User's Guide

Agilent N1900 Series Physical Layer Test Systems

N1930A Physical Layer Test System Software which supports VNA-Based and TDR-Based Physical Layer Test System Hardware

Agilent Technologies

Manufacturing Part Number: N1930-90004

Printed in USA

May 2004

Supersedes N1930-90003

Notice

LIMITED WARRANTY

Assistance

Product maintenance agreements and other customer assistance agreements are available for Agilent Technologies products. For any assistance, refer to "Contacting Agilent" in the "Troubleshooting and Maintenance" chapter of the *PLTS Installation and Reference Guide.*

Software Licensing

The Physical Layer Test System has two licensing options. The options are:

Node-Locked allows use of PLTS software only on a single personal computer. The license resides on and is tied to the local PC via a hardware address.

Network-Server Floating allows the sharing of a licences (or multiple licences) across a network. The PLTS software may be installed on an unlimited number of personal computers. The license(s) reside on a networked drive/folder and are checked out on a first-come, first-served basis.

Refer to the “Installing the Physical Layer Test System Software” chapter of the *PLTS Installation and Reference Guide* for the licensing procedure.

Software Compatibility

This user's guide is compatible with Physical Layer Test System software revisions 2.500 and above.

Document Conventions

This document uses a few conventions to make reading easier.

- Menu and dialog box items are shown in bold face type. When described in text, menus and sub-menus are separated by right arrows, as in **File > Open > Data...**
- Window and dialog box names are shown in *italic* font.
- Keyboard entries are shown in `mono-spaced` typeface.
- Network analyzer keys are displayed in **condensed, bold** font.

Embedded Operating System Risk

PNA Network Analyzers

The Agilent PNA-family of network analyzers make use of Embedded Operating Systems (EOS) technology. The EOS is Windows[1] 2000, a standard personal computer-based operating system. The PNA is essentially a network analyzer with a personal computer behind the front panel.

EOS devices, including the PNA may be connected to computer networks. When connected, these devices are open to numerous security exploits, worms, and viruses. In addition, they can become a threat to other EOS devices, servers, end-user PCs, and to the network itself.

Precautions are taken to ensure that the PNA has no security issues, such as worms or viruses, prior to shipping. To maintain the health of the PNA, it is recommended that you install anti-virus software and the latest service packs on the PNA before connecting the PNA to a computer network.

Infiniium DCA Oscilloscopes

On your 86100C, you can access Microsoft Windows XP Professional just as you would on your personal computer. Use Windows XP Professional to manage files and folders, add, remove, and setup printing, configure networking, and install applications. The ability to access the operating system is not available on 86100A/B instruments.

Although the 86100C is an instrument and not a personal computer, the operating system is accessible. This makes it possible for you to install applications such as virus protection software. If the performance of the Infiniium DCA application decreases while running other applications, you may need to close those applications that are demanding processor resources. It is also suggested that you schedule automatic virus scans for times when you are not making measurements. Refer to the 86100C Help for complete information.

1. Windows® and MS Windows® are U.S. registered trademarks of Microsoft Corporation.

Contents

Contents

I Calibration and Measurements

Part I provides detailed information about making measurements using both the VNA-based and TDR-based physical layer test systems. Part I also provides information about error correcting each system for optimum accuracy.

Chapter 1, “Setting Up and Making Measurements using the VNA-Based PLTS”

Provides information about making a measurement using the VNA-based PLTS.

Chapter 2, “Performing Error Correction on the VNA-Based PLTS”

Provides information about performing error correction on the VNA-based PLTS.

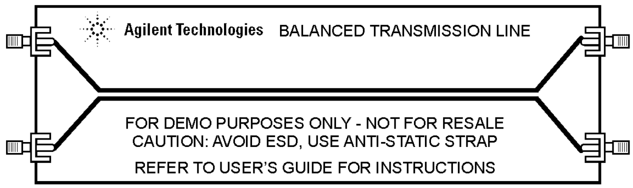
Chapter 3, “Setting Up, Calibrating, and Making Measurements using the TDR-Based PLTS”

Provides information about performing error correction and making a measurement using the TDR-based PLTS.

1 Setting Up and Making Measurements using the VNA-Based PLTS

This chapter guides you through setting up the initial measurement definitions, performing the system calibration, making a measurement, and analyzing the measurement results. It uses a simple balanced transmission line that was supplied with the system as a sample device-under-test (DUT). See Figure 1-1.

Figure 1-1 Balanced Transmission Line Sample DUT



The **Startup Wizard** steps you through the initial setup, the calibration, and the measurement.

1. **Initial setup** includes:
   - System Hardware Verification
   - Calibration Level Selection
   - Calibration and Measurement Parameter Selection
2. **Calibration** includes:
   - Calibration Type Selection and Calibration Kit Selection
   - Calibration (SOLT Mechanical, SOLT Electronic, TRL, or LRM)
   - Adapter Characterization
3. **Measurement** includes:
   - Device Under Test Connection
   - Analysis Type Selection
   - Stimulus Verification
   - Running a Measurement

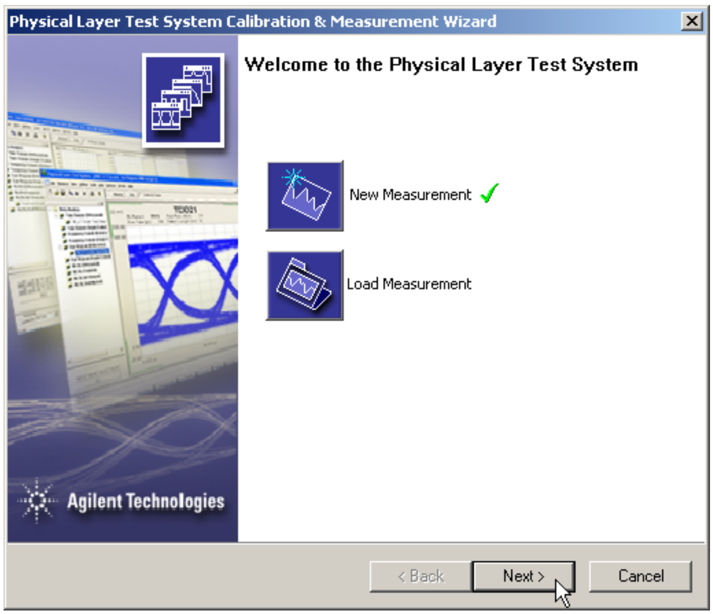
After the measurement is made, the main Physical Layer Test System (PLTS) window gives you the flexibility to perform analysis on the measured data in a variety of ways.

- **Frequency Domain.** Allows for analysis in Balanced and Single-Ended mode.
- **Time Domain.** Allows for analysis in Differential and Single-Ended mode.
- **Eye Diagram.** Allows for analysis in Differential and Single-Ended mode.
- **Transmission Line (RLCG).** Allows for analysis in Differential, Common, W-Element, and Self/Mutual modes.

Navigating the Startup Wizard

When PLTS is first started, the Startup Wizard is displayed. The Startup Wizard Welcome Screen is displayed in Figure 1-2. It is also displayed anytime a new measurement is initiated, such as after selecting **New** from the **File** menu, or clicking on the new file icon on the toolbar.

Figure 1-2 Startup Wizard Welcome Screen



The Startup Wizard Welcome Screen gives you two choices, **New Measurement** or **Load Measurement**. The selected choice has a green check mark to the right of the label. **New Measurement** leads you through the process of calibrating and making a measurement. **Load Measurement** loads measurement data from an existing file that you must select.

When you select **New Measurement**, the Startup Wizard will guide you through:

- Initial Setup
- Calibration
- Measurement

For this exercise, select **New Measurement** and then select the **Next >** button.

How to Perform the Initial Setup

The Initial Setup process consists of:

1. Verifying that the software recognizes your PLTS system hardware.

   This step uses the left portion of the *Hardware Auto-Detection Summary* dialog box to verify that the software recognizes the correct system hardware. This portion of the dialog box displays the model number, GPIB address, and serial number of the hardware. You may re-scan to look for hardware changes automatically. You may select another recognized PLTS measurement system.

2. Selecting the appropriate level of calibration for the upcoming measurement.

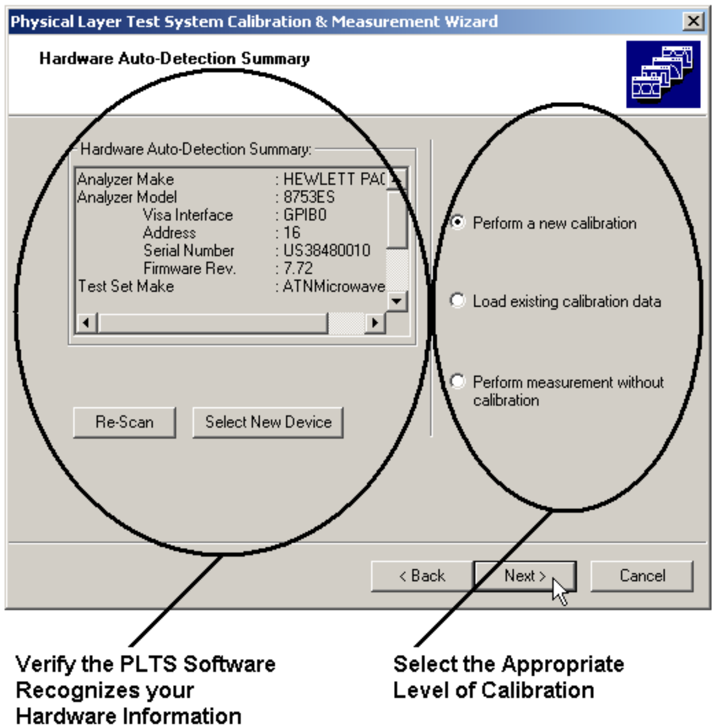
   This step uses the right portion of the *Hardware Auto-Detection Summary* dialog box to select the appropriate level of calibration to be performed. You may perform a new calibration, reuse existing calibration data, or perform measurements without calibration.

3. Setting up the VNA Calibration and Measurement Settings.

   This step uses the *Setup TDR* dialog box to display the default TDR calibration and measurement settings, based on the PLTS hardware recognized by the software. You may modify these default TDR calibration and measurement settings as required.

When you selected **New Measurement** and then clicked the **Next >** button described on page 5, the *Hardware Auto-Detection Summary* dialog box is displayed. See Figure 1-3.

Figure 1-3 Hardware Auto-Detection Summary Dialog Box



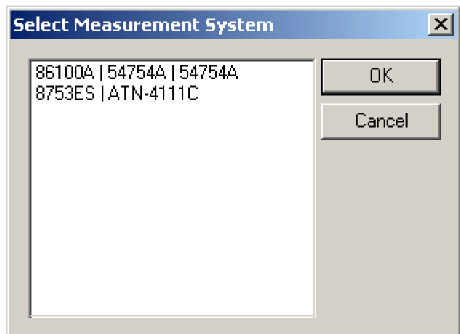
Verifying the Software Recognizes the PLTS Hardware

On the left portion of the *Hardware Auto-Detection Summary* dialog box, the VNA's model number, the GPIB address, the serial number, and the firmware revision is displayed. In addition, the test set's model number, the GPIB address, and the serial number are displayed. This is the test equipment that the software finds connected to the GPIB bus.

If this information is incorrect, check your hardware connections and power status, then select the **Re-Scan** button. The software re-checks for the presence of hardware on the GPIB.

NOTE If you have multiple PLTS systems on the GPIB and you want to select the other equipment, click the **Select New Device** button. The *Select Measurement System* dialog box is displayed. See Figure 1-4. To choose another PLTS system, you may select from the available PLTS systems and then click the **OK** button. The software makes the change and returns to the *Startup Wizard Welcome Screen* of Figure 1-2.

Figure 1-4 Select Measurement System Dialog Box



The **Cancel** button exits without saving a change and returns to the *Startup Wizard Welcome Screen*.

Selecting the Appropriate Level of Calibration

On the right portion of the *Hardware Auto-Detection Summary* dialog box (Figure 1-3), select your calibration strategy:

- **Perform a new calibration** continues with the Startup Wizard performing a new calibration before making a measurement.
- **Load existing calibration data** allows you to select a previously saved calibration (*.cal) file to be used for the new measurement. See Chapter 2, "Performing Error Correction on the VNA-Based PLTS," for guidance on calibration intervals, etc.
- **None. Collect measurement data uncalibrated** allows you skip the calibration, select measurement parameters (see Figure 1-6 on page 10), and then proceed directly to the measurement screen. This option is ***not recommended*** for qualitative data collection.

Select the **Perform a new calibration** choice for this example exercise and then click the **Next >** button.

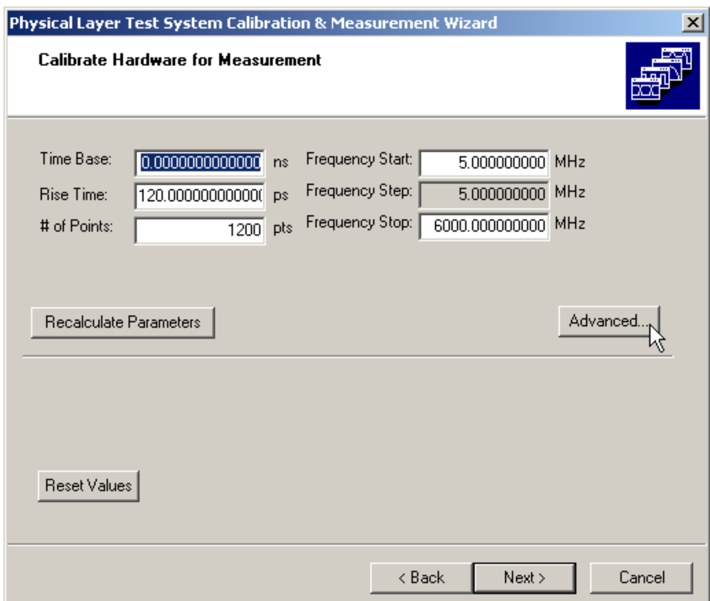
Setting Up the Calibration and Measurement Parameters

After you select **Perform a new calibration** and **Next >**, the first of three *Calibrate Hardware for Measurement* dialog boxes is displayed as shown in Figure 1-6. Use this *Calibrate Hardware for Measurement* dialog box to change the parameters for your calibration and measurement to meet your measurement requirements. This dialog box displays the default measurement parameters which are based on the specifications of your PLTS hardware.

1. The basic measurement parameters are displayed in Figure 1-5. View all of the parameters by clicking the **Advanced** button.

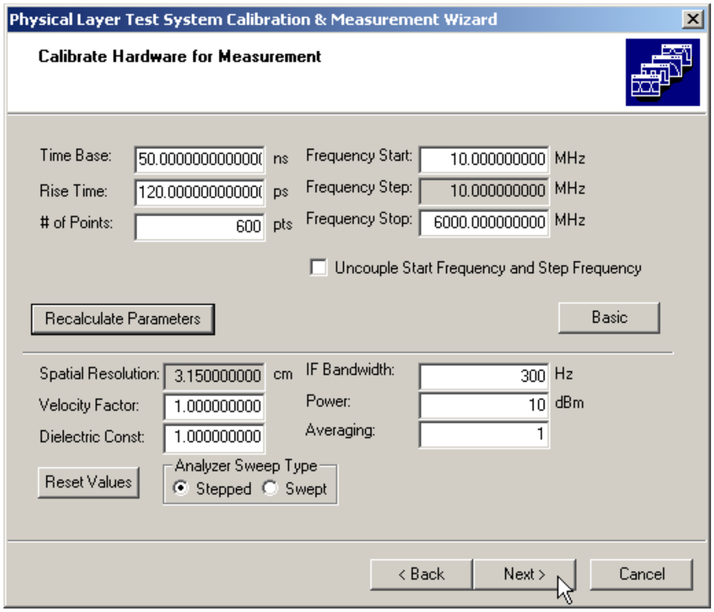
   The **Advanced** button shown in Figure 1-5 toggles between **Advanced** and **Basic**. When in **Advanced**, the entire dialog box is displayed as shown in Figure 1-6.

Figure 1-5 Measurement Parameter Dialog Box (Basic Parameters)



Since the default parameters are based on the specifications of your PLTS hardware, leave the parameters unchanged and click the **Next >** button for this exercise.

Figure 1-6 Measurement Parameter Dialog Box



Modifying these values also modifies other associated values interactively. For example, when you change the **# of Points** entry, the **Time Base** and the **Frequency Start** entries change appropriately. To modify a parameter, enter the value in associated value box and then click the **Recalculate Parameters** button to update the value each parameter. Refer to Table 1-1 for a description of each parameter.

You can return to the default values at any time by selecting the **Reset Values** button.

Table 1-1 Measurement and Calibration Parameter Entry Descriptions

Parameter	**Description**
***Calibrate Hardware for Measurement* Dialog Box Basic Parameters**	
Time Base	Sets the maximum time base (in ns). Time base calculated as $\frac{1}{2 \times F_{min}}$ where F_{min} is the start frequency in Hz.
Rise Time	Sets the transition time (in ps) of the stimulus signal rising from 10 to 90% of the maximum signal amplitude.
# of Points	Sets the number of measured points per sweep.
Frequency Start	Sets the start frequency of the sweep.

Table 1-1 Measurement and Calibration Parameter Entry Descriptions

Parameter	Description
Frequency Step	Sets the step size between points. This value is normally locked to (is the same as) the Frequency Start value.
Frequency Stop	Sets the stop frequency of the sweep.
***Calibrate Hardware for Measurement* Dialog Box Advanced Parameters**	
Uncouple Start Frequency and Step Frequency	Allows you to uncouple the start frequency from the step frequency. See "Uncouple Start Frequency and Step Frequency" below for additional information.
Spatial Resolution	Describes how close in time two responses can be distinguished; depends on the width of the impulse response, which is inversely related to the maximum frequency of the measurement.
Velocity Factor	Is the numerical value related to the speed of energy through a DUT based on the DUTs dielectric material. Some velocity factor (V_f) examples are: Air V_f = 1.000, Surface traces V_f = 0.53146, Buried Traces in $\varepsilon_r \sim 4.3$: V_f = 0.48795
Dielectric Constant	(ε_r) is used to calculate the velocity factor $V_f = 1/(\sqrt{\varepsilon_r})$
IF Bandwidth	Allows you to change the IF bandwidth. Narrow IF bandwidths allow you to view low-level signals, but require more data samples per point and thus slows the measurement time.
Power	Sets the signal level at the source appropriate for measurement of the device. You can measure the signal level available at the test port directly using a power meter.
Averaging	Averages each point of consecutively swept traces until the total number of sweeps is equal to the averaging factor, resulting in a fully averaged trace with better noise reduction. A high averaging factor gives the best signal-to-noise ratio, but slows the measurement time.
Analyzer Sweep Type	**Stepped** takes data while sweeping through defined frequency points. **Swept** takes data while sweeping linearly and continuously across the frequency range.

Uncouple Start Frequency and Step Frequency The Uncouple Start Frequency and Step Frequency selection is available in the *Calibrate Hardware for Measurement* dialog box when the Advanced button has been selected. Normally, the Frequency Step setting is locked (or coupled) to the Frequency Start setting (see Table 1-1).

However, there are situations that you may want to uncouple the **Frequency Step** setting

from the **Frequency Start** setting. For example, uncouple the **Frequency Step** setting from the **Frequency Start** setting if you want to measure more points for better measurement resolution or you may want to measure specific frequency points that would not ordinarily be measured.

When measured data from this uncoupled start frequency/step frequency measurement is converted to the Time Domain and Eye Diagram analysis types, conversion rules must be applied to ensure the data is displayed correctly.

Conversion Rule 1: If **Frequency Start** > **Frequency Step**, then set new **Frequency Step** = **Frequency Start** and:

Condition 1a.

If **Frequency Start** and **Frequency Step** are harmonically related and number of points is > 10, resample the points and mark the plot with the ***Resample*** icon.

Example 1a:

Measured Parameters			**Parameters for Time Domain**		
Start	**Step**	**Stop**	**Start**	**Step**	**Stop**
6MHz	2MHz	9000MHz	6MHz	6MHz	9000MHz
100MHz	50MHz	9000MHz	100MHz	100MHz	9000MHz

Condition 1b.

If **Frequency Start** and **Frequency Step** are harmonically related and the harmonically-related number of points is < 10, use the all of the measured data for time domain and mark the plot with the ***Bad Data*** icon.

Example 1b:

Measured Parameters			**Parameters for Time Domain**		
Start	**Step**	**Stop**	**Start**	**Step**	**Stop**
1000MHz	10MHz	9000MHz	1000MHz	10MHz	9000MHz

Condition 1c.

If **Frequency Start** and **Frequency Step** are not harmonically related, interpolate between measured points for harmonically related points to generate time domain data and mark the plot with the ***Interpolate*** icon.

Example 1c:

Measured Parameters			**Parameters for Time Domain**		
Start	**Step**	**Stop**	**Start**	**Step**	**Stop**
6MHz	5MHz	9000MHz	6MHz	6MHz	9000MHz

Conversion Rule 2: If **Frequency Start** < **Frequency Step**, then set **Frequency Start** = **Frequency Step**, interpolate for step (losing the low end) and mark the plot with the ***Interpolate*** icon.

Example 2:

Measured Parameters			**Parameters for Time Domain**		
Start	**Step**	**Stop**	**Start**	**Step**	**Stop**
6MHz	12MHz	8994MHz	12MHz	12MHz	8988MHz

Table 1-2 shows the icons that may be displayed on the Time Domain or Eye Diagram plots when the previous conversion rules are applied. These icons are displayed in the lower right corner of the plots to which they apply. For plots with multiple traces, only the icon of highest criticality is displayed. The table below shows the icons in increasing order of criticality.

Table 1-2 Time Domain/Eye Diagram Uncoupled Start Frequency/Step Frequency Icons

Icon	**Icon Name**
	**Resampled Data -** This icon indicates that only the harmonically-related data points were used to generate time domain data.
	**Interpolated Data -** This icon indicates interpolation is performed to calculate the harmonically-related points that were not measured. Any measurements that were performed at harmonically-related points are left unchanged. The interpolated data is used, along with the measured harmonically-related data, to perform the Inverse Fast Fourier Transform (IFFT) for the calculated time domain data.
	**Bad Data** – This icon indicates there are less than 10 harmonically-related data points in the measured data. In this case, all of the non harmonically-related data is used to perform the Inverse Fast Fourier Transform (IFFT) to calculate time domain data. This may result in inaccurate time domain data.

How to Perform a Calibration

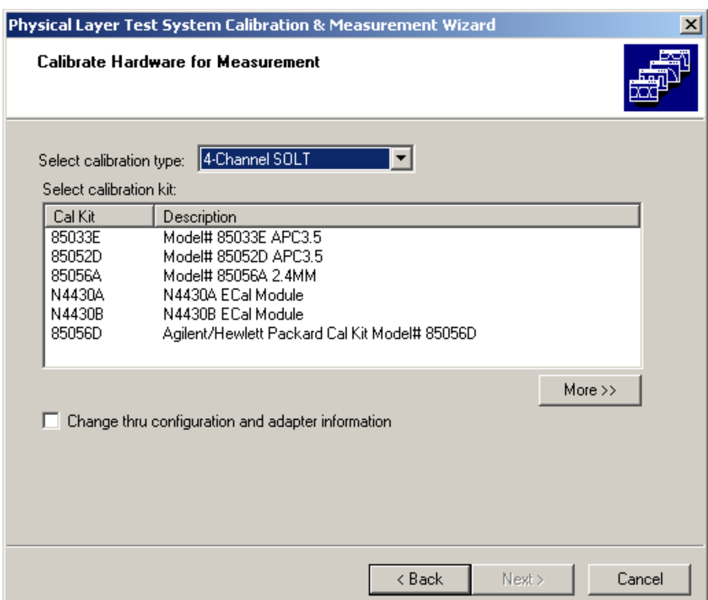
After you review and change the calibration and measurement parameters as needed and click **Next >**, the second of three *Calibrate Hardware for Measurement* dialog boxes is displayed as shown in Figure 1-7. This *Calibrate Hardware for Measurement* dialog box is used to select your calibration kit. However, the calibration procedure is described in Chapter 2, “Performing Error Correction on the VNA-Based PLTS.”

To perform the calibration, continue with “Selecting a Calibration Type” on page 29. The calibration procedure includes:

- Selecting a calibration type and identifying your calibration kit
- Performing one of the following calibration types: SOLT (either mechanical or electronic), TRL, or LRM
- Characterizing any adapters that are used, *if you do not already have the characterization data file*

This procedure guides you through the calibration steps and then, when you have finished, it refers you back to “How to Make a Measurement” on page 15.

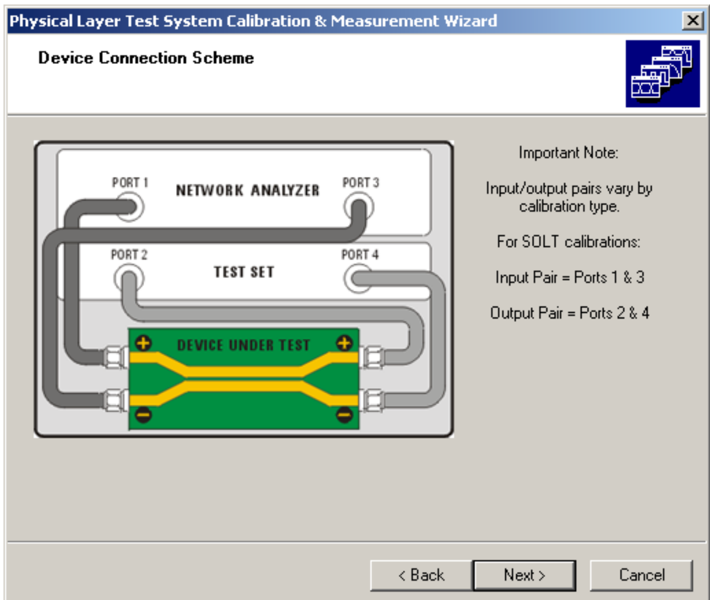
Figure 1-7 Select Calibration Type Dialog Box



How to Make a Measurement

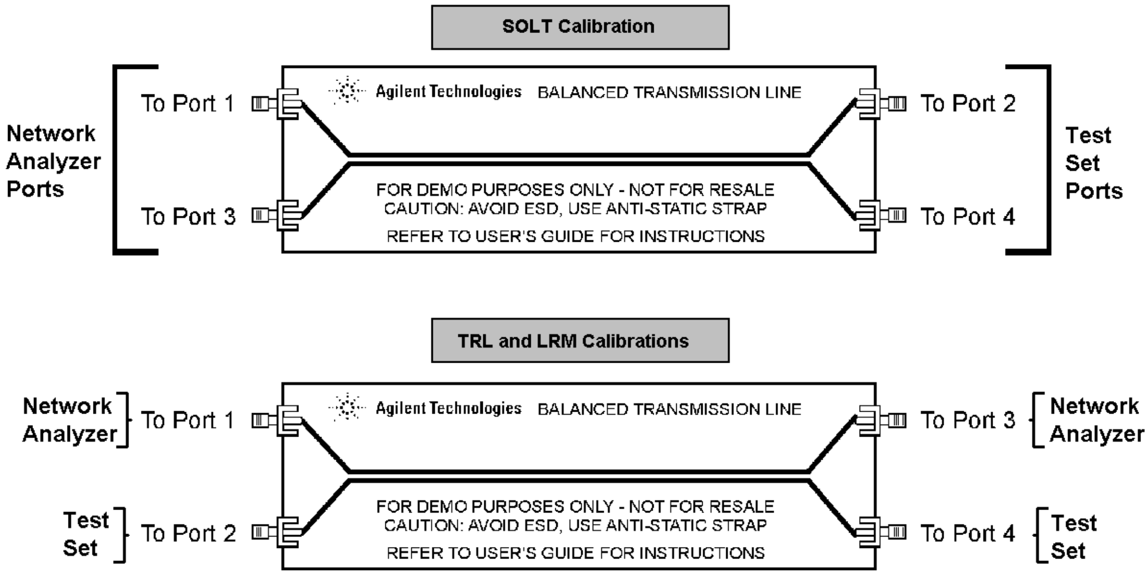
1. After saving the calibration data and selecting the **Next >** button, the *Device connection scheme* window for your calibration type is displayed. See Figure 1-8. This window shows you how the DUT must be connected to the PLTS for the type of calibration data that you are using. Connect the DUT to the PLTS hardware as shown and click the **Next >** button.

Figure 1-8 Device Connection Scheme Window



NOTE SOLT calibration data requires a different connection scheme than either TRL or LRM calibration data. Figure 1-9 shows the difference in the DUT connection between SOLT calibrations and TRL/LRM calibrations.

Figure 1-9 Device Connection Differences by Calibration Type



The Port 2 and Port 3 connections vary by calibration type.

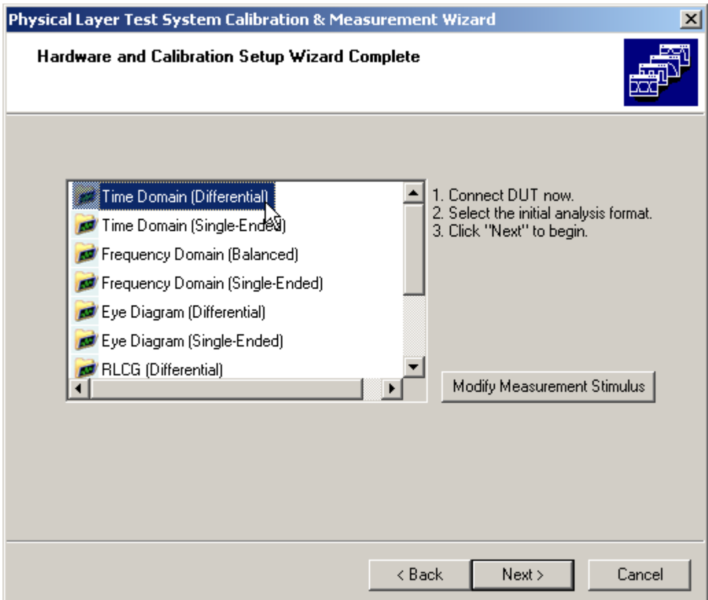
2. The *Hardware and Calibration Setup Wizard Complete* window is displayed. See Figure 1-10.

   This window is the software location to select the one of ten analysis formats that the measurement will be displayed as initially. The analysis formats are:

   - Time Domain (Differential)
   - Frequency Domain (Balanced)
   - Eye Diagram (Differential)
   - RLCG (Differential)
   - RLCG (W-Element)
   - Time Domain (Single-Ended)
   - Frequency Domain (Single-Ended)
   - Eye Diagram (Single-Ended)
   - RLCG (Common)
   - RLCG (Self/Mutual)

For the purpose of this example, the **Time Domain (Differential)** format icon is selected.

Figure 1-10 Hardware and Calibration Setup Wizard Complete Window



The **Modify Measurement Stimulus** button opens the *Measurement Stimulus* dialog box. See Figure 1-11. This dialog box allows you to change some of the measurement stimulus settings that you set up previously.

Settings that would require a recalibration are not active and may not be changed in this dialog box. To change the stimulus settings that are inactive, you must click the **< Back** button until you get back to the window described in "Setting Up the Calibration and Measurement Parameters" on page 9.

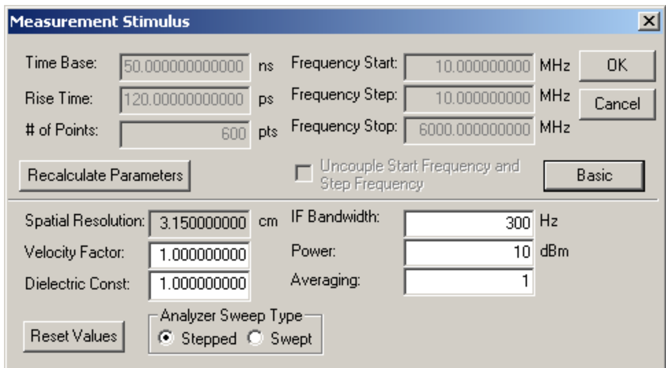
For the purpose of this example, leave the settings unchanged and click the **OK** button.

The **Reset Values** button resets any active values (any values that can be changed without requiring a recalibration) to their original default values prior to being changed in "Setting Up the Calibration and Measurement Parameters".

The **Cancel** button resets any changes that were made after opening this dialog box.

NOTE After the measurement has been made, you may verify these changes were made by printing a characterization report. See "Characterization Report Generator" on page 315 for help.

Figure 1-11 Stimulus Dialog Box

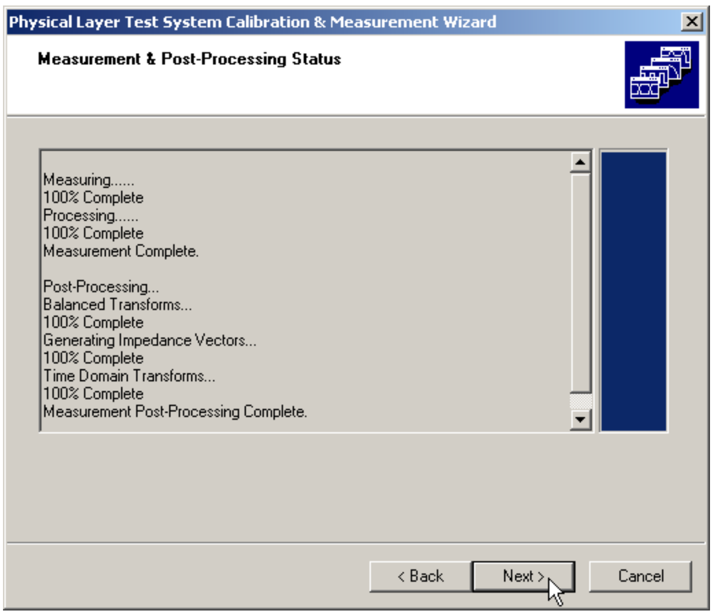


3. Once you exit the *Measurement Stimulus* dialog box, the *Hardware and Calibration Setup Wizard Complete* window is displayed again (see Figure 1-10), select the **Next >** button to start the measurement.

4. The software displays the wizard's *Measurement & Post-Processing Status* window and starts the measurement and the measurement post-processing. See Figure 1-12. The software makes each of the measurements. The status of the measurements and the post-processing is displayed in the status text area. The status is may also be observed by watching the status bar at the right edge of the text area. As the measurements and the post-processing proceed, the color of the bar gradually changes to blue.

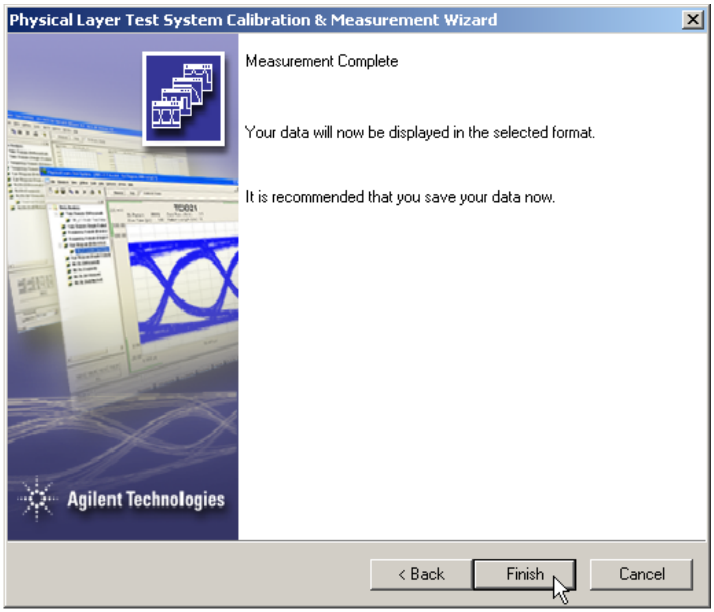
   Once the post-processing is complete, click the **Next >** button to display the *Measurement and Calibration Setup Wizard Complete* window.

Figure 1-12 Measurement & Post-Processing Status Window



5. After selecting the **Next >** button, the *Measurement and Calibration Setup Wizard Complete* window is displayed. See Figure 1-12. Click the **Finish** button to display the measurement results in the main PLTS window.

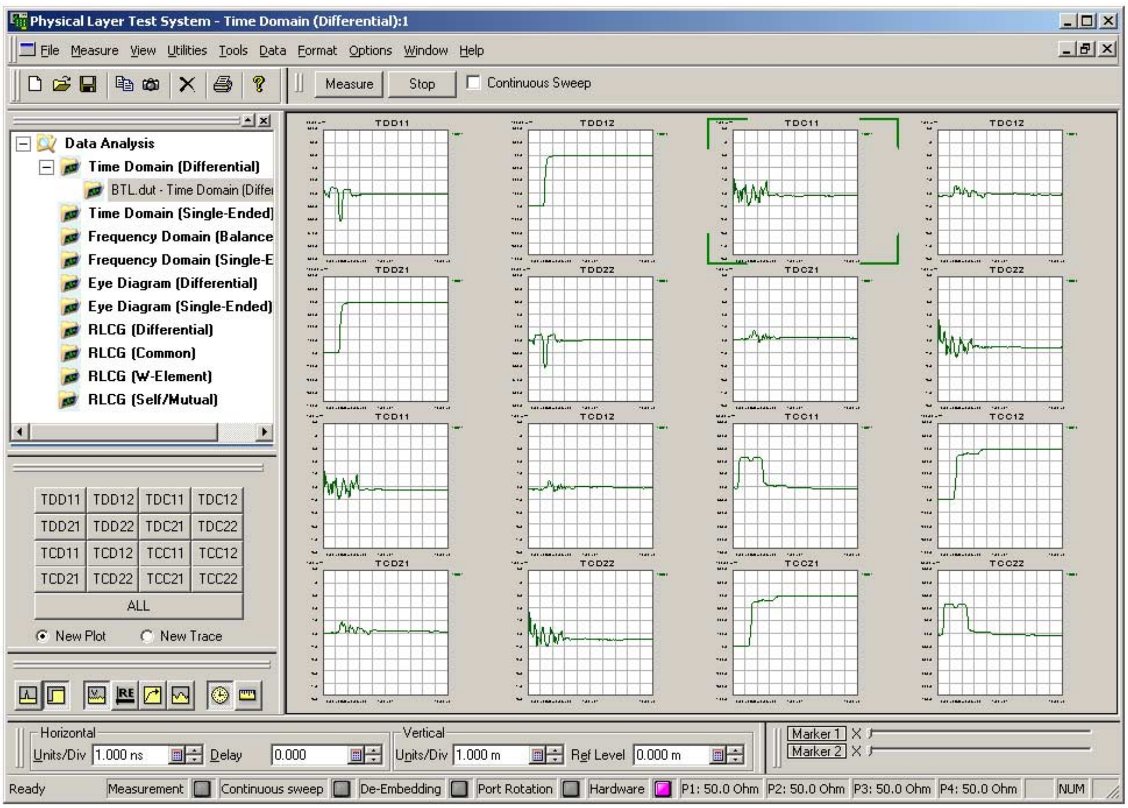
Figure 1-13 Measurement & Calibration Setup Wizard Complete Window



Analyzing the Measurement Results

Now that you have measured the device under test, its measurement results are displayed the main PLTS window as shown in Figure 1-14. This shows the results in the **Time Domain (Differential)** format, the analysis format selected earlier.

Figure 1-14 Displayed Measurement in Time Domain (Differential) Format



Now that the measurement has been made, the main Physical Layer Test System (PLTS) window gives you the flexibility to perform analysis on the measured data in a variety of ways.

- **Frequency Domain.** The Frequency Domain analysis may be analyzed in Balanced or Single-Ended mode.Refer to Chapter 4, “Analyzing Data in the Frequency Domain,” on page 127 for information.
- **Time Domain.** The Time Domain analysis may be analyzed in Differential or

Single-Ended mode. Refer to Chapter 5, “Analyzing Data in the Time Domain,” on page 147 for information.

- **Eye Diagram.** The Eye Diagram analysis may be analyzed in Differential or Single-Ended mode. Refer to Chapter 6, “Analyzing Data using Eye Diagrams,” on page 185 for information.
- **Transmission Line (RLCG).** The RLCG analysis may be analyzed in Differential, Common, W-Element, and Self/Mutual modes. Refer to Chapter 7, “Analyzing Transmission Line Parameters,” on page 205 for information.

However, to ensure the measurement data is not lost, you may want to first save the measurement data by selecting **Save** from the **File** menu. See “Save” under the “File Menu” section in the “Menu Reference” chapter of the *PLTS Installation and Reference Guide.*

2 Performing Error Correction on the VNA-Based PLTS

What Is Measurement Calibration?

Measurement calibration is an accuracy enhancement procedure that effectively removes the systematic errors (repeatable measurement variations) that cause uncertainty in measuring a device under test (DUT). During measurement calibration, the physical layer test system measures actual, well-defined standards and mathematically compares the results with ideal “models” of these standards. Calibration measurements, which characterize the test system, are made with all cables and connections in place but without the DUT.

Systematic Errors are related to signal leakage, signal reflections, and frequency response of the test system. There are six types of systematic errors.

- Directivity and crosstalk related to signal leakage
- Source and load impedance mismatches related to signals being reflected
- Frequency response error caused by reflection and transmission tracking with the test receivers

Other factors that can impact the measurement accuracy of any measurement system are drift errors and random errors.

- **Drift Errors** are due to the instrument or test-system performance changing after a calibration has been done. Drift is primarily caused by temperature variation and it can be removed by recalibration. The timeframe over which a calibration remains accurate is dependent on the rate of drift that the test system undergoes in the test environment. A stable ambient temperature usually minimizes the rate of drift significantly. Allowing equipment to warm up and stabilize prior to calibration and properly ventilating equipment helps reduce drift errors.
- **Random Errors** are unpredictable since they vary with time in a random fashion. Therefore, they *cannot* be removed by calibration. The main contributors to random error are instrument noise such as, source phase noise, sampler noise, and IF noise. The accurate source and phase-locked receiver of the network analyzer greatly minimizes these random errors. There are also external contributors to random errors such as switching power supplies, EMI, etc.

Why Is Calibration Needed?

Components of the measurement setup such as imperfect connectors, cabling, and even the response of the test instruments can introduce errors into measurements. For both transmission and reflection measurements, impedance mismatches within the test setup cause measurement uncertainties that appear as ripples superimposed on the measured data. These errors can distort the signal and make it difficult to determine which reflections are from the DUT and which are from other sources.

Calibration is required for accurate measurements. Even though calibration does take a few minutes to complete, it saves time and money compared to costs associated with erroneous measurement data. Even mechanical (non-electronic) calibration is reasonably quick once you become familiar with the process.

TIP **Understanding How Changes Affect Measurements**

No two measurements and environmental conditions are exactly the same. The best way to understand your conditions is to experiment and see how your test equipment behaves over a period of time. A good way of doing this is to measure the same device (i.e., a known standard) hourly throughout the day. Save or print the measurement results of each measurement. Compare these results to gain an understanding of how the ambient environment and drift affect the measurements. Watch for trends with regard to the device meeting specifications or measuring within guard band limits.

When Is Calibration Needed?

We recommend that you perform a calibration on your physical layer test system when the following conditions occur:

- When connectors are cleaned, repaired, or replaced.
- If test cables have any changes, such as:
  - When a test cable is replaced
  - When any connection is changed except the connections to the DUT
  - When test cables are flexed excessively (kinked or unkinked)
- If the frequency range is changed beyond the limits of the previous calibration
- If the number of measurement points is increased to more than the number of points of the previous calibration
- When ambient temperature changes more than ±3 °C
- Any other ambient environmental changes of significance
- If none of the previous conditions apply, calibrate according to the intervals shown below due to drift:
  - Check the calibration daily at a minimum (twice daily is recommended). Refer to "How to Verify a Calibration" on page 27.
  - Calibrate weekly (daily calibration recommended).

How to Verify a Calibration

A good method of checking calibration is to establish a *Golden Device*, which is a device that meets all specifications and is saved for comparison of the measured results in the future.

Establishing a Golden Device

Follow these steps to establish a golden device:

1. Calibrate the system.
2. Perform the complete set of measurements on the golden device.
3. Save and print all of the test results from these initial measurements.

Now you can measure the golden device when you suspect that your system may need to be calibrated. Compare the results of these measurements against the results that you saved and printed from the initial measurements.

How to Perform a Calibration

You will first select a calibration kit and then you will perform a calibration (either mechanical or electronic, depending on your calibration kit). After you finish the calibration, you will be ready to make a measurement.

Use the following procedure to calibrate the Physical Layer Test System.

1. If you are referring to this chapter from the example measurement in Chapter 1, skip to "Selecting a Calibration Type" on page 29.
2. Open the Physical Layer Test System Startup Wizard by doing any of the following:
   - Start the Physical Layer Test System software.
     (The startup wizard will open at start up unless the "**Do not show this wizard again at start up**" option has been selected.)
   - Select **New** from the **File** menu.
   - Select **Calibration** from the **Utilities** menu. Then select **Calibrate**.
3. Press the **Next >** button until you reach the startup wizard's *Calibrate Hardware for Measurement* dialog box displaying the default parameters, such as the maximum time base, minimum rise time, and number of point settings. See Figure 1-6 on page 10.

   Use this dialog box to check these calibration parameter settings. These default settings are based on the equipment limitations of your Physical Layer Test System.

   Selecting the **Advanced**... button to display all of the calibration parameters in the *Modify Time and Frequency Parameter* dialog box.

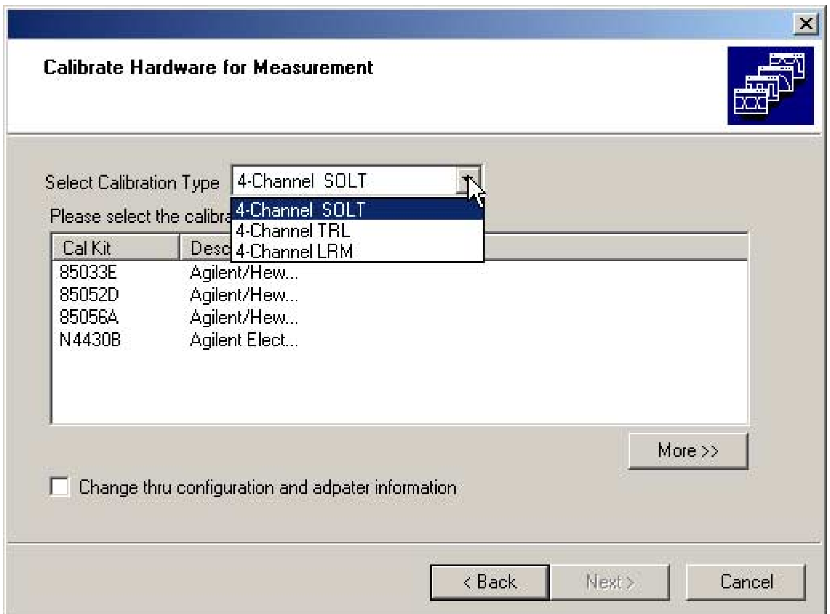
   You can change any of the time domain or frequency domain parameters, and then by selecting the **Recalibrate Parameters** button, the remaining parameters are recalibrated and displayed. Once all of the parameters are set to your satisfaction, select **OK** to return to the *Calibrate Hardware for Measurement* dialog box.

