## Measurement with Option

## Measurement with Options

The following measurement is available depending on the installed option. See the Activating Software Option for activating option.

- Frequency Offset Measurement (Option 008)
- Measurement Wizard Assistant (Option 790)
- Enhanced Time Domain Analysis (Option TDR)
- Time Domain Analysis (Option 010)

# Frequency Offset Measurement

## Frequency Offset Measurement (Option 008)

- Overview
- Measurement of Harmonic Distortion
- Measurement of Mixers

Other topics about Measurement with Options

## Overview

- Introduction
- Measurement consideration for mixers and frequency converter devices

Other topics about Frequency Offset Measurement

### Introduction

The E5071C option 008 provides a frequency-offset function and an absolute measurement function.

Generally with network analyzers, the frequency must be the same both at the signal source and at the receiver, so frequency-translating devices such as mixers or converters cannot be measured. To enable the analyzer to measure frequency conversion devices, the receiver frequency must be synchronized with a frequency consisting of the input frequency and a certain added offset, which is called frequency-offset sweep.

The E5071C option 008 provides not only this frequency-offset sweep but state-of-the-art mixer calibration features: vector-mixer calibration and scalar-mixer calibration. Vector-Mixer Calibration allows you to measure the conversion loss, phase, reflection parameter, and group delay of the mixer. Scalar-Mixer Calibration performs high-accuracy measurement for the magnitude and reflection parameter of mixer conversion loss.

Moreover, the E5071C option 008 supports an absolute measurement function. The network analyzers normally measure the ratio of the test signal's magnitude and phase to the reference signal. In contrast, the absolute measurement function performs a measurement of the absolute value of the test signal's magnitude. In combination with the frequency-offset function, the absolute measurement function provides measurements that include measurement of harmonic distortion. It also provides high-accuracy measurement when combined with Power Calibration and Receiver Calibration.

NOTE When you measure the 3rd harmonics over 8.5 GHz with options 2D5, 4D5, 2K5 and 4K5, it is recommended to suppress the carrier signal using filter etc before executing the measurement for accurate results.

### Measurement consideration for mixers and frequency converter devices

In mixers and frequency converter devices measurements, you have RF and LO inputs and an IF output. Also appearing from the IF port are several other mixing products of the RF and LO signals. In mixers and frequency converter devices measurements, leakage signals from one

mixer port propagate and appear at the other two mixer ports. These unwanted mixing products or leakage signals can cause distortion by mixing with the harmonic of the analyzer's first down-conversion stage and cause measurement error.

To ensure successful mixer measurements, the following measurement challenges must be taken into consideration:

### Reducing the Effect of Spurious Responses

NOTE By choosing test frequencies (frequency list mode), you can reduce the effect of spurious responses on measurements by avoiding frequencies that produce IF signal path distortion. To avoid frequency spacings equal to the analyzer's IF is an important factor. In the case of the E5071C, the IF is 3 MHz and 3.25 MHz with fundamental mixing under 8.5 GHz measurements, and 3rd harmonic mixing over 8.5 GHz measurements. Therefore, avoid the frequencies which meet the following formula. (*1)

m ×RF ± n×LO = l × E5071C_Local ± 3 MHz or 3.25 MHz (m, n, l = 0, 1, 2 ...)
where, E5071C_Local is

NOTE Measured Signal Frequency + 3 MHz or 3.25 MHz (Under 8.5 GHz)

NOTE 1/3 × (Measured Signa Frequency + 3 MHz or 3.25 MHz) (Over 8.5 GHz)

(*1): Except the target measuring frequency, that is m=n=1

NOTE Another effective technique is to reduce the instrument's IF bandwidth. Reducing the IF bandwidth will more selectively filter signals in the instrument's IF signal path.

NOTE If some spurious is inevitable, then one of the practical way is to neglect or delete the presumed frequency data.

### Eliminating Unwanted Mixing and Leakage Signals

By placing filters between the mixer's IF port and the receiver's input port, you can eliminate unwanted mixing and leakage signals from entering the analyzer's receiver. Filtering is required in both fixed and broadband measurements. Therefore, when configuring broad-band (swept) measurements, you may need to trade some measurement bandwidth for

the ability to more selectively filter signals that enters the analyzer receiver.

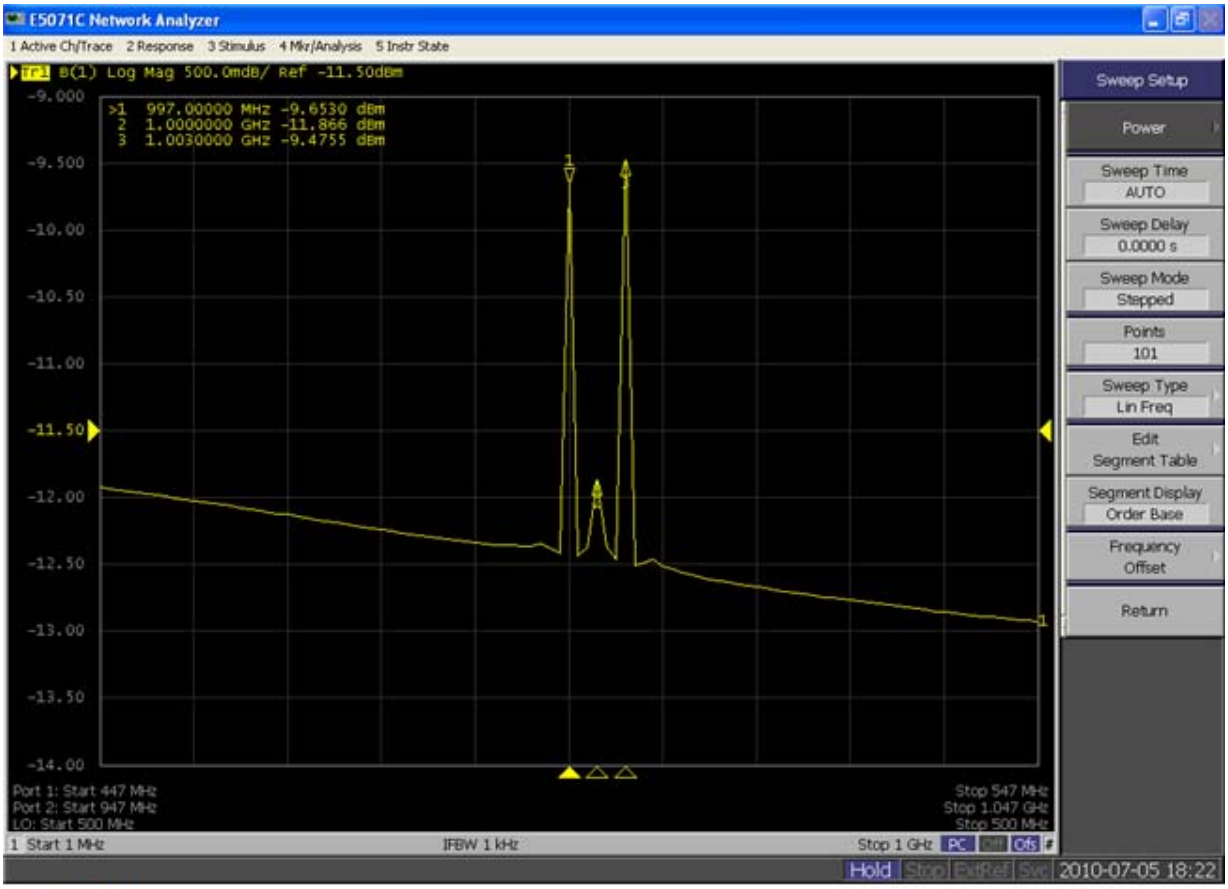
### Minimizing Source and Load Mismatches

Source and load mismatches cause measurement error and will add to overall measurement uncertainty. The E5071C's scalar mixer calibration (SMC) or vector mixer calibration (VMC) removes the error caused by these mismatches.

NOTE Output power level calibration is not included in the SMC and VMC. An independent power calibration is required to apply power calibration.

### Example of Symptom

This following figure shows an example of symptom.



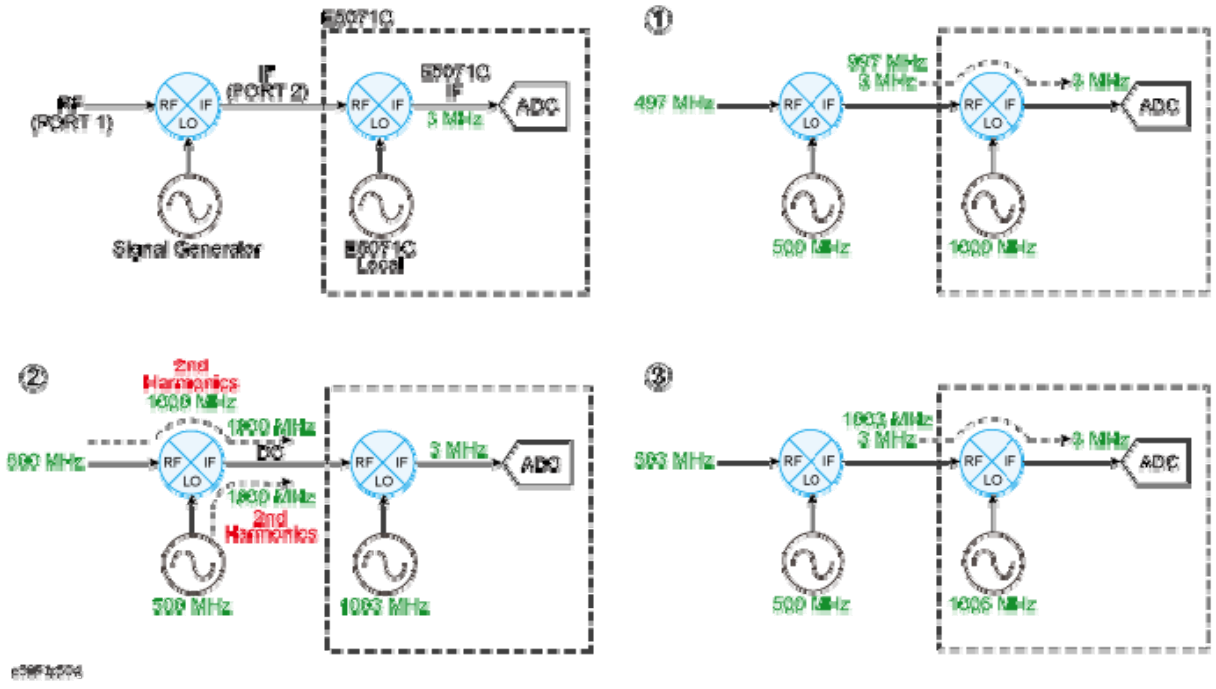

e5071c505

- In the configuration at the markers 1 and 3, the measured signal becomes 3 MHz. A signal leakage through the E5071C mixer affects the E5071C IF signal.
- In the configuration at the marker 2, the 2nd harmonic of RF and IF is the same as the measured signal. A signal leakage of the 2nd harmonic through the mixier (DUT) affects the measured signal. (This

configuration is not a common usage. This is a mathematical example case to explain the symptom.)

| Marker No. in Example result | RF (Port 1) | LO (from Signal Generator) | Measured Signal (Port 2) | E5071C_Local | E5071C_ IF |
|---|---|---|---|---|---|
| 1 | 497 MHz | 500 MHz | 997 MHz and 3 MHz | 1000 MHz | 3 MHz |
| 2 | 500 MHz | | 1000 MHz and DC | 1003 MHz | |
| 3 | 503 MHz | | 1003 MHz and 3MHz | 1006 MHz | |

**Signal Leakage through mixers**



## Measurement of Harmonic Distortion

- Overview
- Procedure

Other topics about Frequency Offset Measurement

### Overview

Using the frequency-offset function and absolute measurement function in combination allows you to measure harmonic distortion of nonlinear devices such as mixers and amplifiers.

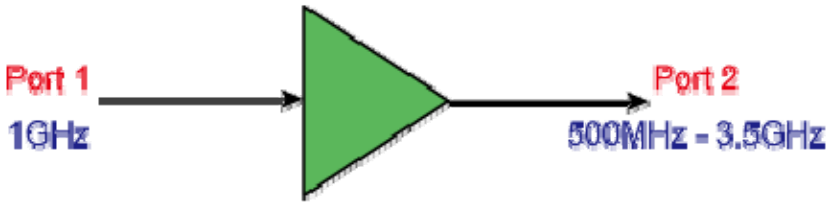
### Procedure

| Item | Description |
|---|---|
| 1. Setting Frequency-Offset | Sets frequency-offset function |
| 2. Implementing Receiver Calibration | Implements receiver calibration |
| 3. Setting Absolute Measurement Parameters | Sets absolute measurement parameters<br>• Procedures for setting absolute measurement parameters |
| 4. Harmonic Distortion Measurement | Implements harmonic distortion measurement<br>• Connection of DUT<br>• Procedures for setting measurement parameters |

### 1. Setting Frequency-Offset Function

The frequency-offset function allows you to make measurements while the frequencies are different at each test port. In this case, measurement is done by using the setting example for measurement frequency in the following figure. For the setting of frequency-offset sweep, see Setting Frequency-Offset.

Set Base frequency: 500MHz - 3.5GHz
Set Port 1 frequency: $M_1$=0, $D_1$=1, $O_1$=1GHz
Set Port 2 frequency: $M_2$=1, $D_2$=1, $O_2$=0Hz




e5071c463

NOTE When you measure the 3rd harmonics over 8.5 GHz with options 2D5, 4D5, 2K5 and 4K5, it is recommended to suppress the carrier signal using filter etc before executing the measurement for accurate results.

### 2. Implementing Receiver Calibration

Receiver calibration is required for the absolute measurement. Receiver calibration of the port to be used for the measurement increases the accuracy in the absolute measurement.

For detailed information on receiver calibration, see Receiver Calibration.

### 3. Setting Absolute Measurement Parameters

The frequency-offset function has an added absolute measurement function.

**Procedures for setting absolute measurement parameters**

Follow the steps below to set the absolute measurement parameters.

1. Press **Channel Next**/**Channel Prev** keys and **Trace Next**/**Trace prev** keys to activate the trace for which you want to implement absolute measurement.
2. Press **Meas** key to display the Measurement Menu.
3. Click **Absolute** to select measurement parameters.

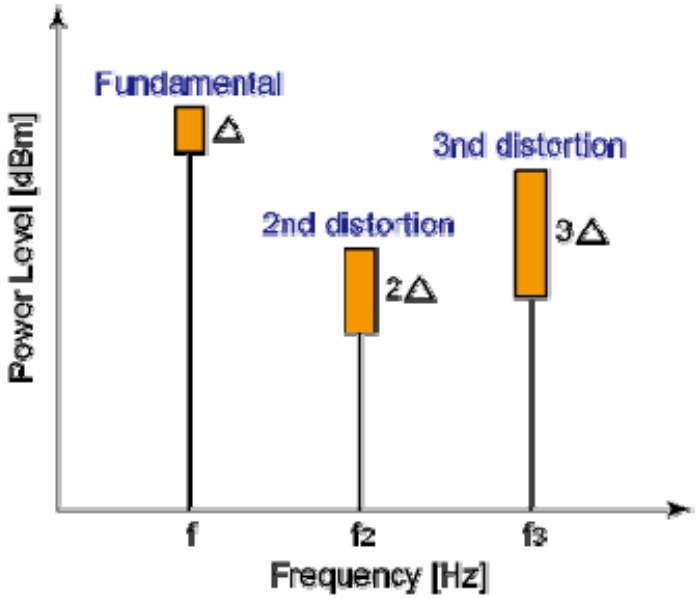
| Softkey | Function |
|---|---|
| **A (n)** | Absolute measurement in Port 1, test receiver |
| **B (n)** | Absolute measurement in Port 2, test receiver |

| | |
|---|---|
| **C (n)** | Absolute measurement in Port 3, test receiver |
| **D (n)** | Absolute measurement in Port 4, test receiver |
| **R1 (n)** | Absolute measurement in Port 1, reference receiver |
| **R2 (n)** | Absolute measurement in Port 2, reference receiver |
| **R3 (n)** | Absolute measurement in Port 3, reference receiver |
| **R4 (n)** | Absolute measurement in Port 4, reference receiver |

n in the parentheses is the stimulus port number.

### 4. Harmonic Distortion Measurement

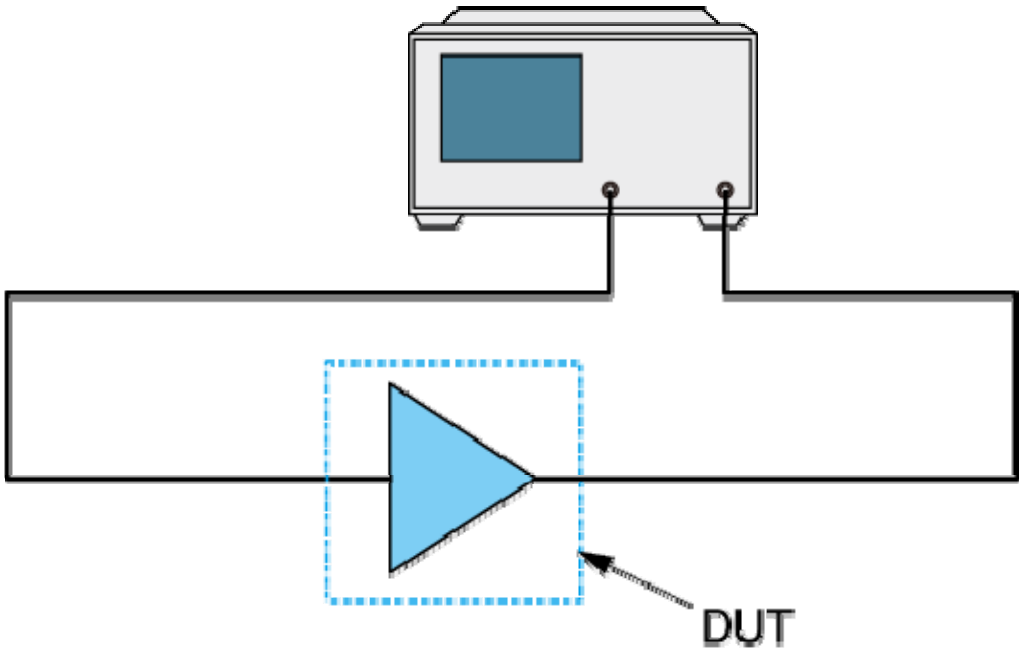
Harmonic distortion is a typical measurement parameter of nonlinear devices such as mixers and amplifiers. As shown in the following figure, the second and third distortions increase by square and cube values of the fundamental signal, respectively. Thus the distortion component increases as a function of the fundamental signal power, and the harmonic distortion can be measured as the proportion of the 2nd and 3rd harmonic power levels to the fundamental signal power level.




e5071c464

#### Connection of DUT

Connect the DUT as shown below.




e5071e344

### Procedures for setting measurement parameters

Follow the steps below to set the measurement parameters. In this case, we measure the harmonic distortion magnitude by using the frequency-offset sweep and the absolute measurement function, based on the E5071C's internal signal source.

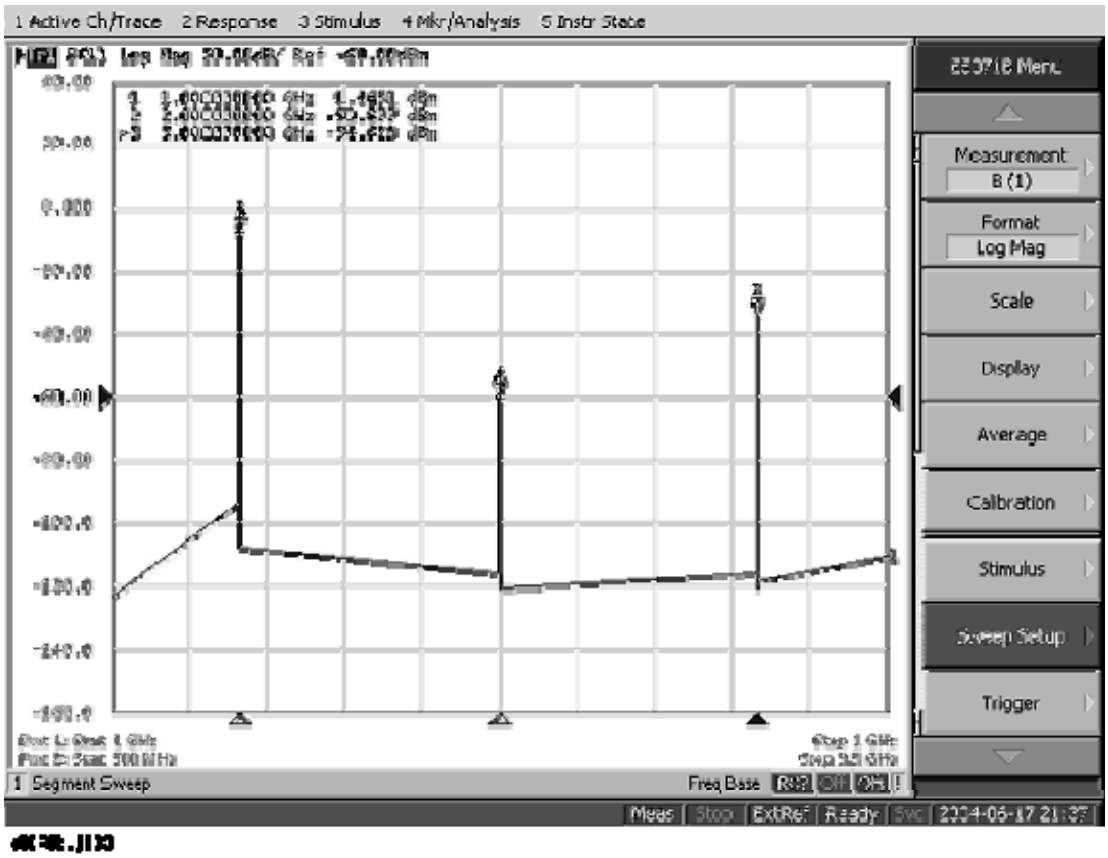
NOTE Receiver calibration is required before starting absolute measurement. For detailed information on receiver calibration, see Receiver Calibration.

1. Press **Channel Next**/**Channel Prev** keys and **Trace Next**/**Trace prev** keys to activate the trace for which you want to implement absolute measurement.
2. Press **Meas** key to display the Measurement Menu.
3. Press **Absolute** to select absolute measurement parameters (**B (1)**). Accordingly, the stimulus port and test port are set to 1 and 2, respectively.
4. In this case, the segment sweep function is used to perform faster measurement. For detailed information on segment sweep, see Performing a Segment-by-Segment Sweep (segment sweep).
5. Press **Sweep Setup**, then click **Edit Segment Table** to display the Edit Segment Table Menu.

6. Set **Frequency Mode** to **Center/Span**. Refer to the following table to input numbers in the segment table, and then press **Return** to return to the Sweep Setup Menu.

| Center | Span | Point |
|---|---|---|
| 500 MHz | 0 Hz | 1 |
| 1 GHz | 1 MHz | 5 |
| 2 GHz | 1 MHz | 5 |
| 3 GHz | 1 MHz | 5 |
| 3.5 GHz | 0 Hz | 1 |

7. Set **Segment Display** to **Freq Base**.
8. Press **Sweep Type** to select **Segment**.
9. Press **Scale** key to use **Auto Scale All** for trace scale optimization.



10. If you use the network analyzer as a tuning receiver like a spectrum analyzer, the setting must be the same at both the frequency point at which the harmonic is generated and the measurement point. Spectrum analyzers can detect all of the signals residing in the range of sweep frequencies, but network analyzers can only detect the signals residing in the measurement frequency point.

NOTE For additional information about the absolute measurements, see Agilent application note 1463-6 Accurate Mixer Measurements Using the Frequency-Offset Mode, 5989-1420EN (Internet Connection is required to open).

## Measurement of Mixers

- Overview
- Procedure

Other topics about Frequency Offset Measurement

### Overview

The frequency-offset function allows you to measure frequency-translating devices such as mixers.

### Procedure

| Item | Description |
|---|---|
| 1. Setting Frequency-Offset | Sets frequency-offset<br>• Procedures for setting frequency at each port<br>• Procedures for enabling frequency-offset function |
| 2. Setting External Signal Source | Sets external signal source (source of LO signal)<br>• Preparations for controlling external signal source<br>• Procedures for setting external signal source GPIB address<br>• Procedures for selecting external signal source<br>• Procedures for setting external signal source frequency<br>• Procedures for setting external signal source power level<br>• Procedures for controlling external signal source |
| 3. Changing Frequency Data | Changes frequency data.<br>• Procedures for changing frequency data |
| 4. Implementing Mixer Calibration | Implements mixer calibration |
| 5. Conversion Loss Measurement | Measures conversion loss of mixer<br>• Connection of measured mixer<br>• Procedures for setting measurement parameters<br>• Procedures for changing measurement data |

### 1. Setting Frequency-Offset

The frequency-offset function allows you to implement the measurements in which frequencies are different for each port, such as in mixer measurement. The frequency set for each port can be applied to the port even if it is for stimulus or receiver.

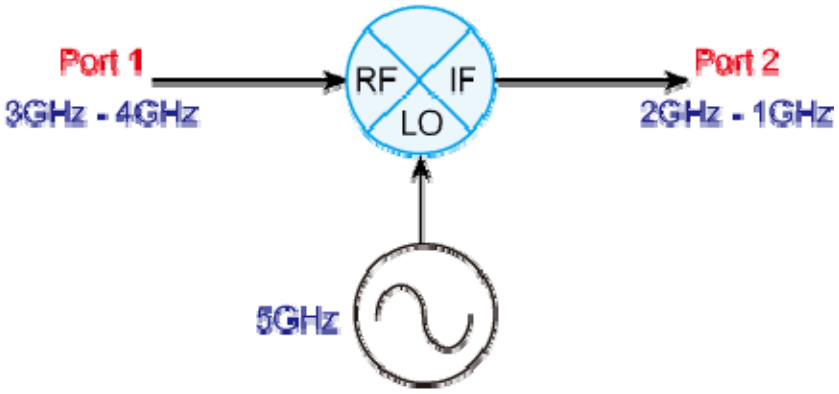
NOTE You can set the frequency-offset for each channel independently.

**Procedures for setting frequency at each port**

The frequencies set at Port-n (n: any port number) should be a multiplier (Mn), divisor (Dn) and offset (On) for the basic frequency setting shown in the following formula.

$$\langle Port\ (n)\ Frequency \rangle = \langle Basic\ Frequency \rangle \times Mn / Dn + On$$

Set Base frequency: 3GHz - 4GHz
Set Port 1 frequency: $M_1$=1, $D_1$=1, $O_1$=0Hz
Set Port 2 frequency: $M_2$=-1, $D_2$=1, $O_2$=5GHz




e5071c461

NOTE The basic frequency is a frequency range set by the **Start** and **Stop** keys in the STIMULUS block.

Follow the steps below to set the frequency at each port.

1. Press **Channel Next**/**Channel Prev** keys to activate the channel on which the frequency-offset is to be set.
2. Press **Sweep Setup** key, then click **Frequency Offset** to display the Frequency Offset Menu.
3. Click **Port n** (n: any port number) to display Port n Menu.

4. Set the frequency for Port-n by using the following softkeys:

| Softkey | Function |
|---|---|
| **Multiplier** | Sets a multiplier for the basic frequency |
| **Divisor** | Sets a divisor for the basic frequency |
| **Offset** | Sets an offset frequency for the basic frequency |
| **Start** | Sets a start frequency of the sweep range for Port-n |
| **Stop** | Sets a stop frequency of the sweep range for Port-n |

NOTE You can use both the **Multiplier**/**Divisor**/**Offset** key and **Start**/**Stop** key to set the frequency for each port. The use of the **Multiplier**/**Divisor**/**Offset** key is recommended as a rule. In this way, the offset value is automatically retained even if you have changed the setting range of the basic frequency, since the correlation of the frequencies among ports is defined as a formula. Using **Start**/**Stop** key will set M and O, which can be determined from the specified frequency and the normal frequency, while maintaining the preset D.

NOTE When you measure the 3rd harmonics over 8.5 GHz with options 2D5, 4D5, 2K5 and 4K5, it is recommended to suppress the carrier signal using filter etc before executing the measurement for accurate results.

NOTE When FOM is used with options 2D5, 4D5, 2K5 and 4K5, the overload & power trip detection function might have to be turned OFF. For more information, see Overload and Power Trip Detection.

### Procedures for enabling frequency-offset function

Follow the steps below to enable the frequency-offset function.

1. Press **Channel Next**/**Channel Prev** keys to activate the channel on which the frequency-offset is to be set.
2. Press **Sweep Setup** key, then click **Frequency Offset** to display Frequency Offset Menu.
3. Click **Frequency Offset** to enable the frequency-offset function (**ON**).
4. Once the frequency-offset function is enabled, the frequencies set at each port are displayed for each measured trace in the lower part of the screen.

e5071c178

Keep in mind the following when you turn **ON** the frequency-offset function:

- If **Swept** is selected for the **Sweep Mode**, it will be changed to **Stepped**.
- The phase information of the measurement value becomes meaningless. Therefore, the view of **Phase**, **Group Delay**, **Smith**, **Polar**, **Real**, **Imaginary**, **Expand Phase** or **Positive Phase** specified with the **Format** key has no actual meaning. In addition, any other function in the **Conversion** menu of the **Analysis** key that cannot be calculated without phase information, including the parameter conversion function, will not operate correctly.
- **Fixture Simulator** and **Time Domain** (**Transform**, **Gating**) are changed to **OFF**.
- If the measured frequency exceeds the measurable range, an error will occur during the measurement.
- The number of sweeps during measurement will increase. For example, you can use two sweeps to measure the S-parameters of a 2-port setup in a normal frequency sweep; however, four sweeps are needed when the frequency-offset function is turned **ON**. This holds true regardless of whether the ports have the same frequency setting.

## 2. Setting External Signal Source

The E5071C allows you to control the external signal source that is connected to USB/GPIB interface.

### Preparations for controlling external signal source

To control the external signal source from the E5071C, you must connect its USB port and the external signal source's GPIB connector via the USB/GPIB interface. Then on the E5071C you must specify the GPIB address and the type of external signal source.

NOTE The settings of the USB/GPIB interface must be made in advance. See Setting the GPIB for more information.

NOTE To use an external signal source, we recommend that you connect the E5071C's internal reference signal output

connector and the external signal source's external reference signal input connector with BNC cable. This ensures stable measurement because the external signal source is phase-locked on the E5071C's frequency reference signal.

**Procedures for setting external signal source GPIB address**

Follow the steps below to set GPIB address of the external signal source.

1. Press **System** key.
2. Click **Misc Setup**> **GPIB Setup** > **Signal Generator Address**> **Address**.

Type the GPIB address of the external signal source you want to use.

**Procedures for selecting external signal source**

The E5071C can use the external signal sources shown below.

| Type | Model |
|---|---|
| 1 | User-defined Commands |
| 2 | 8643A, 8644B, 8664A, 8665A/B |
| 3 | 8648A/B/C/D, ESG Series, PSG Series & MXG Series |

NOTE For type 1, the user-defined commands can be used to control the external signal source. Four commands can be defined: preset, turning on RF output, setting frequency, and setting power. Definitions are required for frequency and power level settings.

NOTE The factory-state definitions are shown below.

| Function | Command | Description |
|---|---|---|
| Preset | "" | Not defined |
| Turning on RF output | "R3" | |
| Frequency setting | "FR %f% HZ" | Frequency (Hz) is set to %f% |
| Power level setting | "AP %p% DM" | Power level (dBm) is set to %p% |

Follow the steps below to select an external signal source.

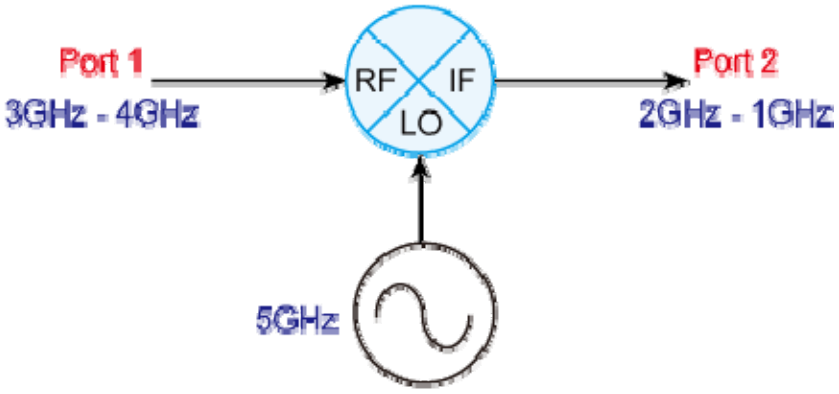
1. Press **System** key.
2. Click **Misc Setup** > **GPIB Setup** > **Signal Generator Address**.
3. Select the external signal source you want to use.
4. **Switching Time** allows you to set the wait time after setting the external signal source's frequency and power in ms.

**Procedures for setting external signal source frequency**

The frequencies set to the external signal source should be a multiplier (MLO), divisor (DLO), and offset (OLO) for the basic frequency setting as shown in the following formula:

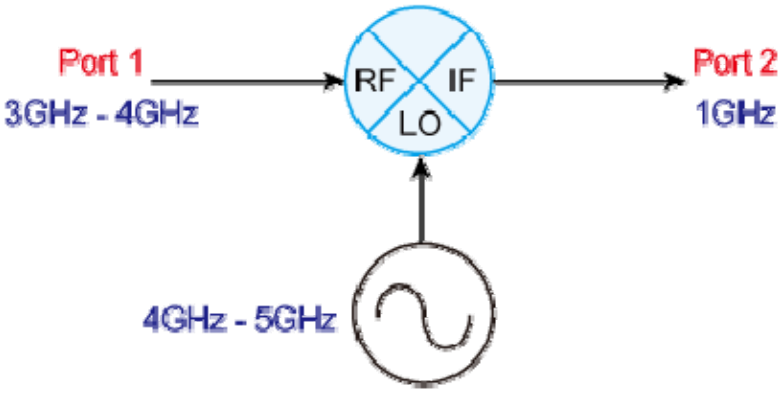
$$<LO\ Frequency> = <Basic\ Frequency> \times MLO / DLO + OLO$$

Set Base frequency: 3GHz - 4GHz
Set Port 1 frequency: $M_1$=1, $D_1$=1, $O_1$=0Hz
Set Port 2 frequency: $M_2$=-1, $D_2$=1, $O_2$=5GHz
Set LO frequency: $M_{LO}$=0, $D_{LO}$=1, $O_{LO}$=5GHz




e5071c459

Set Base frequency: 3GHz - 4GHz
Set Port 1 frequency: $M_1$=1, $D_1$=1, $O_1$=0Hz
Set Port 2 frequency: $M_2$=0, $D_2$=1, $O_2$=1GHz
Set LO frequency: $M_{LO}$=1, $D_{LO}$=1, $O_{LO}$=1GHz




e5071c460

NOTE The basic frequency is a frequency range set by the **Start** and **Stop** keys in the STIMULUS block.

Follow the steps below to set the external signal source frequency.

1. Press **Sweep Setup** key, the click **Frequency Offset** to display the Frequency Offset Menu.
2. Click **External Source** > **Power**.
3. Set a frequency for the external signal source by using the **softkeys**:

| Softkey | Function |
|---|---|
| **Multiplier** | Sets a multiplier for the basic frequency |
| **Divisor** | Sets a divisor for the basic frequency |
| **Offset** | Sets an offset frequency for the basic frequency |
| **Start** | Sets a start frequency of the sweep range for the external signal source |
| **Stop** | Sets a stop frequency of the sweep range for the external signal source |

NOTE You can use both the **Multiplier**/**Divisor**/**Offset** key and the **Start**/**Stop** key to set the external signal source frequency. The use of the **Multiplier**/**Divisor**/**Offset** key is recommended as a rule. This allows you to automatically retain the offset value even if you have changed the setting range of the basic frequency, since the external signal source frequency is defined as a formula. Using the **Start**/**Stop** key will set M and O, which can be determined from the specified frequency and the normal frequency, while maintaining the preset D.

### Procedures for setting external signal source power level

Follow the steps below to set the external signal source power level.

1. Press **Sweep Setup** key, the click **Frequency Offset** to display the Frequency Offset Menu.
2. Click **External Source** > **Power**.
3. Set the power level.

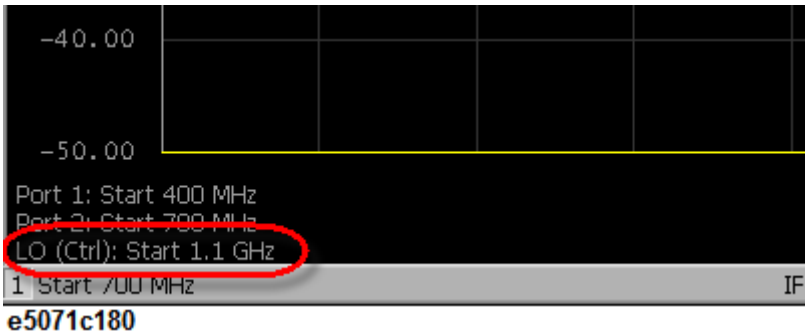
   NOTE The mixer's conversion loss and reflection coefficients are significantly affected by the power level variation of the external signal source (LO signal). The **Slope** provided in the External Source Menu allows you to set a

correction in dB/GHz for the power level variation that may occur when you change the external signal source frequency.

**Procedures for controlling external signal source**

Follow the steps below to control an external signal source.

1. Press **Sweep Setup** key, then click **Frequency Offset** to display the Frequency Offset Menu.
2. Click **External Source** to display the External Source Menu.
3. Click **LO Frequency** to turn it **ON**. The frequency setting of the external signal source (LO) is displayed.
4. Click **Control** to turn the external signal source control **ON**, which allows you to send setting values such as frequency to the external signal source and start the control. If the external signal source control is turned **ON**, (Ctrl) will be displayed on the screen as shown below.




e5071c180

NOTE If the external signal source control fails, the GPIB setting may be wrong. See Setting the GPIB.

### 3. Changing Frequency Data

The frequency-offset function offers the capability to change the frequency data of each trace to any desired frequency.

**Procedures for changing frequency data**

1. Press **Channel Next**/**Channel Prev** keys and **Trace Next**/**Trace prev** keys to activate the trace for which you want to change the frequency.
2. Press **Sweep Setup** key, then click **Frequency Offset** to display the Frequency Offset Menu.
3. Use the **X-Axis** key to set the frequency data.
4. The basic frequency is a frequency range set by the **Start** and **Stop** keys in the STIMULUS bloc.

NOTE If you change the frequency data, it will affect all of the measurement values, such as the marker's read value, conversion parameter, and conversion into time domain.

### 4. Implementing Mixer Calibration

The E5071C provides a vector-mixer calibration function and a scalar calibration function for measuring frequency conversion devices.

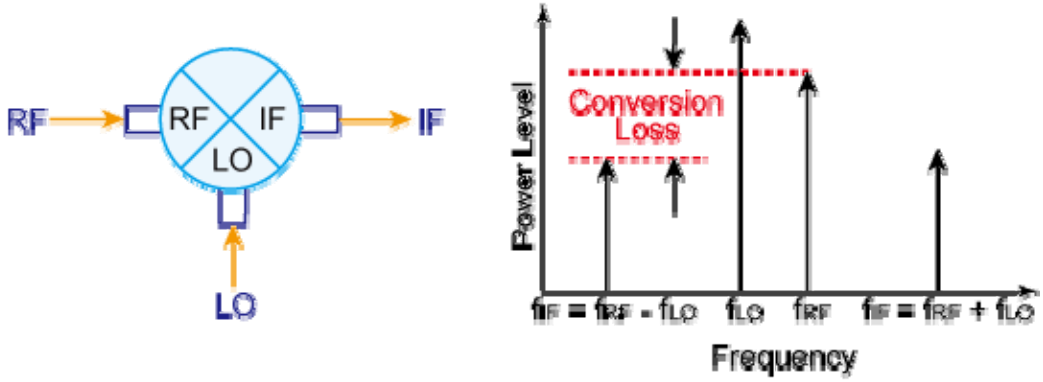
For detailed information on mixer calibration, see Vector-Mixer Calibration and Scalar-Mixer Calibration.

### 5. Conversion Loss Measurement

Conversion loss is a typical measurement parameter of a mixer. As shown below, conversion loss indicates the level of efficiency in which input frequency is converted to another frequency. Furthermore, conversion loss can be defined as a proportion of output frequency power to input frequency power at a given LO signal level. The following description is based on an example of conversion loss measurement by using the vector-mixer calibration.

NOTE You must implement vector-mixer calibration in advance. For detailed information on vector-mixer calibration, see Vector-Mixer Calibration.

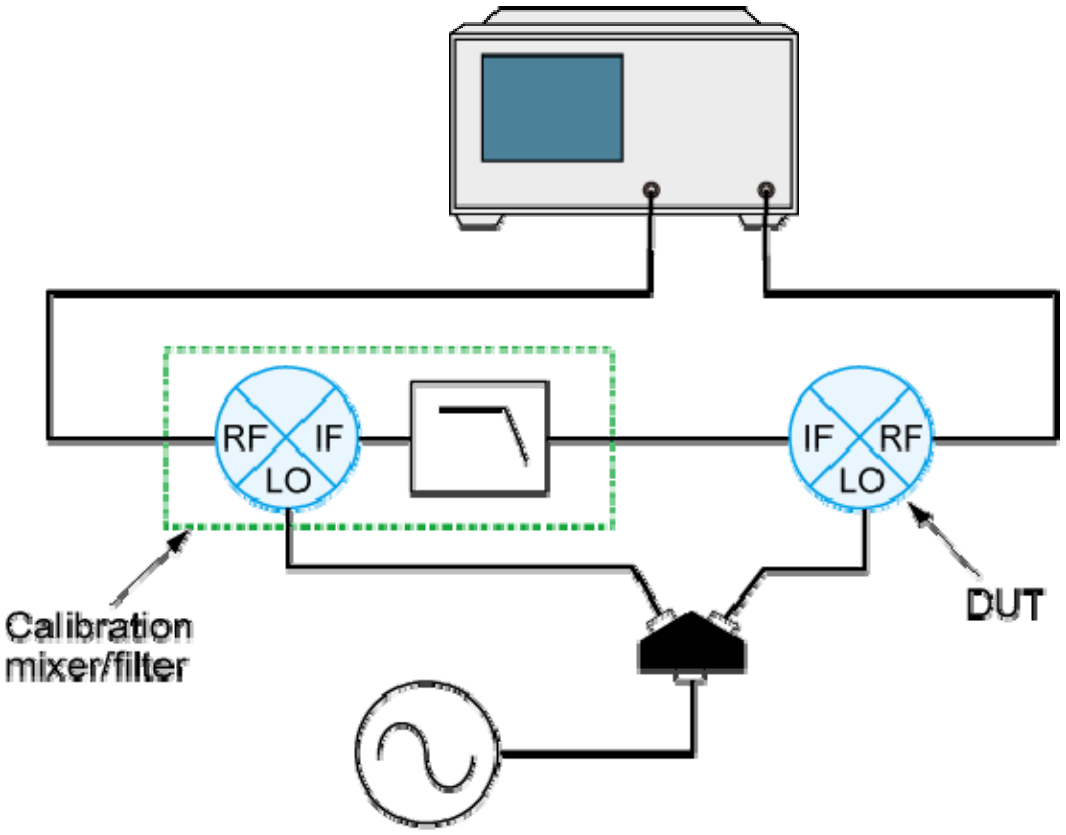
$$\text{Conversion loss [dB]} = 10 \log \frac{\text{Mag (fIF) mW}}{\text{Mag (fRF) mW}}$$




e5071c462

#### Connection of measured mixer

Connect the calibration mixer with the IF filter, measured mixer and external signal source, as shown below.




e5071c343

NOTE If two or more spectrum components are involved at a similar level, it is necessary to use a filter in order to prevent unnecessary spectrum components from entering the receiver port; in this way, you can prevent the total magnitude from reaching a damage level of 6.3 Vp-p.

**Procedure for setting measurement parameters**

Follow the steps below to set the measurement parameters. In this case, we measure each parameter of magnitude, phase, and group delay in conversion loss (S12).

1. Press **Channel Next**/**Channel Prev** keys to activate the channel you want to measure.
2. Press **Display** key to set **Num of Traces** to **3**.
3. Click **Allocate Traces** to select the Graph Layout ( ).
4. Press **Meas** to set the measurement parameter to **S12**.
5. Press **Format** to set **LogMag** (magnitude) to Trace 1.
6. Press **Trace Next** key to select Trace 2.

7. Press **Meas** to set the measurement parameter to **S12**.
8. Press **Format** to set **Phase** (phase) to Trace 2.
9. Press **Trace Next** key to select Trace 3.
10. Press **Meas** key to set the measurement parameter to **S12**.
11. Press **Format** key to set **Group Delay** (group delay) to Trace 3.
12. Press **Scale** key to use **Auto Scale All** for trace scale optimization.

Change the frequency data to the desired frequency as required. For detailed information on changing the frequency data, see Procedures for changing frequency data.

NOTE For additional information about the absolute measurements, see Agilent application note 1463-6 Accurate Mixer Measurements Using the Frequency-Offset Mode, 5989-1420EN (Internet Connection is required to open).

# Measurement Wizard Assistant

## Measurement Wizard Assistant

- Introduction of the MWA Software
- MWA Operational Requirements
- Installation of the MWA Software
- Creating Spec Sheets using the MWA Front-end Application
- Executing the Back-end Application in ENA
- Measurement Example of a Multiport Switch
- Overview and Restrictions of Group

Other topics about Measurement with Options

## Introduction of the MWA Software

- Overview
- Front-end Application
- Back-end Application

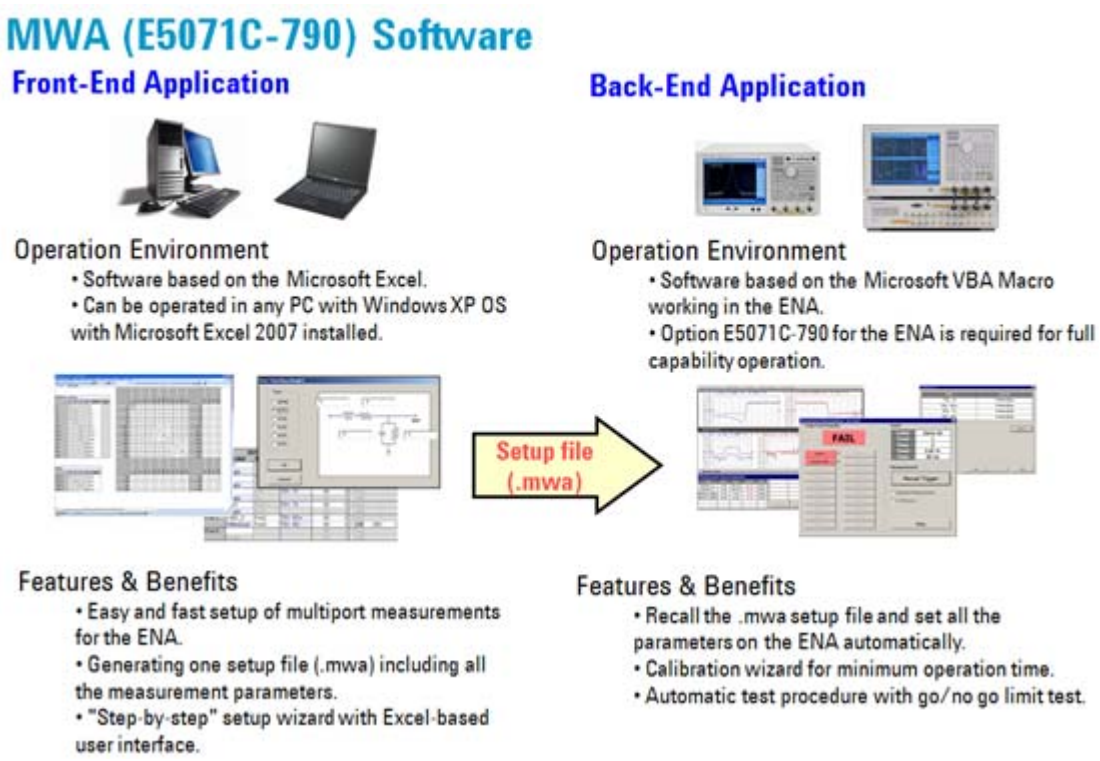
Other topics about Measurement Wizard Assistant

### Overview

The Measurement Wizard Assistant (MWA) is the software option on the ENA that enables easy and fast setup of whole measurement. The software consists of two main applications – front-end and back-end application. The front-end application running on the Microsoft Excel creates a setup file (“spec sheet”) with the .mwa extension which includes all the measurement parameters. A spec sheet is then recalled by the back-end application of the Microsoft VBA program running on the ENA. All the necessary parameters are automatically set up within the ENA by the back-end application. The back-end application also provides the calibration wizard and controls the whole measurement procedure. Operation time of network analysis can dramatically reduced by using the MWA software.

NOTE More information about MWA is available at www.agilent.com/find/mwa and in the Application Note "Measurement Wizard Assistant software for the ENA".

NOTE The Measurement Wizard Assistant (MWA) is only available on the ENA with the option 790. See ENA options for more details.




### Front-end Application

The front-end application is provided as a VBA macro program on Microsoft Excel and can be operated in any PCs with the software installed. The front-end application generates a spec sheet (.mwa) for the ENA and multiport test sets that includes all the necessary parameters of measurement.

The front-end application provides the following functions:

- Defining measurement configuration
- Defining the connection between the DUT and measurement instruments
- Setting commands to external peripherals via GPIB interface
- Setting all measurement parameters in network analysis
- Creating a setup file (“spec sheet”) for the back-end application

### Back-end Application

The back-end application is a VBA macro program that runs on the ENA. A spec sheet created by the front-end application is recalled and all the measurement parameters are automatically set up in the ENA. The back-end application has the calibration wizard function that provides instructions of calibration steps necessary for all measurements. The necessary procedure of total measurements is controlled by the back-end

application and the detailed results of the measurement are saved within the ENA.

The back-end application provides the following functions:

- Recalling a spec sheet to setup all the measurement parameters automatically
- Calibration wizard
- Instruction of connection information
- Controlling external peripherals via GPIB interface
- Customizing the measurement procedure
- Performing limit tests on all measurement paths
- Reporting detailed results of measurement

## MWA Operational Requirements

Requirements for operating the MWA software (Rev 1.1) are as follows:

**A PC for running the front-end application**

- OS: Windows XP
- Microsoft Excel 2007

**The ENA Network Analyzer (E5070B/E5071B/E5071C)**

- The back-end application of the MWA must be installed.
  - The MWA operates only when the number of channels is three or more (four or more for the ENA settings). Set the number of channels of the ENA in advance. Refer to Setting Channels and Traces for channel setup information.

**Multiport Test Set**

If you wish to make multiport measurements, the following multiport test sets are supported by the MWA.

- **E5092A**
  - The option 020 (E5092A-020).
  - Configurations of E5092A 13-port, 16-port, 22-port and X-10-port crossbar are supported by the MWA for the E5092A-020.
- **E5091A**
  - The 9-port option (E5091A-009) and the 16-port option (E5091A-016).
  - Both 13-port & 16 port configuration is supported by the MWA for the E5091A-016.
- Up to two multiport test set can be used. The E5092A & E5091A can be combined with the MWA.

**ECal Module**

When using the ECal module with calibration wizard in the back-end application of the MWA, the 4-port ECal module supported by the ENA is required. Refer to ECal for supported ECal modules by the ENA.

Other topics about Measurement Wizard Assistant

## Installation of the MWA Software



Other topics about Measurement Wizard Assistant

### Overview

The MWA consists of a front-end application, which creates spec sheets on the PC, and a back-end application, which makes measurements with the E5071C. Each of the installation procedures is described below.

### Installing Front-end Application

Follow the procedure below to install the front-end application program of the MWA on the PC:

1. Download MWA software evaluation version from http://www.agilent.com/find/mwa/.
2. Extract the MWA software package.
3. Copy **mwa_xxxx.xls** (**x** in **mwa_xxxx.xls** means revision number. e.g.: **mwa_0106.xls)** to an appropriate folder on the PC.

### Installing Back-end Application

When your E5071C is equipped with option 790, the MWA back-end application has already been installed on the **D:/Agilent/MWA** at the factory shipment. If you retrofit the option, follow the following procedure:

1. Download MWA software evaluation version from http://www.agilent.com/find/mwa/.
2. Extract the MWA software package.
3. Copy **mwa_xxxx.vba** to the **D:/Agilent/MWA** folder on E5071C using an USB memory, external floppy disk drive, or LAN.

   NOTE The application file name (**mwa_xxxx.xls**, **mwa_xxxx.vba**) may change when the version is updated.

   NOTE The back-end application of ENA-B and ENA options 2D5, 4D5, 2K5 and 4K5 model turns OFF the Setting Power Level with Auto Power Range function, to protect the hardware of ENA. In such cases, if the power values set at front-end application are out of range, then the values will be

automatically adjusted by the ENA within the power range limit. If the adjusted power value exceeds the maximum power input of the DUT can cause damage to ENA.

### MWA License Validation and Key Code Entry Procedure

The MWA validates license and a key code on the initial start-up. Follow the procedure below to validate license and enter a key code.

1. Start the back-end application according to Starting MWA (Back-end Application).
2. When starting the MWA for the first time, the license validation dialog appears. Read the contents and press the **AGREE** button to validate license. If you press the **DO NOT AGREE** button, the back-end application does not start and the measurement screen of the E5071C appears.
3. The message window named "Do you have a key code?" dialog box appears. Press the **Yes** button to enter a key code. When you press the **NO** button, the back-end application starts in the evaluation mode.

   NOTE When incorrect MWA license key code is entered, the E5071C firmware initiates a forced restart.
4. The key code entry dialog appears. Enter the key code specified in the certificate that is provided when you order the E5005A, and then press **OK**. If you press the **Cancel** button or press the **OK** button without entering a key code, the back-end application starts in the evaluation mode.

   NOTE Restart E5071C after entering the key code.
5. The initial screen of the MWA's back-end application appears.
6. If the hard disk is initialized (when repaired, etc), you need to enter the key code again.

### Evaluation Mode

The evaluation mode is the mode in which the back-end application can be used without entering the key code and has the following restrictions.

- Manual and External Trigger: Up to 100 measurements.
- Continuous Trigger: Up to 500 measurements.

When the MWA starts in the evaluation mode, the dialog shown below appears before the initial screen of the back-end application is displayed. This evaluation mode enables the evaluation of the MWA without purchasing MWA. MWA can be retrofitted by the E5005A.


Evaluation Mode
MWA requires a key code.
The MWA can be used in evaluation mode without a key code.
The evaluation mode has the following restrictions.
Manual, External Trigger : Max 100 measurements
Continuous Trigger : Max 500 measurements
To obtain a key code, please contact Agilent and provide the following model name and serial number.
Model Name
E5071C
Serial Number
MY00000001
OK

e5071c252

## Creating Spec Sheet using the MWA Front-end Application



Other topics about Measurement Wizard Assistant

### Overview

The front-end application of the MWA creates a setup file named "spec sheet" (.mwa) which is used in the back-end application on the ENA. The procedure of creating a spec sheet is described below:

### Starting MWA (Front-end Application)

1. Start Excel and select **File** > **Open**.
2. In the window that opens, select **mwa_xxxx.xls** and click **Open**.
3. When you are asked whether to enable macros, select the option to enable them.

NOTE When opening the front-end application of MWA 1.1 in Microsoft Office Excel 2007, the macros of the front-end are disabled by default as indicated by "Security Warning" tab. In order to use the front-end application, its macros must be enabled. To enable the macros, click "**option**" button on "**Security Warning**" tab and select either "**Enable this content**" or "**Trust all documents from this publisher**" on the pop-up "Security Alert – Macros & ActiveX" window.

### Selecting Test Set

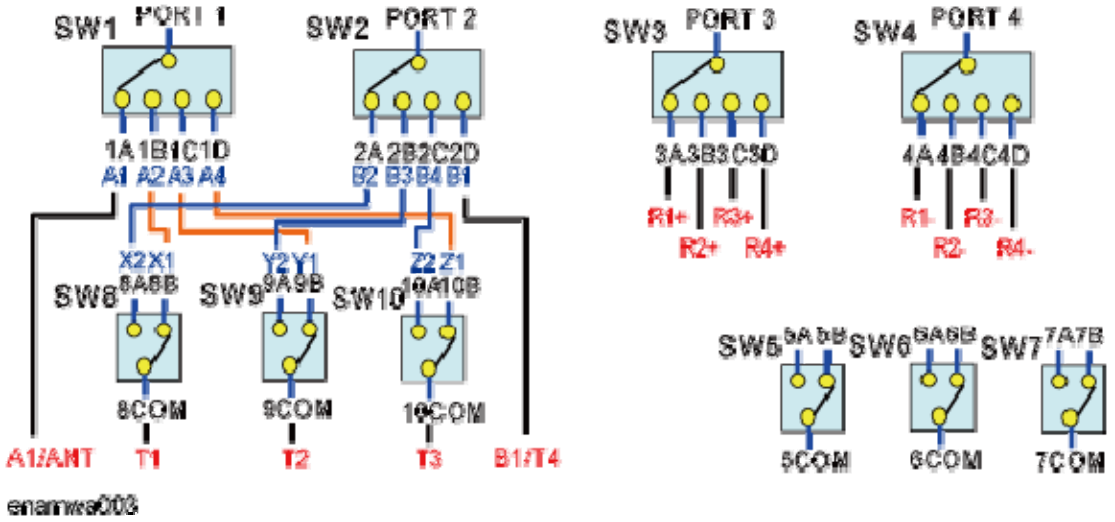
The procedure for selecting the configuration of measurement in the "Multiport Test Set" sheet is described below:

1. Select the "Multiport Test Set" tab ([1] in Multiport Test Set Sheet)
2. Double-click the selection cell for test set 1 ([2] in Multiport Test Set Sheet) and select the first multiport test set in the drop-down list.

