VN Series PN Series

VECTOR NETWORK ANALYZER
USER GUIDE




# Vector Network Analyzer User Guide

V1.0.6
06/11/2026

KAROGIC

# Contents


1. Quick Start ..... 1

1.1 Operating Environment and Interfaces ..... 1
1.1.1 Operating Environment ..... 1
1.1.2 VNB Series Interfaces ..... 2
1.1.3 PNB Series Interfaces ..... 3
1.2 Connecting the Instrument ..... 5
1.2.1 VNB Series ..... 5
1.2.2 PNB Series ..... 5
1.3 Setting the Measurement Conditions ..... 5
1.4 Performing Calibration ..... 6
1.5 Connecting the DUT ..... 6
1.6 Starting the Measurement and Reading the Results ..... 6

2. Interface Overview ..... 8

2.1 Interface Layout ..... 8
2.2 Instrument Status Area ..... 8

3. Measurement Configuration ..... 9

3.1 Graph and Trace Configuration Logic ..... 9
3.2 Frequency Setting ..... 10
3.3 Power Settings ..... 11
3.4 Sweep Settings ..... 11
3.4.1 Sweep Setting Parameters ..... 11
3.4.2 Conventional Sweep Modes ..... 12
3.4.3 CW Mode ..... 12
3.4.4 Segment Sweep Mode ..... 13
3.5 Calibration Parameters ..... 15
3.6 System Clock ..... 15
3.7 Other System Settings ..... 16

4. Calibration ..... 17

4.1 Overview ..... 17
4.2 Calibration Using an Electronic Calibration Module ..... 17
4.3 Calibration Using a Mechanical Calibration Kit ..... 19
4.3.1 Calibration Kit Import and Editing ..... 20

5. Measurement and Result Analysis ..... 21

5.1 Trace Settings ..... 21
5.1.1 Display Formats ..... 21
5.1.2 Trace Memory, Math, and Display ..... 22
5.2 Marker Readout ..... 23

5.3 Bandwidth Search and X dB Search ..... 24
5.3.1 Bandwidth Search ..... 24
5.3.2 X dB Search ..... 26
5.4 Graph Display Settings ..... 28
6. Data and Files ..... 29
6.1 Saving and Opening Instrument Configurations ..... 29
6.2 Exporting Measurement Data ..... 29
6.3 Saving Images ..... 29
6.3.1 Saving the Full-Screen Interface ..... 29
6.3.2 Saving an Individual Measurement Window ..... 29
7. Advanced Functions ..... 30
7.1 Using the Port Extension Function ..... 30
7.2 Using the DTF Function ..... 32
7.2.1 DTF Distance-to-Fault Example ..... 33
7.3 Port Wave Data Display ..... 34
8. Quick Operations ..... 37
8.1 Using the Quick Menu ..... 37
8.2 Using the Mouse Wheel / Two-Finger Gestures to Quickly Adjust the Plot ..... 38
8.3 Using Right-Click Quick Operations ..... 39
8.4 Using Double-Click to Quickly Create a Marker ..... 39
8.5 Maximizing a Plot ..... 40
8.6 Using the Magnifier for Local Zoom ..... 41
8.7 Maximizing the Plot Area by Hiding the Right-Side Settings Panel ..... 41
9. System Settings and Maintenance ..... 42
9.1 Setting the Startup State ..... 42
9.2 Theme Settings ..... 42
9.3 Single and Continuous Measurements ..... 42
9.4 Preset and Full Screen ..... 43
9.5 Software and Firmware Update ..... 43
9.5.1 Parameter Description ..... 43
9.5.2 Online Download Update Example ..... 44
Appendix A VN Series AUXIO Pin Description ..... 45
Appendix B PN Series AUXIO Pin Description ..... 46
Appendix C Version History ..... 47

# 1. Quick Start

This chapter guides you through a complete measurement procedure, from powering on the instrument to obtaining measurement results, using a specific device under test (DUT) as an example. The DUT is a cavity bandpass filter with a center frequency of approximately 1.65 GHz, a −3 dB bandwidth of approximately 103 MHz, and out-of-band rejection ≥ 95 dB.

The entire procedure consists of six steps. Each step provides the specific settings and acceptance criteria used in this example. For more detailed information, please refer to the corresponding sections.

## 1.1 Operating Environment and Interfaces

### 1.1.1 Operating Environment

VNStudio software is used to control VN and PN Series vector network analyzers, providing measurement configuration, data display, and result analysis. For the VN Series, VNStudio must be installed and run on a host computer.

The recommended host computer environment is listed below. The table provides only the basic recommended configuration. For systems below the recommended configuration, actual performance should be determined through testing.

**Table 1 Operating Environment Requirements**

| Item | Requirement |
|---|---|
| **Operating System** | Windows 11/10/8/7 (requires VS2019 C++ redistributable) |
| **Architecture** | Windows: x64 |
| **Processor** | Intel i3 or above |
| **Memory** | 8 GB or higher |
| **Data Port** | USB 2.0, USB 3.0 |
| **Display Resolution** | At least 1280 × 800 pixels |
| **Others** | Some antivirus software may affect the normal operation of VNStudio. |

### 1.1.2 VNB Series Interfaces

The interfaces of the VNB Series are shown below. The two RF ports are used to connect DUT. The USB data port provides both power and data transfer through a single cable. For the pin definitions of the multifunction AUXIO port, please refer to Appendix A.

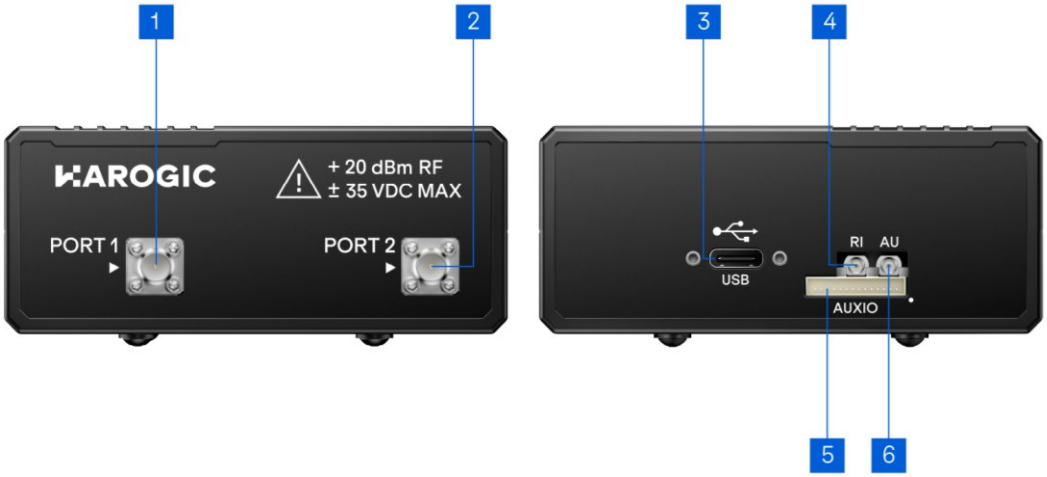



Figure 1 VNB Series Interfaces

**Table 2 VNB-60 Interface Description**

| No. | Interface | Description |
|---|---|---|
| 1 | **Port1** | SMA (F) |
| 2 | **Port2** | SMA (F) |
| 3 | **Data Port** | Type-C, USB2.0 or USB3.0 |
| 4 | **Reference Clock Input** | MMCX (F), supports 10 MHz reference clock |
| 5 | **Multifunction AUXIO** | Refer to Appendix A for details |
| 6 | **Reserved Port** | / |

### 1.1.3 PNB Series Interfaces

The interfaces of the PNB Series are located on the top panel. The interface definitions are shown below:

