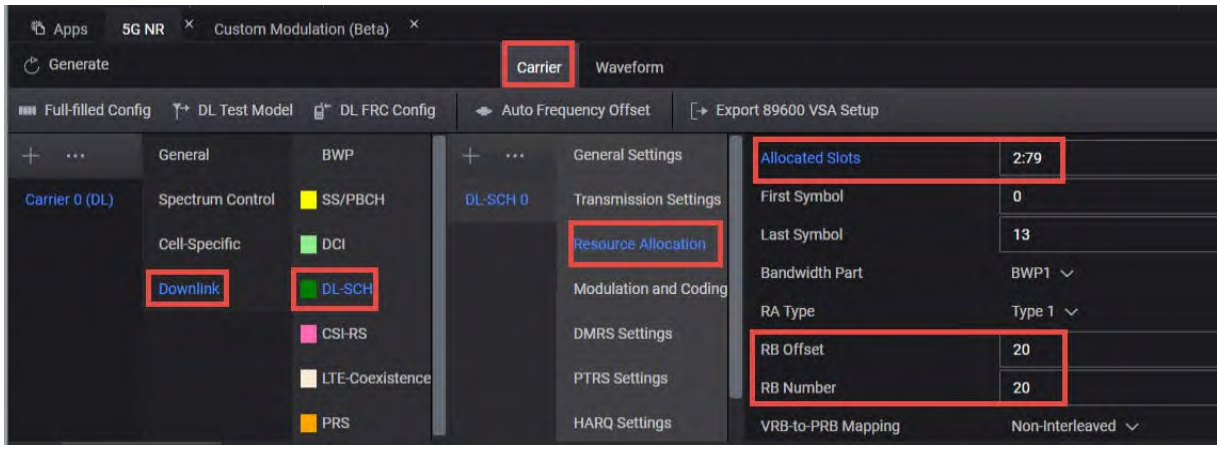
8. Select the **Downlink** node > **DL-SCH**> **Transmission Settings** and confirm DMRS ports is set to **0,1**. Set Number of DMRS CDM groups without data to **2**.

   You will see the layer number is updated to 2 and each layer is assigned with a particular DMRS port. You'll also see that Antenna Ports Generated is automatically set to P0,P1, which will map the multiple antenna port signals to different antennas (instruments).



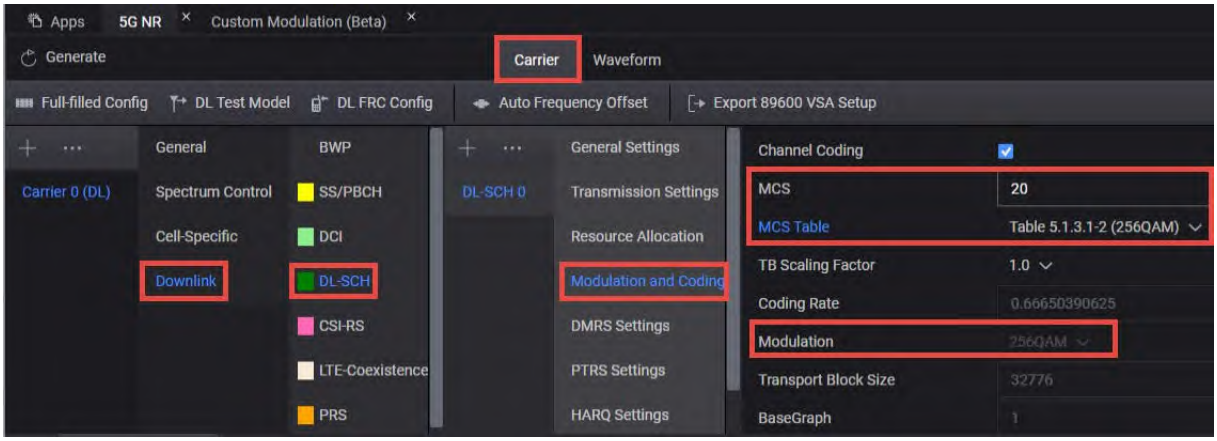
9. Select the **Resource Allocation** node and set:
   - Allocated Slots to **2:79**
   - RB Offset to **20**
   - RB Number to **20**

DL-SCH0 will occupy the RBs 20-40 and DL-SCH1 will occupy the rest of the resources.



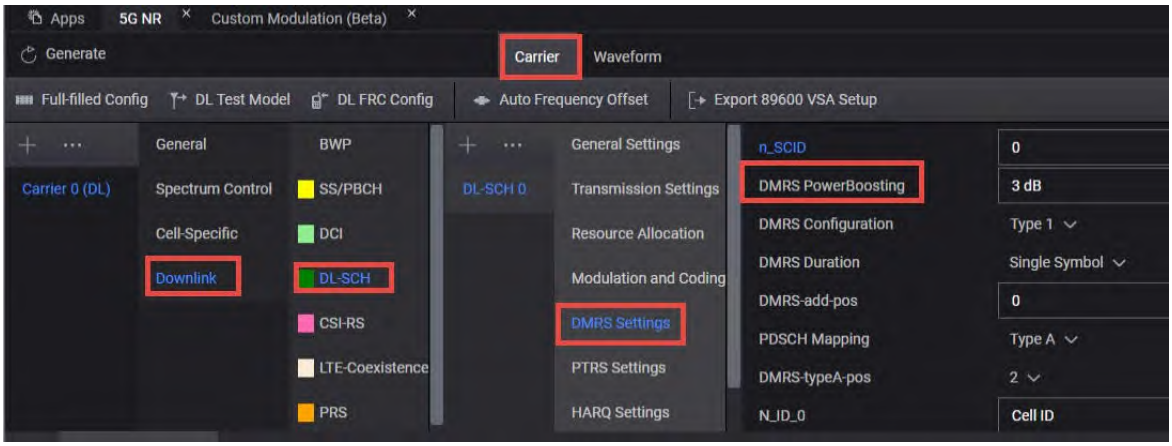
10. Select the **Modulation and Coding** node and set:
    - MCS to **20**
    - MCS Table to **5.1.3.1-2 (256 QAM)**

    Notice that modulation is updated to 256 QAM.

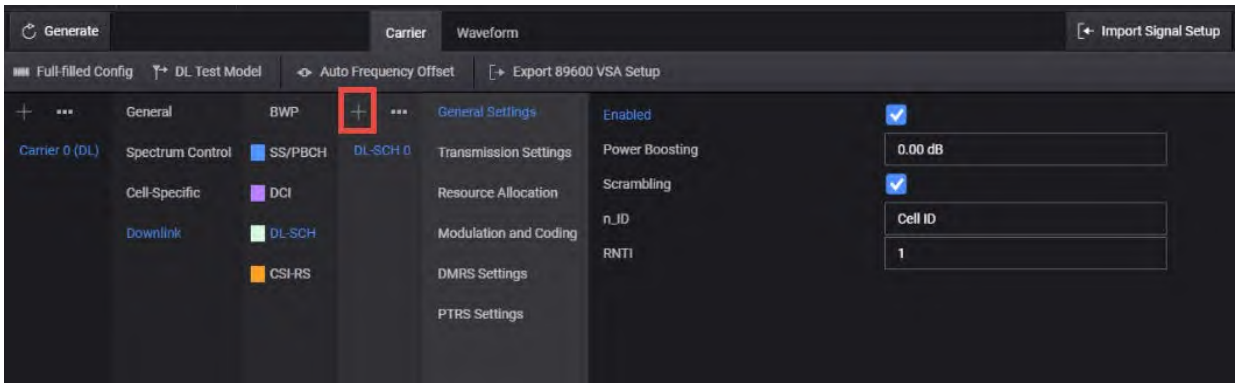


11. Select the **DMRS Settings** node and notice that DMRS Power Boosting is set to **3 dB**.

When CDM Group Number is 2, DMRS power boosting becomes 3 dB per 3GPP.

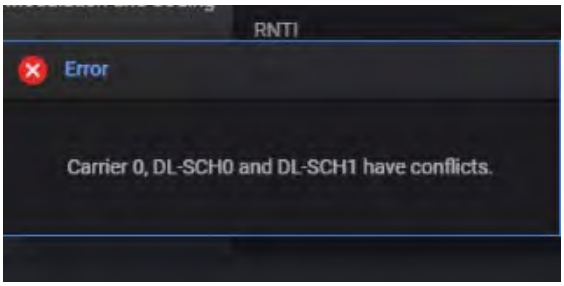


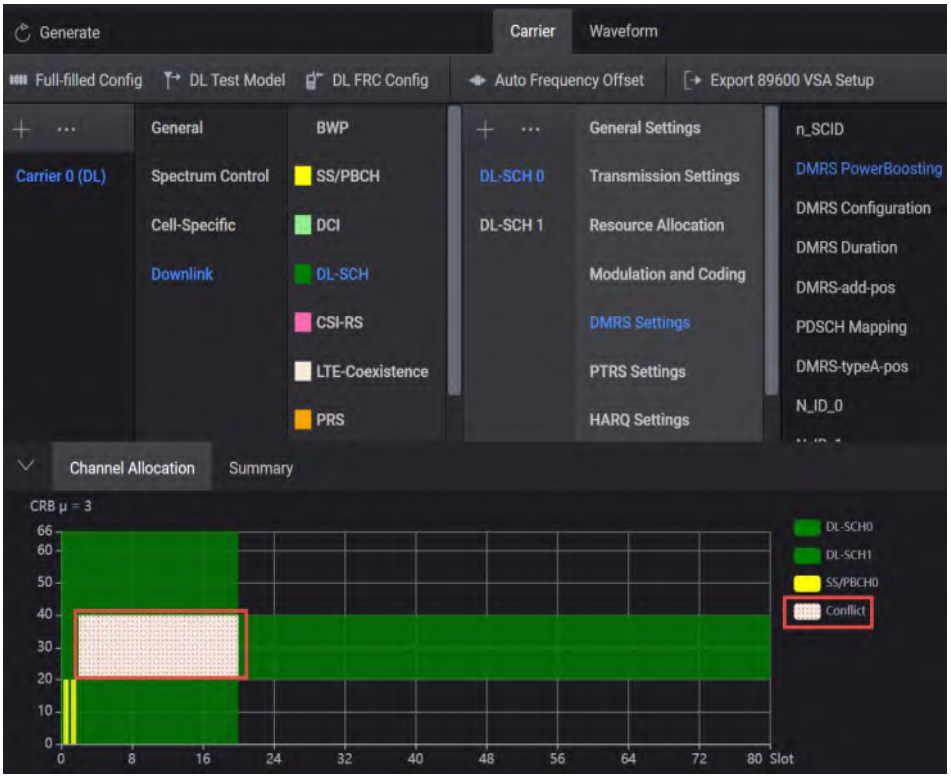
12. Under the DL-SCH 0 node, select the + icon to add a new **+DL-SCH** (DL-SCH1).



NOTE

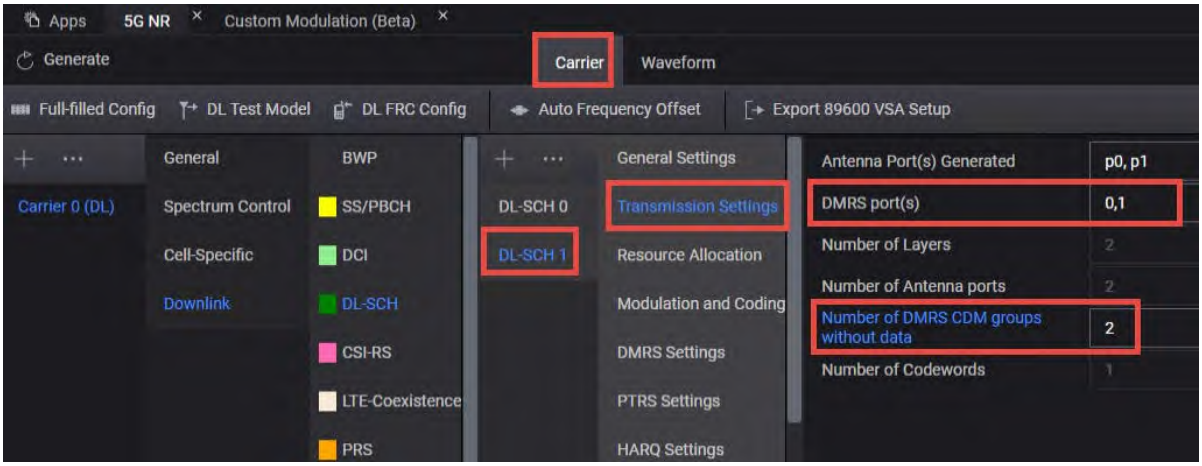
This will create a conflict between DL-SCH0 and DL-SCH1. This will be resolved in a later step.





13. From the Channels node, select **DL-SCH** > **DL-SCH1** > **Transmission Settings** and set DMRS port(s) to **0,1.** Change the Number of DMRS CDM groups without data to **2**.

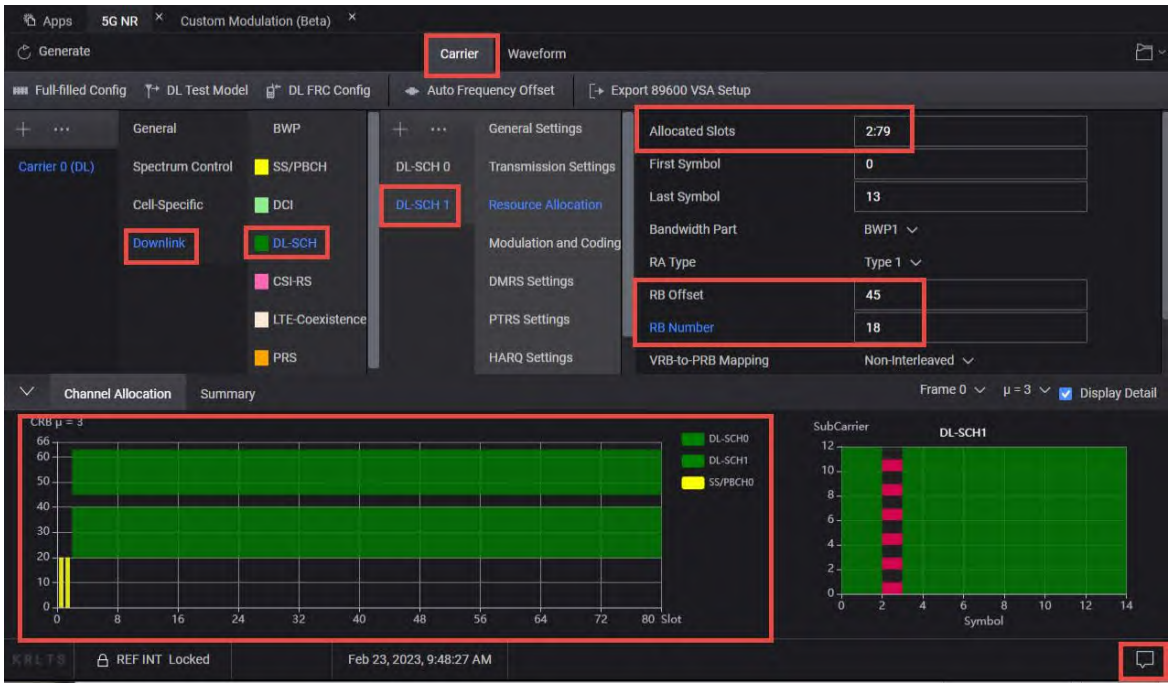
Notice that Number of Layers is updated to 2 and each layer is assigned with a particular DMRS port. Antenna Ports Generated is automatically assigned P0,P1, which will map generated multiple antenna port signals to different antennas (instruments).



14. Select the **Resource Allocation** node and set:

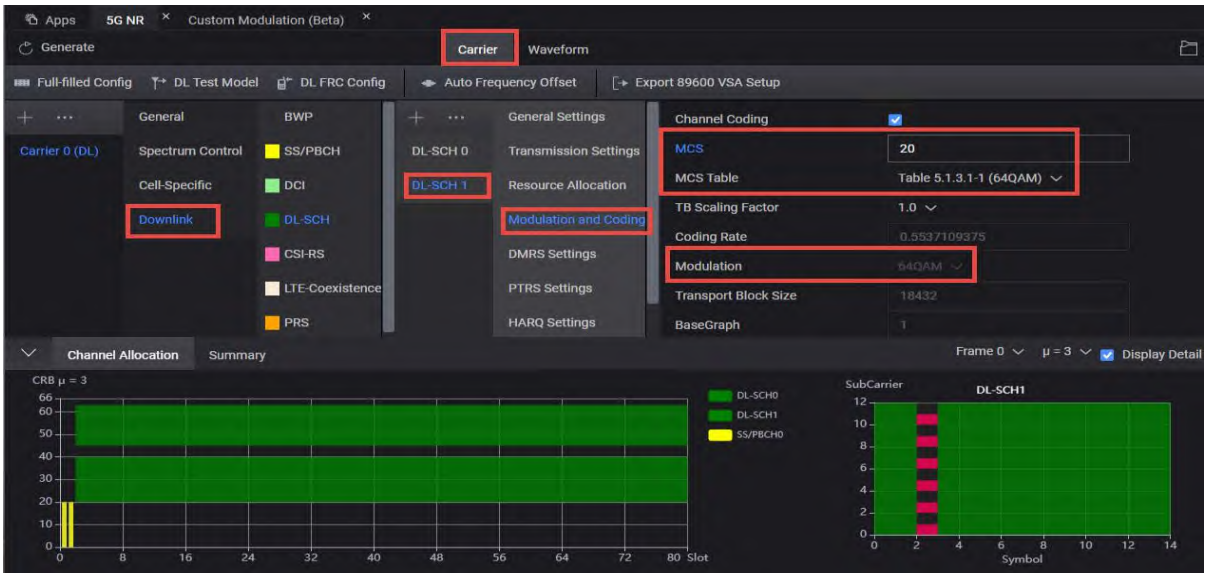
- Allocated Slots to **2:79**
- RB Offset to **45**
- RB Number to **18**

Notice that this resolves the conflict between DL-SCH channels. (The pink shading is removed from the Channel Allocation graph. To clear the conflict error message at the bottom of the display, select the Message icon (bottom, right corner) and select **Clear**.

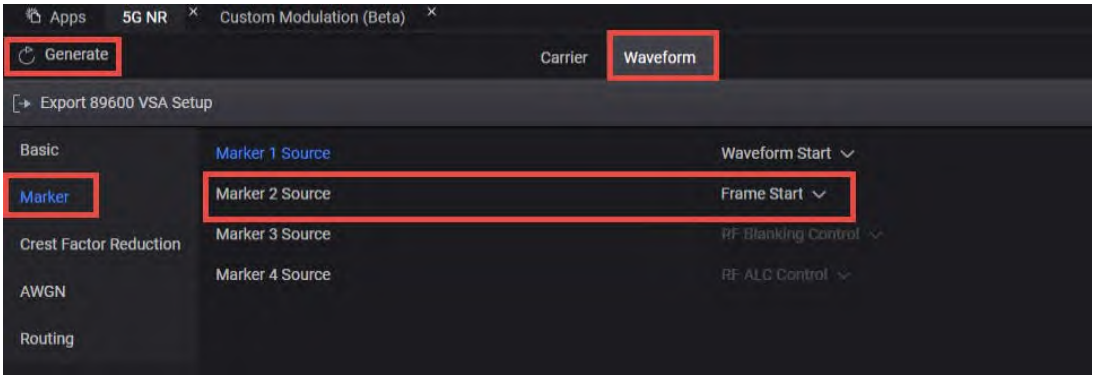


15. Select the **Modulation and Coding** node, and set MCS to **20**.

Use the default MCS Table to Table 5.1.3.1-1 (64QAM). You will see the modulation is updated to 64QAM.