The following configurations are available with the MWA:

| No. | Test Sets | Description |
|---|---|---|
| 0 | None | Uses the 4-port ENA only. |
| 1 | E5091A 9-Port | Uses the 9-port option of the E5091A (E5091A-009). |
| 2 | E5091A 13-Port | Uses the 16-port option of the E5091A (E5091A-016) with the 13-port configuration. |
| 3 | E5091A 16-Port | Uses the 16-port option of the E5091A (E5091A-016) with the 16-port configuration. |
| 4 | E5092A 13-Port | Uses the 13-port configuration of the E5092A option 020 (E5092A-020). |
| 5 | E5092A 16-Port | Uses the 16-port configuration of the E5092A option 020 (E5092A-020). |
| 6 | E5092A 22-Port | Uses the 22-port configuration of the E5092A option 020 (E5092A-020). |
| 7 | E5092A X10-Port | Uses the 10-port full crossbar configuration of the E5092A option 020 (E5092A-020). |
| 8 | Z5623AK64 | This is a test set designed for the PNA-L (N5230A option 245). It cannot be used with the ENA. |
| 9 | Z5623AK66 | This is a test set designed for the PNA-L (N5230A option 245). It cannot be used with the ENA. |

### *E5092A 13-Port Switching Configuration*



### *E5092A 13-Port Configuration Parameter Matrix & Front Panel Connection*

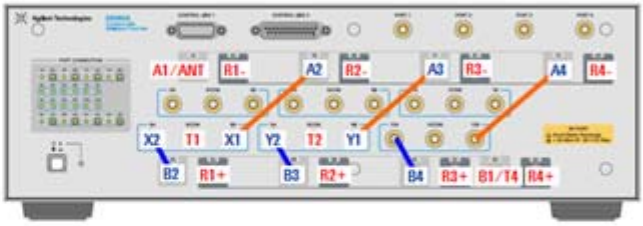
•Parameter matrix

| Output Port \ Input Port | A1:ANT | T1 | T2 | T3 | B1:T4 | R1+ | R2+ | R3+ | R4+ | R1- | R2- | R3- | R4- |
|---|---|---|---|---|---|---|---|---|---|---|---|---|---|
| A1:ANT | ○ | ○ | ○ | ○ | ○ | ○ | ○ | ○ | ○ | ○ | ○ | ○ | ○ |
| T1 | ○ | ○ | ○ | ○ | ○ | ○ | ○ | ○ | ○ | ○ | ○ | ○ | ○ |
| T2 | ○ | ○ | ○ | ○ | ○ | ○ | ○ | ○ | ○ | ○ | ○ | ○ | ○ |
| T3 | ○ | ○ | ○ | ○ | ○ | ○ | ○ | ○ | ○ | ○ | ○ | ○ | ○ |
| B1:T4 | ○ | ○ | ○ | ○ | ○ | ○ | ○ | ○ | ○ | ○ | ○ | ○ | ○ |
| R1+ | ○ | ○ | ○ | ○ | ○ | ○ | | | | ○ | ○ | ○ | ○ |
| R2+ | ○ | ○ | ○ | ○ | ○ | | ○ | | | ○ | ○ | ○ | ○ |
| R3+ | ○ | ○ | ○ | ○ | ○ | | | ○ | | ○ | ○ | ○ | ○ |
| R4+ | ○ | ○ | ○ | ○ | ○ | | | | ○ | ○ | ○ | ○ | ○ |
| R1- | ○ | ○ | ○ | ○ | ○ | ○ | ○ | ○ | ○ | ○ | | | |
| R2- | ○ | ○ | ○ | ○ | ○ | ○ | ○ | ○ | ○ | | ○ | | |
| R3- | ○ | ○ | ○ | ○ | ○ | ○ | ○ | ○ | ○ | | | ○ | |
| R4- | ○ | ○ | ○ | ○ | ○ | ○ | ○ | ○ | ○ | | | | ○ |

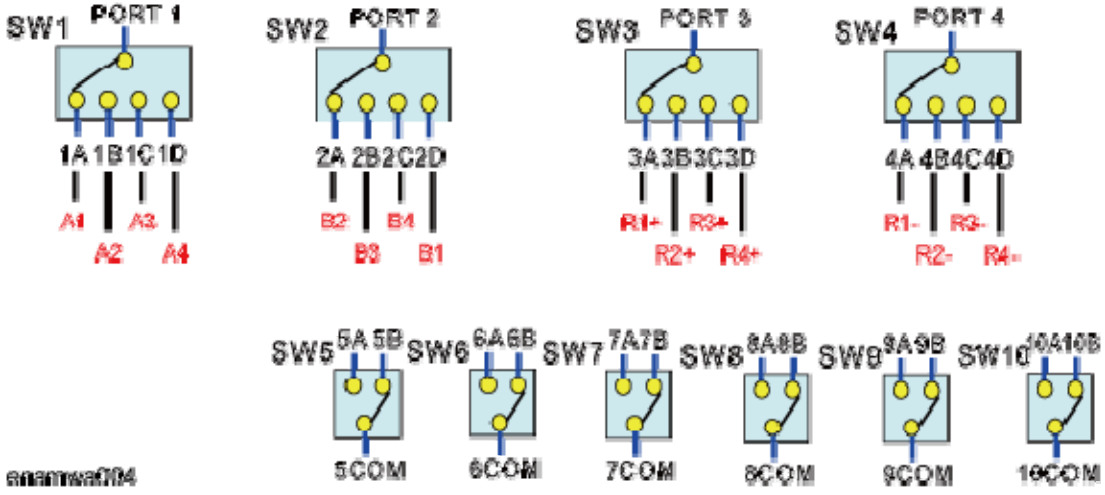
○ : Measurement Possible

enamwa012

•Connection on front panel



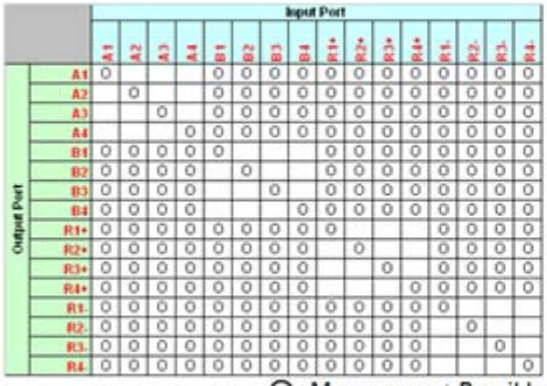

### *E5092A 16-Port Configuration*




### *E5092A 16-Port Configuration Parameter Matrix & Front Panel Connection*

•Parameter matrix

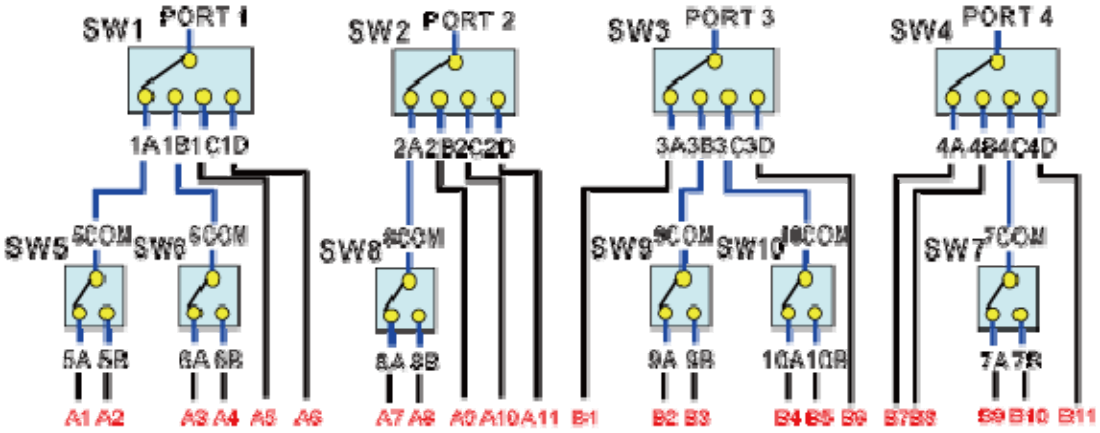
•Connection on front panel





enamwa013

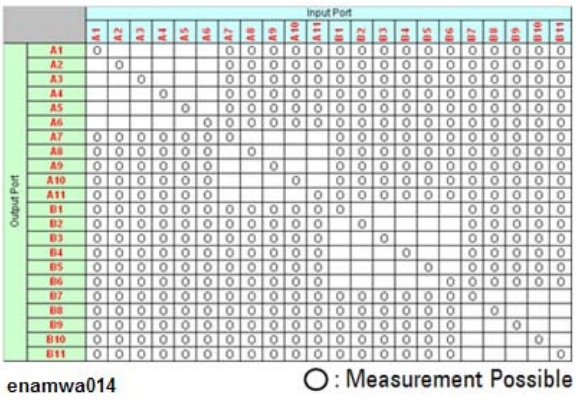
### *E5092A 22-Port Configuration*



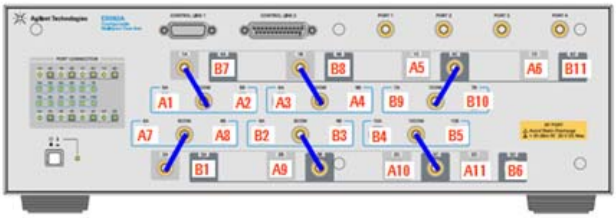

enamwa005

### *E5092A 22-Port Configuration Parameter Matrix & Front Panel Connection*

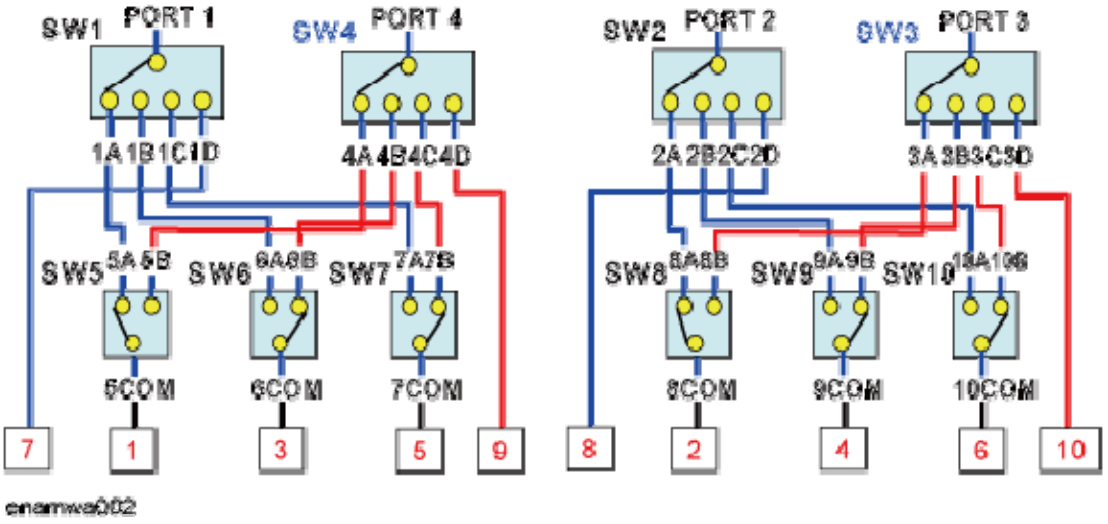
•Parameter matrix




•Connection on front panel



***E5092A 10-Port Full Crossbar Configuration***




***E5092A 10-Port Full Crossbar Configuration Parameter Matrix & Front Panel Connection***

•Parameter matrix

| Output port \ Input Port | 1 | 2 | 3 | 4 | 5 | 6 | 7 | 8 | 9 | 10 |
|---|---|---|---|---|---|---|---|---|---|---|
| 1 | ○ | ○ | ○ | ○ | ○ | ○ | ○ | ○ | ○ | ○ |
| 2 | ○ | ○ | ○ | ○ | ○ | ○ | ○ | ○ | ○ | ○ |
| 3 | ○ | ○ | ○ | ○ | ○ | ○ | ○ | ○ | ○ | ○ |
| 4 | ○ | ○ | ○ | ○ | ○ | ○ | ○ | ○ | ○ | ○ |
| 5 | ○ | ○ | ○ | ○ | ○ | ○ | ○ | ○ | ○ | ○ |
| 6 | ○ | ○ | ○ | ○ | ○ | ○ | ○ | ○ | ○ | ○ |
| 7 | ○ | ○ | ○ | ○ | ○ | ○ | ○ | ○ | ○ | ○ |
| 8 | ○ | ○ | ○ | ○ | ○ | ○ | ○ | ○ | ○ | ○ |
| 9 | ○ | ○ | ○ | ○ | ○ | ○ | ○ | ○ | ○ | ○ |
| 10 | ○ | ○ | ○ | ○ | ○ | ○ | ○ | ○ | ○ | ○ |

○: Measurement Possible

enamwa011

•Connection on front panel


3. Type the label names of ports in a cell of the Label column for test set 1 ((3) in Multiport Test Set Sheet). The label names typed here will be used in the following procedure of the front-end and the back-end application.

4. The functionality to Copy & Paste is not available in the front-end application of the MWA. Make sure to type characters and values or select the option from the drop-down list.

4. Double-click the selection cell for test set 2 ((4) in Multiport Test Set Sheet) and select another multiport test set in the drop-down list. Following configurations (following table) are available for the second test set:

| No. | Test Sets | Description |
|---|---|---|
| 0 | None | Does not use a second test set. |
| 1 | E5091A 9-Port | Uses the 9-port option of the E5091A (E5091A-009). |
| 2 | E5091A 13-Port | Uses the 16-port option of the E5091A (E5091A-016) with the 13-port configuration. |
| 3 | E5091A 16-Port | Uses the 16-port option of the E5091A (E5091A-016) with the 16-port configuration. |
| 4 | E5092A 13-Port | Uses the 13-port configuration of the E5092A (E5092A-020). |
| 5 | E5092A 16-Port | Uses the 16-port configuration of the E5092A (E5092A-020). |

| | | |
|---|---|---|
| 6 | E5092A 22-Port | Uses the 22-port configuration of the E5092A (E5092A-020). |
| 7 | E5092A X10-Port | Uses the 10-port full crossbar configuration of the E5092A (E5092A-020). |

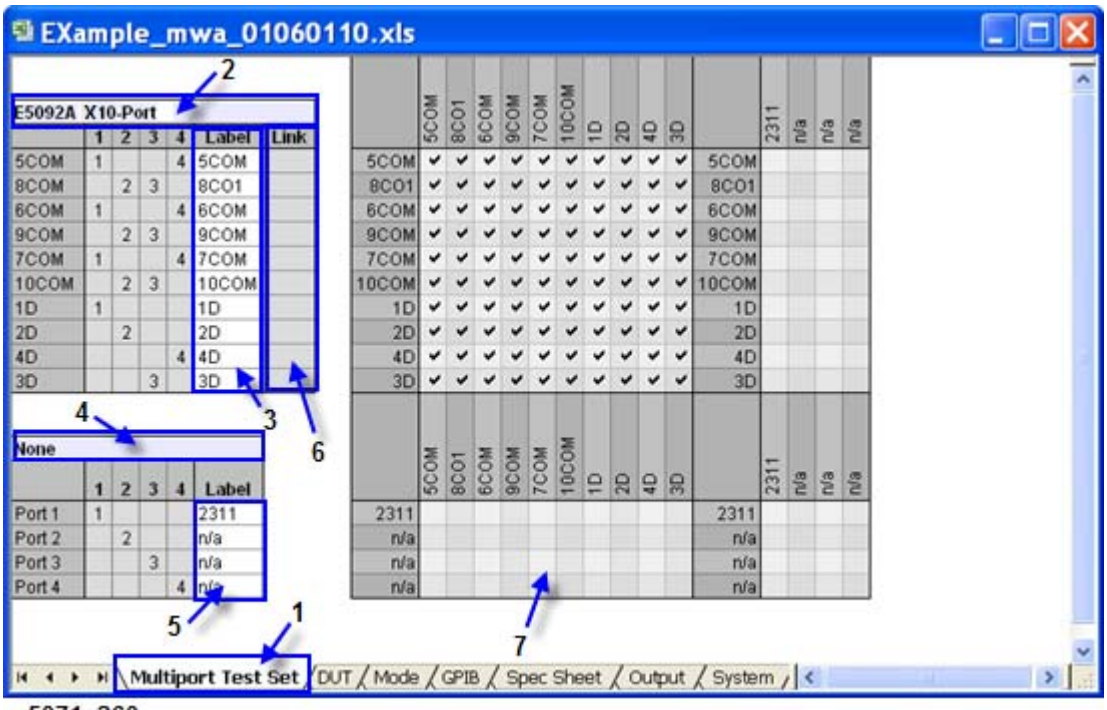
NOTE Step 4 through Step 6 is required when two multiport test sets are used for measurement. When only one test set is used, select None and proceed to Step 7.

5. Type the label names of the ports in cells in the Label column for the test set 2 ((5) in Multiport Test Set Sheet).

6. Set the number 1 to 4 in the cells in the Link column ((6) in Multiport Test Set Sheet), which correspond to the ports 1-4 of the second test set.

7. When the selection of measurement configuration is completed, the connectivity matrix in the right ([7] in Multiport Test Set Sheet) is automatically updated. Those checked in the table are the paths that can be measured.

***Multiport Test Set Sheet***




e5071c260

## Defining DUT and Connecting to a Test Set

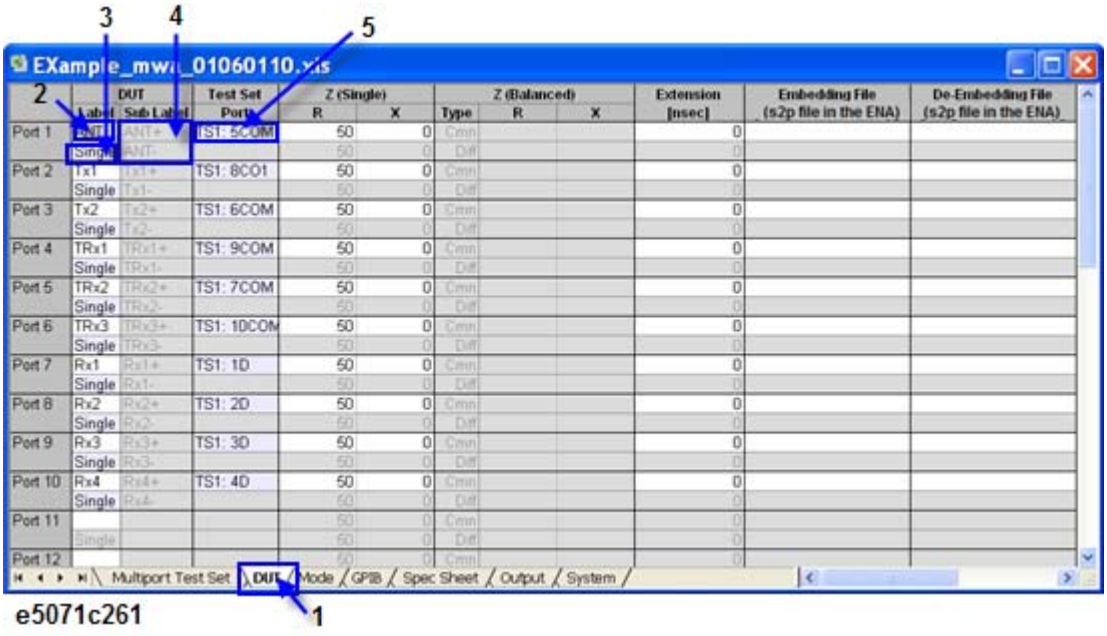
The procedure for defining the characteristics of the DUT and the connection with the ports on the ENA or the multiport test set are explained below.

1. Select the "DUT" tab ([1] in DUT Sheet (1/3)).
2. Type a label name in a cell in the DUT Label column ((2) in DUT Sheet (1/3)) that defines the connection between the DUT and a test set. Typing the label name in the cell enables the corresponding port of the ENA or the multiport test set.
3. Double-click the port type selection cell ((3) in DUT Sheet (1/3)) below the DUT Label and select a port type from the drop-down list or directly input the port type number (1-3). The following types are available:

| No. | Port Types | Description |
|---|---|---|
| 1 | Single | Sets the port type to single-ended port. |
| 2 | Common | Sets the port type to common-mode port. |
| 3 | Differential | Sets the port type to differential-mode port. |

4. If you select Common or Differential as the port type, type into a cell in the Sub Label column ((4) in DUT Sheet (1/3)) a label name that will be used when the ports are connected to the single-ended ports on the ENA or the test set.
5. Double-click the port selection cell ((5) in DUT Sheet (1/3)) and select a port of the test set from the drop-down list. Selection can be made from the ports that have been pre-defined in the Multiport Test Set sheet.

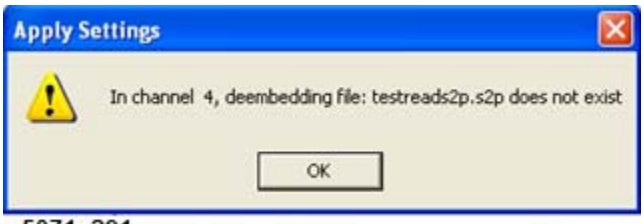
***DUT Sheet (1/3)***



6. When you set the port type to Single, enter port impedance in cells in the R (for the real part) and X (for the imaginary part) columns ([1] in DUT Sheet (2/3)) under Z (Single). When you set the port type to Common or Differential, enter impedance in cells of the R and X columns ((2) in DUT Sheet (2/3)) under Z (Balanced).
7. Enter into a cell in the Extension column ((3) in DUT Sheet (2/3)) an extension time that is applied when the port is extended. If you set the port type to something other than Single, enter an extension time for each of the two ports.

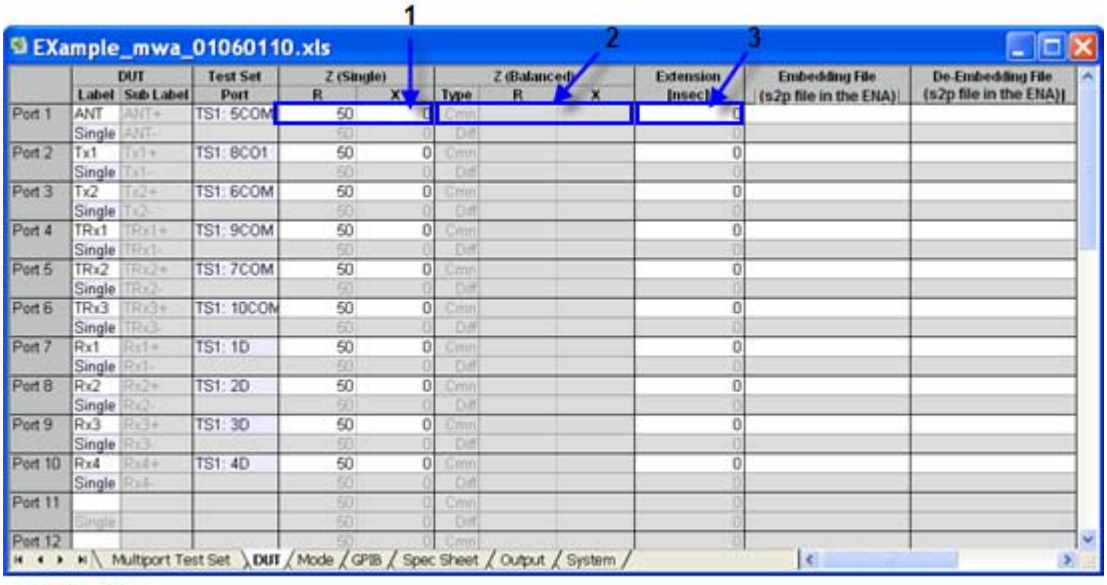
   NOTE From MWA rev.1.1, embedding and de-embedding functions are available in front end application.
   Embedding and Port Matching cannot be used at the same time because of the ENA firmware specification.

***Error Message Screen***




e5071c291

***DUT Sheet (2/3)***




e5071c262

8. Double-click a cell in the Type column under Port Matching (Single) ((1) in DUT Sheet (3/3)) to display a dialog box for setting the port matching circuit (Dialog box for setting the port matching circuit). Select a type, enter values for C, G, L and R and click **OK** ((1) in

Dialog box for setting the port matching circuit). To return to the DUT sheet without setting the type and the values, click **Cancel** ((2) in Dialog box for setting the port matching circuit).

The units of C, G, L and R are F (farad), S (siemens), H (henry) and ohm, respectively. An exponent is acceptable as an input format.

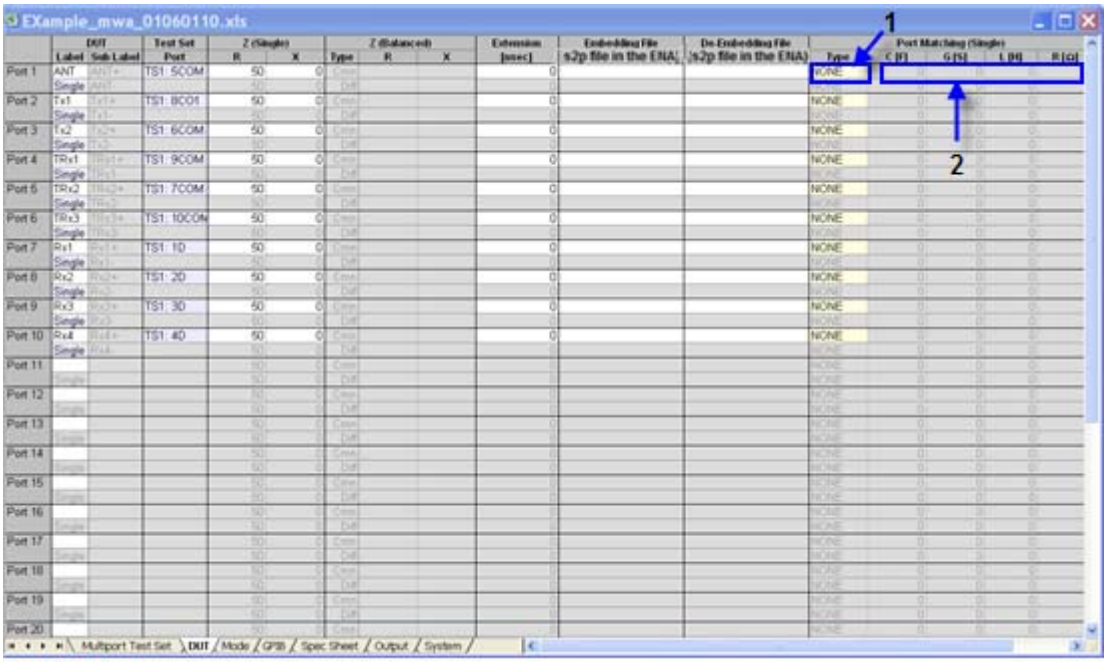
The types of the port matching circuits are as follows:

| Types of Port Matching Circuits | Description |
|---|---|
| None | No matching circuit is selected. |
| SLPC | Selects the circuit that consists of series L and shunt C. |
| PCSL | Selects the circuit that consists of shunt C and series L. |
| PLSC | Selects the circuit that consists of shunt L and series C. |
| SCPL | Selects the circuit that consists of series C and shunt L. |
| PLPC | Selects the circuit that consists of shunt L and shunt C. |

NOTE Values can be entered directly in cells in the C, G, L and R columns ((2) in DUT Sheet (3/3)) instead of using the dialog box.

1. Repeat Step 2 through Step 8 as many times as needed to set all the required parameters.

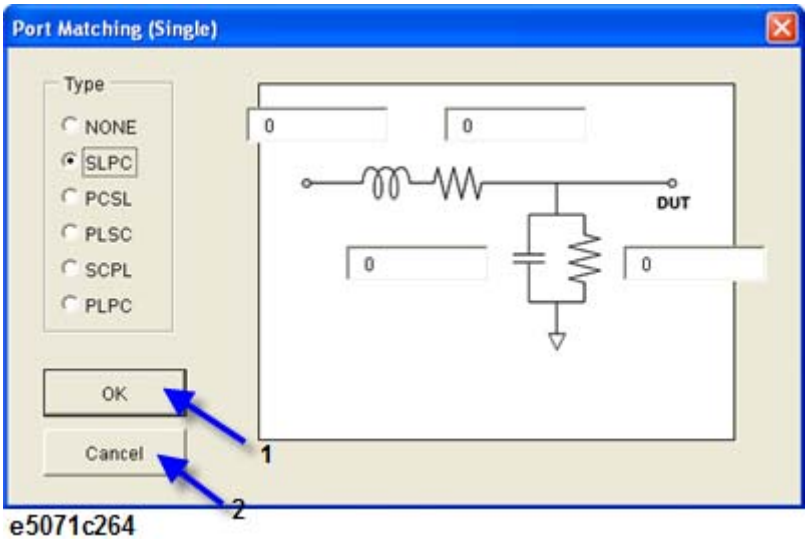
***DUT Sheet (3/3)***




e5071c263

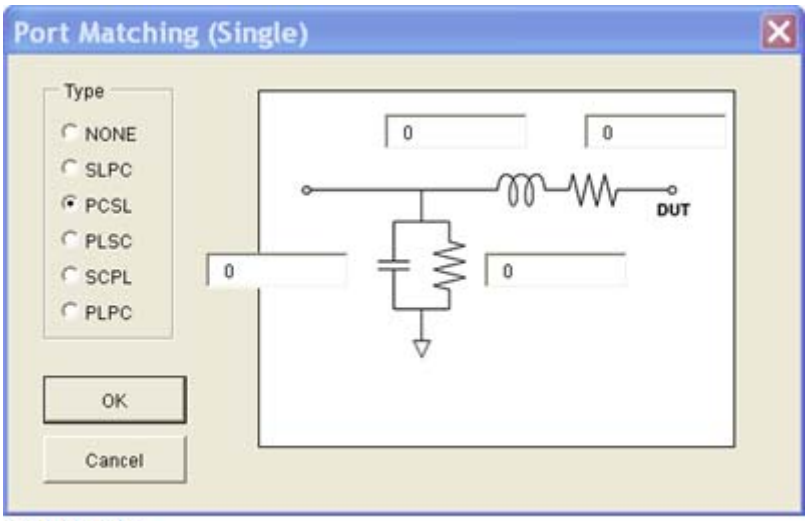
***Dialog box for setting the port matching circuit***

***SLPC***



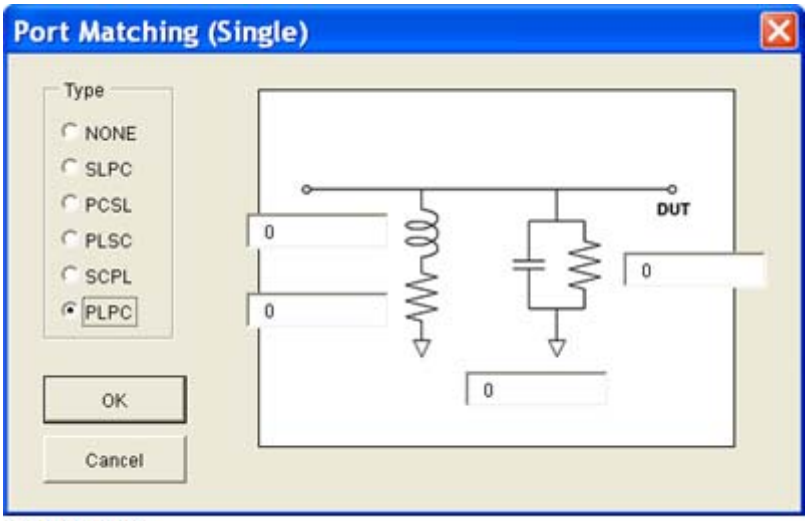

e5071c264

***PCSL***



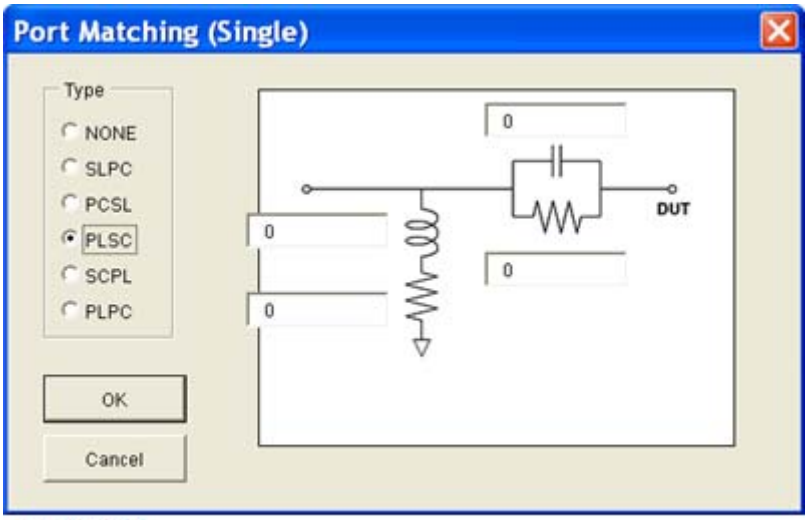

e5071c287

***PLPC***



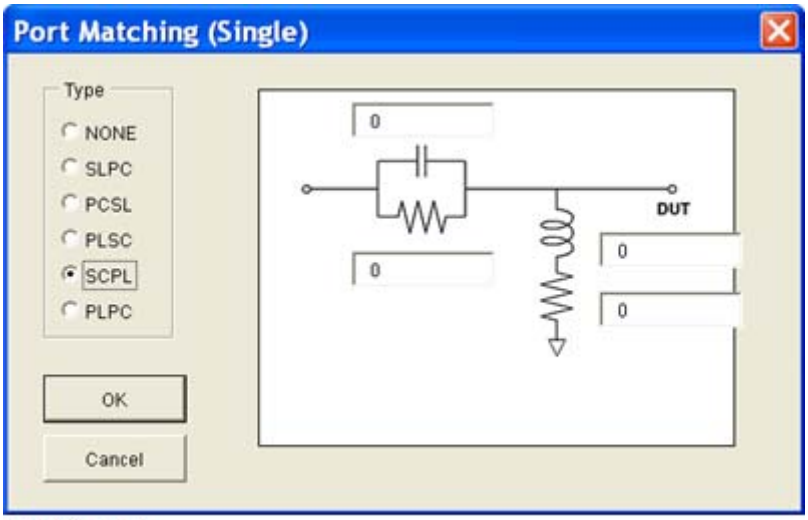

e5071c288

***PLSC***




e5071c289

***SCPL***




e5071c290

### Setting Operation Mode of the DUT

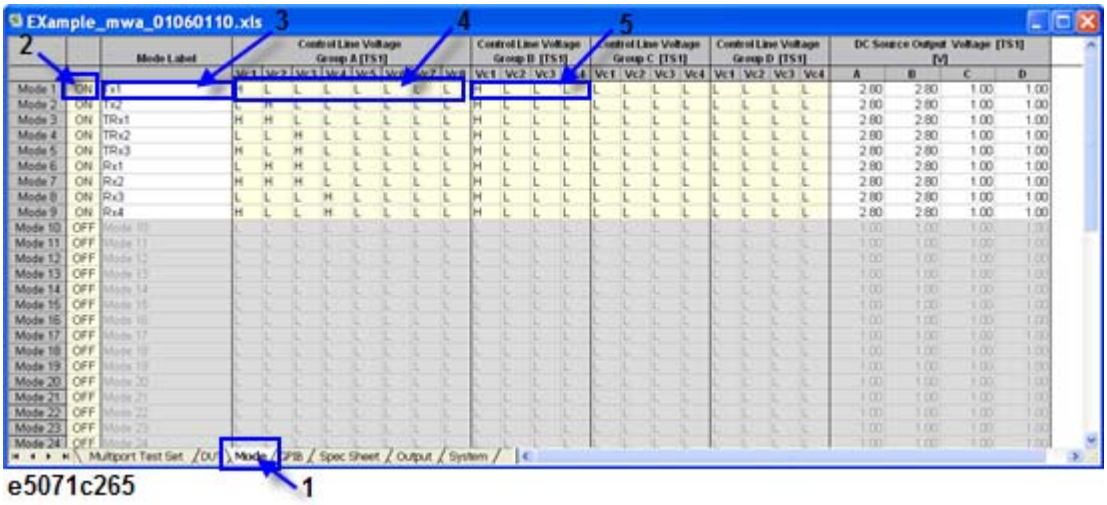
The E5092A multiport test set has the capability of DC source that can supply the bits of DC control voltage to a DUT. By selecting L (Low) or H (High) voltage for each control bit, the operation mode of the DUT can be determined. The procedure for setting operation mode in the "Mode" sheet is explained below:

1. Select the "Mode" tab ((1) in Mode Sheet).
2. Turn ON each cell for the status of operation mode of the DUT ((2) in Mode Sheet).
3. Enter a preferred label name in a cell in the Mode Label column ((3) in Mode Sheet). The label name entered here will be used as a tag

name of a generated spec sheet. It will also be used by the back-end application as a label.

4. Double-click cells for control voltage (Vc1 through Vc8) under Control Line Voltage [TS1] of the first test set ((4) in Mode Sheet) and set L (Low) or H (High) voltage for each control line. It is also possible to directly input the number 0 or 1 (0 for Low, 1for High).
5. When you use two test sets, you can set control voltage outputs from the second test set. Set cells for Vc1 through Vc8 under Line Voltage [TS2] of the second test set ((5) in Mode Sheet) the same way as in Step 4.
6. Repeat Step 2 through Step 5 as many times as needed to set all desired modes.

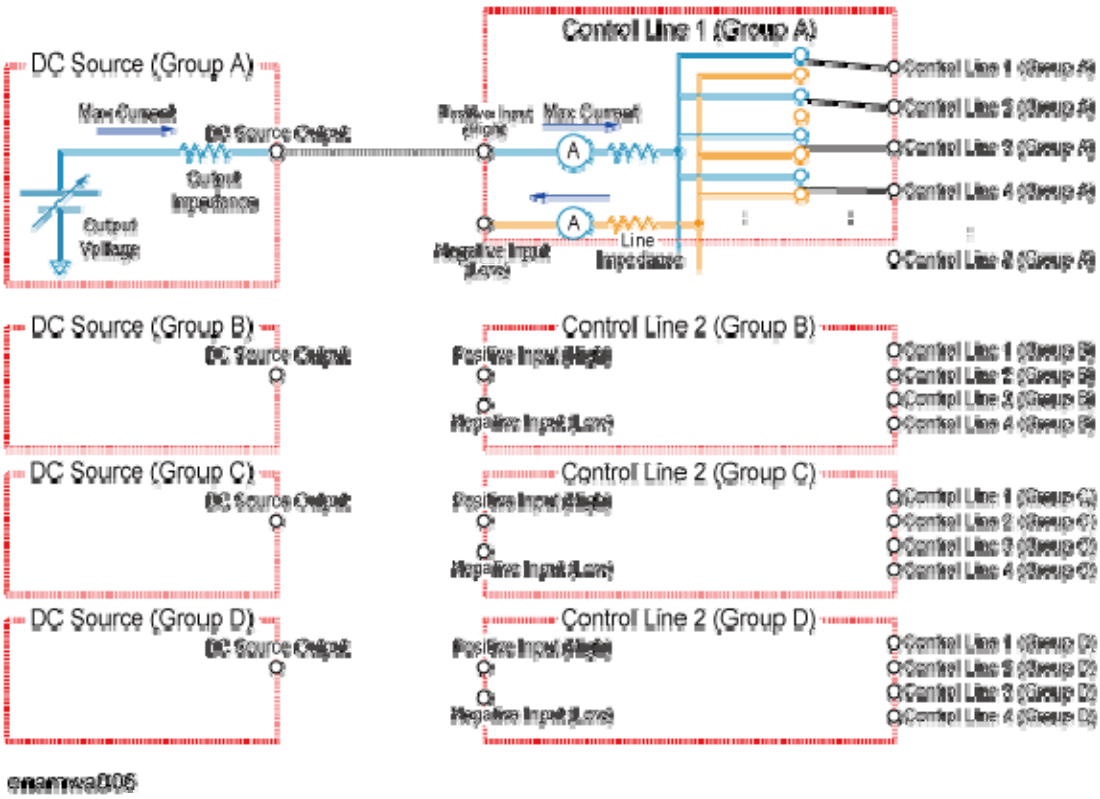
***Mode Sheet***




NOTE Control Line Voltage Group (A to D) and DC Source Output Voltage options are available only when E5092A test set has been selected in the Multiport test set tab. The DC source output voltage range varies between 1V-5V with a resolution of 0.01V depending upon the specification of E5092A.

NOTE The maximum number of modes is 40.

***DC Source and Control Line Voltage Diagram***




### Setting GPIB Commands in Each Mode

The procedure explained below is for setting GPIB commands to external peripherals in the "GPIB" sheet. A USB/GPIB interface (i.e. Agilent 87357B) to connect the ENA and peripherals with the GPIB interface is necessary for this function.

1. Select the "GPIB" tab ((1) in GPIB Sheet).
2. Under GPIB Command, enter into a cell in the Address column ((3) in GPIB Sheet) the GPIB address of the external peripherals to which GPIB commands will be sent.

   NOTE To add a new command row to a mode, double-click the cell in the Mode column((2) in GPIB Sheet). To delete a command row from a mode, place the cursor at the row that you want to delete, right-click to display a list, and select Delete Command.

   To add multiple rows at one time, place the cursor at the row of the command to which you want to add the new rows, right-click to display the Command Count dialog and set the number of rows as needed in the particular mode.

   NOTE GPIB commands can be sent to the ENA itself by entering -1 into a cell in the Address column.

3. Enter into a cell in the Command column ((4) in GPIB Sheet) a GPIB command to be sent.

4. Enter the wait time into a cell in the Wait [ms] column ((5) in GPIB Sheet) under GPIB Command. This wait time (in msec) should be set to insert the data transaction time between the ENA and a peripheral in the testing system, when a query command is sent to the peripheral during the measurement.

5. Double-click a cell in the Type column ((6) GPIB Sheet) to display a drop-down list and select the desired command type. The following options are available:

| No. | Command Type | Description |
|---|---|---|
| 1 | Limit | Set this where a query command is sent via GPIB. The returned value is used in the limit test by the back-end application. |
| 2 | Pre | Set this where a GPIB command is sent before measurement starts in the back-end application. |
| 3 | Post | Set this where a GPIB command is sent after measurement ends in the back-end application. |

NOTE The command type can be set only for GPIB commands that have been set in Setting Operation Mode of the DUT. GPIB command types cannot be set in INIT mode.

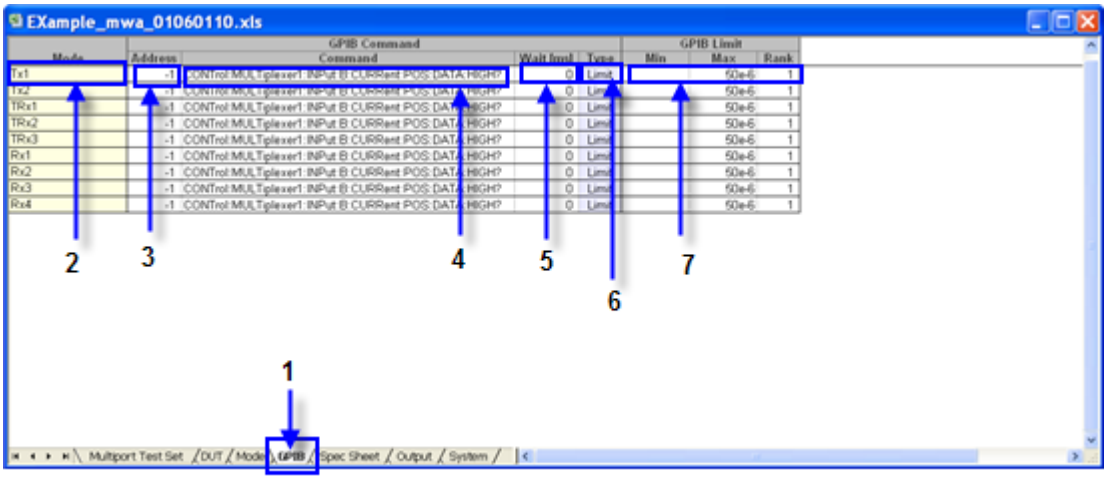
NOTE In INIT mode, a command is executed only once when spec sheets are read by the back-end application. Since this execution of the commands takes place before measurement, set such commands to initialize the state of an external peripheral.

6. When Limit is selected for the command type, enter the lower limit value, the upper limit value and a rank number in cells in the Min, Max and Rank columns ((7) in GPIB Sheet), respectively. These figures are used in the limit test by the back-end application on a returned value of a query command to the peripherals. For details on the rank, refer to Step 11 in Setting Measurement Parameters in Each Mode.

NOTE The lower limit values, the upper limit values, and the ranks can be set in cells under GPIB Limit only if the selected GPIB command type is Limit. This does not apply to Pre or Post GPIB commands. This setting cannot be performed for GPIB commands in the INIT mode either.

7. Repeat Step 2 through 6 as many times as needed to set GPIB commands in each mode.

***GPIB Sheet***



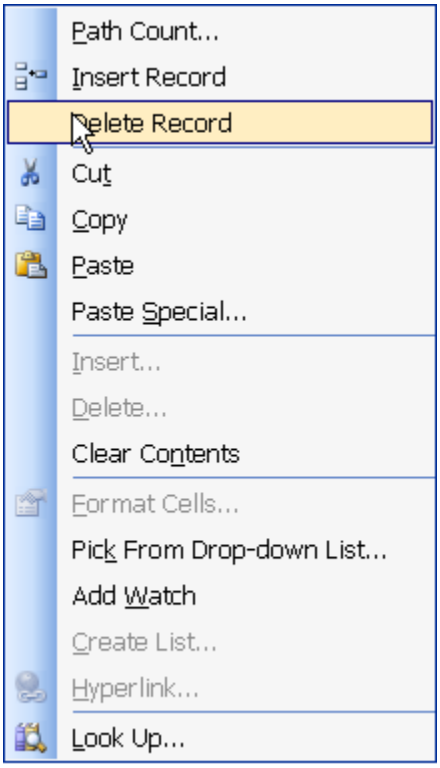

e5071c266

## Setting Measurement Parameters in Each Mode

The procedure described below is for setting all the measurement parameters of modes that are pre-defined in Setting Opeartion Mode of the DUT.

1. Select the "Spec Sheet" tab ((1) in Spec Sheet (1/2)).
2. Double-click the From column under Path ((3) in Spec Sheet (1/2)) and select a DUT port that is connected to the source port of the ENA from the drop-down list. The selection can be made from the ports that have been defined in Defining the DUT and Connecting to a Test Set.

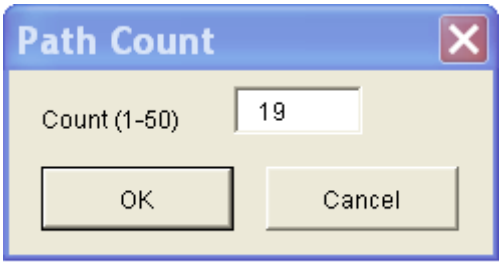
   NOTE To add a new row to a measurement path, double-click the cell in the Mode column ((2) in Spec Sheet (1/2)). To delete a row of a measurement path, place the cursor at the row you want to delete, right-click to display the list, and select Delete Record.




e5071c286

To add multiple new measurement paths, place the cursor at the mode, right-click and select Path Count...In the displayed Path Count dialog, set the number of paths as needed.

***Path Count Dialog***




e5071c486

3. As in Step 2, double-click a cell in the To column under Path ((3) of Spec Sheet (1/2)) to display a drop-down list and select a DUT port that is connected to the receiver port of the ENA.

4. Double-click a cell in the Type column ((4) in Spec Sheet (1/2)) to display a drop-down list and select measurement type for each path. The following types can be selected:

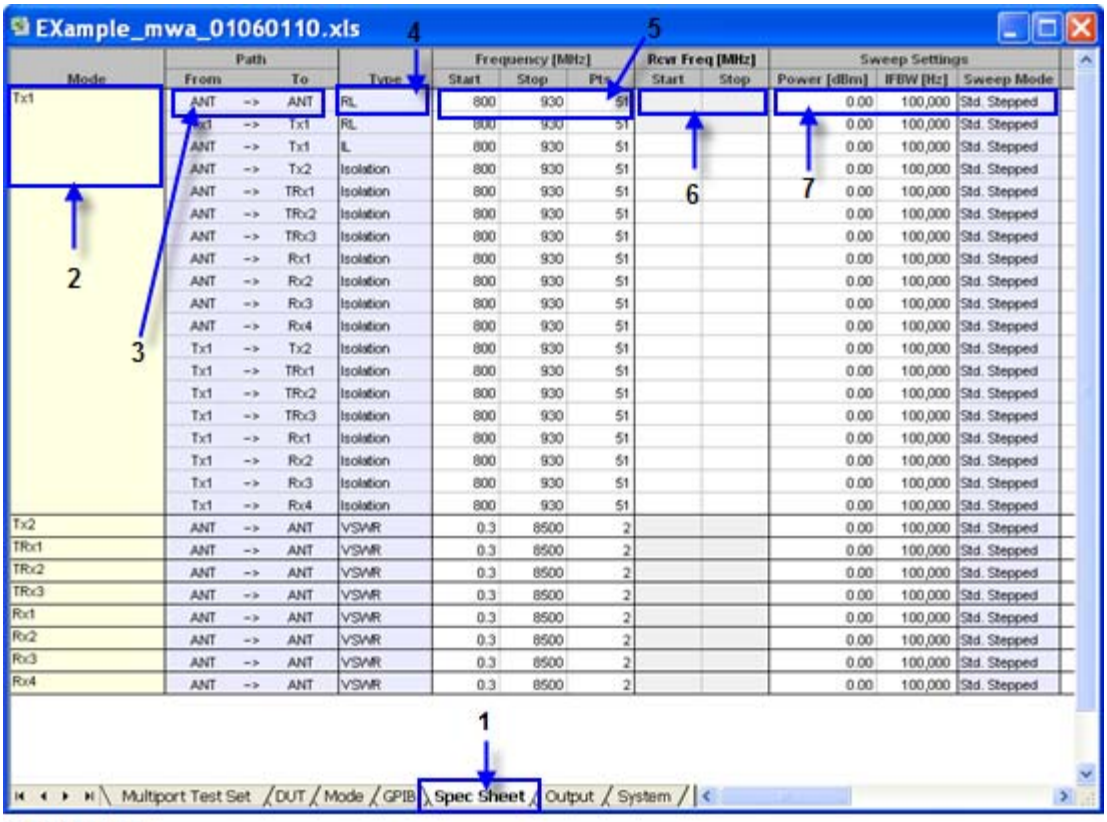
| No. | Measurement | Description | Parameter | Data |
|---|---|---|---|---|

| | Type | | | Format |
|---|---|---|---|---|
| 1 | IL | Performs the Insertion Loss measurement | Transmission | LogMag |
| 2 | Ripple | Performs the Ripple measurement | Transmission | LogMag |
| 3 | VSWR | Performs the Voltage Standing Wave Ratio measurement | Reflection | VSWR |
| 4 | RL | Performs the Return Loss measurement | Reflection | LogMag |
| 5 | ATT | Performs the Attenuation measurement | Transmission | LogMag |
| 6 | Isolation | Performs the Isolation measurement | Transmission | LogMag |
| 7 | BalAmp | Performs the Imbalance measurement with balanced port. (in magnitude) | Imbalance | LogMag |
| 8 | BalPhase | Performs the Imbalance measurement with balanced port. (in phase) | Imbalance | Phase |
| 9 | Phase | Performs the Phase measurement. | Reflection / Transmission | Phase |
| 10 | Group Delay | Performs the Group Delay measurement. | Reflection / Transmission | Group Delay |
| 11 | Lin Mag | Performs the measurement in linear | Reflection / Transmission | LinMag |

| | | | | |
|---|---|---|---|---|
| | | magnitude. | | |
| 12 | Real | Performs the measurement in real. | Reflection / Transmission | Real |
| 13 | Imag | Performs the measurement in imaginary. | Reflection / Transmission | Imaginary |

NOTE The limit test is performed by the back-end application with the limit values in the spec column. (Described in Step 8). The sign change is required for the limit test value in case of the IL, RL, ATT or Isolation measurement type. For example, the maximum -5dB of insertion loss is the allowable specification of the DUT, input 5 (dB) in the maximum (Max) value in the spec column. If you need the isolation at least below -80dB in the measured path, you should input 80 (dB) in the minimum (Min) value in the spec column for the limit test.

***Spec Sheet (1/2)***




e5071c267

5. Enter the start frequency, the stop frequency and the number of measurement points in the cells in Start, Stop and Pts columns ((5)

in Spec Sheet (1/2)) under Frequency [MHz]. These figures will be the sweep settings that are used to perform measurement by the back-end application of the ENA.

6. Frequency offset mode is available with the MWA by entering the start and stop frequency at the receiver port on the ENA in cells in the Start and Stop columns ((6) in Spec Sheet (1/2)) under Rcv Freq [MHz].
7. Set the output level, the IF bandwidth and the sweep mode of a port on the output side using cells in the Power, IFBW [Hz] and Sweep Mode columns under Sweep Setting ((7) in Spec Sheet (1/2)). Double-click the sweep mode cell to display a drop-down list and select a sweep mode of the ENA. The following options are available:

| No. | Sweep Modes | Description |
|---|---|---|
| 1 | Std. Stepped | Sets the sweep mode to the stepped mode. |
| 2 | Std. Swept | Sets the sweep mode to the swept mode. |
| 3 | Fast Stepped | Sets the sweep mode to the fast stepped mode (This mode is only for the E5070B/E5071B.) |
| 4 | Fast Swept | Sets the sweep mode to the fast swept mode (This mode is only for the E5070B/E5071B.) |

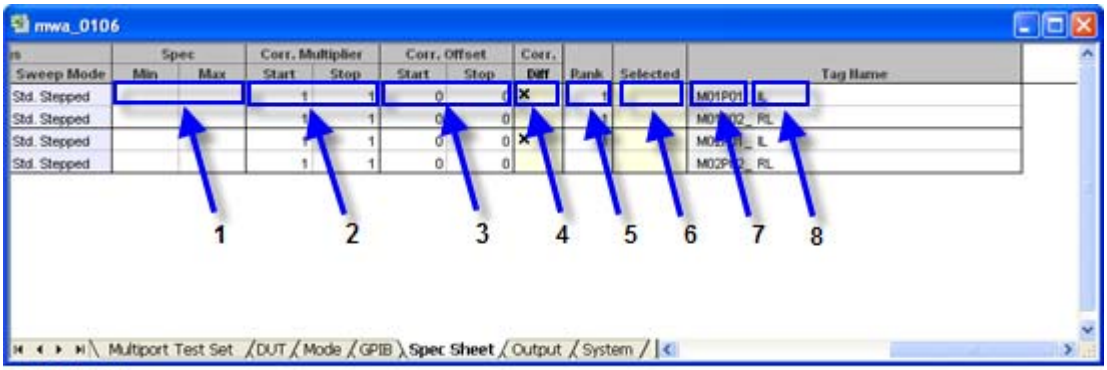
8. Enter the limit values of a measurement parameter in cells in the Min and Max columns ((1) in Spec Sheet (2/2)) under Spec column. The values entered here will be used for the limit test in the back-end application to evaluate measured parameters. If no values are entered in the cells, -999 is assumed for Min (lower limit) and 999 for Max (upper limit).
9. The measured data can be converted in the back-end application. In such cases enter correction values in cells in the Start and Stop columns under Corr. Multiplier ((2) in Spec Sheet (2/2)) and cells in the Start and Stop columns under Corr. Offset((3) in Spec Sheet (2/2)). The correction is done on the measured data at the start frequency and the stop frequency with the entered multiplier and offset values. The measured data on each point in between the two frequencies is linearly interpolated.
10. A cell in the Diff (difference) column under Corr. ((4) in Spec Sheet (2/2)) can be checked by double clicking. When a path has a check mark in this cell, the difference between the corrected value in

the first path and the other checked paths of the mode can be tested between in the back-end application.

This feature can be used for harmonic measurement (in dBc) by entering the parameters at the carrier frequency in the first path and the parameters at the harmonic frequencies in the next paths.

NOTE The first path in each mode cannot be checked because it is automatically assigned as the measurement reference for calculating differences from the other paths.

***Spec Sheet (2/2)***




e5071c268

11. Enter a rank number into a cell in the Rank column ((5) in Spec Sheet (2/2)) Rank can be used to prioritize the test result of measurement paths of the DUT. The rank number can be ranged from 0 to 255 and the lowest number of the rank is outputted to the handler I/O on the ENA by the back-end application after the completion of the limit test. This will indicate the most critical failure of the measurement path of the DUT in the limit test. When the test result gives the pass for all measurement paths, the returned rank number is 0.

12. A cell in the Selected column ((6) in Spec Sheet (2/2)) can be checked by double-clicking. The measurement of the path with the checked mark in the cell is performed in the back-end application. You can select in the back-end application whether to apply the selective measurement or not.

13. Enter the tag name of a measurement path in cells into the Tag Name columns ((7) and (8) in Spec Sheet (2/2)). The first part of the tag name ((7) in Spec Sheet (2/2)) is assigned automatically with the corresponding mode and path number, and cannot be modified. The second part of the name ((8) in Spec Sheet (2/2)) can be changed as needed.

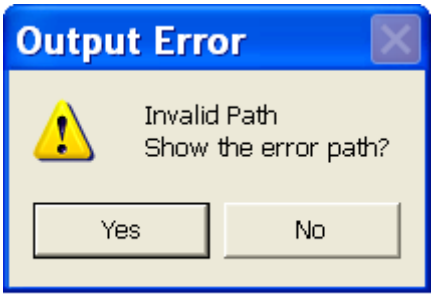
14. Repeat Step 2 through Step 13 as many times as needed to set up measurement path details in each mode.

### Generating a Spec Sheet

The procedure explained below is for verifying the contents of the settings and generating a spec sheet for the back-end application in the “Output” sheet.

1. Select the "Output" tab ((1) in **Output Sheet**).The channel and trace allocation of each measurement path on the ENA is automatically displayed in the sheet

   NOTE An error message dialog is displayed if there is any invalid input in the previous sheets of the front-end application. Press **Yes** in the dialog to return to the portion that caused the error.




e5071c487

2. Enter the maximum channel number of the ENA in the Max Channel cell ((4) in **Output Sheet**), the maximum number of traces per channel in the Max Trace cell ((5) in **Output Sheet**), and the maximum number of measurement points in the Max Point cell ((6) in **Output Sheet**) at the top of the "Output" sheet.

   Based on the values entered here, all the measurement paths in all the modes are automatically allocated to appropriate channels.

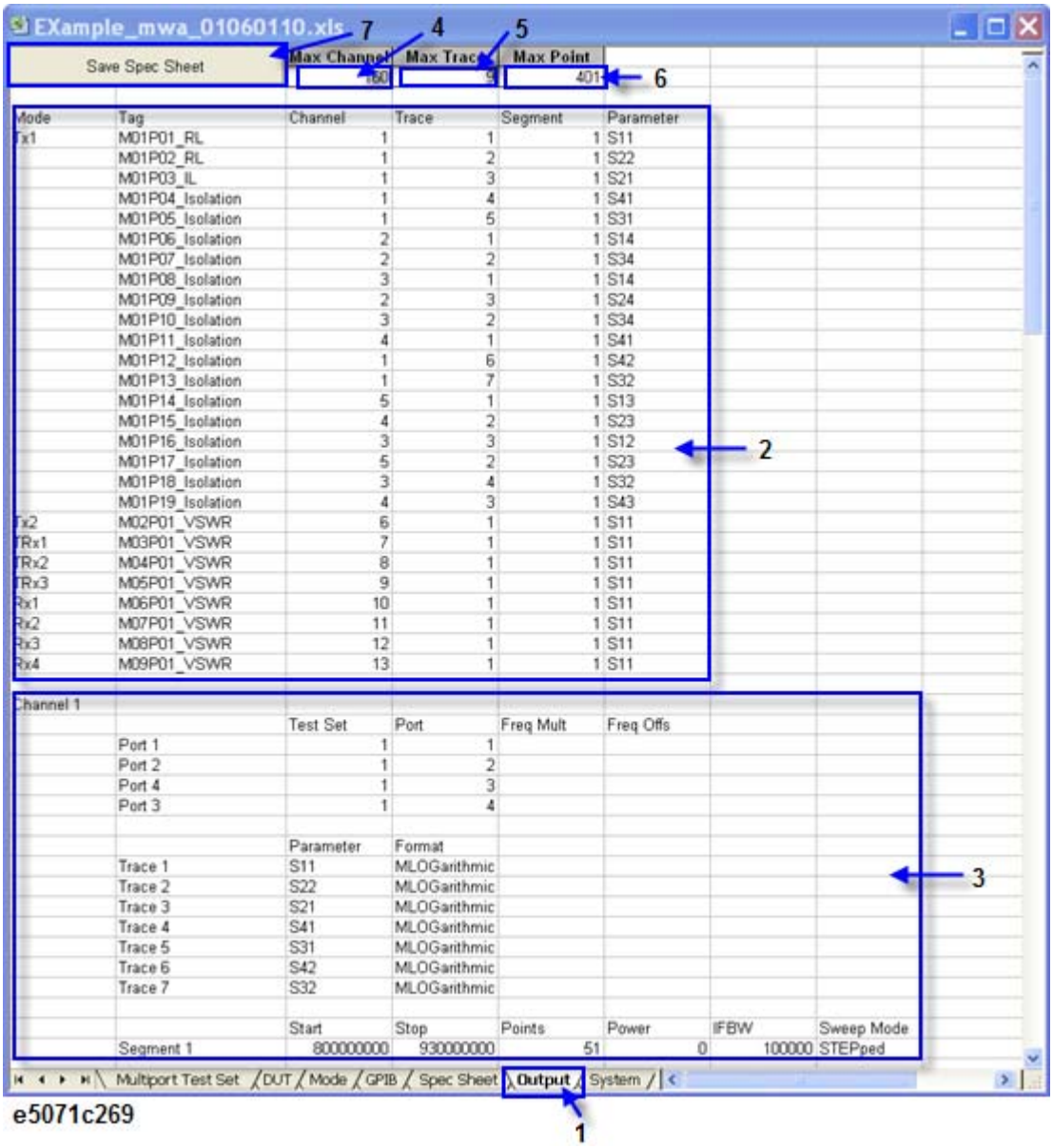
3. Information on all the measurement paths in all the modes and corresponding channels, traces, and segments is shown in the Output sheet ((2) in **Output Sheet**). Check the information and correct the corresponding sheet if there are any errors.