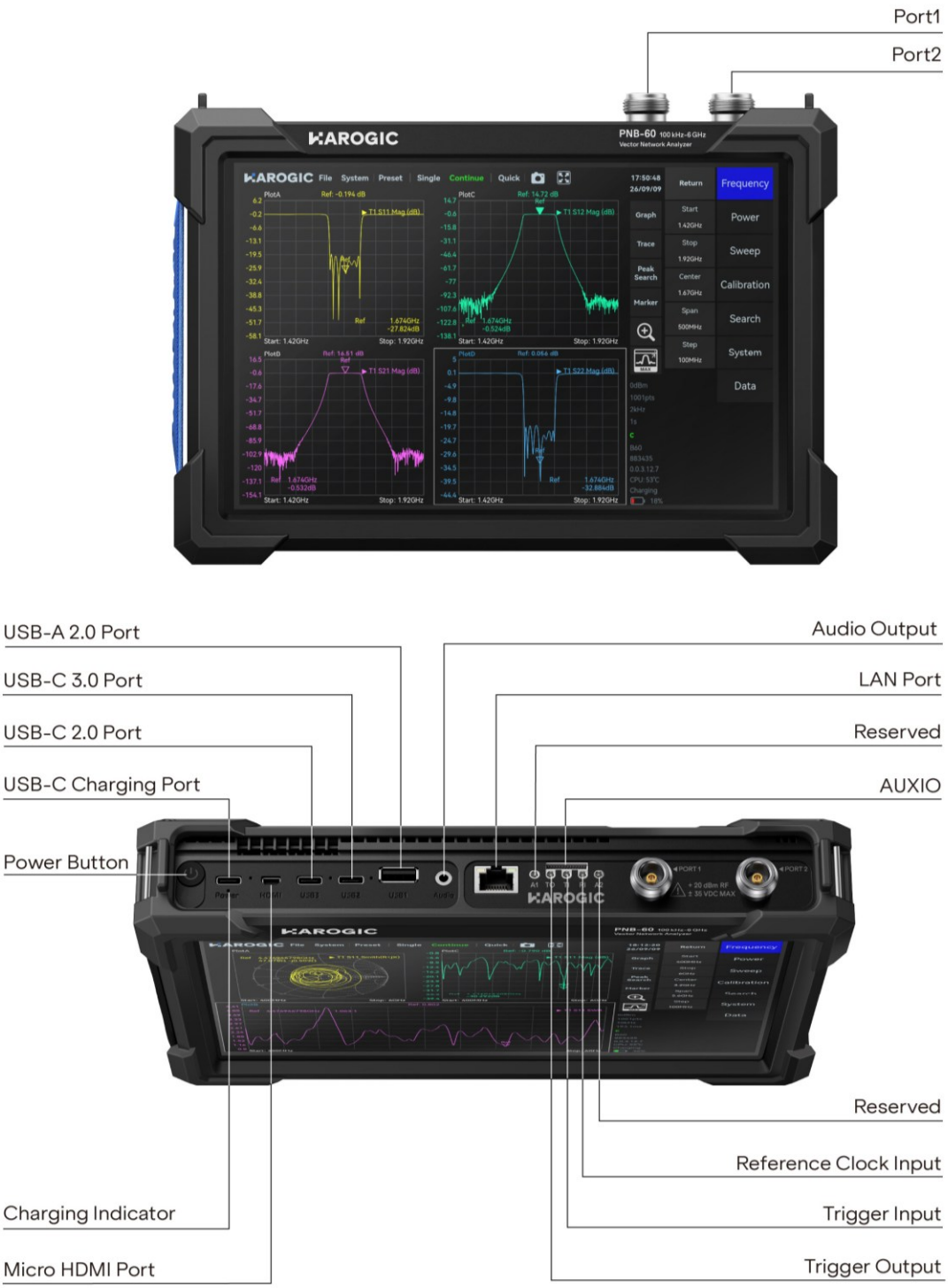



Figure 2 PNB Series Interfaces

**Table 3** PNB-60 Interface Description

| No. | Interface | Description |
|---|---|---|
| 1 | **Power Button** | Turn the instrument on/off. |
| 2 | **Charging port** | Instrument charging port, USB PD 3.0 (65 W). Connect a power supply that meets the operating voltage requirements of the instrument. |
| 3 | **Micro HDMI** | For extended display. |
| 4<br>5<br>6 | **USB3**<br>**USB2**<br>**USB1** | USB interface. USB2 is a USB 3.0 port, while USB1 and USB3 are USB 2.0 ports. These ports can be used to connect an external storage device, a USB keyboard, and a mouse. |
| 7 | **Audio Output** | 3.5 mm headphone jack. |
| 8 | **Ethernet Port** | Gigabit Ethernet port for remote control of the instrument from a PC over Ethernet. |
| 9 | **Multifunctional AUXIO** | Please refer to Appendix B for details. |
| 10 | **Charging Indicator** | Flashing green indicates charging; solid green indicates fully charged. |
| 11 | **Reserved** | / |
| 12 | **Trigger Output** | MMCX (F), 3.3 V CMOS, high impedance. |
| 13 | **Trigger Input** | MMCX (F), 3.3 V CMOS, high impedance.<br>Trigger pulse amplitude: 2.31–3.3 Vpp |
| 14 | **Reference Clock Input** | MMCX (F), 10 MHz<br>Allowable input level: −8 dBm to +10 dBm. |
| 15 | **Reserved** | / |
| 16 | **Port1** | N (F), 50 Ω impedance |
| 17 | **Port2** | N (F), 50 Ω impedance |

## 1.2 Connecting the Instrument

### 1.2.1 VNB Series

The latest version of VNStudio software for the VNB Series is available at the following link:

- HAROGIC VNStudio Software | Software defined radios-HAROGIC

1. Connect the instrument data port to a USB port on the host computer using a USB Type-C data cable. A single cable provides both power and data transfer. A 5 V, 1 A power supply is required;
2. Launch VNStudio (hereinafter referred to as VNS) and confirm that the instrument is correctly detected and connected. The instrument model and serial number should be displayed in the status bar.

### 1.2.2 PNB Series

The PNB Series supports automatic startup. Press the power button on the top panel of the instrument and wait for VNS to launch automatically.

## 1.3 Setting the Measurement Conditions

First, configure the number and layout of Plots under "Graph". Then configure the measurement parameters and display format for each Plot under "Trace", and finally set the standard parameters such as frequency and power (see **Section 3.1** for the configuration logic).

For the filter used in this example, the following settings are recommended:

- **Plot and Trace:** Four-plot layout displaying S11, S21, S12, and S22 respectively; set the display format to Mag (dB)
- **Frequency Range:** 1.4 GHz to 1.9 GHz, covering the 1.65 GHz center frequency and the transition bands and stopbands on both sides
- **Output Power:** 0 dBm (passive device)
- **Sweep Points:** 401
- **IF Bandwidth:** 2 kHz. A narrow IF bandwidth is required to observe stopband rejection as deep as 95 dB; excessive bandwidth may cause the response to be obscured by noise.

For detailed settings, see **Chapter 3 (Measurement Configuration)**.

## 1.4 Performing Calibration

Before measurement, calibrate the measurement reference plane under the current test configuration to remove the effects of test cables and adapters.

- **Electronic calibration is preferred when an electronic calibration module is available:** This example uses a two-port measurement; select SOLT electronic calibration.
- **After calibration, a quick verification with known standards is recommended:** S21 should be approximately 0 dB for a thru connection, while S11 should be very low with a matched load.

For detailed settings, see **Chapter 4 (Calibration)**.

## 1.5 Connecting the DUT

1. Disconnect the electronic calibration module or mechanical calibration standards;
2. Keep the test cables, adapters, and calibration reference planes unchanged from the calibration setup;
3. Connect the two ports of the cavity filter to Port 1 and Port 2 respectively.

## 1.6 Starting the Measurement and Reading the Results

Click "Single" to perform a single measurement, or click "Continue" to start continuous sweeping. Read the following key parameters using markers or automatically extract them using the bandwidth search function:

- **Passband Insertion Loss:** approximately 0.5 dB (S21 ≈ -0.5 dB), a smaller value indicates lower transmission loss.
- **Passband Input Matching (S11):** S11 ≤ −10 dB, corresponding to a return loss ≥ 10 dB, indicating good input matching performance.
- **−3 dB Bandwidth:** approximately 103 MHz, which should agree with the nominal value.
- **Out-of-Band Rejection:** ≥ 95 dB, a larger value indicates greater attenuation of out-of-band signals.
- **Q Factor:** approximately 16 (center frequency / −3 dB bandwidth).

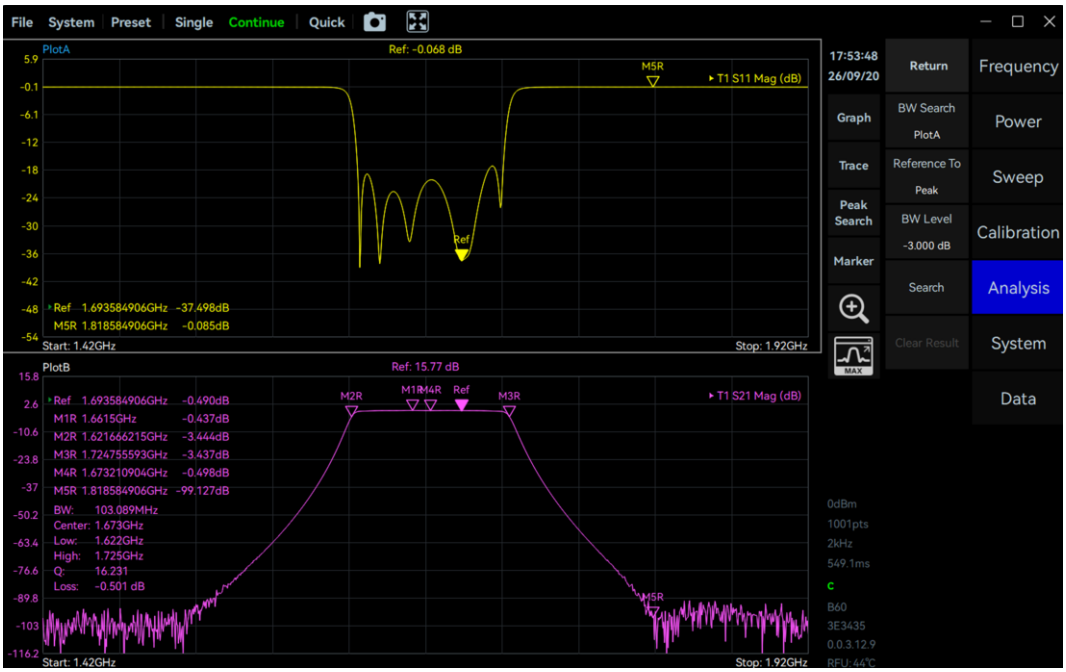



Figure 3 S-Parameter Measurement Results of the Bandpass Filter

For detailed settings, see **Chapter 5 (Measurement and Result Analysis)**.

# 2. Interface Overview

## 2.1 Interface Layout

The VNStudio interface consists of the following areas:

- Menu
- Measurement Display Area
- Graph Setting Area
- Parameter Setting Area
- Instrument State




Figure 4 VNStudio Interface Layout

## 2.2 Instrument Status Area

The instrument status area displays the following information:

- Measurement Parameters (output power, sweep points, and IF bandwidth)
- Sweep Time (sweep time for all traces displayed on the current screen)
- Calibration Correction Status
- Instrument Model / UID
- GUI Version
- Real-Time Instrument Temperature

# 3.Measurement Configuration

This chapter describes the parameter settings for configuring measurement conditions.

## 3.1 Graph and Trace Configuration Logic

VNS uses a "measurement follows display" approach. Based on the Plot and Trace settings, the software automatically determines which S-parameters need to be measured. Unlike some network analyzers, there is no need to configure the measurement parameters and displayed traces separately.

The recommended configuration sequence is as follows:

1. Set the number and layout of Plots under "Graph";
2. Configure the Trace settings for each Plot, including the number of enabled traces, the S-parameter assigned to each trace, and its display format;
3. Set the frequency, output power, and sweep parameters according to the measurement requirements.

For detailed Trace settings, see **Section 5.1**.




Figure 5 Graph Layout Settings




Figure 6 Trace Settings

## 3.2 Frequency Setting

Typical setting: Set the "Start Frequency" / "Stop Frequency" (or "Center Frequency" / "Span") to cover the operating frequency range of the DUT, with sufficient margin to observe the passband and stopband edges. For example, when measuring a bandpass filter with a center frequency of 1.65 GHz, the frequency range can be set to 1.4 GHz to 1.9 GHz.