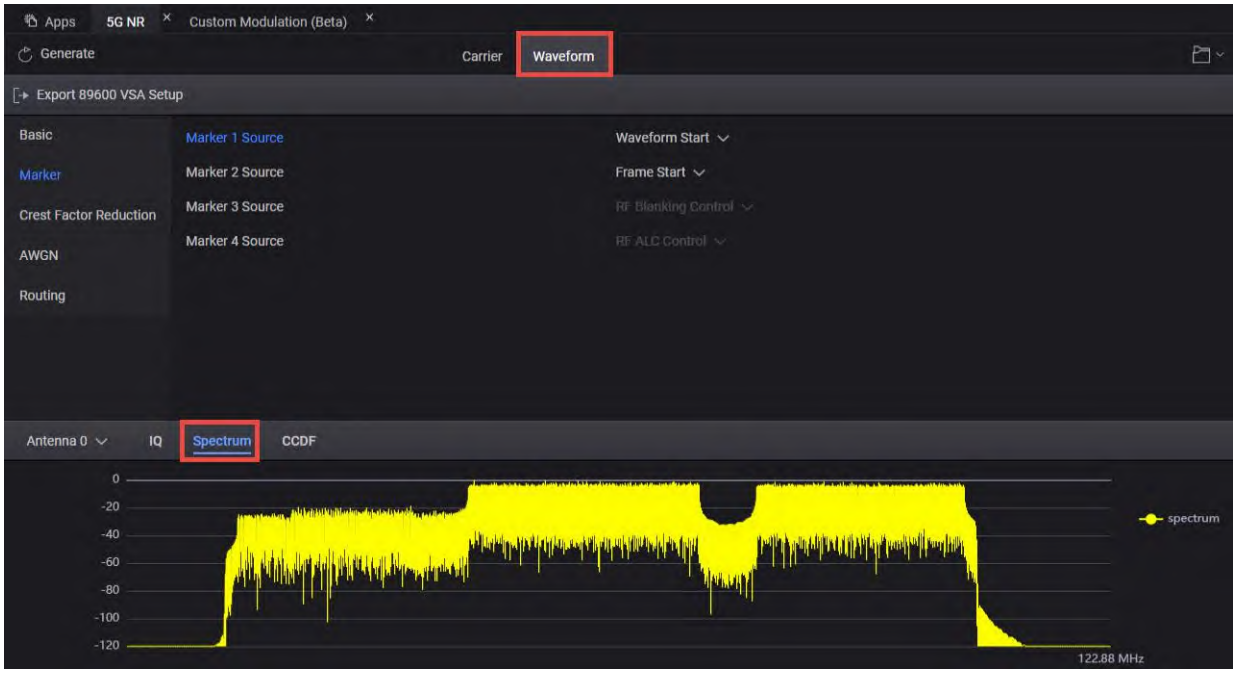


16. Select the **Waveform** tab > **Marker** and notice that Marker 2 Source is set to **Frame Start**. Select **Generate**.



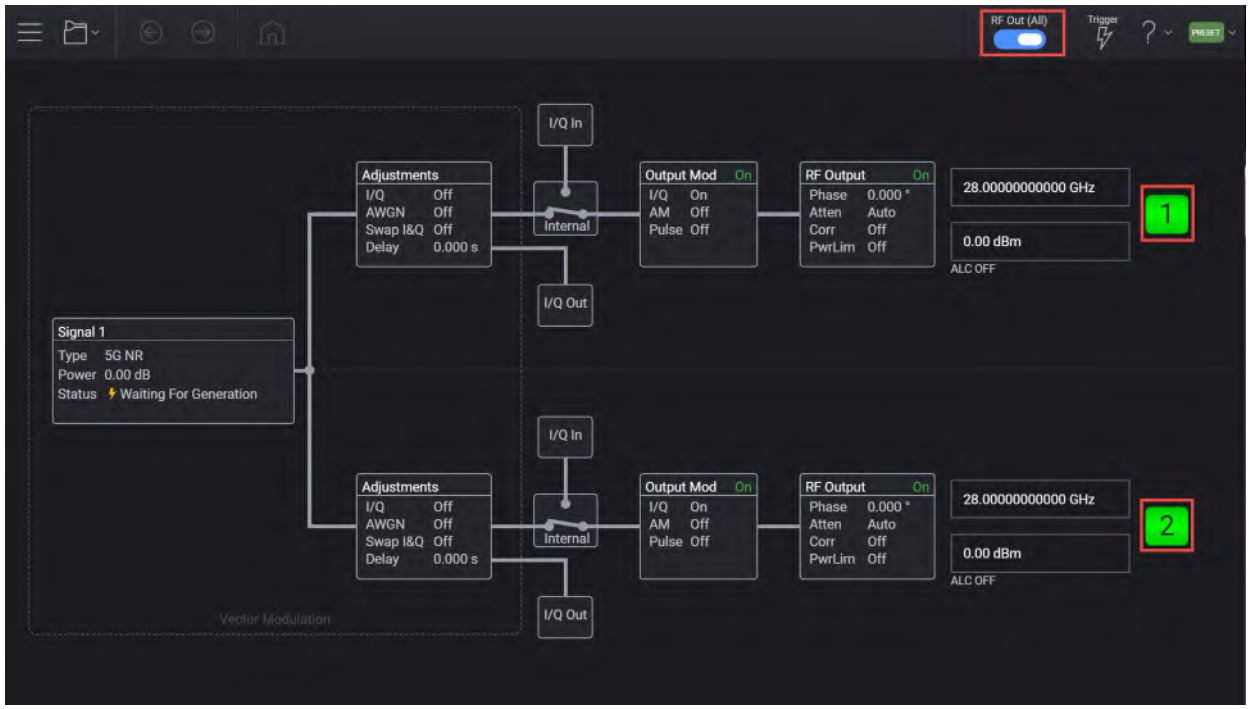
17. In the bottom panel, select **Spectrum**.

    You should see a spectrum like the one below.



18. Return to the **Carriers** tab and select **Export 89600 VSA Setup**. Save the .setx file to a USB drive to transfer to the VSA. We will also set up the 89600 VSA manually in the next section.

19. Select the Home icon and set RF Out to **On** by selecting the numbered channel indicator switches, and then select **RF Out (All)** master control switch to turn the RF output On for both channels or off for both channels.



To Setup the VSA Using the Infiniium UXR Real Time Scope

NOTE

There are two different VSA setup process.

- If you have access to a Keysight Infiniium UXR Real Time Scope continue to the first step below.
- If you do not have access to a Keysight Infiniium UXR Real-Time Oscilloscope, a UXR recorded waveform file has been included in the Example waveform file folder of the VXG. Go to "To setup the VSA Using a UXR Recorded Waveform File" on page 615

To analyze the signal using the VSA on the UXR307944

1. Open the VSA software.

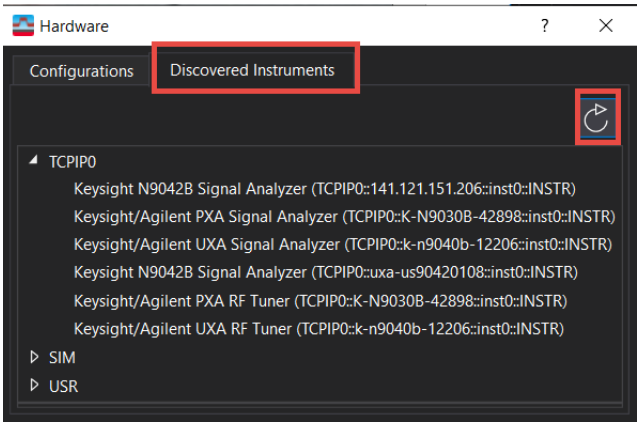
To access the VSA software, go to the Windows Start menu and find Keysight 89600 Software (latest installed version) folder and run the software.

NOTE

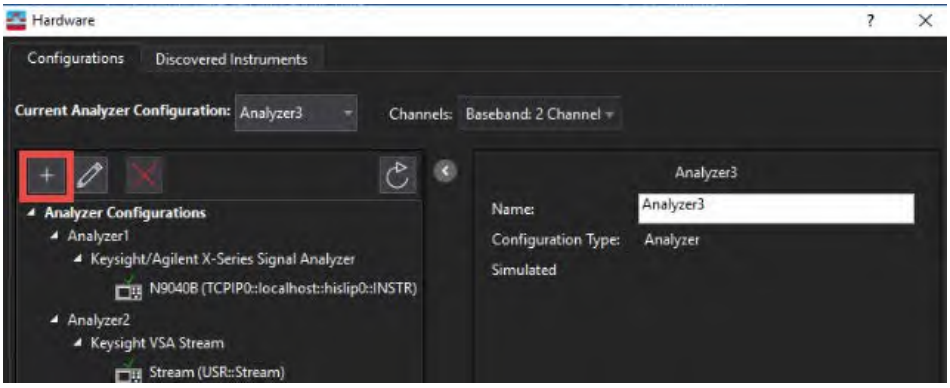
If accessing the instrument via a Remote Desktop connection, select the Screen tab (at the top of the display) to open the Mode/Measurement/View Selector window.

2. From the VSA menu bar, select **File** > **Preset** > **Setup and Traces** to set the VSA to a known state.
3. To configure the hardware, go to **Utilities** > **Hardware** > **Configurations**.

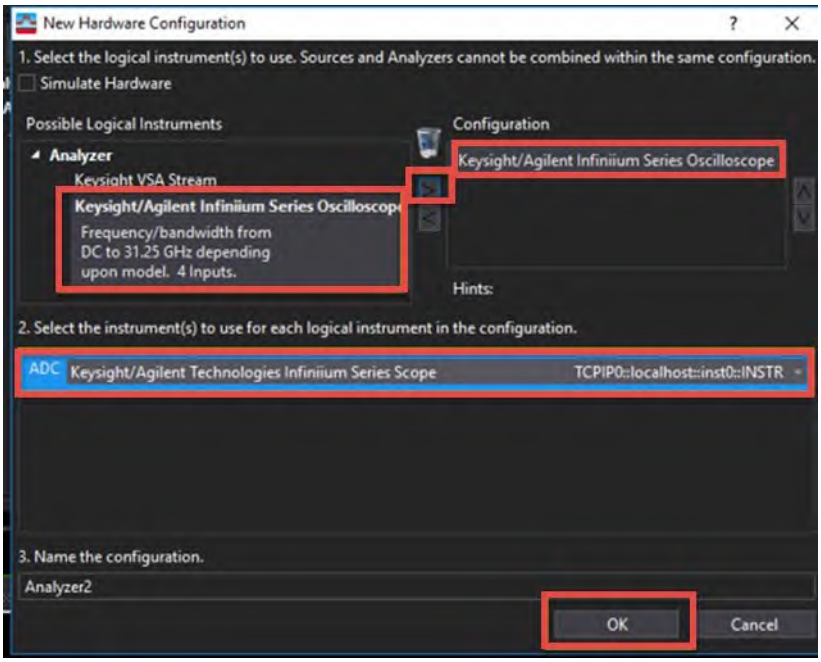
   If it is already configured, go to step 10.
4. Open the **Discovered Instruments** tab and verify that the UXR is listed, if not, select the **Rediscover Instruments** icon.



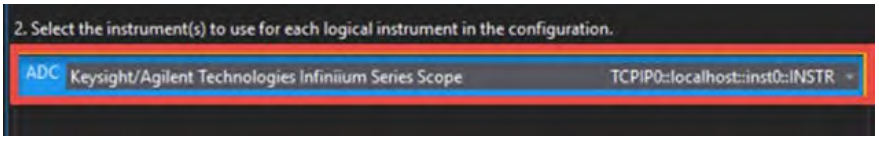
5. In the **Configuration** tab, select the **+ icon**.



6. Scroll down the Possible Logical Instruments and select **Keysight/Agilent Infiniium Series Oscilloscope**, and then select the right arrow to move it under Configuration.

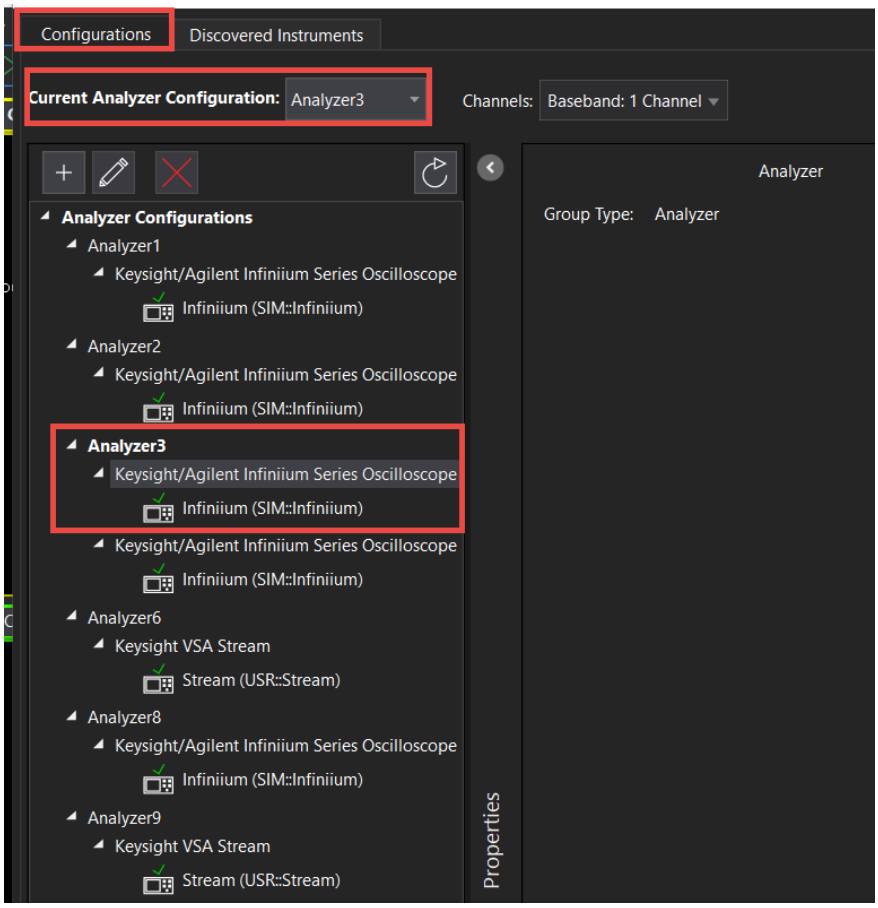


7. From the middle of the dialog box, select the UXR to be used.



8. Select **OK** to create the UXR configuration.

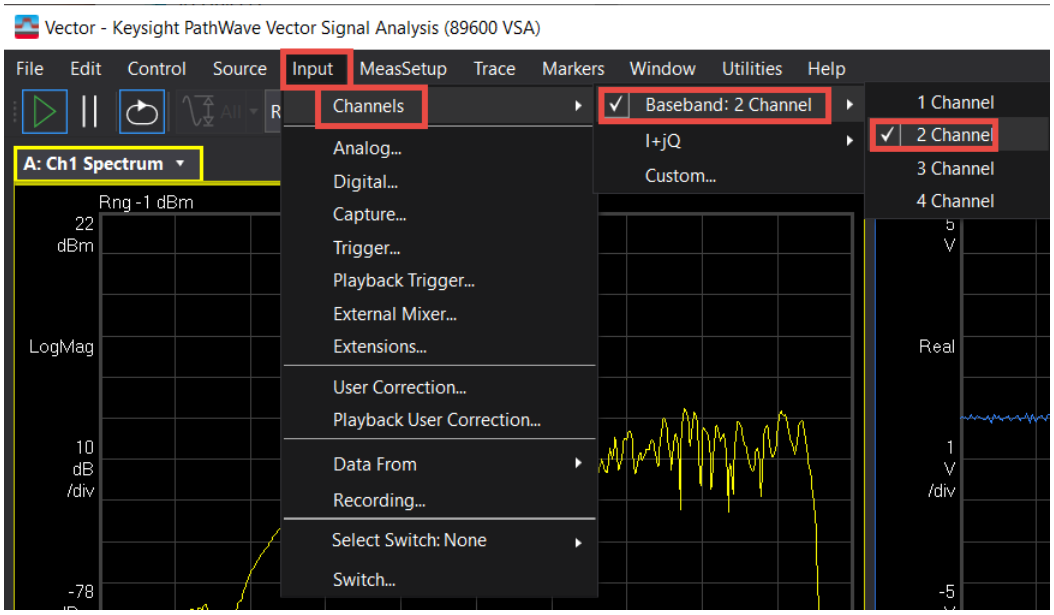
9. In the **Configurations** tab, set the Current Analyzer Configuration, select the Analyzer number for the new configuration. In this example **Analyzer 3**.



NOTE

The following steps perform the exact same function as loading the setup file you exported the VSA Setup File. At this point you can follow the steps below, or simply select File > Recall > Recall Setup and select the file you saved to the VXG.

10. From the menu bar, select **Input** > **Channels** > **RF** > **2 Channels**.

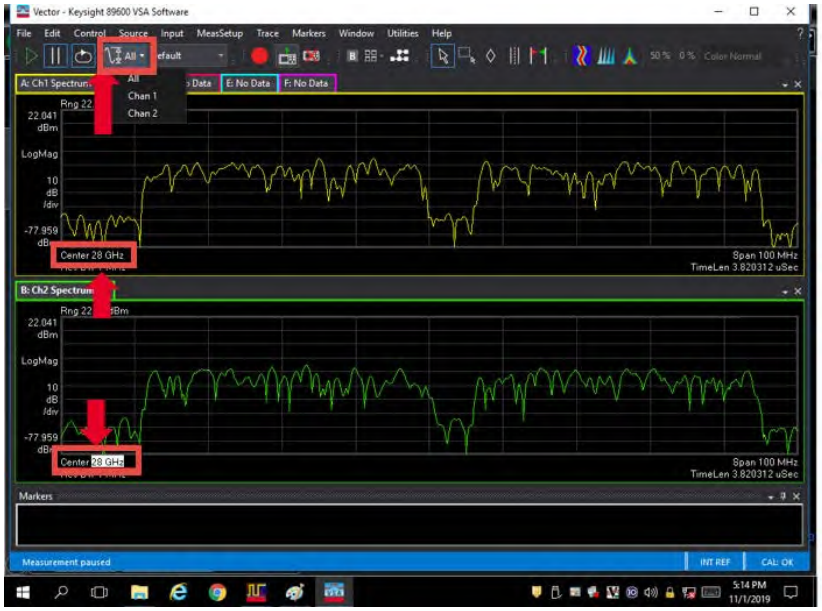


11. Set the center frequency of both channels to **28 GHz**, or the frequency you are using, and span to **100 MHz**.

You can also set the frequency and span of both channels under **Meas Setup** > **Frequency**.

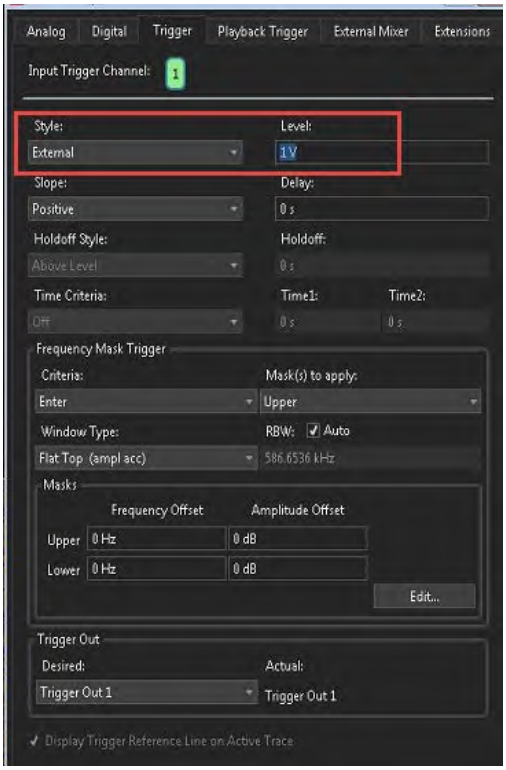
If you are using a UXR, Select Autorange .

Auto Range samples the current input signal and then sets the full scale input range to the minimum range that includes the peak voltage sample of the input signal.