4. Detailed information on each channel including the test set ID, connected port number of the test set, measurement parameters on each trace is displayed ((3) in **Output Sheet**). Check the information and correct the corresponding sheet if there are any errors.

5. If all information shown in the Output sheet is correct, press **Save Spec Sheet** ((7) in **Output Sheet**) at the top of the sheet to display the Save dialog box. Select a folder and enter a preferred file name and press **Save**

to generate and save a spec sheet, which will be used by the back-end application.

***Output Sheet***




**NOTE** The maximum channel of the ENA is 160 from rev.A.09.10.

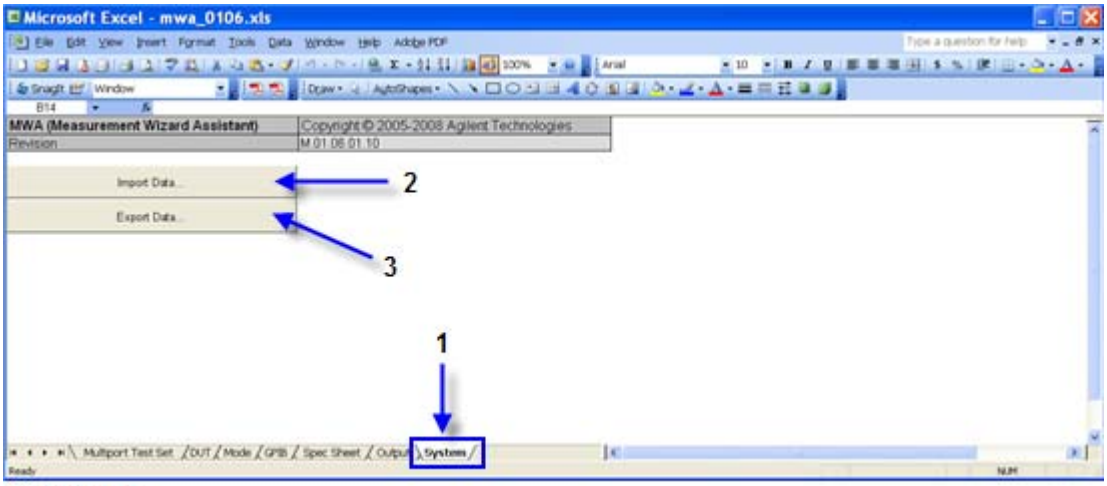
## Importing/Exporting Parameters

The procedure explained below is for importing/exporting the parameters in the csv format with the "System" sheet.

1. Select the "System" tab ((1) in System Sheet).

2. Press **Export** ((3) in System Sheet) to display the Export Data dialog box.
3. Enter a file name and press **OK**. All the entered parameters in the front-end application are exported to a file in csv format.
4. To import the parameters from a csv files, press **Import** ((2) in System Sheet) to display the Import Data dialog.
5. Select a csv file to read and press **OK**. The parameters in the csv file can be imported in the front-end application.

***System Sheet***




e5071c270

NOTE Because the exported file (.csv) is not a spec sheet,(.mwa) generated in the "Output" sheet, it cannot be recalled in the back-end application.

## Executing the Back-end Application in E5071C

- Overview
- Starting MWA Back-end Application
- Description of Windows
- Selecting and Clearing Spec Sheets
- Checking Connections
- Calibration Procedure
- Saving State File
- Measurement Procedure
- Created Result Files
- Controlling the Back-End Application From An External PC

- Operation of Handler I/O by the MWA
- Error messages in the Back-End Application

## Other topics about Measurement Wizard Assistant

### Overview

The back-end application of the MWA is a VBA program running on the E5071C. The back-end application reads a spec sheet (.mwa) created by the MWA front-end application, and automatically sets up all the measurement parameters on the E5071C. Based on the information included in the spec sheet,the back-end application sets up the E5071C multiport test set, or the external peripherals,and executes measurement procedure the detailed analysis report of the results.

### Starting MWA (Back-end Application)

A method for starting the back-end application of the MWA on the E5071C is explained below.

1. Press **Macro Setup** on the front panel.
2. Click the softkey **Load Project...** to display the open dialog.
3. Select the back-end application (**mwa_xxxx.vba**) and click **Open**.
4. Click the softkeys **Select Macro** > **Module1 Main**.
5. The main window of the back-end application is displayed.
6. If **mwa_xxxx.vba** is located in the **D:/Agilent/MWA** folder of the E5071C, the back-end application can be launched by the softkeys **Load & Run** > **mwa_xxxx**.

### Description of Windows

The back-end application of the MWA has the following windows:

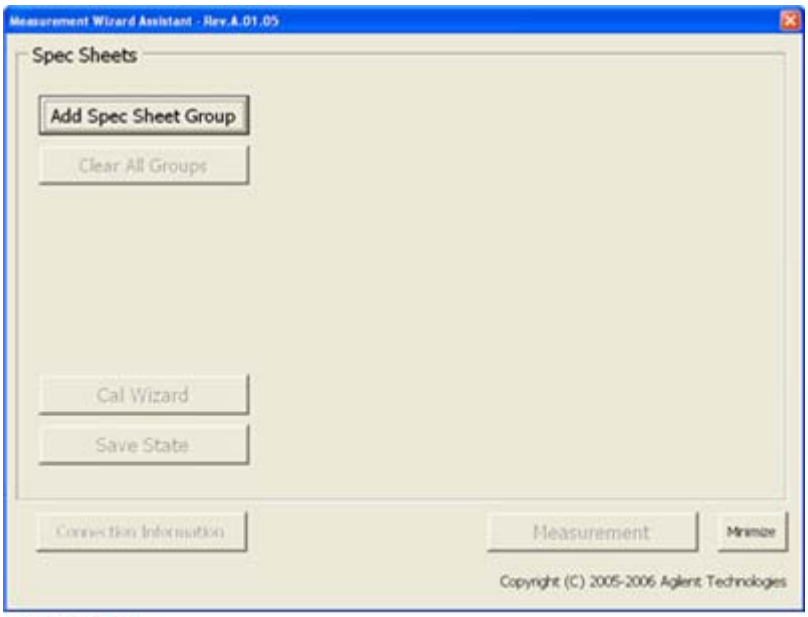
1. Main Window
2. Spec Sheet Setting Window
3. Connection Information Window
4. Calibration Wizard Window
5. All Spec Sheets Measurement Window
6. Measurement by Spec Sheet Window
7. Results Verification by Mode Window

#### 1. Main Window

This is the first window displayed after startup.

In this window, the following buttons can be selected for enabling functionality available in the back-end application

### *Main Window*




e5071c254

- **Add Spec Sheet Group** button (Main Window )

Displays Spec Sheet Setting Window.

- **Clear All Spec Groups** button (Main Window )

Clears group files that have been already read in the Spec Sheet Setting Window.

- **Cal Wizard** button (Main Window )

Displays Calibration Wizard Window (1/2)for automatic calibration procedure.

- **Save State** button (Main Window )

Saves a status file on the E5071C. As a status file is automatically saved in the E5071C when saving a group file or the necessary calibration is completed in the Calibration Wizard Window, it is not necessary to press this button if the setting on the E5071C is not changed.

- **Connection Information** button (Main Window )

Displays **Connection Information Window** for verifying the connectivity of the network analyzer, the multiport test set, and the DUT.

- **Measurement** button (Main Window )

Displays All Spec Sheet Measurement Window for launching measurement procedure.

- **Minimize** button (Main Window )

Minimizes the window of the back-end application. This function provides easy access to the display of the E5071C for changing parameters after recalling a spec sheet.

- Spec Sheet List (Main Window )

The recalled group files with the list of spec sheets are displayed in the Main Window after the setup in the Spec Sheet Setting Window (link inserted). The name of spec sheets, the number of group files, and the allocated channel number of the E5071C are included. The option button for each group file can be enabled for selecting the active group file used in the measurement.

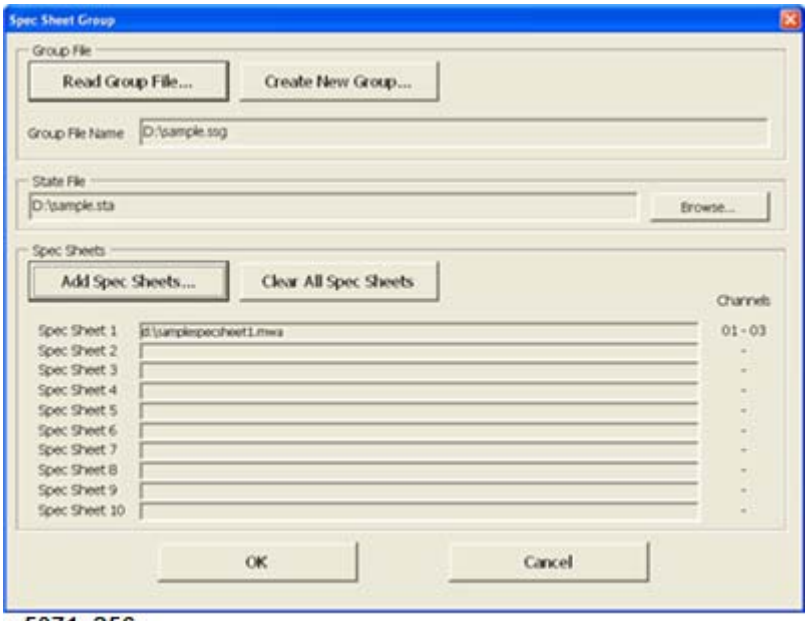
If multiple spec sheets are bundled in a group file, the option button of the first sheet can be enabled for the group file (It is not necessary to select the other spec sheets).

The group with the circle mark (● ) on the option button is the current active group. The calibration performed in the Calibration Wizard Window and the measurement is effective for this active group. The state file is saved by the Save State button (Main Window ) for the active group file as well.

### 2. Spec Sheet Setting Window

In this window, spec including all the parameters for the setup on the E5071C are recalled. The multiple spec sheets are bundled into one group and handled as a group file (.ssg) in the following procedure of the back-end application. Up to 10 different spec sheets can be added in one group file.

***Spec Sheet Setting Window***




e5071c256

- **Read Group File...** button

Recalls a saved group file (.ssg).

NOTE An error occurs when power value is set to different value by E5071C firmware.
In front-end application, the resolution of power value which can be set is 0.05dBm.

- **Create New Group...** button (Spec Sheet Setting Window (x))

Creates a new group file (.ssg).

- **Browse...** button (Spec Sheet Setting Window (x))

Displays the file selection dialog for status files of the group.

- **Add Spec Sheets...** button (Spec Sheet Setting Window (x))

Adds a spec sheet (.mwa) under the group file. The default name of the group file is named after the recalled spec sheet.

- **Clear All Spec Sheets** button (Spec Sheet Setting Window (x))

Clears all the spec sheets that have been already recalled in the window.

- **OK** button (Spec Sheet Setting Window (x))

Reflects all the settings on the E5071C according to the recalled spec sheets and returns to the Main Window. The state file for the group is saved automatically.

- **Cancel** button (Spec Sheet Setting Window (x))

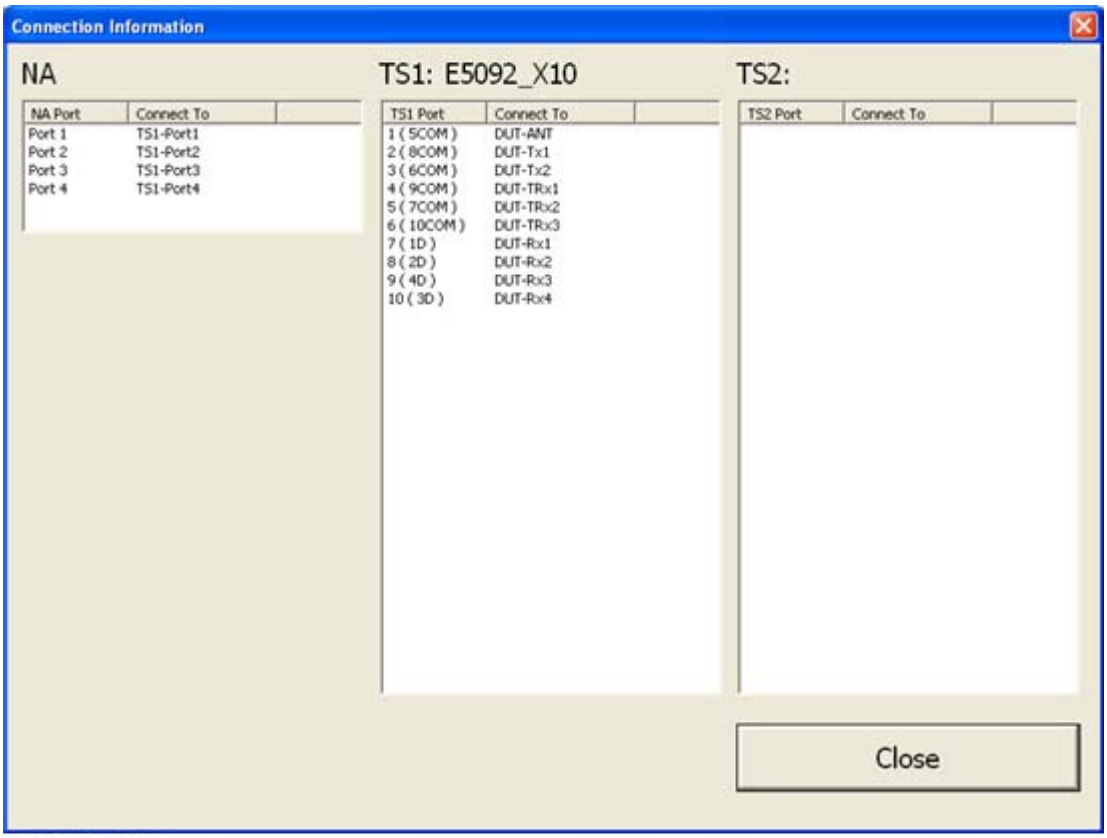
Discards the settings and returns to Main Window.

### 3. Connection Information Window

This window is displayed when **Connection Information** in the Main Window is selected.

Using the label names defined in the front-end application, this window shows information on the connection between the E5071C, the multiport test set(s),and the DUT.

***Connection Information Window***




e5071c257

- **Close** button

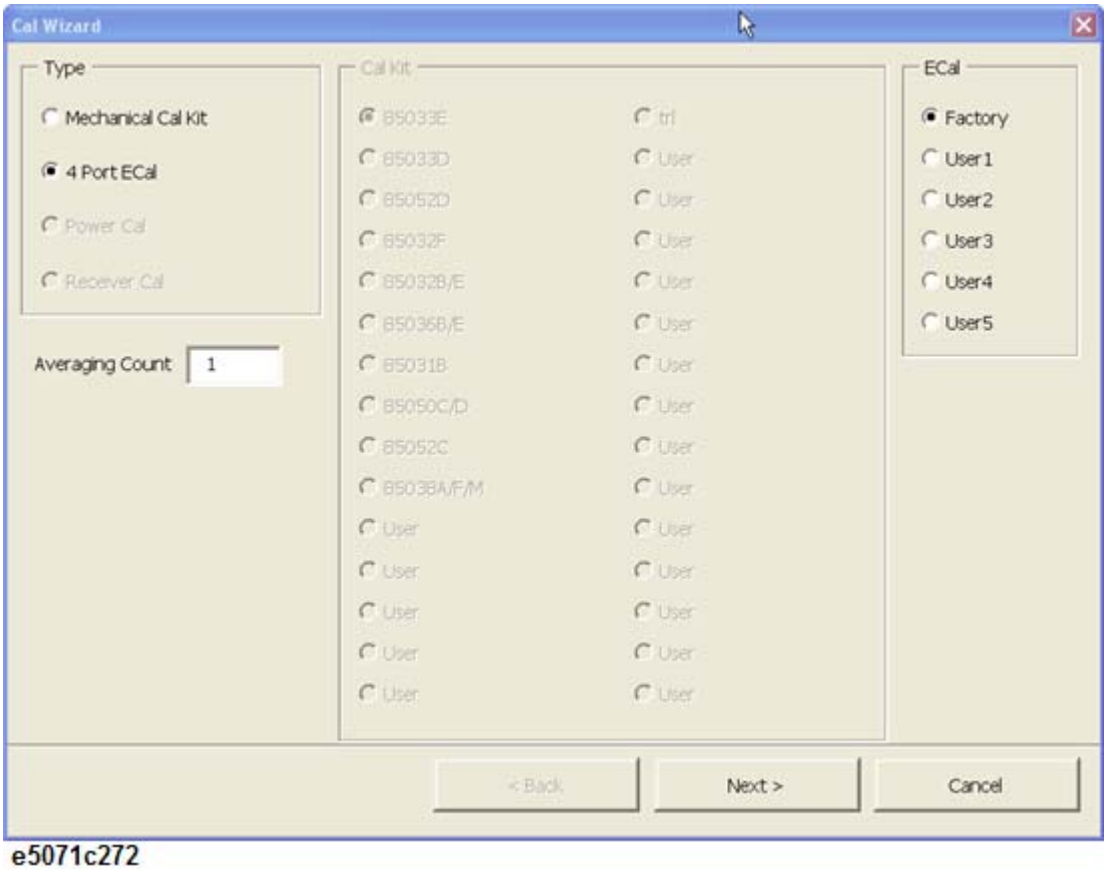
  Closes the window and returns to the Main Window.

#### 4. Calibration Wizard Window

This window is displayed when **Cal Wizard** in the Main Window is selected.

The wizard provides the necessary steps of the calibration procedure with the E5071C and the multiport test set in accordance with the contents of recalled spec sheets.

***Calibration Wizard Window (1/2)***




e5071c272

- List of Calibration Types

Displays the calibration types that can be performed with the calibration wizard.

These include Mechanical Cal Kit (calibration with the mechanical calibration kit), 4-Port ECal (calibration with the 4-port ECal module), Source Power Cal and Receiver Power Cal (Receiver calibration).

NOTE Source Power Cal is applied only to the measurement paths whose Receiver frequencies in the spec sheet of front-end application has been filled.

- List of Calibration Kits

Displays the list of mechanical calibration kits available on the E5071C for the calibration. This can be selected when Mechanical Cal Kit is selected as the calibration type.

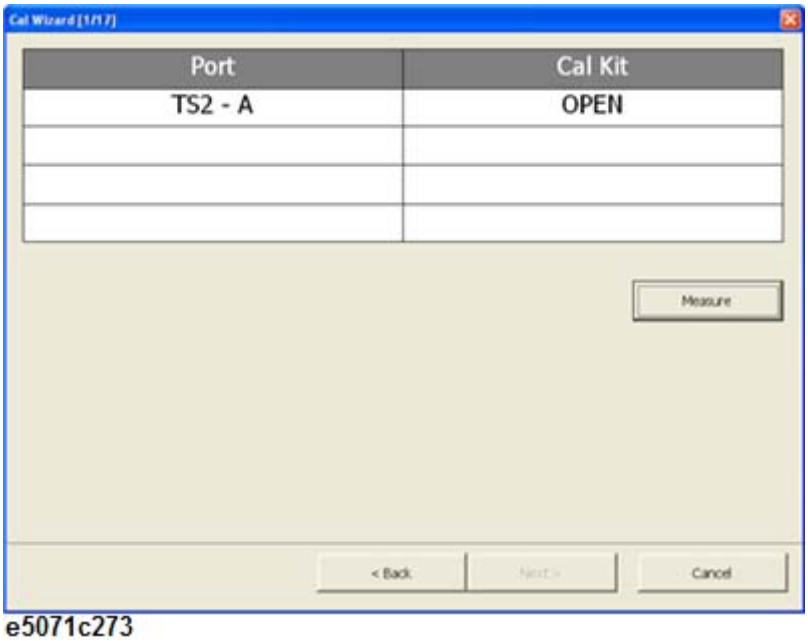
- List of ECals

Displays the list of characteristics registered in the ECal modules. User-defined characteristics of the ECal module (User 1 to 5) can be selected for calibration with different types of connectors.

- Averaging Count Entry

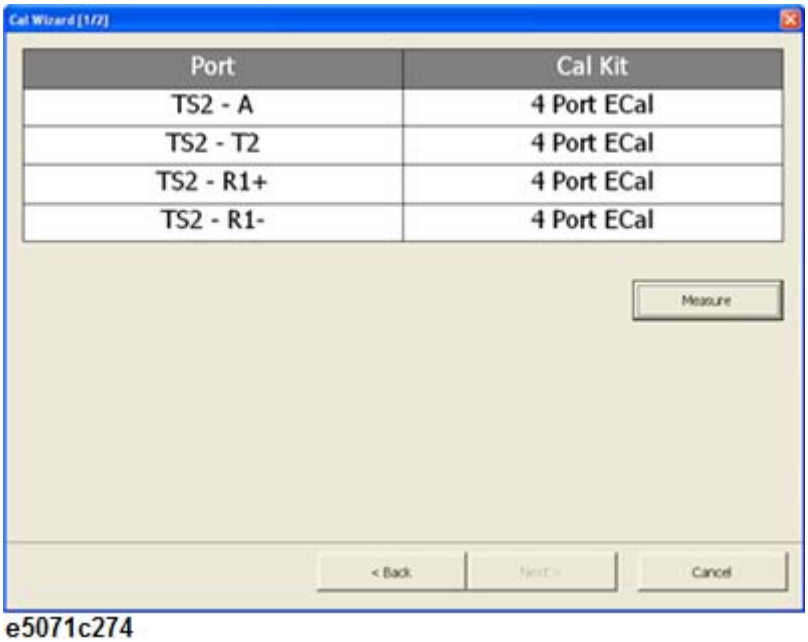
Sets the averaging number for measurements during calibration. This averaging number is used only during calibration and returns to the original number registered in the E5071C when calibration is completed.

***Calibration Wizard Window (2/2): mechanical calibration kit selected***




e5071c273

***Calibration Wizard Window (2/2): 4-port ECal selected***




e5071c274

NOTE During execution of the calibration wizard, the port name in Port or the calibration standard in CalKit in Calibration Wizard Window may be grayed-out. This means "the connection between the port and the calibration standard is the same as step in calibration procedure"; therefore it is not necessary to disconnect the standards for the calibration measurement.

- **Measure** button

Performs calibration measurement with the displayed connection.

- **<Back** button

Goes back to the previous step in the Calibration Wizard.

- **Next >** button

Goes to the next step in the Calibration Wizard after the calibration measurement is completed.

- **Cancel** button

Aborts the Calibration Wizard and returns to the Main Window.

- **Done** button (**Cancel** button changes)

Performs the correction of error coefficients on every necessary port then returns to Main Window.

NOTE Calibration data are automatically saved in the status file (.sta) of the group.

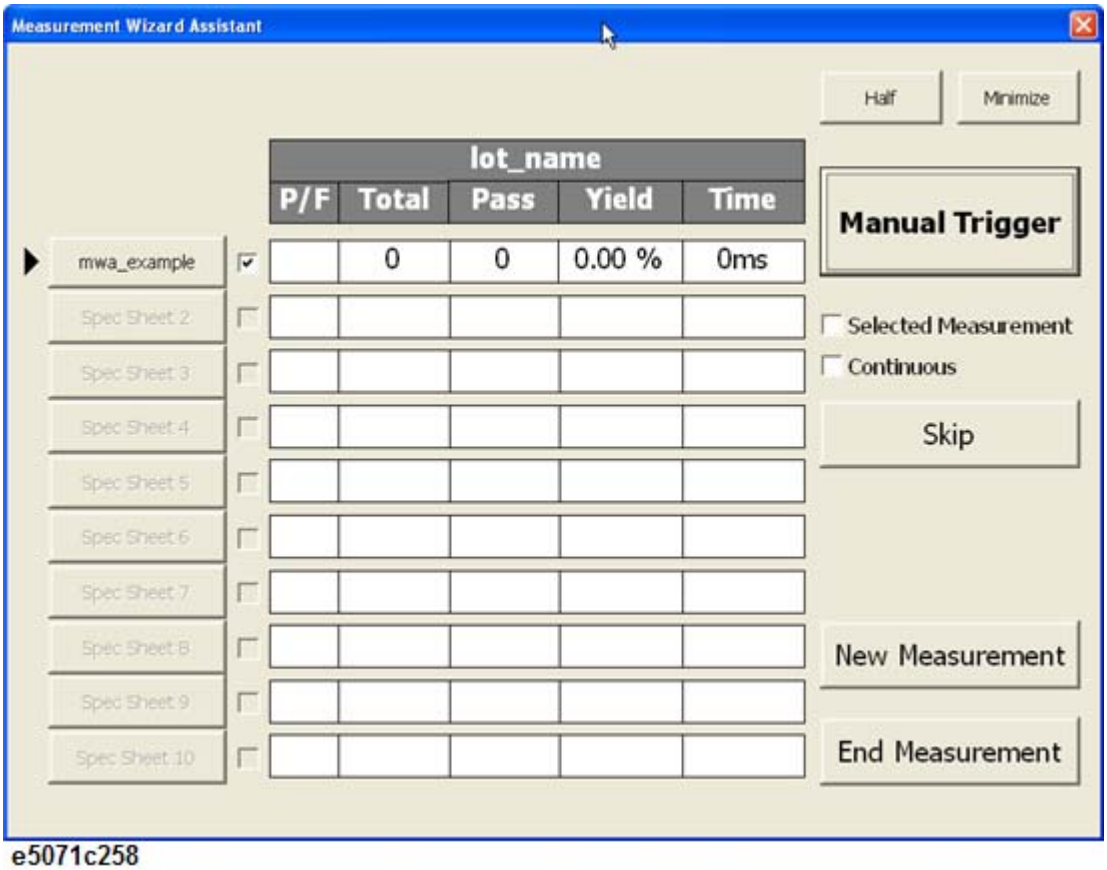
5. Measurement Window

All Spec Sheets Measurement Window

This window is displayed after the setup in the Lot Input Window when the **Measurement** button in the Main window is selected.

Spec sheets and summaries of their measurement results are listed in this window. To check the results by spec sheet, select each spec sheet in this window and press **Manual Trigger**.

***All Spec Sheet Measurement Window***




e5071c258

- **Spec Sheet** button (All Spec Sheet Measurement Window (x))

Displays Measurement by Spec Sheet Window.

The spec sheet button with the black triangle icon to its left indicates the next sheet to be measured.

- Measurement ON/OFF check box (All Spec Sheet Measurement Window (x))

Only the spec sheet with a checked mark is measured.

- **Half** button

Rescale the Measurement Window of the back-end application with the half size of the screen. The window is located at the bottom half of the screen and the measurement results of the E5071C is displayed in the top half. See the Measurement Window in half mode.

- **Minimize** button

Minimizes the Measurement Window. See the minimized window.

- **Manual Trigger** button

Makes an immediate measurements with manual trigger on the checked spec sheets in the window. (The black triangle icon is displayed to the left of the **Spec Sheet** button.)

- **Skip** button

Skip the measurement of the spec sheet with a black triangle icon on the left.

- **New Measurement** button

Ends measurement in the current window, saves the results, and displays All Spec Sheet Measurement Window with a new lot name.

- **End Measurement** button

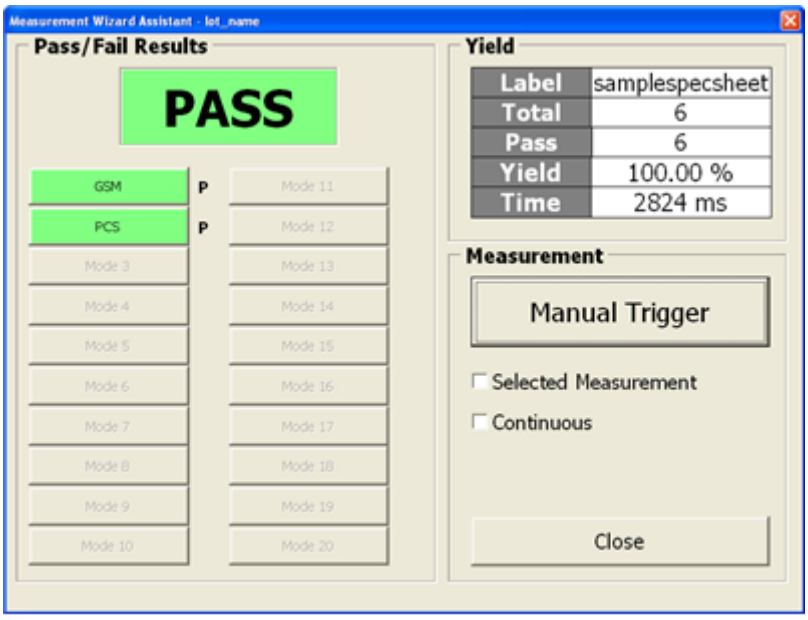
Ends measurement in the current window and returns to the Main Window. The measurement results are saved in the designated directory of the E5071C.

**Measurement by Spec Sheet Window**

This window is displayed by selecting one of the buttons for the spec sheets that are listed in the All Spec Sheets Measurement Window.

This window shows a list of the measurement modes in the selected spec sheet. To make measurements while checking measurement results in different measurement modes, press **Manual Trigger** in this window.

***Measurement by Spec Sheet Window***




e5071c259

- **Manual Trigger** button (Measurement by Spec Sheet Window (1))

Makes an immediate measurement with manual trigger on the checked spec sheets in the All Spec Sheets Measurement window.(The same function on the Manual Trigger button in the All Spec Sheets Measurement Window).

- **Close** button (Measurement by Spec Sheet Window (1))

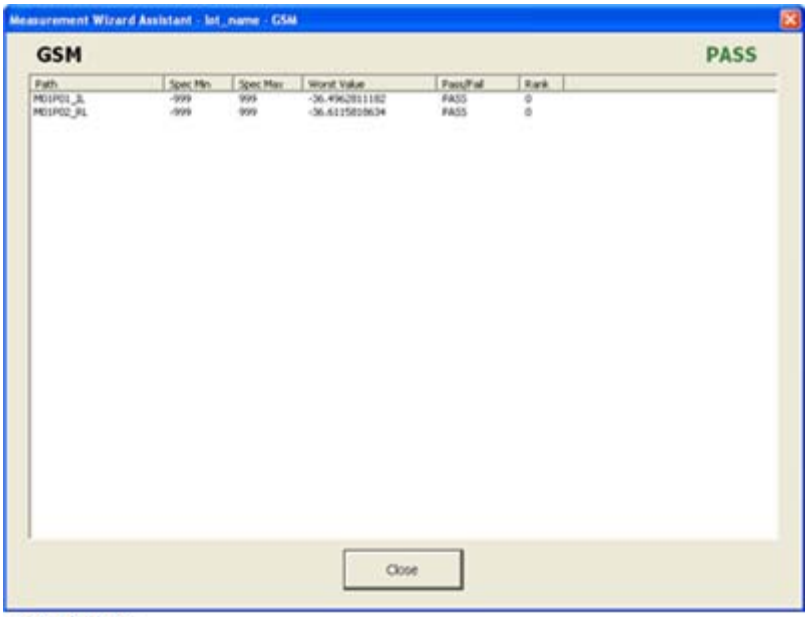
Returns to the All Spec Sheet Measurement window.

**Results Verification by Mode Window**

This window is displayed by selecting one of the buttons for the measurement modes listed in the Measurement by Spec Sheet Window.

This window shows a list of measurement results for the measurement paths specified in the selected measurement mode.

***Results Verification by Mode Window***




e5071c277

- Measurement results display area

  Path: The label name of a measurement path that is defined in the "Spec Sheet" in the front-end application.

  Spec Min/Max: The maximum and minimum limit values of the DUT specification for the measurement paths. These values are entered in the "Spec Sheet" in the front-end application.

  Worst Value: The limit test by the back-end application is performed for each measurement point and the worst value of the DUT is displayed in the measurement result. The worst value is defined as the farthest value from the

average of the maximum and minimum limit values, which corresponds to the typical measured data of the DUT.

Pass/Fail: Indicates the result of the limit test for all data in the measured path.

Rank: The rank value of each measurement path (specified in the Spec Sheet sheet in the front-end application). The result of the limit test can be prioritized and sorted in all the measurement paths with the fail. The back-end application outputs the minimum rank value among the failed measurement paths. For example, by assigning the smallest rank value for a specific measurement path, you can identify the worst performance of the DUT with the output of the rank value after the measurement.

If the result of the limit test gives pass for all the measurement paths, the output rank value is "0".

The definition of the pass/fail result of the measurement can be ranged by the Max Fail Rank specified in the Lot Input Window of the back-end application. When the rank value is greater than the max fail rank value, the failed result of the limit test is treated as pass, and the corresponding rank values for measurement paths are outputted in the result. The following table shows an example of the output rank values, when (the Max Fail Rank is set to 200).

| Rank value for path | Pass/Fail | Rank to be output |
|---|---|---|
| 1 - 200 | Fail | 1 - 200 |
| 201 - 255 | Pass | 201 - 255 |

For the measurement value, Worst Value, and Pass/Fail judgment, refer to step10 of the Setting Up Measurement Parameters in Each Mode.

- **Close** button

Returns to Measurement by Spec Sheet Window.

6. Other Windows

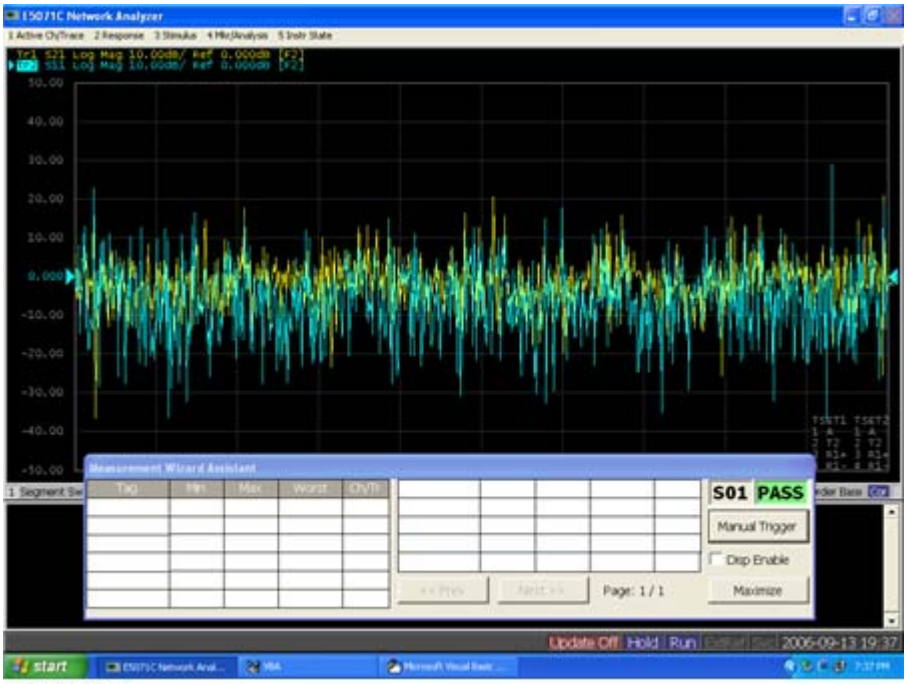
Measurement Window in **Half** mode: This window is shown when **half** in the All Spec Sheets Measurement Window is pressed.

You can use this window when you want to use the MWA while watching the measurement result displayed in the screen of the E5071C. In this window, only a failed measurement path is displayed and you can check the corresponding channel and trace for further investigation of the result.

If the measurement paths cannot be displayed in one screen, the **<<Prev** button and the **Next>>** button become effective.

If you check the Disp enable check box, Display Update function of E5071C is activated for drawing the image on the screen. It is recommended to disable this function and save time for updating the display when the fast testing speed is really required.

***Measurement Window in Half Mode***




e5071c275

**Lot Input Window**

The lot input window is used for the setup of the measurement.  For more information, refer to the Measurement Procedure (All Spec Sheets Measurement Window).

***Lot Input Window***


e5071c271

## Selecting and Clearing Spec Sheets

The procedure described below is for selecting or clearing spec sheets in the Main Window or in the Spec Sheet Setting Window.

### Selecting Spec sheets

1. Press **Add Spec Sheet Group** in the Main Window to display the Spec Sheet Setting Window.
2. To create a new group, press **Create New Group...** in the window, type the name of the new group in the file dialog, and press **Save**.

   To recall an existing group, press **Read Group File...** in the window, select the group file in the file dialog and press **Open**.

   The full path name of the group file entered or selected is displayed in the window
3. To specify the state file to be associated with the group, press **Browse...** in the State File frame, select the state file to recall and press **Open**.

   The full path name of the selected state file is displayed in the window.
4. Press **Add Spec Sheets...** in the window. Select the spec sheet to use in measurements and press **Open**.

The name of the selected spec sheet and allocated channel numbers in E5071C will be displayed in the window.

NOTE To select multiple spec sheets, repeat Step 4 as many times as needed to read all necessary spec sheets.

NOTE When you select a spec sheet without selecting a group file or a status file, the displayed names of these files are the same as that of the spec sheet.

NOTE If an error message appears, refer to Error messages in the Back-End Application.

5. After selecting all the necessary spec sheets, press **OK** to return to the Main Window. The name of the spec sheet and the allocated channel information will be displayed in the Main Window.

   All the functions in the Main Window become available after the spec sheet is imported in the back-end application.

6. When no spec sheet is imported or all the spec sheets are cleared by **Clear All Groups**, the buttons in the Main Window are Disabled and the related their functions become unavailable.

Clearing group files

1. Select **Clear All Groups** in the Main Window after setting up the group file in the Spec Sheet Setting Window.
2. All the group files are cleared and all the buttons in the Main Window become Disabled and the related functions become unavailable.
3. It is not available to clear each group file individually.

### Checking Connections

The procedure described below is for checking the connection between the measurement instruments and the DUT in the Connection Information Window.

1. Select **Connection Information** in the Main Window after recalling a spec sheet.
2. The Connection Information Window is displayed. Based on the information on the screen, connect the cables between the E5071C, the multiport test set(s) and the DUT.
3. After checking that the connection is correct, press **Close** to close the window and return to the Main Window.

NOTE The connection should be confirmed before the calibration procedure in the Calibration Wizard Window. It is also recommended to check the connection again before starting the measurement in the Measurement Window.

### Calibration Procedure

The procedure described below is for performing the calibration procedure by the Calibration Wizard Window of the back-end application.

1. Select a group file to calibrate with the radio button in the Main Window.
2. Select **Cal Wizard** in the Main Window.
3. The calibration wizard starts up and the Calibration Wizard window (1/2) is displayed. From the list of types of calibration shown in the window, select the type of calibration to perform. The following options are available:

| Type of Calibration | Description |
|---|---|
| Mechanical Cal Kit | Uses a calibration kit to perform calibration. For information on the selecting a calibration kit, refer to Step 4. |
| 4 Port ECal | Uses 4-port ECal to perform calibration. |
| Receiver Cal | Performs Receiver calibration while making FOM measurements. |

4. If Mechanical Cal Kit is selected in Step 3, the list of calibration kits available for selection is displayed in the window. Select the appropriate calibration kit to use.

If 4-port Ecal has been selected in Step 3, the list of user-defined characteristics of the ECal module is displayed in the window. Select the appropriate characteristics of the ECal module to use. Refer to User-characterized ECal for more detail.

NOTE The information on displayed calibration kits has been imported from the E5071C firmware. When the parameters of the calibration kit are changed in the E5071C, corresponding information will be reflected in the back-end application as well.

5. Enter the number of averaging during the calibration measurement into the Averaging Count input box. All the parameters on the calibration kit are selected, press **Next** for the next step in the calibration procedure.

6. The Calibration Wizard Window (2/2) will be displayed. Both the ports of the E5071C or the multiport test set and the calibration standards used for the calibration are displayed. Connect the ports with the standard and press **Measure**. After the calibration measurement is completed, press **Next >** for the next step.

7. Repeat Step 6 as many times as needed for all the calibration steps necessary for the selected spec sheets. After all the calibration measurement is completed, press **Done** and return to the Main

Window. The Calibration data will be saved in the state file of the selected group file automatically.

NOTE When performing calibration with ECal,the MWA back-end application shows "Execution error" if active channel is selected for trigger scope on the E5071C firmware. Hence, select **All Channels** for the trigger scope.

### Saving State File

The procedure for saving state files in the Main Window is described below.

The State file for a group is automatically saved when you create a group file in the Spec Sheet Setting Window or you perform calibration in the Calibration Wizard Window. If parameters on the E5071C are changed without using the functions of the back-end application, a state file should be saved manually in the Main Window.

1. Press **Save State** in the Main Window.
2. The State Files (.sta) of the current active groups are updated.

### Measurement Procedure (All Spec Sheets Measurement Window)

The procedure described below is for making measurements by spec sheet in the All Spec Sheets Measurement Window.

1. After the setup of spec sheets in the Spec Sheet Setting Window, press **Measurement** in the Main Window. The lot Input Window is displayed.
2. Enter the lot name of the DUT. Check optional functions in the measurement as necessary and press **OK**.

- Output Start/Stop Values

This option allows you to write measured values at the start and stop frequencies to a result file of the measurement. The back-end application automatically generates the result file for each spec sheet after the measurement. Refer to Measurement result by spec sheet.

- Correction (Limit Test)

This option allows you to reflect the correction on measured values specified in the “Spec Sheet” front-end application, refer to Step 9 in the Setting Measurement Parameters in Each Mode.

- Enable Manual Measurement

This option enables the **Manual Trigger** button in the All Spec Sheets Measurement Window and the Measurement by Spec Sheet Window. When this option is not checked, manual measurements using the **Manual Trigger** button cannot be made.

- Overwrite Output File

This option allows you to overwrite or add the measurement result when a result output file already exists. When this option is checked, the existing file is overwritten.

- Max fail rank (1-255)

This option sets the maximum rank value for the fail result in the limit test. For example, when "max fail rank" is set to 10, the result for measurement paths with ranks 1 through 10 treated as fail and 11 through 255 treated as pass. For more information, refer to rank in Measurement results display area.

When you press the **OK** button, the All Spec Sheet Setting Window is displayed.

3. Press **Skip** to place the black triangle icon to the left of the desired spec sheet with which measurements are made.
4. With the black triangle icon placed to the left of the spec sheet for use during measurement, press **Manual Trigger** to perform measurement.

To do the measurement for the selected measurement paths that have been checked in the "Spec Sheet" of the front-end application, check **Selected Measurement** and press **Manual Trigger**.

To make consecutive measurements using the displayed spec sheets, check the **Continuous** check box.

5. When measurements for the spec sheets are completed, the measurement result (Pass or Fail) is displayed to the right of each spec sheet.
6. To check measurement result for each spec sheet, press the button of the desired spec sheet to display the Measurement by Spec Sheet Window. For information on the Measurement by Spec Sheet Window, refer to Measurement Procedure (Measurement by Spec Sheet Window).

**Measurement Procedure (Measurement by Spec Sheet Window)**

The procedure described below is for selecting one of the spec sheets in the measurement window, and making detailed measurement in the Measurement by Spec Sheet Window for each measurement mode included in the spec sheet.

1. In the All Spec Sheets Measurement Window, press the button of the desired spec sheet to display the Measurement by Spec Sheets Window for the spec sheet.

2. Press **Manual Trigger** to launch the measurement for each mode in the selected spec sheet.

To do the measurement for the selected measurement paths that have been checked in the Spec Sheet" sheet of the front-end application, check the box for **Selected Measurement** and press **Manual Trigger**.

To make consecutive measurements with continuous sweep for all the measurement modes, check the **Continuous** check box.

3. When measurements are completed, the letter "P" (for Pass) or "F" (for Fail), is shown to the right of the measurement mode. Press **Close**, to return to the All Spec Sheets Measurement Window.
4. To check detailed measurement results for each measurement mode, press the button for the desired measurement mode to display the Results Verification by Mode Window. For information on the Results Verification by Mode Window, refer to Measurement Procedure (Results Verification by Mode Window).

**Measurement Procedure (Results Verification by Mode Window)**

The procedure described below is for selecting one of the measurement modes and checking results by measurement path in the Results Verification by Mode Window.

1. To display the Results Verification by Measurement Mode Window, press the button of the desired measurement mode in the Measurement by Spec Sheet Window.
2. Measurement results for each measurement path are listed in the Results Verification by Measurement Mode Window. For more details in the listed parameters, refer to measurement results display area in the Results Verification by Mode Window.

### Created Results File

This section describes measurement result files that are created at the end of a measurement.

1. Measurement result summary by lot

This file is created at the end of a measurement or when the name of a lot is changed by selecting New Measurement in the All Spec Sheets Measurement Window. It is created in csv format and saved automatically in the E5071C.

The content of the created file is shown in Measurement result summary by lot.

***Measurement result summary by lot***

The first line is the header of a file and the second line or below contains the measurement result for each spec sheet. The following parameters are included in the result:

- Spec Sheet x: The name of a spec sheet used in the measurement.
- Total Number: The total number of the measurement for the lot of the DUT.
- Pass Number: The total number of the pass result for the lot of the DUT.
- Mode x: The label name of the mode in a spec sheet that gives the fail result in the measurement.
- Mode x Fail Number: The total number of the fail result for the mode in a spec sheet.

2. Measurement result by spec sheet

The file of measurement result is created separately for each spec sheet. Each file is created in csv format ("Lot name"-"spec sheet name".csv) and saved automatically in the same folder as each spec sheet (.mwa) of E5071C. The content of the created file is changed depending on the selection of the "output start/stop value" check box in the Lot Input Window.

The content of the created file is shown in Measurement result by spec sheet.

***Files of measurement results by spec sheet (start/stop value: ON)***

"Index","Rank","Mode1-Path1",,,"Mode1-Path2",,,"[GPIB**Address] GPIB Command, ... ,"ModeX-PathY",,

,,"Start","Stop","Worst", ... ,"Meas", ... ,"Start","Stop","Worst", ...

1,Output Rank,Start_Value,Stop_Value,Worst_Value, ... ,GPIB_Response Value, ... ,Start_Value,Stop_Value,Worst_value, ...

2,Output Rank,Start_Value,Stop_Value,Worst_Value, ... ,GPIB_Response Value, ... ,Start_Value,Stop_Value,Worst_Value, ...

...

N,rOutput Rank,Start_Value,Stop_Value,Worst_Value, ... ,GPIB_Response Value, ... ,Start_Value,Stop_Value,Worst_Value, ...

The first and second lines are the header of a file, and the third line or below contains all the measurement result for the spec sheet. The following parameters are included in the file:

- Index: The number of measurement for the spec sheet.

- ModeX-PathY: The tag namme of the measurement path for the specific mode in a spec sheet.
- Output Rank: The output rank value for the spec sheet. See Rank for the details.
- Start Value: The measured value at the start frequency for the measurement path.
- Stop Value: The measured value at the stop frequency for the measurement path.
- Worst Value: The worst value in the measured frequency range for the measurement path. See Worst Value for the details.
- GPIB Response Value: The measured value that is imported from an external peripheral by a GPIB command. The result value is inserted at the end of results for a measurement mode. The query command is shown in the header with the GPIB address of the used peripheral.

All the measurement values of a single DUT are summarized in one line. When multiple measurements are performed for a spec sheet, the multiple measurement results in additional lines are included in the file.

As the measurement value at the start/stop frequency is not available for the measurement type such as “mean” or “ripple”, no data is recorded in the start/stop values for the path.

NOTE Even when the measurement for a spec sheet is skipped by the Skip button in the All Spec Sheets Measurement Window, the result is recorded in the file as well. However, the output rank is fixed to -1 and no measured values are recorded for the start/stop or worst values.

When "Output Start/Stop value" is disabled by the Lot Input Window in advance of the measurement, the contents of the output is shown below:

***Measurement result by spec sheet (start/stop value: OFF)***

```
"Index","Rank","Mode1-Path1",,,"Mode1-Path2",,,"[GPIB**Address]GPIB Command", ... ,"ModeX-PathY",,
1,Output Rank,Worst_value, ... ,GPIB Response Value, ... ,Worst_Value, ...
2,Output Rank,Worst_value, ... ,GPIB Response Value, ... ,Worst_Value, ...
...
N,Output Rank,Worst_value, ... ,GPIB Response Value, ... ,Worst_Value, ...
```

### Controlling the Back-End Application From An External PC

The procedure for controlling the back-end application from an external PC is described below.

1. Send the following commands to the E5071C via GPIB.

DISP:WIND1:TITL:DATA (Command Parameter)

The following command parameters are available with the MWA.

***Command Parameter***

| cmd_para function | Argument 1 | Argument 2 | Window that accepts the command |
|---|---|---|---|
| cmd_read | Group file name (.ssg) or spec sheet name (.mwa) | | Main Window |
| cmd_clear | - | | Main Window |
| cmd_begin | Lot name | Parameters in the Lot Input Window. For more details, refer to Measurement Procedure (All Spec Sheets Measurement Window) | Main Window, Measurement Window |
| cmd_initiate | - | | Measurement Window |
| cmd_skip | - | | Measurement Window |
| cmd_settings | Correction values, Manual measurement | | Measurement Window |
| cmd_end | - | | Measurement Window |

Each of the cmd_para functions as follows.

| cmd_para functions | Description | Corresponding buttons |
|---|---|---|
| cmd_read | Reads the group file or spec sheets specified by the argument 1. If you send this command when no file exists or no more files can be read, an error occurs.<br>To specify multiple spec sheets, specify spec sheets separated with commas when assigning an argument 1 in Step 2. | Main Window<br>**Add Spec Sheet Group** |
| cmd_clear | Clears all spec sheets that are currently | Main Window |

| | | |
|---|---|---|
| | read. | **Clear All Groups** |
| cmd_begin | Displays the All Spec Sheets Measurement Window, specifying a lot name of the DUT by using argument 1.<br>Argument 2 specifies the option for the measurement (the same setup in the Lot Input Window). For more details, refer to Measurement Procedure (All Spec Sheets Measurement Window). For each option, refer to that page.<br>• Output Start/Stop Values<br>The same as Start/Stop values. Specifies 0 (OFF) or 1 (ON).<br>• Correction values<br>The same as Correction (Limit Test). Specifies 0 (OFF) or 1 (ON).<br>• Manual Measurement<br>The same as enable Manual Measurement. Specifies 0 (OFF) or 1 (ON).<br>• Overwrite Output File<br>The same as Overwrite Output File. Specifies 0 (OFF) or 1 (ON).<br>• Max Fail Rank<br>The same as max fail rank (1-255). Specifies the value from 1 to 255.<br>If you send this command when each measurement window is displayed, a summary of the results is generated to a file, the current measurement is closed, and the All Spec Sheets Measurement window opens with a new lot name. | Main Window<br>**Measurement**<br>All Spec Sheets Measurement Window<br>**New Measurement** |
| cmd_initiate | Initiates measurement immediately. | All Spec Sheets Measurement Window<br>**Manual Trigger**<br>Measurement by Spec Sheet Window<br>**Manual Trigger** |
| cmd_skip | Skips the measurement of the next spec sheet. | All Spec Sheets Measurement Window<br>**Skip** |

| | | |
|---|---|---|
| cmd_settings | Changes the option of correction values and manual measurement during the measurement. For correction values and manual measurement, refer to the descriptions of cmd_begin. | None |
| cmd_end | Outputs summary of the results and returns to the Main Window. | All Spec Sheets Measurement Window<br>**End Measurement** |

2. If it is necessary to set the argument for a command, the following command is required before sending the command in Step 1. If each argument has several items, set all the parameters separated by a comma (,).

- Setting command of Argument 1

DISP:WIND2:TITL:DATA parameter

- Setting command of Argument 2

DISP:WIND3:TITL:DATA parameter

For example, to start new measurement using the lot name "lot_gsm001," send the following command. In the following example, the output start/stop value and overwrite output file are turned ON and the max fail rank is set to 100.

```
10 output 717;"DISP:WIND2:TITL:DATA lot_gsm001"
20 output 717;"DISP:WIND3:TITL:DATA 1,0,0,1,100"
30 output 717;"DISP:WIND1:TITL:DATA cmd_begin"
```

NOTE If you send a command with incorrect arguments, the command is not executed.

When the command process finishes, the return value of DISP:WIND1:TITL:DATA? becomes "cmd_done" If you want to wait for the command process to finish, confirm this by whether the returned value of DISP:WIND1:TITL:DATA? is "cmd_done"

### Operation of Handler I/O Port by MWA

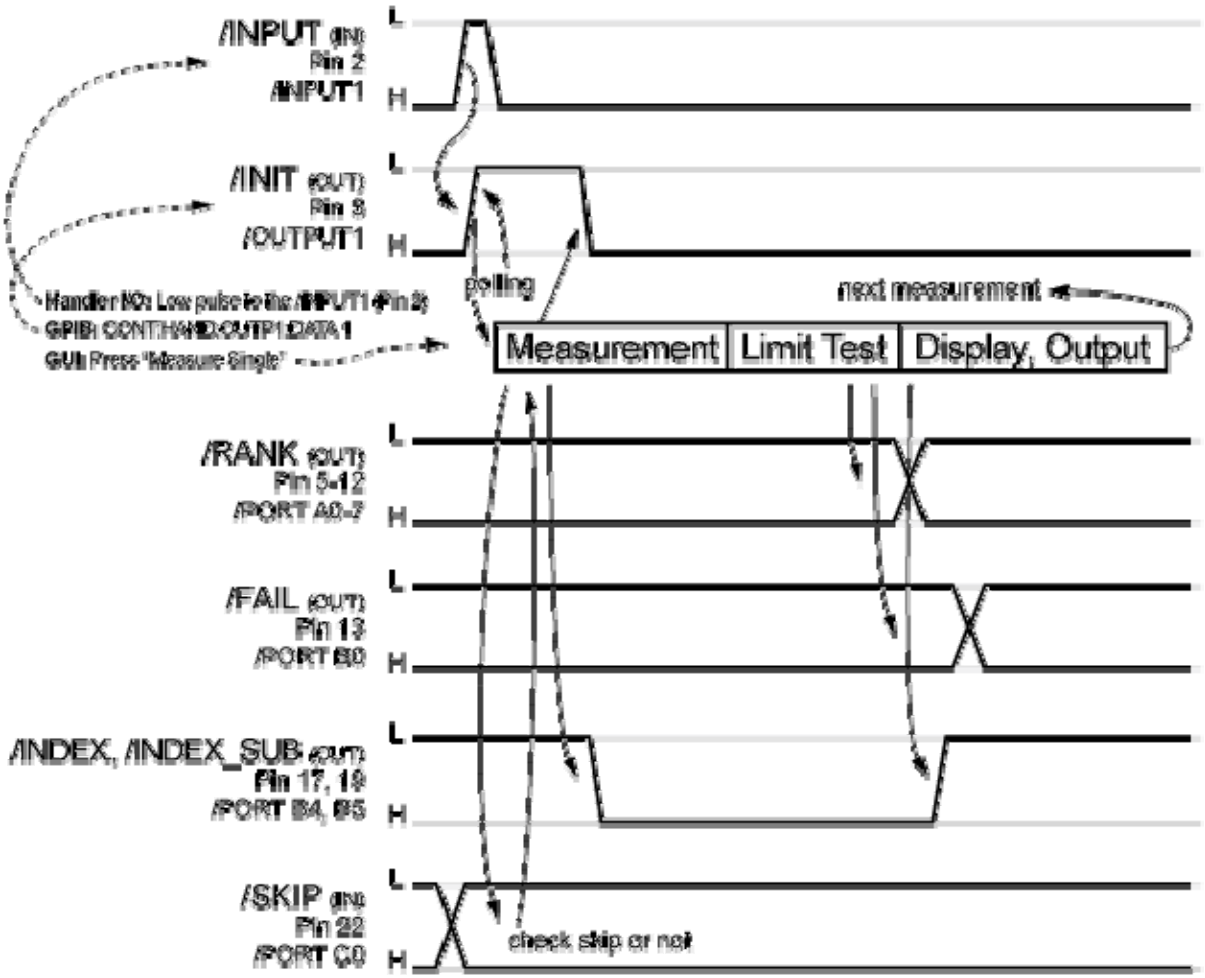
A list of pins of the handler I/O operated by the MWA is shown below. The signal notations in the parentheses correspond to those of the E5071C. For details, refer to I/O Signal Pin Layout and Description.

| Pin number | Signal Name in MWA (signal name in normal mode) | Direction of signal | Description |
|---|---|---|---|

| | | | |
|---|---|---|---|
| 2 | /INPUT<br>(/INPUT1) | Input | When a negative pulse is applied to this pin, the level of the signal /OUTPUT1 becomes Low. |
| 3 | /INIT<br>(/OUTPUT1) | Output | When a negative pulse is applied to pin 2, the level of /OUTPUT1 becomes Low.<br>The back-end application monitors /OUTPUT1 and when it turns Low, starts measurement. /OUTPUT1 returns to High at the start of a measurement.<br>In addition, the "CONT:HAND:OUTP1:DATA 1" command can be used to set the signal level to Low or High. |
| 5 | /RANK A0<br>(/PORT A0) | Output | Outputs the minimum rank value for the measurement. The output rank is the 8-bit value (0-255) with A0 as LSB and A7 as MSB.<br>/PORT A0 is bit 0 of the output rank. |
| 6 | /RANK A1<br>(/PORT A1) | Output | Bit 1 of the output rank. |
| 7 | /RANK A2<br>(/PORT A2) | Output | Bit 2 of the output rank. |
| 8 | /RANK A3<br>(/PORT A3) | Output | Bit 3 of the output rank. |
| 9 | /RANK A4<br>(/PORT A4) | Output | Bit 4 of the output rank. |
| 10 | /RANK A5<br>(/PORT A5) | Output | Bit 5 of the output rank. |
| 11 | /RANK A6<br>(/PORT A6) | Output | Bit 6 of the output rank. |
| 12 | /RANK A7<br>(/PORT A7) | Output | Bit 7 of the output rank. |
| 13 | /FAIL<br>(/PORT B0) | Output | If the limit test fails, /PORT B0 becomes enabled (Low level). If the output rank value (/PORT A0 - /PORT A7)is 0 (which corresponds to the |

| | | | |
|---|---|---|---|
| | | | pass for all measurements), /PORT B0 becomes disabled (High level).<br>This signal transits after the output /RANK (/PORT A0 to A7) transition, but transits almost the same timing of the /INDEX (/PORT B4) transition. Be sure to check /FAIL with some delay after /INDEX transition. |
| 17 | /INDEX<br>(/PORT B4) | Output | A signal that indicates the end of measurement.<br>The signal transits after the output rank (/PORT A0 - /PORT A7) transition.<br>The back-end application sets /PORT B4 back to inactive (High level) at the start of measurement. |
| 19 | /INDEX_SUB<br>(/PORT B5) | Output | A reserved signal pin with the same function as /PORT B4. Either pin can be used. |
| 22 | /SKIP<br>(/PORT C0) | Input/Output | This is a skip signal.<br>If /PORT C0 is enabled (Low level) before measurement, the measurement is not performed and the status shifts to the mode of waiting for the next measurement. |

Timing chart of handler I/O ports by the MWA




## Error messages in the Back-End Application

While working with MWA, an error message may appear due to improper E5071C settings. This section explain the various error messages and the method to avoid these errors.

| Error Message | Description |
|---|---|
| Channel Max | This error message appears if the total number channels required for a spec sheet (.mwa) is more than the number of channels set in the E5071C.<br>• This error can be avoided by increasing the maximum number of channels in E5071C. Refer to Setting Upper Limits of Number of Channels/Traces. |

| | |
|---|---|
| Configuration and test set are mismatched. | This error message appears if the configuration of the test set specified in a spec sheet (.mwa) generated by the Front-End Application is different from the test set that is being connected to the E5071C.<br>This error can be avoided by changing the configuration of the Multiport Test Set sheet in the Front-End Application. For more info, refer to Selecting Test Set.<br><br>This error may appear if:<br>• The configuration in a spec sheet is for the E5091A (E5092A), but the E5092A (E5091A) is connected to the E5071C.<br>• The configuration in a spec sheet is with the multiport test set (E5091A or E5092A), but no test set is connected to the E5071C.<br>• The configuration in a spec sheet is "None" (no test set), but the test set is connected to the E5071C. |
| GPIB Error | This error message appears if the GPIB address of peripherals connected to the E5071C is different from what you specified in the GPIB sheet of the front-end application. |
| Invalid number of point | This error message appears if the maximum number of points required for a spec sheet (.mwa) is more than the number of points set in the E5071C.<br>• This error can be avoided by increasing the maximum number of channels in E5071C. Refer to Setting Upper Limits of Number of Channels/Traces. |
| Invalid Trace | This error message appears if the maximum number of traces required for a spec sheet (.mwa) is more than the number of traces set in the E5071C.<br>• This error can be avoided by increasing the maximum number of channels in E5071C. Refer to Setting Upper Limits of Number of Channels/Traces. |

## Measurement Example of a Multiport Switch

- Overview
- Device Under Test (DUT)
- Required Equipment
- Measurement Parameters
- Measurement Steps

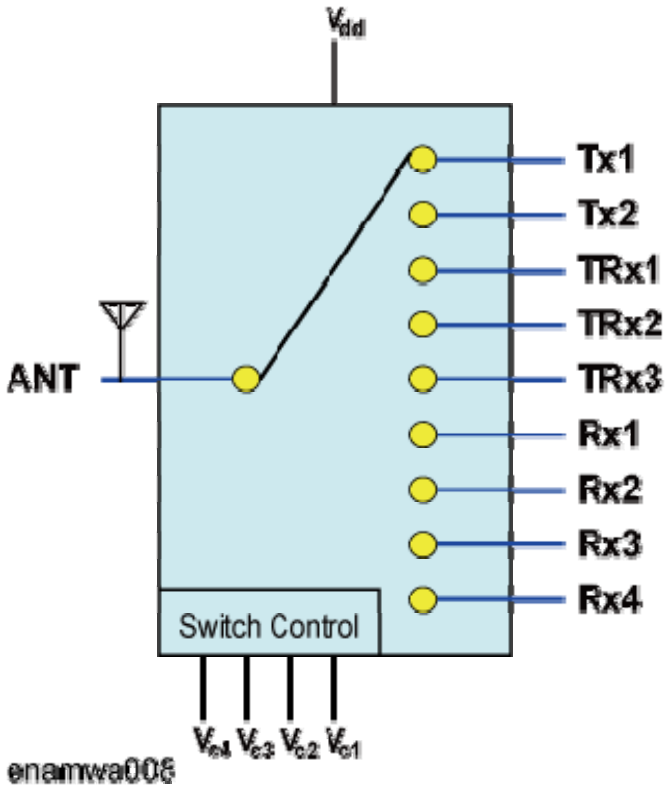
Other topics about Measurement Wizard Assistant

### Overview

This section describes how to measure a multiport switch using the MWA software on the E5071C. The measurement conditions for this measurement example are those suitable for a SP9T multiport switch. To measure another device under test (DUT), change the measurement conditions to suit the particular DUT.