**Table 4 Frequency Setting Parameters**

| Parameter | Description |
|---|---|
| **Start** | Set the start frequency of the frequency sweep. Setting range: [100 kHz, 6 GHz). |
| **Stop** | Set the stop frequency of the frequency sweep. Setting range: (100 kHz, 6 GHz]. |
| **Center** | Set the center frequency of the current measurement frequency or sweep range. In frequency sweep mode, the center frequency and span together determine the sweep range. In fixed-frequency measurement mode, it sets the operating frequency. |
| **Span** | Set the frequency sweep range. |
| **Step** | Set the frequency sweep interval. Minimum: 1 Hz. |

## 3.3 Power Settings

Typical setting: For passive devices, the output power is typically set to 0 dBm. For active devices such as amplifiers, reduce the output power appropriately (for example, −20 dBm or lower) to prevent the output from exceeding the port damage limit or driving the DUT into compression.

**Table 5 Power Setting Parameters**

| Parameter | Description |
|---|---|
| **Power** | Set the current output power. In frequency sweep and CW modes, this parameter sets the fixed output power. |
| **Start** | Set the start power in power sweep mode. |
| **Stop** | Set the stop power in power sweep mode. |
| **Step** | Set the power interval between adjacent sweep points in power sweep mode. Minimum: 1 dB. |

## 3.4 Sweep Settings

Typical setting: Select Linear Frequency as the sweep mode. An IF bandwidth of approximately 10 kHz provides a balance between noise and sweep speed; reduce it to 1 kHz or below when deeper stopband rejection needs to be observed. Use 401 or 1001 points; more points are recommended for steeper transition bands. If averaging is not required, keep the averaging count at 1.

### 3.4.1 Sweep Setting Parameters

**Table 6 Sweep Setting Parameters**

| Parameter | Description |
|---|---|
| **Sweep Mode** | Set the measurement sweep mode, including linear frequency, logarithmic frequency, power, CW, and segment sweep modes. |
| **Trigger** | Set the sweep trigger source, including Internal and External triggers.<br>**Internal Trigger:** The default trigger mode. The instrument automatically controls the sweep process.<br>**External Trigger:** In this mode, the instrument stops automatic continuous sweeping and waits for an external trigger signal. Upon detecting a valid rising-edge trigger signal, the instrument performs one complete sweep. During the sweep, the instrument ignores any additional trigger signals. Once the current sweep is complete, the instrument waits for the next valid trigger signal. |

| | |
|---|---|
| **Trigger Output** | The trigger output function is disabled by default. When enabled, the instrument outputs a high-level pulse through the TRIG OUT pin of the 14-pin AUXIO connector after each completed sweep. The output level is +3.3 V CMOS. |
| **IF Bandwidth** | Set the receiver IF measurement bandwidth to control the measurement noise level and sweep speed. A narrower IF bandwidth generally reduces noise but increases measurement time. |
| **Dwell Time** | Set the dwell time at each measurement point to improve measurement stability. |
| **Averaging** | Set the number of measurements to be averaged. Averaging multiple measurements reduces random noise and improves measurement stability. |
| **Points** | Set the number of data points for a sweep or measurement.<br>**Linear Frequency, Logarithmic Frequency, and Power Sweep:** Set the number of sampling points within the sweep range, affecting the frequency or power interval between adjacent points.<br>**CW Mode:** The stimulus frequency remains constant. The number of points corresponds to the number of measurement data points in the time record, rather than the number of frequency sampling points.<br>**Segment Sweep:** The number of points is configured separately for each segment. The total number of sweep points across all enabled segments must not exceed 10,001. |
| **Sweep Step** | Set the frequency/power interval between adjacent sweep points. |

### 3.4.2 Conventional Sweep Modes

**Linear Frequency Sweep:** Sweep between the specified start and stop frequencies using equally spaced frequency points. The frequency step size is determined by the sweep range and the number of sweep points.

**Logarithmic Frequency Sweep:** Sweep between the specified start and stop frequencies using logarithmically spaced measurement points. The distribution of frequency points is determined by the frequency range and the number of sweep points.

**Power Sweep:** Sweep over the specified power range using equally spaced power points. The power step size is determined by the number of sweep points. This mode is used to observe measurement results at different stimulus power levels.

### 3.4.3 CW Mode

Purpose: CW mode continuously outputs a continuous-wave signal at a specified frequency and acquires measurement data continuously. Measurement results are displayed with time on

the horizontal axis. This mode is suitable for continuous measurements under fixed-frequency excitation and can be used to observe changes in S-parameters over time.

Operating Procedure:

1. Click "Sweep" and set "Sweep Mode" to "CW";
2. Click "Frequency" and set "Center" to the desired stimulus frequency;
3. Configure the output power, IF bandwidth, dwell time, averaging count, number of points, and other parameters according to the measurement requirements;
4. Select the desired S-parameter trace in the chart to observe how the measurement results change over time.

### 3.4.4 Segment Sweep Mode

Purpose: Segment Sweep mode allows the measurement frequency range to be divided into multiple independent sweep segments. The start frequency, stop frequency, number of sweep points, IF bandwidth, and power can be configured separately for each segment. This mode is suitable for measurement applications requiring different settings across different frequency ranges.

**Table 7 Segment Table Parameters**

| Parameter | Description |
|---|---|
| **No.** | Display the number of the current sweep segment. Up to 10 segments are supported. |
| **Enable** | Enable or disable the current sweep segment. Disabled segments are excluded from the sweep. |
| **Start/Stop Frequency** | Set the start and stop frequencies of the current sweep segment. |
| **Points** | Set the number of measurement points for the current sweep segment. The total number of sweep points across all enabled segments must not exceed 10,001. |
| **IF Bandwidth** | Set the receiver IF measurement bandwidth for the current sweep segment, ranging from 15 Hz to 75 kHz. |
| **Power** | Set the stimulus output power for the current sweep segment. |
| **Save** | Save the current segment configuration. The default save path is VNStudio\data\SegmentTableData. |
| **Load** | Load a previously saved segment configuration. |
| **Add** | Add a new sweep segment. |
| **Delete** | Delete the selected sweep segment. |
| **Restore Calibration Parameters** | Restore all settings in the current segment table to the parameter configuration associated with the currently active calibration. |

Operating Procedure:

1. Click "Sweep", set "Sweep Mode" to "Segment Sweep", and open the "Segment Table";
2. Configure the start frequency, stop frequency, number of points, IF bandwidth, and power for each sweep segment according to the measurement requirements. Select Enable to include the corresponding segment in the sweep;
3. To add a sweep segment, click "Add" and configure the parameters for the new segment. To modify an existing segment, select the corresponding segment and adjust its parameters. Segments can also be deleted as needed;
4. To save the current segment configuration, click "Save". To use an existing configuration, click "Load" and select the corresponding segment configuration file;
5. Perform the appropriate calibration procedure according to the measurement requirements (refer to the Calibration section). After calibration, remove the calibration standards while keeping the test cables and adapters in their original positions and maintaining the same connection configuration. Connect the DUT to the test ports;
6. Start the measurement and observe the S-parameter traces of the DUT.




Figure 7 Segment Sweep Parameter Configuration Interface

## 3.5 Calibration Parameters

The following parameters are used to enable, perform, and manage calibration. For detailed procedures, see **Chapter 4**.

Note: The default values of most parameters are suitable for typical applications and normally do not need to be changed. Follow the procedures in Chapter 4 only when calibration needs to be performed or managed.

**Table 8 Calibration Setting Parameters**

| Parameter | Description |
|---|---|
| **Correction** | Enable or disable calibration correction. This function is available only when valid calibration data exists. When enabled, the current calibration data is applied to the measurement results. |
| **Electric Cal** | Perform automatic calibration using an electronic calibration module and generate measurement error correction data. |
| **Manual Cal** | Perform calibration using a mechanical calibration kit and generate measurement error correction data. |
| **Save CAL** | Save the current calibration results for later use under the same measurement conditions. By default, calibration data is saved in the Cal folder under the software installation directory. |
| **Load CAL** | Load existing calibration results and restore the calibration state for the corresponding measurement conditions. |
| **Port Ext.** | Compensate for propagation delay and loss introduced by an additional connection path between the calibration reference plane and the actual DUT connection plane, effectively moving the measurement reference plane to the DUT connection point. |
| **Reset** | Clear the current calibration state and restore the uncalibrated state. |

## 3.6 System Clock

Note: The internal reference clock is sufficient for most applications and can normally be left at its default setting. Use an external reference only when higher frequency stability or synchronization with other instruments is required.

**Table 9 System Clock Parameters**

| Parameter | Description |
|---|---|
| **Reference Clock Source** | Set the instrument reference clock source to Internal or External. The external reference supports a 10 MHz input. |

## 3.7 Other System Settings

Click "System" -> "Preference" on the menu bar to open the settings window, where the following settings can be configured.

Note: For typical applications, these settings can be left at their default values.

**Table 10 Other System Setting Parameters**

| Parameter | Description |
| --- | --- |
| Screen Lock | **On:** An unlocked icon "" appears on the right side of the screen. Click the icon to change it to "" and lock the screen to prevent accidental operation. Click "" again to unlock the screen. |

# 4.Calibration

## 4.1 Overview

Before making measurements, perform a calibration at the measurement reference plane for the current test setup to compensate for the effects of test cables, adapters, and other test accessories. For general measurement applications, a high-performance electronic calibration module provides a convenient calibration process with sufficient accuracy. For more demanding measurements, use a high-performance and reliable mechanical calibration kit to achieve the highest accuracy.

## 4.2 Calibration Using an Electronic Calibration Module

1. Configure the measurement parameters, including frequency range, output power, number of sweep points, and IF bandwidth, according to the measurement requirements;
2. Connect a supported electronic calibration (ECal) module to the host computer. Allow the ECal module to warm up and become ready. The ready indicator will illuminate;
3. Click "Calibration" -> "Electric Cal", and select two-port calibration or one-port calibration according to your calibration requirements:

- Two-Port Calibration: Connect Port A and Port B of the ECB-60 to Port 1 and Port 2 of the instrument, respectively. Click “SOLT” to open the “SOLT Electronic Calibration” window.
- Single-Port Calibration: Calibration can be performed according to the actual connection configuration:

(1) Two-Port Connection: Connect Port A and Port B of the ECB-60 to Port 1 and Port 2 of the instrument, respectively. In the software calibration interface, select the target port to be calibrated (Port 1 or Port 2).

(2) Single-Port Connection: Connect only one port of the ECB-60 to the instrument. The software automatically detects and displays the connection status of the instrument ports. After confirming that the connection has been correctly identified, click the corresponding port to start calibration.