   If you decide you want to reset all changed values to their default settings, select the **Reset Values** button.
4. Once the parameters are set, select the **Next >** button to display the calibration type selection dialog box shown in Figure 2-2.

Selecting a Calibration Type

The dialog box shown in Figure 2-2 is the window that is used to select the calibration type and the calibration kit. It opens in the default SOLT (Short-Open-Load-Thru) calibration type, showing the 4-Channel SOLT calibration type along with the SOLT calibration kits that have been determined by the auto-detection to be appropriate for the system hardware. Some calibration kits listed here may not be displayed.

Figure 2-1 Select Calibration Type Dialog Box



The **Select Calibration Type** list allows you to choose the calibration type. **4-Channel SOLT** is the default calibration type. Opening the list displays that the **4-Channel TRL** (Thru- Reflect-Line) and the **4-Channel LRM** (Line-Reflect-Match) calibration types are also available.

NOTE TRL and LRM calibration types are not supported for physical layer test systems that use the 8753ES or the 872XES network analyzers.

The usage of the PLTS ports 2 and 3 varies by the calibration type that is chosen. Refer to Table 2-1, PLTS Port Configurations for a Balanced Line by Calibration Type, for the

configuration of each port measuring a balanced line.

Table 2-1 PLTS Port Configurations for a Balanced Line by Calibration Type

PLTS Ports	Calibration Type		
	SOLT	TRL	LRM
Port 1	In +	In +	In +
Port 2	Out +	In –	In –
Port 3	In –	Out +	Out +
Port 4	Out –	Out –	Out –

NOTE If you select either 4-Channel TRL or 4-Channel LRM, no calibration kit choices are initially available. You will first need to define a calibration kit that covers your measurement parameter start and stop frequency range. The calibration kit definition procedures are described later in this chapter. Continue as directed in the following list.

Select the calibration type from the list and refer to the appropriate section listed below.

- If you selected **4-Channel SOLT** calibration, refer to "Performing an SOLT Calibration" on page 31.
- If you selected **4-Channel TRL** calibration, refer to "Performing a TRL Calibration" on page 51.
- If you selected **4-Channel LRM** calibration, refer to "Performing an LRM Calibration" on page 65.

Performing an SOLT Calibration

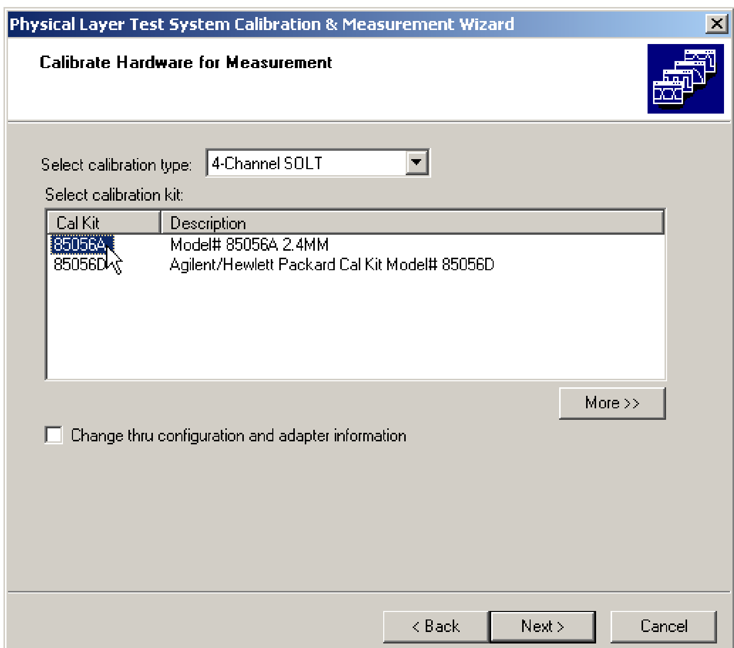
You have selected the **4-Channel SOLT** calibration type. You will first select a calibration kit and then you will perform a calibration (either mechanical or electronic, depending on your calibration kit). After you finish the calibration, you will be ready to make a measurement.

Selecting a SOLT Calibration Kit

The dialog box shown in Figure 2-2 lists the calibration kits that have been pre-defined for use with the VNA-based PLTS hardware.

Additional calibration kits, including user-defined kits, may be added by selecting **Calibration** then **Edit Cal Kit** from the **Utilities** menu. See "Defining a SOLT Calibration Kit" on page 34 for details.

Figure 2-2 Select Calibration Kit Dialog Box



NOTE The frequency range of the pre-defined calibration kits are limited to the frequency of the characterized thru adapter. The thru adapters have been characterized at frequencies used most by customers. Refer to Table 2-2 for the defined PLTS frequency boundaries of the default calibration kits.

If you set the frequency requirements beyond the defined PLTS frequency boundaries of your calibration kit, the calibration kit is not displayed. You will need to either:

- Select as pre-characterized thru file that meets or exceeds the frequency requirements to create a new **Thru Standards** definition of the calibration kit. (Refer to "Defining a SOLT Calibration Kit" on page 34.)
- Characterize your own thru adapters to meet or exceed your frequency requirements (refer to "Characterizing Adapters" on page 79) and select that newly-created thru (adapter) file to create a new **Thru Standards** definition (refer to "Defining a SOLT Calibration Kit" on page 34).

1. Select your calibration kit. Refer to Table 2-2 for the appropriate calibration kit.

Table 2-2 Calibration Kit Frequency Parameters

Calibration Kit Model Number	Nominal Frequency Range of the Calibration Kit[a]		Defined PLTS Frequency Boundaries of the Calibration Kit	
	Minimum Frequency	Maximum Frequency	Start Frequency	Stop Frequency
Mechanical Calibration Kits include:				
85033E	0 MHz	9 GHz	6 MHz	9 GHz
85052D	0 MHz	26.5 GHz	10 MHz	20 GHz
85056A	0 MHz	50 GHz	10 MHz	50 GHz
Electronic Calibration Kits include:				
N4430A[b]	30 kHz	6 GHz	30 kHz	6 GHz
N4430B	30 kHz	9 GHz	30 kHz	9 GHz

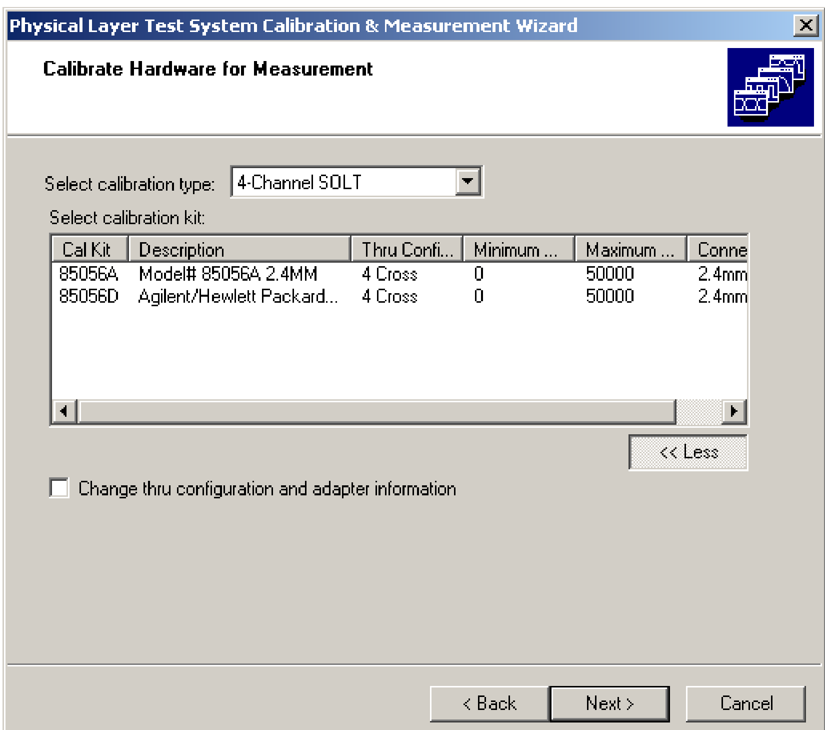
a. Refer to the specific calibration kit documentation for actual specifications and characteristics of each product.
b. Also applies to the ATN-4801 Multiport Calibration Module.

The *Nominal Frequency Range* columns display the minimum and maximum frequencies that the calibration kit will operate. These frequencies are limited by the physical calibration kit. They can *not* be changed.

The *Defined PLTS Frequency Boundaries* columns are the frequencies that the calibration kit are defined in the PLTS software when you first receive it. These frequencies are limited by the PLTS software. They may be changed as explained in the note above.

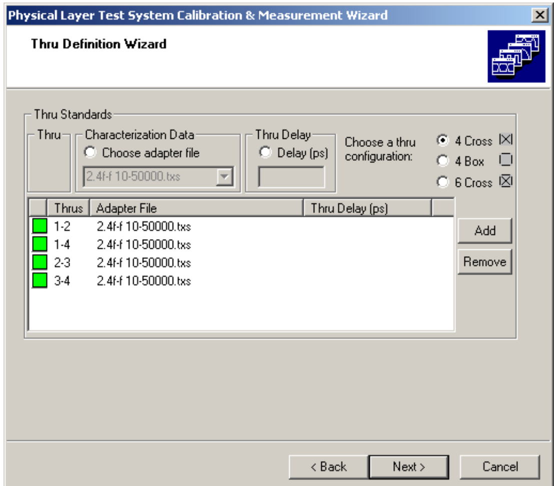
2. Select the **More >>** button to display the thru calibration configuration, the nominal minimum and maximum frequencies of the kit, and the connector type of the four ports. See Figure 2-3.

Figure 2-3 Select Calibration Kit Dialog Box with More Button Selected



3. If the thru calibration configuration is not the configuration that you wish to use, click the **Change thru configuration and adapter information** box and then click the **Next >** button. The *Thru Definition Wizard* is displayed.

Figure 2-4 Thru Definition Wizard



Refer to step 10 on page 37 (part of "Defining a SOLT Calibration Kit") for information that will help you complete the *Thru Definition Wizard*.

After changing the Thru definition, click the **Next >** button to save the change and return to the Calibration Kit selection window of Figure 2-3. Clicking the **Cancel >** button will exit the start up wizard without making any changes to the calibration kit definition.

4. Once the calibration kit is selected, click the **Next >** button to display the calibration window.
   - If you selected a mechanical calibration kit, refer to "Performing a Mechanical SOLT Calibration" on page 40 for additional information.
   - If you selected an electronic calibration kit, refer to "Performing an Electronic Calibration" on page 47 for additional information.

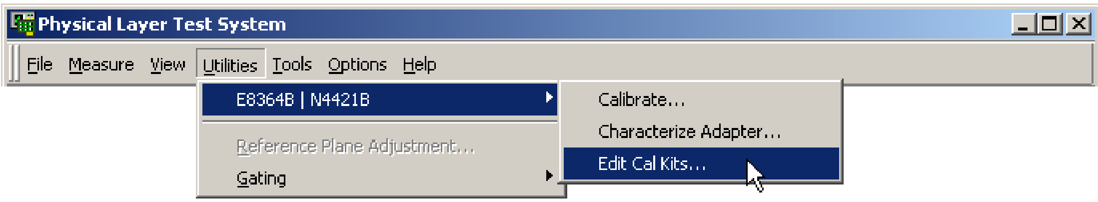
Defining a SOLT Calibration Kit

A SOLT calibration kit may be defined if the calibration kits shown in Figure 2-2 do not match your measurement needs. You will need to exit the startup wizard to define your calibration kit. Use the following procedure. You may also refer to "Edit Cal Kit" under the "Utilities Menu"section in the "Menu Reference" chapter of the *PLTS Installation and Reference Guide* for additional details.

1. Exit the startup wizard by clicking the close button in the upper right corner of the wizard:
   

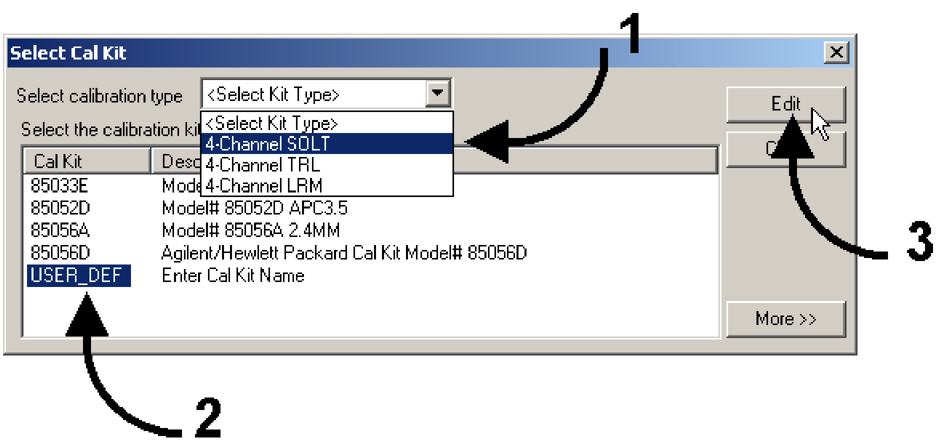
2. From the **Utilities** menu, select **Calibration**, **Edit CalKit** as shown in Figure 2-5.

Figure 2-5 Selecting Edit CalKit



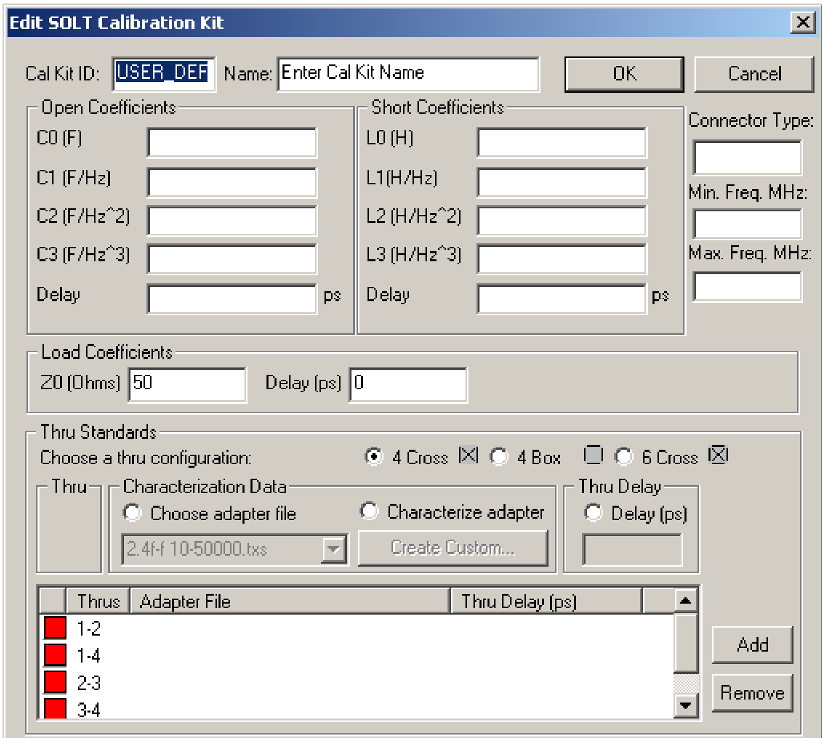
3. In the *Select Cal Kit* dialog box, select **4-Channel SOLT** from the **Select Calibration Type** list, then select the **USER_DEF** selection and click **Edit** to start the process. See Figure 2-6.

Figure 2-6 Select Cal Kit Dialog Box



4. The calibration kit definition is performed in the *Edit SOLT Calibration Kit* dialog box shown in Figure 2-7.

Figure 2-7 Blank Edit SOLT Calibration Kit Dialog Box



5. Enter **Cal Kit ID** and **Name** into the appropriate entry boxes in the header.

   Referring to Figure 2-6, **Cal Kit ID** is listed in the **Cal Kit** column and **Name** is listed in the **Description** column of the calibration kit selection window shown in Figure 2-2.

6. Also enter **Connector Type** along with the minimum (**Min. Freq. MHz**) and the maximum (**Max. Freq. MHz**) frequencies in megahertz.

   The software compares the minimum and maximum frequency entries against the measurement setup frequencies when determining which calibration kits to make available for selection for the calibration as in Figure 2-2.

7. Complete the **Open Coefficients** area by entering the coefficient and delay values displayed. If you are using a standard from a calibration kit, the coefficient and delay values may be found in the calibration kit documentation. The four coefficient values for the open are the capacitance in farads (F). Enter the capacitance value followed by the exponent. For example, if C0 is equal to 49.433×10^{-15}, the value would be entered as: `49.433E-15`

   Enter the **Delay** in picoseconds. This value may also be found in the calibration kit documentation if you are using a standard from a calibration kit.

8. Complete the **Short Coefficients** area by entering the coefficient and delay values displayed. If you are using a standard from a calibration kit, the coefficient and delay values may be found in the calibration kit documentation. The four coefficient values for the short are the inductance in henries (H). Enter the inductance value followed by the exponent. For example, if H0 is equal to -2.0765×10^{-12}, the value would be entered as: `-2.0765E-12`

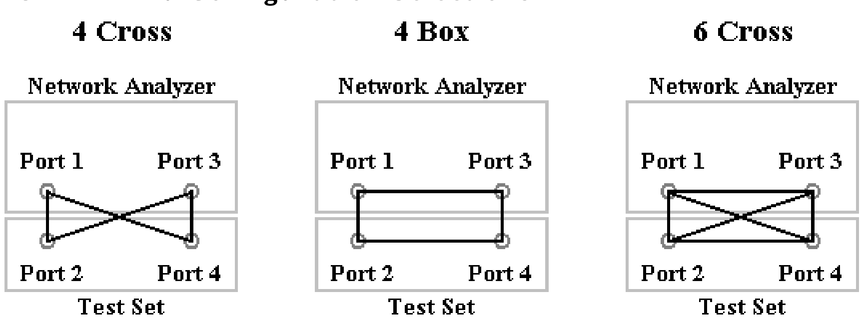
   Enter the **Delay** in picoseconds. This value may also be found in the calibration kit documentation if you are using a standard from a calibration kit.

9. Complete the **Load Coefficients** area by entering the characteristic impedance (**Z0**) in ohms and **Delay** in picoseconds. If you are using a standard from a calibration kit, the coefficient and delay values may be found in the calibration kit documentation.

10. Complete the **Thru Standards** area. Figure 2-9 shows the data that is used in this example.

    First, from the **Choose a thru configuration for this Cal Kit:** selection, select either **4 Cross**, **4 Box**, or **6 Cross**. Figure 2-8 shows the paths that will be calibrated using a thru for each selection.

Figure 2-8 Thru Configuration Selections



Next, select either **Choose Adapter File** in the **Characterization Data** area or **Delay (ps)** in the **Thru Delay** area.

- **Choose Adapter File** allows you to use a characterized file to eliminate the effects of the thru adapter when calibrating.

  If you select **Choose Adapter File**, select the characterization file for the thru adapter from the list of files.

  If you don't have a characterization file for the thru that you will use to calibrate, you can create one by selecting the **Characterize Adapter** selection in the **Characterization Data** area and then select the **Create Custom**... button. This will open the **Custom Adapter Characterization Wizard**. Refer to "Characterizing Adapters" on page 79 for information on the **Custom Adapter Characterization Wizard**.

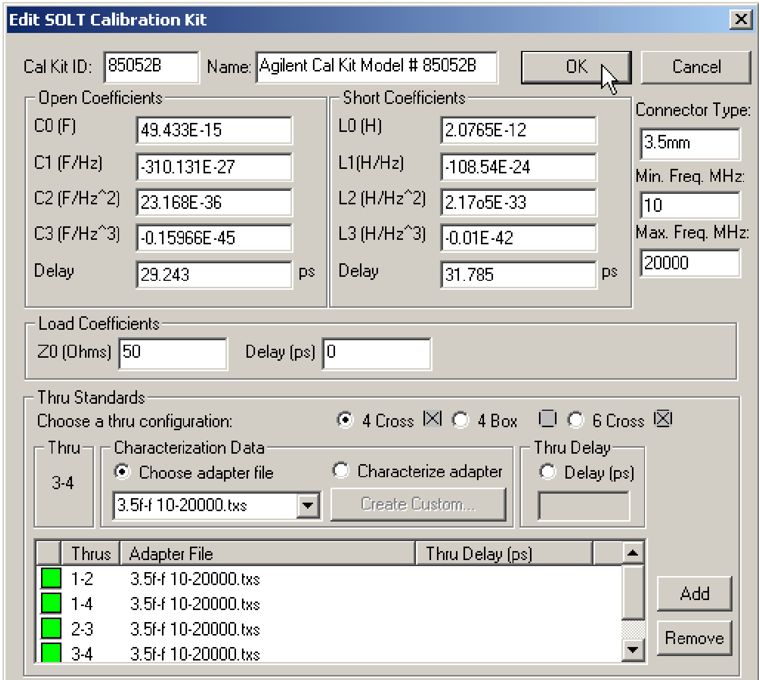
- **Delay (ps)** allows you to insert just the delay (in picoseconds) of the thru adapter, based on the electronic length of the device. If you select **Delay (ps)**, enter the time delay of the thru in the text area. If your thru is part of an Agilent (or Hewlett-Packard) calibration kit, this delay value is listed in the Standard Definitions section of the calibration kit documentation.

Finally, in the list of Thrus at the bottom of this dialog box, select the box at the left of the thru and click the **Add** button to associate the thru to either the selected adapter file or to the entered delay value. The color of the box at the left of the thru changes from red to green as the thru path is completed.

Enter a selected adapter file or a delay value for each thru path. All or the thru paths may use the same characterization file or delay value or each may have a different characterization file or delay value depending on your calibration requirements.

After the *Edit SOLT Calibration Kit* dialog box is complete, click the **OK** button to save the calibration kit with the new information and return to the *Select Cal Kit* dialog box. **Cancel** deletes the information entered and returns to the *Select Cal Kit* dialog box.

Figure 2-9 Completed Edit SOLT Calibration Kit Dialog Box

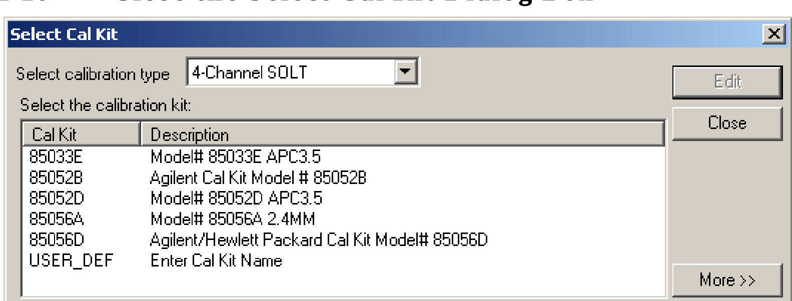


11. Once the SOLT calibration kit data has been entered into the dialog box, select **OK** to save the calibration kit data and exit the *Edit SOLT Calibration Kit* dialog box.

    The **Cancel** button closes the *Edit SOLT Calibration Kit* dialog box without saving the calibration kit data.

12. Once the SOLT calibration kit data has been saved, select **Close** to close the *Select Cal Kit* dialog box. See Figure 2-10.

Figure 2-10 Close the Select Cal Kit Dialog Box



Return to the startup wizard to complete the SOLT calibration procedure. You can select **New** from the **File** menu to restart the wizard. If you need help completing the startup wizard, return to Chapter 1 on page 3. If you do not need assistance, return to “Selecting a SOLT Calibration Kit” on page 31.

Performing a Mechanical SOLT Calibration

NOTE For maximum accuracy and repeatability, the system (network analyzer and test set) should be stabilized at room temperature for a minimum of 24 hours before calibration.

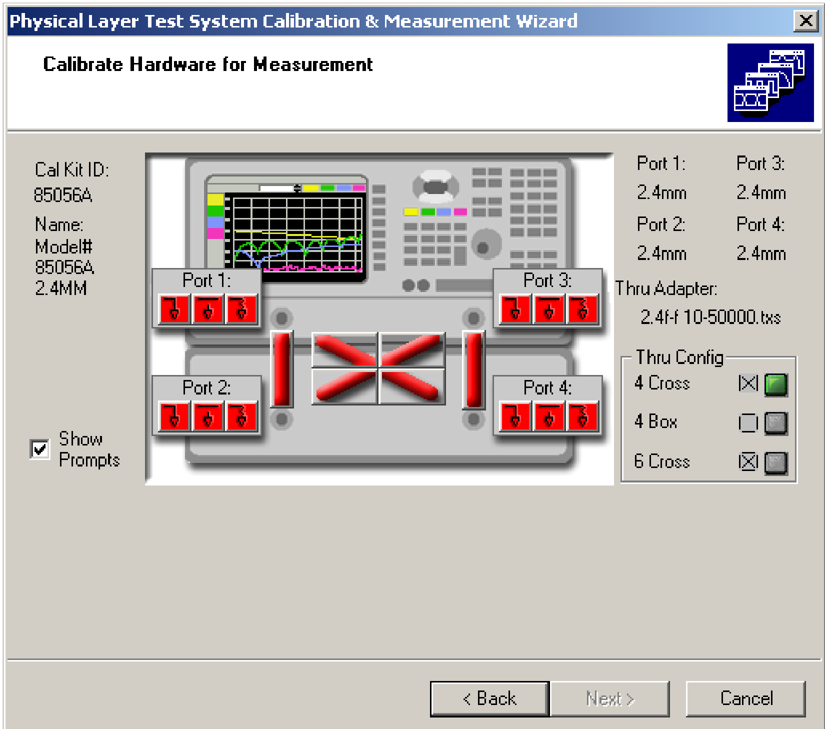
The 4-port (channel) Short/Open/Load/Thru (SOLT) calibration type is one of the most comprehensive calibrations. This calibration effectively removes the directivity error, crosstalk, source match error, load match error, frequency response reflection tracking error, and frequency response transmission tracking error from the test setup in a transmission or reflection test using these ports.

It has two unique components: the Thru component and the Short/Open/Load (SOL) component.

- The Thru component consists of connecting a through (Thru) adapter between the ends of the test cables connected to the system ports. The Thru adapter is also part of the calibration kit.
- The SOL component consists of connecting a short standard, an open standard, and a 50-ohm load standard to the end of the test cable connected to each system port. Each of these standards is part of the calibration kit.

Figure 2-11 is the mechanical calibration interface that steps you through the SOLT calibration. It displays calibration configuration information—such as the calibration kit selection, the test port connector types, the selected thru adapter, and the thru configuration—as well as a graphical representation of the test ports. This graphical representation includes a set of icons for each port, corresponding to a short, open, and load (see “The Short/Open/Load (SOL) Component” on page 44). In the center of the screen, the selected thru configuration is displayed (see “The Thru Component” on page 41).

Figure 2-11 The SOLT Calibration Interface

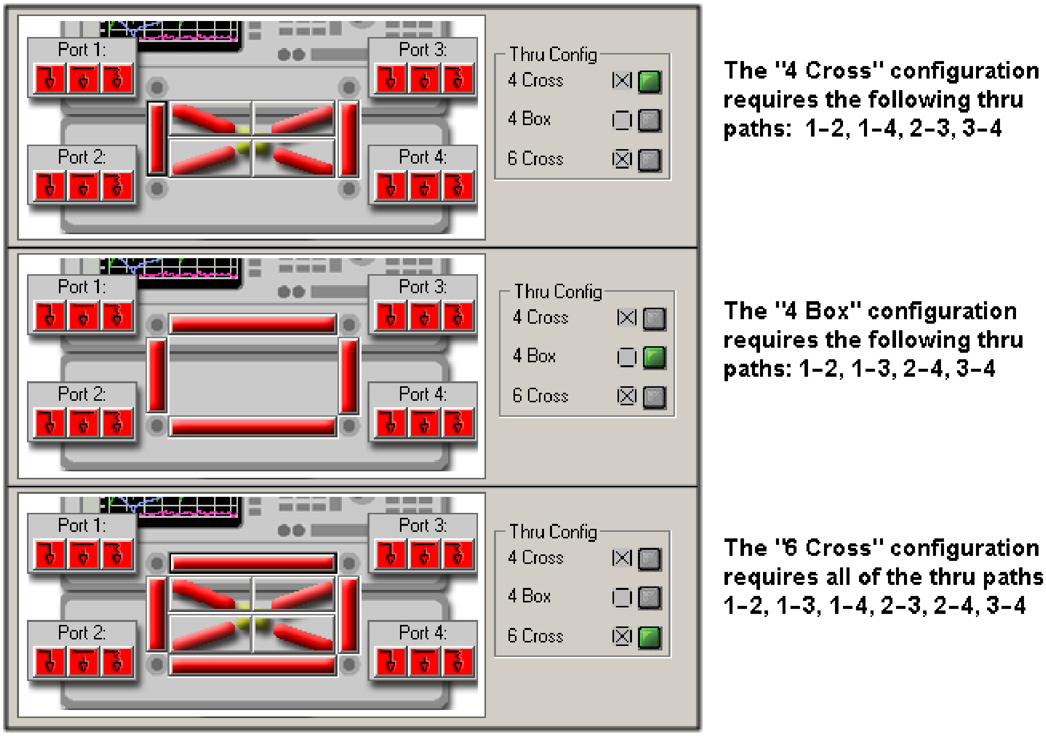


Note that all of the icons and the thru configuration paths are displayed in red at the start of the calibration. As each calibration standard is connected and measured, the color of its corresponding icon will change to blue. Once the measurement is complete, the color of the corresponding icon will change from blue to green. This change of color from red to blue to green also occurs for the thru configuration.

The Thru Component In addition to the SOL component of the calibration, a series of through (thru) measurements are required. For a 4-port calibration, it can be assumed that six thru-path calibrations are required (1-2, 1-3, 1-4, 2-3, 2-4, 3-4). However, some test setups, particularly probing setups, may make thru connections for all six paths physically impossible.

PLTS provides an alternative, and only requires four thru-paths measurements to perform an accurate four-port calibration. These alternatives can be selected in calibration kit definition process. See “Defining a SOLT Calibration Kit” on page 34.

Figure 2-12 "Thru Config" Options



The selection in the **Thru Config** area of the *Calibrate Hardware for Measurement* window (shown in Figure 2-11 on page 41) dictates the calibration pattern of the thru calibration. Figure 2-12 shows the Thru calibration pattern for each **Thru Config** selection.

There are minor trade-offs to consider when deciding between a four or six thru-path calibration. With a four thru-path calibration, the un-measured thru-paths are accounted for as the PLTS software calculates the missing transmission tracking error terms (ETF and ETR). Although this calculation has been proven to be extremely accurate, the simulated paths have a fractional amount of ripple across the band that would not be present if the full six thru-path calibration were performed. This ripple is insignificant in all but extremely low-loss DUT specifications (< 0.1 dB for example).

TIP **Comparing Thru Path Calibrations**

For the sake of convenience, the four thru-path calibrations are recommended in most situations. If you are concerned about the difference, you may choose to

perform two calibrations, one four thru-path and one six thru-path, and compare the results with your particular DUT specifications in mind.

Note that when a **Thru Config** calibration path is complete, the path color has changed from red (before the measurement) to blue (during the measurement) to green (after the measurement is complete).

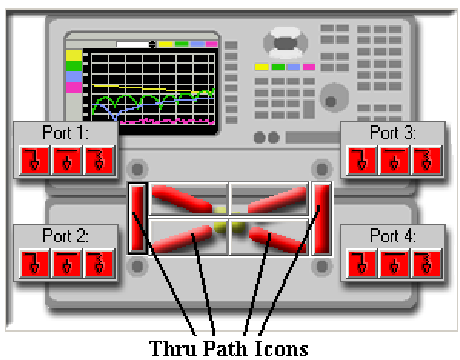
To Perform the Thru Calibration Component Use the following steps to perform the Thru calibration:

1. Connect the test cables and any adapters that you will use for your measurements to the Physical Layer Test System test ports.

   The calibration is made with all cables and connectors in place but without the DUT.

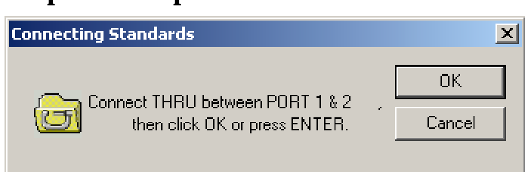
2. With the *Calibrate Hardware for Measurement* window displayed, select one of the Thru path icons on the window.

Figure 2-13 Thru Path Icons



3. Once a Thru path icon is selected, a prompt is displayed telling you which connections must be made for the Thru calibration. An example is shown in Figure 2-14.

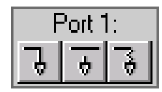
Figure 2-14 Example Prompt for Thru



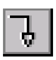
4. Continue to select the Thru icon following the instructions as each prompt is displayed. The Thru component of the calibration is complete once the color of all of the Thru paths have changed to green.

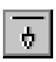
5. Continue with “The Short/Open/Load (SOL) Component” to complete the calibration.

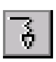
The Short/Open/Load (SOL) Component Performing the SOL portion of the calibration consists of connecting each of the standards to each of the ports for a reference measurement. Any sequence may be used, and multiple connections may be made simultaneously for convenience (for example: connect the short to port 1, load to port 2, and open to port 3, then continue). The icons change color from red (before the measurement) to blue (during the measurement) to green (after the measurement is complete).

Each of the four ports on the window has an associated set of icons as shown: 

Each of the three icons in the set represents a different part of the SOL calibration component.

 represents the short standard from the calibration kit

 represents the open standard from the calibration kit

 represents the 50-ohm load standard from the calibration kit

As each standard is connected to each port and characterized during the calibration, the color of the associated icon changes from red to blue to green.

To Perform the Short/Open/Load (SOL) Calibration Component Use the following steps to perform the SOL calibration:

1. With the *Calibrate Hardware for Measurement* window (see Figure 2-11 on page 41) displayed, ensure that the **Show Prompts** check box is checked.

TIP **To perform the SOL calibration without using prompts:**

Prompts are used to guide you through the steps for this calibration. It is not required that this calibration be performed using prompts, such as the one shown in Figure 2-15. These prompts may be disabled by clearing the **Show Prompts** checkbox. Until you become familiar with the calibration procedure, it is recommended that you use the prompts.

---

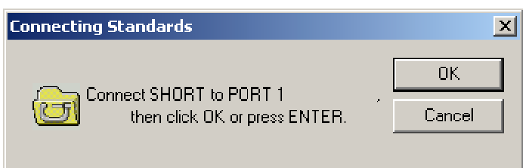
1. Ensure that the **Show Prompts** check box is cleared.
2. Select a short, open, or load standard from the calibration kit and connect it to one of the test ports. Note: You may save time by connecting more than one standard (for example: connect the short to port 1, load to port 2, and open to port 3, then continue).
3. Click the corresponding icon to perform the appropriate measurement.
4. The system will perform a reference measurement, and the color of the icon will change to green, indicating completion.
5. Repeat for the remaining standards and ports until all of the icons have changed to green.

   After you have finished the SOL calibration, select the **Next >** button to the save the calibration data. See "To Save the Calibration" on page 46.

---

2. Click on any one of the icons, at any one of the test ports.
3. A dialog box is displayed confirming the required standard and port. An example is shown in Figure 2-15.

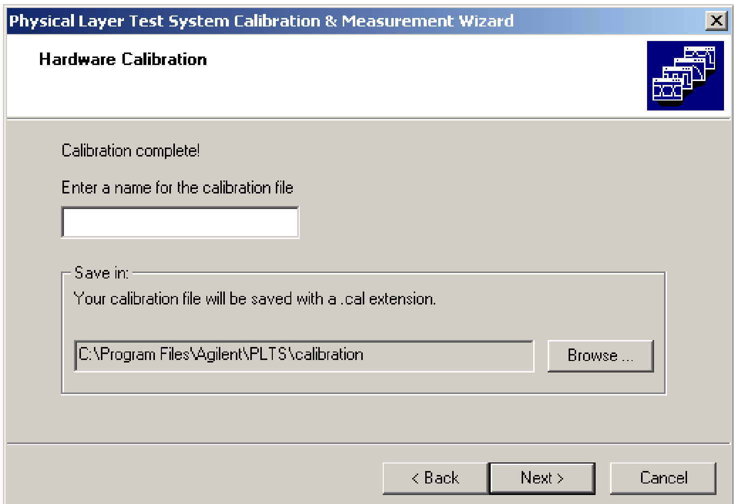
Figure 2-15 Example Prompt for Shorts, Opens, and Loads



4. Make the requested connection and click **OK**.
5. The system will perform a reference measurement, and the color of the icon will change to green, indicating completion.
6. Repeat for the remaining standards and ports until all of the icons have changed to green.
7. After you have finished the SOL calibration, select the **Next >** button to the save the calibration data. See "To Save the Calibration" on page 46.

To Save the Calibration Once you have completed both components of the SOLT calibration, the *Save Calibration Data* dialog box is displayed as shown in Figure 2-16.

Figure 2-16 Save Calibration Data Dialog Box



1. Enter a file name for the calibration data set.

   The calibration data file will be saved in the directory displayed in the **Calibration Data File Path** box. You may change the directory by entering the directory path in the box or selecting the **Browse**... button and navigating through your computer's directory structure to the desired directory. The calibration data is automatically saved with a ".cal" file extension.

2. Save the calibration data by selecting the **Next >** button.

   When the calibration data is saved, the calibration is complete.

3. If you started the calibration:

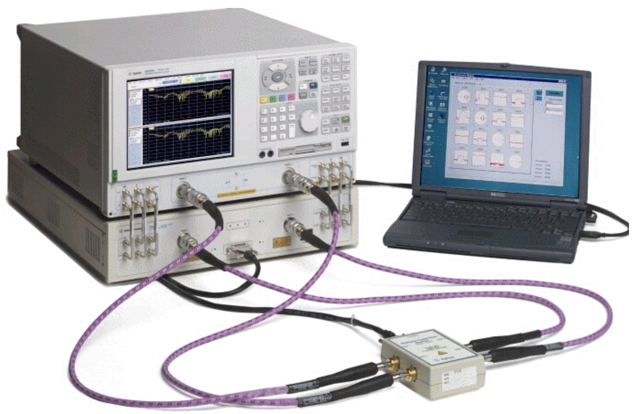
   - As part of the example measurement of Chapter 1, return to "How to Make a Measurement" on page 15.
   - By entering the startup wizard at startup or by selecting **New** from the **File** menu, the software sends you to the Startup Wizard's *Setup & Calibration Complete!* window where you can make a measurement.
   - By selecting **Calibration** from the **Utilities** menu and then selecting **Calibrate**, the software returns to the main software window.

Performing an Electronic Calibration

You can perform a 4-port electronic calibration (ECal) using the Agilent N4430A 6 GHz 4-Port ECal Module[1] or the N4430B 9 GHz 4-Port ECal Module.

With a one-time connection, the ECal procedure cycles through all of the impedance states and six thru paths required for a full, four-port, vector error-corrected SOLT calibration, and can transfer factory-calibration accuracy to the Physical Layer Test System. The software controls the electronic calibration. Refer to Figure 2-17 for a typical equipment setup.

Figure 2-17 Typical Equipment Setup for ECal



NOTE For maximum accuracy and repeatability, the system (network analyzer, test set, and ECal module) should be stabilized at room temperature for a minimum of 24 hours before calibration.

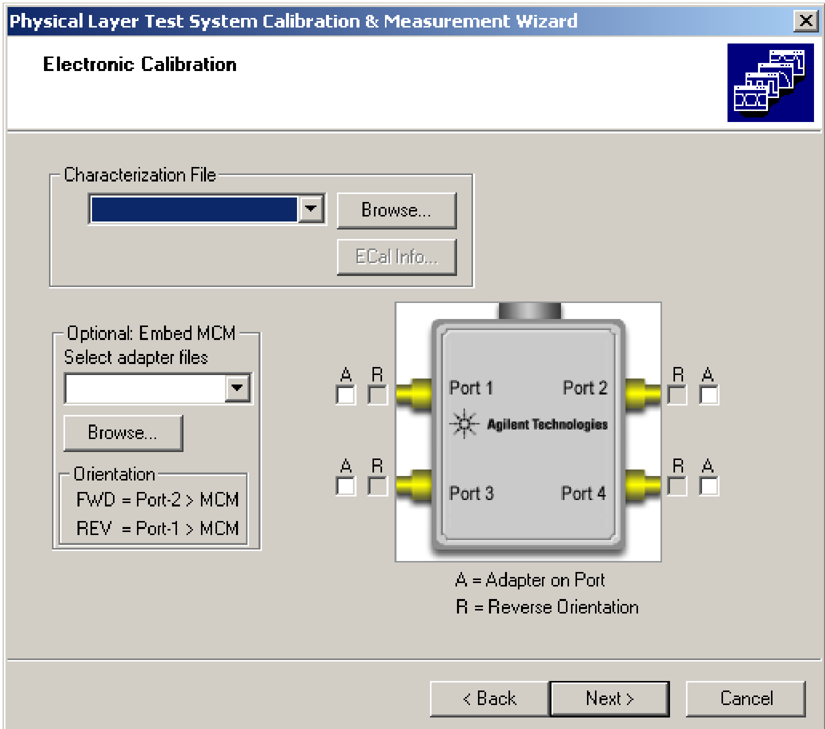
Using standard alignment precautions, the test set may have the ECal module connected and disconnected in any power state. After calibration, the ECal module may remain connected or may be disconnected from the test set Auxiliary connector with no effect on the calibration.

To perform the ECal with the physical layer test system:

1. Once you select an electronic calibration kit and then click the **Next >** button as instructed on the last step of the "Selecting a SOLT Calibration Kit" procedure on page 34, the *Electronic Calibration* is displayed. See Figure 2-18.

1. The ATN-4801 Multiport Calibration Module may also be used.

Figure 2-18 Electronic Calibration Dialog Box



2. Connect the test cables and adapters that you will use for your measurements to the Physical Layer Test System test ports.

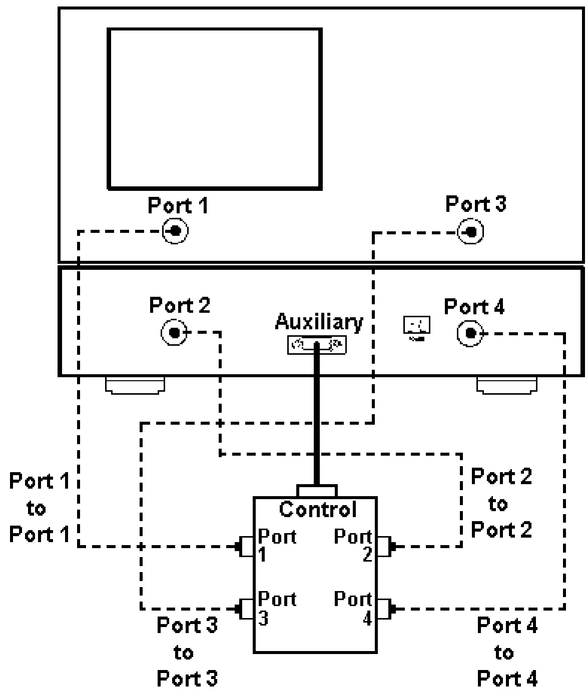
3. Connect the other end of the test cables and adapters to the ECal module so that the connections identified in Table 2-3 are made. If adapters are required to make the connections from the test cable/adapter setup to the ECal module, use the adapters in the ECal module kit.

Table 2-3 ECal Module Connections

ECal Module Connector	PLTS Connector	Cable Type	Specified Torque
CONTROL	AUXILIARY	DB-15	N/A
PORT 1	PORT 1	Coax	8 in/lb
PORT 2	PORT 2	Coax	8 in/lb
PORT 3	PORT 3	Coax	8 in/lb
PORT 4	PORT 4	Coax	8 in/lb

CAUTION Make sure that test port 2 of the system is connected to Port 2 of the ECal module and that test port 3 of the system is connected to Port 3 of the ECal module. Connecting to the wrong port will lead to incorrect calibration results. Refer to Figure 2-19 for test cable connections to the ECal module.

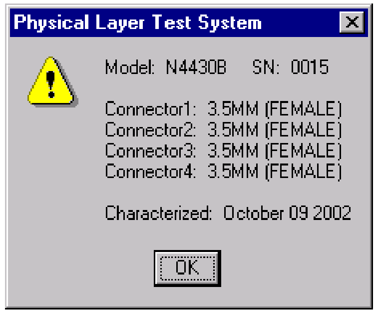
Figure 2-19 Connecting Test Cables to the ECal Module



4. Select the electronic calibration characterization file for your ECal module from the list in the **Characterization File** area. Refer to Figure 2-18. Verify that the file name matched the serial number of your ECal module.

   If your characterization file was not located by the software, load the characterization file from the floppy disk provided with the ECal module. Load the file from the floppy disk by inserting the floppy disk in the PC, selecting the **Browse**... button, and locating the characterization file from the floppy disk directory using the *Open* dialog box. Each module characterization file is unique, so verify that the file name matches the ECal module serial number. The characterization file has an “.ecl” extension. The ECal module should be loaded into the C:\Program Files\Agilent\PLTS\ecal, where C is the hard drive where the PLTS is stored.

NOTE Select the **ECal Info**... button to display the following ECal module information dialog box for the selected module.



5. If your system's test cable setup won't connect directly to the ECal module, you may use an adapter that has already been characterized. To use the adapter, select the adapter's characterization file from the **Select Adapter Files** list. Refer to "Characterizing Adapters" on page 79 for detailed information.

NOTE If you use one or more of the 3.5 mm male to male adapters in the ECal kit, you may select one of the 3.5m-m.txs files from the **Select Adapter Files** list.

- If you are connecting the adapter in the forward orientation (with the adapter's characterized port 2 connected to the ECal module), just select the checkbox labeled **A** (representing Adapter) for the port to which it is being connected.
- If you are connecting the adapter in the reverse orientation (with the adapter's characterized port 1 connected to the ECal module), select the checkbox labeled **A** (representing Adapter) and the checkbox labeled **R** (representing Reverse) for the port to which it is being connected.

NOTE The Adapter **(A)** checkbox must be selected before the program accounts for the Reverse **(R)** checkbox being selected.

6. Once the system is connected to the ECal module, select **Next >** to start the electronic calibration. A small window showing the calibration progress is displayed.

   Once the calibration is complete, it should be saved. See "To Save the Calibration" on page 46.

Performing a TRL Calibration

You have selected the 4-Channel TRL (THRU - REFLECT - LINE) calibration type. TRL calibration is extremely accurate, in some cases more accurate than an SOLT cal. However, very few calibration kits contain TRL standards. TRL calibration is most often performed when you require a high level of accuracy and do not have calibration standards in the same connector type as your DUT. This is usually the case when using test fixtures, or making on-wafer measurements with probes.

The DUT must be physically connected to the PLTS by some kind of transition network or fixture. Therefore, in some cases you must fabricate and characterize standards in the same media type as your DUT configuration. It is easier to manufacture and characterize three TRL standards than the four SOLT standards. A limitation for TRL cal with broad frequency coverage is the requirement for multiple LINE standards. For example, a span from 2 GHz to 26 GHz requires two line standards. Also, for lower frequencies, the LINE standard can be too long for practical use.