### Device Under Test (DUT)

The DUT adopted in this example is a sample SP9T switch with the following configuration and should be connected to the E5092A with 10-port full crossbar configuration:




| DUT Port Name | Port Number | Test Set |
|---|---|---|
| ANT | 1 | 5COM |
| Tx1 | 2 | 8COM |
| Tx2 | 3 | 6COM |
| TRx1 | 4 | 9COM |
| TRx2 | 5 | 7COM |
| TRx3 | 6 | 10COM |
| Rx1 | 7 | 1D |
| Rx2 | 8 | 2D |
| Rx3 | 9 | 4D |
| Rx4 | 10 | 3D |

### Required Equipment

The following test equipment is required for this example:

- E5071C (4-port ENA; option 48x or 4K5) with the firmware rev 9.0.0 or later
- E5092A Configurable Multiport Test Set (option 020)
- RF cables (for the connection between the DUT and the test set)
- Control line cable

### Measurement Parameters

The sample DUT adopted in this example is a SP9T (Single-pole, 9-throw) 10-port switch. Transmission measurement for this switch should be performed for all the transmission paths between arbitrary ports, and reflection measurement for all ports of the DUT. The output port of the DUT is selected by 4-bit logic control DC voltage (2.8V as high and 0V as low).

The truth table for the sample switch is as follows:

| | **Group A** | | | | **Group B** | | | |
|---|---|---|---|---|---|---|---|---|
| **Connected Path** | **Vc1** | **Vc2** | **Vc3** | **Vc4** | **Vc1** | **Vc2** | **Vc3** | **Vc4** |
| Tx1 | **H** | L | L | L | **H** | L | L | L |
| Tx2 | L | **H** | L | L | **H** | L | L | L |
| TRx1 | **H** | **H** | L | L | **H** | L | L | L |
| TRx2 | L | L | **H** | L | **H** | L | L | L |
| TRx3 | **H** | L | **H** | L | **H** | L | L | L |
| Rx1 | L | **H** | **H** | L | **H** | L | L | L |
| Rx2 | **H** | **H** | **H** | L | **H** | L | L | L |
| Rx3 | L | L | L | **H** | **H** | L | L | L |
| Rx4 | **H** | L | L | **H** | **H** | L | L | L |

### Measurement Steps

The measurement for the sample switch will be done using the front-end and the back-end applications of the MWA software. The measurement procedure is:

1. Generate a spec sheet (.mwa) with the MWA front-end application.
2. Copy the spec sheet on the ENA directory

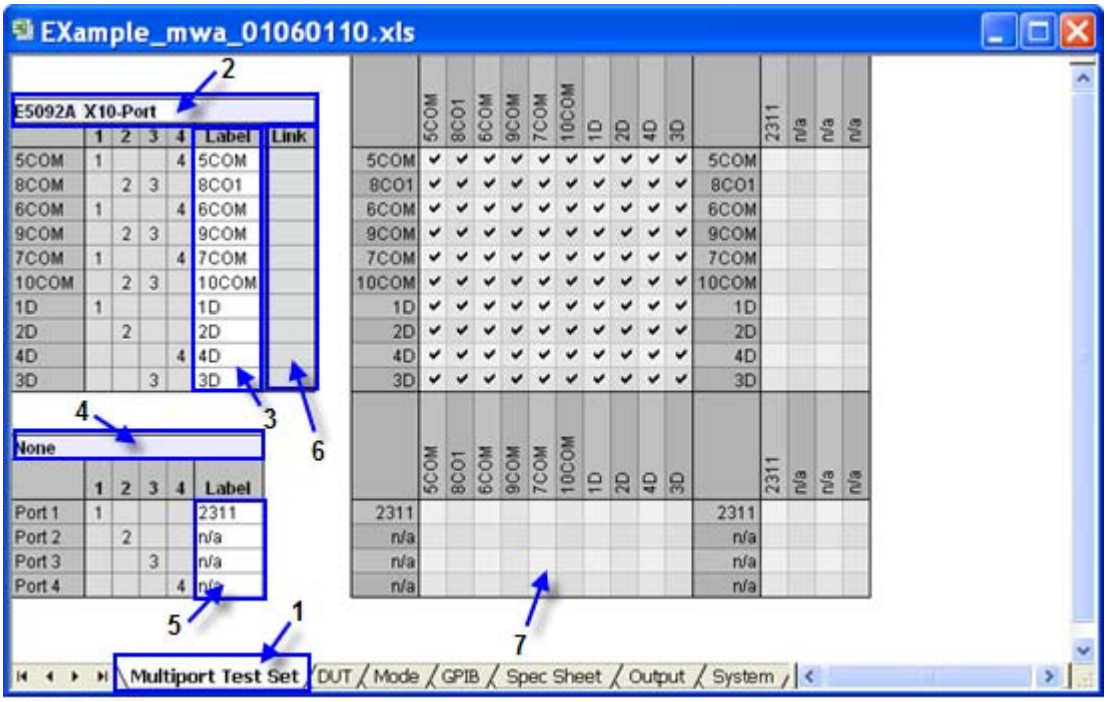
3. Import the spec sheet with the MWA back-end application.
4. Perform necessary calibration measurement by calibration wizard in the back-end application.
5. Connect the DUT with the test set (RF cables, control lines)
6. Perform measurement with the back-end application.

### Front-End Application

The Front-end application consist of six sheets in which data for the sample switch will be entered as per the switch specifications. This sheet can be run on a normal PC:

1. Multiport Test Set sheet

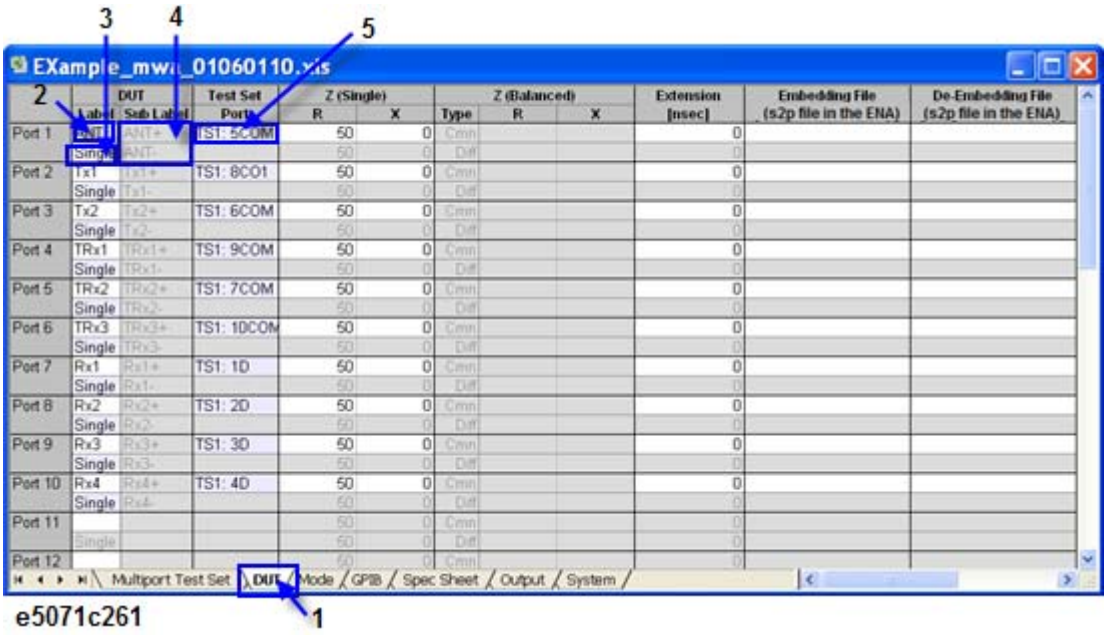
Select a configuration of "E5092A X-10-Port" for 10-port full crossbar measurement.




e5071c260

2. DUT sheet

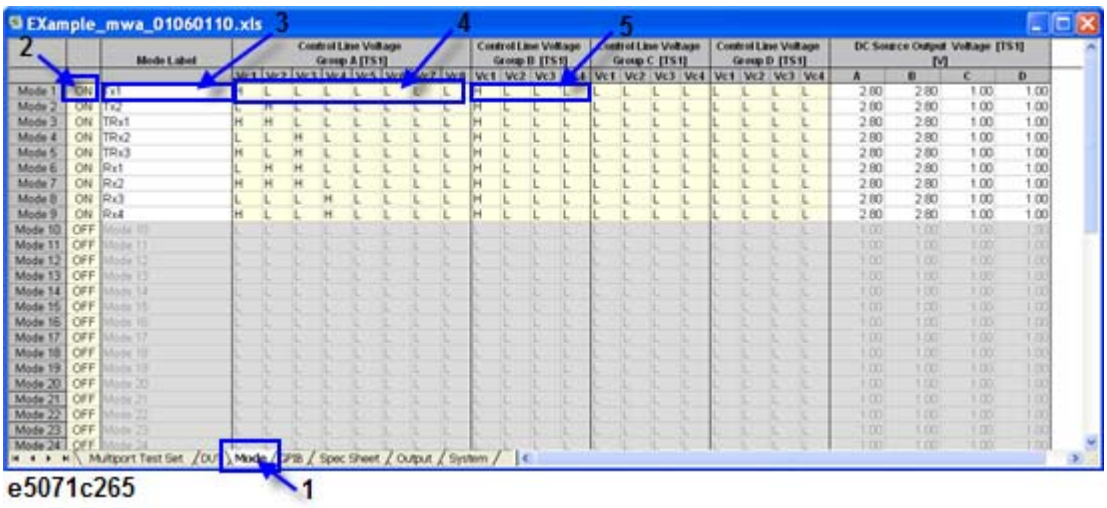
Define the port name of the DUT and the connection with the multiport test set.



e5071c261

3. Mode sheet

Control logic voltage (Vc1 to Vc3) of the control line group A is applied to the DUT (logic A to C) and Vc4 is used for power supply of the DUT.



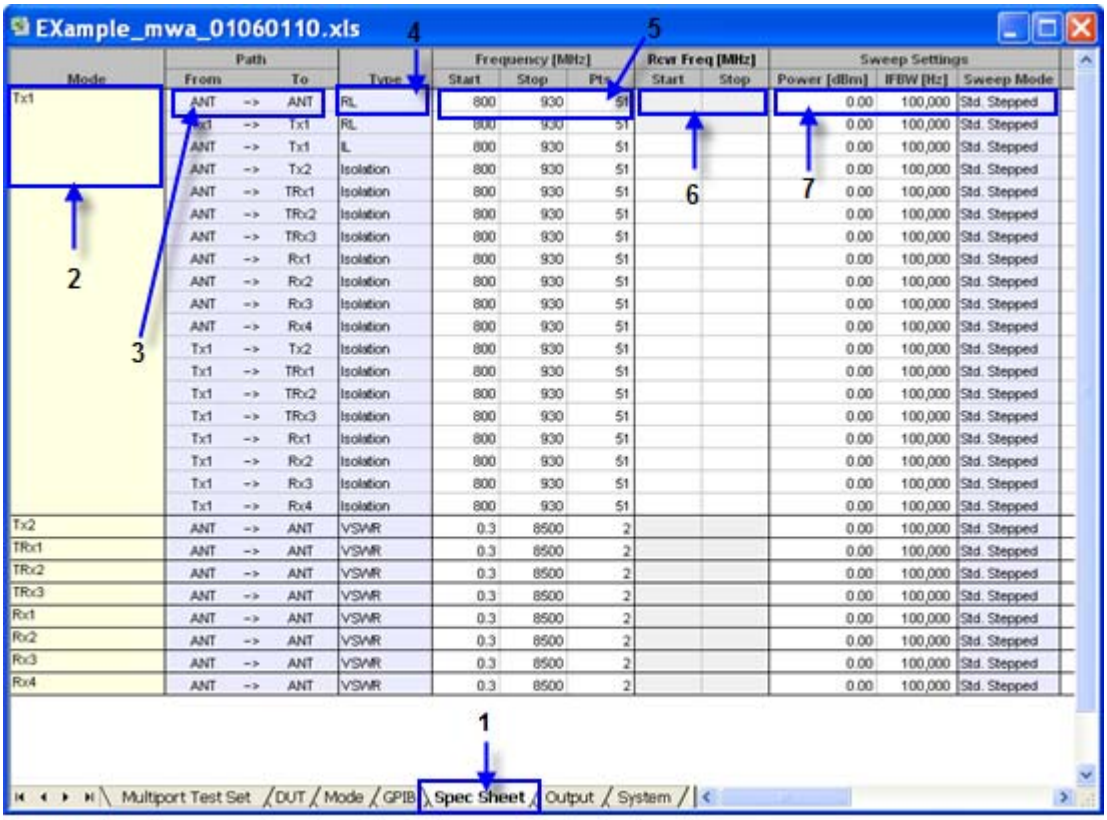
e5071c265

4. GPIB sheet

The SCPI commands can be sent to peripherals connected to the ENA via GPIB.

5. Spec Sheet sheet

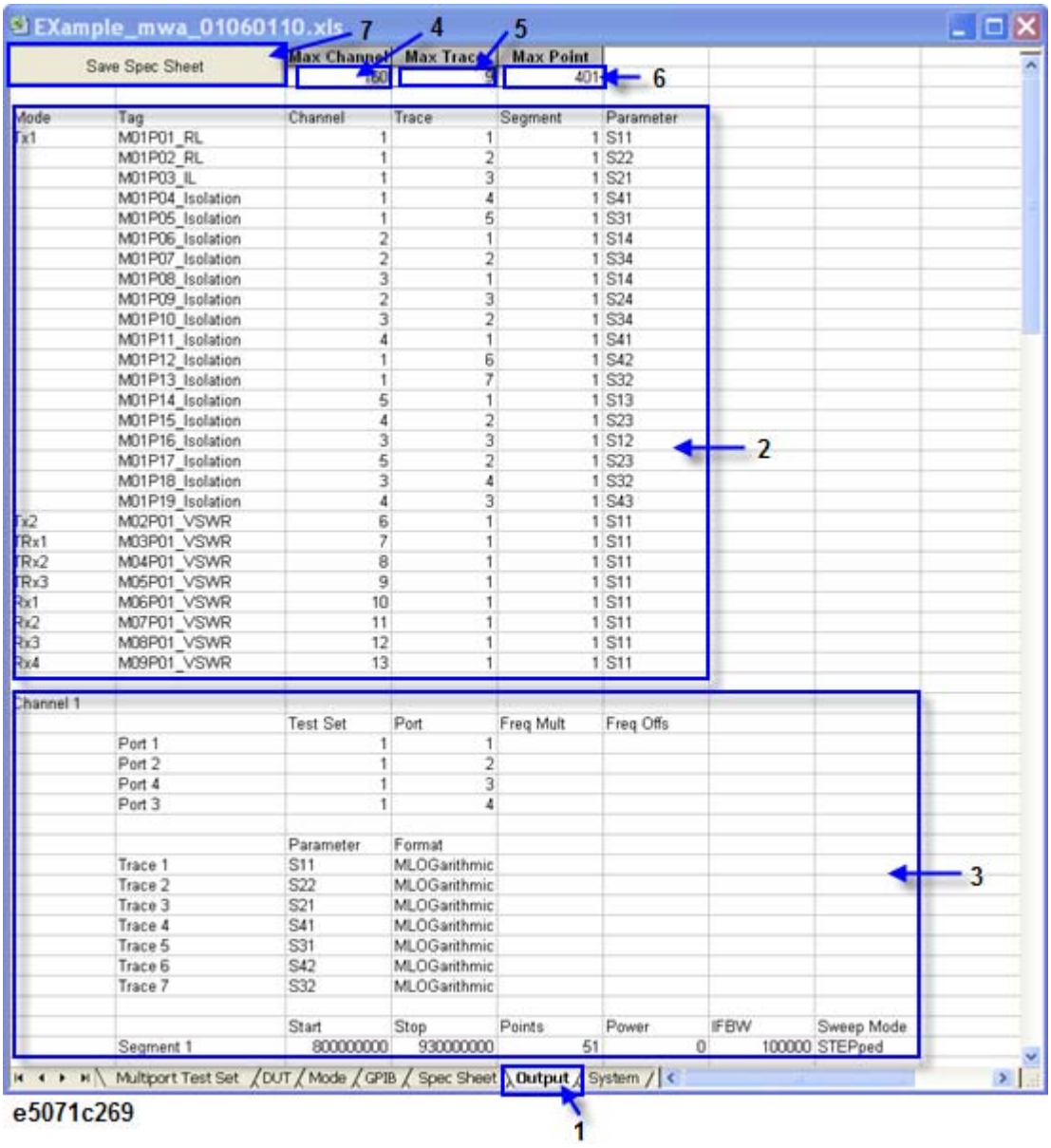
Transmission measurement is performed between all paths for each operation mode of the DUT.




e5071c267

6. Output sheet

A spec sheet is generated by selecting the “Save Spec Sheet” button in the sheet. The traces are allocated automatically in measurement channels.

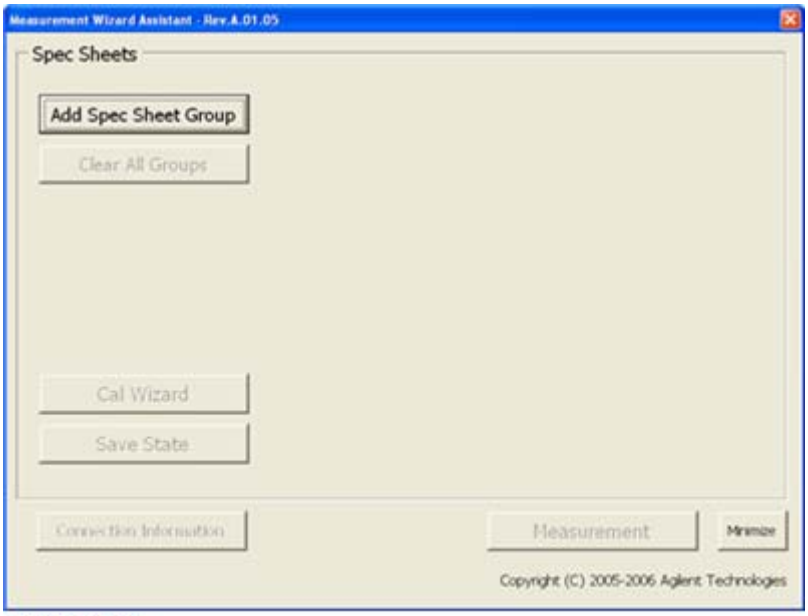



**Back-End Application**

The Back-end application is installed on the E5071C in which the data in mwa file will be imported as per follows:

1. Importing a spec sheet

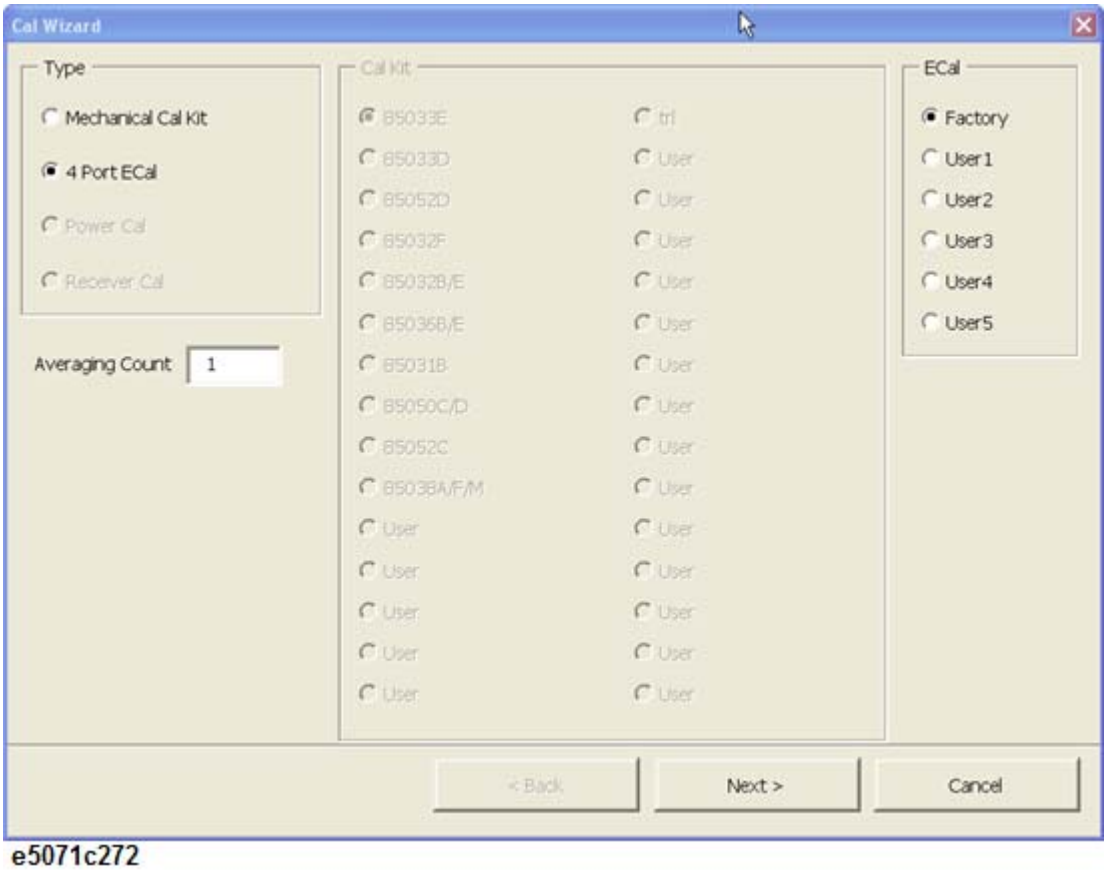
Select "Add Spec Sheet Group" in the main window, and select "Add Spec Sheets.." in the displayed window.



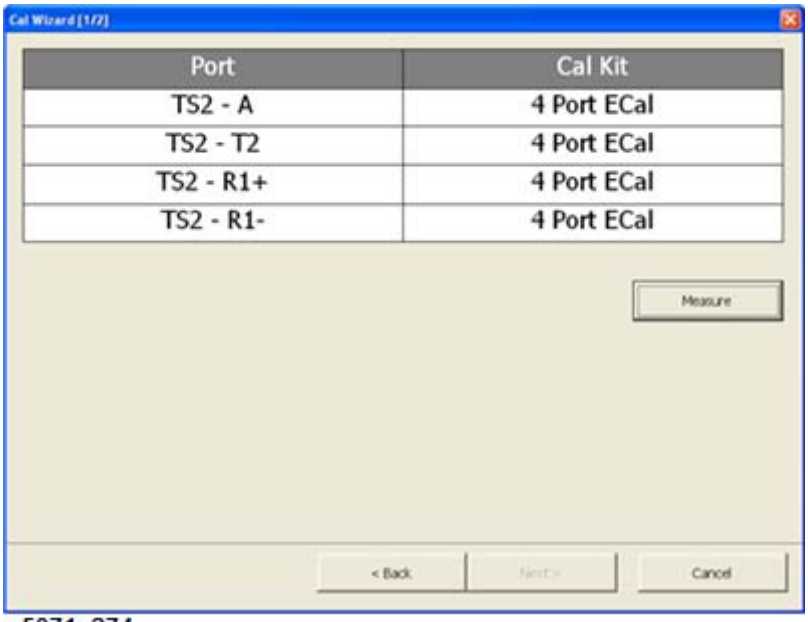

e5071c254

2. Performing calibration

Do a series of calibration measurement by following “step-by-step” calibration wizard.



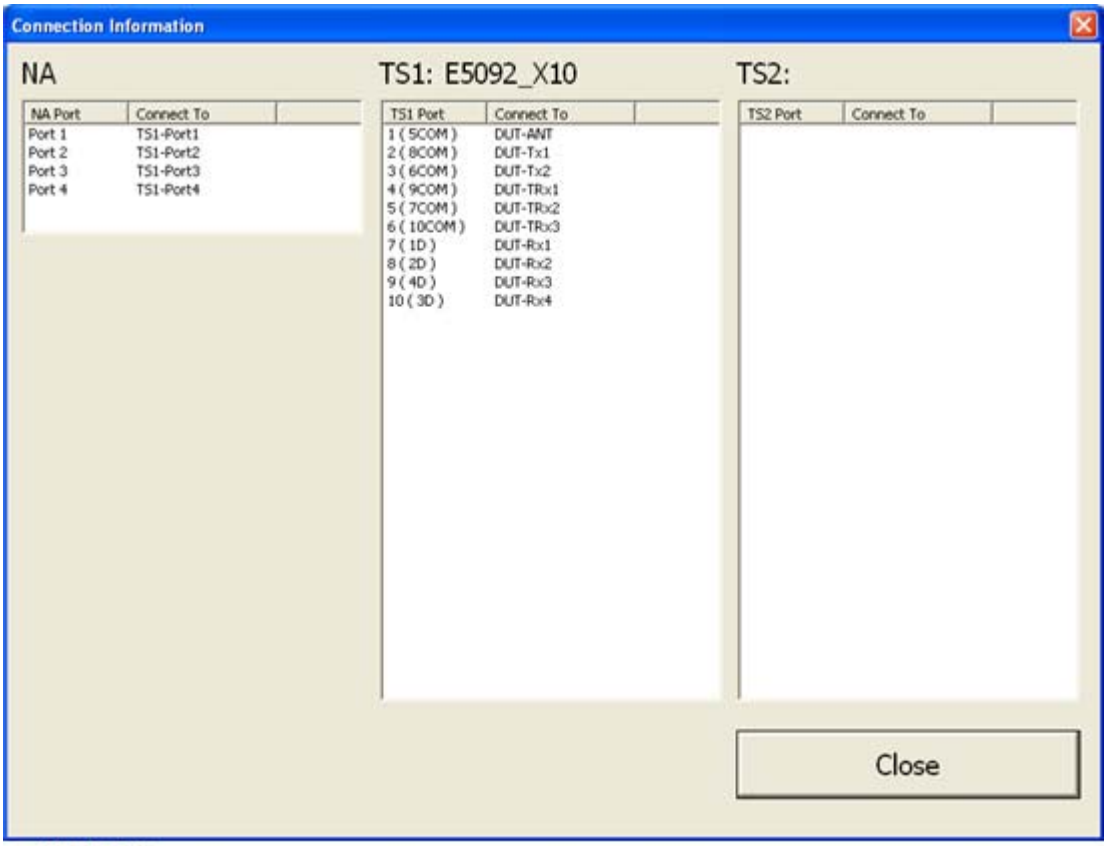

e5071c272




e5071c274

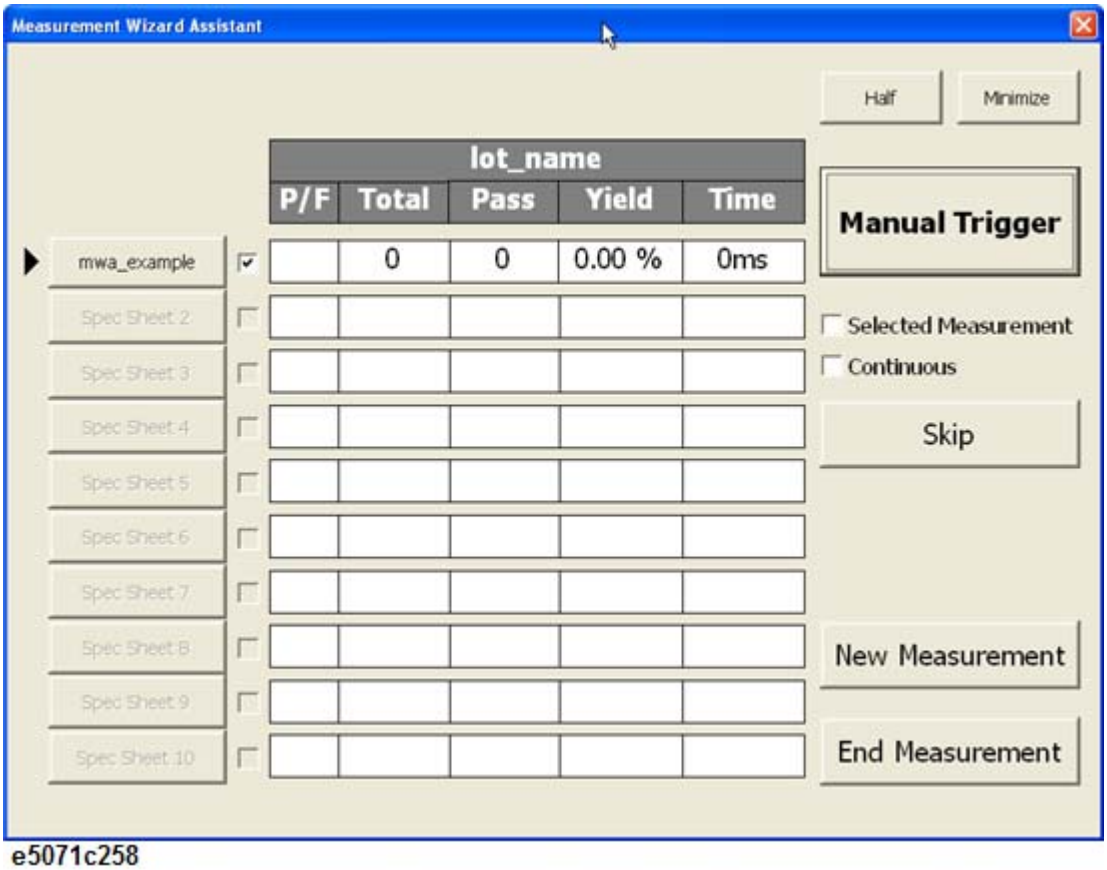
3. Connect the DUT and the test set

The connection between the DUT and the multiport test set is indicated in the connection check window.




e5071c257

4. Performing measurement


Measurement Wizard Assistant
Half
Minimize
lot_name
P/F
Total
Pass
Yield
Time
mwa_example
0
0
0.00 %
0ms
Spec Sheet 2
Spec Sheet 3
Spec Sheet 4
Spec Sheet 5
Spec Sheet 6
Spec Sheet 7
Spec Sheet 8
Spec Sheet 9
Spec Sheet 10
Manual Trigger
Selected Measurement
Continuous
Skip
New Measurement
End Measurement

e5071c258

## Overview and Restrictions of Group



Other topics about Measurement Wizard Assistant

### Overview of Group Function

The back-end application has a "Group" function that enables multiple measurements using spec files.

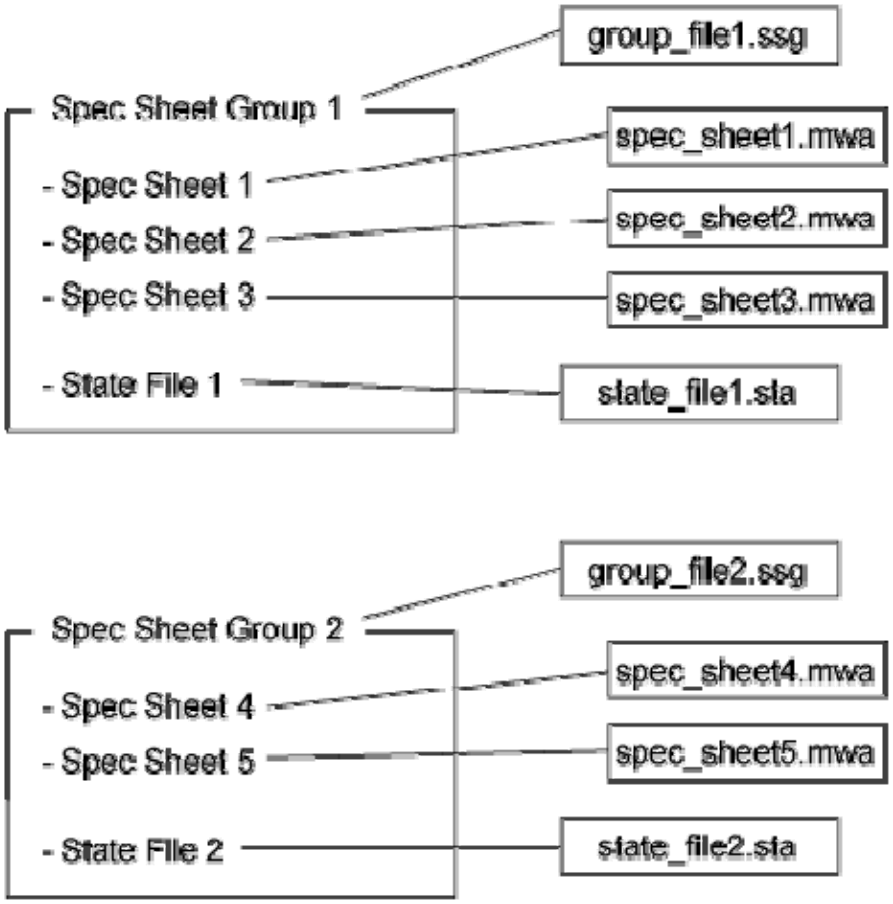
For the back-end application, the following information items are gathered together as a group.

- Multiple spec files
- E5071C settings (status file)

Each group requires a group file (extension: ssg) that has links to files associated with each group file as shown in the following figure. The group file has no information on the contents of spec files nor the E5071C settings but has link information to those files.

NOTE If there is no linked file, an error occurs in the process of recalling the group file.

***Concept of Group and Link***




### Restrictions of Group Function

Restrictions on using the group function are given below.

- A maximum of 10 groups can be defined at a time.

Because the MWA can use up to 10 spec files, assuming one spec file is assigned to each group, the maximum number of groups that can be recalled at a time is 10.

- The file name of a spec file used in one group cannot be used for spec files of other groups.

The MWA cannot recall spec files of the same name at one time, so groups that have spec files of the same name cannot be recalled simultaneously.

- The maximum number of channels for each group is the same as that of the E5071C.

Because the status file associated with the group is recalled with the E5071C, the maximum number of channels for one group is limited to that of the E5071C.

### Active Group

Each group may have a quite different E5071C state, so it is necessary to know which group's state is indicated by the current E5071C status.

This "current E5071C state" is called the "active group." The E5071C keeps the same settings as the active group.

To change the active group, select the radio button to the left of the spec file name displayed in the main window.

If a group file contains more than two spec files, select the spec file at the top of the group. Other spec files are grayed-out and cannot be selected.

When the active group is changed, the E5071C state also changes. This means that the status file associated with each group is recalled by using the E5071C recall function.

NOTE The recall process of the E5071C takes a certain time, as does changing the active group.

The calibration wizard function and the status file save function are executed for the active group. When running the calibration wizard with multiple groups recalled, the calibration is performed for the current active group. When you need to calibrate other groups, change the active group and run the calibration wizard function again. This process must also be done for the status file save function.

NOTE The active group functions during the measurement. You need to switch the active group in order to measure other groups. When multiple groups are recalled, however, the active group automatically changes when the group is changed.

### Relationship between Group File and Status File

One status file is associated with each group. To ensure smooth use of the MWA, read the following information on the status file.

#### Relationship between Group File and Status File

The results of the calibration wizard function and the E5071C settings are recorded and saved in "The status file associated with the group" when the **Save State** button is pressed. The status file is recalled when the active group changes.

NOTE The status file is recalled without confirmation of the file name. Also, the file is overwritten without confirmation.

The timing of saving and recalling the status file is given below. All processes are executed for the status file associated with the group.

1. Timing of saving the status file
   - When the group is recalled (after the contents of the spec files are input to the E5071C).

- When the calibration wizard function is terminated
- When the **Save State** button is pressed.

2. Timing of recalling the status file
   - When the group is recalled (before the contents of the spec files are input to the E5071C).
   - When the active group changes.

#### Cautions for Status File

You may have a trouble if the recalled channel/trace setting is different from the current channel/trace setting.
This is because the status file is overwritten in the following process.

1. Recall the status file when recalling the group.
2. Recall fails.
3. The E5071C enters the preset state when the recall fails.
4. Recall the spec file.
5. Change the E5071C state according to the contents of the spec file.
6. Write the E5071C state in the status file.

For example, assume the following conditions:

- The setting of the status file is correct other than the channel/trace setting of the E5071C.
- The calibration is performed normally.

Recall of this status file fails because it differs from the channel/trace setting of the current E5071C, and then the E5071C enters the preset state. At this time, the calibration data are discarded.

In process 6, the status file associated with the group is saved. However, this file name is the same as the original status file name in process 1; therefore, the original status file is overwritten with the contents of "preset state and the settings in accordance with the spec file." The original calibration data are lost as well.

To prevent this from happening, a confirmation dialog is displayed between the processes 3 and 4. Confirm the ENA setting when you fail to recall the status file.

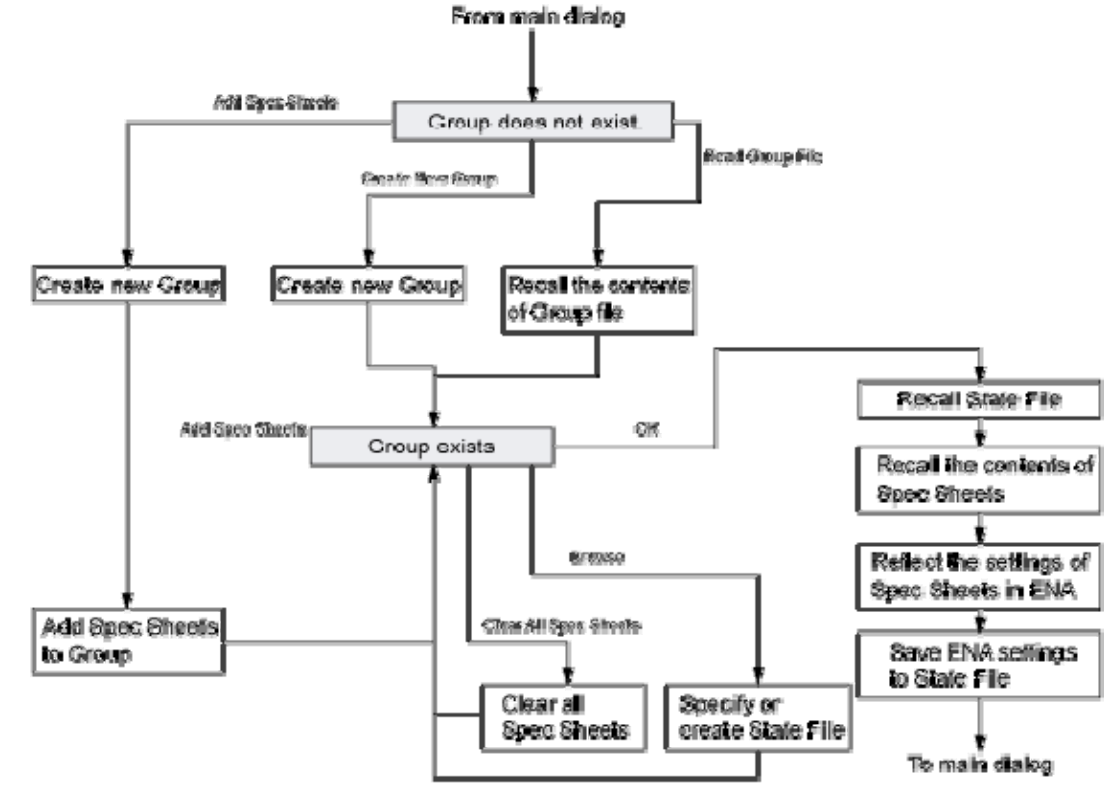
### Process Flow

This section describes the process flow of three states when using the back-end application.

#### Process flow when recalling the spec file

The process flow when recalling spec files is shown below. When using multiple groups, this process is repeated for the number of groups.

***Process flow when recalling the spec file***

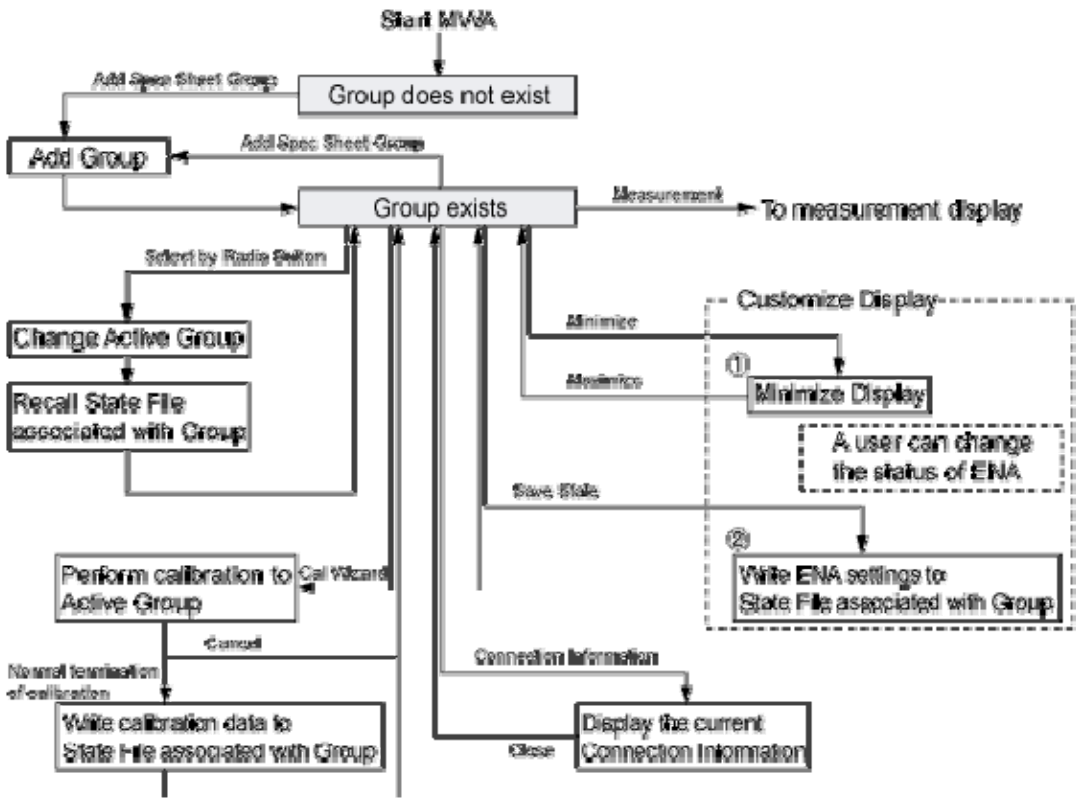



**NOTE** During recalling the status file after the **OK** button is pressed, if the status file associated with the group does not exist, the process goes to the next step, "Recall the spec sheets." There is no intermediary process that changes the E5071C settings such as preset.

**Process flow before the measurement**

The process flow before measurement is shown below.

***Process flow before measurement***




The following five steps are required before starting measurement.

1. Recall spec files (group file)
2. Set up the display with the **Minimize** function and save the state with the **Save State** button.
3. Connect a connector to the calibration edge while checking the connection information screen.
4. Perform calibration with the calibration wizard function.
5. Connect a DUT to the calibration edge while checking the connection information screen.

The display setup and calibration are performed to the current active group. If you use the multiple groups, you need to setup the display and perform the calibration for each group by changing the active group with the radio button.

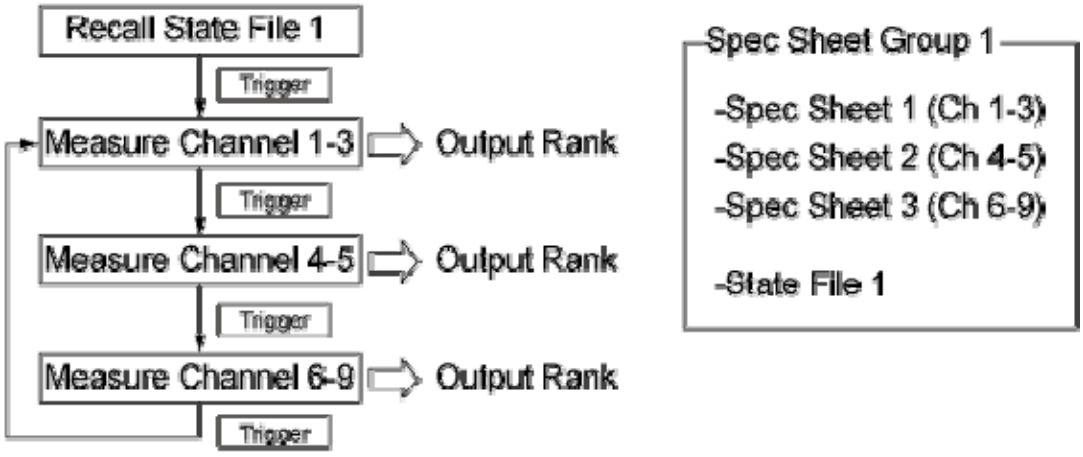
As written in Relationship between Group File and Status File and Cautions, before recalling the spec files of each group, the status file associated with the group is recalled. Therefore, the display setup and the calibration data remain the same as the last time, and thus you can skip this process from the next time.

NOTE It is recommended that you perform calibration every time.

Process flow during the measurement

The process flow for a single group is shown below.

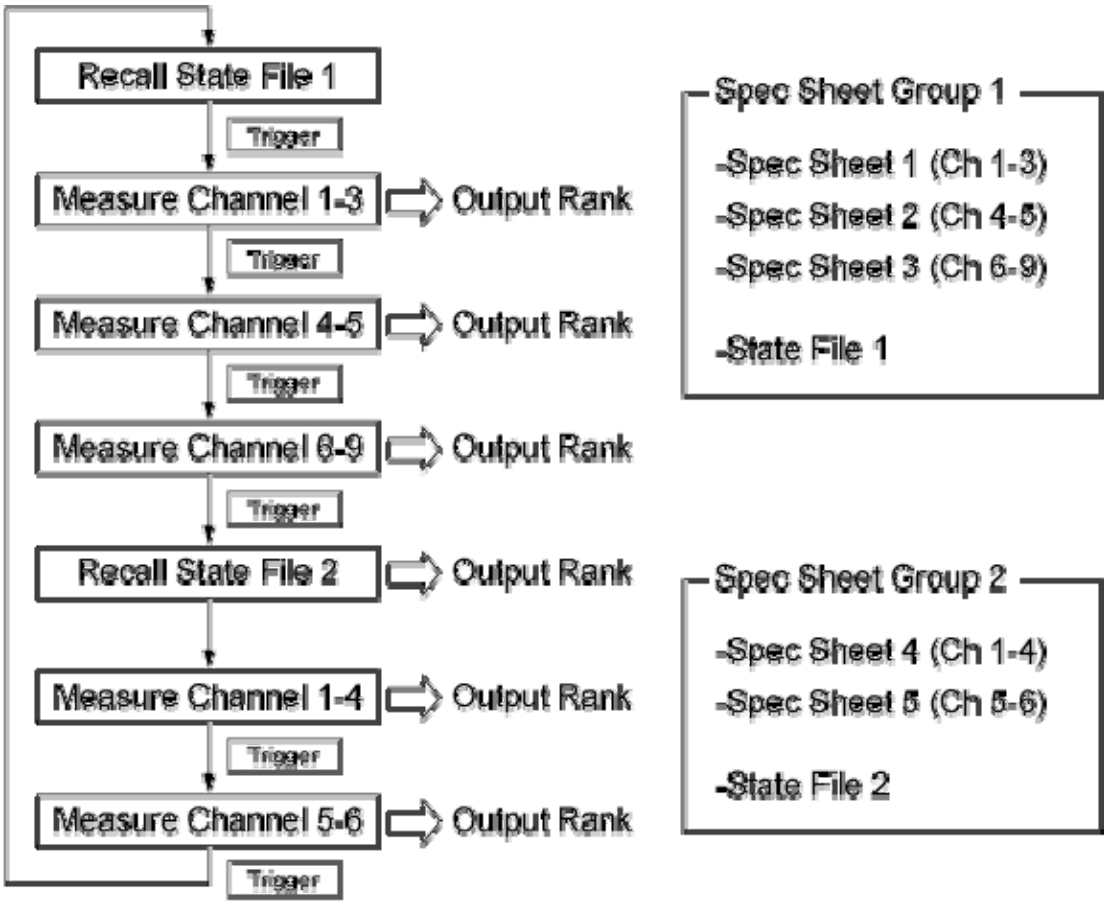
***Process flow during the measurement (single group)***




In this example, three spec files are used, but the process is the same even if there is only one spec file.

The process flow for multiple groups is shown below. The following figure shows the state where State File 1 had been recalled when starting the measurement and the trigger makes each measurement.

***Process flow during the measurement (multiple groups)***




When measuring multiple groups, the active group switches at the following timing.

- After measuring Channel 6-9 of Group 1, the MWA becomes "the Waiting for Trigger" state. When a trigger is generated, the active group switches to Group 2.
- After measuring Channel 5-6 of Group 2, the MWA becomes "the Waiting for Trigger" state. When a trigger is generated, the active group switches to Group 1.

Even after measurement of the last sheet of each group, the active group does not switch to the next group. Since you can switch on/off each spec file during the measurement, if all sheets in the group are off, you do not have to activate the group.

If both Spec Sheet 4 and 5 are off, it is not necessary to move to Group 2, and the measurement of Spec Sheet 1 can be started without recalling

State File 2. This process is the same as that of measuring the single group previously described.

# Enhanced Time Domain Analysis

## Enhanced Time Domain Analysis (Option TDR)

- Overview
- Setting Up Measurement (Setup Tab)
- Making Measurements
- Eye Diagram and Mask Test (Eye/Mask Tab)
- Storing Data and Setting
- Advanced Waveform Analysis
- Advanced Mode
- Measurement Examples

Programming

- Option TDR Remote Control
- Option TDR Command Reference

Other topics about Measurement with Options

## Overview

### Overview

- Features
- TDR Quick Start
- TDR Screen Area
- Consideration of E5071C-TDR Measurements
- Starting and Exiting TDR Application

Other topics about Enhanced Time Domain Analysis

### Features

Agilent E5071C Option TDR provides the following features:

- Up to 20 GHz (Option 2K5/4K5) of bandwidth with 22.3 ps (10%-90%) or 15.9 ps (20%-80%) rise time enables measurement on the latest high speed serial standards.
- Wide dynamic range to observe the true performance of the DUT
- Low noise floor for accurate and repeatable measurements
- Fast measurement speed for real-time analysis
- State-of-the art calibration techniques reduce measurement errors
- Automatic deskew ensures easy removal of fixture and probe effects
- Full calibration available for the utmost in measurement accuracy
- Quickly obtain accurate TDR/TDT and S-parameter measurements
- Easily locate source of loss, reflections and crosstalk by simultaneous analysis of both time and frequency domain
- Single connection forward and reverse transmission and reflection measurements
- All possible modes of operation (single-ended, differential and mode conversion)
- Measure just the device by utilizing advanced calibration techniques to remove cable, fixtures and probe effects
- Gain insight into high speed interconnect performance through simulated eye diagram analysis and manual scale of eye diagram.
- Apply industry standard (PRBS, K28.5) or used specified patterns using the virtual bit pattern generator
- Pre-defined masks for many high speed serial standards
- No need for pulse generators as the eye diagram is synthesized from measurement results
- Hot TDR measurement which allows TDR measurement to be performed while the device is powered ON.

See http://www.agilent.com/find/ena_tdr for more information.

Other topics about Overview

## TDR Quick Start

TDR Quick Start

TDR Quick Start helps you to understand the E5071C-TDR option's operation quickly.

- TDR/TDT Measurement
- Simulated Eye Diagram

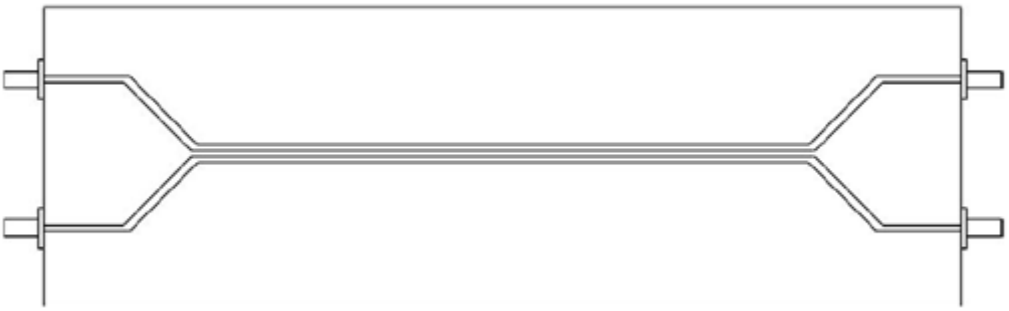
Other topics about Overview

TDR/TDT Measurement

To better understanding how to use the E5071C-TDR option, this section describes the basic TDR/TDT measurement procedure using the E5071C.

Operation procedure

1. Connect 3.5 mm SMA cables to all test ports.
2. Click the **Setup** tab.
3. Click the **Setup Wizard** (under **Basic**) at **Setup** area.
4. Set the measurement condition using the **Setup Wizard**:
    a. Select **Deskew & Loss Compensation** (under **Error Correction**) and click **Next >**.
    b. Click the **Differential 2-Port** button, and click **Next >**.
    c. Click the **Deskew** button, and click **Next >**.
    d. Connect a Thru between Port 1 and Port 3. Click the **Measure** button, and click **Next >**.
    e. Connect a Thru between Port 2 and Port 4. Click the **Measure** button, and click **Next >**.
    f. Connect a Load to Port 1 and click the **Port 1** button to measure the Load. Repeat this step for Port 2, Port 3 and Port 4. When finish, click **Apply** to save the measurement. Click **Next >**.
    g. Connect the DUT to cables. Click the **Measure** button, and click **Next >**.

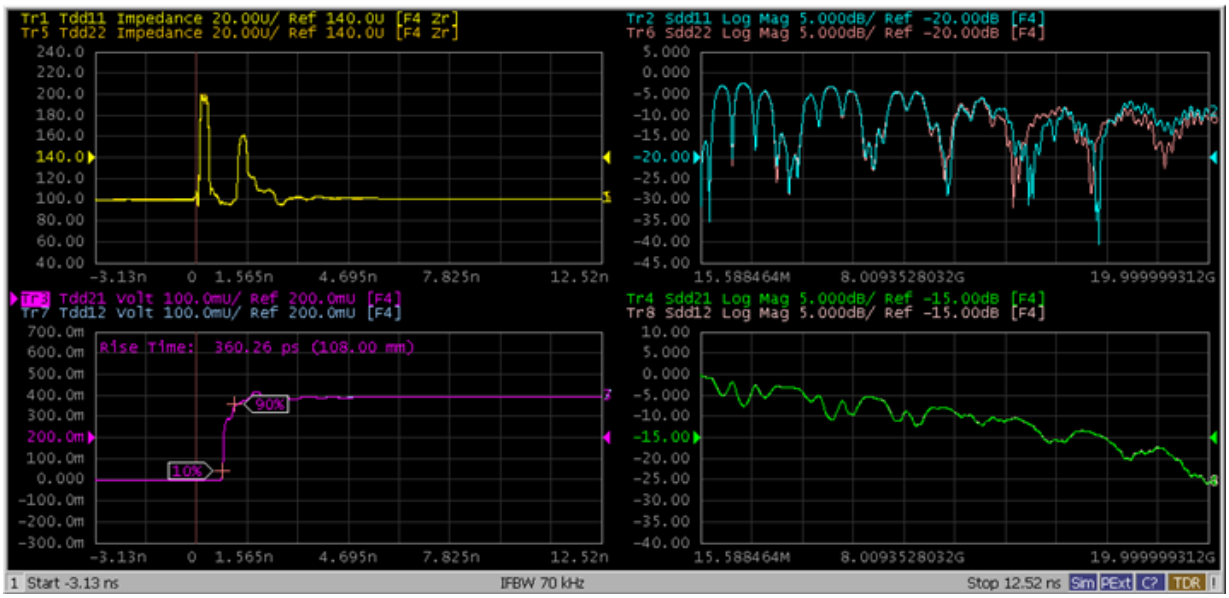


    a. Set the **Rise Time** to "35 ps" and select "10-90%" from the **Definition** drop-down list. When finish, click **Apply** to save the settings.
    b. Click the **Finish** button.
5. Click the **Auto Scale** button.
6. Select **All Traces** from the drop-down list.
7. Select Trace 3.
8. Click the **Marker Search** button.

9. Select **Rise Time (10-90%)** from the drop-down list.

Expected result

Trace 1 shows the TDR measurement in terms of impedance and Trace 3 shows TDT measurement in terms of voltage. The rise time of Trace 3 will be shown.




Other topics about TDR Quick Start

### Simulated Eye Diagram

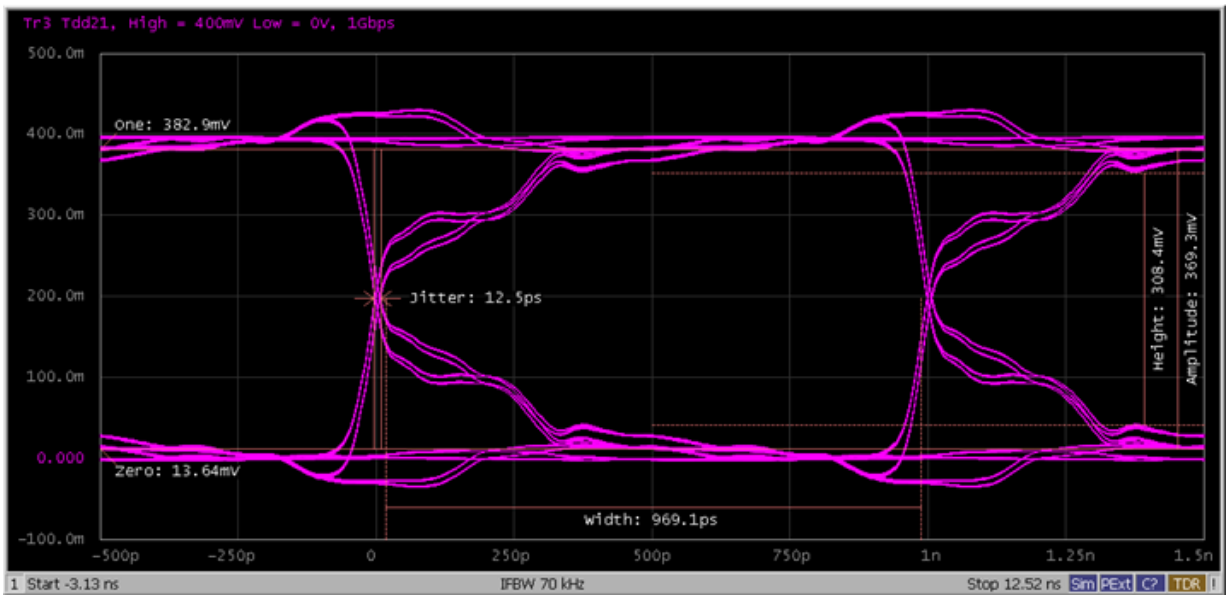
This section describes the procedure of simulated eye diagram using the E5071C with TDR option.

