Vertex® Channel Emulator

Release 4.71

User Manual

Limited Warranty

Spirent Communications, Inc. ("Spirent") warrants that its Products will conform to the description on the face of order, that it will convey good title thereto, and that the Product will be delivered free from any lawful security interest or other lien or encumbrance.

Spirent further warrants to Customer that hardware which it supplies and the tangible media on which it supplies software will be free from significant defects in materials and workmanship for a period of twelve (12) months, except as otherwise noted, from the date of delivery (the "Hardware Warranty Period"), under normal use and conditions. To the extent the Product is or contains software ("Software"), Spirent also warrants that, if properly used by Customer in accordance with the Software License Agreement, the Software which it supplies will operate in material conformity with the specifications supplied by Spirent for such Software for a period of ninety (90) days from the date of delivery (the "Software Warranty Period"). The "Product Warranty Period" shall mean the Hardware Warranty Period or the Software Warranty Period, as applicable. Spirent does not warrant that the functions contained in the Software will meet a specific requirement or that the operation will be uninterrupted or error free. Spirent shall have no warranty obligations whatsoever with respect to any Software which has been modified in any manner by Customer or any third party. Defective Products and Software under warranty shall be, at Spirent's discretion, repaired or replaced or a credit issued to Customer's account for an amount equal to the price paid for such Product provided that: (a) such Product is returned to Spirent after first obtaining a return authorization number and shipping instructions, freight prepaid, to Spirent's location in the United States; (b) Customer provides a written explanation of the defect or Software failure claimed by Customer; and (c) the claimed defect actually exists and was not caused by neglect, accident, misuse, improper installation, improper repair, fire, flood, lightning, power surges, earthquake, or alteration. Spirent will ship repaired Products to Customer, freight prepaid, based on reasonable best efforts after the receipt of defective Products. Except as otherwise stated, any claim on account of defective materials or for any other cause whatsoever will conclusively be deemed waived by Customer unless written notice thereof is given to Spirent within the Warranty Period. Spirent reserves the right to change the warranty and service policy set forth above at any time, after reasonable notice and without liability to Customer.

Page Part Number: 71-009016, Version A0

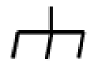
Safety Summary

If the equipment is used in a manner not specified by the manufacturer the protection provided by the equipment may be impaired.

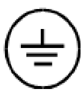
Safety Symbols

The following safety symbols are used throughout this manual and may be found on the instrument. Familiarize yourself with each symbol and its meaning before operating this instrument.

Instruction manual symbol. The product is marked with this symbol when it is necessary for you to refer to the instruction manual to protect against damage to the instrument.



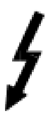
Frame terminal. A connection to the frame (chassis) of the equipment which normally includes all exposed metal structures.



Protective ground (earth) terminal. Used to identify any terminal which is intended for connection to an external protective conductor for protection against electrical shock in case of a fault, or to the terminal of a protective ground (earth) electrode.

Caution

The caution sign denotes a hazard. It calls attention to an operating procedure, practice, condition or the like, which, if not correctly performed or adhered to, could result in damage to or destruction of part or all of the product or your data.



Indicates dangerous voltage (terminals fed from the interior by voltage exceeding 1000 volts must be so marked).

Alternating current (power line).

Symbol when movement with two people is required. When this symbol is noted on our product, two people are required to move it without accident.

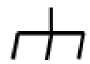
Résumé des règles de sécurité

Si le matériel est utilisé d’une façon non conforme aux spécifications du constructeur, la protection assurée par le matériel peut être mise en défaut.

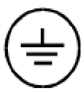
Symboles de sécurité

Les symboles suivants sont utilisés dans tout le manuel et peuvent être trouvés sur le matériel. Il est recommandé de se familiariser avec chaque symbole et sa signification avant de manipuler le matériel.

Symbole « manuel d’instruction ». Ce symbole apparaît sur le produit lorsqu’il est nécessaire de se référer au manuel d’instruction pour éviter une détérioration du matériel.



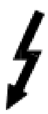
Masse. Ce symbole identifie une connexion au châssis du matériel (ce châssis inclut normalement toutes les structures métalliques exposées).



Terre : ce symbole identifie la connexion de terre chargée de protéger le matériel contre les chocs électriques. Cette connexion doit être raccordée vers un conducteur externe de protection ou vers une électrode de type terre.

Caution

Ce symbole désigne une opération ou une condition dite « sensible », qui, si elle n’est pas correctement réalisée, pourrait entraîner de sérieuses détériorations au matériel ou aux données utilisateur.



Ce symbole indique un voltage dangereux (connexion alimentée en interne par un voltage excédant 1000 volts).

Courant alternatif (ligne de puissance).

Symbole de déplacement avec deux personnes requises. Lorsque ce symbole est noté sur notre produit, deux personnes sont requises afin de le déplacer sans accident

Table of Contents

1. Introduction

1.1. Overview

The Spirent Vertex® channel emulator simplifies high-density MIMO testing for technologies such as LTE, LTE-Advanced, massive MIMO, WiFi, and mesh networks. These technologies rely on large numbers of antennas, higher bandwidth, and band aggregation to deliver high-speed data. The Vertex channel emulator provides integrated, bidirectional RF channels and supports carrier aggregation. With high-fidelity channel and long simulation repetition rates, the Vertex channel emulator ensures reliable and accurate performance evaluation.

The Vertex channel emulator has a modular structure of RF and digital subsystems and can be configured with different channel density or frequency coverage for different applications. The Vertex channel emulator also features an easy-to-use graphical user interface (GUI) that provides one-click access to the most commonly used functions. The step-by-step design of the GUI along with graphical feedback at each step ensures that even the most novice user can quickly set up and run the complex RF environments needed to test mobiles and base stations used in LTE, LTE-Advanced, and beyond.

The Vertex channel emulator includes support for MIMO over-the-air (OTA) testing using the MIMO OTA Environment Builder software. The Vertex channel emulator also possesses the capabilities necessary to evaluate a broad range of local and wide-area wireless network technologies.

Supported technologies include:

- 5G
- WiFi6
- Satellite and aeronautical applications
- Location-based services
- GSM/GPRS/EDGE
- WCDMA
- WCDMA HSPA (HSDPA/HSUPA)
- HSPA+
- LTE
- LTE-Advanced

- CDMA2000 1x
- CDMA2000 1xEV-DO
- CDMA2000 1xEV-DV
- 802.11.a/b/g/n/ac
- 802.16 (WiMAX)
- HiperLAN

1.2. Vertex Channel Emulator Description

With its modular structure, the Vertex channel emulator can be configured to support different channel densities. The following figure shows a fully loaded Vertex channel emulator.

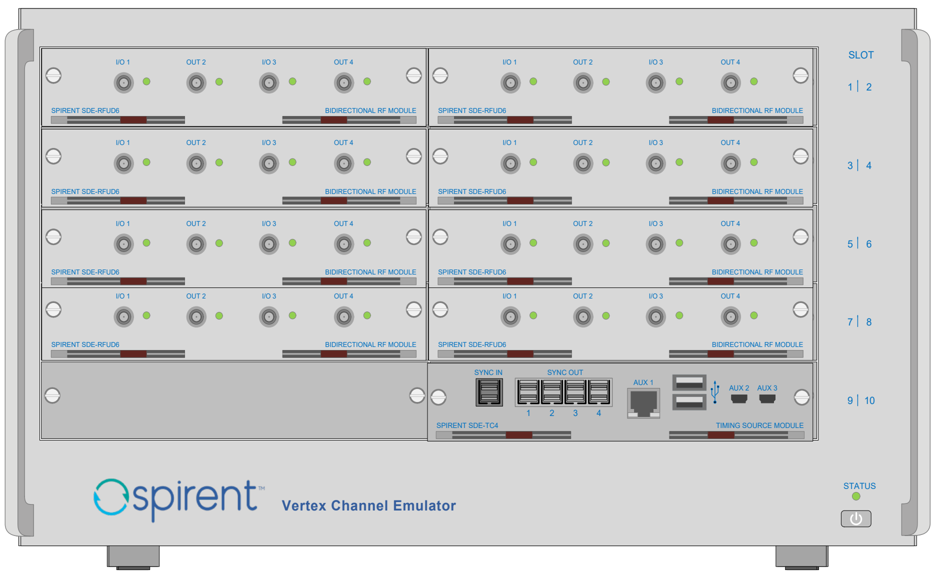


Figure 1. Vertex Channel Emulator front panel.

1.2.1. Front Panel Description

The front panel of the Vertex channel emulator consists of the following components:

- RF module

  The RF modules are installed in slot 1 to slot 9 in the front panel of the Vertex instrument. The slot number is shown at the right side of the front panel. The RF module can be unidirectional or bidirectional. The ports in the bidirectional module are a combination of simplex (OUT) and duplex (I/O) ports.

  CAUTION!
  The RF IN and OUT Ports can accept a limited power range. Refer to the technical specifications to ensure absolute maximum levels are not exceeded.

  - SDE-RFUD4: Bidirectional RF Module (4GHz/40MHz BW) with N-type RF connection

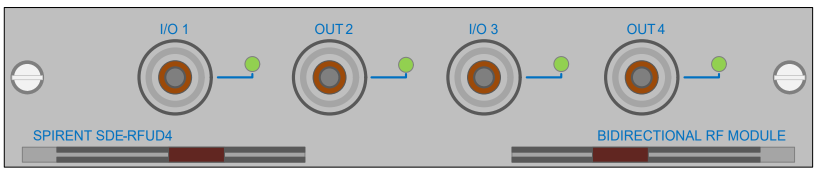


Figure 2. SDE-RFUD4: Bidirectional RF module (4GHz/40MHz BW).

  - SDE-RFUP4: Unidirectional RF module (4GHz/40MHz BW) with N-type RF connection

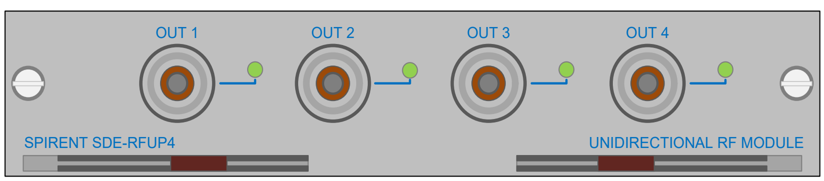


Figure 3. SDE-RFUP4: Unidirectional RF module (4GHz/40MHz BW).

  - SDE-RFUD6: Bidirectional RF module (6GHz/100MHz BW) with SMA-type RF connection

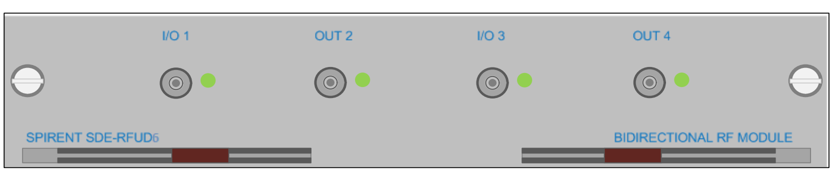


Figure 4. SDE-RFUD6: Bidirectional RF module (6GHz/100MHz BW).

- Timing Card (SDE-TC4)

  The timing card installed in slot 10 provides 10MHz internal reference, sample and sync clocks to RF modules and DSPMs. This card also provides the supervisory master function.

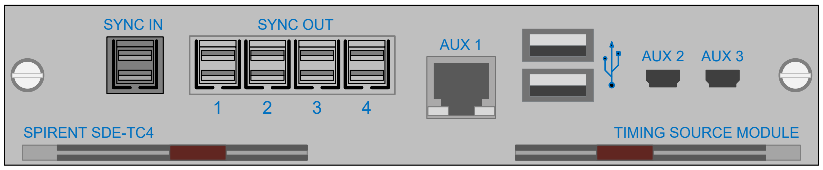


Figure 5. SDE-TC4 timing card.

- AUX 1 through AUX 3

  These ports are reserved.

- USB ports

  The USB ports enable you to insert a thumb drive or external hard drive.

- SYNC IN & SYNC OUT

  SYNC IN and SYNC OUT are mini SAS HD interfaces for synchronizing fading operation between multiple Vertex instruments.

- Power button

  There is one push button to initiate power up and power down on the front panel of the Vertex channel emulator.

1.2.2. Rear Panel Description

The following figure shows the rear panel of the Vertex channel emulator.

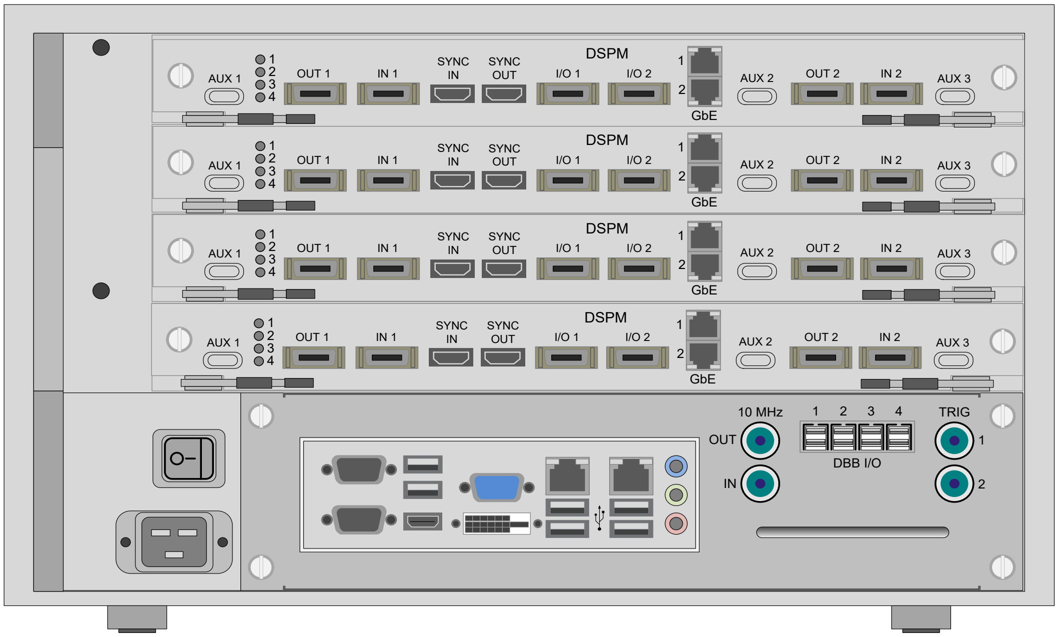


Figure 6. Vertex Channel Emulator rear panel.

The rear panel of the Vertex channel emulator consists of the following components:

- DSPM modules

  DSPM modules are installed in the slots on the rear panel of the Vertex instrument. The slots are numbered 1 to 4 from top to bottom.

  Vertex supports the following DSPM modules:

  - DSPM
  - SDE-DSPM2

  See the section “DSPM Modules” on page 17 for more information about the supported DSPM modules.

- 10 MHz IN

  BNC Type Connector (50 Ω) that accepts externally supplied 10 MHz reference signal.

- 10 MHz OUT

  BNC Type Connector (50 Ω) that provides a 10 MHz reference signal as output.

- TRIG 1 & TRIG 2

  TRIG1 is for Trigger in. TRIG2 is for Trigger out.

- DBB IO 1 through 4

  These ports are used for multi-instrument integration.

- Embedded PC

  The embedded PC has the following ports:

  - **6x USB ports** - To connect either a USB flash drive or USB external hard drive. The blue interface is for USB 3.0 devices, and the black interface is for USB 2.0 devices.
  - **2x LAN ports** - For connection to Ethernet and the controller PC.
  - **2x Serial ports** - For IP address configuration.
  - **HDMI** - Video stream for external display.
  - **VGA** - For external display.
  - **DVI-D** - For external display.
  - **Audio Connectors** – Not supported.

- Power Input Receptacle

  The power input receptacle is type IEC-60320 C20 (mates to IEC-60320 C19 plug).

1.2.3. DSPM Modules

DSPM modules are installed in the slots on the rear panel of the Vertex instrument. The slots are numbered 1 to 4 from top to bottom.

Vertex supports the following DSPM modules:

- DSPM
- SDE-DSPM2

1.2.3.1 DSPM Module

The following figure shows a DSPM module.

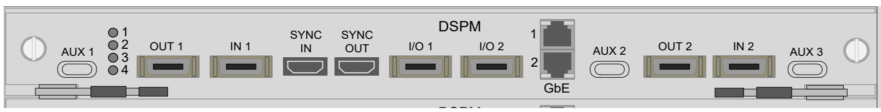


Figure 7. DSPM module.

A DSPM module consists of the following components:

- AUX 1, AUX 2, and AUX 3

  These ports are reserved.

- GbE1 and GbE2

  These ports are reserved.

- SYNC IN and SYNC OUT

  HDMI-type cable for synchronizing fading operation between multiple Vertex instruments.

- OUT 1 and OUT 2

  These ports are reserved.

- IN 1 and IN 2

  These ports are reserved.

- I/O 1 and I/O 2

  These ports are reserved.

- LEDs 1-4

  Shows the power status of the chipsets on the DSPM.

1.2.3.2 SDE-DSPM2 Module

The following figure shows an SDE-DSPM2 module.

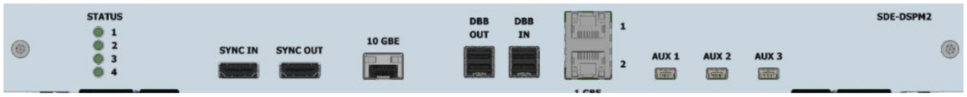


Figure 8. SDE-DSPM2 module.

An SDE-DSPM2 module consists of the following components:

- AUX 1, AUX 2, and AUX 3

  These ports are reserved.

- GbE1 and GbE2

  These ports are reserved.

- SYNC IN and SYNC OUT

  HDMI-type cable for synchronizing fading operation between multiple Vertex instruments.

- 10 GBE

  This port is reserved.

- Status LEDs 1-4

  Shows the power status of the chipsets on the DSPM2 module.

1.2.4. Hardware Configuration

The Timing Card (SDE-TC4) is installed in slot 10 of the front panel for all hardware configurations.

Model	Description	RFM on Front Panel	DSPM on Rear Panel
VCE6-1B4-1D1	2-channel bidirectional, 16 digital links	SDE-RFUD4: Slot 8	DSPM: Slot 1
VCE6-2B4-1D1	4-channel bidirectional, 16 digital links	SDE-RFUD4: Slot 7, 8	DSPM: Slot 1
VCE6-4B4-1D1	8-channel bidirectional, 16 digital links	SDE-RFUD4: Slot 5, 6, 7, 8	DSPM: Slot 1
VCE6-4B4-2D1	8-channel bidirectional, 32 digital links	SDE-RFUD4: Slot 5, 6, 7, 8	DSPM: Slot 1, 2
VCE6-4B4-4D1	8-channel bidirectional, 64 digital links	SDE-RFUD4: Slot 5, 6, 7, 8	DSPM: Slot 1, 2, 3, 4
VCE6-6B4-2D1	12-channel bidirectional, 32 digital links	SDE-RFUD4: Slot 3, 4, 5, 6, 7, 8	DSPM: Slot 1, 2
VCE6-6B4-3D1	12-channel bidirectional, 48 digital links	SDE-RFUD4: Slot 3, 4, 5, 6, 7, 8	DSPM: Slot 1, 2, 3
VCE6-6B4-4D1	12-channel bidirectional, 64 digital links	SDE-RFUD4: Slot 3, 4, 5, 6, 7, 8	DSPM: Slot 1, 2, 3, 4
VCE6-8B4-2D1	16-channel bidirectional, 32 digital links	SDE-RFUD4: Slot 1, 2, 3, 4, 5, 6, 7, 8	DSPM: Slot 1, 2
VCE6-8B4-4D1	16-channel bidirectional, 64 digital links	SDE-RFUD4: Slot 1, 2, 3, 4, 5, 6, 7, 8	DSPM: Slot 1, 2, 3, 4
VCE6-1B4-2U4-1D1	2x8 unidirectional, 16 digital links	SDE-RFUD4: Slot 9 SDE-RPUP4: Slot 7, 8	DSPM: Slot 1
VCE6-2B4-4U4-2D1	Dual 2x8 unidirectional, 32 digital links	SDE-RFUD4: Slot 1, 9 SDE-RPUP4: Slot 5, 6, 7, 8	DSPM: Slot 1, 2
VCE6-1B4-4U4-2D1	2x16 unidirectional, 32 digital links	SDE-RFUD4: Slot 9 SDE-RPUP4: Slot 5, 6, 7, 8	DSPM: Slot 1, 2

Model	Description	RFM on Front Panel	DSPM on Rear Panel
VCE6-2B4-7U4-4D1	2x32 or Dual 2x16 unidirectional, 64 digital links	SDE-RFUD4: Slot 1,9 SDE-RPUP4: Slot 2, 3, 4, 5, 6, 7, 8	DSPM: Slot 1, 2, 3, 4
VCE6-1B4-8U4-4D1	2x32 unidirectional, 64 digital links	SDE-RFUD4: Slot 9 SDE-RPUP4: Slot 1, 2, 3, 4, 5, 6, 7, 8	DSPM: Slot 1, 2, 3, 4
VCE6-1B6-1D1	2-channel bidirectional, 16 digital links	SDE-RFUD6: Slot 8	DSPM: Slot 1
VCE6-2B6-1D1	4-channel bidirectional, 16 digital links	SDE-RFUD6: Slot 7,8	DSPM: Slot 1
VCE6-4B6-1D1	8-channel bidirectional, 16 digital links	SDE-RFUD6: Slot 5, 6, 7, 8	DSPM: Slot 1
VCE6-4B6-2D1	8-channel bidirectional, 32 digital links	SDE-RFUD6: Slot 5, 6, 7, 8	DSPM: Slot 1, 2
VCE6-4B6-4D1	8-channel bidirectional, 64 digital links	SDE-RFUD6: Slot 5, 6, 7, 8	DSPM: Slot 1, 2, 3, 4
VCE6-6B6-2D1	12-channel bidirectional, 32 digital links	SDE-RFUD6: Slot 3, 4, 5, 6, 7, 8	DSPM: Slot 1, 2
VCE6-6B6-3D1	12-channel bidirectional, 48 digital links	SDE-RFUD6: Slot 3, 4, 5, 6, 7, 8	DSPM: Slot 1, 2, 3
VCE6-6B6-4D1	12-channel bidirectional, 64 digital links	SDE-RFUD6: Slot 3, 4, 5, 6, 7, 8	DSPM: Slot 1, 2, 3, 4
VCE6-8B6-2D1	16-channel bidirectional, 32 digital links	SDE-RFUD6: Slot 1, 2, 3, 4, 5, 6, 7, 8	DSPM: Slot 1, 2
VCE6-8B6-4D1	16-channel bidirectional, 64 digital links	SDE-RFUD6: Slot 1, 2, 3, 4, 5, 6, 7, 8	DSPM: Slot 1, 2, 3, 4
VCE6-1B6-1D2	2-channel bidirectional, 64 digital links (100MHz) 16 digital links (200MHz)	SDE-RFUD6:Slot 8	SDE-DSPM2: Slot 1

Model	Description	RFM on Front Panel	DSPM on Rear Panel
VCE6-2B6-1D2	4-channel bidirectional, 64 digital links (100MHz) 16 digital links (200MHz)	SDE-RFUD6: Slot 7, 8	SDE-DSPM2:Slot 1
VCE6-4B6-1D2	8-channel bidirectional, 64 digital links (100MHz) 16 digital links (200MHz)	SDE-RFUD6: Slot 5, 6, 7, 8	SDE-DSPM2:Slot 1
VCE6-4B6-2D2	8-channel bidirectional, 128 digital links (100MHz) 32 digital links (200MHz)	SDE-RFUD6: Slot 5, 6, 7, 8	SDE-DSPM2: Slot 1, 2
VCE6-8B6-2D2	16-channel bidirectional, 128 digital links (100MHz) 32 digital links (200MHz)	SDE-RFUD6: Slot 1, 2, 3, 4, 5, 6, 7, 8	SDE-DSPM2: Slot 1, 2
VCE6-8B6-4D2	16-channel bidirectional, 128 digital links (100MHz) 32 digital links (200MHz)	SDE-RFUD6: Slot 1, 2, 3, 4, 5, 6, 7, 8	SDE-DSPM2: Slot 1, 2, 3, 4
VCE6-9B6-2D2	18-channel bidirectional, 128 digital links (100MHz) 32 digital links (200MHz)	SDE-RFUD6: Slot 1, 2, 3, 4, 5, 6, 7, 8, 9	SDE-DSPM2: Slot 1, 2
VCE6-9B6-4D2	18-channel bidirectional, 256 digital links (100MHz) 64 digital links (200MHz)	SDE-RFUD6: Slot 1, 2, 3, 4, 5, 6, 7, 8, 9	SDE-DSPM2: Slot 1, 2, 3, 4

1.3. Lifting and Carrying the Vertex Channel Emulator

CAUTION!
Lifting and carrying the Vertex channel emulator safely requires two (2) people.

There is a warning label “Two-Person Lift” on top of the Vertex instrument. There are lifting handles on both the front and rear of the instrument. Two people are required for lifting and carrying a Vertex instrument. Attempting to lift the Vertex instrument with less than two people may result in personal injury or damage to the unit.

1.4. Intended Audience

This manual is intended for those who have a working knowledge of wireless communication technologies.

1.5. Documentation

To access the latest version of this document, perform the following steps:

1. Log into the Spirent Customer Service Center website (**http://support.spirent.com**) using the email address and password assigned to you by Spirent.
2. In the Search Knowledge Base box, enter ***DOC10797*** and click on **Search KB**.

   The results list appears.
3. Click on **Spirent Vertex® Channel Emulator Documentation**.

   The Spirent Vertex® Channel Emulator Documentation page appears.
4. Click on the link for the document in which you are interested.

   The page for the selected document appears.
5. Click on the link in the Attachment area to view the corresponding PDF.

1.6. How to Contact Us

To obtain technical support for any Spirent Communications product, please contact our Support Services department using any of the following methods:

Americas
E-mail: support@spirent.com
Web: http://support.spirent.com
Toll Free: +1 800-SPIRENT (+1 800-774-7368) (North America)
Hours: Monday through Friday, 05:30 to 18:00 Pacific Time

Europe, Africa, Middle East
E-mail: support@spirent.com
Web: http://support.spirent.com
EMEA Phone: +33 (1) 6137 2270
UK Phone: +44 1803 546333
Toll Free Phone: +1 818-676-2616
Hours: Monday through Thursday, 09:00 to 18:00, 9:00 to 17:00 Friday, Paris Time

Asia Pacific
E-mail: support@spirent.com
Web: http://support.spirent.com
In China Mainland Phone: +86 (800) 810-9529 (toll-free)
Out of China Mainland Phone: +86 (10) 8233 0033
India Phone: 1800-419-2111
Operating Hours: Monday through Friday, 09:00 to 18:00 Beijing Time

The Spirent Knowledge Base (**http://support.spirent.com**) is designed to serve your technical information needs. The Knowledge Base gives you access to tens of thousands of documents that help answer your network analysis and measurement questions. New content is added daily by Spirent's communications and networking experts. Sign in with your user ID and password to gain access to additional content that is available only to customers – user manuals, Help files, release notes, Tech Bulletins, and more. When you sign in, you can also use the Knowledge Base to download software and firmware, and to manage your SRs.

Information about Spirent Communications and its products and services can be found on the main company website at **http://www.spirent.com**.

Company Address
Spirent Communications, Inc.
26750 Agoura Road
Calabasas, CA 91302
USA

2. Operation Reference

2.1. Instrument Control

The Vertex channel emulator has two interfaces that allow you to control and monitor the instrument in real-time:

- Laptop-based graphical user interface (GUI)
- Remote programming interface (RPI)

2.1.1. Laptop-Based GUI

The laptop-based GUI gives you the maximum flexibility to configure a test setup.

2.1.2. Remote Programming Interface (RPI)

The RPI gives you the ability to remotely control the Vertex channel emulator through commands sent over a TCP/IP connection. For more information on the RPI, refer to the section “Remote Programming Interface (RPI)” on page 234.

2.2. Operating Conditions

The Vertex instrument should be placed in a location that can properly support its weight and where it can be connected to a power source that is rated for the required power of the instrument. Additionally, care must be taken to ensure that the ambient temperature is within the operating range of the Vertex channel emulator. Refer to the technical specifications for details regarding the weight, power, and ambient temperature requirements of the instrument.

2.2.1. Bench-Top Positioning

When positioning a Vertex instrument for operation on a bench top, care must be taken to ensure that the bench:

- Has enough space for easy access to the power connector on the rear panel for emergency disconnection,
- Has proper ventilation on both sides of the Vertex instrument to maintain airflow, and
- Can support the weight of the Vertex instrument safely.

Refer to the Technical Specifications section in this manual for details.

2.2.2. Rackmount Positioning

CAUTION!
When installing the Vertex channel emulator in a rack configuration, care must be taken to ensure that the rack is properly configured to support the weight of the Vertex instrument safely and to maintain an ambient temperature inside the rack that is within the specified operating temperature range of the Vertex instrument. Because the Vertex instrument ventilation system operates from side to side, Spirent recommends that you use vented (or no) side panels in any rack configuration that includes the Vertex instrument. When using vented side panels, Spirent recommends that they have a minimum of 65% venting. Such a configuration allows air to enter and exit the rack more easily. Unless properly designed to adequately circulate cool air into the rack, and in particular to the inflow side of the Vertex instrument, the use of a Vertex instrument in a rack configuration with solid side and rear panels may result in higher than specified temperatures within the rack.

Failure to maintain an ambient temperature within the specified operating range of the Vertex instrument will lead to excessive fan noise and possible thermal shutdown of the Vertex instrument.

2.3. Ventilation

For proper ventilation, ensure that the rack is in an area with the following conditions:

- A minimum of 18 inches of space between the sides of the rack and any objects that may restrict air flow (for example, a wall, cabinet, or another rack).
- A minimum of 12 inches of space between the rear door and any object that may restrict air flow (for example, a wall, cabinet, or another rack).
- There is nothing that blocks the airflow from the exhaust fans on the top of the rack.
- Do not allow other equipment to exhaust into the rack, as this may cause the equipment to overheat.
- If insufficient ventilation is provided, the equipment in the rack may overheat. This can cause improper operation.
- Refer to the Technical Specifications section in this manual for information about the required operating temperature.

2.4. Connecting the Controller Laptop to the Vertex Channel Emulator

Perform the following steps:

1. Using an Ethernet cable, connect the controller laptop to the **User Control** Ethernet port on the rear panel of the Vertex channel emulator, as shown in the following figure. If the IP address of the Vertex channel emulator is unknown, a USB to RS-232 cable (null modem cable) is required to connect from the laptop's USB port to the RS-232 port on the rear panel of the Vertex instrument.

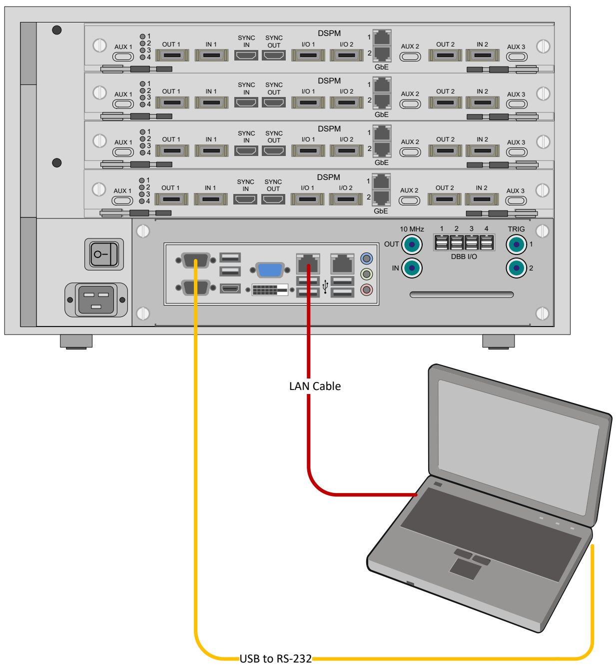


Figure 9. Vertex Channel Emulator rear panel.

2. Ensure that the Vertex instrument is running properly.

3. Launch the Vertex channel emulator software on the remote laptop using .
   The Spirent Vertex Channel Emulator window appears.

Figure 10. Spirent Vertex Channel Emulator window.

4. Perform one of the following steps:
   - If you know the IP address of the Vertex channel emulator, enter the IP address in the Primary Fader IP Address box, and click the **Connect** button.

     The Vertex Channel Emulator GUI appears.
   - If you do not know the IP address of the Vertex channel emulator, perform the following steps:

     a. Click the **Get IP** button.

        The IP address will appear in the Primary Fader IP Address box, as shown in the following figure.

Figure 11. Primary fader IP address.

If the connection is not successful, go to the section “Checking the COM Port Setting” on page 32.

     b. Click the **Connect** button.

        The Vertex window appears.

- If you want to configure a new IP address for the Vertex channel emulator, perform the following steps:

  a. Click the **Config Interface** button.

     The IP Configuration dialog box appears.

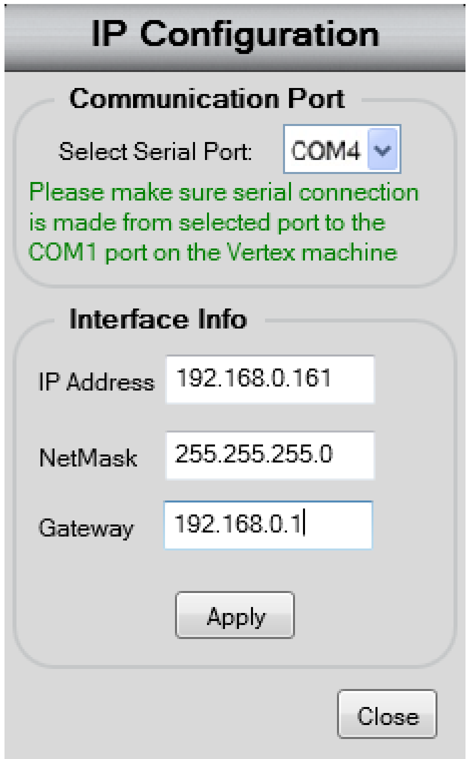


Figure 12. IP Configuration dialog box.

  b. From the Select Serial Port box, select the COM port that is connected to the serial cable.

  c. In the IP Address box, enter the IP address of the Vertex channel emulator.

  d. In the NetMask box, enter the net mask.

  e. In the Gateway box, enter the IP address of the gateway.

  f. Click the **Apply** button.

     The new IP address is set to the Vertex instrument. You can click the **Get IP** button to show the IP address.

g. Click the **Connect** button.

The Vertex window appears.

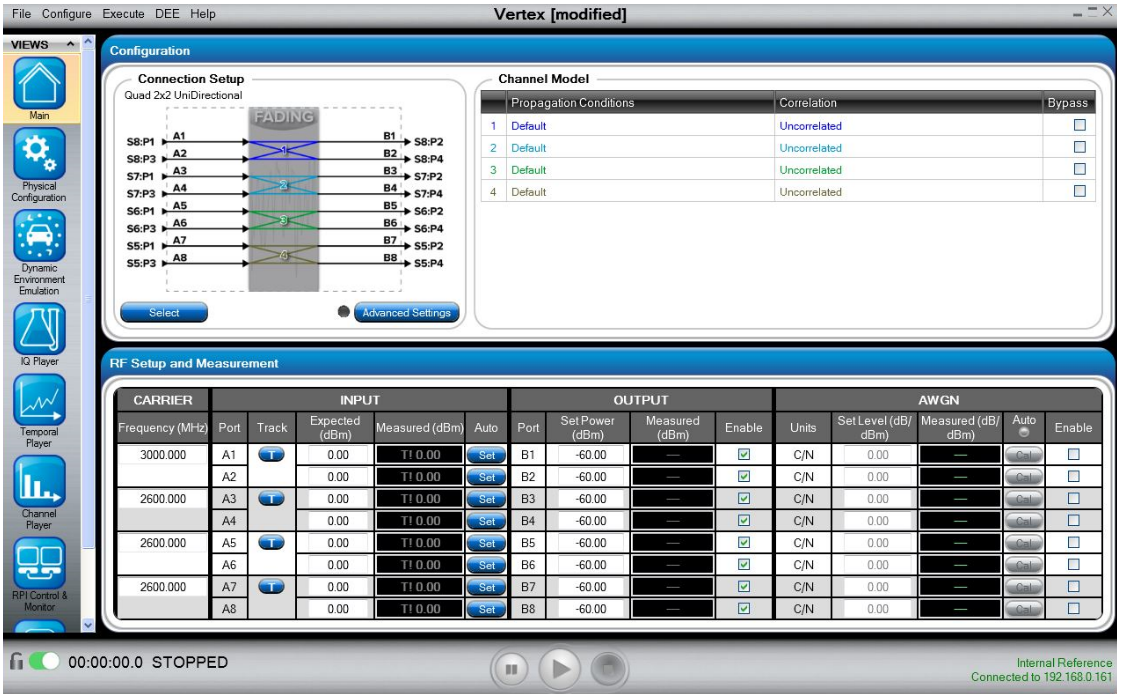


Figure 13. Vertex window.

2.4.1. Checking the COM Port Setting

To check the COM port setting, perform the following steps:

1. Access **Control Panel**.

   The All Control Panel Items window appears.

2. Click on **Device Manager**.

   The System window appears.

3. Click on **Device Manager** as shown in the following figure.

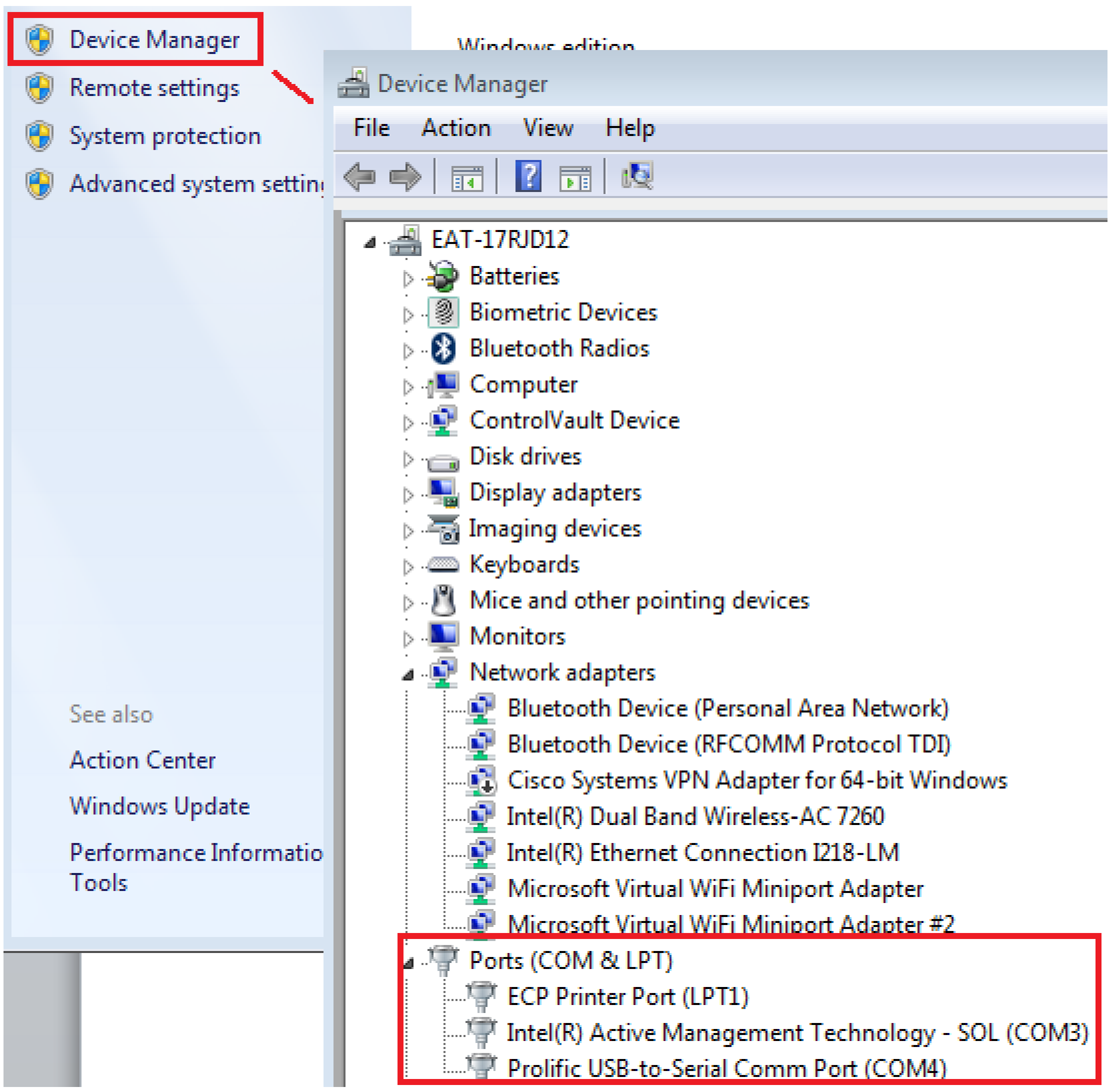


Figure 14. Device Manager window.

The Device Manager window appears. In the Device Manager window, you can check which COM port is connected to the USB-to-Null modem cable.

4. Double-click on the COM port that is connected to the USB to Null modem cable. (In the preceding figure, this is COM4.)

   The Properties dialog box appears for the selected COM port.

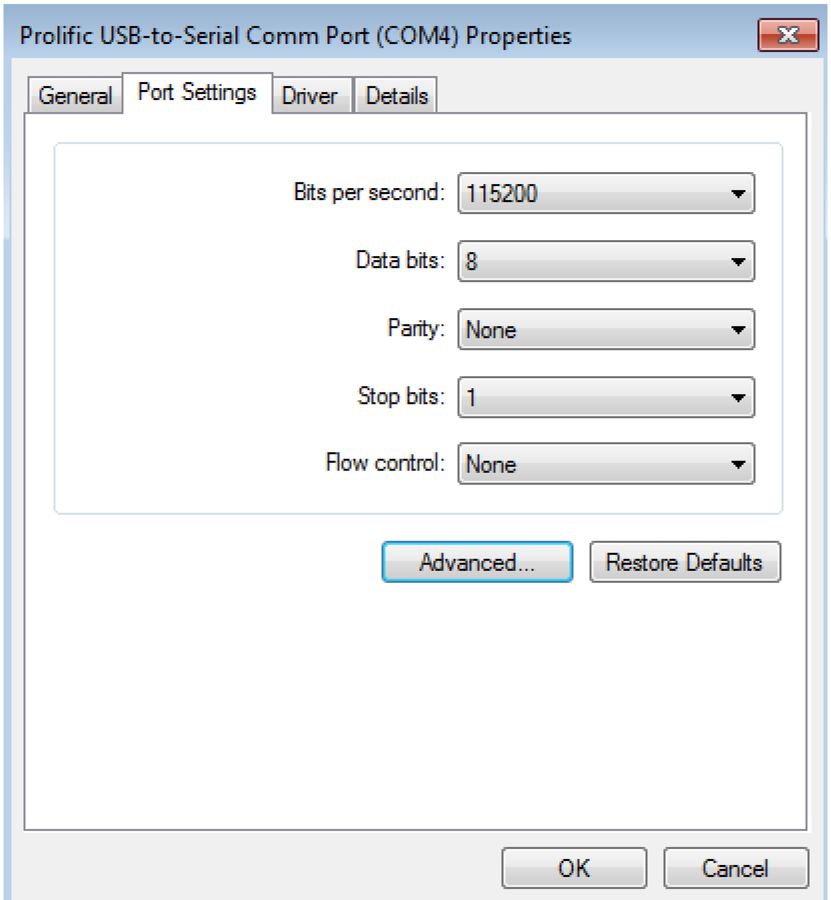


Figure 15. Sample Port Settings tab in the COM Port Properties dialog box.

5. From the Bits per second box on the Port Settings tab, select **115200** to set the data rate to 115200 bits per second.
6. Click the **OK** button.

2.5. Overview of the Vertex GUI

The input parameters for the Vertex channel emulator are organized into the following groups of parameters:

- Connection Setup
- Channel Model
    - Propagation Conditions
    - Correlation (applicable only to Classical Channel Models)
- RF Parameters

The Vertex GUI, shown in the following figure, allows you to view and set these parameters.

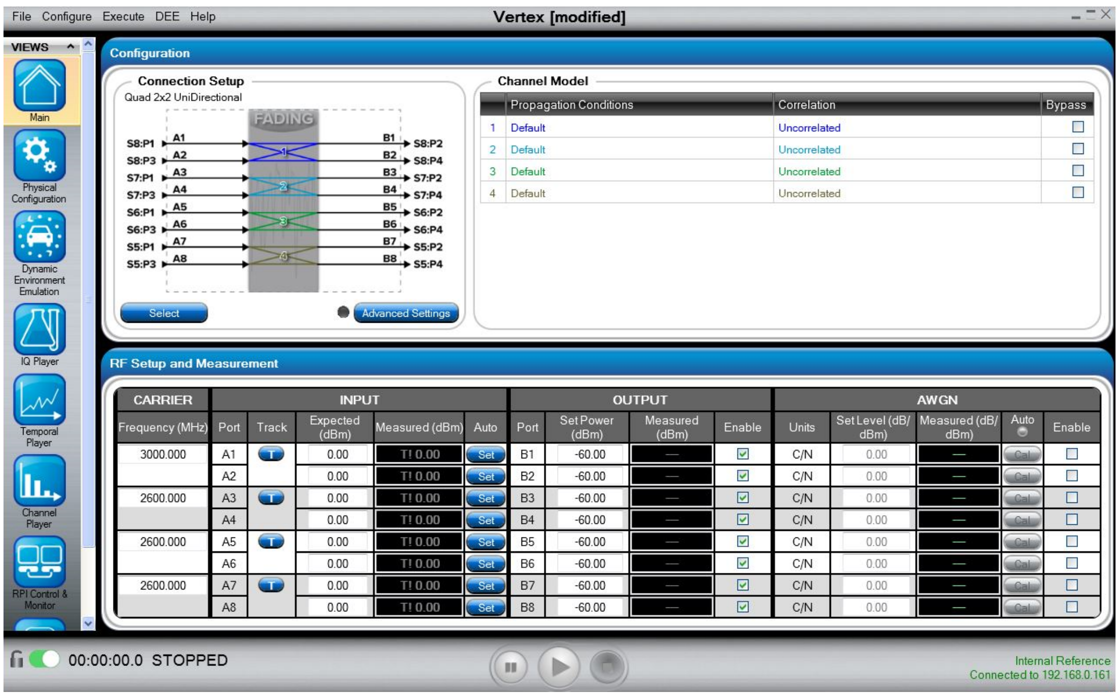


Figure 16. Vertex GUI.

2.5.1. Tooltips

The Vertex GUI provides "tooltips" for quick explanations on certain parameters when you roll the mouse cursor over text in the window. To use tooltips, keep the mouse cursor over a text label to view a pop-up window that provides a short explanation of the corresponding parameter.

2.5.2. Play/Pause/Stop Fading

Use the **Play** button to start/resume the fading emulation. Use the **Stop** button to stop the fading emulation and reset the Elapsed Time to zero. Use the **Pause** button to temporarily suspend the fading emulation. While paused, use the **Play** button to resume fading emulation.

While stopped or paused, the signal passing through Vertex does not vary. The signal is subjected to the exact fading conditions now indicated by the Elapsed Time indicator.

2.5.2.1 Stop Mode Phase Relationships

When Vertex is in the “Stopped” state, phases are automatically applied to each of the radio links to provide maximum MIMO throughput. The actual phases applied vary based on the connection setup and are also referred to as the *Butler matrix*.

Sample phase settings applied during the “Stopped” state are provided in the following tables.

NOTE:
The following tables imply A ports as the inputs and B ports as the outputs. For the reverse MIMO channel in a bidirectional connection setup, the A and B columns are swapped in the following tables.

2x2 MIMO

	B1	B2
A1	0	-90
A2	-90	0

3x3 MIMO

	B1	B2	B3
A1	0	-120	120
A2	0	0	0
A3	120	-120	0

4x4, 4x2, 2x4 MIMO

	B1	B2	B3	B4
A1	-45	180	45	-90
A2	0	-45	-90	-135
A3	-135	-90	-45	0
A4	-90	45	180	-45

8x8 (8x2, 8x4) MIMO

	B1	B2	B3	B4	B5	B6	B7	B8
A1	157.5	-45	112.5	-90	67.5	-135	22.5	-180
A2	157.5	-90	22.5	135	-112.5	0	112.5	-135
A3	135	-157.5	-90	-22.5	45	112.5	-180	-112.5
A4	90	112.5	135	157.5	-180	-157.5	-135	-112.5
A5	-112.5	-135	-157.5	-180	157.5	135	112.5	90
A6	-112.5	-180	112.5	45	-22.5	-90	-157.5	135
A7	-135	112.5	0	-112.5	135	22.5	-90	157.5
A8	-180	22.5	-135	67.5	-90	112.5	-45	157.5

2.5.3. SDE File Saving and Loading

The SDE file allows you to store and recall a complete set of parameters describing the Vertex configuration.

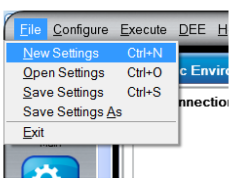


Figure 17. File menu options.

From the File menu, you have the following file-related options:

- **New Settings** - Resets the Vertex instrument to the default state. This is equivalent to *RST over the RPI interface.
- **Open Settings** - Opens the File Load dialog box, which allows you to select an SDE file to recall on the unit.
- **Save Settings** - Saves the changes to the currently loaded file. When no file has been loaded, it prompts you to save the file as a new or existing file.
- **Save Settings As** - Saves the settings to a new file or overwrites an existing SDE file.

When you save or load an SDE file, the name of the file is shown in the title bar of the Vertex window on the controller laptop. You can view the full file path by hovering the mouse cursor over the filename, as shown in the following figure.

Figure 18. Example of the full file path for a demo file.

When a setting has been changed in the configuration since the file was last saved or loaded, the **[modified]** tag appears in the title bar. The **[modified]** tag does not appear when the settings are saved or when a different file is loaded.

Figure 19. Example of a filename in which settings were changed.

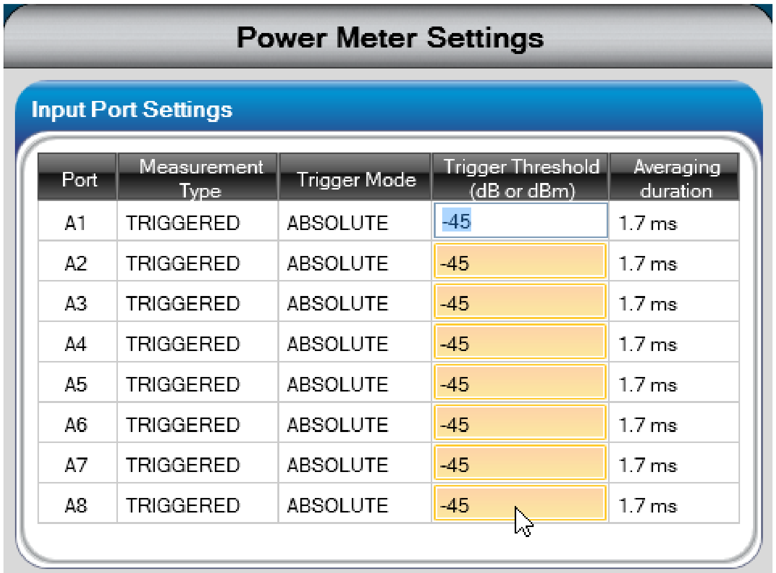
2.5.4. Group Parameter Editing

Group Parameter editing allows you to edit many of the parameter tables by group, including:

- Power Meter Settings
- AWGN Settings
- Port Phase Parameters
- Advanced Port Settings
- Propagation Conditions

To select a group of parameters in a column, select a field in the column and hold down the left mouse button to highlight other fields in the same column. After selecting the fields, changing one of the fields will change all other fields. You must leave the edited field before all the other highlighted fields change.

For example, select **Power Meter Settings**, left-click the first cell in the Trigger Threshold column, and drag to the last cell while holding down the left mouse button, as shown in the following figure.



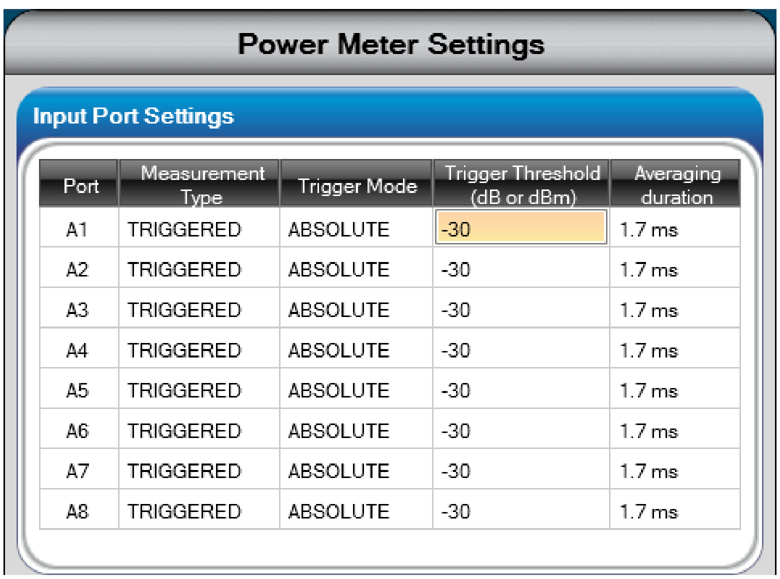
Power Meter Settings

Input Port Settings

Port	Measurement Type	Trigger Mode	Trigger Threshold (dB or dBm)	Averaging duration
A1	TRIGGERED	ABSOLUTE	-45	1.7 ms
A2	TRIGGERED	ABSOLUTE	-45	1.7 ms
A3	TRIGGERED	ABSOLUTE	-45	1.7 ms
A4	TRIGGERED	ABSOLUTE	-45	1.7 ms
A5	TRIGGERED	ABSOLUTE	-45	1.7 ms
A6	TRIGGERED	ABSOLUTE	-45	1.7 ms
A7	TRIGGERED	ABSOLUTE	-45	1.7 ms
A8	TRIGGERED	ABSOLUTE	-45	1.7 ms

Figure 20. Group parameter editing.

In the Trigger Threshold cell, enter a new Trigger Threshold of **-30 dBm**, and press the ENTER key. The entire column changes to **-30 dBm**, as shown in the following figure.



Power Meter Settings

Input Port Settings

Port	Measurement Type	Trigger Mode	Trigger Threshold (dB or dBm)	Averaging duration
A1	TRIGGERED	ABSOLUTE	-30	1.7 ms
A2	TRIGGERED	ABSOLUTE	-30	1.7 ms
A3	TRIGGERED	ABSOLUTE	-30	1.7 ms
A4	TRIGGERED	ABSOLUTE	-30	1.7 ms
A5	TRIGGERED	ABSOLUTE	-30	1.7 ms
A6	TRIGGERED	ABSOLUTE	-30	1.7 ms
A7	TRIGGERED	ABSOLUTE	-30	1.7 ms
A8	TRIGGERED	ABSOLUTE	-30	1.7 ms

Figure 21. Changing the Trigger Threshold.

2.5.4.1 Group Edit in the RF Setup and Measurement Grid

Group Edit in the RF Setup and Measurement grid is aided by the Edit indicator shown in the following figure.

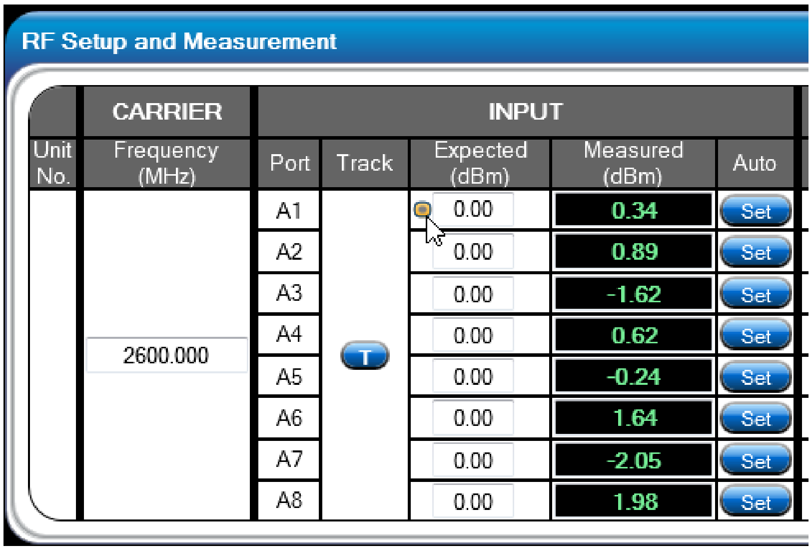


Figure 22. Group Edit indicator for the RF Setup and Measurement Grid.

Holding the cursor over an editable cell causes the Edit indicator to appear.

Hold the cursor over the indicator and left-click to edit the cell. Enter the new value in the cell, and press the ENTER key. The new value updates in all cells related to the group.

2.5.5. RF Setup and Measurement Window Scrolling

Since the Vertex channel emulator supports up to 32 RF channels in one instrument, the RF Setup and Measurement window size is not big enough to show all the RF channels. To view all the RF channels, click in the RF Setup and Measurement window, and use the mouse wheel or the up/down arrows for vertical scrolling.

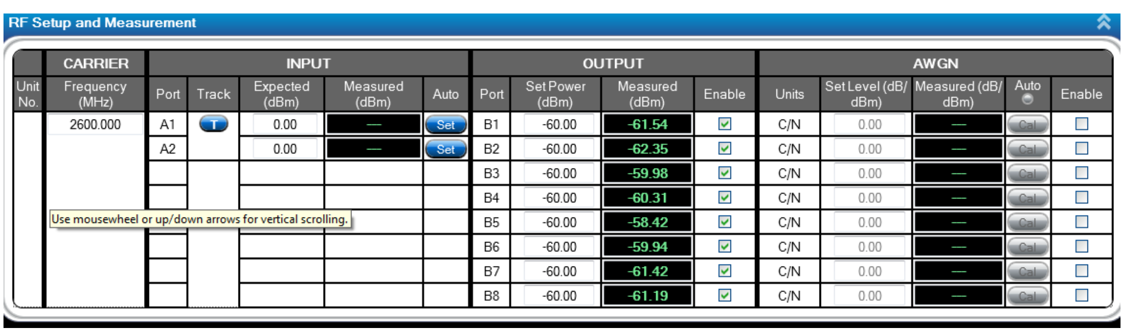


Figure 23. Sample RF Setup and Measurement window.

2.5.6. Connection Setup

The Vertex channel emulator supports a variety of MIMO and SISO configurations. The cabling is done internally within the hardware. The different configurations are selected through the connection setup.

To select a connection setup:

1. In the Main view of the Vertex window, under the Connection Setup area, click the **Select** button, as shown in the following figure.

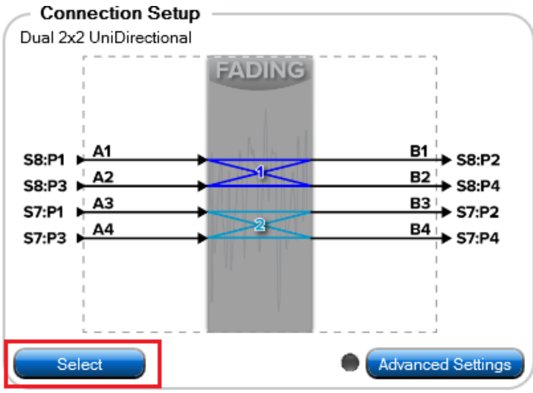


Figure 24. Select button in Connection Setup area.

The Connection Setup Selection dialog box appears.

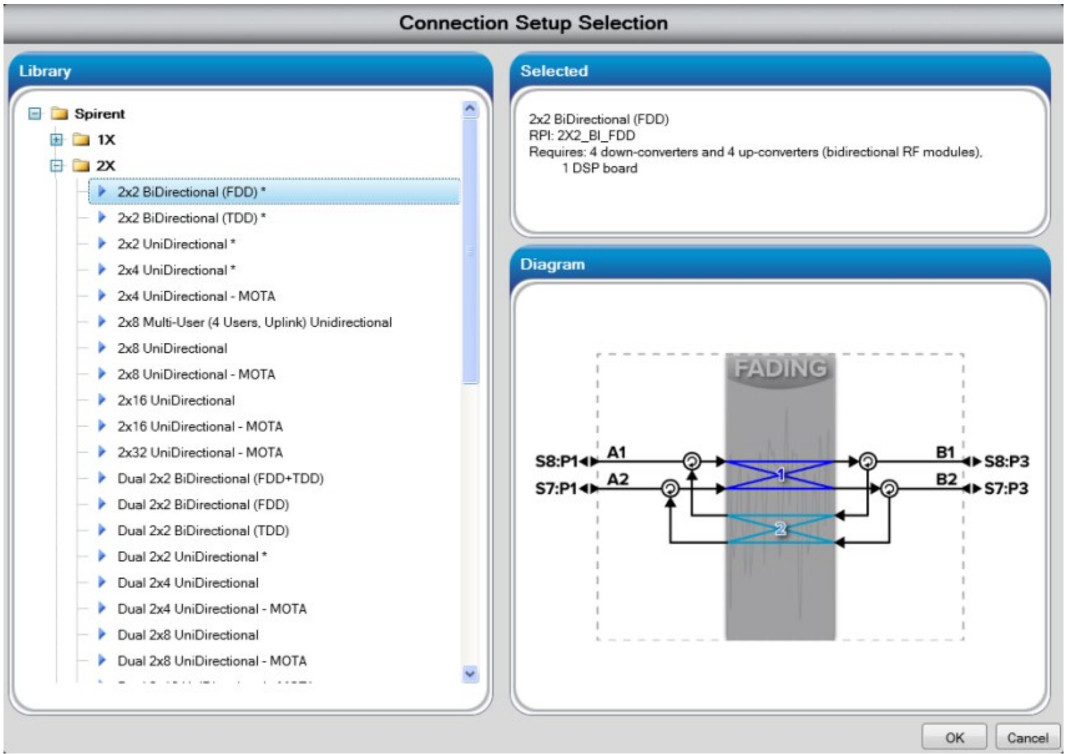


Figure 25. Connection Setup dialog box.

NOTE:
Vertex can automatically check which connection setups are supported with the current hardware configurations. An asterisk “*” appears next to each loadable connection setup.

The Connection Setup Selection dialog box displays all available connection setup configurations, as shown in the following figure.

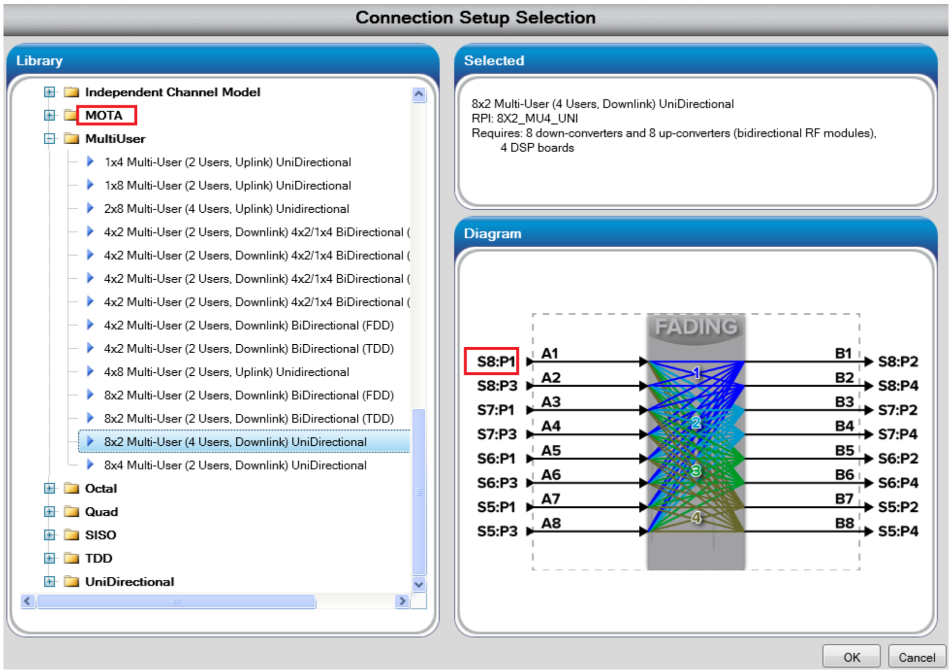


Figure 26. Connection Setup Selection dialog box.

NOTE:
Some connection setup types require multiple Vertex units (for example, 8x8 MIMO Bidirectional). In these cases, the connection diagram specifies which ports belong to which Vertex unit.

NOTE:
Since unidirectional and bidirectional modules may be installed in the Vertex channel emulator, some connection setups (like 2x16 for MIMO OTA application) may be available for both hardware configurations. If Vertex contains unidirectional RF modules, the connection marked with "MOTA" should be used.

2. Select the connection setup you would like to learn more about. Note that a setup does not take effect in the hardware until you click the **OK** button.

   In the Selected area, a verbal and visual description of the selected connection setup are provided. The "RPI: 8X8_UNI" text in the figure above is used to refer to this connection setup from the RPI.

   In the Diagram area, an illustration of the setup displays the following information:

   - Physical cabling and connections inside the Vertex hardware for the highlighted connection setup. Note that these connections are automatically made inside the Vertex when you select a particular connection setup. You do not need to connect anything manually.
   - The number of Vertex units required
   - The logical and physical ports used

     Ensure that you connect to the Vertex instrument on the physical ports indicated to the left-hand and right-hand sides of the diagram. The physical port is named **S**x:**P**y as shown in the following figure. "x" is the number of the slot on the front panel. "y" is the RF port number on the RF module in that slot. The RF port can be **OUT** or **I/O**.

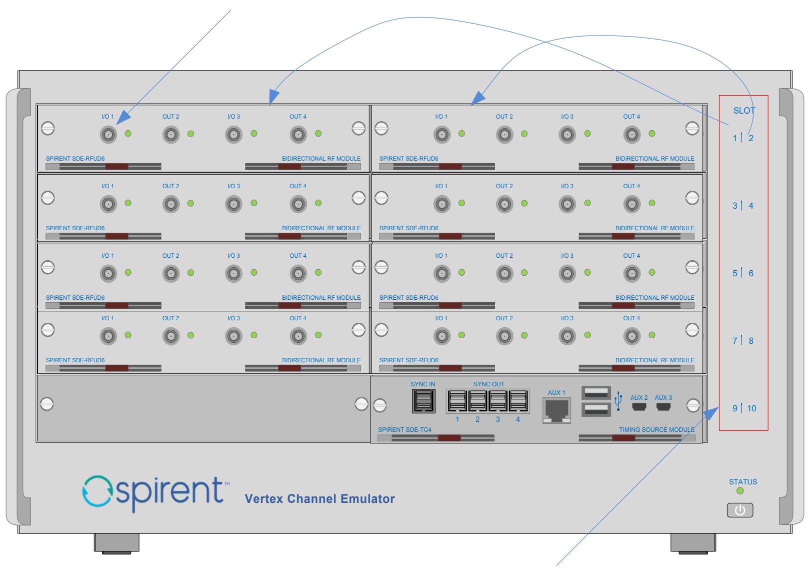


Figure 27. Slot and port number.

- Number of supported channel models
- The colored arrows inside the Fading box denote the various radio links. Each set of radio links with the same color shares the same channel model. The numbers in the relative radio links present the channel model used for that set of radio links. For more information on channel models, refer to Section 2.7.
- Whether the setup is unidirectional or bidirectional.

3. After selecting the connection setup, click the **OK** button.

   The unit is now configured according to the selected connection setup.

2.6. Multi-Instrument Operation

The Vertex GUI provides the following method for configuring scenarios that require more than one Vertex instrument. Vertex supports the following multi-instrument configurations:

- Dual-Vertex configuration
- Quad-Vertex configuration

2.6.1. Dual-Vertex Configuration

You can configure and control two Vertex instruments by a single Vertex GUI instance. This is necessary when correlation properties and channel reciprocity must be consistent across instruments. A dual-Vertex configuration can support up to a 32x4 bidirectional connection setup.

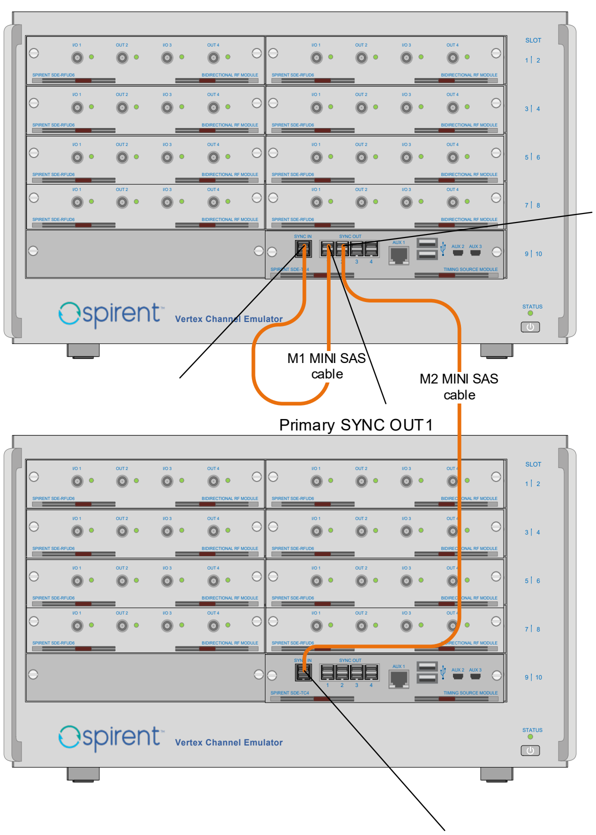


Figure 28. Dual-Vertex configuration - front panel.

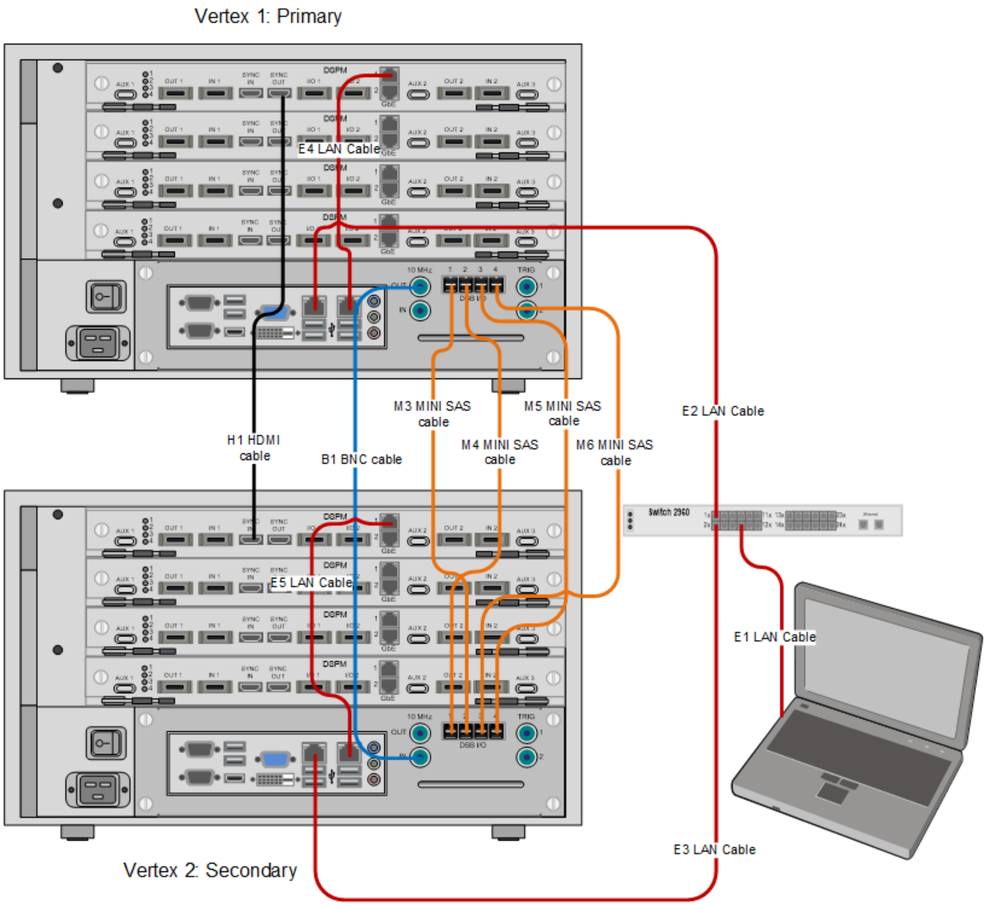


Figure 29. Dual-Vertex configuration - rear panel.

NOTE:
As a prerequisite to adding an additional instrument, you must properly connect sync cables, data cables, Ethernet cables, and 10 MHz reference cables between the primary and secondary instruments.

You must connect all instruments to the same 10 MHz reference source, either from an external source or from the primary (Vertex 1) Vertex instrument. If you are using the reference source from the Vertex instrument, connect the 10MHz OUT port of Vertex1 to the 10MHz IN port of Vertex2 with a BNC cable.

All instruments must be connected with the same Gbit Ethernet switch and in the same subnet. You must configure sync cables and DBB I/O cables as follows:

1. Connect the HDMI cable from the SYNC OUT port of DSPM 1 on Vertex1 to the SYNC IN port of DSPM 1 on Vertex2.
2. Connect the MINISAS cable from the SYNC OUT1 port of the Timing card on the Vertex1 front panel to the SYNC IN port of Vertex1.
3. Connect the MINISAS cable from the SYNC OUT2 port of the Timing card on the Vertex1 front panel to the SYNC IN port of Vertex2.

4. Connect the MINISAS cable from the DBB I/O 1 port of Vertex1 to the DBB I/O 2 port of Vertex 2 on the rear panel.
5. Connect the MINISAS cable from the DBB I/O 2 port of Vertex1 to the DBB I/O 1 port of Vertex 2 on the rear panel.
6. Connect the MINISAS cable from the DBB I/O 3 port of Vertex1 to the DBB I/O 4 port of Vertex 2 on the rear panel.
7. Connect the MINISAS cable from the DBB I/O 4 port of Vertex1 to the DBB I/O 3 port of Vertex 2 on the rear panel.

2.6.2. Quad-Vertex Configuration

The quad-Vertex configuration requires the Spirent Vertex Baseband Synchronizer (VBS) instrument, which integrates the four Vertex instruments. The Vertex Baseband Synchronizer facilitates the distribution of high-speed digital data along with critical clocks and synchronization signals within a Vertex system.

A quad-Vertex configuration can support up to a 64x8 bidirectional connection setup.

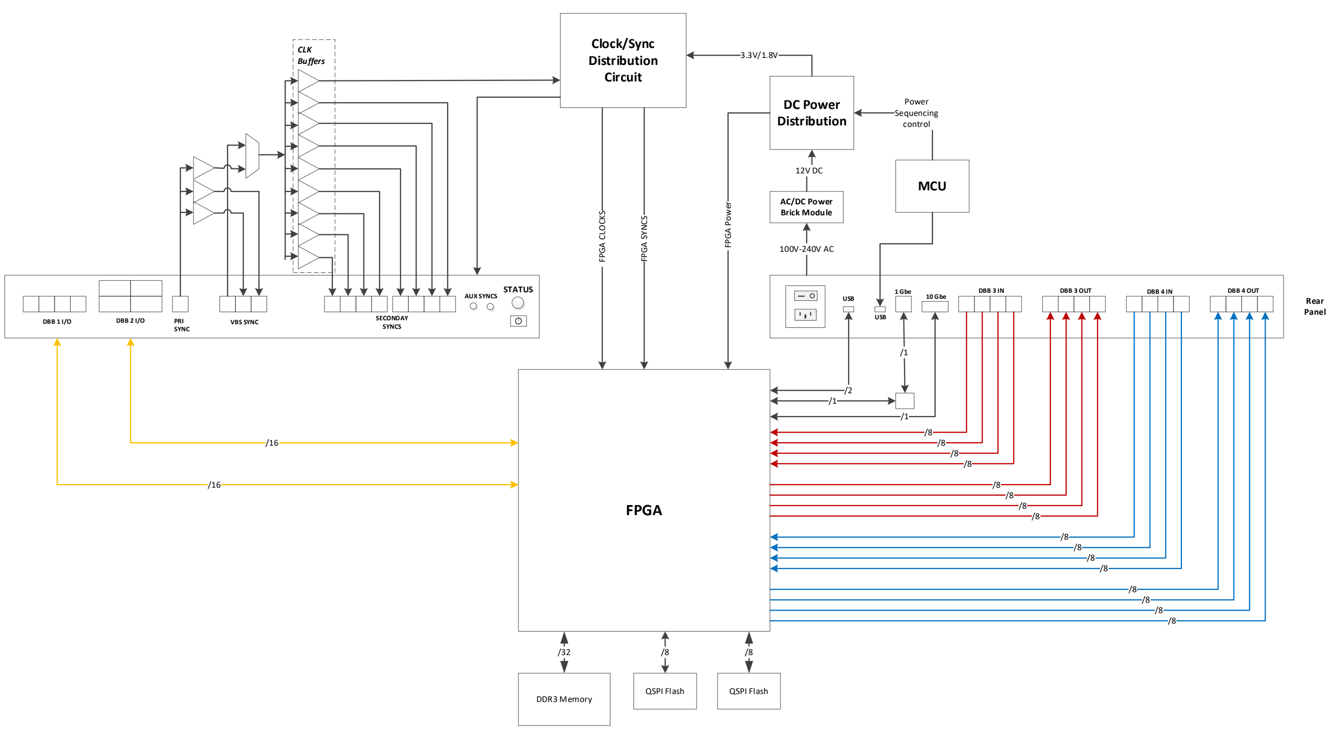


Figure 30. Vertex Baseband Synchronizer.