TRL Calibration Kits

Agilent Technologies offers two cal kits that include the required standards to perform a TRL calibration: 85050C (APC 7mm) and 85052C (3.5mm). Both kits include the traditional Short, Open, and Load standards. (The Thru standard, not actually supplied, assumes a zero-length Thru). In addition, the kits include an airline which is used as the LINE standard. To use the airline, the kits include an airline body, center conductor, and insertion/extraction tools.

TRL Calibration Standards

These standards must be defined in your TRL cal kit:

THRU

- The THRU standard can be either a zero-length or non-zero length. However, a zero-length THRU is more accurate because it has zero loss and no characteristic impedance.
- The THRU standard cannot be the same electrical length as the LINE standard.
- If the insertion phase and electrical length are well-defined, the THRU standard is used to set the reference plane.

REFLECT

- The REFLECT standard can be anything with a high reflection, as long as it is the same when connected to all ports.
- The actual magnitude of the reflection need not be known.

- The phase of the reflection standard must be known within 1/4 wavelength.

LINE

The LINE standard establishes the reference impedance for the measurement after the calibration is completed. TRL calibration is limited by the following restrictions of the LINE standard:

- Limited to 4 Lines.
- Must be of the same impedance as the THRU standard.
- The electrical length need only be specified within 1/4 wavelength.
- Cannot be the same length as the THRU standard.
- Must be an appropriate electrical length for the frequency range: at each frequency, the phase difference between the THRU and the LINE should be greater than 20 degrees and less than 160 degrees. This means in practice that a single LINE standard is only usable over an 8:1 frequency range (Frequency Span / Start Frequency). Therefore, for broad frequency coverage, multiple lines are required.
- At low frequencies, the LINE standard can be too long for practical use. Two 50-ohm terminations can be used in place of a long LINE standard at low frequencies.

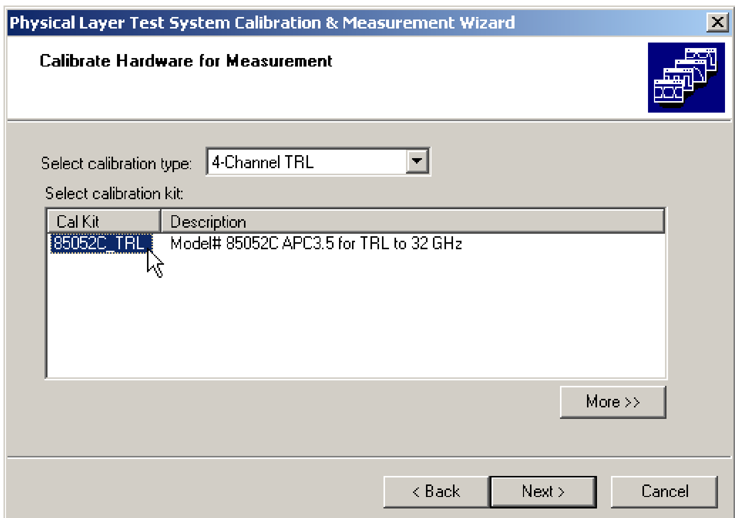
NOTE TRL calibration is not supported for physical layer test systems that use the 8753ES or the 8720ES-series network analyzers.

Start with "Selecting a TRL Calibration Kit".

Selecting a TRL Calibration Kit

The dialog box shown in Figure 2-20 lists the defined TRL calibration kits that have been determined by an auto-detection process to be appropriate for the system hardware and the frequency range that was defined in the measurement parameters.

Figure 2-20 Select TRL Calibration Kit Dialog Box



NOTE No TRL calibration kits are defined in the software when you receive it.

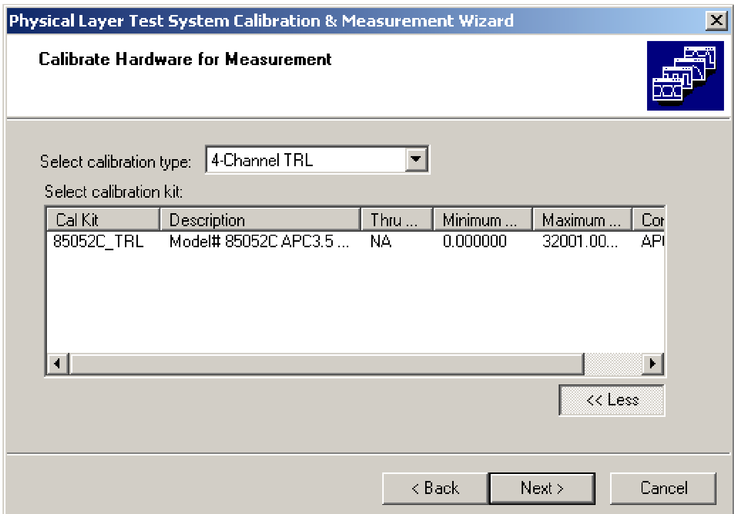
- If you *have not* defined your TRL calibration kit, select the **Cancel** button and define your TRL calibration kit using the procedure described in "Defining a TRL Calibration Kit" on page 54.
- If you *have* defined a TRL calibration kit but it is not displayed in the dialog box:

  1. Make sure that you have selected **4-Channel TRL** from the **Select Calibration Type** list. See "Selecting a Calibration Type" on page 29.
  2. Make sure that the start and stop frequencies that were defined for the TRL calibration kit ("Defining a TRL Calibration Kit" on page 54) match or exceed the start and stop frequencies (≤ the start frequency and ≥ the stop frequency) that were defined in the measurement parameters ("Setting Up the Calibration and Measurement Parameters" on page 9).

---

1. Select your calibration kit in the **Cal Kit** column.
2. Select the **More >>** button to display the thru calibration configuration, the minimum and maximum frequencies of the kit, and the connector type of the four ports. See Figure 2-21.

Figure 2-21 Select Calibration Kit Dialog Box with More Button Selected



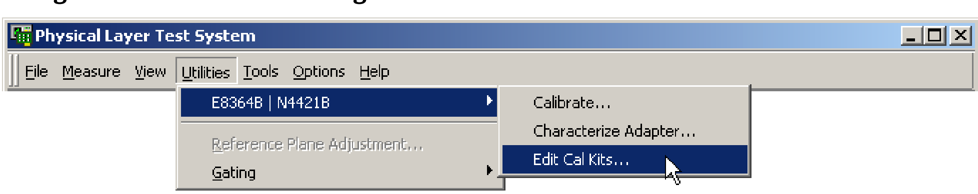
3. Once the calibration kit is selected, click the **Next >** button to display the calibration window and continue with “Performing a TRL Calibration” on page 57.

Defining a TRL Calibration Kit

The dialog box shown in Figure 2-23 shows only the **USER_DEF** selection. This indicates that a TRL calibration kit must be defined before a TRL calibration may be performed.

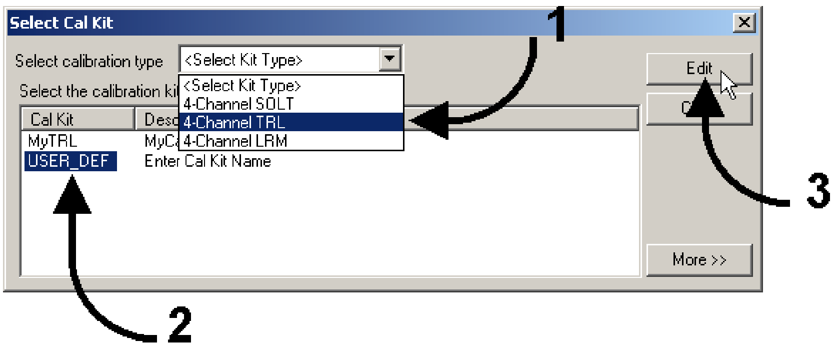
1. From the **Utilities** menu, select your PLTS system name and then select **Edit Cal Kits**... as shown in Figure 2-22.

Figure 2-22 Selecting Edit Cal Kits...



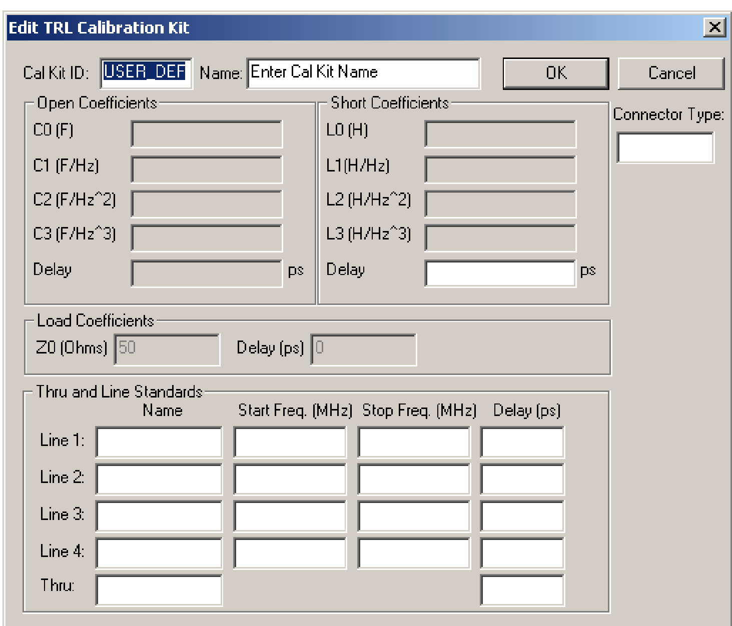
2. In the *Select Cal Kit* dialog box, select **4-Channel TRL** from the **Select Calibration Type** list, then select the **USER_DEF** selection and click **Edit** to start the definition process. See Figure 2-23.

Figure 2-23 Select Cal Kit Dialog Box



3. The TRL calibration kit is defined using the *Edit TRL Calibration Kit* dialog box. See Figure 2-24.

Figure 2-24 Blank Edit TRL Calibration Kit Dialog Box



4. Enter **Cal Kit ID**, **Name**, and **Connector Type** into the appropriate entry boxes in the header.

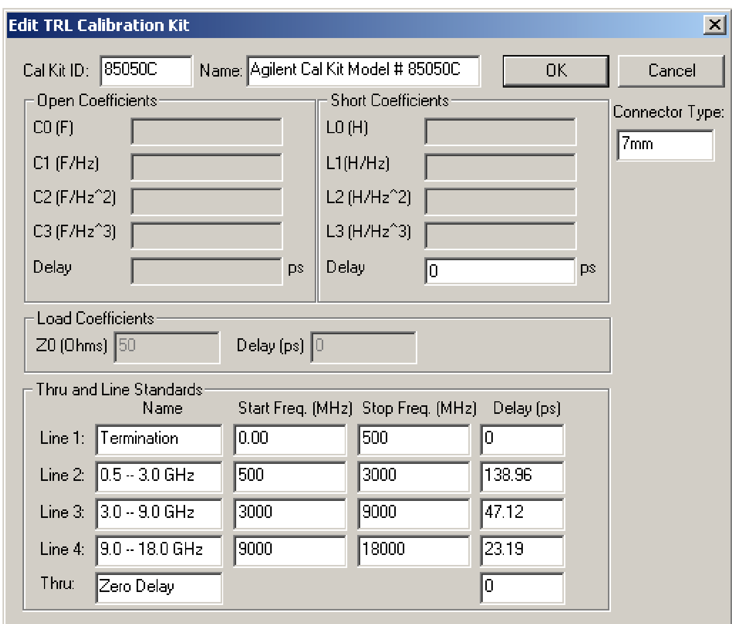
   Referring to Figure 2-25, **Cal Kit ID** is listed in the **Cal Kit** column and **Name** is listed in the **Description** column of the calibration kit selection window.

   In this example, **Cal Kit ID** is defined as "`85050C`", **Name** is defined as "`Agilent Cal Kit Model # 85050C`", and **Connector Type** is defined as "`7mm`". Refer to Figure 2-21.

5. Enter the delay value of the device you are using as the Reflect in the **Short Coefficients** area. Figure 2-25 shows the data that is used in this example.

6. Complete the *Edit TRL Calibration Kit* dialog box. For the **Thru and Line Standards** area, enter a name for each **Line** along with each line's start and stop frequency (in MHz) and delay (in ps). Information for up to four lines can be entered into this area. If you are using a standard from a calibration kit, the coefficient and delay values may be found in the calibration kit documentation. If the delay of a line is entered as 0 ps, the line is assumed to be a load (50Ω termination). Also enter the name and delay of the **Thru** device, which is typically 0 ps. Figure 2-25 shows the data that is used in this example.

Figure 2-25 Completed Edit TRL Calibration Kit Dialog Box

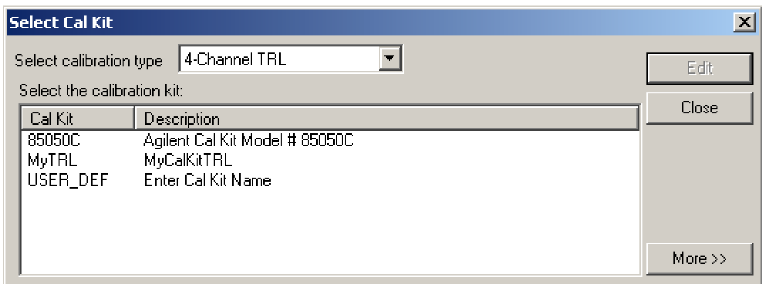


7. Once the TRL calibration kit data has been entered into the dialog box, select the **OK** button to save the calibration kit data and exit the *Edit TRL Calibration Kit* dialog box.

   The **Cancel** button closes the *Edit TRL Calibration Kit* dialog box without saving the calibration kit data.

8. Once the TRL calibration kit data has been saved, select **Close** to close the *Select Cal Kit* dialog box. See Figure 2-26.

Figure 2-26 Close the Select Cal Kit Dialog Box



9. Return to the startup wizard to complete the TRL calibration procedure. You can select **New** from the **File** menu to restart the wizard.

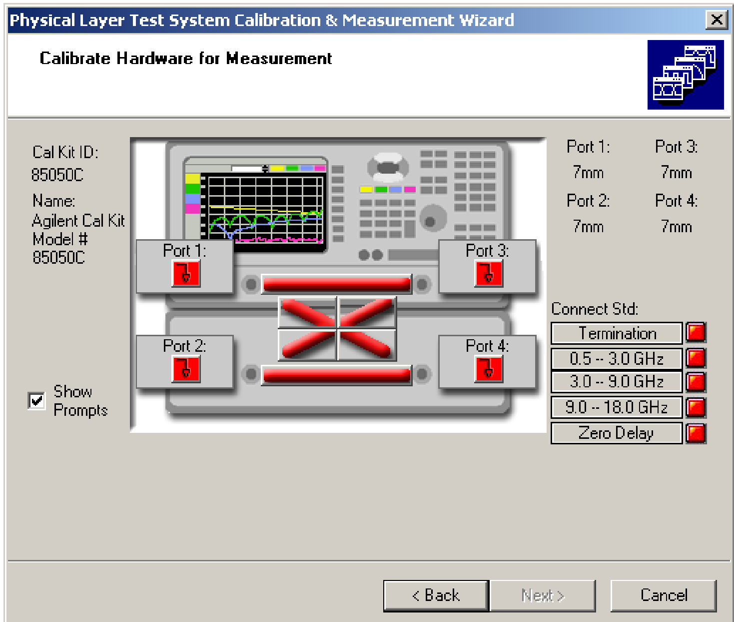
   - If you need help completing the startup wizard, return to Chapter 1 on page 3.
   - If you do not need assistance, continue with "Selecting a TRL Calibration Kit" on page 53.

Performing a TRL Calibration

NOTE For maximum accuracy and repeatability, the system (network analyzer and test set) should be stabilized at room temperature for a minimum of 24 hours before calibration.

As indicated by its name, the Thru-Reflection-Line (TRL) calibration consists of three portions. First, the reflection portion is performed by connecting a short to each system port. The Line portion is performed by connecting each line and a thru between system ports 1 and 3 and then by connecting the same lines and thru between ports 2 and 4. Finally, the Thru portion is performed by connecting the thru between system ports 1 and 4 and then by repeating the connection between ports 2 and 3.

Figure 2-27 Initial TRL Calibration Display



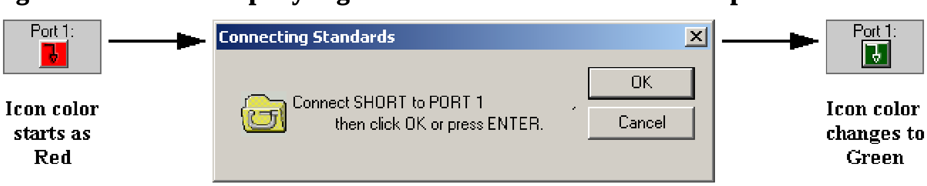
The TRL calibration is performed in two parts which may be performed in any order. These instructions show you how to perform the Reflect portion of the calibration first. The Thru-Line portion of the calibration is documented next.

Reflect Portion of TRL Calibration The following three steps are the Reflect portion of the TRL calibration.

1. Select the following icon to start the Reflection portion of the calibration: 

2. As described in the prompt shown in Figure 2-28, connect the reflective device (in this case, defined as a short) from your calibration kit to Port 1 and click **OK**.

Figure 2-28 Displaying the Port 1 Reflection Prompt



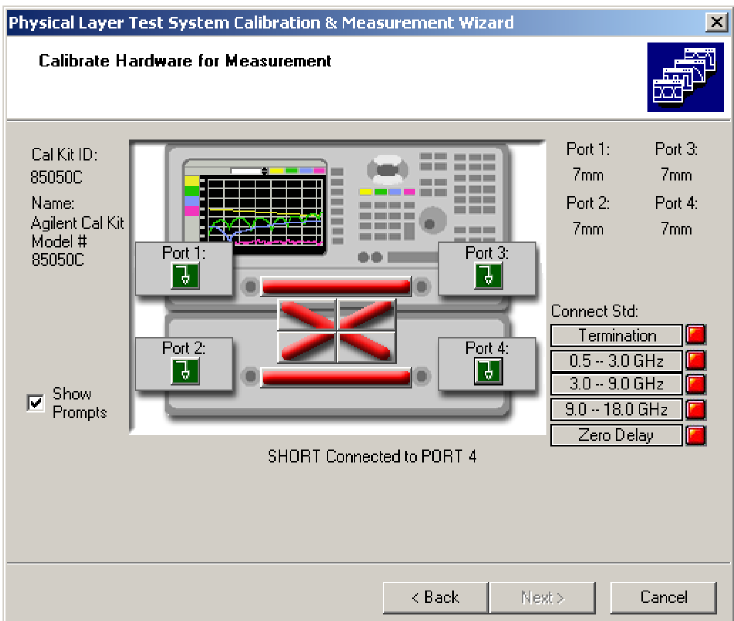
As the portion of the calibration is complete, the Port 1 icon color changes from red to green. Actually, as the measurement is being made, the icon is blue, however, the measurement process may be so quick that you do not see the icon turn blue before turning to green.

3. Repeat steps 1 and 2 for each of the remaining ports.

   The order that the ports are calibrated does not matter.

   Once all four of the ports are calibrated using the Reflect device, the color of each reflect icon has changed to green. See Figure 2-29.

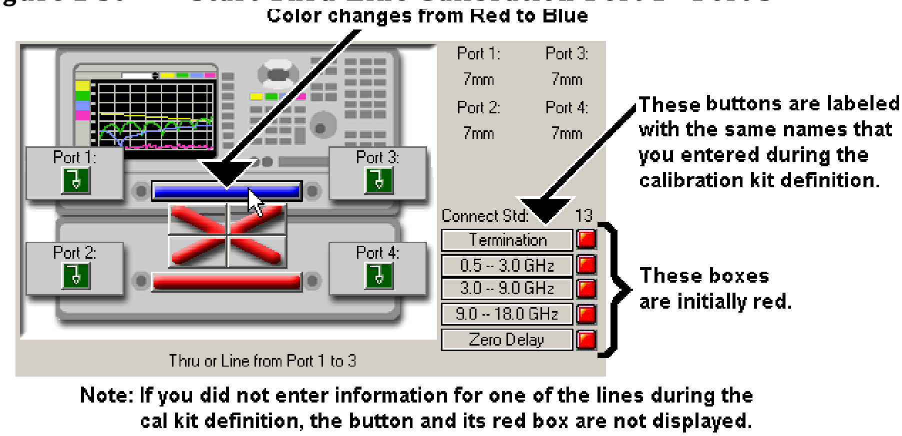
Figure 2-29 Reflection Portion of Calibration Complete



Thru-Line Portion of TRL Calibration The following steps are the Thru-Line portion of the TRL calibration.

1. Click the Thru-Line icon that extends from Port 1 to Port 3. When this icon is clicked, the color changes from red to blue. See Figure 2-30.

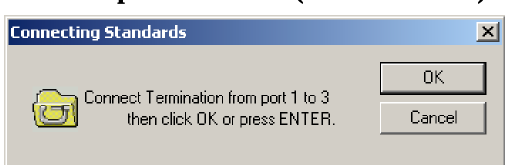
Figure 2-30 Start Thru-Line Calibration Port 1 - Port 3



2. Click the top button on the right side of the display. In this case, the button is called **Termination** because the TRL calibration kit was defined with that label for **Line 1 Name**. See Figure 2-25 on page 56. If you enter another name for Line 1, that label is used.

   When you click the **Termination** button, the following prompt window is displayed:

Figure 2-31 Prompt for Line 1 (Termination) Calibration



3. Make the connections as described in the prompt window and click **OK** to start the calibration measurement.

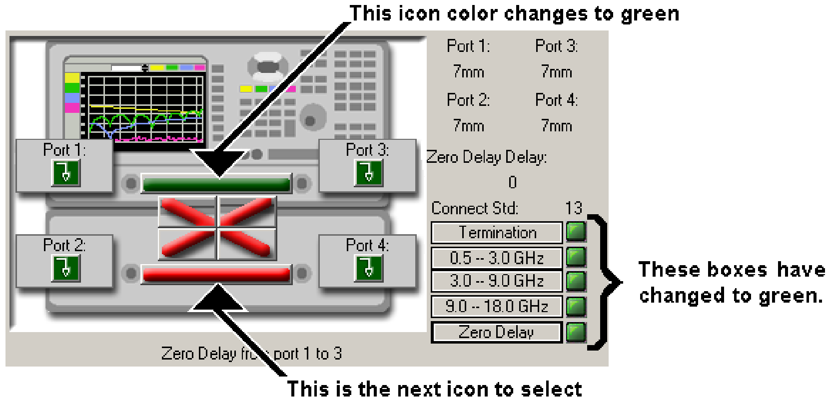
   Once this measurement is made, the color of the box at the right of the **Termination** button changes from red to green.

4. Repeat steps 2 and 3 for Line 2 (**0.5 -- 3.0GHz**), Line 3 (**3.0 - 9.0GHz**), Line 4 (**9.0 - 18.0GHz**), and the Thru (**ZeroDelay**).

Once these measurements are taken, the calibration between ports 1 and 3 is complete. See Figure 2-32. On the display, the color of:

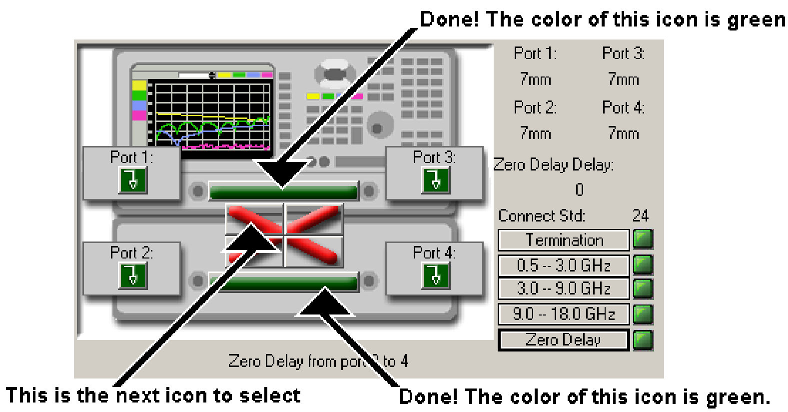
- Each box to the right of the buttons have changed from red to green.
- The line between ports 1 and 3 changes from blue to green indicating that this path is complete.

Figure 2-32 Port 1 - Port 3 Line Portion of Calibration Complete



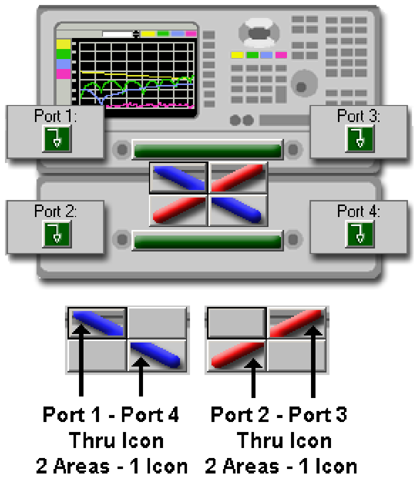
5. Repeat steps 1 through 4 for the Port 2 to Port 4 path.

Figure 2-33 Line Portion of Calibration Complete



6. Select the Port 1 - Port 4 Thru icon.

Figure 2-34 Thru Icons for Port 1 - Port 4 and Port 2 - Port 3



The icon (both halves) turns blue and the box at the right of the **Zero Dela**y Thru button turns red.

7. Click the **Zero Delay** thru button which displays the prompt window.

   All of the line buttons are inactive.

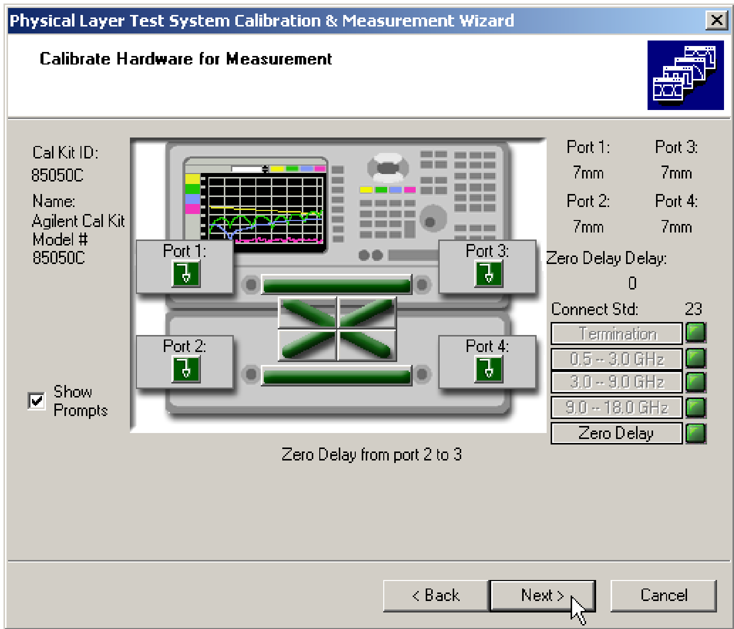
8. After the thru is connected between ports 1 and 4 as indicated on the prompt and click the **OK** button.

   The calibration is performed and the color of the box to the right of the thru button and the thru icon are changed to green.

9. Repeat steps 6 through 8 to complete the Thru calibration between Port 2 and Port 3.

10. Once the Port 2 to Port 3 Thru calibration is made, the TRL calibration measurements are complete. The color of all icons has changed to green. Refer to Figure 2-35.

Figure 2-35 TRL Calibration Complete

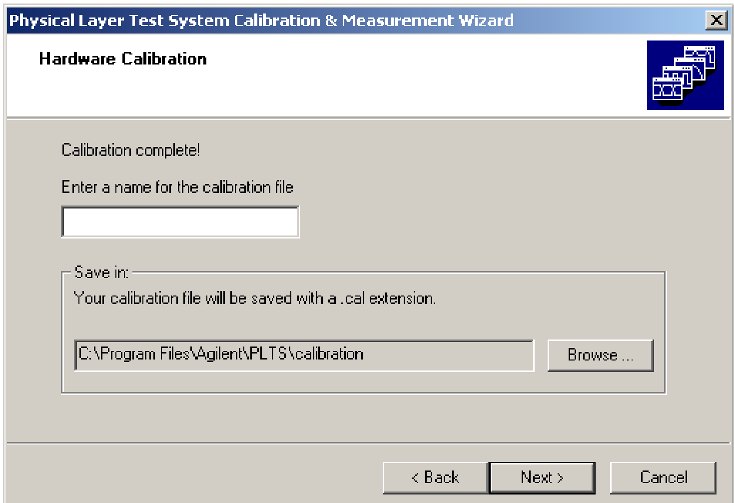


11. Click the **Next >** button.

Continue at "To Save the TRL Calibration" on page 64.

To Save the TRL Calibration Once you have completed the TRL calibration, the *Save Calibration Data* dialog box is displayed as shown in Figure 2-36.

Figure 2-36 Save Calibration Data Dialog Box



1. Enter a file name for the calibration data set.

   The calibration data file will be saved in the directory displayed in the **Calibration Data File Path** box. You may change the directory by entering the directory path in the box or selecting the **Browse**... button and navigating through your computer's directory structure to the desired directory. The calibration data is saved with a ".cal" file extension.

2. Save the calibration data by selecting the **Next >** button.

   When the calibration data is saved, the calibration is complete. A window is displayed showing how to make the connections to measure your DUT following the TRL calibration.

3. If you started the calibration:

   - As part of the example measurement of Chapter 1, return to "How to Make a Measurement" on page 15.
   - By entering the startup wizard at startup or by selecting **New** from the **File** menu, the software sends you to the Startup Wizard's *Setup & Calibration Complete!* window where you can make a measurement.
   - By selecting the system name from the **Utilities** menu and then selecting **Calibrate**..., the software returns to the main software window.

Performing an LRM Calibration

You have selected the 4-Channel LRM (LINE - REFLECT - MATCH) calibration type. LRM calibration is convenient in that calibration standards can be fabricated for a specific measurement environment, such as a transistor test fixture or microstrip. Microstrip devices cannot be connected directly to the coaxial ports of the analyzer. The device under test (DUT) must be physically connected to the PLTS by some kind of transition network or fixture. Calibration for a fixtured measurement in microstrip presents additional difficulties.

A calibration at the coaxial ports of the PLTS removes the effects of the network analyzer, test set, and any cables or adapters before the fixture; however, the effects of the fixture itself are not accounted for. An in-fixture calibration is preferable, but high-quality SHORT - OPEN - LOAD - THRU (SOLT) standards may not be readily available to allow a calibration of the system at the desired measurement plane of the device. In microstrip, a short circuit is inductive, an open circuit radiates energy, and a high-quality purely resistive load is difficult to produce over a broad frequency range. The 4-Channel LRM calibration is an alternative to the traditional SOLT calibration technique that utilizes simpler, more convenient standards for device measurements in the microstrip environment.

For coaxial, waveguide and other environments where high-quality impedance standards are readily available, the traditional SOLT method provides the most accurate results since all of the significant systematic errors are reduced.

In all measurement environments, you must provide calibration standards for the desired calibration to be performed. The advantage of LRM is that only three standards need to be characterized as opposed to four standards in the traditional SOLT calibration. Further, the requirements for characterizing the L, R, and M standards are less stringent and the standards are more easily fabricated.

Unless you have defined an LRM calibration kit previously, you will need to define an LRM calibration kit before proceeding. Refer to “Defining an LRM Calibration Kit”. Select **Launch Startup Wizard** from the **Tools** menu after you complete the definition to return to the calibration and measurement process.

The LRM calibration kit contains the following:

- zero length LINE
- “flush” short for the REFLECT standard (0 second offset)
- 50-ohm termination of the MATCH (infinite length line)

NOTE LRM with a zero length line is sometimes referred to as TRM (THRU - REFLECT - MATCH).

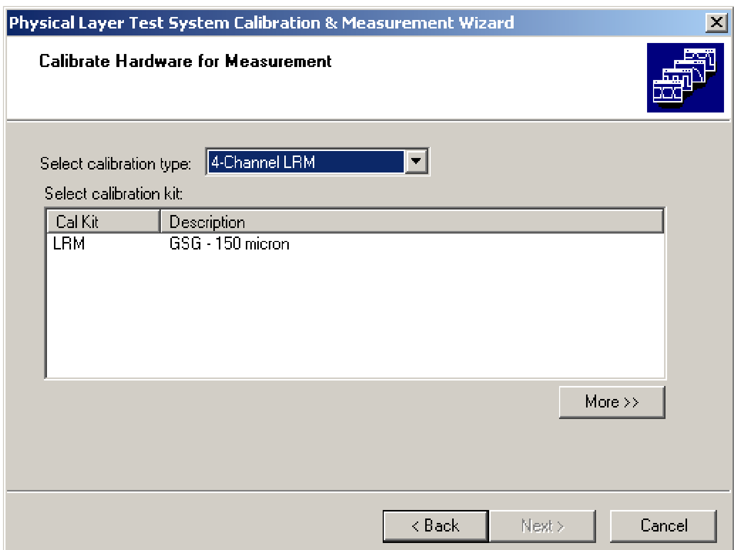
NOTE LRM calibration is not supported for physical layer test systems that use the 8753ES or the 872XES network analyzers.

Start with "Selecting an LRM Calibration Kit".

Selecting an LRM Calibration Kit

The dialog box shown in Figure 2-37 lists the defined LRM calibration kits that have been determined by the auto-detection to be appropriate for the system hardware and the frequency range that was defined in the measurement parameters.

Figure 2-37 Select LRM Calibration Kit Dialog Box



NOTE No LRM calibration kits are defined in the software when you receive it.

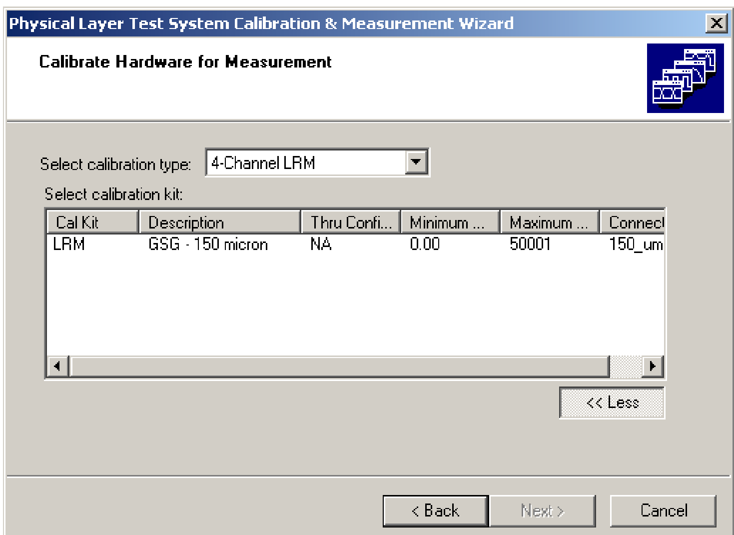
- If you *have not* defined your LRM calibration kit, select the **Cancel** button and define your LRM calibration kit using the procedure described in "Defining an LRM Calibration Kit" on page 68.
- If you *have* defined a LRM calibration kit but it is not displayed in the dialog box:

1. Make sure that you have selected **4-Channel LRM** from the **Select Calibration Type** list. See “Selecting a Calibration Type” on page 29.
2. Make sure that the start and stop frequencies that were defined for the LRM calibration kit (“Defining an LRM Calibration Kit” on page 68) match or exceed the start and stop frequencies that were defined in the measurement parameters (“Setting Up the Calibration and Measurement Parameters” on page 9).

---

1. Select your calibration kit in the **Cal Kit** column.
2. Select the **More >>** button to display the thru calibration configuration, the minimum and maximum frequencies of the kit, and the connector type of the four ports. See Figure 2-38.

Figure 2-38 Select Calibration Kit Dialog Box with More Button Selected



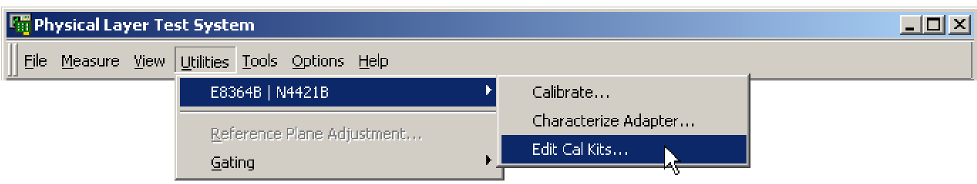
3. Once the calibration kit is selected in the **Cal Kit** column, click the **Next >** button to display the calibration window and continue with “Performing an LRM Calibration” on page 71.

Defining an LRM Calibration Kit

The dialog box shown in Figure 2-40 shows only the **USER_DEF** selection. This indicates that a LRM calibration kit must be defined before a LRM calibration may be performed.

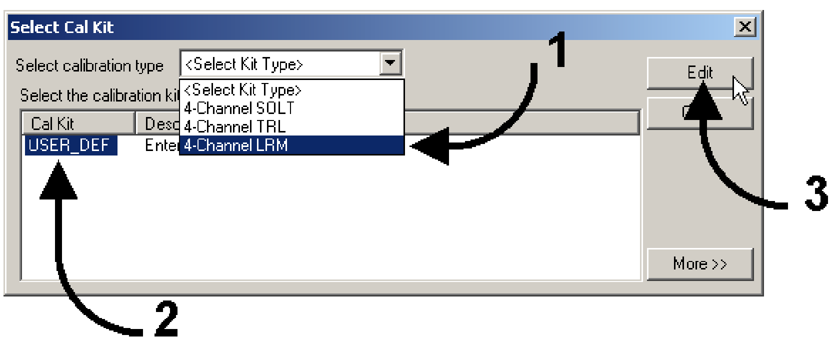
1. From the **Utilities** menu, select your PLTS system name and then select **Edit Cal Kits**... as shown in Figure 2-39.

Figure 2-39 Selecting Edit Cal Kits...



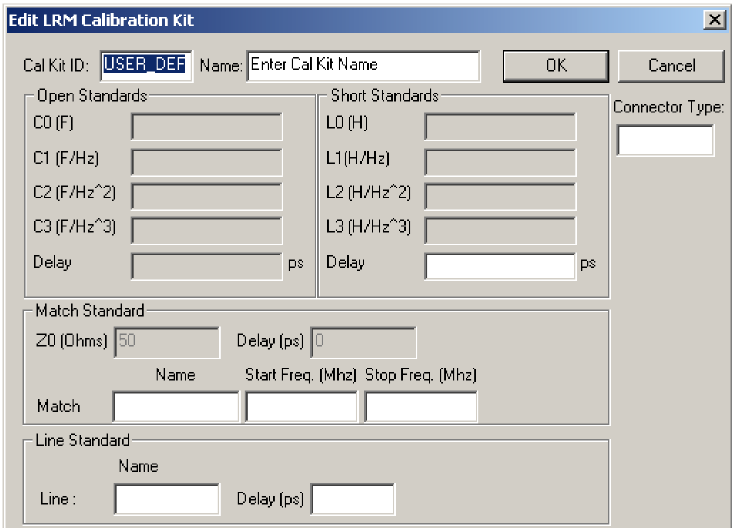
2. In the *Select Cal Kit* dialog box, select **4-Channel LRM** from the **Select Calibration Type** list, then select the **USER_DEF** selection and click **Edit** to start the definition process. See Figure 2-40.

Figure 2-40 Select Cal Kit Dialog Box



3. The LRM calibration kit is defined using the *Edit LRM Calibration Kit* dialog box. See Figure 2-41.

Figure 2-41 Blank Edit LRM Calibration Kit Dialog Box



4. Enter **Cal Kit ID**, **Name**, and **Connector Type** into the appropriate entry boxes in the header. Referring to Figure 2-42, **Cal Kit ID** is listed in the **Cal Kit** column and **Name** is listed in the **Description** column of the calibration kit selection window.

   In this example, **Cal Kit ID** is defined as “`LRM`”, **Name** is defined as “`GSG - 150 micron`”, and **Connector Type** is defined as “`150 um`”. Refer to Figure 2-38.

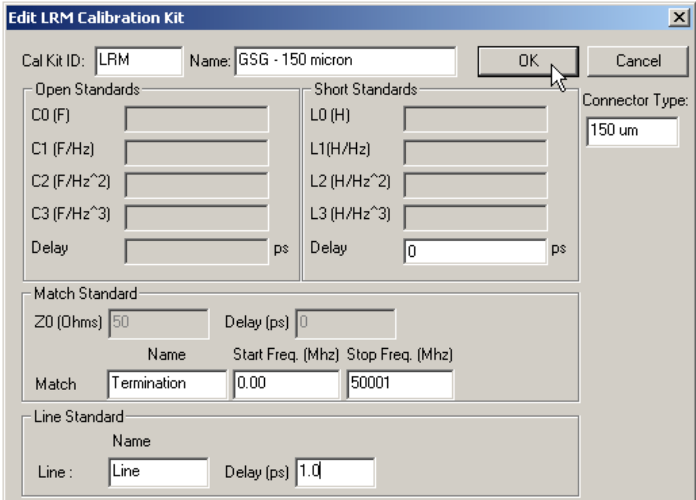
5. For the Reflect: Enter the delay value (in picoseconds) of the short that you are using in the **Short Coefficients** area. Figure 2-42 shows the data that is used in this example.

6. For the Match: Complete the **Load Standards** area by entering the characteristic impedance (**Z0**) in ohms and **Delay** in picoseconds. Also enter a **Load Name** for the Match as well as the start and stop frequencies in MHz of the load.

   In this example, **Z0** is defined as “`500`” ohms, **Delay** is defined as “`-0.0034`” ps, the **Load Name** is defined as “`Termination`”, **Start Freq.** is defined as “`0.00`” MHz, and **Stop Freq.** is defined as “`50001`” MHz. Figure 2-38 shows the data that is used in this example.

7. For the Line: In the **Enter Cal Thru Characterization Data** area, enter a name for the **Thru** along with its delay (in ps). In this example, the **Thru Name** is defined as “`Line`” and the **Delay** is defined as “`1.0`” ps. Figure 2-42 shows the data that is used in this example.

Figure 2-42 Completed Edit LRM Calibration Kit Dialog Box

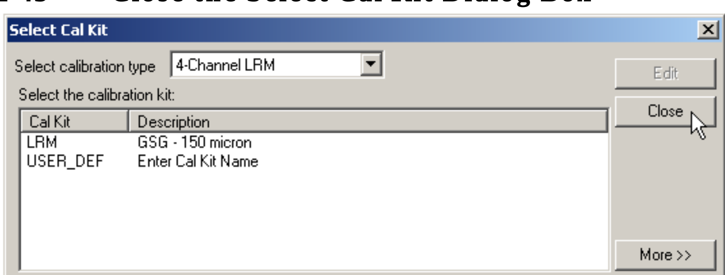


8. Once the LRM calibration kit data has been entered into the dialog box, select the **OK** button to save the calibration kit data and exit the *Edit LRM Calibration Kit* dialog box.

   The **Cancel** button closes the *Edit LRM Calibration Kit* dialog box without saving the calibration kit data.

9. Once the LRM calibration kit data has been saved, select **Close** to close the *Select Cal Kit* dialog box. See Figure 2-43.

Figure 2-43 Close the Select Cal Kit Dialog Box



10. Return to the startup wizard to complete the LRM calibration procedure. You can select **New** from the **File** menu to restart the wizard.

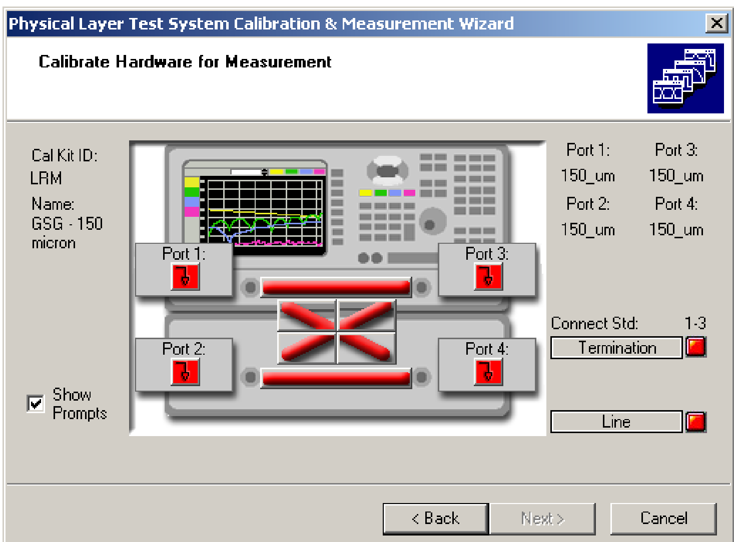
- If you need help completing the startup wizard, return to Chapter 1 on page 3.
- If you do not need assistance, continue with "Selecting an LRM Calibration Kit" on page 66.

Performing an LRM Calibration

NOTE For maximum accuracy and repeatability, the system (network analyzer and test set) should be stabilized at room temperature for a minimum of 24 hours before calibration.

As indicated by its name, the LINE - REFLECT - MATCH (LRM) calibration consists of three portions. First, the Reflect portion is performed by connecting a short to each system port. The Line portion is performed by connecting a thru between system ports 1 and 3 and then between ports 2 and 4. Finally, the Match portion is performed by connecting a load between ports 1 and 3, ports 2 and 4, ports 1 and 4, and ports 2 and 3.

Figure 2-44 Initial LRM Calibration Display



The LRM calibration is performed in two parts. First the Reflect portion of the calibration is done. Then the Line-Match portion of the calibration is done.

Reflect Portion of LRM Calibration The following three steps are the Reflect portion of the LRM calibration.

1. Select the following icon to start the Reflection portion of the calibration: 

2. As described in the prompt shown in Figure 2-45, connect the reflective device (in this case, defined as a short) from your calibration kit to Port 1 and click **OK**.

Figure 2-45 Displaying the Port 1 Reflection Prompt

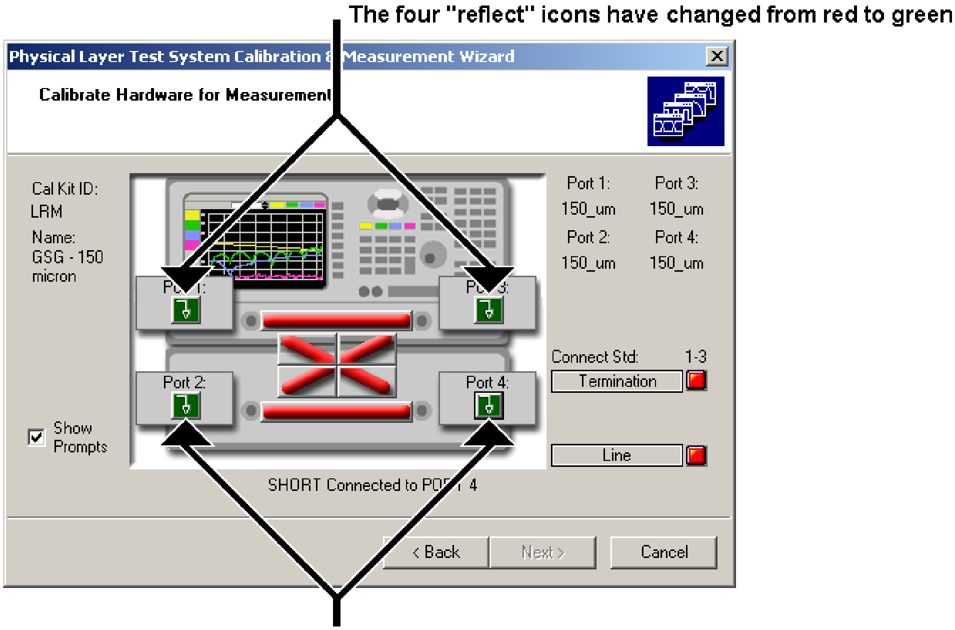
As the portion of the calibration is complete, the Port 1 icon color changes from red to green. Actually, as the measurement is being made, the icon is blue, however, the measurement is so quick you may not even see the icon turn blue before turning to green.