Operation procedure

1. Connect 3.5 mm SMA cables to all test ports.
2. Click the **Setup** tab.
3. Click the **Setup Wizard** (under **Basic**) at **Setup** area.
4. Set the measurement condition using the **Setup Wizard**:
    a. Select **Deskew** (under **Error Correction**) and click **Next >**.
    b. Click the **Differential 2-Port** button, and click **Next >**.
    c. Click the **Deskew** button, and click **Next >**.
    d. Connect the DUT to cables. Click the **Measure** button, and click **Next >**.
    e. Set the **Rise Time** to "35 ps" and select "10-90%" from the **Definition** drop-down list. When finish, click **Apply** to save the settings.
    f. Click the **Finish** button.
5. Click **Trace** button at the tool bar and select "3" to activate Trace 3.
6. Click **Eye/Mask** tab.
7. Under the **Bit Pattern** tab:
    a. Select "PRBS" from the **Type** drop-down list.
    b. Select "2^7-1 bits" from the **Length** drop-down list.
    c. Set the **One Lv.** (amplitude) to "200 mV".
    d. Set the **Data Rate** to "1 Gb/s".
8. Click the **Draw Eye** button under **Eye Control**.

Expected result

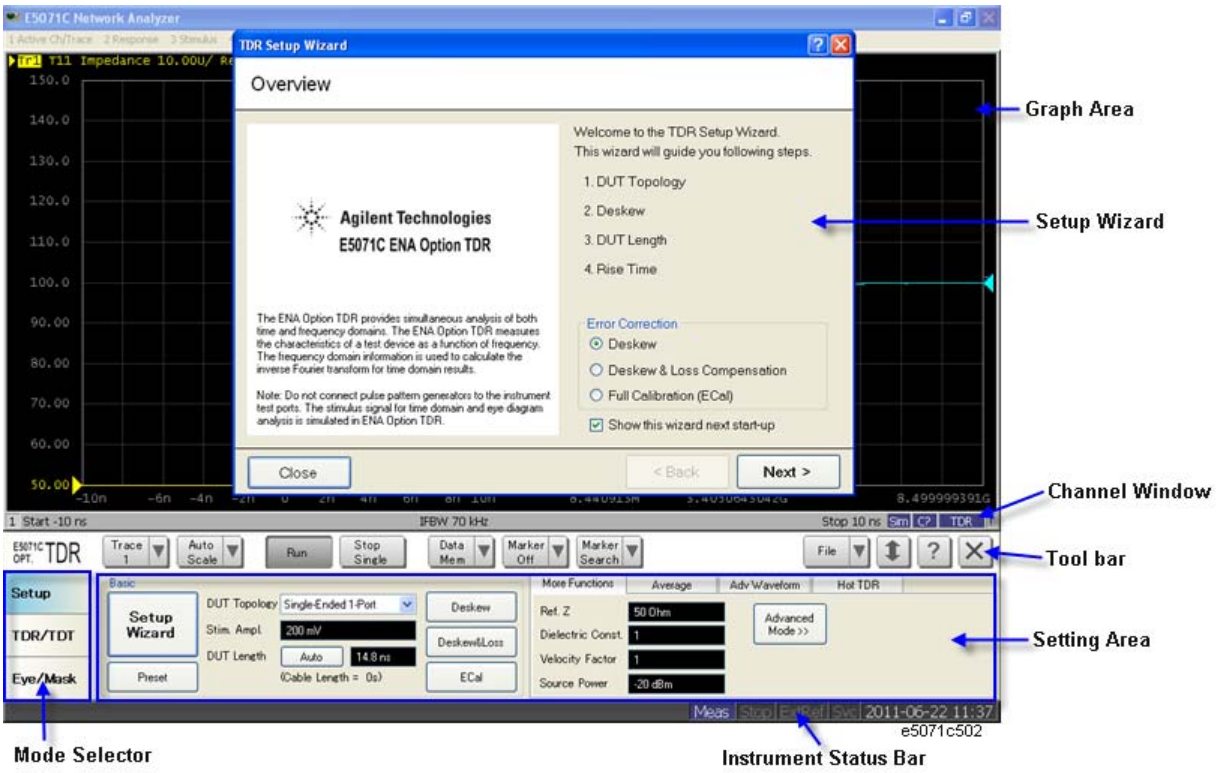
Eye diagram (Trace 3) appears at the graph window.




## Other topics about TDR Quick Start

## TDR Screen Area

TDR screen area appears at the bottom of the standard E5071C screen.




### Graph Area

The results of measurements such as Time Domain, S-Parameter, and Eye are displayed here. Operation with mouse is accepted.

### Setup Wizard

Using Setup Wizard

### Instrument Status Bar

Instrument Status Bar

### Channel Window

In addition to the standard E5071C status bar, there's TDR indicator at the right corner of the channel window. This indicator is turned ON only when TDR option is turned ON. The TDR indicator is blue in basic mode and changes to yellow when advanced mode is turned ON. This indicator also changes according to the Cal status, as shown in the following:

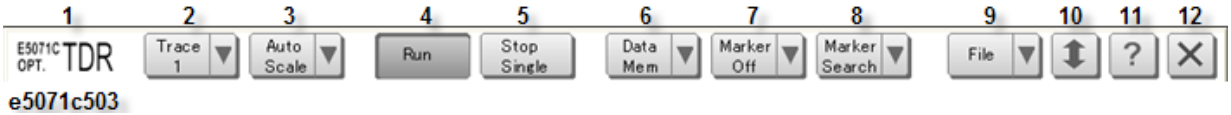
| Cal | Indicator |
|---|---|
| Off | TDR |
| Deskew | TDR [Deskew] |

| | |
|---|---|
| DLC | TDR [DLC] |
| ECal | TDR [Full] |
| ECal&Deskew | TDR [Full+] |

| Others | Indicator |
|---|---|
| Hot TDR Mode | TDR ? |

**Tool Bar**

Frequently used standard E5071C functions are displayed here. These functions are synchronized with E5071C-TDR options and modes.




e5071c503

| No | Functions | Details |
|---|---|---|
| 1 | About E5071C-TDR | About E5071C-TDR |
| 2 | Active Trace | Setting up Parameters on Each Traces |
| 3 | Auto Scale | Using Scale/Zooming |
| 4 | Run | Controlling Trigger |
| 5 | Stop/Single | Controlling Trigger |
| 6 | Data Mem | Using Data/Memory |
| 7 | Marker | Using Marker/Marker Search |
| 8 | Marker Search | Using Marker/Marker Search |
| 9 | File | Storing Data and Setting |
| 10 | Minimize | Minimizes the Setting Area. Only tool bar is displayed. |
| 11 | Help | Executes E5071C-TDR Online Help |
| 12 | Exit | Exiting E5071C-TDR application GUI |

NOTE Buttons 1 to 5 and 9 to 12 are always displayed regardless of the selected mode. Button 6 to 8 does not appear in the **Eye** mode, and the measurement parameter is displayed on the surface of Button 3 (instead of **Auto Scale**).

**About E5071C-TDR**

Click this first icon on the tool bar and the **About** dialog box appears.

**Mode Selector**

You can selected one of the three available E5071C-TDR modes:

- Setup
- TDR/TDT
- Eye Mask

The mode changes when one of these tabs are selected. Once selected, the mode is highlighted. The displayed setting area changes corresponding to the selected mode.

**Setting Area**

The setting area changes and is displayed according to the selected mode.

Other topics about Overview

## Consideration of E5071C-TDR Measurements

- Channel and Trace Concepts
- Device Under Test (DUT) Considerations
- Test Cable and Connector Considerations

### Other topics about Overview

#### Channel and Trace Concepts

It is important for oscilloscope users to note the difference in the concept of channel in oscilloscope and network analyzer.

In network analyzer, channel is referred to the window for displaying traces. Because a channel corresponds to a window, it is called a channel window.

On the other hand, the points on the front panel of network analyzer where cables and DUTs are connected are called test ports. These test ports are not channels and do not directly associate with channel windows.

The E5071C, for example allows you to use up to 160 channels (when the number of traces is up to 9) to perform measurement under 160 different stimulus conditions.

For each channel, up to 4, 6, 9, 12 or 16 traces (measurement parameters) can be displayed. Because multiple traces can be displayed for each channel, no feature is provided to link the stimulus conditions between channels, and each channel is always independent of the others. In other words, for the E5071C, you need to set the measurement conditions and execute calibration for each channel you use for measurement.

To learn more about setting the channel and traces in E5071C, refer to Setting Channels and Traces.

Enhanced Time Domain Analysis option allows you to use channel 1 only. Up to 16 traces can be displayed in this option (when Differential 2-Port DUT topology and all T or all S traces are selected).

#### Device Under Test (DUT) Considerations

##### Maximum DUT Lengths

Refer to the technical overview (http://cp.literature.agilent.com/litweb/pdf/5990-5237EN.pdf) for maximum DUT lengths

To convert from DUT length in seconds to distance in free space, multiply the value in time by c, the speed of light in free space. To calculate the actual physical length, multiply this value in free space by Vf, the relative

velocity of propagation in the transmission medium. (Most cables have a relative velocity of 0.66 for a polyethylene dielectric or 0.7 for a PTFE dielectric.)

$$DUT\ length\ (m) = DUT\ length\ (s) \times c \times Vf$$

#### AC coupled (DC cut) DUTs

Caution is required when measuring AC coupled (DC cut) DUTs.

The response after the capacitor cannot be measured correctly, because the capacitor response overlaps the response from the input and may cause measurement errors.

When measuring AC coupled devices, make sure that the following two conditions are satisfied for proper measurements.

- Capacitance should be equal to or greater than 10 nF
- DUT length should be equal to or less than 6.25 nsec (1.25 meters at ε=2.25)

  DUT length is an actual length of DUT, not "DUT Length" in the Setup Tab in the TDR GUI.

When the above conditions are not satisfied, either:

- Short circuit the capacitor
- Provide open termination at one end of the capacitor and measure from the opposite side.

### Test Cable and Connector Considerations

Using high quality cables to connect the DUT is recommended in order to minimize measurement degradation. The cables should have low loss, low reflections, and minimum performance variation when flexed.

Refer to the technical overview (http://cp.literature.agilent.com/litweb/pdf/5990-5237EN.pdf) for maximum test cable lengths.

NOTE When performing deskew and loss compensation, the same type of cable must be used for all test ports, but they do not necessarily have to be of the same length.

When performing deskew or deskew and loss compensation, use a single connector interface type, such as SMA, or Type-N, for all of the ports to obtain the best results.

### Starting and Exiting TDR Application

- Starting TDR Application
- Exiting TDR Application, GUI (Graphical User Interface)

Other topics about Overview

Ensure that TDR application is installed and E5071C-TDR option is enabled.

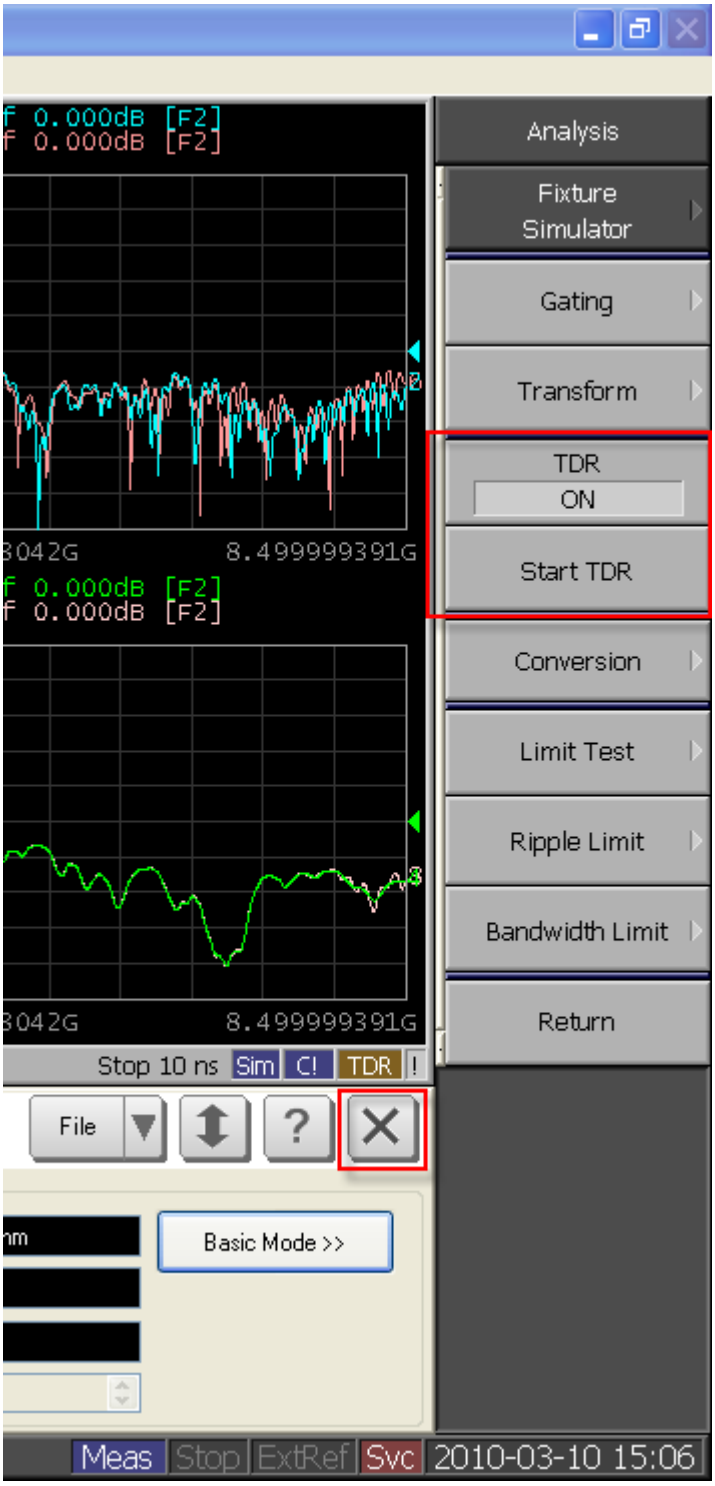
#### Starting TDR Application

- Click **Analysis** > **TDR** to turn on the TDR. The E5071C application restarts.

#### Exiting TDR Application, GUI (Graphical User Interface)

1. Turn on the Advanced Mode.
2. Click **Analysis** > **TDR** to turn off the TDR application. The E5071C application restarts.

Click on the **X** button on E5071C-TDR application GUI to close it. However, the E5071C-TDR application (MODE) is not exited unless you turned off **Analysis** > **TDR**. You can restart the E5071C-TDR application GUI by clicking on the **Analysis** > **Start TDR** key.


0.000dB [F2]
0.000dB [F2]
8.499999391G
0.000dB [F2]
0.000dB [F2]
8.499999391G
Stop 10 ns
Sim
C!
TDR
!
File
Basic Mode >>
Meas
Stop
ExtRef
Svc
2010-03-10 15:06
Analysis
Fixture Simulator
Gating
Transform
TDR
ON
Start TDR
Conversion
Limit Test
Ripple Limit
Bandwidth Limit
Return

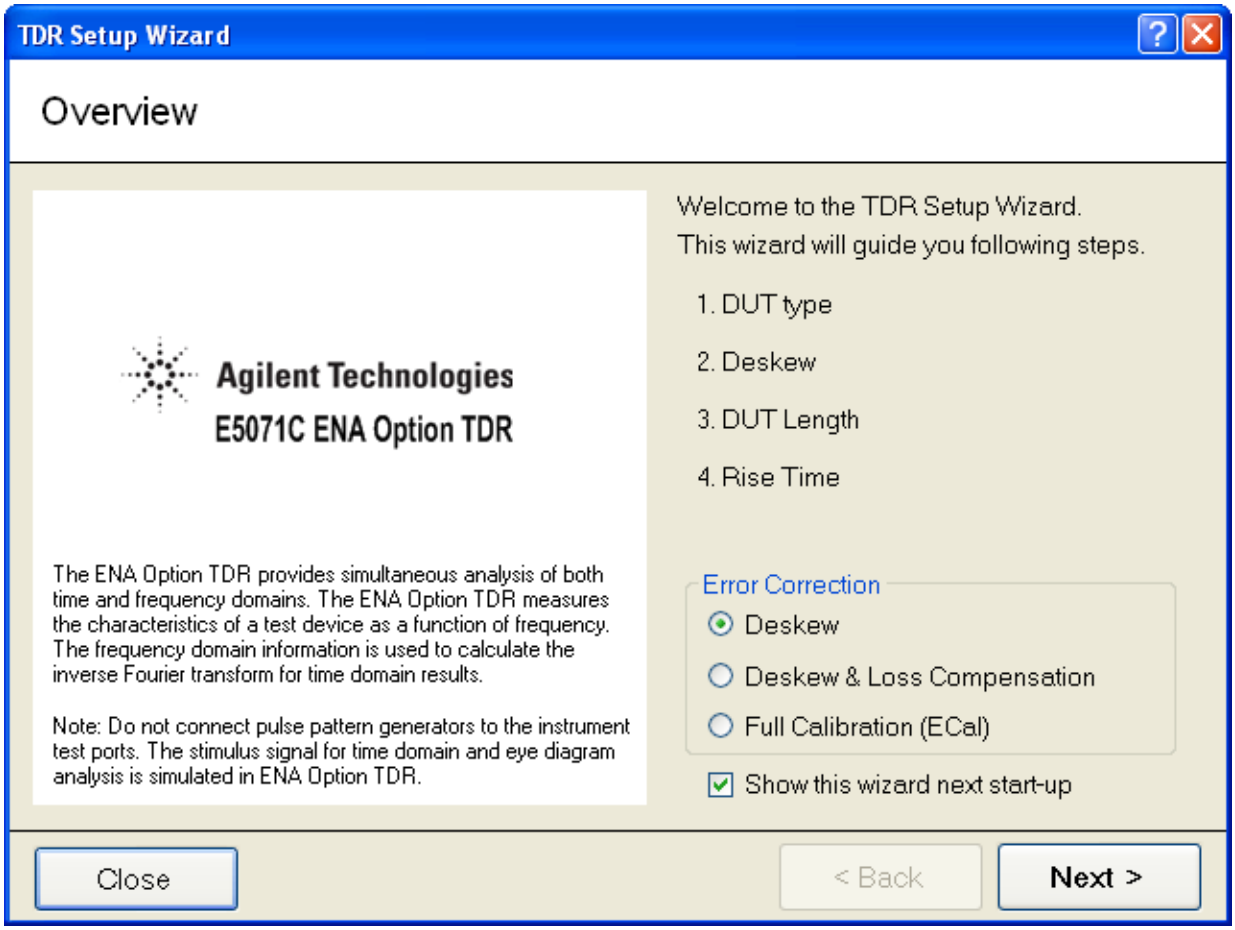
## Setting Up the Measurement

### Setting up Measurement

- Using Setup Wizard
- Performing Manual Setup
- Performing Error Corrections

Other topics about Enhanced Time Domain Analysis

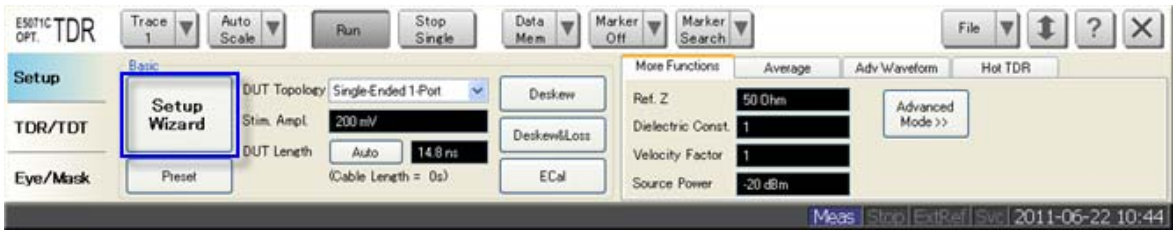
### Using Setup Wizard




**Setup Wizard** guides you to perform the Enhanced Time Domain Analysis measurement setup step-by-step. The wizard appears automatically the first time you execute the Enhanced Time Domain Analysis option. If you check the **Show this wizard next start-up** option, the setup wizard will appear automatically in the next start-up as well.

Another option to start up the Setup Wizard is from the **Setup** tab:

1. Click the **Setup** tab.
2. Click the **Setup Wizard** button under **Basic**, as shown below.



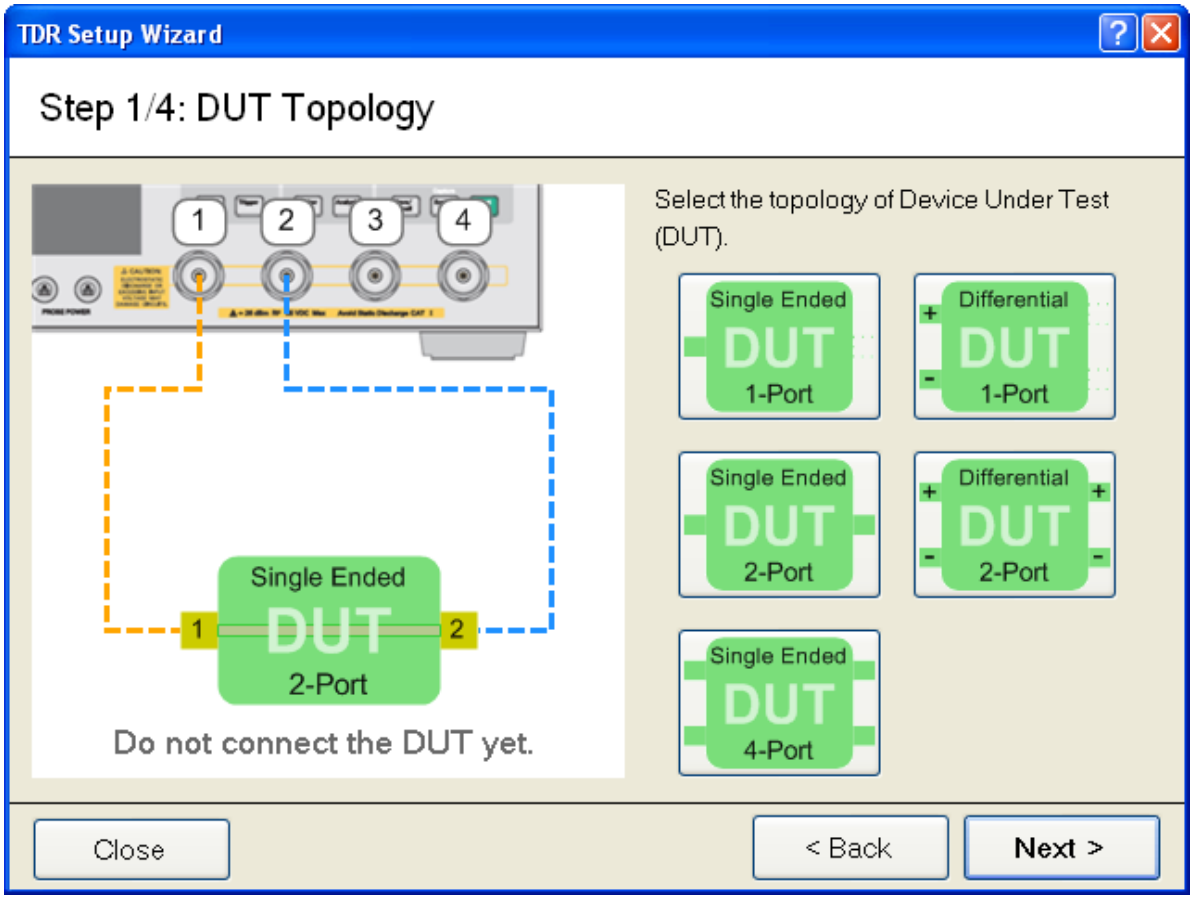

3. If you do not want to perform the Enhanced Time Domain Analysis setup by using the wizard, click **Close** to close the Setup Wizard.

4. Otherwise, select the type of **Error Correction** among the available options:
   - **Deskew**
   - **Deskew & Loss Compensation**
   - **Full Calibration (ECal)**

5. If you check the **Show this wizard next start-up** option, the setup wizard will appear automatically in the next start-up.

6. Click **Next >** to start the setup process.

7. Select the topology of the device under test (DUT).




8. Click **Next >**.

9. The next screen of the wizard varies depends on the selection of **Error Correction** option and topology of the DUT.

10. Follow the instruction on the wizard to complete the setup.

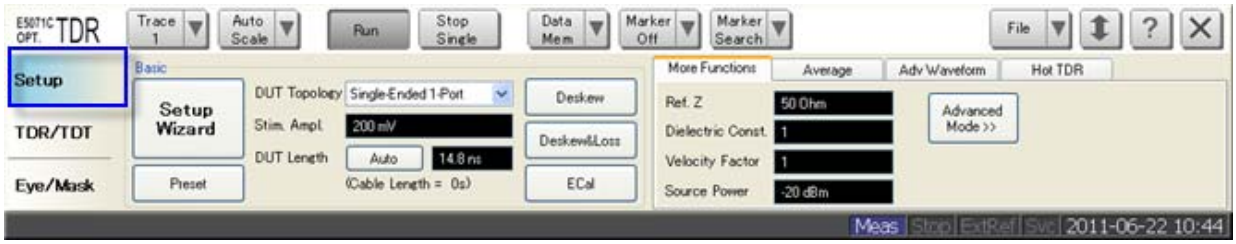
Other topics about Setting Up Measurement

### Performing Manual Setup

Besides Using the Setup Wizard, you can also perform the Enhanced Time Domain Analysis measurement setup manually. The manual setup is performed at the **Setup** tab.

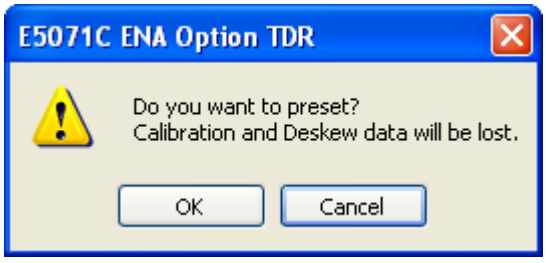
- Preset
- Selecting DUT Topology
- Setting Stimulus Amplitude Level
- Setting DUT Length
- Performing Error Correction
- More Functions
- Average
- Advanced Waveform
- Hot TDR

Other topics about Setting Up Measurement




**Preset**

1. Click **Preset** under **Basic** to preset the E5071C.
2. A dialog box appears requesting for confirmation. Click **OK** to proceed.

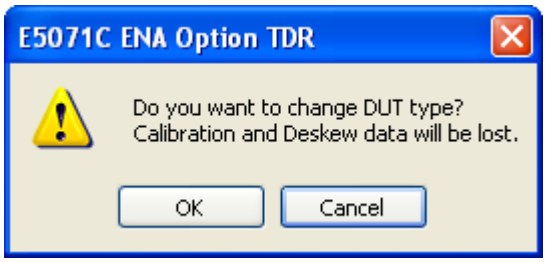



3. All the settings shown in the **Basic** area are changed to default except for the DUT Topology.

NOTE When you click **Preset**, the calibration and deskew data is deleted. Preset sets all of the settings at default except for the DUT Type.

**Setting DUT Topology**

1. At the **DUT Topology** under **Basic**, select one of the available option from the drop-down list box. This is the same function as Step 1/4: DUT Topology in the **Setup Wizard**.
2. A dialog box appears requesting for confirmation. Click **OK** to proceed.



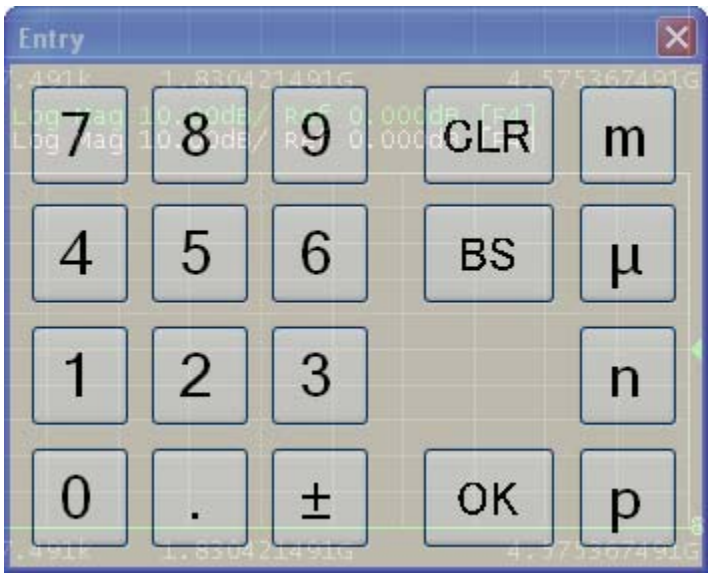

NOTE Selecting the DUT topology executes the preset of the E5071C. Hence, when you change from one DUT topology to another, the calibration and deskew data is deleted.

### Setting Stimulus Amplitude Level

Stimulus Amplitude Level is the value for scaling the time domain measurement result.

NOTE Stimulus Amplitude Level is not related to the actual applied voltage.

1. Click the text box of **Stim. Ampl.** (**Stimulus Amplitude**) under **Basic**. An **Entry** dialog box appears. Type the stimulus amplitude level value and click **OK**. The new value is displayed at **Stim. Ampl.**.




### Setting DUT Length

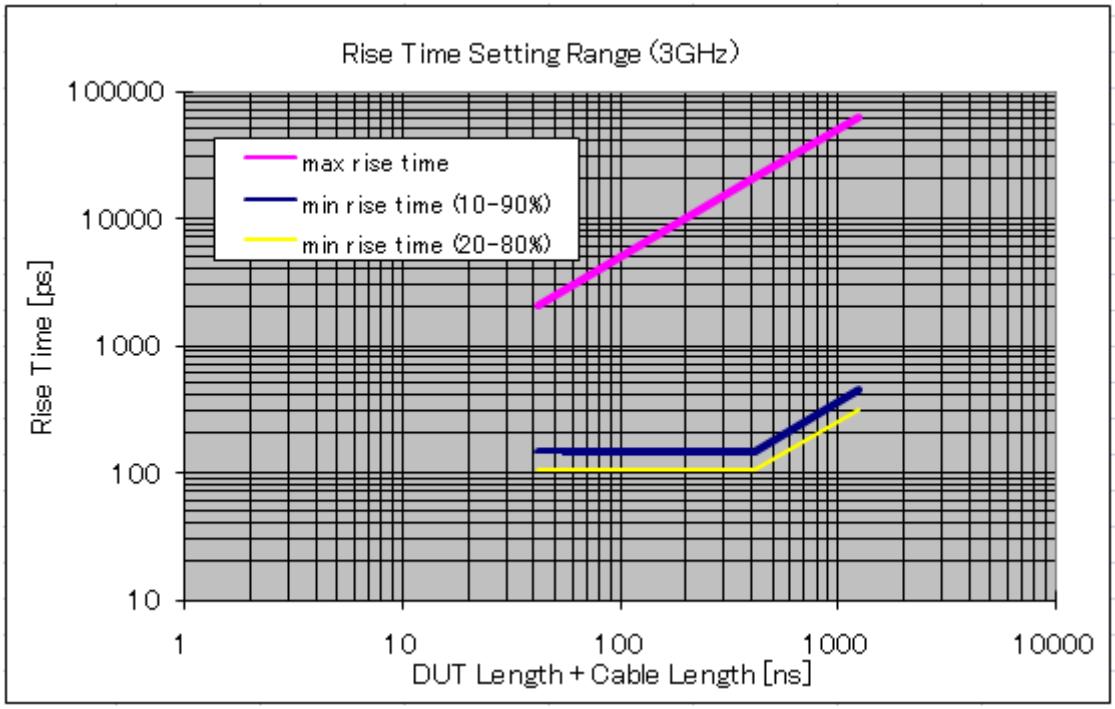
The DUT length setting is used to set the time span for time domain measurements. Measurement can be made on longer DUTs, but minimum rise time values may be limited.

**Minimum and Maximum Rise Time Guideline**

The guideline of minimum and maximum rise time for E5071C 3 GHz, 4.5 GHz, 6.5 GHz, 8.5 GHz, 14 GHz and 20 GHz is outlined below:

E5071C-235/435 (100 kHz to 3 GHz)

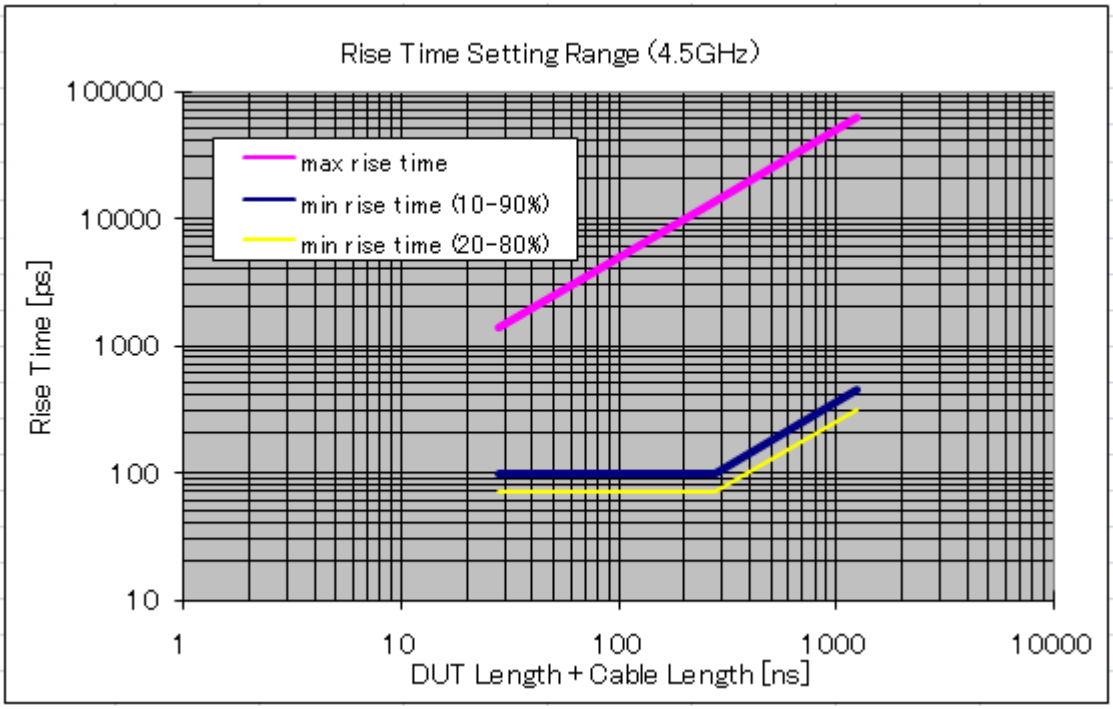
| DUT + Cable [ns] | min rise time (10-90%)[ps] | min rise time (20-80%) [ps] | max rise time [ps] |
|---|---|---|---|
| 41.8 | 149 | 106 | 2090 |
| 416 | 149 | 106 | 20800 |
| 1250 | 446 | 317 | 62500 |




E5071C-245/445 (100 kHz to 4.5 GHz)

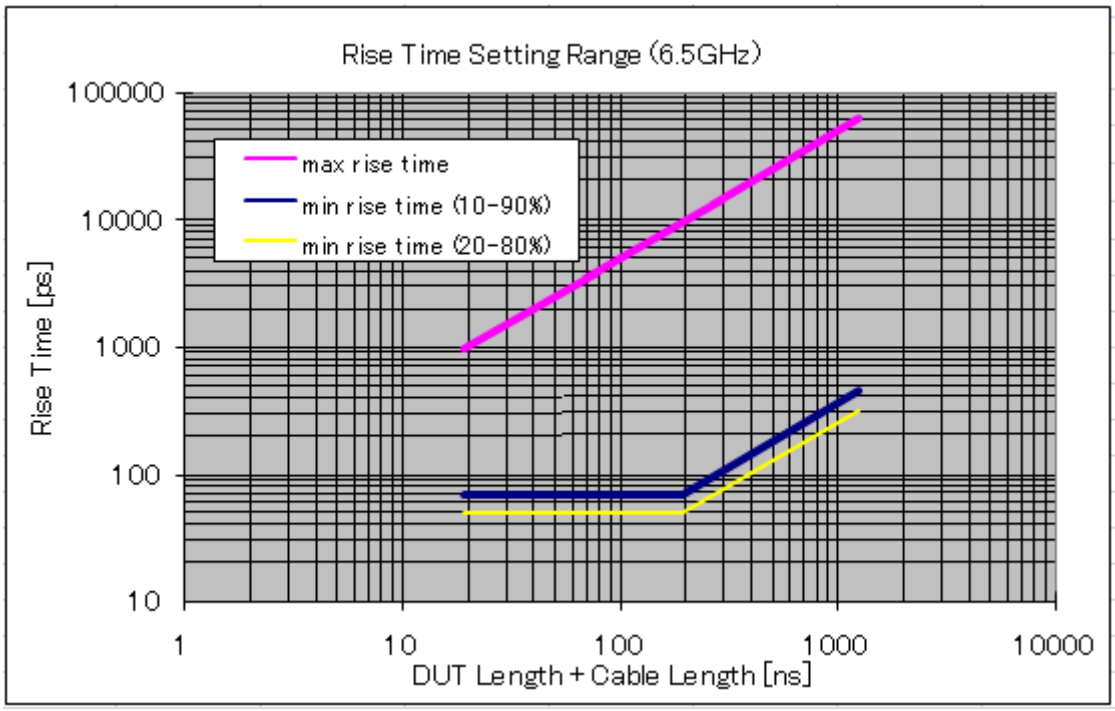
| DUT + Cable [ns] | min rise time (10-90%)[ps] | min rise time (20-80%) [ps] | max rise time [ps] |
|---|---|---|---|
| 27.9 | 99.1 | 70.5 | 1395 |
| 277 | 99.1 | 70.5 | 13850 |

| | | | |
|---|---|---|---|
| 1250 | 446 | 317 | 62500 |



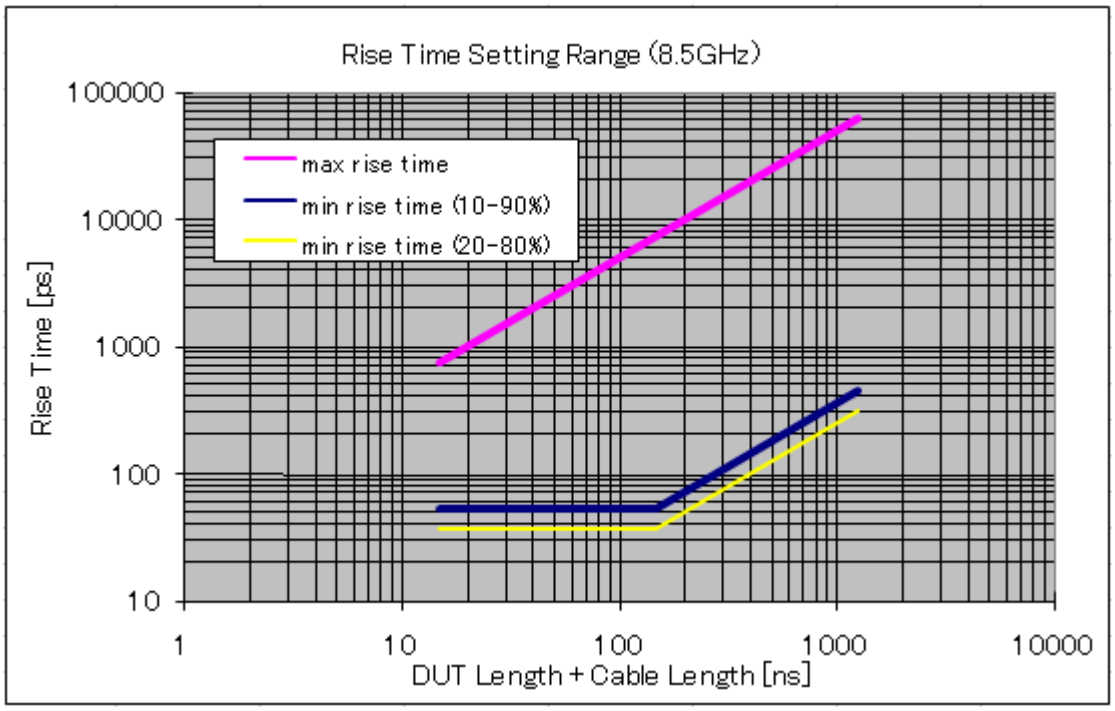

E5071C-265/465 (100 kHz to 6.5 GHz)

| DUT + Cable [ns] | min rise time (10-90%)[ps] | min rise time (20-80%) [ps] | max rise time [ps] |
|---|---|---|---|
| 19.3 | 68.6 | 48.8 | 965 |
| 192 | 68.6 | 48.8 | 9600 |
| 1250 | 446 | 317 | 62500 |



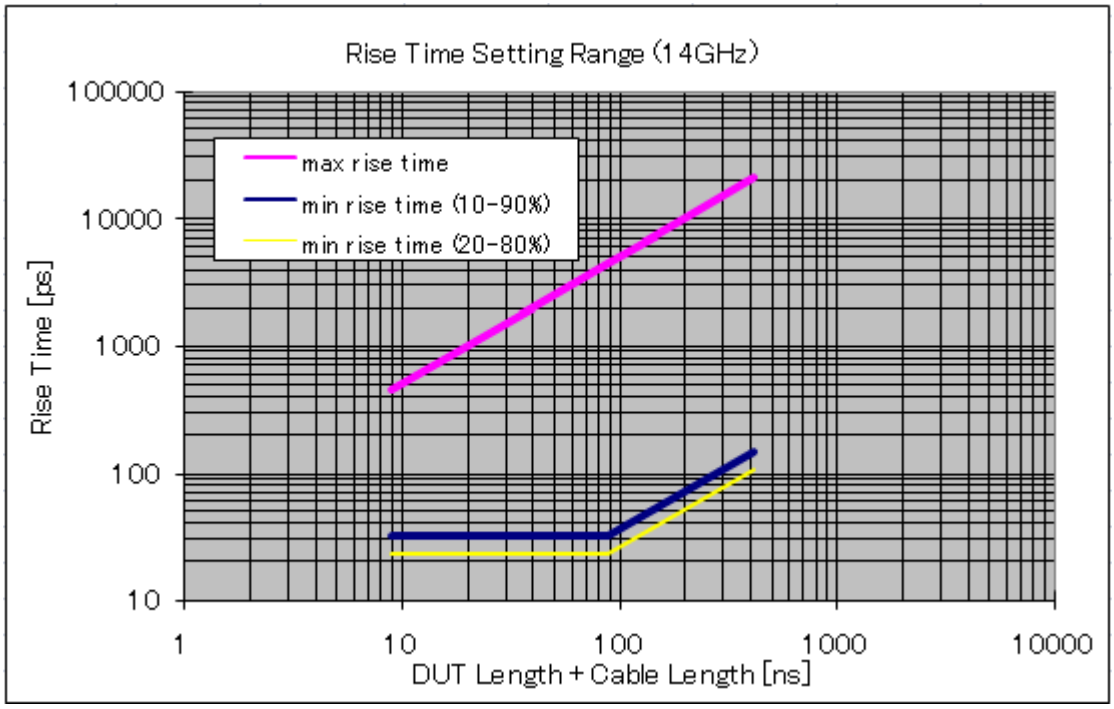

E5071C-285/485 (100 kHz to 8.5 GHz)

| DUT + Cable [ns] | min rise time (10-90%)[ps] | min rise time (20-80%) [ps] | max rise time [ps] |
|---|---|---|---|
| 14.8 | 52.5 | 37.3 | 740 |
| 147 | 52.5 | 37.3 | 7350 |
| 1250 | 446 | 317 | 62500 |



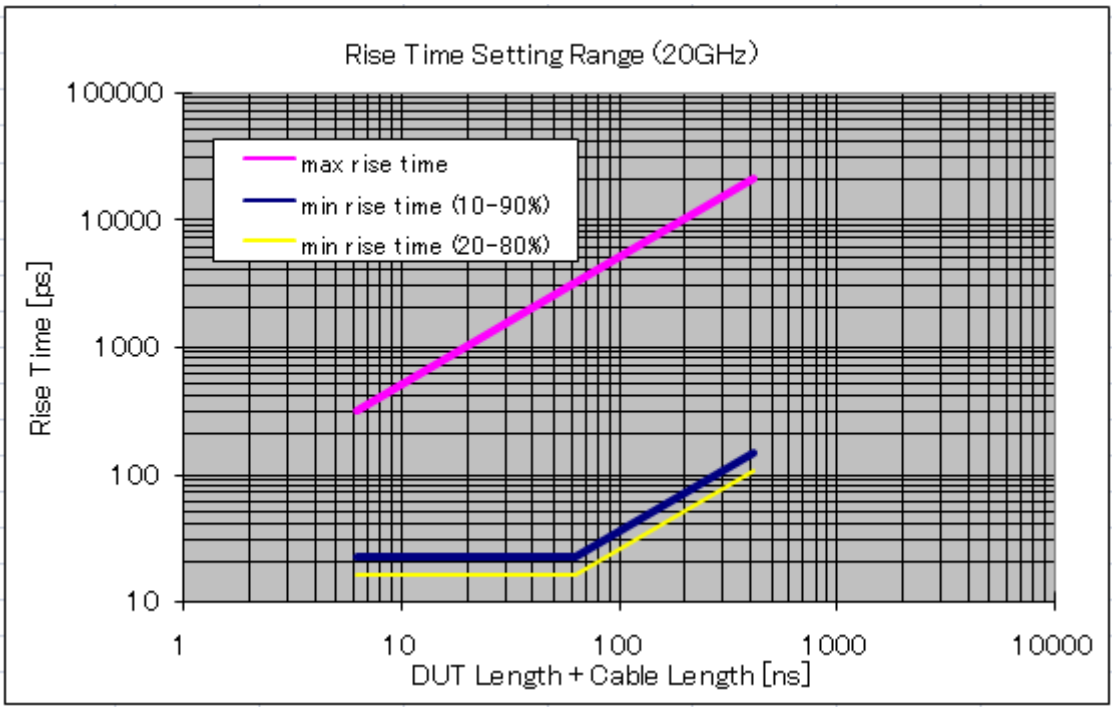

E5071C-2D5/4D5 (300 kHz to 14 GHz)

| DUT + Cable [ns] | min rise time (10-90 %) [ps] | min rise time (20-80 %) [ps] | max rise time [ps] |
|---|---|---|---|
| 8.94 | 31.9 | 22.7 | 447 |
| 89.2 | 31.9 | 22.7 | 4460 |
| 416 | 149 | 106 | 20800 |




E5071C-2K5/4K5 (300 kHz to 20 GHz)

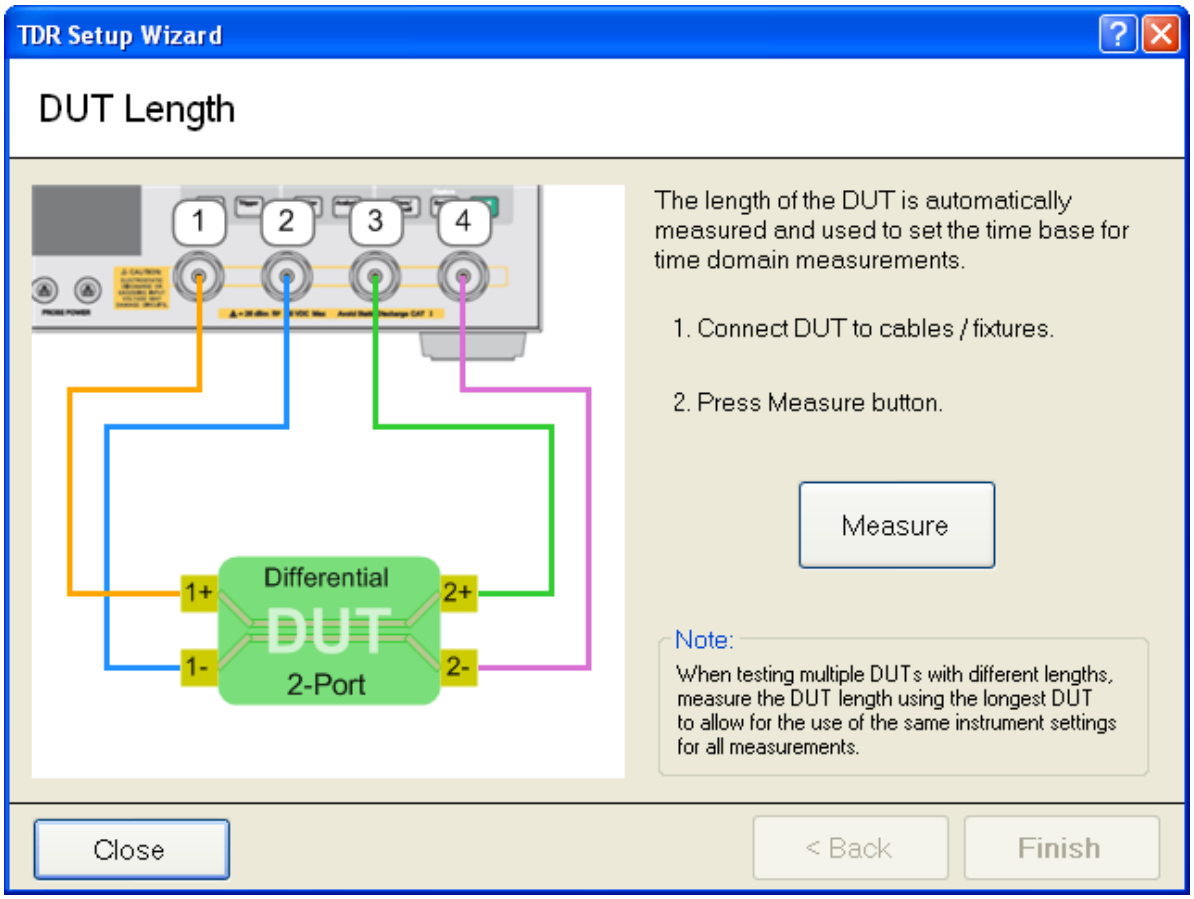
| DUT + Cable [ns] | min rise time (10-90%) [ps] | min rise time (20-80%) [ps] | max rise time [ps] |
|---|---|---|---|
| 6.26 | 22.3 | 15.9 | 313 |
| 62.5 | 22.3 | 15.9 | 3125 |
| 416 | 149 | 106 | 20800 |




The length of the DUT can be set either automatically or manually.

**Automatic measurement - recommended**

1. Click **Auto** at the **DUT Length** under Basic.
2. **DUT Length** dialog box of the **Setup Wizard** appears.
3. Click **Measure**.
4. The length of the DUT is measured automatically and used to set the time base.
5. Once complete, a check mark appears beside the **Measure**.




NOTE The automatic measurement feature is available as one of the steps in the **Setup Wizard.**

**Manual entry**

If the length of the DUT is known, the DUT length can be set manually.

1. Enter the DUT length in the **DUT length text** box under **Basic**.

NOTE Any DUT shorter than the DUT length setting can be measured. Therefore, when testing multiple DUTs with different lengths, set the DUT length using the longest DUT to allow for the use of the same instrument settings for all measurements.

**Performing Error Correction**

There are three error correction options available, as stated below. Refer to Performing Error Correction for more information.

- **Deskew**
- **Deskew and Loss Compensation**
- **Full Calibration (ECal)**

**More Functions**

**Ref. Z**

1. At the **Ref. Z** (**Port Reference Impedance**) under **More Functions**, left-click once. An **Entry** dialog box appears. Type the post reference impedance value and click **OK**. The new value is displayed at **Ref. Z**.

#### Dielectric Const. and Velocity Factor

Velocity Factor = 1 / sqrt(Dielectric Constant).

As such, when you change either one, the value of the other changes automatically.

1. To change the value of dielectric constant, at the **Dielectric Const**. under **More Functions**, left-click once. An **Entry** dialog box appears. Type the Dielectric Constant value and click **OK**. The new value is displayed at **Dielectric Const**.
2. The value of the velocity factor is changed in a similar way at **Velocity Factor** under **More Functions**.

#### Source Power

The signal source level is changed during HOT TDR measurement to avoid device malfunction:

1. At the **Source Power** under **More Functions**, left-click once. An **Entry** dialog box appears. Type the source power value and click **OK**. The new value is displayed at **Source Power**.

#### Advanced Mode

Refer to Advanced Mode.

## Average

The averaging function allows you to reduce the trace noise. It executes the sweep the number of times specified by the averaging factor when the sweep averaging function is turned ON. To activate the averaging option, go to **Average**.

1. To turn ON the averaging factor, select the **Averaging** check box.
2. Type the averaging factor (number of times the sweep needs to be executed).
3. To turn ON the averaging trigger, select the **Average Trigger** check box.
4. At the **IF Bandwidth**, left-click once. An **Entry** dialog box appears. Type the IF bandwidth value and click **OK**. The new value is displayed at **IF Bandwidth**. Reducing IF bandwidth increases the dynamic range.

Advanced Waveform

Refer to Advanced Waveform Analysis.

**Hot TDR**

Refer to Hot TDR Measurement.

## Performing Error Corrections



Other topics about Setting Up Measurement

### Overview

There are many different approaches of removing the effects of the test fixture and cables from the measurement. The level of difficulty for each error correction technique is related to the accuracy of each method. As such, E5071C-TDR gives you the flexibility of selecting the desired error correction.

There are three error correction options available, as stated below:

- **Deskew**
- **Deskew and Loss Compensation**
- **Full Calibration (ECal)**

If you use the **Setup Wizard**, you will be guided though step-by-step to perform the Enhanced Time Domain Analysis measurement setup. This includes performing the error correction. Optionally, you can perform the error correction manually, at the **Setup** tab.

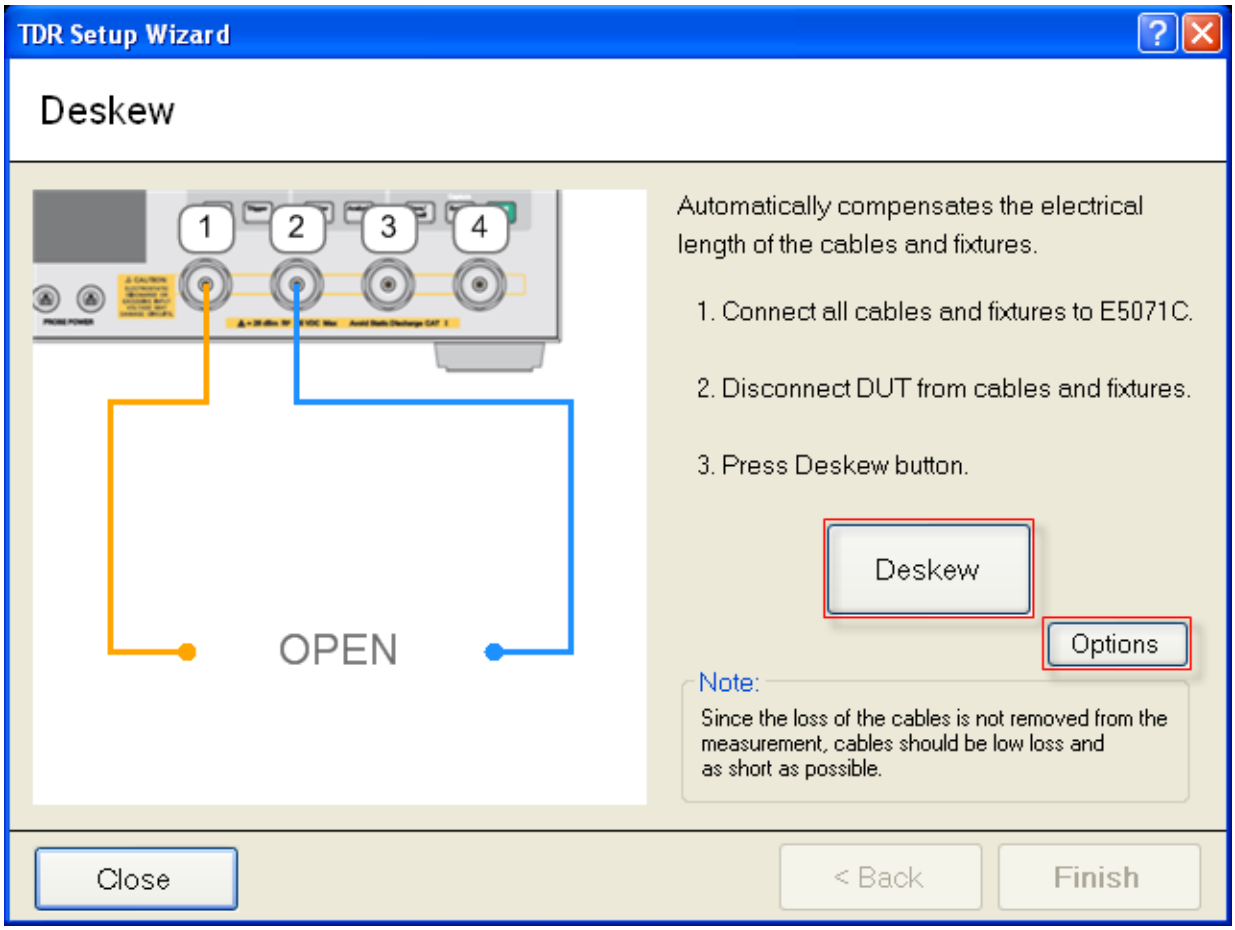

NOTE The TDR indicator at the channel window also shows the TDR deskew and calibration status.

### Deskew

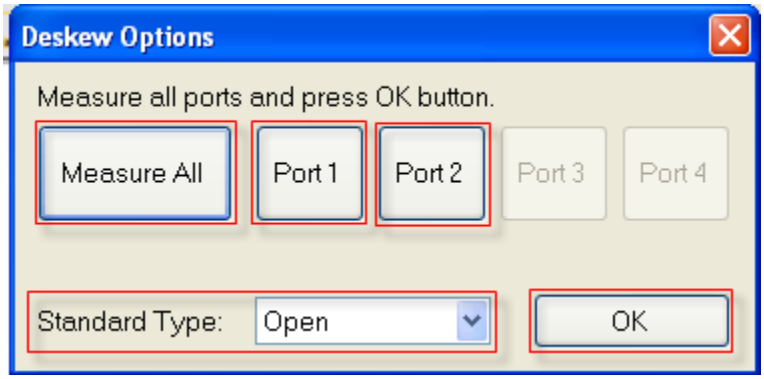
Deskew automatically compensates the electrical length of the cables and fixtures. Deskew mathematically extends the calibration reference plane to the DUT, hence removes the delay from the test setup effectively. This error correction technique provides good results if the cable and fixture are very well designed.

NOTE As deskew does not compensate the loss of cables in the measurement, cables should be low loss and as reasonably short as possible.

1. Click the **Deskew** button under **Basic**.
2. **Deskew** dialog box of the **Setup Wizard** appears.
3. Follow the instruction on the wizard and click the **Deskew** button in the dialog box.




5. Deskew is performed and a check mark appears beside the **Deskew** button.
6. Alternatively, you can deskew one port at a time. To perform this, instead of clicking the **Deskew** button, click the **Options** button in the dialog box.
7. At the **Deskew Option** dialog box, you can click one port at a time to deskew it.