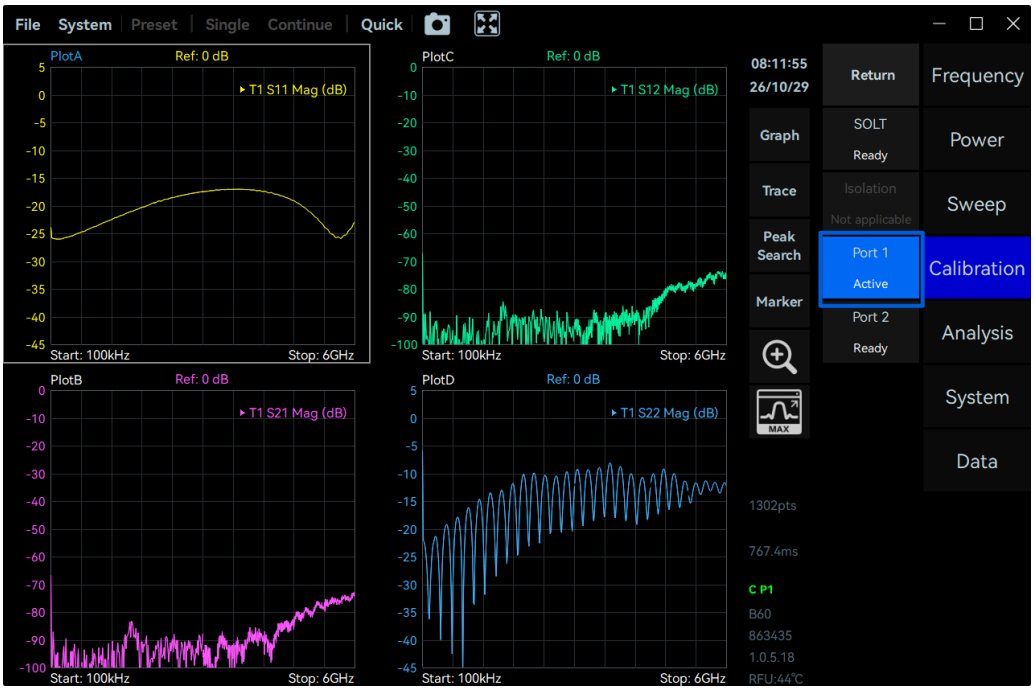



Figure 8 Example of Single-Port Electronic Calibration Status (Port 1)

4. Confirm that the ECal module is connected according to the connection diagram. Tighten all RF connections using a torque wrench set to the appropriate torque;
5. Click "Start" in the pop-up window to initiate the automatic SOLT calibration process. The interface displays the calibration progress in real time. During calibration, do not disconnect the ECal module, modify measurement parameters, or change the port connections. Once SOLT calibration is complete, the status changes from "Ready" to "Active";




Figure 9 SOLT Electronic Calibration Interface and Status Display

6. For measurements requiring a dynamic range greater than 90 dB, additional isolation

calibration is recommended. Click "Isolation" and follow the on-screen instructions to complete the isolation calibration. If isolation calibration is not required, skip this step.

7. Disconnect the ECal module while keeping the test cables and adapters in their original positions and maintaining the same connection configuration. Connect the DUT and perform the measurement.

Note: To save the current calibration data, click "Return" -> "Save CAL". The saved calibration can be loaded later under the same measurement conditions.

### 4.3 Calibration Using a Mechanical Calibration Kit

Manual calibration uses mechanical calibration standards to acquire calibration data and generate system error correction data for the current measurement conditions. The software loads the calibration kit file for the supplied mechanical calibration kit by default. When using another mechanical calibration kit, select the .calkit file that matches the actual calibration kit.

Before performing manual calibration, configure the frequency range, output power, sweep points, IF bandwidth, and other measurement parameters. During calibration, do not change the measurement parameters or the configuration of the test cables and adapters, except when prompted to change calibration standards. The available calibration types are described below.

**Table 11 Manual Calibration Types**

| Manual Calibration Type | Description |
|---|---|
| **2Port SOLT** | Performs a complete two-port SOLT calibration. Follow the on-screen instructions to connect the Open, Short, and Load standards to Port 1 and Port 2, respectively, and complete the Thru calibration between the two ports. Isolation calibration can also be performed as required. Suitable for two-port measurements of S11, S12, S21, and S22. |
| **1Port SOL** | Performs a one-port reflection calibration on the specified port. After selecting the target port, connect the Open, Short, and Load standards in sequence as instructed. Suitable for one-port reflection measurements such as S11 or S22. |
| **2Ports Through** | Performs a Thru calibration between Port 1 and Port 2. |

1. Set the measurement parameters, including the frequency range, output power, sweep points, and IF bandwidth, according to the measurement requirements;
2. Click "Calibration" -> "Manual Cal" to enter the manual calibration interface;
3. Check whether the currently loaded calibration kit matches the mechanical calibration kit being used. To change the calibration kit, click "Load Kit", then import and select the

corresponding .calkit file;

4. Select the appropriate calibration type (2Port SOLT, 1Port SOL, or 2Ports Through) according to the measurement requirements. Follow the on-screen instructions to connect the corresponding calibration standards to the specified ports. After confirming that the connections are correct and stable, click the corresponding item to acquire the calibration data;
5. After all required calibration data has been acquired, click "Done". The software automatically calculates and generates the calibration data and enables "Calibration Correction";
6. Remove the mechanical calibration standards. Keep the positions and bending conditions of the test cables and adapters unchanged, then connect the DUT to the test system and begin the measurement.

### 4.3.1 Calibration Kit Import and Editing

The software supports loading user-defined calibration kits and provides functions for viewing, creating, and editing calibration standard model parameters. The model parameters of Short, Open, Load, Through, and other calibration standards can be configured according to the specifications of the actual mechanical calibration kit. After editing, the current calibration kit definition can be overwritten, or the modified parameters can be saved as a new calibration kit file to accommodate different mechanical calibration kits.

Operating Procedure:

1. Click "Calibration" -> "Manual Calibration" to open the manual calibration interface;
2. Select one of the following operations as needed:

- Load an Existing Calibration Kit: Place a .calkit file that meets the software format requirements in the Cal folder under the VNStudio software directory. Click "Calibration Kit" and select the desired file;
- Edit the Current Calibration Kit: Click "Edit Calibration Kit" to view or modify the model parameters of the currently loaded calibration kit.

3. Modify the corresponding model parameters of the Short, Open, Load, Through, and other calibration standards according to the specifications of the actual calibration kit;
4. Select "Overwrite Current" to save the modifications to the current calibration kit, or select "Save As" to save the modified parameters as a new calibration kit file;
5. Confirm that the desired calibration kit definition has been loaded, and repeat the corresponding mechanical calibration using this definition.

# 5.Measurement and Result Analysis

## 5.1 Trace Settings

Traces are used to display measurement results for specified S-parameters in the graph area. Click Trace on the graph toolbar to select the trace to be configured and set its measurement parameter, display format, vertical scale, and reference position.

Each graph provides four traces, T1–T4. Each trace can be assigned to S11, S12, S21, or S22, with its display status, format, and vertical-axis parameters configured independently.

**Table 12 Trace Parameters**

| Trace | |
|---|---|
| **Enabled** | Show or hide the selected trace. Up to four traces can be enabled in each graph. |
| **Meas** | Set the measurement parameter for the enabled trace, including S11, S12, S21, and S22. |
| **Format** | Set the display format of the current trace, such as logarithmic magnitude, linear magnitude, phase, real, imaginary, SWR, or Smith chart. See Format Parameters for details. |
| **Ref** | Set the vertical-axis reference value for the current trace. |
| **RefPos** | Set the position of the reference value on the vertical axis. |
| **Auto Scale** | Automatically adjust the vertical scale and display range according to the current trace data. |
| **Auto Scale All** | Automatically adjust the display scales of all traces in the selected graph so that the traces are fully displayed. |
| **Default Scale All** | Restore the display scales of all traces in the selected graph to their default settings. |

### 5.1.1 Display Formats

The "Format" button under Trace is used to select how the measurement data of the current trace is displayed. The available formats are described below:

**Table 13 Format Parameters**

| Format | |
|---|---|
| **Mag (dB)** | Display the S-parameter magnitude in decibels. |
| **Mag (Lin)** | Display the S-parameter magnitude on a linear scale. |
| **Phase (°)** | Display the S-parameter phase in degrees. The trace may show discontinuities when the phase crosses a periodic boundary. |
| **Unwrap Phase (°)** | Display unwrapped phase in degrees, removing discontinuities caused by phase wrapping for easier observation of continuous phase variations. |
| **Real** | Display the real part of the complex S-parameter. |
| **Imaginary** | Display the imaginary part of the complex S-parameter. |
| **SWR** | Display the standing wave ratio, primarily for evaluating the impedance matching of a port or DUT. |
| **Smith (R+jX)** | Display impedance characteristics on a Smith chart in terms of resistance R and reactance X. |
| **Smith (G+jB)** | Display admittance characteristics on a Smith chart in terms of conductance G and susceptance B. |
| **Smith (Lin/Ph)** | Display the reflection coefficient on a Smith chart, with measurement values expressed as linear magnitude and phase. |
| **Smith (Log/Ph)** | Display the reflection coefficient on a Smith chart, with measurement values expressed as logarithmic magnitude and phase. |

### 5.1.2 Trace Memory, Math, and Display

The software supports storing the current measurement trace as a reference trace and performing mathematical operations or comparative display between the current measurement data and the stored trace. The stored trace is a copy of the current measurement data at the time it is stored and is not updated during subsequent continuous sweeps.

**Table 14 Trace Memory, Math, and Display Functions**

| | |
|---|---|
| **Memory Trace** | Store the measurement data of the selected trace as a memory trace for subsequent display, comparison, or mathematical operations. |
| **Math** | |
| **Data / Memory** | Divide the current measurement data by the memory trace at corresponding measurement points. This can be used for normalization or comparison of relative changes between the current and reference responses. |
| **Data + Memory** | Perform complex addition of the current measurement data and memory trace at corresponding measurement points. |
| **Math Off** | Disable trace math and restore the display of the current measurement data without mathematical processing. |
| **Display** | |
| **Data** | Display only the current measurement data. During continuous sweeping, the trace is updated in real time with the measurement results. |
| **Memory** | Display only the stored memory trace. |
| **Data & Memory** | Display both the current measurement data and the memory trace for comparison between two measurement results. |

## 5.2 Marker Readout

Markers are used to read the horizontal-axis coordinate and corresponding measurement value at a specified position on a trace. The user can select the target trace and marker number, then position the marker by entering a location or dragging it directly on the graph. Marker functions include enable, delta, and coupling settings, allowing measurement results to be read at specified frequency points and compared across different positions or traces.