3. Repeat steps 1 and 2 for each of the remaining ports.

   The order that the ports are calibrated does not matter.

   Once all four of the ports are calibrated using the Reflect device, the color of each reflect icon has changed to green. See Figure 2-46.

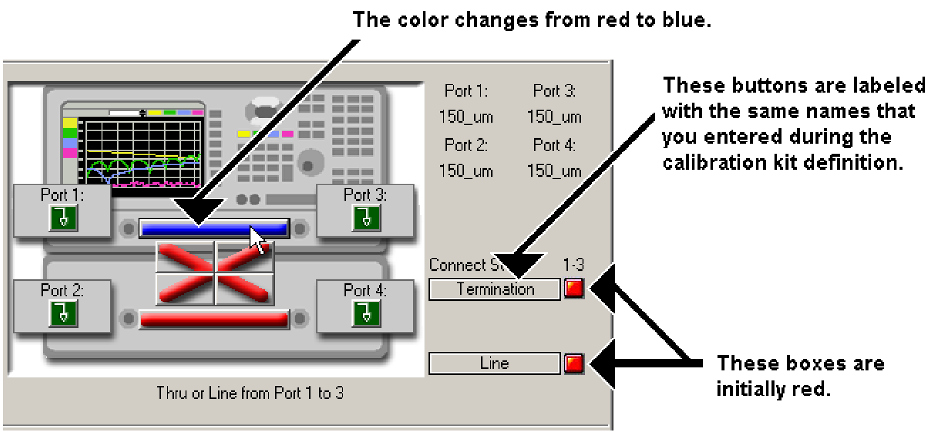
Figure 2-46 Reflection Portion of Calibration Complete



Line-Match Portion of LRM Calibration The following steps are the Line-Match portion of the LRM calibration.

1. Click the Line-Match icon that extends from Port 1 to Port 3. When this icon is clicked, the color changes from red to blue. See Figure 2-47.

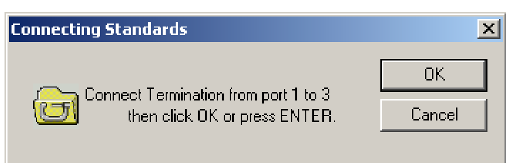
Figure 2-47 Start Thru - Line Calibration Port 1 - Port 3



2. Click the top button on the right side of the display. In this case, the button is called **Termination** because the LRM calibration kit was defined with that label for **Load Name** (this is the name of the match). See Figure 2-42 on page 70. If you would have entered another name for the load, that label would be used.

   When you click the **Termination** button, the following prompt window is displayed:

Figure 2-48 Prompt for Load (Termination) Calibration



3. Make the connections as described in the prompt window and click **OK** to start the calibration measurement.

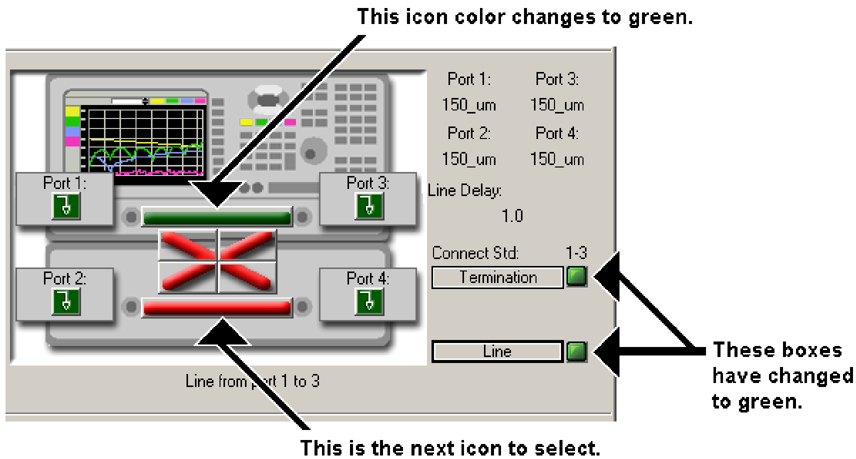
   Once this measurement is made, the color of the box at the right of the **Termination** button changes from red to green.

4. Repeat step 2 and 3 for the thru (**Line**).

Once these measurements are taken, the calibration between ports 1 and 3 is complete. See Figure 2-49. On the display, the color of:

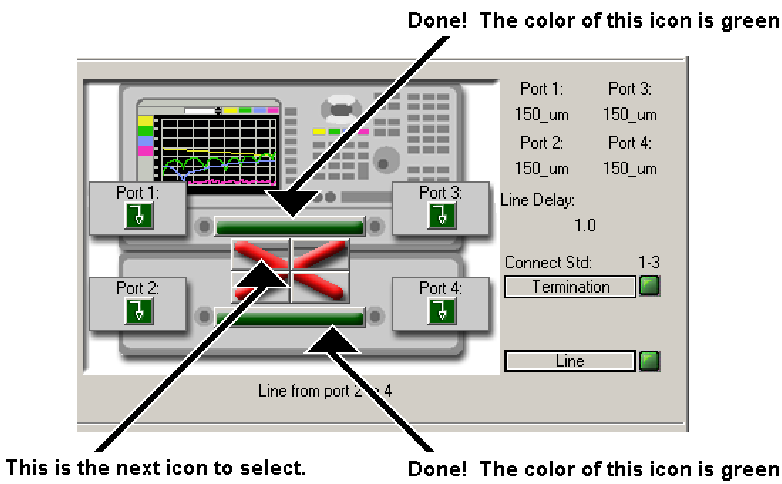
- Each box to the right of the buttons have changed from red to green.
- The line between ports 1 and 3 changes from blue to green indicating that this path is complete.

Figure 2-49 Port 1 - Port 3 Line Portion of Calibration Complete



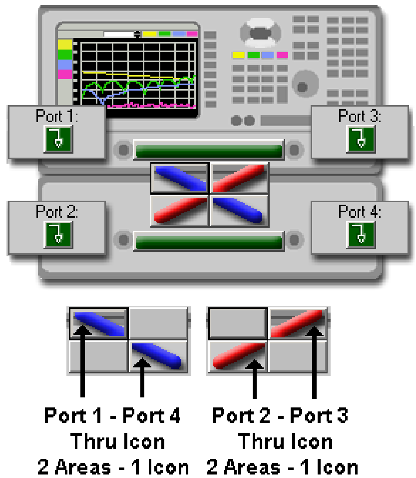
5. Repeat steps 1 through 4 for the Port 2 to Port 4 path.

Figure 2-50 Line Portion of Calibration Complete



6. Select the Line icon that extends from Port 1 to Port 4.

Figure 2-51 Thru Icons for Port 1 - Port 4 and Port 2 - Port 3



The icon (both halves) turns blue and the box at the right of the **Line** button turns red.

7. Click the **Line** button which displays the prompt window. (The **Termination** button is inactive.)

8. After the **Line** (thru) is connected between ports 1 and 4 as indicated on the prompt and click the **OK** button.

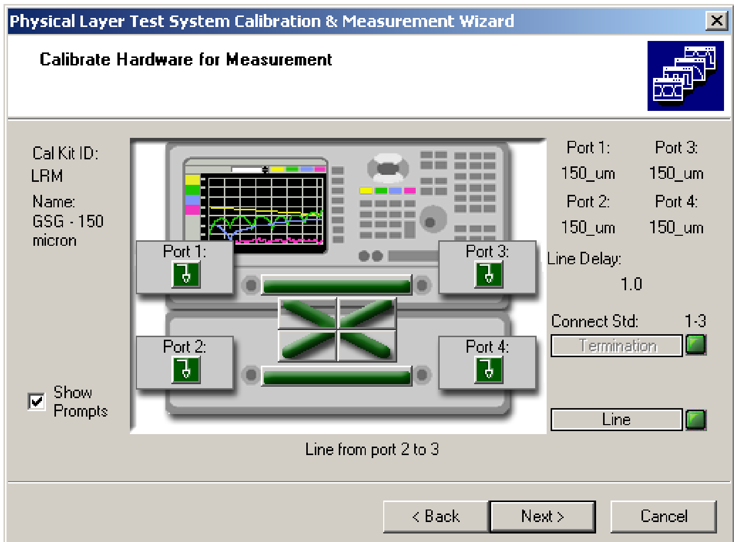
   The calibration is performed and the color of the box to the right of the thru button and the thru icon are changed to green.

9. Repeat steps 6 through 8 to complete the Line calibration between Port 2 and Port 3.

10. Once the Port 2 to Port 3 Line calibration is made, the LRM calibration measurements are complete. Refer to Figure 2-52.

    The color of all icons has changed to green.

Figure 2-52 LRM Calibration Complete

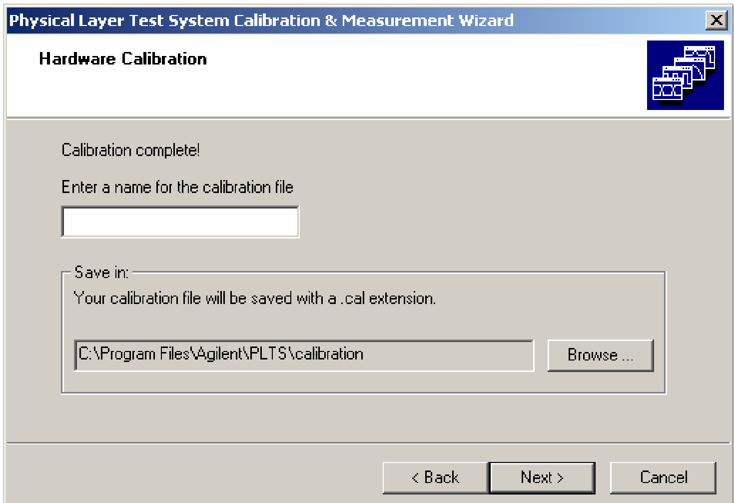


11. Click the **Next >** button.

    Continue at “To Save the LRM Calibration” on page 78.

To Save the LRM Calibration Once you have completed the LRM calibration, the *Save Calibration Data* dialog box is displayed as shown in Figure 2-53.

Figure 2-53 Save Calibration Data Dialog Box



1. Enter a file name for the calibration data set.

   The calibration data file will be saved in the directory displayed in the **Calibration Data File Path** box. You may change the directory by entering the directory path in the box or selecting the **Browse**... button and navigating through your computer's directory structure to the desired directory. The calibration data is saved with a ".cal" file extension.

2. Save the calibration data by selecting the **Next >** button.

   When the calibration data is saved, the calibration is complete. A window is displayed showing how to make the connections to measure your DUT following the LRM calibration.

3. If you started the calibration:

   - As part of the example measurement of Chapter 1, return to "How to Make a Measurement" on page 15.
   - By entering the startup wizard at startup or by selecting **New** from the **File** menu, the software sends you to the Startup Wizard's *Setup & Calibration Complete!* window where you can make a measurement.
   - By selecting your PLTS system name from the **Utilities** menu and then selecting **Calibrate**..., the software returns to the main software window.

Characterizing Adapters

For non-insertable[1] calibrations, the thru adapter to be used must be characterized by itself so that its effects can later be removed from the calibration measurement results.

Several standard thru adapter characterization files have been provided with PLTS (see the ..\PLTS\adapters directory) and are automatically selected based on the calibration kit to be used. These files may be used with very good results, but for the very highest accuracy, it is recommended that you characterize your own adapters using the following procedure.

To perform the characterization, first, a short/open/load calibration is performed directly at the front panel of the system. Either one or two test ports will be used depending on the adapter category selected during the process. Then the adapter is inserted and the short/open/load calibration is repeated. The resulting adapter S-parameters are saved in Citifile format, which can later be de-embedded from the device measurement.

For adapters that will be used for broadband measurements, characterize the adapter over the entire frequency range of the instrument with as many points as possible. This allows for interpolation of adapter data if the frequency points used in a later DUT measurement are not exactly the same as the adapter frequency points.

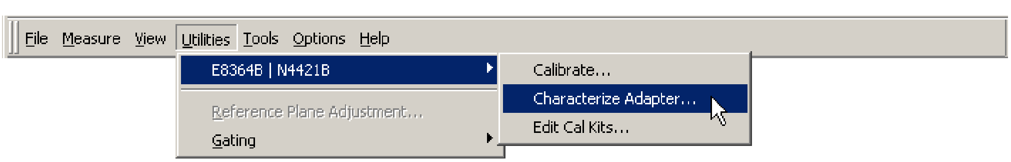
Adapter characterization is performed directly at the instruments front panel (either Port 1 or Ports 1 and 4). If an additional adapter is needed between the front panel and the adapter to be characterized (for type or gender change), install the additional adapter first (metrology grade recommended), and perform all calibrations with it installed.

For purposes of characterization, your adapters must have an orientation. Mark the connectors on the adapter as ports 1 and 2, and treat them as such during the characterization procedure. Forward orientation has the lower-numbered adapter port connected to the test-set port.

Follow these steps to characterize your adapters:

1. Select **Characterize Adapter** from the **Utilities** menu to start the *Custom Adapter Characterization Wizard*.

Figure 2-54 Characterize Adapter Selection



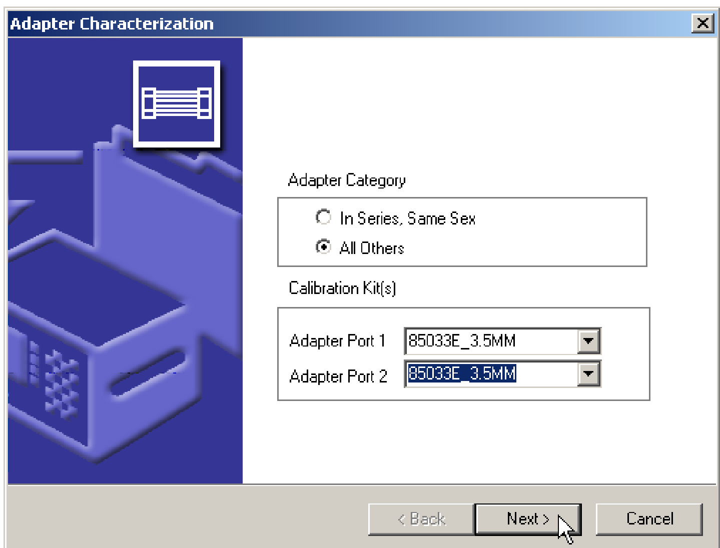
1. Non-insertable calibrations are those in which the test port connectors are of the same gender (male-to-male or female-to-female).

2. In the *Custom Adapter Characterization Wizard* dialog box, select the appropriate adapter category and the calibration kit to be used. See Figure 2-55. Press **Next >** to continue to the next window.

   The two adapter characterization categories are:

   - **“In-Series, Same-Sex”** uses a single port (Port 1) and applies only to adapters having the same type and gender on both ends (3.5 mm male-to-male, for example). Select the calibration kit to be used on Port 1 from the pull-down menu.
   - **“All Others”** uses two ports (Ports 1 and 2) and allows the adapter types to be specified independently. Select the calibration kits to be used on Port 1 and on Port 2 from the pull-down menus.

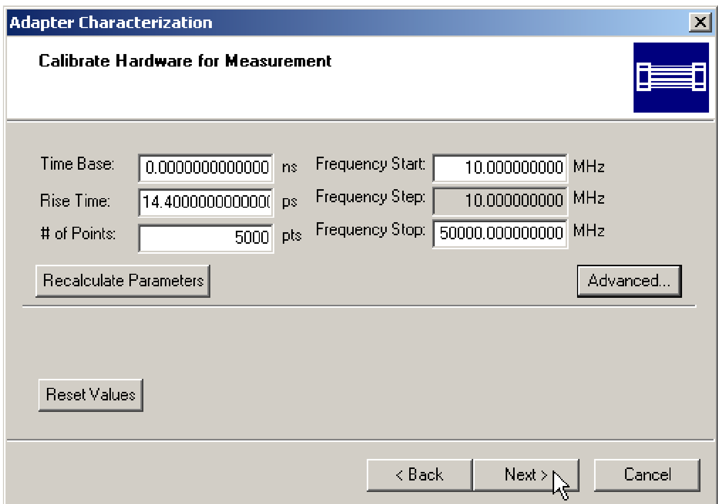
Figure 2-55 Custom Adapter Characterization Wizard



3. Set up the stimulus to measure the adapter in the *Adapter Characterization* wizard's *Calibrate Hardware for Measurement* window. See Figure 2-56. When you are finished, click the **Next >** button to continue.

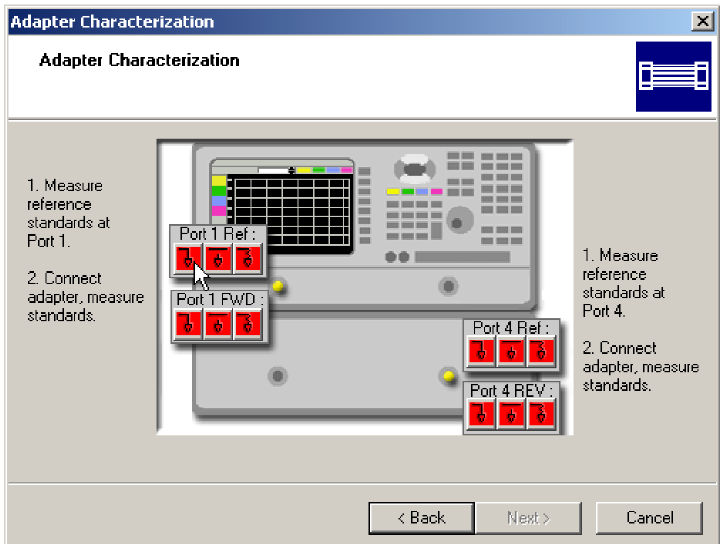
   Refer to "Setting Up the Calibration and Measurement Parameters" on page 9 for information on this window's parameters.

Figure 2-56 Measurement Stimulus Parameter Setup Window

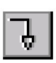


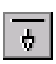
4. Connect one of the standards (Short - Open - Load) to Port 1 on the PLTS and click the icon that represents the standard in the **Port 1 Ref** area. The PLTS will measure the adapter and when it has finished, the icon color changes from red to green.

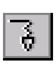
Figure 2-57 Measuring the Standards (Short - Open - Load) on Port 1



Each of the three icons represents a different calibration standard.

 represents the short standard

 represents the open standard

 represents the 50-ohm load standard

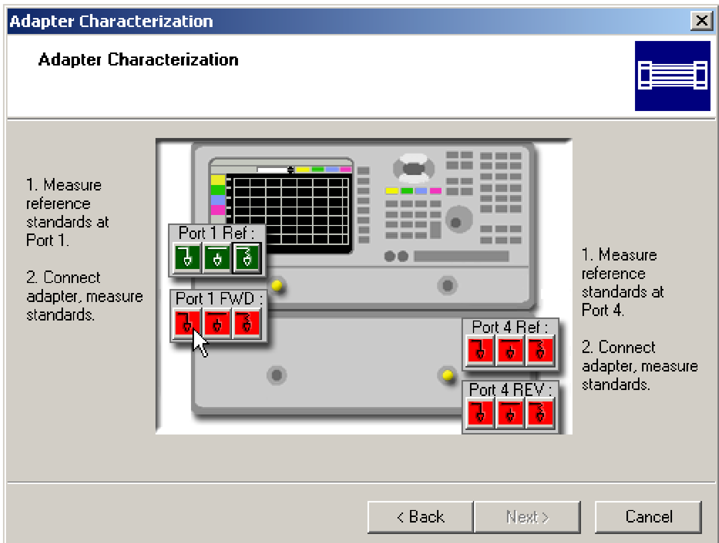
5. Repeat step 4 for the two remaining icons. The color of all three icons in the **Port 1 Ref** area have changed from red to green.

6. Connect the adapter that is being characterized to Port 1.

   Mark the end that is connected to Port 1. The opposite end will be connected to Port 4 later in this procedure.

7. Repeat step 4 and step 5 connecting to the standards to the adapter. Click the icons in the **Port 1 FWD** area to make the measurements (Figure 2-58). As before, as each standard is measured, the color of the icon changes from red to green. When you have finished, the color of all three icons in the **Port 1 FWD** should have changed from red to green.

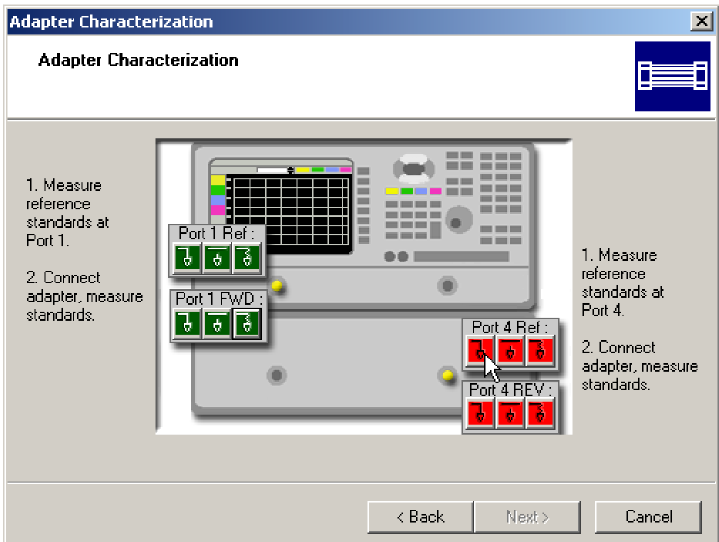
Figure 2-58 Remeasuring the Standards with the Adapter on Port 1



8. When the standards have been measured on the adapter at Port 1 and all of the Port 1 standard icon colors have changed from red to green, repeat step 4 through step 7 on Port 4.

   As mentioned in step 6, remember to connect opposite end of the adapter to Port 4 (the adapter end that you did *not* connect to Port 1).

Figure 2-59 Measuring the Standards/Adapter on Port 4

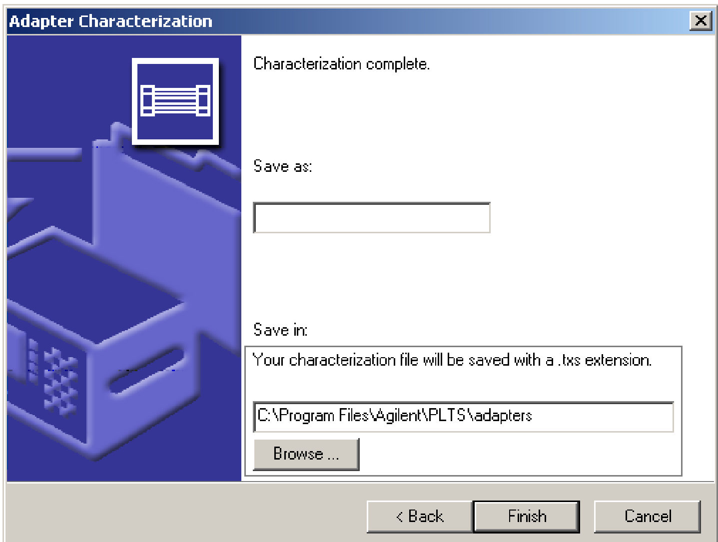


9. When you have finished, click the **Next >** button to continue.

   All 12 standard icon colors have changed from red to green.

10. When the characterization is complete, save the adapter file using the dialog box shown in Figure 2-60. Enter a file name for the adapter data and click **Finish**. The file is automatically saved as a ".txs" file. The default directory for saving adapter characterization information is C:\Program Files\Agilent\PLTS\adapters, where C is the hard drive where the PLTS is stored.

Figure 2-60 Save Adapter Characterization File Dialog Box



11. Since the characterization file is in Citifile format, you can import the data and make a visual check of the quality of the characterization. Select **Import** from the **File** menu and then select **Citifile**. You may open and inspect the adapter characterization file.

3 Setting Up, Calibrating, and Making Measurements using the TDR-Based PLTS

This chapter guides you through setting up the initial measurement definitions, performing the system calibration, and making a measurement with the TDR-based PLTS hardware. PLTS supports the TDR-based PLTS hardware using coaxial test cables.

You will need four coaxial test cables for the TDR-based PLTS measurements.

- For the Agilent 86100A/B/C, we recommend you use:
  - Four 3.5 mm (male-female) cables (such as the Agilent N4419A-B20)
    -or-
  - Four 3.5 mm (male-male) cables (Agilent part number 8120-4948) with a 3.5 mm (female-female) adapter used as a connector saver
- For the Tektronix CSA8000 and TDS8000, we recommend you use:
  - Four 3.5 mm (male-male) cables (Agilent part number 8120-4948)
    -or-
  - Four 3.5 mm (male-female) cables (such as the Agilent N4419A-B20) with a 3.5 mm (male-male) adapter (Agilent part number 1250-1748)

You also need the following calibration standards to perform the TDR calibration. As Table 3-1 shows, different standards are required depending on the TDR-based PLTS hardware and the desired calibration type.

Table 3-1 Required 3.5mm (SMA) Calibration Standards[a]

TDR System	Module Calibration	Calibration For Single-Ended Measurements		Calibration for Differential Measurements	
		Reference Plane Cal Only	Normalization plus Reference Plane Cal	Reference Plane Cal Only	Normalization plus Reference Plane Cal
Agilent 86100A/B/C	1 50Ω Load	1 Thru	1 50Ω Load 1 Short 1 Thru	2 Thrus	2 50W Loads 2 Shorts 2 Thrus
Tektronix CSA8000/ TDS8000	2 50Ω Loads	1 Thru	Not Applicable	2 Thrus	Not Applicable

a. **Recommended Agilent Part Numbers:** Loads, 3.5mm (f), 1250-2151; 3.5 mm (m) 1810-0118
Shorts, 3.5mm (f), 1250-2152; 3.5mm (m) 1250-2153
Thrus, 3.5mm (f-f), 5061-5311;

Agilent Probe Model Numbers:

- N1020A 6 GHz Time Domain Reflectometry Probes (Set of 4)
- N1020A-K05 Calibration Substrate for use with N1020A Probes

CAUTION **Avoiding ESD Damage to TDR Plug-In Modules**

The input connectors are very sensitive to electrostatic discharge (ESD). When you connect a device or cable that is not fully discharged to the input connector, you risk damage to the module and expensive instrument repairs. Refer to your TDR documentation for detailed information regarding ESD susceptibility.

The **Startup Wizard** steps you through the following steps to make a measurement.

1. **Initial setup** includes:
    - System Hardware Verification and Calibration Level Selection
    - TDR Calibration and Measurement Parameter Setup
    - Calibration and Measurement Parameter Selection
2. **Calibration** includes:
    - Calibration Kit Selection
    - Calibration Kit Definition
      (if you need to use other than one of the predefined calibration kits)
    - Calibration, which includes:
      — Module Calibration (may be required or may be recommended)
      — Reference Plane Calibration Only -or- Normalization and Reference Plane Calibration
3. **Measurement** includes:
    - Device Under Test Connection
    - Initial Analysis Type Selection
    - Stimulus and Parameter Verification
    - Running a Measurement

After the measurement is made, the main Physical Layer Test System (PLTS) window gives you the flexibility to perform analysis on the measured data in a variety of ways.

- **Time Domain.** The Time Domain analysis may be analyzed in Differential or Single-Ended mode.
- **Frequency Domain.** The Frequency Domain analysis may be analyzed in Balanced or Single-Ended mode.
- **Eye Diagram.** The Eye Diagram analysis may be analyzed in Differential or Single-Ended mode.
- **Transmission Line (RLCG).** The RLCG analysis may be analyzed in Differential, Common, W-Element, and Self/Mutual modes.

Starting the Startup Wizard

The Startup Wizard is displayed when the PLTS application is started. The *Startup Wizard Welcome* screen is displayed in Figure 3-1. It is also displayed anytime a new measurement is initiated, such as after selecting **New** from the **File** menu, or clicking on the new file icon on the toolbar.

Figure 3-1 Startup Wizard Welcome Screen

The Startup Wizard Welcome Screen gives you two choices, **New Measurement** or **Load Measurement**. The selected choice has a green check mark to the right of the label. **New Measurement** leads you through the process of calibrating and making a measurement. **Load Measurement** loads measurement data from an existing file that you must select.

When you select **New Measurement**, the Startup Wizard will guide you through:

- Initial Setup
- Calibration
- Measurement

For this exercise, select **New Measurement** and then select the **Next >** button.

Performing the Initial Setup

The Initial Setup process includes:

1. Verifying that the software recognizes your PLTS system hardware.

   This step uses the left portion of the *Hardware Auto-Detection Summary* dialog box to verify that the software recognizes the correct system hardware. This portion of the dialog box displays the model number, GPIB address, and serial number of the hardware. You may re-scan to look for hardware changes automatically. You may select another recognized PLTS measurement system.

2. Selecting the appropriate level of calibration for the upcoming measurement.

   This step uses the right portion of the *Hardware Auto-Detection Summary* dialog box to select the appropriate level of calibration to be performed. You may perform a new calibration, reuse existing calibration data, or perform measurements without calibration.

3. Setting up the TDR Calibration and Measurement Settings.

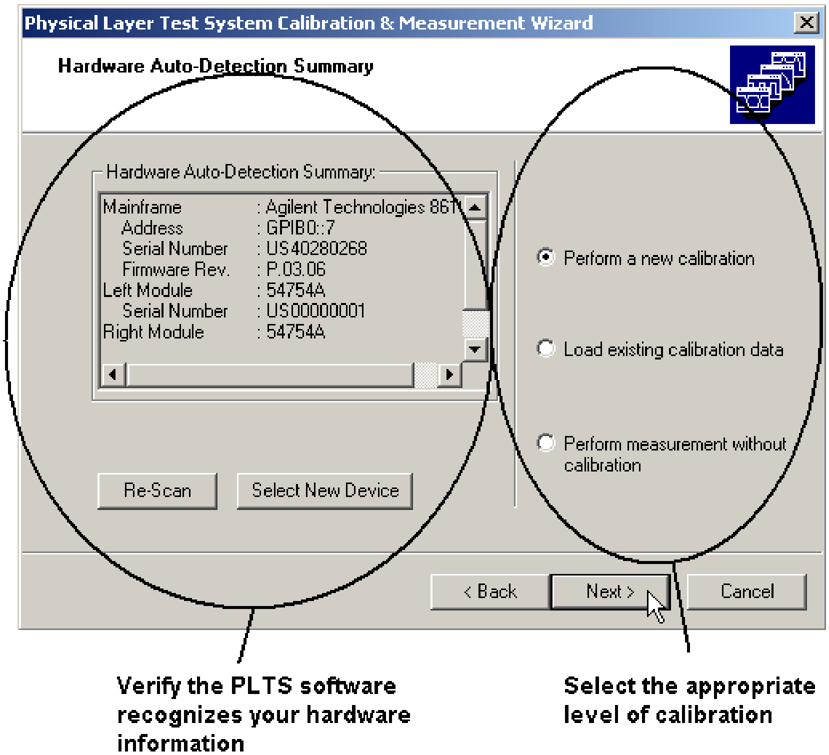
   This step uses the *Setup TDR* dialog box to display the default TDR calibration and measurement settings, based on the PLTS hardware recognized by the software. You may modify these default TDR calibration and measurement settings as required.

4. Selecting the calibration and measurement parameters.

   This step uses the *Select Calibration and Measurement Parameters* dialog box to allow you to select the parameters that will be measured. A calibration will be performed only where required for selected parameters.

The *Hardware Auto-Detection Summary* dialog box (see Figure 3-2) is displayed when you select **New Measurement** and then click the **Next >** button described on page 90.

Figure 3-2 Hardware Auto-Detection Summary Dialog Box



To Verify the Software Recognizes the PLTS Hardware

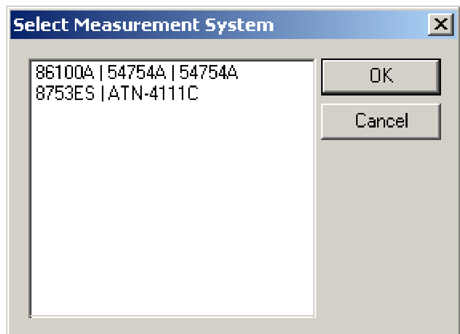
On the left portion of the *Hardware Auto-Detection Summary* dialog box, the model number, the GPIB address, the serial number, and the firmware revision for the TDR mainframe is displayed. In addition, the model numbers and serial numbers for the left side and right side modules are also displayed. This is the test equipment that the software finds connected to the GPIB bus.

If this information is incorrect, check your hardware connections and power status, then select the **Re-Scan** button. The software re-checks for the presence of hardware on the GPIB.

NOTE

If you have multiple PLTS systems on the GPIB and you want to select the other equipment, click the **Select New Device** button. The *Select Measurement System* dialog box is displayed. See Figure 3-3. To choose another PLTS system, you may select from the available PLTS systems and then click the **OK** button. The software makes the change and returns to the *Startup Wizard Welcome Screen* of Figure 3-1.

Figure 3-3 Select Measurement System Dialog Box



The **Cancel** button exits without saving a change and returns to the *Startup Wizard Welcome Screen*.

To Select the Appropriate Level of Calibration

On the right portion of the *Hardware Auto-Detection Summary* dialog box (Figure 3-2), select your calibration strategy:

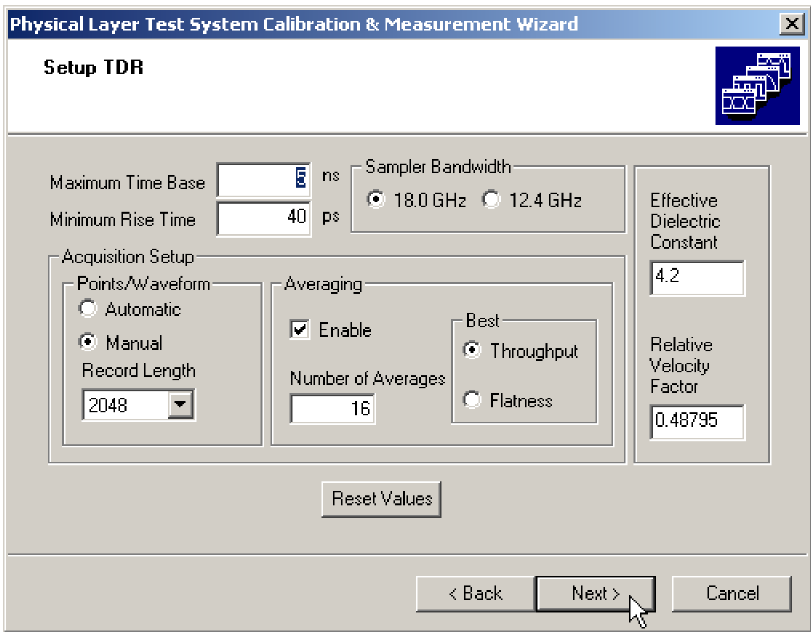
- **Perform a new calibration** continues with the Startup Wizard performing a new calibration before making a measurement.
- **Load existing calibration data** allows you to select a previously saved calibration (*.cal) file to be used for the new measurement. See "To Perform a Module Calibration" on page 109 for guidance on calibration intervals, etc.
- **None. Collect measurement data uncalibrated** allows you skip the calibration, select measurement parameters (see Figure 3-4 on page 94), and then proceed directly to the measurement screen. This option is ***not recommended*** for qualitative data collection.

Select the **Perform a new calibration** choice for this example exercise and then click the **Next >** button.

To Set Up the TDR

The *Setup TDR* wizard screen is displayed. See Figure 3-4. This screen allows you to set the TDR parameters for the calibration and measurement. Each of these parameters is described in Table 3-2, "TDR Parameter Descriptions."

Figure 3-4 Setup TDR Dialog Box



This wizard screen is initially displayed with default values for each entry. You may accept or modify these values. Modifying some of these values directly may also modify other associated values interactively. For example, when changing the **Effective Dielectric Constant**, the **Relative Velocity Factor** will change appropriately.

The **Reset Values** button returns all selections and values to their default value.

When the *Setup TDR* dialog box values are correct, click the **Next >** button to continue.

Table 3-2 TDR Parameter Descriptions

Parameter	Description
Maximum Time Base	Sets the desired measurement range in nanoseconds.The default setting is 5 ns. Typically the time base should be set to 2.5–3 times the electrical length of the DUT. Note: reflection measurements are “round trip”.
Minimum Rise Time	Sets the rise time of normalized measurements in picoseconds. This is a function of normalization and is available to only the Agilent 86100-based TDR system.
Sampler Bandwidth	Sets the Agilent 86100-based TDR system receiver to either a **18.0 GHz** bandwidth (which gives the highest fidelity and the fastest response time) or a **12.4 GHz** bandwidth (which gives the best sensitivity by reducing the noise). This bandwidth is set at 20 GHz for other TDR systems.
Points/Waveform	Sets the number of points for a waveform. Select **Automatic** or **Manual**. **Automatic** allows the TDR to select the record length for the input waveform. The TDR selects a record length that optimizes the amount of acquired data and the display update rate. This is available to only the Agilent 86100-based TDR system. **Manual** allows you to define a record length from a list of points. **Record Length** allows you to select the number of points from a list. Depending on the TDR system, the available number of points are: – 16, 32, 64, 128, 256, 512, 1024, 2048, or 4096 -or- – 20, 50, 100, 250, 500, 1000, 2000, or 4000.
Averaging	**Enable** turn the trace averaging on and off. **Number of Averages** sets the number of sweeps to be averaged. Increasing the number of averages improves the signal-to-noise ratio at the cost of increased measurement time. **Best Throughput** allows you to view the waveform as it is acquired. The TDR displays any noise and feed through error on the signal. It has a faster measurement time. This is available to only the Agilent 86100-based TDR system. **Best Flatness** turns on the feed through compensation circuit which reduces the amount of feed through error. This is useful when the measurement time base is in the 1µs range or greater. It has a slower measurement time. This is available to only the Agilent 86100-based TDR system.

Table 3-2 TDR Parameter Descriptions

Parameter	Description
Effective Dielectric Constant	Specifies the dielectric constant for your transmission medium. If the DUT is an embedded stripline printed circuit board trace, then the same material is on each side of the DUT and the effective dielectric constant is simply the value of the dielectric material. This device configuration is known as a “homogenous dielectric system”. If the DUT is a microstrip printed circuit board trace with one side exposed to air, then the effective dielectric constant is a combination of air dielectric (1.0) and the dielectric material. This device configuration is known as a “nonhomogenous dielectric system”. If the DUT is a complex channel consisting of PCB traces, connectors, vias and/or cables, then the effective dielectric constant will also be complex.
Relative Velocity Factor	Specifies the relative velocity for your transmission medium. Some velocity factor (VF) examples are: Air VF=1.000, Surface traces VF=0.53146, Buried Traces in ε_r~4.3: VF=0.48795

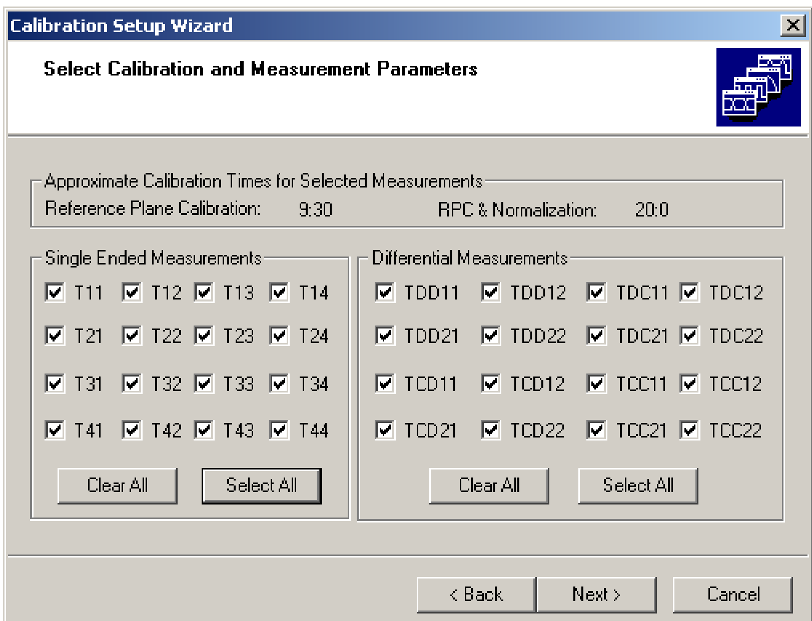
To Select the Calibration and Measurement Parameters

The *Select Calibration and Measurement Parameters* dialog box is displayed. See Figure 3-5. The left side of the dialog box lists all 16 of the available single-ended measurements. The right side of the dialog box lists all 16 of the available differential measurements.

NOTE If you are using only one TDR module, your available measurements will be limited only eight parameters will be available.

For example, if your 54754A TDR module is installed in the left slot, you will only be able to measure four single-ended parameters (T11, T12, T21, and T22) and four differential parameters (TDD11, TDC11, TCD11, and TCC11).

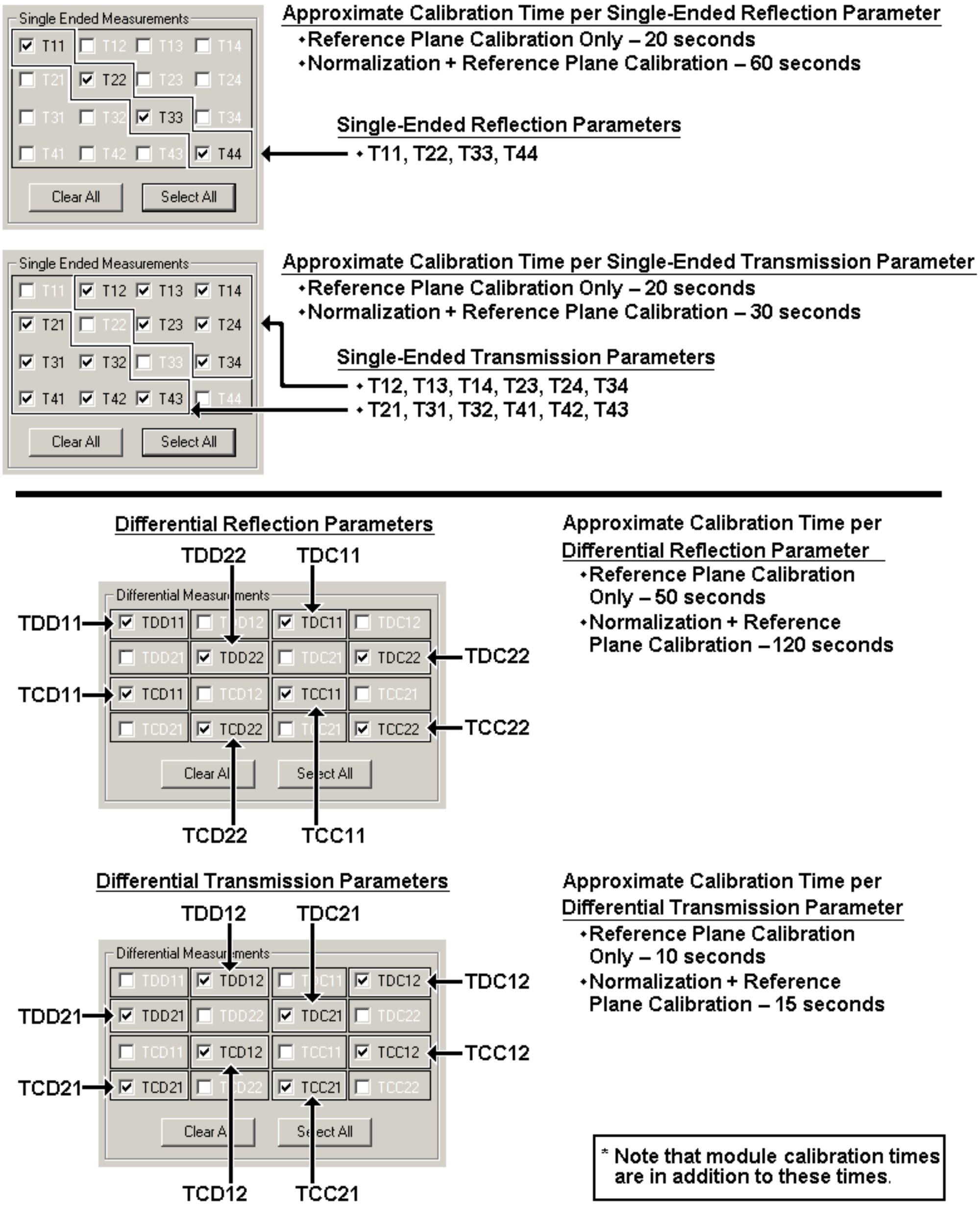
Figure 3-5 Select Calibration and Measurement Parameters Dialog Box



The default status for each of these measurements is selected. When a measurement is selected, it has a check in the check box for the corresponding measurement. However, you may not need or desire data for all 32 parameters (16 single-ended and 16 differential parameters). As a general rule, the more parameters that you select, the longer the calibration and the measurement takes to perform.

Figure 3-6 provides a tool for estimating calibration times for each selected parameter.

Figure 3-6 Approximate Calibration Times for Single-Ended Parameters



Leave all parameters selected for this example exercise and then click the **Next >** button.

Performing a Calibration

The PLTS software adds value and capability to both Agilent and Tektronix TDR systems by automating the calibration process which saves time, reduces the chance of calibration errors, eliminates tedious procedures, and improves the calibration results. The PLTS calibration process includes:

1. Selecting a calibration kit and defining the calibration file name.

   This step uses the *Select Calibration Kit and Filename* dialog box to select the calibration standards and connectors that are used during the calibration process. The file name that the calibration information is saved to is also defined.

2. Defining a calibration kit.

   This optional step allows you to define a calibration kit if none of the pre-defined calibration kits match your calibration needs.

3. Choosing the calibration type.

   This step uses the *Calibrate TDR and Modules* wizard window to define the type of TDR calibration that will be performed, either a “Reference Plane Calibration Only” or a “Normalization and Reference Plane Calibration”.

4. Performing the calibration for the TDR modules.

   This step uses the *Calibrate TDR and Modules* wizard window to perform the calibration for the modules within your system.

5. Setting the reference plane at the end of the cables.

   This step uses the *Calibrate TDR and Modules* wizard window to perform the standards calibrations, either the Reference Plane Calibration Only or the Normalization and Reference Plane Calibration.

Once the **Next >** button in the *Select Calibration and Measurement Parameters* dialog box is selected, the *Select Calibration Kit and Filename* wizard screen is displayed. See Figure 3-7.

Start the calibration.

To Select a Calibration Kit and Define the Calibration File Name

The PLTS software has five default TDR calibration kits. It also stores the calibration kit information.