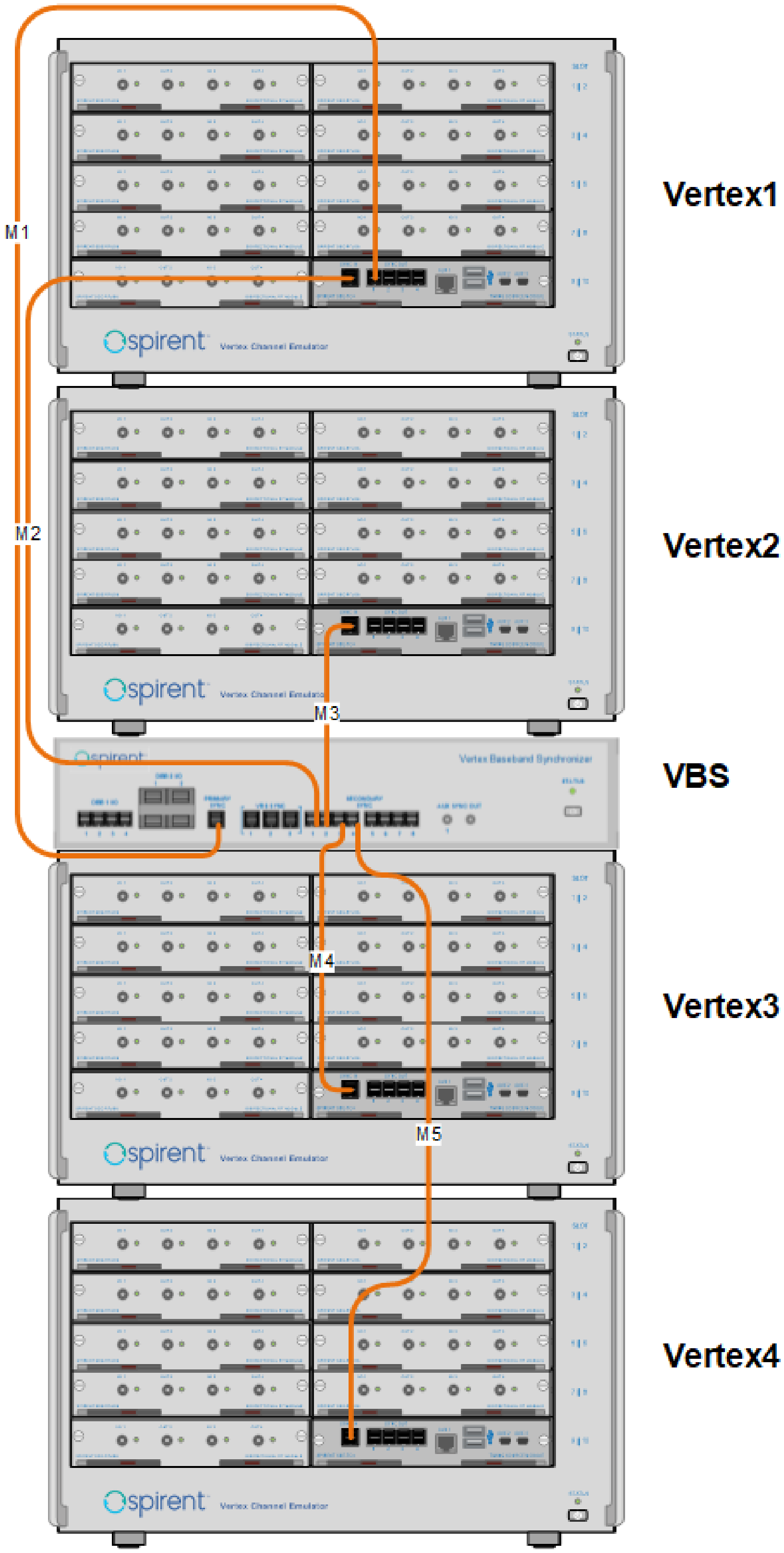


Figure 31. Quad-Vertex configuration - front panel.

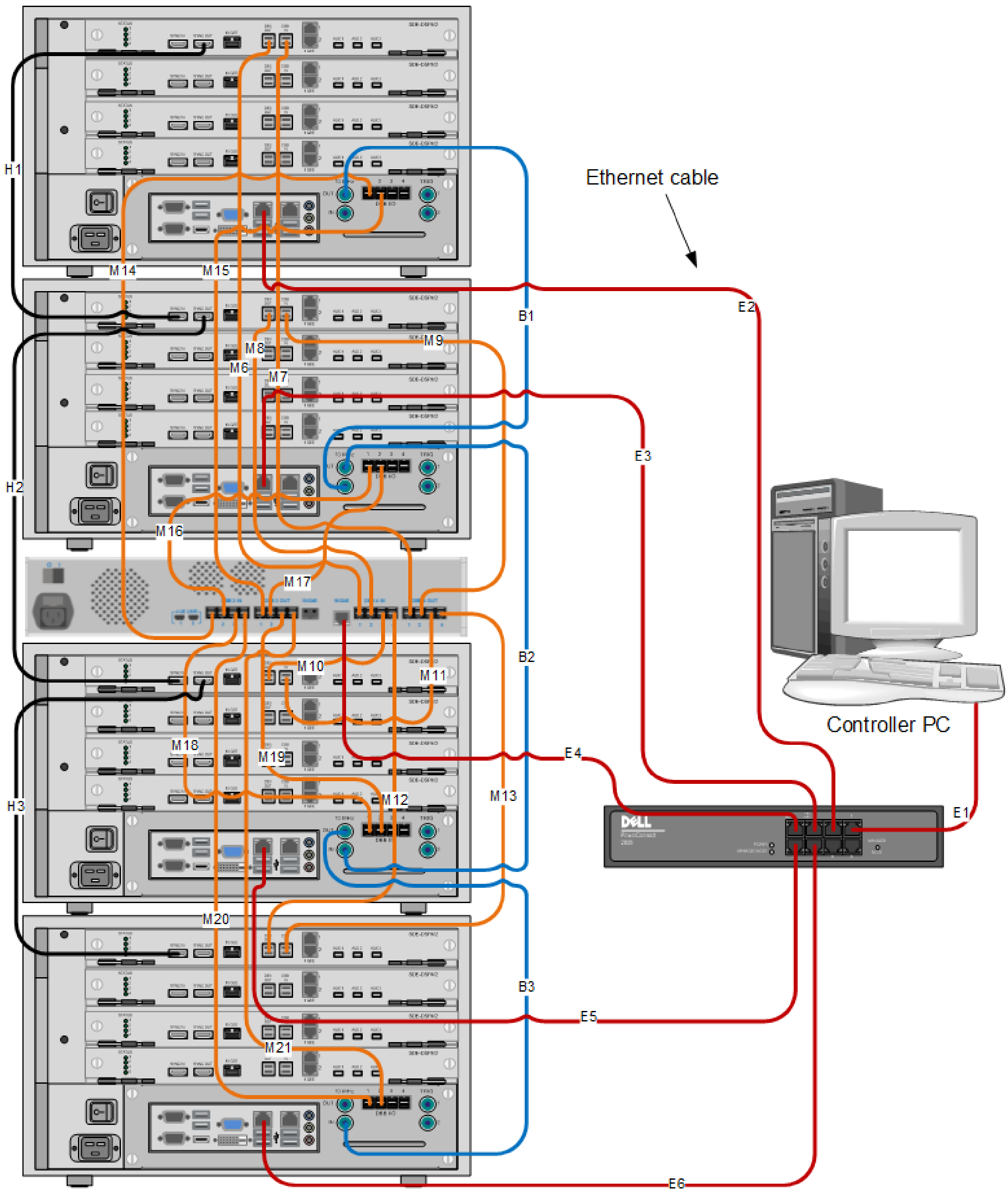


Figure 32. Quad-Vertex configuration - rear panel.

In the quad-Vertex configuration, you must connect all instruments to the same 10 MHz reference source, either from an external source or from the primary (Vertex 1) Vertex instrument. If you are using the reference source from the Vertex instrument, you must perform the following steps:

1. On the rear panel, connect a BNC cable from the 10MHz OUT port on Vertex1 to the 10MHz IN port on Vertex2. (See cable B1 in Figure 32.)
2. On the rear panel, connect a BNC cable from the 10MHz OUT port on Vertex2 to the 10MHz IN port on Vertex3. (See cable B2 in Figure 32.)
3. On the rear panel, connect a BNC cable from the 10MHz OUT port on Vertex3 to the 10MHz IN port on Vertex4. (See cable B3 in Figure 32.)

All instruments must be connected with the same Gbit Ethernet switch and in the same subnet. You must configure sync cables and DBB I/O cables as follows:

1. On the front panel, connect a 1M mini SAS HD cable from the SYNC OUT1 port on Vertex1 to the PRI SYNC IN port on the Vertex Baseband Synchronizer. (See cable M1 in Figure 31.)
2. On the front panel, connect a 1M mini SAS HD cable from the SEC SYNC1 port on the Vertex Baseband Synchronizer to the DIV SYNC IN port on Vertex1. (See cable M2 in Figure 31.)
3. On the front panel, connect a 1M mini SAS HD cable from the SEC SYNC2 port on the Vertex Baseband Synchronizer to the DIV SYNC IN port on Vertex2. (See cable M3 in Figure 31.)
4. On the front panel, connect a 1M mini SAS HD cable from the SEC SYNC3 port on the Vertex Baseband Synchronizer to the DIV SYNC IN port on Vertex3. (See cable M4 in Figure 31.)
5. On the front panel, connect a 1M mini SAS HD cable from the SEC SYNC4 port on the Vertex Baseband Synchronizer to the DIV SYNC IN port on Vertex4. (See cable M5 in Figure 31.)
6. On the rear panel, connect a 1.5M mini SAS HD cable from the DSPM1 DBB OUT port on Vertex1 to the DBB4 IN1 port on the Vertex Baseband Synchronizer. (See cable M6 in Figure 32.)
7. On the rear panel, connect a 1.5 mini SAS HD cable from the DBB4 OUT1 port on the Vertex Baseband Synchronizer to the DSPM1 DBB IN port on Vertex1. (See cable M7 in Figure 32.)
8. On the rear panel, connect a 1M mini SAS HD cable from the DSPM1 DBB OUT port on Vertex2 to the DBB4 IN2 port on the Vertex Baseband Synchronizer. (See cable M8 in Figure 32.)
9. On the rear panel, connect a 1M mini SAS HD cable from the DBB4 OUT2 port on the Vertex Baseband Synchronizer to the DSPM1 DBB IN port on Vertex2. (See cable M9 in Figure 32.)
10. On the rear panel, connect a 1M mini SAS HD cable from the DSPM1 DBB OUT port on Vertex3 to the DBB4 IN3 port on the Vertex Baseband Synchronizer. (See cable M10 in Figure 32.)

11. On the rear panel, connect a 1M mini SAS HD cable from the DBB4 OUT3 port on the Vertex Baseband Synchronizer to the DSPM1 DBB IN port on Vertex3. (See cable M11 in Figure 32.)
12. On the rear panel, connect a 1.5M mini SAS HD cable from the DSPM1 DBB OUT port on Vertex4 to the DBB4 IN4 port on the Vertex Baseband Synchronizer. (See cable M12 in Figure 32.)
13. On the rear panel, connect a 1.5M mini SAS HD cable from the DBB4 OUT4 port on the Vertex Baseband Synchronizer to the DSPM1 DBB IN port on Vertex4. (See cable M13 in Figure 32.)
14. On the rear panel, connect a 1.5M mini SAS HD cable from the DBB I/O P1 port on Vertex1 to the DBB3 IN1 port on the Vertex Baseband Synchronizer. (See cable M14 in Figure 32.)
15. On the rear panel, connect a 1.5M mini SAS HD cable from the DBB3 OUT1 port on the Vertex Baseband Synchronizer to the DBB I/O P2 port on Vertex1. (See cable M15 in Figure 32.)
16. On the rear panel, connect a 1M mini SAS HD cable from the DBB I/O P1 port on Vertex2 to the DBB3 IN2 port on the Vertex Baseband Synchronizer. (See cable M16 in Figure 32.)
17. On the rear panel, connect a 1M mini SAS HD cable from the DBB3 OUT2 port on the Vertex Baseband Synchronizer to the DBB I/O P2 port on Vertex2. (See cable M17 in Figure 32.)
18. On the rear panel, connect a 1M mini SAS HD cable from the DBB I/O P1 port on Vertex3 to the DBB3 IN3 port on the Vertex Baseband Synchronizer. (See cable M18 in Figure 32.)
19. On the rear panel, connect a 1M mini SAS HD cable from the DBB3 OUT3 port on the Vertex Baseband Synchronizer to the DBB I/O P2 port on Vertex3. (See cable M19 in Figure 32.)
20. On the rear panel, connect a 1.5M mini SAS HD cable from the DBB I/O P1 port on Vertex4 to the DBB3 IN4 port on the Vertex Baseband Synchronizer. (See cable M20 in Figure 32.)
21. On the rear panel, connect a 1.5M mini SAS HD cable from the DBB3 OUT4 port on the Vertex Baseband Synchronizer to the DBB I/O P2 port on Vertex4. (See cable M21 in Figure 32.)
22. On the rear panel, connect an HDMI cable from the DSPM1 SYNC OUT port on Vertex1 to the DSPM1 SYNC IN port on Vertex2. (See cable H1 in Figure 32.)
23. On the rear panel, connect an HDMI cable from the DSPM1 SYNC OUT port on Vertex2 to the DSPM1 SYNC IN port on Vertex3. (See cable H2 in Figure 32.)
24. On the rear panel, connect an HDMI cable from the DSPM1 SYNC OUT port on Vertex3 to the DSPM1 SYNC IN port on Vertex4. (See cable H3 in Figure 32.)
25. Connect an Ethernet cable from the Ethernet switch to the Ethernet port on the Controller laptop. (See cable E1 in Figure 32.)

26. Connect an Ethernet cable from the Ethernet switch to the User Ethernet port on the rear panel of Vertex1. (See cable E2 in Figure 32.)
27. Connect an Ethernet cable from the Ethernet switch to the User Ethernet port on the rear panel of Vertex2. (See cable E3 in Figure 32.)
28. Connect an Ethernet cable from the Ethernet switch to the 10 GbE Ethernet port on the rear panel of the Vertex Baseband Synchronizer. (See cable E4 in Figure 32.)
29. Connect an Ethernet cable from the Ethernet switch to the User Ethernet port on the rear panel of Vertex3. (See cable E5 in Figure 32.)
30. Connect an Ethernet cable from the Ethernet switch to the User Ethernet port on the rear panel of Vertex4. (See cable E6 in Figure 32.)
31. Power up Vertex1.
32. Power up Vertex2.
33. Power up Vertex3.
34. Power up Vertex4.

2.6.3. Additional Instruments Setup

You must use the Additional Vertex Instruments dialog box to configure certain connection setup configurations that require more than one Vertex unit to achieve the desired MIMO configuration.

To configure a connection setup that contains more than one Vertex unit:

1. Select **Configure>Additional Vertex Instruments**.

   The Additional Vertex Instruments dialog box appears.

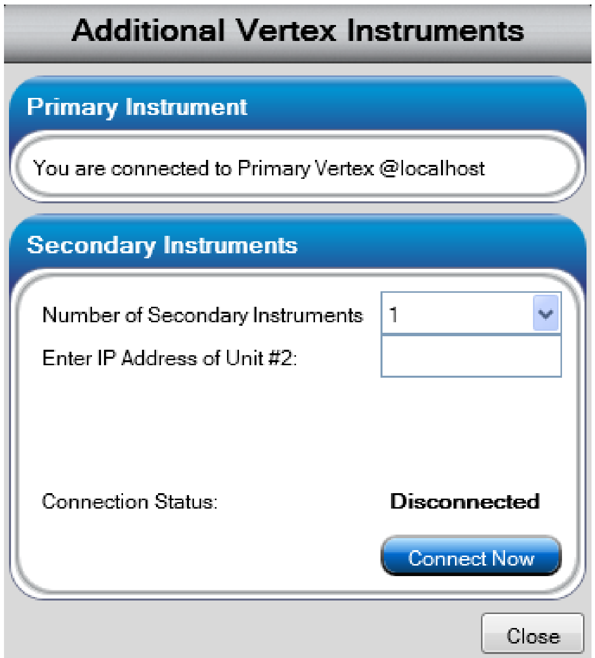


Figure 33. Additional Vertex Instruments dialog box.

2. Perform one of the following steps:
   - If you are using a dual-Vertex configuration, enter the IP address of the secondary Vertex unit in the Enter IP Address of Unit #2 box, and click the **Connect Now** button.

     When the connection process is complete, a default connection setup loads onto the Vertex unit. This setup spans the configured number of instruments.
   - If you are using a quad-Vertex configuration:
     a. From the Number of Secondary Instruments box, select **3** as shown in the following figure.

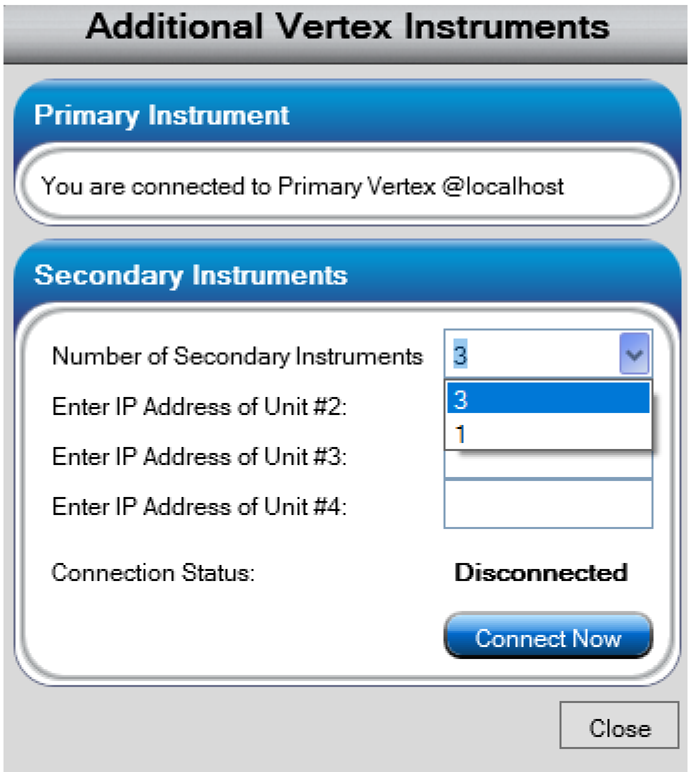


Figure 34. Additional Vertex Instruments dialog box.

     b. In the Enter IP Address of Unit #2 box, enter the IP address of Vertex2.
     c. In the Enter IP Address of Unit #3 box, enter the IP address of Vertex3.
     d. In the Enter IP Address of Unit #4 box, enter the IP address of Vertex4.
     e. Click the **Connect Now** button.

     When the connection process is complete, a default connection setup loads onto the Vertex unit. This setup spans the configured number of instruments.

2.7. Channel Model Setup

2.7.1. Selecting the Fading Mode

The Vertex channel emulator operates in several different fading modes. The fading mode determines the nature of the parameters entered in the channel model.

Classical Channel Models: These channel models are suitable for narrowband technologies. The input parameters consist of propagation conditions and the correlation.

Geometric Channel Models: These channel models are suitable for wide bandwidth, multiple antenna technologies.

The input parameters consist of the geometric (spatial) characteristics of the wireless environment emulated. Use this fading mode to emulate channel models based on SCM, SCME, and WINNER.

3D GCM mode is used to emulate the 3D geometrical channel models specified by 3gpp 39.901 and 36.873 standards.

The correlation between the radio links (MIMO branches) is implicit in the geometric settings selected for the channel model. The correlation is not specified separately for geometric channel models.

MIMO OTA Channel Models: These channel models are suitable for MIMO OTA environment building in a chamber.

To configure the fading mode:

1. In the Vertex window, select **Configure>Fading Mode**, as shown in the following figure.

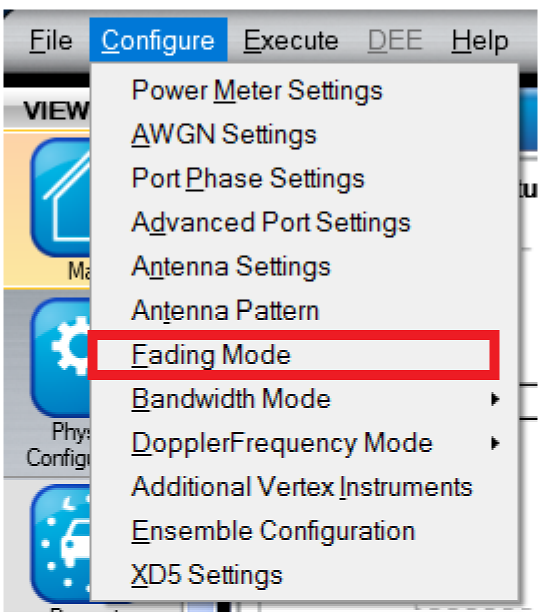


Figure 35. Fading Mode option in the Configure menu.

The Fading Mode dialog box appears, as shown in the following figure.

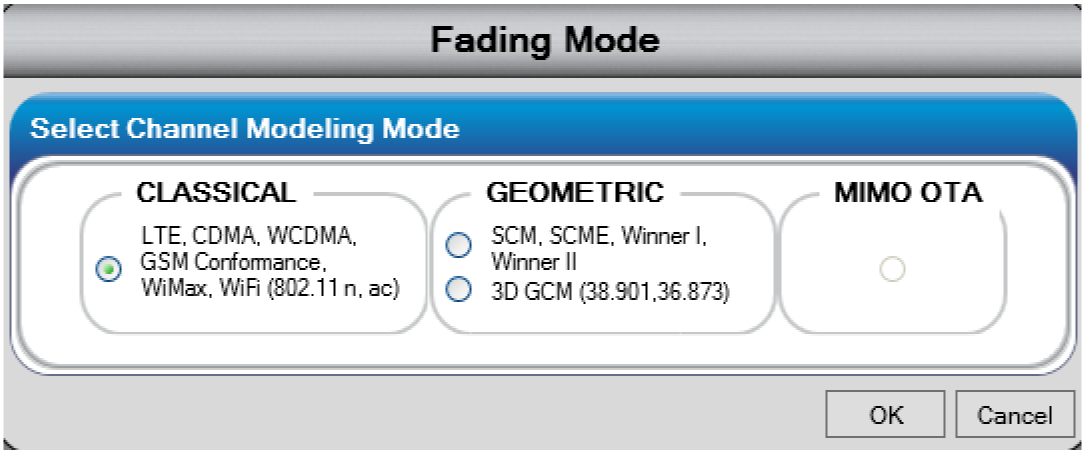


Figure 36. Fading Mode dialog box.

2. Click on the option button for the fading mode you want to use.

NOTE:
To put the Vertex unit in MIMO OTA mode, you must either launch the MIMO OTA Environment Builder software or use the RPI.

3. When finished, click the **OK** button.

2.7.2. Configuring the Channel Model

Each set of radio links with the same color in the Connection Setup Diagram area on the Main View of the Vertex window can have its own channel model. The numbers in the radio links indicate the index of the channel model. To view or edit the corresponding channel models, select **Channel Model** in the Main View of the Vertex window.

The propagation conditions and correlation matrix for each of the channel models can be edited, saved, and recalled in separate files independent of each other.

Each row in the table under the Channel Model box corresponds to the channel model for the set of radio links with the same index in the connection setup.

The channel model includes the following parameter groups:

- Propagation Conditions

  This is the multi-path power delay profile along with all the other fading parameters. You can specify one set of propagation conditions corresponding to each channel model.

- Correlation

  This is a matrix that represents the correlation between radio links. You can specify one correlation matrix corresponding to each channel model. This applies only to classical channel models.

- Bypass

  This allows you to bypass the propagation conditions and enable a clean path through the Vertex with one click.

These parameters are explained in detail in the following sections.

2.7.2.1 Propagation Conditions

To edit the propagation conditions:

1. Click the row corresponding to the propagation conditions you want to view or change.

   A drop-down arrow and an **Edit** button display in that row, as shown in the following figure.

Figure 37. Selecting the propagation conditions.

2. Click the **Edit** button.

   The Interactive Propagation Conditions Editor dialog box appears.

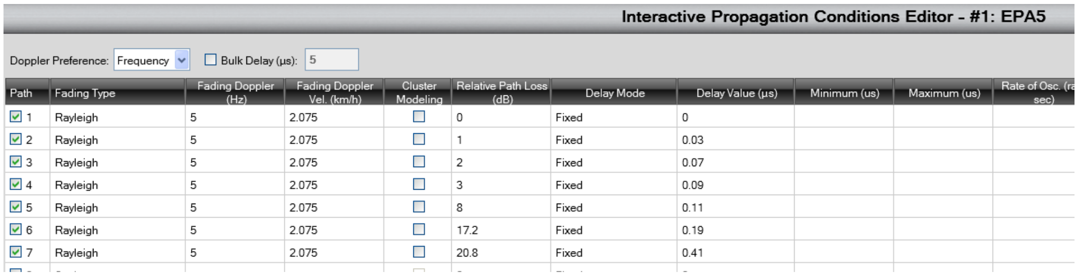


Figure 38. Sample Interactive Propagation Conditions Editor dialog box.

The table displays the following information:

- The rows correspond to multi-paths.
- The columns correspond to the various fading parameters.

The parameters that appear outside the table apply to all the paths.

The check boxes in the Path column enable you to include or exclude a path from the channel model. If the checkbox for a particular path is checked, it will be included in the channel model.

NOTE:
Any changes made to Propagation Conditions Parameters in this dialog box take effect in the Vertex hardware immediately.

3. Make your changes.
4. When you are finished, perform one of the following steps:

   - If you want to continue your tests without saving your changes, click the **Close** button to exit without saving the propagation conditions in a particular name for future recall.

     Note that the Vertex instrument still retains the altered propagation conditions under "Unsaved Profile."

Figure 39. Unsaved Profile.

   - If you want to save your changes in a file in the library:

     a. Click the **Save As** button.

        The Save Propagation Conditions dialog box appears.

     b. In the Propagation Conditions Name box, type a name for this file.
     c. In the Propagation Conditions Descriptions box, type a description for the propagation conditions in this file.

d. Click the **Save** button.

This file is now available in the Propagation Conditions library in the User Created folder.

In either case, the Vertex instrument reflects the propagation conditions as specified in the editor after exiting.

To apply standardized or previously saved propagation conditions:

1. Click the row corresponding to the propagation conditions you want to view or change.

   A field displays, as shown in the following figure. Clicking the arrow displays a list of the 10 most recently accessed propagation conditions.

Figure 40. Selecting propagation conditions.

2. If you want to access the entire library of Propagation Conditions:

   a. Scroll down to the bottom of the list, and click on **More**.

      The Propagation Conditions Selection dialog box appears. The Library area display both the “canned,” standardized Propagation Conditions supplied with the Vertex instrument and all user-created Propagation Conditions.

   b. Click on the folder that contains the propagation conditions file you want to use.

   c. Click on the propagation conditions file you want to use.

      The Selected area displays the information for each path, and the Power Delay Profile area displays the power delay for the selected propagation conditions file.

3. Click the **OK** button.

   The folder and name of the selected propagation conditions file appears in the corresponding cell in the Channel Model area.

2.7.2.2 Classical Channel Model Propagation Condition Parameters

This section describes the propagation condition parameters available with the Classical channel models in the editor.

Fading Type

The Fading Type can be set to **Static**, **Rayleigh**, or **Rician**.

Velocity and Doppler

This parameter sets the velocity for each path. The Carrier Frequency, Doppler, and Velocity parameters are interdependent. When the Carrier Frequency is changed, the Vertex channel emulator calculates the Doppler to maintain the currently set Velocity.

Spectrum Shape

The Vertex instrument allows you to select the Fading Spectrum Shape for each path with independently set status. You can only set the Fading Spectrum Shape for paths that are set to **Rayleigh** or **Rician**.

Rician Parameters

The following path parameters apply when the Path Status is set to **Rician**.

- Line of Site Angle of Arrival (LOS AOA)
- Line of Site Doppler (LOS Doppler)
- Rician K Factor (Rician K)

The LOS AOA and LOS Doppler are dependent. Setting one of these parameters causes the other to be reset to the appropriate calculated value. The Carrier Frequency, LOS AOA, and LOS Doppler parameters are interdependent. When the Carrier Frequency is changed, the Vertex channel emulator calculates the LOS Doppler to maintain the currently set LOS AOA.

Frequency Shift

Each path can have an independent Frequency Shift associated with it. If the Fading Type is set to **static**, this is sometimes called “Pure Doppler.”

Phase Shift

Each path can have an independently associated Phase Shift setting.

Delay Mode

The delay mode can be set to **Fixed**, **Moving Propagation**, or **Birth Death**.

Fixed Delay

Sets the amount of fixed delay associated with each path.

Moving Propagation

Moving Propagation can be applied to a path by setting the Sliding Delay Parameters grid.

Birth Death Delay

The Vertex unit allows any number of paths to have Birth Death Delay. To setup a path for Birth Death Delay, use the Delay Mode column and select **Birth Death**.

You can also configure the Birth Death settings parameters from the Birth Death Settings dialog box. To access the Birth Death Settings dialog box, click on the **Birth Death Settings** button as shown in the following figure.

Figure 41. Birth Death Settings button.

The Birth Death Settings dialog box appears.

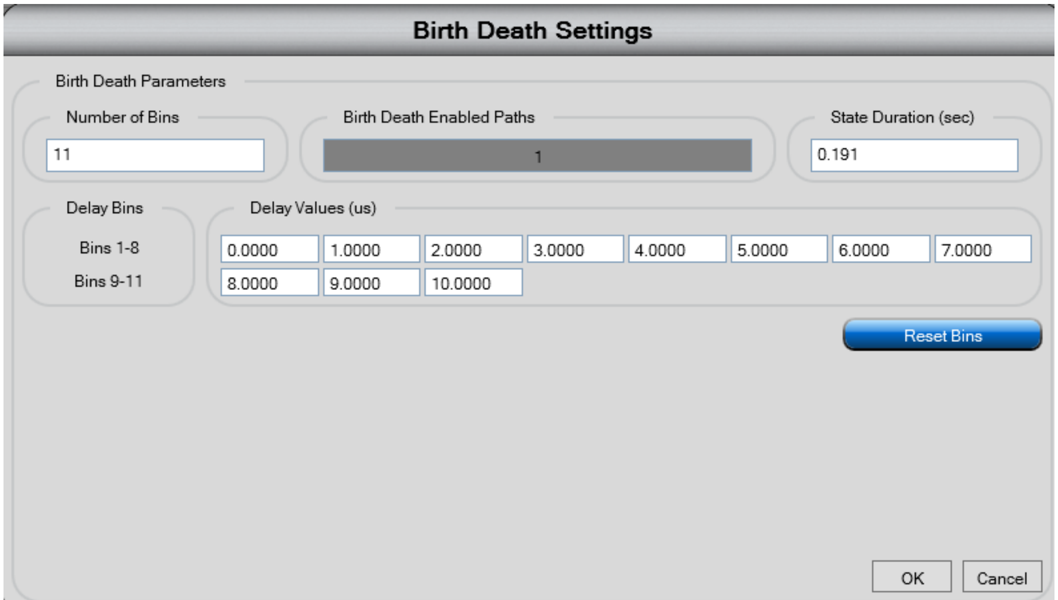


Figure 42. Birth Death Settings dialog box.

Relative Path Loss

Each path can have its own relative fixed loss. To set the Relative Path Loss, use the corresponding column.

NOTE:
The Vertex channel emulator normalizes the power of each path to maintain a composite channel power that equals the Set Output Level. The Path Loss value indicates the path power relative to other paths in the Power Delay Profile.

If only one path is enabled, the Relative Path Loss setting is not relevant.

2.7.2.3 Log-Normal Parameters

Each path can have Log-Normal fading enabled. You can also set the Log Normal Rate and Log Normal Std (Standard Deviation).

NOTE:
Enabling Log Normal on any path reduces the available output power setting for the channel and degrades system noise and spurious performance. This is due to the additional headroom requirements of Log-Normal. This is true even if the path is not enabled.

Bulk Delay

Defines an additional amount of delay applied to each path in the model. If you want to simulate the Bulk Delay (Group delay of all paths) in a larger range (5us to 4000us), check the Bulk Delay check box, and then set the value in the corresponding text box as shown in the following figure.

Figure 43. Bulk Delay check box.

2.7.2.4 Cluster Modeling Parameters

To support 802.11 n/ac/ax Indoor Channel Models, you can configure each path with parameters representing multiple paths from different clusters arriving at the same delay value. These parameters determine the Correlation Matrix for each path where Cluster Modeling is enabled, as well as the Relative Path Loss.

These parameters include on a per-cluster basis:

- Angle Of Arrival/Departure
- Angle Spread (Transmit/Receive)
- Power

NOTE:
There are only a few parameters set through the Cluster Modeling Parameters Editor that are geometric in nature. However, many other properties of cluster-based channel models, such as spectral shape, are not affected by the geometric parameters. This is why Cluster Modeling falls under the Classical Channel Models fading mode.

NOTE:
When Cluster Modeling is enabled on a given path, the Correlation Matrix for that path is automatically generated and becomes read-only. In addition, the relative path loss is generated based on the sum of the cluster powers.

2.7.2.5 Correlation

To view/edit the correlation matrix:

If you have selected “Classical Channel Models” as the fading mode, a Correlation column displays next to Propagation Conditions.

1. To edit the matrix, click the row corresponding to the Channel Model with the desired Correlation.

   A field displays as shown in the following figure. Clicking the arrow displays a list of the 10 most recently accessed Propagation Conditions.

Figure 44. Selecting propagation conditions.

2. Click the **Edit** button.

   The Correlation Editor dialog box appears, as shown in the following figure.

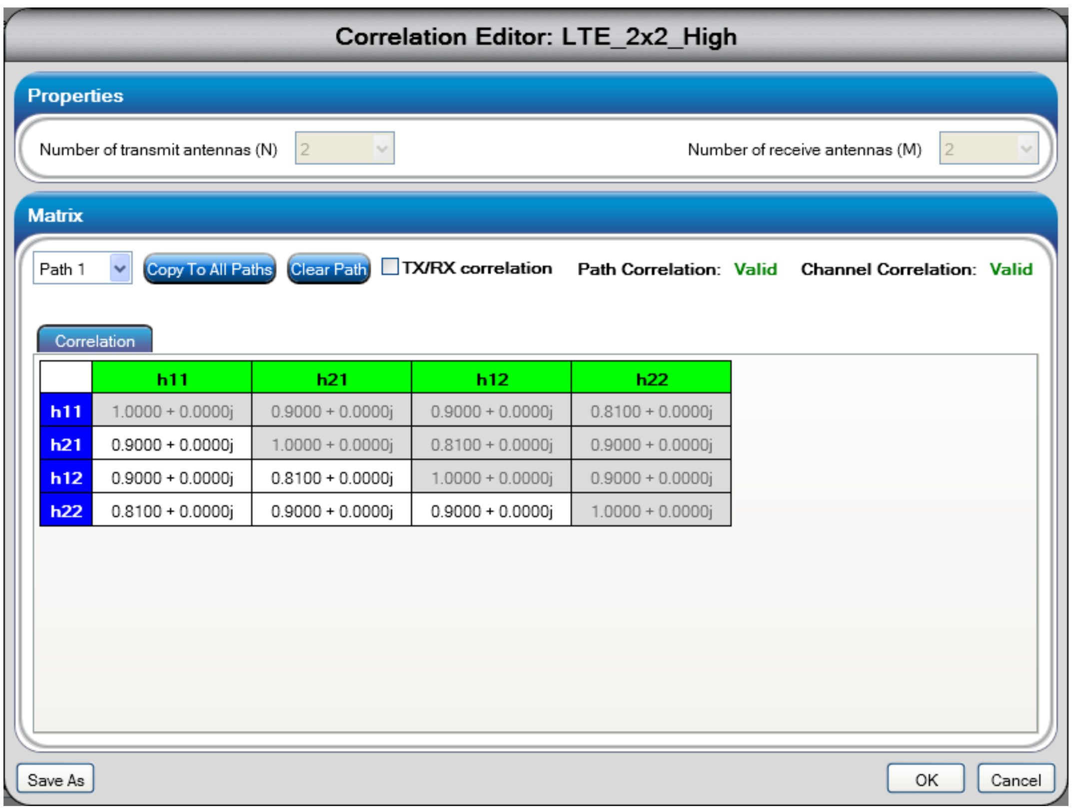


Figure 45. Correlation Editor dialog box.

The Properties area indicates the number of transmit antennas and receive antennas for the current connection setup.

The Matrix area allows you to view and edit the correlation matrix. You can set a unique correlation matrix for each path of your channel model. Select the path to view/edit by selecting it from the drop-down box at the top of the Matrix area.

Correlation matrices have the property that the lower triangle of the matrix is the complex conjugate of the upper triangle of the matrix. It is necessary for you to enter the correlation values for the lower triangle only. The upper triangle is automatically populated so that the entries are the complex conjugate of the corresponding entries on the lower triangle.

A correlation matrix should be positive semi-definite to be valid. The Vertex instrument automatically verifies the validity of the correlation matrix as you enter each element. If the matrix is valid, **Valid** appears, as shown in the following figure. If it is invalid, **Invalid** appears.

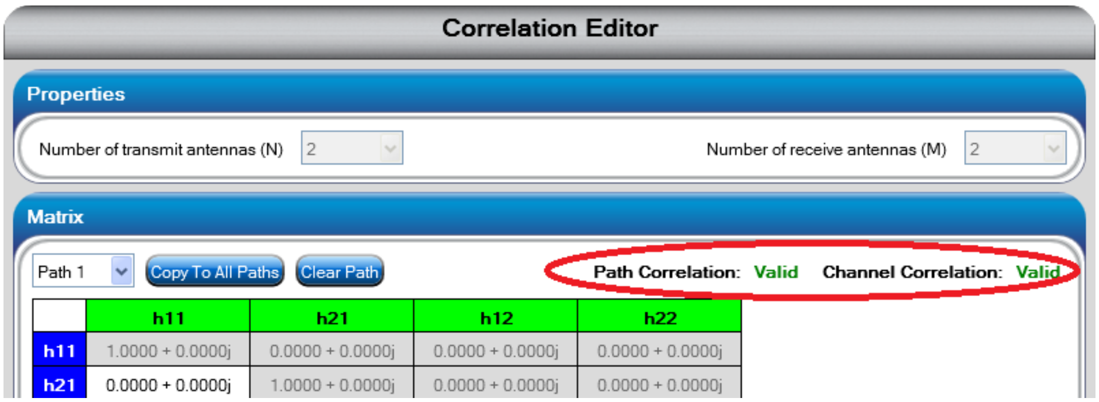


Figure 46. Correlation Validation.

If the correlation matrix for the current path is valid, but that of another path is invalid, that is indicated by the “Channel Correlation” label.

You can enter values into the correlation matrix in any order. However, keep in mind that the values entered in the top-left of the matrix affect the range of values in the bottom-right of the matrix. Because of these range dependencies, the easiest way to enter the values is from top to bottom by tabbing through the matrix.

You can also use the TX/RX correlation method to calculate the correlation matrix. To use the TX/RX correlation method, perform the following steps:

a. Check the **TX/RX correlation** check box.

   The Alpha and Beta boxes appear as shown in the following figure.

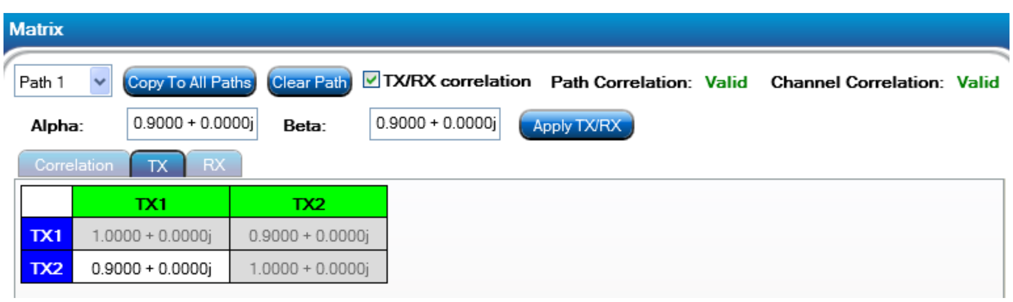


Figure 47. TX/RX Correlation Method.

b. In the Alpha box, enter the value of Alpha. (Refer to 3GPP TR36.101 Annex B.2.3 about the definition of Alpha and Beta.)

c. In the Beta box, enter the value of Beta.

d. Click the **Apply TX/RX** button.

The Correlation matrix is calculated automatically.

Figure 48. Correlation applied.

3. If you would like to use the same correlation matrix for all paths, click the **Copy To All Paths** button.

4. To reset the correlation matrix of the current path to zero, click the **Clear Path** button.

5. When you are finished making changes in the Correlation Editor, perform one of the following steps:

   - If you want to continue your tests without saving changes, click the **Close** button to exit without saving the correlation matrix in a particular name for future recall.

     Note that the Vertex channel emulator still retains the altered matrix conditions under “Unsaved Correlation.”

   - If you want to save your changes into a file in the library:

     a. Click the **Save As** button.

        The Save Correlation dialog box appears.

     b. In the Correlation Name box, type the name for the correlation matrix.

     c. In the Correlation Description box, type a description for the correlation matrix.

     d. Click the **Save** button.

        This file is available in the Correlation library in the **User Created** folder.

In either case, the Vertex channel emulator reflects the correlation matrix as specified in the editor after exiting.

To apply a standardized or previously saved correlation:

1. In the appropriate cell of the Correlation column, click the arrow to display a list of the ten most recently used Correlation Matrices for the current connection setup, as shown in the following figure.

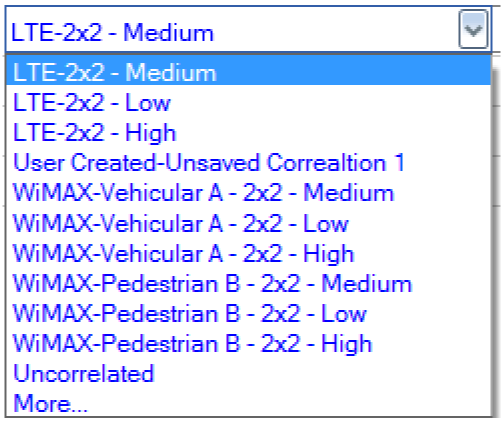


Figure 49. Selecting a correlation matrix.

2. To access to the entire library of Correlation Matrices for the current connection setup, scroll down to the bottom of the list and select **More**.

   The Correlation Selection dialog box appears. The Library area displays both “canned,” standardized correlation matrices supplied with the Vertex unit and user-created correlation matrices.

   NOTE:
   In a MIMO system, the dimensions of correlation matrices are directly related to the number of transmit and receive antennas. For an M x N MIMO system, the size of the correlation matrix is MN x MN.

   The Vertex unit filters the list of correlation matrices in the **More** drop-down menu to show only the matrices that are relevant to the current connection setup. The same applies for when you access the Library through the **More** drop-down menu. This ensures that only the matrices that you need for your current setup are displayed.

3. In the Library area, click on the folder that contains the correlation matrix you want to use.

4. Click on the correlation matrix you want to use.

   The Matrix area shows the matrix information for the selected correlation matrix.

5. Click the **OK** button.

2.7.2.6 Geometric Channel Model Propagation Condition Parameters

The following propagation condition parameters are available in the Geometric Channel Model:

Fading Type: Rayleigh and static modulation types are supported.

BS PAS: Defines the Power Azimuth Spectrum at the base station: Laplacian, Gaussian, or Uniform, for a path.

BS Angle Spread: This refers to the angle spread of each path at the BS.

MS PAS: Defines the Power Azimuth Spectrum at the mobile station: Laplacian, Gaussian, or Uniform, for a path.

MS Angle Spread: This refers to the angle spread of each path at the MS.

> **NOTE:**
> The higher the angle spread, the bigger the cluster from which the signal is arriving/departing. The higher the angle spread, the lower the correlation between channels for a given antenna separation and AoA/AoD.

AOD: The Angle of Departure (AoD) is defined to be the mean angle with which a departing path's (cluster's) power is transmitted by the BS array with respect to the antenna array broadside. The antenna broadside refers to the direction in which the antenna gain is the highest.

By setting different AoDs for each path, you can model the signal arriving/departing from the BS through clusters at different locations in space.

AOA: The Angle of Arrival (AoA) is defined to be the mean angle with which an arriving path's (cluster's) power is received by the MS array with respect to the antenna array broadside.

By setting different AoAs for each path, you can model the signal arriving at the MS from clusters at different locations in space.

MS Direction: Defines the angle of the MS velocity with respect to the MS broadside.

MS Velocity: Defines the Velocity of the MS.

LOS Enabled: Allows the addition of a Line of Sight (LOS) path.

LOS AOD: The LOS Angle of Departure (AOD) is the Angle of LOS direction between BS and MS, with respect to the broadside of the BS antenna array.

LOS AOA: The LOS Angle of Arrival (AOA) is the Angle of LOS direction between MS and BS, with respect to the broadside of the MS antenna array. This determines the Angle of Arrival of the LOS component.

LOS Doppler: This is the component of the Doppler in the LOS direction.

LOS K Factor: The K Factor is the ratio of power between the LOS component and the non-LOS (NLOS) components for a path.

The K Factor setting has a valid range of -30 dB (faded spectrum will dominate) to +30 dB (LOS component will dominate).

The power is divided such that the LOS component will have a relative power of K/K+1, and the power of the NLOS components is 1/K+1.

Mid-Paths Enabled: Enables mid-paths on the path. Refer to the SCME specification for more information on mid-paths.

Number of Mid-paths: When mid-paths is enabled, each path is composed of a number of mid-paths. The total path numbers per radio link will be reduced because the reduced paths are used for mid-paths.

Number of Scatterers per Mid-path: When mid-paths is enabled, each path is composed of a number of mid-paths.

Relative Power per Mid-path: This is a read-only field. It defines the relative power of each mid-path. The sum of mid-path powers for each path will be unique.

Excess Delay per Mid-path: Defines how much additional delay (beyond the delay set for the path) each mid-path has.

Num Scatterers: This is the number of scatterers per cluster in your environment. Presently, this number is fixed at 20. This value is read-only.

Delay Value: Defines the delay for the path. When mid-paths are enabled for a path, the maximum delay for the path is 1999.8 microseconds.

Relative Path Loss: Defines the relative power of path compared to path 1.

XPD: Defines the Cross-Polarization Discrimination value for the channel.

Bulk Delay: Defines an additional amount of delay to be applied to each path in the model.

Frequency of One Scatterer: Winner II B5 models define this parameter.

2.7.2.7 Geometric Channel Model Antenna Settings

Access the Geometric Channel Model Antenna settings from the **Configure** menu by selecting **Configure > Antenna Settings**. The Connection Setup Antenna Settings dialog box appears, as shown in the following figure.

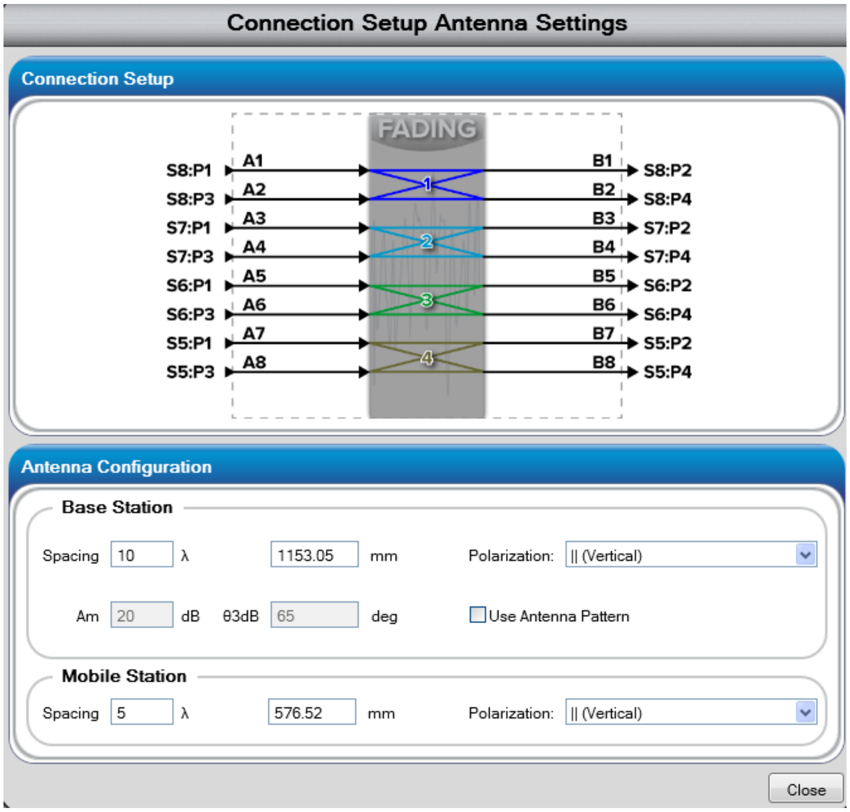


Figure 50. Connection Setup Antenna Settings dialog box.

In the Connection Setup Antenna Settings dialog box, you can set antenna-related parameters for the Geometric Channel Models.

The following parameters are available:

Base Station (BS)

Spacing: Defines the spacing, in wavelengths, of the BS antennas. Antenna spacing is also displayed in meters. This depends on the carrier frequency.

Polarization: Defines the polarity of the BS antenna as either parallel vertical antennas or cross-polarized antenna pairs.

Use Antenna Pattern: Allows antenna pattern parameters to be input for the BS.

Mobile Station (MS)

Spacing: Defines the spacing, in wavelengths, of the MS antennas. Antenna spacing is also displayed in meters. This depends on the carrier frequency.

Polarization: Defines the polarity of the MS antenna as either parallel vertical antennas or cross-polarized antenna pairs.

2.7.2.8 Geometric Channel Model Antenna Patterns

You can access the Geometric Channel Model Antenna patterns from the **Configure** menu by selecting **Configure > Antenna Pattern**. The Antenna Pattern Settings dialog box is shown in the following figure.

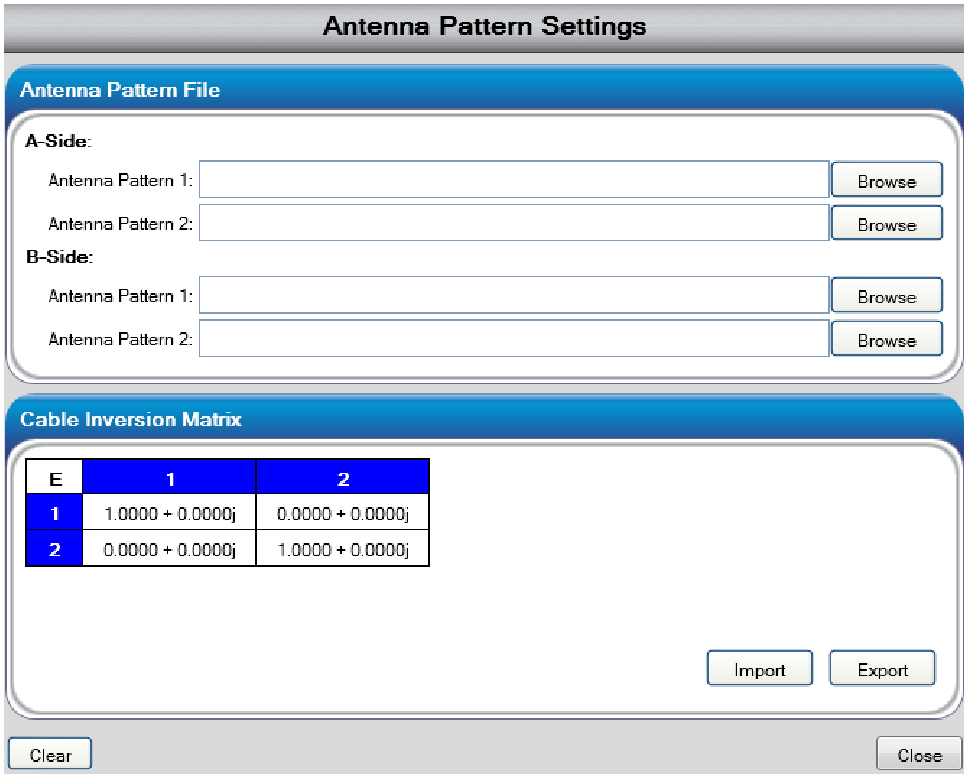


Figure 51. Antenna Pattern Settings window.

In the Antenna Pattern Settings window, you can set an antenna pattern file for vertical and horizontal antenna elements and a coupling matrix for use with Virtual OTA.

The following parameters are available:

Antenna Pattern File

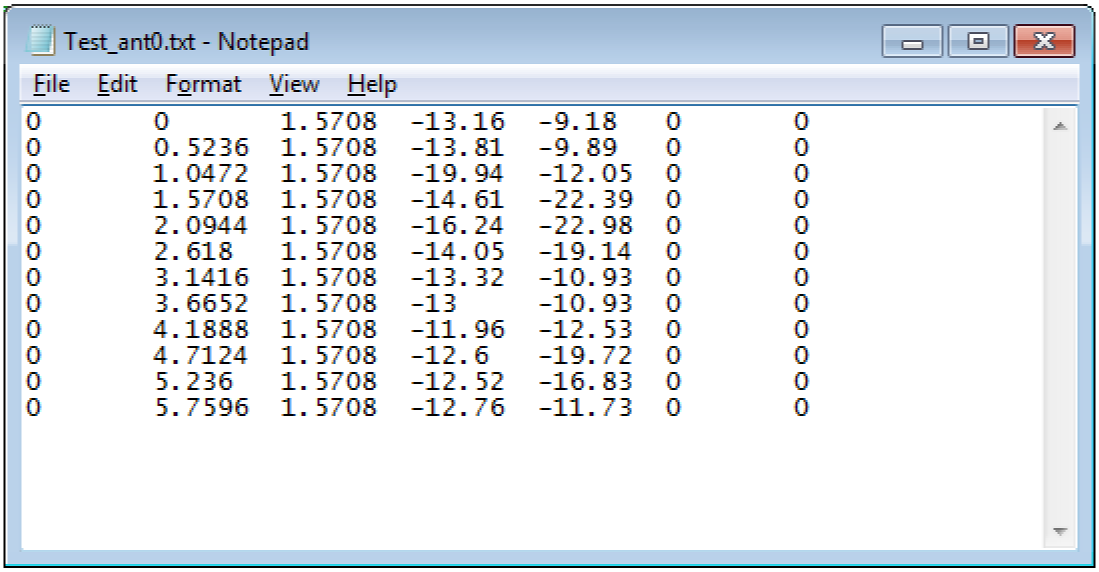
Pattern files are available for the Mobile Station and the Base Station.

Pattern File 1: Defines the antenna pattern for odd-indexed antennas elements of like orientation (i.e., 1, 3, 5, and 7).

Pattern File 2: Defines the antenna pattern for even-indexed antenna elements of like orientation (i.e., 2, 4, 6, and 8).

Antenna Pattern File Format

The following figure shows a sample antenna pattern file.


Test_ant0.txt - Notepad

File Edit Format View Help

0	0	1.5708	-13.16	-9.18	0	0
0	0.5236	1.5708	-13.81	-9.89	0	0
0	1.0472	1.5708	-19.94	-12.05	0	0
0	1.5708	1.5708	-14.61	-22.39	0	0
0	2.0944	1.5708	-16.24	-22.98	0	0
0	2.618	1.5708	-14.05	-19.14	0	0
0	3.1416	1.5708	-13.32	-10.93	0	0
0	3.6652	1.5708	-13	-10.93	0	0
0	4.1888	1.5708	-11.96	-12.53	0	0
0	4.7124	1.5708	-12.6	-19.72	0	0
0	5.236	1.5708	-12.52	-16.83	0	0
0	5.7596	1.5708	-12.76	-11.73	0	0

Figure 52. Sample Antenna Pattern File.

The format is tab-delimited text.

Column 1: This is an index identifier and can be left set to zeroes.

Column 2: Azimuth Angle (in radians) of the antenna pattern data point. (In this example, data points are spaced every 30 degrees.)

Column 3: Altitude Angle (in radians). Note: Altitude Angle is not supported yet.

Column 4: Horizontal Gain (in dB) of the antenna at the specified azimuth angle.

Column 5: Vertical Gain (in dB) of the antenna at the specified azimuth angle.

Column 6: Horizontal Phase (in radians) of the antenna at the specified azimuth angle.

Column 7: Vertical Phase (in radians) of the antenna at the specified azimuth angle.

Clear: Clears the antenna pattern and coupling matrix.

Cable Inversion Matrix

The cable inversion matrix allows you to calibrate the Virtual OTA wireless connection to the DUT in a shielded box.

NOTE:
Antenna pattern only works with the GCM channel model. If the path is set to **Static**, or channel emulation is **STOPPED**, the antenna pattern will not be taken into effect.

2.7.2.9 Optimized Geometric Channel Models

The Vertex channel emulator has the capability to optimize certain Geometric Channel Models to match theoretical correlation and branch powers to a higher degree of accuracy.

The Vertex software looks in the **C:\ProgramData\Spirent Communications\Vertex\db\PhaseOpt** directory for a file that conforms to the naming convention shown in the following table.

Model	Model Type	Line of Sight	BS Antenna Config	MS Antenna Config
IMT-A	_UMi	_LOS	_X	_X
SCME	_UMa	_NLOS	_V	_+
Winner II	_RMa			_V
	_SMa			

For example, to optimize phases for IMT-A UMi NLOS X to +, the **IMT-A_UMi_NLOS_X_+.txt** file must reside in the **PhaseOpt** directory.

The file contains a tab-delimited set of phases 4 columns by (20 x Number of Paths), as shown in the following figure.

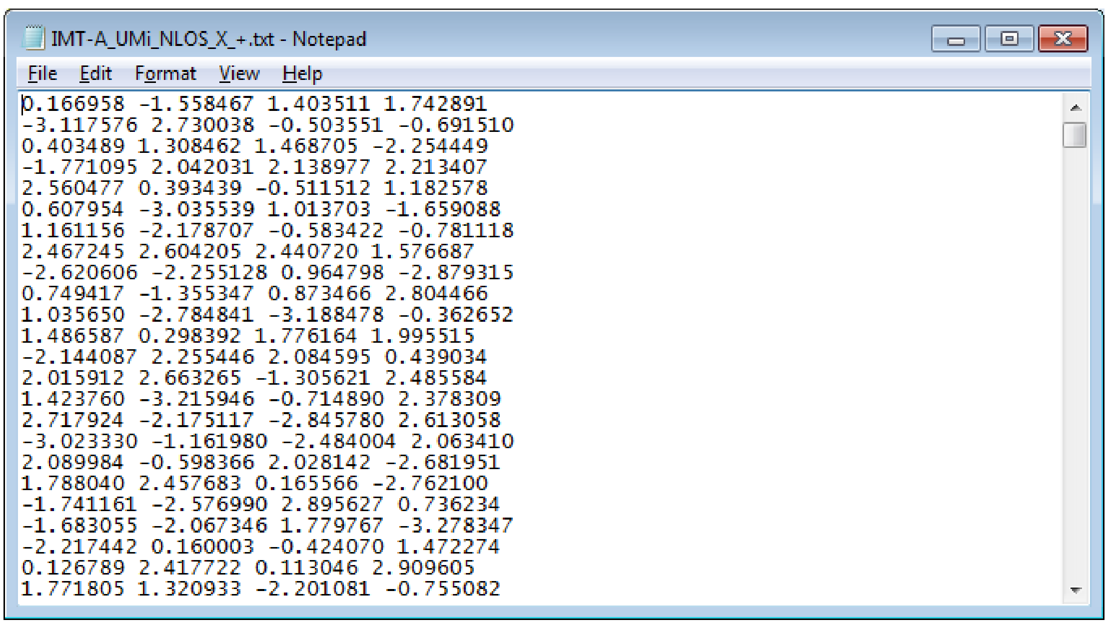


Figure 53. Phase table.

Phase values are in radians from -pi to +pi. If a Spirent-generated phase table does not exist for your setup, you can generate and drop a table into the **PhaseOpt** directory. You can also request a file from Spirent by contacting Spirent Customer Service.

When a valid phase table is found by the Vertex software, a “o” is shown next to the index number to indicate that the channel model has been optimized, as shown in the following figure.

Figure 54. Optimized propagation condition.

2.7.2.10 3D Geometric Channel Model Settings

NOTE:
DEE does not support 3D Geometric Channel model in Vertex release 4.70.

3D GCM (Geometric Channel Model) is different from classical or 2D GCM models.

In 3D GCM mode, if you click **Edit** in the Channel model window as shown in the following figure, you will see the model editor specified for 3G GCM model setting.

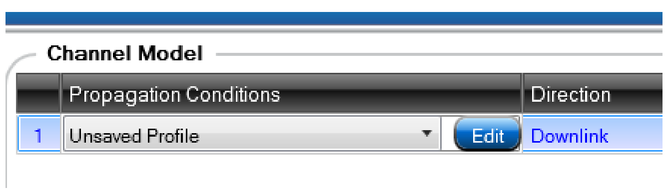


Figure 55. Edit button.

The 3D GCM model editor is called the Interactive Propagation Conditions Editor.

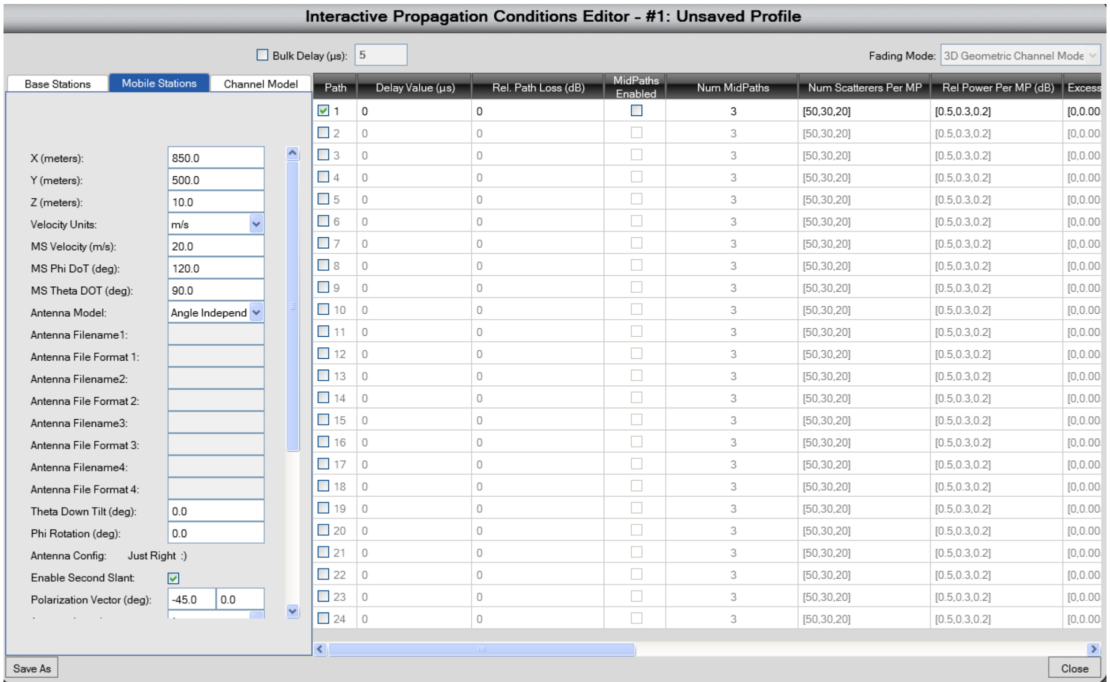


Figure 56. Interactive Propagation Conditions Editor.

The Interactive Propagation Condition Editor includes base stations, mobile stations, channel model settings and propagation editor window.

Base Stations setting

You can click the **Base Stations** tab to access the settings for the base station. The base station parameters include physical location (X,Y,Z), antenna model , antenna array angle , number and distance between antenna elements, polarization of antenna elements, antennal pattern property, and element antenna pattern simulation setting.

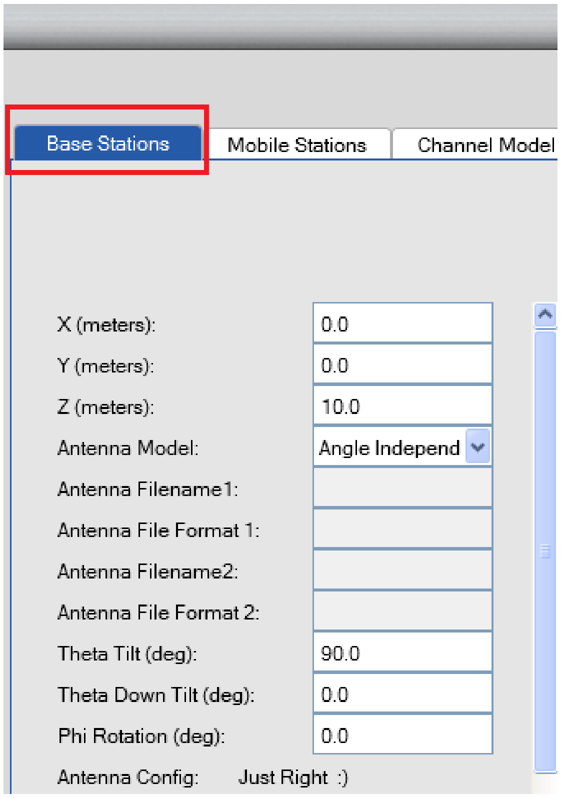


Figure 57. Base Stations tab.

Location area

Enables you to specify the X,Y coordinates for the base station and the height of the antenna array above ground level.

X (meters)

Specify the distance from origin in the X direction. Range is -10000 to 10000. Default is **0**.

Y (meters)

Specify the distance from origin in the Y direction. Range is -10000 to 10000. Default is **0**.

Z (meters)

Specify the height of the antenna array above ground level. Range is 0 to 10000. Default is **10**.

Antenna Parameters

Antenna Model

Specify the type of antenna model. Choices are ForeShortening, Angle Independent, and Read From File. Default is **ForeShortening**.

Antenna Filename1

Select the text file that contains the antenna pattern of the vertical slants. This parameter is available when **Antenna Model** is set to **Read From File**. Default is **Landscape_Ant0.txt**.

Antenna File Format 1

Enables you to arrange the columns of data in the selected file to match the expected order for ACM.

Antenna Filename2

Select the text file that contains the antenna pattern of the horizontal slants. This parameter is available when **Antenna Model** is set to **Read From File**. Default is **Landscape_Ant1.txt**.

Antenna File Format 2

Enables you to arrange the columns of data in the selected file to match the expected order for ACM.

Theta Tilt (deg)

Specify the electrical tilt of the antenna pattern. Range is 0 to 180 degrees. Default is **90** degrees. **0** indicates no tilt.

Theta Down Tilt (deg)

Specify the mechanical tilt of the antenna pattern. Range is -90 to 90 degrees. Default is **0** degrees. **0** indicates the antenna pattern points to the horizon. **90** indicates the antenna pattern points to the south pole. **-90** indicates the antenna pattern points to the north pole.

> **NOTE**:
> The overall antenna pattern is tilted down by the sum of both Theta Tilt and Theta Down Tilt. When the sum of both Theta Tilt and Theta Down Tilt is 0 degrees, the antenna pattern is pointing to the zenith. When it is tilted 90 degrees, the antenna pattern is pointing to the horizon.

Phi Rotation (deg)

Specify the mechanical rotation of the antenna array. Range is -180 to 180 degrees. Default is **45** degrees. **0** indicates the antenna array points to the X axis.

Enable Second Slant

Specify whether each antenna array location contains 2 slants. Choices are **Yes** and **No**. Default is **Yes**. **Yes** indicates each antenna array location contains 2 slants and they are co-centered (that is, “++”, ”XX”). **No** indicates each antenna array location contains 1 slant (“//”, ”\\”, ”||”, ”--“).

Polarization Vector (deg)

Specify the polarization vector for the antenna array slant. The left box sets the polarization vector for the first antenna array slant. The right box sets the polarization vector for the second antenna array slant. The right box is enabled if **Enable Second Slant** is set to **Yes**.

Range is -180 to 180 degrees. Default for the first antenna array slant is **45** degrees. Default for the second antenna array slant is **-45** degrees.

Number of Rows

Specify the number of rows in the antenna array. Range is 1 to 16. Default is **1** row.

Number of Columns

Specify the number of columns in the antenna array. Range is 1 to 16. Default is **1** column.

Distance Units

Specify the distance unit you want to use for the Distance Y and Distance Z settings. Choices are **Lambda** and **Meters**. Default is **Lambda**.

Distance Y

Specify the horizontal distance between adjacent antenna array elements that are in the same row of the antenna array. Range is > or = to 0. Default is **0.50**.

Distance Z

Specify the vertical distance between adjacent antenna array elements that are in the same column of the antenna array. Range is > or = to 0. Default is **0.50**.

Slant Start

Specify which of the 2 Polarization Vector parameters is indexed first. Choices are **First Element** and **Second Element**. **First Element** is the polarization vector for the first antenna array slant. **Second Element** is the polarization vector for the second antenna array slant. Default is **First Element**.

If you set the slant angles to 45, -45, you do not want the first numbered slant to be 45 or -45.

Count Style

Specify how the two antenna array slants are indexed. Choices are **Count in Order** and **Count Same Slant First**. Default is **Count in Order**. **Count in Order** indicates the two slants in the same location are indexed sequentially. **Count Same Slant First** indicates similarly slanted elements are indexed sequentially.

First Value

Specify whether the antenna elements indexing starts with 0 or 1. Choices are **Count Starts at 0** and **Count Starts at 1**. Default is **Count Starts at 1**.

Antenna Array Center

Specify the phase reference, which can be located at any slant location. If set to -1, it will reference the Antenna Array Center, which is not a slant location when the antenna array includes an even number of slants. Default is **-1**.

Normalized Vertical Gain

Specify whether you want to use normalized vertical gain. Choices are **Yes** and **No**. Default is **No**.

Remove self-normalization

Specify whether you want to remove self-normalization. Choices are **Yes** and **No**. Default is **Yes**.

Force AoDs To 0

Specify whether you want to ignore the paths azimuth angles of departure and replace them with 0 degrees. Choices are **Yes** and **No**. Default is **No**.

Force ZoDs To 90

Specify whether you want to ignore the paths zenith angles of departure and replace them with 90 degrees. Choices are **Yes** and **No**. Default is **No**.

Force UnCorrelated

Specify whether there is no correlation between neighboring slants. Choices are **Yes** and **No**. Default is **No**.

Normalize Output Power

Specify whether you want to normalize output power. Choices are **Yes** and **No**. Default is **No**.

Normalize Power Per Tap

This box is enabled if **Normalized Output Power** is set to **Yes**. When this setting is enabled, the total power is set to 1, and the power tap is set to exactly match what was described by the Relative Path Loss on the Channel Model tab for each cluster. This feature is not available under some conditions. ACM will display a message for these conditions.

Output Power Scale Factor

Specify the output power scale factor. This box is enabled if **Normalize Output Power** is set to **Yes**. Range is 0 to 100. Default is **0.5**.

Apply Ant Pattern

Specify whether to use the antenna pattern. Choices are **Yes** and **No**. Default is **No**.

Beam Width 3dB (deg)

Specify the degrees at which the antenna pattern falls by 3dB. Range is 0 to 360 degrees. Default is **65.0** degrees.

Max attenuation (dB)

Specify the attenuation in the back side of the pattern. Range is 0 to 100dB. Default is **30.0dB**.

Max Directional Gain (dB)

Specify the maximum gain of an individual element. Range is 0 to 100dB. Default is **8.0dB**.

Mobile Stations setting

Click the **Mobile Stations** tab to access the settings of a mobile station. You can configure the physical location and antenna property of mobile station.

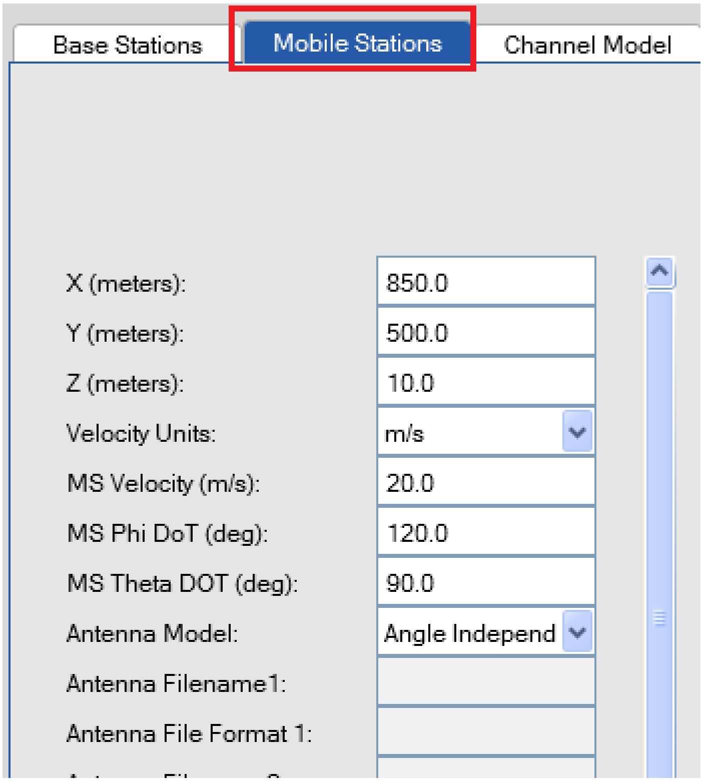


Figure 58. Mobile Stations tab.

X (meters)

This parameter is used when **Motion Type** is set to **Static**. Specify the distance from origin in the X direction. Range is -10000 to 10000. Default is **850**.

Y (meters)

This parameter is used when **Motion Type** is set to **Static**. Specify the distance from origin in the Y direction. Range is -10000 to 10000. Default is **500**.

Z (meters)

This parameter is used when **Motion Type** is set to **Static**. Specify the height of the antenna array above ground level. Range is 0 to 10000. Default is **10**.

Velocity Units

Specify how you want to measure the velocity of the mobile station. Choices are **m/s**, **Km/h**, and **mph**. Default is **m/s**.

Velocity

This parameter is used when **Motion Type** is set to **Linear Motion** or **Circular Motion**. Specify the speed at which the mobile station travels. Range is 0.01 to 500. Default is **20**.

Virtual Phi DoT (deg)

This parameter is used when **Motion Type** is set to **Static**. Specify the virtual direction of the mobile station in the XY plane. Range is -180 to 180 degrees. Default is **120** degrees.

Virtual Theta DoT (deg)

This parameter is used when **Motion Type** is set to **Static**. Specify the virtual direction of the mobile station on the Z axis. . Range is 0 to 180 degrees. Default is **0** degrees. **0** points to the zenith. **90** points to the horizon.

Antenna Parameters

Antenna Model

Specify the type of antenna model. Choices are **ForeShortening**, **Angle Independent**, and **Read From File**. Default is **ForeShortening**.

Antenna Filename1

Select the text file that contains the antenna pattern of the vertical slants. This parameter is available when **Antenna Model** is set to **Read From File**. Default is **Landscape0_Ant0.txt**.

Antenna File Format 1

Enables you to arrange the columns of data in the selected file to match the expected order for ACM. This parameter is available when **Antenna Model** is set to **Read From File**.

Antenna Filename2

Select the text file that contains the antenna pattern of the horizontal slants. This parameter is available when **Antenna Model** is set to **Read From File**. Default is **Landscape0_Ant1.txt**.

Antenna File Format 2

Enables you to arrange the columns of data in the selected file to match the expected order for ACM. This parameter is available when **Antenna Model** is set to **Read From File**.

Antenna Filename3

Select the text file that contains the dipole settings. This parameter is available when **Antenna Model** is set to **Read From File**. Default is **Ideal_dipole.txt**.

Antenna File Format 3

Enables you to arrange the columns of data in the selected file to match the expected order for ACM. This parameter is available when **Antenna Model** is set to **Read From File**.

Antenna Filename4

Select the text file that contains the loop settings. This parameter is available when **Antenna Model** is set to **Read From File**. Default is **Ideal_loop.txt**.

Antenna File Format 4

Enables you to arrange the columns of data in the selected file to match the expected order for ACM. This parameter is available when **Antenna Model** is set to **Read From File**.

Theta Down Tilt (deg)

Specify the mechanical tilt of the antenna array. Range is 0 to 180 degrees. Default is **90** degrees. **0** indicates the antenna array points to the zenith. **90** indicates the antenna array points to the horizon.