**Table 15 Marker Functions**

| | |
|---|---|
| **Marker** | Select the marker to be configured. Ref is the reference marker, and M1–M9 are measurement markers. |
| **Trace** | Select the target trace for the current marker, T1–T4. |
| **Enabled** | Show or hide the marker selected under Marker. |
| **Delta** | Switch the enabled measurement marker to delta mode. When enabled, the marker designation changes from MxR to MxD, and the frequency difference and measurement value difference relative to Ref are displayed. |
| **Coupling** | When enabled, markers with the same number on different traces remain at the same frequency position. Moving one marker moves the corresponding markers on the other traces simultaneously, allowing results on different traces to be compared at the same frequency. |
| **Peak / Min Search** | If markers are already enabled in the current graph, searches for the maximum/minimum value on each associated trace and moves the corresponding marker to that position. If no marker is enabled, searches the selected trace and moves the current marker to the maximum/minimum position. |
| **Previous Peak** | Depending on the selected maximum or minimum search type, searches from the current marker position toward lower frequencies for the adjacent feature point that meets the search criteria. |
| **Next Peak** | Depending on the selected maximum or minimum search type, searches from the current marker position toward higher frequencies for the adjacent feature point that meets the search criteria. |
| **Advance** | Set the threshold and prominence used as filtering criteria for previous/next feature point searches. |

## 5.3 Bandwidth Search and X dB Search

VNS provides Bandwidth Search and X dB Search functions, which automatically locate characteristic points on the current measurement trace that meet specified amplitude conditions. Bandwidth Search is used to calculate parameters such as bandwidth, center frequency, and Q factor. X dB Search is used to locate the frequency positions corresponding to a specified amplitude change relative to a reference value.

### 5.3.1 Bandwidth Search

Purpose: Bandwidth Search uses a specified peak or marker position as the reference. Based on the configured bandwidth level, it searches for the low-frequency and high-frequency intersections on both sides of the reference point that meet the specified condition, and

calculates parameters such as bandwidth, center frequency, and Q factor. This function can be used to analyze the bandwidth characteristics of filters, resonators, and other devices.

Procedure:

1. Click "Analysis" -> "BW Search" to open the Bandwidth Search settings, and select the target plot on which to perform the bandwidth search;
2. Set "Reference To" to "Peak" or "Marker";
3. When "Marker" is selected as the reference, enable the reference marker "M1R" and move it to the desired reference position;
4. Set the "BW Level" For example, to measure the -3 dB bandwidth, set the bandwidth level to -3 dB;
5. Click "Search". Based on the measurement value of the selected reference and the bandwidth level, the software searches for the low-frequency and high-frequency intersections and displays the bandwidth, center frequency, Q factor, and other results;
6. To adjust the marker reference position, move "M1R" and click "Search" again to update the search results;
7. To clear the current search results, click "Clear Result".

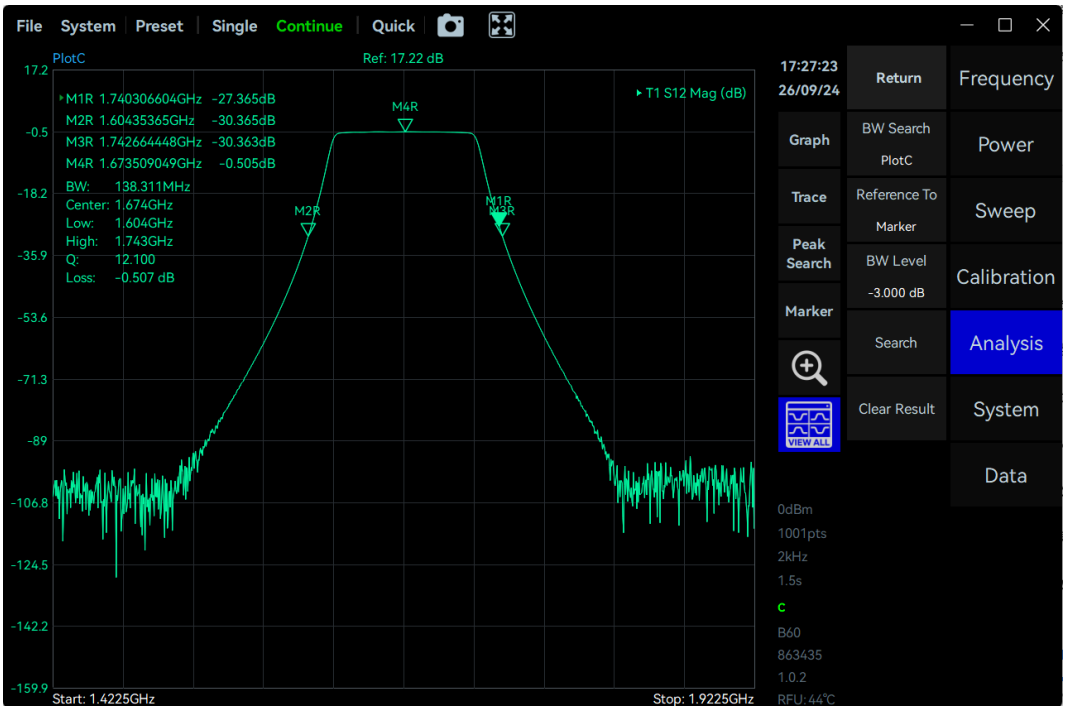



Figure 10 Bandwidth Search

Table 16 Bandwidth Search Setting Parameters

| Parameter | Description |
|---|---|
| BW Search | Select the target plot on which to perform the bandwidth search, from PlotA to PlotD. |
| Reference To | Set the reference for the bandwidth search.<br>Peak: The software automatically uses the peak of the target trace as the reference.<br>Marker: Use the measurement value at the current marker position as the reference. |
| BW Level | Set the amplitude offset relative to the reference. |
| Search | Performs the bandwidth search and displays the search results on the plot. |
| Clear Result | Clear the current bandwidth search results and associated markers. |

After the search is completed, the software automatically marks the **reference point (M1R), low-frequency intersection (M2R), high-frequency intersection (M3R), and center position (M4R)** on the plot and displays the corresponding bandwidth analysis results. The displayed results are described below.

Table 17 Bandwidth Search Result Parameters

| Parameter | Description |
|---|---|
| BW | Frequency difference between the high-frequency and low-frequency intersections. |
| Center | Center frequency between the low-frequency and high-frequency intersections. |
| Low | Frequency of the low-frequency intersection that meets the specified bandwidth level condition. |
| High | Frequency of the high-frequency intersection that meets the specified bandwidth level condition. |
| Q | Q factor calculated from the center frequency and bandwidth. |
| Loss | Display the loss result corresponding to the current bandwidth search. |

### 5.3.2 X dB Search

Purpose: X dB Search uses a specified peak or marker position as the reference and searches in the specified direction for the frequency position that meets the target amplitude condition based on the configured X dB amplitude offset. This function can be used to locate frequency points at which the response of a filter or other device has decreased by a specified amplitude relative to the peak.

Procedure:

1. Click "Analysis" -> "X dB Search" to open the X dB Search settings, and select the target plot

on which to perform the X dB search;

2. Set "Reference To" to "Peak" or "Marker";
3. When "Marker" is selected as the reference, enable the reference marker "M1R" and move it to the desired reference position;
4. Set the "X dB" value. For example, to search for a position 3 dB below the reference value, set X dB to -3 dB;
5. Set the search "Direction" to "Left", "Right", or "Both";
6. Click "Search". Based on the measurement value of the selected reference and the X dB setting, the software searches in the specified direction for the frequency position that meets the target amplitude condition and displays the search results;
7. To adjust the marker reference position, move "M1R" and click "Search" again to update the search results. To clear the current search results, click "Clear Result";

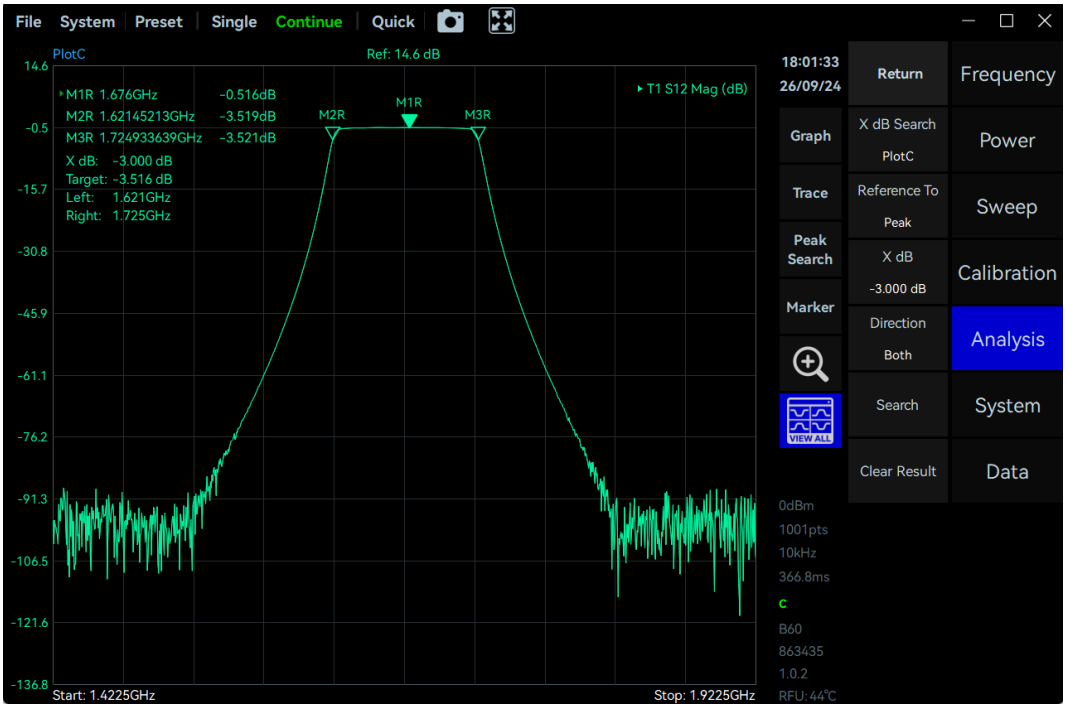



Figure 11 X dB Search

**Table 18 X dB Search Setting Parameters**

| Parameter | Description |
|---|---|
| **X dB Search** | Select the target plot on which to perform the X dB search, from PlotA to PlotD. |
| **Reference To** | Set the reference for the X dB search.<br>Peak: The software automatically uses the peak of the target trace as the reference.<br>Marker: Use the measurement value at the current marker position as the reference. |
| **X dB** | Set the amplitude offset relative to the reference. For example, when set to -3 dB, the software searches for the low-frequency and high-frequency intersections 3 dB below the reference value. |
| **Direction** | Set the search direction.<br>Left: Searches toward lower frequencies.<br>Right: Searches toward higher frequencies.<br>Both: Searches in both directions. |
| **Search** | Perform the X dB search and display the search results on the plot. |
| **Clear Result** | Clear the current X dB search results and associated markers. |

The target amplitude for the X dB search is determined by the measurement value of the reference and the configured X dB offset:

$$A_{Target} = A_{Ref} + X$$

where $A_{Ref}$ is the amplitude value corresponding to the reference, and $X$ is the configured X dB value.