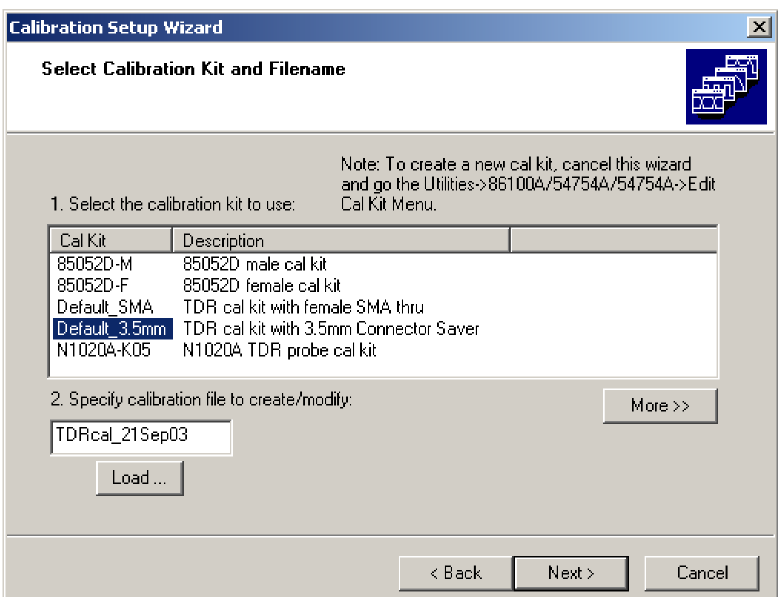
Table 3-3 Pre-Defined TDR Calibration Kits

Name	Description
85052D-M	85052D male cal kit
85052D-F	85052D female cal kit
Default SMA	TDR cal kit with female SMA thru
Default_3.5mm	TDR cal kit with 3.5mm Connector Saver
N1020A-K05	N1020A TDR Probe cal kit

1. Select the calibration kit to use for the TDR calibration from the **Select the calibration kit to use:** list of the *Select Calibration Kit and Filename* wizard screen.

   The PLTS software has several calibration kits already defined for the TDR calibration. All of these calibration kits are listed in the **Select the calibration kit to use:** list. PLTS is set to the **Default 3.5mm** selection.

Figure 3-7 Select Calibration Kit and Filename Wizard Screen



The **Select the calibration kit to use:** list area displays each of the defined calibration kits and its description.

When the **More >>** button is selected, the list area also displays the thru configuration, the minimum frequency, the maximum frequency, and the calibration kit connector type as shown in Figure 3-8. The button label toggles between **More >>** and **<< Less**.

Figure 3-8 Calibration Kit List with More >> Selected

Cal Kit	Description	Thru Confi...	Minimum ...	Maximum ...	Connector...
85052D-M	85052D m...	6 Cross	0	20000	3.5mm_M
85052D-F	85052D fe...	6 Cross	0	20000	3.5mm_F
Default_SMA	TDR cal ki...	6 Cross	0	20000	SMA_F
Default_3.5mm	TDR cal ki...	6 Cross	0	20000	3.5mm_F
N1020A-K05	N1020A T...	6 Cross	0	20000	Probe_Tip

If the calibration kits listed in the **Select the calibration kit to use:** list area do not match your calibration kit, you may define a new calibration kit to meet your requirements. Refer to "To Define a Calibration Kit" on page 103.

2. In the **Specify calibration file to create/modify:** text box of the *Select Calibration Kit and Filename* wizard screen, specify the file that you will use to save the calibration data.

   The text box displays a default calibration file name based on the current date. It uses the string "TDRcal_" followed by the current date. For this example, "TDRcal_21Sep03" is displayed. You may enter another file name if you like. This calibration file name is used as a base name for saving calibration files on the PC and on the Agilent 86100-based TDR (if this TDR is being used).

   - **Files Stored on the PC**

     This calibration file name is used as a base name for saving calibration files on the PC for the TDR. The calibration will be saved to the calibration directory. If you installed PLTS to the default C:/ directory, you may follow this path to the calibration directory: **C:/Program Files/Agilent/PLTS/calibration**

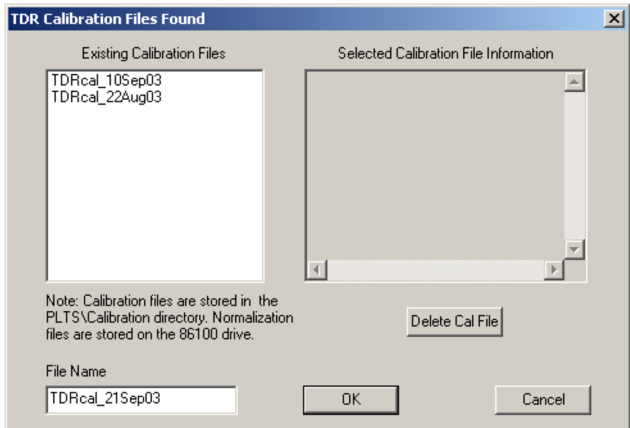
   - **Files Stored on the 86100-based TDR**

     This calibration file name is also used as a base name to store additional calibration and setup files if you are using an Agilent 86100A/B/C TDR. The files stored on the 86100-based TDR are dependent on the calibration and measurement parameters that you selected in the initial setup procedure. For each parameter calibration, a setup file and a calibration file are stored on the 86100 hard drive in the "setups" directory and the "TDR normalization" directory, respectively.

   If you want to overwrite a previously saved calibration file, you may select the **Load**... button to display the *TDR Calibration Files Found* dialog box. This dialog box displays

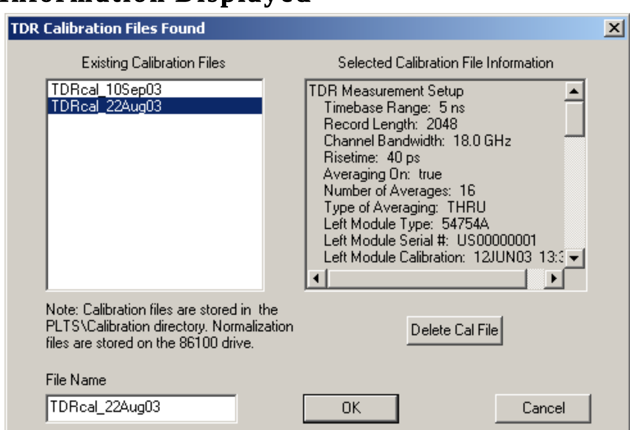
each TDR calibration file that has been saved in the **Existing Calibration Files** list. See Figure 3-9.

Figure 3-9 TDR Calibration Files Found Dialog Box



You may review the calibration data that is saved in a calibration file by selecting a file in the **Existing Calibration Files** list. When a file is selected, its calibration information is displayed in the **Selected Calibration File Information** area as shown in Figure 3-10.

Figure 3-10 TDR Calibration Files Found Dialog Box with Calibration Information Displayed



Calibration files may be deleted by selecting the calibration kit in the **Existing Calibration Files** list and then selecting the **Delete Cal File** button. This also deletes the corresponding normalization files on the 86100-based TDR.

NOTE Normalization files are stored on the Agilent 86100 DCA mainframe drive.

Click the **OK** button to return to the *Select Calibration Kit and Filename* wizard screen.

3. After the calibration kit is selected and the calibration file name is defined, click the **Next >** button to continue with the calibration.

   For this example, the selected calibration kit is set to the **Default 3.5mm** selection and the default calibration file name.

To Define a Calibration Kit

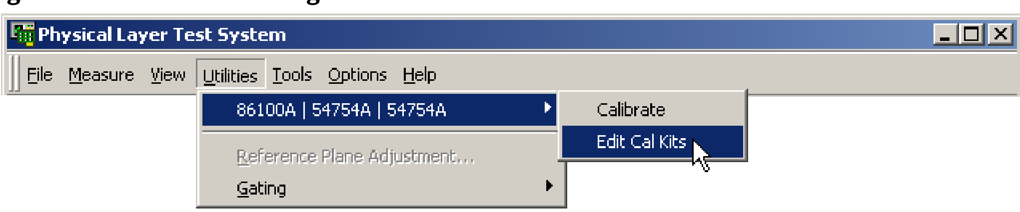
If none of the previously defined calibration kits in the **Select the calibration kit to use:** list meets your requirements, you can define your calibration kit. To define a calibration kit:

1. Exit the startup wizard by clicking the close button in the upper right corner of the wizard:
   

   If you exit the wizard, all previously-defined setup definitions are lost.

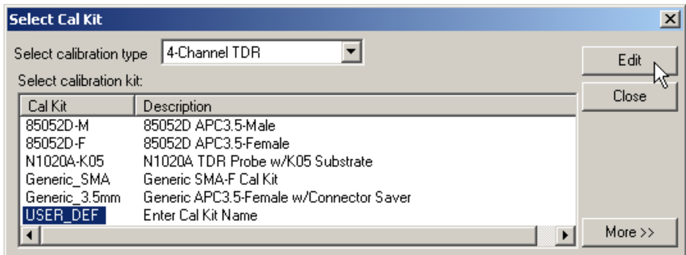
2. From the **Utilities** menu, select the TDR System Model (in this case, **86100A|54754A|54754A**), the click **Edit Cal Kits** as shown in Figure 3-11.

Figure 3-11 Selecting Edit Cal Kits



3. In the *Select Cal Kit* dialog box, make sure **4-Channel TDR** is selected in the **Select Calibration Type** list, then select the **USER_DEF** selection and click **Edit** to start the process. See Figure 3-12.

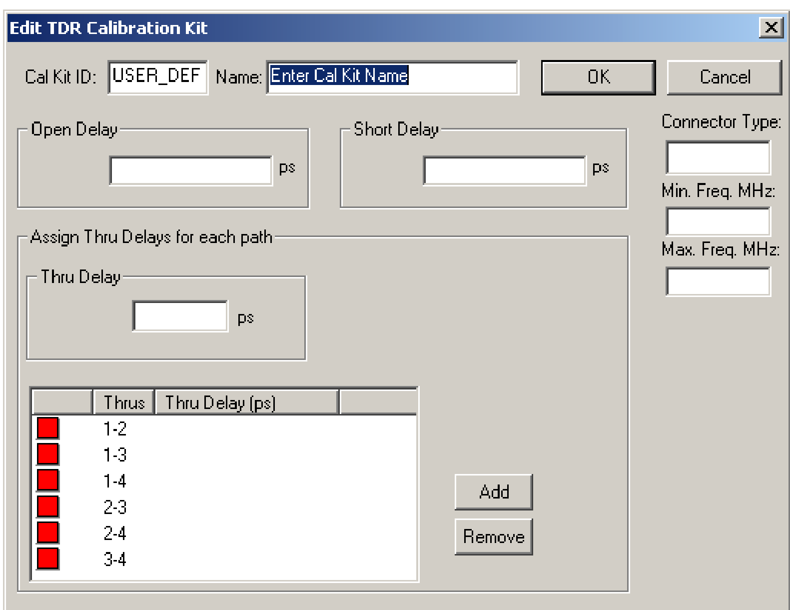
Figure 3-12 Select Cal Kit Dialog Box



4. The calibration kit definition is performed in the *Edit TDR Calibration Kit* dialog box shown in Figure 3-13. Enter **Cal Kit ID** and **Name** into the appropriate entry boxes in the header.

   The **Cal Kit ID** is listed in the **Cal Kit** column and **Name** is listed in the **Description** column of the calibration kit selection window shown in Figure 3-7.

Figure 3-13 Blank Edit TDR Calibration Kit Dialog Box



5. Also enter **Connector Type**. You may optionally specify the minimum (**Min. Freq. MHz**) and the maximum (**Max. Freq. MHz**) frequencies in megahertz.

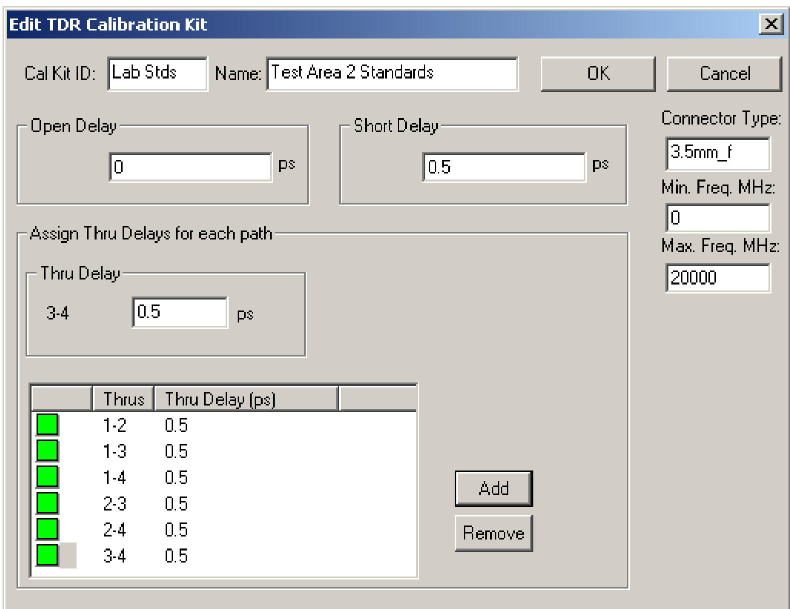
   The PLTS software does not currently use the frequency range information.

6. Enter the delay values for the open and short standards **Open Delay** and **Short Delay** entry boxes.

   Enter the delay values in picoseconds. The value may be found in the calibration kit documentation if you are using a standard from a calibration kit.

   Typically, if you are using an un-terminated connector or cable as the open, you may enter "0" (zero) ps as the value for the open.

Figure 3-14 Completed Edit TDR Calibration Kit Dialog Box



7. Enter the delay value (in picoseconds) of the thru in the **Thru Delay** entry box. Then, in the list of thrus, select the box at the left of the thru and click the **Add** button to associate the thru to the entered delay value. Repeat this until all six thru paths are completed.

   The color of the box at the left of the thru changes from red to green as the thru path is completed. Figure 3-14 shows the data that is used in this example. Be sure to enter a delay value for all six thru paths.

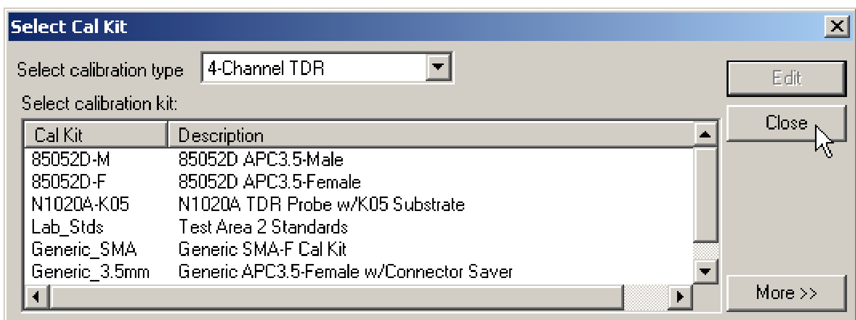
   If your thru is part of an Agilent (or Hewlett-Packard) calibration kit, the delay value is listed in the Standard Definitions section of the calibration kit documentation.

The **Remove** button disassociates the delay value with the thru path allowing you to change the delay value for the path.

All of the thru paths may use the same delay value or you may have multiple thru delay values if you are using a thru having a different delay. One instance of using more than one delay value might be when you are creating a calibration kit definition for probing and not all of the thru paths would use the same thru delay value.

8. After the *Edit TDR Calibration Kit* dialog box is complete, click the **OK** button to save the calibration kit with the new information and return to the *Select Cal Kit* dialog box shown in displaying the newly defined calibration kit in the list. **Cancel** deletes the information entered and returns to the *Select Cal Kit* dialog box.
9. Then click the **Close** button to close the *Select Calibration Kit* dialog box.

Figure 3-15 Select Cal Kit Dialog Box with Newly Defined Cal Kit

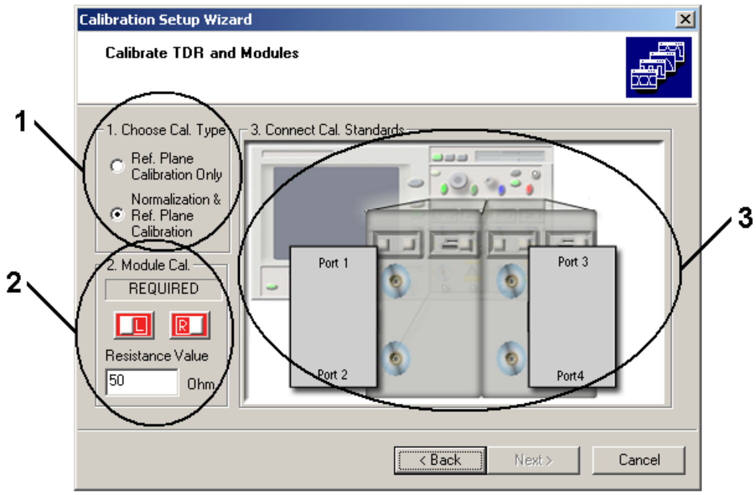


Return to the *Startup Wizard* by selecting **New** from the **File** menu to make your measurement. Refer back to "Starting the Startup Wizard" on page 90.

To Choose the Calibration Type

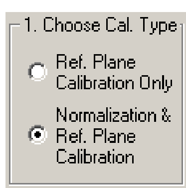
After selecting the calibration kit, verifying the data file, and selecting the **Next >** button, the *Calibrate TDR and Modules* wizard window is displayed. See Figure 3-16. This wizard window is used for the entire calibration.

Figure 3-16 Calibrate TDR and Modules Wizard Screen: Initial Display



To select the calibration type, you will only use the **Choose Cal. Type** area which is labeled "**1**" in Figure 3-16. This area is shown in Figure 3-17. The **Choose Cal. Type** area allows you to select between the two calibration types listed.

Figure 3-17 Choose Cal. Type Area



The **Ref. Plane Calibration Only** selection:

- Is available to all compatible TDR systems
- Sets the reference plane at the end of the cables
- Automatically de-skews the cables when differential measurements are selected
- Positions the trace on the display for easy viewing such that the step is one-half of a division from the left edge of the display and displays the complete maximum time base

value to the right of the step

- Recommends the module calibration

The **Normalization & Ref. Plane Calibration** selection:

- Is the most complete and accurate TDR calibration
- Has all of the features as the reference plane calibration selection listed above
- Removes cable loss, mismatch, and reflection errors which can be significant

NOTE TDR Normalization

— Is available only with the Agilent 86100-based TDR system using the 86100’s built-in differential TDR/TDT normalization feature

— Requires the module calibration

Choose the Calibration Type

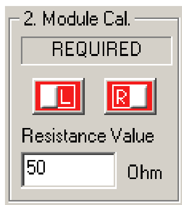
1. In the **Choose Cal. Type** area (see Figure 3-17), select either **Ref. Plane Calibration Only** or **Normalization & Ref. Plane Calibration**.

To Perform a Module Calibration

The module calibration, also called the vertical channel calibration, calibrates the gains, offsets, and timing for each channel. This calibration is valid for all supported TDR systems.

The module calibration is not always a requirement. The **Module Cal.** area (see Figure 3-18) has a text box that indicates the status of the module calibration. It displays: REQUIRED, RECOMMENDED, or VALID

Figure 3-18 Module Cal. Area



The module calibration is not required if you selected if you selected **Ref. Plane Calibration Only** in the **Choose Cal. Type** area. However, it is recommended.

Once a module calibration is performed, it remains valid for ten hours, unless:

- the instrument power has been cycled
- the module has been removed and then reinserted.
- the ambient temperature changes more than 5°C since the last module calibration

There are two icons which represent the two modules, the icon labeled “L” representing the left module (channels 1 and 2) and the icon labeled “R” representing the right module (channels 3 and 4). The color of the module calibration icon shows the status of the module calibration.

If an icon is:

- **Red**, the module calibration is required.
- **Yellow**, the module calibration is recommended.
- **Green**, the module calibration is valid.

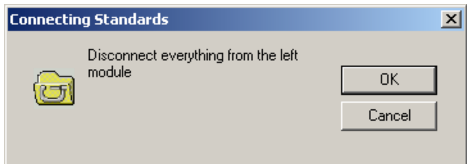
The **Resistance Value** text box allows you to enter the resistance value of your load. The default value is 50 ohms.

Perform the Module Calibration

1. Enter the resistance value of your load in the **Resistance Value** text box if it varies from the current value.
2. Click the left module’s “L” icon to start the module calibration.

3. Follow the PLTS software prompts to perform the module calibration for the left module. Figure 3-19 shows the first prompt that is displayed.

Figure 3-19 First Module Calibration Prompt



NOTE Follow the PLTS software prompts on your PC rather than following any prompts that is displayed on your TDR to ensure your calibration is correct.

The module calibration gives you five prompts.

NOTE This procedure requires that all cables be removed from the module.

The left module calibration prompts are:

i. Disconnect everything from the left module
ii. Connect a 50 ohm load to Channel 1 at the reference plane
iii. Connect a 50 ohm load to Channel 2 at the reference plane
(Note: You may use the load that was on channel 1 or you may use another load.)
iv. Disconnect everything from the left module
v. Done Re-connect test port Cables to Ports 1 and 2

When the module calibration is complete, the left module icon color changes to green.

4. Repeat steps 2 and 3 to perform the module calibration for the right module using the icon labeled “R”.

Once the right module calibration is complete, the color of both icons has changed to green. Also the text box which displayed either REQUIRED or RECOMMENDED, now has changed to VALID.

CAUTION Make sure the module calibration passes. If either of the module calibrations fail, you may still be able to complete the calibration and make a measurement. However, the accuracy of the measurement will be in doubt. It is recommended that you reattempt the module calibration before proceeding and correct the equipment problem, if there is one.

The module calibration procedure is now complete and the module calibration factors are stored in the non-volatile memory in the modules.

To Set the Reference Plane at the End of the Cables

This procedure de-skews, performs a reference plane calibration, performs a normalization (if selected) using the **Connect Cal. Standards** area.

Both calibration types (Ref. Plane Calibration Only and Normalization & Ref. Plane Calibration) require that the standards be connected to perform the calibration. After the calibration type is chosen and all required module calibrations are complete, the calibration icons are displayed in the **Connect Cal. Standards** area of the *Calibrate TDR and Modules* wizard screen shown in Figure 3-20.

The calibration first calibrates the reference plane, setting the reference plane at the end of the cables. Then the calibration automatically de-skews the cables if differential measurements are selected. If the **Normalization & Ref. Plane Calibration** type is selected, then the normalization is performed which removes errors such as, cable loss, mismatch, and reflections.

NOTE **Automatic De-Skewing**

Unmatched cables and differences in the path of the TDR module cause skew in differential and common signals. Skew must be removed to provide accurate differential measurements. De-skewing is required for TDR and TDT measurements. PLTS performs this tedious, error-prone task automatically and correctly. PLTS can remove skew of up to 780 ps in the Agilent TDR system and of up to 340 ps in the Tektronix TDR systems.

PLTS removes the TDR skew by measuring the amount of skew, and then adding 1/2 of the amount to the appropriate channel. PLTS then repeats the process, iteratively adjusting the channels until the rest of the skew is removed.

After TDR is de-skewed, Channel 1 is connected to channel 3 while channel 2 is connected to channel 4 using thrus. The time difference is measured in the two TDT signal paths and de-skewing is applied to the appropriate channel correcting the TDT response.

This section guides you through performing the calibrations for both calibration types. Both types are divided into two calibrations:

- Single-Ended, Differential-Mode, and Common-Mode Calibration
- Thru Path Calibration

Figure 3-20 Calibrate TDR and Modules Wizard Screen

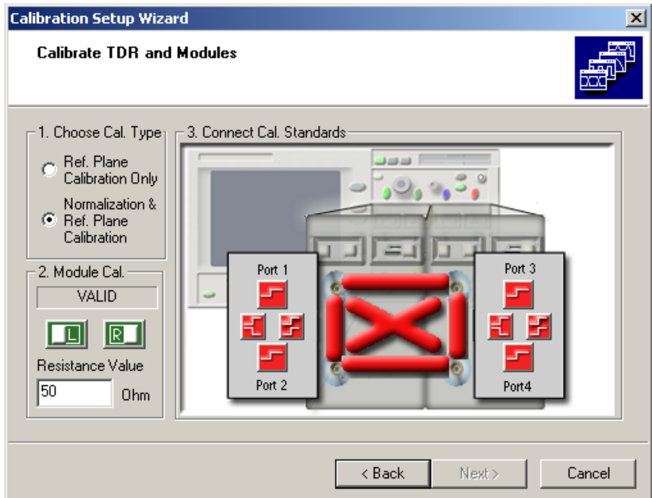
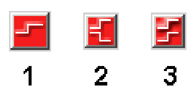


Figure 3-20 shows the *Calibrate TDR and Modules* wizard screen when all calibration and measurement parameters selected. The **Connect Cal Standards** area displays several icons. All available icons are displayed in this illustration. Both modules are shown with four icons. There are three unique icons:



The icon labeled “1” is the single-ended icon. This icon is displayed twice in each module; once for each TDR channel. The icon labeled “2” is the differential-mode icon. It is displayed in each module. The icon labeled “3” is the common-mode icon. It is also displayed in each module.

Some of these icons may not be displayed when you are performing a calibration. The icons that are displayed depend on the calibration and measurement parameters that are selected during the initial setup in “To Select the Calibration and Measurement Parameters” on page 97. Table 3-4 shows when each of these icons are displayed and which calibrations are required based on the selected parameters.

Table 3-4 Single-Ended, Differential-Mode, and Common-Mode Calibration Parameters

Icon	Definition	Selected Parameters that Display Icon
	**Single-Ended**	T11, T22, T33, and T44: All single-ended TDR parameters (T11: icon on Channel 1, T22: icon on Channel 2, T33: icon on Channel 3, T44: icon on Channel 4)

Table 3-4 Single-Ended, Differential-Mode, and Common-Mode Calibration Parameters

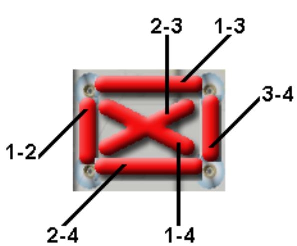
Icon	Definition	Selected Parameters that Display Icon
	**Differential Mode**	TDD11, TCD11, TDD22, TCD22: All differential TDR parameters having a differential stimulus (TDD11 and TCD11: icon on module 1, TDD22, TCD22: icon on module 2.
	**Common Mode**	TDC11, TCC11, TDC22, TCC22: All differential TDR parameters having a common stimulus (TDC11 and TCC11: icon on module 1, TDC22, TCC22: icon on module 2.

The color of the icons displayed in the **Connect Cal Standards** area is red if the calibration has not performed. While the calibration is being performed, the icon color is blue. When the calibration is complete, the icon color turns to green.

The calibration also requires a calibration between the ends of the TDR channel cables. These calibrations are called Thru Path calibrations because a “thru” connector is used to connect the ends of the cables. Refer to Table 3-5 for an illustration showing each of the six thru path icons representing the each thru path between the four channels.

Some of these thru path icons may not be displayed when you are performing a calibration. As with the single-ended, differential-mode, and common-mode icons shown above, the icons that are displayed depend on the calibration and measurement parameters that are selected during the initial setup in “To Select the Calibration and Measurement Parameters” on page 97. Table 3-5 also shows when each of these icons are displayed and which calibrations are required based on the selected parameters.

Table 3-5 Thru Paths Required by the Parameters Selected

Thru Paths Icons Displayed on Wizard	Thru Paths	Single-Ended Parameters	Differential Parameters
	**1 – 2**	T12 and T21	N/A
	1 – 3	T13 and T31	TXX12 or TXX21, *where XX is DD, DC, CD, or CC*[a]
	1 – 4	T14 and T41	N/A
	2 – 3	T23 and T32	N/A
	2 – 4	T24 and T42	TXX12 or TXX21, *where XX is DD, DC, CD, or CC*[a]
	3 – 4	T34 and T43	N/A

a. When a differential parameter warrants the thru path calibration on either thru path 1 – 3 or 2 – 4, the calibration of the other thru path is required and is performed at the same time.

Perform the Single-Ended, Differential-Mode, and Common-Mode Calibrations

This section guides you through performing the Single-Ended, Differential-Mode, and Common-Mode Calibrations for both calibration type selections (**Ref. Plane Calibration Only** and **Norm & Ref. Plane Calibration**).

Selecting an icon in the **Connect Cal Standards** area starts the calibration. The order that the calibration is performed is completely arbitrary. You may select the icons in any order.

NOTE TDR calibration has an incremental calibration feature. This allows you to recalibrate a specific parameter if you suspect there is an error. To recalibrate just that suspect calibration parameter, select the icon that represents the suspect parameter and follow the prompts.

In this example, the calibration is performed using all of the icons. See Figure 3-20 for an illustration. The left module is calibrated first. The right module is calibrated next.

1. Select the single-ended icon  at Channel 1 to display the prompt box that reads: **Leave open Channel 1 at the reference plane**

   The prompt box is displayed in Figure 3-21. These prompts are used throughout the TDR calibration.

Figure 3-21 Single-ended Calibration Prompt

NOTE Follow the PLTS software prompts on your PC rather than following any prompts that is displayed on your TDR to ensure your calibration is correct.

---

2. Make sure that nothing is connected to the end of the Channel 1 cable and click the **OK** button to calibrate.

   This is the only connection for the **Ref. Plane Calibration Only**. However, if you are performing a **Norm & Ref. Plane Calibratio**n, there are two additional calibration prompts to follow. These prompts are:

   i. Connect a short to Channel 1 at the reference plane
   ii. Connect a 50 ohm load to Channel 1 at the reference plane

   The calibration is complete when the single-ended icon color changes to green.

3. Repeat steps 1 and 2 for the single-ended icon at Channel 2.

4. Select the differential-mode icon of the left module to display the prompt box that reads: **Leave open Channels 1 and 2 at the reference plane**

5. Make sure that nothing is connected to the end of the Channel 1 and Channel 2 cables and click the **OK** button to calibrate.

   This is the only connection for the **Ref. Plane Calibration Only**. However, if you are performing a **Norm & Ref. Plane Calibratio**n, there are four additional calibration prompts to follow. These prompts are:

   i. Connect shorts to Channels 1 and 2 at the reference plane
   ii. Remove short from Channel 1 at the reference plane
   iii. Connect a 50 ohm load to Channel 1 at the reference plan
   iv. Replace short on Channel 2 with a 50 ohm load

   The calibration is complete when the differential-mode icon color changes to green.

6. Select the common-mode icon of the left module to display the prompt box that reads: **Leave open Channels 1 and 2 at the reference plane**

7. Make sure that nothing is connected to the end of the Channel 1 and Channel 2 cables and click the **OK** button to calibrate.

   This is the only connection for the **Ref. Plane Calibration Only**. However, if you are performing a **Norm & Ref. Plane Calibration**, there are four additional calibration prompts to follow. These prompts are:

   i. Connect a short to Channel 1 at the reference plane
   ii. Connect a 50 ohm load to Channel 1 at the reference plane
   iii. Connect a short to Channel 2 at the reference plane
   iv. Connect a 50 ohm load to Channel 2 at the reference plane

   The calibration is complete when the common-mode icon color changes to green.

With all of the left module calibrations complete, the right module is calibrated next.

8. Repeat steps 1 through 7 for Channel 3 and Channel 4 on the right module.

   With all of the right module calibrations complete, perform the thru path calibration next.

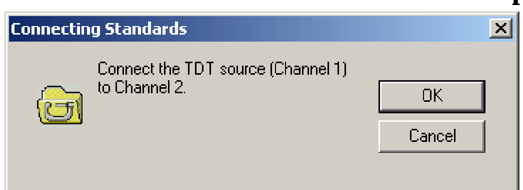
Perform the Thru Path Calibration

This section guides you through performing the thru path calibrations for both calibration type selections (**Ref. Plane Calibration Only** and **Norm & Ref. Plane Calibration**).

Selecting an icon in the **Connect Cal Standards** area starts the calibration. The order that the calibration is performed is completely arbitrary. You may select the icons in any order. However, for this example, start with step 1.

1. Select the thru path icon between Channel 1 and Channel 2 (see Table 3-5, if necessary) and follow the PLTS software prompts to perform the thru path calibration between these channels. Figure 3-22 shows the prompt that is displayed.

Figure 3-22 Channel 1 - Channel 2 Thru Path Prompt



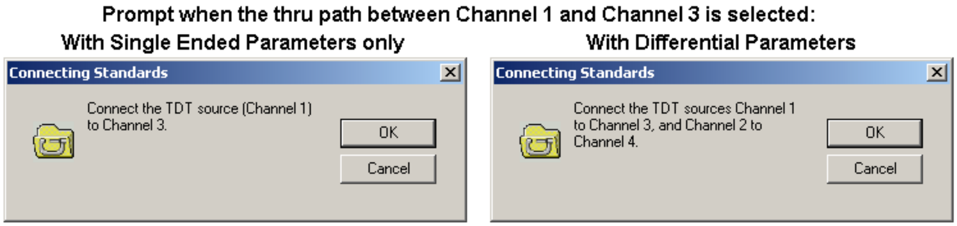
NOTE Follow the PLTS software prompts on your PC rather than following any prompts that is displayed on your TDR to ensure your calibration is correct.

The calibration is complete when the thru path icon color changes to green.

2. Select the thru path icon between Channel 1 and Channel 3.

   Depending on the selected parameters, the prompt box message can vary as shown in Figure 3-23 when the thru icon between channels 1 and 3 (or between channels 2 and 4) is selected.

Figure 3-23 Channel 1 - Channel 3 Thru Path Prompts



3. Using thru connectors, make the cable connections indicated by the prompt and select the **OK** button to calibrate.

   The calibration is complete when the thru path icon colors change to green.

4. Repeat step 1 for the remaining uncalibrated thru paths.

   You may repeat any portion of this calibration by clicking the icon and following the prompt that is displayed.

5. When the calibration is complete and the color of all of the icons have changed to green, click the *Calibrate TDR and Modules* wizard screens **Next >** button to complete the calibration. Continue at "Making a Measurement" on page 118.

NOTE TDR calibration has an incremental calibration feature. This allows you to recalibrate a specific parameter if you suspect there is an error. To recalibrate just that suspect calibration parameter, select the icon that represents the suspect parameter and follow the prompts.

This completes the TDR calibration.

NOTE As mentioned above, TDR calibration has an incremental calibration feature. In addition to recalibrating a suspect parameter during a calibration, this feature allows you to revisit the calibration wizard and select additional parameters to calibrate (Figure 3-5) and continue through the calibration (Figure 3-20) and while following the prompts, calibrate just those incremental, new parameters.

Making a Measurement

The Measurement process includes:

1. Connecting the DUT

   This step shows you how to connect your DUT to the PLTS.

2. Selecting the initial displayed format of the measurement

   This step guides you through selecting which of the ten analysis formats that the upcoming measurement will initially be displayed in. The measurement can be displayed in any of the 10 formats at any time.

3. Modifying the measurement stimulus and measured parameters

   This optional step allows you to make last minute changes to the measurement stimulus dialog box and the measurement parameter dialog box. Only changes that will not require a recalibration may be made.

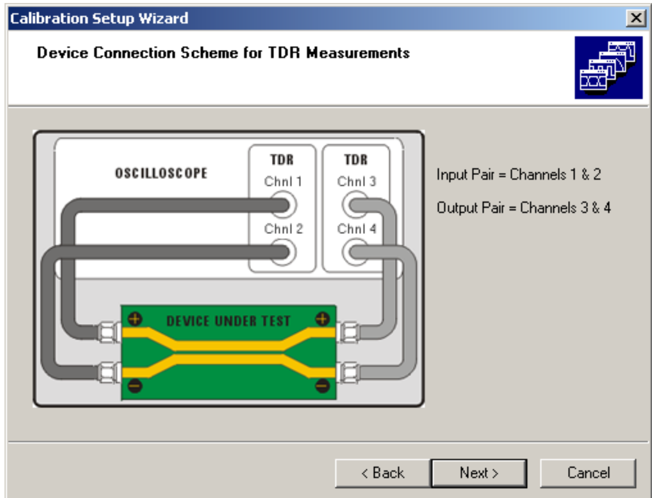
4. Running the measurement

   This step guides you through starting the measurement and displaying the data when the measurement is complete. It also directs you to save the measurement and points you to the chapters that will assist in your data analysis.

To Connect the DUT

1. After saving the calibration data and selecting the **Next >** button, the *Device connection scheme for TDR calibrations* window is displayed. See Figure 3-24. This wizard window shows how to connect your DUT to the TDR system to make the measurement. Connect the DUT to the TDR system as shown and then click the **Next >** button.

Figure 3-24 Connecting the DUT to the TDR System



To Select the Initial Displayed Format of the Measurement

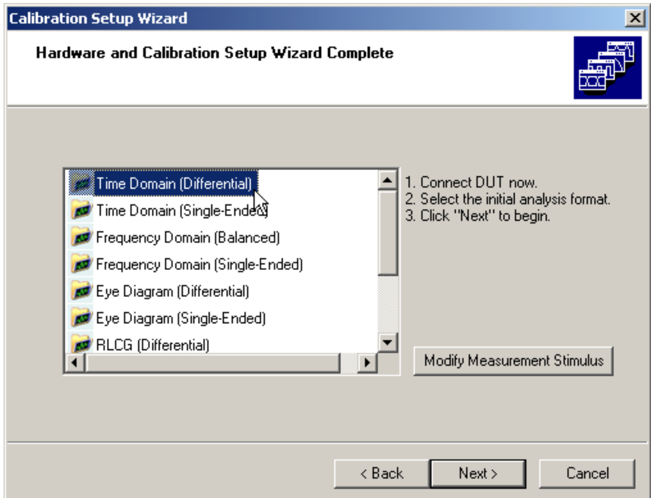
1. The *Hardware and Calibration Setup Wizard Complete* window is displayed. See Figure 3-25.

   This window is the software location to select the one of ten analysis formats that the measurement will be displayed as initially. The analysis formats are:

   - Time Domain (Differential)
   - Frequency Domain (Balanced)
   - Eye Diagram (Differential)
   - RLCG (Differential)
   - RLCG (W-Element)
   - Time Domain (Single-Ended)
   - Frequency Domain (Single-Ended)
   - Eye Diagram (Single-Ended)
   - RLCG (Common)
   - RLCG (Self/Mutual)

   For the purpose of this example, the **Time Domain (Differential)** format icon is selected.

Figure 3-25 Hardware and Calibration Setup Wizard Complete

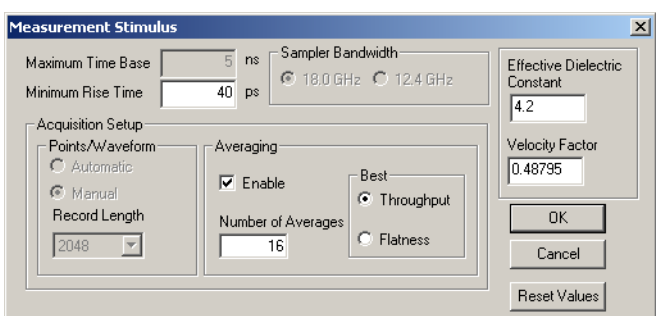


To Modify the Measurement Stimulus and Measured Parameters

1. The **Modify Measurement Stimulus** button opens the *Measurement Stimulus* dialog box. See Figure 3-26. This dialog box allows you to change some of the measurement stimulus settings that you set up previously.

   Settings that would require a recalibration are not active and may not be changed in this dialog box. To change the stimulus settings that are inactive, you must click the **< Back** button until you get back to the window described in “To Set Up the TDR” on page 94.

Figure 3-26 Measurement Stimulus Dialog Box



For the purpose of this example, leave the settings unchanged and click the **OK** button.

The **Reset Values** button resets any active values (any values that can be changed without requiring a recalibration) to their original default values prior to being changed in "To Set Up the TDR".

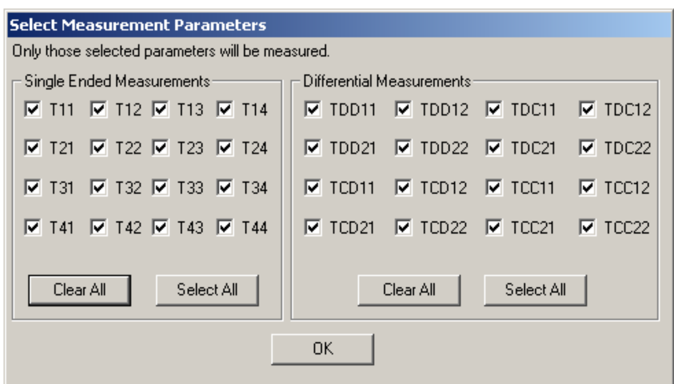
The **Cancel** button resets any changes that were made after opening this dialog box.

NOTE After the measurement has been made, you may verify these changes were made by printing a characterization report. See "Characterization Report Generator" on page 315 for help.

---

2. Once you exit the *Measurement Stimulus* dialog box, the *Select Measurement Parameters* dialog box is displayed. See Figure 3-27. The left side of the dialog box lists all 16 of the TDR single-ended measurements. As with the dialog box described in "To Select the Calibration and Measurement Parameters" on page 97, the right side of the dialog box lists all 16 of the TDR differential measurements. Only the parameters for which the calibration is valid are active. You may select all of the active parameters or you may select a subset of these parameters. Selecting a subset of the parameters may save you some measurement time.

Figure 3-27 Select Measurement Parameters Dialog Box



To measure parameters that are inactive, you must click the **< Back** button until you get back to the window shown in To Select the Calibration and Measurement Parameters and reselect the parameters. However, note that if you change the parameters in this window, you may be required to perform additional portions of the calibration.

For the purpose of this example, leave the parameters unchanged and click the **OK** button.

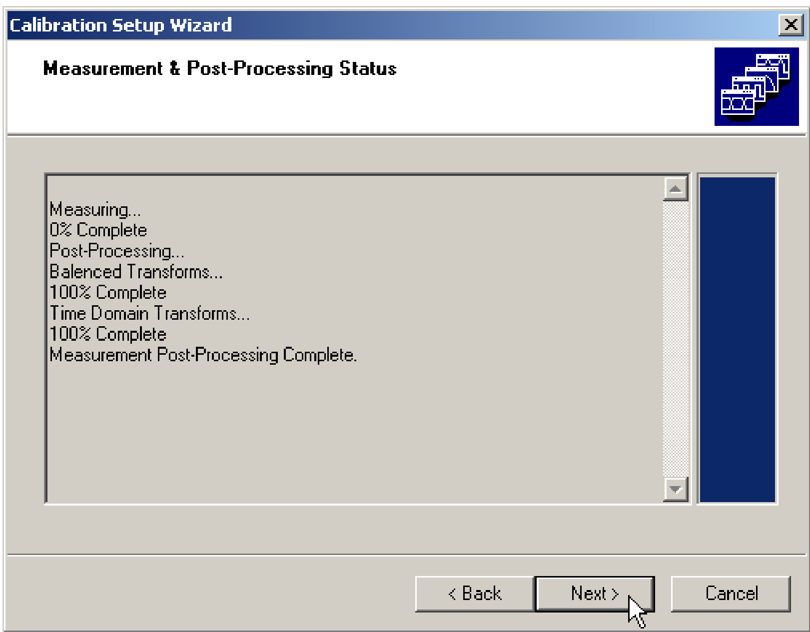
3. With the *Hardware and Calibration Setup Wizard Complete* window is displayed (see Figure 3-25), select the **Next >** button to start the measurement.

To Run the Measurement

1. The software displays the wizard's *Measurement & Post-Processing Status* window and starts the measurement and the measurement post-processing. See Figure 3-28. The software makes each of the measurements. The status of the measurements and the post-processing is displayed in the status text area. The status may also be observed by watching the status bar at the right edge of the text area. As the measurements and the post-processing proceed, the color of the bar gradually changes to blue.

   Once the post-processing is complete, click the **Next >** button to display the measurement results in the main PLTS window.

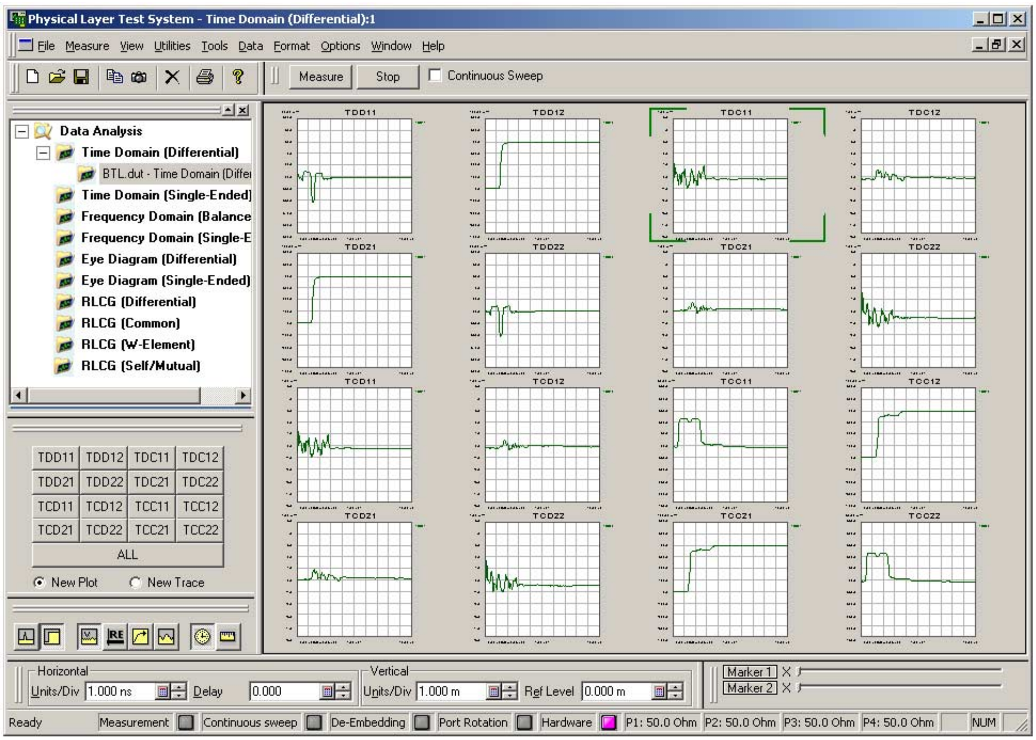
Figure 3-28 Measurement & Post-Processing Status Window



Analyzing the Measurement Results

Now that you have measured the device under test, the measurement results are displayed in the main PLTS window as shown in Figure 3-29. This shows the results in the **Time Domain (Differential)** format, the analysis format selected earlier.

Figure 3-29 Displayed Measurement in Time Domain (Differential) Format



Now that the measurement has been made, the main Physical Layer Test System (PLTS) window gives you the flexibility to perform analysis on the measured data in a variety of ways.

- **Frequency Domain.** The Frequency Domain analysis may be analyzed in Balanced or Single-Ended mode.Refer to Chapter 4, “Analyzing Data in the Frequency Domain,” on page 127 for information.
- **Time Domain.** The Time Domain analysis may be analyzed in Differential or Single-Ended mode. Refer to Chapter 5, “Analyzing Data in the Time Domain,” on page 147 for information.

- **Eye Diagram.** The Eye Diagram analysis may be analyzed in Differential or Single-Ended mode. Refer to Chapter 6, “Analyzing Data using Eye Diagrams,” on page 185 for information.
- **Transmission Line (RLCG).** The RLCG analysis may be analyzed in Differential, Common, W-Element, and Self/Mutual modes. Refer to Chapter 7, “Analyzing Transmission Line Parameters,” on page 205 for information.