Phi Rotation (deg)

Specify the mechanical rotation of the antenna array. Range is -180 to 180 degrees. Default is **0** degrees. **0** indicates the antenna array points to the X axis.

Enable Second Slant

Specify whether each antenna array location contains 2 slants. Choices are **Yes** and **No**. Default is **Yes**. **Yes** indicates each of the antenna array locations contains 2 slants. **No** indicates each antenna array location contains 1 slant.

Polarization Vector (deg)

Specify the polarization vector for the antenna array slant. The left box sets the polarization vector for the first antenna array slant. The right box sets the polarization vector for the second antenna array slant. The right box is enabled if **Enable Second Slant** is set to **Yes**.

Range is -180 to 180 degrees. Default for the first antenna array slant is **0** degrees. Default for the second antenna array slant is **90** degrees.

Antenna Locations

Specify the antenna location. Choices are **Array** and **Arbitrary**. Default is **Array**.

Number of Rows

Specify the number of rows in the antenna array. This parameter is used when **Antenna Locations** is set to **Array**. Range is 1 to 16. Default is **1** row.

Number of Columns

Specify the number of columns in the antenna array. This parameter is used when **Antenna Locations** is set to **Array**. Range is 1 to 16. Default is **1** column.

Distance Units

Specify the distance unit you want to use for the Distance Y and Distance Z settings. Choices are **Lambda** and **Meters**. Default is **Lambda**.

Distance Y

Specify the horizontal distance between adjacent antenna array elements that are in the same row of the antenna array. Range is > or = to 0. Default is **0.50**.

Distance Z

Specify the vertical distance between adjacent antenna array elements that are in the same column of the antenna array. Range is > or = to 0. Default is **0.50**.

X Position Vector

Specify the X coordinate for each antenna. This parameter is used when **Antenna Locations** is set to **Arbitrary**.

Y Position Vector

Specify the Y coordinate for each antenna. This parameter is used when **Antenna Locations** is set to **Arbitrary**.

Z Position Vector

Specify the Z coordinate for each antenna. This parameter is used when **Antenna Locations** is set to **Arbitrary**.

Antenna Array Center

Specify the phase reference, which can be located at any slant location. Range is -1 to array dimension. If set to **-1**, it will reference the Antenna Array Center, which is not a slant location when the antenna array includes an even number of slants. Default is **-1**.

Force UnCorrelated

Specify whether there is no correlation between neighboring slants. Choices are **Yes** and **No**. Default is **No**.

Channel Model Settings

Click the **Channel Model** tab to access the settings of the channel model. You can select a pre-defined 3D GCM model or create a new model.

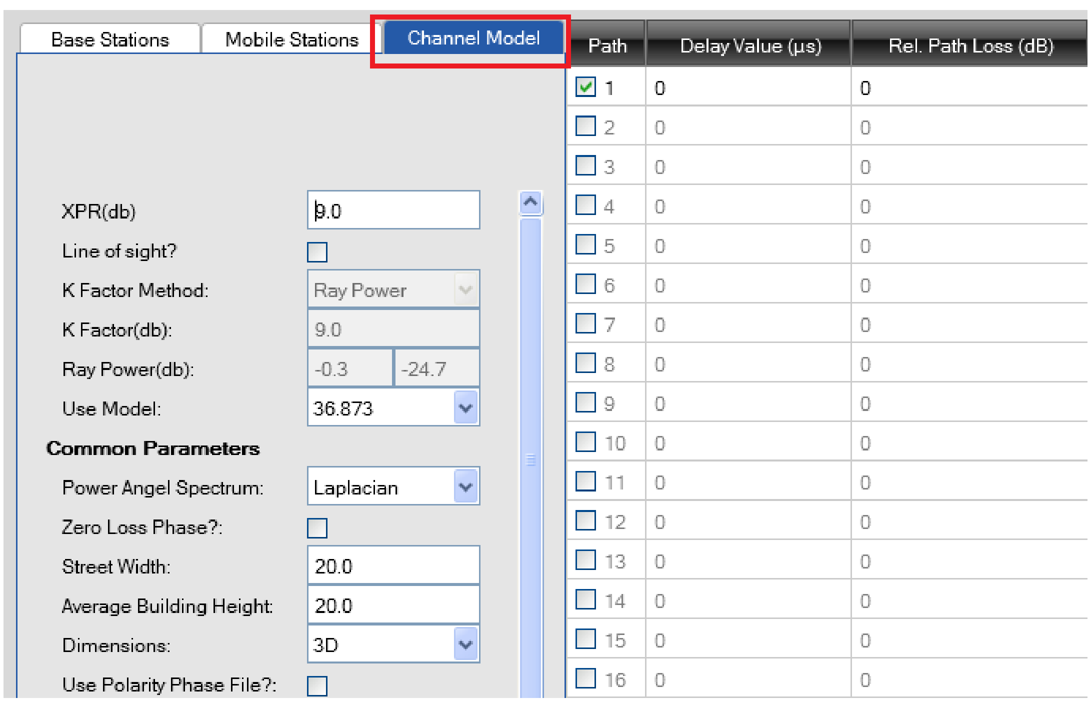


Figure 59. Channel Model tab.

XPR (dB)

Specify the cross polarization ratio, which measures the correlation between the horizontal and vertical elements. Range is 0 to 200 dB. Default is **8** dB.

Line of sight?

Specify whether there is line of sight beam between the base station and the mobile station. Choices are **Yes** and **No**. Default is **No**.

K Factor Method

This setting is used when **Channel Model** is set to **High Speed Train**, **CDL-D**, and **CDL-E**. (There are 3 different methods to define the K factor when a channel is set to LOS.) Choices are:

- **Path 1 Only**: The K factor is equal to the power in LOS part divided by the power in the NLOS part of the first path only.
- **All Paths**: The K factor is equal to the power in the LOS part in the first path divided by the power in the NLOS part of all paths.
- **Ray Power**: WINNER and WINNER II style.

Default is All Paths.

K Factor (dB), Overall (dB)/Path 1 (dB)

This setting is used when **K-Factor Method** is set to **Path 1 Only** or **All Paths**.

Ray Power (dB)

This setting is used when **K-Factor Method** is set to **Ray Power**.

Use Model

Specify whether the geometric model is based on 3GPP recommendation 36.873 or 38.901. Choices are **36.873** and **38.901**. Default is **36.873**.

Common Parameters

You can modify these parameters when **Channel Model** is set to **Custom**. These settings are only available when **Channel Model** is set to **Custom**.

Power Angle Spectrum

Statistical distribution of angle spread. Setting is **Laplacian**.

Zero LOS Phase?

Choices are **Yes** and **No**. Default is **Yes**.

Street Width

Specify the street width that the mobile is traversing in meters. Only applicable when **36.873** is selected. Range is 5 to 50. Default is **20**.

Average Building Height

Specify the average building height in the vicinity of the mobile in meters. Only applicable when **36.873** is selected. Range is 5 to 50. Default is **20**.

Dimensions

Specify whether the geometrical model is 2D (ignoring heights) or 3D. Choices are **2D** and **3D**. Default is **3D**.

Paths and Midpaths

Use Polarity Phase File?

Choices are Yes and **No**. Default is **No**.

Polarity Phase Filename

Select the text file that contains the polarity phase. This parameter is available when **Use Polarity Phase File?** is set to **Yes**. Default is **Pol_phase_table_20_sines_6_path.txt**.

Subpath Assign Type

Choices are **Rand**, **Read File**, and **Linear**.

Subpath Assign Filename

Select the text file that contains subpath assign. This parameter is available when **Subpath Assign Type** is set to **Read File**. Default is **Subpath_assignment_20_sines_6_path.txt**.

Use Subpath Angles File?

Choices are **Yes** and **No**. Default is **No**.

Subpath Angles Filename

Select the text file that contains subpath angles. This parameter is available when **Use Subpath Angles File?** is set to **Yes**. Default is **Subpath_angles_20_sines_6.txt**.

Use Midpath Map File?

Choices are **Yes** and **No**. Default is **No**.

Midpath Map Filename

Select the text file that contains the midpath map. This parameter is available when **Use Midpath Map File?** is set to **Yes**. Default is **Midpath_map_20_sines.txt**.

Cluster

These parameters are only available when **Use Model** is set to **38.901**.

Cluster DS (nSec)

Specify the intra-cluster delay spread. This parameter is only available when **Use Model** is set to **38.901**. Range is 0 to 300 nSec. Default is **3.91 nSec**.

Distance 3D (m)

Specify the 3D distance between two points. This parameter is only available when **Use Model** is set to **38.901**. Range is 0 to 300 m. Default is **100 m**.

Delay Spread Scaling?

When set to **Yes**, this setting enables you to set the desired rms delay spread. The resulting delays in the channel model are calculated as the product of the DS Desired parameter and the delays of the channel model. Delay values are entered in ns. For example, to set the resulting DS to 135ns, enter the number **135**. This setting is only available when **Use Model** is set to **38.901**.

DS Desired (nSec)

Specify the desired channel delay spread. This parameter is only available when **Use Model** is set to **38.901**. Range is 0 to 300 nSec. Default is **100 nSec**.

Scaling Factors

These parameters are only available when **Use Model** is set to **38.901**.

ASA Desired (deg)

Specify the desired angle of arrival spread. This parameter is only available when **Use Model** is set to **38.901**. Range is -1 to 180 degrees. Default is **-1 degree**.

ASD Desired (deg)

Specify the desired angle of departure spread. This parameter is only available when **Use Model** is set to **38.901**. Range is -1 to 180 degrees. Default is **-1 degree**.

ZSA Desired (deg)

Specify the desired elevation angle of arrival spread. This parameter is only available when **Use Model** is set to **38.901**. Range is -1 to 180 degrees. Default is **-1 degree**.

ZSD Desired (deg)

Specify the desired elevation angle of departure spread. This parameter is only available when **Use Model** is set to **38.901**. Range is -1 to 180 degrees. Default is **-1 degree**.

AoA Offset (deg)

Specify the angle of arrival offset. This parameter is only available when **Use Model** is set to **38.901**. Range is -180 to 180 degrees. Default is **0 degree**.

AoD Offset (deg)

Specify the angle of departure offset. This parameter is only available when **Use Model** is set to **38.901**. Range is -180 to 180 degrees. Default is **0 degree**.

ZoA Offset (deg)

Specify the elevation angle of arrival offset. This parameter is only available when **Use Model** is set to **38.901**. Range is -180 to 180 degrees. Default is **0 degree**.

ZoD Offset (deg)

Specify the elevation angle of departure offset. This parameter is only available when **Use Model** is set to **38.901**. Range is -180 to 180 degrees. Default is **0 degree**.

Channel/path view Window

The Channel/path view window enables you to edit the model information.

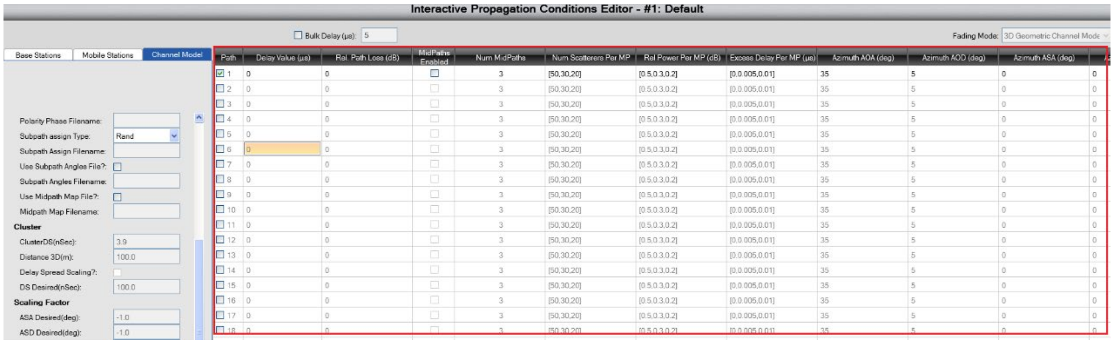


Figure 60. Channel/path view window.

Path ID

Displays the ID of the path.

Enabled?

Allows you to enable or disable the associated path. When the path is enabled, it is displayed in the polar graphs at the bottom of the tab.

Delay (ns)

Specify the delay for the associated path. Range is 0 to 100000 ns. Default is 0 ns.

Relative Path Loss (dB)

Specify the relative path loss for the associated path. Range is 0 to 32 dB. Default is **0 dB**.

Midpaths Enabled?

Allows you to enable or disable the midpaths for the associated path.

Azimuth AoA (deg)

Specify the azimuth (rotation) angle arriving at the mobile station for the associated path. Range is -180 to 180 degrees. Default is **0.7 degrees**. **0** indicates same as line of sight.

Azimuth AoD (deg)

Specify the azimuth (rotation) angle departing the base station for the associated path. Range is -180 to 180 degrees. Default is **6.6 degrees**. **0** indicates same as line of sight.

Azimuth Angle Spread Arrival (deg)

Specify the azimuth angle spread arrival for the associated path. Range is 1 to 75 degrees. Default is **35 degrees**.

Azimuth Angle Spread Departure (deg)

Specify the azimuth angle spread departure for the associated path. Range is 1 to 75 degrees. Default is **5 degrees**.

Zenith AoA (deg)

Specify the zenith (elevation) angle arriving at the mobile station for the associated path. Range is 0 to 180 degrees. Default is **90 degrees**. **90** indicates same as line of sight.

Zenith AoD (deg)

Specify the zenith (elevation) angle arriving at the base station for the associated path. Range is 0 to 180 degrees. Default is **90 degrees**. **90** indicates same as line of sight.

Zenith Angle Spread Arrival (deg)

Specify the zenith angle spread arrival for the associated path. Range is 0 to 75 degrees. Default is **0 degree**.

Zenith Angle Spread Departure (deg)

Specify the zenith angle spread departure for the associated path. Range is 0 to 75 degrees. Default is **0 degree**.

After you finished editing, you can save the model file into the propagation library using the **Save As** button as shown in the following figure.

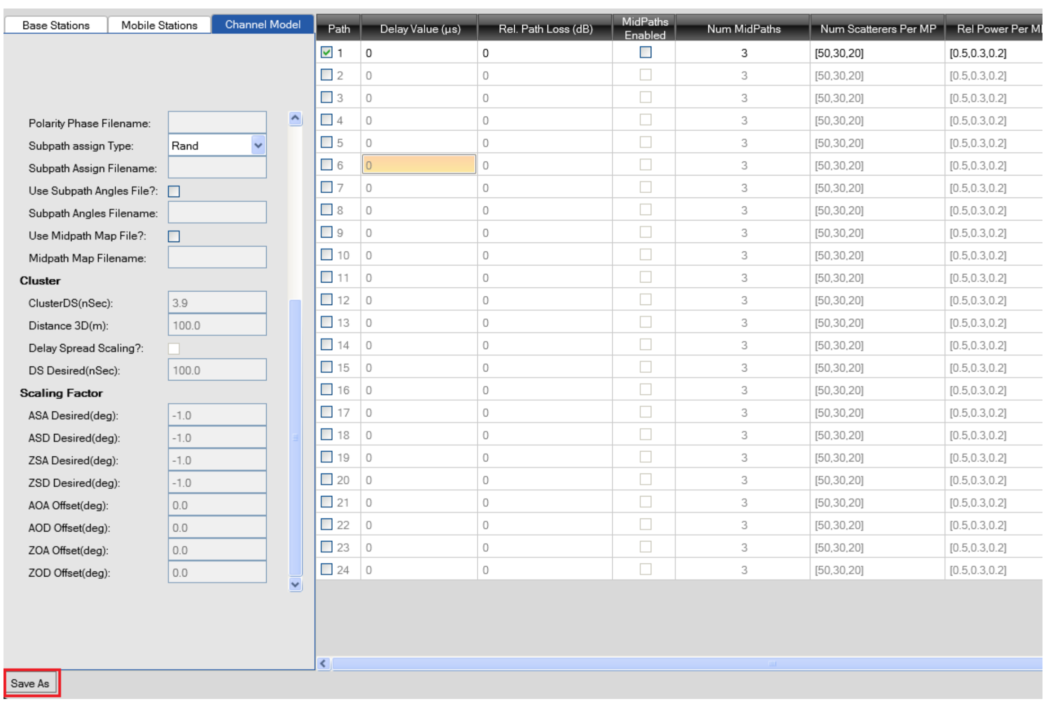


Figure 61. Location of the Save As button in the Channel/path view window.

2.7.2.11 Channel Bypass

While the Vertex instrument is in the "Playing" state, you can bypass the MIMO channel propagation conditions and configure them by selecting the **Bypass** option, as shown in the following figure.

Figure 62. Selecting the Bypass option.

To configure the Bypass function, click outside the **Bypass** check box in a row of the Channel Model to display the **More** button, as shown in the following figure.

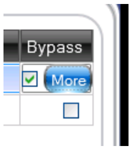


Figure 63. More button.

The Bypass Configuration window appears, as shown in the following figure.

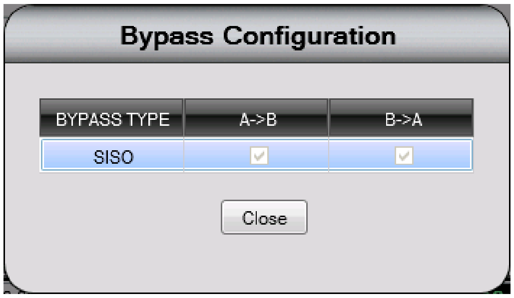


Figure 64. Bypass Configuration options.

Bypass Type

When the channel is bypassed, the behavior of the Radio Links is specified by the "Bypass Type":

- **SISO (default)**: All cross links (hij, i!=j) are disabled, and the channel behaves as a clean SISO channel.
- **MIMO-BUTLER**: The channel has the same properties as in the "Stopped" state, with Butler matrix phase relationships between the radio links. (See section 2.5.2.1 for more information on the phases applied).

- **A->B, B->A**: Enabling bypass in a particular direction (A->B or B->A) is primarily intended for TDD scenarios where the channel model spans both the A->B and B->A directions. However, you may not want to bypass in both directions simultaneously.

  For example:
  Connection Setup - 2x2 BiDirectional (1 Channel Model)
  Deselecting **A->B** allows the configured propagation conditions to be active while emulation is in the “Playing” state, as shown in the following figure.

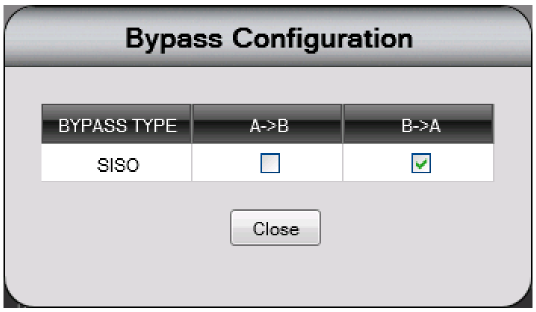


Figure 65. Bypass Configuration – B->A.

This configuration is indicated in the GUI by the color of the directional arrows next to the Bypass option, as shown in the following figure.

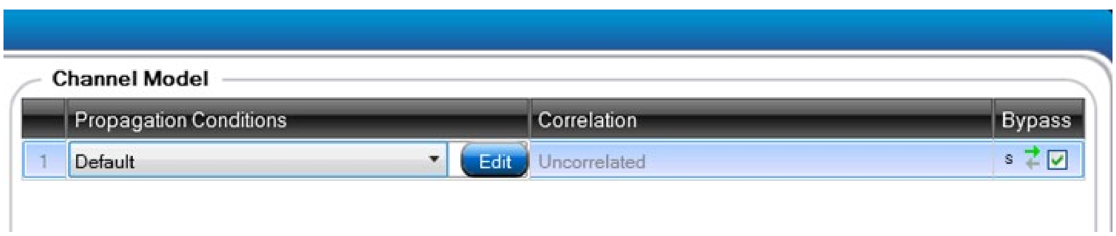


Figure 66. Bypass direction indicator.

The arrow pointing in the **A->B** direction is GREEN, indicating that Propagation conditions are active in that direction. The arrow pointing in the **B->A** direction is GRAY, indicating that Propagation conditions are bypassed in this direction.

2.8. Working with Libraries

The Library is a repository of both user-created and pre-installed standardized configuration files. It allows you to view, edit, and create configuration files.

> **NOTE:**
> You cannot select a configuration file to apply to the Vertex instrument through the Library. You can only view, edit, and create new files.

There are three sections in the Library:

1. Connection Setup
2. Propagation Conditions
3. Correlation (applicable only to Classical Channel Models)

For Propagation Conditions and Correlation, you can perform the following actions:

- Create your own files from scratch and save to the Library under your own folder.
- Edit standardized files available pre-installed on the Vertex and save to the Library under your own folder.

These files can be loaded for use on the Vertex from the Main View under **Configuration**.

To access the Library:

In the Vertex GUI, click the **Library** button, as shown in the following figure.

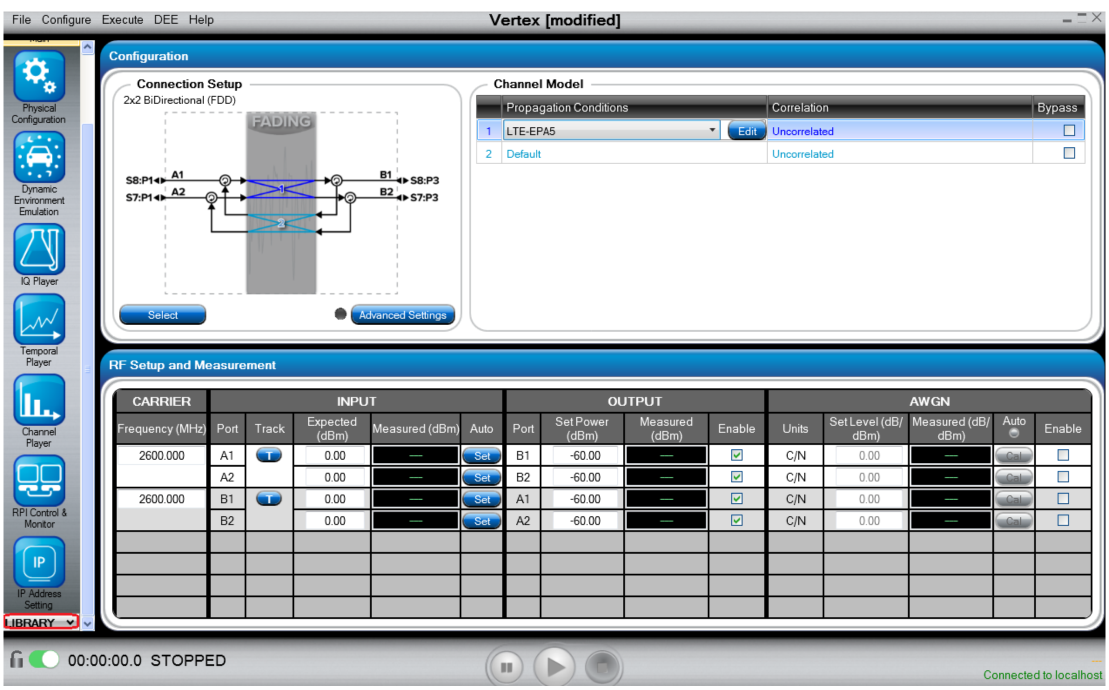


Figure 67. Library button.

The following sections appear in the library:

- Propagation Conditions
- Correlation
- Connection Setup

These sections are explained in detail in the following sections.

2.8.1. Propagation Conditions

This section explains how to use the Propagation Conditions section of the Library to view existing standardized/user created files and to create your own file.

Click **Propagation Conditions**.

The Propagation Conditions Library appears.

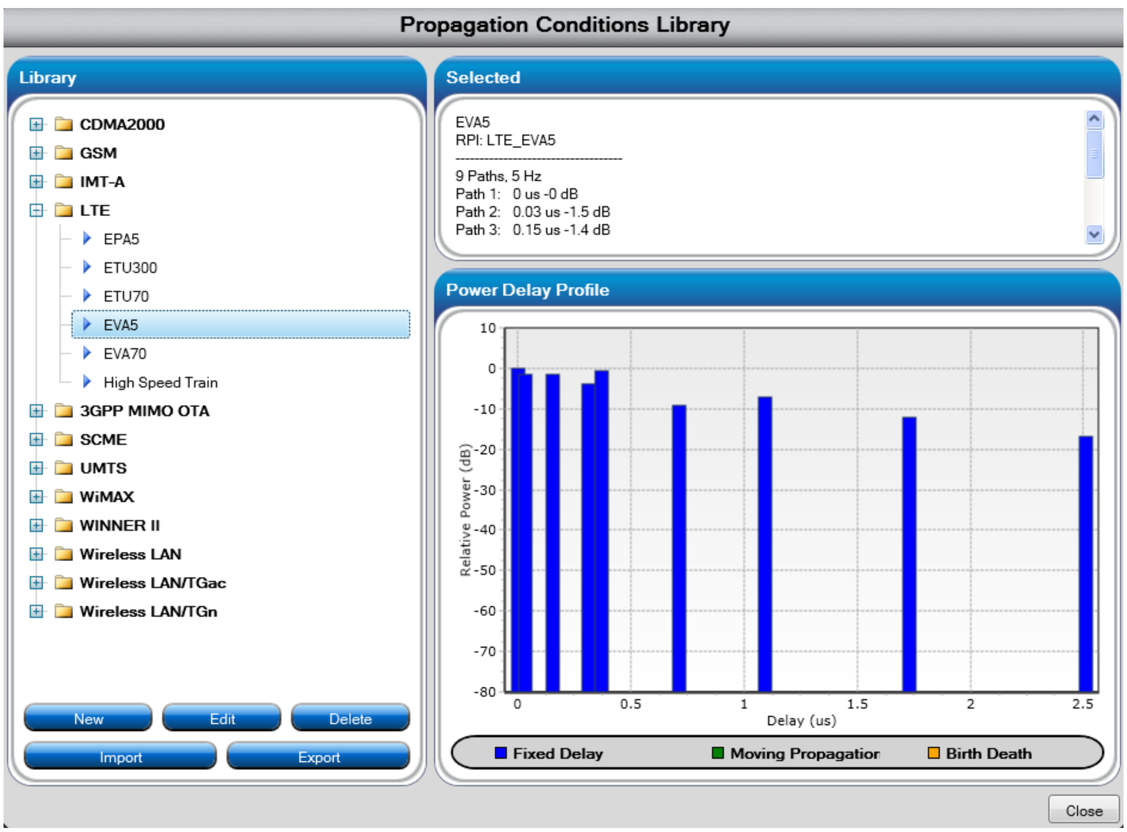


Figure 68. Propagation Conditions Library.

On the left of the window is a collapsible list of propagation conditions classified by industry standards/technology. If you select a particular propagation condition, the right side of the window provides an overview of the chosen conditions along with a visual display of its power delay profile.

NOTE:
Any change you make to the files while in the Library view will not be reflected on the Vertex hardware. The Library is strictly for viewing, editing, and creating new files.

2.8.1.1 Creating a New Propagation Conditions File

Perform the following steps:

1. Click the **New** button.

   The Propagation Conditions Editor displays.

2. Create your desired propagation conditions, and click the **Save** button.
3. Enter a name and description for the propagation conditions, and click the **OK** button.

   The new propagation conditions are saved in the Library under the **User Created** folder.

2.8.1.2 Editing a Standardized Propagation Conditions File

Perform the following steps:

1. Select the propagation conditions, and click the **Edit** button.

   The Propagation Conditions Editor displays.

2. Make your changes to the propagation conditions, and click the **Save** button.
3. If necessary, enter a name and description for the edited propagation conditions, and click the **OK** button.

   The edited propagation conditions are saved in the Library under the **User Created** folder.

2.8.2. Correlation

This section explains how to use the Correlation section of the Library to view existing standardized/user created correlation matrices and to create your own matrices.

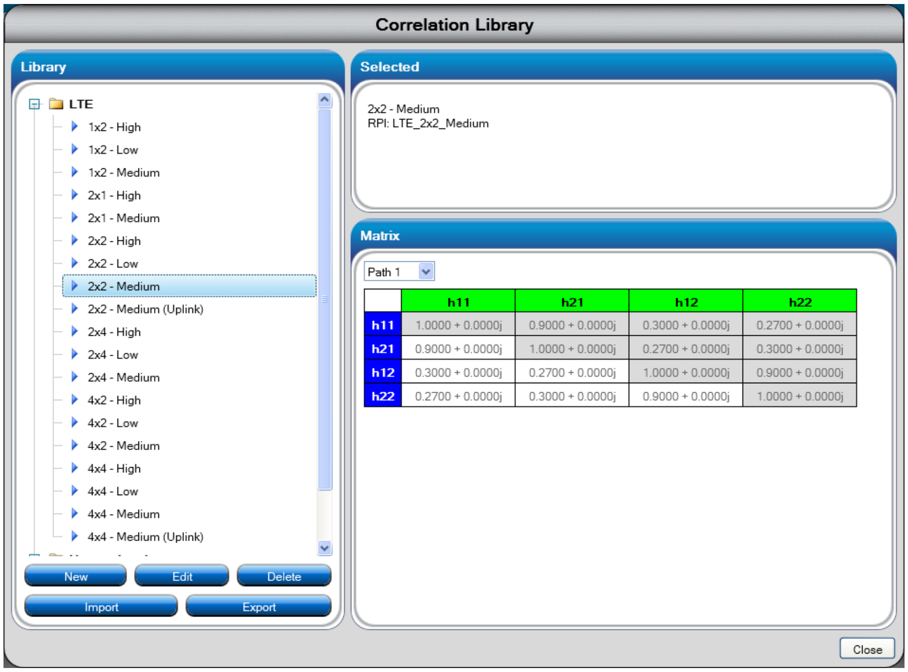


Figure 69. Correlation Library window.

On the left of the window is a collapsible list of correlation matrices classified by industry standards/technology. If you select a particular correlation, the right side of the window displays the corresponding correlation matrix.

2.8.2.1 Creating a New Correlation File

Perform the following steps:

1. Click the **New** button.

   The Correlation Editor displays.

2. Under the “Properties” section of the editor, select the number of transmit and receive antennas in the setup.

   The size of the correlation matrix in the “Matrix” section is updated according to the number of Tx and Rx antennas chosen.

3. Create the desired correlation, and click the **Save** button.
4. Enter a name and description for the Correlation file, and click the **OK** button.

   The new correlation is saved in the Library under the **User Created** folder.

2.8.2.2 Editing a Standardized Correlation File

Perform the following steps:

1. Select the standardized correlation, and click the **Edit** button.

   The Correlation Editor displays.

2. Make changes to the correlation matrix, and click the **Save** button.
3. If necessary, enter a name and description to the correlation, and click the **OK** button.

   The correlation is saved in the Library under the **User Created** folder.

2.8.3. Transferring Library Files

All files created in the Library reside on the Vertex hardware. To transfer files from one Vertex instrument to another, use the **Import**/**Export** buttons provided on Library window.

This applies to library files for Propagation Conditions and Correlation.

Perform the following steps:

1. Connect your controller laptop running the Vertex GUI to the Vertex unit with the configuration file you want to transfer.
2. From the Vertex GUI:
   a. Click the **Export** button.
   b. Navigate to the location on the hard drive of the laptop in which you want to save the file.
   c. Name the file, and click the **Save** button.
3. Connect your controller laptop to the Vertex unit to which you want to transfer the library file.
4. From the Vertex GUI:
   a. Click the **Import** button.
   b. Navigate to the location where you saved the library file.
   c. Select the file, and click the **Open** button.

      The file is now part of your library

NOTE:
The name of the file is for the purpose of the Export/Import operation. When the file is imported into your Library, it is included in the same folder with the same name as in the previous Library. For example, if you export propagation conditions named **EVA5_test** from the **User Created** folder into a file named **file_EVA5_test** and import this file on a new Vertex instrument, it will be found under the **User Created** folder with the name **EVA5_test**.

The pre-defined propagation and correlation models cannot be exported. It is unnecessary to export these models because every Vertex has the models in the Library.

2.9. RF Setup and Measurement

Configure the RF settings for your test through the RF Setup and Measurement section in the Vertex GUI Main view. This section, shown in the following figure, allows you to view and set RF input, output levels, carrier frequencies, and AWGN settings.

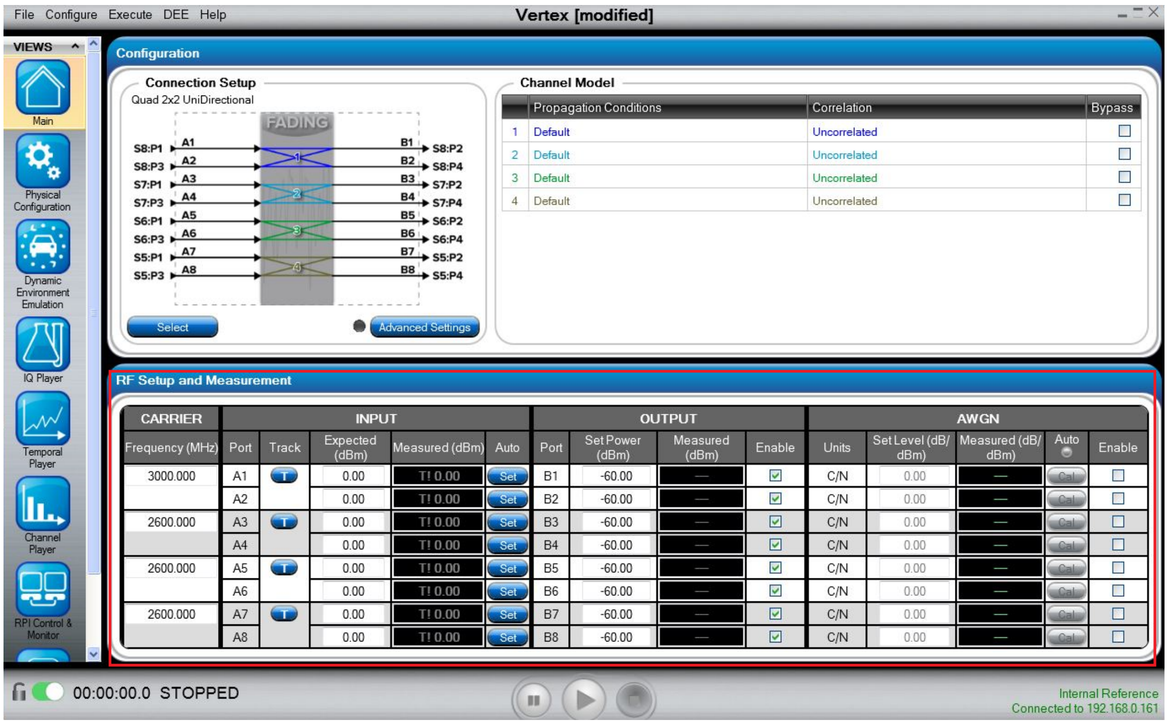


Figure 70. Vertex GUI – RF Setup and Measurement.

2.9.1. Carrier Frequency

You can configure a carrier frequency for each unique transmission frequency in the current setup. This number is equal to the number of channel models indicated on the Connection Setup Diagram. A sample diagram is shown in the following figure.

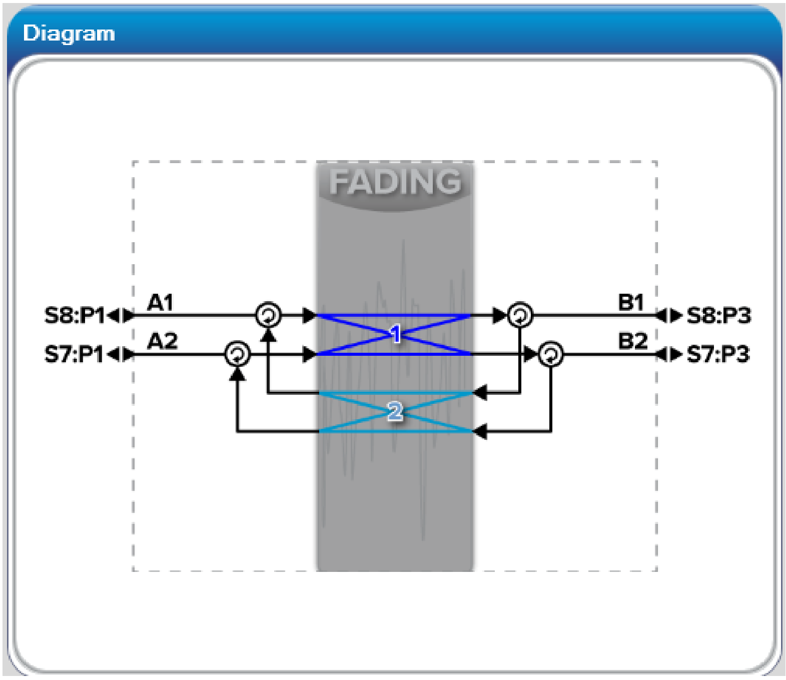


Figure 71. Connection Setup diagram.

For example, if the current connection setup is **MIMO 2x2 BI_FDD**, you are required to set two carrier frequencies: one for the Downlink and one for the Uplink. You can configure the carrier frequency based on the RF input port as shown in the following figure.

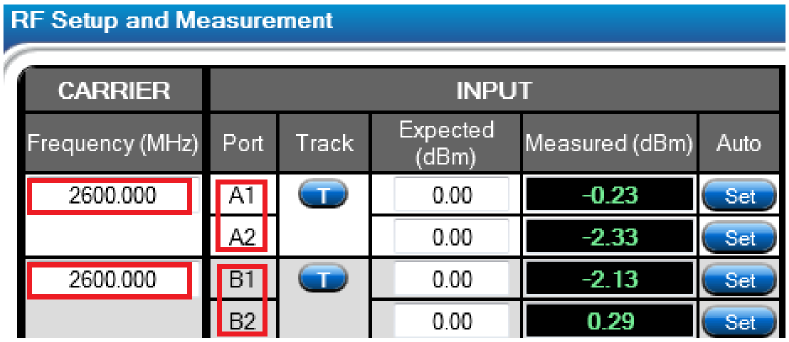


Figure 72. Carrier frequency based on the RF input port.

NOTE:
With the 6GHz RF module, Vertex can support from 30MHz to 5925MHz. This means that the lowest frequency to pass through Vertex is 30MHz, and the highest frequency is 5925MHz. For example, if you want a signal with 20MHz bandwidth, the lowest carrier frequency you can set is 40MHz, and the highest frequency you can set is 5915MHz.

2.9.2. Input

Set the input levels to match the RMS signal power at each input port of the Vertex unit.

2.9.2.1 Port

This is the logical name of the input port. The logical port name corresponds to a physical port as shown in the Connection Setup diagram in the Main View.

2.9.2.2 Input Tracking

In testing mobile devices, the input to the Vertex channel emulator often varies based on the channel characteristics configured through the Vertex. Estimating or adjusting the input power while testing can be challenging, so Vertex includes Input Tracking. Now, input can be tracked to ensure that the set and measured input levels of a port are consistent. This ensures the accuracy of the output power and SNR for the respective ports, as well as maximizes EVM.

There are two Input Tracking mechanisms: Automatic and Predicted. These are explained in detail in the following sections.

Automatic Input Tracking

When a port has Automatic Input Tracking enabled, the Vertex instrument measures the input power and updates the Expected Input Power accordingly. The tracking cannot respond to fast abrupt changes in input power level, but only gradual trends over time.

Three settings allow you to customize input tracking: Resolution, Minimum Period, and Maximum Step Size:

1. Resolution (dB): The range over which the input level will not be adjusted. For example, a resolution of 2 dB means that the input power will not be adjusted unless it is measured to be +/- 2 dB from the current setting.
2. Minimum Period (ms): The minimum time between power measurements.
3. Maximum Step Size (dB): The biggest step by which the input power can be changed to match the measured value.

> **NOTE:**
> Automatic Input Tracking is only supported in DEE while in Power Level Output mode.

Automatic Input Tracking in DEE

Automatic Input Tracking is allowed in DEE under certain circumstances/caveats:

- Output Mode is set to Power Level Mode.
- The speed of the input tracking is gated by the state duration. Because DEE is a time synchronous engine, the input level changes must be registered to state boundaries. This introduces a latency of one additional state transition for the input powers to be applied. Therefore, it is recommended to use the minimum state duration for testing.
- Input Tracking must be set to **Allowed** before DEE is enabled. However, while DEE is playing, you can change the input levels and input tracking settings “on the fly.”
- After DEE is enabled, you can only apply inputs to the unit while DEE is playing. If input levels are adjusted before emulation has started, the input measurements and calculated output measurements displays are inaccurate or frozen until emulation is started.

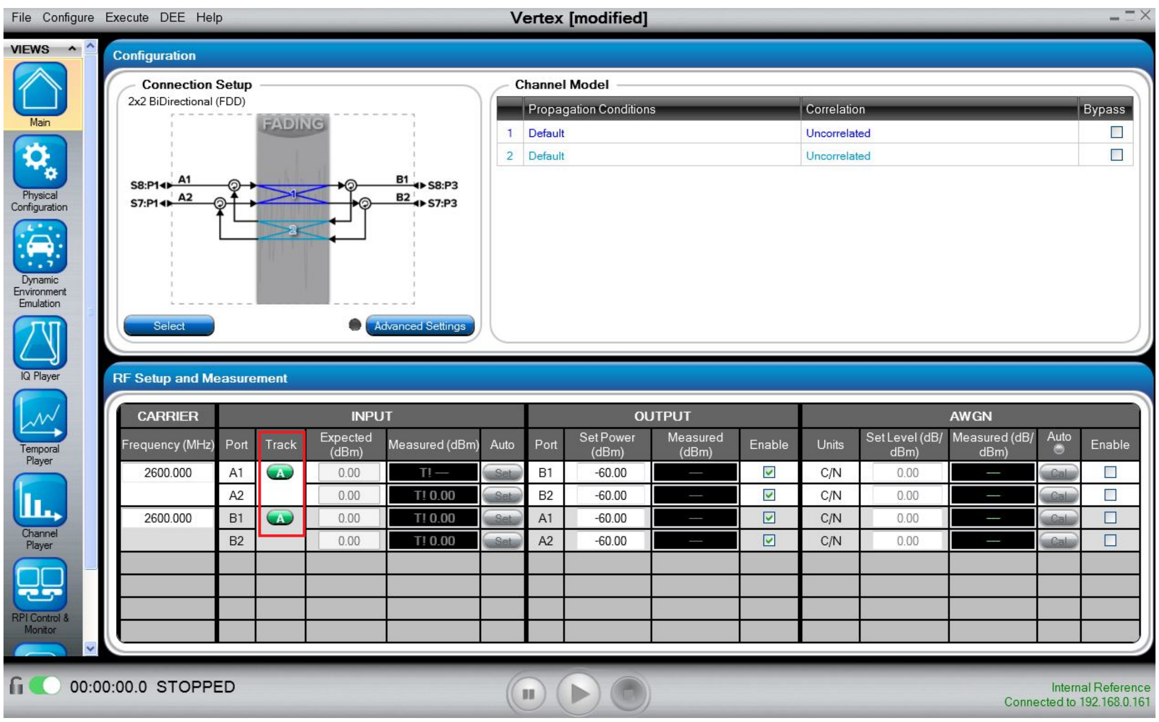


Figure 73. DEE View – Input Tracking configuration before DEE Enable.

When Input Tracking is set to **Allowed** prior to enabling DEE, you can enable Automatic Input Tracking after enabling DEE. Enabling DEE disables Automatic Input Tracking on all ports, but it can be enabled after DEE is enabled.

When you change the input power while DEE is enabled, by explicitly setting the set power or by turning on Automatic Input Tracking, any Absolute Trigger on the affected ports gets changed to a Relative Trigger. We recommend setting such input ports to Relative Trigger or Continuous Measurement Type before running DEE.

Predicted Input Tracking

For many applications, such as LTE, the transmit power of the UE can be estimated based on the loss in the channel (Open Loop Power Control). In Predicted Input Tracking mode, a port uses the received loss to determine the necessary transmit input power. For example, when Predicted Input Tracking is enabled on the B ports, the Loss associated with the B ports (loss from A to B direction) is used to calculate the input power.

You must enter a table specifying Set Loss values and the corresponding Expected Input Powers. If an Expected Input Power is being determined, and the desired Set Loss value is not present in the table, the Vertex interpolates to obtain the necessary value, or if the Set Loss is outside the minimum/maximum table values, uses the Expected Input Power corresponding to the nearest Set Loss.

NOTE:
Predicted Input Tracking is only available for bidirectional connection configurations and when the Output Mode is set to Loss. Predicted mode is supported in DEE mode.

The Input Tracking column is shown in the following figure. The same Input Tracking setting is applied to all input ports that belong to the same MIMO connection.

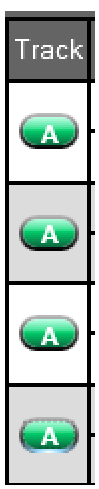


Figure 74. Input Tracking selected for Share column.

To configure Input Tracking:

1. Select the button for the desired port(s).

   The Input Tracking Settings window appears, as shown in the following figure.

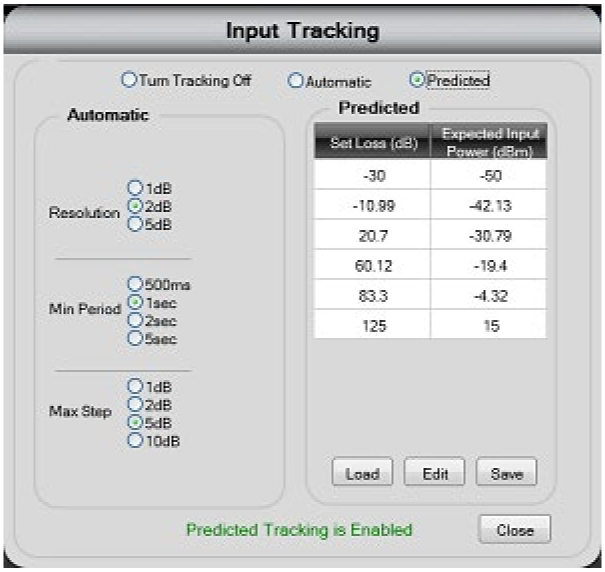
   

   Figure 75. Input Tracking window.

2. Select the appropriate option button to set the Tracking mode to **Off**, **Automatic**, or **Predicted**.

   You can configure the automatic settings of Resolution, Min Period, and Max Step in this window.

   Because Predicted Input Tracking must use a table to obtain the Expected Input Power values, you must load the table before selecting the Predicted option. If a table is configured, it displays in this window. You can click the **Load** button to load a previously created table, the **Edit** button to edit the configured table, or the **Save** button to save the table.

   The Predicted Input Tracking window displays, as shown in the following figure. From this window, you can add and delete rows, apply the new values, or cancel the table changes. The range of Expected Input Power is -50.00 to 15.00 dBm. The maximum range of Set Loss is -30.00 to 125.00 dB, but it is limited by the value of the Expected Input Power.

Figure 76. Predicted Input Tracking table window.

2.9.2.3 Expected Input

To achieve the best performance from the Vertex channel emulator, you must properly configure the input powers at the Vertex RF ports to match the RMS power of the input signals. This can be done manually, through the Autoset function, or with Input Tracking.

For bursty signals, triggered mode input power measurements must be enabled through **Configure>Power Meter Settings**.

2.9.2.4 Measured

This is the measured input power on the port. Averaging and triggering for this measurement can be set in the **Configure>Power Meter Settings** menu. (Refer to Section 2.10 "Power Meter Settings.")

2.9.2.5 Autoset

Autoset optimally adjusts the RF front end of the port based on power levels entering the port at that time. It ensures accurate power levels at the output of the instrument and optimizes dynamic range. It may not be appropriate for bursty signals or signals that vary widely in average power (for example, UE Transmitters).

In each port, Autoset is disabled if Input Tracking is enabled. When the Input Tracking function is enabled, the **Autoset** buttons are disabled, as shown in the following figure.

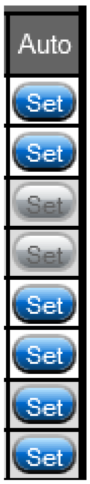


Figure 77. Autoset with Input Tracking enabled.

2.9.3. Output

2.9.3.1 Port

This is the logical name of the output port. The logical port name corresponds to a physical port as shown in the Connection Setup diagram in the Main View.

2.9.3.2 Set Power

Sets the desired output power level for this port.

In order for the output power level to be accurate, the Expected Input power field should be correctly set manually or through the Autoset function.

For bursty signals, triggered mode input power measurements must be enabled in the **Configure** menu.

2.9.3.3 Measured

This is the measured output power on the port. You can set the averaging and triggering for this measurement by selecting **Configure>Power Meter Settings**. Refer to Section 2.10 "Power Meter Settings" for more information.

2.9.3.4 Enable

Enables and disables the signal on the output port. A checked box indicates that the output port is enabled.

2.9.4. AWGN

You can view and set the AWGN settings for each RF output port using the columns under AWGN.

2.9.4.1 Units

Available AWGN units are C/N, C/N0, and Eb/N0. Refer to Section 2.9.4.10 for more information.

2.9.4.2 Set Level

To set the ratio of the signal to noise, specify the value in dB.

2.9.4.3 Measured

This is the measured signal-to-noise ratio of the signal on the output port. This measurement is derived from the measured output level and measured noise level on the output port. The measurement bandwidth and units can be adjusted in the **Configure** menu.

If this value is different from the Set Ratio, it is likely because the Expected Input level does not match the Measured Input level. This in turn results in the carrier output power being inaccurate, and hence the measured signal to noise ratio does not match the Set Ratio. This can be solved by clicking **Autoset** for the particular RF port.

2.9.4.4 Autoset/Calibrate AWGN

Autoset AWGN adjusts the noise level to account for any drift in input power. When the actual input power drifts from the set input power, the actual output power drifts from the set output power. Consequently, the actual AWGN can drift from the set AWGN (C/N and Eb/No drift because they are ratios dependent on the output power, but Noise does not drift). Executing Autoset AWGN on an output port adjusts the AWGN so that the actual value matches the set value. The Autoset AWGN column is shown in the following figure.

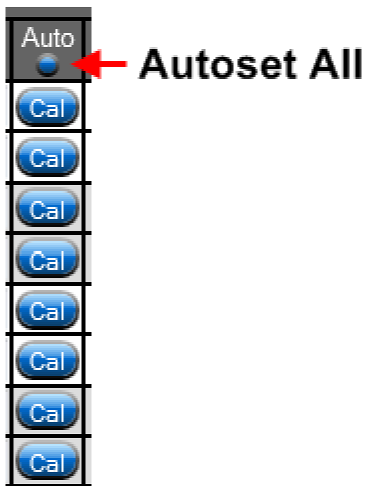


Figure 78. Autoset AWGN.

Autoset All will attempt an autoset AWGN operation on all AWGN-enabled ports. The GUI **Autoset All** button is enabled as soon as one port has AWGN enabled. A pop-up indicates when Autoset AWGN or Autoset All completes and then automatically closes after a few seconds.

The range of the autoset varies depending on the current setting for Output Power and AWGN.

The AWGN compensation is cleared in the following circumstances:

- Connection setup is changed.
- Fading mode is changed.
- DEE is enabled.
- Set Input Power is changed.
- AWGN is disabled.
- AWGN units are changed.
- Antenna pattern state is changed.
- Base Station antenna configuration is changed.
- Mobile Station antenna configuration is changed.

2.9.4.5 Enable

The Enable option enables or disables AWGN for a given output port.

2.9.4.6 AWGN Settings

To configure other AWGN interferer properties, such as measurement bandwidth and units, select **Configure>AWGN Settings**. The AWGN Settings window appears, as shown in the following figure.

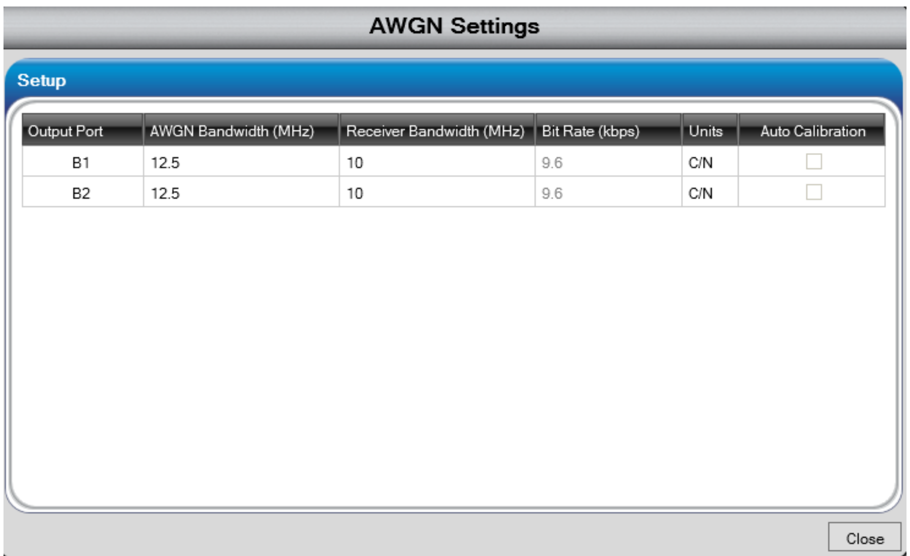


Figure 79. AWGN Setup window.

2.9.4.7 Out Port

This is the logical name of the output port. The logical port name corresponds to a physical port as shown in the Connection Setup diagram in the Main View.

2.9.4.8 AWGN Bandwidth

To set the bandwidth of the AWGN noise for a particular output port, enter the value in MHz.

NOTE:
The AWGN bandwidth must be greater than or equal to the Receiver Bandwidth.

2.9.4.9 Receiver Bandwidth

Receiver Bandwidth is the bandwidth over which the total AWGN power is equal to the value as required by set signal to noise ratio.

Example:

For the following AWGN settings, set output power = -40 dBm, C/N = 10 dB, Receiver Bandwidth = 20 MHz, AWGN Bandwidth = 25 MHz.

This implies AWGN power required = N = -30 dBm. The AWGN power over 20 MHz is-30 dBm. However, the AWGN extends over the entire specified 25 MHz.

2.9.4.10 Units

There are three different units to configure the signal to noise ratio:

1. Carrier to Noise (C/N)

   The power of the band-limited noise is set as a ratio of carrier power to noise in the bandwidth of the receiver.

2. Carrier Bit Power/Noise Power Spectral Density (E_b/N_o)

   The power of the band-limited noise is set as a ratio of carrier bit energy to noise power spectral density.

3. Noise Level (N)

   The power of the band-limited noise in the receiver bandwidth in dBm. With a selection of N, C/N ratio changes as output power is adjusted, but the absolute Noise power remains consistent.

Use the drop-down menu to specify the units you would like to use.

Relationship between C/N and E_b/N_o:

The carrier bit power and noise power in dBm can be calculated based on the following formula:

```
Eb (dBm/bps) = C (dBm) - 10*log10(Bit Rate (bps))

where:

Eb = Bit power in dBm/bps

C = carrier power in dBm

Bit Rate = bit rate of the carrier

and

N (dBm) = No + 10 log10 (Receiver Bandwidth)

where:

N = noise power in the receiver bandwidth in dBm

No = noise power spectral density in dBm/Hz

Receiver Bandwidth = carrier bandwidth in Hz
```

2.9.4.11 Bit Rate

Specify the bit rate of the carrier in kbps. This is applicable only when the units chosen for the port is Eb/No.

2.9.4.12 AWGN Auto Calibration

If the **Auto Calibration** check box is checked, Vertex will automatically calibrate the AWGN to C/N setting level based on measured power at the output port. It only works when AWGN is enabled at the port.

2.10. Power Meter Settings

Access the Power Meter Settings by selecting **Configure>Power Meter Settings**. This window allows you to:

- Set Continuous or Triggered Input Level Measurements
- Set Relative or Absolute Trigger Threshold
- Set Measured or Calculated Output Level Measurements
- Set Averaging on Input or Output Measurements

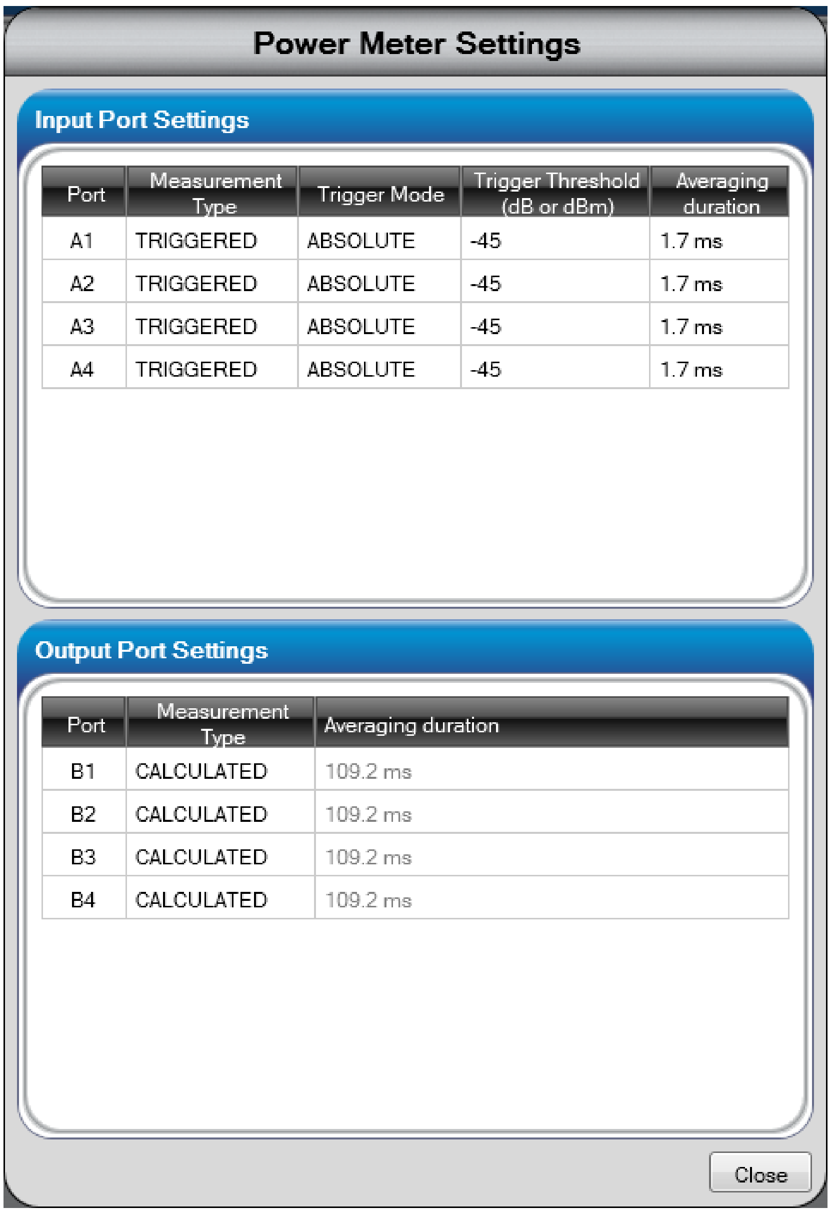


Figure 80. Power Meter Settings window.

2.10.1. Input Port Settings

Measurement Type is Continuous or Triggered. When the Measurement Type is set to Continuous, the Power Meter constantly triggers and measures the signal.

When the Measurement Type is set to Triggered, the Power Meter measures the signal when the input power is detected over the Trigger Threshold.

Trigger Mode can be set to Absolute or Relative. When the Trigger Mode is set to Absolute, the Trigger Threshold is taken as an absolute value. When the Trigger Mode is set to Relative, the Trigger Threshold is taken relative to the Set Input Level.

Averaging Duration sets the time window for averaging. Averaging Duration on the Input Port can be set from 0.2 microseconds to 6.9 seconds.

2.10.2. Output Port Settings

Measurement Type can be set to Calculated or Measured. When the Measurement Type is set to Calculated, the Output Level is calculated as a function of Set and Measured Input Level and loss through the channel. When the Measurement Type is set to Measured, the actual power measured by the power meter is given.

Averaging Duration sets the time window for averaging. On the Output Port, averaging is only available when the Measurement Type is set to Measured. Averaging Duration on the Output Port can be set from 13.6 milliseconds to 6.9 seconds.

2.11. Advanced Settings

The Advanced Settings window allows you to:

- Set relative powers between radio links connected to the same output port, to model branch imbalance.
- Enable or disable individual radio links.
- Set the phases of individual radio links.
- Apply a preconfigured Static Channel Model defining the phase relationships between the radio links.

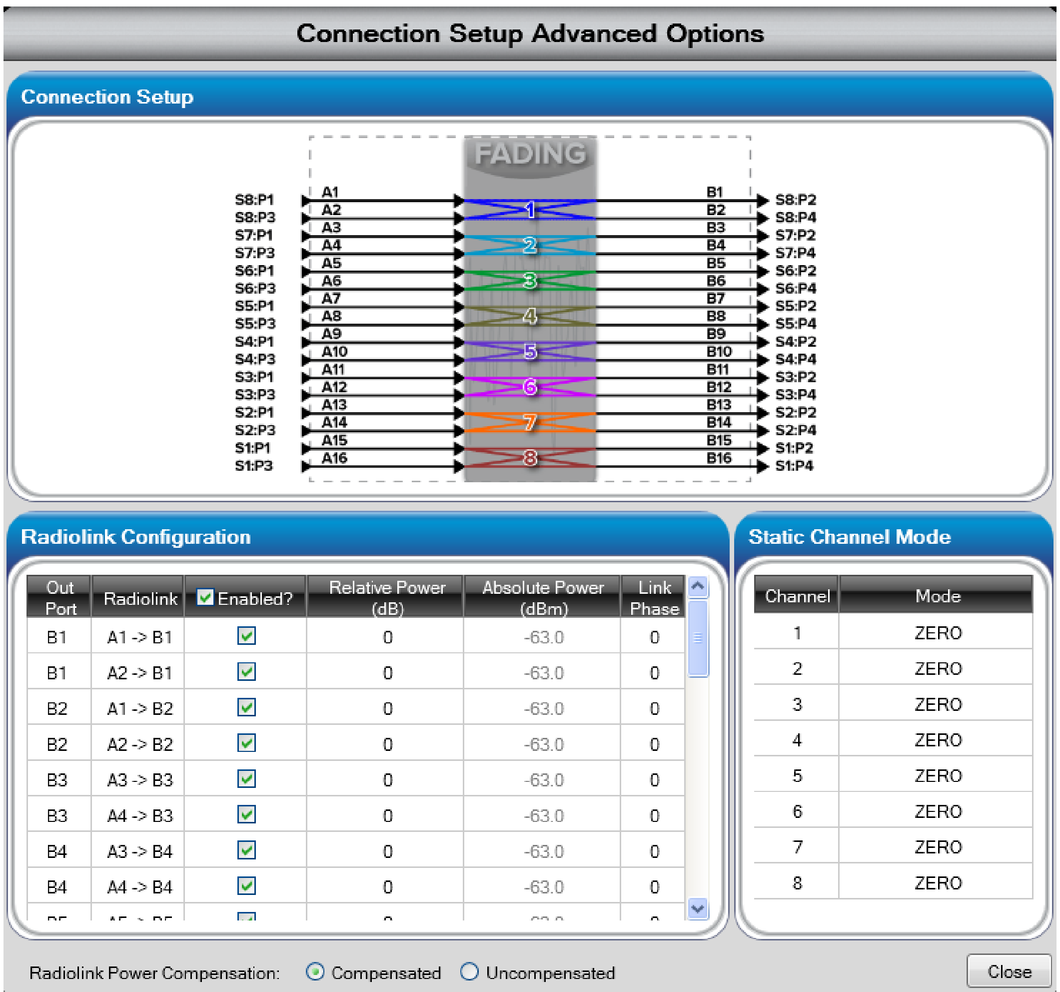
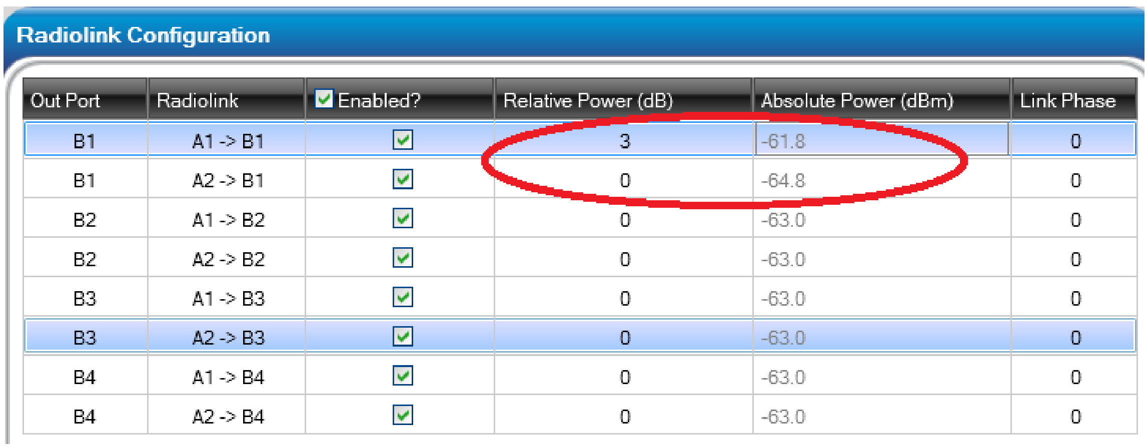


Figure 81. Connection Setup Advanced Options window.

2.11.1. Relative Power Between Radio Links

In the example shown in the preceding figure, if you set the relative power of Radio link A1->B1 to 3 dB, it results in the absolute power of A1->B1 and A2->B1 being adjusted so that the following result occurs:

- There is a 3 dB difference between them as shown in the following figure.
- The total output power of that port matches the set power in the RF Setup and Measurements panel. The total output power of any port is governed solely by the settings in Section 2.8: RF Setup and Measurement.



Out Port	Radiolink	Enabled?	Relative Power (dB)	Absolute Power (dBm)	Link Phase
B1	A1 -> B1	☑	3	-61.8	0
B1	A2 -> B1	☑	0	-64.8	0
B2	A1 -> B2	☑	0	-63.0	0
B2	A2 -> B2	☑	0	-63.0	0
B3	A1 -> B3	☑	0	-63.0	0
B3	A2 -> B3	☑	0	-63.0	0
B4	A1 -> B4	☑	0	-63.0	0
B4	A2 -> B4	☑	0	-63.0	0

Figure 82. Radiolink Configuration.

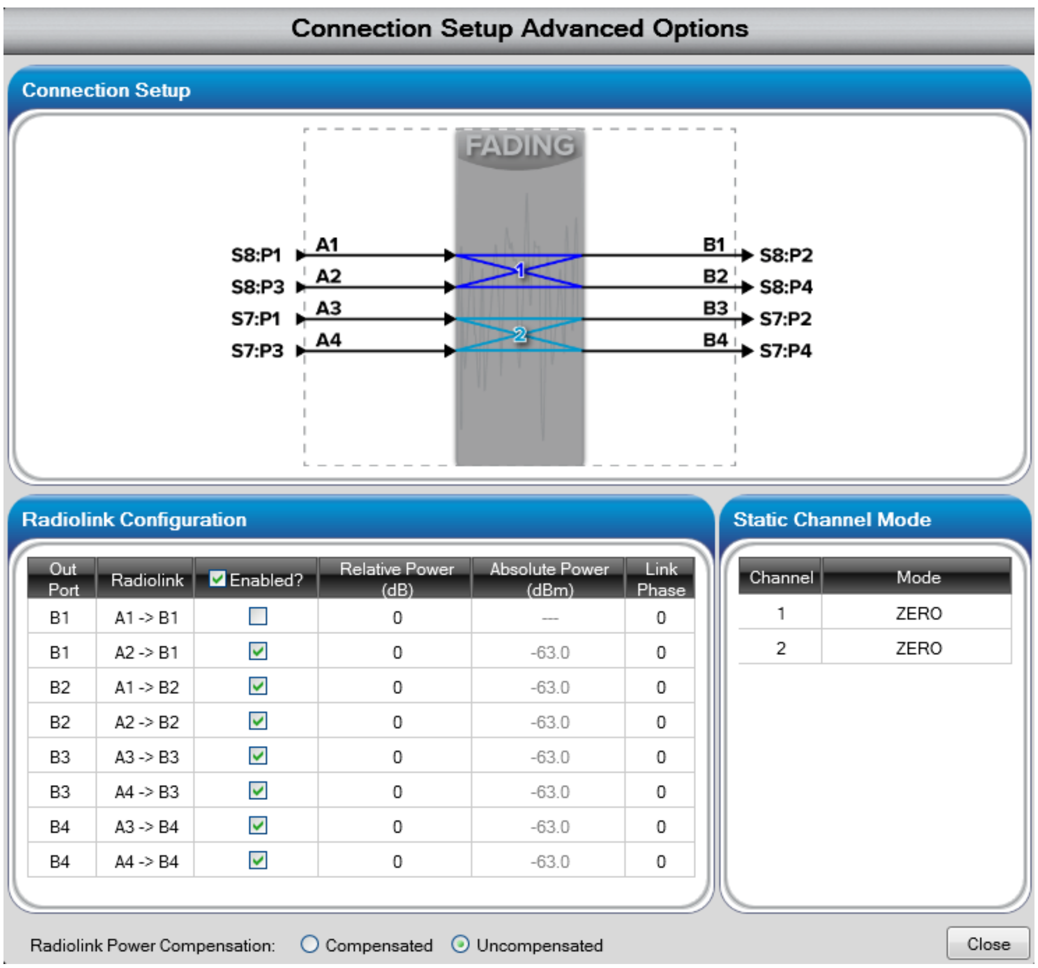
NOTE:
Similar to the rest of the Vertex GUI, any value you enter or change you make in this window takes effect in the Vertex immediately.

2.11.2. Enabling and Disabling Radio Links

When you disable a radio link, that radio link will have zero power, and the powers of the other radio links connected to that output port are adjusted up so that the total power of the port matches the set power in the RF Setup and Measurements panel.

To prevent this renormalization behavior, select the **Uncompensated** option, as shown in the following figure. The compensation options are described below:

- **Compensated**: When a Radio link in the MIMO configuration is disabled, readjust the Radio link powers of the other links to ensure full power is configured at the output port.
- **Uncompensated**: When a Radio link in the MIMO configuration is disabled, do not renormalize, so the output power is not adjusted.



Out Port	Radiolink	Enabled?	Relative Power (dB)	Absolute Power (dBm)	Link Phase
B1	A1 -> B1	☐	0	---	0
B1	A2 -> B1	☑	0	-63.0	0
B2	A1 -> B2	☑	0	-63.0	0
B2	A2 -> B2	☑	0	-63.0	0
B3	A3 -> B3	☑	0	-63.0	0
B3	A4 -> B3	☑	0	-63.0	0
B4	A3 -> B4	☑	0	-63.0	0
B4	A4 -> B4	☑	0	-63.0	0

Channel	Mode
1	ZERO
2	ZERO

Figure 83. Radiolink Configuration – Uncompensated option.

Radio link Power Compensation provides the selection to renormalize the absolute powers of the enabled Radio links when connected Radio links are disabled (Compensated), or to prevent renormalizing the Radio links power (Uncompensated).

2.11.3. Link Phase

Link Phase is an absolute phase in degrees that is applied to each radio link when emulation is playing.

2.11.4. Static Channel Mode

By default, when the Vertex channel emulator is playing emulation, the relative phases between the radio links follow the Link Phase column in the Radio link Configuration table. The Static Channel Mode allows you to apply pre-defined static channel behavior to the radio link phases with a single click.

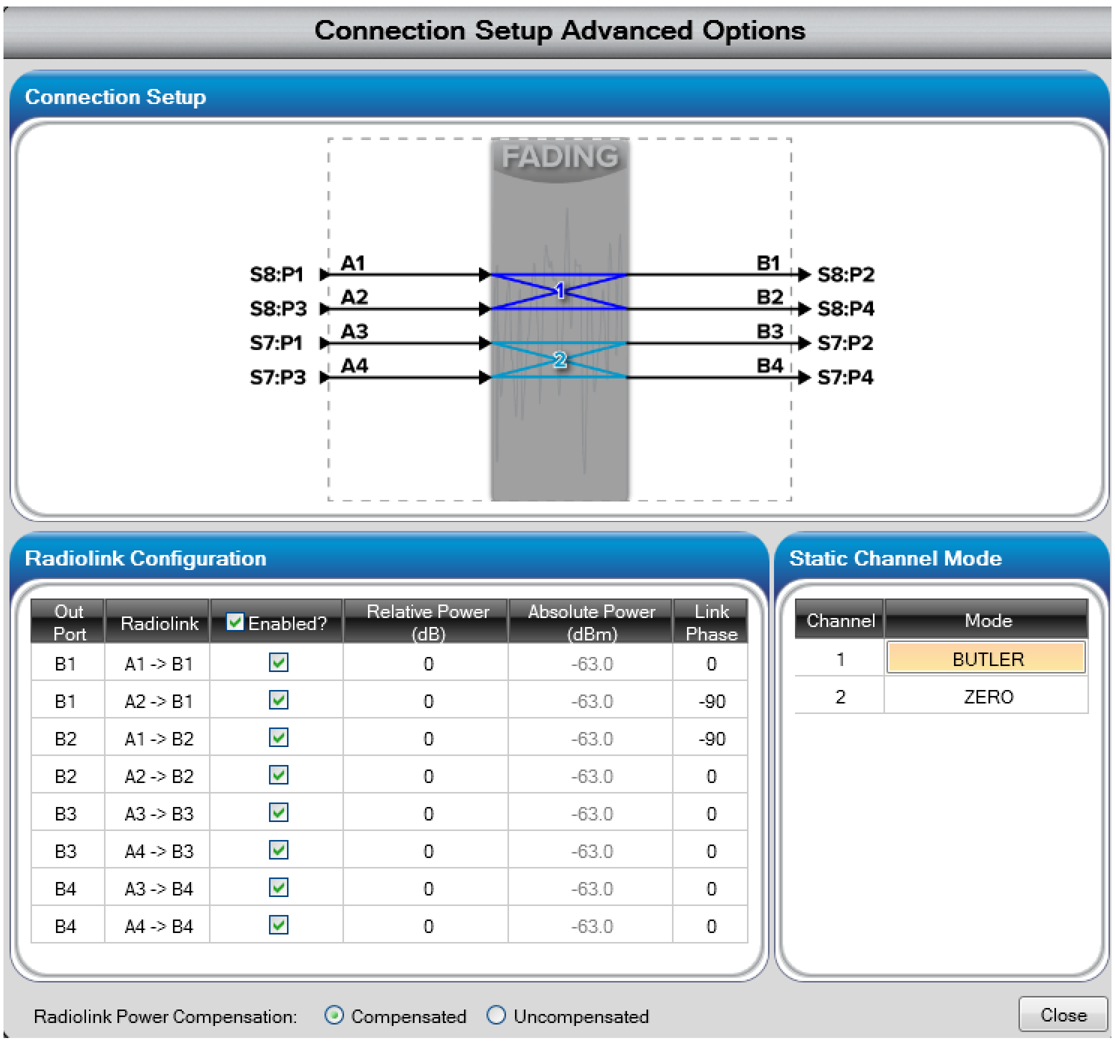


Figure 84. Static Channel Mode application to the Link Phases.

The Static Channel mode is applied to all radio links belonging to the channel grouping as indicated in the Connection Setup diagram.

By Default, ZERO is applied, but selecting a different option applies different phase offsets.

- **BUTLER** - Applies the Butler Matrix with the same phase relationships at the Vertex normally applies when emulation is stopped.
- **BUTLERUL** - Applies the reciprocal of the Butler Matrix for Uplink Testing.
- **ZERO** - All relative Link Phases are set to 0 degrees (Default).

2.12. Phase Calibration

The Vertex channel emulator has the ability to apply phases to individual input and output ports to calibrate your test setup for phase.

Calibrate the phase by selecting **Configure>Port Phase Settings**. The Port Phase Parameters window appears, as shown in the following figure.

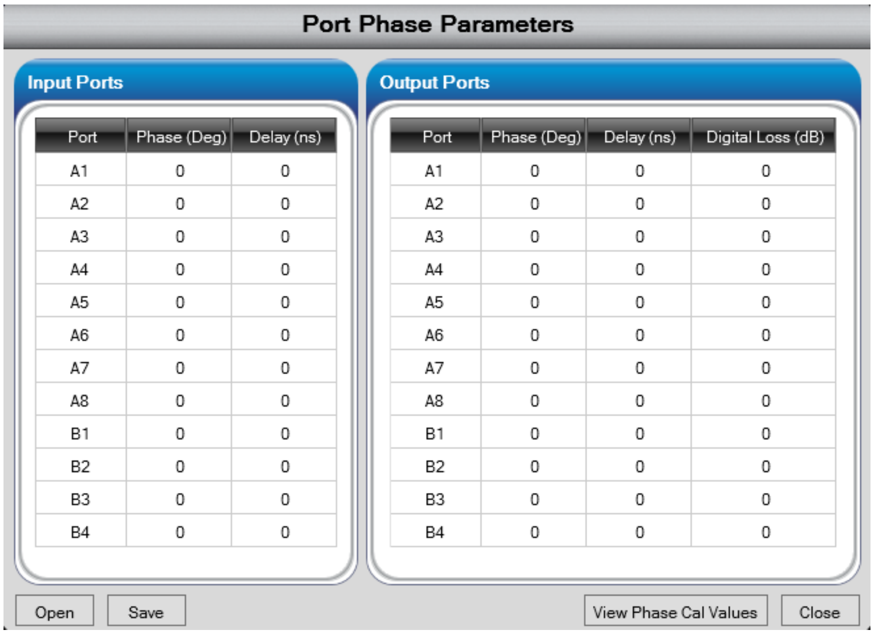


Figure 85. Port Phase Parameters window.