**Table 19 X dB Search Result Parameters**

| Parameter | Description |
|---|---|
| **X dB** | Current amplitude offset setting. |
| **Target** | Target amplitude calculated from the reference value and the X dB offset. |
| **Left** | Frequency position on the left (low-frequency) side that meets the target amplitude condition |
| **Right** | Frequency position on the right (high-frequency) side that meets the target amplitude condition. |

## 5.4 Graph Display Settings

**Table 20 Graph Zoom and Maximization Functions**

| | |
|---|---|
|  | When enabled, drag to select an area of the current trace for an enlarged view. Click the icon again to restore the original display. |
|  | Maximizes the currently selected graph. Click the icon again to restore the original interface layout. |

# 6.Data and Files

## 6.1 Saving and Opening Instrument Configurations

1. Save the Current Configuration

1) Click "File" -> "Save State" on the menu bar;

2) In the "Save Setup" dialog box, specify the save path and file name, then click "Save". By default, instrument configuration files are saved in the "/data" folder.

2. Open a Saved Configuration

1) Click "File" -> "Recall State" on the menu bar;

2) In the "Please select file" dialog box, select a configuration file and click "Open" to apply the saved configuration.

## 6.2 Exporting Measurement Data

Click "Data" in the parameter settings area on the right to expand the submenu containing the following export options, as described in the table below.

**Table 21 Measurement Data Export Functions**

| Function | Description |
|---|---|
| **Export.csv** | Export Trace: Exports the measurement data of the currently selected trace.<br>Export All: Exports the data of all measurement traces. |
| **Export.s2p** | Export two-port S-parameter data. |
| **Export.s1p** | Export the measurement data of the currently selected reflection parameter (S11 or S22). |

## 6.3 Saving Images

### 6.3.1 Saving the Full-Screen Interface

Click the shortcut button "" on the menu bar, or select "File" -> "Save Image", to save a screenshot of the entire software interface to the "/images" folder. The save path cannot be changed.

### 6.3.2 Saving an Individual Measurement Window

1. In the measurement display area, click to select the trace window to be saved;
2. Click "Graph" -> "DataExport" -> "Image", select a save path, and save the current measurement window image to the specified location.

# 7. Advanced Functions

This chapter describes advanced functions intended for specific analysis and diagnostic applications. It is not necessary for routine S-parameter measurements and result interpretation.

## 7.1 Using the Port Extension Function

Purpose: Cables and adapters in the test path introduce additional electrical length, causing the calibration reference plane to remain at the end of the cable rather than at the DUT interface. Port Extension compensates for this delay to electrically shift the reference plane to the DUT interface in software, eliminating phase errors introduced by the cable without requiring recalibration.

Procedure:

1. Complete port calibration using electronic or mechanical calibration;
2. Connect the additional adapter, short interconnect cable, or other component to the end of the calibrated test path;
3. Click "Calibration" -> "Port Ext. ", select the port to be extended, and enable "Port Ext.";
4. Connect an open or short calibration standard to the end of the added connection path. Click "Measure Open" or "Measure Short", as appropriate for the connected standard;
5. After the measurement is completed, the software automatically calculates the port extension parameters. Click "Configuration" to view parameters including distance, time, velocity factor, and loss compensation;
6. If compensation for the insertion loss of the added connection path is required, configure the corresponding loss compensation parameters;
7. Click "Apply" to complete the port extension;
8. Remove the calibration standard and connect the DUT to the end of the added connection path for measurement.

**Table 22 Port Extension Parameter Description**

| Function | Description |
|---|---|
| **Port** | Select the test port for port extension. |
| **Port Ext.** | Enable or disable port extension for the selected port. |
| **Velocity Factor** | The ratio of the signal propagation velocity in the cable to the speed of light, used to convert between delay and electrical length. |
| **Measure Short** | Perform a measurement after a short calibration standard is connected to the end of the added connection path. The software automatically calculates the corresponding port extension parameters. |
| **Measure Open** | Perform a measurement after an open calibration standard is connected to the end of the added connection path. The software automatically calculates the corresponding port extension parameters. |
| **Configure** | Display the port extension parameters, including distance, time, velocity factor, and loss compensation parameters. |
| **Apply** | Apply the currently calculated or configured port extension parameters. |
| **Reset** | Clear the current port extension parameters and restore the default settings. |




Figure 12 Port Extension Settings

## 7.2 Using the DTF Function

Purpose: The DTF (Distance to Fault) function transforms frequency-domain measurement results into the time domain and displays the results as time or distance information. It is used to analyze reflection characteristics in cables, connectors, and transmission paths, and to locate reflection events caused by open circuits, short circuits, impedance discontinuities, and other conditions. This function is suitable for cable fault location, connection integrity checks, and transmission path analysis.

Procedure:

1. Set the Frequency Range: Configure the frequency range according to the required measurement distance and resolution. A wider frequency range generally provides higher distance resolution;
2. Click "Analysis" -> "DTF" to enable the DTF function for the selected trace;
3. Configure the transform mode, velocity factor, window function, X-axis display mode, and start and stop distances;
4. Calibrate the measurement port. DTF supports analysis of measurement results obtained after either one-port or two-port calibration;
5. Connect the device under test (DUT) to the calibrated measurement port and perform a single or continuous sweep;
6. Read the Results: The horizontal axis represents distance or time, while the vertical axis displays DTF parameters such as reflection coefficient, reflection step response, or impedance, depending on the selected display format. The positions of reflection peaks indicate the locations of reflection events and can be used to identify open circuits, short circuits, or impedance discontinuities. In reflection coefficient display mode, positive reflection peaks typically indicate open circuits or high impedance, while negative reflection peaks typically indicate short circuits or low impedance.

**Note:** After configuring the measurement parameters and completing calibration, the DTF analysis parameters can still be adjusted as needed, including the transform mode, velocity factor, window function, X-axis display mode, and start and stop distances. Changing these DTF analysis parameters does not require recalibration.

Table 23 DTF Parameter Description

| Function | Description |
|---|---|
| **Trace** | Select the target trace for DTF analysis. For distance-to-fault measurements, a reflection trace such as S11 or S22 is typically selected. |
| **Transform** | Set the time-domain transform mode. Low-pass and band-pass modes are available.<br>Low-pass: Requires frequency points to satisfy the corresponding harmonic relationship. The software determines the start frequency based on the stop frequency and number of sweep points. This mode is suitable for time-domain or distance-domain analysis of transmission paths such as cables and connectors.<br>Band-pass: Performs the transformation using the currently configured frequency range and is suitable for DUTs with a limited frequency range. |
| **Format** | Set the display format of the DTF results, including Pulse dB, Pulse Linear, Transmission Step, and Impedance. |
| **Window** | Set the window function used for the time-domain transform, including Rectangular, Gaussian, Hanning, Hamming, and Blackman windows. Different window types affect distance resolution and sidelobe characteristics. |
| **Vel. Factor** | Set the ratio of the propagation velocity in the transmission medium to the speed of light in vacuum. It is used to convert propagation time into distance and should be set according to the actual cable specifications. |
| **X Axis** | Set the horizontal-axis display mode. Distance or time can be selected. |
| **Start / Stop** | Set the display range of the current time-domain or distance-domain results. This setting adjusts the display range of the DTF results without changing the previously acquired frequency-domain measurement data. |

### 7.2.1 DTF Distance-to-Fault Example

This example uses an RF cable approximately 0.5 m in length as the device under test (DUT). By measuring the reflected signal from the end of the cable, it demonstrates how to use the DTF function to locate the cable end.

1. Select the S11 trace, click "Analysis" -> "DTF" in the parameter settings area to enable the DTF function, and set "Transform Mode" to "Bandpass".
2. Set the frequency range to 100 MHz–6 GHz and the number of sweep points to 1001.
3. Click "Calibration" -> "Manual Calibration" -> "Calibrate" -> "1 Port SOL".
4. Select Port 1 and perform the Open, Short, and Load calibrations in sequence.

5. Click the chart maximize button and select "Auto Scale" under "Trace" settings to display the complete S11 trace. Click "Peak Search" to locate the reflection peak near the calibration reference plane.
6. Connect the approximately 0.5 m cable under test and leave its far end open. Double-click the trace near the newly appearing reflection peak to activate the reference marker, then read the reflection distance displayed by the marker.