CAUTION To ensure the measurement data is not lost, you may want to first save the measurement data by selecting Save from the File menu. See “Save” under the “File Menu” section in the “Menu Reference” chapter of the PLTS Installation and Reference Guide.

II Data Analysis, Exporting, and Utilities

Part II provides detailed information about optimizing and analyzing the measured data using the physical layer test system.

Chapter 4, "Analyzing Data in the Frequency Domain"

Provides information about analyzing measured data in the frequency domain.

Chapter 5, "Analyzing Data in the Time Domain"

Provides information about analyzing measured data in the time domain.

Chapter 6, "Analyzing Data using Eye Diagrams"

Provides information about analyzing measured data using the eye diagram.

Chapter 7, "Analyzing Transmission Line Parameters"

Provides information about extracting transmission line R, L, C, G parameters from the measured data.

Chapter 8, "Importing and Exporting Data"

Provides information about importing data files into PLTS and exporting measured data for use with other software packages, such as TDA IConnect[a] MeasureXtractor[b], Avanti HSPICE, or Agilent ADS.

Chapter 9, "Removing Unwanted Effects from the Measurement"

Provides information about using measurement enhancement features such as gating, port rotation/extension, and de-embedding.

Chapter 10, "Using Analysis Tools and Utilities"

Provides information about using the PLTS tools, such as markers, trace math, overlaying traces, zooming, printing, and report generator and the file converter utility.

a. IConnect is a registered trademark of TDA Systems, Inc.
b. MeasureXtractor is a trademark of TDA Systems, Inc.

4 Analyzing Data in the Frequency Domain

Physical layer test systems can provide measurement-based frequency domain information in two ways:

- Making frequency domain measurements directly utilizing a vector network analyzer (VNA) and an S-parameter test set to sweep the DUT with an RF signal and measuring the RF response.
- Making the time domain measurements utilizing a Time Domain Reflectometer (TDR) to apply a synthesized step waveform to a DUT and observing the response. Then the measured time domain information is converted to the frequency domain using the Fast Fourier Transform (FFT).

  In a linear network, the Fourier Transform describes the relationship between a time domain measurement and its corresponding frequency domain response in detail. Therefore, given the measured time domain response of a DUT, it is possible to determine its frequency domain response mathematically by performing a Fourier Transform.

There is information to be gained from the frequency domain that other analysis types do not provide. Frequency domain information can help you verify and validate your modeling and simulation procedures by providing:

- Vector error-corrected data which allows you to de-embed fixtures and signal launches
- More accurate simulation for frequency dependent effects, such as bandwidth and impedance
- Insight into common analog problems, such as crosstalk, reflections, and loss
- Better information more efficiently when you are analyzing the effects of transmission lines, studying power/ground distribution, and investigating EMI effects as a function of frequency
- S-parameter data which can be used over the widest range of applications and frequency bandwidths

About S-Parameters

At high frequencies, S-parameters (scattering parameters) are commonly used to describe the performance of microwave and RF devices. These parameters can be used to completely describe the electrical behavior of the device (or network). For those not familiar with S-parameters, they are simply the energy that is reflected off of, or transmitted through, a device under test. While S-parameter data is formatted differently than TDR/TDT data, the underlying information is the same.

S-parameters relate to familiar measurements such as reflection coefficient (input/output match), and transmission coefficient (gain or loss, and isolation). They are the shared language between simulation and measurement and are easily imported into electronic-design automation (EDA) tools like HSPICE, ADS, and other simulators.

Conventional *single-ended* parameters describe the performance of a single-ended device when it is stimulated on a single port, and the corresponding responses are observed on all of the ports. For a detailed explanation, refer to "Single-Ended (Unbalanced) S-Parameters" on page 130. *Mixed-mode* (or *balanced*) S-parameters describe the performance of devices with balanced ports. For a detailed explanation, refer to "Mixed Mode (Balanced) S-Parameters" on page 132.

Common Frequency Measurements with S-Parameters

Reflection Measurements	**Transmission Measurements**
Return Loss	Insertion loss
Standing Wave Ratio (SWR)	Gain/loss
Reflection coefficient	Transmission coefficient
Impedance	Electrical delay
Sxx (x = stimulus port and response port)	Syz (z = stimulus port, y = response port)

How to Interpret S-Parameters

A multi-terminal device can be viewed in different ways, depending on how it is meant to be operated. For a device that is designed to be a single-ended four-port device, its conventional (single-ended) four-port S-parameters can be measured and displayed. In a balanced device, two terminals constitute a single balanced port. Each balanced port will support both a common-mode and a differential-mode signal. This performance is described using mixed-mode (balanced) S-parameters.

Single-Ended (Unbalanced) S-Parameters

Conventional single-ended S-parameters are defined as the ratio of two normalized power waves (response/stimulus), defined in terms of the voltages and current at each port of a device (see Figure 4-2).

S-parameter notation identifies these quantities using the following convention:

SAB

where:

The first number (represented by A) refers to the test-device port where the signal is received. This received signal is referred to as the *response*.
- and -
The second number (represented by B) refers to the test-device port where the signal is sourced. This signal is referred to as the *stimulus*.

Figure 4-1 S-Parameter Notation

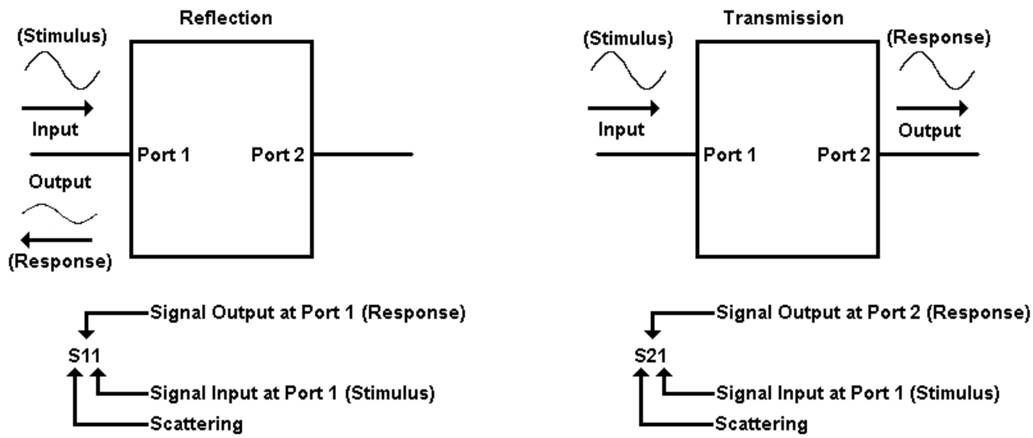
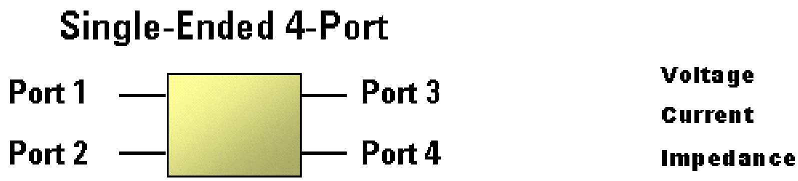


Figure 4-2 **Single-Ended S-Parameter Review**

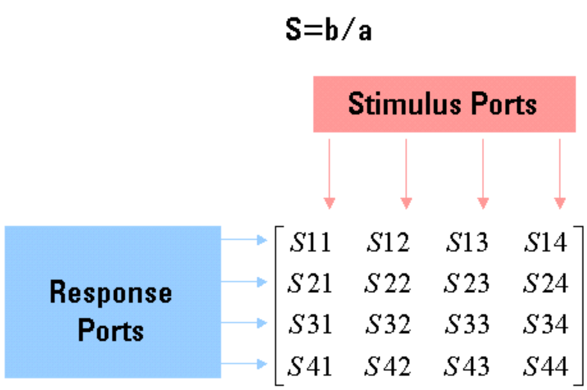


	Normalized Power Waves
stimulus	$an = \frac{1}{2 \cdot \sqrt{\mathrm{Re}\{Zn\}}}(Vn + In \cdot Zn)$
response	$bn = \frac{1}{2 \cdot \sqrt{\mathrm{Re}\{Zn\}}}(Vn - In \cdot Zn)$

S=b/a

Figure 4-3 shows the naming convention for a single-ended 4-port S-matrix, showing the ratio of all possible combinations of response/stimulus.

Figure 4-3 **Single-Ended S-Matrix**



Mixed Mode (Balanced) S-Parameters

Mixed-mode S-parameters are used to describe the performance of balanced circuits by independently considering each mode of operation. The mixed-mode S-parameter concept is similar to conventional S-parameter definitions, except that instead of stimulating a single terminal of the DUT, we consider pairs of terminals to be stimulated in either a differential (anti-phase) or a common (in-phase) mode.

For a balanced device, we are not necessarily interested in voltages and current referenced to ground. Instead, we can define differential and common mode voltages and currents on each balanced port. Likewise, we can also define differential-mode and common-mode impedances.

We can define normalized power waves on the ports of a balanced device having the exact same form as the single-ended case. Only the definitions of “voltage” and “current” are changed. Both are defined as ratios of normalized power waves.

Figure 4-4 Mixed Mode S-Parameter Basics

Balanced 2-Port

	Differential	Common
Voltage	$V_{an} - V_{bn}$	$0.5 \times (V_{an} + V_{bn})$
Current	$0.5 \times (I_{an} - I_{bn})$	$I_{an} + I_{bn}$
Impedance	$Z_{Dn} = \frac{V_D+}{I_D+}$	$Z_{Cn} = \frac{V_C+}{I_C+}$

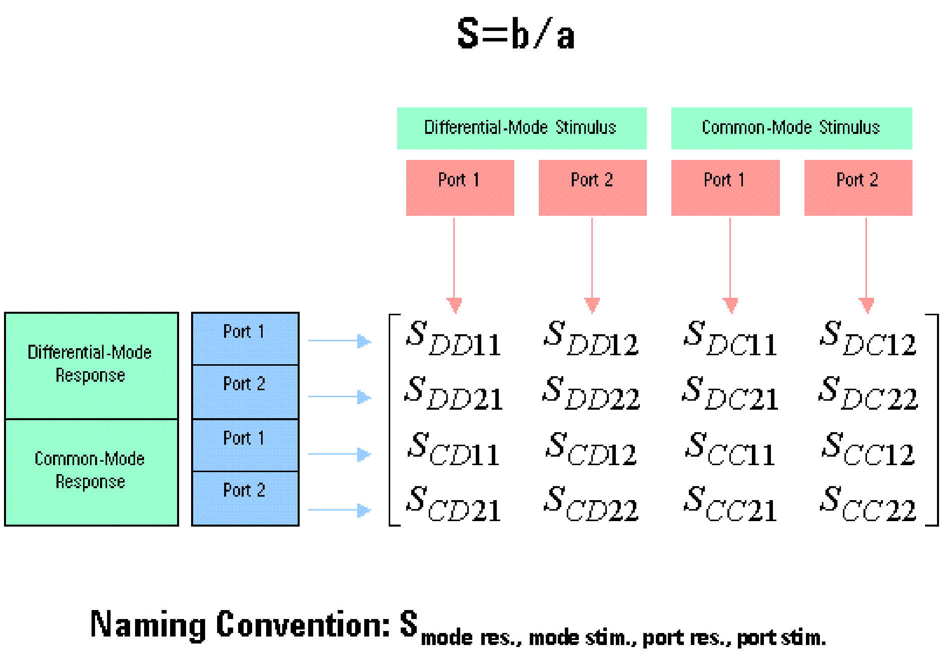
	Normalized Power Waves	
	Differential-Mode	**Common-Mode**
Stimulus	$b_{dn} = \frac{1}{2 \cdot \sqrt{Re(Zdn)}}(Vdn + Idn \cdot Zdn)$	$b_{cn} = \frac{1}{2 \cdot \sqrt{Re(Zcn)}}(Vcn + Icn \cdot Zcn)$
Response	$b_{dn} = \frac{1}{2 \cdot \sqrt{Re(Zdn)}}(Vdn - Idn \cdot Zdn)$	$b_{cn} = \frac{1}{2 \cdot \sqrt{Re(Zcn)}}(Vcn - Icn \cdot Zcn)$

S = b/a

The naming convention for mixed-mode S-parameters includes mode information as well as port information. Unlike the single-ended example, though, in the mixed-mode S-matrix, we are not only considering the port, but we are also considering the mode of the signal at each port. Therefore, the first two subscripts describe the mode of the response and stimulus, respectively, and the next two subscripts describe the ports of the response and stimulus.

Again we can take the ratio of all possible combinations of response over stimulus for the differential and common mode normalized power waves to calculate the mixed-mode S-parameters. The mixed-mode matrix fully describes the linear performance of a balanced two-port measurement. Figure 4-5 shows the naming convention for a mixed mode S-matrix, showing the ratio of all possible combinations of response/stimulus.

Figure 4-5 Mixed-Mode (Balanced) S-Matrix



Viewing Data in the Frequency Domain

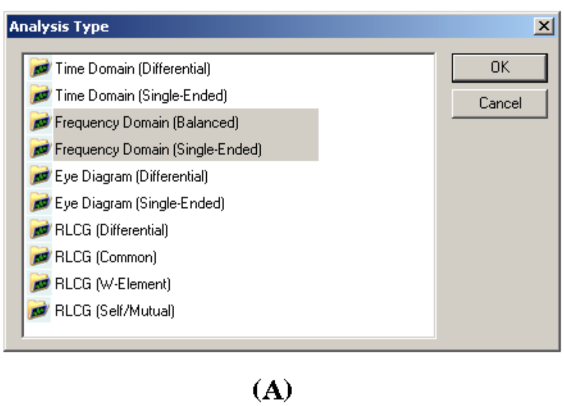
This section guides you with opening measured data in the frequency domain and viewing the data in the way that best suits your requirements. As discussed earlier, there are 16 4-port S-parameters for both single-ended and balanced devices. See Figure 4-3 and Figure 4-5. You may elect to view any number of these S-parameters.

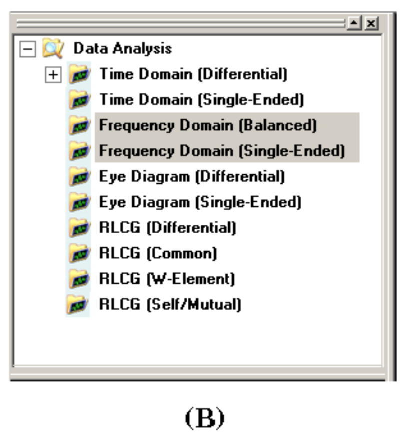
Opening a Frequency Domain Plot Window

The frequency domain may be viewed in either balanced or single-ended mode. You may open the frequency domain plot window in one of four ways.

- From the Startup Wizard immediately before selecting the **Measure** button where you must select the analysis type - see (A) of Figure 4-6
- From the Startup Wizard Welcome Screen by selecting **Load Measurement** where you must select the analysis type - see (A) of Figure 4-6
- From the **Open** selection in the **File** menu or the **Open** icon in the **Toolbar** where you must select the analysis type - see (A) of Figure 4-6
- From the **Browser** when data is already being viewed in another analysis type by selecting one of the Frequency Domain choices - see (B) of Figure 4-6

Figure 4-6 Opening the Frequency Domain Plot Window





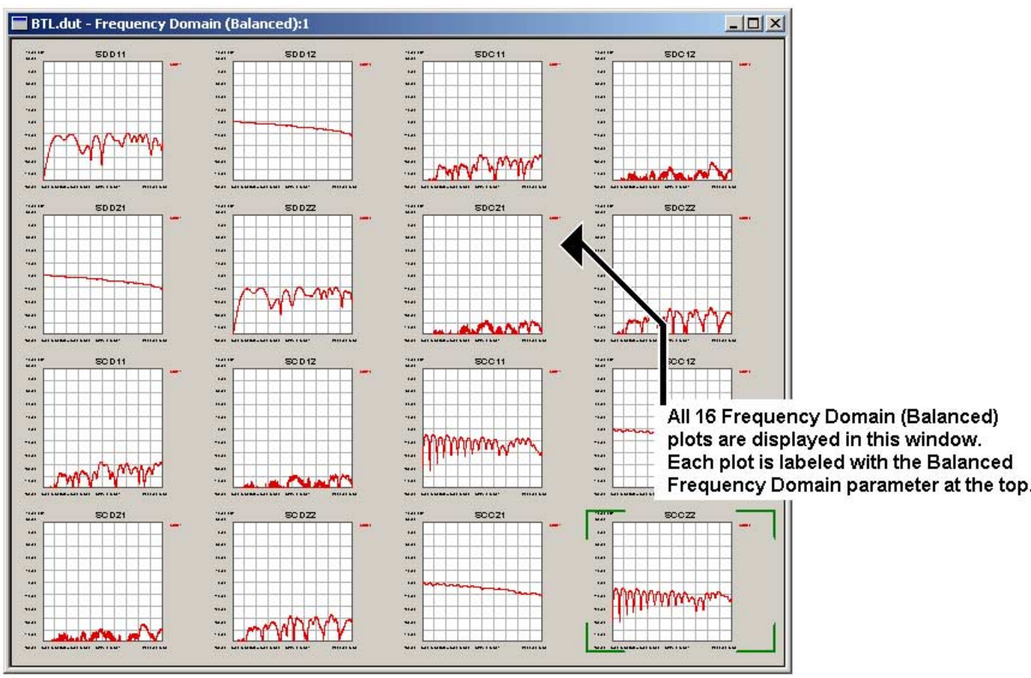
NOTE Using the **Browser** method to open a window requires that you select the parameters that you want to view from the **Parameter Bar** or the **Data** menu when the blank plot window is displayed.

Viewing All 16 S-Parameters

Except when you open the plot window using the **Browser**, all 16 frequency domain parameter plots are displayed. The parameter plots are displayed in the same orientation as shown in Figure 4-3 and Figure 4-5. Each plot is labeled with its parameter at the top. See Figure 4-7.

As mentioned previously, when you open the plot window from the **Browser**, an empty plot window is displayed. View all 16 plots by selecting **All** from the **Parameter Bar** or from the **Data** menu with **New Plot** selected.

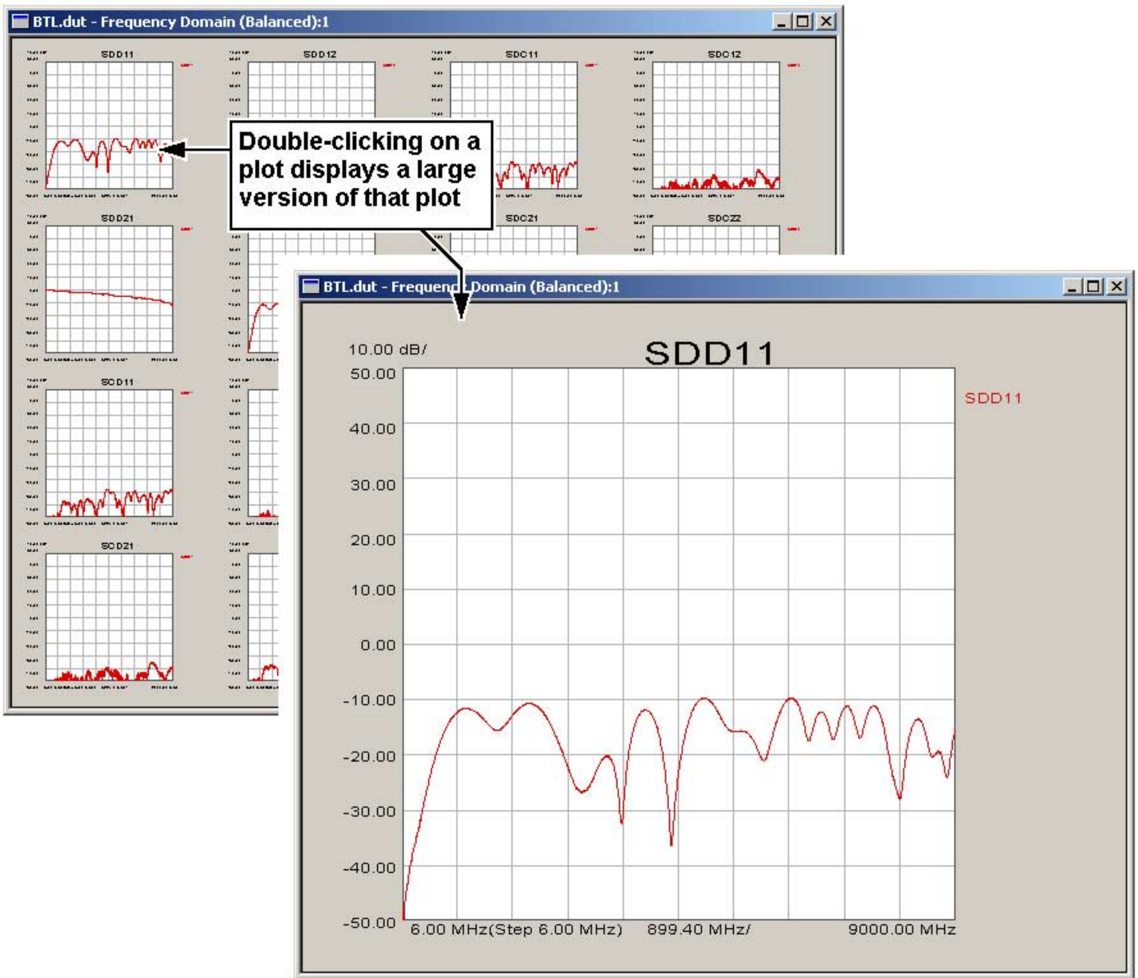
Figure 4-7 All 16 Balanced Frequency Domain Plots



Viewing a Single S-Parameter

Viewing a single plot gives better resolution. To display a single plot, from the window with all 16 plots (or with multiple plots if you have a custom plot window displayed), double-click on the plot that you would like to view. See Figure 4-8.

Figure 4-8 Opening a Single Plot

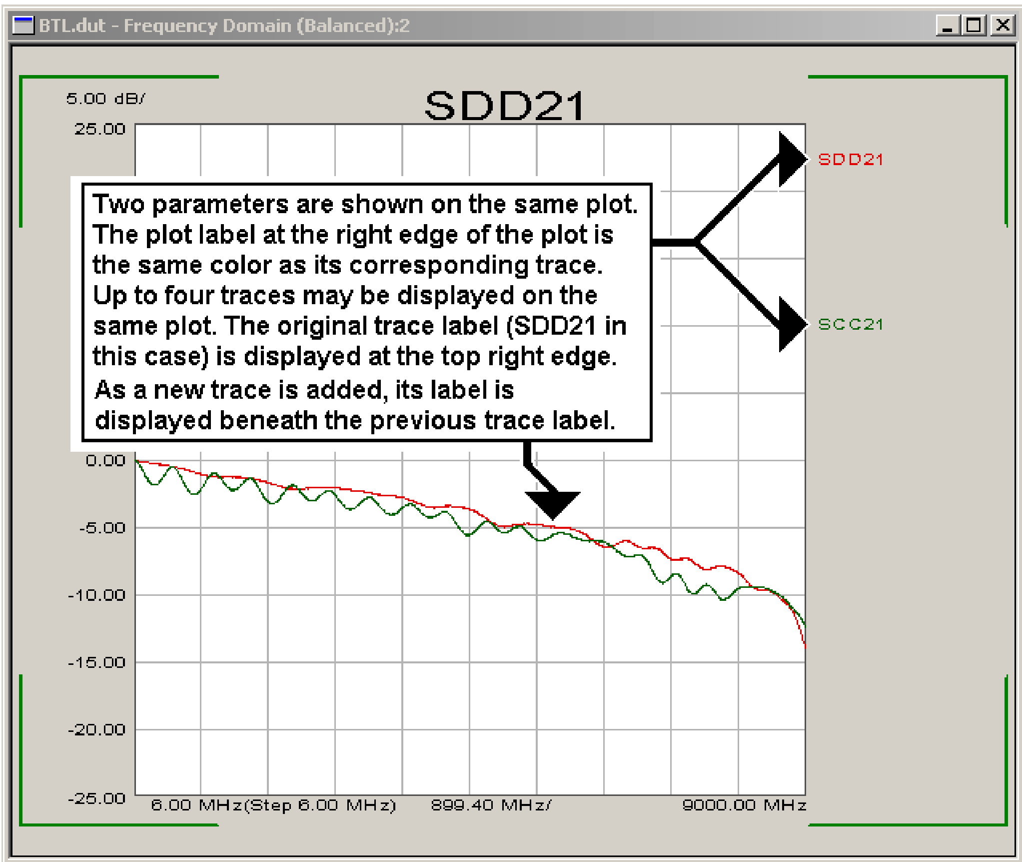


Double-click on the single plot to return to the original view with multiple plots.

View Multiple Traces on a Single Plot

This single plot may have just one parameter or if you choose, it could contain up to four parameters. For example, you could show how SDD21 compares with SCC21. To do this double-click on the SDD21 plot so that is now displaying a single plot similar to Figure 4-8. With SDD21 displayed as a single plot, make sure **New Trace** is selected in the **Parameter Bar** (or the **Data** menu), then click any parameters you wish to add, SCC21 in this example.

Figure 4-9 A Single Plot with Multiple Traces

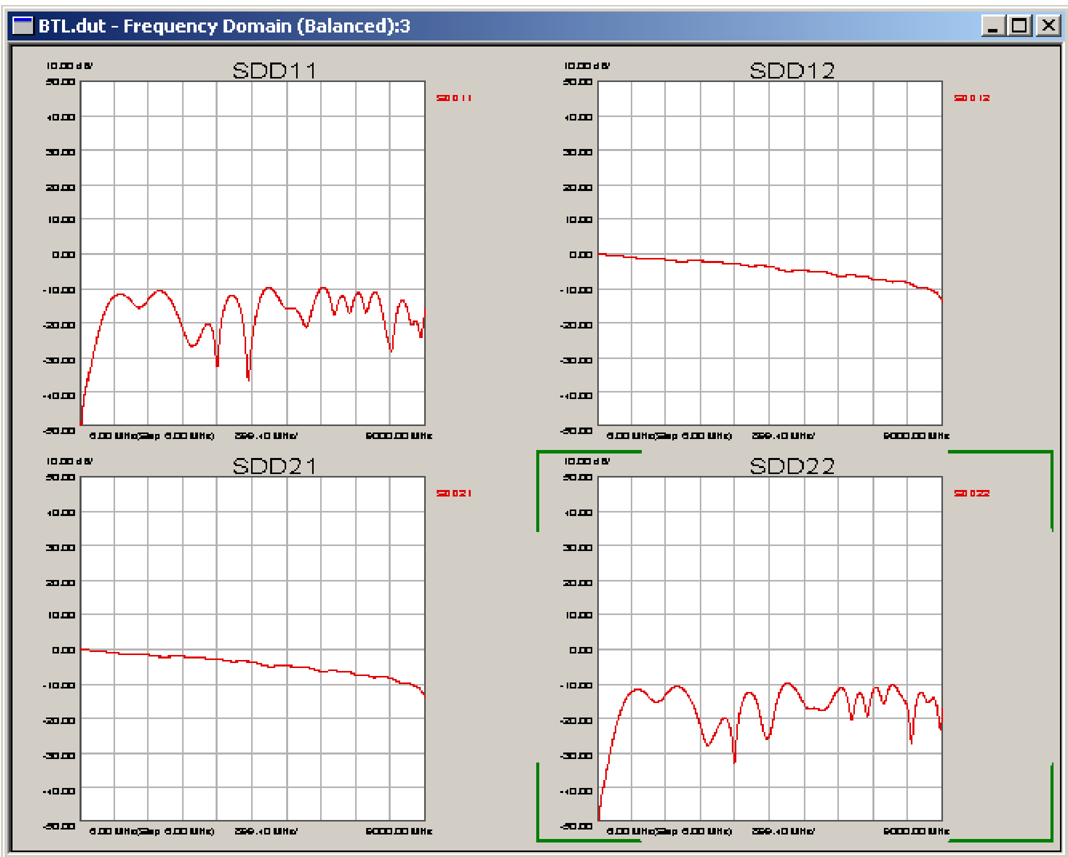


It could contain as many as 4 of the parameters. All trace parameters are labeled at the right edge of the plot. The color of the trace labels are the same color as the traces they represent.

Creating a Custom S-Parameter Plot Window

You can also create a plot window with just the plots you desire. For example, you may want your plot window to show just the four SDD*xx* plots. To create a custom window, open the data file in any analysis type. Then, in the **Browser**, select the data type that you want to display the plots. In this example, select **Frequency Domain (Balanced)**. A blank plot window is displayed. With **New Plot** selected in the **Parameter Bar** (or the **Data** menu), click the desired parameters (SCC11, SCC12, SCC21, and SCC22 in this example). As each parameter is selected, a new plot is added to the plots window. See Figure 4-10.

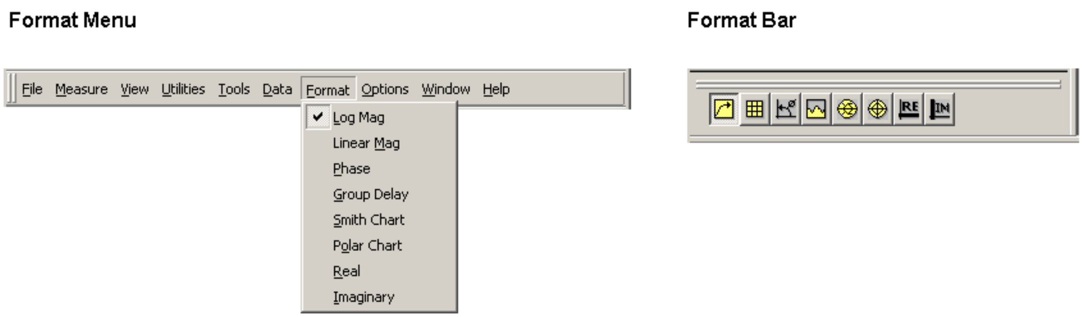
Figure 4-10 Custom S-Parameter Plots Window with Four Plots



Selecting Frequency Domain Display Formats

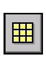
A data format is the way the physical layer test system presents the measurement graphically. Select a data format appropriate to the information you want to learn about the device. You may use either the **Format Bar** or the **Format** menu to select the format.

Figure 4-11 Format Menu and Format Bar for Frequency Domain



Frequency Domain Format Bar

Log Mag displays Cartesian logarithmic magnitude (no phase) in dB. Typical measurements are return loss and gain. This is the default format.



Linear Mag displays positive values only. Typical measurements are transmission, reflection coefficients, time domain transfer.

Phase displays phase (no magnitude).

Group Delay displays signal transmission (propagation) time through a device.

Frequency Domain Format Bar

Smith Chart displays an impedance plane that is mapped onto the polar plane. Every point on the Smith Chart represents a complex impedance made up of a real resistance (r) and an imaginary reactance (r + jX). The dotted circles represent constant resistance. The horizontal line through the middle is purely resistive (no reactive component).

- To the far right, the value is zero ohms (short). To the far left, the value is infinite ohms (open).

The dotted arcs represent constant reactance.

- The reactance arcs in the upper (positive) half of the circle represent inductive reactance ($Z_L = j\omega L$).
- The reactance arcs in the lower (negative) half of the circle represent capacitive reactance ($Z_C = 1/j\omega C$).
- Typical measurements: Impedance profile

When in this format, the cursor bar allows you to choose the cursor value in either Mag + Phase or Inductance style. Refer to "Frequency Domain Polar and Smith Chart Markers" on page 296 for additional information.

Polar Chart plots the measurement result in a vector representation.

The magnitude at any point is determined by its displacement from the center (or zero value). Magnitude is scaled linearly, with the value of the outer circle set to a ratio value of 1.

The radial lines scale the phase angle from 0 degrees to +180 degrees (counterclockwise) or –180 degrees (clockwise).

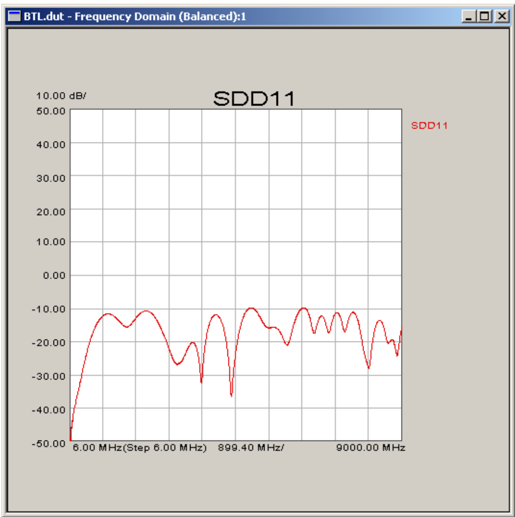
When in this format, the cursor bar allows you to choose the cursor value in either Mag + Phase or Inductance style. Refer to "Frequency Domain Polar and Smith Chart Markers" on page 296 for additional information.

Real displays only the real (resistive) portion of the measured complex data. Can show both positive and negative values. Typical measurement: time domain

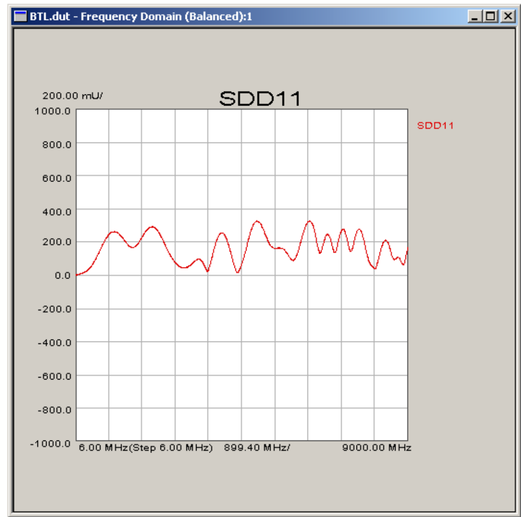
Imaginary displays only the imaginary (reactive) portion of the measured complex data. Typical measurements are impedance for designing matching circuits.

Table 4-1 Frequency Domain Formats

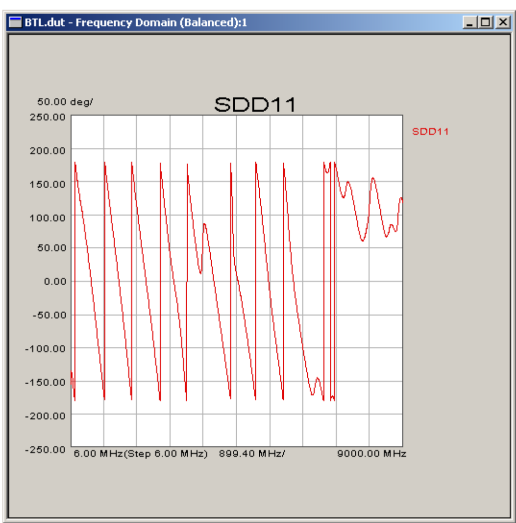
Log Mag



Linear Mag



Phase



Group Delay

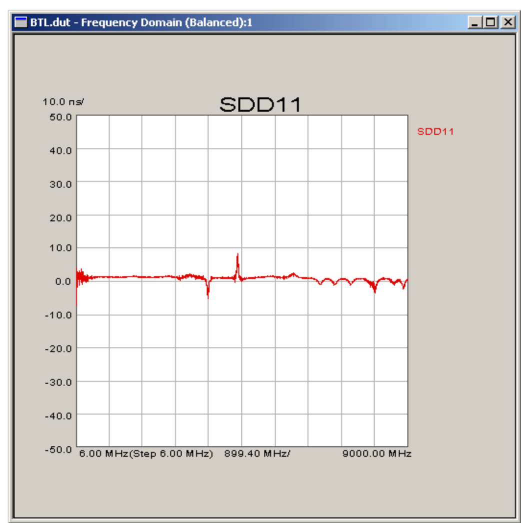
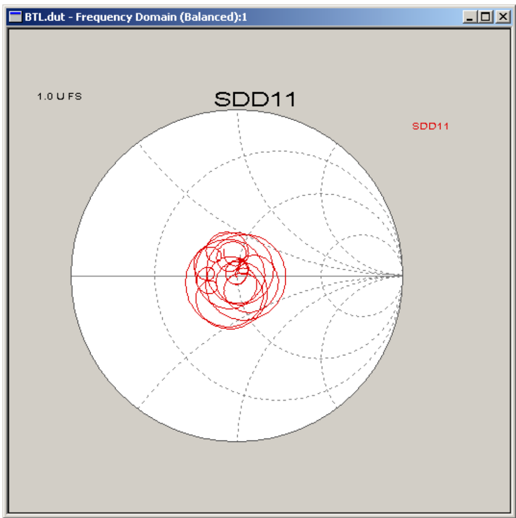
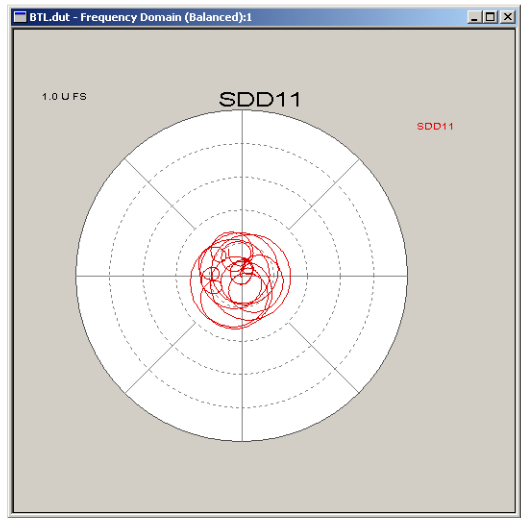
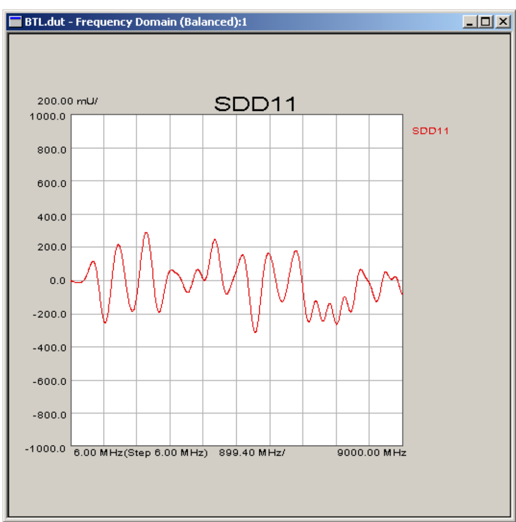
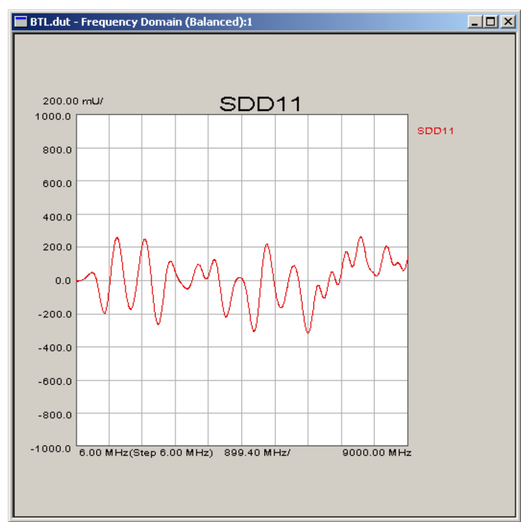


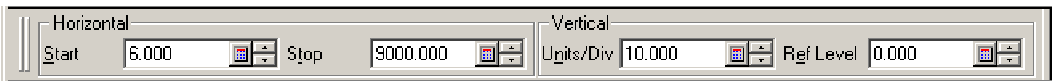
Table 4-1 Frequency Domain Formats

Smith Chart	Polar Chart
	
Real	**Imaginary**
	

Setting the Scale

The PLTS software allows you to change the horizontal and vertical scale of the plots using the **Scaling Bar**.

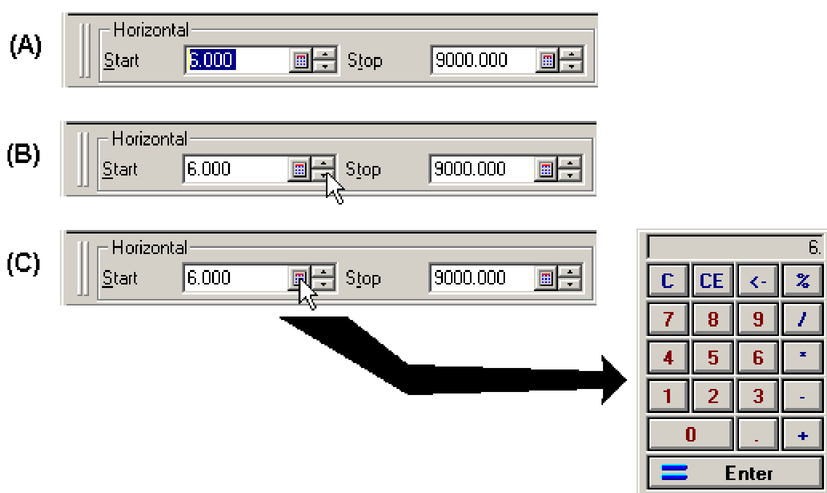
Figure 4-12 Scaling Bar



Change the Scaling Bar values by:

- Clicking and dragging within a scaling bar entry box to highlight the current value and then typing the new value. See (A) of Figure 4-13.
- Selecting the up/down arrow buttons to the right of each entry. See (B) of Figure 4-13.
- Selecting the calculator icon to the right of each entry to display a keypad. Click the keypad's numeric buttons to enter a new value and click the **Enter** button to save the new value. See (C) of Figure 4-13.

Figure 4-13 Entering a Scale Value



The horizontal scale is changed by changing the start and stop frequencies in megahertz (MHz). Note that you can not extend the start and stop frequencies beyond the start and stop frequencies used in the measurement. The horizontal scale may not be changed in Smith Chart and Polar formats.

The vertical scale units vary depending on the format.

Format	**Vertical Scale Units**
Log Mag	decibels (dB)
Linear Mag	mU
Phase	degrees
Group Delay	nanoseconds (nS)
Smith Chart	Not Applicable
Polar	Not Applicable
Real	mU
Imaginary	mU

Quick Scale Features

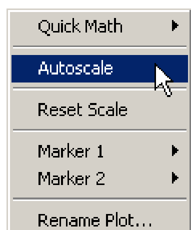
PLTS has three features that make scaling changes quickly and easily. The three features are:

- **Autoscale**
- **Reset Scale**
- **Copy Plot Format** used with **Paste Plot Format** (see "Copying and Pasting Plot Formats" on page 320)

Autoscale

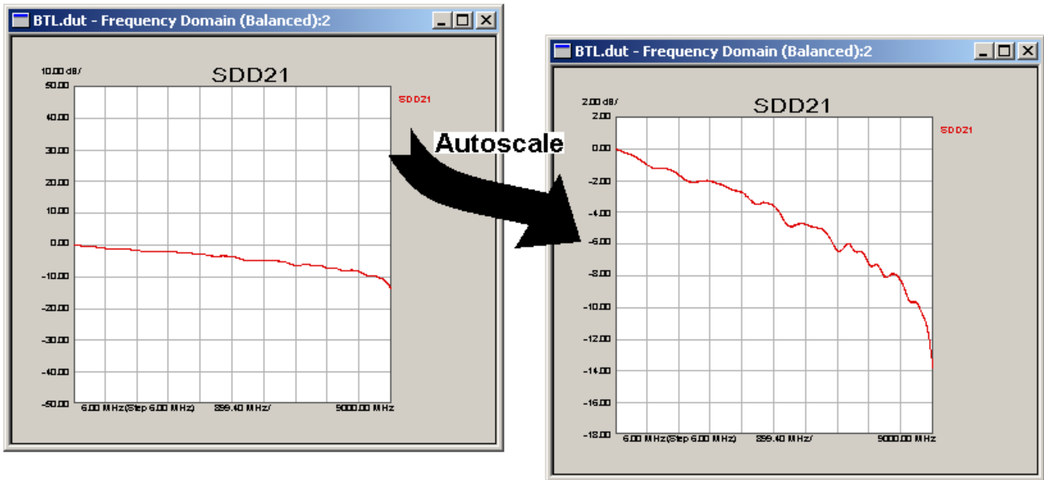
Autoscale changes the vertical scale of the active plot to allow the trace to occupy approximately 80% of the vertical axis of the display. It places the display such that the graticule values are numbers that are easy to work with.

Figure 4-14 Autoscale



To autoscale a plot, select the plot, then right click on the plot to display the quick menu displayed in Figure 4-16. Click **Autoscale** to change the vertical scale of the plot. Figure 4-15 shows a frequency domain plot that has **Autoscale** applied to it.

Figure 4-15 A Plot that has been Autoscaled

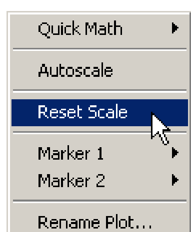


Reset Scale

Reset Scale resets the vertical and horizontal scale of the active plot to the default settings. This is useful when you are adjusting the scale and the trace is moved off screen and can no longer be seen.

To reset the scale of a plot, select the plot, then right click on the plot to display the quick menu displayed in Figure 4-16. Click **Reset Scale** to reset the plot to the default settings.

Figure 4-16 Reset Scale



5 Analyzing Data in the Time Domain

Physical layer test systems can provide measurement-based time domain information in two ways:

- Making the time domain measurements directly utilizing a Time Domain Reflectometer (TDR) to apply a synthesized step waveform to a DUT and observing the response.
- Making frequency domain measurements utilizing a vector network analyzer (VNA) and an S-parameter test set to sweep the DUT with an RF signal and measuring the RF response. Then the measured frequency domain information is converted to the time domain using the Inverse Fast Fourier Transform (IFFT).

  In a linear network, the Fourier Transform describes the relationship between a frequency domain measurement and its corresponding time domain response in detail. Therefore, given the measured frequency domain response of a DUT, it is possible to determine its time domain response mathematically by performing an inverse Fourier Transform. PLTS accomplishes the frequency domain transformation to time domain by utilizing the inverse chirp Z Fourier transform[1].

The type of information that can be observed in time domain mode is quite different than the information that can be observed in frequency domain mode. If the network is thought of in terms of its equivalent circuit model, then the frequency domain response describes the composite behavior of all of the circuit elements at any given operating frequency.

By contrast, the time-domain response shows the contribution of each individual circuit element. Since there is a direct relationship between time and distance, this mode allows each element to be separated spatially. With an understanding of the unique signature characteristics of different circuit elements, this view of the DUT can provide considerable insight into the device.

The advantages of using the PLTS measurement approach for TDR data are listed below. While the traditional TDR measurement technique provides fast measurement speed, the measurement technique used by PLTS provides:

- Superior accuracy
- Significantly better dynamic range (important for crosstalk and mode-conversion terms)
- Ability to de-embed fixtures and signal launchers
- Access to both frequency and time domain information (as vector quantities)
- Single setup for forward and reverse transmission and reflection, single-ended, differential-, and common-mode, and mode-conversion terms
- No need for DUT to have DC return path
- No large voltage steps applied to DUT

1. The advantage of the chirp z-transform is that it enables calculation of the sample of the z-transform equally spaced over an arc or a spiral contour with an arbitrary starting point and arbitrary frequency range. In contrast, the frequency range of the discrete Fourier Transform is strictly related to the sampling frequency.