The phase behavior of the Vertex is dependent on the RF settings: input power level, output power level, and the carrier frequency. In addition, these values apply only to the current connection setup. For any given RF setting, you can save the phase calibration values by clicking the **Save** button.

If you set all output power levels equal during the calibration, the phase calibration values are accurate for a range of output power levels around the calibration point.

We recommend that you phase calibrate your Vertex instrument for the different RF settings and connection setup configurations you are using for the day and store them. Note that the Vertex needs to be "Playing" when you perform phase calibration. In the "Stopped" state, the Butler matrix phases are in place, and the phase calibration is not valid after you begin "Playing" fading.

For any given RF setting and connection setup, if there are stored phase calibration values, you can open the stored phase values when you use them with the same RF setting and connection setup. If there are no stored phase calibration values, the current phase calibration values in the Port Phase Parameters window are applied.

NOTE:
The phase behavior of RF equipment varies over time with temperature. Spirent recommends that you do not use phase calibration values obtained more than eight (8) hours prior.

The phase calibration data is not valid across power cycles. Any stored values are lost after you reboot the Vertex instrument.

2.12.1. Input Phase Calibration

The Vertex channel emulator is capable of automatically phase aligning the input ports. This is useful to prevent highly correlated inputs from destructively interfering when combined in the MIMO channel.

To execute Input Phase Calibration, connect a Signal Generator through a splitter to each input port to be phase calibrated. The figure shows an example of a 2-port Input Phase Calibration setup.

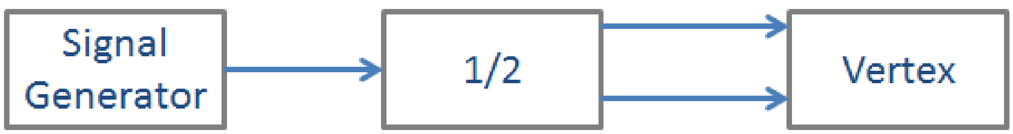


Figure 86. Input phase calibration setup.

An input port can only be calibrated against another input port in the same port group, where a port group is defined as all input and output ports that are related to each other, even through other ports. All the input ports selected in a group must be connected to a common output port. Multiple groups of input ports can be calibrated simultaneously.

NOTE:
Input Ports of a Directional Bypassed channel cannot be calibrated.

For each group, the same signal must be fed into the input ports to be calibrated and an Autoset be performed for each of these ports. The calibration uses all existing Power Meter settings except the Output Port Measurement Type, so these values must be set appropriately for the input signal (including the Output Port Averaging duration).

Access the Input Phase Calibration window shown in the following figure by selecting **Execute>Input Phase Calibration**. The port groups are identified by color.

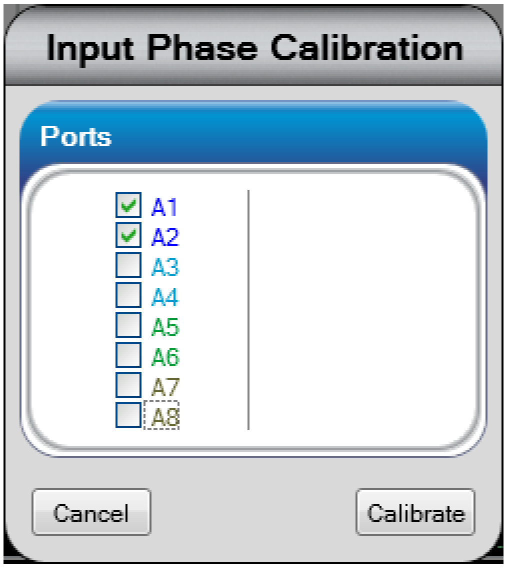


Figure 87. Input Phase Calibration window.

Select the ports to be calibrated by selecting the associated check box, and then click the **Calibrate** button. When Input Phase Calibration begins, a progress bar displays. You can cancel the calibration at any point by clicking the **Cancel** button. If the calibration is canceled, the input phases will be restored to their pre-calibration values.

NOTE:
Input Phase Calibration is not allowed if DEE, the IQ Player, or Ensemble Mode is enabled.

If the measured input power is more than 20 dB below the set power, the calibration will not run. The further the measured power is from the set power, the greater possibility that the calibration will be inaccurate. The calibration stops any emulation in progress. The calibration can take multiple minutes, depending on the number of ports and port groups to be calibrated and on the Power Meter settings. When the calibration is complete, the first input port selected in each port group will have a phase of 0, and all the others will have the phase calibrated with respect to the first input port selected for that port group.

2.12.2. Automatic Phase Calibration

MIMO performance is very sensitive to phase of signal. For MIMO tests, the phase alignment between radio links is critical for the channel emulator. The phase shift of each radio link will change at different frequency and power level, so the phase deviation between radio links must be calibrated at all frequencies and power levels that may be used.

Vertex enables you to align the phase between radio links with the automatic phase calibration software feature. The Vertex instrument's internal link phase must be calibrated in the factory or service center before enabling the auto automatic phase calibration feature. Once the internal link phase is calibrated, the calibration data for different frequency and power levels are saved into the instrument. Then you can load the data to align radio link phase after the RF parameter (like frequency and power level) is configured.

NOTE:
You can load the factory phase calibration data only when the Auto Phase Calibration software option is enabled,

To perform auto phase calibration, perform the following steps:

1. In the Vertex GUI, select Help>Instrument Options to make sure the auto phase calibration software option is available in the instrument.

   The Instrument Options dialog box appears.

Instrument Options

Name	SW/HW	Feature ID	Present
Annual Service Agreement	ASA	ASA	present
Geometric Channel Modes Fading Engine	SW	GCM	present
Wifi Cluster Modeling	SW	WIFICM	present
MIMO OTA - 8 Channel	SW	MOTA-8	present
MIMO OTA - 32 Channel	SW	MOTA-32	not present
Independent Channel Models	SW	ICM	present
Virtual OTA	SW	VOTA	present
Automatic Input Tracking	SW	AUTO-ITR	present
Predicted Input Tracking	SW	PRED-ITR	present
VDT-CT Playback	SW	VDT-CT-PLAYBACK	present
DEE	SW	DEE	present
Primary	SW	PRIMARY	present
Secondary	SW	SECONDARY	not present
Number of Input RF Channels	HW	INPUT-RF-CHANNELS	16
Number of Output RF Channels	HW	OUTPUT-RF-CHANNELS	16
Number of Digital Links	HW	DIG-LINKS	48
Automatic Phase Cal	SW	AUTO-PCAL	present
10 MHz Reference			INTERNAL

Figure 88. Instrument Options dialog box.

2. Configure connection setup and RF parameters.

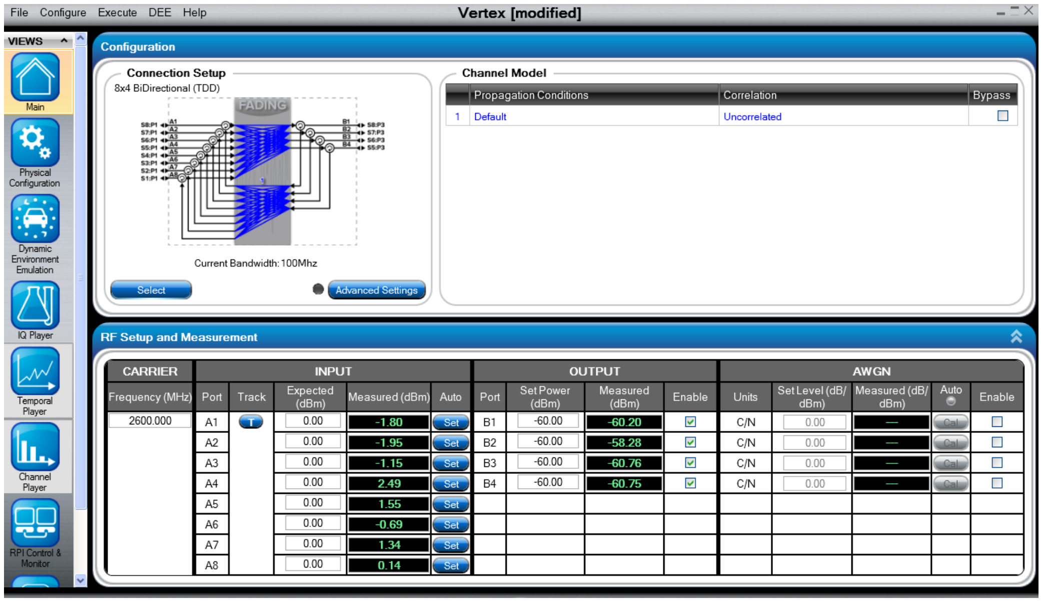


Figure 89. Connection setup and RF configurations.

3. Load the factory phase calibration data:
   a. Select **Configure>Port Phase Setting**.
      The Port Phase Parameters dialog box appears.

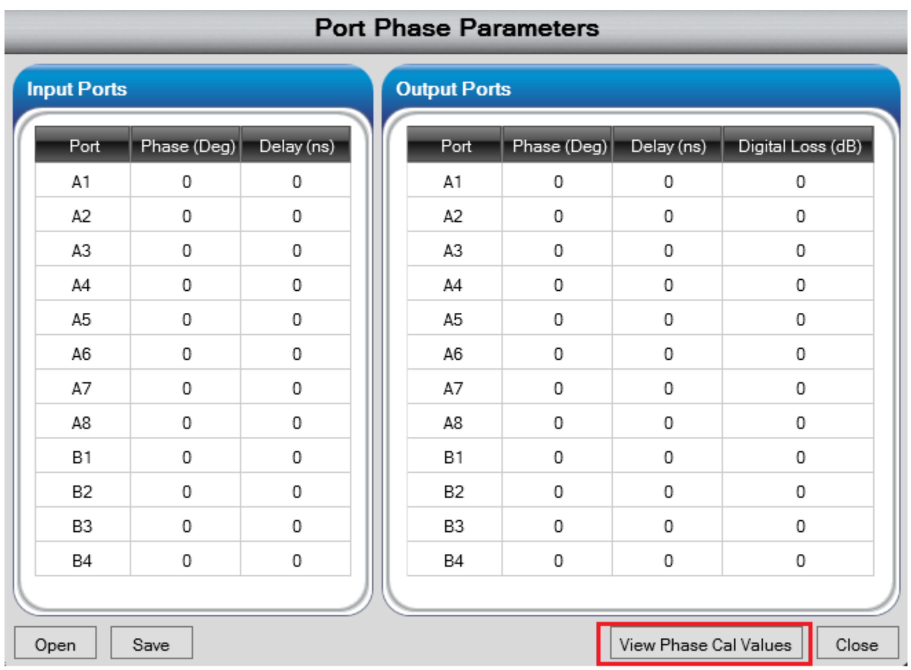


Figure 90. Port Phase Parameters dialog box.

b. Click the **View Phase Cal Values** button.

The Phase Calibration Values dialog box appears.

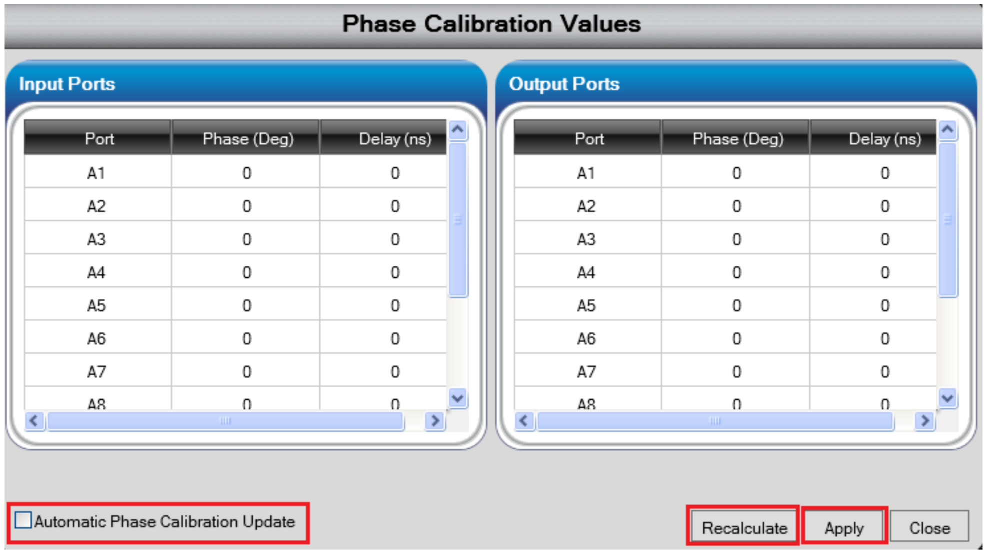


Figure 91. Phase Calibration Values dialog box.

c. Click the **Recalculate** button.

d. Click the Apply button to apply the factory phase data.

> **NOTE**:
> You can load this factory phase calibration data only when the Auto Phase Calibration software option is enabled,

> **NOTE**:
> You have the option to check the **Automatic Phase Calibration Update** check box. When you change the carrier frequency or output power level, you must recalculate the phase calibration data and click the **Apply** button. If you check the **Automatic Phase Calibration Update** check box, when you change the output power level, Vertex will automatically update the phase calibration data with the power level change.

The internal link phase and delay will be automatically aligned after you load the phase calibration data.

Since the Auto phase calibration feature only aligns the internal link phase, if you use different lengths of external cables or different attenuators, you may also need to align the external cables and attenuators together with internal radio links of Vertex.

Vertex enables you add the phase and delay of external phase and delay offset, and save it into the system. If you do not change external cables, you can just load the offset data after you reboot the instrument. Perform the following steps to add cables phase and delay offset:

1. Locate the internal factory phase calibration data.
2. Select **Configure>Port Phase Setting**.

   The Port Phase Parameters dialog box appears.

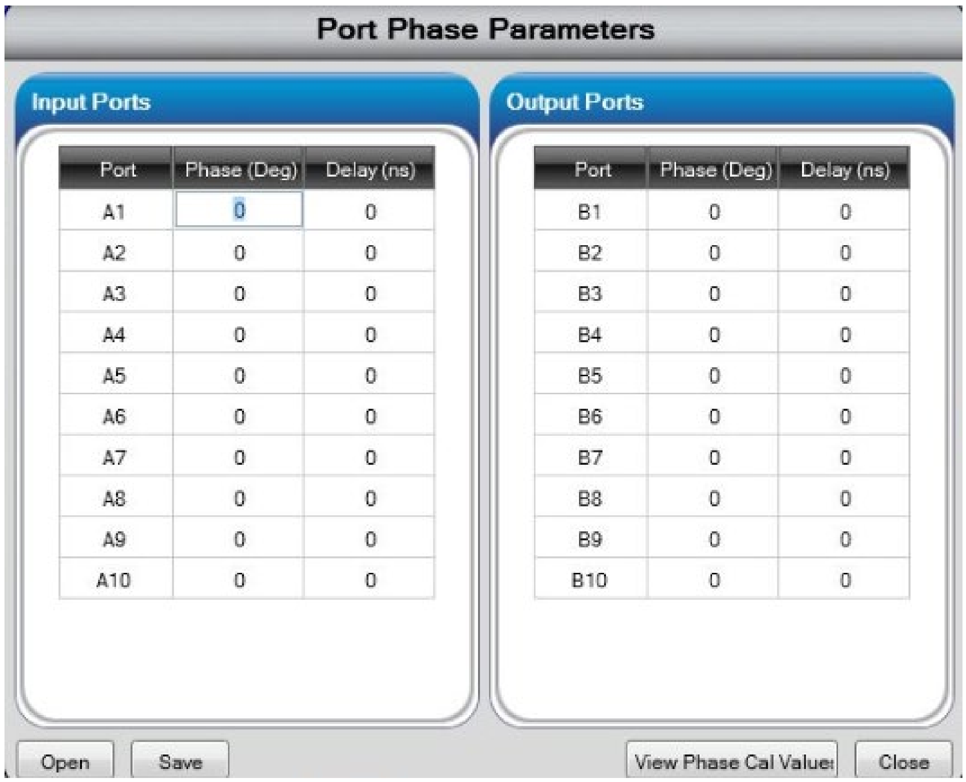


Figure 92. Port Phase Parameters dialog box.

3. Measure external components (input ports components, output port components) for absolute delay and phase measurement. Refer to the application notes on how to conduct external components phase calibration for Vertex units.
4. Add the relative phase and delay to each port in the port phase parameter table.
5. Click the **Save** button to save the data into the Vertex instrument.

   The link phase and delay with external cables is aligned.

Since the external phase and delay offset is only valid to the current frequency, if you want to use Vertex at different frequencies, you must measure the offset data at different frequencies, and save them into respective files. When you change frequency, you just need to open the data at that frequency. Then the phase and delay with external cables will be aligned.

2.13. Advanced Port Settings

You can configure Advanced Port settings by selecting **Configure>Advanced Port Settings**, as shown in the following figure.

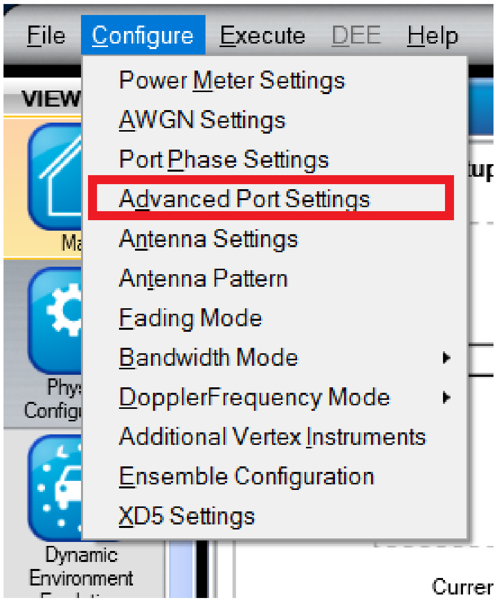


Figure 93. Selecting the Advanced Port Settings option.

The Advanced Settings window is shown in the following figure.

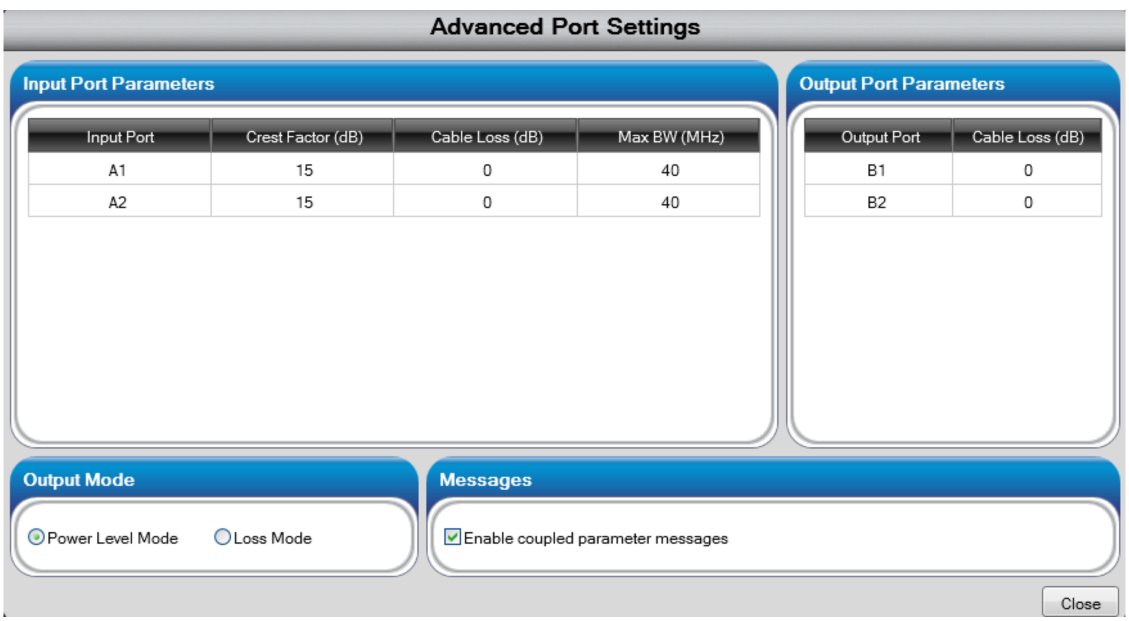


Figure 94. Advanced Port Settings window.

The settings that can be configured in the Advanced Settings window are described in detail in the following sections.

2.13.1. Crest Factor

The Crest Factor is a measure of the maximum peak/avg power ratio the Vertex channel emulator can accept without causing an overload condition. You can configure each port of the Vertex with a larger crest factor than the default setting.

The Vertex Crest Factor is set to 15 dB by default, which is sufficient for most applications. You can set larger values when required by the application.

NOTE:
The Crest Factor of a port is tied to the crest factor of all ports connected by Radio Links. For example, in a Dual 2x2 UniDirectional connection setup, changing the Crest Factor for Port A2 changes Port A1 to the same Crest Factor, but does not affect the Crest Factor for Ports A3 and A4.

NOTE:
Increasing this value limits the maximum output power of the Vertex. Additionally, system noise and spurious performance will be degraded.

2.13.2. Cable Loss

The Vertex channel emulator provides the ability to set an offset compensating for cable loss in both the Set and Measured Power indicators. Cable loss can be configured separately for the input path and output path to and from each Vertex port. This setting allows you to enter a value for the loss associated with a cable or other RF components connected to a Vertex port.

For example, in a UniDirectional connection setup, if a cable connecting the device has an associated 1.2 db loss, enter this value in the Cable Loss column for the Output Port. If you then set the Vertex output level to -50 dBm, the actual level of the Vertex RF Output Port is set to -48.8 dBm, and the level of the DUT is -50 dBm.

2.13.3. Max BW (MHz)

There are four bandwidth modes with Vertex: 40MHz ,100MHz, 200MHz, and 400 MHz. You can switch the bandwidth mode using either the GUI or an RPI command.

200MHz and 400MHz bandwidth modes are only supported with SDE-DSPM2 hardware. You also need a software license to switch to 100MHz bandwidth mode 200MHz bandwidth mode, and 400MHz bandwidth mode.

NOTE:
After you change the bandwidth mode among 40MHz, 100MHz, 200MHz, and 400MHz, you must reboot the Vertex instrument.

To change the bandwidth mode using the GUI, perform the following steps:

1. Select **Configure->Bandwidth Mode** and select the appropriate bandwidth as shown in the following figure.

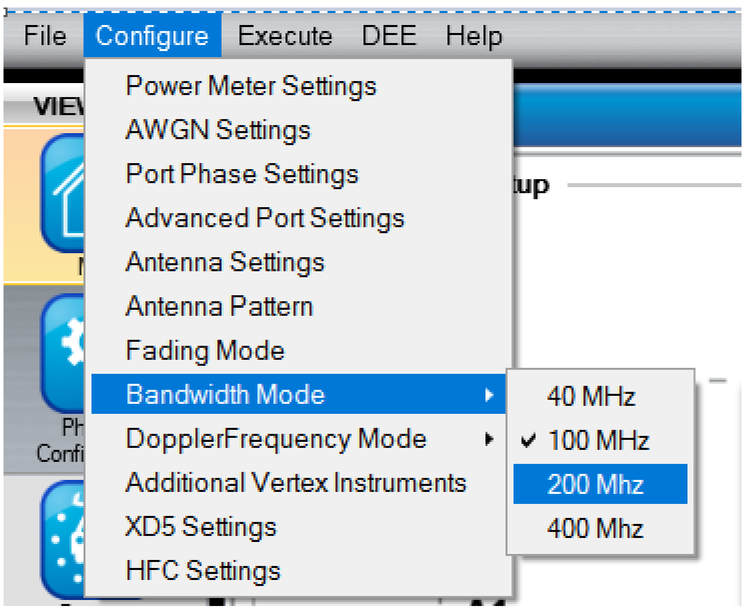


Figure 95. Selecting the Bandwidth Mode option using the GUI.

The following Warning dialog box appears.

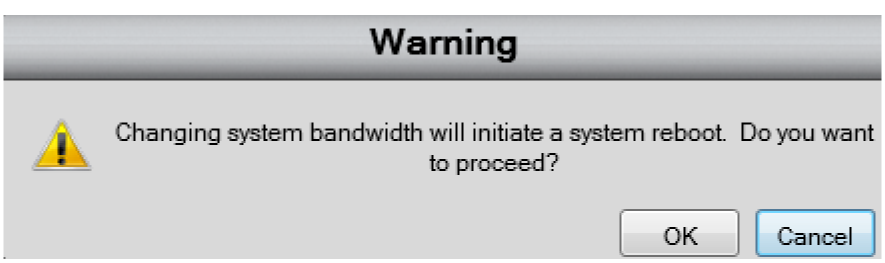


Figure 96. Warning dialog box.

2. Click the **OK** button to reboot the instrument.
3. Accept the firmware update, and then wait until the Vertex instrument boots up completely.

After the Vertex instrument boots up, the current bandwidth mode is displayed in the Connection Setup window.

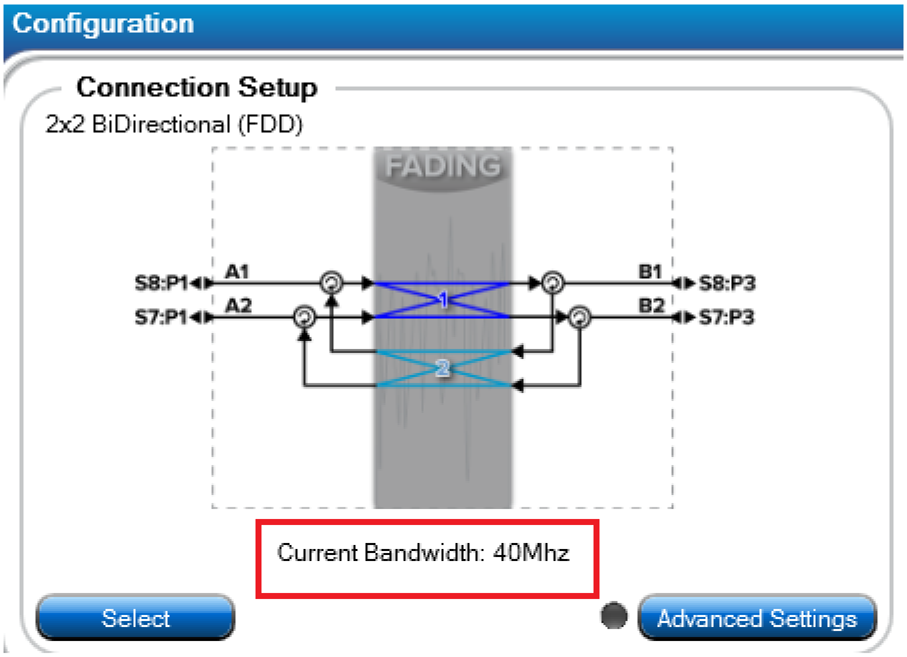


Figure 97. Current bandwidth displayed in Connection Setup window.

In an FDD bidirectional connection setup, you can improve the isolation performance of the test system by limiting the maximum signal bandwidth input into the digital processing section of the Vertex.

NOTE:
The Vertex unit default setting is 40 MHz maximum bandwidth on each port in 40MHz bandwidth mode.

You can only switch to 100MHz bandwidth mode, 200MHz bandwidth mode, or 400MHz bandwidth mode when a 6GHz RF module is installed.

If a 4GHz RF module is installed, the maximum bandwidth is fixed at 40MHz.

2.13.4. Using 400MHz Bandwidth Mode

Vertex release 4.70 and later supports 400MHz bandwidth.

NOTE:
Currently, DEE, Input Port Autotest, and AWGN cannot support 400MHz bandwidth.

2.13.4.1 Switching Vertex to 400 MHz Bandwidth Mode

You can switch Vertex from another bandwidth mode to 400MHz bandwidth mode from the GUI or with an RPI command.

NOTE:
A 400MHz Bandwidth software license is required to enable Vertex to operate in 400MHz bandwidth mode.

To set Vertex to operate in 400MHz bandwidth mode using the GUI, perform the following steps:

1. Select **Configure->Bandwidth Mode** and select **400 Mhz** as shown in the following figure.

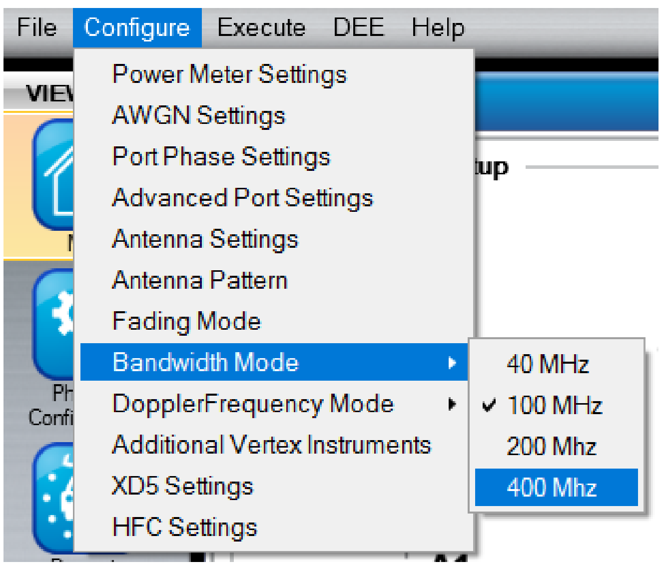


Figure 98. Selecting 400MHz bandwidth mode using the GUI.

The following Warning dialog box appears.

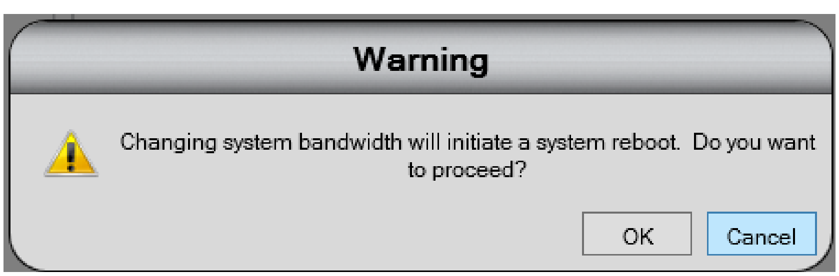


Figure 99. Warning dialog box.

2. Click the **OK** button to reboot the instrument.

   The following message box appears.

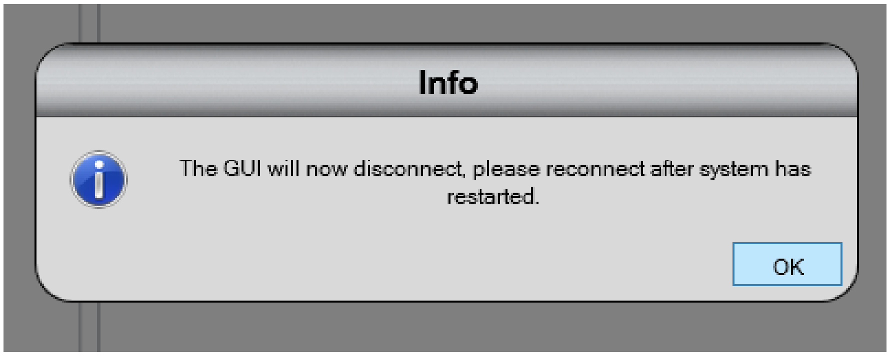


Figure 100. Info message box.

3. Click the **OK** button.
4. Accept the firmware update, and then wait until the Vertex instrument boots up completely.

After the Vertex instrument boots up, Vertex is operating in 400MHz bandwidth mode. The current bandwidth mode is displayed in the Connection Setup window.

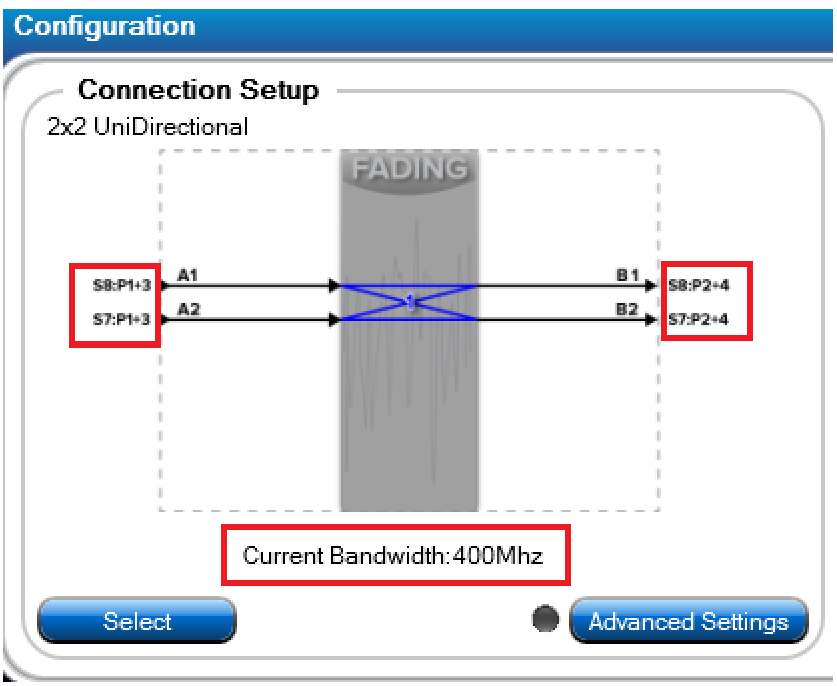


Figure 101. Current bandwidth displayed in Connection Setup window.

When use Vertex in 400MHz bandwidth mode, two RF ports must be combined together with a calibrated RF cable and a power combiner to create one logical port. For example, for a 2x2 Uni directional connection setup:

- the two I/O ports on the RFM of slot 8 are combined together as A1 (S8:P1+3)
- the two I/O ports on the RFM of slot 7 are combined together as A2 (S7:P1+3)
- the two output ports on RFM of slot 8 are combined together as B1 (S8:P2+4)
- the two output ports on RFM of slot 7 are combined together as B2 (S7:P2+4)

You can check the Physical Layout on the Vertex GUI to determine the correct RF port location and mapping as shown in the following figure.

Figure 102. Sample Physical Layout to determine RF port location and mapping.

2.13.4.2 Phase and Amplitude Calibration for 400MHz Bandwidth Mode

To achieve good phase and amplitude flatness over 400MHz bandwidth and to eliminate the discontinuity of phase and delay between the two RF ports that are combined together, you must perform phase and amplitude calibration when you use Vertex in 400MHz bandwidth mode.

You can use a network analyzer to calibrate each port. For a unidirectional connection setup, perform the following steps:

1. Connect A ports to port 1 on the network analyzer.
2. Connect B ports to port 2 on the network analyzer.
3. Measure the amplitude, phase, and delay at frequencies over the whole band and align them.

Both input and output must be aligned and independently. You must perform the following calibration sequence:

1. A* to B1 for level
2. A1 to B* for level
3. A* to B1 for delay
4. A1 to B* for delay
5. A* to B1 for phase
6. A1 to B* for phase

The following figure shows the calibration setup.

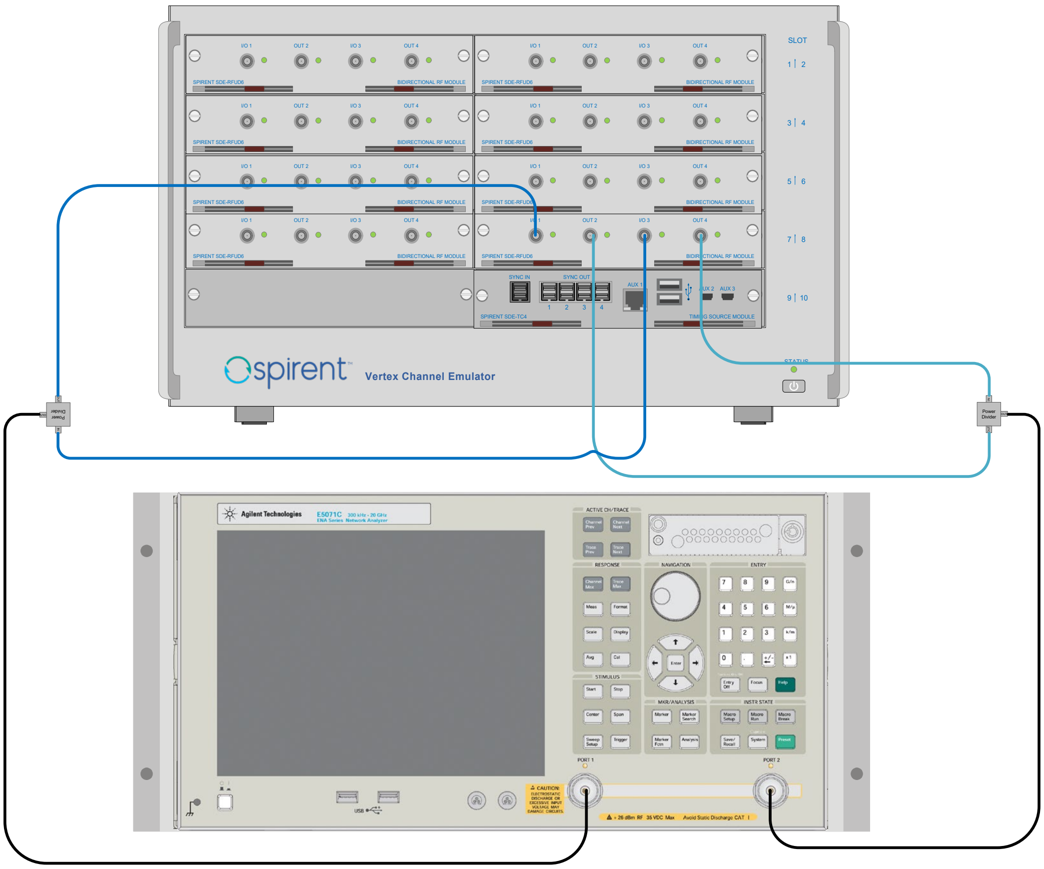


Figure 103. Sample calibration setup.

System Calibration (field):

- Only single level calibration is supported at this time.
- Both input and output must be aligned and independently. You must perform the following calibration sequence:
    a. A* to B1 for level
    b. A1 to B* for level
    c. A* to B1 for delay
    d. A1 to B* for delay
    e. A* to B1 for phase
    f. A1 to B* for phase
    g. A1 to B1

When you use Vertex in 400MHz BW mode, you can save the calibration data to a file in Vertex. To load a saved calibration data file, select **Execute->Port Combine Calibration->Load Port Combine Calibration** and then select the file you want to load.

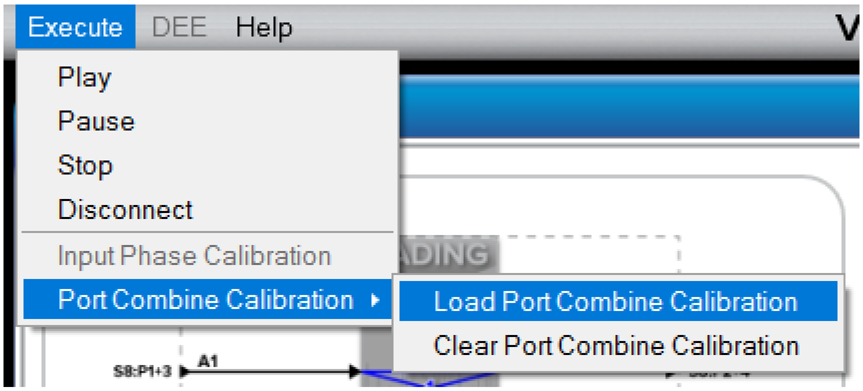
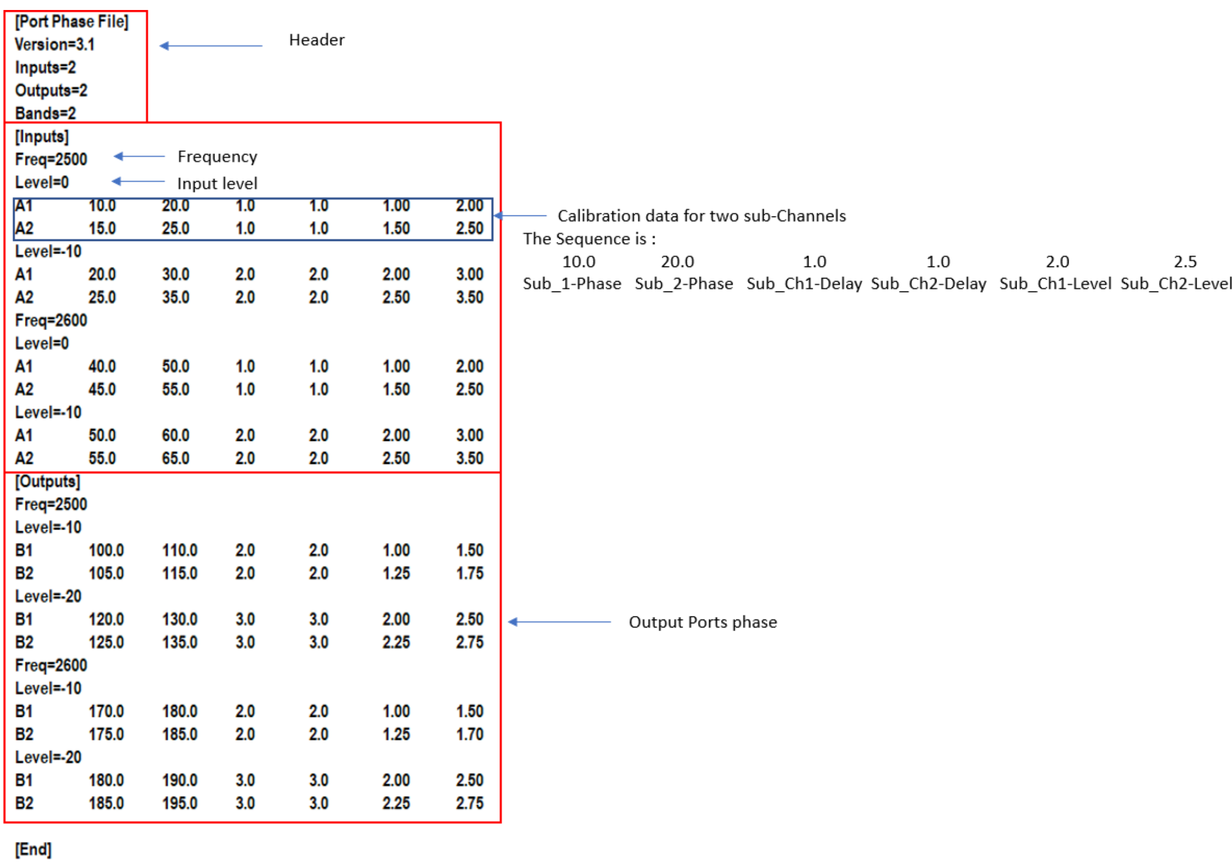


Figure 104. Load Port Combine Calibration option.

If you want to exit 400MHz BW mode or clear the Port combine calibration data, select **Execute->Port Combine Calibration->Clear Port Combine Calibration**.

The following figure shows the format of port combine calibration data:



```
[Port Phase File]
Version=3.1
Inputs=2
Outputs=2
Bands=2
[Inputs]
Freq=2500
Level=0
A1   10.0   20.0   1.0   1.0   1.00   2.00
A2   15.0   25.0   1.0   1.0   1.50   2.50
Level=-10
A1   20.0   30.0   2.0   2.0   2.00   3.00
A2   25.0   35.0   2.0   2.0   2.50   3.50
Freq=2600
Level=0
A1   40.0   50.0   1.0   1.0   1.00   2.00
A2   45.0   55.0   1.0   1.0   1.50   2.50
Level=-10
A1   50.0   60.0   2.0   2.0   2.00   3.00
A2   55.0   65.0   2.0   2.0   2.50   3.50
[Outputs]
Freq=2500
Level=-10
B1   100.0  110.0  2.0   2.0   1.00   1.50
B2   105.0  115.0  2.0   2.0   1.25   1.75
Level=-20
B1   120.0  130.0  3.0   3.0   2.00   2.50
B2   125.0  135.0  3.0   3.0   2.25   2.75
Freq=2600
Level=-10
B1   170.0  180.0  2.0   2.0   1.00   1.50
B2   175.0  185.0  2.0   2.0   1.25   1.70
Level=-20
B1   180.0  190.0  3.0   3.0   2.00   2.50
B2   185.0  195.0  3.0   3.0   2.25   2.75

[End]
```


Figure 105. Format of port combine calibration data.

NOTE:
Keep in mind:

- Each port above consists of 2 subchannels (or 2 physical ports) for 400MHz.
- The RFMs must be calibrated by the factory with Vertex release 4.71.
- Each level must be calibrated separately.

Contact the Spirent Support team for details about 400MHz bandwidth calibration.

2.13.5. Doppler Frequency Mode

In Vertex release 4.50 and later, if a Vertex instrument is installed with SDE-DSPM2 hardware, the instrument can support up to 12KHz fading doppler frequency. You can switch the doppler frequency mode between 4KHz and 12KHz.

NOTE:
The Vertex unit default setting is **4KHz maximum fading doppler frequency**. You need a software license to support 12KHz doppler frequency.

To change the Doppler frequency mode using the GUI, perform the following steps:

1. Select **Configure->Doppler Frequency Mode** and select the appropriate frequency as shown in the following figure.

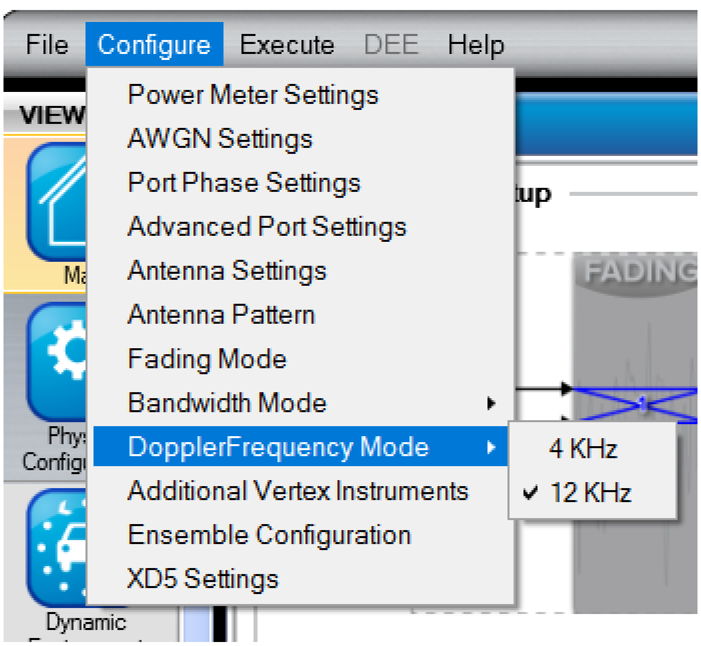


Figure 106. Selecting the Doppler Frequency Mode option using the GUI.

The Warning dialog box appears. When you change the Doppler frequency mode from 12KHz to 4KHz, the Vertex system will reset to the default topology .

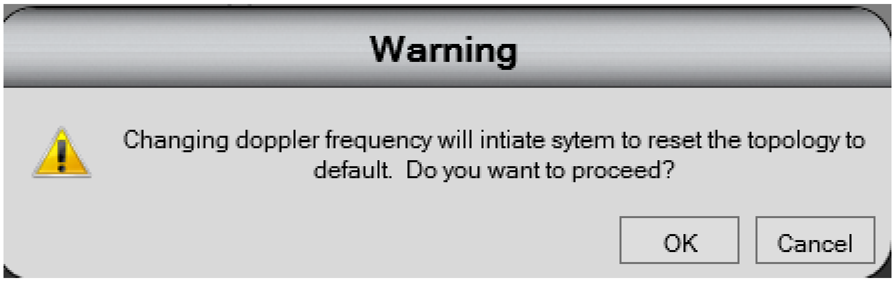


Figure 107. Warning dialog box.

2. Click the **OK** button.

2.13.6. Output Mode

2.13.6.1 Power Level Mode

The Power Level Mode parameter allows you to configure expected input and set output power levels separately for maximum flexibility. The loss through a particular Vertex channel in its simplest form (when all expected input powers are set equal, and all set output powers are set equal) then becomes:
Loss = Expected Input - Set Output

2.13.6.2 Loss Mode

The Loss Mode parameter allows you to set the Loss through the Channel directly, as opposed to separately setting the Input and Output Power Levels. This mode forces all Expected Input Powers and all Set Output Powers to be set to the same value for port groups on each side of the connection setup that are connected by a MIMO channel.

2.13.7. Coupled Parameter Messages

Certain Vertex instrument operations can cause parameters to change based on range precedence. In these instances, the Coupled Parameter Message Log appears.

For example, if you change the receiver bandwidth of the AWGN source, it could cause the set C/N ratio to become unachievable. In this case, the set C/N ratio is modified to the closest achievable value.

You can hide these messages by deselecting the **Enable Coupled Parameter Messages** option.

2.14. Dynamic Environment Emulation (DEE)

The DEE feature allows you to change the state of the Vertex channel emulator dynamically at specified time intervals.

NOTE:
The DEE feature is only supported with the bidirectional RF module-based hardware platform. A Vertex instrument that has unidirectional RF modules installed cannot use DEE feature.

This section details the parameters that control the Dynamic Environment Emulation (DEE) function available with the Vertex.

You can create DEE scenarios with Vertex DEE Template, which is supported by Microsoft Excel. You must make sure the PC has a Microsoft Excel license if you want to run the DEE Template.

You can access the DEE template in the following two ways:

- From the installation folder

  The DEE Template is stored in the root directory of the Vertex installation and can usually be found in the following location:
  C:\Program Files (x86)\Spirent Communications\Vertex.

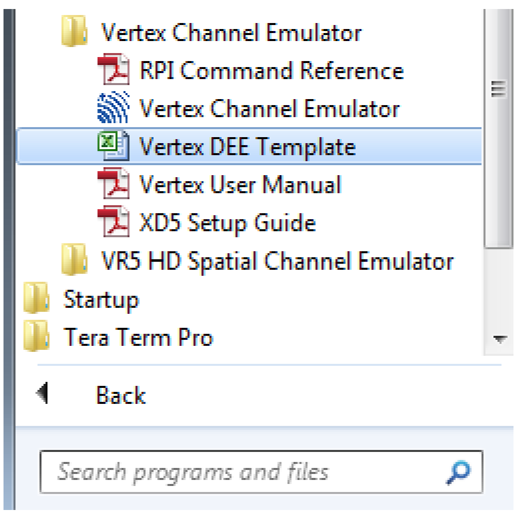


Figure 108. Vertex DEE Template in the Program shortcut folder.

The **Vertex DEE Template** icon appears on the desktop after Vertex is installed. You can open the Vertex DEE template by double-click on this icon.

Figure 109. Vertex DEE Template icon on the desktop.

- From the Vertex GUI

  You can access the DEE template from the Start menu or from the DEE menu in the GUI. When accessed through the GUI, the DEE template is automatically set to match the selected connection setup, Fading Mode, and Output Mode of the GUI.

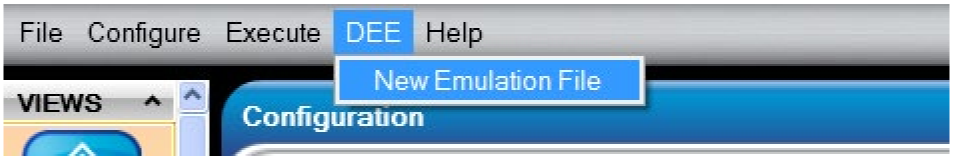


Figure 110. Creating a new DEE template.

The following figure shows the DEE Template open in Microsoft Excel:

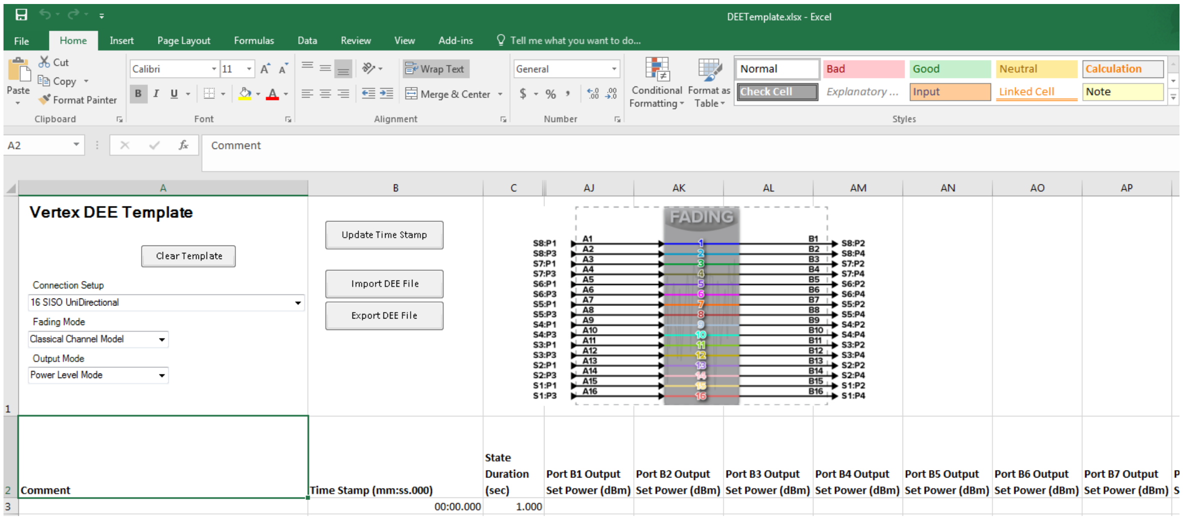


Figure 111. A New DEE template in Microsoft Excel.

You can configure DEE scenarios for Classical Channel and Geometric Channel Models by selecting the Fading Mode. You can also select an Output Mode of either Power Level or Loss, as shown in the following figure.

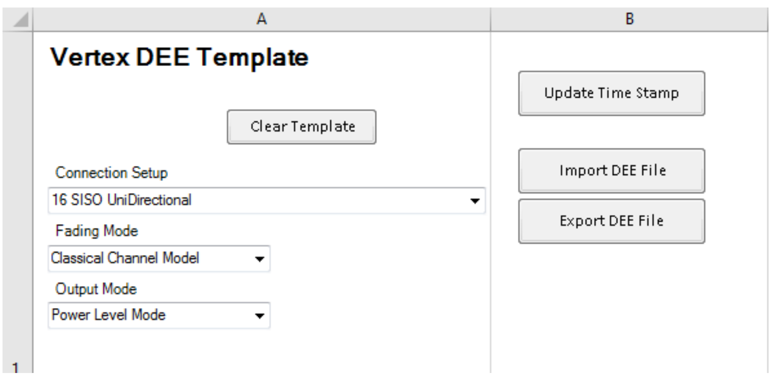


Figure 112. Selecting the Connection Setup, Fading Mode, and Output Mode.

You can change the following parameters in the Classical Channel and Geometric Channel Fading modes:

- Output Port Settings
  - Output Power/Loss
  - AWGN Enabled (ON/OFF)
  - C/N Ratio, Noise Level
- Radio Links
  - Radio Link Enabled (ON/OFF)
  - Radio Link Relative Power
  - Radio Link Phase
- Path Enabled (ON/OFF)
- Path Delay
- Relative Path Loss

The following parameters are available in Classical Channel Models Mode only:

- Bulk Delay Enabled (ON/OFF)
- Bulk Delay
- Rician Line of Sight Angle of Arrival
- Rician K Factor
- Frequency Shift
- Doppler Frequency, Doppler Velocity
- Correlation

The following parameters are available in Geometric Channel Models Mode only:

- BS Angle Spread
- MS Angle Spread
- Angle of Departure (AoD)
- Angle of Arrival (AoA)
- MS Direction
- MS Velocity
- LOS AoD
- LOS AoA
- LOS K Factor

2.14.1. Setting up a Dynamic Environment Test

The following sections provide detailed instructions on setting up a test with dynamically changing environment conditions.

To set up a dynamic environment test:

1. Define the Non-DEE Vertex state.

   The non-DEE state refers to the setup of the Vertex when you are not using the DEE feature. Set the non-DEE state of the instrument using the Vertex GUI or through RPI commands as you would under normal operation. This information combined with State 1 of the State Emulation file describes the state of the Vertex in “State 1” of DEE.

   All parameters set up statically remain in effect unless the particular parameter is changed in DEE. Only certain parameters are capable of being changed in DEE. Parameters not controllable in DEE remain in whatever state they are in prior to starting a DEE run. For example, the Fading Type associated with a path can be set through the GUI or RPI in non-DEE mode. Even though the Fading Type cannot be changed dynamically with DEE, the Fading Type set prior to entering DEE remains active during the DEE run.

   NOTE:
   SPECIAL CASE: Delay Mode – If the delay mode for a particular path is set to **Birth/Death or Sliding-Delay**, the delay for that path cannot be changed in DEE. Sliding Delay and Birth/Death operate as set in non-DEE mode while running a DEE simulation.

2. Defining the Dynamic Environment.

   Define the dynamic channel conditions you would like to emulate in the DEE File. Refer to Section 2.7.2 for details.

   NOTE:
   This file needs to define changes only from the non-DEE state of the unit. If information in the template is left blank, it assumes there is no change from the last state.

3. Enter the Dynamic Environment Emulation view in the Vertex GUI, as shown in the following figure.

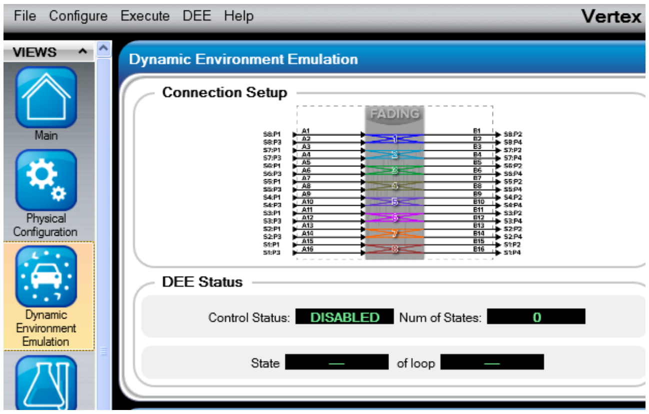


Figure 113. Dynamic Environment Emulation (DEE) view.

4. Select the DEE File by clicking the **DEE File** button and selecting the file from the saved location, as shown in the following figure.

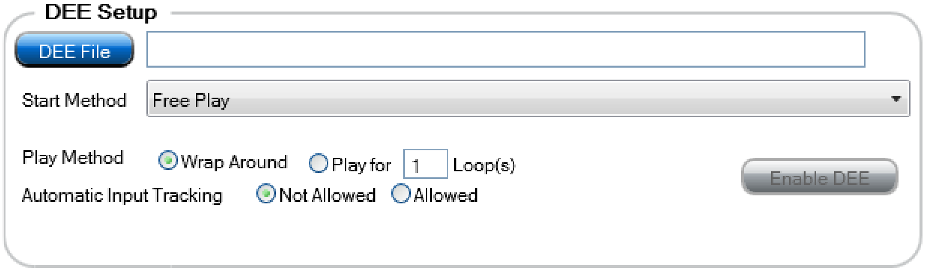


Figure 114. Selecting the DEE file.

5. Configure Playback Controls:
   a. Select the Start Method (Free Play or Triggered Start):
      - **Free Play** – DEE will start immediately when a start command is issued.
      - **Triggered Start** – A rising edge must be detected on BNC TRIG port 1 prior to the start of DEE simulation.
   b. Select the Play Method (Wrap Around or Play for X Loops):
      - **Wrap Around** – The DEE file continues to loop indefinitely.
      - **Play for X Loops** – The DEE file plays to the end, resets, and loops for X times. After completing X loops, the DEE run ends and remains stopped at the beginning of State 1. This means that, statistically, State 1 will be the same each time DEE loops, but the instantaneous phase and amplitude distortion will differ. This is done to avoid any glitches when wrapping from the last state to the first.
   c. Select the Automatic Input Tracking (Allowed or Not Allowed):
      - **Allowed** – Input Level changes (including those from Automatic Input Tracking) can be applied while the DEE scenario is running.
      - **Not Allowed** – Input changes are not allowed after DEE is enabled.

   NOTE:
   When the file loops back to State 1, the state of the instrument will be the same as it was the first time in State 1, except that the random number generator creating Rayleigh fading will not reset.

   NOTE:
   You can also perform Steps A through C via the remote programming interface. Refer to Chapter 3 for details.

6. Enable DEE by clicking the **Enable DEE** button, as shown the following figure.

Figure 115. Enable DEE button.

7. Play the DEE file by clicking the **Play** button at the bottom of the GUI, as shown in the following figure.

Figure 116. Play button.

8. Disable DEE. To disable and restore the instrument to its original state before entering DEE:

   a. Click the **Stop** button at the bottom of the GUI, as shown in the following figure.

Figure 117. Stop button.

   b. Click the **Disable DEE** button, as shown in the following figure. This stops the DEE engine and restores the instrument to its original state before entering DEE.

Figure 118. Disable DEE button.

2.14.2. Creating a DEE File

The DEE Template helps you easily create dynamically varying channel conditions by filling out a simple spreadsheet.

2.14.2.1 Using the DEE Template

The emulation file can be modified using standard Excel methods. The emulation file contains five types of tabs: RF Setup, Propagation, RLink, Correlation and XD5.

The RF Setup Tab

The RF Setup tab displays and allows you to select the connection setup, fading mode and output mode, and import and export DEE Files. Additionally, this tab contains the following parameters:

- State Duration – The duration for each state in units of seconds. Note that although state duration is available for viewing in all of the tabs, the settings in the main tab are used to determine the set state duration.
  The minimum state duration you can set is 10 ms*.

> **NOTE**:
> The Minimum State Duration in Automatic Input Tracking is set to Allowed is 40 milliseconds.

- Port X Output Set Power – For each of the output ports used in the connection setup, the output power may be set. This setting is only visible when Output Mode is set to Power Level.
- Port X Set Loss – For each of the output ports used in the connection setup, the power loss may be set. This setting is only visible when Output Mode is set to Loss.
- Port X AWGN Enabled – For each of the output ports used in the connection setup, the AWGN may be enabled or disabled.
- Port X C/N Ratio (dB) – The Carrier to Noise Ratio for the port can be set (in dB) for each of the output ports used in the connection setup.

> **NOTE**:
> The C/N value is only applied if AWGN units is set to C/N prior to DEE compile.

- Port X Noise Level (dBm) – The Fixed Noise level for the port can be set (in dBm) for each of the output ports used in the connection setup.

> **NOTE**:
> Noise Level value is only applied if AWGN units is set to Noise Level prior to DEE compile.

In loss output mode, one value of Input Set Loss can apply to more than one port. This is indicated by an additional “Group *” designator in the column heading. Each such column heading has a comment detailing the ports that use the parameter.

The RF Setup tab also includes the following buttons that can be used to perform actions associated with generating a DEE file.

- The **Update Timestamp** button updates the Timestamp Column in each of the tabs. This column is useful when determining how much time it takes to reach state X, especially when the state duration of individual states vary.
- The **Export DEE** button is used to export the information in the template to .vstb format. This is the format the Vertex software uses to import the state change information.
- The **Import DEE** button is used to import a previously exported VSTB file back into the template.
- The **Clear Template** button is used to clear the template of all user-entered data.

NOTE:
When exporting a file, if a row is encountered without any data, it is treated as the end of the file. If you want to have a number of states where nothing changes, fill in the state duration column for all of these states, as shown in the following two figures. The data does not need to change, but it does need to exist.

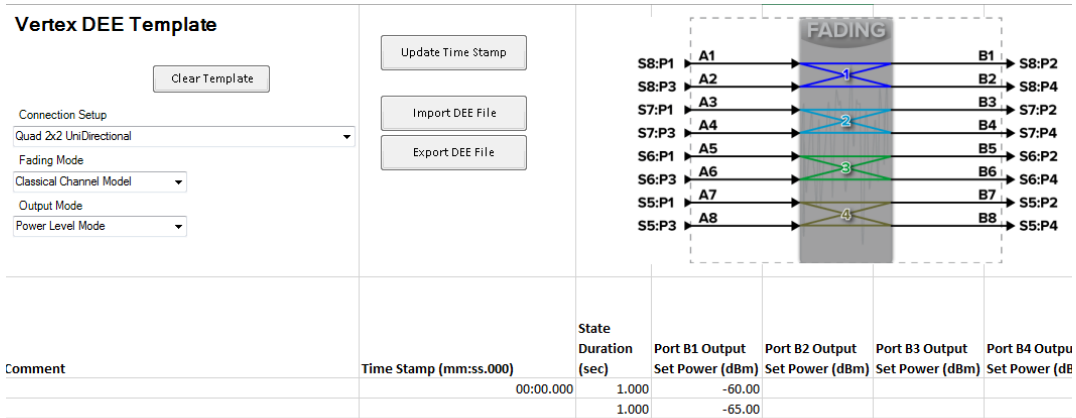


Figure 119. Sample column with two states.

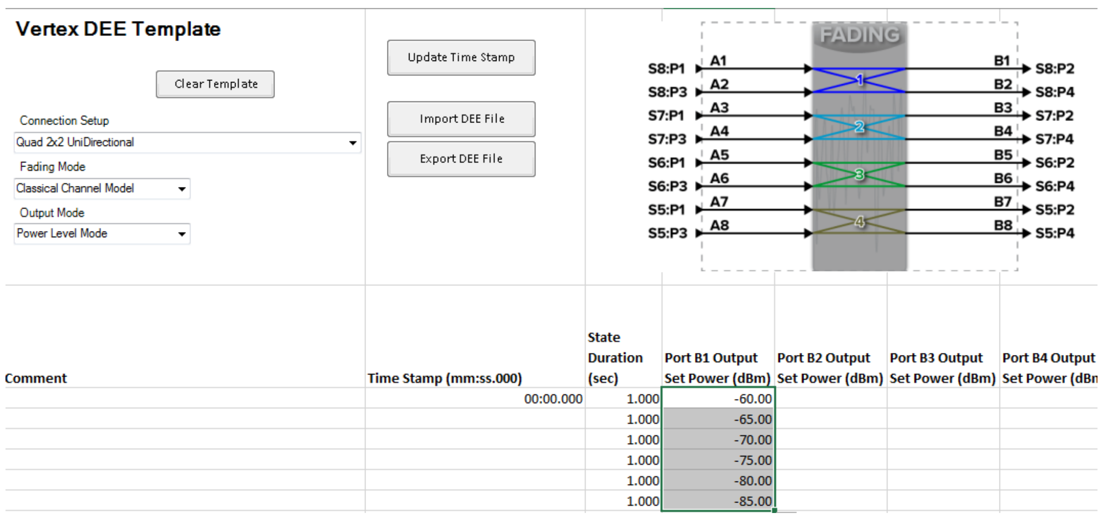


Figure 120. Sample column with six states.

Propagation Tab

The Propagation X tabs defines dynamic changes associated with each of the propagation conditions. There is a separate tab available for each Channel Model defined in the selected connection setup. For example, the 2x2 BiDirectional (FDD) connection setup has two channel models and two associated separate propagation conditions, as shown in the following figure.

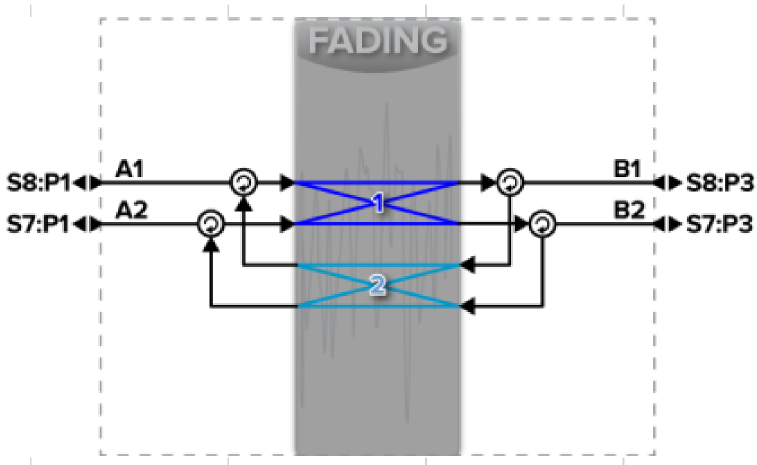


Figure 121. 2x2 Bidirectional connection setup diagram.

If this connection setup is selected, there are two tabs available in the template (Propagation 1 and Propagation 2).

Each tab can be used to set the following parameters:

- PX Doppler Preference (Frequency/Velocity) – Defines whether the Doppler Frequency or Doppler Velocity parameter is used as the source of the Doppler for this particular path.
- Bulk Delay Enabled-Enable or disable Bulk delay.
- Bulk Delay (us)-Defines the value of Bulk delay.
- PX Enabled (ON/OFF) – Defines whether the particular path is enabled.
- PX Delay (µs) – Defines the delay of the particular path.
- PX Relative Path Loss (dB) – Defines the relative path loss of the particular path.
- PX LOS AOA (Deg) – Defines the line of sight angle of arrival of the particular path.
- PX Rician K Factor (dB) – Defines the Rician K factor of the particular path. This setting is only meaningful if the path Fading Type was set to Rician in the non-DEE mode prior to running DEE.
- PX Frequency Shift (Hz) – Defines the frequency shift of the particular path.

- PX Doppler Frequency (Hz) – Defines the Doppler frequency of the particular path. This setting is only meaningful if the path Fading Type was set to Rayleigh or Rician in the non-DEE mode prior to running DEE and the PX Doppler Preference is set to Frequency.
- PX Doppler Velocity (km/h) – Defines the Doppler velocity of the particular path. This setting is only meaningful if the path Fading Type was set to Rayleigh or Rician in the non-DEE mode prior to running DEE and the PX Doppler Preference is set to Velocity.

Each of the above parameters can be manually set by editing the template. You can also right-click a row in the Excel spreadsheet to open the Import Fading Profile window, as shown the following figure. From this window, you can import a previously exported fading profile using the library view. Note that this window will only display if the selected row is before the end of the emulation information.

NOTE:
You must use the keyboard shortcuts (**Ctrl-C** and **Ctrl-V**) to copy or paste because right-clicking under the **Propagation Conditions** tab opens the File window.

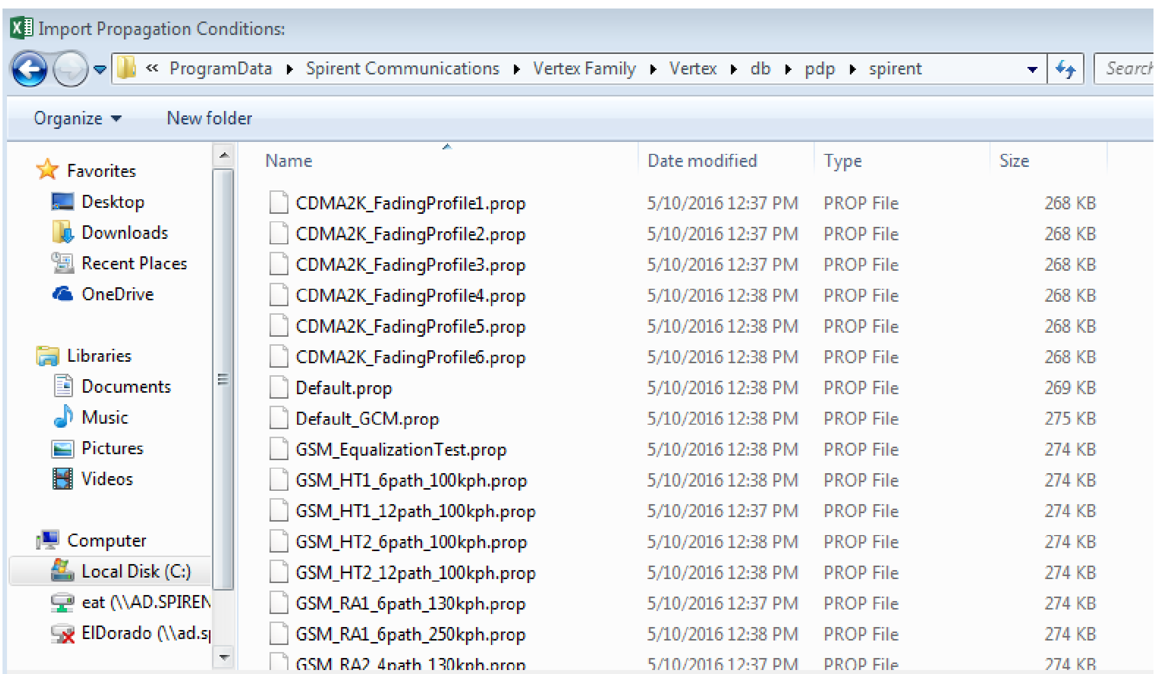


Figure 122. Import Propagation Conditions window.

RLink Tab

The RLink tab is used to set parameters associated with each of the radio links in the selected connection setup. For example, the 2x2 BiDirectional (FDD) connection setup has a total of eight associated radio links. For each radio link, you can modify the following parameters:

- AX->BY Enabled – This field can be used to selectively enable or disable a given Radio Link.
- AX->BY Relative Power (dB) – Sets the power of the particular radio link relative to other enabled Radio Links that are connected to the same output ports.
- AX->BY Link Phase (Deg) – Sets the phase offset associated with the particular radio link.

Correlation Tab

The Correlation X tabs are used to set the correlation matrices associated with each set of propagation conditions. Similar to the Propagation tabs, there is a separate correlation tab available for each channel model in the chosen connection setup.

To edit the correlation matrix associated with a given state, right-click the desired row in the "Complex Correlation All" column to open the Import Correlation window, shown the following figure. This window allows you to select a .corr file similar to the selection of Fading Profiles under the Propagation tab. These .corr files can be created in the Vertex GUI Correlation Library View by exporting the Correlation information, which spans 24 paths.

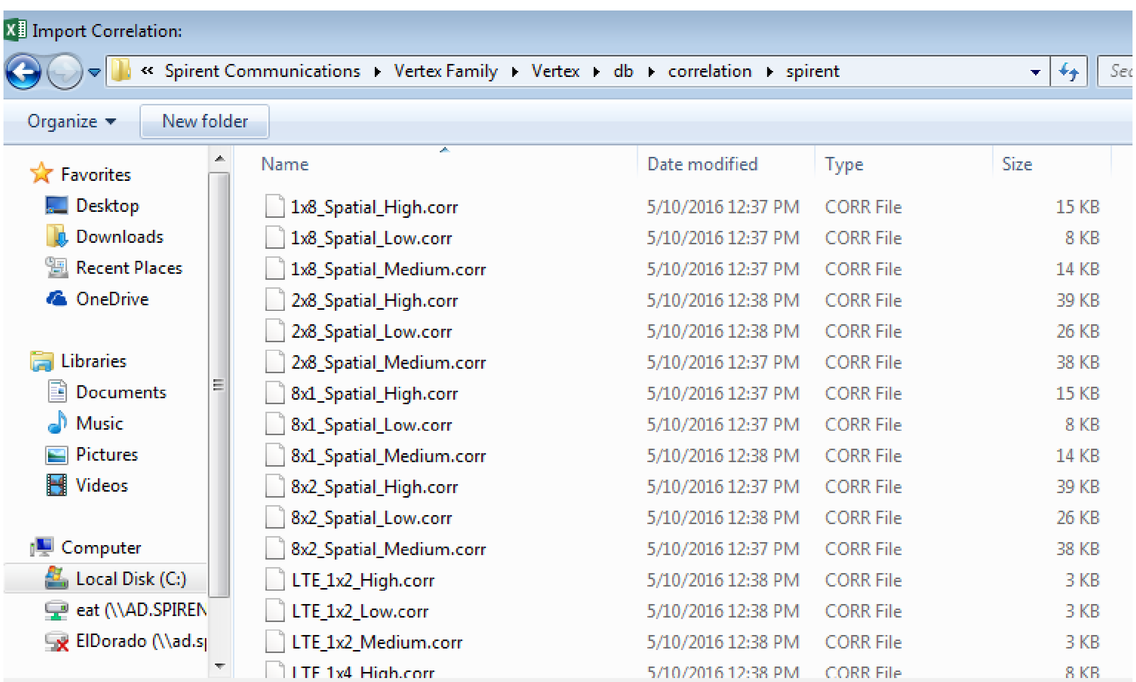


Figure 123. Import Correlation window – Selection .corr files.

XD5 Tab

The XD5 tab can only be seen when the connection setup is supported for XD5 application. For example, if you select **Quad 2x2 UniDirectional** connection setup, you will see the XD5 tab after the last Correlation tab. The XD5 tab is used to configure the uplink attenuation.

2.14.2.2 Sample DEE File

An example DEE File is shown in the following figures. Each figure displays a different tab in the DEE file.