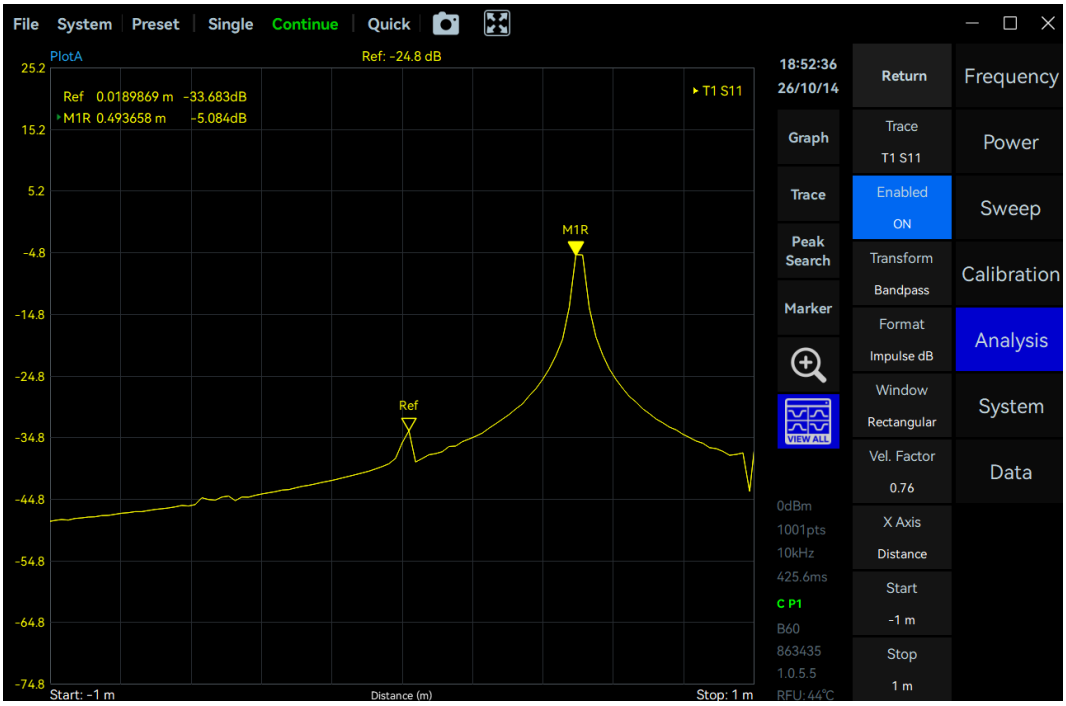



Figure 13 DTF Distance-to-Fault Measurement Result for a 0.5 m Cable

As shown in the figure, a distinct reflection peak appears at approximately 0.5 m, corresponding to the end of the cable under test. The distance indicated by the marker is approximately equal to the actual cable length.

## 7.3 Port Wave Data Display

Purpose: The wave data display function allows users to view measurement data from different receiver channels under excitation from different ports. The software supports wave measurement quantities including A1, A2, B1, B2, R1, and R2, with display formats such as magnitude, phase, real, and imaginary. This function can be used to analyze the amplitude and phase characteristics of received signals.

**Table 24 Wave Measurement Quantities**

| Measurement Quantity | Description |
|---|---|
| **A1** | Reflected wave data measured by the Port 1 receiver when Port 1 provides the excitation signal. |

| | |
|---|---|
| **B1** | Transmitted wave data measured by the Port 2 receiver when Port 1 provides the excitation signal. |
| **R1** | Incident reference data measured by the reference receiver when Port 1 provides the excitation signal. |
| **A2** | Transmitted wave data measured by the Port 1 receiver when Port 2 provides the excitation signal. |
| **B2** | Reflected wave data measured by the Port 2 receiver when Port 2 provides the excitation signal. |
| **R2** | Incident reference data measured by the reference receiver when Port 2 provides the excitation signal. |

Note: Conventional S-parameters are calculated from the ratios of the wave quantities described above, as follows:

Forward parameters: $S_{11} = \frac{A1}{R1}$, $S_{21} = \frac{B1}{R1}$; Reverse parameters: $S_{22} = \frac{B2}{R2}$, $S_{12} = \frac{A2}{R2}$

Procedure

1. Click "Graph" and enable the "Wave Data" switch.
2. Select the target plot (PlotA–PlotD) for displaying the wave data.
3. Click "Trace", select the trace to configure (T1–T4), and enable the selected trace.
4. Under "Measurement", select the wave measurement quantity to display, such as A1, A2, B1, B2, R1, or R2.
5. Under "Format", select the desired display format.
6. To display multiple wave quantities simultaneously, enable additional traces in the same plot and configure their measurement quantities and display formats separately. The software displays the corresponding measurement curves for each trace.

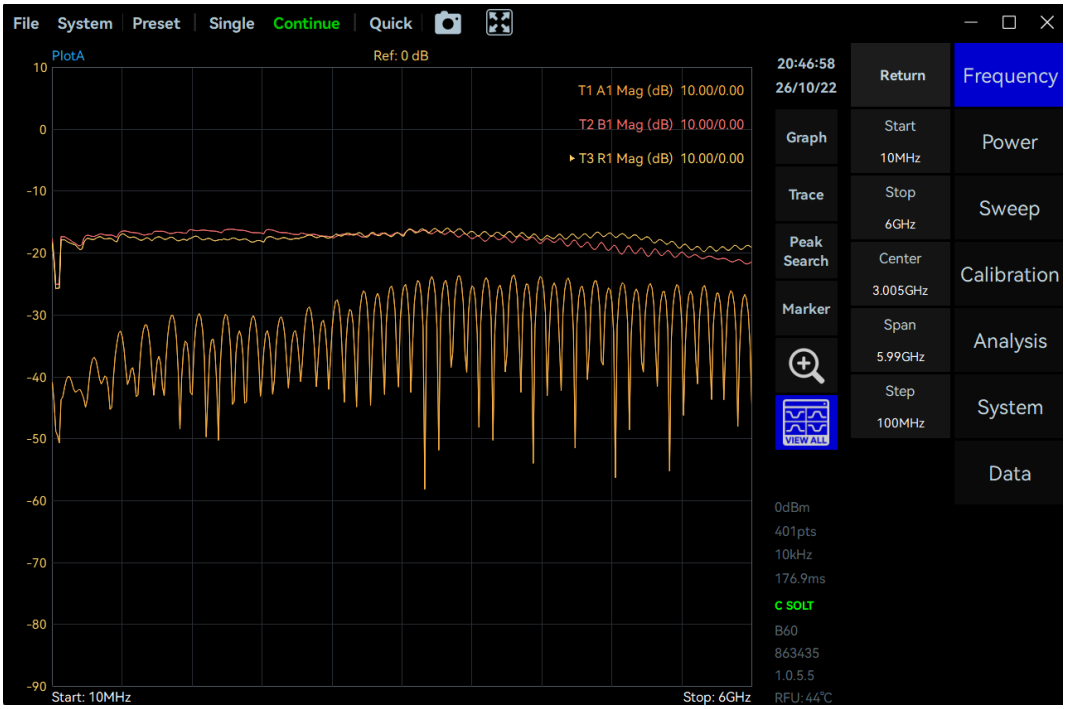



Figure 14 Simultaneous Display of A1, B1, and R1 Wave Quantities

# 8.Quick Operations

VNS provides a range of interactive features designed to improve ease of operation. This chapter introduces these convenient operations to help users improve efficiency during actual measurements.

## 8.1 Using the Quick Menu

Purpose: In full-screen mode, VNS hides the right-side settings panel. To adjust commonly used parameters, users would normally need to exit full-screen mode first. The Quick menu consolidates the most commonly used network analyzer settings into a floating window. Click the "Quick" button on the menu bar to open the window and quickly configure commonly used settings without exiting full-screen mode.

Procedure:

1. Click the "Quick" button on the menu bar. The Quick window appears in the center of the screen;
2. Configure the required parameters in the window as needed (see the table below for details);
3. Click the pin icon "" in the upper-right corner of the window title bar to pin the window and keep it open; click the close icon "" to close the window.




Figure 15 Quick Settings Interface

Table 25 Quick Menu Parameter Description

| Function | Item | Description |
|---|---|---|
| **Frequency** | Start / Stop / Center / Span | Quickly set the start frequency, stop frequency, center frequency, and span. |
| **Sweep** | Power / IF Bandwidth / Points | Set the output power, IF bandwidth, and number of sweep points. |
| **Preset Scenarios** | Passive Device - General / Passive Device - Better Dynamic / Passive Device - High Dynamic / Cable - Connector / Amplifier - Small Signal / Antenna | Apply the recommended measurement configuration for the corresponding device type with one click. |
| **Calibration Status** | Electronic Calibration SOLT / Electronic Calibration ISO (Optional) / Electronic Calibration SOL | Display the readiness status of the electronic calibration items. |
| **Restore** | Restore | Restore the current configuration to the default settings. |

## 8.2 Using the Mouse Wheel / Two-Finger Gestures to Quickly Adjust the Plot

Purpose: Zoom and move the plot directly without entering a menu, allowing the displayed range to be adjusted quickly.

- **Zoom (Adjust Scale):** Scroll the mouse wheel, or use a two-finger pinch gesture on the touchscreen.
- **Move the Display Position:** Click and drag vertically with the mouse, or drag with one finger on the touchscreen.

## 8.3 Using Right-Click Quick Operations

Purpose: Right-click the plot to open the quick operation menu for the current plot.

**Table 26 Right-Click Quick Operation Description**

| Function | Description |
|---|---|
| Create Marker | Creates a marker at the position where the plot is right-clicked. |
| Clear All Markers | Clears all markers from the current plot. |
| Auto Scale | Automatically scales the axes so that the entire trace is displayed at an appropriate scale. |
| Trace | Opens trace management to select, add, delete, or configure traces in the current plot. |
| Data Export | Opens the data export submenu, including Image, S2P, and CSV. |




Figure 16 Right-Click Quick Operations

## 8.4 Using Double-Click to Quickly Create a Marker

Purpose: Double-click a plot to quickly create a new marker at the clicked position. The software also performs a local peak search and automatically positions the new marker at the peak.

Operation: Double-click the plot with the mouse, or double-tap the plot on the touchscreen.




Figure 17 Creating a Marker and Performing a Local Peak Search by Double-Clicking

## 8.5 Maximizing a Plot

Purpose: Maximizes a single plot to fill the entire plot area, making it easier to focus on an individual S-parameter.

1. Click the Maximize Plot button "" for the corresponding plot;
2. Double-click the title area at the top of the plot (for example, Plot X), or click the Maximize Plot button "" again to restore the multi-plot layout.