TDR/TDT Mode

The time domain mode shows the contribution of each individual circuit element. Using time domain reflectance (TDR), you can measure the location, electrical length, nature of discontinuities (resistive, capacitive, inductive), and amount of reflection from discontinuities. Time domain transmission (TDT) response parameters typically measured are gain, propagation delay, and crosstalk between traces.

The physical layer test system can measure and display any of the single-ended (unbalanced) or mixed-mode S-parameters in the time domain and display the response of a device as if it were stimulated with either a step or an impulse waveform. For those not familiar with S-parameters, they are simply the energy that is reflected off of, or transmitted through, a device under test. S-parameters are defined as the ratio of two normalized power waves (response/stimulus), defined in terms of the voltages and current at each port of a device. For more information, see “How to Interpret S-Parameters” on page 130.

- In TDR/TDT mode, the horizontal axis displays:
  — Reflection parameters showing the characteristics of the DUT at a certain time delay into the device.
  — Transmission parameters showing the propagation delay through the device.
- The vertical axis displays:
  — An impulse response that is a reflection or transmission coefficient on either a linear or logarithmic scale. This parameter can be displayed as an absolute number, or relative to a minimum or maximum value of the response.
  — A step response on either a linear or a logarithmic scale. Alternatively, a reflection parameter can be displayed as impedance versus time rather than as a reflection coefficient.

The following table shows the relationship between frequency domain parameters to their time domain equivalents.

Table 5-1 Relationship of Frequency Domain Parameters to Time Domain Equivalents

Mode	Direction	Type	Parameter
Single-ended	N/A	TDR	S11
Single-ended	Reverse	TDT	S12
Single-ended	Reverse	TDT	S13

Table 5-1 Relationship of Frequency Domain Parameters to Time Domain Equivalents

Mode	Direction	Type	Parameter
Single-ended	Reverse	TDT	S14
Single-ended	Forward	TDT	S21
Single-ended	N/A	TDR	S22
Single-ended	Reverse	TDT	S23
Single-ended	Reverse	TDT	S24
Single-ended	Forward	TDT	S31
Single-ended	Forward	TDT	S32
Single-ended	N/A	TDR	S33
Single-ended	Reverse	TDT	S34
Single-ended	Forward	TDT	S41
Single-ended	Forward	TDT	S42
Single-ended	Forward	TDT	S43
Single-ended	N/A	TDR	S44
Differential	Forward	TDR	SDD11
Differential	Reverse	TDT	SDD12
Differential	Forward	TDT	SDD21
Differential	Reverse	TDR	SDD22
Common	Forward	TDR	SCC11
Common	Reverse	TDT	SCC12
Common	Forward	TDT	SCC21
Common	Reverse	TDR	SCC22
Differential-to-Common	Forward	TDR	SCD11
Differential-to-Common	Reverse	TDT	SCD12

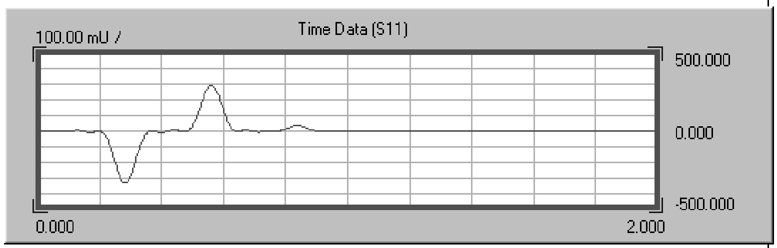
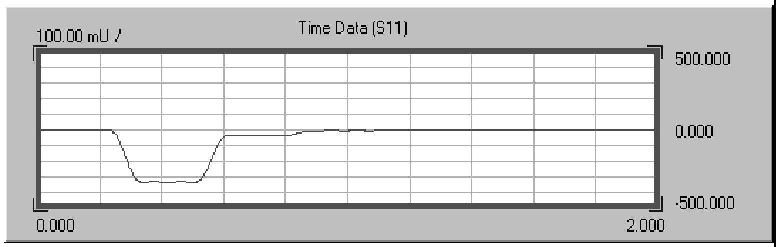
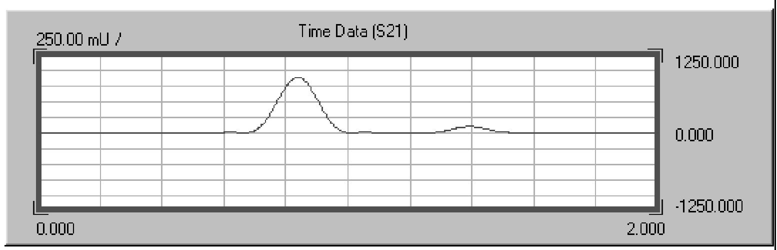
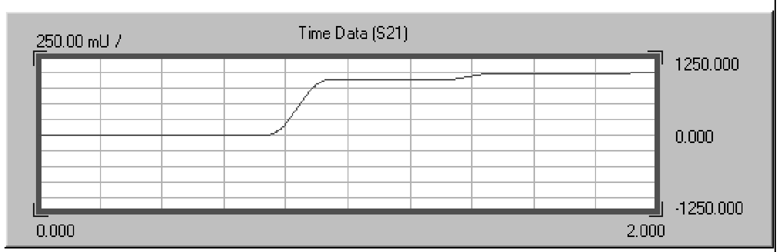
Table 5-1 Relationship of Frequency Domain Parameters to Time Domain Equivalents

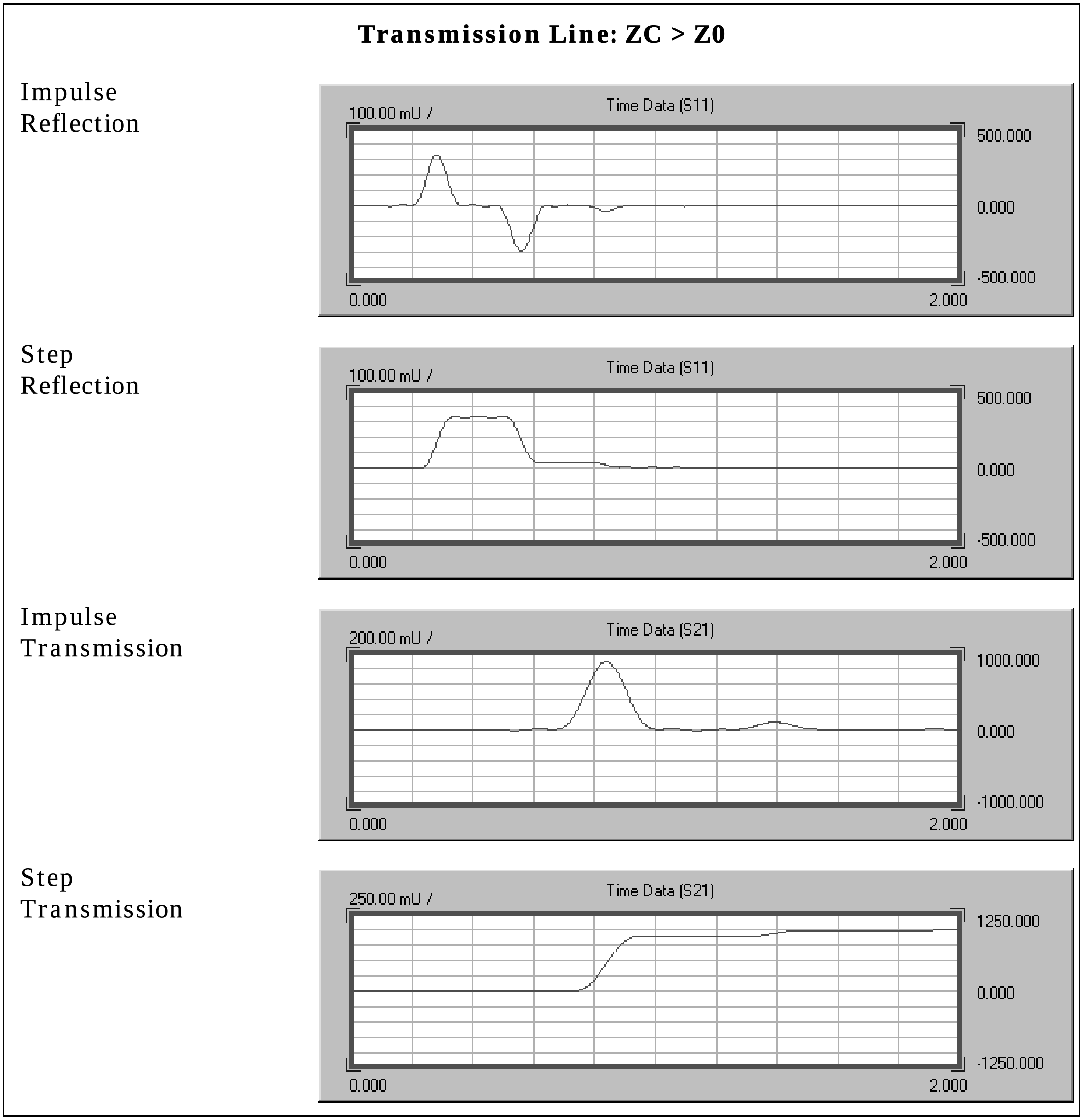
Mode	Direction	Type	Parameter
Differential-to-Common	Forward	TDT	SCD21
Differential-to-Common	Reverse	TDR	SCD22
Common-to-Differential	Forward	TDR	SDC11
Common-to-Differential	Reverse	TDT	SDC12
Common-to-Differential	Forward	TDT	SDC21
Common-to-Differential	Reverse	TDR	SDC22
Differential-to-Single-ended	Reverse	TDT	SSD12
Differential-to-Single-ended	Reverse	TDT	SSD13
Single-ended-to-Differential	Forward	TDT	SDS21
Single-ended-to-Differential	Forward	TDT	SDS31
Common-to-Single-ended	Reverse	TDT	SSC12
Common-to-Single-ended	Reverse	TDT	SSC13
Single-ended-to-Common	Forward	TDT	SCS21
Single-ended-to-Common	Forward	TDT	SCS31

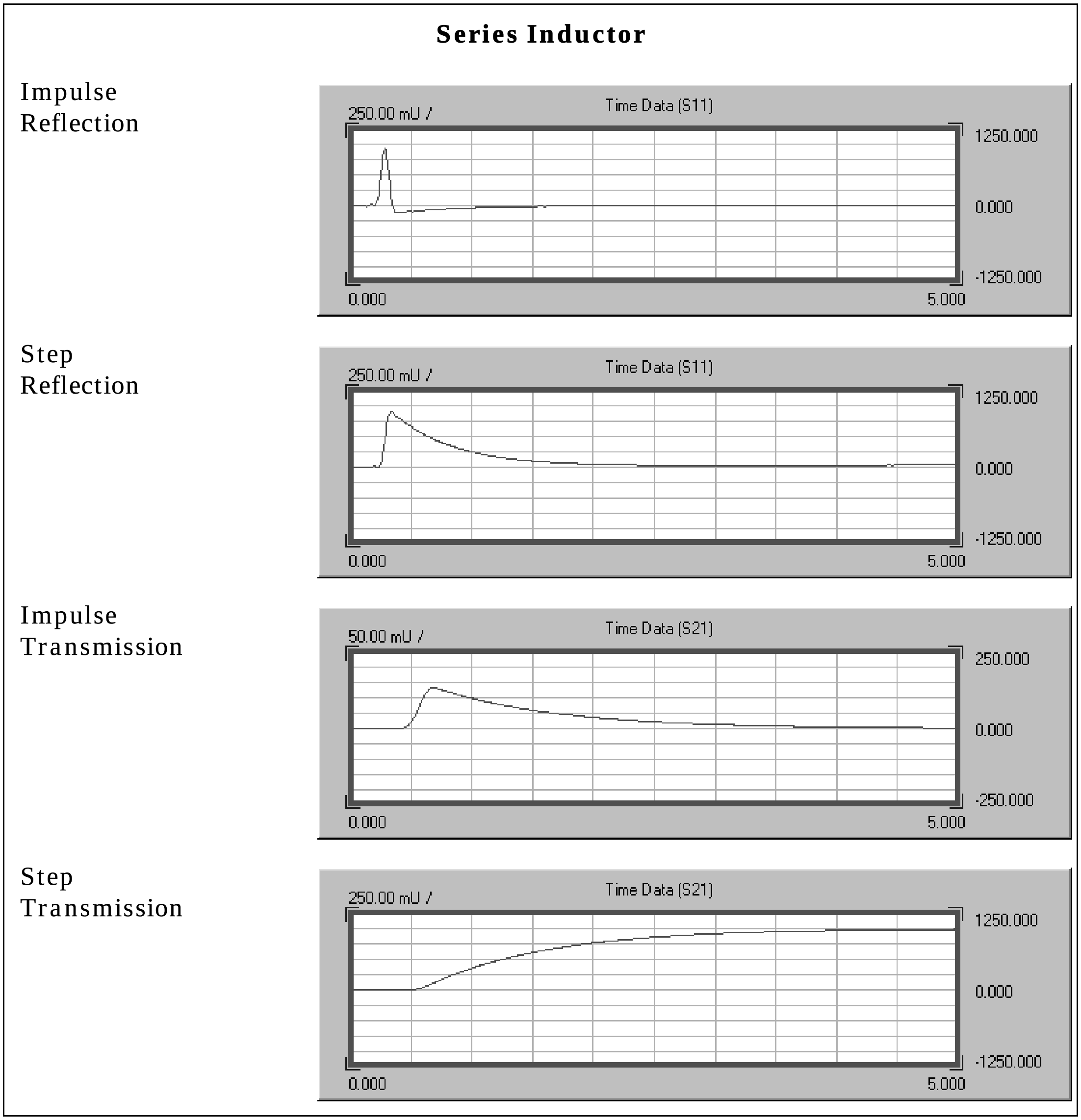
Analyzing Time-Domain Signatures

The time-domain response of a device, its signature, provides specific circuit detail. The shape of the response indicates the element type and configuration (series or shunt). Its value and location can be determined from the size of the reflection and its time delay. In general, a wider measurement bandwidth will provide finer response resolution. Table 5-2 shows various circuit elements and associated time-domain signatures.

Table 5-2 **Time Domain Signatures**

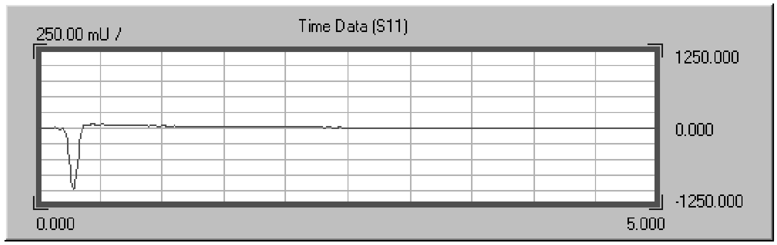
Transmission Line: ZC < Z0	
Impulse Reflection	
Step Reflection	
Impulse Transmission	
Step Transmission	


Transmission Line: ZC > Z0
Impulse Reflection
100.00 mU /
Time Data (S11)
500.000
0.000
-500.000
0.000
2.000
Step Reflection
100.00 mU /
Time Data (S11)
500.000
0.000
-500.000
0.000
2.000
Impulse Transmission
200.00 mU /
Time Data (S21)
1000.000
0.000
-1000.000
0.000
2.000
Step Transmission
250.00 mU /
Time Data (S21)
1250.000
0.000
-1250.000
0.000
2.000

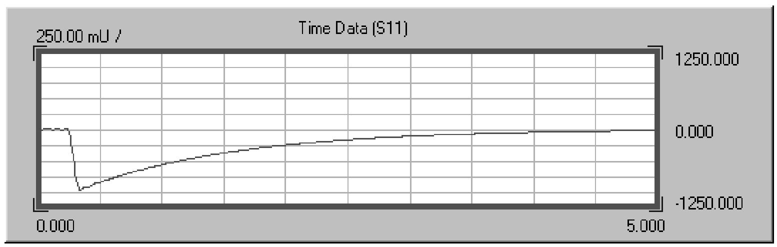

Series Inductor
Impulse Reflection
250.00 mU /
Time Data (S11)
1250.000
0.000
-1250.000
0.000
5.000
Step Reflection
250.00 mU /
Time Data (S11)
1250.000
0.000
-1250.000
0.000
5.000
Impulse Transmission
50.00 mU /
Time Data (S21)
250.000
0.000
-250.000
0.000
5.000
Step Transmission
250.00 mU /
Time Data (S21)
1250.000
0.000
-1250.000
0.000
5.000

Shunt Capacitor

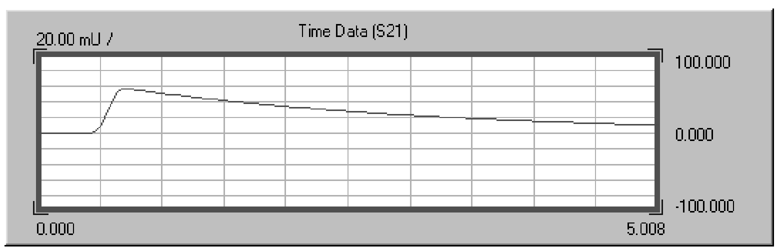
Impulse Reflection



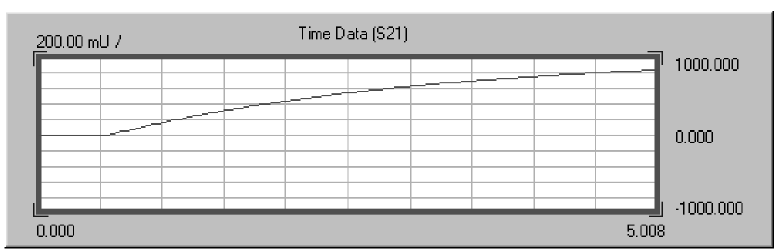
Step Reflection



Impulse Transmission



Step Transmission



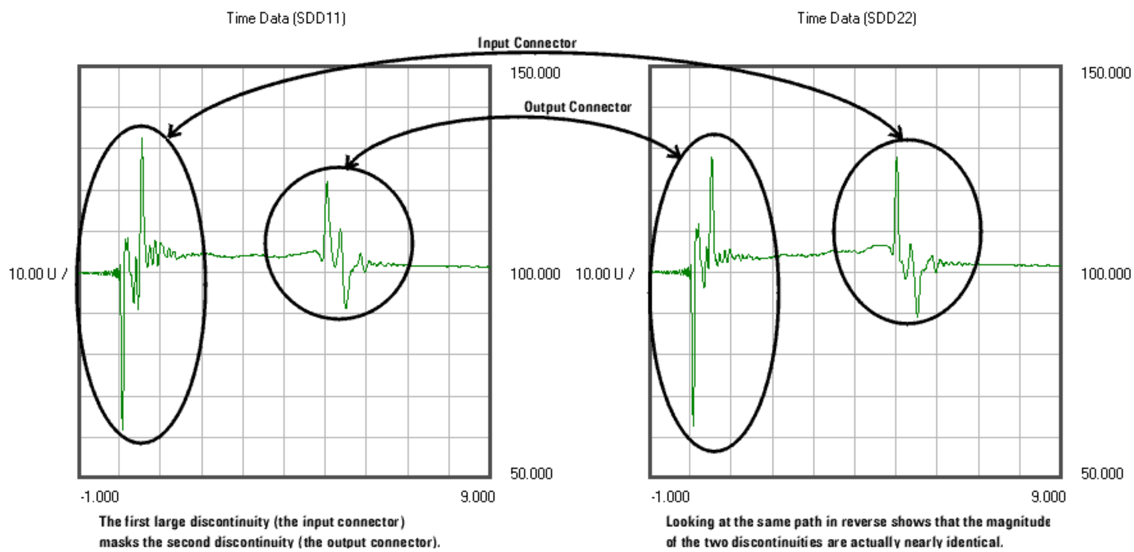
Practical Considerations

In general, better accuracy of the measured frequency domain data will provide for better accuracy of the time domain data. Using the step mode rather than the sweep mode provides additional frequency stability of the source which can greatly improve the time domain data. The following are several other important considerations.

Masking

Time domain responses are most accurate closest to the location of the source. A discontinuity in the DUT will reflect some power back to the source, meaning less power is transmitted to the rest of the DUT. This loss of power going away from the source is referred to as masking, and allows the true impedance of the next discontinuity to be misrepresented.

Figure 5-1 Masking Effects



Masking effects can be seen in Figure 5-1. The plot on the left shows the differential-mode input reflection of a device (SDD11). The first large discontinuity is the input connector; the second is the output connector. Because these connectors are physically identical, the apparent impedance difference between the two can be attributed to masking. The power level at the output connector has been decreased (masked) by the input connector. The plot on the right, output reflection (SDD22), proves this. Looking backwards into the device, the output

connector now exhibits the greater apparent impedance. Were it not for masking, these two plots, and the measured impedance of the input and output connectors, would be identical.

The high dynamic range of a VNA-based PLTS system extends the ability of the instrument to accurately characterize devices that have several discontinuities or high loss. TDR-based PLTS systems may not be as accurate.

Time Domain Windowing

The PLTS software has a feature called *Time Domain Windowing* that is designed to enhance Time Domain measurements. The need for windowing is due to the abrupt transitions in the Frequency Domain measurement at the Start and Stop frequencies. This band limiting of the frequency domain response causes overshoot and ringing in the Time Domain response. It causes the un-Windowed Impulse stimulus to have a sin (kt)/kt shape (k=π/frequency span), which has two effects that limit the usefulness of the Time Domain measurement:

- Finite Impulse Width limits the ability to resolve between two closely spaced responses. The effects of the finite impulse width cannot be improved with increasing the frequency span of the measurement.
- Impulse side lobes limit the dynamic range of the Time Domain measurement by hiding low-level response within the side lobes of the higher-level responses. The effects of side lobes can be improved by windowing.

Windowing improves the dynamic range of the Time Domain measurement by modifying (filtering) the Frequency Domain data prior to conversion to the Time Domain to produce an impulse stimulus with lower side lobes. This greatly enhances the effectiveness in viewing Time Domain responses that are very different in magnitude. The side lobe reduction is achieved, however, as the tradeoff with increased impulse width. The effect of windowing on the STEP stimulus is a reduction of overshoot and ringing while the tradeoff is increased rise time.

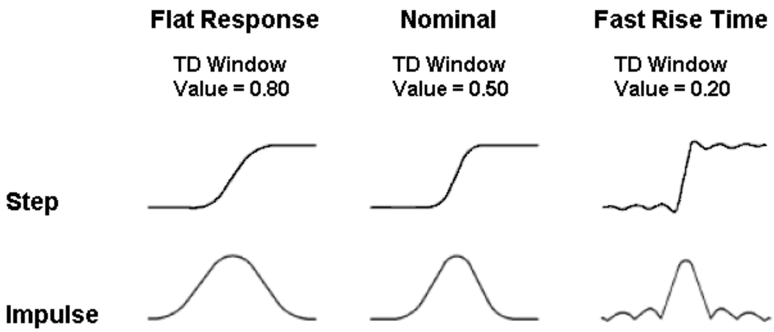
PLTS defines three window selections: **Flat Response**, **Nominal**, and **Fast Rise Time**. Each of these selections has an associated *Time Domain Window Value*. See Figure 5-3. The side lobe levels of the Time Domain stimulus depend only on the window that is selected.

PLTS also allows you to define your own window using the **Custom** selection where you will set you own *Time Domain Window Value.*

The purpose of windowing is to make the Time Domain response more useful in isolating and identifying individual responses. The window does not affect the displayed Frequency Domain response. The Time Domain function is only applied to time domain data. It does not affect frequency domain data. Figure 5-2 shows typical effects of windowing on the Time Domain response of the reflection measurement of a short circuit.

NOTE The windowing setting also has an effect on the transformation of eye diagrams.

Figure 5-2 Windowing Effect on Time Domain Responses of a Short Circuit



The following formula can be used to determine the equivalent 10 percent to 90 percent system rise time for the step function transformation and the eye diagram simulation:

$$T_r = (1000 / F_{max}) \times RTEC$$

Where, T_r = Rise time of the step response form 10% to 90% (in picoseconds)

1000 = Factor to convert frequency to picoseconds

F_{max} = Maximum stop frequency used in the measurement (in GHz)

RTEC = Rise time equation coefficient

As an example: System rise time = 27 ps = (1000/ 20 GHz) $\times$ 0.54 (see table below)

Table 5-3 shows the equivalent rise times for a maximum frequency of 20 GHz for each window setting.

Table 5-3 Equivalent Rise Times for a Maximum Frequency of a 20 GHz System

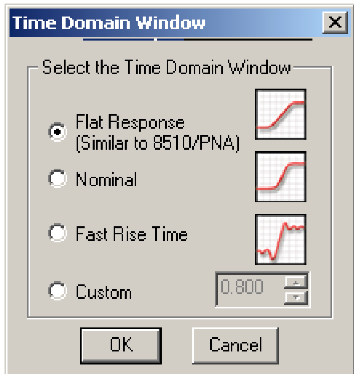
Window Selection	TD Window Value[a]	Maximum Frequency (F_{max})	Rise Time Equation Coefficient (RTEC)	Equivalent Rise Time (T_r)
Flat Response	0.80	20 GHz	0.91	45.5 ps
Nominal	0.50	20 GHz	0.71	35.5 ps
Fast Rise Time	0.20	20 GHz	0.54	27 ps

a. The relationship between the TD window value and the rise time equation coefficient is not made available.

By increasing or decreasing the filter value above or below the default, a trade-off can be made between rise time and side-lobe level (dynamic range).

When **Time Domain Window**... is selected from the **Tool** menu, the *Time Domain Window* dialog box is displayed allowing a choice of the three Time Domain Window settings.

Figure 5-3 Time Domain Window Dialog Box



- **Flat Response** gives the minimum side lobes and this provides the greatest dynamic range. The time domain window value is 0.80. The rise time equation coefficient value is 0.91.
- **Nominal** gives reduced side lobes and is normally the most useful. The time domain window value is 0.50. The rise time equation coefficient value is 0.71. This is the default setting.
- **Fast Rise Time** is essentially no window and therefore gives the highest side lobes. The time domain window value is 0.20. The rise time equation coefficient value is 0.54.
- **Custom** allows you to create your own time domain window. Set the time domain window value (range 0.000 to 1.000) by selecting the spinner or entering a value in the box.

NOTE When you open measurement data in time domain format, the previously selected windowing is used. To change the windowing selection, select Time Domain Window... from the Tool menu, which displays the *Time Domain Window* dialog box. Make your windowing selection from the dialog box and click OK. The new windowing choice is then applied to the data.

Response Resolution

The response resolution describes how close in time two responses can be distinguished. This depends on the width of the impulse response, which is inversely related to the measurement bandwidth. The relationship between the three is approximately R = T = 1.25/BW; where R is the response resolution in picoseconds, T is the effective impulse width in picoseconds, and BW is the frequency span in GHz.

Range Resolution

As described previously in "Analyzing Time-Domain Signatures" on page 151, the TDR signature provides specific circuit detail. Range resolution (TD span/Number of points, or Stop-Start/Number of points) will define how accurately the signature of a response can be identified. In general, a wider measurement bandwidth will provide finer spatial resolution.

To improve range resolution, zoom in on the section of interest and adjust the start- and stop-points to be as narrow as possible without compromising the agreement in the frequency domain.

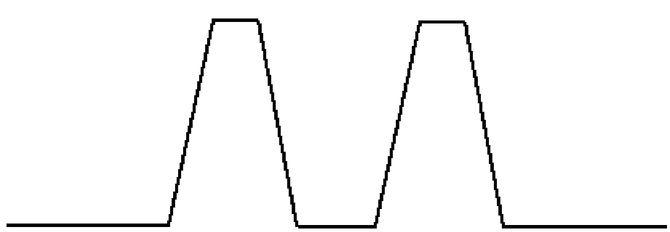
Spatial Resolution

When measuring short electrical devices, matching the spatial resolution to the minimum discontinuity distance is critical.

Impulse Response (IR) is the waveform that results at the output of a device when the input is excited by a unit impulse. As the maximum measurement frequency of a VNA increases, the pulse-width of the IR gets narrower. The pulse-width of the IR must be narrower than the two adjacent discontinuities in order to properly characterize the discontinuities in the time domain.

To illustrate this, assume we are measuring a device that has two discontinuities that are 1 cm apart and the expected response in the time-domain is as pictured in Figure 5-4.

Figure 5-4 Expected Response for Two Discontinuities 1 cm Apart



If the IR has a spatial resolution is greater than 1 cm and is swept across the DUT as shown in Figure 5-5, the response in the time domain is dramatically different than what is expected. It looks like the picture on the right side of the illustration because the IR is larger than the two adjacent discontinuities, and the power levels from the multiple discontinuities are being added together.

Figure 5-5 Example 1: Displayed Response

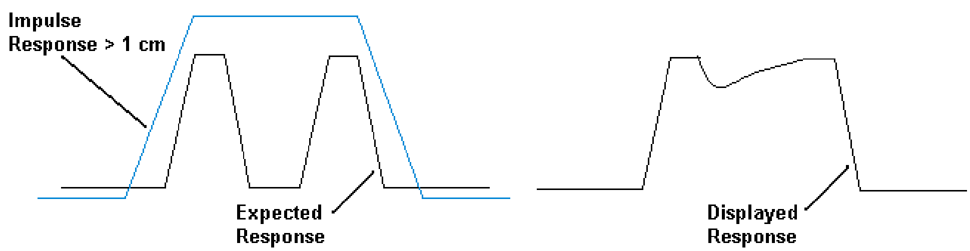
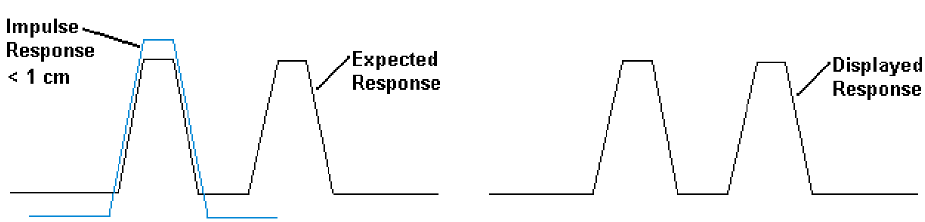


Figure 5-6 shows the same device using an IR with a pulse-width that is less than 1 cm. The response you see in the time domain is much more like what is expected. It looks like the picture on the right side of the illustration. This is because the IR is narrower than the two adjacent discontinuities, and the IR is able to capture only the power level of the individual discontinuities.

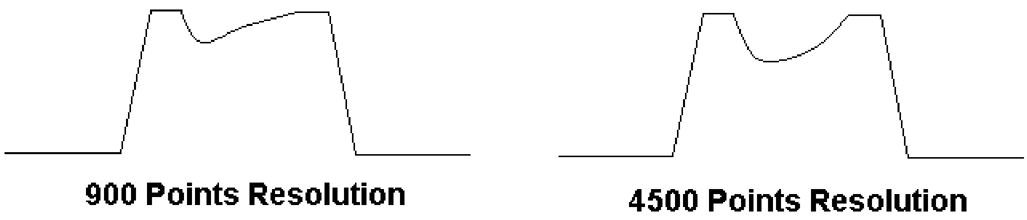
Figure 5-6 Example 2: Displayed Response



Understanding the spatial resolution requirements of a device is extremely important as devices become shorter in length.

An increased number of points can help provide *some* additional resolution to data, but will never make up for using a system with insufficient spatial resolution. Figure 5-7 shows the same device but measured with the number of points set at 900 and at 4500. While there is some improvement, the improvement is not significantly better.

Figure 5-7 **Sampling Resolution: 900 Points Versus 4500 Points**



The idea to remember is that the spatial resolution of the PLTS system ***must*** be a narrower length than the expected minimum length of any adjacent discontinuities on the device.

Automated Start and Stop Settings In Time Domain

For Measurements Made in the Frequency Domain Only

When a measurement is made in the frequency domain and is converted to the time domain, the time domain start and stop frequencies can be ambiguous. This describes the algorithm for displaying the start and stop frequencies using the PLTS automated process.

This algorithm is best described using a flow diagram (see Figure 5-8) and a few real life examples of this process.

Example 5-1 **Standard Transmission Measurement**

Parameter = SDD21 (Transmission)

Location of the SDD21 Peak Value in Time Domain = 2 ns

At Point 1,

Tstart = 0 ns

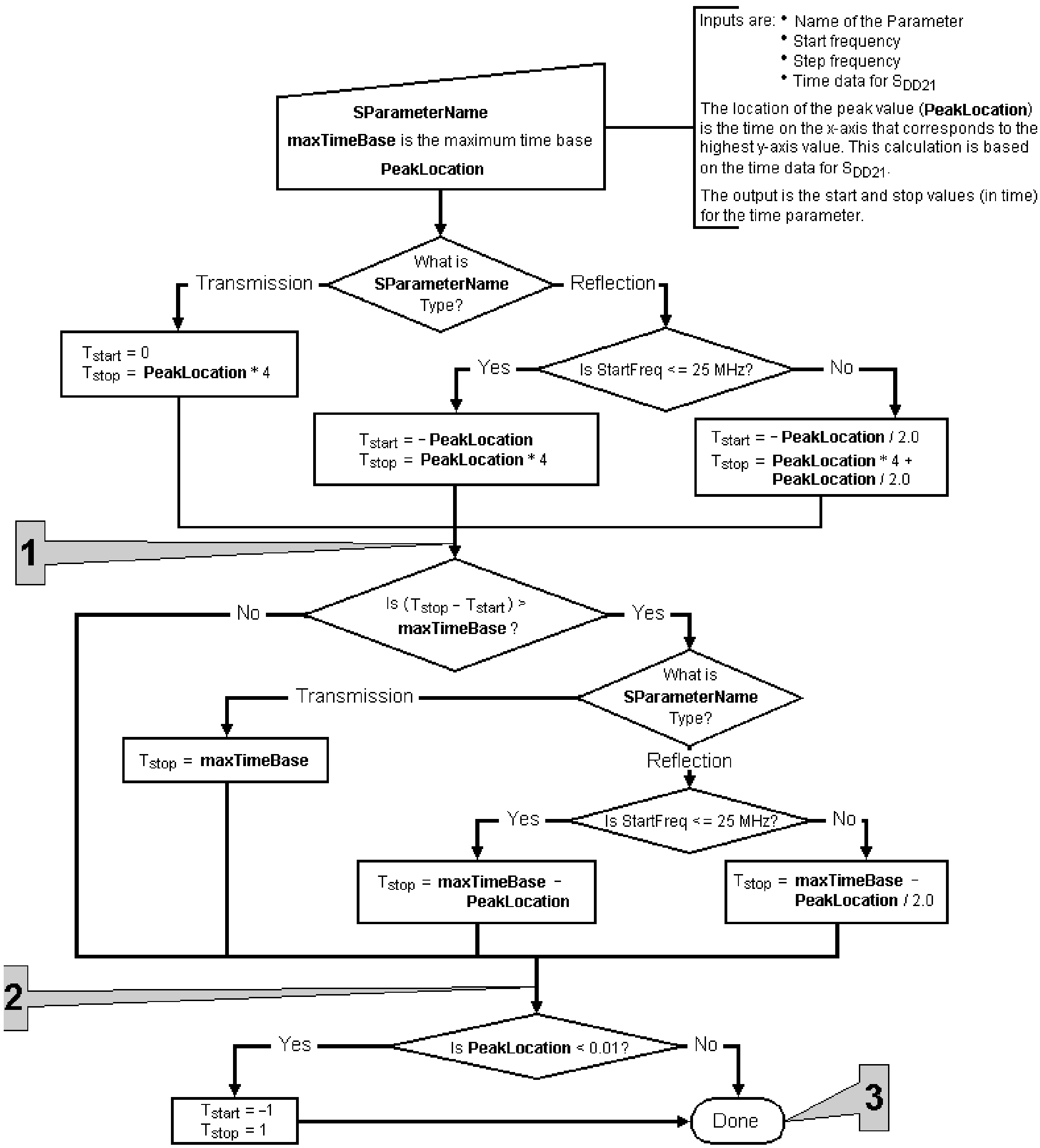
Tstop = 8 ns

The result is:

Tstart = 0 ns

Tstop = 8 ns

Figure 5-8 Automated Start and Stop Algorithm Flow Diagram



Also using Figure 5-8, you can follow these examples:

Example 5-2 Standard Reflection Measurement

Parameter = SDD11 (Reflection)

Location of the SDD21 Peak Value in Time Domain = 2 ns

At Point 1,

Tstart	= –1 ns
Tstop	= 4 + 1 = 9 ns

The result is:

Tstart	= –1 ns
Tstop	= 4 + 1 = 9 ns

Example 5-3 Potential Limitation of the Algorithm

Parameter = SDD11 (Reflection)

Location of the SDD21 Peak Value in Time Domain = 5 ns

At Point 1,

Tstart	= –2.5 ns
Tstop	= 20 + 2.5 = 22.5 ns (greater than maximum time base of 10 ns)

At Point 2,

Tstart	= –2.5 ns
Tstop	= 10 – 2.5 = 7.5 ns

The result is:

Tstart	= –2.5 ns
Tstop	= 7.5 ns

Comments: The results in the start and stop calculations are not suitable to resolve the SDD21 with a 5 ns measurement. This is because the stop time is 7.5 ns while the device needs 10 ns for the two-way travel of in reflection. The system is capable of resolving 10 ns, however, due to the current calculation of the -2.5 ns start time, an artificial limitation has been imposed. This can be worked around by manually changing the start and stop in the gating mode.

Example 9-4 No Transmission Data

Parameter = SDD11 (Any parameter)

Location of the SDD21 Peak Value in Time Domain = 0 ns

At Point 1,

Tstart	= 0 ns
Tstop	= 0 ns

At Point 3,

Tstart	= –1 ns
Tstop	= 1 ns

The result is:

Tstart	= –1 ns
Tstop	= 1 ns

Comments: Due to the absence of a strong signal in SDD21, the algorithm defaults to start of –1.0 ns and stop of 1.0 ns. This can be worked around by manually changing the start and stop in the gating mode.

Uncoupled Start Frequency and Step Frequency Setting

The Uncouple Start Frequency and Step Frequency selection is available in the *Calibrate Hardware for Measurement* dialog box for the VNA-based measurement setup when the Advanced button has been selected. Normally, the Frequency Step setting is locked (or coupled) to the Frequency Start setting.

However, there are situations that you may want to uncouple the **Frequency Step** setting from the **Frequency Start** setting. For example, uncouple the **Frequency Step** setting from the **Frequency Start** setting if you want to measure more points for better measurement resolution or you may want to measure specific frequency points that would not ordinarily be measured.

When measured data from this uncoupled start frequency/step frequency measurement is converted to the Time Domain analysis type, conversion rules that change the data must be applied to ensure the data is displayed correctly. See “Uncouple Start Frequency and Step Frequency” on page 11 for additional information regarding each of these conversion rules.

When these conversion rules change the measured data to be displayed in the Time Domain analysis type, one of three icons is displayed in the lower right corner of the plot to notify you of the change. These icons are displayed in Table 5-4.

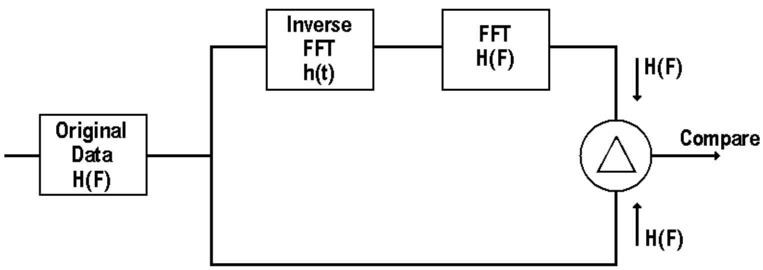
Table 5-4 Time Domain/Eye Diagram Uncoupled Start Frequency/Step Frequency Icons

Icon	Icon Name
	**Resampled Data -** This icon indicates that only the harmonically-related data points were used to generate time domain data.
	**Interpolated Data -** This icon indicates interpolation is performed to calculate the harmonically-related points that were not measured. Any measurements that were performed at harmonically-related points are left unchanged. The interpolated data is used, along with the measured harmonically-related data, to perform the Inverse Fast Fourier Transform (IFFT) for the calculated time domain data.
	**Bad Data** – This icon indicates there are less than 10 harmonically-related data points in the measured data. In this case, all of the non harmonically-related data is used to perform the Inverse Fast Fourier Transform (IFFT) to calculate time domain data. This may result in inaccurate time domain data.

Checking the Validity of a Time-Domain Calculation

There are a number of practical considerations in examining time domain data, as described previously. Therefore, it is very important to have a method of validating the data. This can be accomplished by comparing the original frequency domain data to the data after it is inverse Fourier transformed into the time domain, and then Fourier transformed back into the frequency domain, as shown in Figure 5-9. Ideally, these data should be identical. Changing the time domain start- and stop-points, the filter value, and the value of the DC parameter may improve the agreement.

Figure 5-9 Time Domain Data Validation Model



Viewing Data in the Time Domain

This section guides you with opening measured data in the time domain and viewing the data in the way that best suits your requirements. There are 16 parameter plots for both single-ended and balanced devices. You may elect to view any number of these plots.

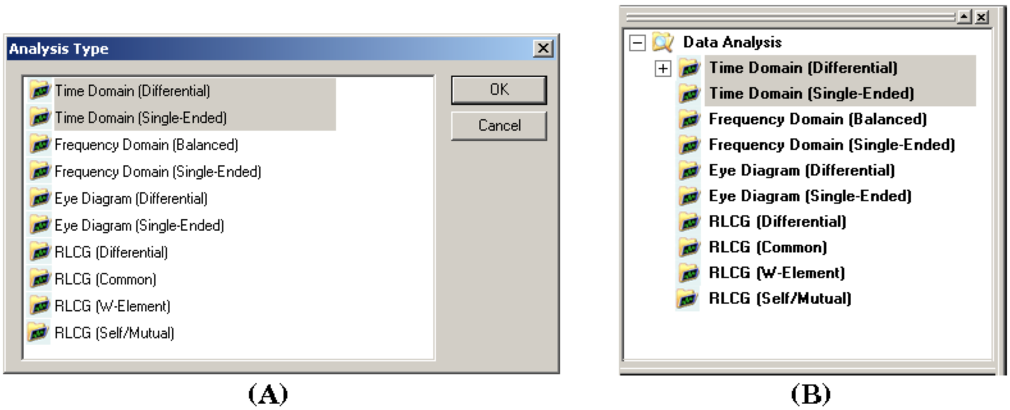
Opening a Time Domain Plot Window

The time domain may be viewed in either balanced or single-ended mode. You may open the time domain plot window in one of four ways.

- From the Startup Wizard immediately before selecting the **Measure** button where you must select the analysis type - see (A) of Figure 5-10
- From the Startup Wizard Welcome Screen by selecting **Load Measurement** where you must select the analysis type - see (A) of Figure 5-10
- From the **Open** selection in the **File** menu or the **Open** icon in the **Toolbar** where you must select the analysis type - see (A) of Figure 5-10
- From the **Browser** when data is already being viewed in another analysis type by selecting one of the Time Domain choices - see (B) of Figure 5-10

NOTE Using the **Browser** method to open a window requires that you select the parameters that you want to view from the **Parameter Bar** or the **Data** menu when the blank plot window is displayed.

Figure 5-10 Opening the Time Domain Plot Window

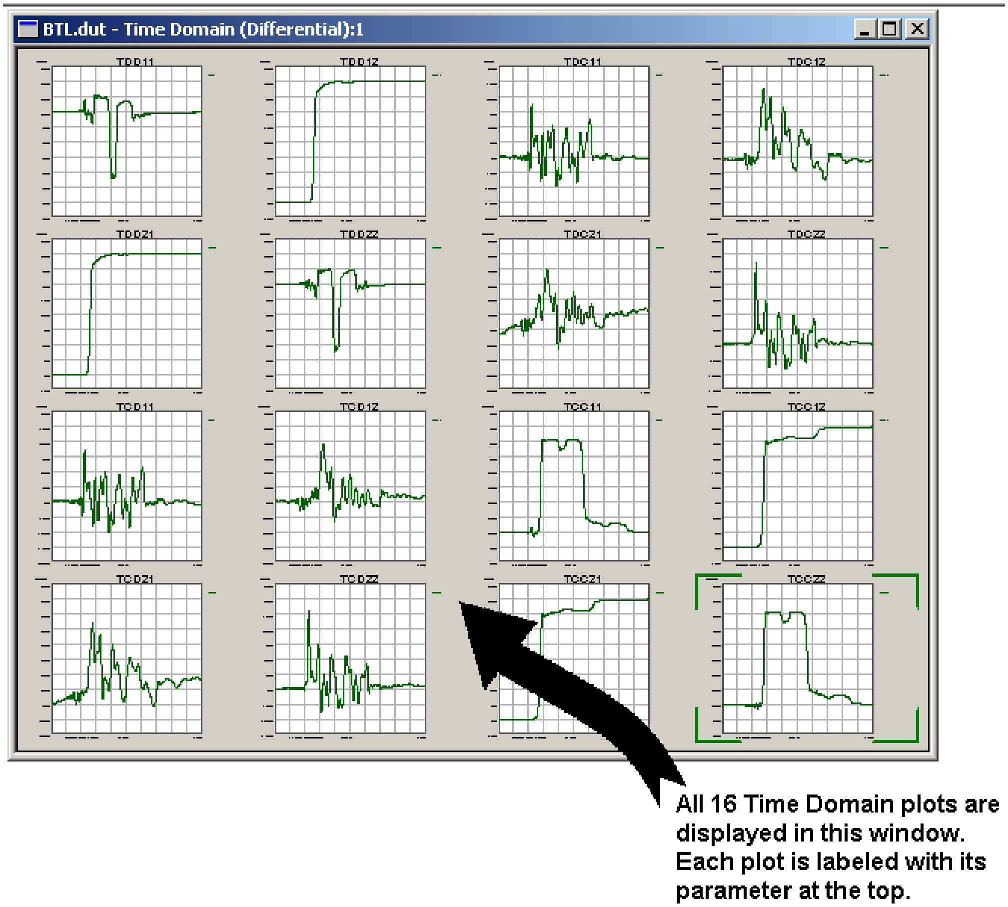


Viewing All 16 Parameters

In all cases, except when you open the plot window using the **Browser**, all 16 time domain parameter plots are displayed. Each of the plots are labeled at the top with their parameter. See Figure 5-11.

As mentioned previously, when you open the plot window from the **Browser**, an empty plot window is displayed. View all 16 plots by selecting **All** from the **Parameter Bar** or from the **Data** menu with **New Plot** selected.