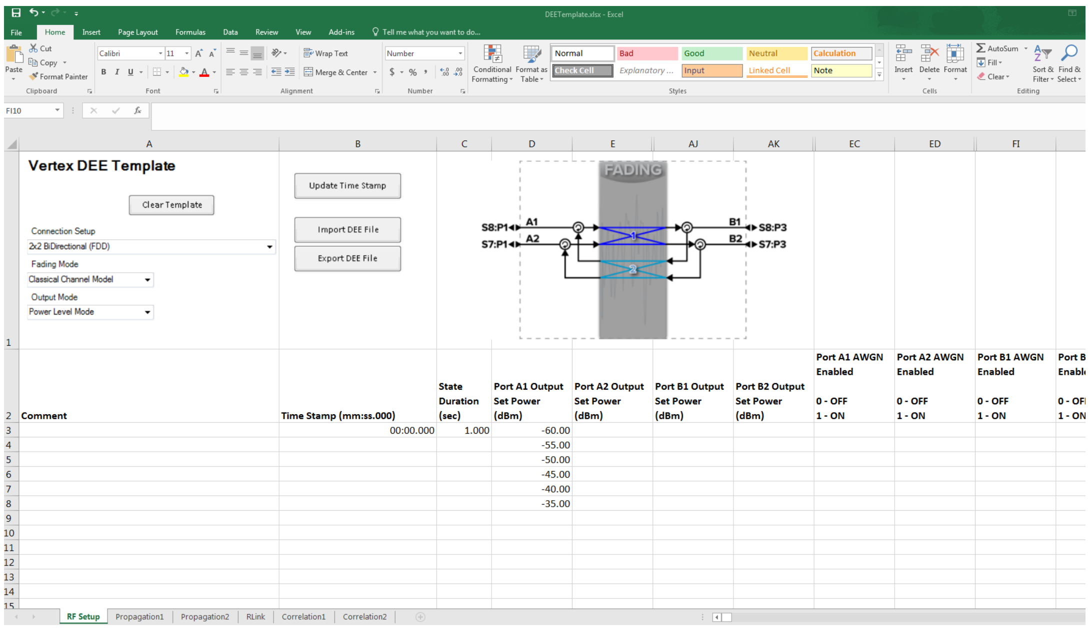


Figure 124. DEE template – sample 1.

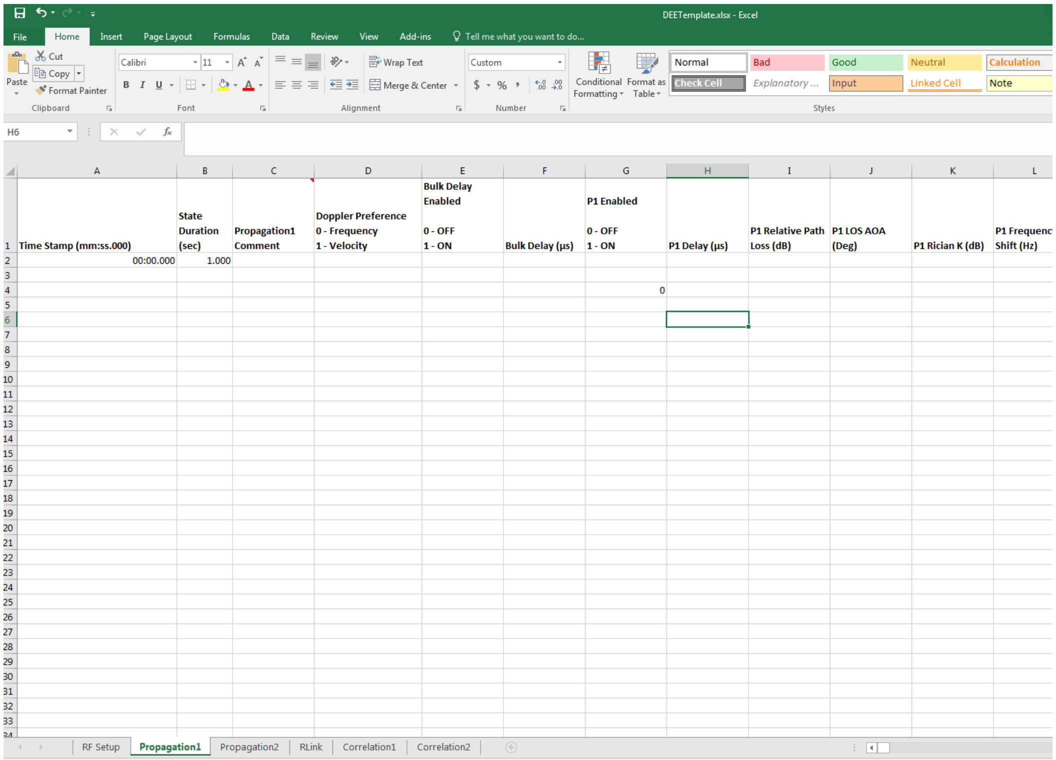


Figure 125. DEE template – sample 2.

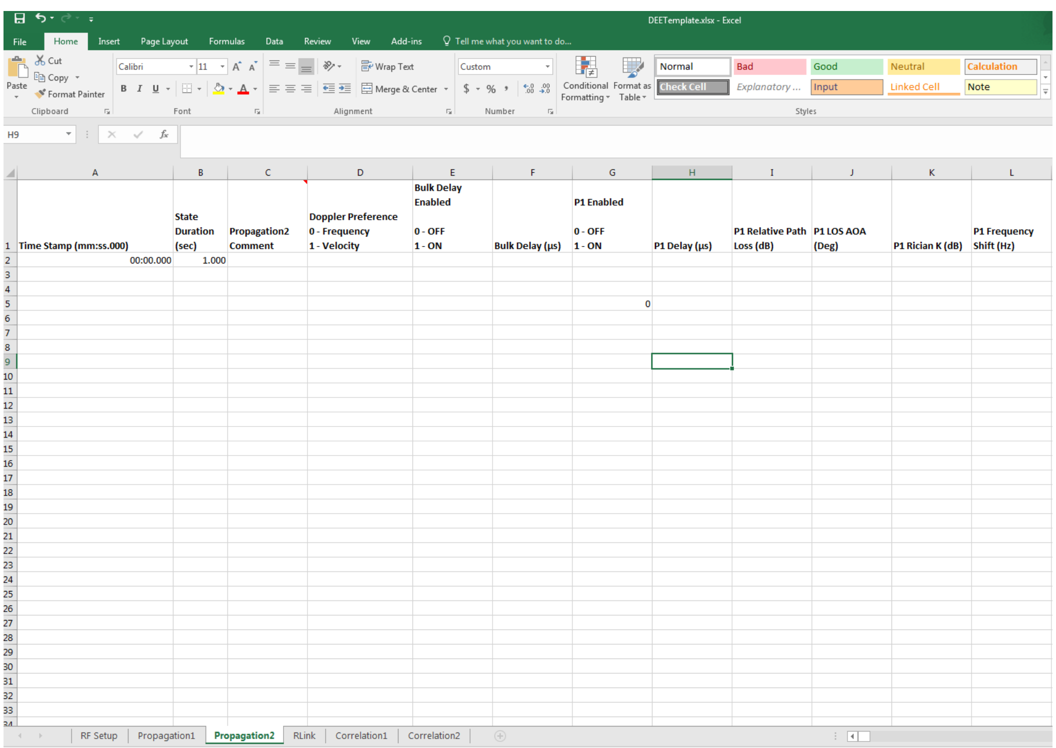


Figure 126. DEE template – sample 3.

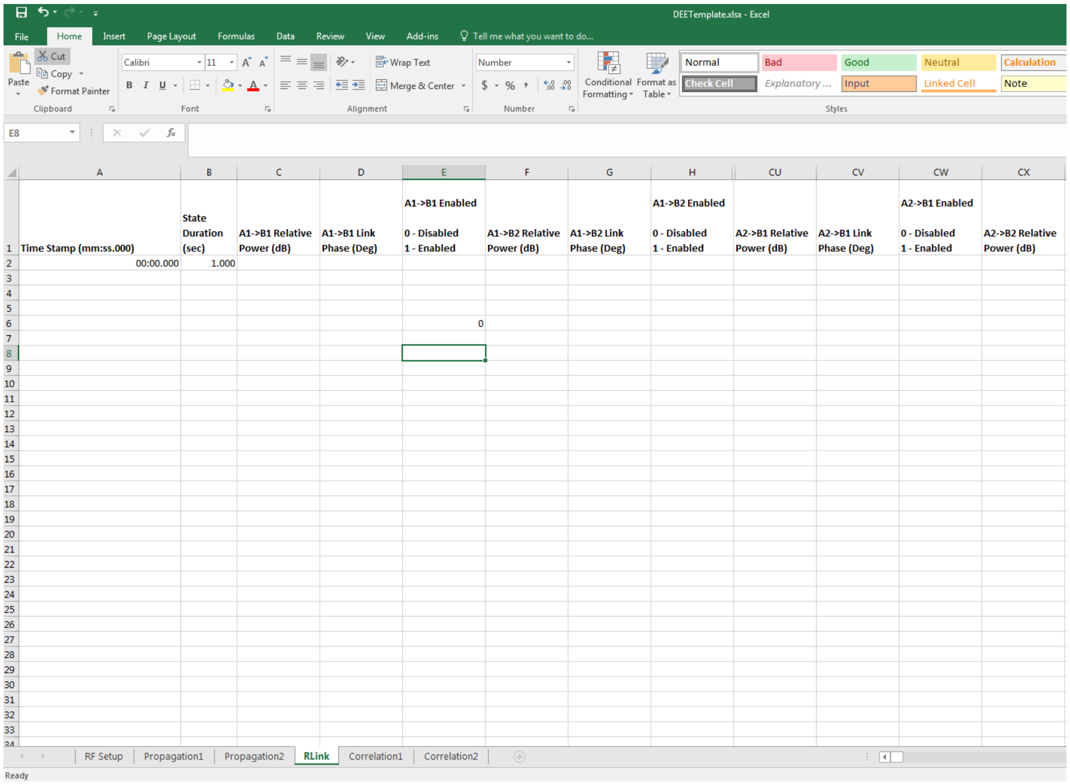


Figure 127. DEE template – sample 4.

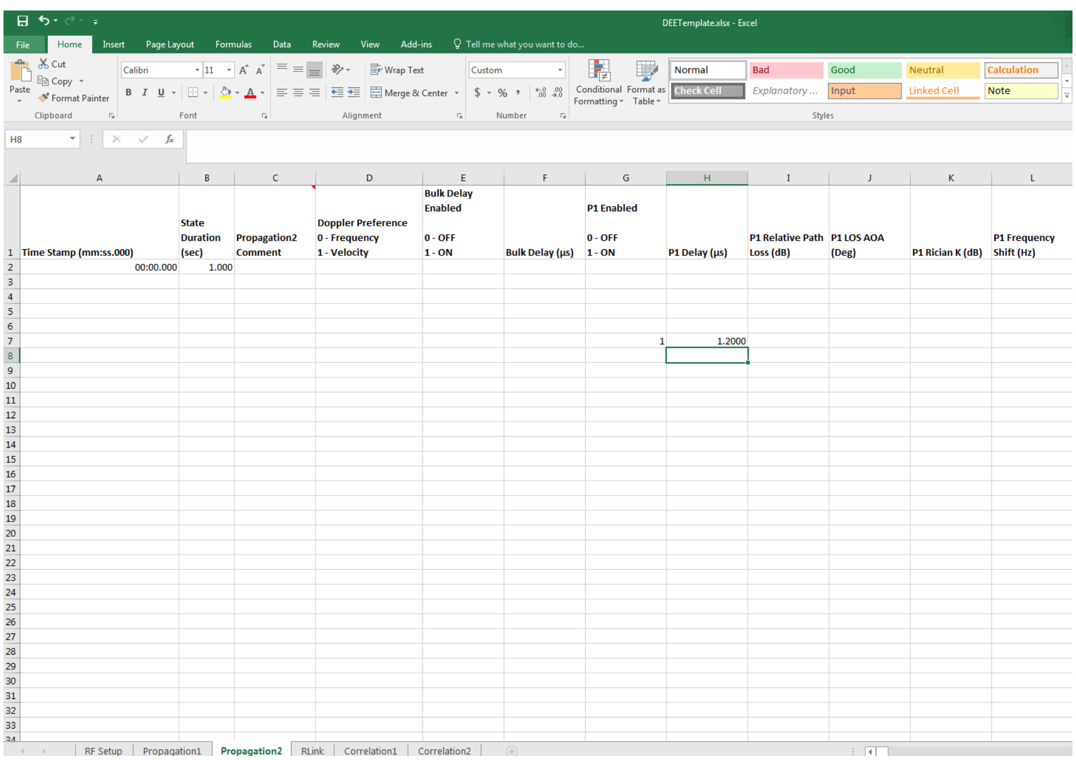


Figure 128. DEE template – sample 5.

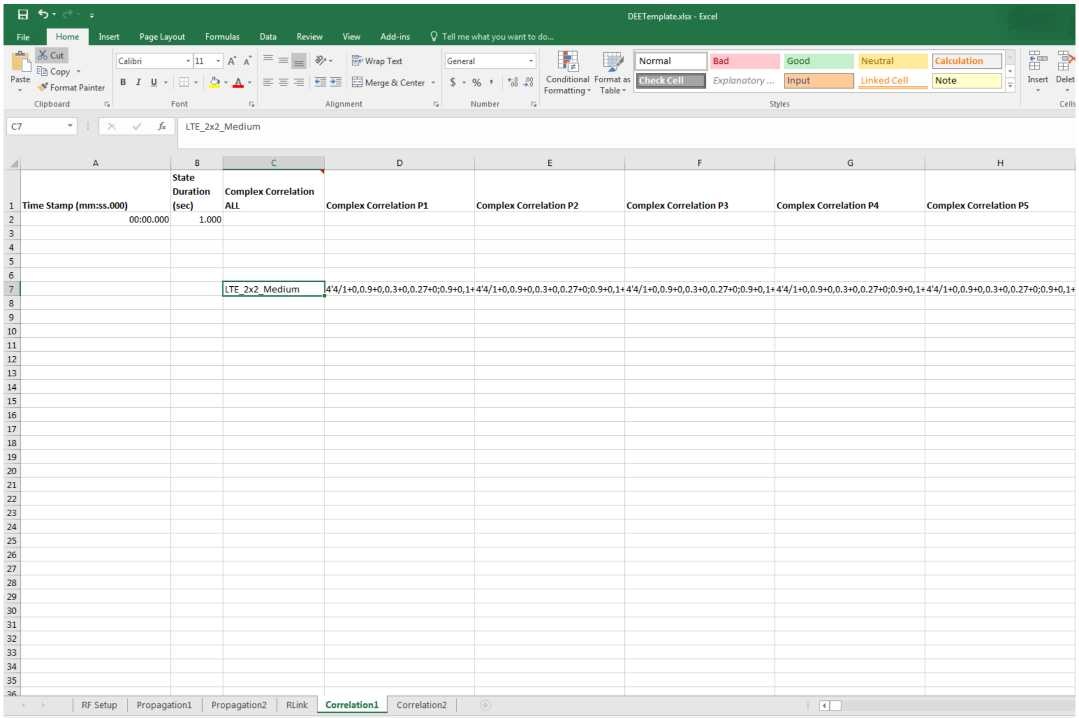


Figure 129. DEE template – sample 6.

> **NOTE**:
> The column title "Complex Correlation PX" is display-only. Do not attempt to modify correlation by editing this column.

This DEE file performs the following functions:

In State 1:

- Set State Duration to 1 second (each state duration thereafter remains 1 second unless the particular state is changed).
- Set the output power of port A1 to -60.00.
- All other parameters remain as defined in non-DEE mode.

In State 2:

- Modify the output power of port A1.

In State 3:

- Modify the output power of port A1.
- Turn Path 1 OFF (1 – ON, 0 – OFF) for propagation condition 1.

In State 4:

- Modify the output power of port A1.
- Turn Path 1 OFF (1 – ON, 0 – OFF) for propagation condition 2.

In State 5:

- Modify the output power of port A1.
- Disable Radio Link A1->B1

In State 6:

- Modify the output power of port A1.
- Turn Path 1 on and change the delay of Path 1 for propagation condition 2.
- Modify the correlation matrix associated with propagation condition 1.

To specify power loss instead of output power, use an Output Mode of Loss. The following figure shows the RF Setup tab for such a DEE file.

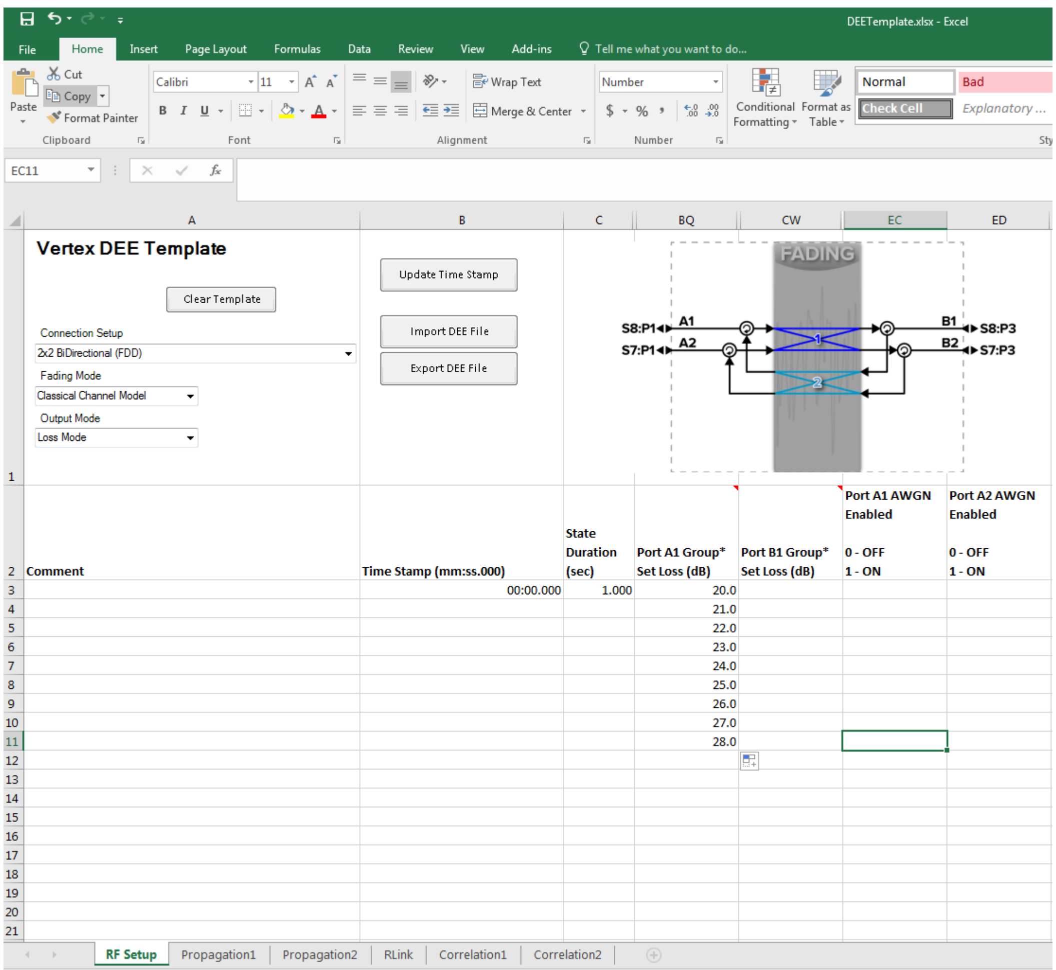


Figure 130. DEE template – sample 1 – Loss mode.

2.14.3. DEE in Detail

2.14.3.1 Selecting an Emulation File

Select the emulation file by clicking the **DEE File** button and selecting the file from the exported location, as shown in the following figure.

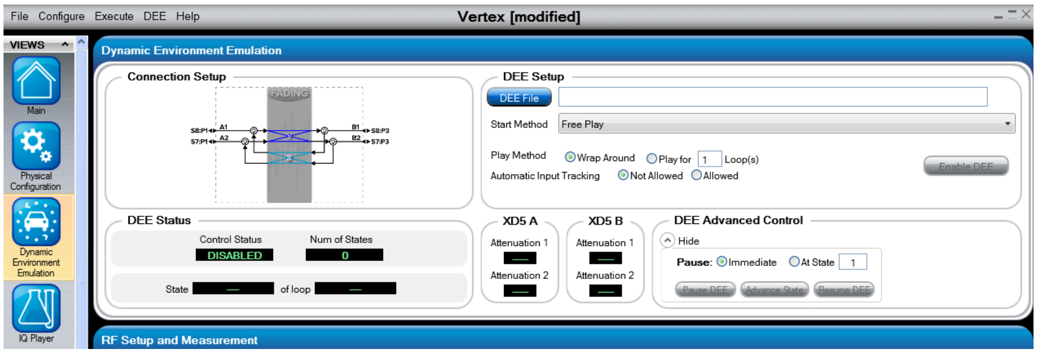


Figure 131. DEE Setup – Selecting the source file.

2.14.3.2 Enabling DEE

Download the DEE File to the Vertex instrument by clicking the **Enable DEE** button. When DEE is enabled, the following sequence of events occurs:

1. The setting of all non-DEE related parameters is disabled. You cannot modify parameters until DEE is disabled. Attempting to set a non-DEE related parameter results in an error message similar to the following figure.

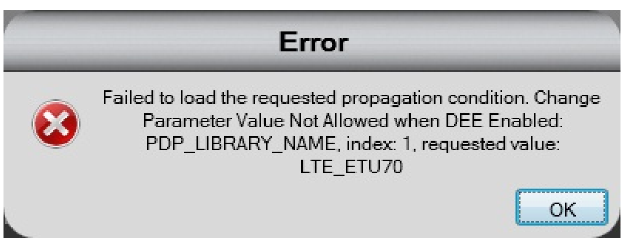


Figure 132. DEE error message.

2. The software compiles the DEE File into a machine-readable format. A Progress dialog box appears displaying the status of the DEE compile, as shown in the following figure.

   If the compile is successful, click the **Enter DEE** button.

   If the compile is unsuccessful, review the error messages provided to determine the nature of the error. Typically, errors are generated when parameters are set to invalid values.

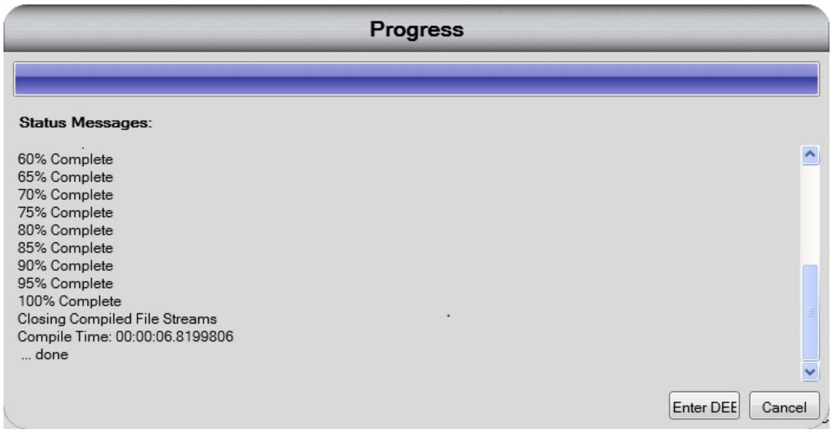


Figure 133. Compile status dialog box.

NOTE:
The Vertex skips this step if you have successfully compiled the file previously.

2.14.3.3 Playing (Running) DEE

After you enable DEE, and before you click the **Play** button, the player is stopped at Time 0 of State 1. When you click the **Play** button, the fading engine begins, and the DEE engine begins cycling through states specified in the DEE File.

2.14.3.4 DEE Status Information

You can monitor the progress of the DEE test through the Vertex GUI. The following information, shown in the following figure, is provided to the Vertex GUI from the DEE engine:

- Control Status – This indicates the status of the DEE Engine (Enabled or Disabled).
- State – This is the current state of the DEE file the Vertex is in.
- Loop – This indicates how many times the states have been looped.
- Total Time – This indicates the total time that DEE has been playing.

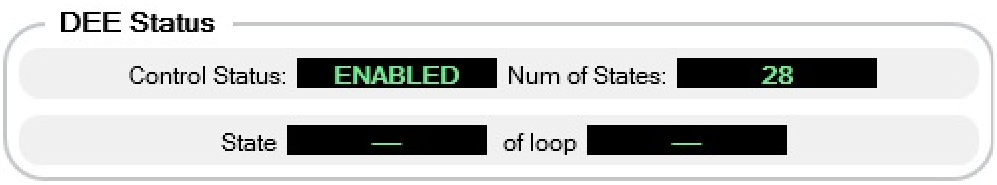


Figure 134. DEE Status panel.

NOTE:
The other views in the GUI are active and provide real-time feedback during the DEE simulation. For example, you can use the Channel Player view to view the real-time propagation conditions. An exception is the Propagation Editor in the Main view, which does not reflect changes during the DEE simulation.

2.14.3.5 DEE Advanced Control

The DEE Advanced Controls, shown in the following figure, can be used to debug a DEE test by playing certain states of the DEE file. Advanced Controls are only available when the Play method is set to “Wrap Around.”

The Advanced Controls allow you to hold the Vertex at a particular DEE state and play it back as in non-DEE mode. You may also step from one state to the next.

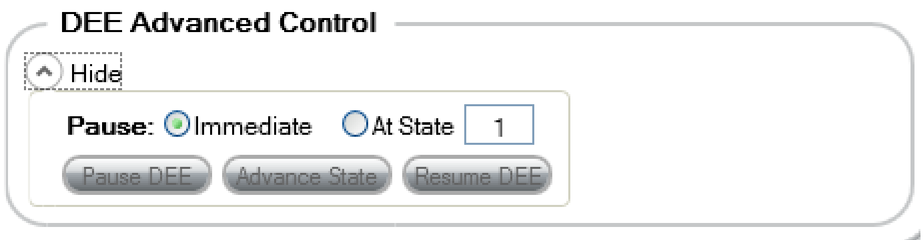


Figure 135. DEE Advanced Control panel.

- The **Pause DEE** button is used to hold the DEE simulation. If the Pause method is set to “Immediate,” DEE simulation pauses immediately after you click the **Pause DEE** button. If the Pause method is set to “Pause at State X,” and the **DEE Pause** button is clicked, DEE pauses when entering the defined state. Note that DEE only pauses on state X once, not on every loop. To have DEE pause on state X a second time, click the **Pause DEE** button again.
- The **Advance State** button advances the state to the next DEE state after a pause action has occurred. This allows you to manually step through each DEE state at a slow rate.
- The **Resume DEE** button continues the DEE simulation. Note that while paused, you can select the next state with the **Pause DEE** button. Clicking the **Resume DEE** button resumes DEE simulation until the selected state is entered, at which point simulation pauses again.

2.14.3.6 DEE Specifications

- Minimum State Duration:

  For Static and Rayleigh Fading type, the minimum state duration is 10ms. For Rician fading type, if LOS path is enabled, the minimum state duration for LOS path and K factor update is 100ms.

- Maximum State Transition Time:
  - RF Output Level Changes 2 ms. (Measured from start of state change to completion of state change.)
- All Other Changes 400 μs (Measured from start of state change to completion of state change.)
- DEE Trigger Characteristics:
  - Trigger Signal TTL, Rising Edge
  - Trigger Signal Power 3.3 V
  - Minimum Trigger Width 90 ns
  - Trigger Delay - from Trigger to
  - Change in Output Level < 5.0 ms
  - Change in other parameters < 1.0 ms

2.15. High-Speed RPI (HSPRI) Mode

2.15.1. Overview

High-Speed Remote Programming Interface (HSRPI) mode is an advanced mode of operation for the RPI to significantly improve the speed of command execution of the Vertex channel emulator.

HSRPI re-uses the existing RPI command interface and its familiar string-based command set.

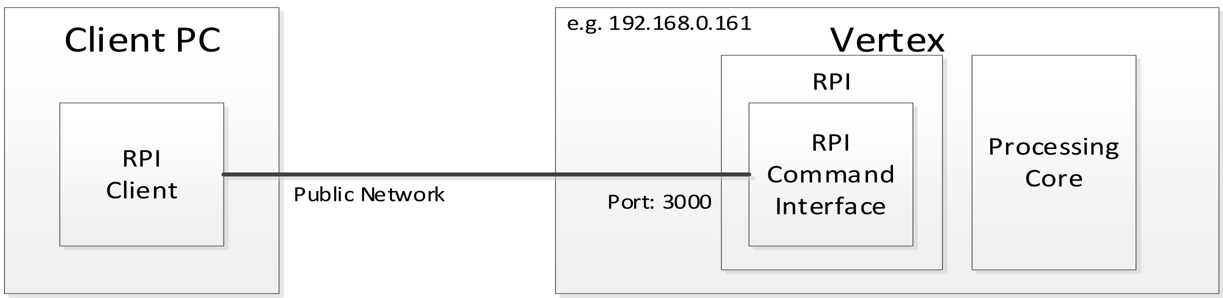


Figure 136. RPI Command Interface.

HSRPI is built on top of the Vertex's pre-existing DEE feature and uses the DEE engine. Therefore, it is subject to similar constraints as DEE and live-streaming DEE features.

Unlike traditional DEE, there is no requirement to pre-define and pre-compile the emulation states. Once you enter HSRPI mode, you will use standard RPI commands to control the supported emulation parameters.

2.15.2. Supported Emulation Parameters

With DEE as the underpinning engine for this feature, the same set of emulation parameters can be supported as with traditional DEE. For this initial release of HSRPI, a subset of the full DEE parameter set is available.

2.15.2.1 Channel Emulation Parameters

Every topology can be viewed as a set of one or more Channel Models with sub-components (paths and radio links), along with RF ports with input and output power controls. The following emulation parameters at each component level for HSRPI are supported:

- **Channel Model**:
  - Bulk Delay Enable State
  - Bulk Delay Value
- **Radio Link**:
  - Link Enable State
  - Relative Power Value
  - Phase Value
- **Path**:

  The following table provides the supported Path emulation parameters.

Parameter	Modulation Type		
	Static	Rayleigh	Rician
Path Enable State	Included	Included	Included
Relative Path Loss	Included	Included	Included
Fixed Delay	Included	Included	Included
Frequency Shift	Included	Included	Included
Doppler Frequency	---	Included	Included
Rician LOS AOA (doppler)	---	---	Included
Rician K-Factor	---	---	Included

- **RF Ports**:
  - Input Power Value
  - Output Power Value

2.15.3. Basic Operation

2.15.3.1 Before Entering HSRPI Mode

The functionality underpinning this mode of operation is the DEE engine (or more specifically the LSDEE engine). As with regular DEE mode there is a limited set of emulation parameters that may be edited once this mode is entered.

2.15.3.1.1 Preconfigure the Unit

You must preconfigure the following non-DEE parameters on the unit:

- Connection topology

  The topology will determine the channel models and associated parameters that will be configurable within HSRPI.

- Path modulation setting

  The path modulation type is not configurable once HSRPI mode is entered. Before entering HSRPI mode, you must select the path modulation types desired for all paths under all channel models.

- Any desired configurations for the initial emulation state

  When you enter HSRPI mode, the pre-existing state will be used as the initial state, with the caveat of mandatory constraints imposed as discussed in next section.

2.15.3.1.2 Mode Initial State

HSRPI re-uses the pre-existing state configured for the topology where possible. Certain configuration constraints must be imposed for HSRPI due to the underlying LSDEE engine. If associated changes are required, these changes will be made automatically when you enter HSRPI mode.

These configuration constraints include:

- Fading Mode: Classic Fading
- Output Mode: Power
- All Radio Links enabled;
- All paths set for:
  - Rician fading
  - Classic 6dB channel spectrum
  - Fixed Delay mode
  - Fixed Frequency Shift mode
  - Log Normal disabled
  - Cluster modelling disabled

2.15.3.2 Entering HSRPI Mode

The Remote Programming Interface command set is expanded to provide a new command for entering and exiting the HSRPI mode as follows.

Command String	**Parameter range**	**Default**
HSRPI[:STATe]	ON, OFF	OFF

To enter HSRPI mode, use the RPI command: “HSRPI:STATE ON”.

When Vertex receives this command, the following actions occur:

1. The Vertex is placed in HSRPI mode.
2. Initial conditions for the topology are configured.
3. Emulation is set to ***PLAYING***.

You can then send RPI commands as usual to control the emulation configuration.

The Vertex GUI is intentionally shut down during HSRPI. This is required to achieve the highest state transition speeds within HSRPI. The HSRPI context is one of automation, and the RPI client application is making the fading state changes and has all the required knowledge of the current state.

A dedicated HSRPI Information Form is provided to replace the Vertex GUI to indicate the mode of operation.

2.15.3.3 Using HSRPI mode

HSRPI uses the same command set as the regular RPI for configuring emulation parameters. Because of the underlying DEE engine in use, there are constraints on which parameters may be changed, as discussed in an earlier section.

The following section describes the associated command set.

2.15.4. Command Overview

This section lists the RPI command set supported when Vertex is in HSRPI mode. Refer to the RPI Command Reference Manual for further details on parameter ranges and usage.

2.15.4.1 PATH Control

Classic Fading model parameters are supported initially.

Parameter	Command Syntax	Range	Res	Default	Units
State	CHM#:PATH#[:STATe]	ON (1), OFF (0)		OFF	
Relative Path Loss	CHM#:PATH#:RPLoss	0 to 40	0.1	0	dB
Fixed Delay	CHM#:PATH#:DELay[:VALue]	0 to 100	0.0001	0	uS
Doppler Frequency	CHM#:PATH#:DFRequency	-4000 to -0.1, 0.1 to 4000*	0.01	41.7	Hz
Frequency Shift	CHM#:PATH#:FSHift[:VALue]	-4000 to 4000*	0.01	0	Hz
LOS Angle of Arrival (doppler)	CHM#:PATH#:LOS:AOA	0 to 360	0.1	0	Deg
LOS K-Factor	CHM#:PATH#:LOS:KRICian	-30 to 30	0.1	0	dB

* Doppler Frequency and Frequency Shift are coupled parameters. The Doppler frequency is dominant and constrains the range of Frequency Shift dynamically to ensure a combined max range of ABS(4000Hz).

2.15.4.2 Radio Link Control

Parameter	Command Syntax	Range	Res	Default	Units
State	RLINK:{AB,BA}##[:STATe]	ON, OFF		OFF	
Relative Power	RLINK:{AB,BA}##:RELPower	-40 to 40	0.1	0	dB
Phase	RLINK:{AB,BA}##:PHAse	-180 to 180	0.1	0	Deg

2.15.4.3 Channel Model Control

Parameter	Command Syntax	Range	Res	Default	Units
Bulk Delay State	CHM#:BULKdelay:STATe	ON, OFF		OFF	
Bulk Delay Value	CHM#:BULKdelay[:VALue]	5 to 4000*	1	5	uS

* Actual range will vary based upon the state of other system parameters including frequency, bidirectional connection setup, AWGN etc.

2.15.4.4 Port Power Control

Parameter	Command Syntax	Range	Res	Default	Units
Input Power	PORT:{A,B}#:INPut	-50 to -15*	0.01	-10	dBm
Output Port Power	PORT:{A,B}#:OUTPut	-120 to -20*	0.01	-60	dBm

* Range will vary based on connection setup, frequency and other system parameters.

2.16. Live Streaming Dynamic Simulation

2.16.1. Overview

Live Streaming Dynamic Environment Emulation (LSDEE) is an advanced feature addition to the Remote Programming Interface (RPI) of the Vertex Channel Emulator that provides a high-speed control mechanism for an extensive set of emulation parameters.

As its name suggests, LSDEE is built on top of the Vertex's pre-existing DEE feature, using the DEE engine. Unlike traditional DEE, there is no requirement to pre-define and pre-compile the emulation states or the inter-state duration.

LSDEE exposes a UDP server interface for streaming full DEE emulation state information. An LSDEE client must connect to the LSDEE server and provide LSDEE-specific datagrams. The LSDEE server parses control datagrams and updates the emulation accordingly.

The emulation state change execution speed is a function of the selected topology. The current state plays indefinitely until the client provides a new state.

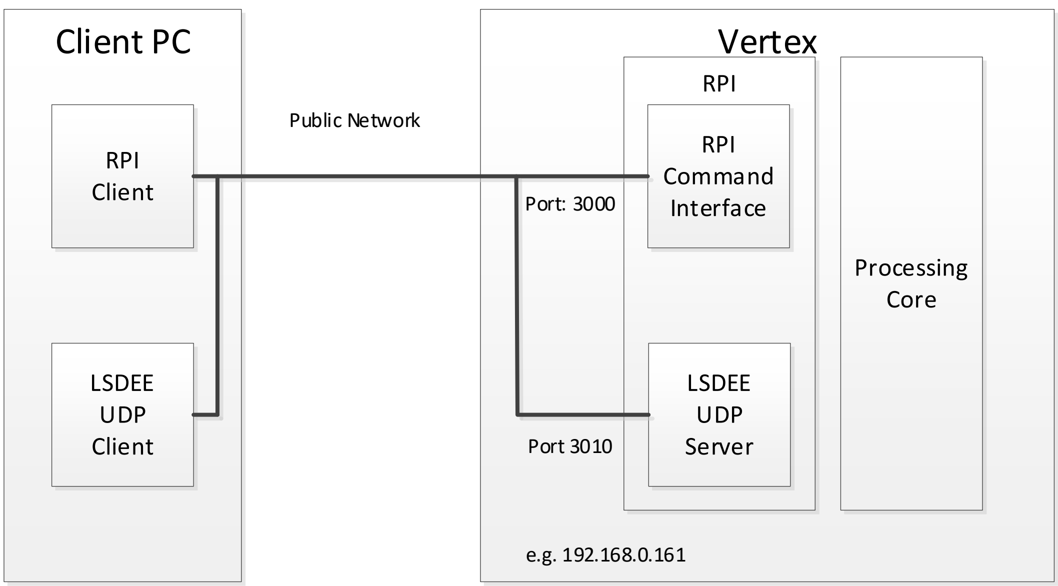


Figure 137. RPI Command Interface for LSDEE client and server.

The UDP datagram takes the form of a header block followed by a byte array payload whose content is a function of the command type, topology in use, and other considerations.

2.16.2. Supported Emulation Parameters

With DEE as the underpinning engine for LSDEE feature, the same set of emulation parameters can be supported as with traditional DEE. For the initial release of LSDEE, a subset of the full DEE parameter set is available.

2.16.2.1 Channel Emulation Parameters

You can view every topology as a set of one or more Channel Models with sub-components (paths and Radio links), along with RF ports with input and output power controls. The following emulation parameters at each component level are supported:

- **Channel Model**:
  - Bulk Delay Enable State
  - Bulk Delay Value
- **Radio Link**:
  - Link Enable State
  - Relative Power Value
  - Phase Value
- **Path**:

  The following table provides the supported Path emulation parameters.

Parameter	Modulation Type		
	Static	Rayleigh	Rician
Path Enable State	Included	Included	Included
Relative Path Loss	Included	Included	Included
Fixed Delay	Included	Included	Included
Frequency Shift	Included	Included	Included
Doppler Frequency	---	Included	Included
Rician LOS AOA (doppler)	---	---	Included
Rician K-Factor	---	---	Included

- **RF Ports**:
  - Input Power Value
  - Output Power Value

2.16.2.2 Frequency Parameters

Traditional DEE does not support changing frequencies. For LSDEE, where the user has a connected client, the functionality is provided to query and change the frequency from within the client-server context. The non-DEE details are handled internally to make this a minimally disruptive action.

2.16.3. Basic Operation

2.16.3.1 Before Entering LSDEE Mode

The functionality underpinning LSDEE is the DEE engine. As with traditional DEE mode, there are some constraints with respect to which emulation parameters may be edited once you enter this mode.

2.16.3.1.1 User Preconfiguring

Once you enter LSDEE mode, the regular RPI command set for configuring the box will be disabled (with the exception of certain LSDEE-specific commands defined later). Therefore, you will need to preconfigure:

- the connection topology

  The topology will determine the Channel Models and associated parameters that will be configurable within LSDEE.

- any desired initial emulation state configuration

  Upon entering LSDEE, the LSDEE server will use the pre-existing state as the initial state, with the caveat of mandatory constraints imposed, as discussed in the next section.

2.16.3.1.2 Mode Initial State

LSDEE will re-use the pre-existing state configured for the topology where possible. Certain configuration constraints must be imposed for LSDEE, and if associated changes are required, they shall be made automatically upon entering LSDEE mode. These constraints include:

- Fading Mode: Classic fading mode is supported in the initial LSDEE release.
- Output Mode: Power
- Doppler Configuration Preference: Frequency
- All paths set for:
  - Classic 6dB channel spectrum
  - Fixed Delay mode
  - Fixed Frequency Shift mode
  - Log Normal disabled
  - Cluster modeling disabled

In addition, path modulation selection is not dynamically configurable within LSDEE. Therefore, a predefined model is required. This is controlled by the Path Modulation Mode command. The default Path Modulation Mode for LSDEE is:

- Path 1 – Static
- Path 2 – Rician modulation
- Path 3 - Total paths available – Rayleigh modulation

From a modeling perspective, only a single Line of Sight (LOS) component is required for a given Radio Link. LOS components are available in the Static and Rician modulation options only. The intended use for this model is that only one of paths 1 or 2 should be enabled at a time. Rayleigh paths can be enabled as needed.

2.16.3.2 Entering LSDEE Mode

The Remote Programming Interface command set is expanded to provide a new command for entering and exiting LSDEE mode.

Note that the prototype name for this feature was Real-Time DEE. Hence the legacy “RTDEE” naming convention held over into the command set.

Command String	Parameter range	Default
RTDEE[:STATe]	ON, OFF	OFF

To enter LSDEE mode, use the RPI command: “RTDEE:STATE ON”.

When Vertex receives this command, the following actions occur:

1. The Vertex is placed in LSDEE mode.
2. Initial conditions for the topology are configured.
3. Emulation is set to ***PLAYING***.
4. The UDP server is launched and awaits a user client connection to all you to interact and control the topology’s emulation state.

The Vertex GUI is intentionally shut down during LSDEE. This is required to achieve the highest state transition speeds within LSDEE. The LSDEE context is one of automation, and your LSDEE client makes the fading state changes and has all the required knowledge of the current state.

A dedicated LSDEE information form is provided to replace the Vertex GUI. When you exit LSDEE, the initial state will be configured again and playing.

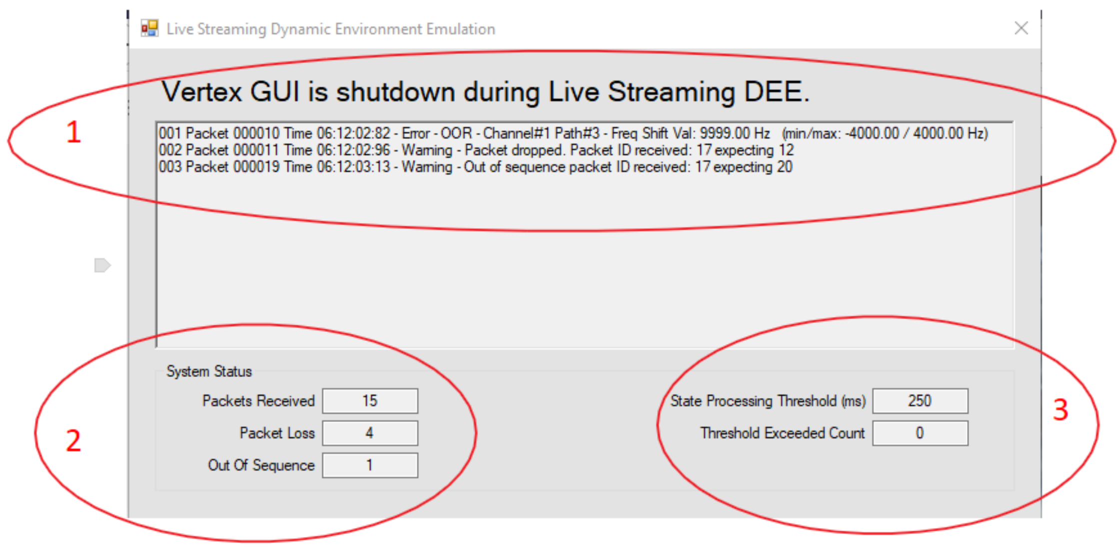
2.16.3.3 LSDEE Client

A client PC must launch a UDP client and connect on port 3010 to the RPI LSDEE UDP server. Once connected, the client can then send byte array datagrams to configure (and query) the state of the programmable emulation parameters for the set topology. The server will execute commands immediately and return acknowledgement upon completion. The server accepts UDP datagrams formatted for LSDEE commands only. The datagram form and content are discussed later in this section.

Spirent provides an example Python code base that facilitates an easier launch for a new user and provides a good reference point for the inner workings of the client-server pairing. The code is complemented by this manual for understating the protocol. Contact Spirent Support to receive the example Python code.

2.16.3.4 LSDEE GUI

When Vertex is in LSDEE mode, the Vertex GUI is shutdown for speed and consistency of state changes, and the LSDEE launches its own simpler GUI or Information Form. The following figure shows the Information Form and highlights some of the resources of the Information Form by intentionally forcing errors that would typically not occur. Under normal circumstances, you would typically expect only the Packets Received counter to increment.



LSDEE Information Form

Figure 138. Sample LSDEE Information Form.

In the preceding figure, the forced error sequence of packets intentionally created first a single out of range parameter instance, followed by mimicking a packet loss drop of 5 packets, and then a recovery of one of those packets to increment the out of sequence field.

The LSDEE Information Form contains the following components:

1. **Text box area**: This area is generally empty but will display any warnings and errors that occur during an LSDEE session. The information displayed in this area will mimic the LSDEE Log Events queue content.
2. **System Status area**: This area tracks the following UDP packet statistics:
   - **Packets Received**: This count reflects the number of commands (of all kinds) received by the server since the LSDEE session started.
   - **Packet Loss**: This count reflects an unexpected number in the message ID field, where an increment of 1 is expected from one command to the next. This field is expected to always be 0. In the preceding figure, Packet Loss is forced to an unexpected error value for illustrative purposes only.
   - **Out Of Sequence**: This count reflects an unexpected number in the message ID field where a previously assumed lost packet has arrived. This field is expected to always be 0. In the preceding figure, Out Of Sequence is forced to an unexpected error value for illustrative purposes only.
3. **State Processing Statistics**: This section shows the specified State Processing Threshold value (in ms) and how many times that threshold was exceeded.

2.16.3.5 LSDEE RPI Support

Once you enter LSDEE mode, the RPI interface will no longer support traditional Vertex configuration commands. There are several LSDEE-related commands available, some of which may be used while in LSDEE mode.

Command Hierarchy	Command	Description	Range
RTDEE	[:STATe] (optional)	LSDEE enable/disable	ON, OFF (default)
RTDEE	:PMMode	Path Modulation Mode Must use ***prior*** to LSDEE mode enable.	1: 1 static path, 1 Rician, remainder Rayleigh (currently only 1 mode supported)
RTDEE:LOG	:TTHreshold	Timing Threshold for state emulation command execution. Must use ***prior*** to LSDEE mode enable.	Positive integer; default 250ms
RTDEE:LOG:EVENts	[:QUERy]? (optional)	Query LSDEE Event Log	Event Log string
RTDEE:LOG:EVENts	:CLEar	Clear LSDEE Event Log	n/a

2.16.3.5.1 LSDEE enable/disable

This command is accepted while in LSDEE mode to allow exit of the mode.

2.16.3.5.2 Path Modulation Mode

Under LSDEE, the path modulation TYPE may not be changed once in LSDEE (as with DEE). Therefore, the path modulation type for each path is hardcoded upon entry in the mode. Only one profile is being supported – namely a single Static path, a single Rician path, and the remaining paths are Rayleigh. This allows for easy modeling with a single LOS component.

2.16.3.5.3 Logging – Timing Threshold

Timing Threshold (in mS) for state emulation command execution. This is a client developer-targeted resource. The average execution time of a given state-emulation command is dependent on the topology in play, and additionally there is some range of randomness around the average. A developer may wish to be informed when a specific timing threshold is exceeded. You can adjust the threshold with the RPI command prior to entering LSDEE mode. This threshold statistic is tracked on the LSDEE Information Form.

2.16.3.5.4 Logging – Events log

LSDEE will track events such as rejected commands (for example, out of range parameters), input power over-range issues, UDP protocol mismatches, and timing threshold breaches. These events are displayed in the Information Form and also in a query-able log events queue. A query of an empty log under Python will provide the following response:

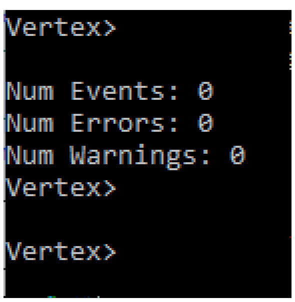


Figure 139. Query of empty log.

An Event is an instance of an error, warning, or informational data item passed to the log.

Taking the forced failure example provided in the previous section on the LSDEE GUI, the following figure represents the Log Event outputs for those errors. In this instance, the queue is queried once an event is detected and cleared once it has been read, so we are seeing these one at a time. Had the queue not been cleared, these events would have accumulated.

```
ERROR... val: 4

Num Events: 1
001 Packet 000010 Time 06:12:02:82 - Error - OOR - Channel#1 Path#3 - Freq Shift Val: 9999.00 Hz   (min/max: -4000.00 / 4000.00 Hz)
Num Errors: 1
Num Warnings: 0
Vertex>

Vertex>
send state... FAILURE!
WARNING... val: 1

Num Events: 1
001 Packet 000011 Time 06:12:02:96 - Warning - Packet dropped. Packet ID received: 17 expecting 12
Num Errors: 0
Num Warnings: 1
Vertex>

Vertex>
WARNING... val: 1

Num Events: 1
001 Packet 000019 Time 06:12:03:13 - Warning - Out of sequence packet ID received: 17 expecting 20
Num Errors: 0
Num Warnings: 1
Vertex>

Vertex>
```

Figure 140. Sample Log Event outputs for errors.

Note that for programmatic reading of the event log:

- The event summary is at the start, the event descriptions follow, and a final summary of error and warning count appears at the end.
- The event description list is limited to the first 100 items currently, but the error and warning counters will continue to increment.
- All log events are cleared upon receipt of a CLEAR command.

2.16.4. LSDEE Client – UDP Protocol

This section describes the details of the UDP protocol defined for LSDEE. Code for a working LSDEE client example implementation is provided by Spirent in addition to this document. The code is the best reference for interpreting the protocol. This section is a complementary description of the basics of the protocol.

The UDP datagrams consist of a fixed-format header block followed by an optional payload. The UDP protocol is a stimulus-response exchange between the user client and the Vertex LSDEE server. The client initiates an exchange and the server responds. The server response is an acknowledgement message with or without additional information. The same header format is used for incoming and outgoing commands.

2.16.4.1 Header Block

A simple header block is defined for use by both the client and server for command exchanges. It includes human-readable tags to assist with detection and readability within network packet analyzer programs.

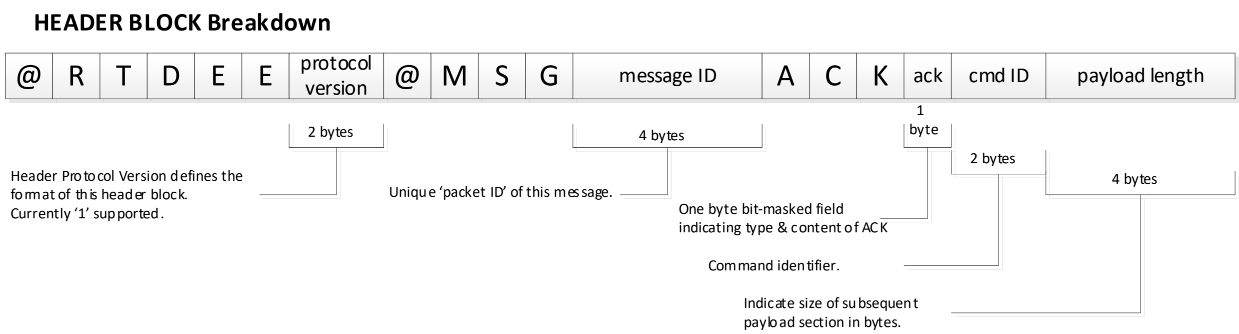


Figure 141. Header block breakdown.

2.16.4.1.1 Header Protocol Version Field

The header protocol version field defines the format and interpretation of the header bytes to follow. This allows for future variations to be defined. Currently only version 1 is supported.

2.16.4.1.2 Message ID Field

Every message from the client should have a unique message (or packet) ID and increment with each new command. The corresponding server response to a client command includes the same message ID in response.

The server tracks the packet IDs to ensure an incrementing counter. If the packet ID is something other than a single increment, a packet sequence warning will be provided to the client. If the packet ID advances by more than a single increment, this will be considered a packet drop. If the packet ID numbering regresses, this will be considered both an out of sequence packet and a recovered dropped packet. Packet sequence issues will not cause the command to be rejected. The statistics of packet sequence issues are tracked in the LSDEE Information Form.

2.16.4.1.3 ACK Field

The acknowledgement field is a bit-wise logical OR field. For protocol version 1, the ACK bits are interpreted as follows:

Bit Mask	Description	Interpretation
0x00	ACK_NONE	Incoming client commands can use this.
0x01	ACK_EXECUTION_COMPLETE	Server indicates command execution is complete.
---	reserved	
0x40	ACK_WARNING_INCLUDED	A 2-byte warning code is included in the response payload.
0x80	ACK_ERROR_INCLUDED	A 2-byte error code is included in the response payload.

For example, a server response ACK field value of 0x81 indicates the command execution is complete but with an error. The payload will begin with a 2-byte error code. The LSDEE Event Log can then be polled for additional information.

2.16.4.1.4 Command ID Field

The Command ID field contains the client command ID. The corresponding server response will duplicate the same command ID. Supported command IDs are as follows:

Command Code	Description	Interpretation
0	No Command	
1	PING	Simple sanity PING. Server responds with execution complete ACK.
2	QUERY_STATE	Query the active state of the LSDEE channel emulation. Server returns the byte array representation of the emulation state as used by the state control command, The format is dictated by the topology used.
3	QUERY_FREQUENCY	Query the active frequency profile. Server returns a byte array representation of the frequency profile for the topology.
20	SET_FREQUENCY_ALL	Configure the active frequencies of the topology with a byte array with appropriate frequency profile formatting.
40	SET_CLASSIC_FADING_STATE	Configure the active emulation state of the topology with a byte array with appropriate state profile formatting.

2.16.4.1.5 Payload Length Field

The Payload Length field indicates the number of bytes to follow as payload for incoming client commands or server responses.

2.16.4.2 Error Handling

Error codes are primarily used to indicate errors in the commands received that cause the command to be rejected. Most likely this will be a parameter out of range issue or a command header or payload protocol mismatch. In these instances, the command is ignored or rejected, but the LSDEE mode is maintained and the server stays alive.

The UNSPECIFIED error is assigned to unexpected or undefined internal errors in the execution of a command. If such an error occurs, this will cause an exit from LSDEE, and the server will be shut down.

In all error cases, the following actions will occur:

- The client will receive an indication of an error in the ACK field (if possible).
- The LSDEE Log Events queue will be informed.

The client should query the events log for any more detailed information that may be available on the warning.

In the case of an UNSPECIFIED error, in addition to the above actions, the RPI Listener Error Queue will be provide an error so that the user's RPI client can gain information on the reason for the LSDEE exit.

The following table provides the server response error codes.

Error Code	Error Name	Description	Action
0	NO_ERROR		None
1	UNKNOWN COMMAND	Unrecognized command ID.	Command is rejected.
2	INVALID LENGTH	Header payload length field mismatches with received payload byte count.	Command is rejected.
3	CMD_FORMAT_MISMATCH	Example is if the payload content did not match the expected number of bytes for the given command (based upon the topology).	Command is rejected.
4	OUT_OF_RANGE	A parameter of the command is out of range.	Command is rejected.
5	UNSPECIFIED	Unexpected or as yet undefined error.	LSDEE mode is forced to exit. Server will be shut down.

2.16.4.3 Warning Handling

Warnings will not prevent a command from being executed. This will include asynchronous alerts or non-critical synchronous events. No action is performed by the server in the warning context other than informing the client and updating the LSDEE events log and LSDEE Information Form, if appropriate. The client should query the events log for any more detailed information that may be available on the warning.

The following table provides the server response warning codes.

Warning Code	Warning Name	Description
0	NO_WARNING	
1	PACKET_SEQUENCE_ISSUE	Packet drop, out of sequence or other Packet ID unexpected numbering detected.
2	OVERLOAD_DETECTED	An Input Port power overload detection has occurred.
3	PROTECTION_TRIP_DETECTED	An Input Port power overload exceeded the protection circuitry threshold and triggered the protection of the port.

2.16.5. LSDEE Client – Emulation Control Payloads

2.16.5.1 Topology Dependency

The emulation parameters available for configuration are a function of the selected topology on Vertex. The topology selection therefore dictates the LSDEE UDP client command formatting for both emulation state and frequency configurations. The LSDEE client must be aware of the ‘profile’ of the current topology in order to correctly format control commands.

Every topology can be viewed as a set of one or more channel models with sub-components (paths and radio links), along with RF ports with input and output power controls. Non-TDD channel models typically also have independent frequency controls, with the exception of independent channel model topologies where radio links share a frequency.

The following figure shows an example of the profile information “fleshed out” for two topologies.

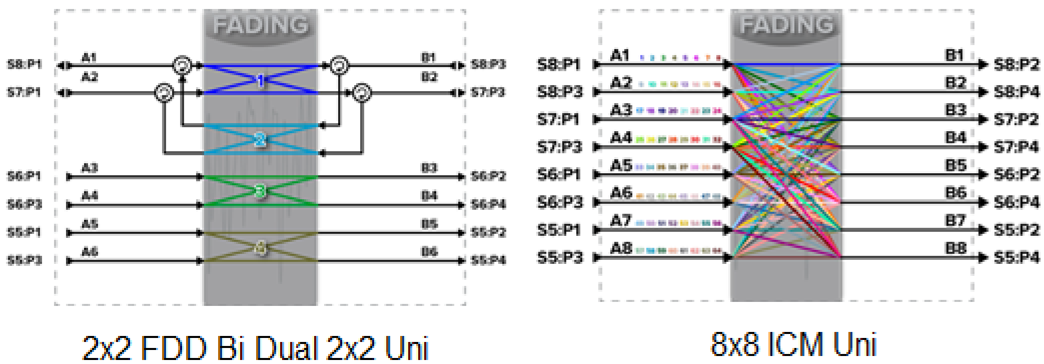


Figure 142. Example of profile information for two topologies.

Profile Parameters	**2x2 FDD Bi Dual 2x2 Uni**				**8x8 ICM Uni**
Number of Channel Models	1	2	3	4	All 64 have
Number of Radio Links per Channel Model of A-to-B Direction	4	0	4	0	1
Number of Radio Links per Channel Model of B-to-A Direction	0	4	0	0	0
Number of Paths per Radio Link	24	24	24	24	6
Number of Input Ports in the A-side	6				8
Number of Input Ports in the B-side	2				0
Number of Output Ports in the A-side	2				0
Number of Output Ports in the B-side	6				8
Number of Independent Frequencies to be Controlled	4				1

Knowledge of this topology profile is sufficient information for creating the UDP command payloads for controlling the Vertex under LSDEE. Most topology profiles can be determined easily from their connection diagrams. Alternatively, the RPI command set can assist to query these configuration details programmatically.

2.16.5.2 Fading State Format

As discussed in the preceding section, for any selected topology there will be one or more channel models. Each channel model will have associated radio links, and each link will have a set of paths. Under a given channel model, the configuration of the path set will be shared by all associated radio links.

A multi-channel topology will be represented as follows:

Channel Model 1

- Channel Model fields
- Associated AB Radio Link fields
- Associated BA Radio Link fields (if applicable)
- Associated Paths in order

Channel Model 2…

…

Channel Model N

Input Port fields for A-side input ports

Input Port fields for B-side input ports (if applicable)

Output Port fields for A-side output ports (if applicable)

Output Port fields for B-side output ports

The payload of a state command datagram to the UDP client presents channel models in sequence with their hierarchical internal breakdown. After channel models are described, the RF port powers are listed in order of ascending A-port numbers followed by B-port numbers, for input followed by output ports.

All subcomponents have a fixed byte format for a given fading model.

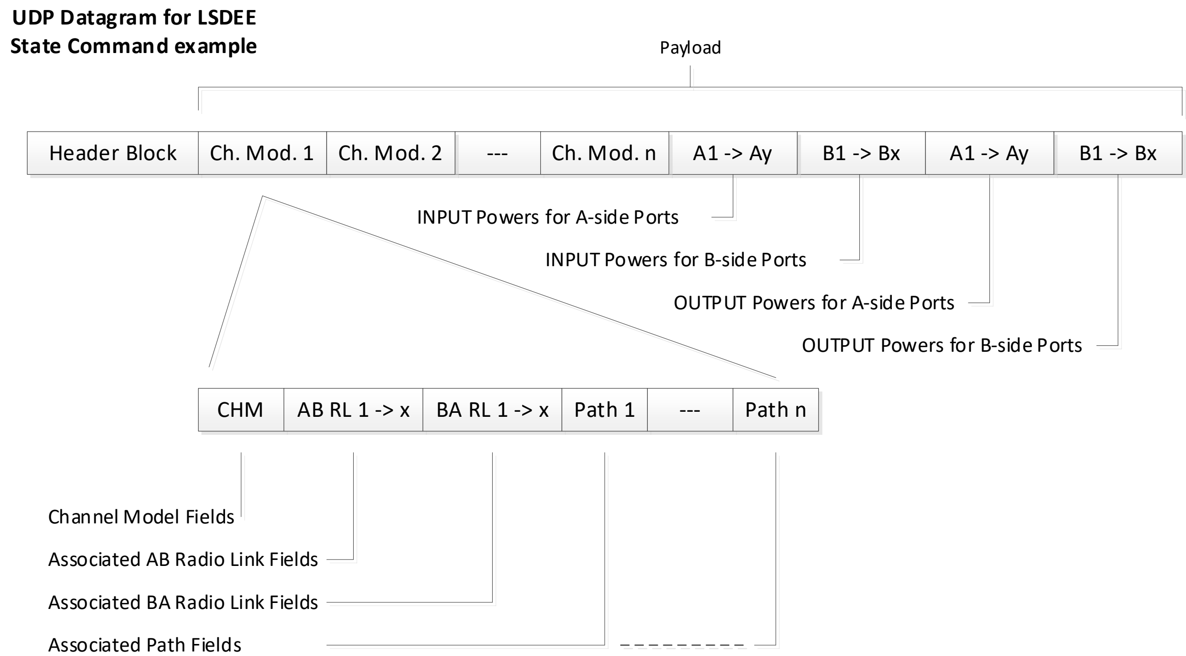


Figure 143. Fading state format.

The Classic Fading State command supported for LSDEE supports configuring all supported DEE parameters at once, or in traditional DEE terminology, changing the full ‘state’ of the emulation. If only a subset or even a single parameter is being changed, the other parameters will be sent with unchanging values.

A state query command is provided by the server. This can be used to retrieve the active state of the unit under LSDEE. It is available immediately upon entering LSDEE mode at which time it contains the initial state as defined by the unit’s configuration prior to entering LSDEE, along with any required LSDEE constraints that may have applied.

2.16.5.3 Frequency Format

Frequency control is provided within the client-server context in LSDEE to avoid the overhead of exiting and re-entering LSDEE mode, as would be required under traditional DEE. For the simpler topologies, the mapping of frequencies to the topology is typically a one-to-one with channel mappings. However, this does not apply across the board (for example, with ICM and other hybrid topologies).

To simplify the modeling for the client, the server exposes a frequency query command. This has the dual role of providing the current frequency settings of the unit and also providing the topology's frequency profile and associated command format.

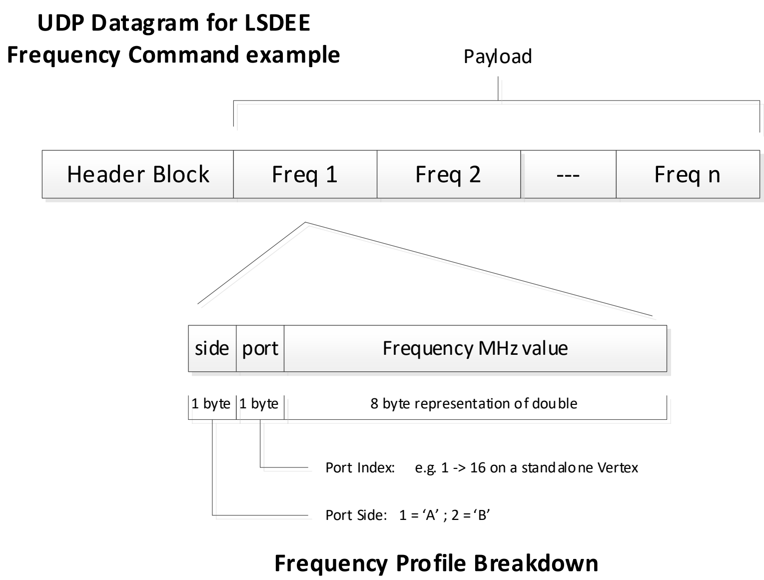
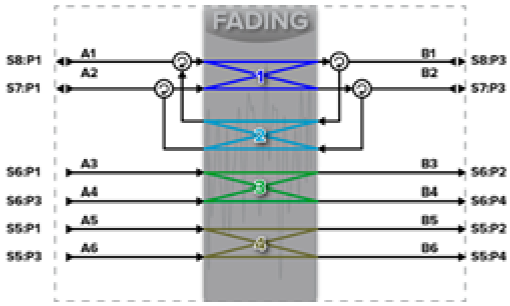


Figure 144. Frequency command example.

The format of the query response is the same as required by the server for configuring the frequencies. Therefore, the client can use the response as the client's frequency profile and byte array format. The client can then edit frequencies as required and resend the byte array to the server to update frequencies.

The port side and index refer to the ***first input port*** associated with that frequency grouping of ports. This is the only one that will be accepted, so simply re-using the query response eliminates any errors in the port ID.

Taking the 2x2_FDD_Bi_Dual_2x2_Uni topology again as an illustration of this, the following mappings apply:



2x2 FDD Bi Dual 2x2 Uni

Frequency ID	Port Side	Port Index	Port
1	1	1	A1
2	2	1	B1
3	1	3	A3
4	1	5	A5

2.16.6. LSDEE Client – Emulation Payload Components

2.16.6.1 Path Control

Parameters included in the path are a function of the Modulation type dictated:

Modulation	Enable/ Disable State	Relative Path Loss	Delay	Freq. Shift	Doppler Freq.	Rician LOS Angle of Arrival	Rician LOS k-factor
Static	Yes	Yes	Yes	Yes			
Rayleigh	Yes	Yes	Yes	Yes	Yes		
Rician	Yes	Yes	Yes	Yes	Yes	Yes	Yes

Datagram command format and order:

Parameter	Range	Res	Default	Units	Command Bytes
State	1 (ON), 0 (OFF)		0		1
Relative Path Loss	0 to 40	0.1	0	dB	8
Fixed Delay	0 to 100	0.0001	0	uS	8
Frequency Shift	-4000 to 4000*	0.01	0	Hz	8
Doppler Frequency	0.1 to 4000*	0.01	41.7	Hz	8
LOS Angle of Arrival (doppler)	0 to 360	0.1	0	Deg	8
LOS K-Factor	-30 to 30	0.1	0	dB	8

* Doppler Frequency and Frequency Shift are coupled parameters. The Doppler frequency is dominant and constrains the range of Frequency Shift dynamically to ensure a combined maximum range of ABS (Maximum Doppler in Hz). Maximum Doppler is a function of the Doppler Frequency Mode setting (4kHz or 12kHz).

2.16.6.2 Radio Link Control

Datagram command format and order:

Parameter	Range	Res	Default	Units	Command Bytes
State	ON, OFF		OFF		1
Relative Power	-40 to 40	0.1	0	dB	8
Phase	-180 to 180	0.1	0	Deg	8

2.16.6.3 Channel Model Control

Datagram command format and order:

Parameter	Range	Res	Default	Units	Command Bytes
Bulk Delay State	ON, OFF		OFF		1
Bulk Delay Value	5 to 4000	1	5	uS	8

2.16.6.4 Port Power Control

Datagram command format:

Parameter	Range	Res	Default	Units	Command Bytes
Input Port Power	-50 to 15*	0.01	-10	dBm	8
Output Port Power	-120 to -20*	0.01	-60	dBm	8

* Actual range varies based upon the state of other system parameters including frequency, bidirectional connection setup, etc.

2.16.6.5 Frequency Control

Datagram command format and order:

Parameter	Range	Res	Default	Units	Command Bytes
Port Side	1, 2 (for 'A' or 'B' resp.)				1
Port Index	1 -> maximum number for topology				1
Frequency	30 to 5925	0.001	2600.0	MHz	8

2.16.6.6 Double-to-Byte Array Format

For LSDEE commands, the conversion from parameters of type double to their byte array equivalent follows the IEEE 754 specifications. This provides an 8-byte array representation. Little-endian storage is used for doubles and throughout byte arrays (least significant byte first).

2.17. IQ Play

2.17.1. Overview

IQ Playback allows you to supply the fading sample data to be applied to the channel. In normal operation, Vertex internally generates the fading modulation signals based on the parameters configured by the user. These signals are used to modulate the user signal and produce the desired channel effects.

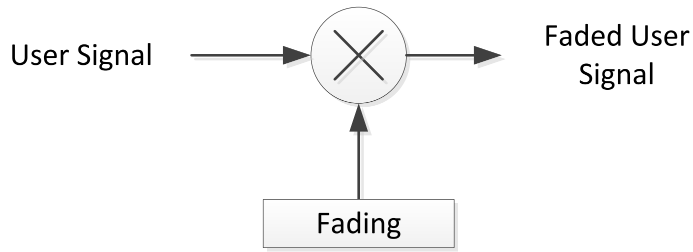


Figure 145. Example of user signal modulated with fading generated from internal fading engine of Vertex.

IQ Playback allows you to disable the internally generated fading in Vertex and supply your own fading sample data.

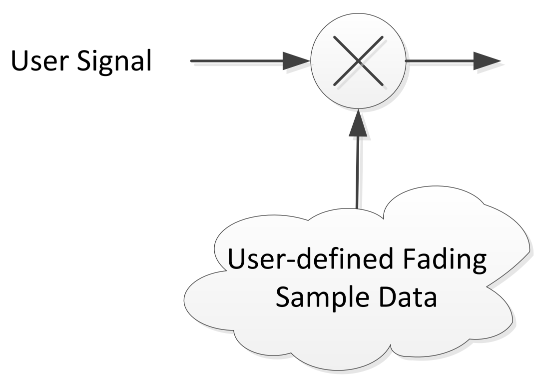


Figure 146. Example of user signal modulated with fading spooled from user-defined sample data.

In IQ Playback, you can change the following parameters:

- Delay
- I data
- Q data

2.17.2. Using IQ Playback

2.17.2.1 Generating Text Input

The first step in using IQ Playback is generating the text input file that defines:

- which radio links will play back the sampled fading
- the sample data

The text file must have the following format.

[Spirent IQ Playback File]											
Version = 1.0.0											
Configuration = A1-B1:1;A2-B2:1;A3-B3:1;A4-B4:1											
Upsample Factor = 2											
[Fading Sample Data]											
1.00	1.00	0.00	1.00	1.00	0.00	1.00	1.00	0.00	1.00	1.00	0.00
1.00	0.70	0.70	1.00	0.70	0.70	1.00	0.70	0.70	1.00	0.70	0.70
1.00	0.00	1.00	1.00	0.00	1.00	1.00	0.00	1.00	1.00	0.00	1.00
1.00	-0.70	0.70	1.00	-0.70	0.70	1.00	-0.70	0.70	1.00	-0.70	0.70
1.00	-1.00	0.00	1.00	-1.00	0.00	1.00	-1.00	0.00	1.00	-1.00	0.00

The file must begin with the tag line **[Spirent IQ Playback File]** to signify to Vertex that this is an IQ Playback file and not a generic text file.

On line 2, the **Version** must be specified. Since this is the 1.0.0 release of IQ Playback, only **1.0.0** is currently supported. Future versions may be specified.

On line 3, the **Configuration** must be specified. It is essentially a semi-colon separated list of radio link path elements consisting of an input port (for example, A1), a dash, an output port (for example, B1), a colon, and then a comma separated list of paths (for example, 1,2,3).

The Configuration syntax is as follows:

configuration:

Configuration = configuration-list

configuration-list:

configuration-element ; configuration-element;…

configuration-element:

radio-link : path-list

radio-link:

radio-link-designator – radio-link-designator

radio-link-designator:

{ A1 | A2 | A3 | … | B1 | B2 | B3 | … }

path-list:

path-num, path-num, …

path-num:

{ 1-24 }

The following figure shows an example of an IQ playback file displayed in the Notepad editor:

```
[Spirent IQ Playback File]
Version = 1.0.0
Configuration = A1-B1:1;A2-B1:1;A1-B2:1;A2-B2:1
Upsample Factor = 2
[Fading Sample Data]
0.0 0.7071  0.0000  0.0 0.7071  0.0000  0.0 0.7071  0.0000  0.0 0.7071  0.0000
0.0 0.5000  0.5000  0.0 0.5000  0.5000  0.0 0.5000  0.5000  0.0 0.5000  0.5000
0.0 0.0000  0.7071  0.0 0.0000  0.7071  0.0 0.0000  0.7071  0.0 0.0000  0.7071
0.0 -0.5000 0.5000  0.0 -0.5000 0.5000  0.0 -0.5000 0.5000  0.0 -0.5000 0.5000
0.0 -0.7071 0.0000  0.0 -0.7071 0.0000  0.0 -0.7071 0.0000  0.0 -0.7071 0.0000
0.0 -0.5000 -0.5000 0.0 -0.5000 -0.5000 0.0 -0.5000 -0.5000 0.0 -0.5000 -0.5000
```

Figure 147. Sample IQ playback file displayed in Notepad.

2.17.2.2 Minimum File Size

There is a minimum file size requirement of 10000 samples. Excessive looping of the data occurs with files containing less than 10000 samples.

2.17.2.3 Setting the IQ Play File on Vertex

To open the IQ Player View, click the  icon. The IQ Player View is shown in the following figure.

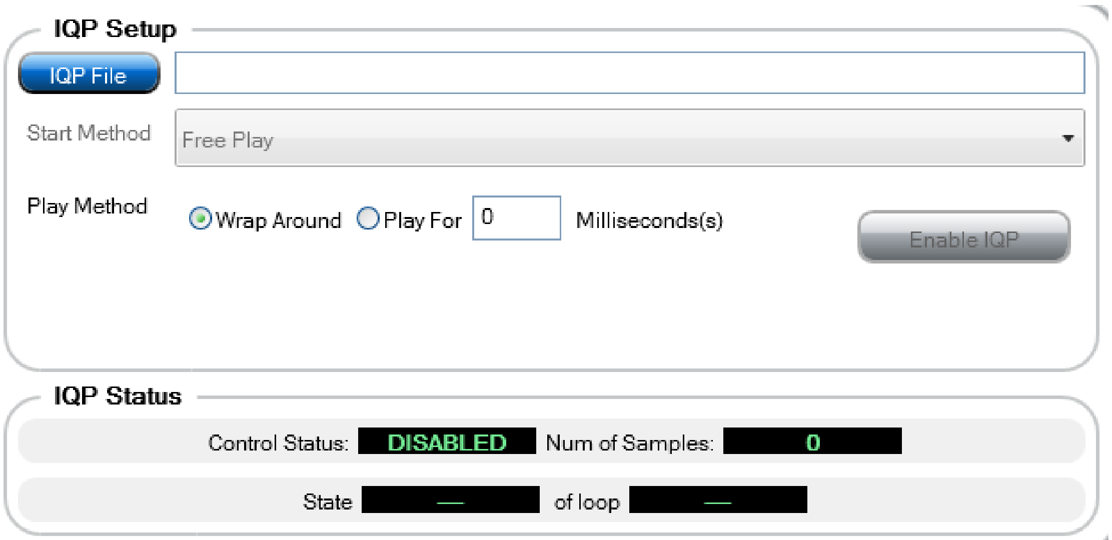


Figure 148. IQ Player view.

Click the **IQP File** button to browse to the text file that contains the fading data you want to play. When a valid file is selected, the file is parsed. If the file is valid, the **Enable IQP** button becomes enabled and the Number of Samples field is updated. The following figure shows a valid IQP file selected with 19200 samples.

You can set the Play Method to **Wrap Around**.

You can also use the **Play for** method to control IQ play for a specific amount of time (in milliseconds). If you select the **Play For** option button, set a value in the corresponding text box, and then click the **Enable IQP** button, each time you play the fading, it will pause after the time duration you specified. If you continue to play, the IQ play starts at the pause point and plays for another duration.

If you want to change the duration for each play, you must disable IQ play and then change the value.