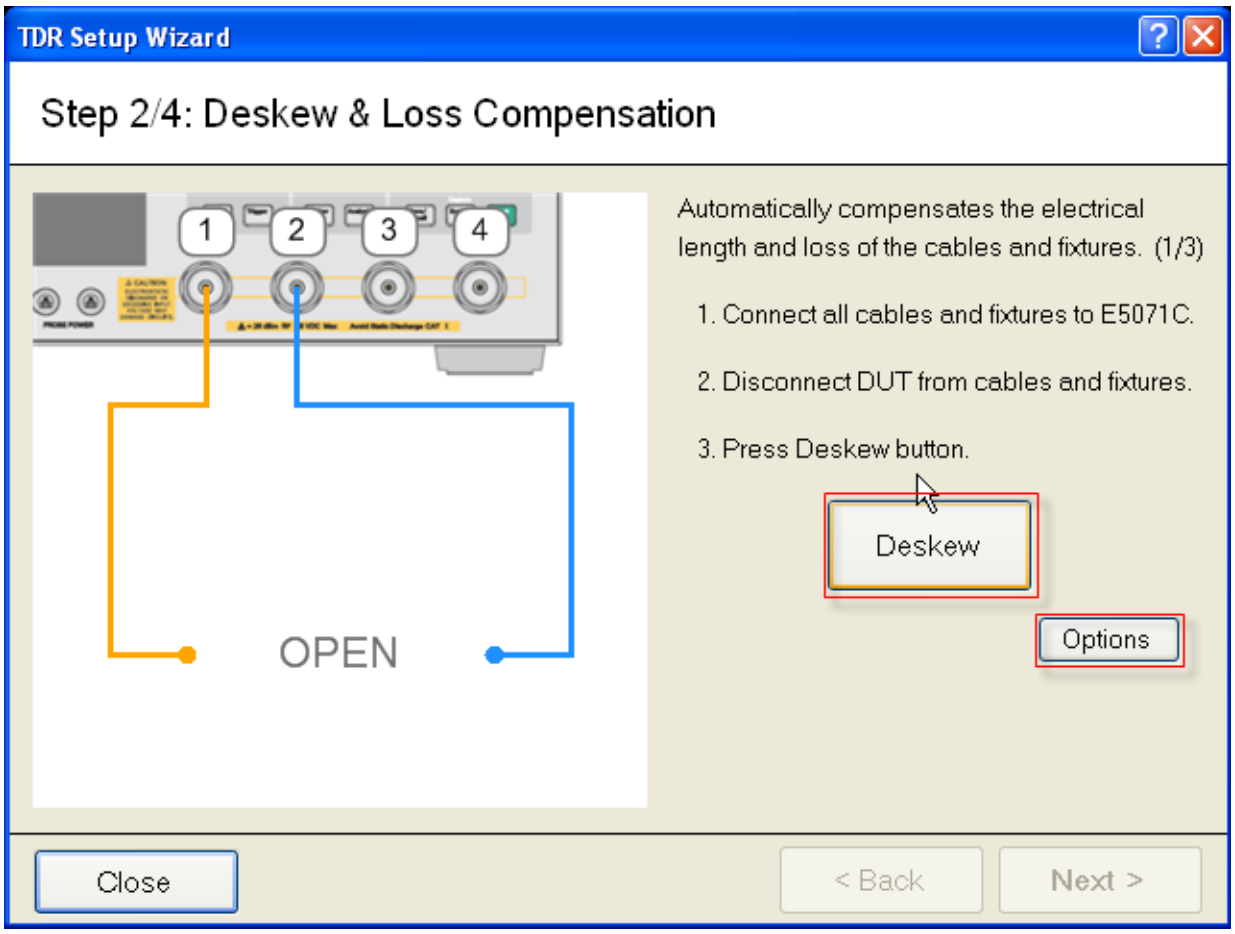

8. Click **Measure All** to perform deskew on all the ports. This action is the same as clicking the **Deskew** button. You must deskew all the active ports.
9. At **Standard Type**, select either **Open** or **Short** deskew.
10. Click **OK** to close the **Deskew Option** dialog box.
11. Once complete, click **Finish** on the dialog box.

### Deskew and Loss Compensation

Deskew and loss compensation mathematically extends the calibration reference plane to the DUT, hence removes the delay and loss from the test setup effectively. This error correction technique is a good compromise between level of difficulty and accuracy.

NOTE The same type of cable must be used for all test ports. However, they do not necessarily have to be of the same length.

1. Click the **Deskew & Loss** button under **Basic**.
2. **Deskew** dialog box of the **Setup Wizard** appears.
3. The Deskew & Loss is a three-step process.
4. Follow the instruction on the wizard and click the **Deskew** button in the dialog box.




5. Alternatively, you can deskew one port at a time, just as in Deskew. To perform this, instead of clicking the **Deskew** button, click the **Options** button in the dialog box.
6. At the **Deskew Option** dialog box, you can click one port at a time to deskew it.
7. Clicking **Measure All** performs deskew on all the ports at the same time. This action is the same as clicking the **Deskew** button. You must deskew all the active ports.
8. At **Standard Type**, select either **Open** or **Short** deskew.
9. Click **OK** to close the **Deskew Option** dialog box.
10. At Step 2, connect a thru between the ports as per the instruction on the dialog box and click **Measure**.

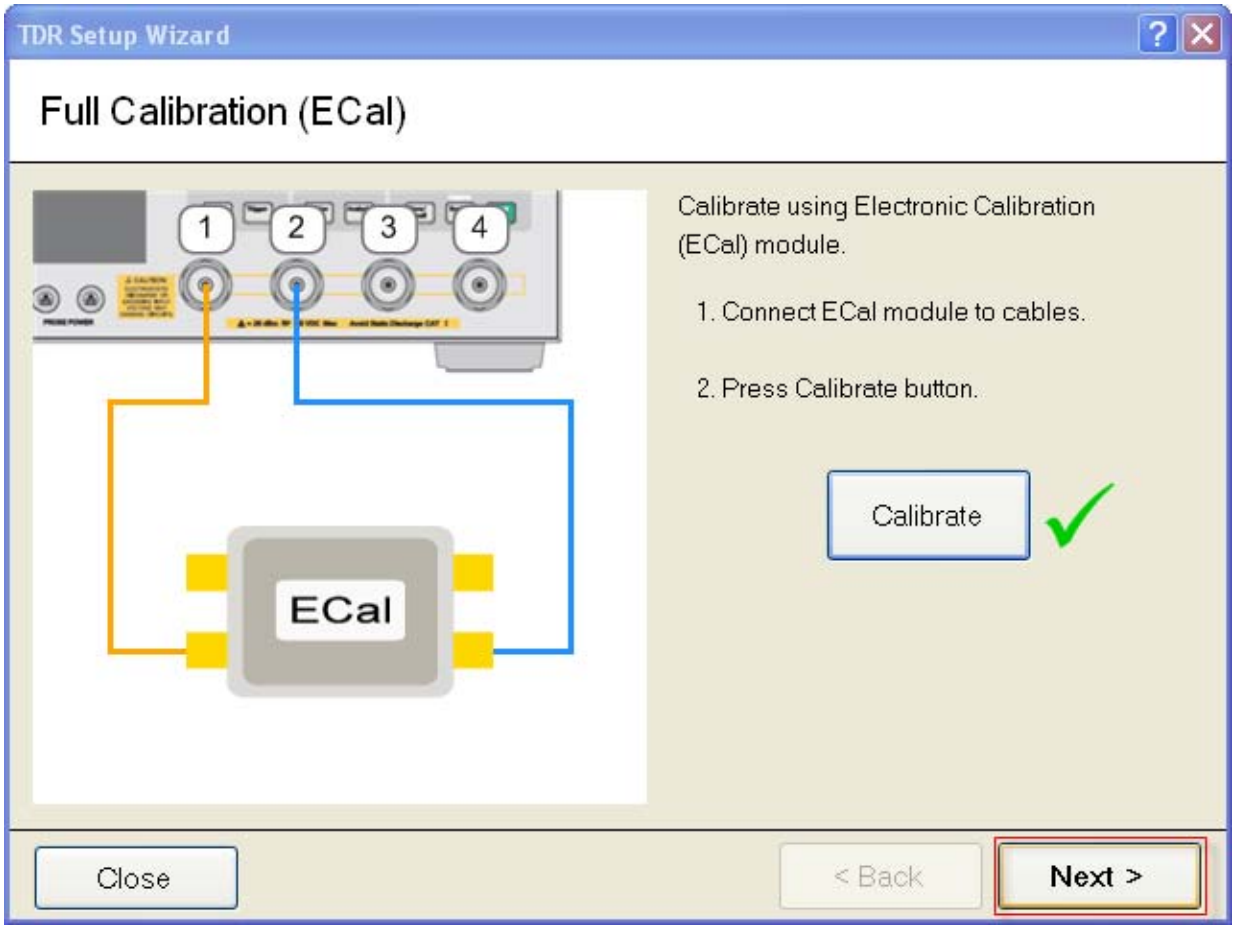
    NOTE Use thru with a short and low loss.
11. At Step 3, connect a load to the ports one by one and click the associate button accordingly. Once complete, click **Apply** to save the measurement.
12. Click **Finish** on the dialog box.

### Full Calibration (ECal)

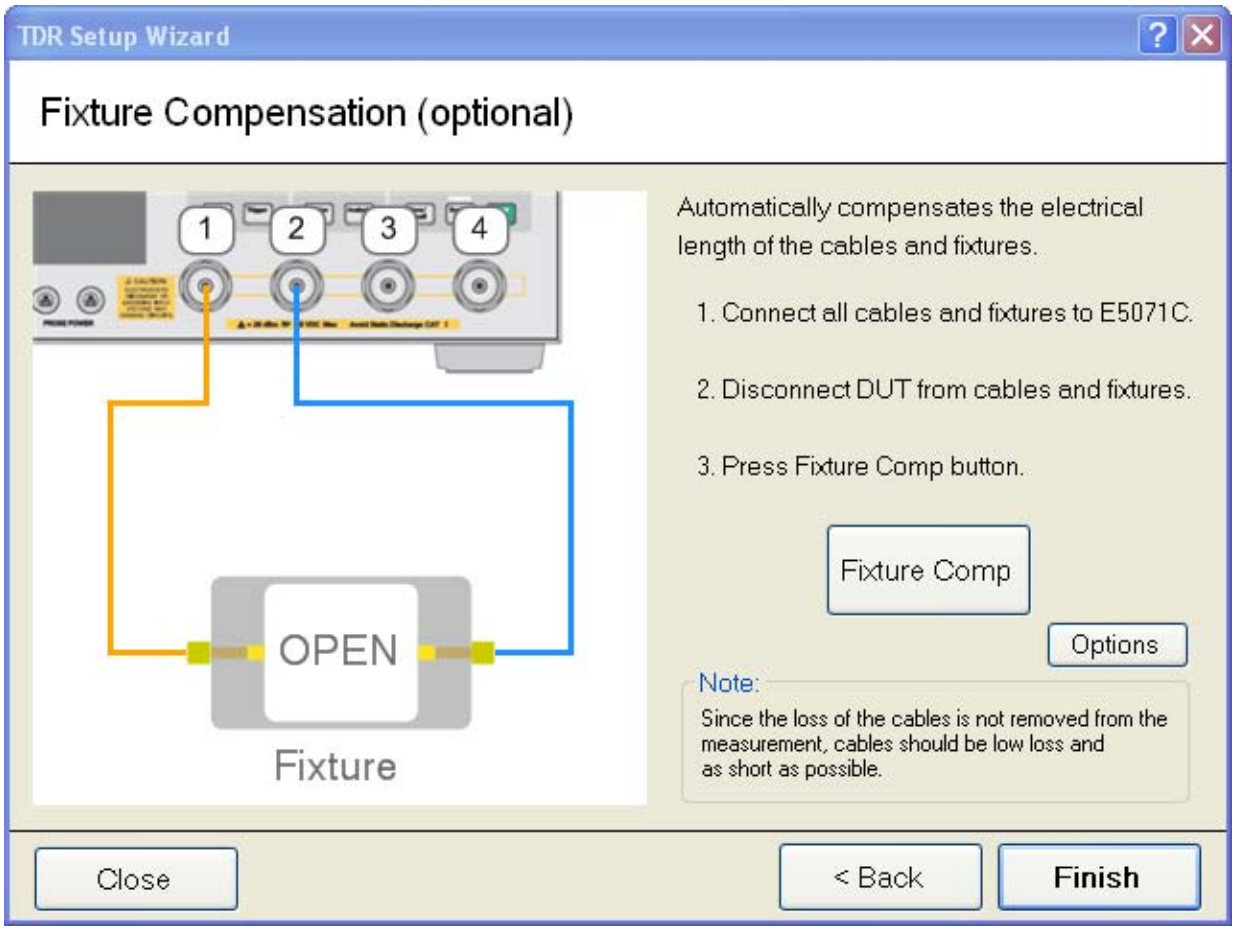
Full Calibration refers to calibration using the Electronic Calbration (ECal) module. It is a complete solid-state calibration solution, which makes calibration fast and easy.

NOTE 2-Port ECal is supported when DUT topology is set at Single Ended-1/2 Port, Differential 1 port.

1. Click the **ECal** button under **Basic**.
2. **Full Calibration** dialog box of the **Setup Wizard** appears.
3. Follow the instruction on the wizard and click the **Calibrate** button in the dialog box.




4. Click **Next >**.
5. Fixture Compensation is an optional process.




6. If you choose to perform fixture compensation, follow the instruction on the wizard and click the **Fixture Comp** button in the dialog box. This automatically compensates the electrical length of the cables and fixtures of all the ports.
7. Alternatively, you can compensate one port at a time. To perform this, instead of clicking the **Fixture Comp** button, click the **Options** button in the dialog box.
8. At the **Fixture Compensation Option** dialog box, you can click one port at a time to compensate it.
9. Clicking **Measure All** compensates all the ports at the same time. This action is the same as clicking the **Fixture Comp** button. You must compensate all the active ports.
10. At **Standard Type**, select either **Open** or **Short** compensation.
11. Click **OK** to close the **Fixture Compensation Option** dialog box.
12. Once complete, **Finish** on the dialog box.

## Making Measurement

### Making Measurements

- Setting up Parameters on Each Traces
- Controlling Trigger
- Using Scale/Zooming
- Using Marker and Marker Search
- Using Data and Memory
- Using Gating
- Using Trace Control
- Hot TDR Measurement

Other topics about Enhanced Time Domain Analysis

### Setting up Parameters on Each Traces

- Selecting Trace
- Changing Displayed Trace
- Selecting Parameters
- Defining the Stimulus
- Peeling
- Smoothing

## Other topics about Making Measurement

**Selecting Trace**

Three ways to select the displayed trace:

Using Mouse

- Double-click on any area of the graph plot to exit from full view.
- Click on the desired trace to select the trace.

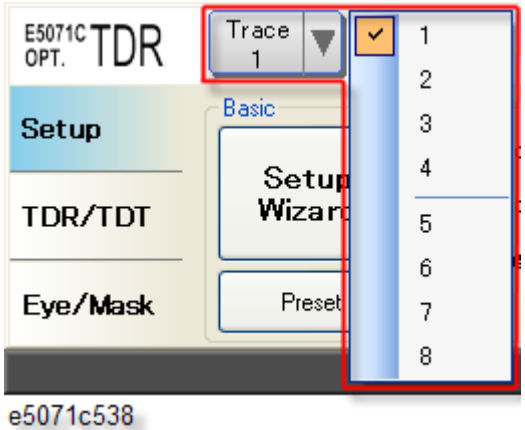
Using Hardkey

1. Press the **Trace Max** key to exit from full view of the active trace.
2. Press the **Trace Prev** or **Trace Next** key to select the desired trace.
3. Press the **Trace Max** key to enter full view.

Using Softkey

- Click on the **Trace** button and select the trace number from the list as shown below.

NOTE The number of trace is changeable and up to 16 traces can be displayed in this option. 16 traces are displayed when Differential 2-port DUT topology, and all T and all S traces are selected.



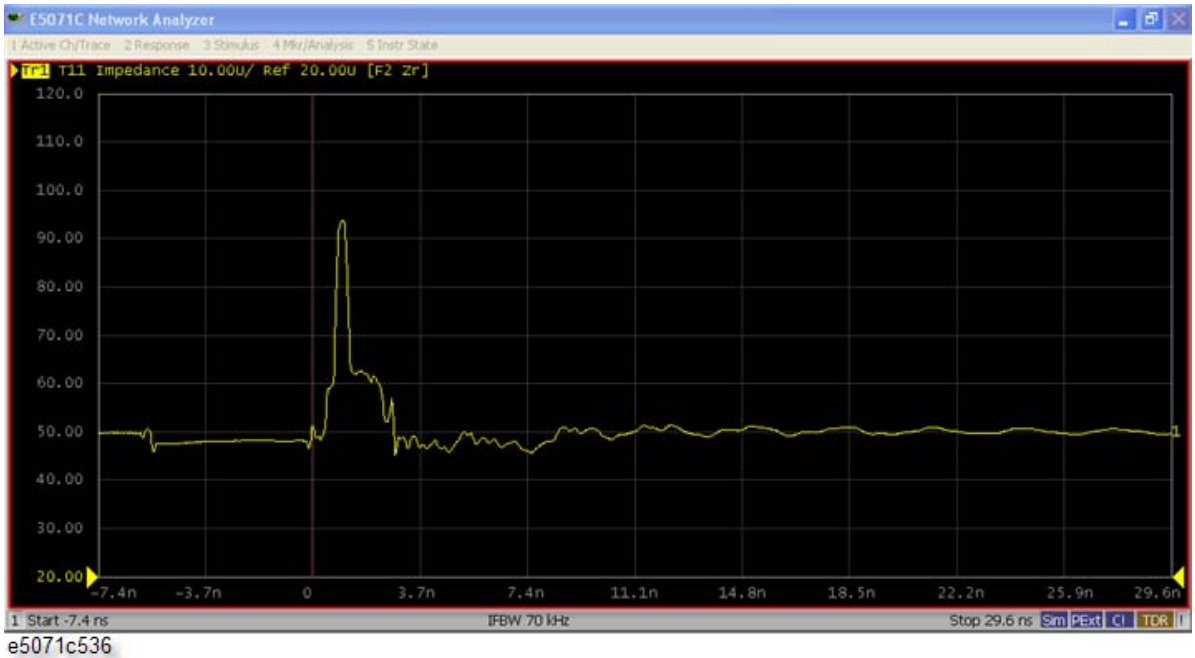

**Changing Displayed Trace**

**Changing Displayed Trace**

Two ways to change the displayed trace:

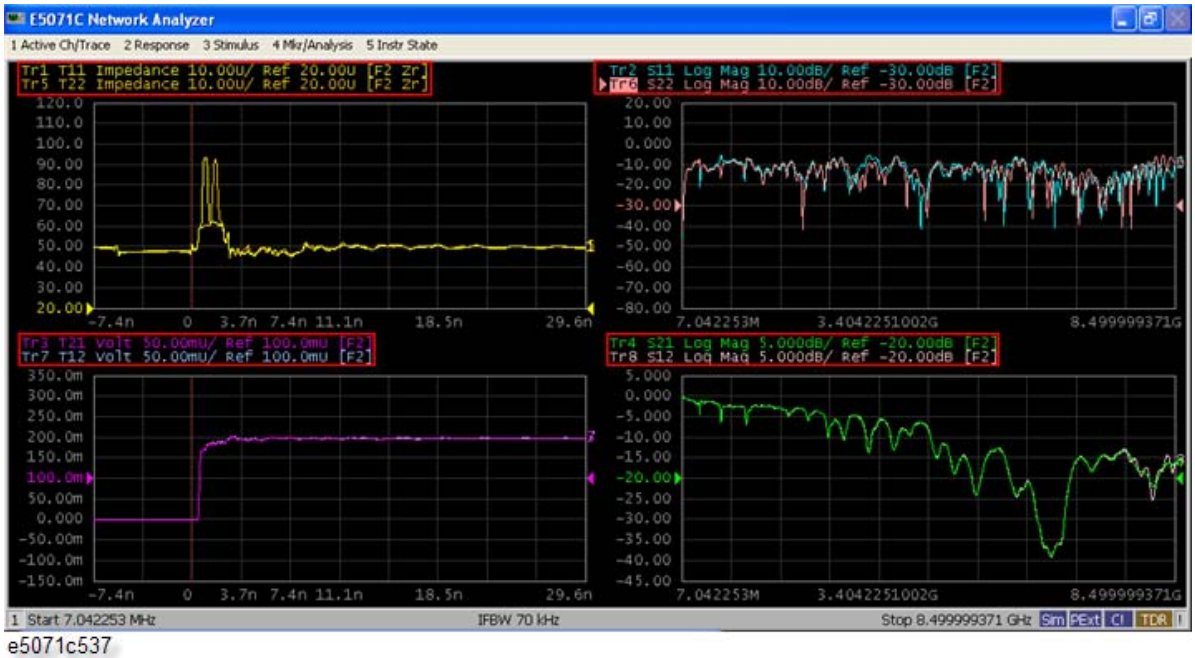
Using Mouse

1. Double-click on the single trace graph window to exit from full view of active trace as shown below.




e5071c536

2. Double-click on the desired trace for full view.




e5071c537

Using Hardkey

1. Press the **Trace Max** key to exit from full view of the active trace.
2. Press the **Trace Prev** or **Trace Next** key to select the desired trace.

3. Press the **Trace Max** key again to display the desired trace in full view.

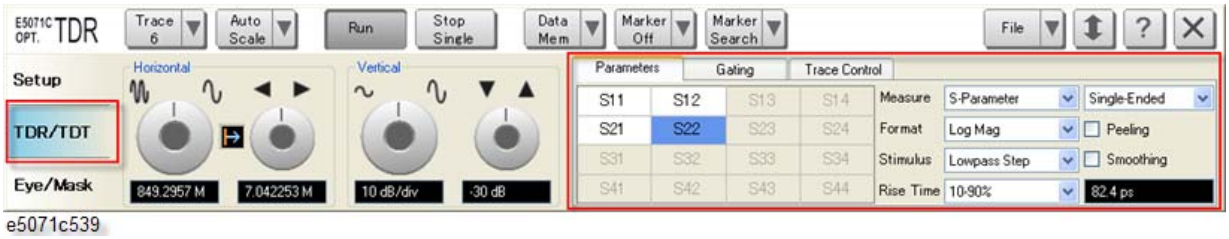
### Selecting Parameters

Click on the **TDR/TDT** tab in order to get to the **Parameters** selection as shown below.

The table below shows the Measurement and Format available under the **Parameter** setting:

| Measure | Format |
|---|---|
| S-Parameter | Log Mag |
| | Linear |
| | Real |
| | Imaginary |
| | Group Delay |
| | SWR |
| | Phase |
| | Expand Phase |
| | Positive Phase |
| | Smith (Lin/Phase) |
| | Smith (Log/Phase) |
| | Smith (Re/Im) |
| | Smith (R + jX) |
| | Smith (G + jB) |
| | Polar (Lin/Phase) |
| | Polar (Log/Phase) |
| | Polar (Re/Im) |
| Time Domain | Impedance |
| | Volt |

| | |
|---|---|
| | Log Mag |
| | Linear |
| | Real |



e5071c539

The table in the **Parameter** area changes as the selection of **Measure** changes. The changes can be referred as below:

| Measure | | Table Content starts with |
|---|---|---|
| S-Parameter | Single-Ended | S |
| | Differential | Sc, Sd |
| Time Domain | Single-Ended | T |
| | Differential | Tc, Td |

### Defining the Stimulus

There are two options to choose from under the Stimulus:

- Lowpass Step
- Lowpass Impulse

Only for the Lowpass Step the selection for Rise Time is active. The two options for Rise Time are:

- 10-90%
- 20-80%

The rise time settings in TDR/TDT mode and EYE/MASK mode are independent.

### Peeling

When a device has two or more impedance discontinuities, reflections from the second discontinuity reflects off the first discontinuity. This complex interaction of secondary reflections from the stimulus pulse compromises the measured impedance profile and decreases the measurement performance.

TDR peeling compensates for the complicated interaction between the discontinuities. TDR peeling analysis reflects the signals at the source and

de-convolves the time domain reflections to create an impedance profile of the device being tested. This option is available when the measurement parameter is set to time domain reflection and the format is impedance.

#### Limitation

- TDR peeling does not account for frequency response losses (for example, PC board transmission lines are lossy devices). The sum of the waves that are incident on a node are assumed to be equivalent to those exiting the node.
- TDR peeling assumes a lossless transmission line (resistance of 0). Any actual resistance (which causes loss, even at DC) degrades the accuracy of peeling.
- Initial impedance mismatch is the most accurate; as distance increases from initial impedance mismatch down the transmission line, the impedance accuracy decreases.
- TDR peeling cannot be used on TDT responses, because the lack of reflections invalidates the algorithm.

#### How to set Peeling

1. Click **TDR/TDT** tab.
2. Select the **Peeling** check box under **Parameter**.

### Smoothing

Smoothing is only available when S-Parameter is measured and only applicable to reflection coefficient only (S11, S22, S33, S44).

#### How to set Smoothing

1. Click **TDR/TDT** tab.
2. Select the **Smoothing** check box under **Parameter**.

## Controlling Trigger

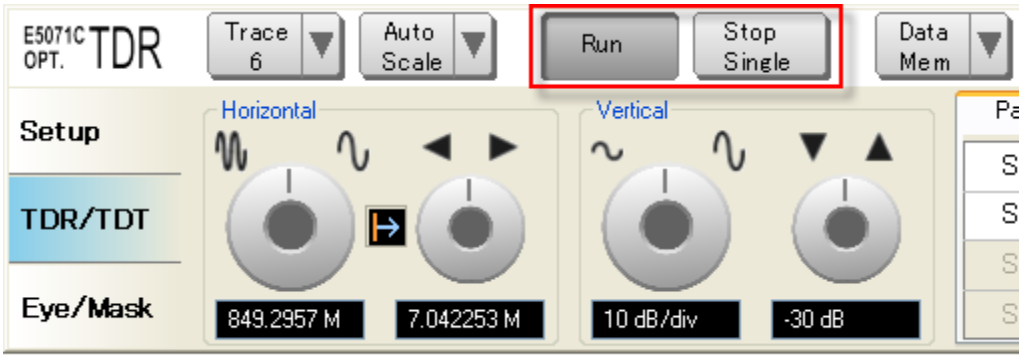
### Making Single/Continuous measurement

#### Single measurement

- Click on the **Stop Single** button to enable single measurement.

#### Continuous measurement

- Click on the **Run** button to enable continuous measurement.




e5071c572

Other topics about Making Measurement

## Using Scale / Zooming

- Using Autoscale
- Changing Scale Manually
- Zooming

Other topics about Making Measurement

### Using Autoscale

1. Select the trace which you want to change the scale.
2. Click on the **Auto Scale** button on the tool bar or right-click in the graph plot area.
3. Select the desired scale. Refer to the table below for the details of each selection.

| Scale | Details |
|---|---|
| Auto Scale X | Auto scaling X axis only, on the active trace |
| Auto Scale Y | Auto scaling Y axis only, on the active trace |
| Auto Scale X & Y | Auto scaling both X & Y axis, on the active trace |
| Auto Scale All | Auto scaling both X & Y axis, on all traces |

### Changing Scale Manually

#### Changing Horizontal Scale

1. Select the trace which you want to change the scale.
2. Click the **TDR/TDT** tab.

- Using Virtual knob
    a. Click the left knob under **Horizontal**. The centre of the knob turns blue.
    b. Scroll up or down the wheel mouse button to change the scale.
- Type in **Entry** box
    a. Click the box below the left knob under **Horizontal**. An **Entry** dialog box appears.
    b. Type the precise value.
- Clicking Wave icons
    a. Click the wave icons above the left knob under **Horizontal**. Details are as table below:

| Function Icon | Details |
|---|---|
|  | Click to increase the scale in a 1-2-5 sequence. The width of the waveform compresses. The default value for basic time units is 1.0 ns/div |
|  | Click to decrease the scale in a 1-2-5 sequence. The width of the waveform expands. |

Reference Position for Horizontal Axis

The reference point of horizontal axis can be selected from the center line or left edge of the displayed graticule.

Click the icon as shown in table below under **Horizontal** to change the reference point:

| Function Icon | Details |
|---|---|
|  | Reference point of the display graticule to the left edge. The position of left edge is not changed even if you change the scale. |
|  | Reference point of the display graticule to the center. The position of center is not changed even if you change the scale. |

**Changing Horizontal position**

1. Select the trace which you want to change the horizontal position.
2. Click the **TDR/TDT** tab.

- Using Virtual knob
    1. Click the right knob under **Horizontal**. The centre of the knob turns blue.
    2. Scroll up or down the wheel mouse button to change the position.
- Type in **Entry** box
    1. Click the box below the right knob under **Horizontal**. An **Entry** dialog box appears.
    2. Type the precise value.
- Clicking Arrow icons

1. Click the arrow icons above the right knob under **Horizontal**. Details are as table below:

| Function Icon | Details |
|---|---|
| ◀ | Click to increase the delay from trigger; the waveform moves to the left of the display. |
| ▶ | Click to decrease the delay from trigger; the waveform moves to the right of the display. |

#### Changing Vertical Scale

1. Select the trace which you want to change the scale.
2. Click the **TDR/TDT** tab.

- Using Virtual knob
  1. Click the left knob under **Vertical**. The centre of the knob turns blue.
  2. Scroll up or down the wheel mouse button to change the scale.
- Type in **Entry** box
  1. Click the box below the left knob under **Vertical**. An **Entry** dialog box appears.
  2. Type the precise value.
- Clicking Wave icons
  1. Click the wave icons above the left knob under **Vertical**. Details are as table below:

| Function Icon | Details |
|---|---|
| ~ | Click to increase the scale in a 1-2-5 sequence; the waveform height compress. |
| ∿ | Click to decrease the scale in a 1-2-5 sequence; the waveform height will expand. |

#### Changing Vertical Position

1. Select the trace which you want to change the vertical position.
2. Click the **TDR/TDT** tab.

- Using Virtual knob
    1. Click the right knob under **Vertical**. The centre of the knob turns blue.
    2. Scroll up or down the wheel mouse button to change the position.
- Type in **Entry** box
    1. Click the box below the right knob under **Vertical**. An **Entry** dialog box appears.
    2. Type the precise value.
- Clicking the Arrow icons
    1. Click the arrow icons above the right knob under **Vertical**. Details are as table below:

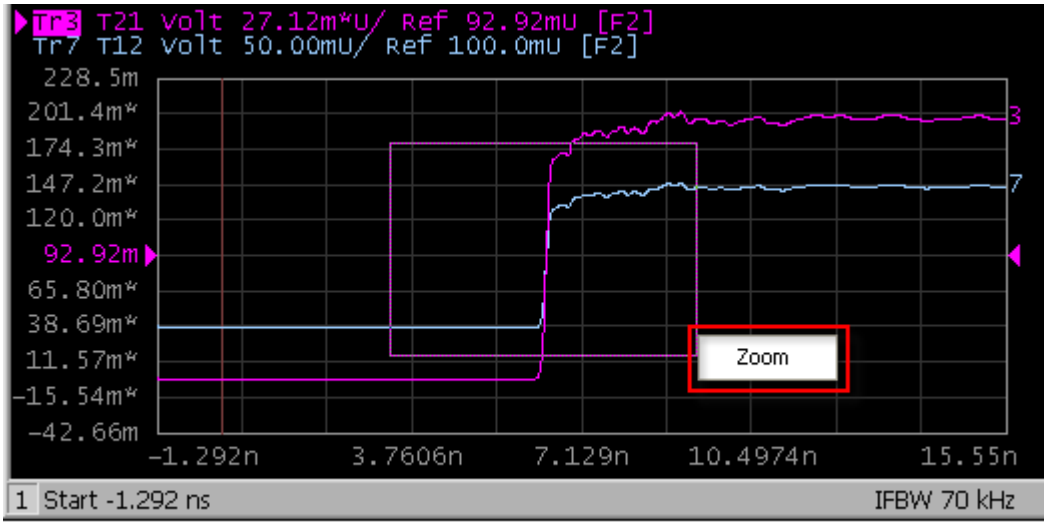
| Function Icon | Details |
|---|---|
|  | Click to increase the offset value; the waveform shift downward. |
|  | Click to decrease the offset value; the waveform shift upward. |

- Drag and drop
    1. Click and hold the y-axis of the graph plot.
    2. Drag up or down to the desired position and release the button.

### Zooming

To zoom in the graph plot:

1. Left-click and hold the mouse button.
2. Drag the area you would like zoom and release the mouse button.
3. Select zoom as shown below.


Tr3 T21 Volt 27.12m*U/ Ref 92.92mU [F2]
Tr7 T12 Volt 50.00mU/ Ref 100.0mU [F2]
228.5m
201.4m*
174.3m*
147.2m*
120.0m*
92.92m
65.80m*
38.69m*
11.57m*
-15.54m*
-42.66m
-1.292n
3.7606n
7.129n
10.4974n
15.55n
3
7
Zoom
1 Start -1.292 ns
IFBW 70 kHz

e5071c565

### Using Marker / Marker Search

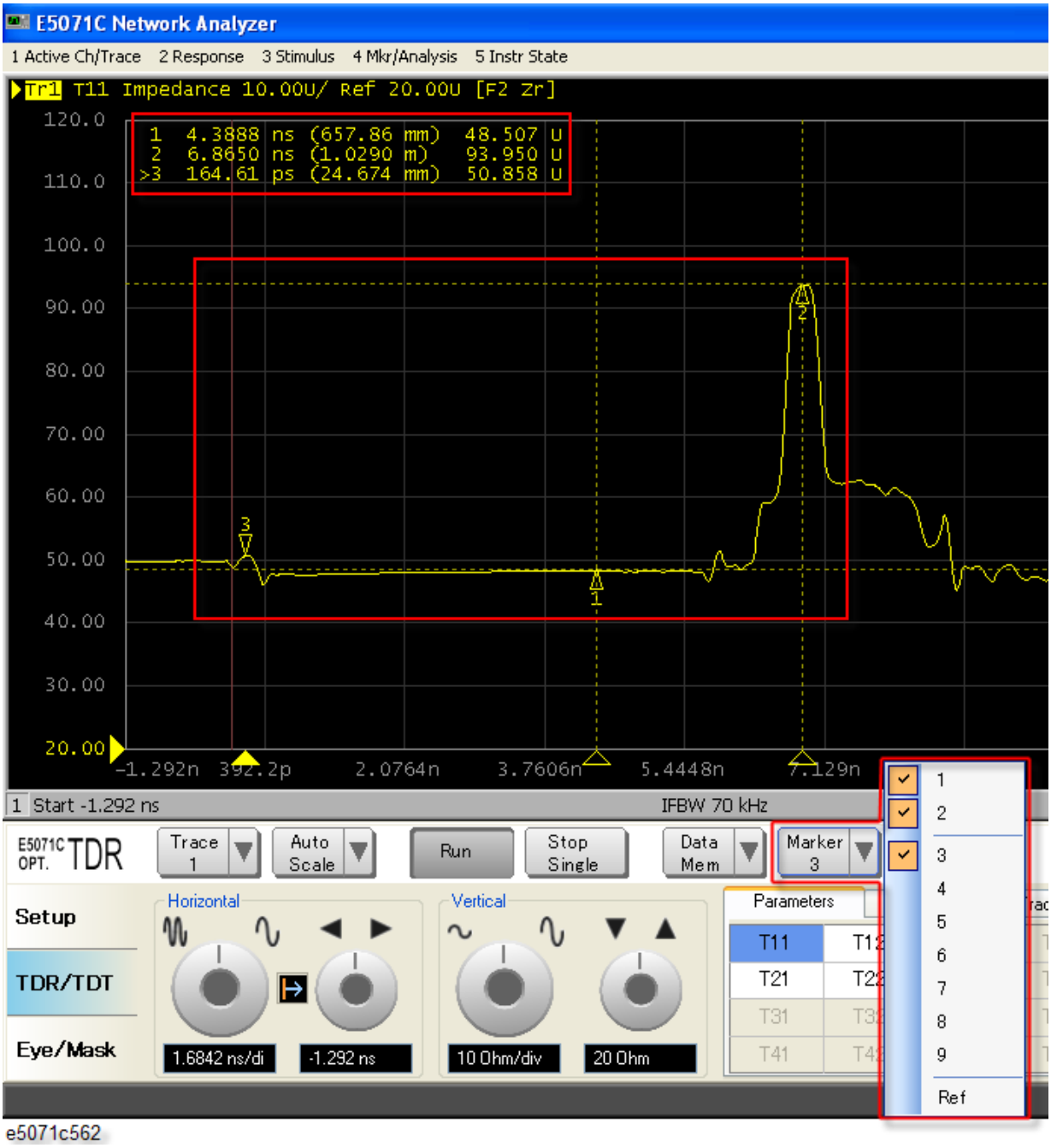
- Activating Marker
- Moving Marker
- Using Reference Marker
- Searching Max/Min Points on Trace
- Measuring Rise Time
- Measuring Delta Time

Other topics about Making Measurement

#### Activating Marker

1. Click on the **Marker Off** button.
2. Select **1** from the list to activate Marker 1.

NOTE The marker button shows **Marker Off** when there is no active markers. The marker button shows **Marker Ref** when reference marker is active. This applies to all markers. Active marker has an arrow head pointing down. Inactive markers have an arrow head pointing up. Only markers 1 and 2 have vertical dotted lines. All marker values are displayed at the upper left corner of the graph view. There are a maximum of 9 markers that can be activated.




e5071c562

### Moving Marker

- Click and hold on the marker, drag the marker to the desired point and release the mouse button.

### Using Reference Marker

When reference marker is used, all 9 markers use reference marker as reference point.

1. Click on the **Marker Off** button.
2. Select **Ref** from the list.

Moving reference marker is similar to moving marker. Simply choose the reference marker instead of the marker number.

### Searching Max/Min Points on Trace

#### Searching Max point on trace

1. Click on the desire trace.
2. Click on the **Marker Search** button.
3. Select **Max** from the list.

#### Searching Min point on trace

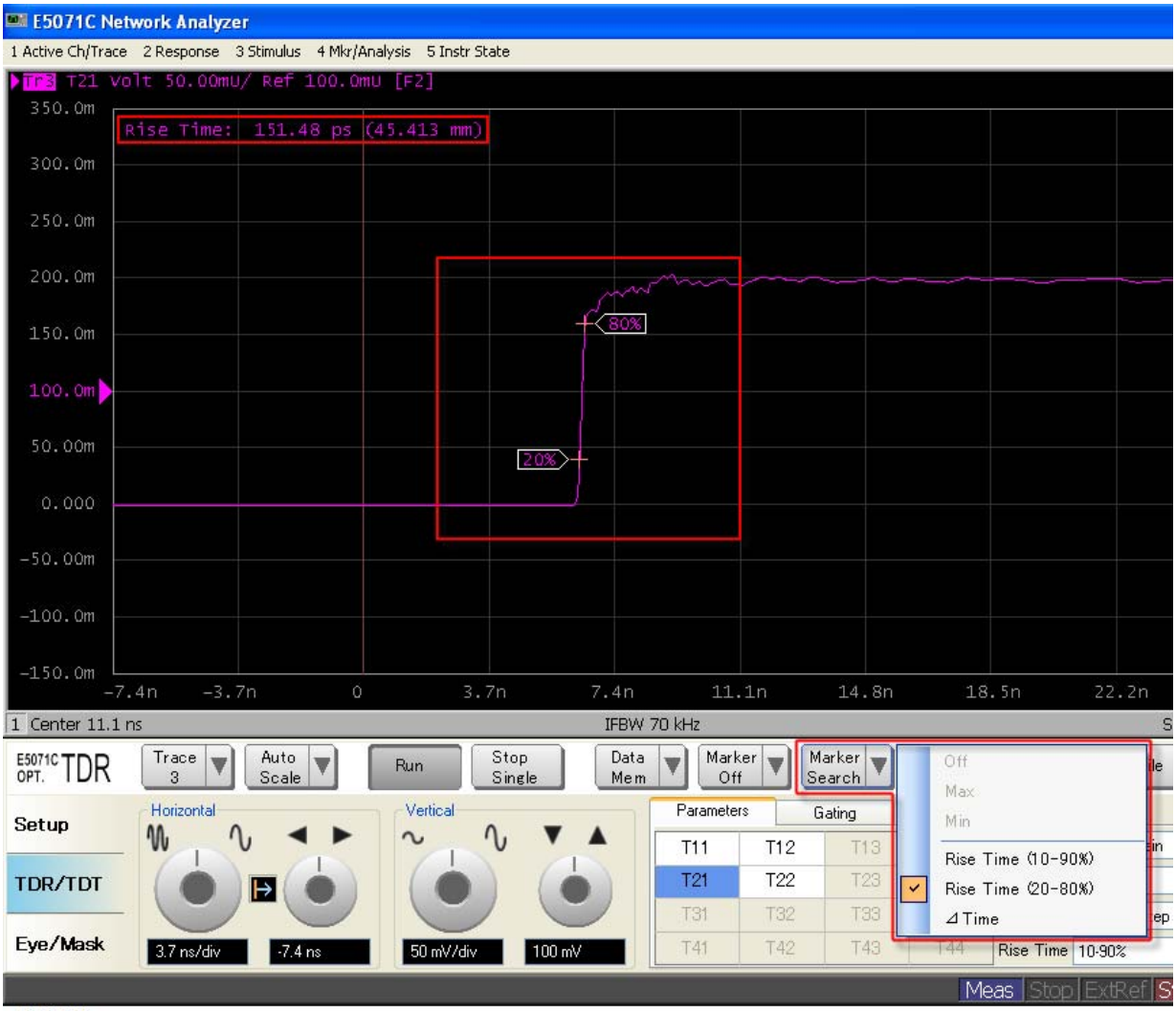
1. Click on the desire trace.
2. Click on the **Marker Search** button.
3. Select **Min** from the pull down menu.

NOTE The marker is constantly on tracking mode when Max or Min is selected. Even moving the marker with the mouse, the marker tracks the Max or Min of the trace and points to it.

### Measuring Rise Time

1. Click on the desire trace.
2. Click on the **Marker Search** button.
3. Select **Rise Time (10-90%)** or **Rise Time (20-80%)** from the list.

The data is displayed at the top left corner of the graph plot.




**Measuring Delta Time**

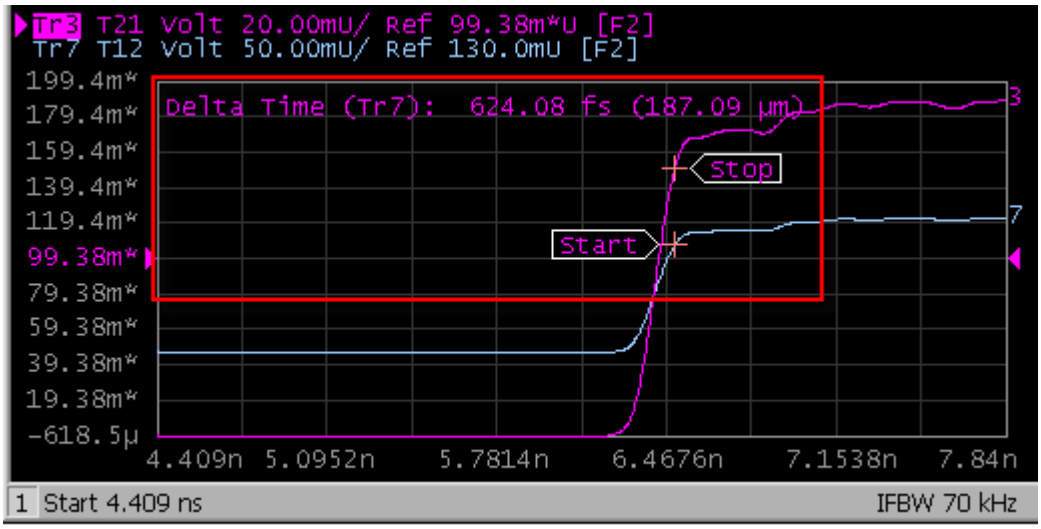
Delta Time measurement available for Time Domain. Traces other than time domain are not able to use for delta time measurement. Delta time is compared with trace save in memory, if available, else it is compared with the trace selected.

1. Click on the trace that is the starting point for the Delta Time.
2. Click on the **Marker Search** button.
3. Select **Δ Time** from the list and select the **Δ Time** checkbox.
4. Select the trace that will be the stopping point.

   NOTE The trace of the starting point can not be changed, therefore it is selected at the beginning. You can compare Data with Memory, by selecting the same trace. (only if Data & Memory is selected)
5. Select the Position to be measured.

6. Click on the **OK** button.

The data is displayed at the top left corner of the graph plot.



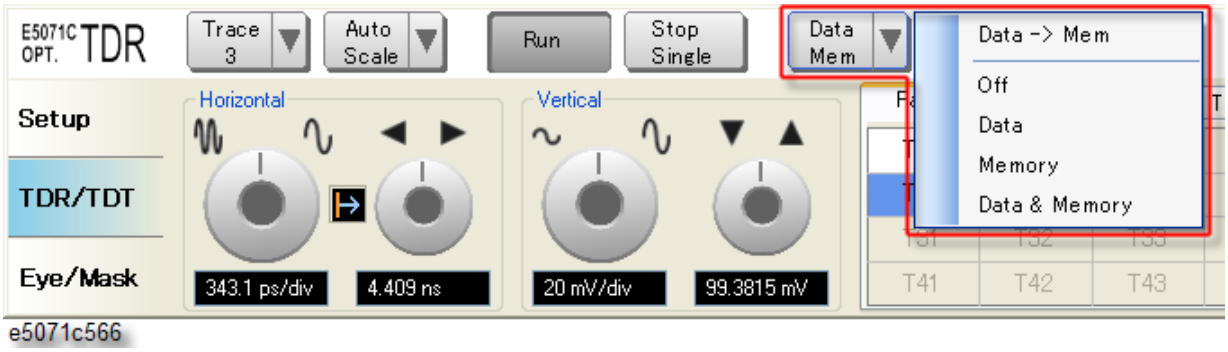

e5071c564

## Using Data and Memory

- Storing Data Traces to Memory
- Display the Memory Traces

### Other topics about Making Measurement

Data displayed on the graph plot can be stored in the memory and recalled for later use. Data traces stored in the memory will be deleted once E5071C-TDR option restarts.




#### Storing Data Traces to Memory

1. Click on the **Data Mem** button.
2. Select **Data -> Mem** from the list.

#### Display the Memory Traces

1. Click on the **Data Mem** button.
2. Select **Memory** from the list.

Table below explains other functions in the Data Mem list:

| Function | Details |
|---|---|
| Data -> Mem | Save Data traces to Memory |
| Off | Display off |
| Data | Display Data traces only |
| Memory | Display Memory traces only |
| Data & Memory | Display Data and Memory traces |

## Using Gating



Other topics about Making Measurement

### Overview

Gating provides the ability to observe the effect of a particular circuit element on frequency domain response by virtually removing undesired responses. When you define a gate on time domain plot, the gated section is removed and replaced mathematically with an ideal transmission line having the same electrical delay as the removed section. Gating is applied to time plots of individual parameters. While gating the time domain plot of a parameter, you can observe the effect that gating has on the frequency domain of the same parameter by coupling the relevant traces using the gate coupling feature. By observing the original frequency domain response and the transformed frequency domain response, the effect of the gating operation on the S-parameter data can be seen.

NOTE When a discontinuity in a test device reflects energy, that energy will not reach subsequent discontinuities. This can "MASK", or hide, the true response which would have occurred if the previous discontinuity were not present. The Gating feature does NOT compensate for this.

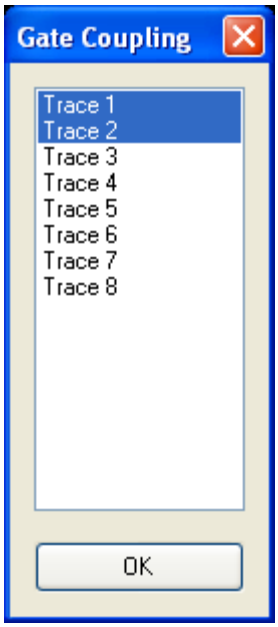
### Coupling Gate on Several Traces

Gate Coupling allows two or more traces to share the same gating values. The gating value of the active trace is coupled to other trace(s).

It is used to observe the gated frequency response while setting a gate on time response.

#### Operational procedure

1. Click the **TDR/TDT** tab.
2. Under **TDR/TDT**, click the **Gating** tab.
3. Under **Gating**, click the **Gate Coupling** button.
4. **Gate Coupling** dialog box appears.




5. Select the time domain trace and the associated S-Parameter trace. For example, trace of T11 and trace of S11.
6. Click **OK**.

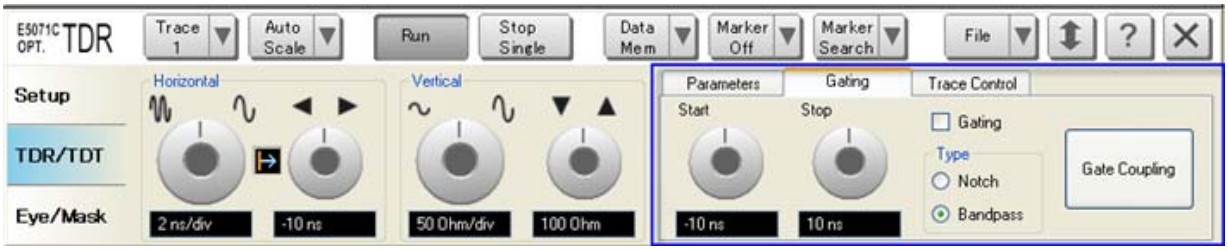
NOTE The gate coupling should not be applied to unrelated responses. For example, TDR (T11) and Insertion Loss (S21).

NOTE Selecting a lot of traces the gate coupling makes the response of virtual knobs slow. Type your desired value instead of rotating virtual knobs to specify the value.

### Setting Start/Stop Points

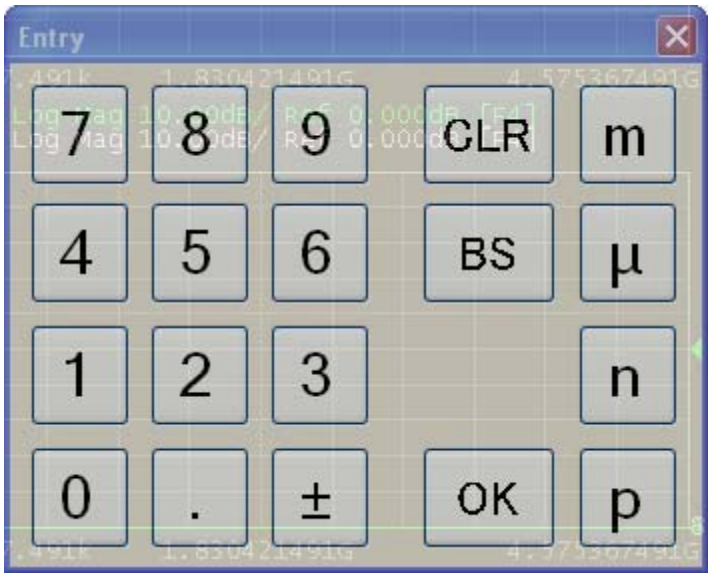
Gating is set on the active trace. There are several methods to set the start and stop points of the gate:

#### Setting points at gating tab




1. Click the **TDR/TDT** tab.
2. Under **TDR/TDT**, click the **Gating** tab.
3. Click on the **Start** or **Stop** virtual knob.
4. The knob is highlighted in blue once its enabled.

5. Click on the virtual several times clockwise to increase the value and anti-clockwise to decrease the value.
6. You can observe the start or stop time in the text box below the virtual knob and gate markers on the active trace.
7. Optionally, you can click the text box under the **Start** or **Stop** virtual knob.
8. An **Entry** dialog box appears.



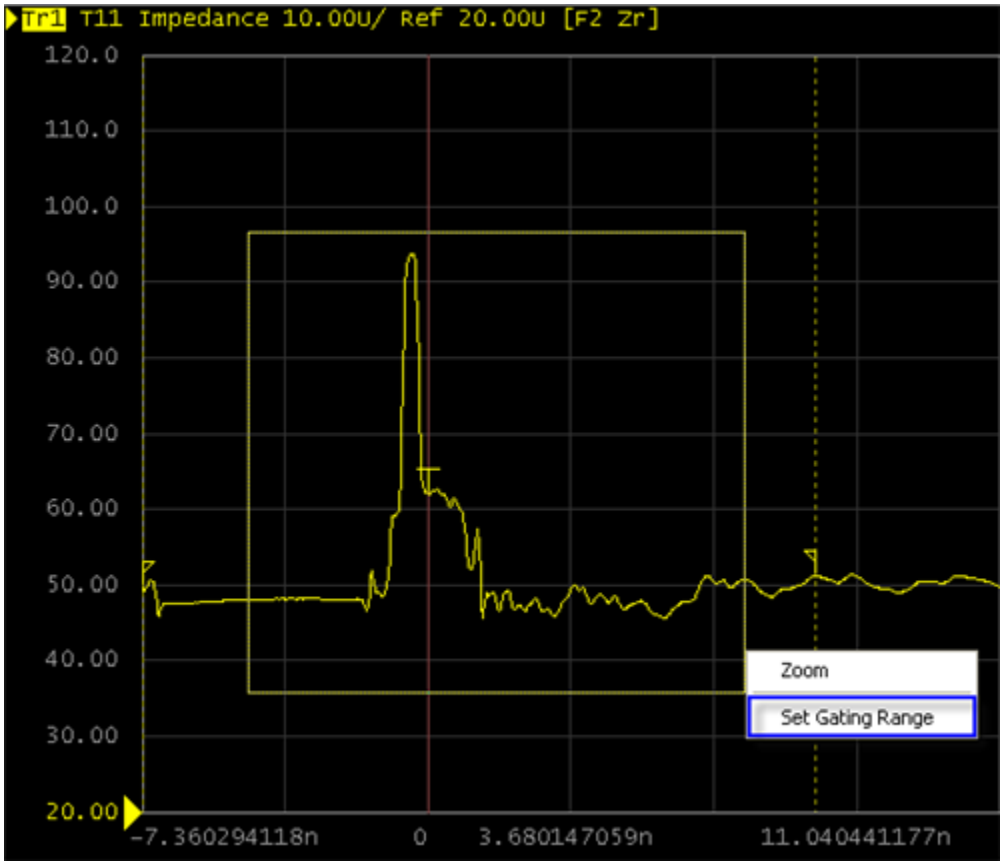

9. Type the start or stop time and click **OK**.
10. The new value will be displayed at text box and x-axis of the active trace.

NOTE The **Start** virtual knob and text box sets the start time and the **Stop** sets the stop time.

NOTE The start time and stop time should be set at a point on which the impedance is close to the reference impedance.

**Setting points at active trace graph plot**

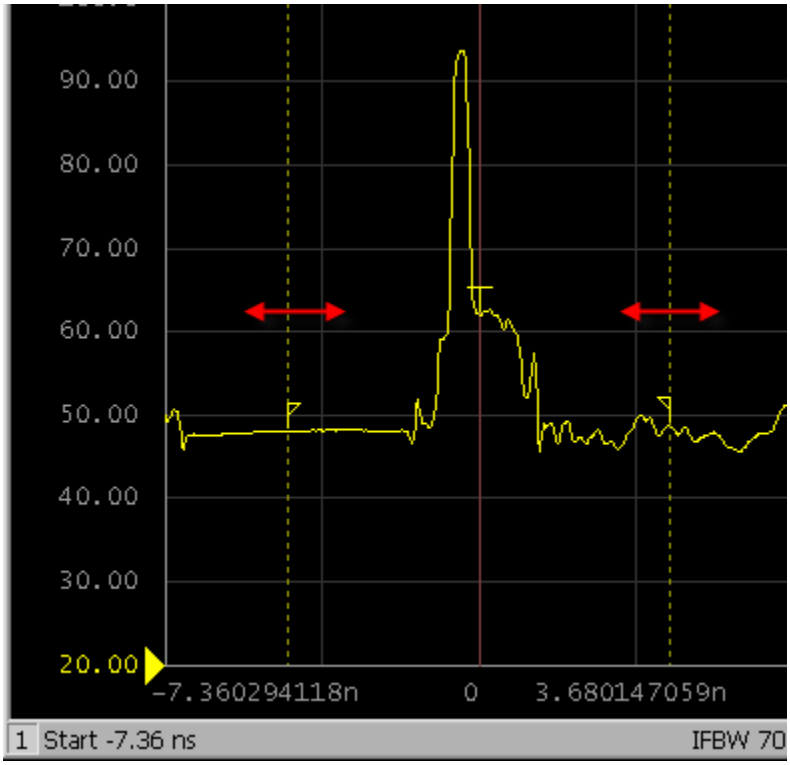
1. On the time domain active trace, drag your mouse across the area to set the gating.
2. Select **Set Gating Range**.




3. You can observe two dashed lines indicating the gate on the graph plot. These are gate markers.
4. You can also observe the time value in the text box below the virtual knob under the **Gating** tab in the **TDR/TDT** area.

**Moving gate at active trace graph plot**

1. Select the gate marker (dashed line).
2. Drag the gate marker, with your mouse to a new location.
3. Release the mouse click, to have the move take effect.




### Selecting Gating Type

The Enhanced Time Domain Analysis module allows you to choose from the following two gate types:

| Gate type | Description |
|---|---|
| Band pass | Removes response outside the gate range |
| Notch | Removes response inside the gate range |

**Operational procedure**

1. Click the **TDR/TDT** tab.
2. Under **TDR/TDT**, click the **Gating** tab.
3. Under **Gating**, select one of the gating type from the **Type** area.

### Activating Gate

Ensure that you have set the start and stop time. Then follow the following procedure:

1. Click the **TDR/TDT** tab.
2. Under **TDR/TDT**, click the **Gating** tab.

3. Under **Gating**, select the **Gating** check box.

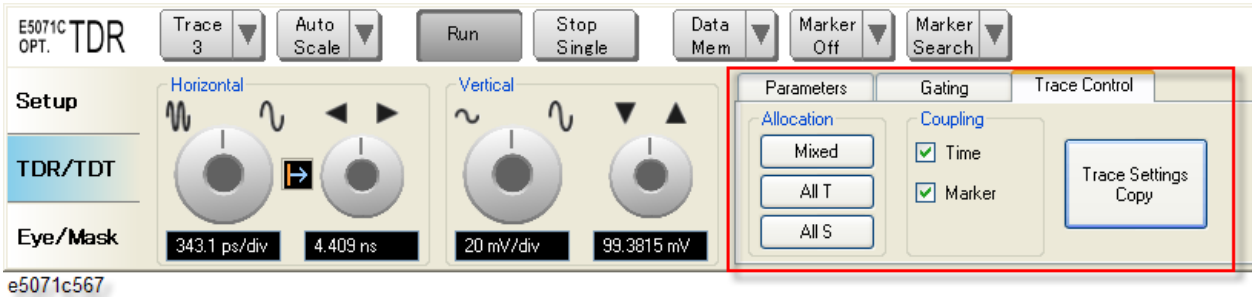
#### Switch Gating State

1. Right-click on the time domain active trace.
2. Select **Switch Gating State**.
3. If gating was turned ON earlier, this step will turn it OFF and vise versa. You can observe the changes in the **Gating** check box.

### Using Trace Control

- Changing Trace Allocation
- Coupling Marker/Time
- Copying Trace Setting

Other topics about Making Measurement




#### Changing Trace Allocation

Changing the trace allocation affects the data display on the graph plot. Table below shows the details of each selection:

| Allocation | Details |
|---|---|
| Mixed | Display mixed of commonly measured time domain and S-parameter data |
| All T | Display all Time Domain data for selected device topology |
| All S | Display all S-Parameter data for selected device topology |

1. Click the **TDR/TDT** tab.
2. Click on the desired trace allocation in **Trace Control** under **Allocation**.

#### Coupling Marker/Time

1. Click the **TDR/TDT** tab.
2. Click on the desired check box in **Trace Control** under **Coupling**.

Selecting Marker under Coupling enables all the marker on other traces to be moved in same alignment.

Selecting Time under Coupling enables all other traces using the same X axis (Time).