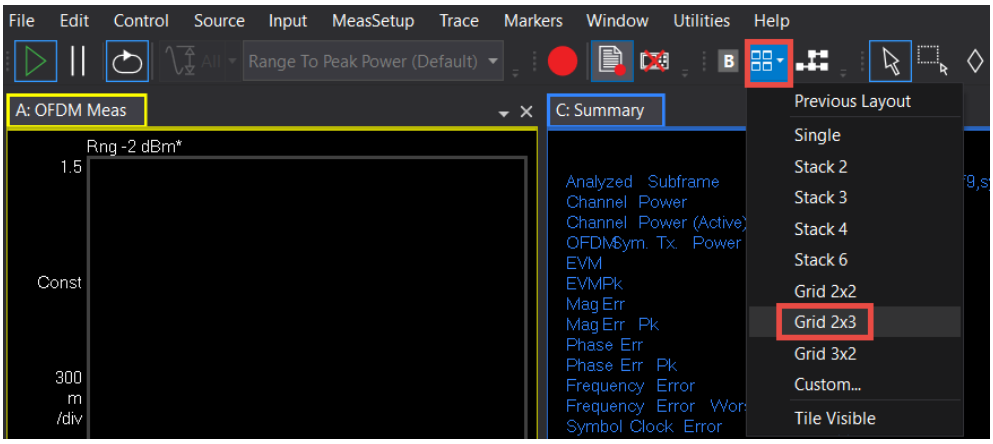
12. From the menu bar, select **Input** > **Trigger** and set:

- Style to **External**
- Level to **1 V**



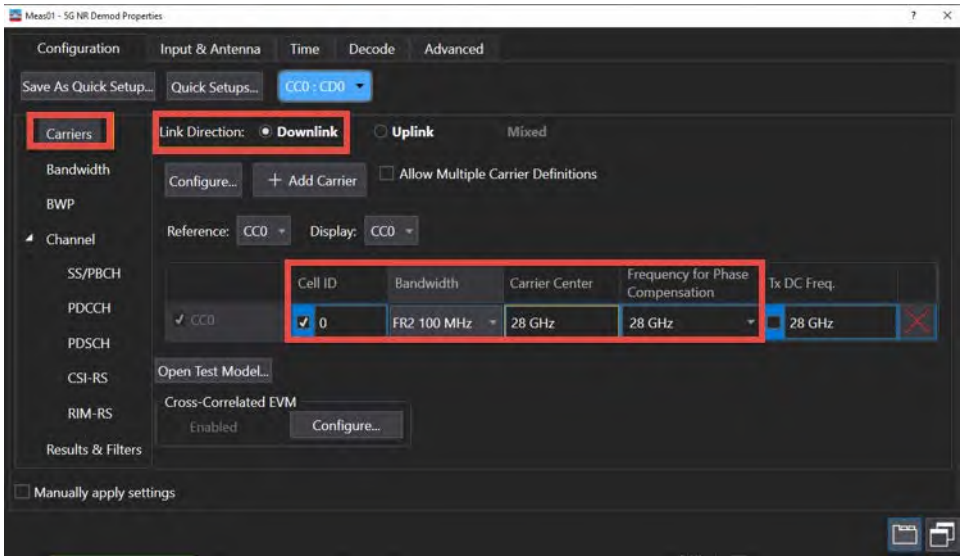
13. From the menu bar, select **MeasSetup** > **Measurement Type** > **Cellular** > **5G NR/5G-Advanced** > **5G NR Modulation Analysis**.

14. Change the trace layout to **Grid 2x3**.



TIP **To improve setup speed, Pause the measurement until all parameters are correctly configured, then run the measurement.**

15. Select **MeasSetup** > **5G NR Demod Properties** > **Configuration** tab and set that Bandwidth **FR2 100 MHz**, Carrier Center and Frequency for Phase compensation are set to **28 GHz**. Select the **Cell ID** checkbox.

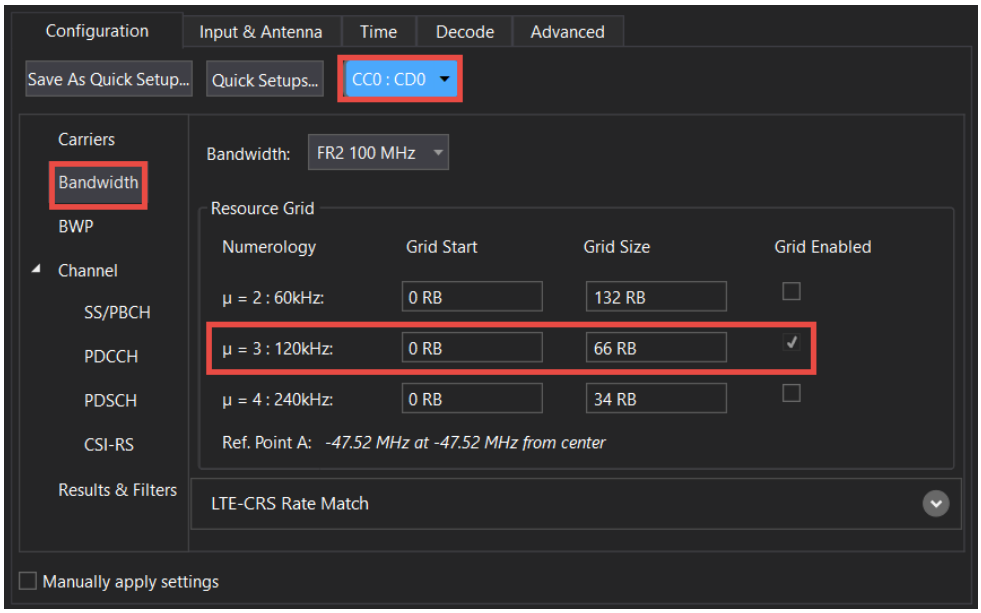


Cell ID is carried on PSS and SSS so that SS/PBCH must be transmitted and enabled for Auto Cell ID to work and must be a Downlink since SS/PBCH is only transmitted in the DL.

Phase compensation is per the 3GPP requirement, and is enabled by default. It is used to compensate for phase differences between symbols caused by upconversion or downconversion. Getting this setting wrong will cause a demodulation issue. For this example, it is ON and it is applied at the center frequency.

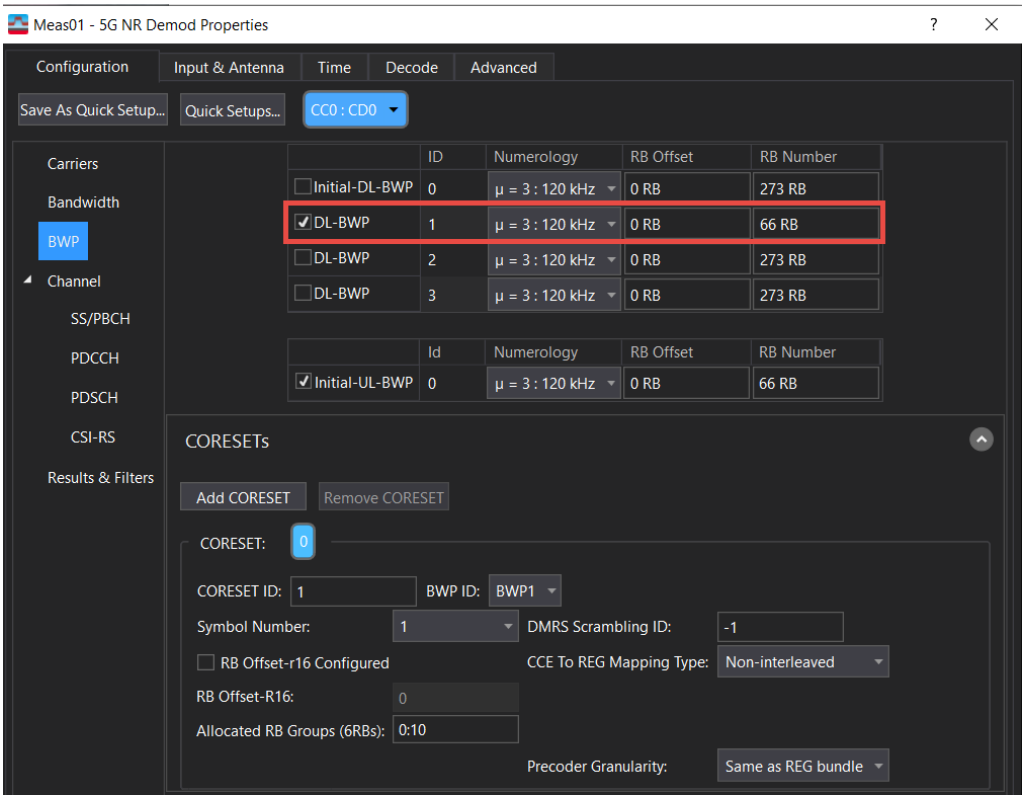
16. Select the **Bandwidth** pane and confirm that numerology is set to **μ= 3:120 kHz**.

The Bandwidth panel is used to configure the Resource Grid for each Numerology. Note that for FR2 100 MHz, the Max RB for 120 kHz Numerology is 66 RB. We will use this value when we configure the PDSCH parameters.



17. Select the **BWP** pane and confirm that **DL-BWP ID 1** is enabled.

For each BWP allocation, configure the numerology (μ) and RB allocation information. In this example, we will use the default 120 kHz numerology, RB Offset = 0 and RB number = 66 RB for a 100 MHz bandwidth signal.



18. Select the **Channel** pane > **SS/PBCH** and set:

a. RB Offset (60 kHz) to **0 RB**

b. Periodicity to **10 ms**

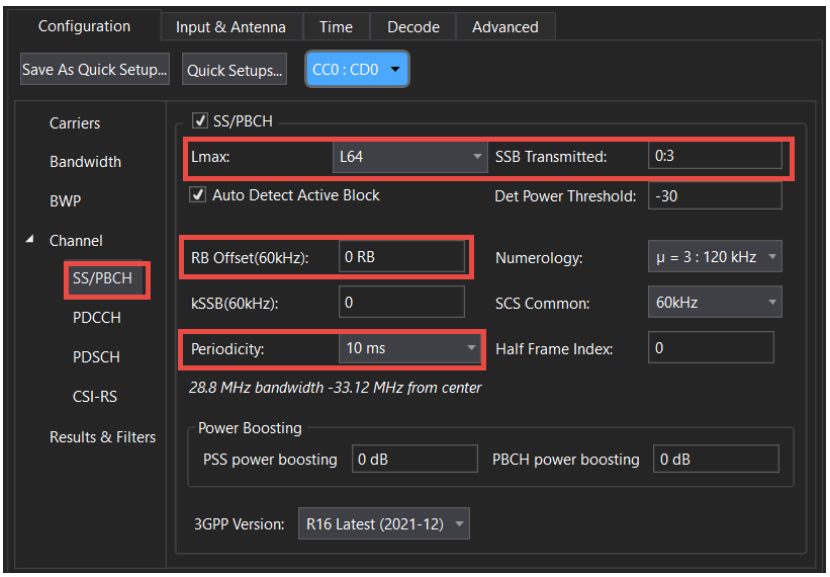
c. Lmax to **L64**

d. SSB Transmitted to **0:3**

For this example we will use 4 of the 64 SS/PBCH Blocks (beams). In an SSBlock, the period is different fr the different numerologies.

- FR1 up to 3 GHz, L=4
- FR1 3 GHz to 7.125 GHz, L8
- FR2-1, FR2-2, L = 64

The frequency location of an SSBlock is not fixed. In the VSA software, a default value of 46 RB and kSSB of 0 subcarriers places it in the center of the carrier bandwidth. Using 0 RB Offset puts the SSB at the lower edge of the carrier.



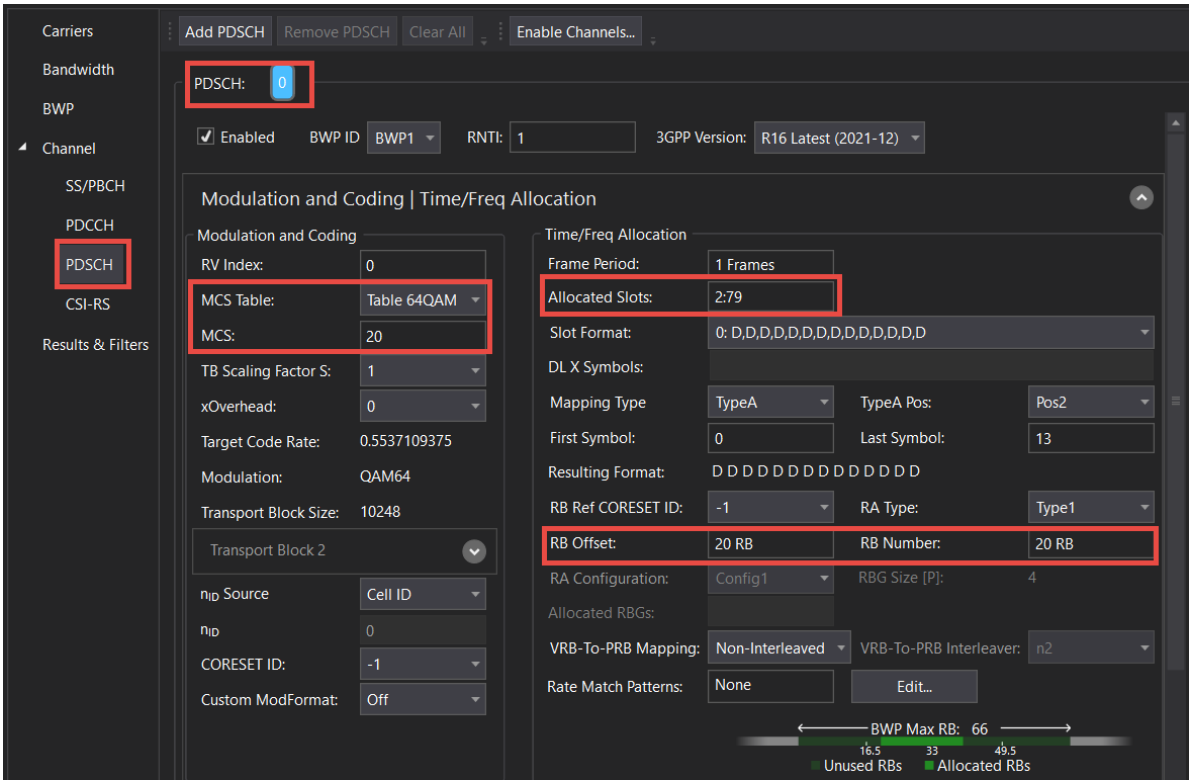
19. To configure the PDSCH0, select the **PDSCH** pane and for PDSCH 0, set:

- MCS Table to **Table 256 QAM**
- MCS to **20**

  3GPP has different Tables for PDSCH MCS.

  - Table 5.1.3.1-1 has 64QAM as max modulation
  - Table 5.1.3.1-2 has 264QAM as max modulation
  - Table 5.1.3.1-4 has 1024QAM as max modulation
  - Table 5.1.3.1-3 has 64QAM as max modulation and is for low spectrum efficiency (LowSE). See 3GPP TS38.214 for more information.
- Allocated Slots to **2:79**
- RB Offset to **20 RB**

– RB Number to **20 RB**



20. Scroll down and select the Antenna Port dropdown and set:

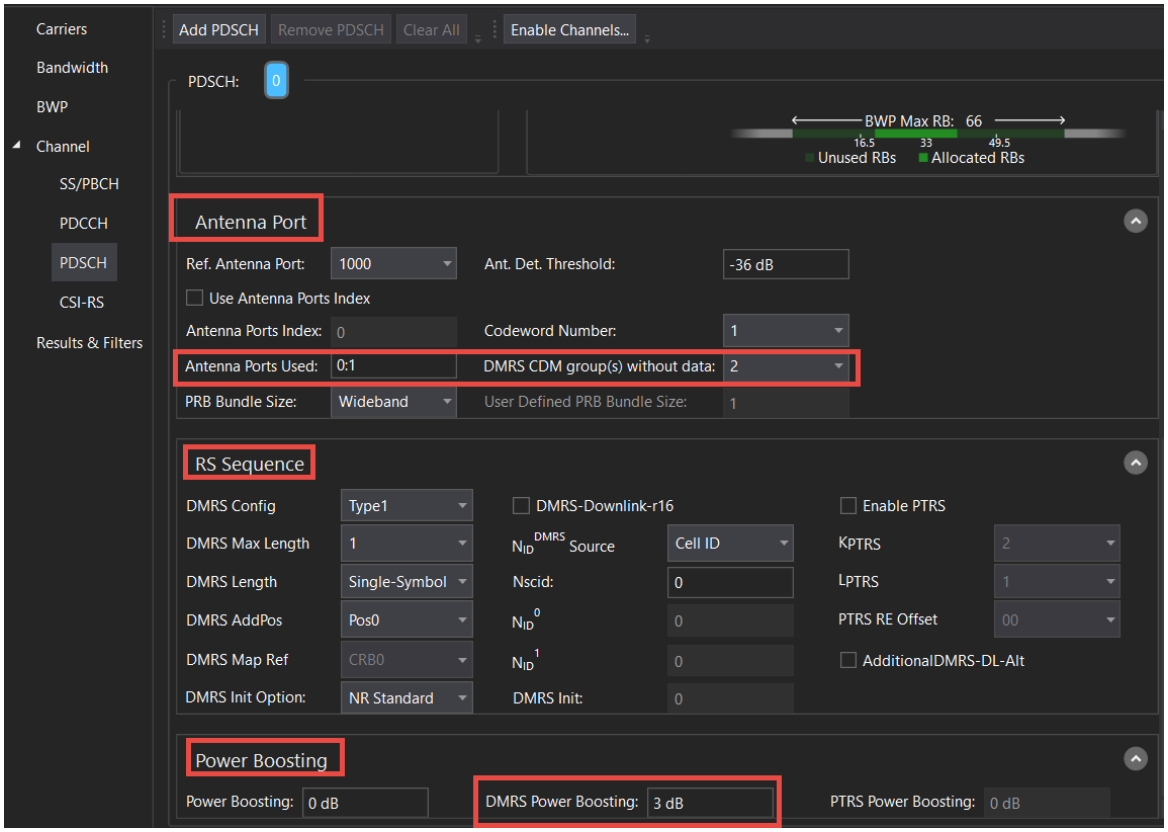
– Antenna Ports Used,1 to **0:1**

– DMRS CDM Group(s) without data to **2** (to match the PathWave setup)

21. Under the RS Sequence dropdown, we will use the default settings for DMRS and PTRS. However, this is where you would change the RS settings.

22. Select the **Power Boosting** dropdown and verify that DMRS Power Boosting has automatically been set to **3 dB**.

When "DMRS CDM group without data" is set to 2, the DMRS power boosting becomes 3 dB per 3GPP definition.