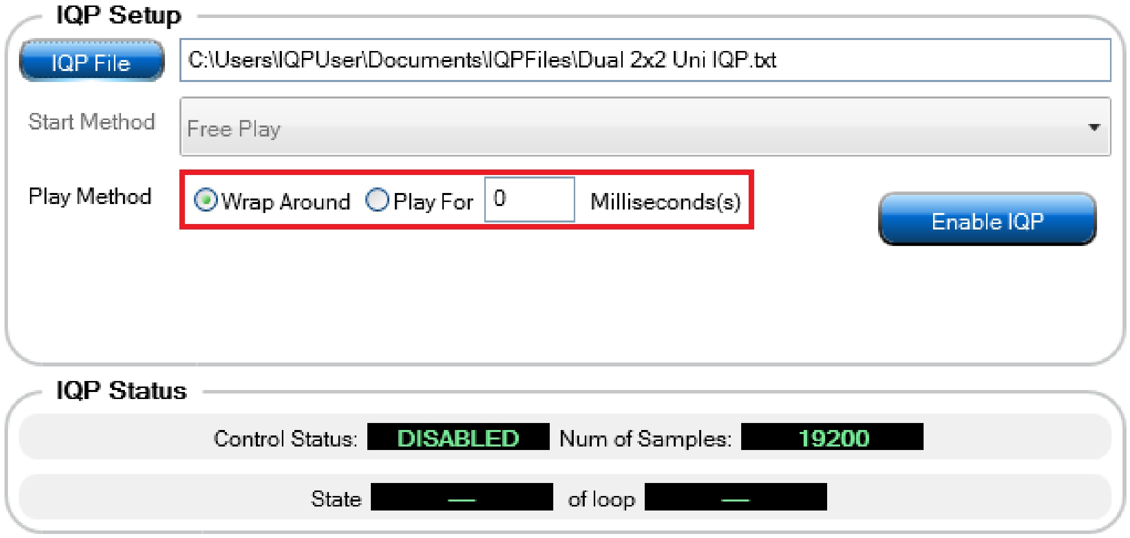


Figure 149. Valid IQP file with 19200 Samples.

The IQ Play Compiler Progress dialog box compiles the fading sample data to a binary format that resides on Vertex after you click the **Enable IQP** button. The following figure shows a sample IQ Play Compiler Progress dialog box.

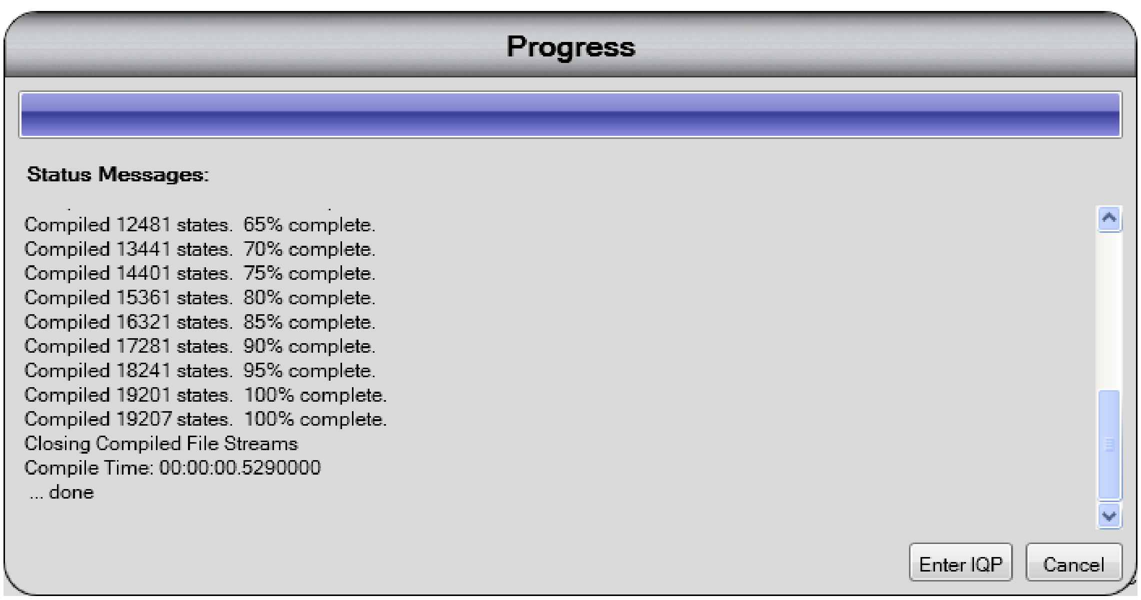


Figure 150. Sample compiler Progress dialog box.

After a successful compile, click the **Enter IQP** button to enter IQ Playback mode. You can click the **Cancel** button to return to the Vertex window with the compiled binary data still residing on Vertex.

When you are in IQ Play mode, click the **Play** button at the bottom of the main Vertex window to apply the fading sample data.

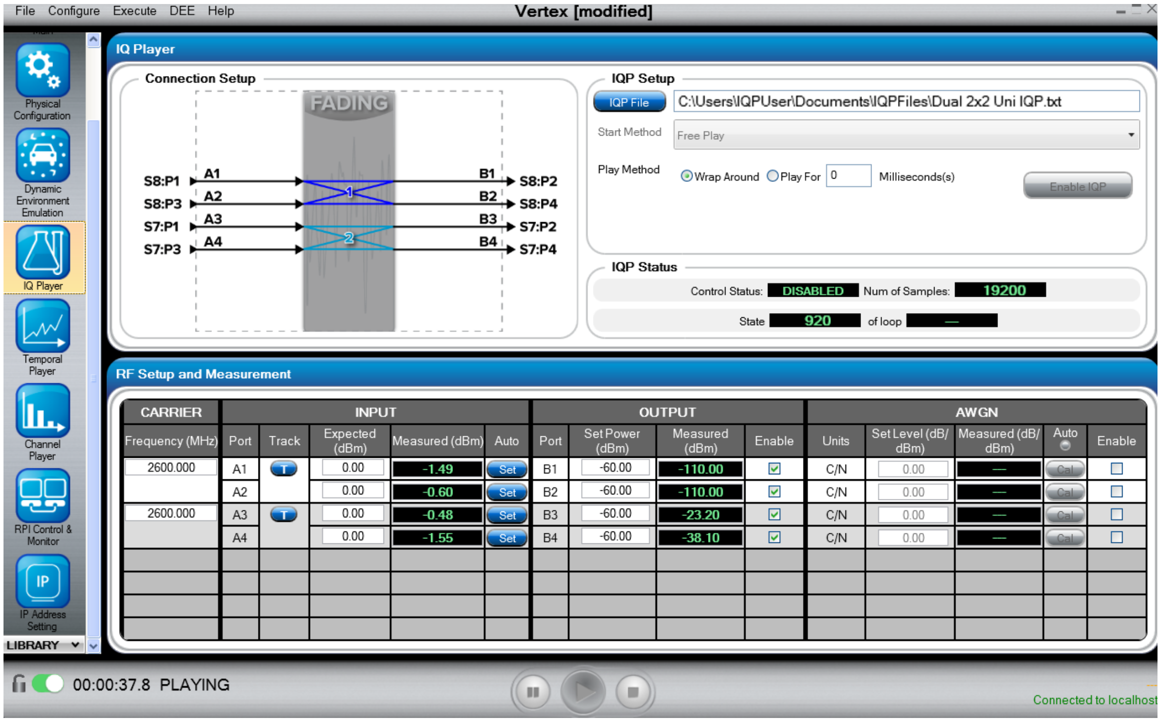
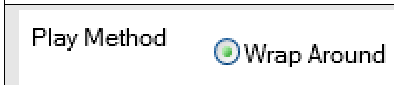
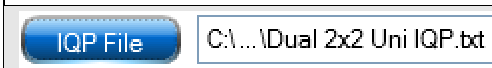


Figure 151. IQ Playback.

2.17.2.4 Controlling IQ Playback

The following table provides details about the controls available in IQ Playback mode. When IQ Playback mode is enabled, no other parameters can be edited. You must set up your levels and radio link parameters before entering IQ Playback mode.

Function	Description
	**Only Free Play is supported at this time.** Free Play – Play the fading sample data when the **Play** button is clicked. Triggered (Future Available)– Play the fading sample data when a trigger is received on the BNC port.
	When the end of the fading sequence is reached, wrap around to the beginning and loop continuously.
	Play for a certain amount of time (in milliseconds).
	Specify the text file that contains the fading sample data you want to use.
	Enter IQ Playback mode. The Vertex software will check to see if the file has already been compiled and if the binary files exist. If a compile is not needed, IQP will be enabled. If a compile is needed, the Compile dialog box will appear, and the binary files will be generated on the Vertex instrument.
	Displays “DISABLED” when IQ Playback mode is disabled and “ENABLED” when IQ Playback mode is enabled.
	Displays the number of samples in the IQ Playback text file.
	State box - When the fading sample data is playing, the State box is updated with the current sample number. of loop box - Displays the number of times the fading scenario has wrapped around.
	Click the **Play** button to begin playback of the fading scenario (after compilation). Click the **Pause** button to pause playback at a given sample. When paused, the radio link becomes a static pass-through channel. From the paused state, clicking the Play button will continue the fading scenario from the current sample. Click the **Stop** button to stop the fading playback and reset to the beginning of the sample file.
	Exit IQ Playback mode. You can then change other parameters and use the built-in fading generator.

2.17.2.5 Sample Files

The following set of sample files are installed with the Vertex software in **C:\Program Files (x86)\Spirent Communications\Vertex\IQ Playback Sample Files**:

- 1x1 Uni 1 path.txt
- 2x2 Bi 1 path.txt
- 2x2 Uni 1 path.txt
- 2x2 Uni 3 path with delays.txt
- 2x2 Uni 3 path.txt
- 4x4 Uni 1 path.txt

There following Matlab script file and output file are also installed:

- IQplay_example.m
- IQplay_example.txt

You can modify the Matlab script to generate custom IQ Playback files.

2.17.3. Sample Generation

Each radio link specified in the sample file can be assigned a stream of I and Q fading samples with one sample per row. For example, consider the fading sequence described by the playback file shown in the following table.

[Fading Sample Data]		
0.0	1.0000	0.0000
0.0	0.7071	0.7071
0.0	0.0000	1.0000
0.0	-0.7071	0.7071
0.0	-1.0000	0.0000
0.0	-0.7071	-0.7071
0.0	0.0000	-1.0000

This is a simple fading sequence describing a phasor rotating clockwise around the unit circle advancing $\frac{\pi}{4}$ radians per sample. The plot diagram shown in the following figure shows the modulation sequence (repeated over 3 cycles).

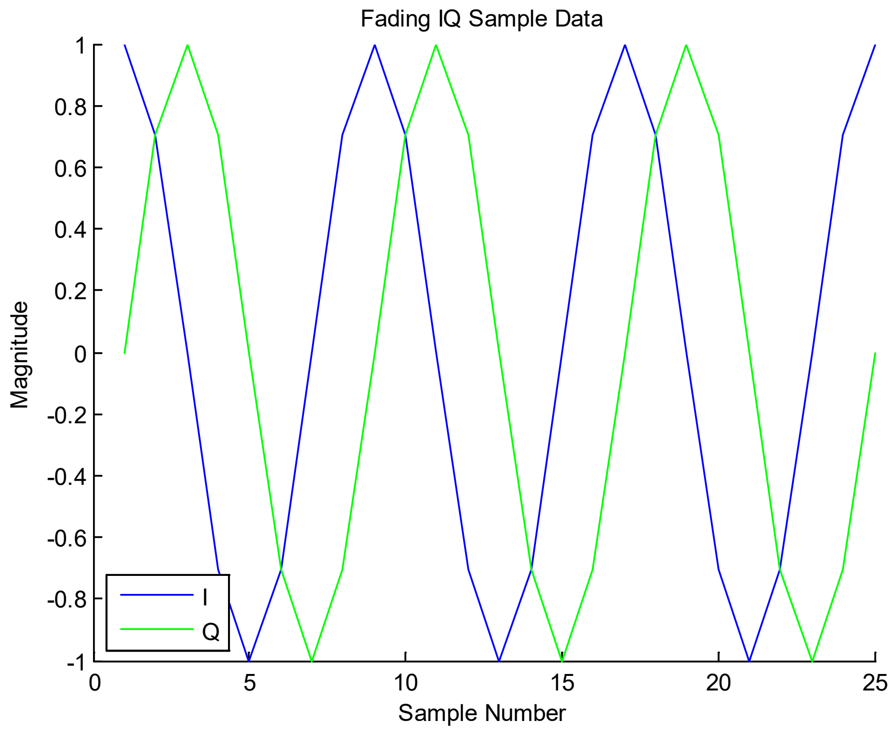


Figure 152. A simple modulation sequence.

Now, consider the case where the user input is a tone at the Carrier Frequency (Fc). The output of this fading sequence is a tone offset from Fc by the frequency of the sinusoid represented in the following figure.

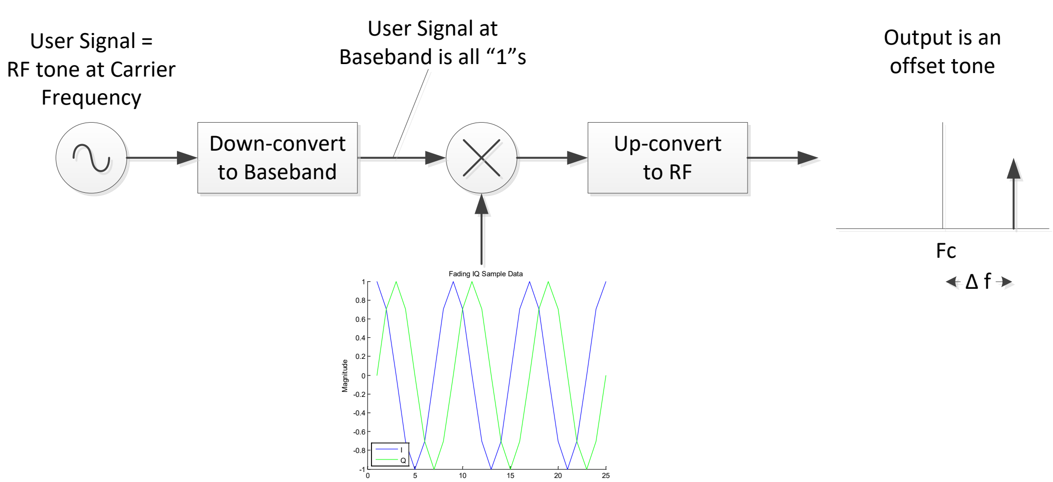


Figure 153. Producing an offset tone at RF.

In this simple scenario, you can calculate the value of Δf by considering the frequency of the sampled sinusoid in radians per sample (in this example $\frac{\pi}{4}$ radians per sample) using the following equation:

$$\Delta f = \frac{Rps}{2\pi} \cdot \frac{Fs}{Usf}$$

Equation 1.

Where Rps is the sinusoid frequency in radians per sample, Fs is the sample rate (10000 samples/second for Vertex), and Usf is the upsample factor (2, in this example).

You can see that, for the $\frac{\pi}{4}$ example, Δf can be found as follows:

$$\Delta f = \frac{Rps}{2\pi} \cdot \frac{Fs}{Usf} = \frac{\frac{\pi}{4}}{2\pi} \cdot \frac{10^3}{2} = 625\ Hz$$

Equation 2.

You can measure this offset using a signal generator to generate the tone at the Carrier Frequency as input to the Vertex unit and specifying the offset tone sample file for IQ Playback and observing the spectrum centered on the carrier frequency using a spectrum analyzer. The following figure shows the un-faded output of the Vertex (IQ Playback is stopped) unit.

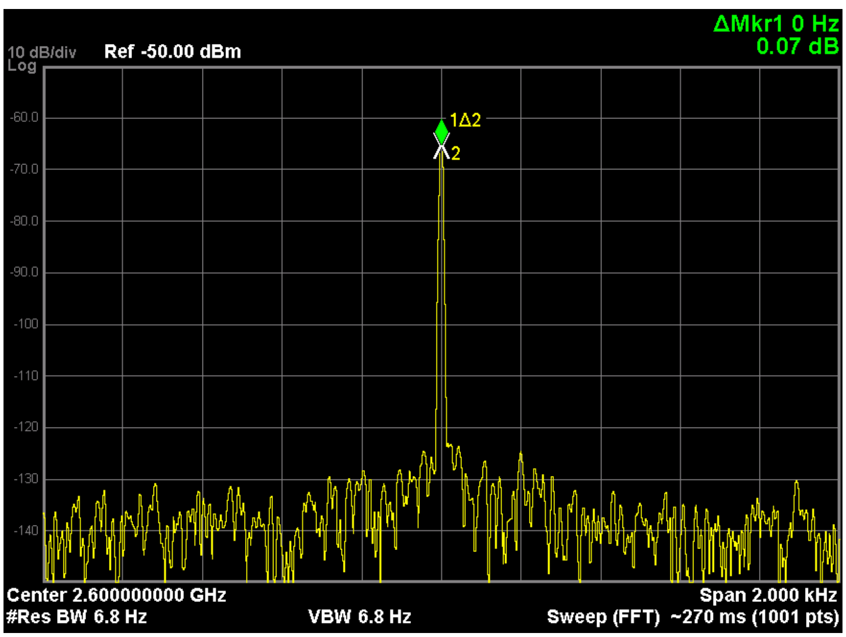


Figure 154. Static un-faded channel (IQ Playback is stopped).

The following figure shows the offset tone sample file applied.

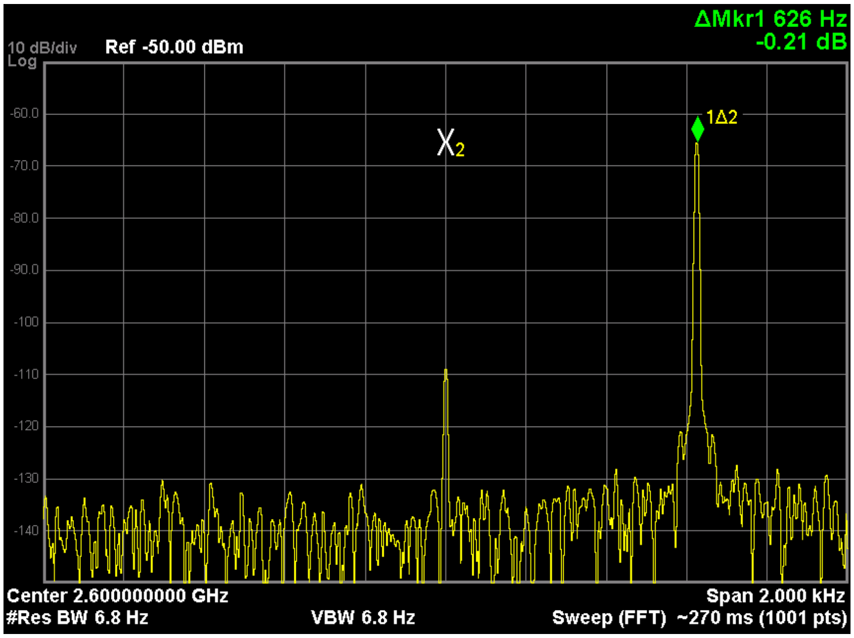


Figure 155. Offset tone fading sample file applied (tone offset of 625 Hz).

2.17.3.1 Upsample Factor

The Upsample Factor provides a way to reduce the data rate required to transmit the sample data to the DSP by allowing the DSP to interpolate points inserted into the sample data.

Consider, for example, the offset tone sample set in the previous figure. For an Upsample Factor of 1 (full DSP rate), the offset tone moves to:

$$\Delta f = \frac{Rps}{2\pi} \cdot \frac{Fs}{Usf} = \frac{\frac{\pi}{4}}{2\pi} \cdot \frac{10^3}{1} = 1250\ Hz$$

Equation 3.

Using an Upsample Factor of 2 yields:

$$\Delta f = \frac{Rps}{2\pi} \cdot \frac{Fs}{Usf} = \frac{\frac{\pi}{4}}{2\pi} \cdot \frac{10^3}{2} = 625\ Hz$$

Equation 4.

However, you can provide a sample file with half the number of samples (for a smaller file size), as shown in the following table.

[Fading Sample Data]		
0.0	1.0000	0.0000
0.0	0.0000	1.0000
0.0	-1.0000	0.0000
0.0	0.0000	-1.0000

For this fading sequence, the frequency of the sampled sinusoid in radians per sample is $\frac{\pi}{2}$ radians per sample, which yields:

$$\Delta f = \frac{Rps}{2\pi} \cdot \frac{Fs}{Usf} = \frac{\frac{\pi}{2}}{2\pi} \cdot \frac{10^3}{2} = 1250\ Hz$$

Equation 5.

In this example, you have specified your sample data using half the number of samples and have maintained the offset frequency.

> **WARNING!**
> Upsample Factor = 1 is risky to the DSP health for large fading scenarios.
> The data rate needed to supply all DSPs in a large fading scenario (for example, MIMO 8x4 24 path) would exceed the capacity supported by the Vertex unit.

If the Vertex instrument is installed with SDE-DSPM2 hardware, the up sample rate is fixed to 16.

2.17.3.2 Maximum Doppler

Nyquist theorem states that the maximum frequency that can be sampled without aliasing is ½ the sample rate of the system. On Vertex, at full 10k samples per second, that frequency is 5 kHz and is the Maximum Doppler frequency that can be supported at this rate. With practical signal processing considerations, the maximum Doppler Frequency is more realistically 0.8 times the maximum theoretical Doppler.

Note that the Maximum Doppler is reduced by the Upsample Factor for the typical connection setups as shown in the following table.

Connection Setup	#RLs	#Paths / RL	Delay Enabled	Minimum Up-Sample Rate Supported	Max Doppler Possible (Hz)
1x4 UNI MU	4	24	Yes	2	2000
Dual 2x2 Uni	8	24	No	2	2000
Dual 2x2 Uni	8	24	Yes	2	2000
Dual 2x4 Uni	16	24	No	4	1000
Dual 2x4 Uni	16	24	Yes	4	1000
Dual 4x2 Uni	16	24	No	2	2000
Dual 4x2 Uni	16	24	Yes	2	2000
Dual 4x4 Uni	32	24	No	4	1000
Dual 4x4 Uni	32	24	Yes	4	1000
Dual 8x2 Uni	32	24	No	2	2000
Dual 8x2 Uni	32	24	Yes	4	1000
Dual 4x8 Uni	64	24	No	4	1000
Dual 4x8 Uni	64	24	Yes	8	500
Dual 8x4 Uni	64	24	No	4	1000
Dual 8x4 Uni	64	24	Yes	8	500
8x8 Uni	64	24	No	4	1000
8x8 Uni	64	24	Yes	8	500

2.17.3.3 Delay

Consider the fading sequence with the addition of delay.

[Fading Sample Data]		
1.0	1.0000	0.0000
1.0	0.7071	0.7071
1.0	0.0000	1.0000
1.0	-0.7071	0.7071
1.0	-1.0000	0.0000

The first column represents the radio link delay in µS with a resolution down to 0.1 nS. This delay is a physical delay of the faded user signal.

WARNING!
Delays are applied to radio link pairs. For example, radio links sourced by a common port must share delays due to architectural reasons on Vertex. For example, in a 2x2 MIMO connection setup, A1-B1 and A1-B2 share delays even though the sample file allows independent delays. In this case, the A1-B2 would take on the delays of A1-B1. Each radio link only has 12 delays available on average.

2.17.3.4 Output Power and Scaling

In general, everything sums. In the simplest case, a single radio link of a SISO channel with a single path, the amplitude of the sinusoid is the exact amplitude to generate the Output Level on the Vertex.

Note that this equates to an RMS of $\sqrt{2}/_2$. For multiple paths, the fading sample data must be scaled by $\sqrt{Number\ of\ Paths}$. For multiple radio links summing (as in MIMO 2x2 or MIMO 4x4), the fading sample data must additionally be scaled by $\sqrt{Number\ of\ Radio\ Links\ Summing}$.

In general, the fading waveform must contain RMS average according to Equation 6 to achieve the correct Output Level on the Vertex.

$$RMS\ of\ Fading\ Waveform = \frac{\sqrt{2}/_2}{\sqrt{Number\ of\ Paths}\sqrt{Number\ of\ Radio\ Links\ Summing}}$$

Equation 6.

If RMS of the fading waveform is less than the value indicated in Equation 6, the Output Level of the Vertex will be lower than expected. If the RMS of the fading waveform is greater than indicated by Equation 6, the Output Level of the Vertex will be higher than expected. If the RMS of the fading waveform is much greater than indicated by Equation 6, you risk clipping, which results in added noise and reduced fidelity of the fading channel.

The following table shows some examples.

Scenario	Number of Radio Links Summing	Number of Paths	RMS of the Fading Waveform	Amplitude of Sinusoidal Fading Waveform
Single SISO radio link with 1 path	1	1	$\frac{\sqrt{2}/2}{\sqrt{1}\sqrt{1}} = 0.7071$	$\frac{1}{\sqrt{1}\sqrt{1}} = 1$
MIMO 2x2 with 1 path	2	1	$\frac{\sqrt{2}/2}{\sqrt{1}\sqrt{2}} = 0.5$	$\frac{1}{\sqrt{1}\sqrt{2}} = 0.7071$
MIMO 4x4 with 1 path	4	1	$\frac{\sqrt{2}/2}{\sqrt{1}\sqrt{4}} = 0.3536$	$\frac{1}{\sqrt{1}\sqrt{4}} = 0.5$
MIMO 4x4 with 24 paths	4	24	$\frac{\sqrt{2}/2}{\sqrt{24}\sqrt{4}} = 0.0722$	$\frac{1}{\sqrt{24}\sqrt{4}} = 0.1021$

2.17.3.5 Clipping

It is possible to clip if instantaneous peaks of the fading sample data are excessively high.

2.17.3.6 Wrap-around Discontinuity

When the IQ Playback reaches the end of the sample file and wraps around to the beginning, depending on the circumstances, the last sample of the file and the first sample of the file may not align and introduce a discontinuity to the playback. This can be seen on a spectrum analyzer as a brief jump in noise that settles down quickly. It can be seen every time the loop number increments and the state number wraps around to zero.

WARNING!
Since the size of IQ playback files are always big, Spirent recommends that you delete old IQ playback files that are no longer be used. Otherwise, it will be extremely easy to run out of space on the SDD drive in the Vertex instrument. The IQ playback files are located in the following folder on the Vertex instrument: **D:\FTPROOOT\Spirent\FDP**.

2.17.4. Dynamic Emulation with IQ Playback Model

You can dynamically change the output power and AWGN level when you play an IQ playback file The dynamic scenario is predefined in the IQ playback file with state duration and dynamic steps as described in the following figure.

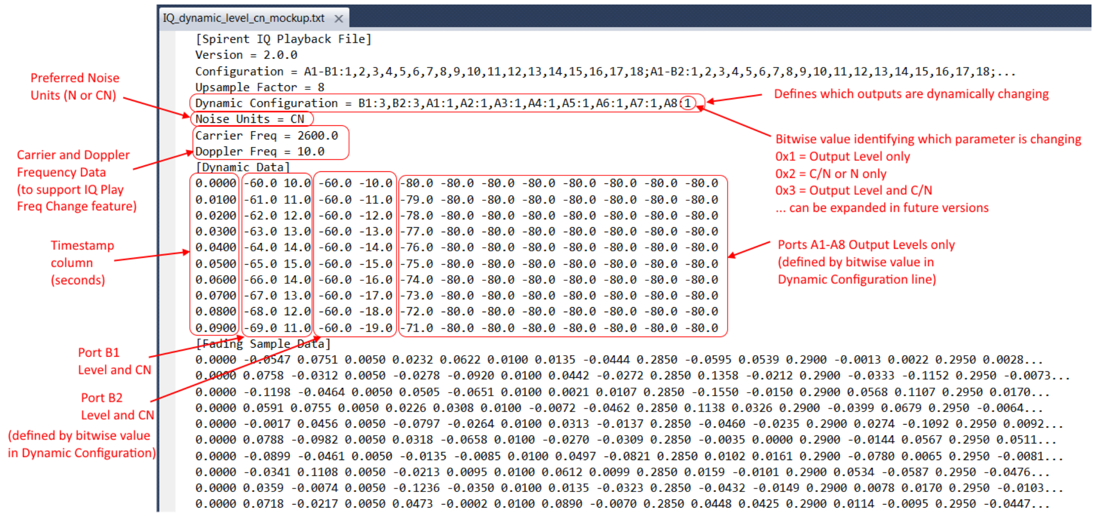


Figure 156. State duration and dynamic steps in IQ Playback file.

You need to declare the dynamic configuration to define which output ports are dynamically changing. After this row, you need to set the preferred units for noise.

After [Dynamic Data] header, you can put all the dynamic data into the IQ play file before the fading sample data.

The first column of dynamic data contains the timestamp (in seconds). The remaining columns contain the dynamic data for each port (power and AWGN).

Vertex will compile the dynamic data together with fading sample data. When you play the IQ playback model, the output power and AWGN will be updated as the file plays.

2.18. Indoor Channel Model Support

The Vertex channel emulator provides support for Indoor Channel Models (802.11n/ac) using the Classical Channel Models Fading Mode. It includes the additional mapping Cluster-based Geometric parameters feature outlined in the TGn and TGac specifications into the appropriate MIMO Correlation properties for each channel model.

To configure Indoor Channel Models:

1. Configure the Antenna Spacing for the devices on the A and B-sides of the Connection Diagram.
2. Configure the Channel Model Properties.

2.18.1. Antenna Configuration

To configure the antenna, select **Configure>Antenna Settings**. The Connection Setup – Antenna Settings window appears, as shown in the following figure. In version 2.00 and earlier, the Antenna Settings were only available in Geometric Channel Modeling Fading Mode.

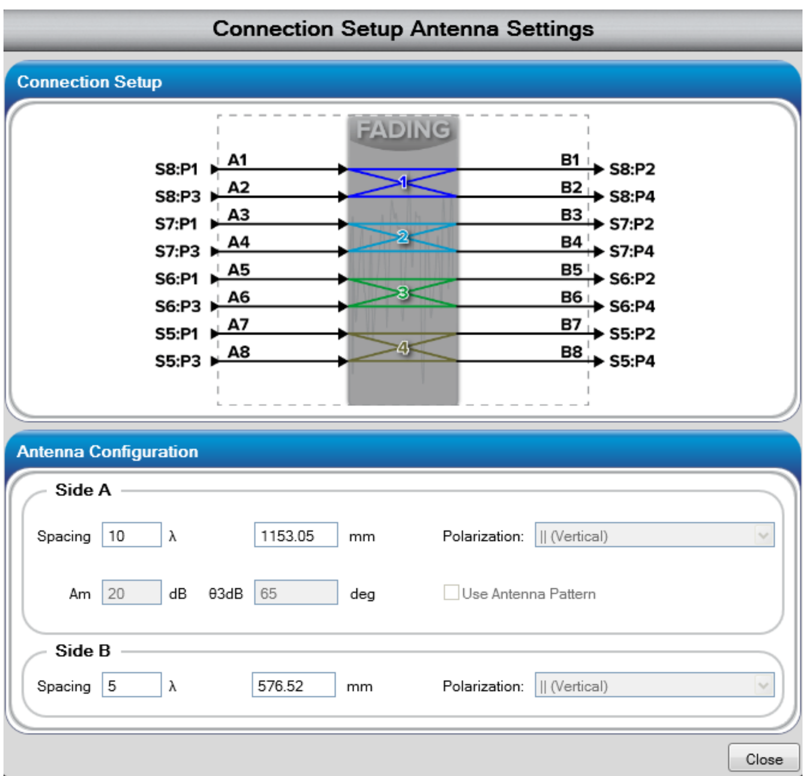


Figure 157. Connection Setup Antenna Settings window.

The Antenna Settings window allows you to configure the following properties:

Spacing: Defines the spacing, in wavelengths, of the device antennas. Antenna spacing is also displayed in meters. This depends on the carrier frequency.

2.18.2. Channel Model Configuration

Indoor Channel Models as defined in the TGn/ac specification are described in terms of up to six clusters, the properties of which overlap on a set of taps with discrete delay values. The following figure displays the representation of Model E from the TGn Channel Model Specification. For example, in this model, Path 11 has three overlapping clusters.

	Tap index	1	2	3	4	5	6	7	8	9	10	11	12	13	14	15	16	17	18
	Excess delay [ns]	0	10	20	30	50	80	110	140	180	230	280	330	380	430	490	560	640	730
Cluster 1	Power [dB]	-2.6	-3.0	-3.5	-3.9	-4.5	-5.6	-6.9	-8.2	-9.8	-11.7	-13.9	-16.1	-18.3	-20.5	-22.9			
AoA	AoA [°]	163.7	163.7	163.7	163.7	163.7	163.7	163.7	163.7	163.7	163.7	163.7	163.7	163.7	163.7	163.7			
AS (receive)	AS [°]	35.8	35.8	35.8	35.8	35.8	35.8	35.8	35.8	35.8	35.8	35.8	35.8	35.8	35.8	35.8			
AoD	AoD [°]	105.6	105.6	105.6	105.6	105.6	105.6	105.6	105.6	105.6	105.6	105.6	105.6	105.6	105.6	105.6			
AS (transmit)	AS [°]	36.1	36.1	36.1	36.1	36.1	36.1	36.1	36.1	36.1	36.1	36.1	36.1	36.1	36.1	36.1			
Cluster 2	Power [dB]					-1.8	-3.2	-4.5	-5.8	-7.1	-9.9	-10.3	-14.3	-14.7	-18.7	-19.9	-22.4		
AoA	AoA [°]					251.8	251.8	251.8	251.8	251.8	251.8	251.8	251.8	251.8	251.8	251.8	251.8		
AS	AS [°]					41.6	41.6	41.6	41.6	41.6	41.6	41.6	41.6	41.6	41.6	41.6	41.6		
AoD	AoD [°]					293.1	293.1	293.1	293.1	293.1	293.1	293.1	293.1	293.1	293.1	293.1	293.1		
AS	AS [°]					42.5	42.5	42.5	42.5	42.5	42.5	42.5	42.5	42.5	42.5	42.5	42.5		
Cluster 3	Power [dB]									-7.9	-9.6	-14.2	-13.8	-18.6	-18.1	-22.8			
AoA	AoA [°]									80.0	80.0	80.0	80.0	80.0	80.0	80.0			
AS	AS [°]									37.4	37.4	37.4	37.4	37.4	37.4	37.4			
AoD	AoD [°]									61.9	61.9	61.9	61.9	61.9	61.9	61.9			
AS	AS [°]									38.0	38.0	38.0	38.0	38.0	38.0	38.0			
Cluster 4	Power [dB]															-20.6	-20.5	-20.7	-24.6
AoA	AoA [°]															182.0	182.0	182.0	182.0
AS	AS [°]															40.3	40.3	40.3	40.3
AoD	AoD [°]															275.7	275.7	275.7	275.7
AS	AS [°]															38.7	38.7	38.7	38.7

Figure 158. 802.11n model E showing cluster to path mapping.

Even though a particular cluster-based channel model describes properties for multiple clusters across a number of paths (18 in the case of TGn Model E), only the number of paths with unique delay values are required to emulate the model. The overlapping clusters get mapped into a single set of correlation properties for each Path. The following figure shows the 18 paths of Model E from the Propagation Conditions Selection.

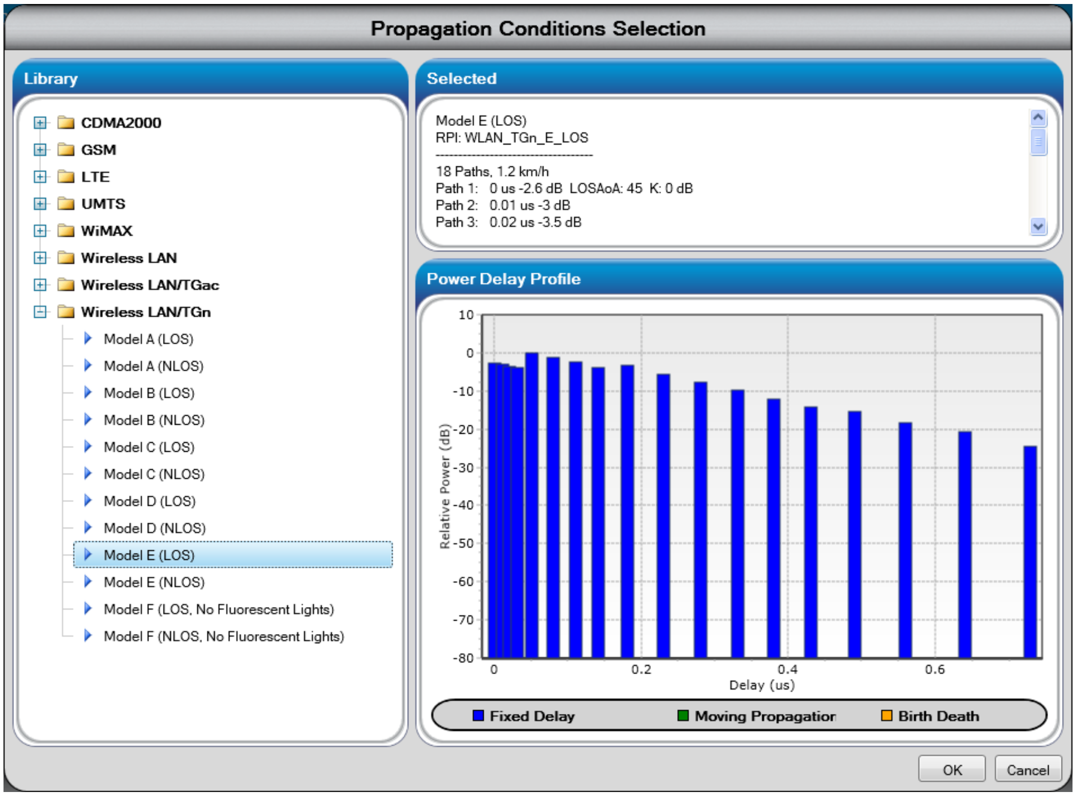
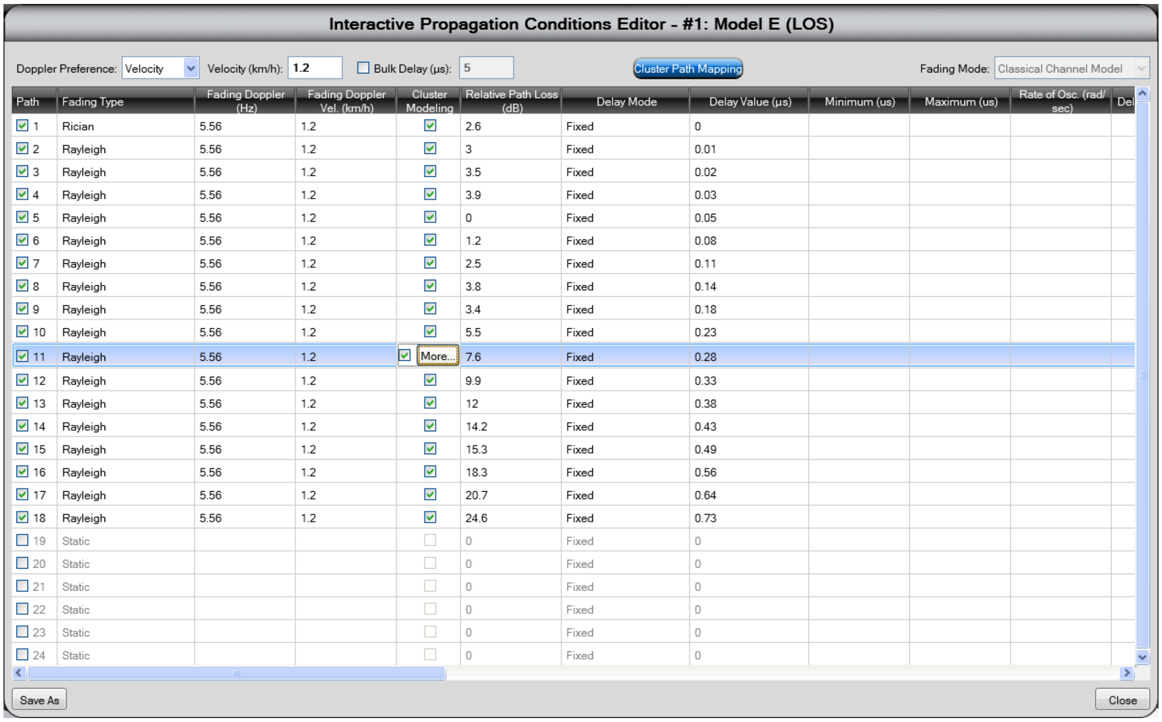


Figure 159. Selection of 802.11n model E from the Propagation Conditions Selection.

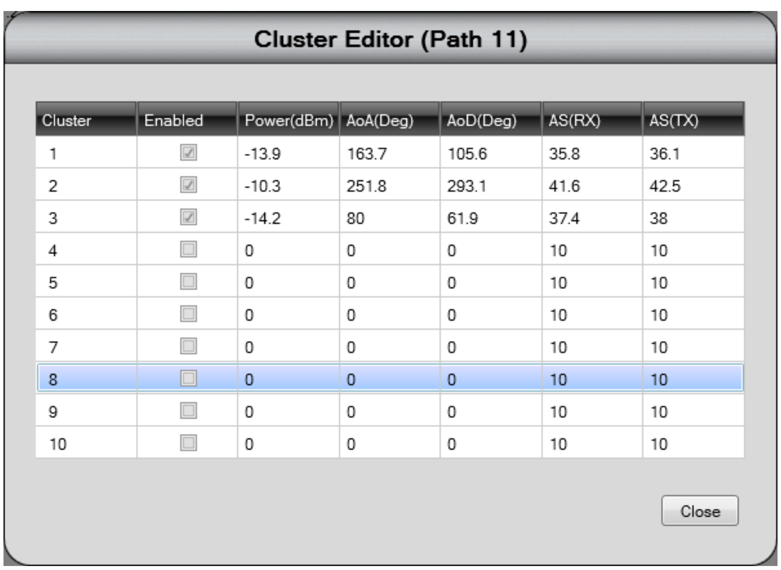
2.18.2.1 Editing Cluster-based Propagation Conditions

The cluster parameters (AoA, AoD, AS) can be configured independently for each path by clicking the **More** button. This button is active when Cluster Modeling is enabled on a given path. Clicking the **More** button displays the Cluster Editor for this specific Path, where you can edit the parameters on a per cluster basis, as shown in the following figure.



Path	Fading Type	Fading Doppler (Hz)	Fading Doppler Vel. (km/h)	Cluster Modeling	Relative Path Loss (dB)	Delay Mode	Delay Value (µs)	Minimum (us)	Maximum (us)	Rate of Osc. (rad/sec)	De
[x] 1	Rician	5.56	1.2	[x]	2.6	Fixed	0				
[x] 2	Rayleigh	5.56	1.2	[x]	3	Fixed	0.01				
[x] 3	Rayleigh	5.56	1.2	[x]	3.5	Fixed	0.02				
[x] 4	Rayleigh	5.56	1.2	[x]	3.9	Fixed	0.03				
[x] 5	Rayleigh	5.56	1.2	[x]	0	Fixed	0.05				
[x] 6	Rayleigh	5.56	1.2	[x]	1.2	Fixed	0.08				
[x] 7	Rayleigh	5.56	1.2	[x]	2.5	Fixed	0.11				
[x] 8	Rayleigh	5.56	1.2	[x]	3.8	Fixed	0.14				
[x] 9	Rayleigh	5.56	1.2	[x]	3.4	Fixed	0.18				
[x] 10	Rayleigh	5.56	1.2	[x]	5.5	Fixed	0.23				
[x] 11	Rayleigh	5.56	1.2	[x] More...	7.6	Fixed	0.28				
[x] 12	Rayleigh	5.56	1.2	[x]	9.9	Fixed	0.33				
[x] 13	Rayleigh	5.56	1.2	[x]	12	Fixed	0.38				
[x] 14	Rayleigh	5.56	1.2	[x]	14.2	Fixed	0.43				
[x] 15	Rayleigh	5.56	1.2	[x]	15.3	Fixed	0.49				
[x] 16	Rayleigh	5.56	1.2	[x]	18.3	Fixed	0.56				
[x] 17	Rayleigh	5.56	1.2	[x]	20.7	Fixed	0.64				
[x] 18	Rayleigh	5.56	1.2	[x]	24.6	Fixed	0.73				
[ ] 19	Static			[ ]	0	Fixed	0				
[ ] 20	Static			[ ]	0	Fixed	0				
[ ] 21	Static			[ ]	0	Fixed	0				
[ ] 22	Static			[ ]	0	Fixed	0				
[ ] 23	Static			[ ]	0	Fixed	0				
[ ] 24	Static			[ ]	0	Fixed	0				

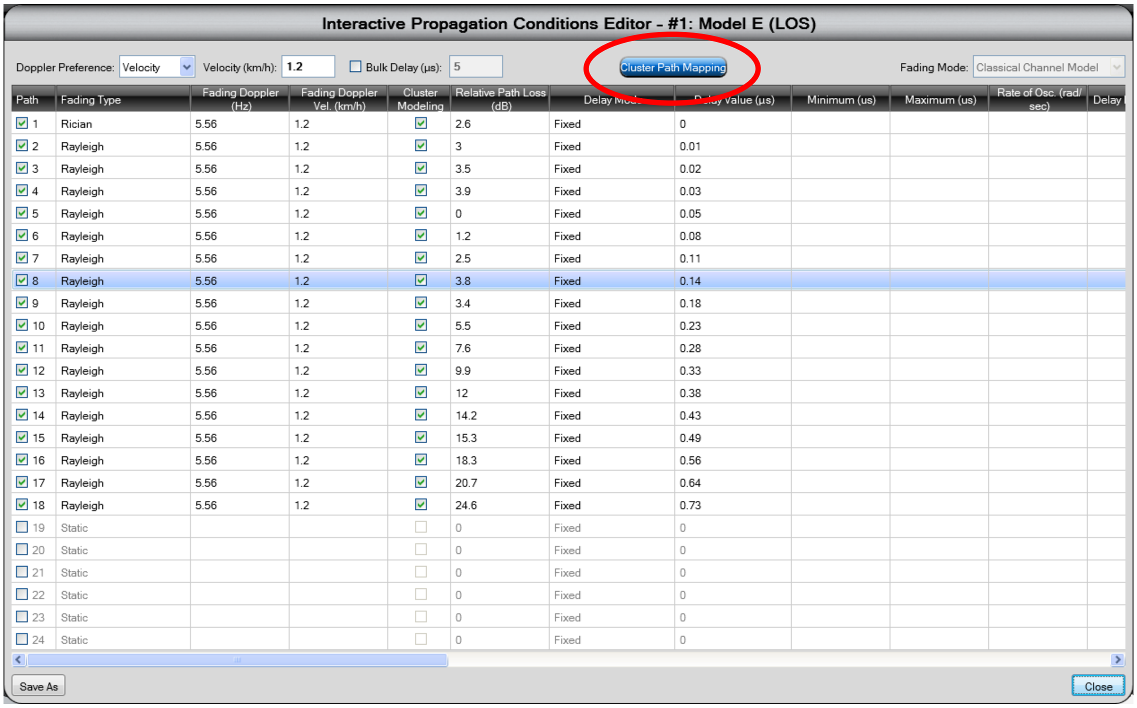
Figure 160. Enabling Cluster Modeling.



Cluster	Enabled	Power(dBm)	AoA(Deg)	AoD(Deg)	AS(RX)	AS(TX)
1	[x]	-13.9	163.7	105.6	35.8	36.1
2	[x]	-10.3	251.8	293.1	41.6	42.5
3	[x]	-14.2	80	61.9	37.4	38
4	[ ]	0	0	0	10	10
5	[ ]	0	0	0	10	10
6	[ ]	0	0	0	10	10
7	[ ]	0	0	0	10	10
8	[ ]	0	0	0	10	10
9	[ ]	0	0	0	10	10
10	[ ]	0	0	0	10	10

Figure 161. Cluster Editor for given path.

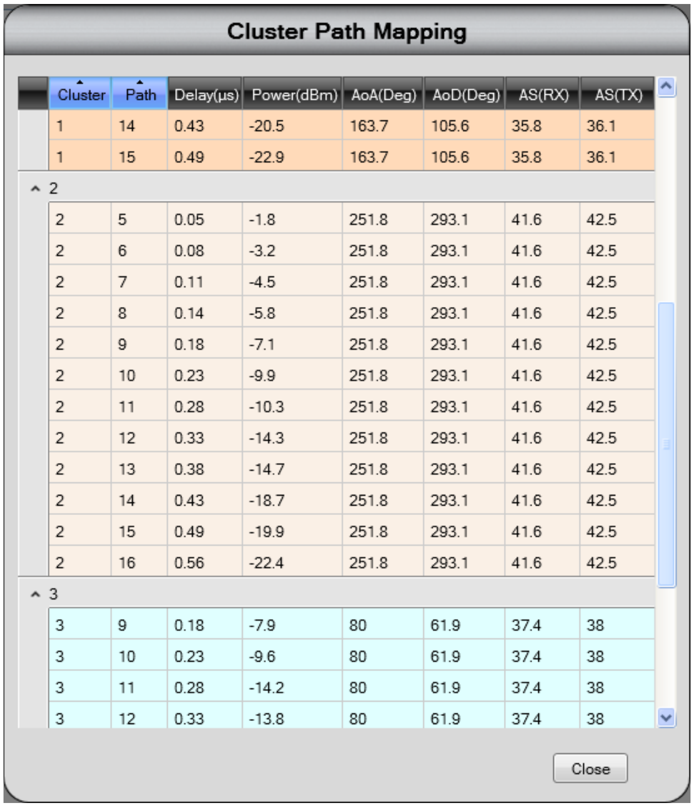
A similar view in information to what is presented in the 802.11 TGn Channel Models specification is available by clicking the **Cluster-Path Mapping** button, as shown in the following figure. This button is visible when any path in the Propagation Conditions has Cluster Modeling Enabled.



Path	Fading Type	Fading Doppler (Hz)	Fading Doppler Vel. (km/h)	Cluster Modeling	Relative Path Loss (dB)	Delay Mo	y Value (µs)	Minimum (us)	Maximum (us)	Rate of Osc. (rad/sec)	Delay
1	Rician	5.56	1.2	☑	2.6	Fixed	0				
2	Rayleigh	5.56	1.2	☑	3	Fixed	0.01				
3	Rayleigh	5.56	1.2	☑	3.5	Fixed	0.02				
4	Rayleigh	5.56	1.2	☑	3.9	Fixed	0.03				
5	Rayleigh	5.56	1.2	☑	0	Fixed	0.05				
6	Rayleigh	5.56	1.2	☑	1.2	Fixed	0.08				
7	Rayleigh	5.56	1.2	☑	2.5	Fixed	0.11				
8	Rayleigh	5.56	1.2	☑	3.8	Fixed	0.14				
9	Rayleigh	5.56	1.2	☑	3.4	Fixed	0.18				
10	Rayleigh	5.56	1.2	☑	5.5	Fixed	0.23				
11	Rayleigh	5.56	1.2	☑	7.6	Fixed	0.28				
12	Rayleigh	5.56	1.2	☑	9.9	Fixed	0.33				
13	Rayleigh	5.56	1.2	☑	12	Fixed	0.38				
14	Rayleigh	5.56	1.2	☑	14.2	Fixed	0.43				
15	Rayleigh	5.56	1.2	☑	15.3	Fixed	0.49				
16	Rayleigh	5.56	1.2	☑	18.3	Fixed	0.56				
17	Rayleigh	5.56	1.2	☑	20.7	Fixed	0.64				
18	Rayleigh	5.56	1.2	☑	24.6	Fixed	0.73				
19	Static			☐	0	Fixed	0				
20	Static			☐	0	Fixed	0				
21	Static			☐	0	Fixed	0				
22	Static			☐	0	Fixed	0				
23	Static			☐	0	Fixed	0				
24	Static			☐	0	Fixed	0				

Figure 162. Interactive Propagation Conditions Editor – Cluster Path Mapping button.

The Cluster Path Mapping window, shown in the following figure, displays the same information as the Cluster Editor on a per-path basis, but also shows all cluster properties for paths in a common view.



Cluster	Path	Delay(µs)	Power(dBm)	AoA(Deg)	AoD(Deg)	AS(RX)	AS(TX)
1	14	0.43	-20.5	163.7	105.6	35.8	36.1
1	15	0.49	-22.9	163.7	105.6	35.8	36.1
2							
2	5	0.05	-1.8	251.8	293.1	41.6	42.5
2	6	0.08	-3.2	251.8	293.1	41.6	42.5
2	7	0.11	-4.5	251.8	293.1	41.6	42.5
2	8	0.14	-5.8	251.8	293.1	41.6	42.5
2	9	0.18	-7.1	251.8	293.1	41.6	42.5
2	10	0.23	-9.9	251.8	293.1	41.6	42.5
2	11	0.28	-10.3	251.8	293.1	41.6	42.5
2	12	0.33	-14.3	251.8	293.1	41.6	42.5
2	13	0.38	-14.7	251.8	293.1	41.6	42.5
2	14	0.43	-18.7	251.8	293.1	41.6	42.5
2	15	0.49	-19.9	251.8	293.1	41.6	42.5
2	16	0.56	-22.4	251.8	293.1	41.6	42.5
3							
3	9	0.18	-7.9	80	61.9	37.4	38
3	10	0.23	-9.6	80	61.9	37.4	38
3	11	0.28	-14.2	80	61.9	37.4	38
3	12	0.33	-13.8	80	61.9	37.4	38

Figure 163. Cluster Path Mapping window.

NOTE:
The Cluster-Path Mapping window is read-only. All cluster parameters must be entered on a per path basis through the Cluster Editor.

2.18.2.2 Correlation View

The geometric parameters of the Clusters on each path lead to a Correlation Matrix Calculation. This causes the Correlation Library name to display “Unsaved Profile” when Propagation Conditions with Cluster Modeling is enabled on any of the paths, as shown in the following figure.

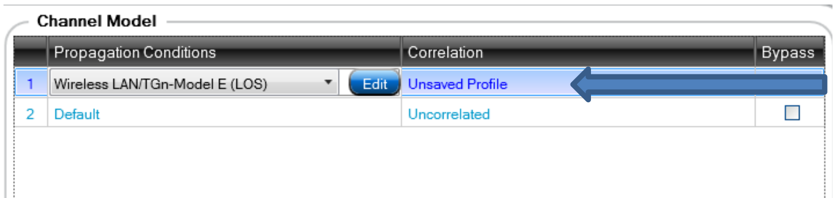


Figure 164. “Unsaved Profile” Correlation Library name.

NOTE:
Because the Correlation Matrix is automatically generated when cluster Modeling is enabled on a path, the matrix is read-only for that given path.

2.19. HFC (High Frequency Converter) Settings

Vertex can work with a Spirent external high-frequency converter (HFC) to simulate propagation conditions at frequencies higher than 6GHz. There are four models of the Spirent HFC available, and each HFC has different LO frequencies. The following table provides the frequency range of each HFC.

Frequency Range	5.9 - 10GHz	9 - 13GHz	24.25 - 29.5GHz	37 - 40.5GHz
Model Number	VCE6-HFC-5C-7GHz	VCE6-HFC-5C-11GHz	VCE6-HFC-5C-27GHz	VCE6-HFC-5C-39GHz
LO	11.75GHz	14.5GHz	23. 5GHz	35GHz
Internal mmWave Filter	5.9-10GHz	9-13GHz	24.25 to 29.5GHz	37 to 40.5GHz
Input Frequency	1.75 to 5.85GHz	1.5 to 5.5GHz	0.75 to 6GHz	2 to 5.5GHz

You can set the higher frequencies (as covered by the different models of HFC) from the Vertex GUI or with RPI commands. Vertex can automatically convert the internal carrier frequency to the right frequency after you select the correct HFC model and set the carrier frequency.

The following figure shows the **HFC Settings** option on the **Configure** menu in the Vertex GUI.

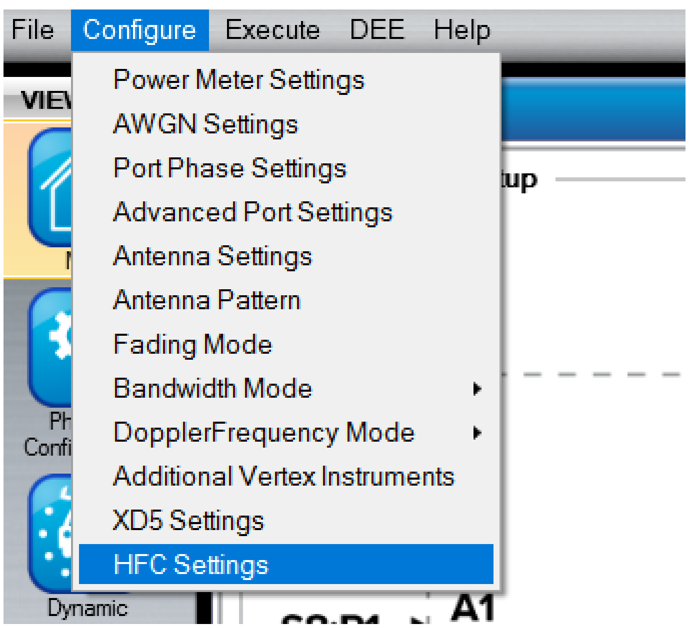


Figure 165. HFC Settings option on the Configure menu in the Vertex GUI.

To configure the HFC settings, perform the following steps:

1. From the Configure menu, select **HFC Settings**.

   The HFC Configuration dialog box appears.

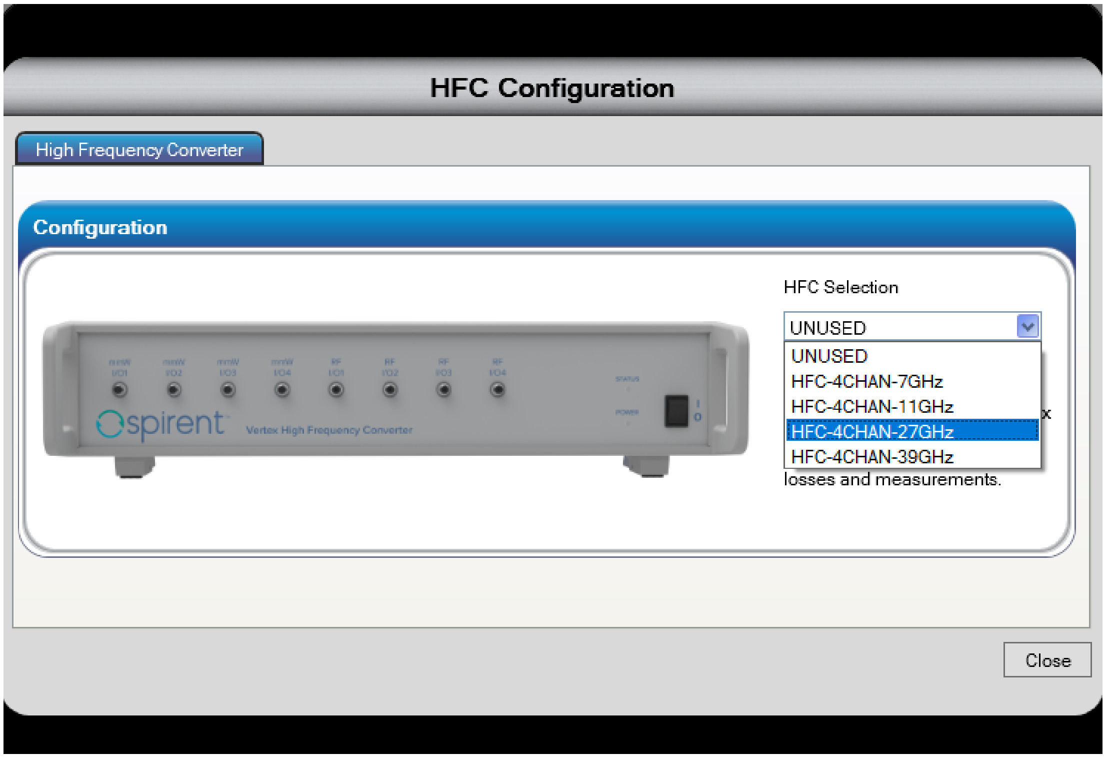


Figure 166. HFC Configuration dialog box.

2. From the HFC Selection box, select the HFC model that is connected to Vertex, and then click the **Close** button.

Once you select the HFC model, you can set the frequency at the frequency range of that HFC model. For example, if you selected the 27GHz HFC, you can set the carrier frequency from 24.25 to 29.5 GHz as shown in the following figure.

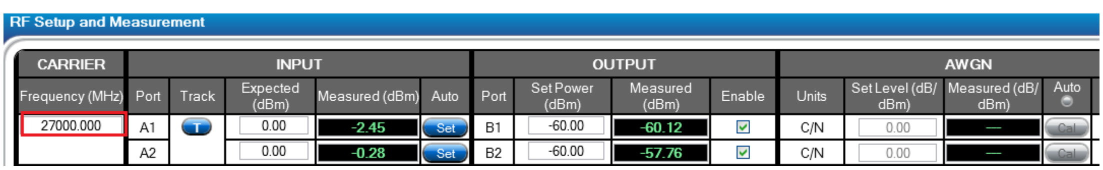


Figure 167. Setting the frequency.

The fading doppler value will also be calculated by the frequency you set as shown in the following figure.

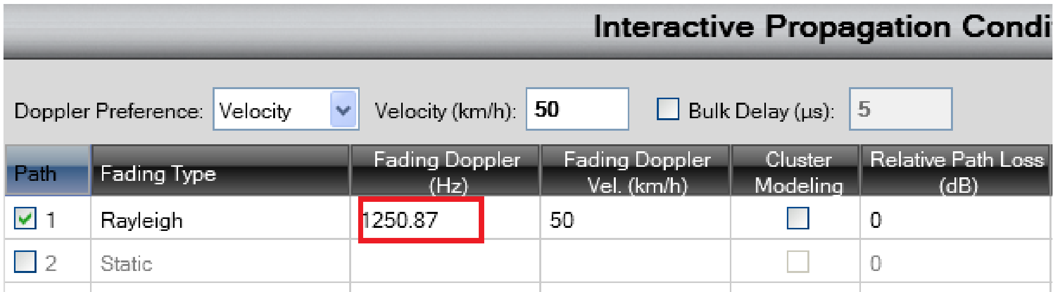


Figure 168. Calculated Fading Doppler.

2.20. Temporal Player

Temporal Player is a graphical view feature for you to monitor or capture the real-time input/out power of RF ports. To capture the measured input and output power, you must set Measurement Type to **MEASURED** for Output Port Settings in the Power Meter Settings dialog box as shown in the following figure.

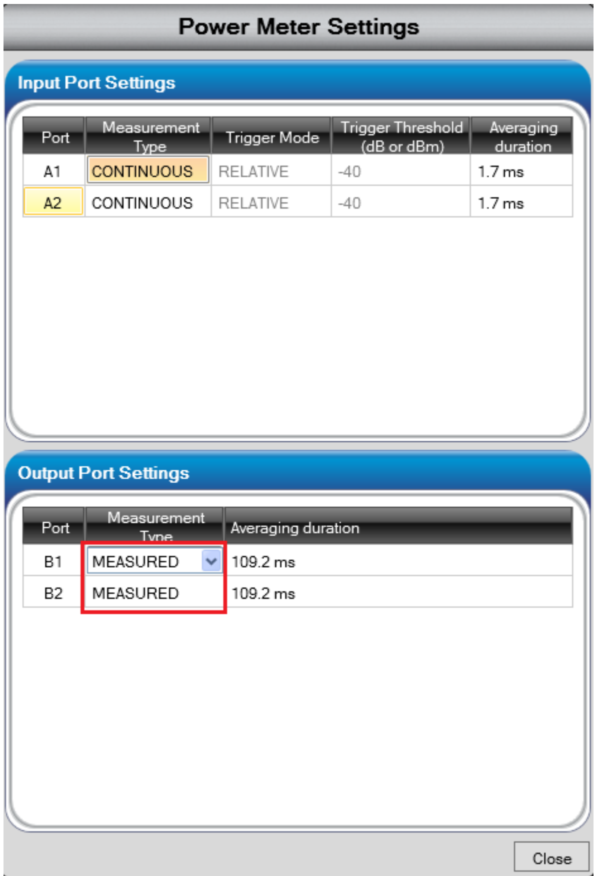


Figure 169. MEASURED setting for Output Port Settings.

To view the real-time power update, perform the following steps:

1. Click the **Temporal Player** button in the Vertex GUI.

   The Temporal Player window appears.

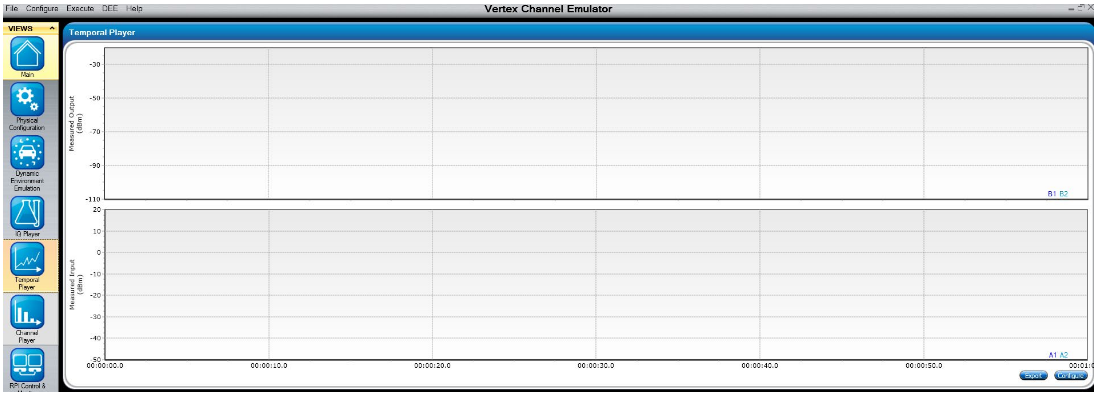


   Figure 170. Temporal Player window.

2. Click the **Configure** button located in the lower, right corner of the Temporal Player window to select the ports or chart type.

   The Temporal Chart Configuration dialog box appears.

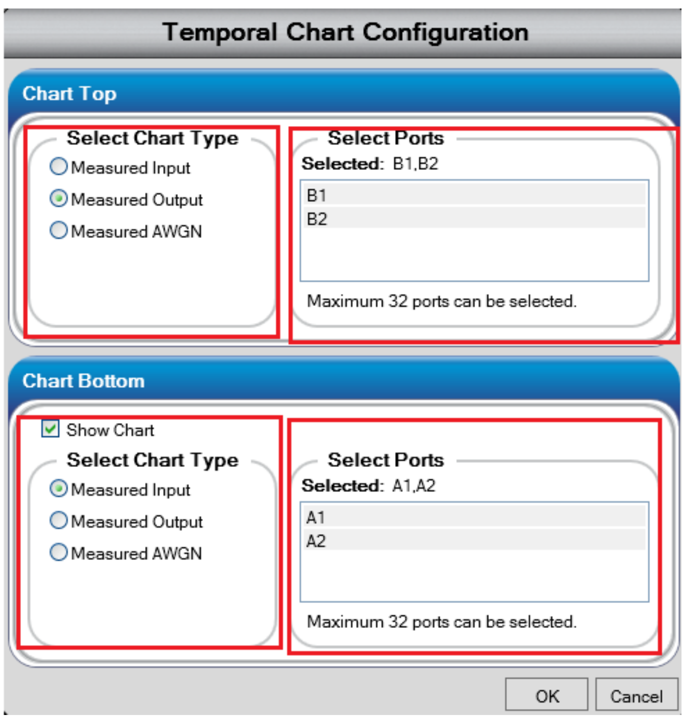


   Figure 171. Temporal Chart Configuration dialog box.

3. In the Select Chart Type area, click the option button for the chart type you want to use.
4. In the Select Ports area, select the ports in which you are interested. The selected ports will be highlighted with light gray color.

   NOTE:
   The maximum number of ports you can select is 32. The minimum number of ports is 1.

5. When finished, click the **OK** button.

When you play the channel emulation, you can see the real-time power update at the selected ports.

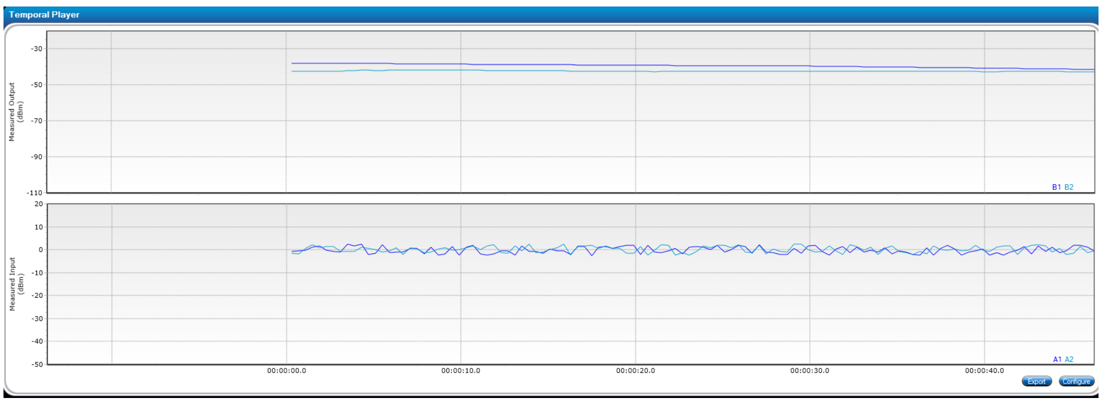


Figure 172. Channel emulation displayed in the Temporal Player window.

If you want to save the power change, click the **Export** button located in the lower, right corner of the Temporal Player window and then save the exported data into an Excel file.

2.21. Vertex Passwords

2.21.1. Updating the Vertex Password

You can update the password through Vertex embedded PC or the Controller laptop.

Perform the following steps:

1. Connect the USB device containing the password into the USB port of Vertex instrument (if you use embedded PC) or Controller Laptop (if you use the Controller Laptop).
2. From Vertex GUI menu bar, select **Help>Password Utility**.

   The Password Utility window appears, as shown in the following figure.

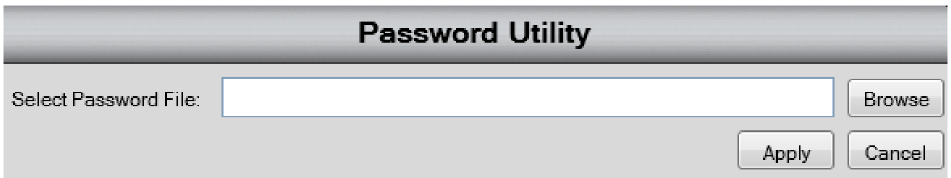
   

   Figure 173. Password Utility window.

3. Click the **Browse** button, and select the .txt password file.
4. Click the **Apply** button.

   A window appears, indicating that the password file upgrade is successful.

2.21.2. Annual Support Agreement

You can purchase an Annual Support Agreement (ASA).

The agreement includes the following for the duration of the agreement:

- Calibration, repair, and technical assistance
- Software/firmware upgrades

The ASA is installed on a Vertex in the same way as a password.

When the ASA expires, the installer no longer allows you to upgrade the Vertex to any revision released after the ASA expiration date, as shown in the following figure.

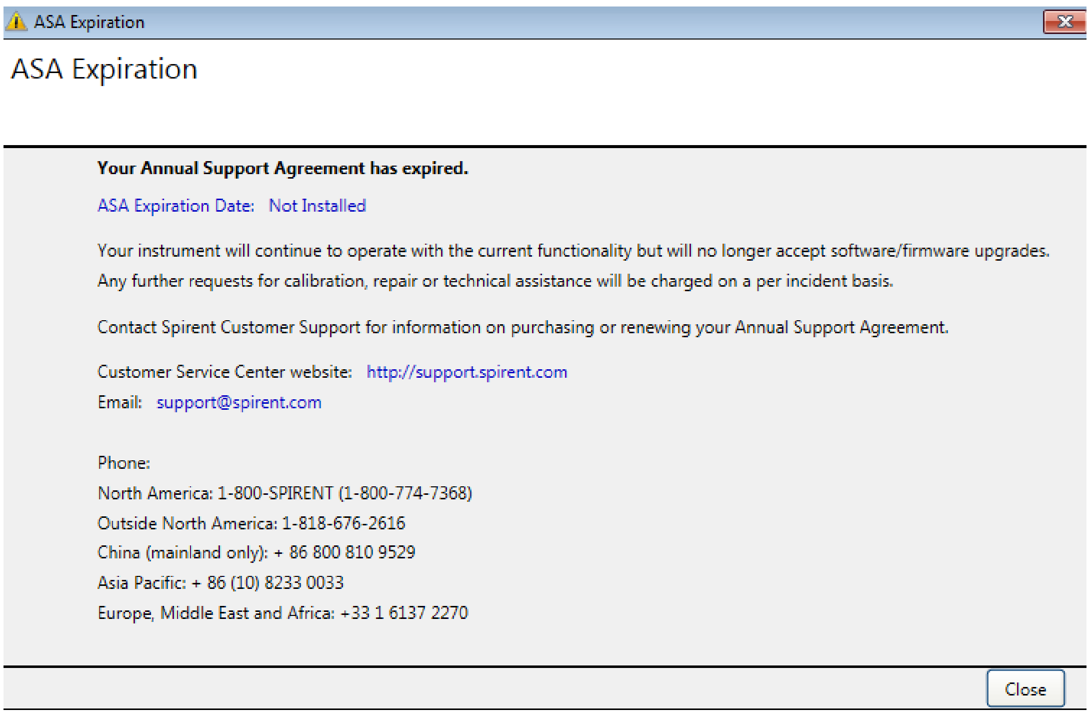


Figure 174. ASA Expiration window.

2.22. Setting the IP Address

The IP address of Vertex can be set with the local embedded PC or remote controller PC.

2.22.1. Setting the IP Address using the Local Embedded PC

If the Vertex instrument is connected with an external monitor, keyboard, and mouse, you can click the **IP Address Setting** button, enter the IP address, Subnet Mask, and Default Gateway in the Set Instrument IP Address window, and then click the **Apply IP Settings** button. The instrument IP address will be updated to the current setting.

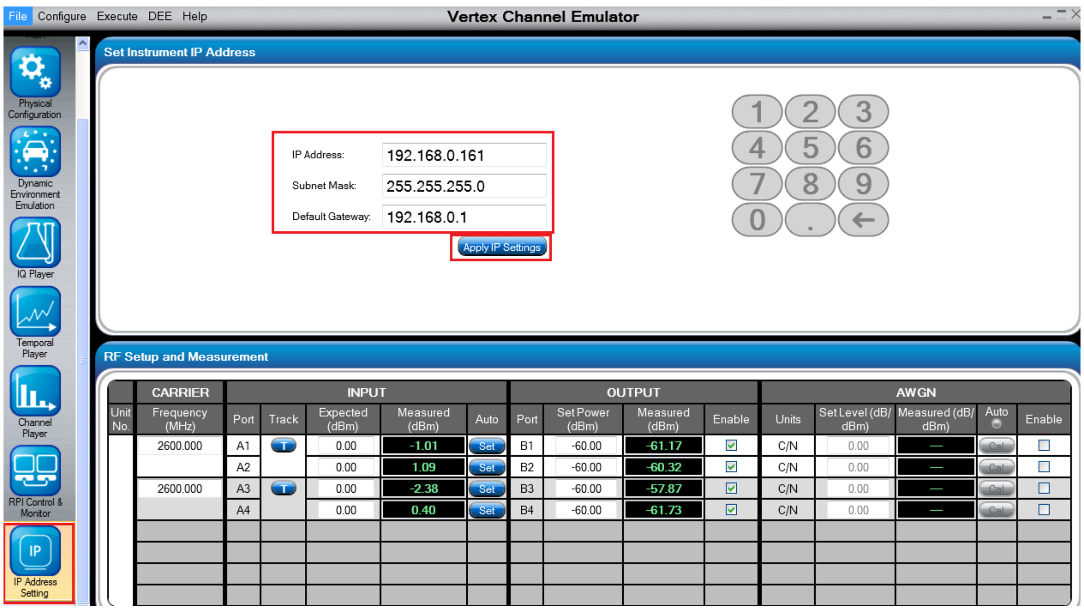


Figure 175. Set Instrument IP Address window.

2.22.2. Setting the IP Address using the Remote Controller PC

If there is no monitor connected to the Vertex instrument, you can also update the Vertex instrument IP address with the remote controller PC.

Using the remote GUI, perform the following steps:

1. Click the **IP Address Setting** button, enter the IP address, Subnet Mask, and Default Gateway, and then click the **Apply IP Settings** button.

   The following warning appears.

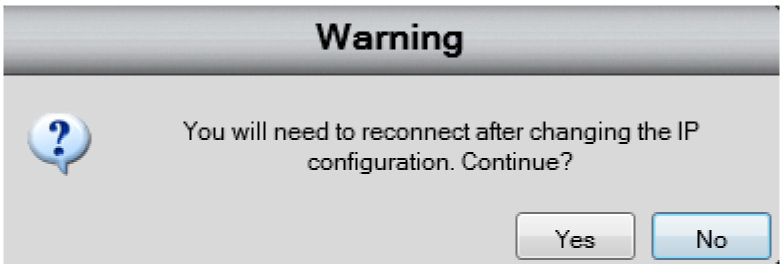


Figure 176. Warning dialog box.

2. Click the **Yes** button.

   The GUI shows the progress of setting the IP address.

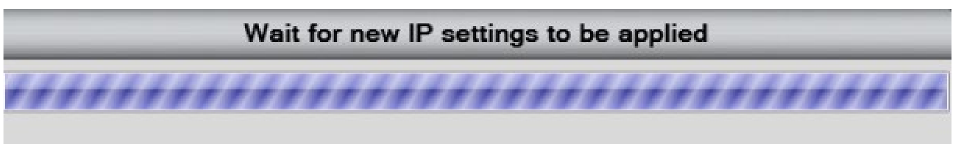


Figure 177. Progress of IP settings.