#### Copying Trace Setting

1. Click on the **Trace Settings Copy** button. Trace Settings Copy dialog box appears.
2. Select the source trace in the From list. Select the desired destination trace in the To list.

3. Click on the **>> Copy >>** button.

**Copied Parameter**

The following parameters for the following functions are copied by the **Trace Settings Copy**.

| Functions | SCPI Commands |
|---|---|
| Parameter, Time Domain/S-Parameter, Single-Ended/Differential | CALC:TRAC:PAR |
| Format | CALC:TRAC:FORM |
| Marker | CALC:TRAC:AMAR:ACT |
| Peeling | CALC:TRAC:CONV:PEEL:STAT |
| Delta Time Dialog | CALC:TRAC:DTIM:POS |
| | CALC:TRAC:DTIM:STAT |
| Delta Time Target [Target is trace for stop] | CALC:TRAC:DTIM:TARG |
| Gating Start | CALC:TRAC:GATE:STAR |
| Gating State | CALC:TRAC:GATE:STAT |
| Gating Stop | CALC:TRAC:GATE:STOP |
| Gating Type | CALC:TRAC:GATE:TYPE |
| Marker Search [ON/OFF], marker [0-9, ref] | CALC:TRAC:MARK[1-10]:FUNC:TRAC |
| Marker Search [MIN/MAX], marker [0-9, ref] | CALC:TRAC:MARK[1-10]:FUNC:TYPE |
| Reference Marker [ON/OFF] | CALC:TRAC:MARK[1-10]:REF:STAT |
| Marker | CALC:TRAC:MARK[1-10]:STAT |
| Marker [x-axis value] | CALC:TRAC:MARK[1-10]:X |
| Smoothing | CALC:TRAC:SMO:STAT |
| Impulse Width Value for Lowpass Impulse | CALC:TRAC:TIME:IMP:WIDT |
| Rise Time (for all traces) | CALC:TRAC:TIME:STEP:RTIM:DATA |

| | |
|---|---|
| | CALC:TRAC:TIME:STEP:RTIM:THR |
| Stimulus | CALC:TRAC:TIME:STIM |
| Search Rise Time | CALC:TRAC:TTIM:STAT |
| | CALC:TRAC:TTIM:THR |
| Horizontal Scale | DISP:TRAC:X:SCAL:PDIV |
| Horizontal Position | DISP:TRAC:X:SCAL:RLEV |
| Vertical Scale | DISP:TRAC:Y:SCAL:PDIV |
| Vertical Position | DISP:TRAC:Y:SCAL:RLEV |

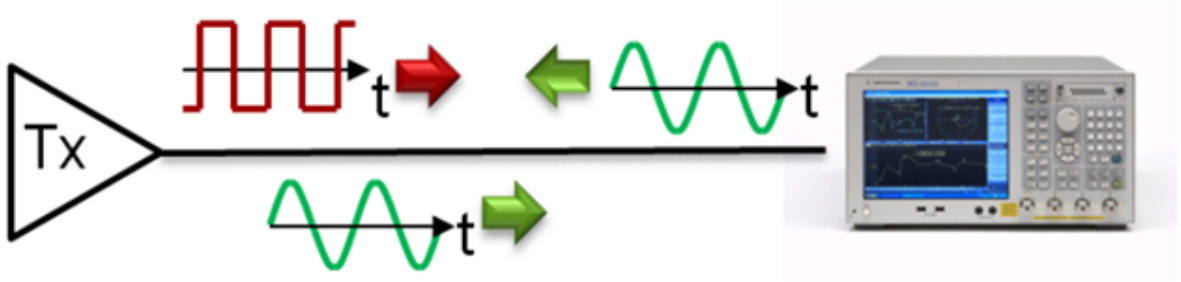
## Hot TDR Measurement

- Overview
- Checking Device Malfunction
- Reducing Measurement Error

Other topics about Making Measurement

### Overview

Hot TDR refers to TDR measurement while the device is powered ON. During measurement, the measurement signal is applied from the E5071C to the transmitter. However, this may cause device malfunction. In addition, the transmittor signal from the DUT device into the E5071C ports may cause measurement error. When you measure Hot TDR, the transmitter should be connected with Port 1 for Single END, Port1/2 for differential.




### Checking Device Malfunction

The following procedure checks if the Signal Source level from the E5071C does not affect the device operation:

1. Measure the reflection on the E5071C.
2. Save the results in memory trace.
3. Reduce the signal source level of the E5071C at Source Power.
4. Check for significant change on the trace. If the change is significant, reduce the signal source level further to avoid device malfunction.

### Reducing Measurement Error

The following procedure reduces the measurement error produced by the device signal.

#### Case 1 (For Periodical Bit Pattern)

If the output signal from device is periodic, set the data rate and execute "Avoid Spurious". This will reduce error due to spurious:

1. Click **Setup** tab and select **Hot TDR**.
2. At the **Data Rate**, left-click once. An **Entry** dialog box appears. Type the data rate value and click **OK**. The new value is displayed at **Data Rate**. The Data Rate accuracy should be within ±0.5%.
3. Click **Avoid Spurious** to execute the option. ENA searches for spurious and changes the stimulus setting to avoid the spurious. If the Avoid Spurious is successfully executed, a check mark appears next the **Avoid Spurious** button. At this point, measurement mode is changed from TDR/TDT to Hot TDR Mode and this is indicated at the channel window as "TDR ?". The blue **SVC** indicator is also turned ON.

   NOTE **Eye/Mask** option tab is disabled in HOT TDR mode.

   NOTE To reset the HOT TDR mode, simply execute preset or change the DUT toplogy.

**Case 2 (For Random Data)**

If the output signal from device is random, increase averaging to reduce the measurement error.

About Avoid Spurious

If Avoid Spurious fails, the "Spurious Not Found" warning message is displayed. Check mark next to the **Avoid Spurious** button will not be displayed.

When the following parameter is changed, Avoid Spurious option should be executed again:

- Data Rate
- DUT Length
- Deskew

The setting of Avoid Spurious cannot be stored. To recall the condition of Avoid Spurious, you must execute the Avoid Spurious again after recalling the status setting.

At the execution of Avoid Spurious:

- If the IF bandwidth is over 10 kHz, the IF bandwidth value will be set at 10 kHz.
- If the source power is over -20 dBm, the source power will be set at -20 dBm.

## Eye Diagram and Mask Test

### Eye Diagram and Mask Test (Eye / Mask Tab)

- Performing Eye Diagram Measurements
- Selecting Bit Pattern
- Using Mask Test
- Available Masks

Other topics about Enhanced Time Domain Analysis

## Performing Eye Diagram Measurements

- Overview
- Showing Eye Diagram
- Displaying Results
- Scaling the Eye Diagram
- Injecting Jitter

### Other topics about Eye Diagram and Mask Test

#### Overview

In the oscilloscope, an eye diagram is often used to analyze signal quality. You can diagnose problems, such as attenuation, noise, jitter, and dispersion that arise or characterize specific parts of the system with one display.

The E5071C option TDR provides simulated eye diagram analysis capability, eliminating the need for a hardware pulse pattern generator. The virtual bit pattern generator is used to define a virtual bit pattern. The defined bit pattern is then convolved with the device impulse response to create an extremely accurate measurement based eye diagram.

#### Showing Eye Diagram

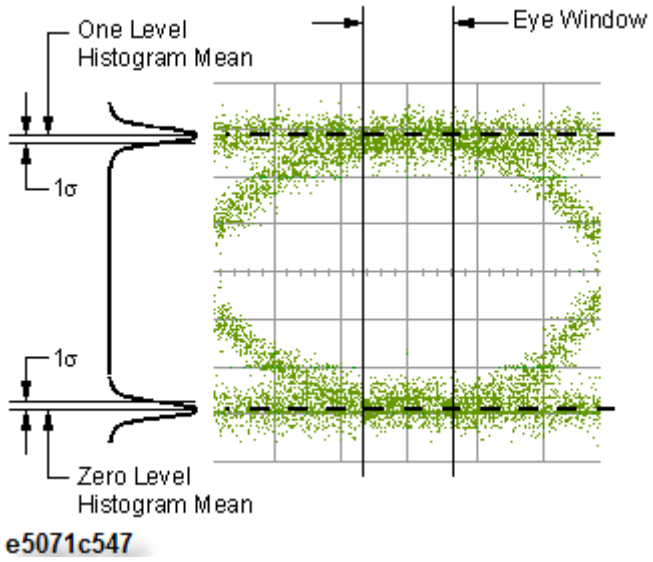
1. Select the trace number which you want to observe the eye diagram.
2. Click **Draw Eye** to display the eye diagram.
3. Whenever you change the setting of data pattern, it is required to click **Draw Eye** to reflect the setting on the waveform.

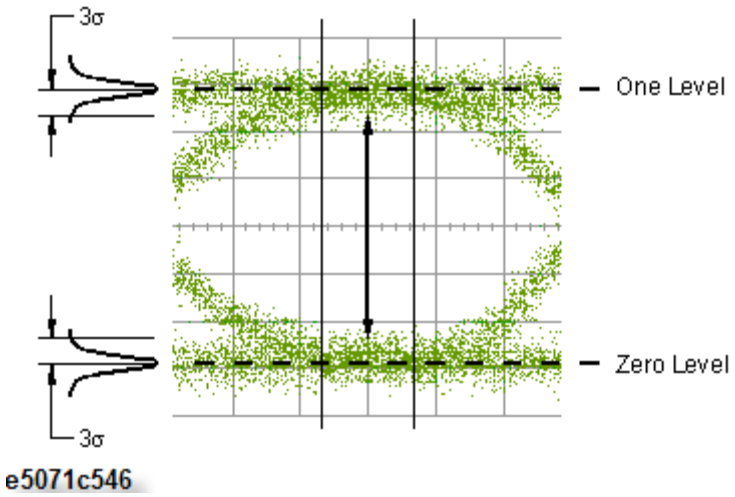
#### Displaying Results

1. Select **Rise Time Def** under **Results**.
2. Click **Draw Eye** to display the result.
   Whenever you change the setting of **Rise Time Def**, it is required to click **Draw Eye** to reflect on the result.

The following results are displayed on the table under **Results**.

| Name | Unit | Description |
|---|---|---|
| Rise Time | Second | Rise Time Def=10%-90%: Time at 90% level - Time at 10% level<br>Rise Time Def=20%-80%: Time at 80% level - Time at 20% level |
| Fall Time | Second | Rise Time Def=10%-90%: Time at 90% level - Time at 10% level<br>Rise Time Def=20%-80%: Time at 80% level - Time at 20% level |

| | | |
|---|---|---|
| Jitter RMS | Second | 1σ width of the histogram at the eye crossing point |
| Jitter p-p | Second | Full width of histogram at the eye crossing point |
| Crossing Percentage | % | Crossing Height / Amplitude × 100 |
| Opening Factor | None | (Level One - $\sigma_{one}$) - (Level Zero + $\sigma_{zero}$) / Amplitude |
| Signal/Noise Ratio | None | (Level One - Level Zero)/( $\sigma_{one}$+ $\sigma_{zero}$)<br>  |
| Duty Cycle Distortion | Second | $\lvert T_{rise\ middle} - T_{fall\ middle} \rvert$ |
| Duty Cycle Distortion (%) | % | Duty Cycle Distortion (s)/ Bit period × 100 |
| Level Zero | Voltage | Histogram mean for level zero |
| Level One | Voltage | Histogram mean for level one |
| Level Mean | Voltage | (Level Zero + Level One) / 2 |
| Amplitude | Voltage | Level One - Level Zero |
| Height | Voltage | (Level One - 3 $\sigma_{one}$) - (Level Zero + 3 $\sigma_{zero}$) |

| | | |
|---|---|---|
| | |  |
| Width | Second | Bit Period - 2 × 3 × Jitter RMS |

- Bit Period = 1/Bit Rate
- Input Amplitude = Setting of Level One - Setting of Level Zero
- $T_{rise\ middle}$ = The time at which the rising edge cross the middle threshold (50%)
- $T_{fall\ middle}$ = The time at which the falling edge cross the middle threshold (50%)

**Overlaying the results on the waveform**

1. Select the **Overlay** check box under Result.
2. The following results are displayed on the screen.
    - Jitter p-p, Level Zero, Level One, Amplitude, Height (V), Width

**Saving Results into File**

You can save the results as text file.

1. Click **Export to Csv**, the **Save Eye Result** dialog box is displayed.
2. Type your desired file name, then click **Save**.

***Example of Result***

```
# E5071C ENA Option TDR Simulated Eye Results
# 7/1/2010 3:10:06 PM
#
Level Zero, 0.00337131636124
Level One, 0.392246236818
Level Mean, 0.19780877659
Amplitude, 0.388874920457
Height, 0.372870737968
```

Width, 9.94598885146E-10
Opening Factor, 0.986281636548
Signal / Noise, 72.894992429
Duty Cycle Distortion, 6.12868274149E-14
Duty Cycle Distortion (%), 0.00612868274149
Rise Time, 4.69421997336E-11
Fall Time, 4.69412762334E-11
Jitter (PP), 6.25E-12
Jitter (RMS), 9.00185809062E-13
Cross Point (%), 49.9268781576

### Scaling The Eye Diagram

By default, the eye diagram is set to **Auto Scale**. You can also set the scale manually.

1. Click on **Scale/Mask**.
2. Select the **Manual** radio button. This will activate the **Scale / Div** and **Offset** options.
3. Click the entry box **Scale / Div** and input the Y axis scale value.
4. Click the entry box for **Offset** and input the Y axis offset value.

#### Mask Pattern

See Using Mask Test for more information.

### Injecting Jitter

See Using Jitter Injection for more information.

## Selecting Bit Pattern

- Overview
- Bit Pattern Type
- Settings Parameters of Bit Stream

### Other topics about Eye Diagram and Mask Test

**Overview**

E5071C-TDR can provides simulated eye diagram analysis capability, eliminating the need for a hardware pulse pattern generator. The virtual bit pattern can be selected from:

- Pseudo-Random Bit Sequence
- K 28.5
- User Custom
- Statistical

**Bit Pattern Type**

The following Bit Patterns can be used to develop an Eye Diagram:

| Bit Pattern | Description |
|---|---|
| PRBS | Pseudo-Random Bit Sequence. An industry standard created from a specified pattern length. For example, when 2^7 is selected, 127 [(2^7) -1] unique data 'words' are assembled according to the industry standard. |
| K 28.5 | Industry standard developed by IBM which includes comma (control) characters. The pattern is "00111110101100000101" (20 bits). |
| User | Bit Patterns that you have created. |
| Statistical | Bit Patterns produced via statistical calculations of jitter specification When this option is selected, eye diagram is displayed as "Statistical" type. When Jitter Injection is turned ON, this option is set as the default selection. Refer to Using Jitter Injection. |

In the user bit pattern, you can set the same bit pattern as the Pseudo-Random Bit Sequence. However you can get much better resolution in result when you use PRBS.

**Selecting Bit Pattern**

1. Select the **Eye/Mask** Tab.
2. Select your desired bit pattern at **Type** under **Bit Pattern**.

3. If you select the **PRBS** option, **length** is activated. Then, select **length** under **Bit Pattern**.

#### Using a User Bit Pattern

You can easily create user (custom) bit patterns. The length of bit should be from 2 to 32768 (2^15). The pattern with only either 0 or 1 can not be accepted (ex. 00, 111, 0000).

Defining/Saving User Bit Pattern

1. Select the **Eye/Mask** Tab.
2. Select **User** at **Type** under **Bit Pattern**, then **User Pattern** is activated.
3. Click **User Pattern**, then **Bit Pattern Editor** is displayed.
4. Type "0" or "1" to create your bit pattern.
5. Click **OK**, then the **Save Bit Pattern** Dialog box is displayed.
6. Type your desired file name, then click **Save**.

NOTE Saving pattern to the file must be required when you use the user pattern.

Recalling User Bit Pattern

1. Select the **Eye/Mask** Tab.
2. Click **User Pattern**, then **Bit Pattern Editor** is displayed.
3. Click **Load**, then **Load Bit Pattern** Dialog box is displayed.
4. Select your desired file name, then click **Open**.
5. Click **Ok** to exit **Bit Pattern Editor**.

#### Settings Parameters of Bit Stream

The following parameter can be set for the bit stream.

| Label | Description |
|---|---|
| One Lv. | Eye Diagram Y-axis scaling for bit "1" in volts. Negative voltages are allowed. For Differential Eye Diagrams, these scale values are doubled. |
| Zero Lv. | Eye Diagram Y-axis scaling for bit "0" in volts. Negative voltages are allowed. For Differential Eye Diagrams, these scale values are doubled. |
| Data Rate | The speed in bits/second which data is transferred over a circuit or a communications line. |
| Rise Time | The time that it takes a signal to transition from a low to a high condition. Maximum value is 40% of Bit width (Bit width =1/Bit Rate). The time can be defined by either "10-90%" or "20-80%". The rise time settings in EYE/MASK mode and TDR/TDT mode are independent. |

**Defining the parameters**

1. Select the **Eye/Mask** Tab.
2. Click the text box of desired parameter under **Stimulus**, then the **Entry** dialog box is displayed.
3. Type your desired number by clicking numeric keys on the **Entry** dialog box.

## Using Mask Test

- Overview
- Defining Mask
- Executing Mask Test

### Other topics about Eye Diagram and Mask Test

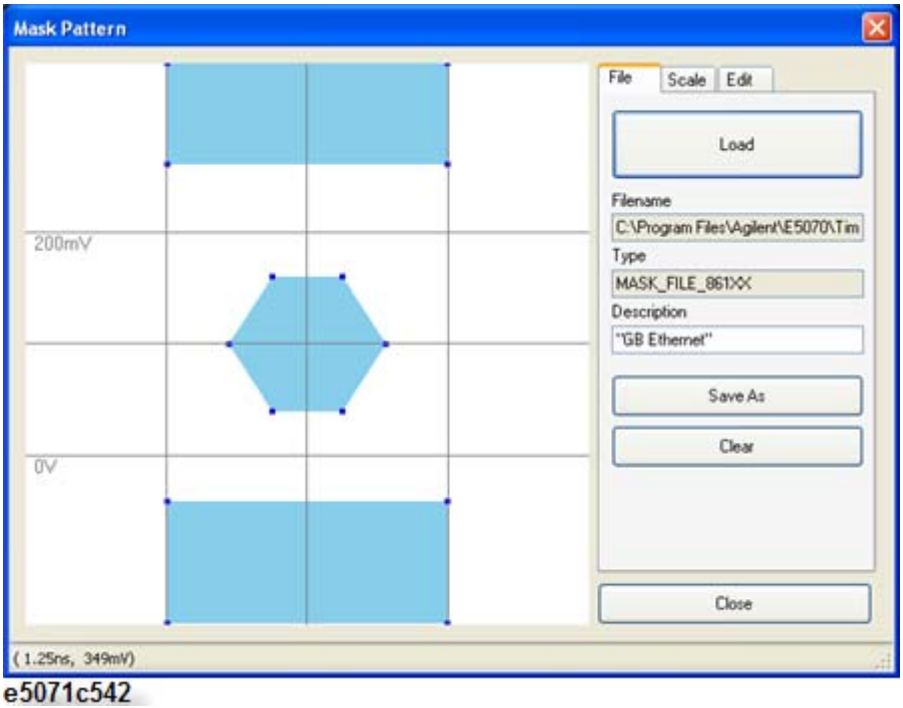
**Overview**

The mask test allows you to verify that a displayed waveform complies with industry-standards definitions for electrical waveforms. To comply with the industry standard, the input waveform must remain outside the shaded mask regions. The mask testing is available in **Scale/Mask** under the **Eye/Mask** tab. The E5071C-TDR uses the same format as Infiniium DCA (86100C), therefore, you can use the MASK file (.msk) stored by DCA (86100C).

The some masks with industry-standards definitions are available in the E5071C directories.

**Defining Mask**

**Opening Mask File**



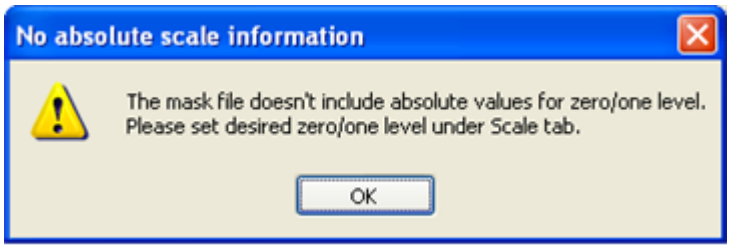

e5071c542

| Items | Description |
|---|---|
| File Name | Shows the file name and location which is currently selected. The pre-defined files and **templates files** are available under C:\Program Files\Agilent\E5070\TimeDomain\Masks. |

| | |
|---|---|
| | <table><tr><th>File Name</th><th>Description</th></tr><tr><td>Template_Deca.msk</td><td>Decagon shape eye mask (10 points)</td></tr><tr><td>Template_Hexa.msk</td><td>Hexagon shape eye mask (6 points)</td></tr><tr><td>Template_Octa.msk</td><td>Octagon shape eye mask (8 points)</td></tr><tr><td>Template_Recta.msk</td><td>Rectangular shape eye mask (4 points)</td></tr></table> |
| Type | Shows the Mask File Identifier. In case of Infiniium DCA, this identifier should be "MASK_FILE_861XX". However, the E5071C does not care of this. Even if this is other than "MASK_FILE_861XX", the E5071C accepts it.<br>As the pre-installed MASK files are compatible with Infiniium DCA, "MASK_FILE_861XX" is displayed when you use pre-installed MASK files. It is not possible to change this in the **Mask Pattern** Dialog box. |
| Description | Shows the description of MASK file. You can change this in the **Mask Pattern** Dialog box. |

Loading Mask File

1. Select the **Eye/Mask** tab.
2. Click **Mask Pattern** under **Scale/Mask**, then the **Mask Pattern** Dialog box is displayed.
3. Select the **File** tab.
4. Click **Load** , then the **Load Mask Pattern** Dialog box is displayed.
5. Select your desired mask file, then click **Open**.
    - When the following message prompts, define the values of the Logic 1 Lv and Logic 0 Lv in the scale tab and save the file.

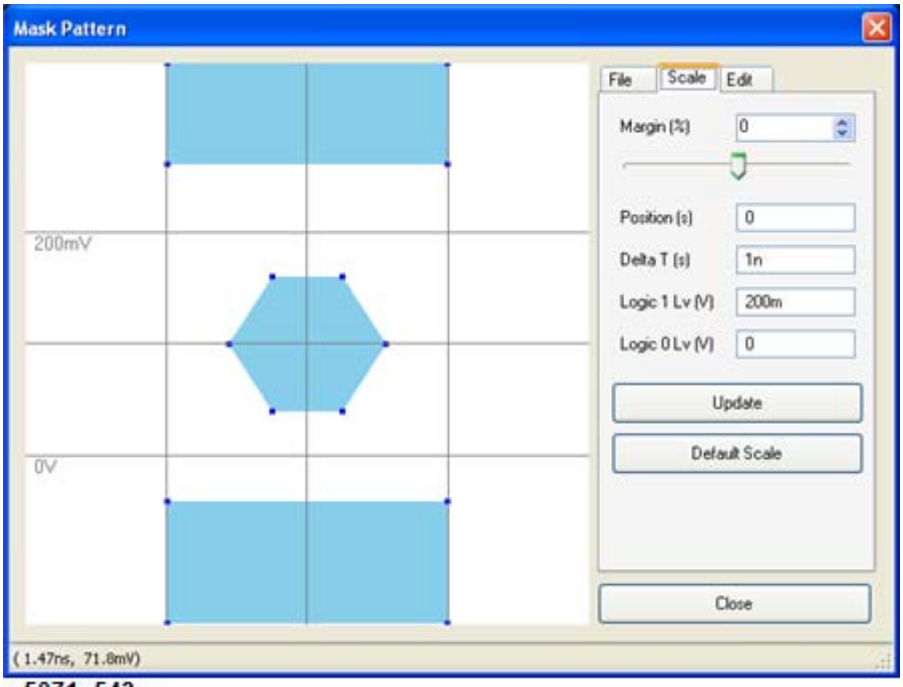


e5071c506

6. Click **Close** to exit the **Mask Pattern** Dialog box.

The pre-defined files are read-only file. As you cannot overwrite on them, save the file by clicking **Save As**.

**Scaling Mask**



e5071c543

| Parameters | Description |
|---|---|
| Margin (%) | Set the size of the mask margin. Mask margins are used to determine the margin of compliance for a standard or scaled mask |
| Position (s) | Move X-axis location of mask |
| Delta T (s) | Change X-axis width of mask |
| Logic 1 Lv (V) | Change voltage of logical 1. |
| Logic 0 Lv (V) | Change voltage of logical 0 |

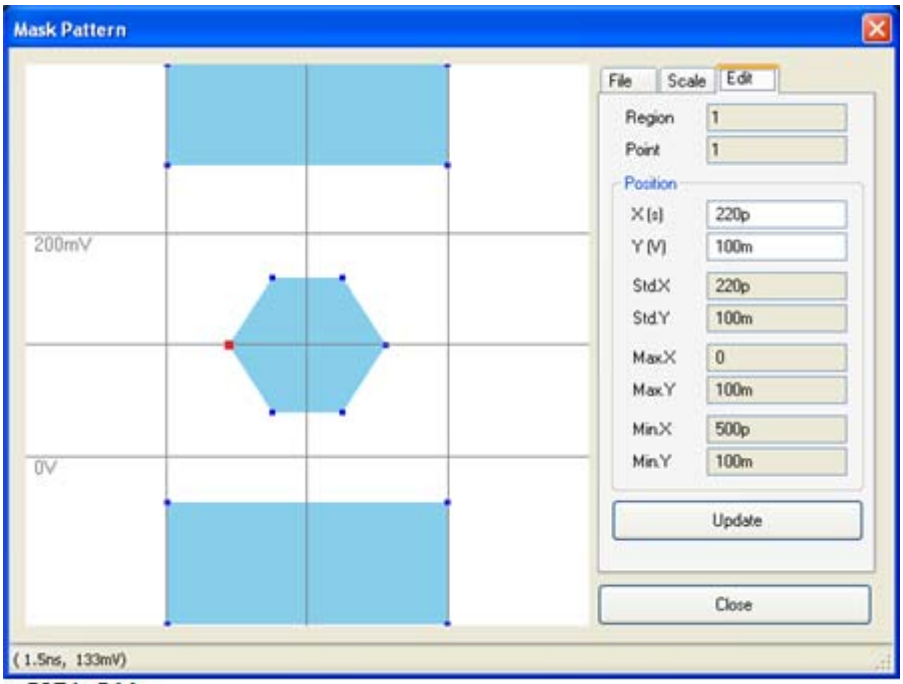
Changing Margin, Position, Delta T and Logic 0/1 Lv

1. Load the Mask Pattern.
2. Select the **Scale** tab.
3. Change the margin number using slider, or typing number.
4. Click the box at your desired parameter of position, delta T and logic 0/1 level.
5. Type the number you want to set.
6. Click **Update** to apply the entered number.
7. Click **Close** > **Yes** to save the modified scale.

Set Value at Default

Click **Default Scale** to set the parameter at default.

**Editing Mask**



e5071c544

| Parameter | Description |
|---|---|
| Region | The currently selected region number. The region number defines a mask violation area (or polygon). |
| Point | The currently selected point number. The point number defines a point in the region. |
| X (s) and Y (V) | The X and positions for the selected point. You can enter the number to change the selected position location. |
| Std.X and Std.Y | The X and Y positions when **Margin** in **Scale** Tab is selected at 0%. This shows the positions of STD in the .msk file. |
| Max.X and Max.Y | The X and Y positions when **Margin** in **Scale** Tab is selected at 100%. This shows the positions of MARGIN_MAX in the .msk file. |
| Min.X and Min.Y | The X and Y positions when **Margin** in **Scale** Tab is selected at -100%. This shows the positions of MARGIN_MIN in the .msk file. |

1. Load the Mask Pattern.
2. Select the **Edit** tab.
3. Move the position of points by either way.
   - Using Mouse

1. Click the desired point on the figure of mask, then the point is selected (the point color becomes red).
2. Move the point with drag and drop on the mouse to your desired position.

- Entering Position
    1. Click desired point on the figure of mask, then the point is selected.
    2. Click entry box of X(s) under **position**, then type number for X axis.
    3. Click entry box of Y(V) under **position**, then type number for Y axis.
    4. Click **Update** to apply the entered number on the selected point.

4. Click **Close** > **Yes** to save the modified pattern.

**Executing Mask Test**

1. Select the **Mask Test** check box under **Scale/Mask**.
2. Click **Draw Eye** to redraw Eye pattern and Mask.
3. The mask and pass/fail result is displayed on the screen.

### Available Masks

The following tables list the available standard masks files that you can use. The E5071C-TDR uses the same format as Infiniium DCA (86100C). The following available masks are the same as ones stored in DCA and stored under C:\Program Files\Agilent\E5070\TimeDomain\Masks.

- SDH/SONET
- Fibre Channel
- Ethernet
- Infiniband
- Serial ATA
- XAUI
- Electrical
- G_984_2

Other topics about Eye Diagram and Mask Test

**SDH/SONET**

| File Name | Standard |
|---|---|
| STM000_OC1.msk | STM0/OC1, 51.8 Mb/s |
| STM001_OC3.msk | STM1/OC3, 155.5 Mb/s |
| STM004_OC12.msk | STM4/OC12, 621.8 Mb/s |
| STM008_OC24.msk | STM8/OC24, 1244 Mb/s |
| STM016_OC48.msk | STM16/OC48, 2.488 Gb/s |
| STM016_G.691_V2.0.msk | STM16/G.691 |
| STM064_OC192.msk | STM64/OC192, 9.953 Gb/s |
| STM256_OC768.msk | STM256/OC-768, 39.812 Gb/s |
| STM64_OC192FEC_10_664.msk | STM-64/OC-192 FEC, 10.664 Gb/s |
| STM64_OC192FEC_10_709.msk | STM-64/OC-192 FEC, 10.709 Gb/s |
| STM64_OC192SuperFEC_12_5.msk | STM-64/OC-192 Super FEC, 12.5 Gb/s |

**Fibre Channel**

| File Name | Standard |
|---|---|
| FC0133.msk | FC133, 133 Mb/s |
| FC0266.msk | FC266, 266 Mb/s |
| FC0531.msk | FC531, 531 Mb/s |
| FC1063.msk | FC1063, 1063 Mb/s (Revision 1 of the standard) |
| FC1063_PI_R13_Dec01.msk | FC1063, 1063 Mb/s (Revision as of Dec., 2001) |
| FC2125.msk | FC2125, 2125 Mb/s (Revision 1 of the standard) |
| FC2125_PI_R13_Dec01.msk | FC2125, 2125 Mb/s (Revision as of Dec., 2001) |
| FC4250_PI_R13_Dec01.msk | FC4250, 2350 Mb/s |
| 008.5000-FC-PI-4 Multimode.msk | FC8.5 Gb/s Fibre Channel |
| 10xFiberChannel.msk | 10X Fibre Channel, 10.51875 Gb/s |
| 010.51875-SFP+_Rx_C'_10GE.msk | 10X Fibre Channel, SFP+Rx C' |
| 010.51875-SFP+_Tx_B_10GE.msk | 10X Fibre Channel, SFP+Tx B |
| 010.51875-XFP_B'_10GE.msk | 10X Fibre Channel, XFP B' |
| 010.51875-XFP_C'_10GE.msk | 10X Fibre Channel, XFP C' |

**Ethernet**

| File Name | Standard |
|---|---|
| 01xGbEthernet.msk | GB Ethernet, 1250 Mb/s |
| 02xGbEthernet.msk | 2XGB Ethernet, 2.500 Gb/s |
| 10GbE_9_953_May02.msk | 10GB Ethernet, 9.953 Gb/s |
| 10GbEthernet_10_3125.msk | 10GB Ethernet, 10.3125 Gb/s (Revision 1 of the standard) |
| 10GbE_10_3125_May02.msk | 10GB Ethernet, 10.3125 Gb/s (Revision as of May, 2002) |
| 10_G_Base_LRM_May_2006.msk | 10G BASE LRM, 10 Gb/s |

| | |
|---|---|
| 10xGbEthernet_12_5.msk | 10XGB Ethernet, 12.5 Gb/s |
| 1000BASE-LX10_September_2004.msk | 1000BASE-LX10, 1.25 Gb/s |
| 100BASE-BX10_September_2004.msk | 100BASE-BX10_September_2004.msk |
| 100BASE-BX_LX10.msk | 100BASE-LX10, 100 Mb/s |
| 100BASE-BX_LX10.msk | 100BASE-BX_LX10.msk |
| 10GBASE-LX4.msk | 10BASE-LX4 |
| 010.3125-SFP+_Rx_C'_10GE.msk | 10.3125 Gb Ethernet SFP+ Rx C' |
| 010.3125-SFP+_Tx_B_10GE.msk | 10.3125 Gb Ethernet SFP+ Tx B |
| 010.3125-XFP_B'_10GE.msk | 10.3125 Gb Ethernet XFP B' |
| 010.3125-XFP_C'_10GE.msk | 10.3125 Gb Ethernet XFP C' |

**Infiniband**

| File Name | Standard |
|---|---|
| InfiniBand_Jun01.msk | 2.500 Gb/s |

**Serial ATA**

| File Name | Standard |
|---|---|
| SATA 1.5Gb TX 250 Cycles.msk | SATA TX 250 Cycles, 1.5 Gb/s |
| SATA 1.5Gb TX 5 Cycles.msk | SATA TX 5 Cycles, 1.5 Gb/s |

**XAUI**

| File Name | Standard |
|---|---|
| XAUI-E_Far_May02.msk | XAUI-E Far |
| XAUI-E_Near_May02.msk | XAUI-E Near |

**Electrical**

| File Name | Standard |
|---|---|
| STS1Eye.msk | STS1Eye, 51.8 Mb/s |
| STS3Eye.msk | STS3Eye, 155.5 Mb/s |

**G.984.2**

| File Name | Standard |
|---|---|
| 2488.32_G.984.2_Downstream_March_2003.msk | 2488.32 Mb/s ONU |
| 1244.16_G.984.2_Downstream_March_2003.msk<br>1244.16_G.984.2_UpStream_March_2003.msk | 1244.16 Mb/s ONU |
| 622.08_G.984.2_Upstream_March_2003.msk | 622.08 Mb/s ONU |
| 155.52_G.984.2_Upstream_March_2003.msk | 155.52 Mb/s ONU |

## Storing Data and Setting

### Storing Data and Setting

- Saving/Recalling Setting
- Saving Data
- Saving Touchstone Data
- Saving Displayed Image

Other topics about Enhanced Time Domain Analysis

## Saving / Recalling Setting

- Saving Setting
- Recalling Setting
- Compatibility of State Files

### Other topics about Storing Data and Setting

#### Saving Setting

The setting of E5071C-TDR can be saved and recalled. The state file of TDR measurements has **.tdr** file extension.

1. Setup your configuration which you want to save.
2. Click **File** in the tool bar.
3. Select **Save State** from menu, then the **Save State As** Dialog Box is displayed.
4. Type desired file name.
5. Click **Save**.

When you use 2-channel measurement in the Advanced Mode, the channel 2 setting is also saved into .tdr file.

#### Recalling Setting

1. Click **File** in the tool bar.
2. Select **Recall State** from menu, then the **Recall State As** Dialog Box is displayed.
3. Select the file name of state file whose file extension is **.tdr**.
4. Click **Open**.

#### Compatibility of State Files

##### Compatibility between Mode

The following table shows the compatibility of state file between mode.

| Mode | | | Recalling | | |
|---|---|---|---|---|---|
| | | | Basic | Advanced | |
| | | | | Without Advanced Calibration Methods | With Advanced Calibration Methods |
| Saved File By | Basic | | Y | Y | N |
| | Advanced | Without Advanced Calibration | N | Y | N |

| | | | | | |
|---|---|---|---|---|---|
| | | **Methods** | | | |
| | | **With Advanced Calibration Methods** | N | N | Y |

Y: Recall is possible.

N: Recall is impossible.

### Saving Data

The trace data can be saved in .csv format.

1. Select the trace which you want to get the data.
2. Click **File** in the tool bar.
3. Select **Save Trace Data** from menu, then the **Save Trace As** Dialog Box is displayed.
4. Type desired file name.
5. Click **Save**.

***Example of File***

```
# Channel 1
# Trace 4
Frequency  Formatted Data  Formatted Data
8.49E+06 -5.84E-02 0.00E+00
1.70E+07 -8.27E-02 0.00E+00
2.55E+07 -1.04E-01 0.00E+00
3.40E+07 -1.32E-01 0.00E+00
4.25E+07 -1.52E-01 0.00E+00
5.09E+07 -1.84E-01 0.00E+00
5.94E+07 -2.18E-01 0.00E+00
6.79E+07 -2.70E-01 0.00E+00
.
(data is continued until the end)
.
```

Other topics about Storing Data and Setting

## Saving Touchstone Data

- File Extension
- Saving Touchstone Data

### Other topics about Storing Data and Setting

#### File Extension (SnP Format)

The SnP file format is changed depending on the selected DUT topology.

| File Extension | DUT Topology |
|---|---|
| .s1p | Single Ended 1 |
| .s2p | Single Ended 2, Differential 1 |
| .s4p | Single Ended 4, Differential 2 |

See the Saving data in Touchstone format for touch stone format.

#### Saving Touchstone Data

The trace data can be saved in touch stone format.

1. Click **File** in the tool bar.
2. Select **Save Touchstone** from menu, then the **Save Touchstone As** Dialog Box is displayed.
3. Type desired file name.
4. Click **Save**.

You can not create touchstone files in the following conditions.

- # is displayed at Channel Measurement Status.
- Some cases in Advanced Mode with Advanced Calibration Method

## Saving Displayed Image

- Saving Image
- Inverting Display Color

Other topics about Storing Data and Setting

### Saving Image

The screen image can be saved in bit map or png format.

1. Click **File** in the tool bar.
2. Select **Save Image** from menu, then the **Save Image As** Dialog Box is displayed.
3. Select the file type in **Save as Type** from either **.bmp** or **.png**.
4. Type desired file name.
5. Click **Save**.

### Inverting Display Color

Inverting display color changes the background color of the screen from black to white. When you print out the saved image on a white paper, the background in white color saves a printer ink.

1. Click **File** in the tool bar.
2. Select **Invert Color** from menu, then the background color of the screen is changed from black and white.
3. Save the screen by following the procedure of Saving Image.

## Advanced Waveform Analysis

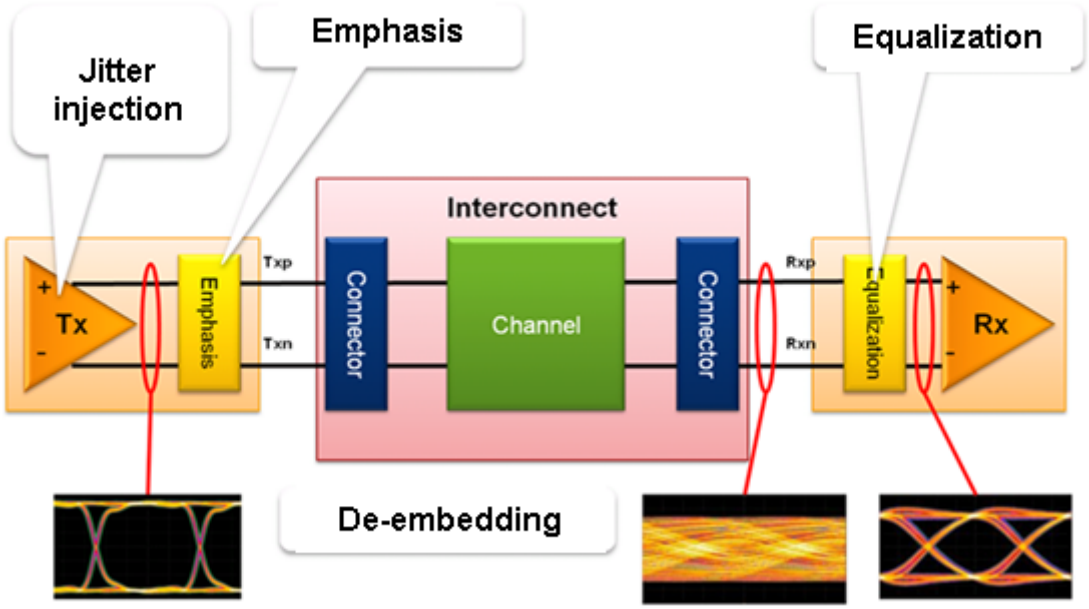
### Advanced Waveform Analysis

- Overview
- Using Jitter Injection
- Using Emphasis
- Using De-embedding
- Using Equalization

Other topics about Enhanced Time Domain Analysis

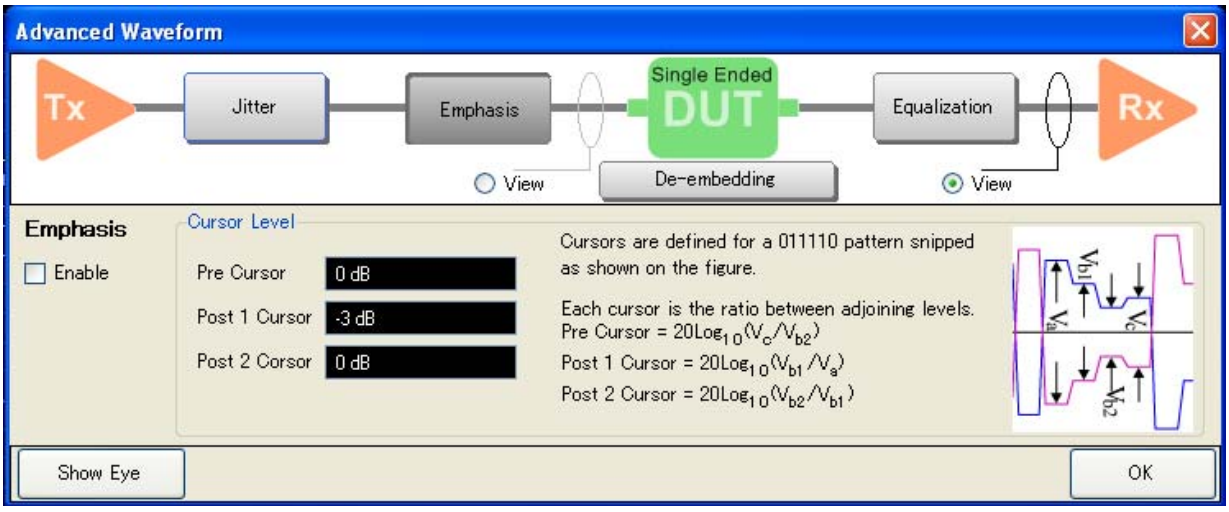
## Overview

In Advanced Waveform Analysis, emphasis and equalization helps to improve the quality of the waveform and subsequently the quality of the eye diagram.




There are two options to display the advance waveform window:

- Click on **Adv Waveform** under **Setup** tab and click one of these buttons: **Emphasis**, **De-embedding**, **Equalization**.
- Click on **Advanced Waveform** button under the Eye/Mask tab



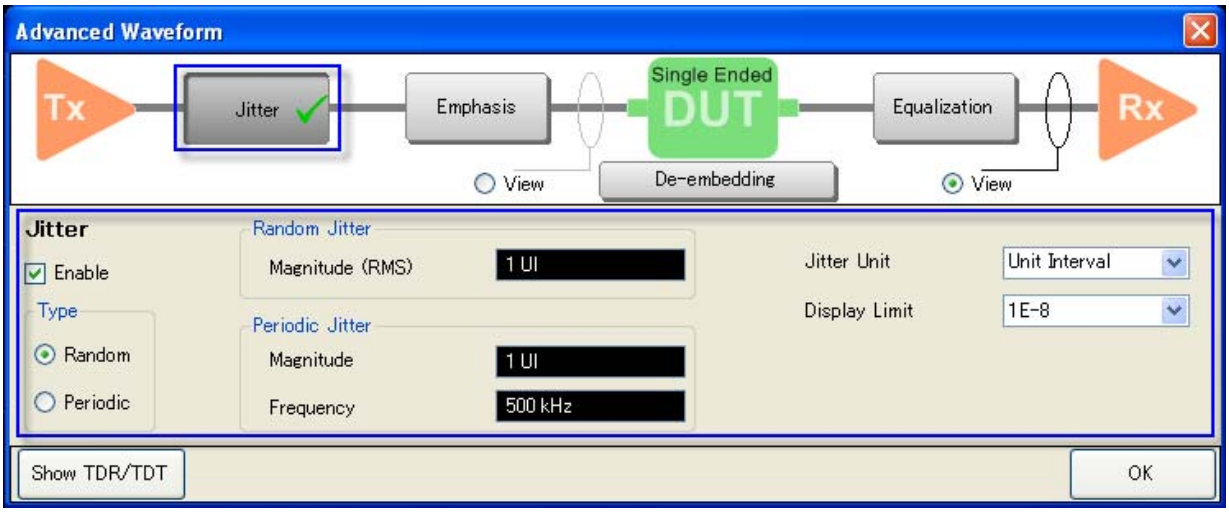
**View** radio button allows you to select the observation point for TDR/TDT or Eye, according to your selection. Changing **View** also affects the trigger mode:

- **View** before the DUT shows the Stimulus View. At this view, trigger mode at the tool bar changes to "STOP".
- **View** after the DUT shows the Response View. At this view, trigger mode changes to "RUN".

NOTE When Stimulus View is selected and the trigger mode "RUN/Single" is selected, the view changes to Response View.

Other topics about Advanced Waveform Analysis

### Using Jitter Injection

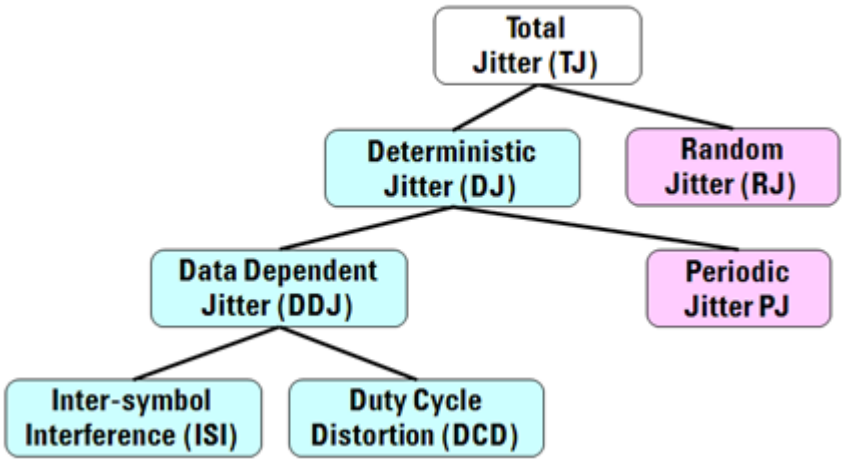



1. To execute jitter injection, click on **Jitter** button and check **Enable**.
2. When Jitter Injection is turned ON, check mark appears on the **Jitter** button. The **Stimulus Type** (under **Eye/Mask** tab) is automatically changed to "Statistical".

   NOTE See also Bit Pattern Type.

There are two types of jitter injection available to choose from:

- Random Jitter - follows the Gaussian distribution and is represented by the rms value of the Random Jitter distribution.
- Periodic Jitter - represented by peak-to-peak value.



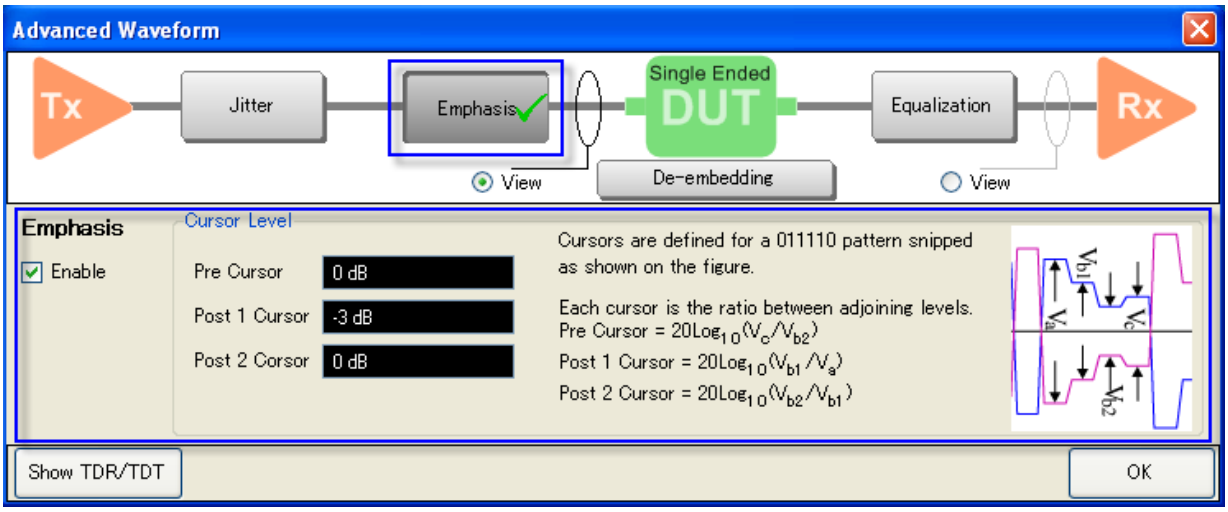

In general, Deterministic Jitter and Random Jitter totals up to Total Jitter (TJ). Deterministic Jitter is bounded by a finite magnitude. It can be broken into jitter which is correlated to the data sequence and jitter that occurs independent of data. Periodic Jitter is data independent.

**Jitter Unit**: You can select from Unit Interval (UI) or Second. Second = Unit Interval / Data Rate

**Display Limit**: This is a probability density limit. Normally, it is not necessary to change this from default setting. This function allows you to execute a fine tuning of skirt of eye diagram. This applied to eye mask result. So that, you can have the same result on the mask test by adjusting this value. This is not applied to the eye result.

Other topics about Advanced Waveform Analysis

## Using Emphasis




1. To execute emphasis, click on **Emphasis** button and check **Enable**.
2. **Cursor Level**:
   - Pre cursor is the ratio between $V_c$ and $V_{b2}$:

   Pre Cursor = 20 $Log_{10}$ ($V_c/V_{b2}$)

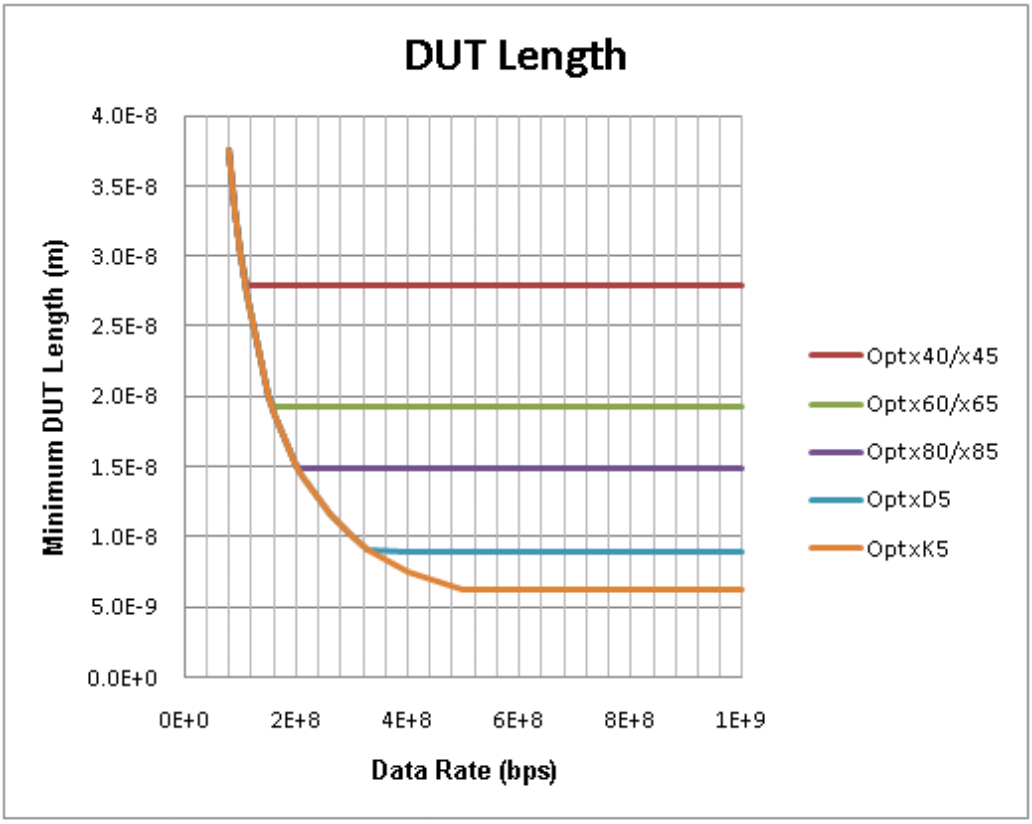
   - Post 1 cursor is the ratio between $V_{b1}$ and $V_a$:

   Post 1 Cursor = 20 $Log_{10}$ ($V_{b1}/V_a$)

   - Post 2 cursor is the ratio between $V_{b2}$ and $V_{b1}$:

   Post 2 Cursor = 20 $Log_{10}$ ($V_{b2}/V_{b1}$)

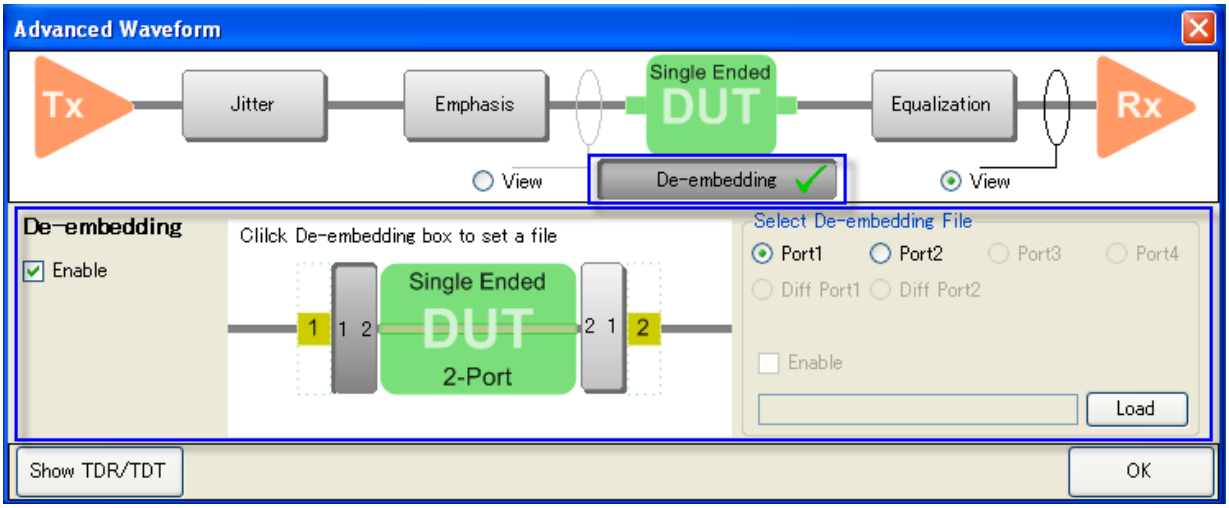
   NOTE When the emphasis is turned on, the DUT length should be larger than (1/Data Rate) × 3. The following graph shows the Data Rate vs. minimum DUT length for each frequency model.




e5071c607

## Other topics about Advanced Waveform Analysis

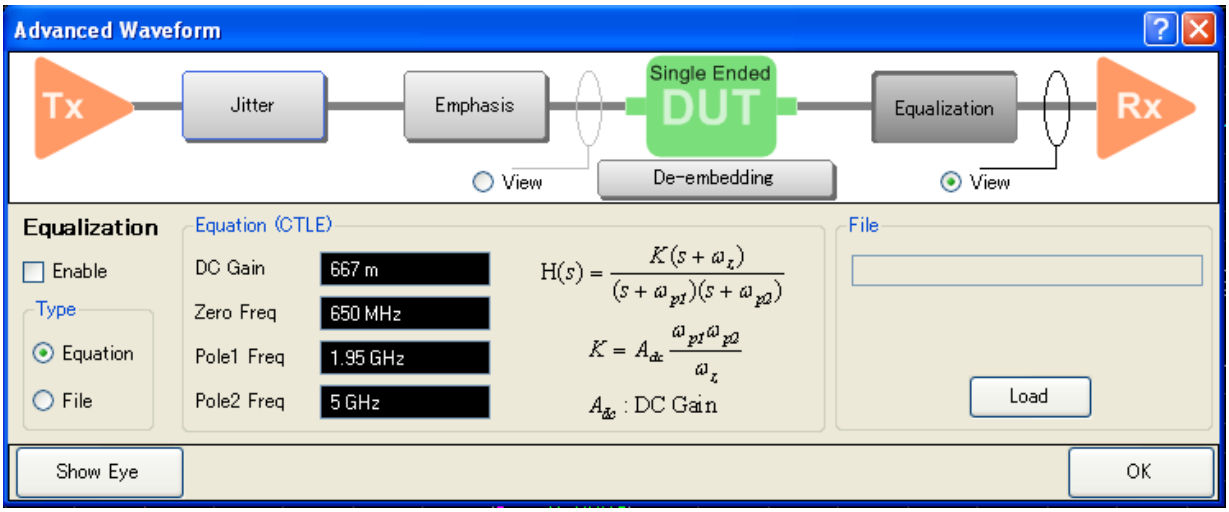
### Using De-embedding




1. To execute de-embedding, click on **De-embedding** button and check **Enable**.
2. If differential topology is selected, load S4P or S2P file. For single ended topology, load S2P file.
3. You can enable each SnP file by checking **Enable** in **Select De-embedding file**.
4. The port orientation is shown and it defers depending on the DUT topology.

Other topics about Advanced Waveform Analysis

## Using Equalization




To execute equalization, click on **Equalization** button and check **Enable**.

You can either select to use the equation by specifying the variables or equation file.

You must specify four variables in this equation in order to generate the filter. These variables are:

- DC Gain
- Zero Frequency
- Pole 1
- Pole 2

***Example of Equalization File***

```
!Agilent Technologies
!E5071C Option ENA-TDR Equalization File
# Hz dB
1000000000,-1.53E+01,-3.27E-01
2000000000,-1.52E+01,-7.34E+01
3000000000,-1.03E+01,-8.63E+01
4000000000,-9.88E+00,-1.00E+02
-----------------------------------
! Comment
# Hz MA or DB or RI
Freq1, Data1a, Data1b
Freq2, Data2a. Data2b
Freq3, Data3a, Data3b
```

```
.
.
```

Where

MA: Dataxa = Magnitude (Linear), Dataxb = Phase

DB: Dataxa = Magnitude (dB), Dataxb = Phase

RI: Dataxa = Real, Dataxb = Image

NOTE

- If there is no Dataxb, Dataxb is taken as 0.
- If there is no MA/DB/RI, MA is selected.
- ! (comment) can be place at any lines (Not only top lines but also in the middle).
- If Freq n > Freq (n+1), then the data at freq (n+1) is ignored.
- If frequency span of setting on the E5071C is wider than frequency range of data, the extrapolation is applied.

Error Messages

- File Name Error: is the same as ENA error. This message appears when an error exists in the file name and hence a command is not executed correctly.
- Failed To Read Error. This message appears when the file cannot be opened or if the file does not exist. It also occurs when the format of the file is incorrect or the number of data line is less than 2 or more than 10001.

Other topics about Advanced Waveform Analysis

## Advanced Mode

### Advanced Mode

- Overview
- Activating and Deactivating Advanced Mode
- Frequently Used Functions in Advanced Mode (ENA Standard Functions)

Other topics about Enhanced Time Domain Analysis

## Overview

- Feature
- Consideration in Advanced Mode

### Other topics about Advanced Mode

**Feature**

The following table shows the comparison of Advanced and Basic Modes.

| Mode | Feature | TDR Application (User Interface) | Hard Key on Front Panel | Soft Key (right sideof the screen) | Indicators |
|---|---|---|---|---|---|
| Advanced | • Customized TDR/S-parameter measurements in TDR (Ch1)<br><br>Such as:<br><br>o More Marker Functions<br>o Limit Test<br>o TRL Calibration<br>o Two channel measurements (Ch1 for TDR/Ch2 for Network Measurement) | Available | Available | Available | TDR indicator: Yellow<br><br>In Instrument status bar, SVC: Blue (the measurement performance is not guaranteed) |
| Basic | Easy of use | Available | Locked excepts **Trace Prev**, **Trace Next**, **Trace Max** and | Hidden | TDR indicator: Blue<br><br>In Instrument status bar, |

| | | | | | |
|---|---|---|---|---|---|
| | | | keys in numeric key. | | SVC: Normal |

The sample measurements are shown in the measurement examples.