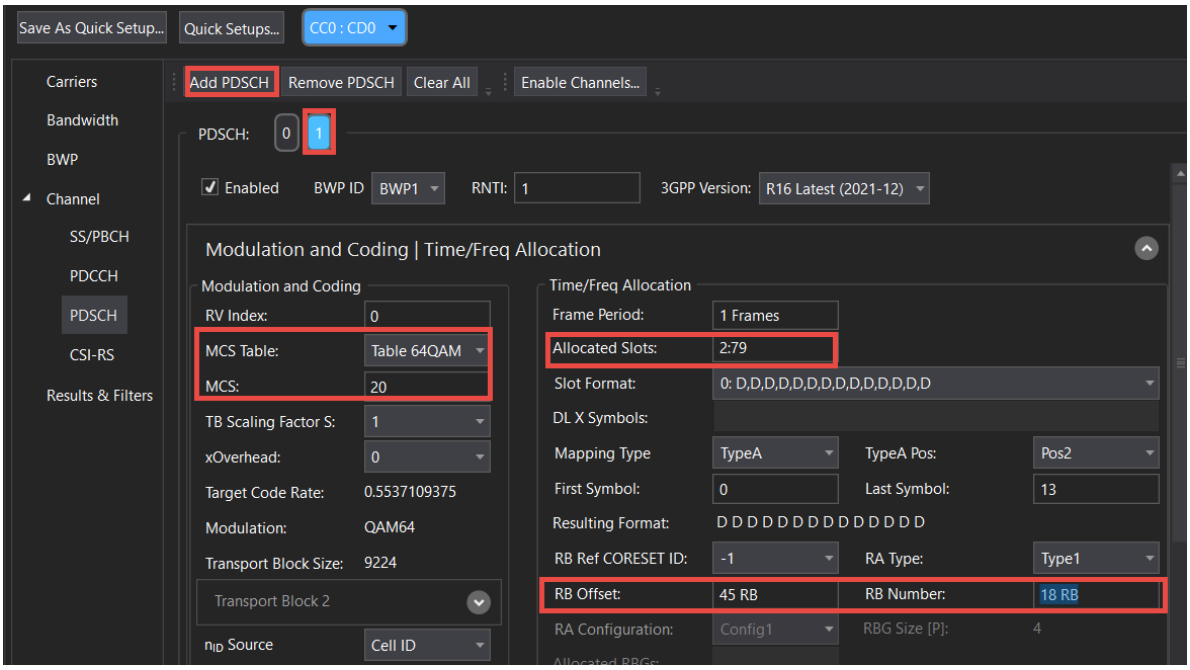


23. To add and configure PDSCH1, select **Add PDSCH**.

24. Select **PDSCH1** and set:

- MCS Table to **Table 64 QAM**
- MCS to **20**
- Allocated Slots to **2:79**
- RB Offset to **45 RB**

– RB Number to **18 RB**

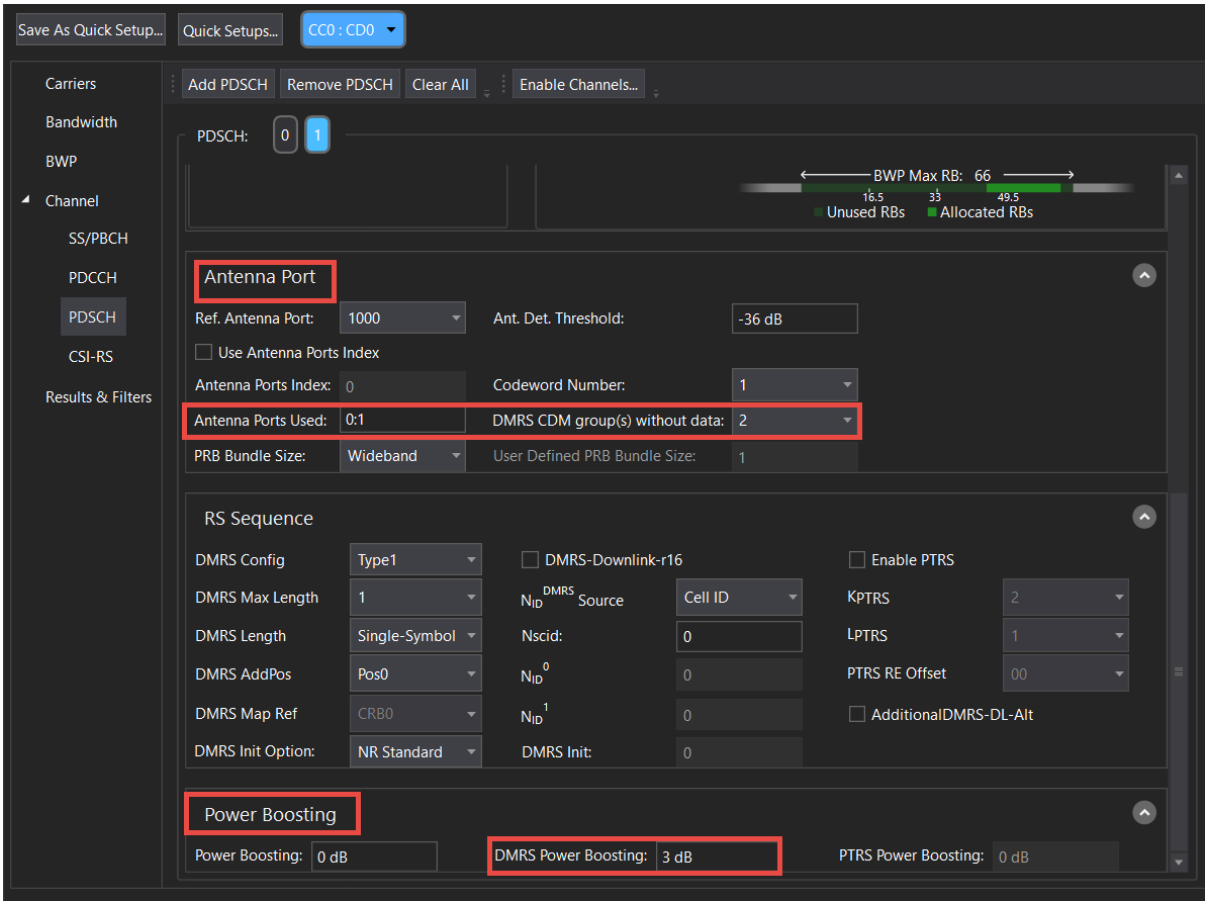


25. Scroll down and open the **Antenna Port** dropdown and set:

– Antenna Ports Used to **0:1**

– DMRS CDM group(s) without data to **2**.

26. Select the **Power Boosting** dropdown and verify that DMRS Power Boosting is set to **3 dB**.

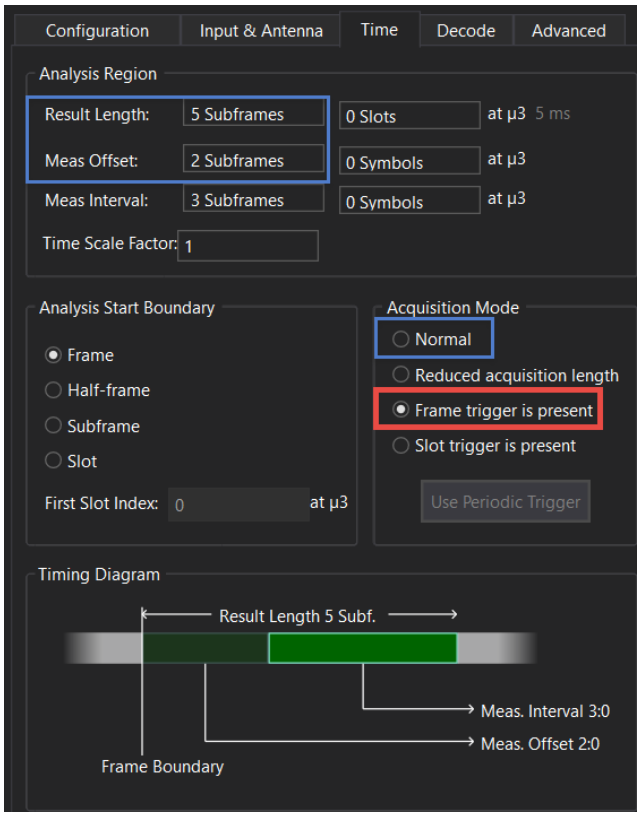


Now you should see the demodulated signal and EVM results per layer in the Frame Summary for PDSCHO and PDSCH1.

27. To configure the Analysis region, select the **Time** tab and set:

- Result Length to **5 Subframes**
- Meas Interval to **2 Subframes**
- Enable **Frame Trigger is present**.

We are using an external trigger for this example so selecting Frame Trigger is Present will use the external trigger and will speed up the measurement significantly. If not using an external trigger, set to Normal.



TIP **To improve setup speed**

Live measurement:

- Use external frame trigger and enable Frame trigger is present. The real frame boundary must be within ±50 µs of the external trigger.
- If external frame trigger is not available, use Reduced acquisition length. The reduced acquisition length may fail to synchronize in scenarios where > 1 frame SSB periodicity are defined, unless hardware triggering is used.
- Change Analysis Start Boundary to Subframe.
- Reduce Result Length and Measurement Interval.

Playback mode

- Reduce the Result Length and Measurement Interval.
- Set Analysis Start Boundary to Subframe.
- Reduce the number of active traces.

28. To configure the PBCH and PDSCH decoding, select the **Decode** tab and set.

- PBCH Decode to **Decoded TB**
- PDSCH Decode to **Decoded TB**
- PDSCH Decoder Algorithm to **Offset Min Sum** (default)

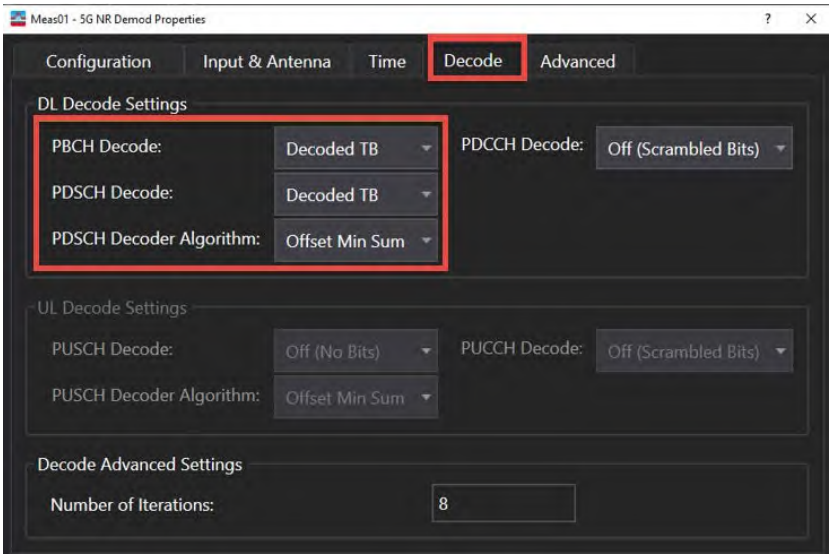
NOTE

VSA2023U2 or later releases support a user selectable Decoder Algorithm.

Offset Min-Sum (Default) - Specifies the layered belief propagation algorithm with offset min-sum approximation. This is faster but less accurate than Belief Propagation.

Belief Propagation - Specifies the belief-passing or message-passing algorithm. This is better for accuracy but slow. Use for receiver sensitivity test where the SNR is very small.

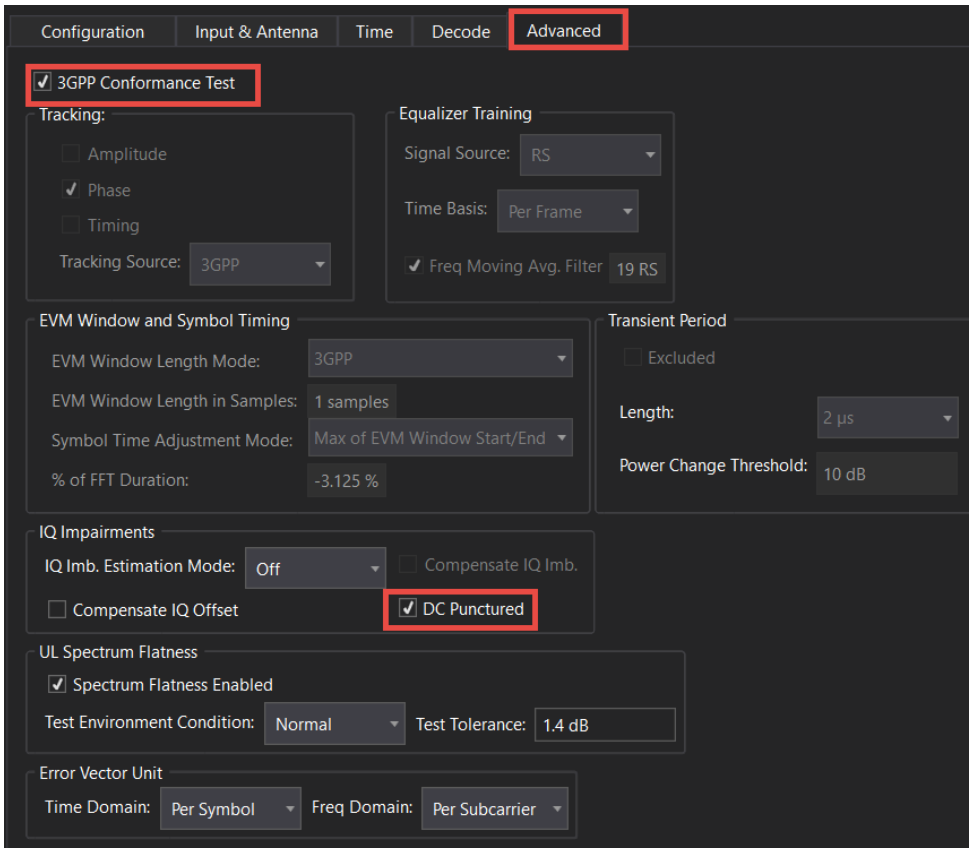
For successful PDSCH decoding (i.e. CRC pass), make sure the RNTI values (used for scrambling) and Transport Block Size matches between the transmitted signal and VSA. (Transport block size depends on MCS table, MCS value, TB Scaling Factor S, and xOverhead under PDSCH channel setting).



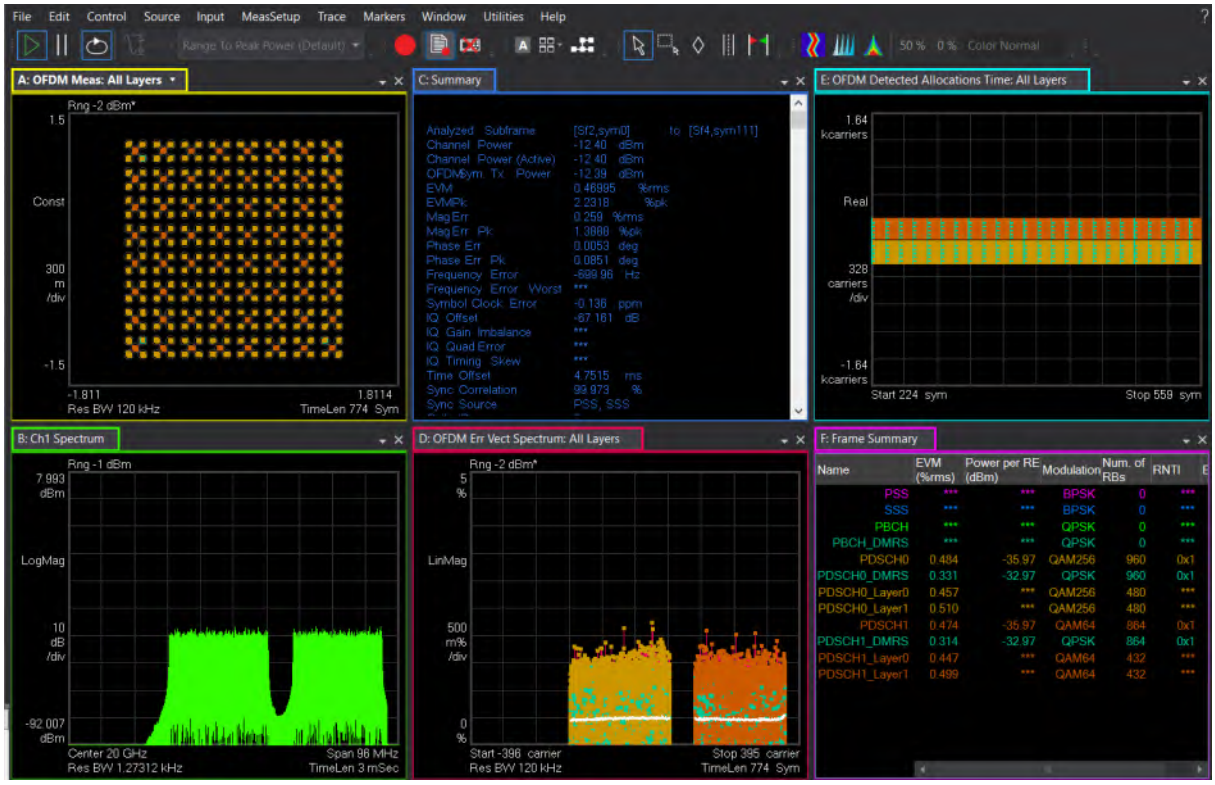
29. Select the **Advanced** tab and select the **DC Punctured** check box.

As part of conformance test, 3GPP has defined different equalizer training and tracking for EVM measurements. 3GPP Conformance Test is enabled by default where Tracking, Equalizer Training and EVM Window, and Symbol Timing is applied per 3GPP conformance test requirement. For FR1, no tracking is applied. For FR2, Phase Tracking using PTRS is applied.

5G NR counts the DC subcarrier as a valid subcarrier for rate-matching purposes. The network decides if the DC subcarrier is modulated or not. High LO feedthrough will impact demodulation and EVM performance of the input signal. This is OFF by default.



30. Close the 5G NR Properties dialog, start and new sweep to update the display, then view the results.



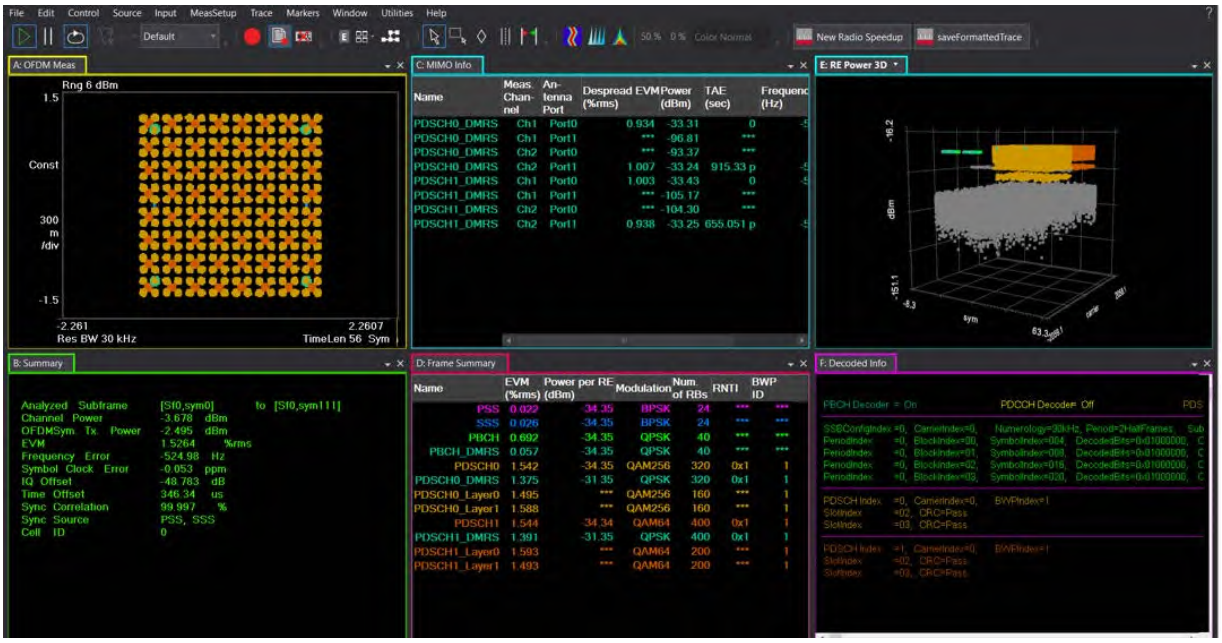
- Trace A: Composite constellation diagram showing different modulation formats for the different channels/signals present in the transmitted signal. See Trace F (Frame Summary) for the list of channels and modulation formats.
- Trace C: Summary trace showing composite error metrics.
- Trace D: Error Vector Spectrum showing EVM versus subcarrier and symbol.
- Trace E: Detected Allocations Time showing the detected allocations of all channels/signals within the measurement interval.
- Trace F: Frame Summary. EVM, per Layer EVM, Power per RE, Mod Format, Number of RB, RNTI, and BWP ID of the individual channels/signals.

One of the most powerful tools in the 89600 VSA is the coupling of markers across different measurements, traces, and domains. Coupled markers allow you to understand the identity and characteristics of a symbol simultaneously in time, frequency, and error.

Place a marker on Traces A, D, and E and couple the markers (Markers > Couple Markers), and then show the results in a markers window (Window > Markers).

Right click on Trace D and select **Peak**. The exact symbol associated with this peak EVM can now be understood in terms of time domain symbol index, frequency domain subcarrier number, channel type, modulation format, IQ magnitude, and phase values. Other parameters are shown in the Marker window.

Change the traces so you can see the Constellation diagram, MIMO Info table, Frame Summary Table (or Slot Summary Table), Summary table, 3D Power, Decoded Info plus any additional traces.



– Trace C: MIMO Info table shows EVM, power, and time, frequency and phase offset for each antenna port.

  MIMO Info is a type of Matrix Table that provides sortable rows by column, selectable column visibility, and copy/paste and export functionality to share rows of content or complete tables to applications like email, text editing or spreadsheet programs.

– Trace E: 3D trace showing resource element (RE) power per symbol and per subcarrier. This is very useful when verifying base station systems, for example, to make sure the power per each RE is balanced. You can use a marker to read the power and channel information. A marker on this trace cannot be coupled with other traces.