After the Vertex instrument IP address is updated, the remote controller PC will be disconnected.

To connect to Vertex with the new IP address, perform the following steps:

1. In the IP Settings window, enter the new IP address.

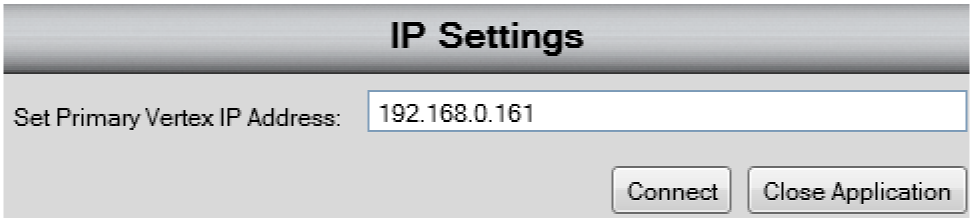


Figure 178. IP Settings window.

2. Click the **Connect** button.

   The Success message box appears, displaying "connected" and the IP address of the Vertex instrument.

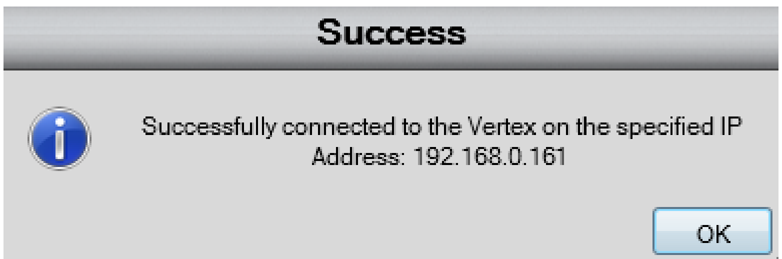


Figure 179. Success message box.

2.23. Upgrading Vertex Software

You must upgrade the Vertex software on both the Vertex instrument and the laptop.

2.23.1. Upgrading Software on the Instrument

NOTE:
It can take up to 45 minutes for a complete upgrade.

To upgrade the Vertex software on the instrument:

1. Connect a display on the video port (VGA, DVI-D, or HDMI) on the rear panel of the Vertex channel emulator.
2. Connect the USB drive containing the unzipped installer to one of the USB ports on the rear panel of the Vertex channel emulator.
3. Connect a keyboard and mouse to the other USB port on the rear panel of the Vertex channel emulator.
4. Using Windows Explorer, navigate to the ***<Vertex Installer Root>*** folder on the USB drive.
5. Double-click on the file **VertexInstall.exe** and follow the installation instructions.

   If this is the first time the Vertex software has been installed on the instrument, the following dialog box will appear.

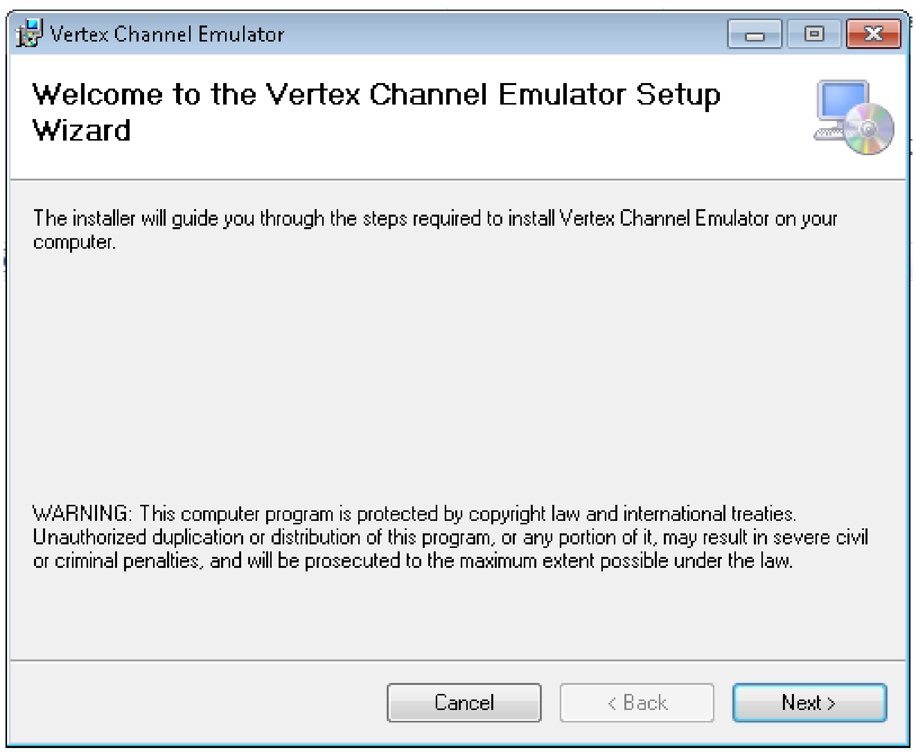


Figure 180. Installation dialog box.

Perform the following steps:

a. Click the **Next** button.

   The Confirm Installation dialog box appears.

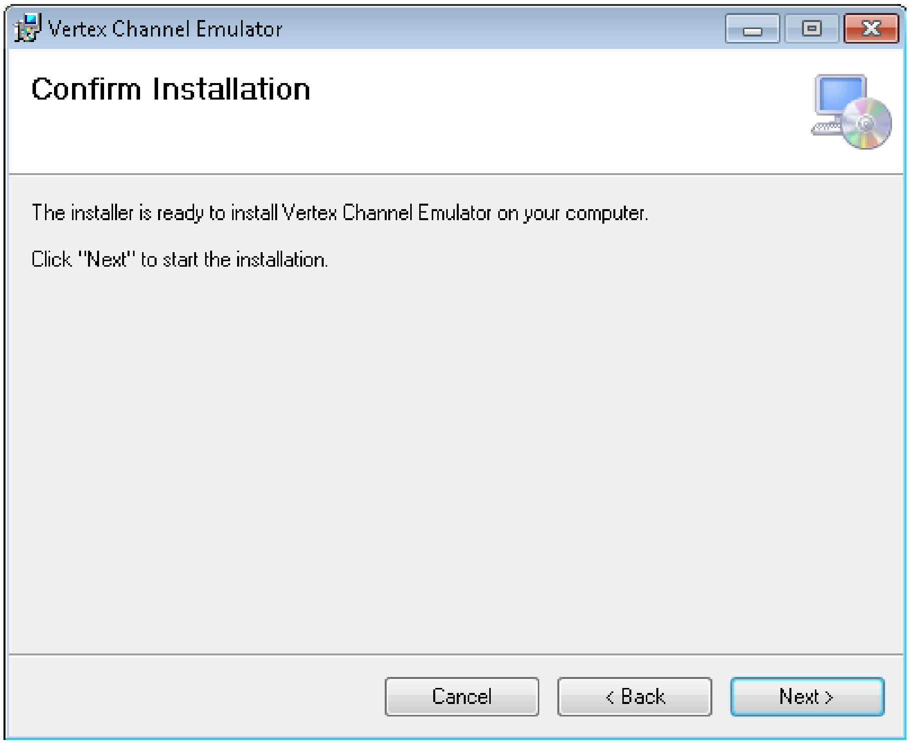


Figure 181. Confirm Installation dialog box.

b. Click the **Next** button.

When the installation is finished, the Installation Complete dialog box will appear.

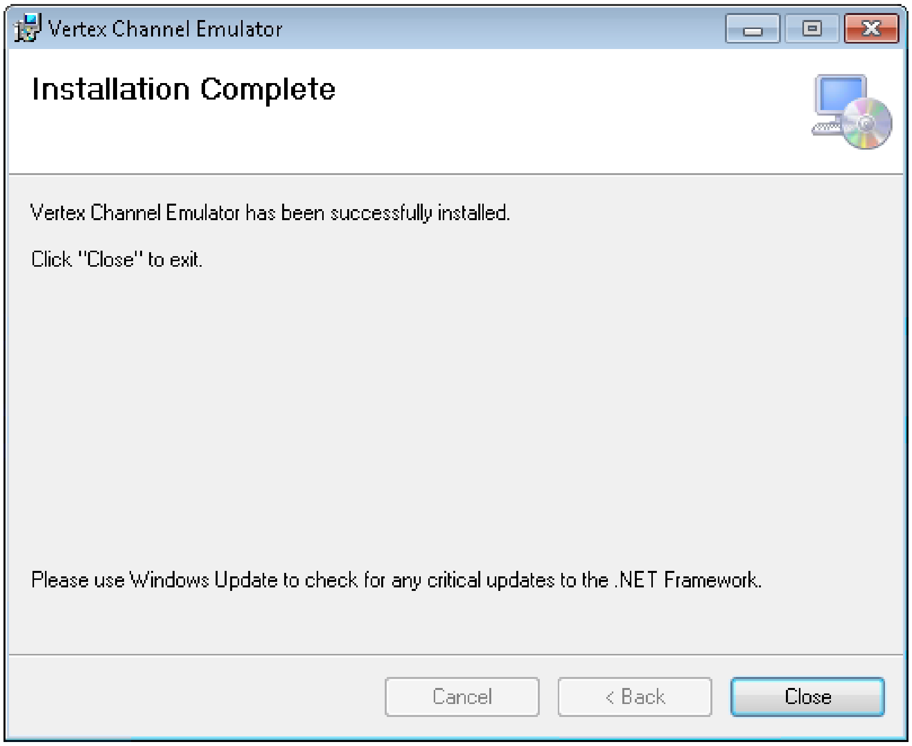


Figure 182. Installation Complete window.

c. Click the **Close** button.

If you are upgrading the software from an earlier version, a dialog box will appear and prompt you to install the instrument software during the installation process.

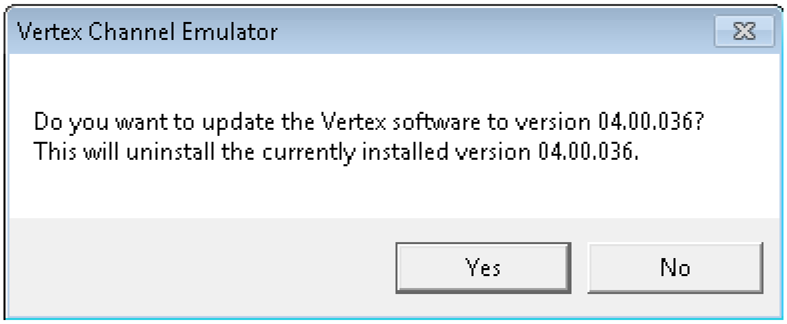


Figure 183. Sample Installation dialog box.

Perform the following steps:

a. Click the **Yes** button.

The License Agreement dialog box also appears.

Figure 184. License Agreement dialog box.

b. In the License Agreement dialog box, read the license agreement, and then click the **I Agree** option button.

The installation starts, unless the ASA is not present or has expired. The embedded GUI will start automatically, and there will be no dialog box informing you that the installation is complete.

If the firmware needs to be updated, the embedded GUI will inform you. In this case, always click the **Yes** button. When the firmware update is finished, a dialog box will appear briefly, informing you that the instrument is about to reboot. Then the instrument reboots.

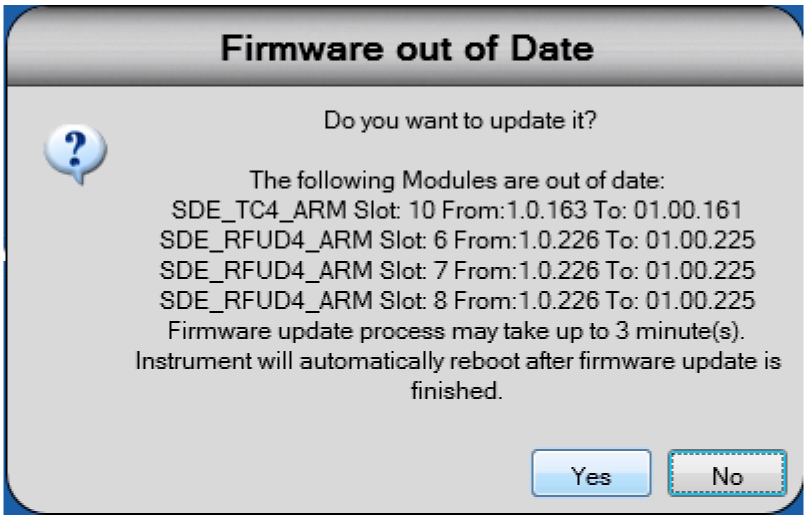


Figure 185. Firmware Out of Date dialog box.

NOTE:
It can take up to 45 minutes for a complete installation/upgrade. At the conclusion of the procedure, if a firmware upgrade has occurred, you will be informed that the instrument is about to reboot in order to complete the installation/upgrade. Then the instrument will reboot of its own accord.

2.23.2. Upgrading Software on the Laptop

To upgrade the Vertex software on the laptop computer provided with the Vertex instrument:

1. Connect the USB drive to one of the USB ports on the laptop.
2. Using Windows Explorer, navigate to the ***<Vertex Installer Root>*** folder.
3. Double-click on the file **VertexInstall.exe** and follow the installation instructions.
4. If you are prompted to upgrade the instrument firmware during the installation process, click the **Yes** button to install the new firmware.

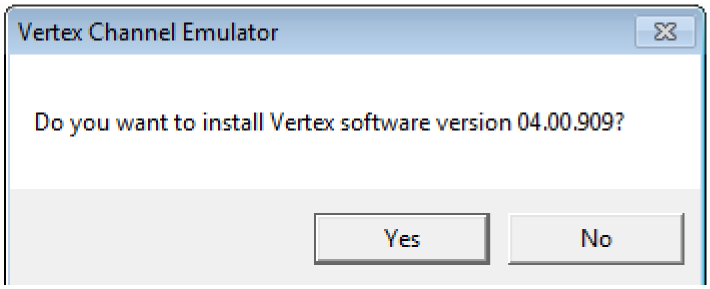


Figure 186. Sample Installation dialog box.

The License Agreement dialog box also appears.

5. Read the license agreement, and then click the **I Agree** option button.
6. Click the **Next** button.

The Welcome to the Vertex Channel Emulator Setup Wizard dialog box appears.

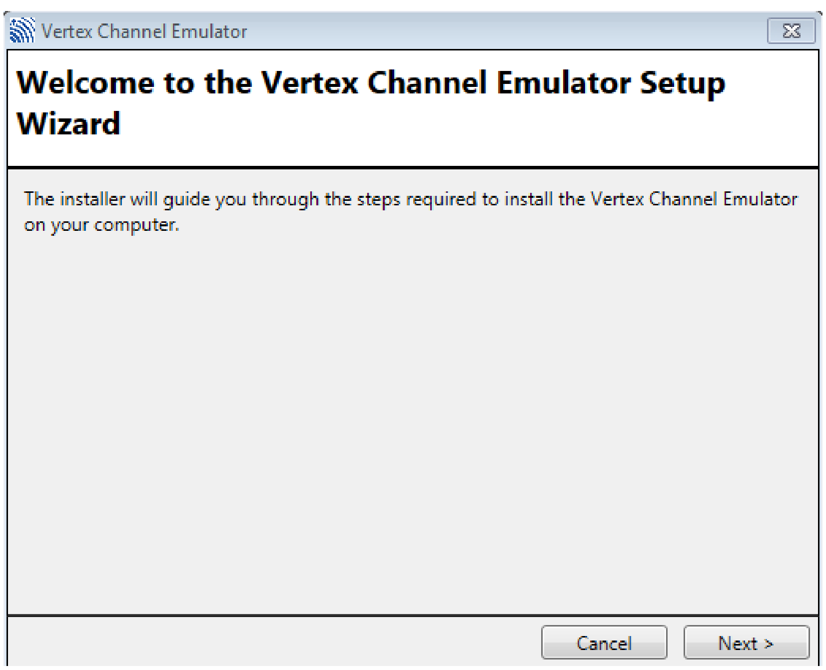


Figure 187. Welcome to the Vertex Channel Emulator Setup Wizard dialog box.

7. Click the **Next** button.

   The Checking Windows Updates dialog box appears as the installer checks the Windows Update to determine whether the Windows Service Pack KB3073630 is installed.

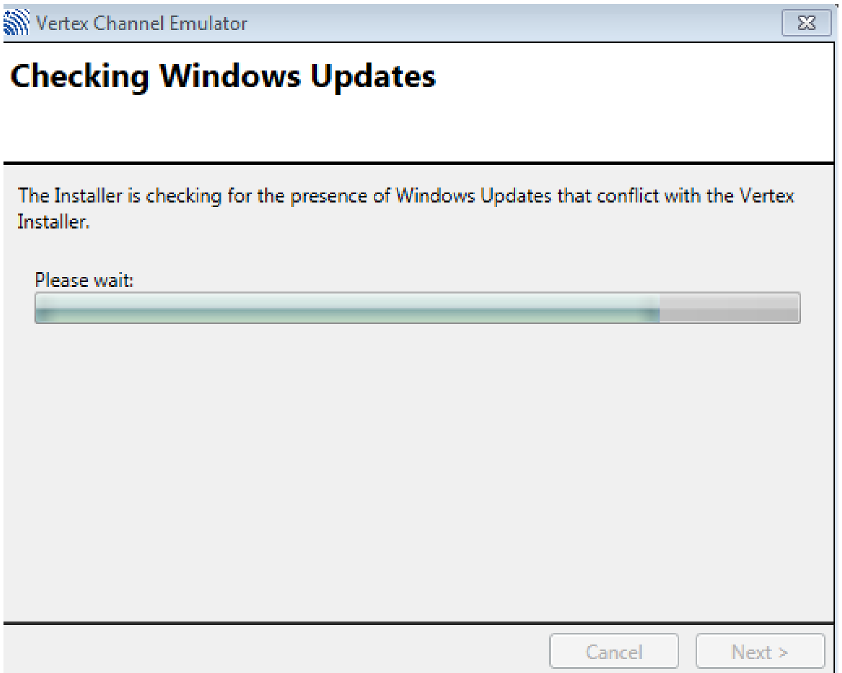


Figure 188. Checking Windows Updates dialog box.

If Windows Service pack KB 3072630 is installed on the PC, the following Warning dialog box appears.

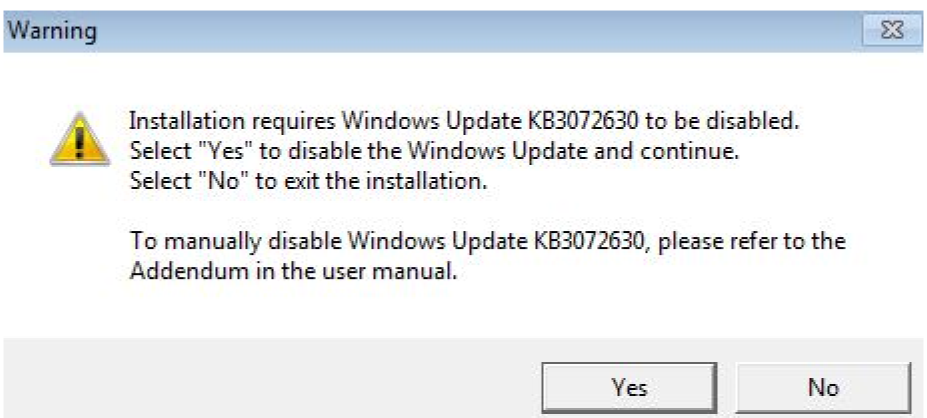


Figure 189. Warning dialog box.

8. Perform one of the following steps:
   - If the Warning dialog box does not appear, go to Step 9.
   - If the warning dialog box appears, click the **Yes** button, and then go to Step 9.
9. Click the **Next** button and confirm until the Installation Complete dialog box appears.

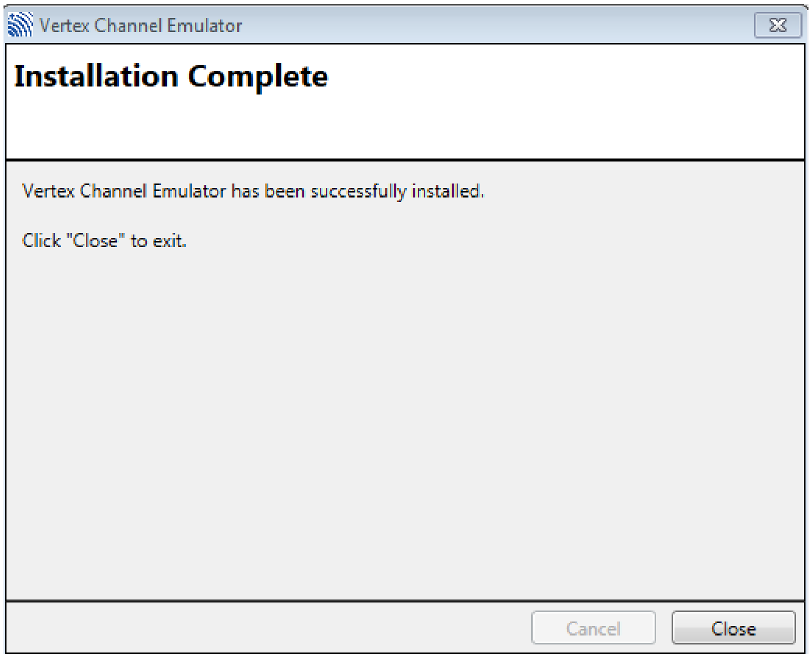


Figure 190. Installation Complete dialog box.

10. Click the **Close** button.

2.23.3. Upgrading Software on the Instrument Remotely from the Controller Laptop

If you upgrade software on the controller laptop first, you can also upgrade the Vertex instrument remotely with the controller laptop.

Perform the following steps:

1. From the remote controller laptop, launch the Vertex GUI.

   The Vertex GUI appears.

Figure 191. Start page of the GUI on the remote controller laptop.

2. Enter the IP address of the Vertex instrument, and click the **Connect** button.

   If the software version on the controller laptop does not match the software version on the Vertex instrument, the Vertex software will show the version mismatch information.

Figure 192. Version mismatch dialog box.

3. Click the **Yes** button to proceed.

   The License Agreement dialog box appears.

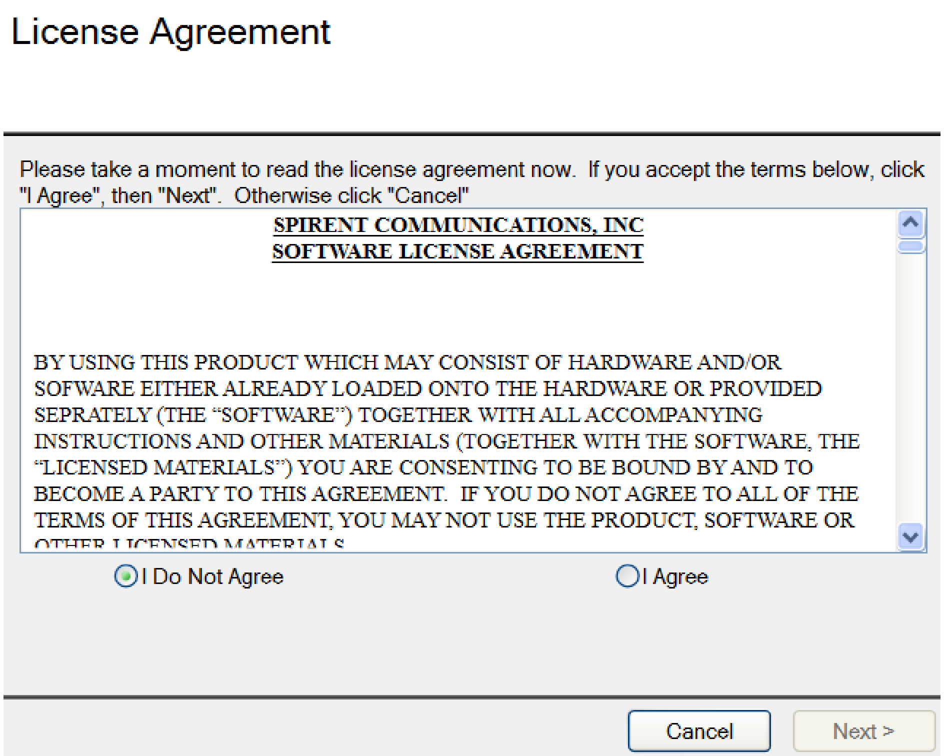


Figure 193. License Agreement dialog box.

4. Click the **I Agree** option button, and click the **Next** button.

   The Remote Update window appears, displaying the status of the update.

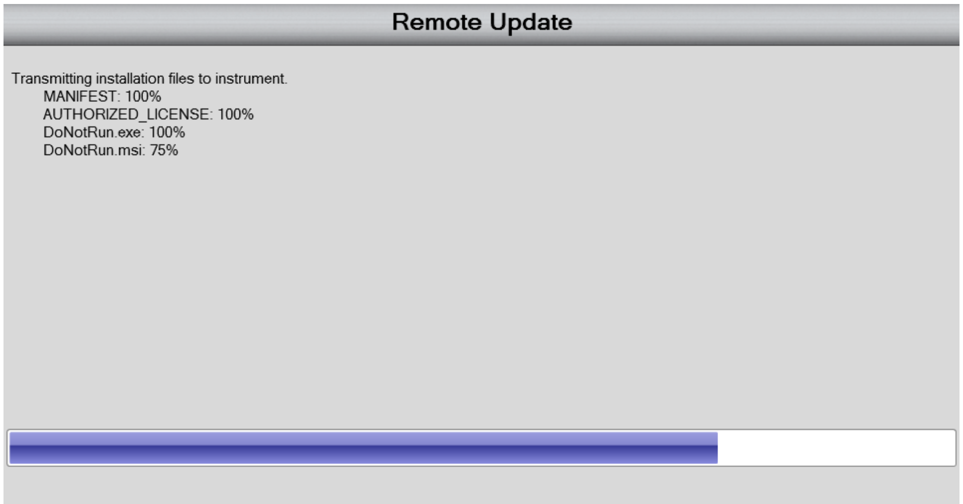


Figure 194. Remote Update window.

The Vertex instrument will automatically reboot once the update is finished.

After the Vertex boots up fully, the Remote Update window displays the message Remote upgrade complete!

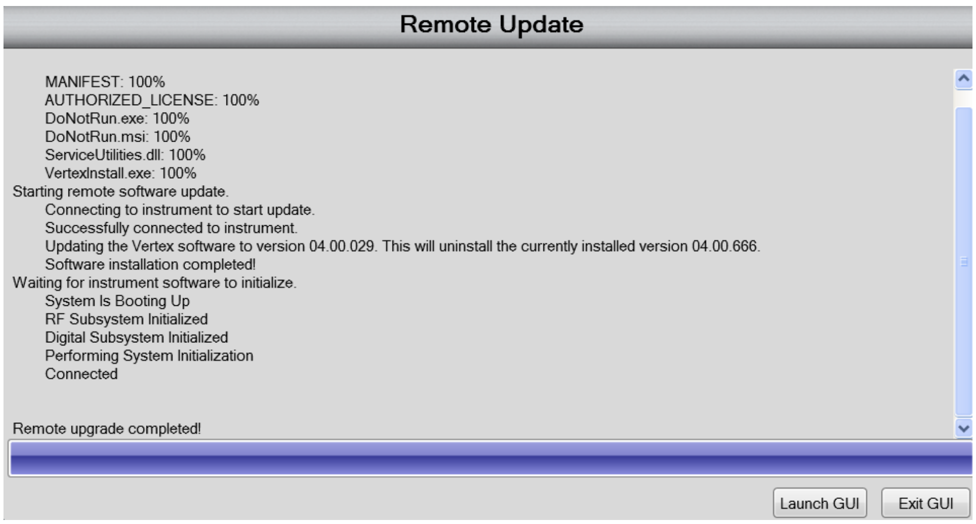


Figure 195. Remote Update window.

5. Perform one of the following steps:
   - If you want to use Vertex, click the **Launch GUI** button.
   - If you do not want to use Vertex, click the **Exit GUI** button.

2.24. Field Troubleshooting Tools

In the event that the Vertex instrument is not performing as expected, provide Spirent with the appropriate log information obtained from the instrument.

2.24.1. System Log File

Vertex continuously records debug information into a system log file located in the **FTPROOT** directory on the Vertex instrument.

To gather log file information to send to your Spirent Representative:

1. Connect a USB drive to the Vertex instrument
2. Close the Vertex GUI.
3. Using Windows Explorer, navigate to **D:\FTPROOT\Spirent\ Log**.
4. Copy the **LOGFILE.LOG** file to the USB drive.
5. Compress the file, and send the file to your Spirent Representative.

NOTE:
To ensure the log files do not surpass 100 MB in size, the logging mechanism switches log files periodically. If the history in **LOGFILE.LOG** does not go back far enough to encompass the field issue, compress and send the subsequent **LOGFILE_1.LOG** log file.

2.24.2. Module Versions

In addition to logging information, the Hardware Information can be valuable in further understanding instrument issues.

To access the Hardware Information:

1. In the Vertex GUI, select **Help>Hardware Information**.

   A window displays the firmware and board revisions of the unit.
2. Click the **Save To File** button, and send this file to your Spirent Representative.

2.25. Multi-Operators Control

Vertex supports multiple operators control, which enables multiple operators to simultaneously access a Vertex instrument remotely to monitor, configure or perform tests.

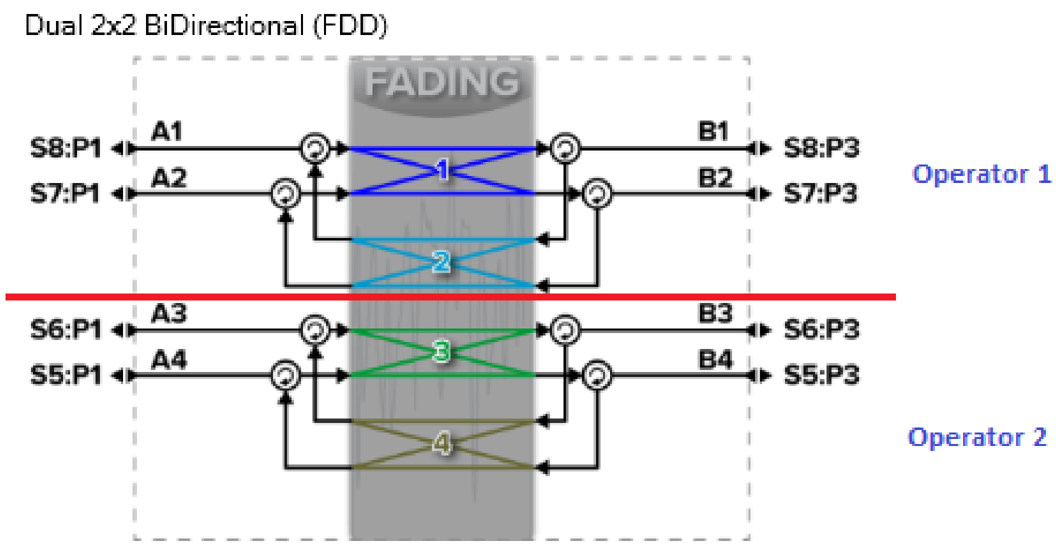


Figure 196. Two operators sharing the same Vertex instrument.

Since the fading engine is the same, if one operator changes parameters that may need fading reset, another test will be impacted. To avoid interruption without notice, Vertex allows the operator to “lock” the GUI by clicking the **Lock** button located in the lower, left corner of the Vertex GUI. The following figure shows the location of the **Lock** button.

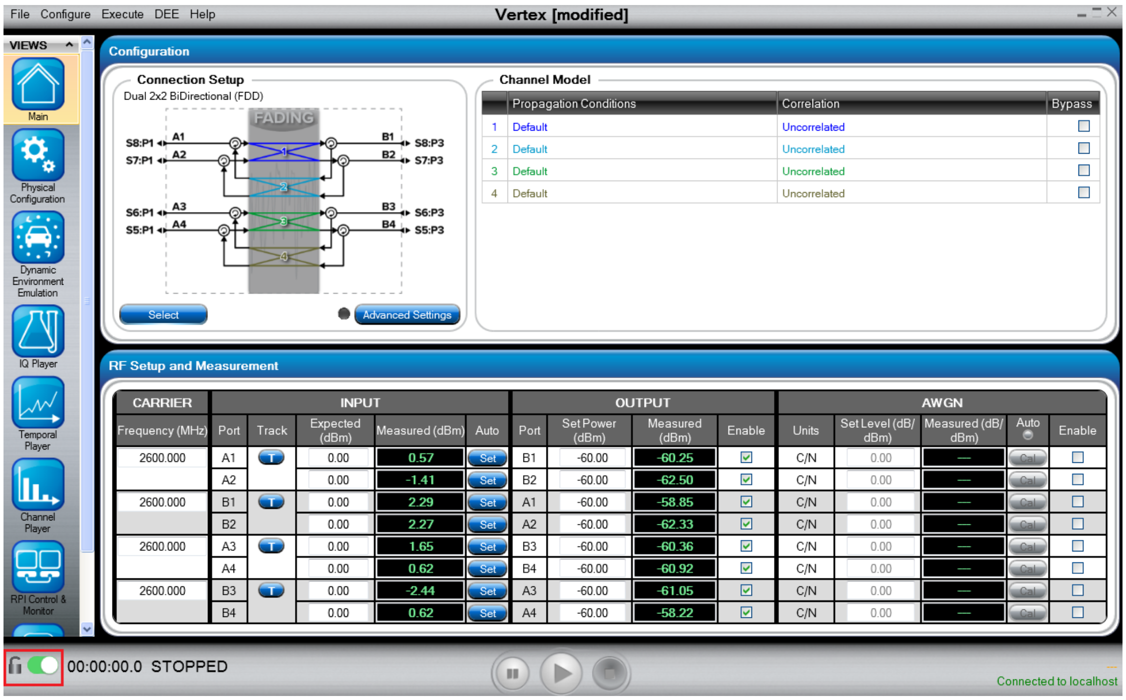


Figure 197. Location of the Lock button in the Vertex GUI.

Once the GUI is locked, other operators cannot change frequency or edit the fading profile until the GUI is unlocked.

You can still update the power level and AWGN when the Vertex GUI is locked because changing these parameters will not cause fading reset.

3. Remote Programming Interface (RPI)

3.1. Overview

The Remote Programming Interface (RPI) gives you the ability to control the Vertex channel emulator remotely. Using a computer or terminal, you can control the Vertex channel emulator by issuing commands through the Ethernet remote control port.

The RPI follows the LAN CR/LF (carriage return/line feed), which is a simple command-line protocol that allows you to control Vertex from a terminal or a computer using a TCP/IP socket connection. Before processing remote commands, you make a connection to the Vertex spatial channel emulator(s).

Vertex channel emulator commands and queries are arranged in a tree structure. The top of this tree contains headers and IEEE 488.2 mandatory commands and queries. Each header can have more headers and Program Messages (commands and queries) under it.

As described above, the Vertex channel emulator command set is made up of IEEE 488.2 mandatory common commands, as well as Vertex-specific commands. These commands adhere loosely to the SCPI protocol. This simplifies learning and using the command set if you are already familiar with other IEEE 488.2 instruments.

3.2. RPI Setup

This section describes how to set up the remote programming interface. For IP connection, you may need a USB to RS232 cable to query or configure the IP address of the Vertex instrument.

To set up the remote programing interface:

1. Connect the controller laptop (running the automation program) to the **User Control** Ethernet port on the rear panel of the Vertex channel emulator, as shown in the following figure.

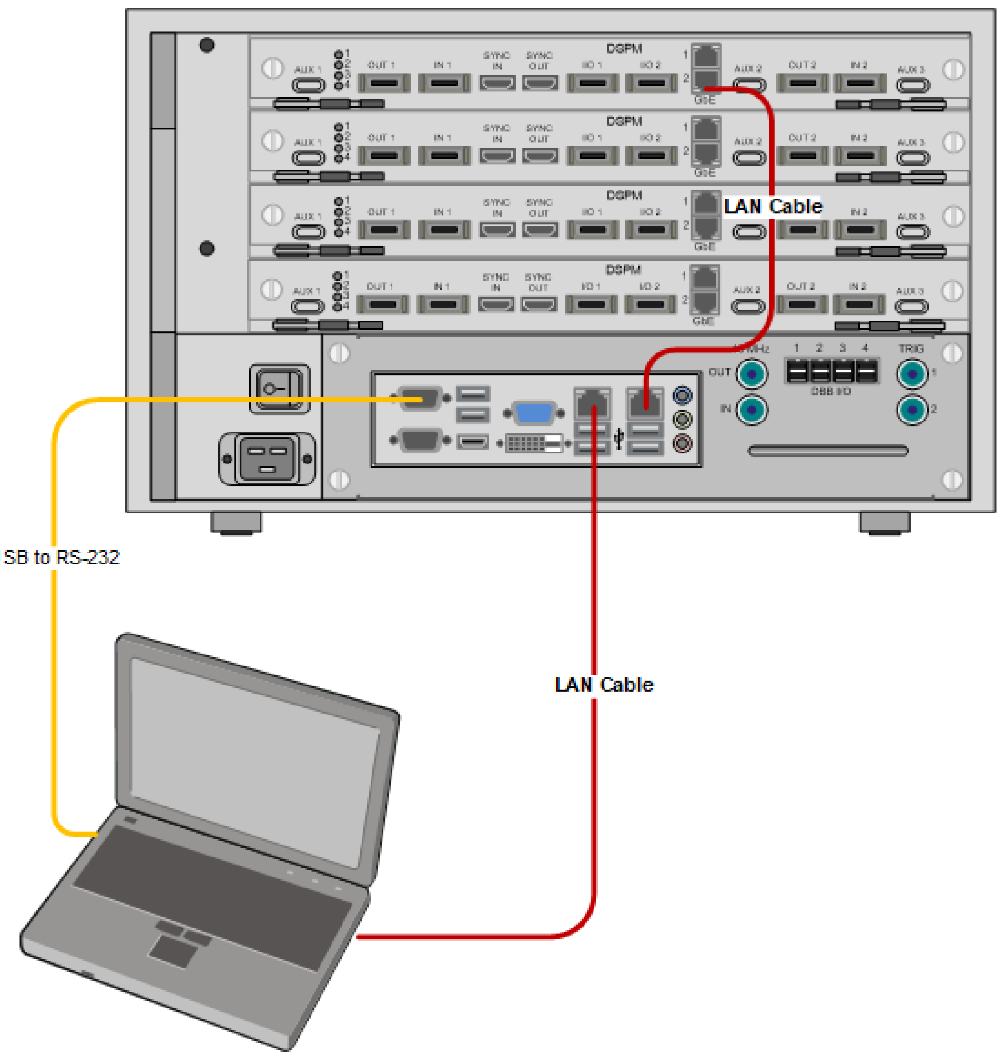


Figure 198. Vertex rear panel.

2. Ensure the unit is running properly.
3. Obtain the IP address and port to establish a Telnet connection to send commands and receive responses. To configure or obtain the IP and port settings, select the **RPI Control & Monitor** view, shown in the following figure.

   You can query or configure the IP address on the start page of the Vertex GUI.

   The port setting for the RPI is shown in the following figure.

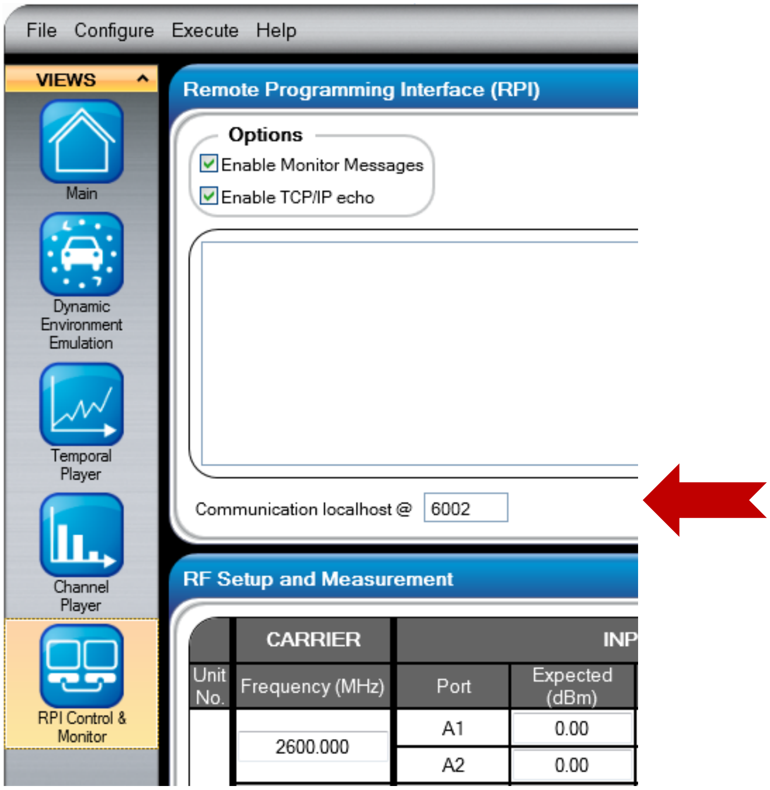


Figure 199. RPI view – port setting.

You can now establish a Telnet connection to control the Vertex through SCPI commands.

For legacy SR5500 users, the Vertex differs from the SR5500 in that the Vertex GUI does not have to be running to send RPI commands to the Vertex instrument.

3.2.1. Enable Monitor Messages

Enabling monitor messages allows the display of status information associated with commands received by the RPI.

NOTE:
Disabling these messages increases the speed of command processing.

3.2.2. Enable TCP/IP Echo

When enabled, the RPI echoes back all characters sent to it. This is useful when you are connected manually to the RPI through a TCP/IP client and need to view what is being typed.

NOTE:
When using automation software to control the Vertex through the RPI, Spirent recommends disabling this function to increase the speed of the RPI operation.

3.3. Vertex RPI Command Protocol

Beginning with release 4.70, Vertex supports both Telnet and SSH protocol. For SSH, Vertex also supports RSA encryption. For RSA encryption, you must place the public key in the folder **D:/FTPROOT/Spirent/RPI** on the embedded PC.

When accessing SSH using RSA, use the corresponding private key on the client application.

The default SSH login credentials are:

Username: spirent_rpi

Password: Sp!rent

Telnet access does not require a password.

3.3.1. Command Types

IEEE 488.2 supports two different program Message Unit types: **queries** and **commands**. Queries request a response from the unit being queried. Commands instruct the unit to carry out an action, such as set a parameter or execute a function.

GPIB common commands are defined by the IEEE 488.2 specification so that every unit that supports IEEE 488.2 responds to the command in the same way. Examples of such commands are ***IDN?**, which is a query requesting model and revision information, and ***RST**, which is a command that resets the unit to a known state.

Other commands are defined specifically for Vertex and loosely follow the SCPI convention. Related commands and queries are grouped together under headers, allowing them to be organized, and thus easier to understand.

3.3.2. Program Messages

A Vertex Program Message consists of one or more Program Message Units, which can be Command Message Units (commands) or Query Message Units (queries). IEEE 488.2 dictates that a semicolon (**;**) separates different Program Message Units sent together in a Program Message. A colon (**:**) indicates that the program mnemonic for the message being sent starts at the root of the tree.

It is possible to send multiple Program Message Units in a single Program Message, as long as they are separated by a semicolon. Only one query should be present in the message; sending two queries at once causes one response to be lost. You can also send each command or query separately.

The first Program Message Unit within a Program Message does need to start with a colon since it is assumed that the command starts at the root of the command tree. Subsequent Program Message Units are assumed to reside under the same header as the preceding command if the next command is not preceded by a colon. The following example shows two commands sent under the same header:

```
Vertex> CHM1:PATH1:DELay 10;MODE FIXed
```

This is equivalent to sending the following two commands separately:

```
Vertex> CHM1:PATH1:DELay 10
Vertex> CHM1:PATH1: MODE FIXed
```

The **`Vertex>`** represents the prompt for RPI commands.

Vertex ignores extra white space within the command frame, and characters are not case sensitive.

Query Message Units (queries) are Program Message Units that ask Vertex to report back a response of some kind, usually a parameter value. Queries have a question mark (**?**) after the command name as shown in the following example:

```
Vertex> CHM1:PATH1: DELay:VALue?
```

A possible response for this query might be:

```
10.0000
```

3.3.3. Command Sequence

To execute a Vertex Program Message Unit:

1. Check for any pending responses. The Vertex channel emulator does not execute a new Program Message Unit if the result from a previous message has not been read.
2. Send the command or query to the Vertex instrument.
3. After a query is sent, execute a serial poll, and then read the query response from Vertex.

In the LAN CR/LF protocol, it is not necessary to check for pending responses. The unit automatically queues any responses and error messages. Nevertheless, it is highly recommended that you query the response before sending the next command. A common practice is to append a **:ERR?** query after each command. For example:

```
Vertex> CHM1:PATH1:DELay 10;:ERR?
```

Notice that a semicolon separates the two actions, and a colon indicates that the following message should start from the root of the command tree. By sending the above message and reading back the response, you will see any errors that might have arisen.

Default Commands and Headers

Many commands, queries, and command groups have defaults associated with them. For example, under each header, there is a default command, query, or another header that does not have to be explicitly stated to be understood. The **CHM1:PATH1:DELay** header has many commands under it. The **VALue** command is the default command. Because of this, the following two commands have the same meaning:

```
Vertex> CHM1:PATH1:DELay:VALue 10
Vertex> CHM1:PATH1:DELay 10
```

In the second example, the fact that **VALue** is being referenced is implied. The same holds true for queries.

3.3.4. Response Format

IEEE 488.2 Command Message Units (commands) do not generate responses. IEEE 488.2-compliant units only generate responses to Query Message Units (queries). The Vertex channel emulator requires you to request the response.

The Vertex response format is a "Headerless" format. This format skips the command header and sends the value being requested.

The following is an example of a Headerless response from the Vertex instrument:

```
Send query:                    CHAN1:PATH1:DELay?
Receive response:              10.000
```

3.3.5. Long Form and Short Form of Mnemonics

Every mnemonic has both a long form and a short form. You can use either of these forms when sending commands and queries. Typically, the short form of the mnemonic is shown in capital letters, and the long form is a combination of the short form plus any lower case letters. IEEE 488.2 dictates that Program Messages can be sent in capital letters, lower case letters, or any combination of the two. An example of the command long form and short form is as follows:

```
Vertex> CHM1:PATH1:DELay 10.0
```

The mnemonic called **DELay** is a command under the PATH1 root command group. Its short form is **DEL** and its long form is **DELay**.

The following are valid combinations:

```
Vertex> CHM1:PATH1:DELay 10.0
Vertex> CHM1:PATH1:DEL 10.0
Vertex> CHM1:PAtH1:dElAy 10.0
```

The first example shows the first mnemonic using the long form. The second example shows the short form. The third example shows a combination of upper and lower case letters.

The following are not valid combinations:

```
Vertex> CHM1:PATH1:DELa 10.0
Vertex> CHM1:P1:DELay 10.0
```

In the first example, the **DELay** mnemonic is neither in long form or short form. In the second example, the **PATH1** mnemonic is invalid.

3.3.6. Hierarchical Default Format

There are default sub-mnemonics for many instances of the hierarchical level of a command structure. When these commands are addressed, you are not required to explicitly enter the default sub-mnemonics. Use the default sub-mnemonics for both parameter set and query commands.

For example, the Channel 1 path 1 delay command is presented as:

```
Vertex> CHM1:PATH1:DELay:VALue?
```

The current setting for this frequency can be queried in several ways:

```
Vertex> CHM1:PATH1:DELay:VALue?
```

```
Vertex> CHM1:PATH1:DEL:VAL?
```

```
Vertex> CHM1:PATH1:DEL?
```

The first example is the long format of the full implementation of the query. The second example is the short form of the full implementation of the query. The third example takes advantage of the defined default sub-mnemonics to shorten the query text.

3.3.7. Error Message Format

The **:ERR?** query reports any errors that may have occurred. Errors are reported in the form “ERROR_NUMBER, ERROR_DESCRIPTION.”

The following is a list of possible error responses:

- “0, No Error.”
- “3, Communications failure.”
- “4, Connection failure.”
- “5, File load failure.”
- “6, File save failure.”
- “11, Autoset in progress.”
- “12, DEE in progress (DEE ON).”
- “13, DEE compile in progress.”
- “14, DEE is not supported under current topology.”
- “18, One or more units are restricted to API usage only.”
- “20, ICS command failed.”
- “21, Cannot supply doc as command does not exist.”
- “-100, Command error”: Command not understood.
- “-200, Execution error”: Unable to execute command.
- “-222, Data out of range”: Parameter data out of range.

- “-224, Parameter error”: Parameter data not understood.
- “-256, File name not found”: The specified file does not exist.
- ”-300, Operational error.”
- “-350, Queue Overflow: There is no more room in the Error queue. An error occurred but has not been recorded.”

3.3.8. Blocking vs Non-Blocking Commands

Most commands in the Vertex RPI are blocking in that the command prompt is not returned until the command has finished execution, and the Vertex is ready to receive the next command. There are some non-blocking commands, however, that start a process in the Vertex, whose status can be queried by the use of another RPI command. These non-blocking commands include:

[SYSTem]:PORT:{A,B}#:BAUToset

[SYSTem]:DEE:COMPile:BEGin

[SYSTem]:PHCalib:SINGle:BEGin

3.3.9. Vectorized RPI Command Indexing

Vertex supports the structure for changing multiple instances of a parameter within a single command. This can speed up automation scripts tremendously when iterating over parameters on each step of the automation. For instance, changing the RF output power of all ports to a common value, or each to different values, can now be accomplished in a single line.

To change all ports to a single value:

```
Vertex> PORT:B(1-8):OUTP -43
```

To change all ports to individual values:

```
Vertex> PORT:B(1-8):OUTP -30,-37,-42,-41,-66,-84,-64,-77
```

Within this vectorized syntax, ports can be selectively indexed using a valid index string including a combination of comma separated values and hyphen separated ranges, strictly increasing. For instance, ports 3,4,5,7 and 9 can be indexed using:

```
Vertex> PORT:B(3-5,7,9):OUTP -43
```

The above command will set the outputs of these ports to -43 and leave the outputs of all the other ports unaffected.

NOTE:
The Vectorized format is currently only valid for commands and not queries.

3.3.9.1 Nested Loops

For commands that have multiple indices in the mnemonic (such as CHM#:PATH#...), multiple vector indexing is supported by way of nested looping. The leftmost index will comprise the outer loop, with each inner loop built from the vectorized indexing moving right in the command. For example, the following command:

```
Vertex> CHM(1-3):PATH(1-2):RPL 1,2,3,4,5,6
```

is equivalent to the following sequence of RPI commands:

```
Vertex> CHM1:PATH1:RPL 1
Vertex> CHM1:PATH2:RPL 2
Vertex> CHM2:PATH1:RPL 3
Vertex> CHM2:PATH2:RPL 4
Vertex> CHM3:PATH1:RPL 5
Vertex> CHM3:PATH2:RPL 6
```

3.3.9.2 Supported Commands

The following commands are supported in the vectorized indexing format:

[SYSTem]:PORT:{A,B}#:INPut

[SYSTem]:PORT:{A,B}#:OUTPut

[SYSTem]:PORT:{A,B}#:LOSS

[SYSTem]:PORT:{A,B}#:RFOUT

[SYSTem]:PORT:{A,B}#:INPPHAse

[SYSTem]:PORT:{A,B}#:OUTPPHAse

[SYSTem]:PORT:{A,B}#:INPDelay

[SYSTem]:PORT:{A,B}#:OUTPDelay

[SYSTem]:PORT:{A,B}#:INFREQuency

[SYSTem]:PORT:{A,B}#:CFACtor

[SYSTem]:PORT:{A,B}#:ICBLloss

[SYSTem]:PORT:{A,B}#:OCBLloss

[SYSTem]:PORT:{A,B}#:INTerferer:[MODe]

[SYSTem]:PORT:{A,B}#:INTerferer:CTON

[SYSTem]:PORT:{A,B}#:INTerferer:EBNO

[SYSTem]:PORT:{A,B}#:INTerferer:NOISElevel

[SYSTem]:PORT:{A,B}#:INTerferer:BITRate

[SYSTem]:PORT:{A,B}#:INTerferer:NBWidth

[SYSTem]:PORT:{A,B}#:INTerferer:RBWidth

[SYSTem]:PORT:{A,B}#:MEASure:TTHReshold

[SYSTem]:PORT:{A,B}#:MEASure:IAVGexp

[SYSTem]:PORT:{A,B}#:MEASure:OAVGexp

[SYSTem]:CHM#:BYPass

[SYSTem]:CHM#:BYPAB

[SYSTem]:CHM#:BYPBA

[SYSTem]:CHM#:[PROP]:BULKdelay:[VALue]

[SYSTem]:CHM#:[PROP]:BULKdelay:STATe

[SYSTem]:CHM#:[PROP]:GCM:PATH#:AOA

[SYSTem]:CHM#:[PROP]:GCM:PATH#:AOD

[SYSTem]:CHM#:[PROP]:GCM:PATH#:BSAS

[SYSTem]:CHM#:[PROP]:GCM:PATH#:DELay:[VALue]

[SYSTem]:CHM#:[PROP]:GCM:PATH#:LOS:AOA

[SYSTem]:CHM#:[PROP]:GCM:PATH#:LOS:AOD

[SYSTem]:CHM#:[PROP]:GCM:PATH#:LOS:KFACtor

[SYSTem]:CHM#:[PROP]:GCM:PATH#:LOS:[STATe]

[SYSTem]:CHM#:[PROP]:GCM:PATH#:MODulation

[SYSTem]:CHM#:[PROP]:GCM:PATH#:MSAS

[SYSTem]:CHM#:[PROP]:GCM:PATH#:MSDirection

[SYSTem]:CHM#:[PROP]:GCM:PATH#:MSVelocity

[SYSTem]:CHM#:[PROP]:GCM:PATH#:RPLoss

[SYSTem]:CHM#:[PROP]:PATH#:DELay:[VALue]

[SYSTem]:CHM#:[PROP]:PATH#:DFRequency

[SYSTem]:CHM#:[PROP]:PATH#:DVELocity

[SYSTem]:CHM#:[PROP]:PATH#:FSHift:[VALue]

[SYSTem]:CHM#:[PROP]:PATH#:LOS:AOA

[SYSTem]:CHM#:[PROP]:PATH#:LOS:KRICian

[SYSTem]:CHM#:[PROP]:PATH#:MODulation

[SYSTem]:CHM#:[PROP]:PATH#:PHSHift

[SYSTem]:CHM#:[PROP]:PATH#:RPLoss

[SYSTem]:CHM#:[PROP]:PATH#:[STATe]

4. Channel Modeling Reference

4.1. Overview

Wireless communication is a demanding application that requires complex air interface protocols to seamlessly interact and harsh radio channel effects to be mitigated. When a wireless signal is sent from the transmitter to the receiver, it traverses a complex radio channel that distorts the intended signal transmission. Every aspect of the environment encountered by a signal from the time it is transmitted to when it is received is called the wireless channel. The transmitted signal takes multiple paths to the receiver. These paths are caused by the signal bouncing off reflective surfaces such as the ground, buildings, or trees. Mobility between the transmitter and receiver causes the characteristics of these paths to be time varying.

Multiple copies of the originally transmitted signal arrive at the receiver; each having taken a different route through the wireless channel. So, each copy travels a different length and accordingly has a different phase. The greater the bandwidth of the receiver, the greater its ability to resolve different copies separated finely in time. So, depending on the bandwidth of the receiver, these different copies are seen as arriving at the same or different instant in time. The copies of the signal that are seen as arriving at the same instant in time add vectorially with different phases to produce one path.

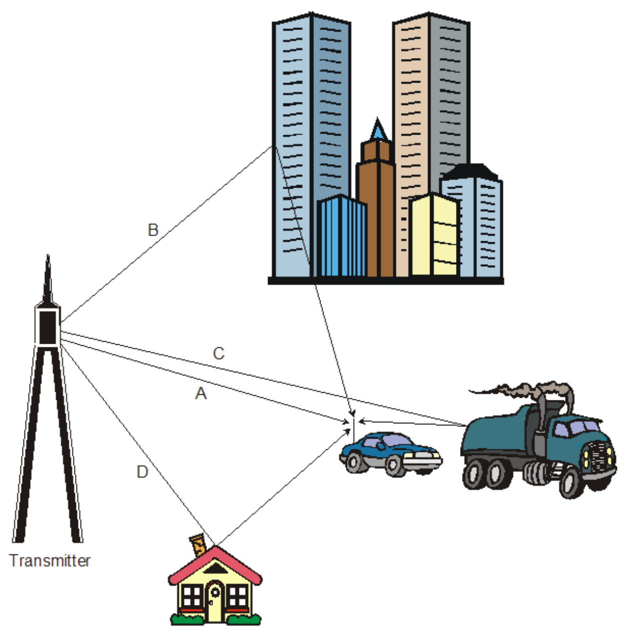


Figure 200. Typical multi-path fading scenario.

The preceding figure is a diagram of a typical mobile receiver (the car) as it drives along a roadway.

A, B, C, and D depict four of the many paths from the transmitter to receiver.

This phenomenon where multiple copies of the transmitted signal are received staggered in time and with different average power levels is called *multi-path*.

In the limit case with infinite bandwidth, every copy of the signal or “path component,” no matter how insignificant, is resolvable and produces thousands of paths. However, actual bandwidths filter the ability to resolve different paths and lead to lower and more practical numbers of paths we use in today’s channel models.

Fast and slow fading describe the time variation of the received signal level around an average power level. Fast fading describes the signal variations of a path that take place over the course of several milliseconds. Each path is the result of the vectorial addition of multiple copies of the signal, each having a different phase. This results in constructive and destructive addition of the different copies, leading to the phenomenon of fast fading. These multiple received transmissions are generated by scattering caused by the small objects in the environment (within a few hundred wavelengths of the receiver).

While fast fading effects are attributed to local scattering of the transmitted signal, large scatterers in the environment introduce slow fading effects that vary over tens (10s) or hundreds (100s) of milliseconds. These signal variations are caused by aspects of the environment, such as a mountain or large building, getting in between the transmitter and receiver and partially blocking signal reception. Slow fading is often described as shadow fading, since in effect, the geographic element casts a shadow on the receiver. Amplitude variation fluctuations happen at a slow rate.

4.2. Power Delay Profile (PDP)

In wireless communications, a signal transmitted to a receiver can arrive having traveled over many different paths through the wireless channel. On its way to the receiver, a transmitted signal may take the direct line of sight path or may bounce off reflecting surfaces before arriving at the receive antenna. Since these multiple copies of the original transmitted signal travel different distances, they arrive at the receiver staggered in time with different average power levels.

The impulse response of the wireless channels is used to characterize what predominant paths are present between the transmitter and receiver at a given time. Using the impulse response method, a short transmit signal is broadcast through the radio channel, and multiple copies of the original signal are captured and measured at the receive antenna. The result is displayed in the form of a Power-Delay Profile.

An example Power Delay Profile is shown in the following figure. This example shows four copies of the original transmitted signal arriving at the receiver. The Y-axis describes the relative power of each of these paths at the receive antenna. The X-axis describes the relative time difference between the paths as they arrived at the receiver. Since the wireless channel is dynamic, the amplitude and relative delay characteristics of the paths in the Power Delay Profile vary over time. The following sections describe various characteristics of the paths illustrated by the radio channel's Power Delay Profile.

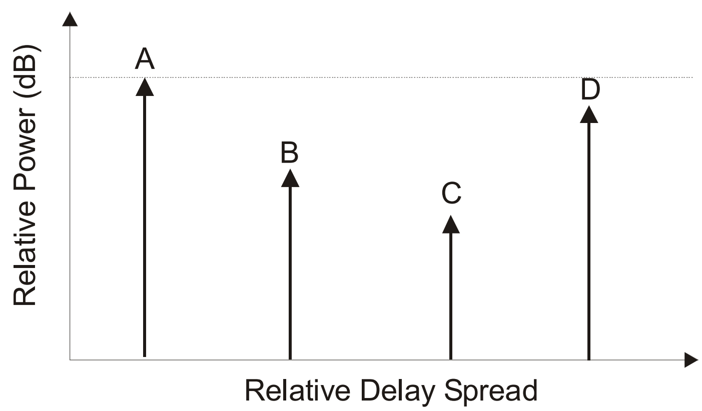


Figure 201. 2D – Power Delay Profile of the typical multi-path fading scenario.

4.3. Static Relative Path Delay

Relative path delay is a phenomenon where individual signal paths from the transmitter to the receiver arrive at different times. An example of this is shown in the preceding figure between Paths (A) and (C). Path (C) arrives at the receiver (the automobile) a finite time after signal Path (A). The net effect of the arrival time difference is to spread the originally transmitted signal at the receiver in time. In digital wireless communications, this causes received symbols to overlap, resulting in inter-symbol interference.

The amount of relative path delay varies with terrain and application. In an indoor application, delays could be in the tens (10s) of nanoseconds (ns), where 10 ns is about 10 feet. In outdoor applications, delays of 10 microseconds (μs) or less are typical (1 μs is about 1000 feet). Delays greater than 50 μs are rare in cellular environments.

Path delay in the Vertex is set relative to the first arriving path. This delay setting is in addition to the absolute electrical delay through the system.

4.4. Bulk Delay

Bulk delay is a larger amount of delay that can be applied to the propagation conditions. For example, the Vertex instrument can add up to four milliseconds (ms) of delay. Any amount of Bulk delay added applies to all of the paths equally. Bulk delay does not affect the relative delay spread of the paths. Bulk delay can be used to emulate environments in which the transmitter and receiver are extremely distant from one another or to stress a receiver design.

4.5. Time-Varying Relative Path Delay

A Power Delay Profile, shown in the following figure, provides a snapshot of the impulse response of a radio propagation channel. In mobile applications, the number of paths in a Power Delay Profile and their location along the delay spread X-axis would remain constant over several meters. In many cases, the impulse response of a radio channel is averaged over this small distance (which translates into a short-period of time with mobility) to provide a “static” or wide sense stationary view of channel conditions. As a mobile wireless terminal moves over a wider area, the shape and characteristics of the Power Delay Profile change dramatically.

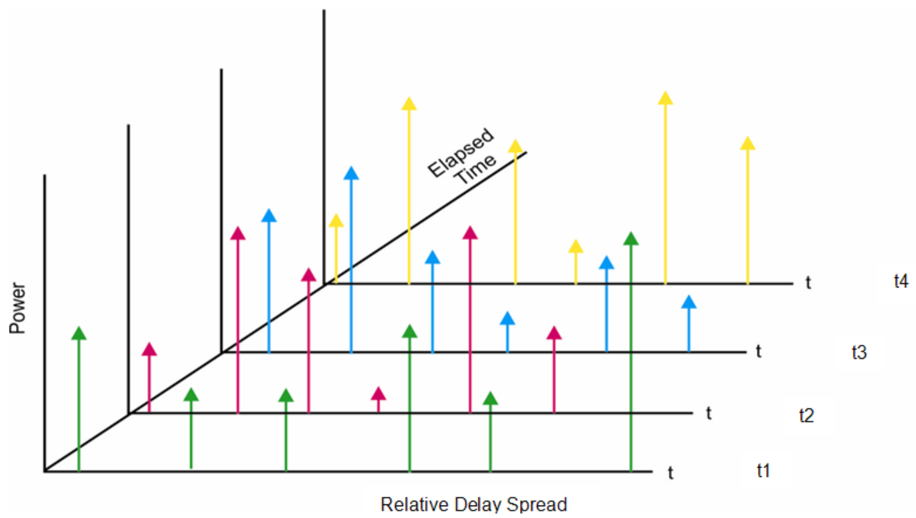


Figure 202. 3D plot showing time-varying PDP.

Modern wireless communications systems must adapt to these dramatic changes to continuously mitigate the impact of multi-path delay spread. To accurately evaluate the performance over a time-varying Power Delay Profile, a fading emulator must be able to emulate the time-varying changes in the paths delay characteristics. The following sections describe popular employed models to emulate dynamic delay spread.

4.5.1. Moving Propagation - Sliding Relative Path Delay

Popular channel models feature Moving Propagation Power Delay Profiles with time-varying delay spread to evaluate the ability of a receiver to adapt to dynamic changes in the radio environment caused by mobility. These models may specify the use of paths with sliding delay characteristics. 3GPP test specifications define Moving Propagation channel models that utilize paths that possess sliding delay with a sinusoidal variation in delay spread. Vertex sliding delay emulation smoothly varies the temporal location of individual multi-path components using a periodic sinusoidal function. A two-path example is shown in the following figure. In this example, Path 1 has fixed delay (t0) while Path 2 has sliding delay oscillating over the delay range of $\Delta\tau$.

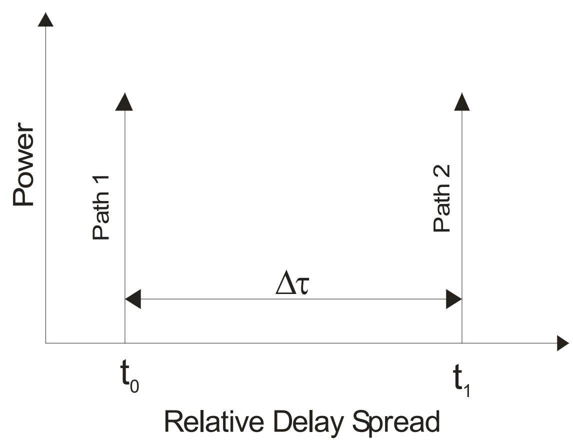


Figure 203. Sliding Delay example.

Several parameters must be defined for paths employing Moving Propagation, including:

- Minimum Delay – minimum delay of the sliding path
- Maximum Delay – maximum delay of the sliding path
- Rate of Oscillation – rate of sliding delay change
- Delay Period – time of one sliding delay period
- Initial Phase - initial phase of the sinusoidal function, which defines the starting delay of the emulation.
- Birth-Death - Time-varying Relative Path Delay

As an alternative to changing the delay spread of a path by sliding the path along the delay axis, some channel models employ Birth-Death time-varying delay emulation. The Birth-Death emulation method randomly varies the location of the paths in the Power Delay Profile along the delay-spread axis. Paths take turns hopping between pre-defined delay spread bins. An example Birth-Death sequence is illustrated in the series of power-delay profiles found in the following figure.

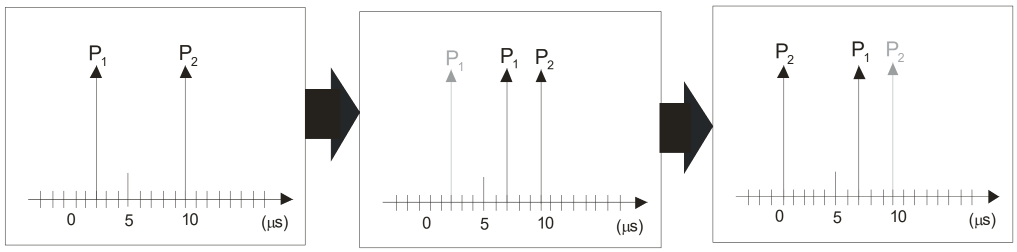


Figure 204. Birth-Delay delay example.

Birth-Death paths have fixed delay value during each defined state but change delay value during a state change. Birth-Death paths participate in the Birth-Death sequence by taking turns changing their location along the delay spread axis. During each state, only one path changes its temporal delay location. This “death” of the path in its current delay bin and subsequent “birth” in a new unoccupied bin is performed using a uniform random distribution. You define the individual delay bins that make up the distribution set.

Several parameters must be defined for paths participating in the Birth-Death sequence. These include:

- **Number of Bins**: Defines the number of bins that paths configured for Birth-Death delay will hop between.
- **State Duration**: Defines the time between delay state changes.
- **Delay Bin Values**: Defines the location of the individual delay bins used in the Birth-Death sequence.

4.6. Relative Path Loss

Relative path loss is a phenomenon where individual paths arriving at the receiver are at different absolute power levels. The difference in power levels between paths is due to the fact that different paths take different routes in the wireless environment. Referring to Paths (A) and (C) in the following figure, Path (C) arrives at a lower power level than Path (A). This occurs since some amount of the power in signal Path (C) is lost when it reflects off the truck. Signal strength also varies as a function of the distance the signal travels. The loss of signal strength should follow the 1/d2 law in free space, where d is the distance between the transmitter and the receiver. In the actual cellular environment, the loss is much worse (between 1/d3 to 1/d6) due mainly to variations in the terrain.

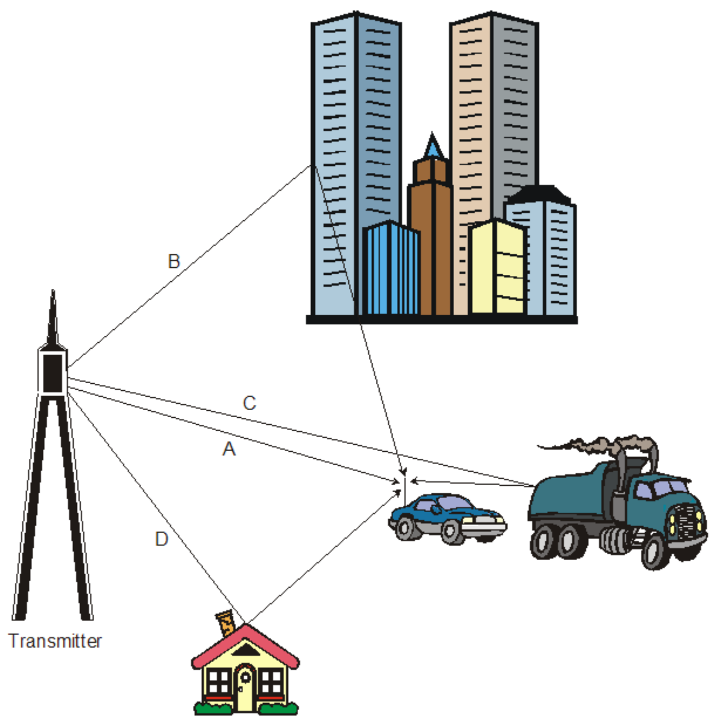


Figure 205. Transmitter to receiver signal diagram.

4.7. Fast Fading

Fast fading is generated by local scattering of the individual paths in the Power-Delay Profile in close proximity to the receiver. This scattering creates a large number of reflected signal transmissions that arrive at the receiver at relatively the same time (with respect to the inverse of receive signal bandwidth) with random phase and amplitude caused by the difference in distance traveled. Several different mathematical distributions are commonly used to model the amplitude and phase characteristics of the fast fading phenomena. These include the Rayleigh and Rician fast fading amplitude distributions.

4.7.1. Rayleigh Fading Amplitude Distribution

Fast fading is commonly referred to as Rayleigh fading. A Rayleigh modulated signal is caused by scattering of the paths in the Power-Delay Profile from man-made and natural obstacles such as buildings and trees in the local geographical area (within a few hundred wavelengths of the receiver). It is formed by a large number of these scattered (reflected) signals combining at the receiver. Each of these signals has a random phase and amplitude at the receiver due to the reflections and difference in distance traveled.

The phenomenon that creates Rayleigh fading can be easily illustrated using a simple two path example. At the receiver, the two paths can be of any amplitude and phase. If the two paths are of the same amplitude, and their phase is 180o apart, there will be total destructive interference and no resultant signal. If the two signal paths are 0o apart in phase, there will be constructive interference, and the signal envelope will be 3 dB larger than the individual path's amplitudes.

The signals rarely combine to greater than 10 dB above the individual path's power. The deep fades (destructive interference) would range from just a few dB to fades of greater than 50 dB. The spacing and amplitude of the fades are a function of the carrier frequency. At 900 MHz, the deep fades will occur at the mobile every few centimeters apart.

The fades and peaks of the signal envelope follow a Rayleigh distribution. This causes the signal strength to fluctuate rapidly between slightly higher levels to deep fades of greater than 50 dB. The following figure shows an example of the Rayleigh faded signal versus time. Rayleigh fading is called fast fading since the fluctuations are so rapid, as compared to log-normal or slow fading.

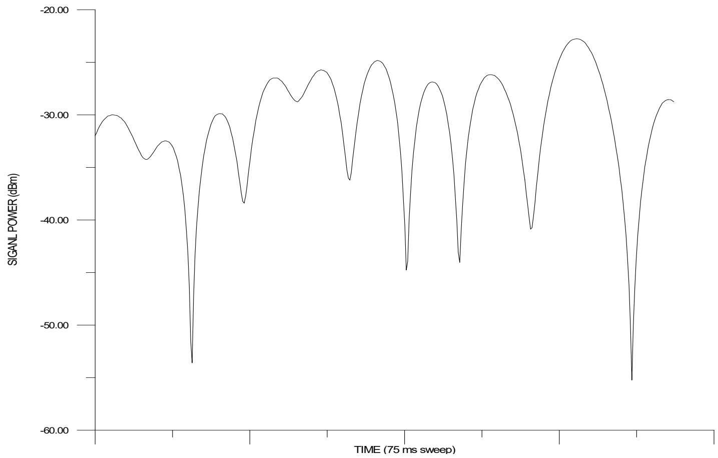


Figure 206. Rayleigh faded signal vs. time.

- Doppler Freq. = 100 Hz
- Center Freq. = 900 MHz
- Span = 0 Hz
- RBW = 100 kHz
- Sweep Time = 75 msec

The Rayleigh distribution is generated using a complex I/Q modulator. The I/Q signals are modulated with two Gaussian distributed signals. Since Rayleigh fading occurs when there is relative movement between the transmitter and receiver, the signal is subjected to a Doppler shift (frequency shift). As a result, the spectrum of Rayleigh fading is limited to plus or minus the Doppler frequency (which is a function of the vehicle velocity) assuming that there is an equal probability that the signal is received with an arrival angle anywhere within the range from 0 to 360 degrees. The theoretical power spectral density of a Rayleigh faded signal is shown in the following figure.

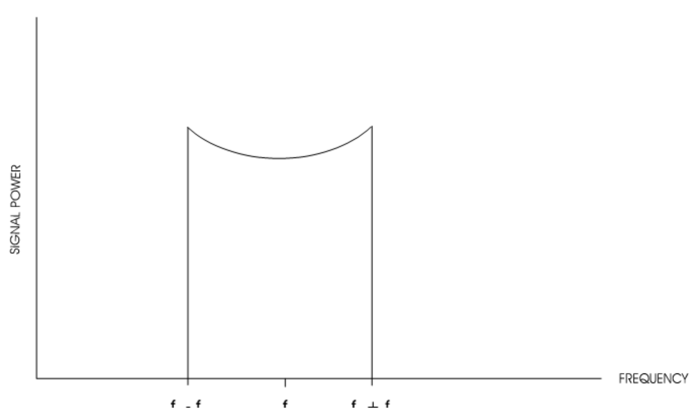


Figure 207. Theoretical Rayleigh power spectral density.

4.7.2. Rician Fading Amplitude Distribution

Rician fading is formed by the sum of a Rayleigh distributed signal and a Line-Of-Site (LOS or direct path) signal, where the LOS signal is typically subjected to a static frequency shift (static Doppler). A fading environment typically associated with Rician fading is that where one strong direct path reaches the receiver at roughly the same delay as multi-path from local scatterers.

The Vertex channel emulator supports the general case of Rician fading with programmable Angle of Arrival (AOA) and K factor. In the general case of Rician fading, the arrival angle of the LOS path at the receiver is programmable, as is the ratio of power between the LOS path and the multi-path. The Vertex channel emulator provides access to both the LOS arrival angle specified as the AOA (expressed in degrees) and the LOS path to multi-path power ratio specified as the K factor (expressed in dB). Changing the LOS arrival angle will move the relative location of the direct path with respect to the faded spectrum by changing the static Doppler shift of this component. This Doppler shift is set according to the following equation:

Doppler direct component = Doppler faded component x cosine (LOS arrival angle)

The K factor setting then controls the relative power of the direct path and the multi-path and has a valid range of -30dB (faded spectrum will dominate) to +30dB (LOS signal will dominate).

An example configuration of Rician fading may have an angle of arrival of the LOS signal path set to be 45o, resulting in a Doppler shift that is ~0.707 of the maximum Doppler shift of the Rayleigh distributed signal (classical Doppler spectrum). Furthermore, if the signal power of Rician fading is split equally between the LOS and multi-paths (where the power envelope of the multi-paths combine to from a Rayleigh distribution), this corresponds to a K factor setting of 0 dB. A theoretical power spectral density for this example of Rician fading is shown in the following figure.

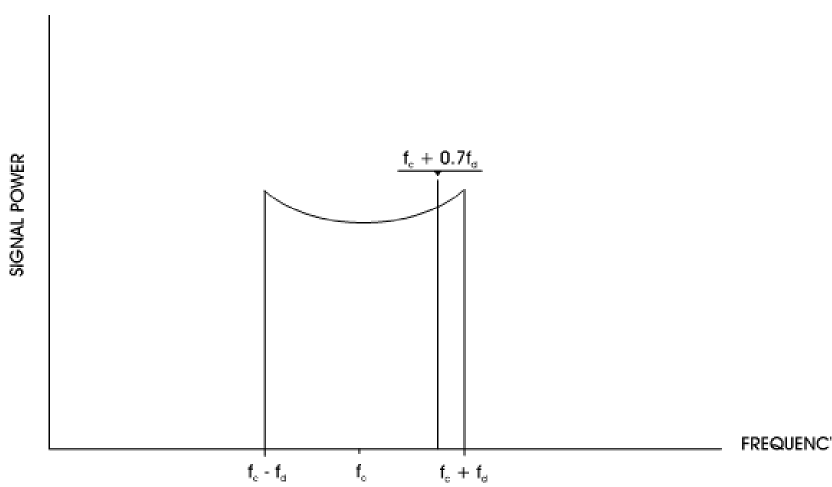


Figure 208. Theoretical power spectral density for Rician fading.