**Consideration in Advanced Mode**

In the Advanced Mode, you can access all setting of E5071C. The setting you changed may affect the measurement unexpectedly. The measurement may not be correct if you have such a case. Therefore, the measurement performance is not guaranteed in the Advanced Mode.

It is known that changing the following setting causes incorrect measurement.

| Hard key | Do not use |
|---|---|
| Meas | When Use Advanced Calibration Mode is not checked:<br>Single end 1 port: Parameters related with ports 2, 3 and 4<br>Single end 2 port/Differential 1 port: Parameters related with ports 3 and 4 |
| Display | Data Math, Equation Editor |
| Calibration | When Use Advanced Calibration Mode is not checked: All |
| Sweep Setup | Start/Stop/Center/Span<br>Points<br>Sweep Type<br>Frequency Offset |
| Marker Function | Marker -> Start, Marker -> Stop |
| System | Measurements with Multi-port Test Set (E5091A/E5092A) |

Here is another caution in the Advanced Mode.

| Function | | Note |
|---|---|---|
| Meas | Change parameter when Port extension on | This may cause to tilt the wave form in the time domain . Use TDR GUI to select parameter. |
| Display | Increasing Num of Trace | This may cause to tilt the wave form in the time domain. |
| Analysis | Fixture Simulator | This may cause to tilt the wave form in the time domain. |
| | Transform > Window > Impulse width or Step | The range of these settings is narrower than one of the Rise Time in the TDR GUI. Use |

| | | |
|---|---|---|
| | Rise | TDR GUI to set the rise time. |
| Calibration | Increasing Port Extension | This may cause to exceed the DUT length limitation. |
| Scale | Increasing Electrical Delay | This may cause to exceed the DUT length limitation. |
| Ave | IF Bandwidth | Even if you have narrow IF Bandwidth, you may not have noise reduction at lower frequency. |

## Activating and Deactivating Advanced Mode

- Activating Advanced Mode
- Deactivating Advanced Mode

### Other topics about Advanced Mode

**Activating Advanced Mode**

1. Select the **Setup** tab.
2. Click **Advanced Mode** under **More Function**, then **Advanced Mode** dialog box is displayed.
3. When you want to perform a calibration such as TRL calibration in Advanced Mode, select the check box named **Use Advanced Calibration Methods** in dialog box.

   NOTE When you select the check box, the calibration you did in basic mode is cleared.
4. Click **Yes** to start the Advanced Mode.
5. The following settings are changed.
   - Softkeys are displayed on the right side of screen
   - All hard keys are unlocked.
   - SVC in the instrument status bar turns in blue.

**Deactivating Advanced Mode (Returning to Basic Mode)**

1. Select the **Setup** tab.
2. Click **Basic Mode** button under **More Function**, then **Advanced Mode** dialog box is displayed.
3. Click **Yes** to start the Basic Mode.
4. E5071C application is re-started and all setting has been reset.

### Frequently Used Functions in Advanced Mode (E5071C Standard Functions)

- For Channel 1
- For Channel 2

## Other topics about Advanced Mode

This section provides the links with the E5071C standard function.

**For Channel 1**

As the channel 1 is for TDR measurement, you can use some of functionality of E5071C standard function.

- Cal (Mechanical, ECal, TRL) (Only when Use Advanced Calibration Mode)
- Marker Search
- Limit Test

**For Channel 2**

As the channel 2 can be used as one channel of E5071C, You can use all of functionality of E5071C.

- Measurement Parameter
  - Single-ended & Differential
- Format
- Scale
- Display Allocation
- IFBW
- Cal (Mechanical, ECal, TRL)
- Stimulus
  - Power
  - NOP
  - Sweep Type
  - Segment Sweep
- Trigger
- Marker, Marker Search, Marker Function
- Limit Test
- Port Z conversion
- De-embedding
- Save/Recall
- Save data (csv, snp)

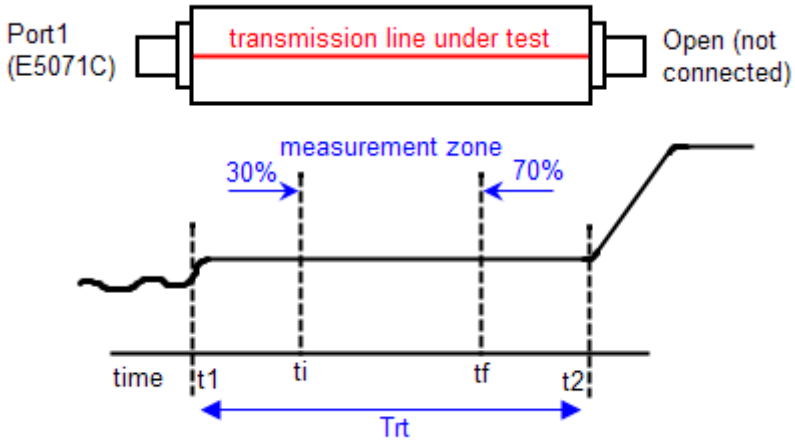
## Measurement Example

### Measurement Examples

- PCB Impedance Measurement (Advanced Mode Example)
- 2 Channel Measurement Example (Advanced Mode - 2 Channel Example)

## PCB Impedance Measurement

Measuring the characteristic impedance of single-ended transmission line within a defined zone.




Ensure E5071C-TDR application is in Advanced Mode in order to follow this example.

Before beginning to make measurement on PCB impedance, we need:

1. Set Measurement Conditions
2. Define Measurement Zone
3. Set Measurement Zone and Measure Characteristic Impedance

Other topics about Measurement Examples

### Set Measurement Conditions

1. Click **Setup** > **Setup Wizard**.
2. Select **Deskew** and click **Next**.
3. Select **Single Ended 1-Port** and click **Next**.
4. Follow the instruction and click **Deskew**. When finish, click **Next**.
5. Follow the instructions and click **Measure**. When finish, click **Next**.
6. Click **Apply** and **Finish**.

### Define Measurement Zone

Using E5071C-TDR application GUI,

1. Disconnect the DUT.
2. Click **Trace** > **1** and double-click on the trace.
3. Click **TDR/TDT** > **Parameters** > **Linear** (under **Format**).
4. Click **AutoScale** > **X & Y** to adjust timebase for the entire response of the DUT is visible.
5. Click **Marker** > **1**.

Using Softkey (on the right side of the screen),

6. Click **Marker Search** > **Target** > **Target Value**.
7. Type 0.5 target value into the entry box. This steps locate the instant t1 on the TDR waveform where the open discontinuity occurs.
8. Connect the DUT.

Using E5071C-TDR application GUI,

9. Click **Marker** > **2**.

Using Softkey (on the right side of the screen),

10. Click **Marker Search** > **Target** > **Target Value**.
11. Type 0.5 target value into the entry box. This steps locate the instant t2 on the TDR waveform where the open discontinuity occurs.

Calculate the measurement zone using the formulas below.

Compute the round trip propagation time of the transmission line:

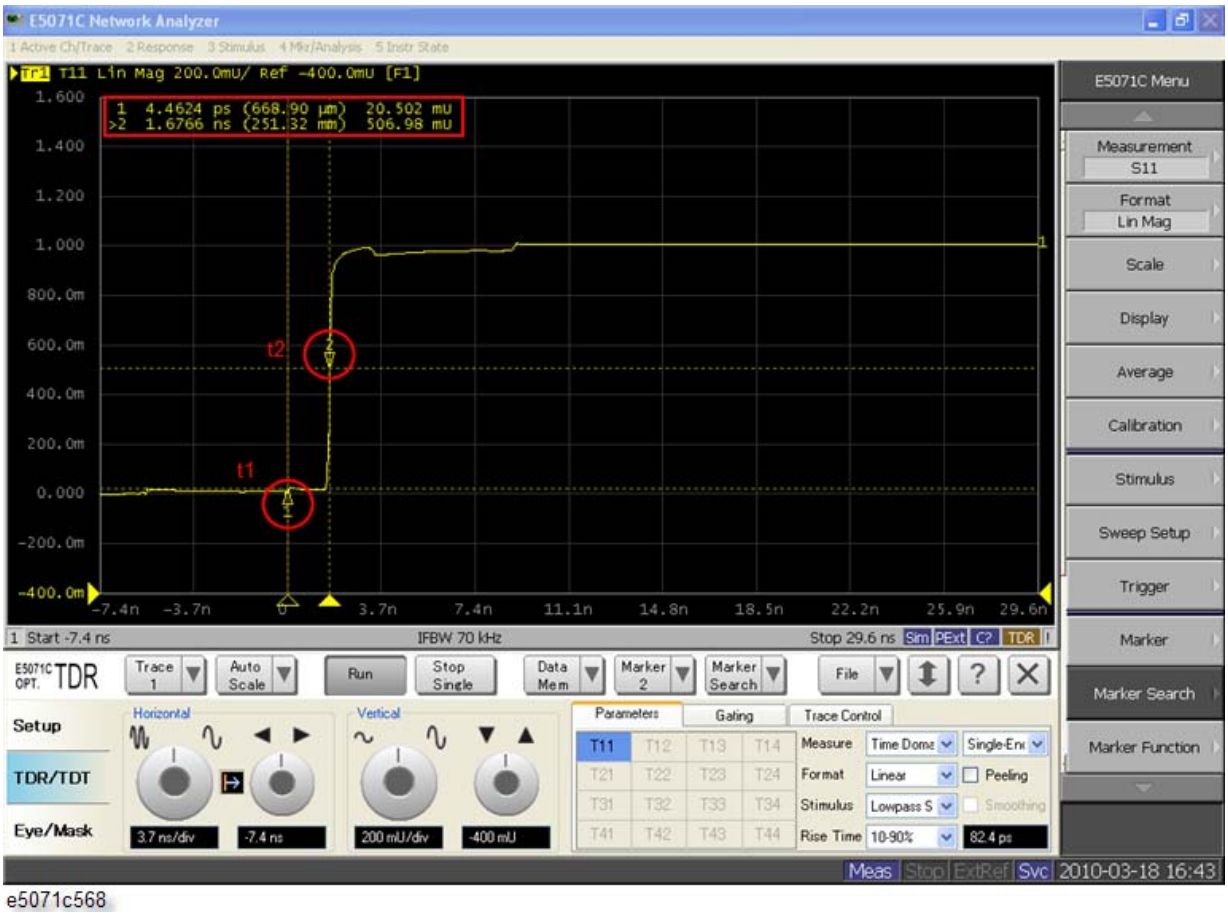
$$Trt = t2 - t1 = 1.68\ ns - 0 = 1.68\ ns \text{ (refer to the figure below)}$$

Determine the initial instant, ti, of the measurement zone:

$$ti = t1 + 30\%Trt = 0 + (0.3) \times 1.68\ ns = 0.50\ ns$$

Determine the final instant, tf, of the measurement zone:

$$tf = t1 + 70\%Trt = 0 + (0.7) \times 1.68\ ns = 1.18\ ns$$



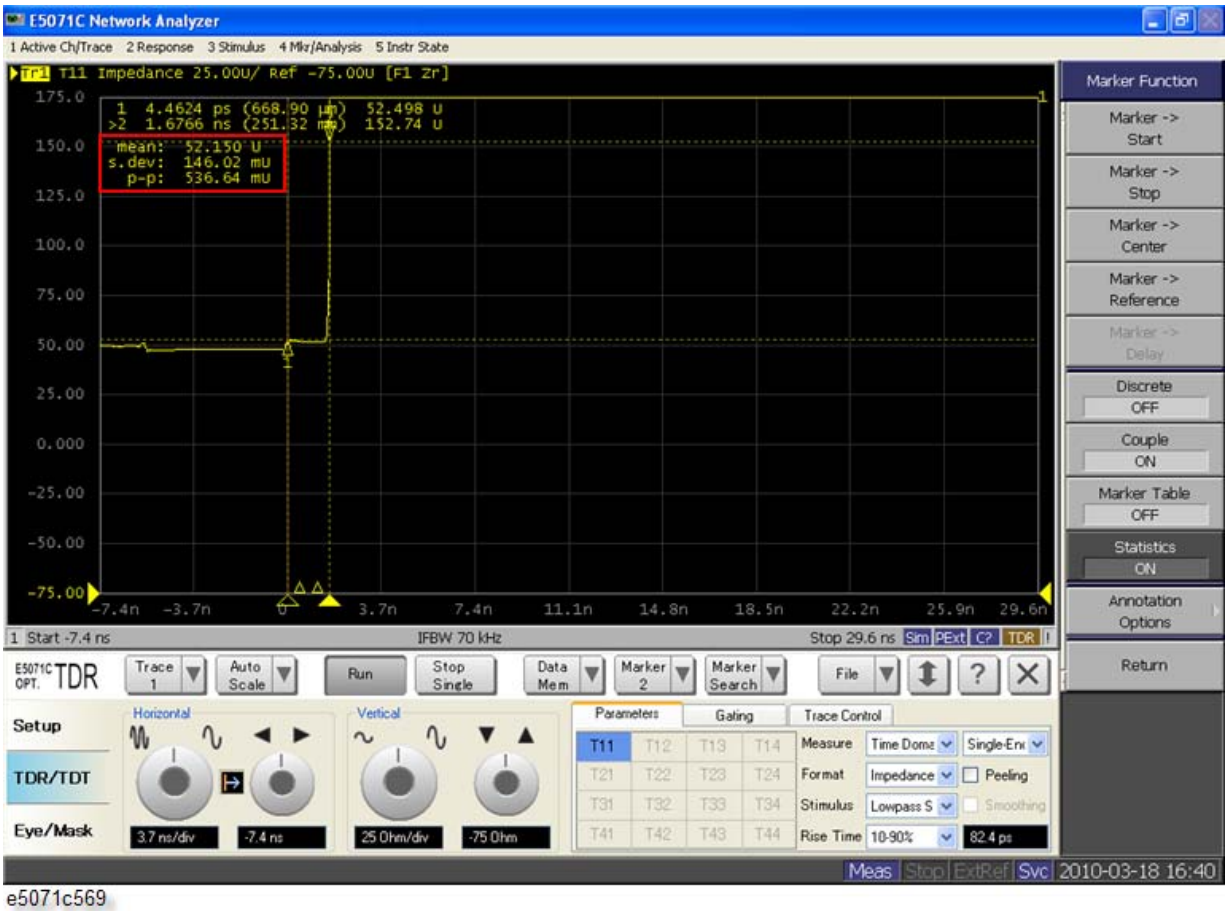
e5071c568

**Set Measurement Zone and Measure Characteristic Impedance**

1. Click **TDR/TDT** > **Parameters** > **Impedance** (Under **Format**).
2. Click **TDR/TDT**. Type in 25 into scale/div (left) entry box under **Vertical**.
3. Click **TDR/TDT**. Type in -75 into position (right) entry box under **Vertical**.

Using Softkey,

4. Click **Marker Search** > **Search Range** > **Start**. Type in 0.5 ns in the entry box.
5. Click **Marker Search** > **Search Range** > **Stop**. Type in 1.18 ns in the entry box.
6. Click **Marker Search** > **Search Range** > **Search Range** to enable search range.
7. Click **Marker Function** > **Statistics** to enable statistics and measure the characteristic impedance.



e5071c569

The characteristic impedance is the mean value, 52.15 U.

## 2 Channel Measurement Example

- Overview
- Procedure

Other topics about Measurement Examples

### Overview

This sample shows a 2 channel measurement in the Advanced Mode.

2 Channel measurement allows you to make measurements:

- TDR measurement on channel 1
- More customized S-parameter measurement on channel 2

Here is the sequence of this sample.

- Setup for channel 1
    1. Deskew
    2. Auto DUT length
    3. Set rise time
    4. Setup limit table for the trace 1
- Setup for channel 2
    1. Setup start and stop frequency and IF Bandwidth.
    2. Setup Sdd11 of Balance-Balance measurement
    3. Setup limit table for the trace 1
    4. Full 4-port calibration with ECal
- Measurement for channel 1
    1. Trigger
    2. Auto scale
- Measurement for channel 2
    1. Trigger

### Procedure

#### Preparation for 2 Channel Measurement

1. Connect E-Cal on the USB port on the front panel.
2. Connect cables to all test ports.
3. Activate Advanced Mode (Do not select the **Use Advanced Calibration Method** check box)
4. Click **Stop Single** to stop the trigger.

Using Hardkey/Softkey

1. Press **Display** > **Allocate Channel** >  to have 2 channels.
2. Press **Trigger** > **Trigger Scope** to select Active Channel.
3. Press **System** > **Misc Setup** > **Beeper** > **Beep Waning** to turn off the beep warning.

#### Setup for Channel 1

1. Click the **Setup** tab on the GUI.
2. Click the **Setup Wizard** (under **Basic**) at **Setup** area.
3. Set the measurement condition using the **Setup Wizard**:
    a. Select **Deskew** (under **Error Correction**) and click **Next >**.
    b. Click the **Differential 2-Port** button, and click **Next >**.
    c. Click the **Deskew** button, and click **Next >**.
    d. Click the **Measure** button, and click **Next >**.
    e. Set the **Rise Time** to "50 ps" and select "20-80%" from the **Definition** drop-down list. When finish, click **Apply** to save the settings.
    f. Click the **Finish** button.

Limit Test Setup

Using Hardkey/Softkey

1. Select Trace 1 with either **Trace Next**/**Trace Prev** or **Trace** on the tool bar of the GUI.
2. Press **Analysis** > **Limit Test** > **Limit Test** to turn on the limit test.
3. Click **Limit Line** to turn on.
4. Click **Edit Limit Line**, then limit table is displayed.
5. Edit the table as shown below, then click **Return**.

| | Type | Begin Stimulus | End Stimulus | Begin Response | End Response |
|---|---|---|---|---|---|
| 1 | MAX | 0 s | 1 ns | 105 U | 105 u |
| 2 | MIN | 0 s | 1 ns | 75 U | 75 U |

#### Setup for Channel 2

1. Press **Channel Next** to select the channel 2.
2. Press **Start**, then type 1G to set the start frequency at 1GHz.

3. Press **Stop**, then type 3G to set the start frequency at 3 GHz.
4. Press **Avg** > **IF Bandwidth**, then type 10k to set the IF bandwidth frequency at 10 kHz.
5. Press **Analysis** > **Fixture Simulator** > **Fixture Simulator** to turn ON.
6. Click **Topology** > **Device** > **Bal-Bal**, then click **Return**.
7. Click **BalUn** to turn ON.
8. Click **Measurement** > **Sdd21**.

Full 4-port Calibration

1. Connect 4 Port Ecal with the cables.
2. Press **Cal** > **Ecal** > **4-Port Cal**.

See Full 4-Port Calibration if you use the mechanical calibration kit.

Limit Test Setup

Using Hardkey/Softkey

1. Press **Analysis** > **Limit Test** > **Limit Test** to turn on the limit test.
2. Click **Limit Line** to turn on.
3. Click **Edit Limit Line**, then limit table is displayed.
4. Edit the table as shown below, then click **Return**.

| | Type | Begin Stimulus | End Stimulus | Begin Response | End Response |
|---|---|---|---|---|---|
| 1 | MIN | 100 MHz | 1.25 GHz | -1.5 dB | -5 dB |
| 2 | MIN | 1.25 GHz | 2.5 GHz | -5 dB | -7.5 dB |
| 3 | MIN | 2.5 GHz | 7.5 GHz | -7.5 dB | -25 dB |

**Measurement for Channel 1**

1. Connect the DUT with the cables.
2. Click **Stop Single** to make one single measurement.
3. Select Trace 1 with either **Trace Next**/**Trace Prev** or **Trace** on the tool bar of the GUI.
4. Click **Auto Scale** (tool bar), then select **All Traces**.

**Measurement for Channel 2**

1. Press **Channel Next** to select the channel 2.
2. Press **Trigger** > **Single** to make one single measurement.

# Time Domain Analysis

## Time Domain Analysis

- Overview
- Characteristics of Response in Time Domain
- Deleting Unnecessary Data in Time Domain (Gating)
- Transformation to Time Domain

Other topics about Measurement with Options

## Overview of time domain measurement

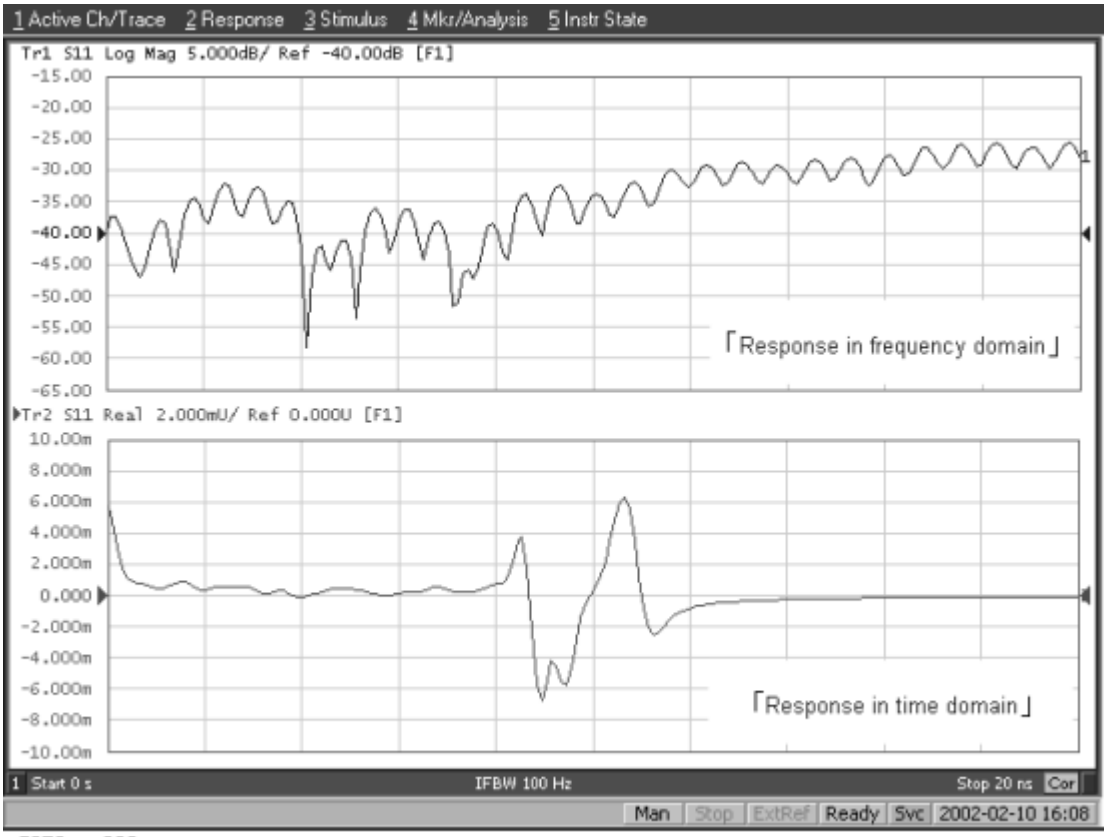
- Overview
- Comparison to Time Domain Reflectometry (TDR) Measurement
- Time Domain Function of E5071C

Other topics about Time Domain Analysis

### Overview

The E5071C Option 010 provides the time domain function, which is used to mathematically transform waveforms in the frequency domain that can be measured with a general network analyzer to waveforms in the time domain.

The following figure shows waveforms in the frequency domain and in the time domain when using the same cable. The waveform in the frequency domain shows ripples due to mismatches, but it is difficult to estimate their locations. On the other hand, from the waveform in the time domain, you can determine the locations and magnitudes of mismatches.



e5070aue202

### Comparison to Time Domain Reflectometry (TDR) Measurement

The time domain function of the E5071C is similar to the time domain reflectometry (TDR) measurement in that it displays the response in the time domain.

In the TDR measurement, a pulse or step signal is inputted to the DUT and the change of the reflected wave over time is measured.

On the other hand, the time domain function of the E5071C changes the frequency of the input signal to the DUT, performs measurement in the frequency domain, and transforms the result to the response in the time domain using the inverse Fourier transform.

### Time Domain Function of E5071C

The time domain function of the E5071C is divided into the following two types of functions. You can use them at the same time.

#### Transformation function

Transforms measurement data in the frequency domain to data in the time domain. For more information, refer to Transformation to time domain.

#### Gating function

Deletes unnecessary data in the time domain from original data in the time domain. For more information, refer to Deleting Unnecessary Data in Time Domain (gating).

## Characteristics of Response in Time Domain



Other topics about Time Domain Analysis

### Overview

This section describes masking and the identification of the mismatch type, which are important for analyzing the response in the time domain.

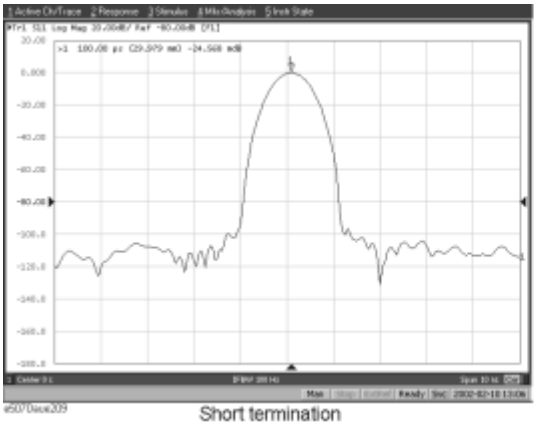
### Masking

Masking is a phenomenon in which a mismatch at a location near the calibration surface affects the response at the next mismatch location. This occurs because energy reflected by a mismatch at the location nearest to the calibration surface does not reach the next mismatch location. For example, when you measure the reflection of a cable that has 2 mismatch locations reflecting 50% of the input voltage, the first mismatch reflects 50% of the measurement signal. The remaining 50% reaches the next mismatch and its 50%, which is 25% of the entire measurement signal, is reflected. Therefore, in the response in the time domain, the second mismatch looks smaller.

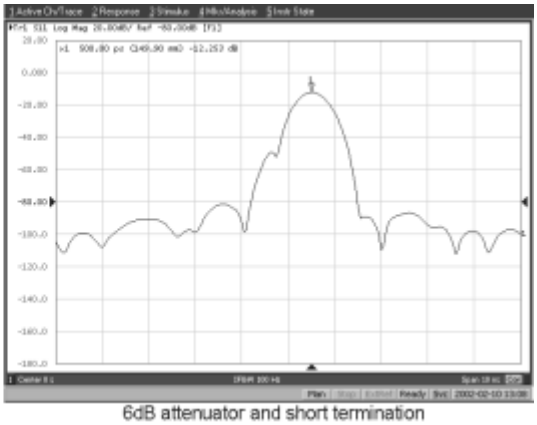
NOTE In this example, the transmission line is assumed to have no loss. However, because there actually is loss, the signal is attenuated as the distance from the calibration surface becomes greater.

The following figure shows an example of masking due to loss. It compares masking when a short termination is directly connected to the calibration surface and masking when a 6-dB attenuator is inserted. In either case, total reflection occurs at the short termination. In the latter case, the signal is attenuated in both ways, and the return loss appears to be - 12 dB.

***Example of masking***



Short termination



6dB attenuator and short termination

### Mismatch Type

The transformation in the low pass mode simulates the response in the TDR measurement. In addition to mismatch locations, the response includes information on the mismatch type.

The following shows each mismatch type and the response waveform corresponding to it. In the low pass mode, you can simulate the response of the step signal and the impulse signal. From the viewpoint of mathematics, the response of the impulse signal is the waveform obtained by differentiating the response of the step signal.

| | Step response | Impulse Response |
|---|---|---|
| Open |  Unity Reflection |  Unity Reflection |
| Short |  Unity Reflection, -180° |  Unity Reflection, -180° |
| Resistor $R > Z_0$ |  |  |
| Resistor $R < Z_0$ |  |  |
| Inductor |  |  |
| Capacitor |  |  |

e5071c415

## Deleting Unnecessary Data in Time Domain (gating)

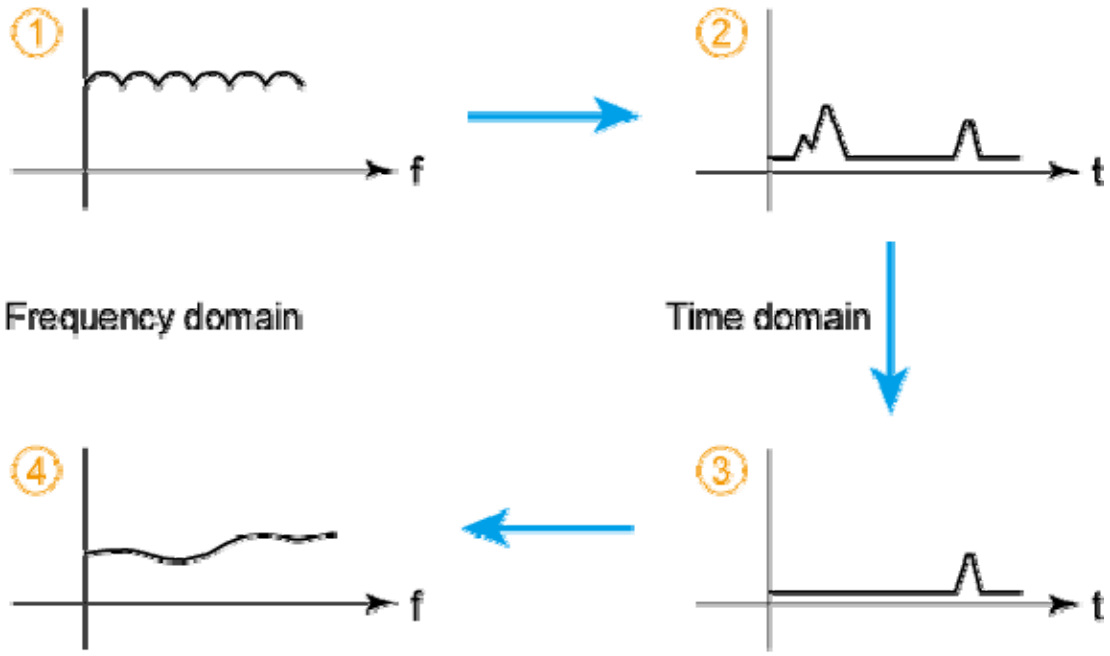
- Measurement Flow
- Setting Gate Type
- Setting Gate Shape
- Setting Gate Range
- Enabling Gating Function

Other topics about Time Domain Analysis

### Measurement Flow

| Item | Description |
|---|---|
| 1. Measurement in frequency domain | Executes measurement in frequency domain |
| 2. Transformation to time domain | Enables transformation function and transforms measurement data to data in time domain |
| 3. Setting the gate | Makes the following settings of the gate to select the necessary domain:<br>• Gate type<br>• Gate shape<br>• Gate range |
| 4. Transformation back to frequency domain | Disables transformation function and displays response in frequency domain corresponding to the data selected with the gate |

The following figure shows the change in the waveform at each step of the flow.




e5071c414

### Setting Gate Type

The E5071C lets you choose from the following two gate types:

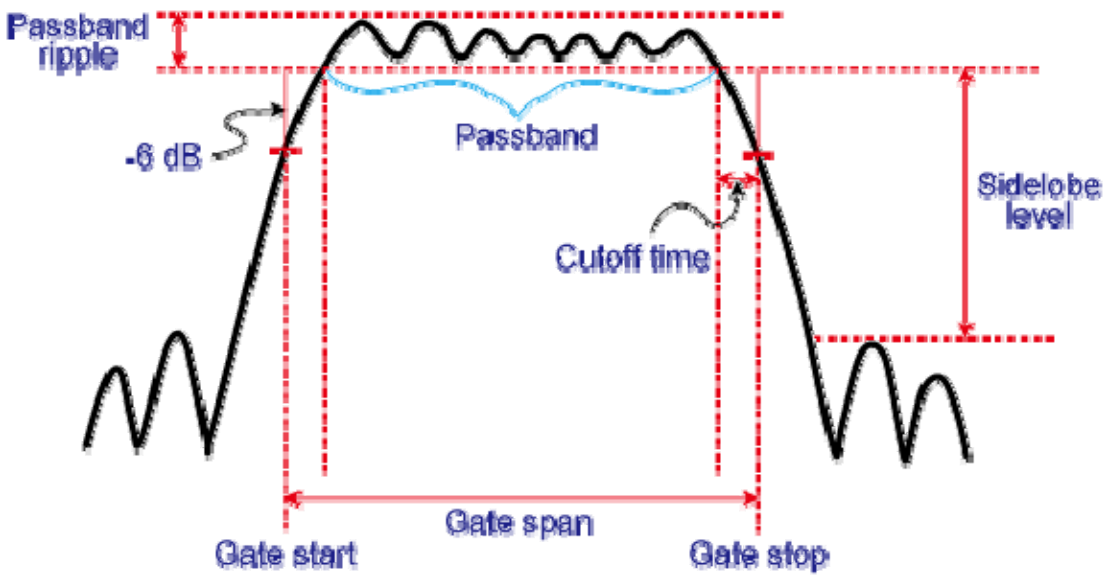
| Gate type | Description |
|---|---|
| Band pass | Deletes response outside the gate range |
| Notch | Deletes response inside the gate range |

Operational procedure

1. Press **Channel Next**/**Channel Prev** keys and **Trace Next**/**Trace prev** keys to activate the trace for which you want to set the gate type.
2. Press **Analysis** key, then click **Gating** to display the Gating menu.
3. Click **Type** to toggle between band pass (**Bandpass**) and notch (**Notch**).

### Setting Gate Shape

The gate is a filter whose shape looks like a band pass filter. There are several parameters that indicate the gate shape. The following figure shows the definition of the gate shape parameters.




e5071c416

The following table compares the characteristics according to the gate shape. When the shape is "minimum," the cutoff time is shorter and the response is deleted abruptly, but the sidelobe level and band pass ripples become larger. When it is "maximum," cutoff is gentler, but the sidelobe level and the band pass ripple become smaller. The minimum gate span in the following table is the minimum gate range you can set. This value is defined as the minimum gate span necessary for the existence of the pass band and is equal to 2 times the cutoff time.

| Gate shape | Passband ripple | Sidelobe level | Cutoff time | Minimum gate span |
|---|---|---|---|---|
| Minimum | ± 0.13 dB | - 48 dB | 1.4/frequency span | 2.8/frequency span |
| Normal | ± 0.01 dB | - 68 dB | 2.8/frequency span | 5.6/frequency span |
| Wide | ± 0.01 dB | - 57 dB | 4.4/frequency span | 8.8/frequency span |
| Maximum | ± 0.01 dB | - 70 dB | 12.7/frequency span | 25.4/frequency span |

**Operational procedure**

1. Press **Channel Next**/**Channel Prev** keys and **Trace Next**/**Trace prev** keys to activate a trace for which you want to set the gate shape.
2. Press **Analysis** key, then click **Gating** to display the Gating menu.

3. Press **Shape** and then select the gate shape.

### Setting Gate Range

Specify the gate range in time. The ends of the range are defined as the -6 dB attenuation points shown in the figure above. You can set the gate range by specifying the start and stop times or the center and span. The E5071C has the following limitations on the gate range you can set.

$$Lower\ limit \quad -T_{span}$$

$$Upper\ limit \quad T_{span}$$

$T_{span}$ is the measurement range expressed in time obtained in Measurement range.

#### Operational procedure

1. Press **Channel Next**/**Channel Prev** keys and **Trace Next**/**Trace prev** keys to activate the trace for which you want to set the gate range.
2. Press **Analysis** key, then click **Gating** to display the "Gating" menu.
3. Press each of the following softkeys to specify the gate range. Distance corresponding to the setting time is displayed at the side of the set value area in the data entry bar. The displayed distance is a value that takes the velocity factor into consideration.

| Softkey | Function |
|---|---|
| **Start** | Sets the start time |
| **Stop** | Sets the stop time |
| **Center** | Sets the center of the gate in time |
| **Span** | Sets the gate span in time |

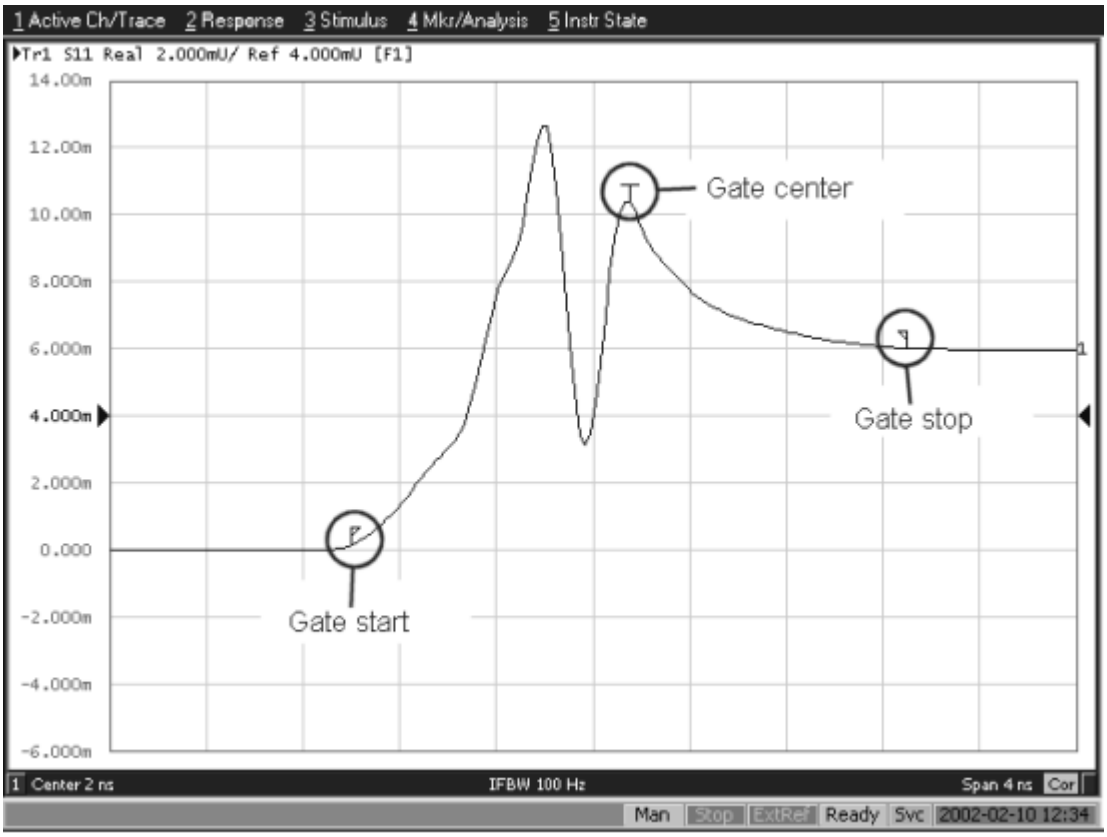
NOTE No Hardkey is provided for this setting. The hardkeys are dedicated to setting the sweep range.

NOTE You can set the center and span by dragging and dropping flags indicating the gate range.

### Enabling Gating Function

When you enable the gating function, data within the specified range is deleted. When the transformation function is enabled, the flags indicating the gate range is displayed as shown in the following figure.

NOTE In the figure, the gate type is set to band pass. When it is set to notch, the directions of the flags indicating the ends of the gate range are reversed.




e5070aue205

**Operational procedure**

1. Press **Channel Next**/**Channel Prev** keys and **Trace Next**/**Trace prev** keys to activate the trace for which you want to use the gate function.
2. Press **Analysis** key, then click **Gating** to display the "Gating" menu.
3. Use **Gating** to enable (**ON**) the gate function.

## Transformation to time domain

- Overview
- Measurement Flow

Other topics about Time Domain Analysis

### Overview

The transformation function lets you transform a response in the frequency domain to the corresponding response in the time domain.

### Measurement Flow

| Item | Description |
|---|---|
| Selecting a type | Selects the transformation type from the following.<br>Band pass mode<br>You can set the sweep range arbitrarily. This is suitable for devices through which signals within a certain frequency range pass such as filters.<br>Low pass mode<br>Simulates the TDR measurement. This mode is suitable for devices through which dc current or signals of many frequencies pass, such as cables.<br>The low pass mode provides two types of modes: low pass step and low pass impulse. |
| Setting the window | Sets the window to reduce the ringing phenomenon, in which a waveform's waves are caused due to the fact that the frequency domain is finite. |
| Calculating necessary measurement conditions | Calculates the following values to obtain the necessary resolution and measurement range in the time domain.<br>Sweep range<br>Number of points<br>Window width |
| Setting the frequency range and the number of points | Sets the sweep range and the number of points to the values calculated above. |
| Setting display range | Sets the range displayed on the graph. |
| Enabling transformation function | Enables the transformation function. |

### Selecting a type

There are two types of transformation to the time domain: band pass and low pass. The appropriate transformation type depends on the DUT.

**Comparison between band pass mode and low pass mode**

| Item | Band pass | Low pass |
|---|---|---|
| Appropriate DUT | DUTs that do not operate with dc current such as band pass filters. | DUTs that operate with dc current such as cables. |
| Input signal | You can simulate the response to the impulse signal. | You can simulate the TDR measurement. You can simulate the response to both the impulse signal and step signal. |
| Selection of the sweep range | You can select the sweep range arbitrarily. | Because dc data are estimated from the first few points, the frequencies of measurement points must be multiples of the start frequency. |
| Transmission/ reflection measurement | You can perform both transmission and reflection measurements. | You can perform both transmission and reflection measurements. |
| Identification of mismatches | You can identify the locations of mismatches. | You can identify the locations of mismatches and the type of impedance (capacitive or inductive). |
| Resolution | | The resolution in the time domain increases by two times over that in the band pass mode. |

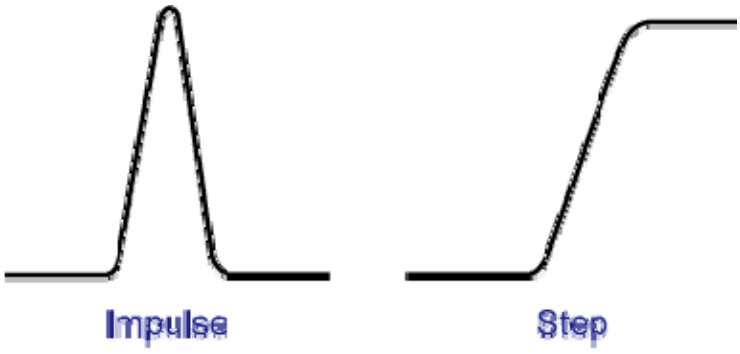
| | | |
|---|---|---|
| Available data format | • Liner magnitude format<br>• In the reflection measurement, it indicates the mean of the reflection coefficient within the frequency sweep range.<br>• In the transmission measurement, it indicates the mean of the transmission coefficient within the frequency sweep range.<br>• Log magnitude format<br>• In the reflection measurement, it indicates the mean of the return loss within the frequency sweep range.<br>• In the transmission measurement, it indicates the mean of the transmission gain within the frequency sweep range.<br>• SWR format<br>• In the reflection measurement, it indicates the mean of SWR (standing wave ratio) within the frequency sweep range. | Real format<br>In the low pass mode, the real format is useful because the time axis data do not have phase information. |

**Impulse signal and step signal**

The E5071C lets you simulate the response from the DUT to two types of signals: impulse signal and step signal. The impulse signal is a pulse-shape signal in which the voltage rises from 0 to a certain value and returns to 0 again. The pulse width depends on the frequency sweep range. The step signal is a signal in which the voltage rises from 0 to a certain value. The rise time depends on the maximum frequency within the frequency sweep range.

NOTE For more information on how the frequency span setting affects the pulse width and the rise time, refer to Calculating necessary measurement conditions.

***Step signal and impulse signal***




e5071c417

**Operation**

1. Press **Channel Next**/**Channel Prev** keys and **Trace Next**/**Trace prev** keys to activate a trace for which you want to set the transformation type.
2. Press **Analysis** key, then click **Transform** to display the Transform menu.
3. **Type** and then press one of the **softkeys** to specify the type.

| Softkey | Function |
|---|---|
| **Bandpass** | Sets the transformation type to "band pass." |
| **Lowpass Step** | Sets the transformation type to "lowpass step." |
| **Lowpass Imp.** | Sets the transformation type to "lowpass impulse." |

4.
5. Press **Format** key to display the "Format" menu and then select the data format.

### Setting the window

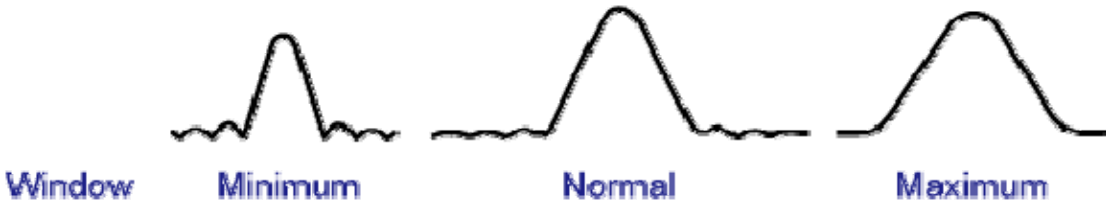
Because the E5071C transforms data within a finite frequency domain to data in the time domain, an unnatural change of data at the end points within the frequency domain occurs. For this reason, the following phenomena occur.

- Width of impulse signal and rise time of step signal
  A time width occurs in the impulse signal and a rise time occurs in the step signal.
- Sidelobe
  Sidelobes (small peaks around the maximum peak) occur in the impulse signal and the step signal. Ringing occurs on the trace due to sidelobes, which reduces the dynamic range.

By using the window function, you can lower the level of sidelobes. However, the width of the impulse and the rise time of the step become larger as a penalty. You can select from three types of windows: maximum, normal, and minimum. The following table shows the approximate relation between the window and the sidelobe/impulse width.

| Window | Sidelobe level of impulse signal | Width of impulse (50% in low pass mode) | Sidelobe level of step signal | Rise time of step signal (10 - 90%) |
|---|---|---|---|---|
| Minimum | - 13 dB | 0.60/frequency span | - 21 dB | 0.45/frequency span |
| Normal | - 44 dB | 0.98/frequency span | - 60 dB | 0.99/frequency span |
| Maximum | - 75 dB | 1.39/frequency span | - 70 dB | 1.466/frequency span |

The window function is available only when the response in the time domain is displayed. It does not have any effect when the response in frequency domain is displayed. The following figure shows the effect of the window when measuring the reflection of a short circuit in the time domain.




e5071c419

1. Press **Channel Next**/**Channel Prev** keys and **Trace Next**/**Trace prev** keys to activate a trace for which you want to set the window.
2. Press **Analysis** key, then click **Transform** to display the Transform menu.
3. Press **Window** and then select a window type.

| Softkey | Function |
|---|---|
| **Maximum** | Sets the window type to maximum. b of the Kaiser Bessel function is set to 13. |
| **Normal** | Sets the window type to normal. b of the Kaiser Bessel function is set to 6. |
| **Minimum** | Sets the window type to minimum. b of the Kaiser Bessel function is set to 0. |
| **Rise Time** | Sets the window by specifying the impulse width or the step rise time. The lower limit can be set to the value when the window is the minimum, the upper limit when the window is the maximum. |
| **Kaiser Beta** | Sets the window by specifying the b value of the Kaiser Bessel function. The Kaiser Bessel function determines the shape of the window. The allowable setting range is 0 to 13. |

NOTE By specifying **Kaiser Beta**, **Impulse Width**, or **Rise Time**, you can specify a window that is not classified into the three window types. Conversely, when you specify a window type, these values are set automatically.

### Calculating necessary measurement conditions

To use the transformation function efficiently, you need to make the following three settings appropriately.

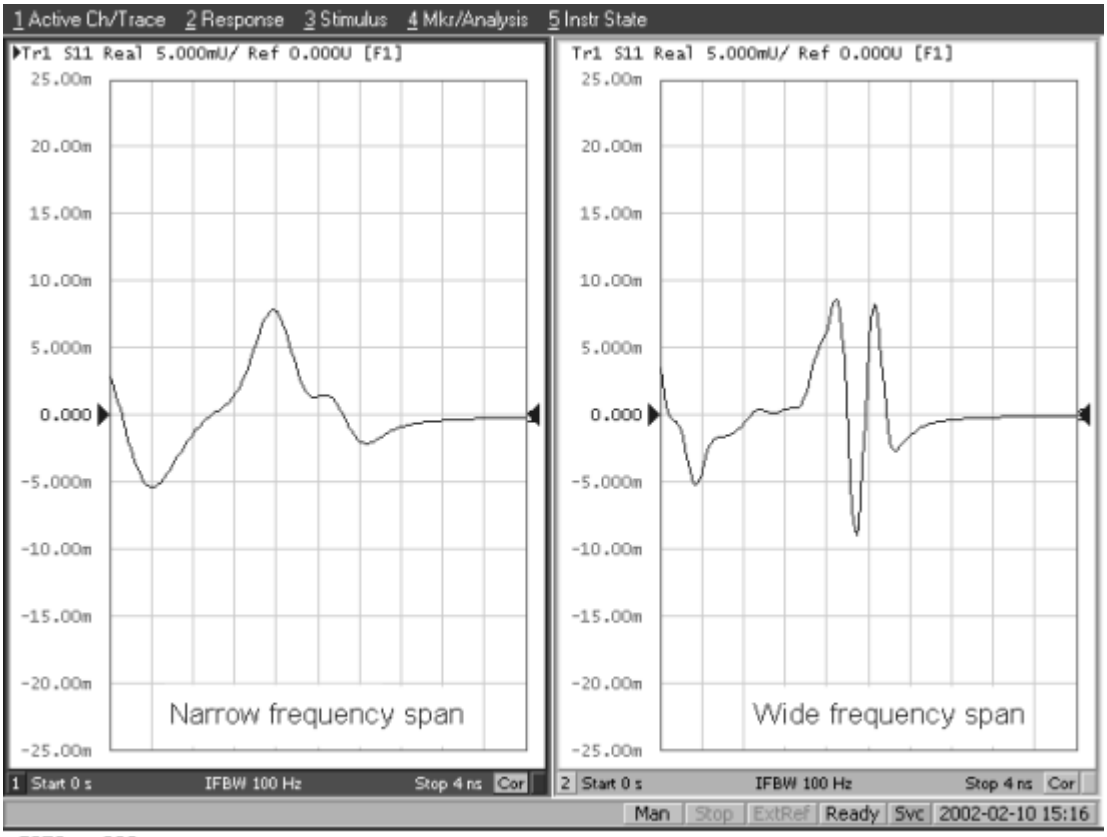
- Resolution of the response
- Measurement range
- Sidelobe

This section describes the details of these settings.

#### Effect of frequency sweep range on response resolution

The following figure shows an example when measuring the same cable while changing the sweep span. When measured in a narrower sweep range, the overlap between two peaks is larger than when measured in a wider sweep range. By performing measurement in a wider sweep range, adjacent peaks can be clearly separated, which means that the response resolution is smaller.

***Effect of frequency sweep range on resolution***

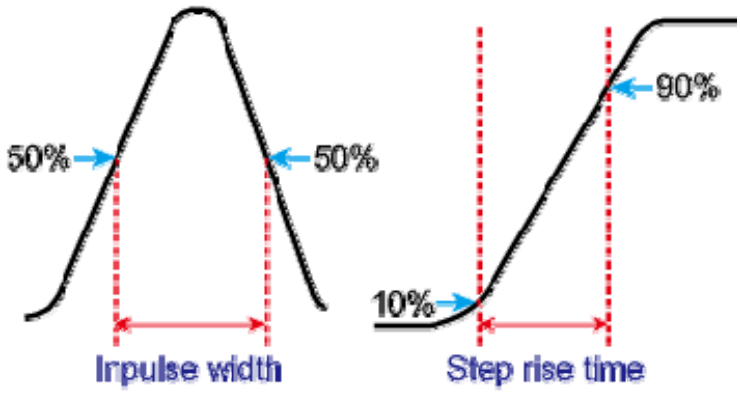



e5070aue208

The sweep range affects the width of the impulse signal and the rise time of the step signal. The width of the impulse signal and the rise time of the step signal are inversely proportional to the sweep range. Therefore, the wider the sweep range is, the shorter these times are.

The resolution is equal to the width defined at the point of 50% of the impulse signal or the rise time defined at the points of 10% and 90% of the step signal.

***Definitions of impulse width and step rise time***




e5071c418

**Effect of the window function on the response resolution**

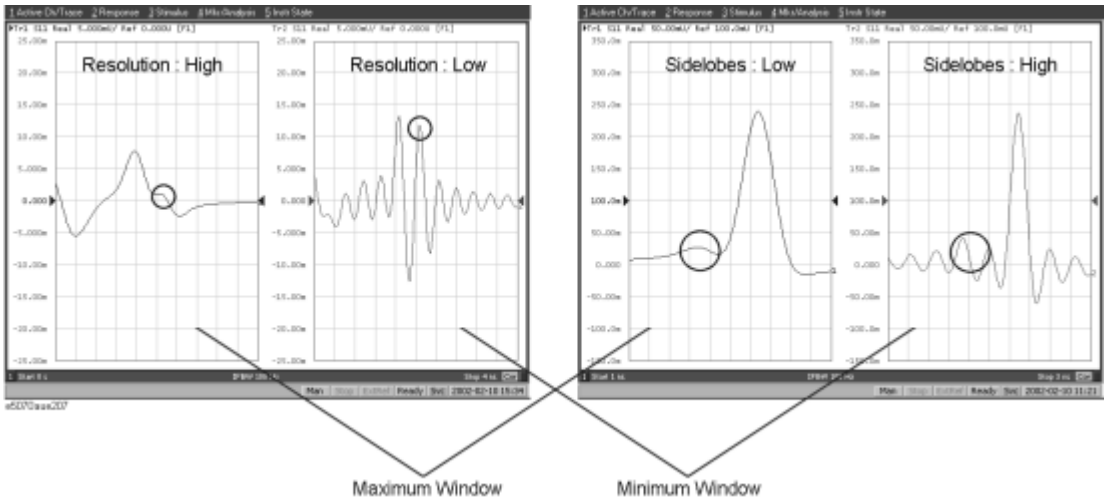
Lowering the sidelobe level with the window function elongates the width of the impulse signal and the rise time of the step signal. As described in Effect of frequency sweep range on response resolution, because the response resolution is equal to the width of the impulse signal and the rise time of the step signal, lowering the sidelobe level enlarges the response resolution. The following table shows the approximate relationship between the response resolution and the window setting.

***Correspondence of Window setting and response resolution***

| Window | Low pass step | Low pass impulse | Band pass |
|---|---|---|---|
| Minimum | 0.45/stop frequency | 0.60/stop frequency | 1.20/frequency span |
| Normal | 0.99/stop frequency | 0.98/stop frequency | 1.95/frequency span |
| Maximum | 1.46/stop frequency | 1.39/stop frequency | 2.77/frequency span |

The following figure shows how the response changes as the window shape changes. If the magnitudes of adjacent peaks are comparable, you need to make the resolution higher; if they differ significantly, you need to set the window so that smaller peaks with lower sidelobes appear.

***Effect of window on response resolution***




**Effect of the transformation type on the response resolution**

Although both transformation types, band pass and low pass impulse, simulate the response of the impulse signal, the impulse width in the low pass impulse mode is half the width of that in the band pass mode as shown in the table above. Therefore, the resolution is better in the low pass mode. If the DUT can be measured in the low pass mode, response data with better resolution is obtained in the low pass mode.

**Measurement range**

In the time domain function, the measurement range means the range within which the response can be measured without repetition. The repetition of the response occurs because measurement in the frequency domain is performed discretely instead of continuously. The measurement range is inversely proportional to the frequency difference between adjacent measurement points. The frequency difference between measurement points *Delta F* is expressed as follows by using the span of the sweep frequency $F_{span}$ and the number of points $N_{meas}$.

$$\Delta F = \frac{F_{span}}{N_{meas} - 1}$$

Therefore, the measurement range is proportional to the number of points-1 and inversely proportional to the span of the sweep range. To enlarge the measurement range, use one of the following methods:

- Increase the number of points.
- Narrow the span of the sweep range.
    - When you change the above settings after performing calibration, you need to perform calibration again.

The sweep range is expressed as time or distance. The time of the measurement range $T_{span}$is as follows:

$$T_{span} = \frac{1}{\Delta F}$$

The distance of the measurement range Lspan is expressed as follows using the velocity factor V and the speed of light in a vacuum c (3*E$^8$ m/s).

$$L_{span} = \frac{Vc}{\Delta F}$$

NOTE The maximum length of the DUT that can be measured in the transmission measurement is $L_{span}$. On the other hand, in the reflection measurement, bemcause the signal goes and returns, it is 1/2 of $L_{span}$.

The velocity factor varies depending on the material through which the signal propagates: 0.66 for polyethylene and 0.7 for PTFE.

**Changes in the settings and the changes in the response**

The following table shows the effect of changes in the measurement conditions on the response resolution and the measurement range.

| Change in setting | Response resolution | Measurement range | Sidelobe |
|---|---|---|---|
| Widening the sweep range | Becomes smaller | Becomes narrower | Does not change |
| Setting the window type to maximum | Becomes larger | Does not change | Becomes lower |
| Increasing the number of points | Does not change | Becomes wider | Does not change |

## Setting the frequency range and the number of points

**Operation**

1. Press **Channel Next**/**Channel Prev** keys to activate the channel you want to set. The frequency range and the number of points are common to all of the traces in the channel. If you want to use different settings, make them on another channel.
2. Press **Sweep Setup** key, then press **Sweep Type** > **Lin Freq** to set the sweep type to linear sweep. When the sweep type is set to other than the linear sweep, the time domain function is not available.
3. Set the sweep range using the **Start**/**Stop** or **Center**/**Span** keys.
4. Press **Sweep Setup** key, then press **Points** and then enter the number of points. When performing measurement in the low pass mode, press **Analysis** > **Transform** > **Set Freq Low Pass** to adjust the frequency range so that it is appropriate for the low pass mode. The

frequency changes depending on the stop frequency as shown below.

| Condition of stop frequency | Frequency setting |
|---|---|
| >= $F_{low}$ * the number of points | Start frequency = stop frequency/number of points |
| < $F_{low}$ * the number of points | Start frequency: $F_{low}$<br>Stop frequency = $F_{low}$ * Number of points |

5. $F_{low}$ =9 kHz or 100 KHz (Depending upon the installed option)

When the frequency settings satisfy the conditions shown above, the **Set Freq Low Pass** key is displayed in gray.

### Setting display range

The E5071C has the following limitations on the display range you can set.

- Lower limit: $-T_{span}$
- Upper limit: $T_{span}$

$T_{span}$ is the measurement range expressed in time obtained in Measurement range.

The number of response points displayed on the graph is the same as the number of points regardless of the response resolution.

#### Operation

1. Press **Channel Next**/**Channel Prev** keys and **Trace Next**/**Trace Prev** to activate the trace for which you want to set the display range.
2. Press **Analysis** key, then click **Transform** to display the "Transform" menu.
3. Press each of the following softkeys and then specify the display range. Distance corresponding to the setting time is displayed at the side of the set value in the data entry bar. The displayed distance is a value taking the velocity factor into consideration.

| Softkey | Function |
|---|---|
| **Start** | Sets the start value of the display range in time |
| **Stop** | Sets the stop value of the display range in time |
| **Center** | Sets the center value of the display range in time |
| **Span** | Sets the span of the display range in time |

NOTE You cannot use hardkeys to set the display. The hardkeys are dedicated to specifying the sweep range.

### Enabling transformation function

Operation

1. Press **Channel Next**/**Channel Prev** keys and **Trace Next**/**Trace Prev** to activate the trace for which you want to use the transformation type.
2. Press **Analysis** key, then click **Transform** to display the "Transform" menu.
3. Press **Transform** to enable (**ON**) the transformation function.
4. The following requirements must be met to enable the transformation function.
   - The sweep mode is the linear sweep.
   - There are three or more measurement points.