  – Press the mouse wheel and rotate the wheel forward to zoom in, backward to zoom out.

  – Hold left-click of the mouse for panning. Note: If marker is enabled (i.e. not a normal pointer), you will need to hold Alt in order to pan.

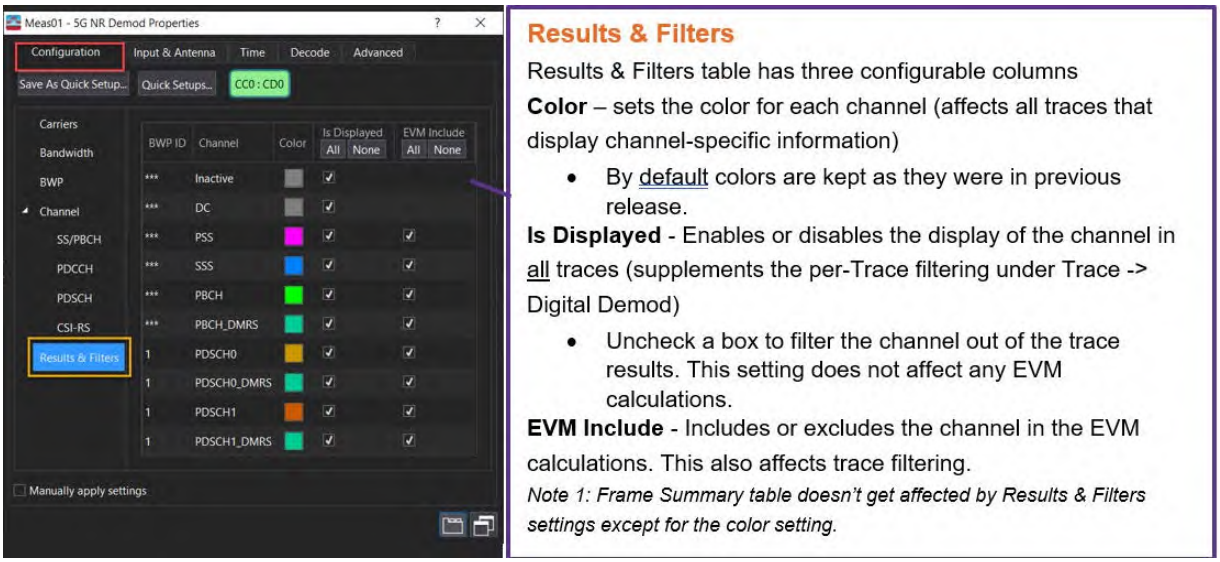
- Use right click to auto scale, enable the marker, and change the display to different axes.

- Trace F: The Decoded Info table provides CRC pass/fail for PDSCH and PBCH plus the high layer information that is carried on PBCH.

  If CRC of PDSCH fails, make sure the Transport Block Size matches between the transmitted signal and VSA (this depends on MCS table, MCS value, TB Scaling Factor S, and xOverhead under PDSCH channel setting). Also, make sure the RNTI values match since RNTI is used for scrambling.

Result and Display Filtering

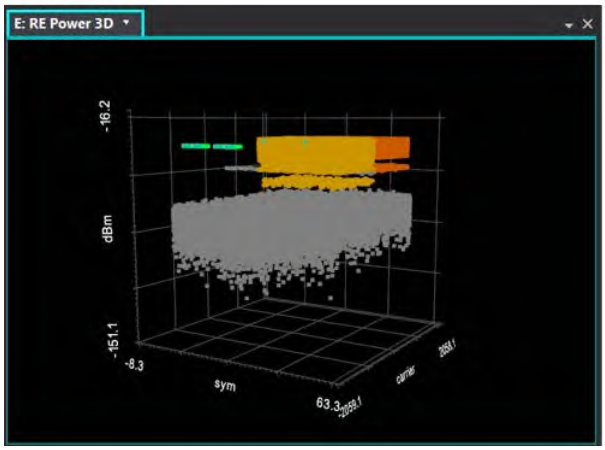
Meas Setup > NR Demod Properties > Configuration tab > Results & Filters panel.



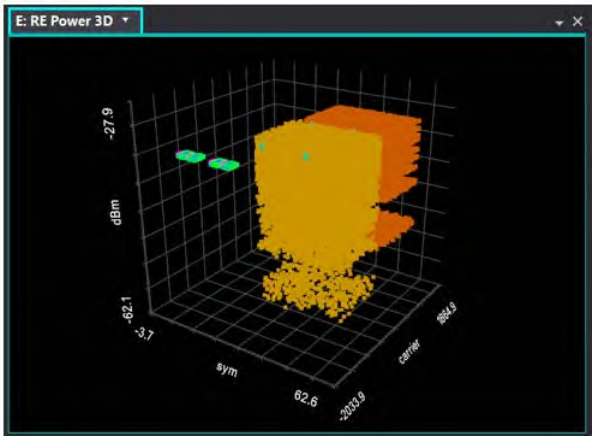
You can try to change colors of some of the channels, and/or filtering out all channels/signals except for SSBlock (PSS, SSS & PBCH) etc.

Clear (Uncheck) "Inactive" and "DC" to remove the inactive resources (subcarriers, symbols) and the DC subcarrier from the display and see the change in the 3D plot:

Before:



After:



You can now auto-scale the 3D plot and only see the active channels/signals being displayed.

Per-Trace Filtering

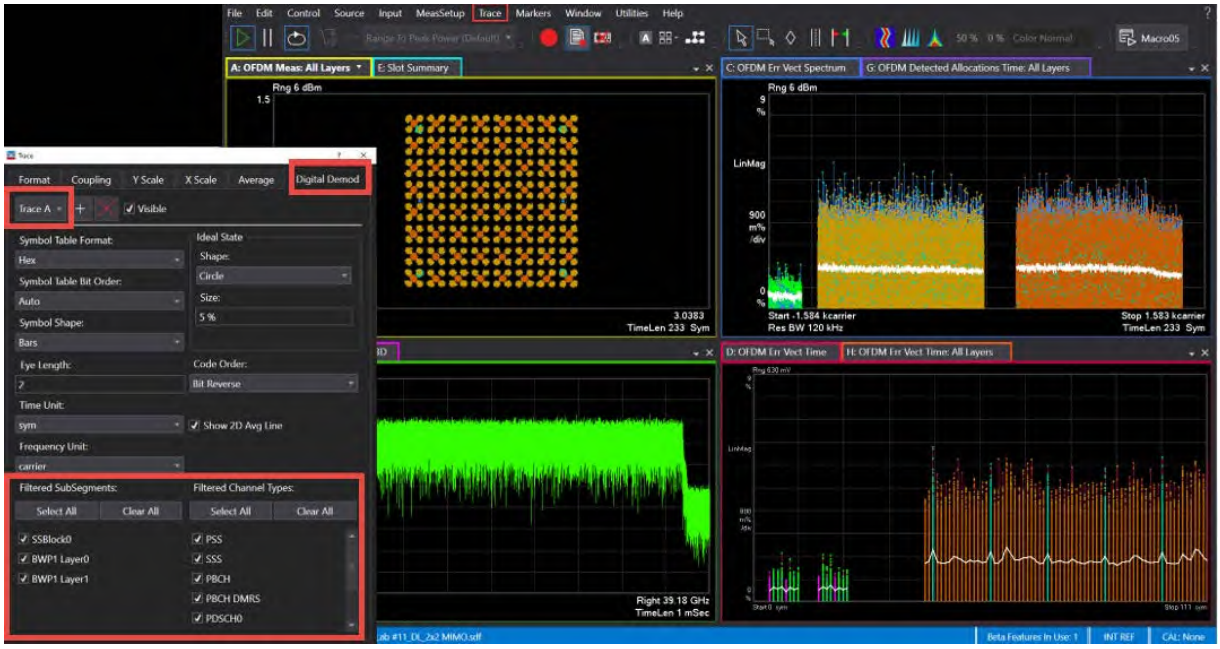
Trace > Digital Demod

In addition to the Results & Filters, VSA also has per-trace filtering to filter by subsegments and/or channels. Subsegment means MIMO Layers, BWP and SSBlock.

The following traces can be filtered by subsegments:

- OFDM Meas
- OFDM Ref
- Error Vector Spectrum
- Error Vector Time

– Detected Allocations Time trace



To setup the VSA Using a UXR Recorded Waveform File

1. Open the VSA software.

   To access the VSA software, go to the Windows Start menu and find Keysight 89600 Software (latest installed version) folder and run the software.

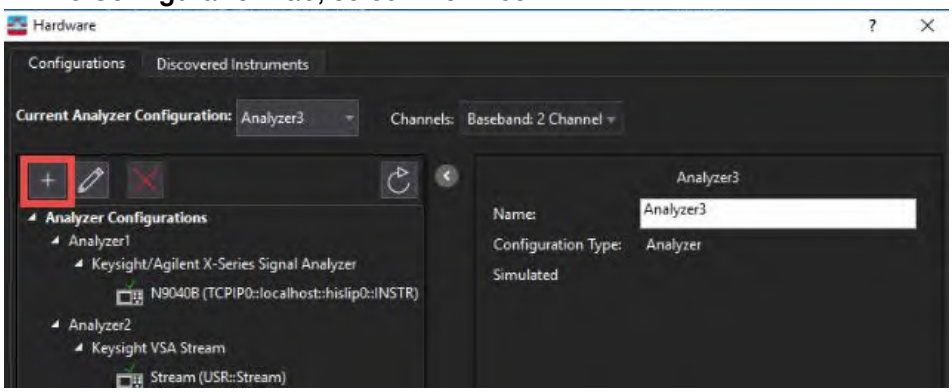
NOTE

If accessing the instrument via a Remote Desktop connection, select the Screen tab (at the top of the display) to open the Mode/Measurement/View Selector window.

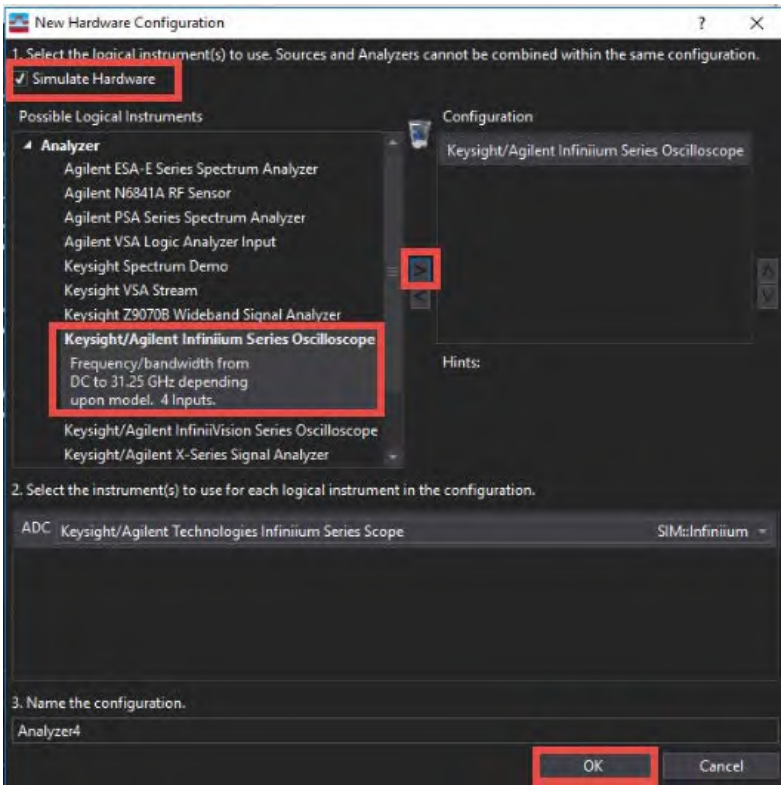
2. From the VSA menu bar, select **File** > **Preset** > **All** to set the VSA to a known state.
3. To configure the hardware, go to **Utilities** > **Hardware** > **Configurations**.

   If it is already configured, continue with step 9.

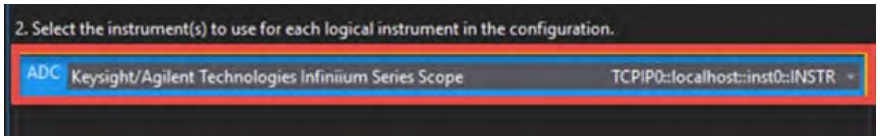
4. In the **Configuration** tab, select the **+ icon**.



5. In the New Hardware Configuration dialog, select **Simulate Hardware** and scroll down the Possible Logical Instruments and select **Keysight/Agilent Infiniium Series Oscilloscope**, and then select the right arrow to move it under Configuration.

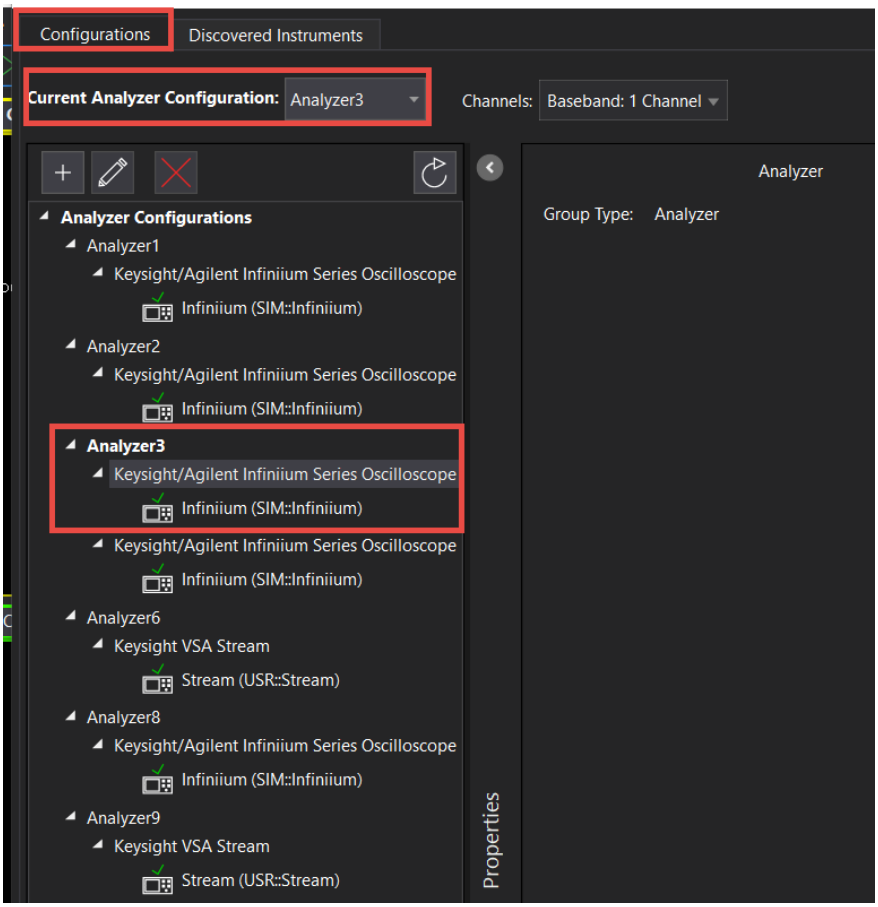


6. From the middle of the dialog box, select the UXR to be used.



7. Select **OK** to create the UXR configuration.

8. In the **Configurations** tab, set the Current Analyzer Configuration, select the Analyzer number for the new configuration. In this example **Analyzer 3**.



9. If you do not have access to a Keysight Infiniium UXR Real-Time Oscilloscope, a UXR recorded waveform file has been included in the Examples waveform file folder on the VXG.

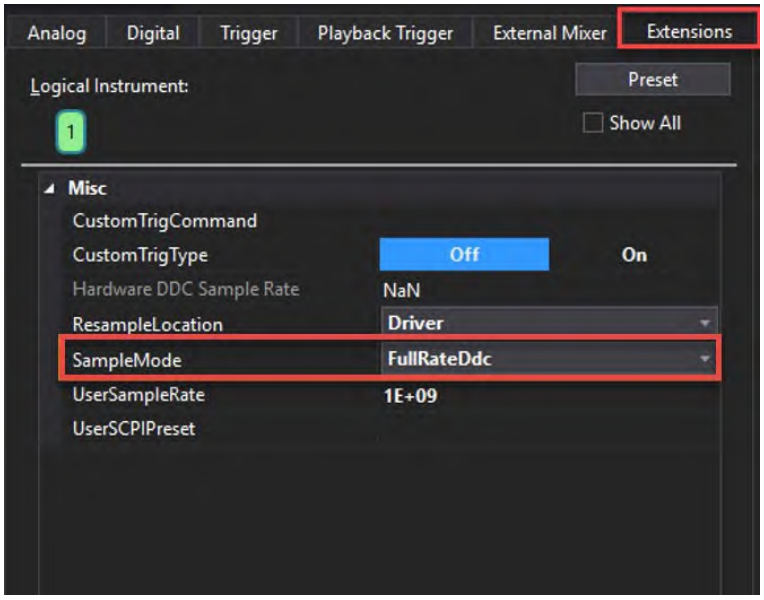
   Go to: `D:\Users\Instrument\Documents\Keysight\PathWave \SignalGenerator\Examples`

   Copy the **MIMO_UXR_Recording.sdf** file to the signal analyzer or the PC running the VSA application.

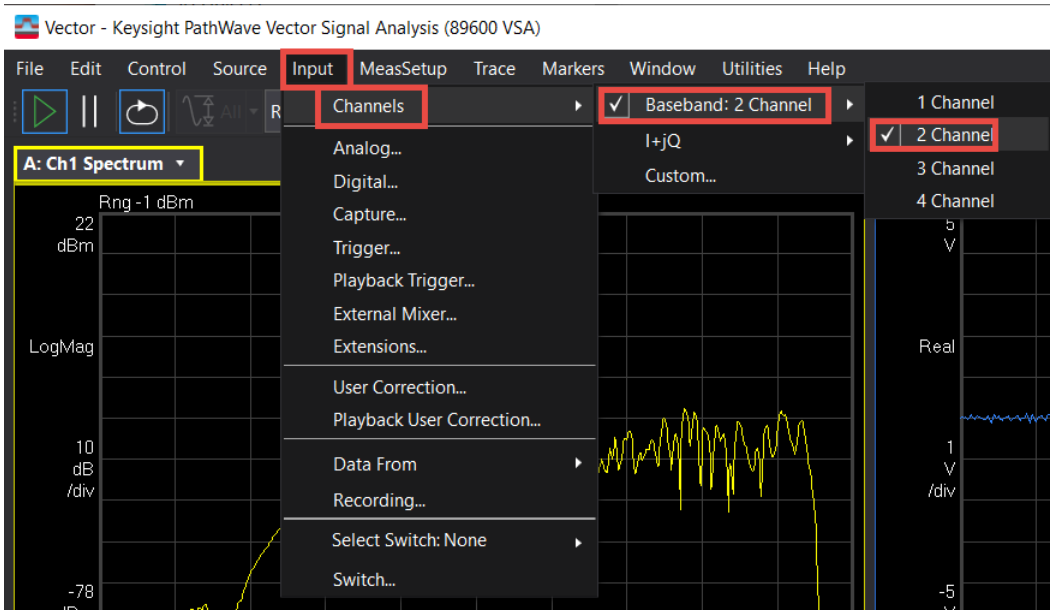
10. From the Menu Bar, select **File** > **Recall** > **Recall Recording** and select `MIMO_UXR_Recording.sdf`.

NOTE Recalling the recording will setup all of the parameters described below. If you want to become familiar with the VSA setup, continue on with the remaining steps.