4.7.3. Fast Fading Power Spectrum Shapes

Rayleigh fading and Rician fast fading describe the amplitude distribution of the faded signal. However, several different frequency domain models can be used to represent the power spectrum shape produced by multi-path fading.

The Vertex channel emulator allows you to select the shape of the power spectrum produced by multi-path fading. The four possible spectrum shapes that can be set are shown in the following figure. The first shape, Classical 6 dB, is the most commonly used model and adheres to the spectral requirements detailed in many mobile communications standards for Rayleigh fading conditions. The Flat spectrum shape has been determined to be representative of the multi-path propagation effects experienced in some indoor applications. The Classical 3 dB, Rounded, and Rounded 12 dB spectrum shapes are also available in the Vertex channel emulator.

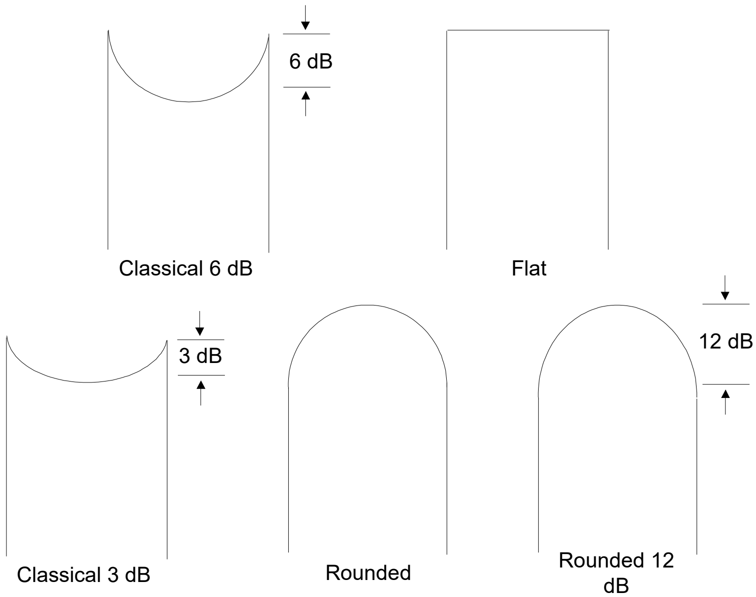


Figure 209. Fading power spectrum shapes.

4.8. Static Amplitude Channel Effects

In some cases, it is desirable to emulate single reflected paths that do not undergo local multi-path scattering and thus have static, or constant, amplitude. While these paths have fixed amplitude versus time, they may be subjected to constant or time-varying phase modulation. These phase modulation effects are described in the following sections.

4.8.1. Fixed Frequency Shift

Fixed frequency shift from the carrier frequency occurs when the distance between the receiver and transmitter is changing. An example of this is when a mobile receiver (car) is driving away from the transmitter. Path (A) in Figure 137 has a static frequency shift due to the movement of the car. The amount of the frequency shift (Doppler frequency) from the carrier is determined by the following formula:

$$\mathrm{Freq}_{\mathrm{Doppler}} = \frac{\mathrm{Velocity}_{\mathrm{mobile}} \times \mathrm{Freq}_{\mathrm{carrier}}}{\mathrm{C}}$$

where:

C ≅ Speed of Light (3x10^8 m/s)

The Doppler frequency, caused by dynamic rotation of the path phase, can be either positive or negative depending whether the mobile receiver is moving away from or towards the transmitter respectively.

4.8.2. High Speed Train Frequency Shift

At the high speeds, both the Doppler shift and the rate of change of Doppler shift are very large, making it very difficult to maintain a mobile connection. Since one of the 3GPP's goals is to maintain mobility up to 350 km/h, HST tests are included in the 3GPP standards. The High Speed Train model focuses on a signal's Doppler Shift as a user equipment (UE) in a high speed train (300 -350 km/h) passes a Base Station (BS).

The model in the standards for the High Speed Train consists of a single static path whose instantaneous Doppler shift is given as:

$$f_s(t) = f_d \cos\theta(t) \qquad (2.2)$$

where fd is defined as:

$$f_d = f_c \frac{v}{C}$$

where fc is the carrier frequency, v is the speed of the mobile (in m/s), and C is the speed of light (3x10^8 m/s).

and cosθ(t) is defined as:

$$\cos\theta(t) = \frac{D_s/2 - vt}{\sqrt{D_{\min}{}^2 + (D_s/2 - vt)^2}}, \quad 0 \le t \le D_s/v$$

$$\cos\theta(t) = \frac{-1.5D_s + vt}{\sqrt{D_{\min}{}^2 + (-1.5D_s + vt)^2}}, \quad D_s/v < t \le 2D_s/v$$

$$\cos\theta(t) = \cos\theta(t \mod (2D_s/v)), \quad t > 2D_s/v \qquad (2.3)$$

where $D_s/2$ is the initial distance of the train from the BS, and $D_{\min}$ is the BS - Railway track distance, both in meters; v is the velocity of the train in m/s, t is time in seconds.

The actual shape of the frequency shift variation depends on the values of fd, Ds/2, Dmin, and v. The following figure shows the instantaneous frequency shift for Ds=1000 m, Dmin=50m, v=350km/h, and fd=1340 Hz (3GPP TS36.104 Scenario 1).

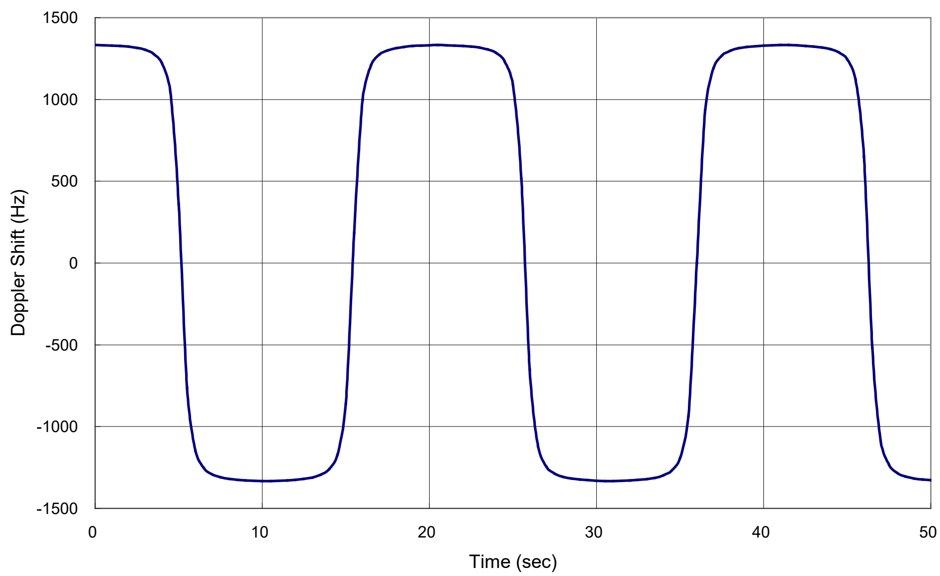


Figure 210. High-speed train frequency shift vs. time.

4.8.3. Phase Shift

A static phase shift is a result of a constant random distance between the transmitter and receiver. This distance is very rarely going to be an integer number of carrier wavelengths; a non-integer value will result in a static phase shift on the signal path. The amount of phase shift can vary between 0 and 360 degrees.

4.9. Slow or Shadow Fading

Slow or Shadow fading is the slow variation of the average signal power over time. A plot of signal power versus time for Shadow fading is shown in the following figure. Shadow fading is often characterized by a log-normal amplitude distribution. The time scale is much larger than that for Rayleigh fading as shown in Figure 142. The variation in signal strength at the receiver is due to blockage or absorption of the signal by large-scale variations in the terrain profile and by changes in the nature of the local topography in the path from the transmitter to the receiver. The blockage of the signal is caused by elements in the environment such as hills or a building. This phenomenon is often called shadowing since the receiver is passing through a large "shadow" of an object. An example of this is shown in the following figure as the mobile receiver (car) passes in the "shadow" of the building, the signal strength would fade.

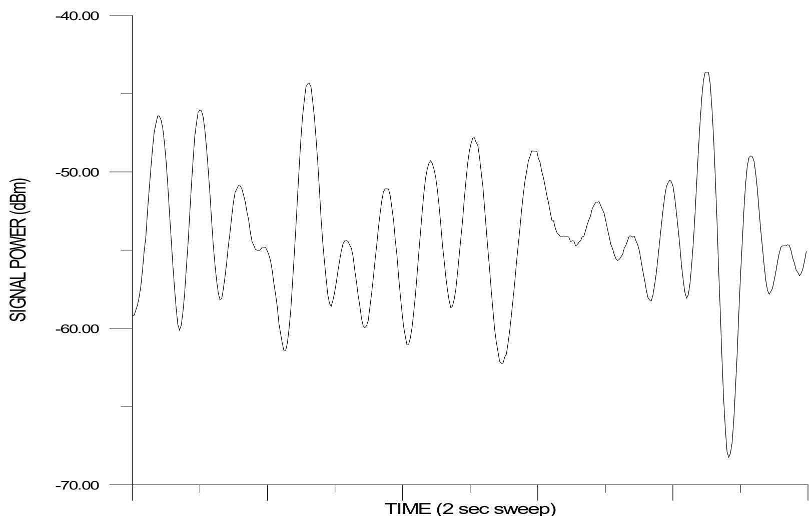


Figure 211. Log-normal fading vs. time.

- Log-Normal Standard Deviation = 10 dB
- Log-Normal Rate = 10 Hz
- Path Loss = 25 dB
- Center Freq. = 900 MHz
- Span = 0 Hz
- RBW = 100 kHz
- Sweep Time = 2 sec.

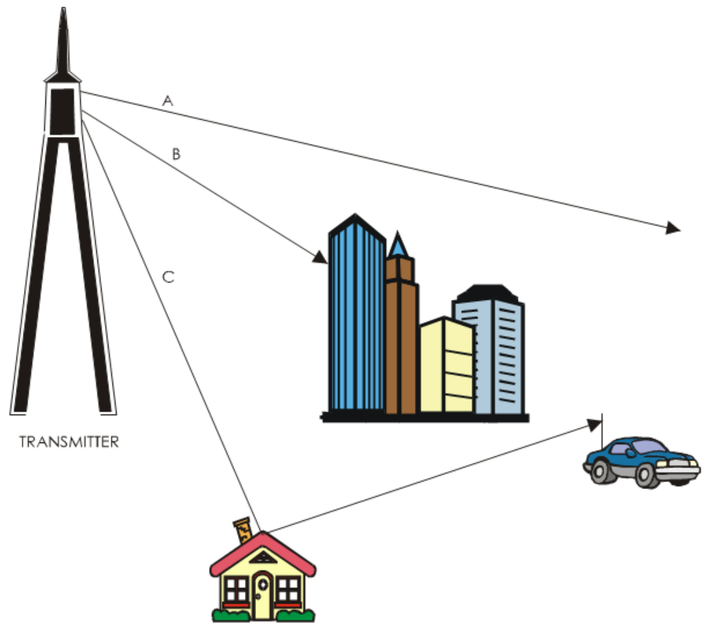


Figure 212. Transmitter to receiver Log-Normal diagram.

This fading has statistical characteristics that are represented by a log-normal distribution of fluctuations in the mean (average) signal power expressed in decibels (dB). The standard deviation of the log-normal distribution is determined by the characteristics of the terrain where the transmitter and receiver are located. For example, a standard deviation of between 6-8 dB is typical for urban areas, while a deviation of 10-12 dB can be observed in rural locations.

The maximum rate of the log-normal fading must also be specified. The rate of log-normal fading is the maximum frequency of the fading spectrum and defines the maximum pace that the mobile will move through the shadow of elements in the terrain. An example can be given of a mobile receiver (car) driving at a fixed speed along a road. If the car is in a rural area behind hills that are far apart, the log-normal rate would be small since the car is moving through “shadows” at a slow rate. If the car is in an urban area behind rows of buildings, the rate would be larger since the mobile would be passing through “shadows” at a higher rate.

The following relationship holds for log-normal fading:

$$\text{Log Normal Rate (Hz)} = \frac{\text{Mobile Velocity (m / s)}}{\text{Min. Shadow Length (m)}}$$

The log-normal frequency in this equation will be the maximum rate that the mobile will move through “shadows”. This corresponds to the maximum frequency of the log-normal fading spectrum that has a span that begins near DC.

5. XD5 Multi-link Duplexer Unit

5.1. Overview

The Spirent XD5 is a Multilink Duplexer unit. When used in conjunction with the Vertex channel emulator, the Spirent XD5 allows configurations to double the number of faded cells by fading only the downlink signals. This provides a clean, isolated uplink signal back from the UE. The XD5 is remotely controlled by the Vertex unit.

One Spirent XD5 has 8 RF ports for the eNB side and 2 RF ports for the UE side. One 16-channel Vertex unit can work with two XD5s to provide 16 RF downlinks.

The schematics shown in the following figure describe the conditions under which you can operate the XD5. Additionally, the XD5 provides summation circuitry so that the connected mobile device receives signal from multiple eNBs simultaneously, and its signal is transmitted back to each eNB.

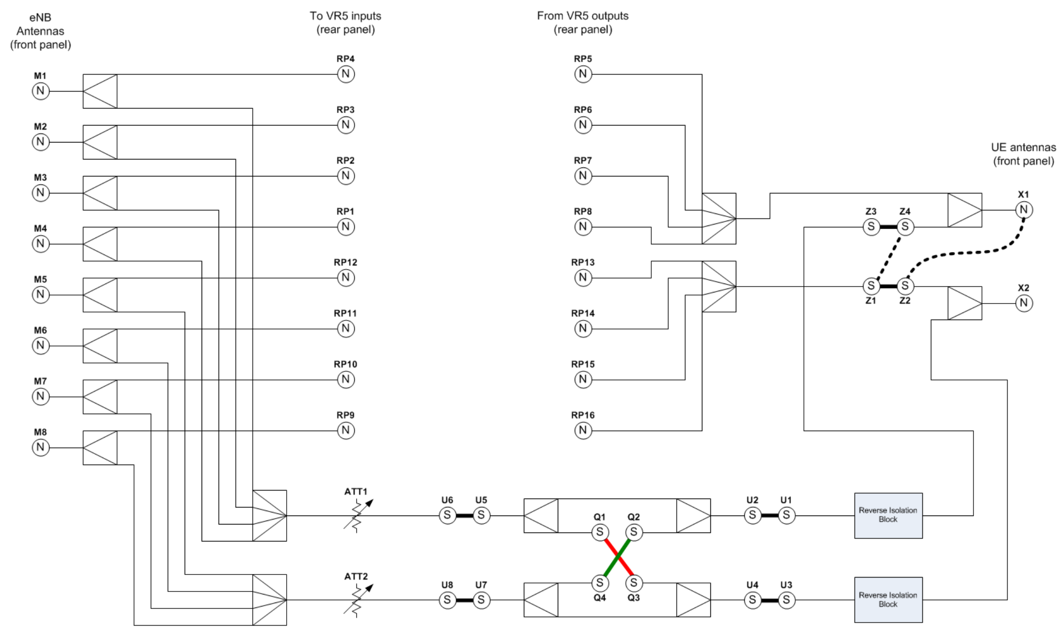


Figure 213. Schematic drawing of the XD5.

5.2. Front-Panel View

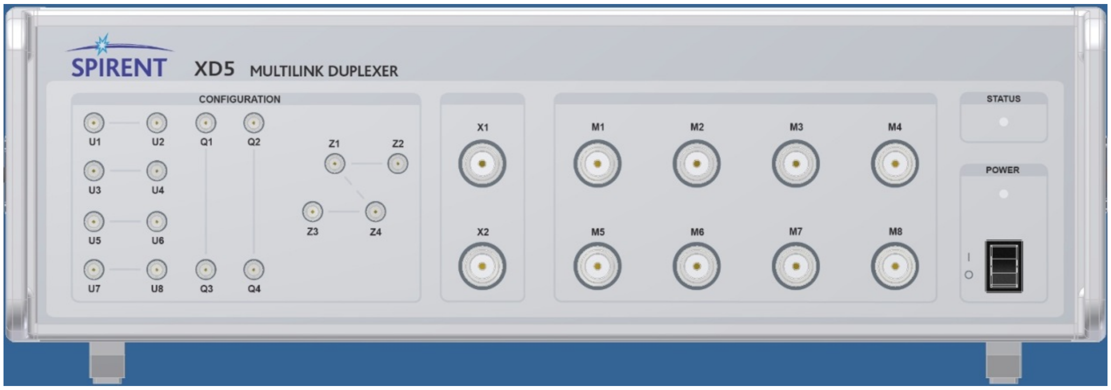


Figure 214. Front panel of the XD5.

5.3. Rear-Panel View

Figure 215. Rear panel of the XD5.

5.4. Connector Descriptions

The XD5 front-panel connectors are:

- M1 – Connects to eNodeB antenna terminal
- M2 – Connects to eNodeB antenna terminal
- M3 – Connects to eNodeB antenna terminal
- M4 – Connects to eNodeB antenna terminal
- M5 – Connects to eNodeB antenna terminal
- M6 – Connects to eNodeB antenna terminal
- M7 – Connects to eNodeB antenna terminal
- M8 – Connects to eNodeB antenna terminal
- Q1 – Uplink hybrid phase breakout
- Q2 – Uplink hybrid phase breakout
- Q3 – Uplink hybrid phase breakout
- Q4 – Uplink hybrid phase breakout
- U1 – Uplink hybrid input breakout
- U2 – Uplink hybrid input breakout
- U3 – Uplink hybrid input breakout
- U4 – Uplink hybrid input breakout
- U5 – Uplink hybrid output breakout
- U6 – Uplink hybrid output breakout
- U7 – Uplink hybrid output breakout
- U8 – Uplink hybrid output breakout
- X1 – UE Duplex Connector
- X2 – UE Duplex Connector
- Z1 – Duplex Breakout
- Z2 – Duplex Breakout
- Z3 – Duplex Breakout
- Z4 – Duplex Breakout

The XD5 rear-panel connectors are:

- RP1 - Connects to Vertex input
- RP2 – Connects to Vertex input
- RP3 – Connects to Vertex input

- RP4 – Connects to Vertex input
- RP5 – Connects to Vertex output
- RP6 – Connects to Vertex output
- RP7 – Connects to Vertex output
- RP8 – Connects to Vertex output
- RP9 – Connects to Vertex input
- RP10 – Connects to Vertex input
- RP11 – Connects to Vertex input
- RP12 – Connects to Vertex input
- RP13 – Connects to Vertex output
- RP14 – Connects to Vertex output
- RP15 – Connects to Vertex output
- RP16 – Connects to Vertex output
- User Connect - Ethernet (RJ45)
- IEC 320-C14 - AC Mains receptacle

5.5. Configuration Panel

The XD5 configuration panel allows for the proper routing and termination of RF paths under different technology configurations using the XD5 with the Vertex unit. Refer to the following diagrams for the correct connections and terminations for the desired antenna configurations. For more detailed configuration instructions, refer to the XD5 Setup Guide for Vertex.

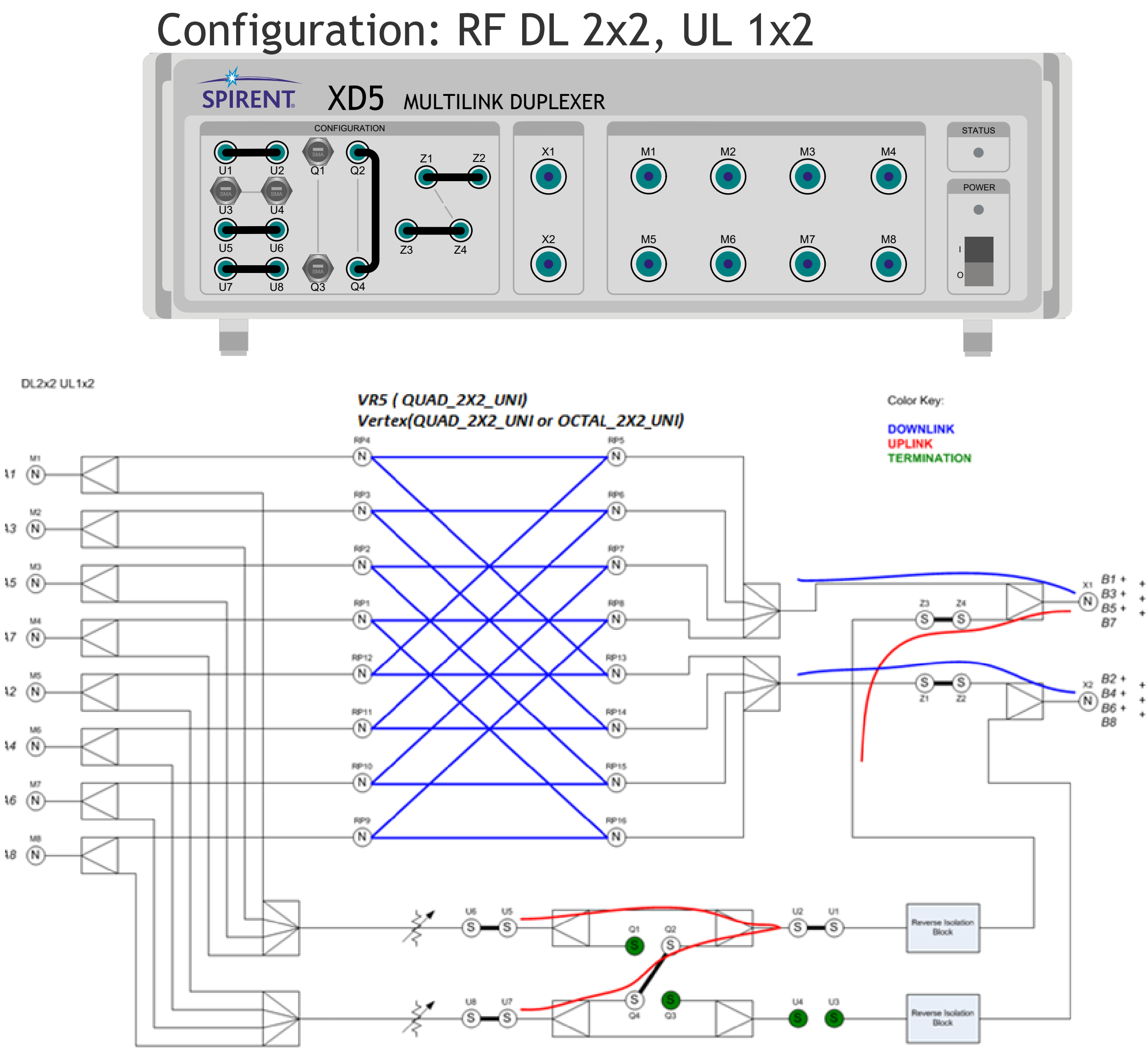


Figure 216. Functional diagram for XD5 – VR5 or Vertex system in DL 2x2, UL 1x2.

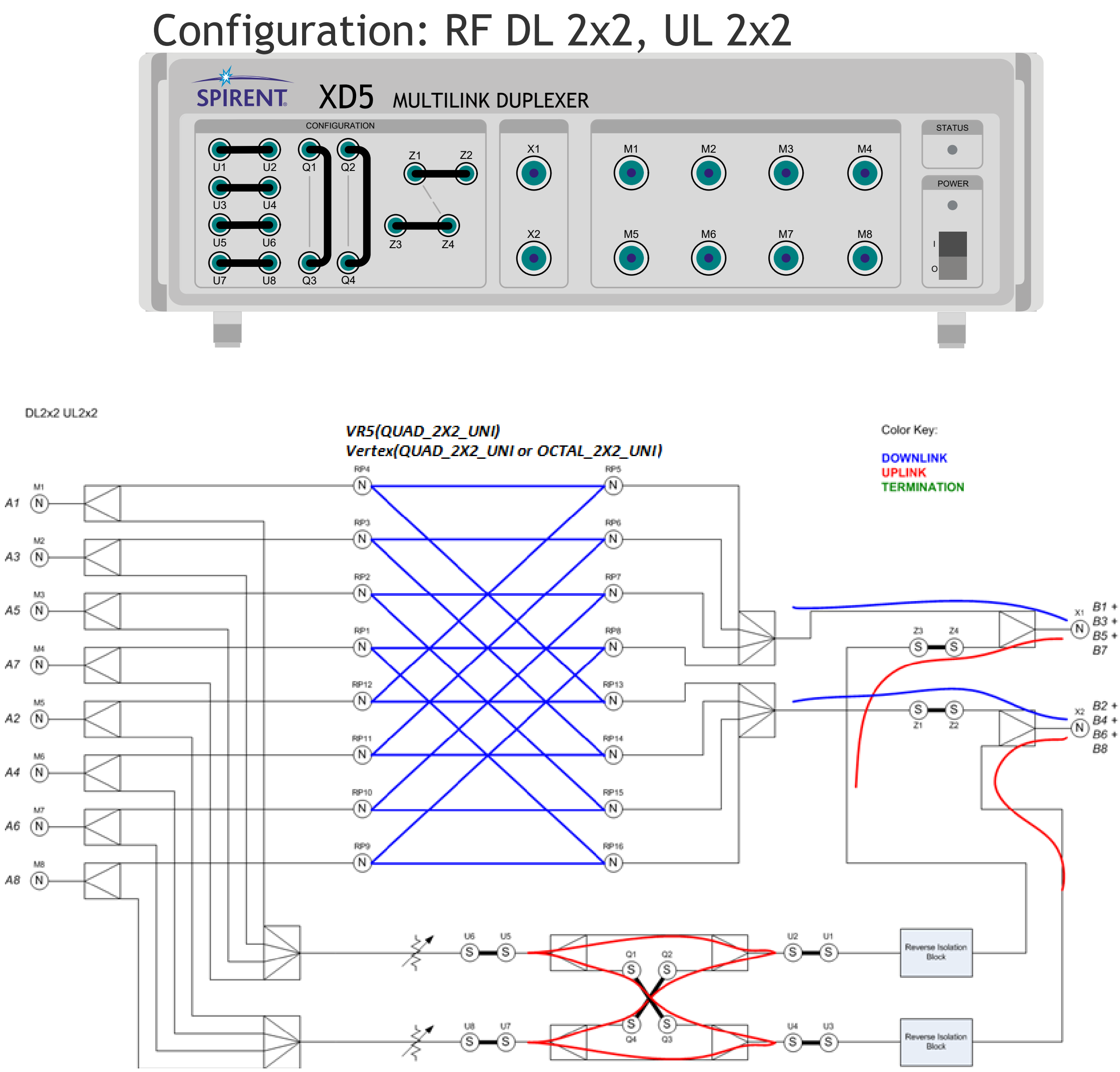


Figure 217. Functional diagram for XD5 – VR5 or Vertex system in DL 2x2, UL 2x2.

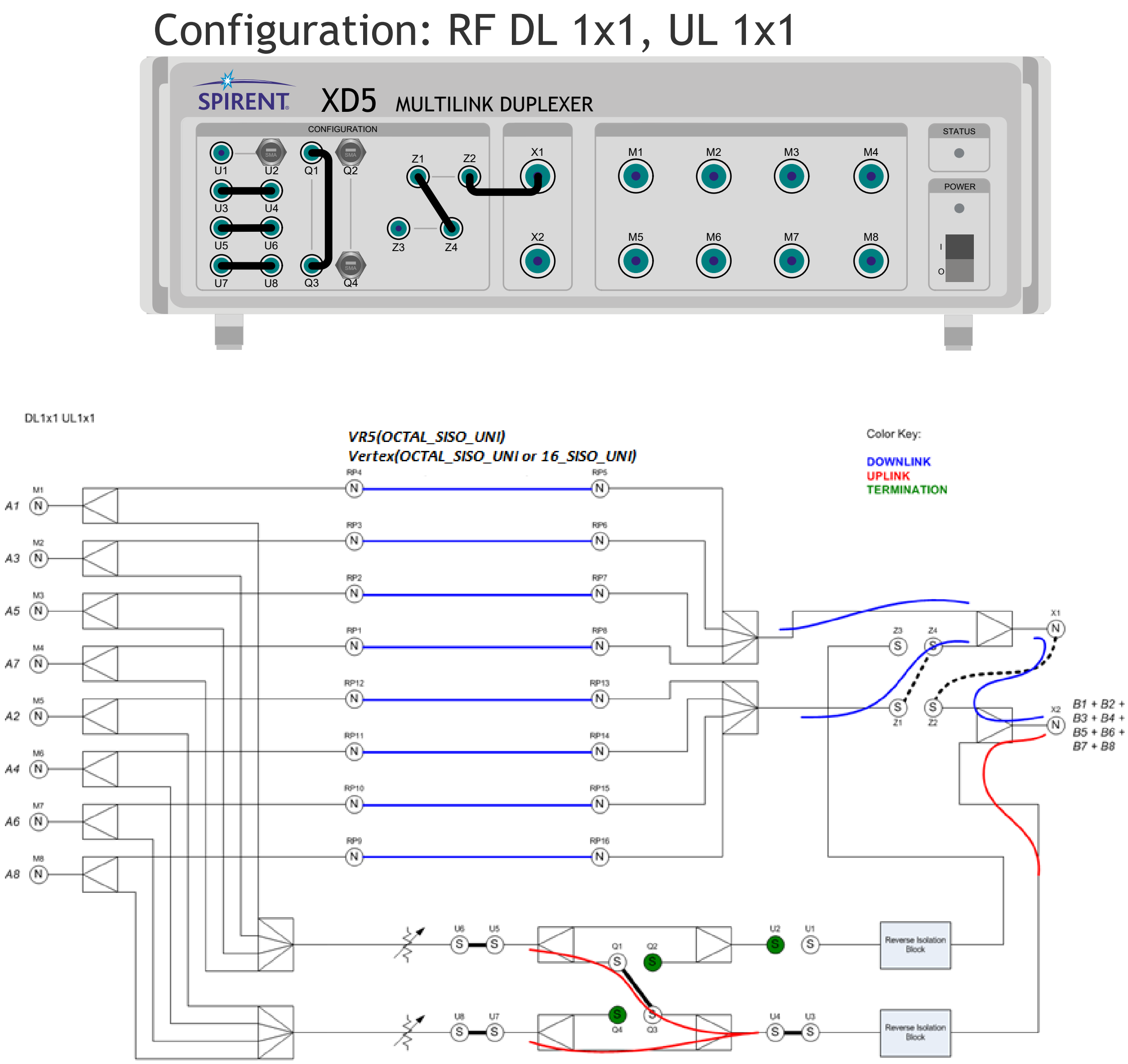


Figure 218. Functional diagram for XD5 – VR5 or Vertex system in DL 1x1, UL 1x1.

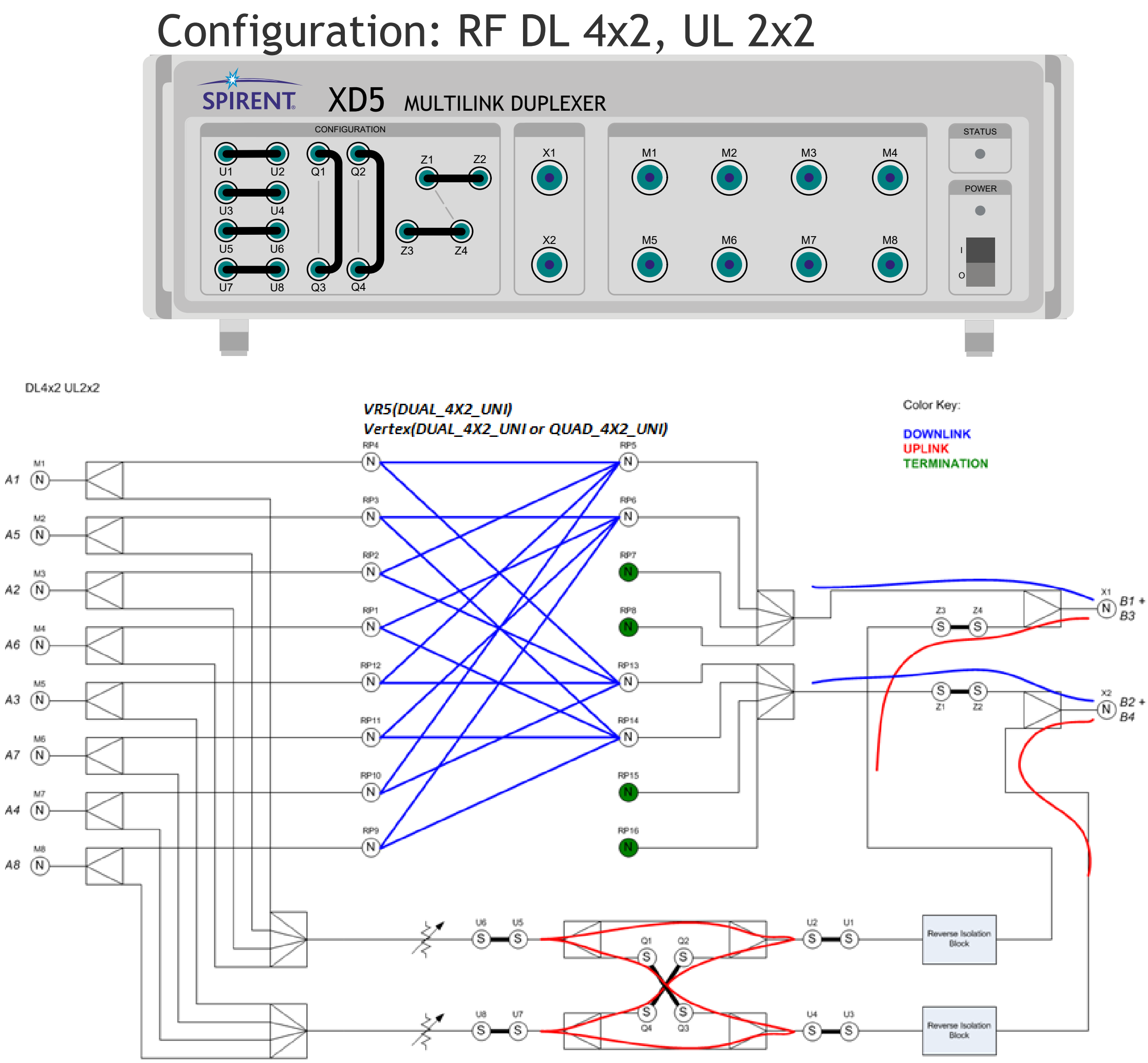


Figure 219. Functional diagram for XD5 – VR5 or Vertex system in DL 4x2, UL 2x2.

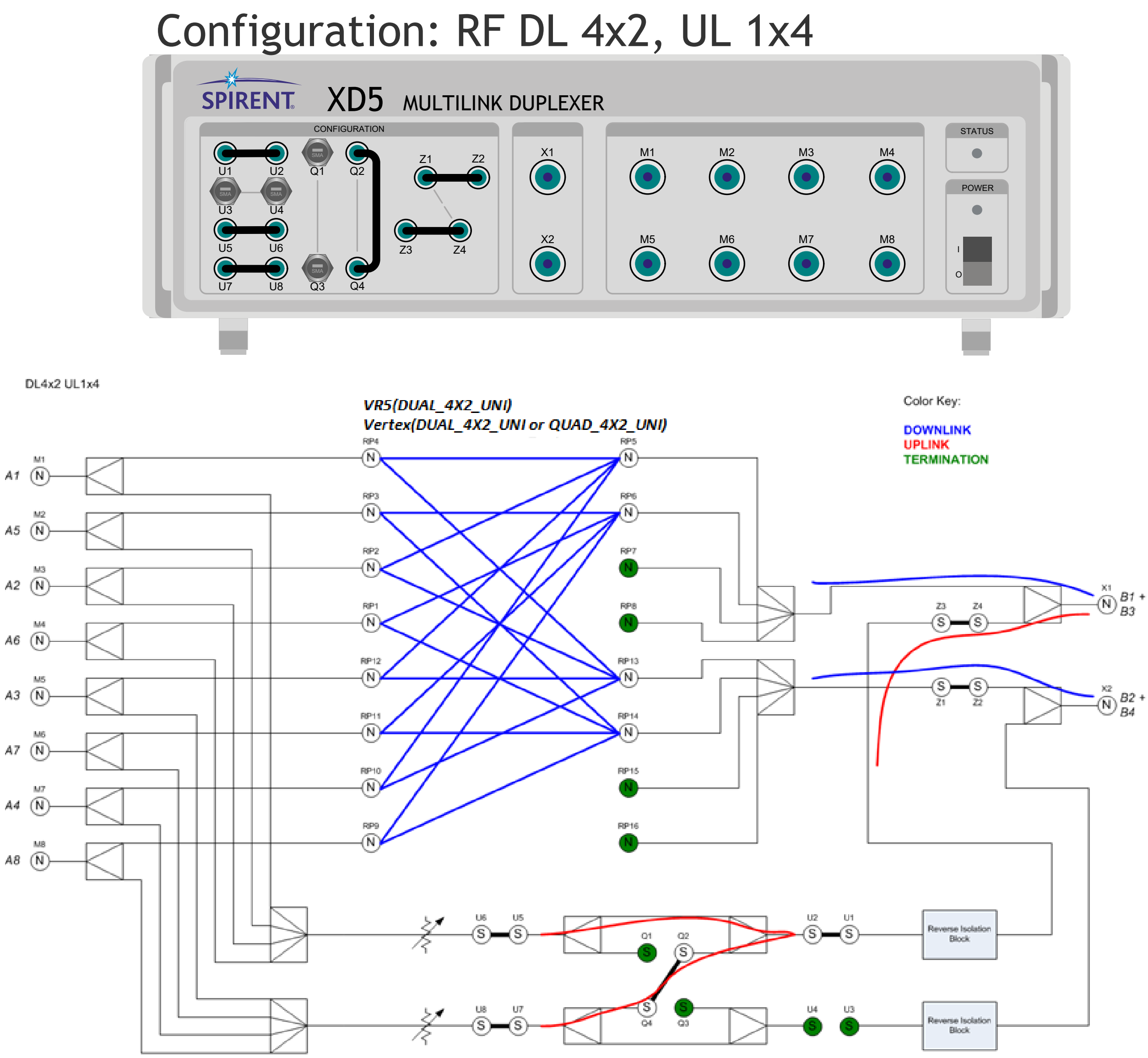
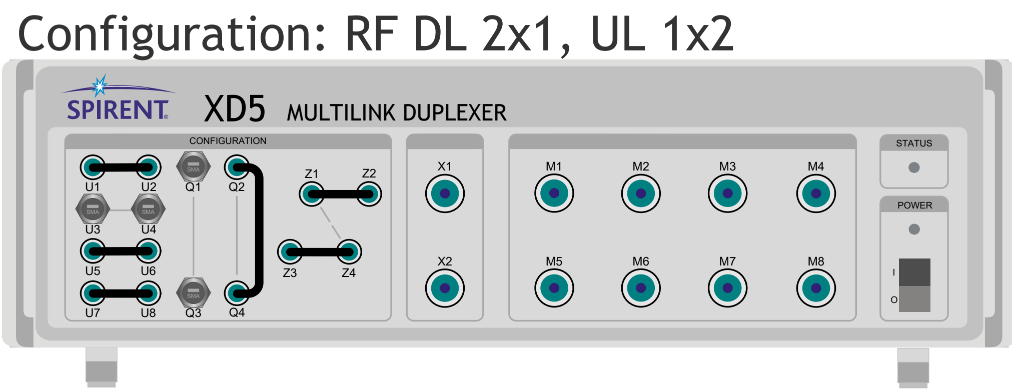


Figure 220. Functional diagram for XD5 – VR5 or Vertex system in DL 4x2, UL 1x4.



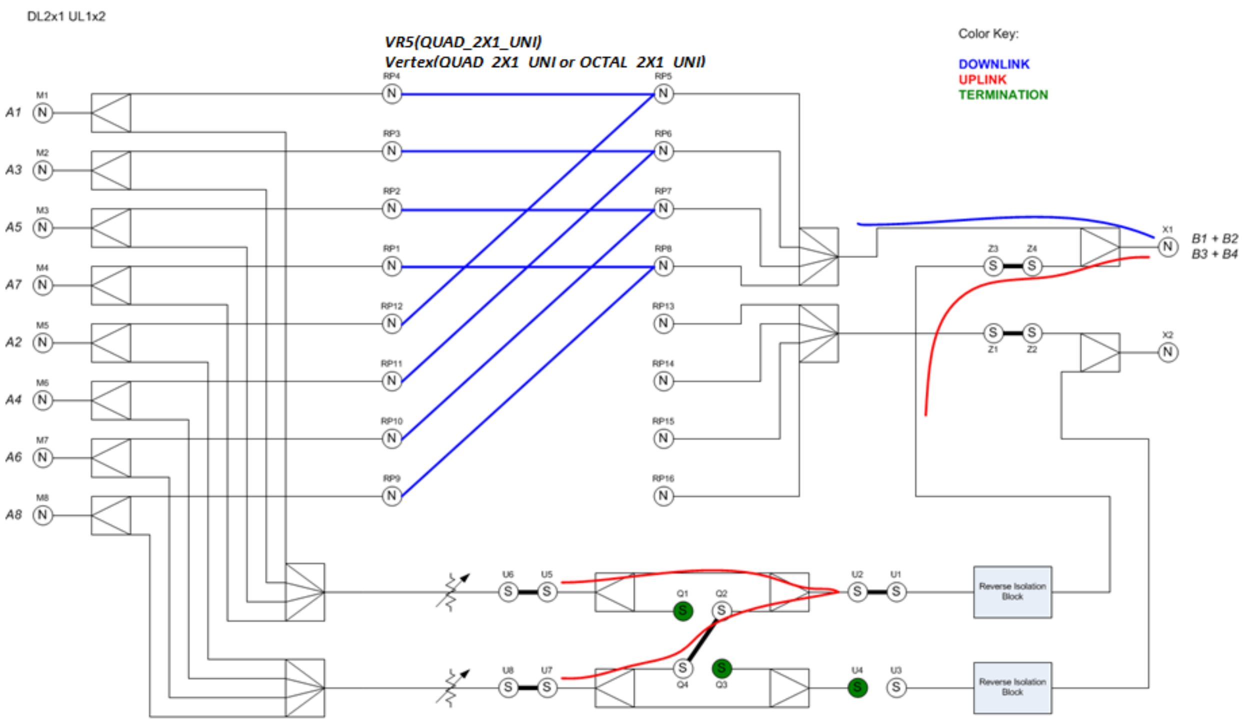


Figure 221. Functional diagram for XD5 – VR5 or Vertex system in DL 2x1, UL 1x2.

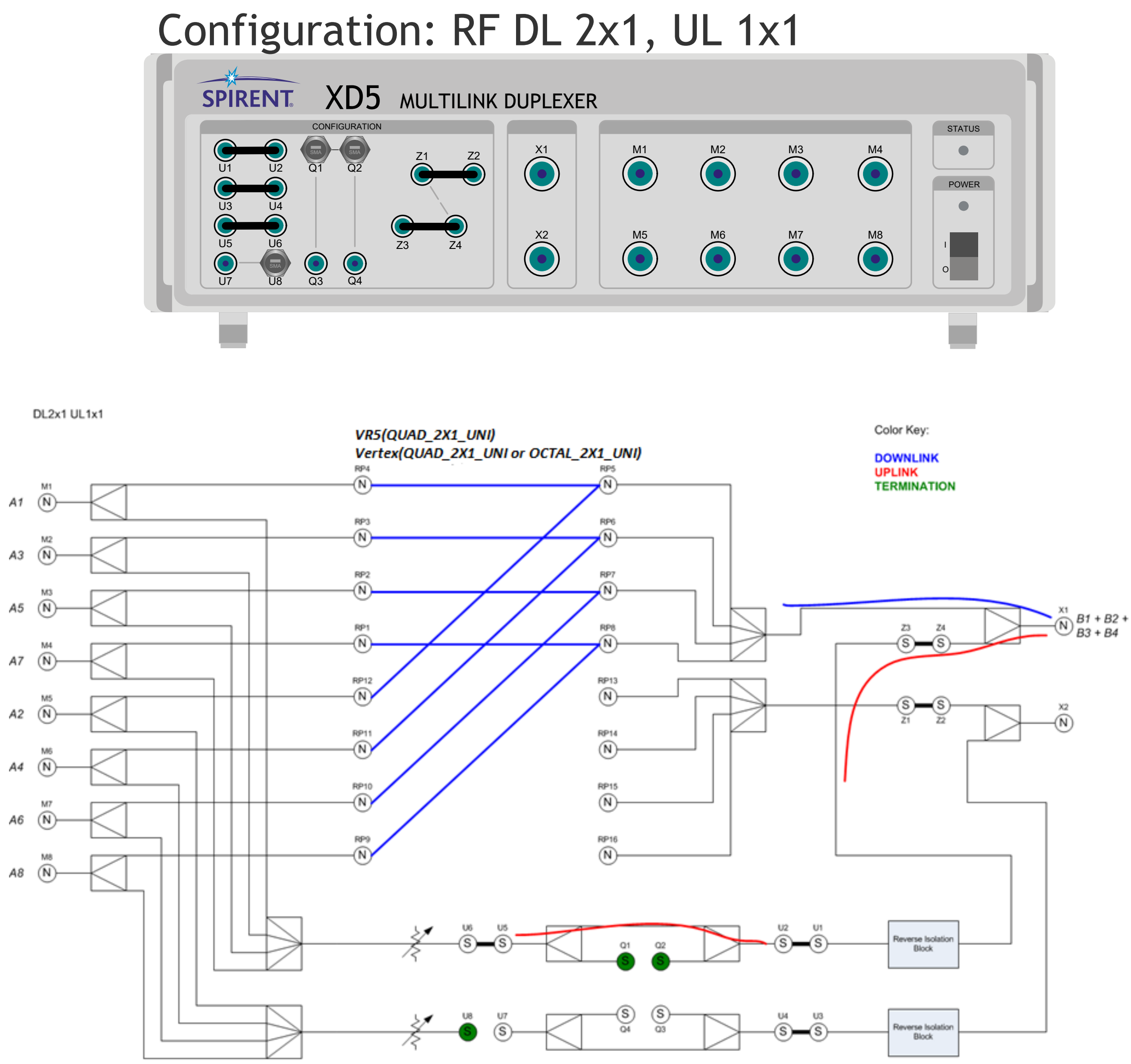
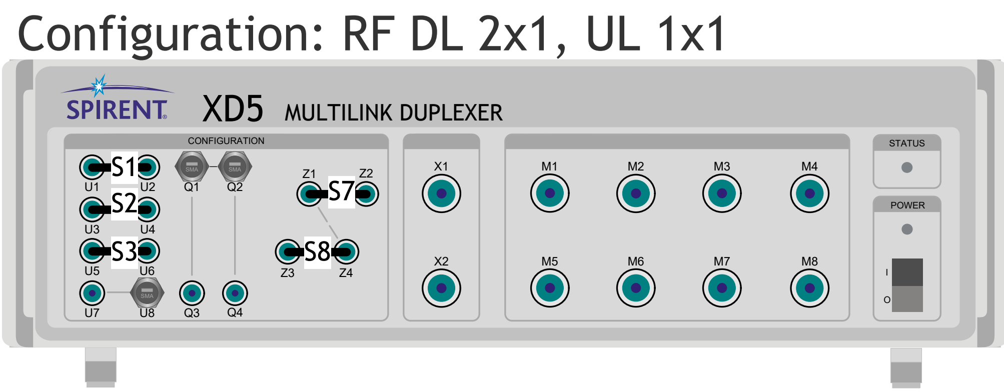


Figure 222. Functional diagram for XD5 – VR5 or Vertex system in DL 2x1, UL 1x1.



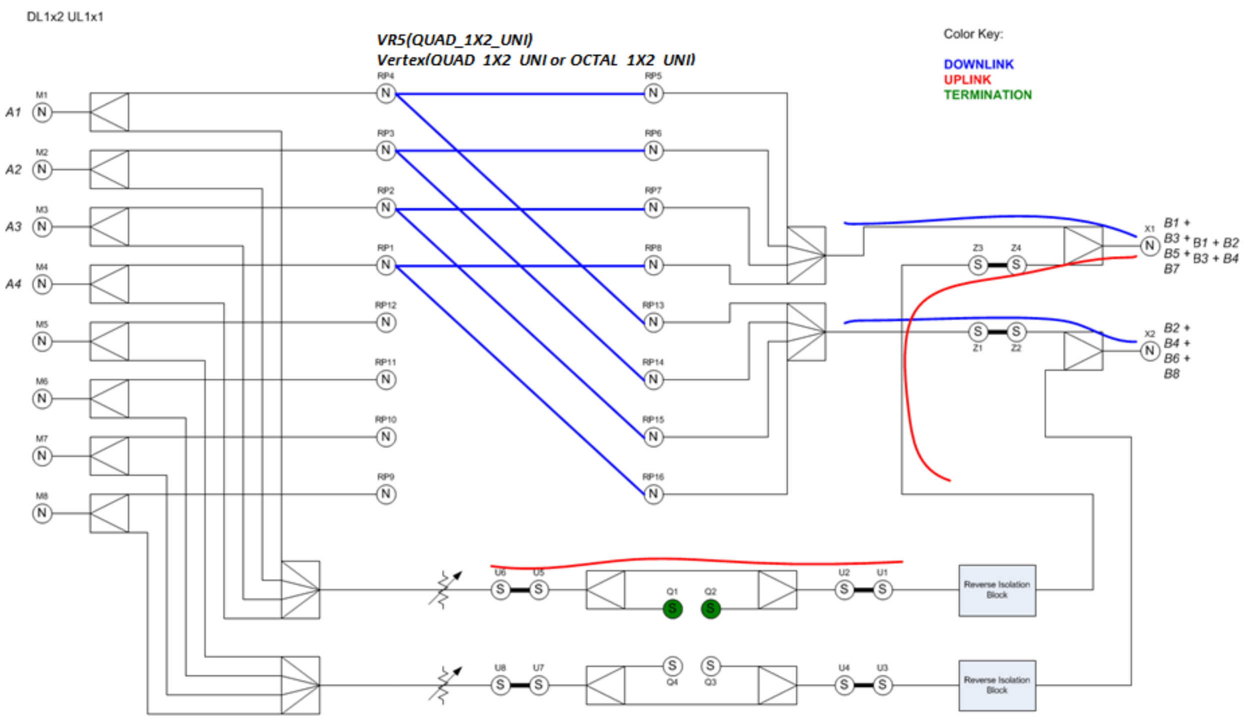


Figure 223. Functional diagram for XD5 – VR5 or Vertex system in DL 1x2, UL 1x1.

5.6. RF Performance Specifications

Frequency Range: 400 MHz to 3000 MHz

Max RF Input Power:

Downlink: +30 dBm max

Uplink: +23 dBm max (damage = > +30 dBm max continuous)

Uplink Minimum RF Input Power: -35 dBm for 50 dBc SNR with 20 MHz BW

Isolation:

Forward Isolation: >125 dB

Uplink Reverse Isolation: 110 dB min, 205 dB max

VSWR 1.50:1 Max

Uplink Attenuation Range: 95.5 dB, 0.5 dB steps

Attenuator Latency: <10 ms from command packet reception

Insertion Loss:

Uplink: 8 dB min, 103 dB max

Duplex Echo Loss: 66 dB

Level Accuracy by Attenuation Range:

0 to 7.5 dB: +/- 0.5 dB

8 to 11.5 dB +/- 1.0 dB

12 to 85.0 dB +/- 1.25 dB or 4%

86 to 95.5 dB 5%

5.7. XD5 Control

5.7.1. Initial Configuration of XD5

Since two XD5s may work together with one Vertex instrument in a system, each XD5 must be configured with a dedicated ID (XD5-A or XD5-B) with the Vertex GUI. Each ID is assigned with an internal static IP address for Vertex to identify which XD5 to control.

The default ID of an XD5 is **XD5-A**. If only one XD5 is in use, it must be configured as an XD5-A unit. If two XD5s are in use by one Vertex, they must be configured as **XD5-A** and **XD5-B** respectively. Connecting two XD5s with the same ID assignment to a single Vertex unit will cause IP conflict problems. You must configure the IDs before connecting two XD5s to a Vertex unit.

NOTE:
Only the Vertex unit can set the ID of an XD5 through the GUI. The VR5 does not have this capability. Since a VR5 can only work with the XD5-A configuration, if you want to use an XD5 with VR5, make sure to configure the XD5 to XD5-A with Vertex.

To set an XD5 ID, perform the following steps:

1. Connect the User Connect port of an XD5 to the GbE1 port on DSP1 of Vertex with a LAN cable as shown in the following figure.

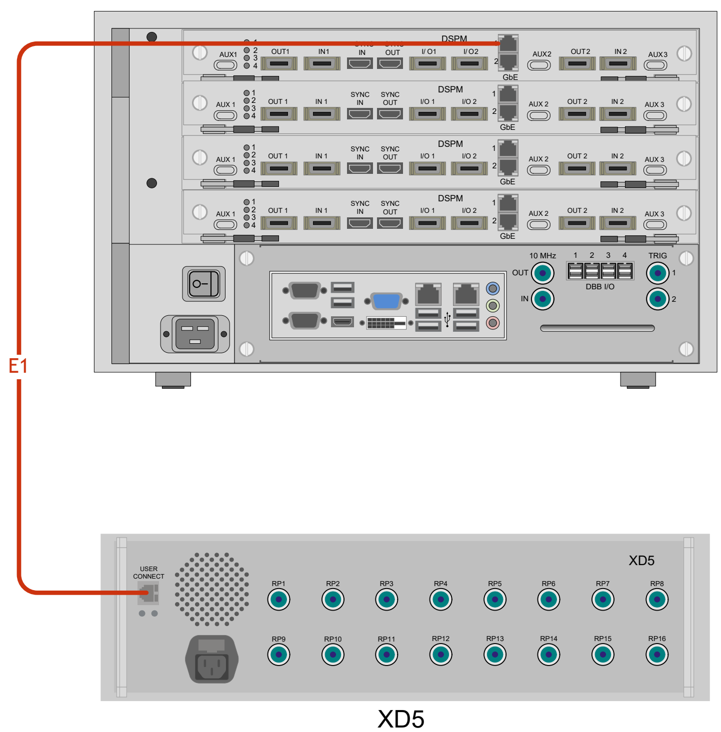


Figure 224. Connect XD5 to Vertex.

2. Power up Vertex.
3. Power up the XD5.
4. From the main menu of the Vertex GUI, select **Configure->XD5 Settings**.

   The XD5 Configuration window appears.
5. Click Settings.

   The Settings page appears, as shown in the following figure.

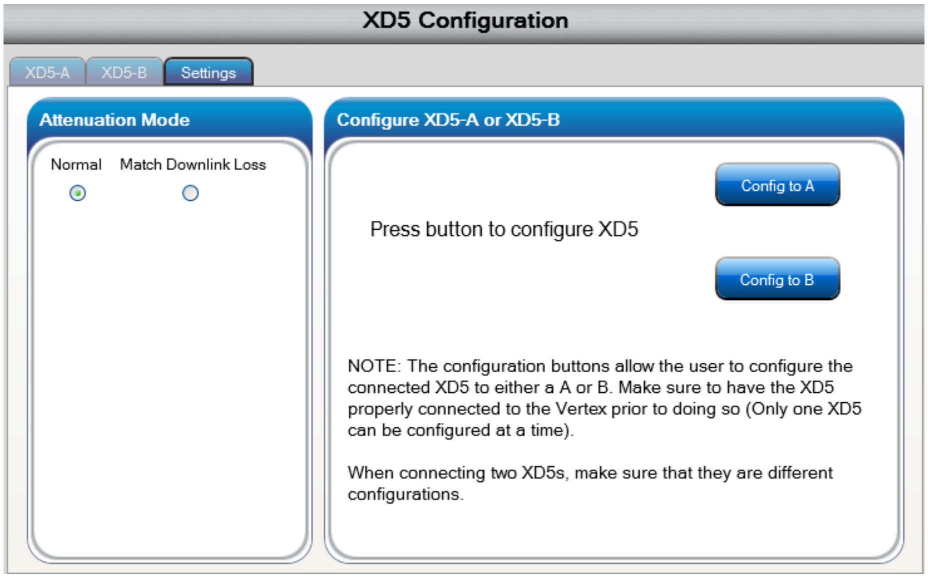


Figure 225. XD5 settings.

6. Click the **Config to A** button.

   This XD5 is assigned with ID XD5-A.
7. Make a tag for this XD5, and mark it as **XD5-A**.
8. Disconnect the XD5-A from Vertex.
9. Connect the other XD5 to the GbE1 port of DSP1 on Vertex.
10. Repeat Steps 4 and 5.
11. Click the **Config to B** button.

    This XD5 is assigned with ID XD5-B.

12. Make a tag for this XD5, and mark it as **XD5-B**.
13. Connect both XD5-A and XD5-B to Vertex as shown in the following figure. XD5-A and XD5-B are connected to GbE port 1 of DSP1 and DSP2.

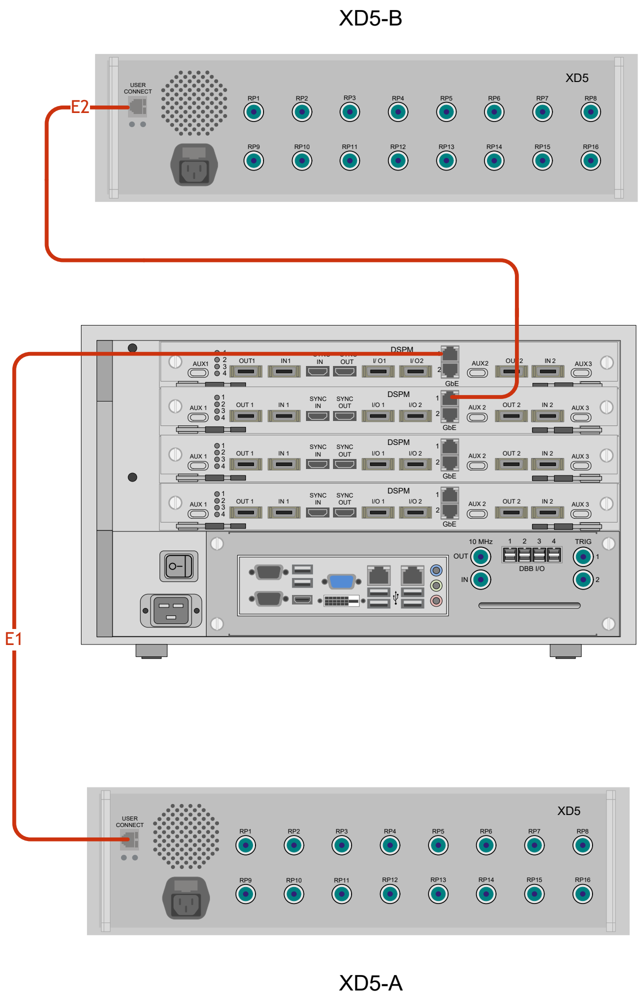


Figure 226. System connection with one Vertex and two XD5s.

5.7.2. Connection Setup

For XD5 system configurations with Vertex, in terms of RF cabling, the user plane becomes the ports of the XD5. (Details are provided in the XD5 setup guide for Vertex). This means that logical ports, such as A1, now map to the physical ports of the XD5 instead of the physical ports of the Vertex.

For example, when configuring a Quad 2x2 UniDirectional connection setup on the Vertex, settings and measurements for Port A1 no longer correspond to Port A1 on the Vertex. Instead, settings and measurements for Port A1 correspond to Port M1 on the XD5, as the losses through the XD5 and accompanying cables are internally compensated.

If two XD5s work together with one Vertex, the logical ports of Vertex may map to the physical ports of different XD5s. XD5-A works with logical ports A1/B1 to A8/B8, and XD5-B works with A9/B9 to A16/B16.

5.7.3. Attenuation Control

Because the XD5 is connected directly to the Vertex over Ethernet, the XD5 attenuators can be controlled from the Vertex GUI through the XD5 Configuration window. You can access the XD5 Configuration window from the main menu on the Vertex GUI by selecting **Configure>XD5 Settings**.

You can set the attenuations of XD5-A or XD5-B by clicking the button with the respective name.

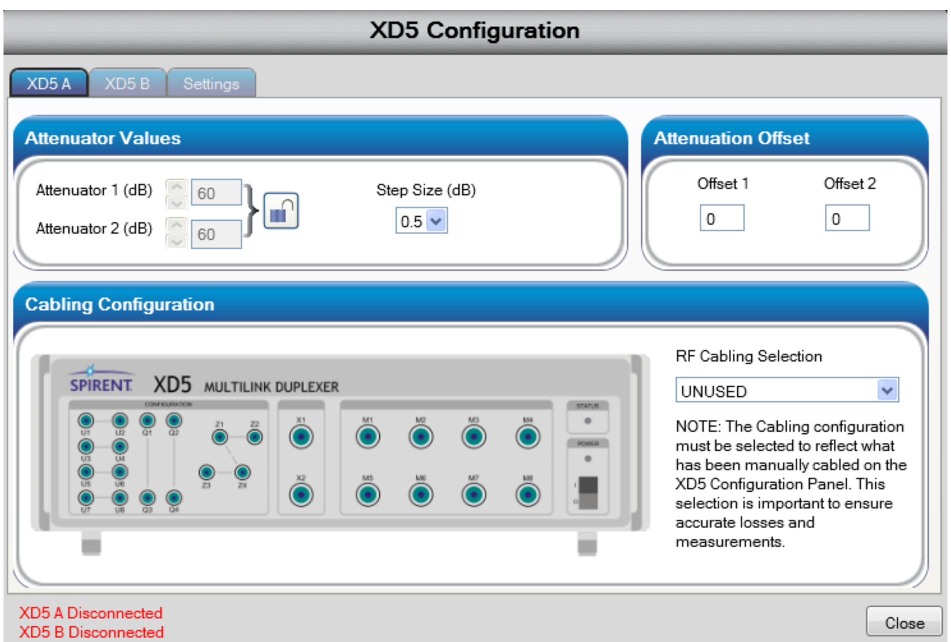


Figure 227. XD5 Configuration window.

5.7.3.1 Attenuator Values

You can change the attenuator values by entering a new value (in 0.5 dB steps) or by using the up/down arrows. By pressing the **Lock** icon, you can move the values of both attenuators up and down together using the up/down arrows of Attenuator 1.

The Attenuator value specifies the attenuation from port X# to port M# in the given configuration. In this regard, it is important to accurately select the Cabling Configuration, as the total loss is internally compensated to achieve a set attenuation between these ports.

5.7.3.2 Attenuation Offset

Attenuation Offset in the XD5 is similar to cable loss on the Vertex. It is used to ensure the losses can be matched between uplink and downlink when cable losses are used on the Vertex. Because there is not a 1:1 mapping between ports on the XD5 and ports on the Vertex due to loss summation, you must take care to calculate the attenuation offset makes sense in accordance with the set cable losses.

For example, if attenuator 1 on the XD5 uplink is set to 50 dB, an attenuation offset of 5 dB will imply that there is an additional 5 dB of loss outside the XD5. In this case, the attenuation through the XD5 will only be set to 45 to account for the additional 5 dB loss.

Offset 1 applies to Attenuator 1, and Offset 2 applies to Attenuator 2.

5.7.3.3 Attenuation Mode

Attenuation Mode provides the following settings:

- **Normal** - The XD5 attenuators are configured independently from the Vertex settings.
- **Match Downlink Loss -** The XD5 attenuators are selected automatically to match the lowest loss from the downlink channels configured on the Vertex. This is useful for DEE scenarios so at compile time the XD5 can be automatically configured to correspond to the loss changes in the Vertex.

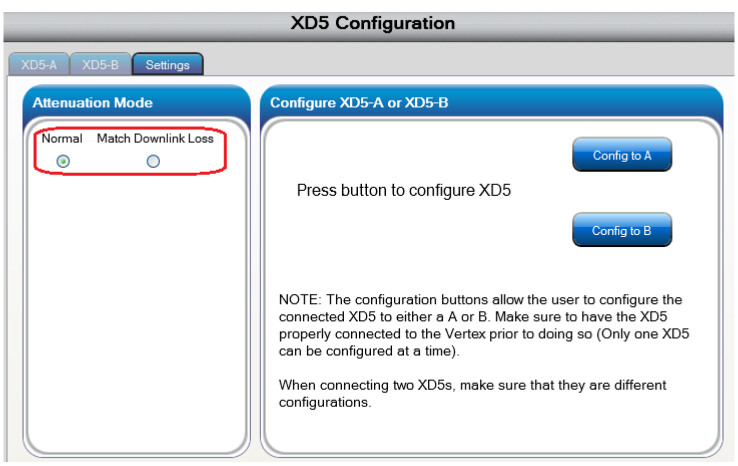


Figure 228. Attenuation Mode settings.

5.7.3.4 Cabling Configuration

For attenuation through the XD5 to be accurate, you must manually select the configuration that represents how the XD5 is cabled. For your convenience, an image of the appropriately cabled XD5 is displayed when you select the RF Cabling Selection on the XD5 Configuration page. You can access the XD5 Configuration page from the main menu on the Vertex GUI by selecting **Configure>XD5 Settings**.

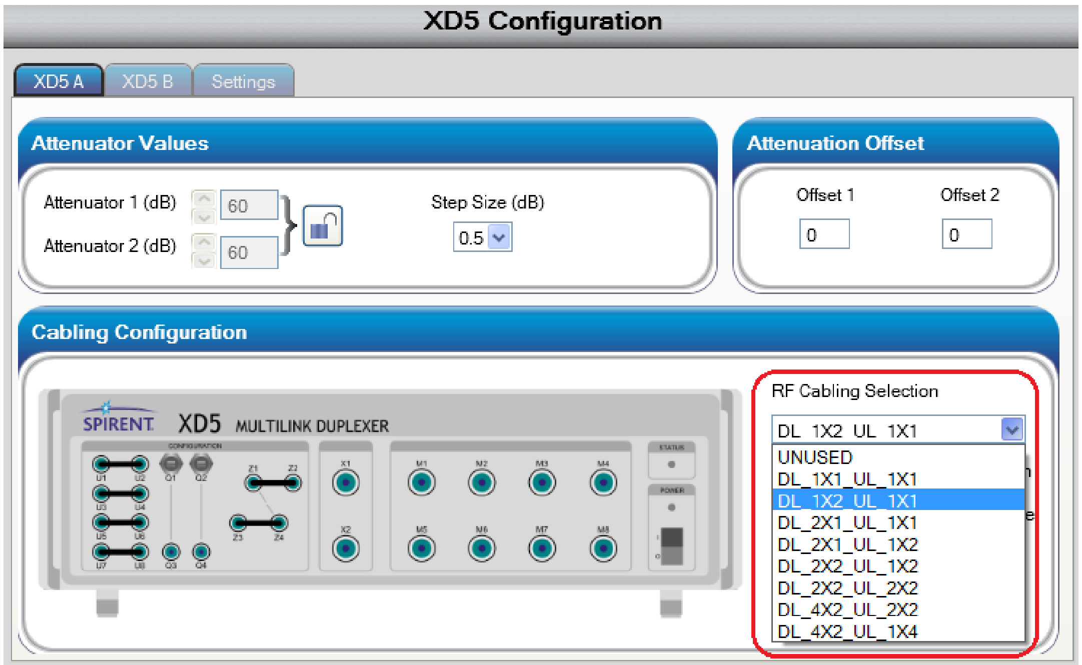


Figure 229. RF Cabling Selection.

5.7.4. RPI Control

You can also control the XD5 remotely using RPI commands. These commands are described in detail in this user manual and the RPI Command Reference Manual.

5.8. Interface and Environmental Characteristics

Front Panel Indicators	
Power	1 Green LED
Status	1 Green LED
Power Requirements	
Voltage	100-240 VAC (auto sensing)
Frequency	50-60 Hz
Power	
110 VAC	24 Watts (Max)
220 VAC	24 Watts (Max)
Fuse Type	2A, 250 VAC, 5x20mm, Time Lag (Slo-Blo)
Number of Fuses	2
Operating Environment	
Temperature	0 to 40 degrees C
Humidity	10% to 90%, noncondensing
Dimensions and Weight	
Height (including feet)	15.24 cm
Width	43.61 cm
Depth (including handles)	40.03 cm
Weight	10 kg
Control Interfaces	
Interfaces Provided	RJ-45 Jack on rear panel for Ethernet

5.9. Installation

5.9.1. Rack Installation

The Spirent XD5 is typically used in a bench-top configuration along with a Vertex and other equipment. However, it can be installed in a rack with the same equipment. For more information about preparing the XD5 for rack mounting, refer to the appropriate Vertex Setup Guide that applies to your specific system and instruction sheet MI5574 supplied with the XD5 rack mount kit. The instructions for cabling and connecting the XD5 are provided in the setup guide.

5.9.2. Bench-Top Installation

The Spirent XD5 is typically in a bench-top configuration. For more information about preparing the XD5 for installation, refer to the appropriate XD5 Setup Guide for Vertex that applies to your specific system and instruction sheet MI5574 supplied with the XD5 rack mount kit. The instructions for cabling and connecting the XD5 are provided in the setup guide. When the system is used this way, be sure that the power-cord connection is accessible at all times in case an emergency power disconnection is required. The cabling and connections are the same as for a rack-mounted installation.

5.10. Care and Maintenance

Cleaning the Spirent XD5 is typically not required. However, for good RF performance, it is important to keep the RF connectors on the front and rear panels free of dust and other debris. Also, for proper internal airflow, ensure the fan vent on the rear panel is clear of debris and not blocked by an object.

6. Technical Specifications

> **NOTE**:
> All technical specifications are typical and subject to change without notice. Unless otherwise indicated, the specifications are measured at room temperature, Vertex has been booted up for more than 30 minutes, the input power level is -10 dBm, and the output power level is the maximum supported in the frequency band.

6.1. RF Channel Specifications

Frequency Ranges	30 MHz to 5925 MHz

6.1.1. Input Signal Level

Input Signal level	-50 to +15 dBm
Max RF Input Power	+15 dBm
Level Resolution	0.1 dB
Damage Level	+33 dBm (peak)

6.1.2. Output Signal Level

Range	-110 to -10 dBm (RMS)
Resolution	0.1 dB
Typical Accuracy	+/-1 dB (Unidirectional connection setup)
	+/-1.5 dB (Bidirectional connection setup)

6.1.3. AWGN Performance

C/N Range	-40 to +40 dB
Resolution	0.01 dB
Typical Accuracy	+/-0.1 dB (|C/N| <= 30 dB)
	+/-0.2 dB (|C/N| > 30 dB)

6.1.4. Spurious Emission Levels

Within Channel	-40 dBc
Outside Channel within 30 to 5925 MHz	-15 dBc

6.1.5. Residual EVM

-40 dB typical per sub-carrier

6.1.6. Noise Floor

Better than -165 dBm/Hz (typical) at a set output level of -45 dBm and set input level of -10 dBm

6.1.7. RF Physical Interface Characteristics

Impedance	50 ohms
RF Port VSWR	< 1.5
Connector Type	N Female (Front Panel, 4GHz RFM)
	SMA Female (Front Panel, 6GHz RFM)

6.2. Interface and Environmental Characteristics

6.2.1. Vertex Channel Emulator Front Panel Indicators

Status	1 LED indicator
RF Ports (Active RF power) (Future feature)	1 LED indicator for each port

6.2.2. Vertex Instrument Power Requirements

WARNING!
Ensure that the Vertex instrument is connected to a grounded receptacle.

Voltage	100-240 VAC (auto sensing)
Frequency	50-60 Hz
Current and Power	
100~125 VAC, 50~60 Hz	20 A (Max), 2000 W (Max)
220~240 VAC, 50~60 Hz	9A (Max), 1950 W (Max)
Overload Protection	1 thermal circuit breaker
	20 Amps, 240 VAC, Thermal, 60C
Circuits Protected	Hot & Neutral conductors

Power Supply Cords

When operating the Vertex instrument between 100 VAC and 125 VAC, use a detachable Mains supply cord rated for 20A. The supply connector on the cord depends on the country where it is being used.

WARNING!
Do not use a 15A power cord with a NEMA 5-15P connector on the instrument configuration with 4 DSPM when operating at voltages below 200 VAC. The power cord may overheat and become a potential fire hazard. It is not recommended to use Vertex with configuration of 9 RFMs and 4 DSPM2s (VCE6-9B6-4D2) with a 100VAC power supply.

For all Mains supply cords, the end that connects to the Vertex instrument must have an IEC 60320 C19 connector.

CAUTION!
The Mains supply cable/connector is the Mains Disconnecting Device. It is important to allow for easy access to the Mains supply cord on the rear of the Vertex instrument for emergency disconnection.

6.2.3. Operating Environment

Temperature	0 to 40 degrees C
Humidity	10% to 90%, noncondensing

6.2.4. Vertex Dimensions and Weight

Height	28 cm, (10.5 inches) (including feet)
Width	43.6 cm, (17.2 inches)
Depth	71.8 cm, (28.3 inches) (including handles)
Weight	107 pounds (maximum configuration with DSPM)
	115 pounds (maximum configuration with SDE-DSPM2)

6.2.5. Vertex Channel Emulator Control Interfaces

Interfaces Provided	Ethernet

6.2.6. 10 MHz Reference Requirements

External 10 MHz Reference Input	
Connector Type	50Ω BNC connector (rear)
Input Level Range	0 dBm +/- 2 dB
Input Level PPM from	10 MHz < 2 PPM
Internal 10 MHz Reference	
Connector Type	50Ω BNC connector (rear)
Output Level Range	0 dBm
Type	OCXO
Accuracy	< 1 PPM

7. Care and Maintenance

7.1. Cleaning

Cleaning the Spirent Vertex channel emulator is typically not required. However, for good performance, it is important to keep the air ventilation holes on the sides clean and clear of anything that could restrict airflow. Also, keep the connectors on the front and rear panels free of dust and debris.

Vertex Circuit Breaker Operation

The circuit breaker on the rear panel is for the safe disconnection of power to the Vertex in the event of an overload condition.

In the event of a circuit breaker trip, perform the following steps:

CAUTION:
Disconnect the power cord before resetting circuit breaker.

1. Unplug the power cord from the wall outlet, or unplug the power cord from the receptacle on the rear panel of the Vertex instrument.
2. Wait 2 minutes.
3. Reset the circuit breaker.
4. Reconnect the power cord.
5. If the circuit breaker stays on, proceed with normal power up and operation.
6. If the circuit breaker trips again, disconnect power cord and do not use the Vertex instrument.
7. Contact Spirent Global Services:

CUSTOMER SERVICE CENTER (CSC)
Online Support: http://support.spirent.com

TELEPHONE SUPPORT
Follow the interactive voice menu to reach support for the proper product line. You may also select administrative assistance or technical support.

North America: 1-800-SPIRENT (1-800-774-7368)

Outside North America: 1-818-676-2616

China (mainland only): + 86 800 810 9529

Asia Pacific: + 86 (10) 8233 0033

Europe, Middle East and Africa: +33 1 6137 2270

E-MAIL SUPPORT
To open a new service request for technical or administrative issues via e-mail, contact: support@spirent.com

CAUTION:
Do not open the Vertex instrument. It contains no internal user serviceable parts.

Vertex Utilisation du Disjoncteur

Le disjoncteur sur le panneau arrière est là pour déconnecter le Vertex en toute sécurité en cas de surcharge. Dans le cas d'un déclenchement du disjoncteur, exécutez les étapes suivantes:

ATTENTION:
Débranchez le cordon d'alimentation avant de réinitialiser le disjoncteur.

1. Débranchez le cordon d'alimentation du Vertex de la prise murale, ou débranchez le cordon d'alimentation de la prise du panneau arrière du Vertex
2. Attendez 2 minutes.
3. Réinitialisez le disjoncteur.
4. Rebranchez le cordon d'alimentation.
5. Si le disjoncteur reste allumé, procédez à la mise sous tension normale de fonctionnement.
6. Si le disjoncteur se déclenche à nouveau, débranchez le cordon d'alimentation et ne plus utiliser le Vertex.
7. Contactez alors Spirent Global Services:

CUSTOMER SERVICE CENTER (CSC)
support en ligne: http://support.spirent.com

SUPPORT TELEPHONIQUE
Suivez le menu vocal interactif afin d'atteindre le support du produit recherché. Vous pouvez également sélectionner une assistance administrative ou un support technique aux numéros de téléphone suivant:

Amérique du Nord: 1-800-SPIRENT (1-800-774-7368)

Hors Amérique du Nord: 1-818-676-2616

Chine (continentale seulement): + 86 800 810 9529

Asie-Pacifique: + 86 (10) 8233 0033

Europe, Moyen-Orient et Afrique: +33 1 6137 2270

SUPPORT PAR EMAIL
Pour ouvrir une nouvelle demande de service par e-mail pour des questions techniques ou administratives contactez: support@spirent.com.

ATTENTION:
Ne pas ouvrir le Vertex. Il ne contient aucune pièce réparable par les utilisateurs.