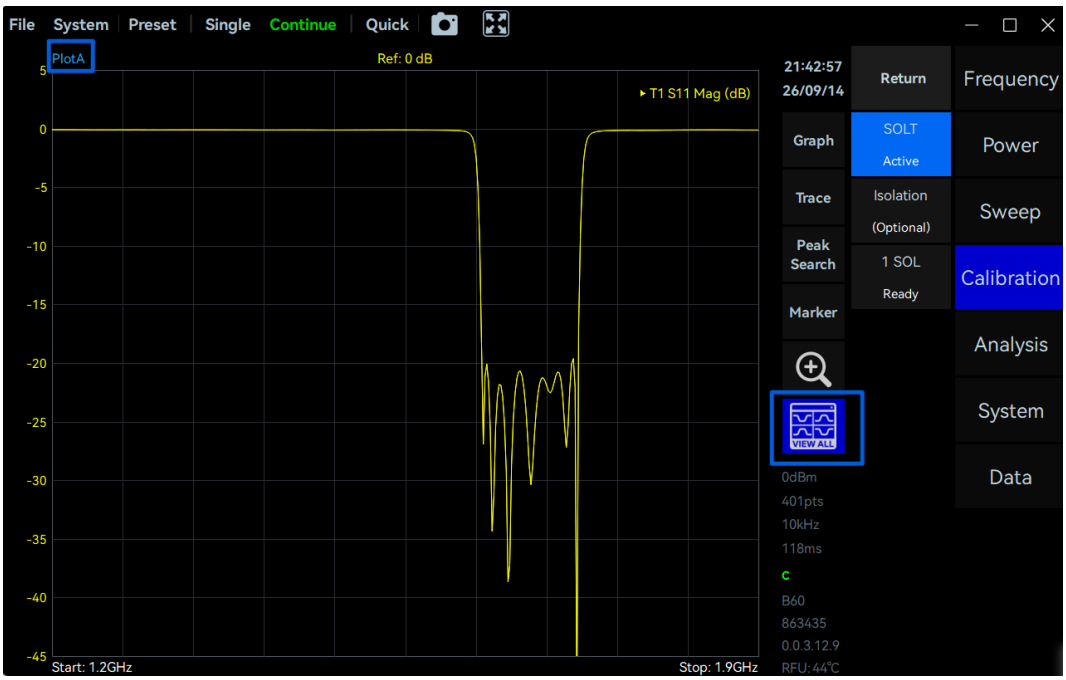



Figure 18 Maximizing a Plot

## 8.6 Using the Magnifier for Local Zoom

Purpose: Magnifies local details of a plot while retaining visibility of both the overall trend and local waveform details.

1. In multi-plot view: Click the Magnifier button "" to magnify the plot.
2. When a single plot is maximized: In addition to the magnified display, a picture-in-picture navigation window appears in the upper-right corner of the plot. The red box in the window indicates the position of the currently magnified area within the main plot.

## 8.7 Maximizing the Plot Area by Hiding the Right-Side Settings Panel

Purpose: Hides the right-side settings panel so that the plot area fills the window, providing the maximum display area for measurement results.

Operation: Click the Maximize button "" on the menu bar.

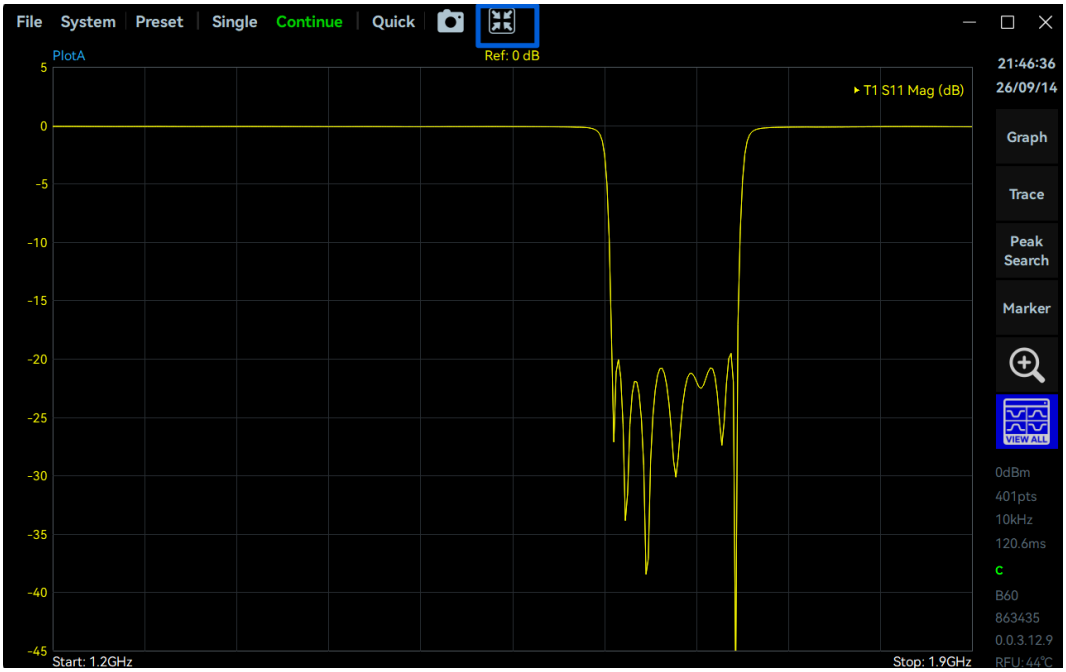



Figure 19 Hiding the Right-Side Settings Panel

# 9.System Settings and Maintenance

## 9.1 Setting the Startup State

The instrument supports user-defined startup states. The available options are described below:

**Table 27 VNStudio Startup States**

| No. | Startup State | Description |
|---|---|---|
| 1 | **Default** | Use the instrument default configuration |
| 2 | **Last State** | Use the parameter configuration from the previous software session as the initial configuration at startup |
| 3 | **User Preset** | Use a user-saved configuration file as the initial configuration at startup |

To configure the startup state:

1. Click "File" -> "Power on State" on the menu bar;
2. For "Default" or "Last State", simply select the desired option. The software will use the selected state as the initial configuration the next time it starts;
3. Select "User Preset". In the "Please select file" dialog box, choose a user-saved configuration file. The next time the software starts, it will open using the specified configuration.

## 9.2 Theme Settings

Click "System" -> "Theme" on the menu bar to switch between the "Dark" and "Light" themes.

## 9.3 Single and Continuous Measurements

VNS supports two measurement modes: Single and Continuous. By default, the software starts in Continuous measurement mode.

**Single:** Perform one complete frequency sweep. After the measurement is completed, sweeping stops and the measurement traces are updated. To perform another measurement, click "Single" or "Start".

**Continue:** Enter continuous measurement mode, repeatedly performing frequency sweeps and updating the measurement traces in real time. Click "Stop" to pause the current continuous measurement, and click "Start" again to resume.

## 9.4 Preset and Full Screen

**Preset:** Click "Preset" on the menu bar to restore the current software configuration to the instrument default settings.

**Full Screen:** Click the "Full Screen" button on the menu bar to enter full-screen display mode. The enlarged viewing area makes it easier to observe and analyze measurement traces. Click the button again to exit full-screen mode and restore the original interface.

## 9.5 Software and Firmware Update

This section describes how to update the instrument software and firmware.

### 9.5.1 Parameter Description

**Information:** Displays the current and target software and firmware version information.

**Update Content:** Displays detailed information about the changes included in the target version.

**Update Notification:** Sets whether the software automatically displays an update notification when a new version is detected.

**Update Method:** The available update methods are described below.

**Table 28 Update Methods**

| Update Method | Description |
|---|---|
| Online Download | Automatically connect to the remote server to download and install the latest software and firmware versions. |
| Use Built-In Firmware | Use the firmware package included with the current software to update the instrument firmware to the version matched with the current software. |
| Import Offline Package | Manually load and install an update package from a USB drive or local storage device. |

### 9.5.2 Online Download Update Example

1. Click "System" -> "Update" to enter the update interface;
2. Select the "Update notification" checkbox. When the host computer is connected to the Internet, if a new version is detected during software startup or operation, the system automatically displays the "Update" window;
3. Set "Update to" to "Online Download". The system starts downloading the update package, and the "Update" button is temporarily disabled. After the installation package has been successfully downloaded and parsed, detailed update information is displayed and the "Update" button becomes enabled;
4. Carefully compare the current and target version information and review the list of new features and fixes. After confirming the information, click "Update" in the lower-right corner;
5. After clicking "Update", the software automatically exits and enters the update process. Keep the update window open until the progress bar is complete and the software automatically restarts and returns to the main interface.

# Appendix A VN Series AUXIO Pin Description

**Table 29 Multifunction AUXIO Pin Description (Pin Order Shown from Right to Left)**

| Pin | Name | Direction | Logic Level | Description |
|---|---|---|---|---|
| 1 | TRG_In | I | 3.3 V | External trigger input |
| 2 | GIO0 | / | / | / |
| 3 | TRG_OUT | O | 3.3 V | Trigger output |
| 4 | GND | / | / | Ground |
| 5 | NC | / | / | / |
| 6 | 3V3_FPGA | O | / | Power output, 3.3 V |
| 7 | UART_TX | / | / | / |
| 8 | GND | / | / | Ground |
| 9 | UART_RX | / | / | / |
| 10 | NC | / | / | / |
| 11 | NC | / | / | / |
| 12 | VBUS_P | O | / | USB power output |
| 13 | GND | / | / | Ground |
| 14 | VBUS_P | O | / | USB power output |

# Appendix B PN Series AUXIO Pin Description

**Table 30 Multifunction AUXIO Pin Description (Pin Order Shown from Left to Right)**

| Pin | Name | Direction | Logic Level | Description |
|---|---|---|---|---|
| 1 | TRG_EXTIN | / | / | Reserved |
| 2 | REM_TRG_OUT | / | / | Reserved |
| 3 | RFM_TRG_IO3 | O | / | Trigger output |
| 4 | GND | / | / | Ground |
| 5 | RFM_TRG_IO4 | O | 3.3V | 1PPS output |
| 6 | 3V3D | O | / | Power output, 3.3 V |
| 7 | USART1_RX | / | / | Reserved |
| 8 | GND | / | / | Ground |
| 9 | USART1_TX | / | / | Reserved |
| 10 | GPIO3 | O | 3.3V CMOS | Programmable General-Purpose Digital Output |
| 11 | GPIO2 | O | 3.3V CMOS | Programmable General-Purpose Digital Output |
| 12 | GPIO1 | O | 3.3V CMOS | Programmable General-Purpose Digital Output |
| 13 | GND | / | / | Ground |
| 14 | GPIO0 | O | 3.3V CMOS | Programmable General-Purpose Digital Output |

# Appendix C Version History

| Version | Description | Date |
| --- | --- | --- |
| **V1.0.4** | 1. Initial Version | 23/10/2026 |
| **V1.0.6** | 1. Added: Calibration Kit Import and Editing Section.<br>2. Added: DTF Distance-to-Fault Example and Port Wave Data Display section.<br>3. Refactored: Sweep Settings section. | |

www.harogic.com
info@harogic.com
+65-8299 8857