11. From the menu bar, select **Input** > **Extensions** > **Sample Mode** >**FullRate DDC**.



12. From the menu bar, select **Input** > **Channels** > **RF** > **2 Channels**.



13. Set the center frequency of both channels to **28 GHz** and span to **100 MHz**.

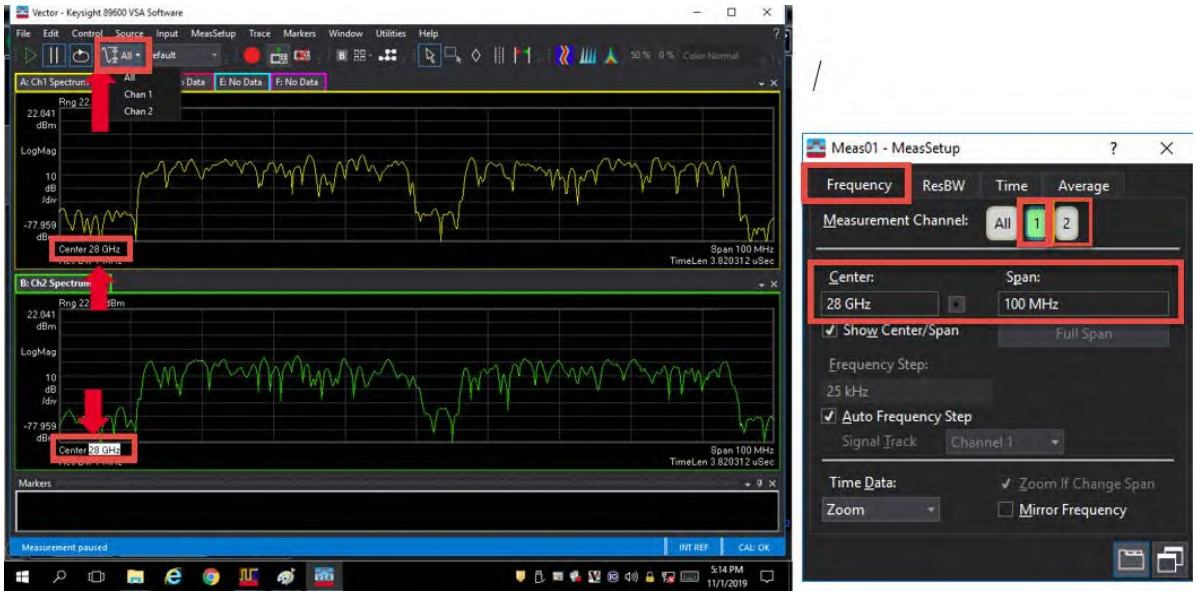
    You can also set the frequency and span of both channels under Meas Setup > Frequency.

If you are using a UXR, Select Autorange .

NOTE

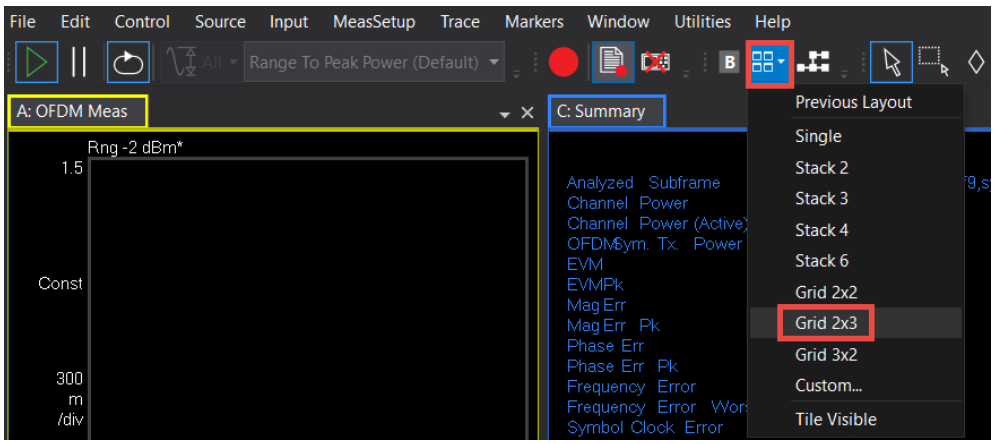
Autorange does not work if you are simulating the waveform with the UXR recording. Continue with the next step.

Auto Range samples the current input signal and then sets the full scale input range to the minimum range that includes the peak voltage sample of the input signal.

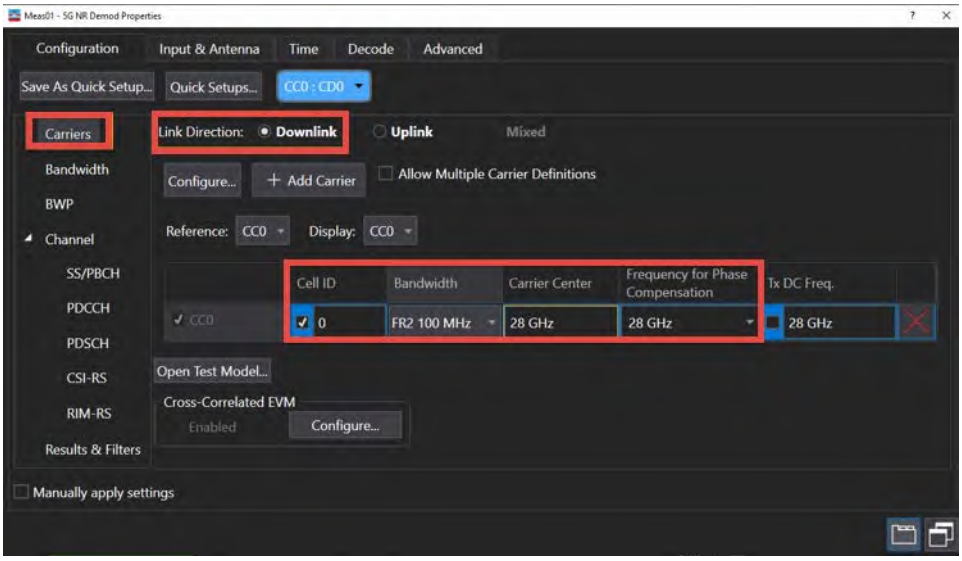


14. From the menu bar, select **MeasSetup** > **Measurement Type** > **Cellular** > **5G NR/5G-Advanced** > **5G NR Modulation Analysis**.

15. Change the trace layout to **Grid 2x3**.



16. Select **MeasSetup** > **5G NR Demod Properties** > **Configuration** tab and set Bandwidth to **FR2 100 MHz**, Carrier Center and Frequency for Phase compensation are set to 28 GHz. Select the **Cell ID** checkbox.

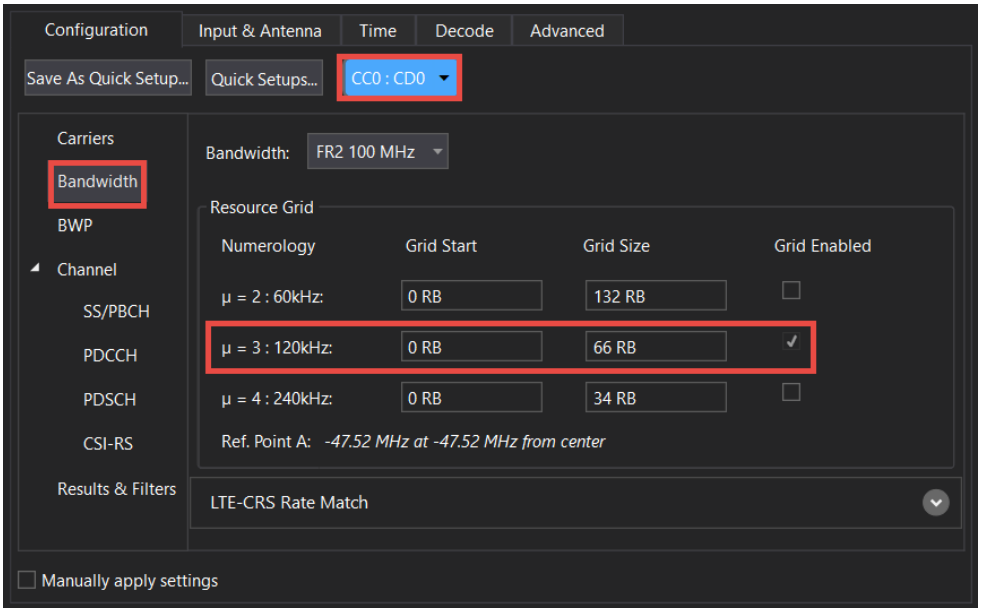


Cell ID is carried on PSS and SSS so that SS/PBCH must be transmitted and enabled for Auto Cell ID to work and must be a Downlink since SS/PBCH is only transmitted in the DL.

Phase compensation is per the 3GPP requirement, and is enabled by default. It is used to compensate for phase differences between symbols caused by upconversion or downconversion. Getting this setting wrong will cause a demodulation issue. For this example, it is ON and it is applied at the center frequency.

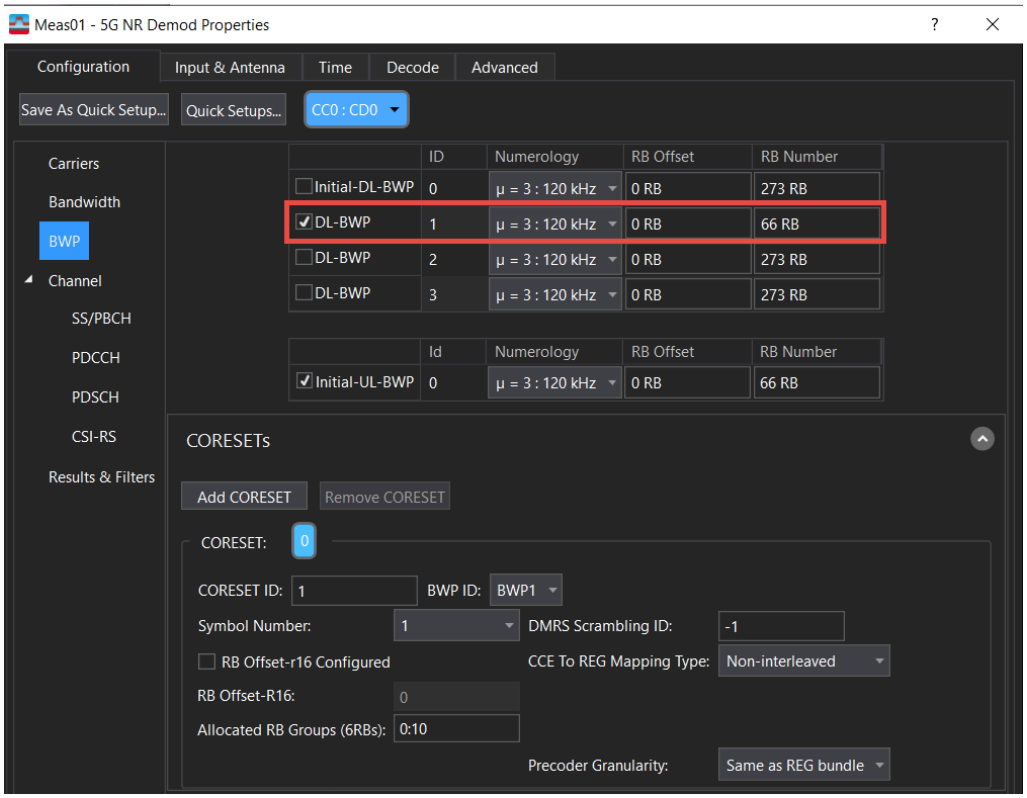
17. Select the **Bandwidth** pane and confirm that numerology is set to **μ= 3:120 kHz**.

The Bandwidth panel is used to configure the Resource Grid for each Numerology. Note that for FR2 100 MHz, the Max RB for 120 kHz Numerology is 66 RB. We will use this value when we configure the PDSCH parameters.



18. Select the **BWP** pane and confirm that **DL-BWP ID 1** is enabled.

For each BWP allocation, configure the numerology (μ) as well as RB allocation information. In this lab, we will use the default 120 kHz numerology, RB Offset = 0 and RB number = 66 RB for a 100 MHz bandwidth signal.



19. Select the **Channel** pane > **SS/PBCH** and set:

a. RB Offset (60 kHz) to **0 RB**

b. Periodicity to **10 ms**

c. Lmax to **L64**

d. SSB Transmitted to **0:3**

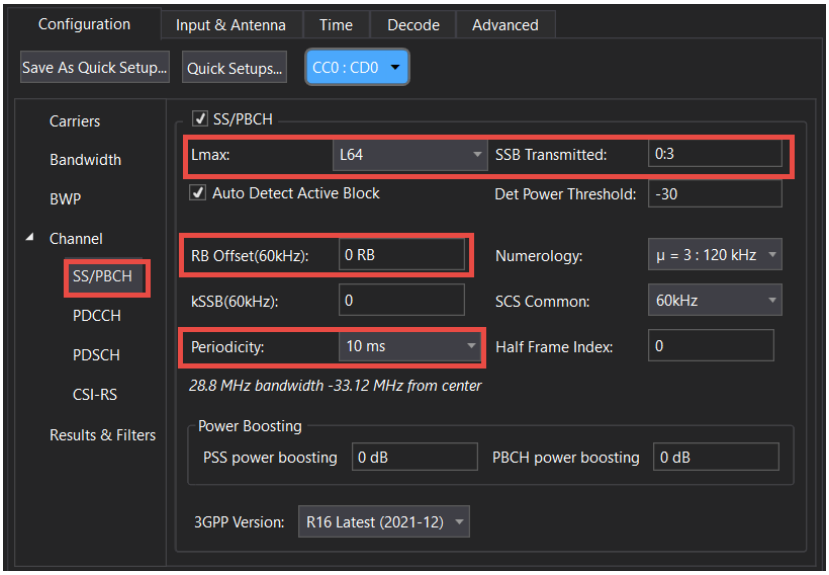
For this example we will use 4 of the 64 SS/PBCH Blocks (beams). In an SSBlock, the period is different fr the different numerologies.

– FR1 up to 3 GHz, L=4

– FR1 3 GHz to 7.125 GHz, L8

– FR2-1, FR2-2, L = 64

The frequency location of an SSBlock is not fixed. In the VSA software, a default value of 46RB and kSSB of 0 subcarriers places it in the center of the carrier bandwidth. Using 0 RB Offset puts the SSB at the lower edge of the carrier.



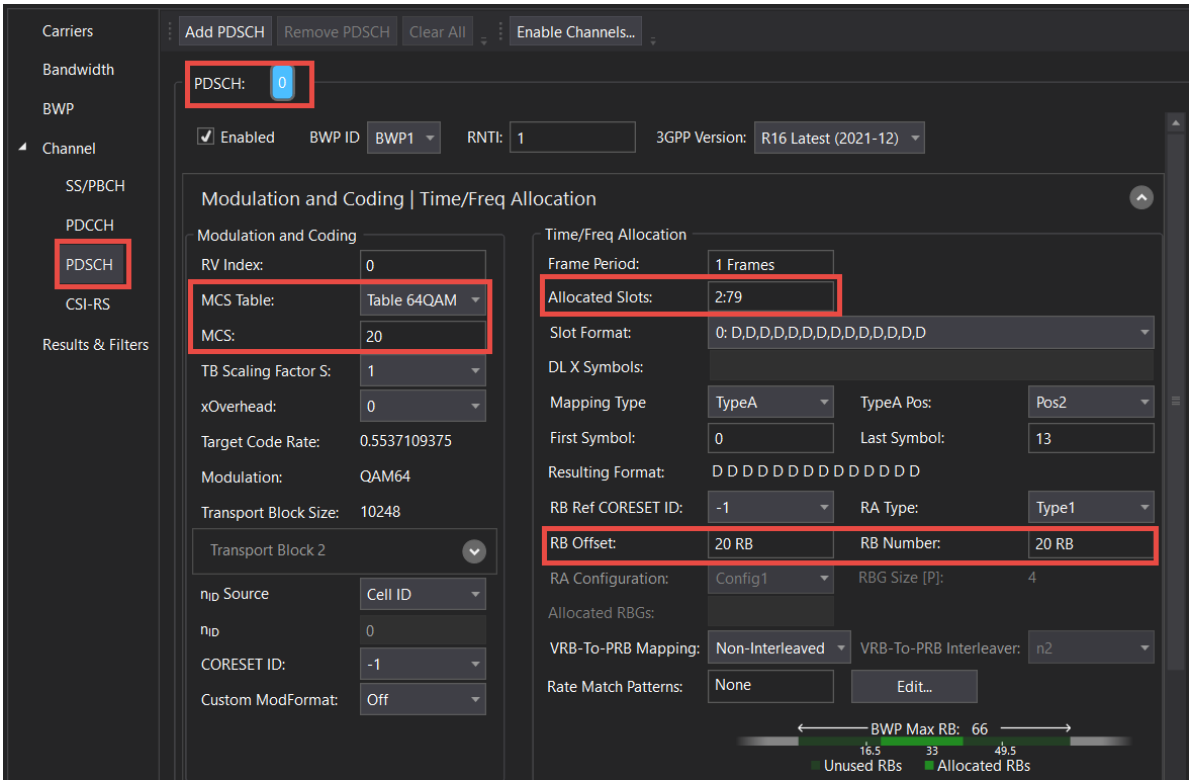
20. To configure the PDSCH0, select the **PDSCH** pane and for PDSCH 0, set:

- MCS Table to **Table 256 QAM**
- MCS to **20**

  3GPP has different Tables for PDSCH MCS

  - Table 5.1.3.1-1 has 64QAM as max modulation
  - Table 5.1.3.1-2 has 264QAM as max modulation
  - Table 5.1.3.1-4 has 1024QAM as max modulation
  - Table 5.1.3.1-3 has 64QAM as max modulation and is for low spectrum efficiency (LowSE). See 3GPP TS38.214 for more information.
- Allocated Slots to **2:79**
- RB Offset to **20 RB**

– RB Number to **20 RB**



21. Scroll down and select the Antenna Port dropdown and set:

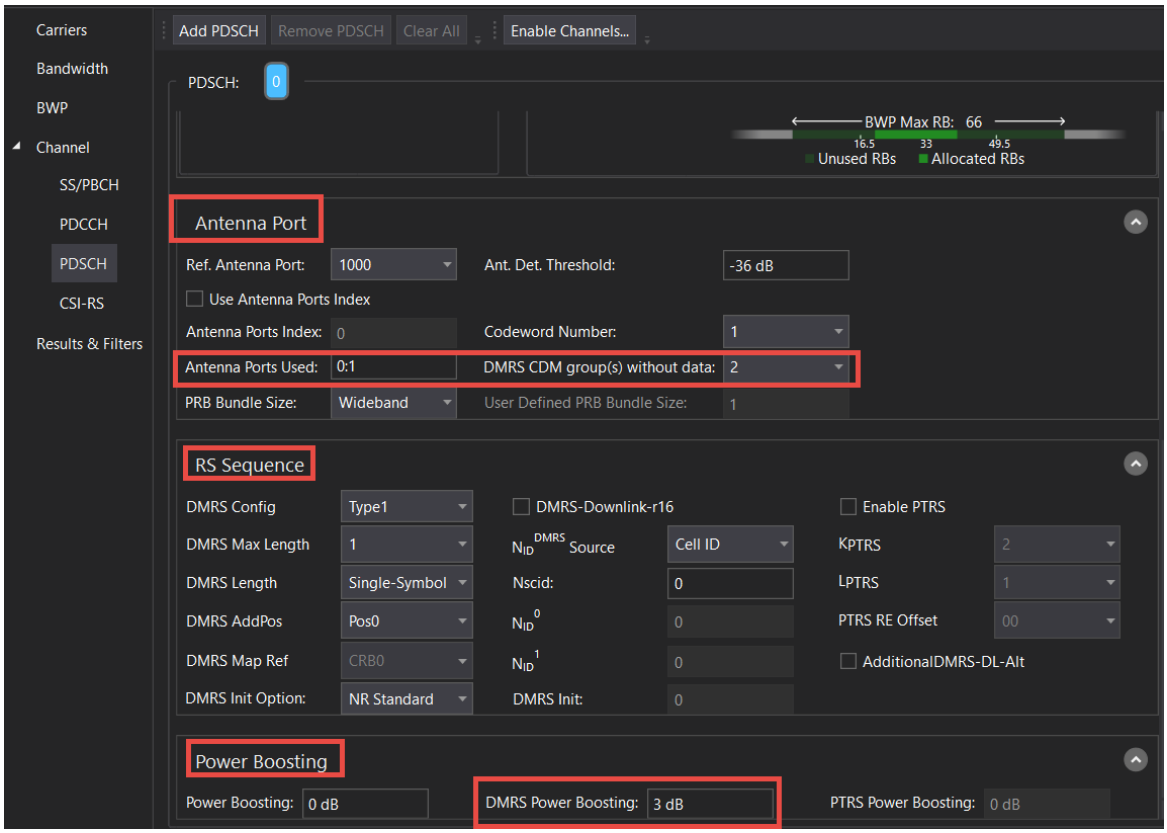
– Antenna Ports Used,1 to **0:1**

– DMRS CDM Group(s) without data to **2** (to match the PathWave setup)

22. Under the RS Sequence dropdown, DMRS and PTRS are default settings, but this is where you would change the RS settings.

23. Select the **Power Boosting** dropdown and verify that DMRS Power Boosting has automatically been set to **3 dB**.

When "DMRS CDM group without data" is set to 2, the DMRS power boosting becomes 3 dB per 3GPP definition.

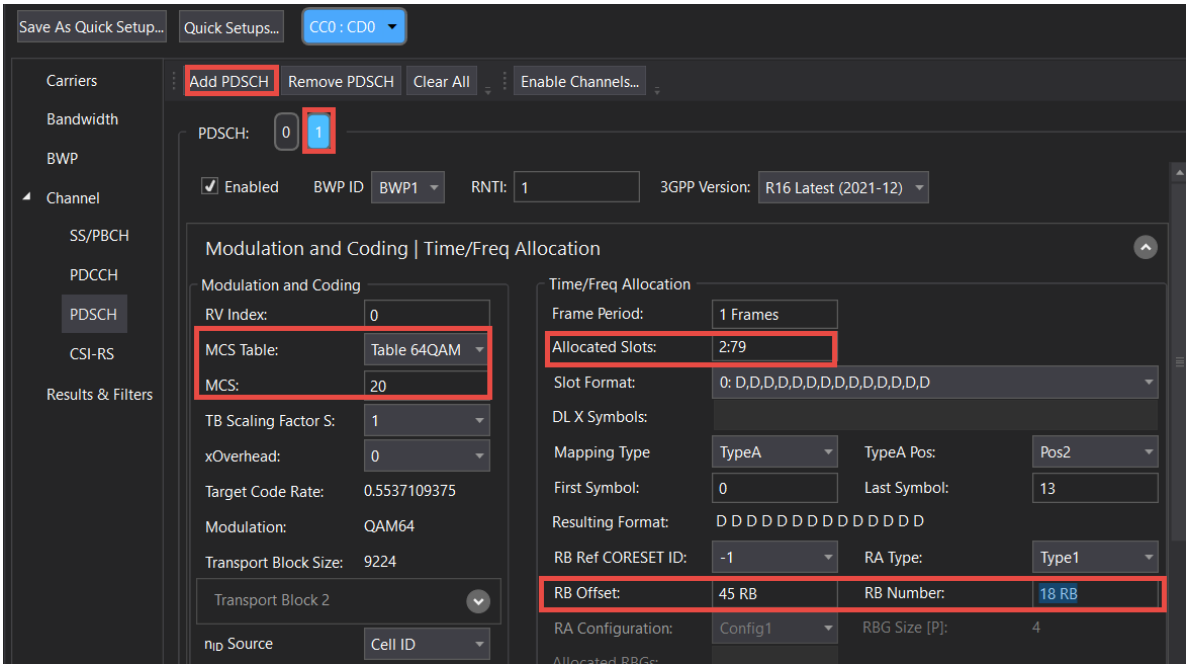


24. To add and configure PDSCH1, select **Add PDSCH**.

25. Select **PDSCH1** and set:

- MCS Table to **Table 64 QAM**
- MCS to **20**
- Allocated Slots to **2:79**
- RB Offset to **45 RB**

– RB Number to **18 RB**

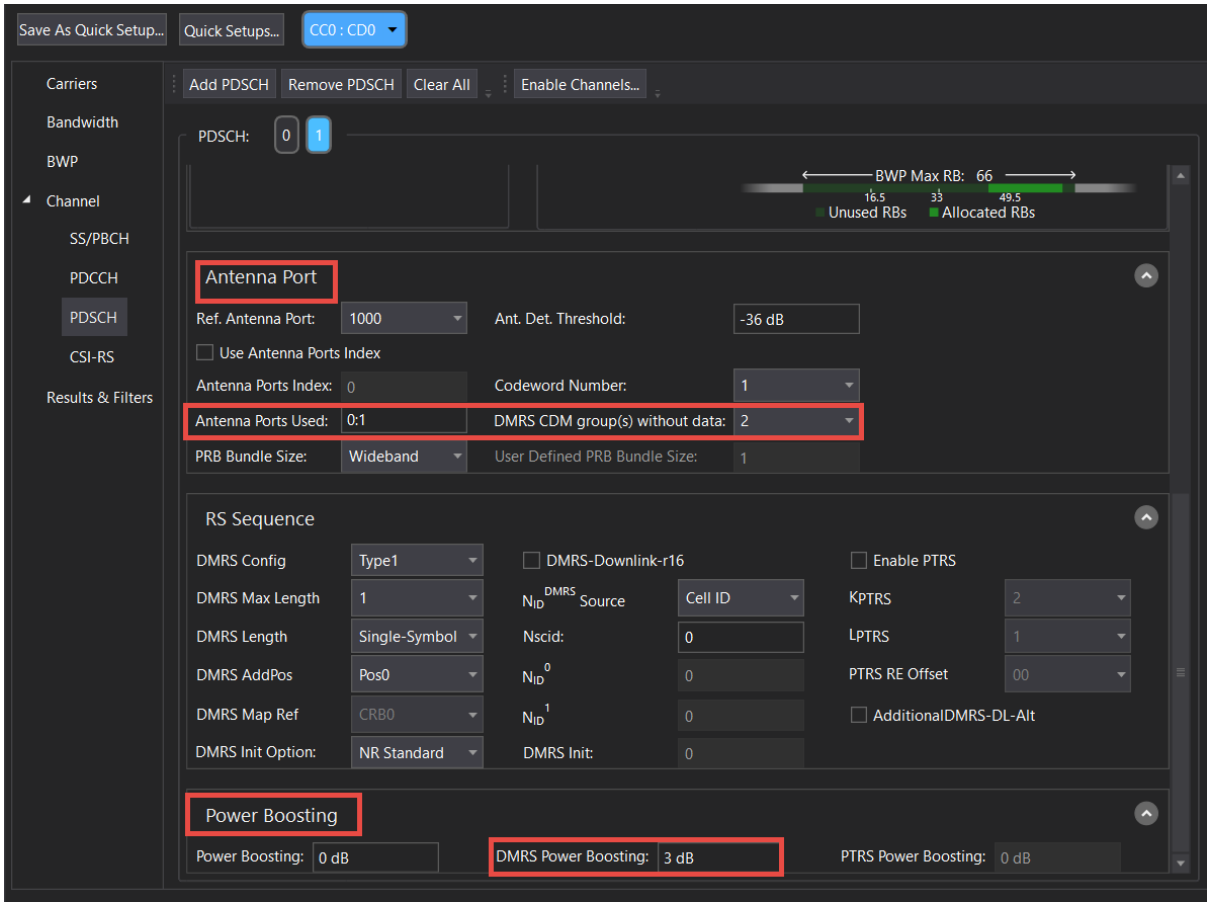


26. Scroll down and open the **Antenna Port** dropdown and set:

– Antenna Ports Used to **0:1**

– DMRS CDM group(s) without data to **2**.

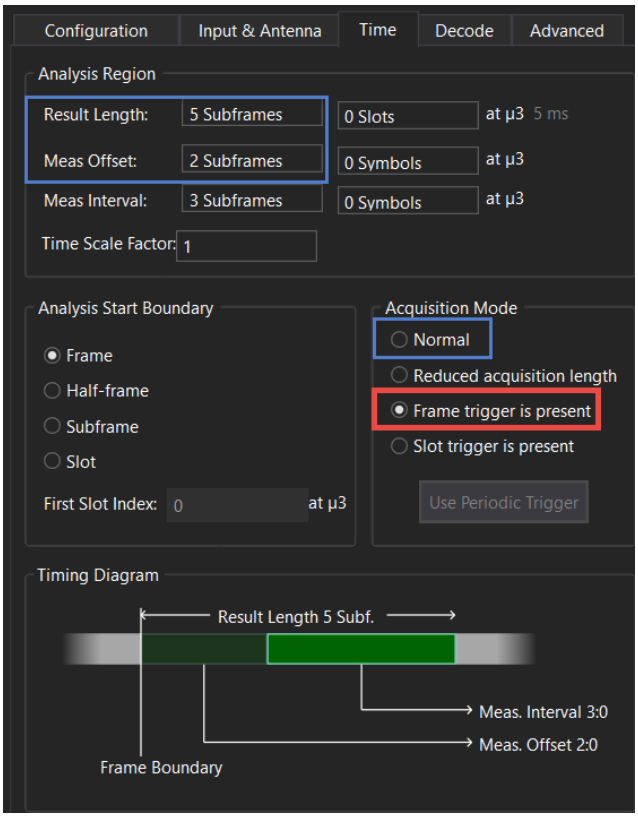
27. Select the **Power Boosting** dropdown and verify that DMRS Power Boosting is set to **3 dB**.



28. To configure the Analysis region, select the **Time** tab and set:

- Result Length to **5 Subframes**
- Meas Interval to **2 Subframes**
- Enable **Frame Trigger is present**. **NOTE:** if using the UXR recording, leave Frame Trigger set to **Normal**.

The example measurement above uses an external trigger thus we would use Frame Trigger is Present to speed up the measurement significantly. If not using an external trigger or are using the UXR recording, set to Normal.



TIP **To improve setup speed**

Live measurement:

- Use external frame trigger and enable Frame trigger is present. The real frame boundary must be within ±50 µs of the external trigger.
- If external frame trigger is not available, use Reduced acquisition length. The reduced acquisition length may fail to synchronize in scenarios where > 1 frame SSB periodicity are defined, unless hardware triggering is used.
- Change Analysis Start Boundary to Subframe.
- Reduce Result Length and Measurement Interval.

Playback mode

- Reduce the Result Length and Measurement Interval.
- Set Analysis Start Boundary to Subframe.

– Reduce the number of active traces.

29. To configure the PBCH and PDSCH decoding, select the **Decode** tab and set.

– PBCH Decode to **Decoded TB**

– PDSCH Decode to **Decoded TB**

– PDSCH Decoder Algorithm to **Offset Min Sum** (default)

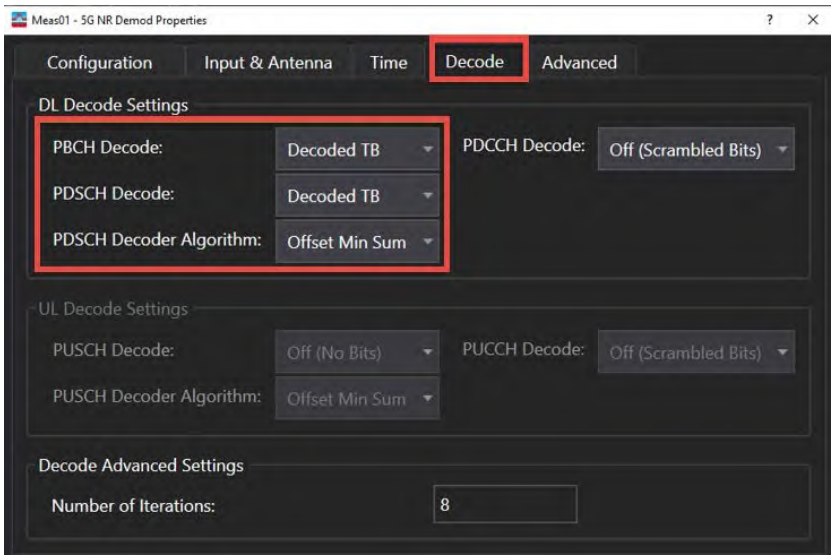
NOTE

VSA2023U2 or later releases support user selectable Decoder Algorithm.

Offset Min-Sum (Default) - Specifies the layered belief propagation algorithm with offset min-sum approximation. This is faster but less accurate than Belief Propagation.

Belief Propagation - Specifies the belief-passing or message-passing algorithm. This is better for accuracy but slow. Use for receiver sensitivity test where the SNR is very small.

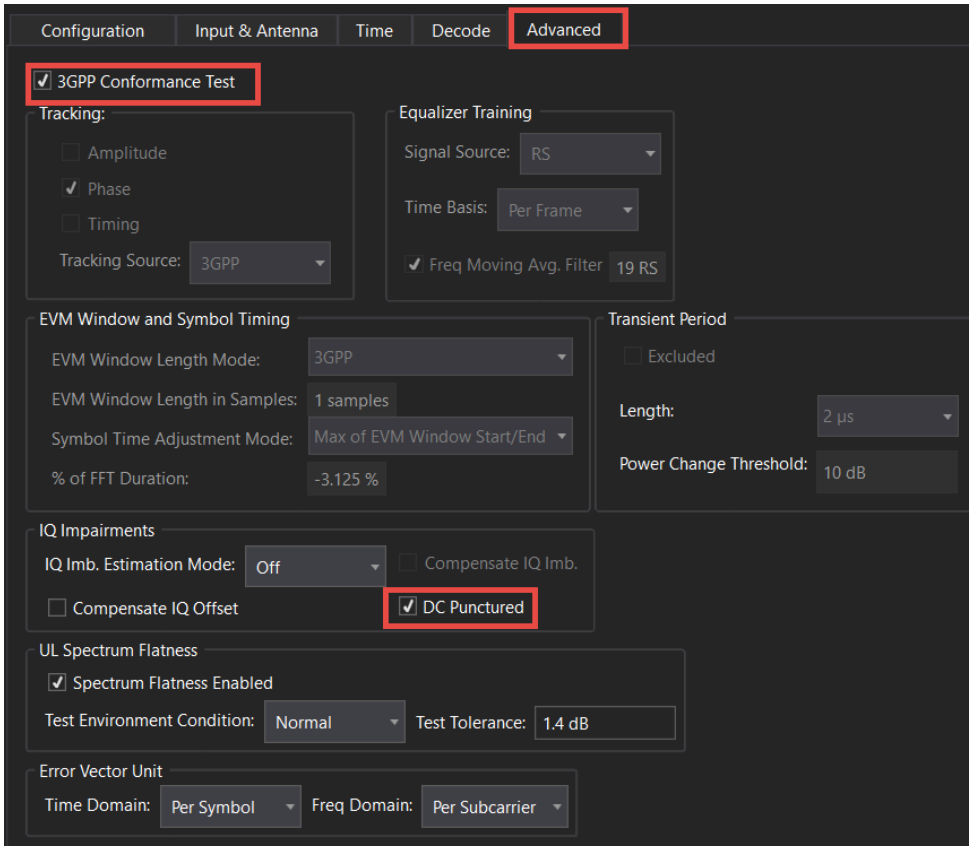
For successful PDSCH decoding (i.e. CRC pass), make sure the RNTI values (used for scrambling) and Transport Block Size matches between the transmitted signal and VSA. (Transport block size depends on MCS table, MCS value, TB Scaling Factor S, and xOverhead under PDSCH channel setting).



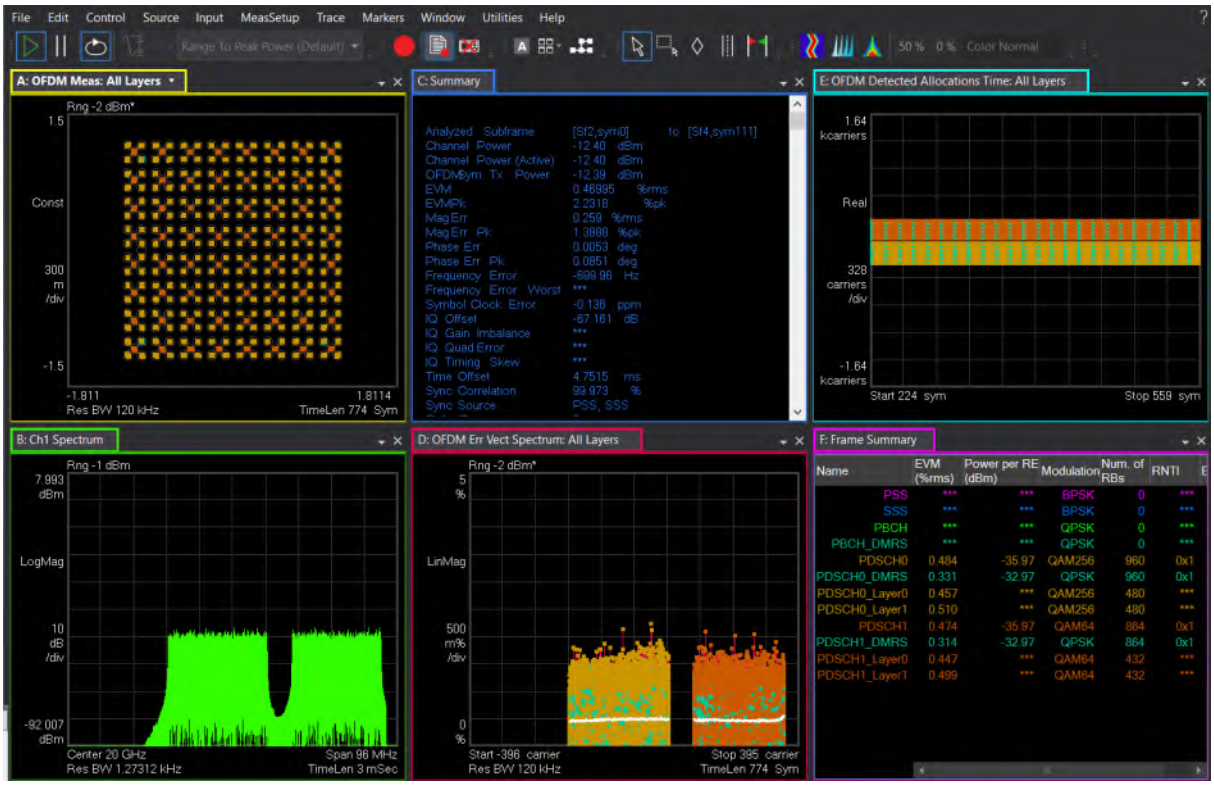
30. Select the **Advanced** tab and select the **DC Punctured** check box.

As part of conformance test, 3GPP has defined different equalizer training and tracking for EVM measurements. 3GPP Conformance Test is enabled by default where Tracking, Equalizer Training and EVM Window, and Symbol Timing is applied per 3GPP conformance test requirement. For FR1, no tracking is applied. For FR2, Phase Tracking using PTRS is applied.

5G NR counts the DC subcarrier as a valid subcarrier for rate-matching purposes. The network decides if the DC subcarrier is modulated or not. High LO feedthrough will impact demodulation and EVM performance of the input signal. This is OFF by default.



31. Close the 5G NR Properties dialog to view the results.



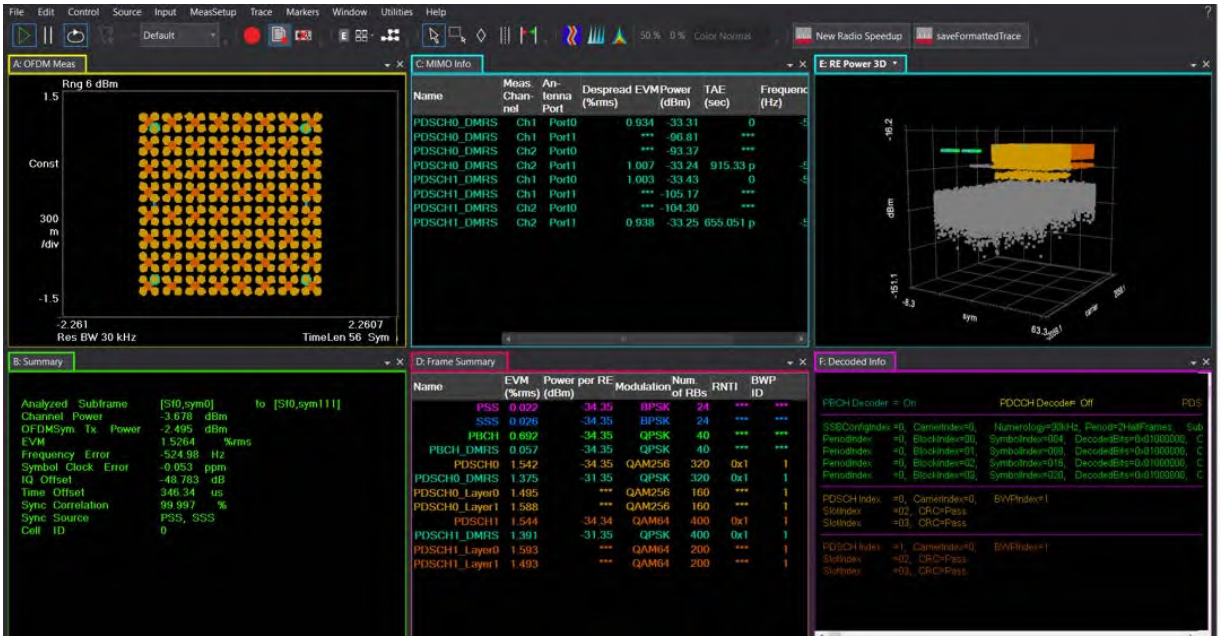
- Trace A: Composite constellation diagram showing different modulation formats for the different channels/signals present in the transmitted signal. See Trace F (Frame Summary) for the list of channels and modulation formats.
- Trace C: Summary trace showing composite error metrics.
- Trace D: Error Vector Spectrum showing EVM versus subcarrier and symbol.
- Trace E: Detected Allocations Time showing the detected allocations of all channels/signals within the measurement interval.
- Trace F: Frame Summary. EVM, per Layer EVM, Power per RE, Mod Format, Number of RB, RNTI, and BWP ID of the individual channels/signals.

One of the most powerful tools in the 89600 VSA is the coupling of markers across different measurements, traces, and domains. Coupled markers allow you to understand the identity and characteristics of a symbol simultaneously in time, frequency, and error.

Place a marker on Traces A, D, and E and couple the markers (Markers > Couple Markers), and then show the results in a markers window (Window > Markers).

Right click on Trace D and select **Peak**. The exact symbol associated with this peak EVM can now be understood in terms of time domain symbol index, frequency domain subcarrier number, channel type, modulation format, IQ magnitude, and phase values. Other parameters are shown in the Marker window.

Change the traces so you can see the Constellation diagram, MIMO Info table, Frame Summary Table (or Slot Summary Table), Summary table, 3D Power, Decoded Info plus any additional traces.



– Trace C: MIMO Info table shows EVM, power, and time, frequency and phase offset for each antenna port.

  MIMO Info is a type of Matrix Table that provides sortable rows by column, selectable column visibility, and copy/paste and export functionality to share rows of content or complete tables to applications like email, text editing or spreadsheet programs.

– Trace E: 3D trace showing resource element (RE) power per symbol and per subcarrier. This is very useful when verifying base station systems, for example, to make sure the power per each RE is balanced. You can use a marker to read the power and channel information. A marker on this trace cannot be coupled with other traces.

  – Press the mouse wheel and rotate the wheel forward to zoom in, backward to zoom out.

  – Hold left-click of the mouse for panning. Note: If marker is enabled (i.e. not a normal pointer), you will need to hold Alt in order to pan.

- Use right click to auto scale, enable the marker, and change the display to different axes.

- Trace F: The Decoded Info table provides CRC pass/fail for PDSCH and PBCH plus the high layer information that is carried on PBCH.

  If CRC of PDSCH fails, make sure the Transport Block Size matches between the transmitted signal and VSA (this depends on MCS table, MCS value, TB Scaling Factor S, and xOverhead under PDSCH channel setting). Also, make sure the RNTI values match since RNTI is used for scrambling.

Working with Traces

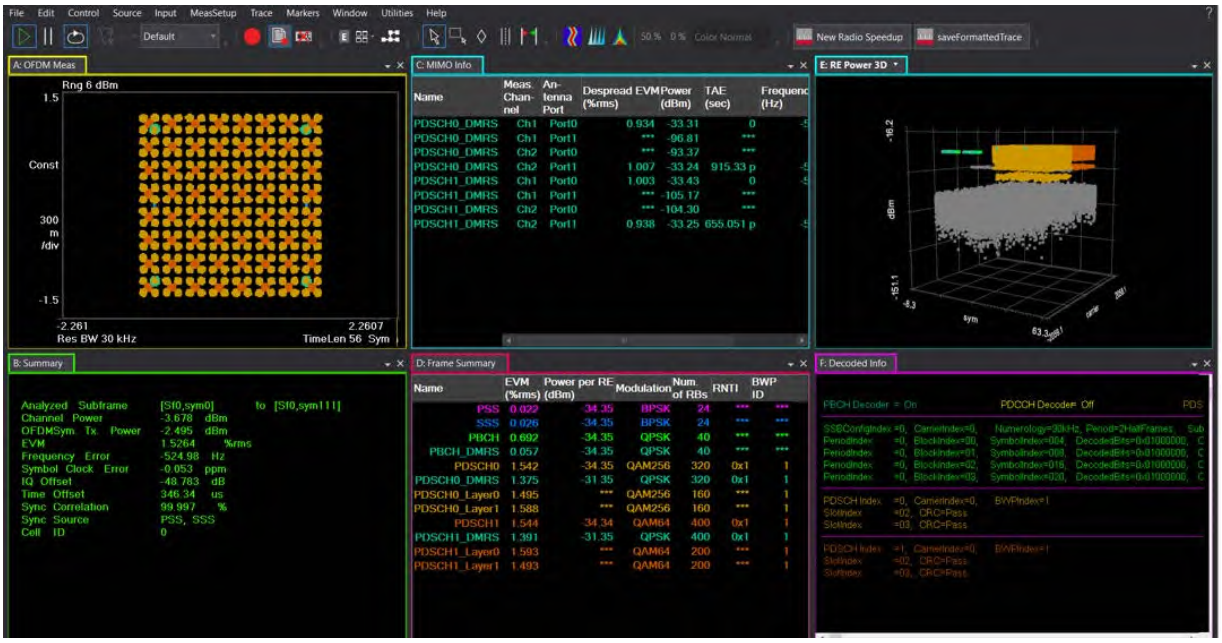
- Trace A: Composite constellation diagram showing different modulation formats for the different channels/signals present in the transmitted signal. See Trace F (Frame Summary) for the list of channels and modulation formats.
- Trace C: Summary trace showing composite error metrics.
- Trace D: Error Vector Spectrum showing EVM versus subcarrier and symbol.
- Trace E: Detected Allocations Time showing the detected allocations of all channels/signals within the measurement interval.
- Trace F: Frame Summary. EVM, per Layer EVM, Power per RE, Mod Format, Number of RB, RNTI, and BWP ID of the individual channels/signals.

One of the most powerful tools in the 89600 VSA is the coupling of markers across different measurements, traces, and domains. Coupled markers allow you to understand the identity and characteristics of a symbol simultaneously in time, frequency, and error.

Place a marker on Traces A, D, and E and couple the markers (Markers > Couple Markers), and then show the results in a markers window (Window > Markers).

Right click on Trace D and select **Peak**. The exact symbol associated with this peak EVM can now be understood in terms of time domain symbol index, frequency domain subcarrier number, channel type, modulation format, IQ magnitude, and phase values. Other parameters are shown in the Marker window.

Change the traces so you can see the Constellation diagram, MIMO Info table, Frame Summary Table (or Slot Summary Table), Summary table, 3D Power, Decoded Info plus any additional traces.



– Trace C: MIMO Info table shows EVM, power, and time, frequency and phase offset for each antenna port.

  MIMO Info is a type of Matrix Table that provides sortable rows by column, selectable column visibility, and copy/paste and export functionality to share rows of content or complete tables to applications like email, text editing or spreadsheet programs.

– Trace E: 3D trace showing resource element (RE) power per symbol and per subcarrier. This is very useful when verifying base station systems, for example, to make sure the power per each RE is balanced. You can use a marker to read the power and channel information. A marker on this trace cannot be coupled with other traces.

  – Press the mouse wheel and rotate the wheel forward to zoom in, backward to zoom out.

  – Hold left-click of the mouse for panning. Note: If marker is enabled (i.e. not a normal pointer), you will need to hold Alt in order to pan.

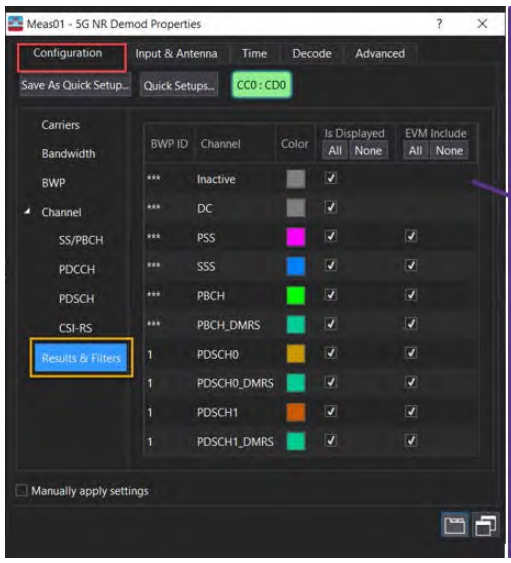
  – Use right click to auto scale, enable the marker, and change the display to different axes.

– Trace F: The Decoded Info table provides CRC pass/fail for PDSCH and PBCH plus the high layer information that is carried on PBCH.

If CRC of PDSCH fails, make sure the Transport Block Size matches between the transmitted signal and VSA (this depends on MCS table, MCS value, TB Scaling Factor S, and xOverhead under PDSCH channel setting). Also, make sure the RNTI values match since RNTI is used for scrambling.

Result and Display Filtering

Meas Setup > NR Demod Properties > Configuration tab > Results & Filters panel.



Results & Filters

Results & Filters table has three configurable columns

Color – sets the color for each channel (affects all traces that display channel-specific information)

- By default colors are kept as they were in previous release.

Is Displayed - Enables or disables the display of the channel in all traces (supplements the per-Trace filtering under Trace -> Digital Demod)

- Uncheck a box to filter the channel out of the trace results. This setting does not affect any EVM calculations.

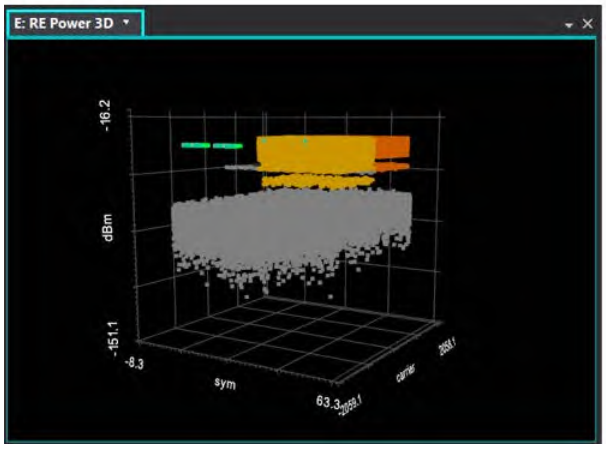
EVM Include - Includes or excludes the channel in the EVM calculations. This also affects trace filtering.

Note 1: Frame Summary table doesn't get affected by Results & Filters settings except for the color setting.

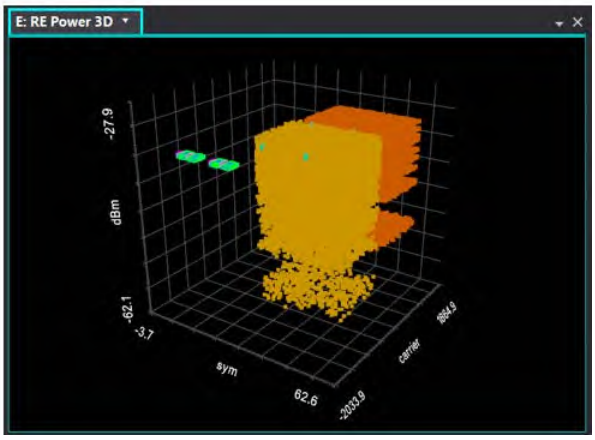
You can try to change colors of some of the channels, and/or filtering out all channels/signals except for SSBlock (PSS, SSS & PBCH) etc.

Clear (Uncheck) "Inactive" and "DC" to remove the inactive resources (subcarriers, symbols) and the DC subcarrier from the display and see the change in the 3D plot:

Before:



After:



You can now auto-scale the 3D plot and only see the active channels/signals being displayed.

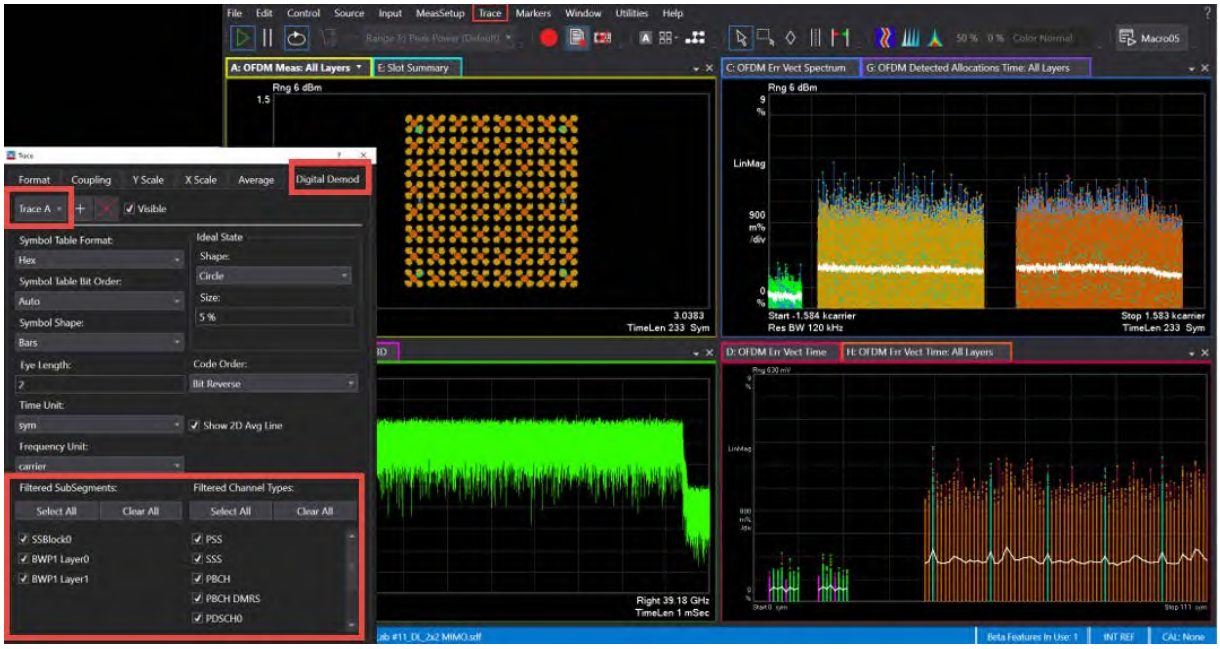
Per-Trace Filtering

Trace > Digital Demod

In addition to the Results & Filters, VSA also has per-trace filtering to filter by subsegments and/or channels. Subsegment means MIMO Layers, BWP and SSBlock.

The following traces can be filtered by subsegments:

– OFDM Meas
– OFDM Ref
– Error Vector Spectrum
– Error Vector Time
– Detected Allocations Time trace



Using the equivalent SCPI commands

On the VXG:

```
SYSTem:PRESet
RF1:FREQuency:CW 28GHz
RF1:POWer:AMPLitude 0dBm
RF2:FREQuency:CW 28GHz
RF2:POWer:AMPLitude 0dBm
CONFigure TX2
SIGNal:MODE NR
SIGNal:NR5G:CCARrier0:TYPE DL
SIGNal:NR5G:CCARrier0:CIDentity 0MIMO
SIGNal:NR5G:CCARrier0:NUM:MODE SINGLE
SIGNal:NR5G:CCARrier0:BWID FR1BW100M
SIGNal:NR5G:CCARrier0:SNUM MU1
SIGNal:NR5G:CCARrier0:SNUM:RB:NUMB 273
SIGNal:NR5G:CCARrier0:DLINk:SSBL:PERiodicity P10MS
SIGNal:NR5G:CCARrier0:DLINk:SSBL:RB:OFFSet 0
SIGNal:NR5G:CCARrier0:DLINk:SCH0:DMRS:PORT '0,1'
SIGNal:NR5G:CCARrier0:DLINk:SCH0:SLOT "2:19"
SIGNal:NR5G:CCARrier0:DLINk:SCH0:RB:OFFSet 2
SIGNal:NR5G:CCARrier0:DLINk:SCH0:RB:NUMBer 80
SIGNal:NR5G:CCARrier0:DLINk:SCH0:MCS:TABL TABL52
SIGNal:NR5G:CCARrier0:DLINk:SCH0:MCS 20
SIGNal:NR5G:CCARrier0:DLINk:SCH0:DMRS:POWer 3
SIGNal:NR5G:CCARrier0:DLINk:SCH0:ADD
SIGNal:NR5G:CCARrier0:DLINk:SCH1:DMRS:PORT '0,1'
SIGNal:NR5G:CCARrier0:DLINk:SCH1:DMRS:CGWD:COUN 2
SIGNal:NR5G:CCARrier0:DLINk:SCH1:SLOT "2:19"
SIGNal:NR5G:CCARrier0:DLINk:SCH1:RB:OFFSet 150
SIGNal:NR5G:CCARrier0:DLINk:SCH1:RB:NUMBer 100
SIGNal:NR5G:CCARrier0:DLINk:SCH1:MCS:TABL TABL51
SIGNal:NR5G:CCARrier0:DLINk:SCH1:MCS 20
```

```
SIGNal:NR5G:WAVeform:GENerate
SIGNal ON
SIGNal:NR5G:TRIGger:SYNC:MARKer M2
RF1:OUTPut ON
RF2:OUTPut ON
RFAL1:OUTPut ON
```

On the Analyzer:

```
INSTrument:SELect VSA89601
SYSTem:PRESet
FREQuency:CENTer 28 GHz
FREQuency:SPAN 122.88 MHz
INPut:ANALog:RANGe:AUTO
INITiate:PAUSe
INP:CHAN:CONF RF,RF
MEASure:CONFigure NR5G
NR5G:DBWP:ENABled 1
NR5G:SSBLock:ENABled 1
NR5G:SSBLock:ROFFset 0
NR5G:SSBLock:PERiodicity "Period10Milliseconds"
NR5G:DBWP:PDSCh1:MCS:TABLe "Table2"
NR5G:DBWP:PDSCh1:MCS 20
NR5G:DBWP:PDSCh1:SLOT:ALLocated "2:19"
NR5G:CCARrier:PDSCh1:ROFFset 40
NR5G:CCARrier:PDSCh1:RNUMber 80
NR5G:CCARrier:PDSCh1:ANTenna:PORT:USED 3
NR5G:CCARrier:PDSCh1:RCGNumber "Two"
NR5G:DBWP:PDSCh1:BPOWer:DMRS 3
NR5G:DBWP:PDSCh2:MCS:TABLe "Table1"
NR5G:DBWP:PDSCh2:MCS 20
NR5G:DBWP:PDSCh2:SLOT:ALLocated "2:19"
NR5G:CCARrier:PDSCh2:ROFFset 150
```

```
NR5G:CCARrier:PDSCh2:RNUMber 100
NR5G:CCARrier:PDSCh2:ANTenna:PORT:USED 3
NR5G:CCARrier:PDSCh2:RCGNumber "Two"
NR5G:DBWP:PDSCh2:BPOWer:DMRS 2
NR5G:RLENgth 10
NR5G:SUBFrame:INTerval 2
NR5G:FRAMe:TRIGger:ENABled 1
NR5G:DC:PUNCtured 1
NR5G:MCFilter:ENABled 1
NR5G:COMPensate:SYMBol:CLOCk:OFFset 1
INPut:TRIGger:STYLe "External"
INPut:TRIGger:LEVel:EXTernal 1V
INPut:EXTension:PARameters:SET "ExtTriggerLoc", 2
INPut:EXTension:PARameters:SET "PhaseNoiseOptDualLoop", 1
NR5G:FRAMe:TRIGger:ENABled 1
NR5G:DECode:MODE "DecodedTB"
DISPlay:LAYout 3,2
```

This information is subject to change without notice.
© Keysight Technologies 2021- 2026

Edition 1, June 2026

M9484-90002

www.keysight.com