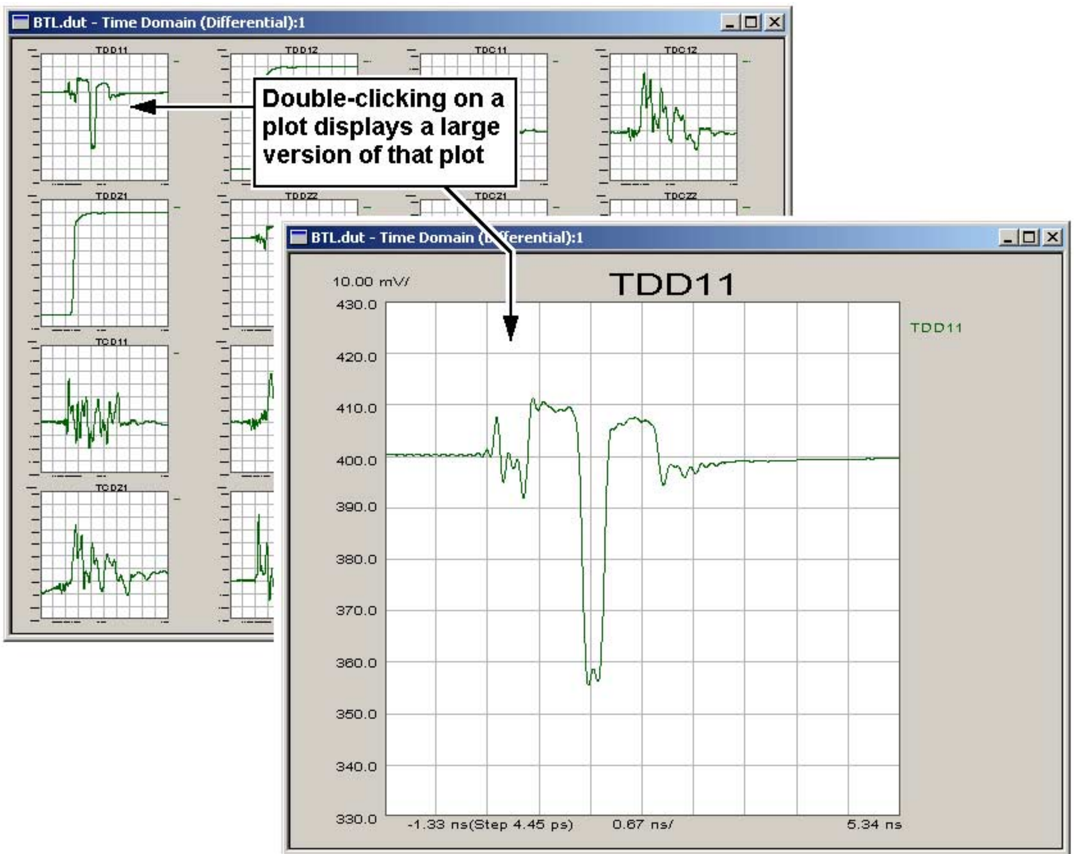
Figure 5-11 All 16 Differential Time Domain Plots



Viewing a Single Parameter

Viewing a single plot gives better resolution. To display a single plot, from the window with all 16 plots (or with multiple plots if you have a custom plot window displayed), double-click on the plot that you would like to view. See Figure 5-12.

Figure 5-12 Opening a Single Plot

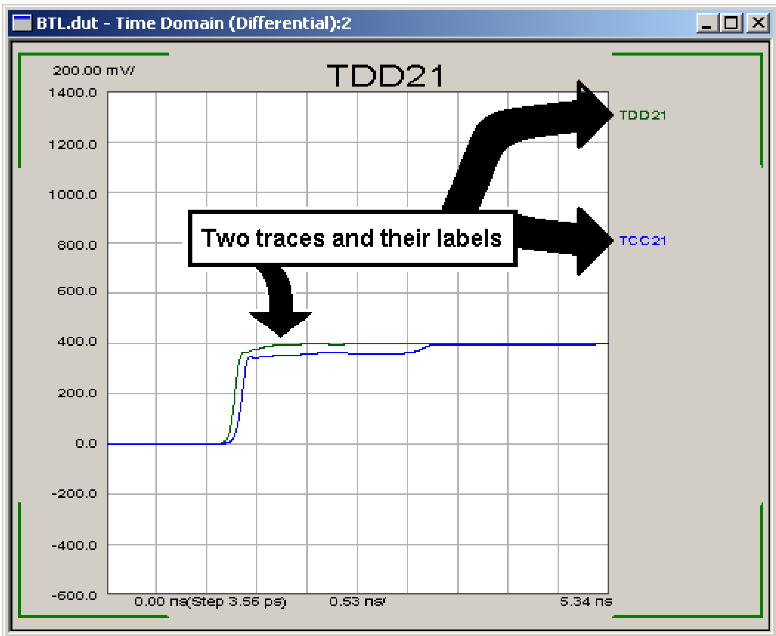


Double-click on the single plot to return back to the original view with multiple plots.

View Multiple Traces on a Single Plot

This single plot may have just one parameter or if you choose, it could contain up to four parameters. For example, you could show how TDD21 compares with TCC21. To do this double-click on the first plot so that is now displaying a single plot similar to Figure 5-12. For this example, TDD21 was double-clicked and is displayed as a single plot. With **New Trace** selected in the **Parameter Bar** (or the **Data** menu), click the remaining parameters (TCC21 in this example).

Figure 5-13 A Single Plot with Multiple Traces

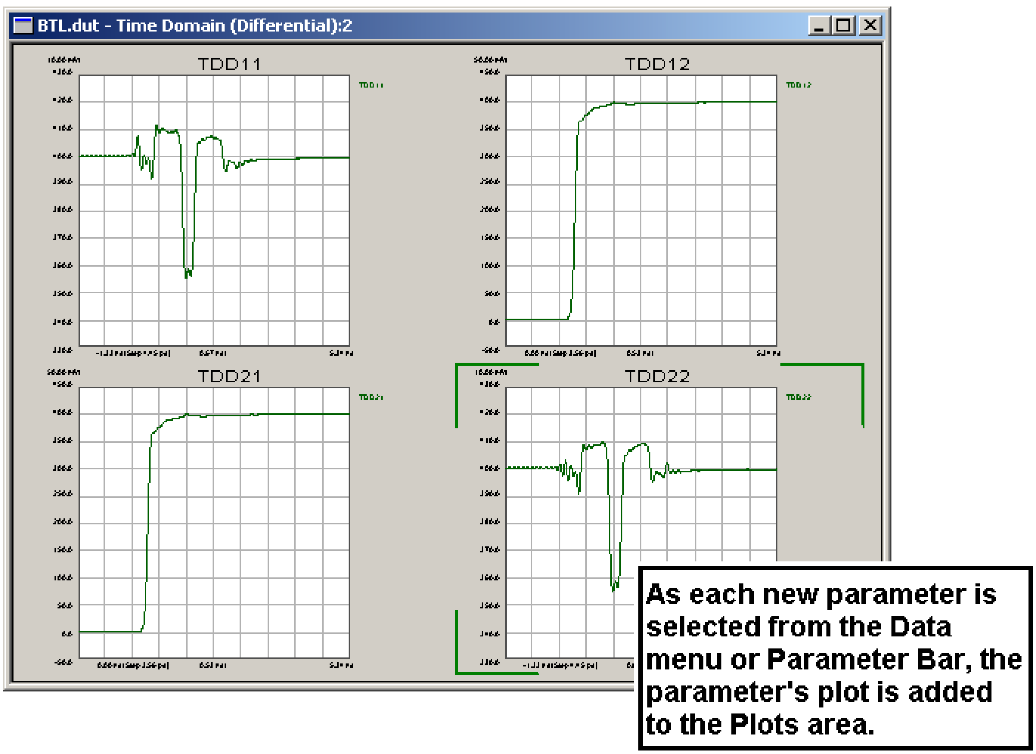


It could contain as many as 4 of the parameters. All trace parameters are labeled at the right edge of the plot. The color of the trace labels are the same color as the traces they represent.

Creating a Custom Time Domain Plots Window

You can also create a plot window with just the plots you desire. For example, you may want your plot window to show just the four TDD*xx* plots. To create this custom window, open the measured data file in any analysis type. Then, in the **Browser**, select the data type that you want to display the plots. In this example, select **Time Domain (Balanced)**. A blank plots window is displayed. With **New Plot** selected in the **Parameter Bar** (or the **Data** menu), click the desired parameters (TDD11, TDD12, TDD21, and TDD22 in this example). As each parameter is selected, the new plot is added to the plots window. See Figure 5-14.

Figure 5-14 Custom Time Domain Plots Window with Four Plots



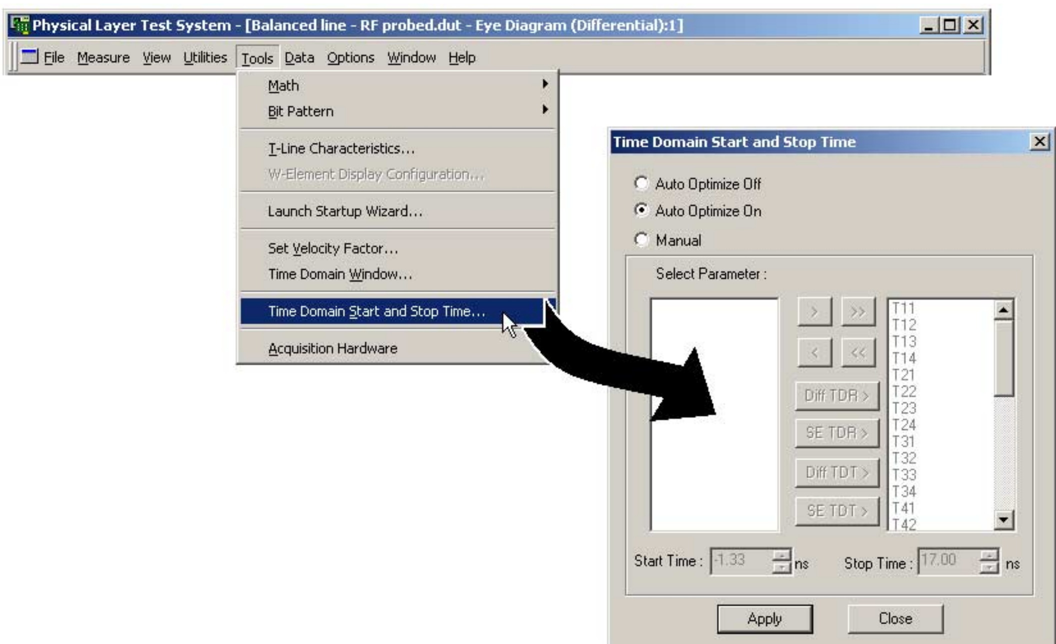
Optimizing the Time Domain Time Scale for Viewing

The *Time Domain Start and Stop Time* dialog box allows you to change and view the start times and stop times of time domain plots three ways.

NOTE The feature is used only for frequency domain S-parameter files that are calculated to show time domain. If your measurement was taken using a TDR, this feature does not apply.

From the **Tools** menu, select **Time Domain Start and Stop Time**... to open the *Time Domain Start and Stop Time* dialog box.

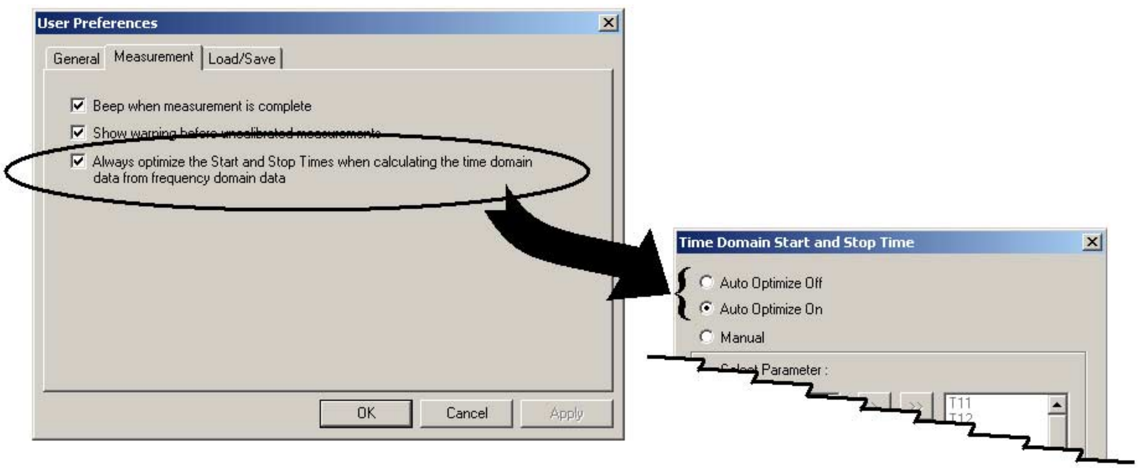
Figure 5-15 Opening the Time Domain Start and Stop Time Dialog Box



NOTE While most dialog boxes will not allow you to work between them and the main PLTS window, the *Time Domain Start and Stop Time* dialog box will allow you to make changes and then go to the main PLTS window to see the change without closing the dialog box.

The *Time Domain Start and Stop Time* dialog box is displayed with either **Auto Optimize Off** or **Auto Optimize On** selected. The state depends on the **Always optimize the Start and Stop Times when calculating the time domain data from frequency domain data** selection in User Preferences dialog box dictates whether the Auto Optimize feature is turned on or off when the calculations are made from frequency domain to time domain. See Figure 5-16. Refer to "User Preferences" under the "Options Menu" section in the "Menu Reference" chapter of the *PLTS Installation and Reference Guide* for additional information.

Figure 5-16 User Preferences Dialog Box Determines if Auto Optimize is On



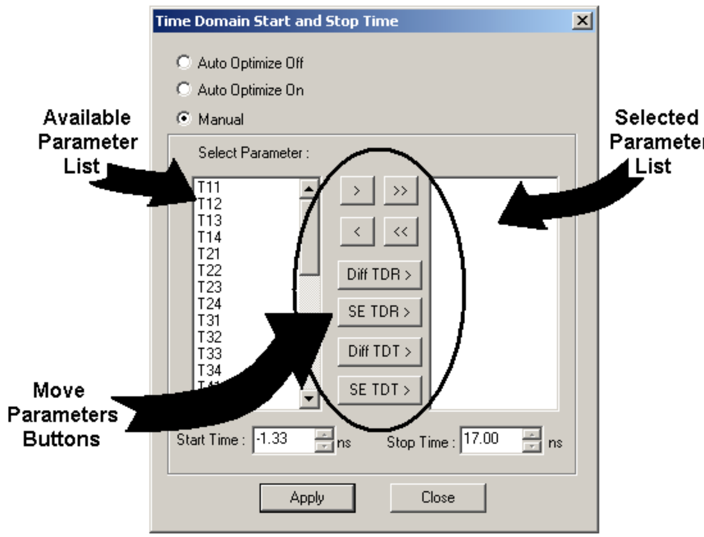
If **Auto Optimize Off** is selected, the time domain data is not optimized in any parameter for viewing and the start time and the stop time is set to show the full time range of the measurement. Often, no usable information is displayed over much of the time.

If **Auto Optimize On** is selected, the time domain data is optimized in all parameters for viewing and the start time and stop time are changed to show the time domain time range such that there is not a lot of unusable information displayed.

Changing the Start Time and Stop Time Manually

When **Manual** is selected, you can change as many parameters that you like to the start and stop time values that you choose. When **Manual** is selected, all differential and single-ended time domain parameters are placed in the list at the left, the **Available Parameter List**. Move the parameters that have a start or stop time that you would like to change to the right list at the right, the **Selected Parameter List**. Move the parameters using the eight **Move Parameters Buttons** located between the two lists. A description of each of these buttons is located below.

Figure 5-17 Time Domain Start and Stop Time Dialog Box in Manual Mode

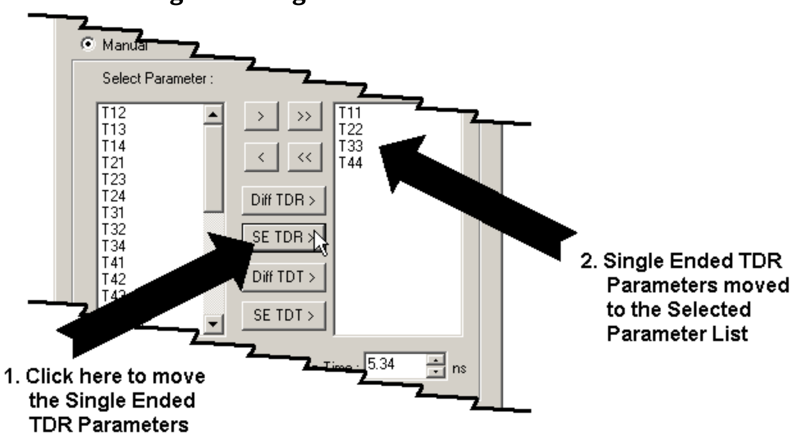


Button Label	Description
>	Moves the highlighted parameters in the available parameters list to the right into the selected parameters list
<	Moves the highlighted parameters in the selected parameters list to the left into the available parameters list
>>	Moves all parameters in the available parameters list to the right into the selected parameters list
<<	Moves all parameters in the selected parameters list to the left into the available parameters list

Button Label	Description
Diff TDR >	Moves all differential TDR parameters in the available parameters list to the right into the selected parameters list. The different TDR parameters are: TCC11, TCC22, TCD11, TCD22, TDC11, TDC22, TDD11, and TDD22.
SE TDR >	Moves all single-ended TDR parameters in the available parameters list to the right into the selected parameters list. The single-ended TDR parameters are: T11, T22, T33, and T44.
Diff TDT >	Moves all differential TDT parameters in the available parameters list to the right into the selected parameters list. The different TDT parameters are: TCC12, TCC21, TCD12, TCD21, TDC12, TDC21, TDD12, and TDD21.
SE TDT >	Moves all single-ended TDT parameters in the available parameters list to the right into the selected parameters list. The single-ended TDT parameters are: T12, T13, T14, T21, T23, T24, T31, T32, T34, T41, T42, and T43.

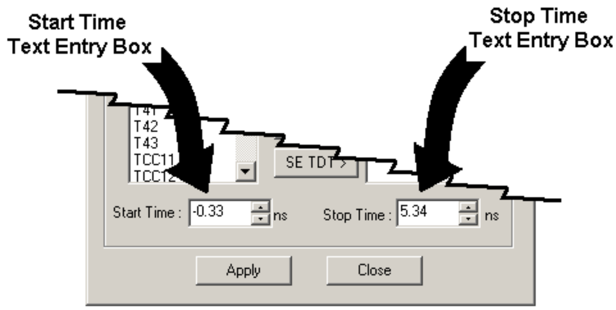
For example, to change the start and stop time of the single-ended TDR parameter plots, you must move the parameters to the **Selected Parameter List**. See Figure 5-18.

Figure 5-18 Moving the Single-Ended TDR Parameters



Once the parameters are listed in the **Selected Parameter List**, change the Start Time and the Stop Time using the **Start Time** and the **Stop Time** text entry boxes. See Figure 5-19.

Figure 5-19 Changing the Start Time and Stop Time



The times can be changed two ways:

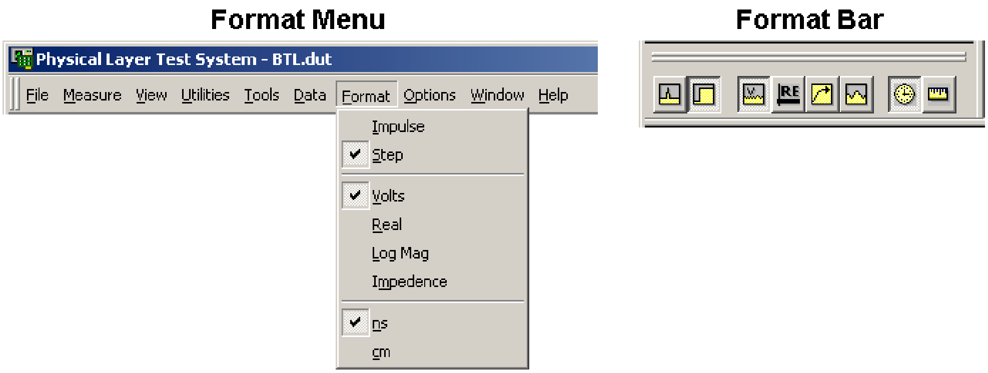
1. By typing in a new value for one or both of the times and then clicking the **Apply** button.
2. By dragging over the value to change with the mouse, and clicking the spinner to the new value. This method changes the value with clicking the **Apply** button.

NOTE The spinners in the method are unique. They increment or decrement only the most significant digit that was highlighted with the mouse. For example, the Stop Time in Figure 5-19 is 5.34 ns. If you highlight the entire “5.34” value or just the “5”, clicking the up spinner increments the “5”. However, if you highlight the “34” or only the “3”, clicking the up spinner increments the “3”.

Selecting Time Domain Display Formats

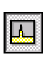
A data format is the way the physical layer test system presents the measurement graphically. Select a data format appropriate to the information you want to learn about the device. You may use either the **Format Bar** or the **Format** menu to select the format.

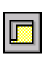
Figure 5-20 Format Menu and Format Bar for Time Domain



Time Domain Format Bar

Stimulus - Type of the input to the DUT


Impulse inputs an impulse waveform as the stimulus.


Step inputs a step waveform as the stimulus. This is the default format.

Vertical Format - Units used on the vertical axis


Volts selects volts as the vertical unit of measure. This is the default format.


Real displays only the real (resistive) portion of the measured complex data. Real can show both positive and negative values. This is the default format.


Log Mag displays Cartesian logarithmic magnitude (no phase) in dB. Typical measurements are return loss and gain.


Impedance selects ohms as the vertical unit of measure. This choice is active only for reflection plots with a Step stimulus.

Time Domain Format Bar

Horizontal Format - Units used on the horizontal axis

	**ns**	selects time units (in nanoseconds) for the horizontal format. This is the default format.
	**cm**	selects distance units (in centimeters) for the horizontal format.

NOTE When opening measured DUT files in Time Domain to view in the Time format

1. Go into the time domain/distance format and set your velocity factor accordingly (Example: Air VF=1.000, Surface traces VF=0.53146, Buried traces in a dielectric constant (ε_r) ~ 4.3: VF=0.48795).

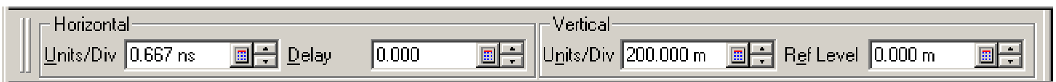
2. Select to view data in time, and then switch to view data in distance.

This triggers the correct computations to take place and you will now be able to set markers and measure device lengths and discontinuities accurately.

Setting the Scale

The PLTS software allows you to change the horizontal and vertical scale of the plots using the **Scaling Bar**.

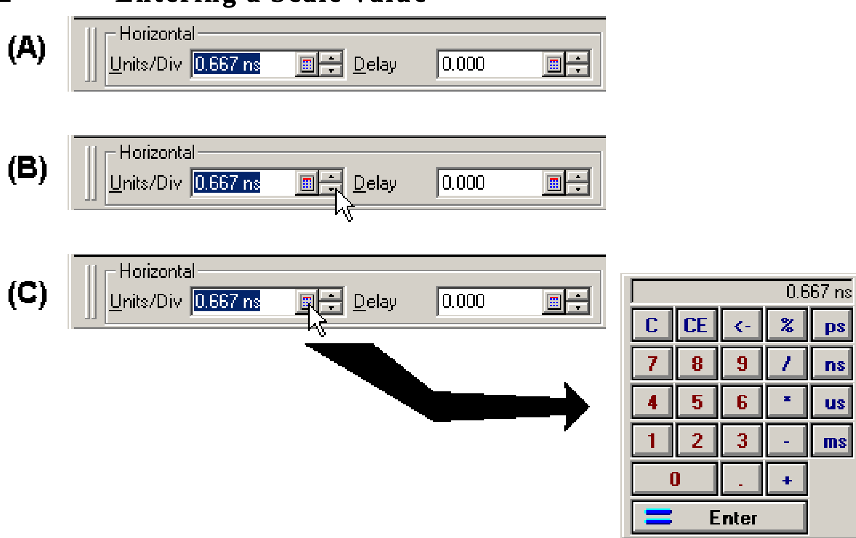
Figure 5-21 Time Domain Scaling Bar



Change the **Scaling Bar** values by:

- Clicking and dragging within a scaling bar entry box to highlight the current value and then typing the new value. See (A) of Figure 5-22.
- Selecting the up/down arrow buttons to the right of each entry. See (B) of Figure 5-22.
- Selecting the calculator icon to the right of each entry to display a keypad. Click the keypad's numeric buttons to enter a new value and click the **Enter** button to save the new value. See (C) of Figure 5-22. The scaling calculator icon varies slightly between scaling entries meet the requirements of the specific entry.

Figure 5-22 Entering a Scale Value



The horizontal scale is changed by changing the start and stop frequencies in megahertz

(MHz). Note that you can not extend the start and stop frequencies beyond the start and stop frequencies used in the measurement.

The horizontal scale units are either nanoseconds (ns) or centimeters (cm) depending on the **Format Bar** or **Format** menu selection.

NOTE When the horizontal units per division is changed, the Delay value is reset to zero.

The vertical scale is either volts, units, ohms, or decibels depending on the **Format Bar** or **Format** menu selection. The vertical scale selections are:

Volts	selects volts as the vertical unit of measure.
Real	displays only the real (resistive) portion of the measured complex data. Real can show both positive and negative values. This is the default format.
Log Mag	displays Cartesian logarithmic magnitude (no phase) in dB. Typical measurements are return loss and gain.
Impedance	selects ohms as the vertical unit of measure. This choice is active only for reflection plots with a Step stimulus.

Quick Scale Features

PLTS has three features that make scaling changes quickly and easily. The three features are:

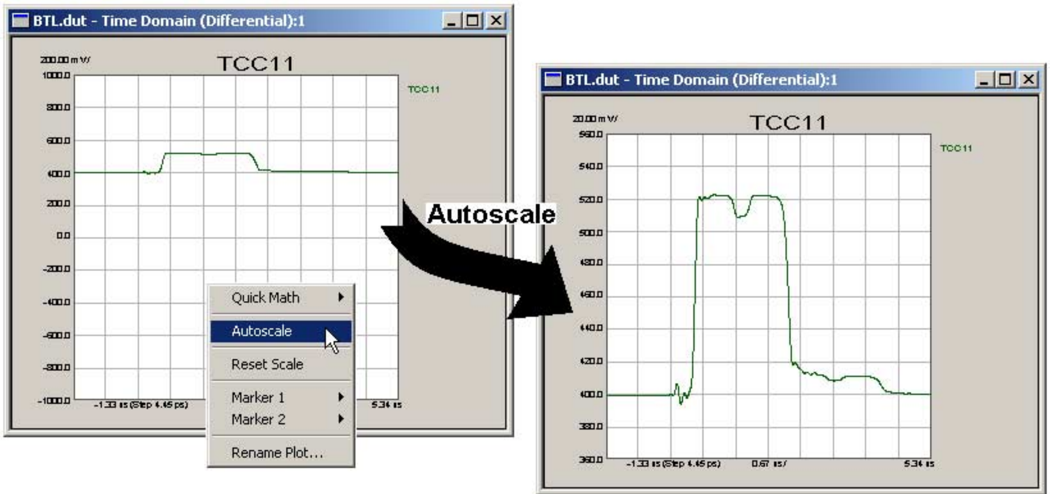
- **Autoscale**
- **Reset Scale**
- **Copy Plot Format** used with **Paste Plot Format** (see “Copying and Pasting Plot Formats” on page 320)

Autoscale

Autoscale changes the vertical scale of the active plot to allow the trace to occupy approximately 80% of the vertical axis of the display. It places the display such that the graticule values are numbers that are easy to work with.

To autoscale a plot, select the plot, then right click on the plot to display the quick menu. See Figure 5-23. Click **Autoscale** to change the vertical scale of the plot. The figure shows a time domain plot that has **Autoscale** applied to it.

Figure 5-23 Autoscale

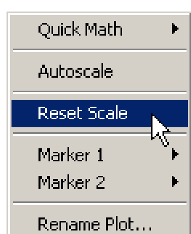


Reset Scale

Reset Scale resets the vertical and horizontal scale of the active plot to the default settings. This is useful when you are adjusting the scale and the trace is moved off screen and can no longer be seen.

To reset the scale of a plot, select the plot, then right click on the plot to display the quick menu displayed in Figure 5-24. Click **Reset Scale** to reset the plot to the default settings.

Figure 5-24 Reset Scale



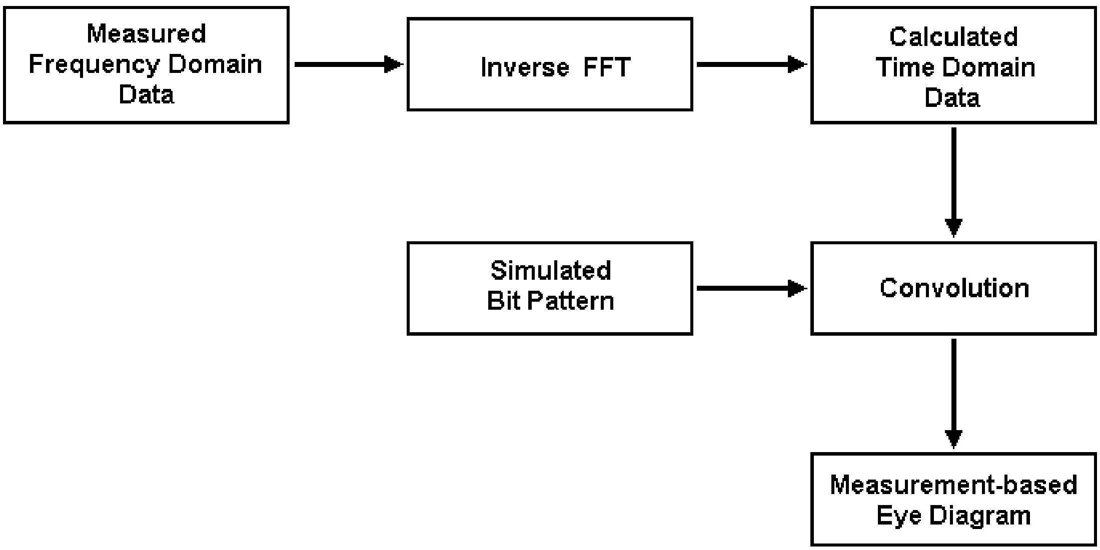
Gating

Gating provides the ability to remove the effect of a particular circuit element mathematically from time-domain plots. The gated section removes a section of the plot that you define, replacing it with an ideal transmission line having the same electrical delay as the removed section. 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. For detailed information on gating, refer to "Gating" on page 276.

6 Analyzing Data using Eye Diagrams

The physical layer test system software constructs measurement-based eye diagrams (or patterns) by convolving the calculated time domain impulse response (generated from frequency domain measurement data) with a simulated pattern of bit sequences. Refer to Figure 6-1 for a simplified block diagram of the eye diagram creation process.

Figure 6-1 Eye Diagram Process Simplified Block Diagram



With eye diagrams you can see signal quality with one display, you can diagnose problems, such as attenuation, noise, jitter, and dispersion that arise or characterize specific parts of the system. You can then view the measurement in the Time Domain mode to help isolate the source of the problem.

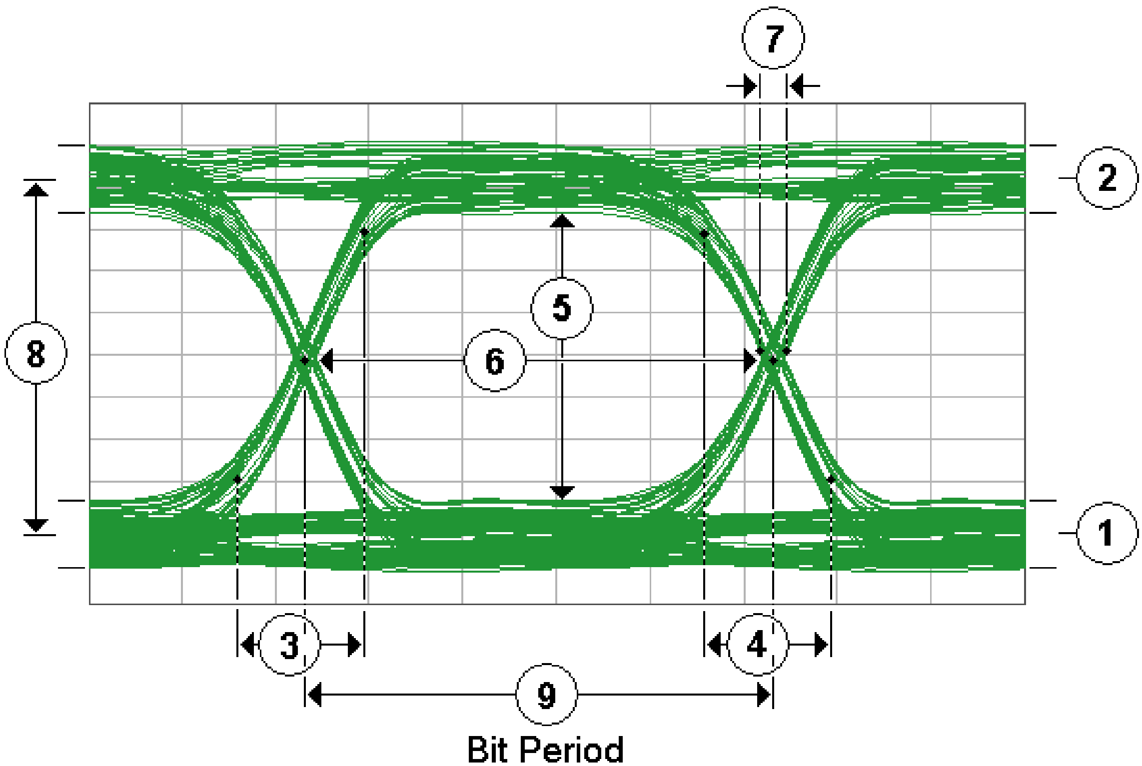
The software allows you to specify a simulated bit pattern (sequence) of between 8 and 32 bits that is convolved with the calculated time domain data to generate the eye diagram. You may select from three simulated bit patterns types: 1) a pre-defined bit pattern, 2) a bit pattern that you have defined, or 3) an arbitrary bit stream (a random-like bit pattern).

The Eye Diagram

The eye diagram shown in Figure 6-2 identifies key eye diagram definitions. These definitions are listed below this illustration. The eye diagram of the TCC21 parameter shown in Figure 6-2 was created using the BTL.dut file located in the PLTS data folder. An Arbitrary Bitstream bit pattern with the following settings was also used:

Rise/Fall Time = 0 ps	Data Rate = 2.5 Gb/s
Pattern Length = 32 bits	Number of Patterns = 12

Figure 6-2 The Eye Diagram



1	**Zero Level**	Zero Level is a measure of the mean value of the logical 0 of an eye diagram.
2	**One Level**	One Level is a measure of the mean value of the logical 1 of an eye diagram.

3	**Rise Time**	Rise time is a measure of the transition time of the data from the 10% level to the 90% level on the upward slope of an eye diagram.
4	**Fall Time**	Fall time is a measure of the transition time of the data from the 90% level to the 10% level on the downward slope of an eye diagram.
5	**Eye Height**	Eye height is a measure of the vertical opening of an eye diagram. An ideal eye opening would be measured from the one level to the zero level. However, noise on the eye will cause the eye to close. The eye height measurement determines eye closure due to noise.
6	**Eye Width**	Eye width is a measure of the horizontal opening of an eye diagram. Ideally, the eye width would be measured between the crossing points of the eye. However, jitter may appear on the waveform and influence the eye opening.
7	**Deterministic Jitter**	Deterministic jitter is the deviation of a transition from its ideal time caused by reflections relative to other transitions.
8	**Eye Amplitude**	Eye amplitude is the difference between the logic 1 level and the logic 0 level histogram mean values of an eye diagram.
9	**Bit Rate**	Bit rate (data rate) is the inverse of bit period (1 / bit period). The bit period is a measure of the horizontal opening of an eye diagram at the crossing points of the eye.

Time Domain Windowing

The PLTS software has a feature called *Time Domain Windowing* that is designed to enhance Time Domain measurements. The need for windowing is due to the abrupt transitions in the Frequency Domain measurement at the Start and Stop frequencies. This band limiting of the frequency domain response causes overshoot and ringing in the Time Domain response. It causes the un-Windowed Impulse stimulus to have a sin (kt)/kt shape (k=π/frequency span), which has two effects that limit the usefulness of the Time Domain measurement:

- Finite Impulse Width limits the ability to resolve between two closely spaced responses. The effects of the finite impulse width cannot be improved with increasing the frequency span of the measurement.
- Impulse side lobes limit the dynamic range of the Time Domain measurement by hiding low-level response within the side lobes of the higher-level responses. The effects of side lobes can be improved by windowing.

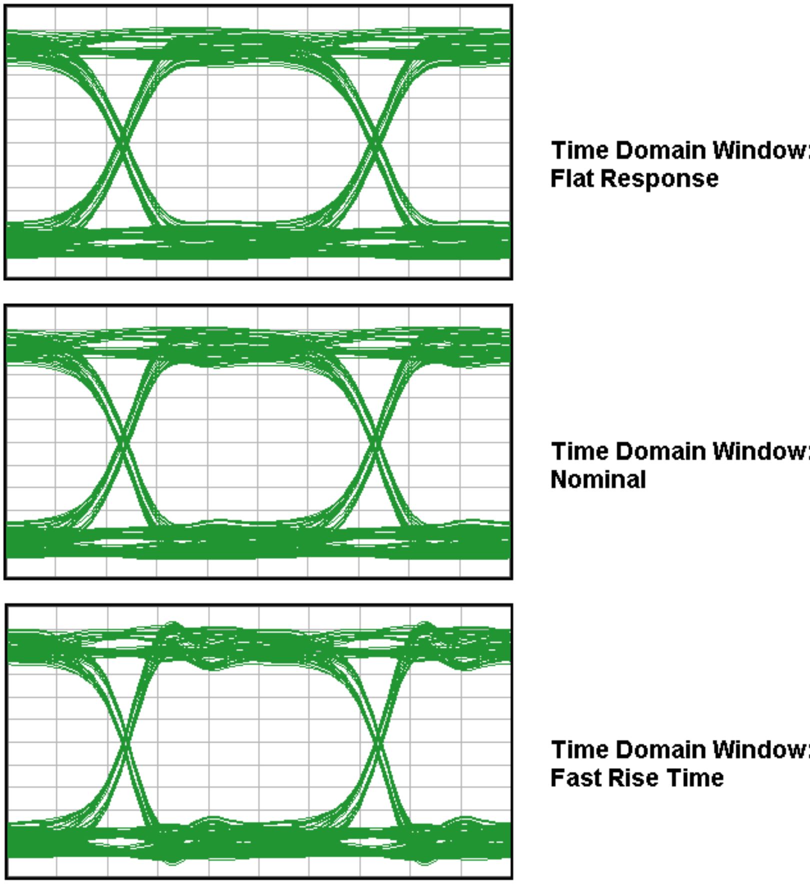
Windowing improves the dynamic range of the Time Domain measurement by modifying (filtering) the Frequency Domain data prior to conversion to the Time Domain to produce an impulse stimulus with lower side lobes. This greatly enhances the effectiveness in viewing Time Domain responses that are very different in magnitude. The side lobe reduction is achieved, however, as the tradeoff with increased impulse width. The effect of windowing on eye diagrams is a reduction of overshoot and ringing while the tradeoff is increased rise time.

PLTS defines three window selections: **Flat Response**, **Nominal**, and **Fast Rise Time**. Each of these selections has an associated *Time Domain Window Value*. The side lobe levels of the Time Domain stimulus depend only on the window that is selected. Figure 6-3 shows the same eye diagram that was displayed in Figure 6-2 with each of the three window settings.

PLTS also allows you to define your own window using the **Custom** selection where you will set you own *Time Domain Window Value*.

The purpose of windowing is to make the Time Domain response more useful in isolating and identifying individual responses. The window does not affect the displayed Frequency Domain response. The Time Domain function is only applied to time domain data. It does not affect frequency domain data.

Figure 6-3 The Effect of the Time Domain Windowing Selection



The following formula can be used to determine the equivalent 10 percent to 90 percent system rise time for the step function transformation and the eye diagram simulation:

$$T_r = (1000 / F_{max}) \times RTEC$$

Where,	T_r	= Rise time of the step response form 10% to 90% (in picoseconds)
	1000	= Factor to convert frequency to picoseconds
	F_{max}	= Maximum stop frequency used in the measurement (in GHz)
	RTEC	= Rise time equation coefficient

As an example: System rise time = 27 ps = (1000/ 20 GHz) × 0.54 (see table below)

Table 6-1 shows the equivalent rise times for a maximum frequency of 20 GHz for each window setting.

Table 6-1 Equivalent Rise Times for a Maximum Frequency of a 20 GHz System

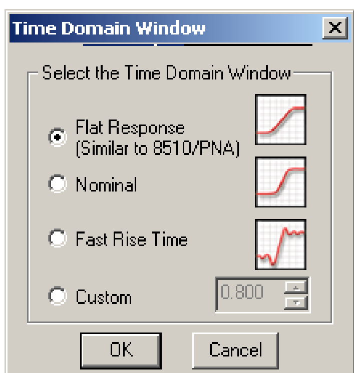
Window Selection	TD Window Value[a]	Maximum Frequency (F_{max})	Rise Time Equation Coefficient (RTEC)	Equivalent Rise Time (T_r)
Flat Response	0.80	20 GHz	0.91	45.5 ps
Nominal	0.50	20 GHz	0.71	35.5 ps
Fast Rise Time	0.20	20 GHz	0.54	27 ps

a. The relationship between the TD window value and the rise time equation coefficient is not made available.

By increasing or decreasing the filter value above or below the default, a trade-off can be made between rise time and side-lobe level (dynamic range).

When **Time Domain Window**... is selected from the **Tool** menu, the *Time Domain Window* dialog box is displayed allowing a choice of the three Time Domain Window settings.

Figure 6-4 Time Domain Window Dialog Box



- **Flat Response** gives the minimum side lobes and this provides the greatest dynamic range. The time domain window value is 0.80. The rise time equation coefficient value is 0.91.
- **Nominal** gives reduced side lobes and is normally the most useful. The time domain window value is 0.50. The rise time equation coefficient value is 0.71. This is the default setting.
- **Fast Rise Time** is essentially no window and therefore gives the highest side lobes. The

time domain window value is 0.20. The rise time equation coefficient value is 0.54.

- **Custom** allows you to create your own time domain window. Set the time domain window value (range 0.000 to 1.000) by selecting the spinner or entering a value in the box.

NOTE When you open measurement data in time domain format, the previously selected windowing is used. To change the windowing selection, select **Time Domain Window**... from the **Tool** menu, which displays the *Time Domain Window* dialog box. Make your windowing selection from the dialog box and click **OK**. The new windowing choice is then applied to the data.

Uncoupled Start Frequency and Step Frequency Setting

The Uncouple Start Frequency and Step Frequency selection is available in the *Calibrate Hardware for Measurement* dialog box for the VNA-based measurement setup when the Advanced button has been selected. Normally, the Frequency Step setting is locked (or coupled) to the Frequency Start setting.

However, there are situations that you may want to uncouple the **Frequency Step** setting from the **Frequency Start** setting. For example, uncouple the **Frequency Step** setting from the **Frequency Start** setting if you want to measure more points for better measurement resolution or you may want to measure specific frequency points that would not ordinarily be measured.

When measured data from this uncoupled start frequency/step frequency measurement is converted to the Eye Diagram analysis type, conversion rules that change the data must be applied to ensure the data is displayed correctly. See "Uncouple Start Frequency and Step Frequency" on page 11 for additional information regarding each of these conversion rules.

When these conversion rules change the measured data to be displayed in the Eye Diagram analysis type, one of three icons is displayed in the lower right corner of the plot to notify you of the change. These icons are displayed in Table 6-2.

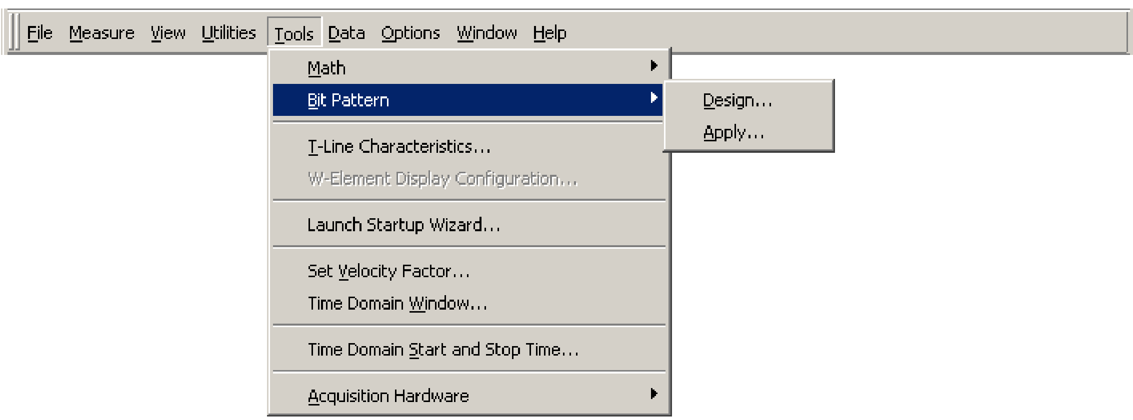
Table 6-2 Time Domain/Eye Diagram Uncoupled Start Frequency/Step Frequency Icons

Icon	Icon Name
	**Resampled Data -** This icon indicates that only the harmonically-related data points were used to generate time domain data.
	**Interpolated Data -** This icon indicates interpolation is performed to calculate the harmonically-related points that were not measured. Any measurements that were performed at harmonically-related points are left unchanged. The interpolated data is used, along with the measured harmonically-related data, to perform the Inverse Fast Fourier Transform (IFFT) for the calculated time domain data.
	**Bad Data** – This icon indicates there are less than 10 harmonically-related data points in the measured data. In this case, all of the non harmonically-related data is used to perform the Inverse Fast Fourier Transform (IFFT) to calculate time domain data. This may result in inaccurate time domain data.

Designing a Bit Pattern for Eye Diagrams

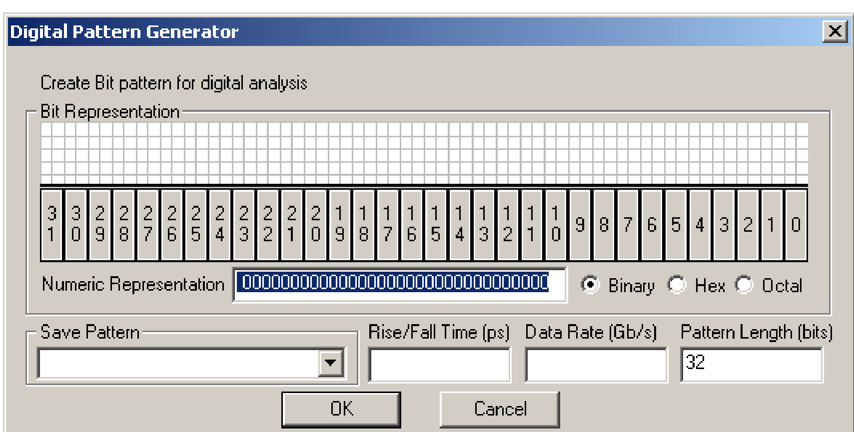
The **Bit Pattern** feature in the **Tools** menu allows you to design a digital bit pattern and to apply a bit pattern that has been saved to eye diagram plots. Only the industry standard K 28.5 bit pattern has been created and included in the software. However, the PLTS software allows you to create bit patterns, from 8 to 32 bits very easily.

Figure 6-5 Tools Menu with Bit Pattern Expanded



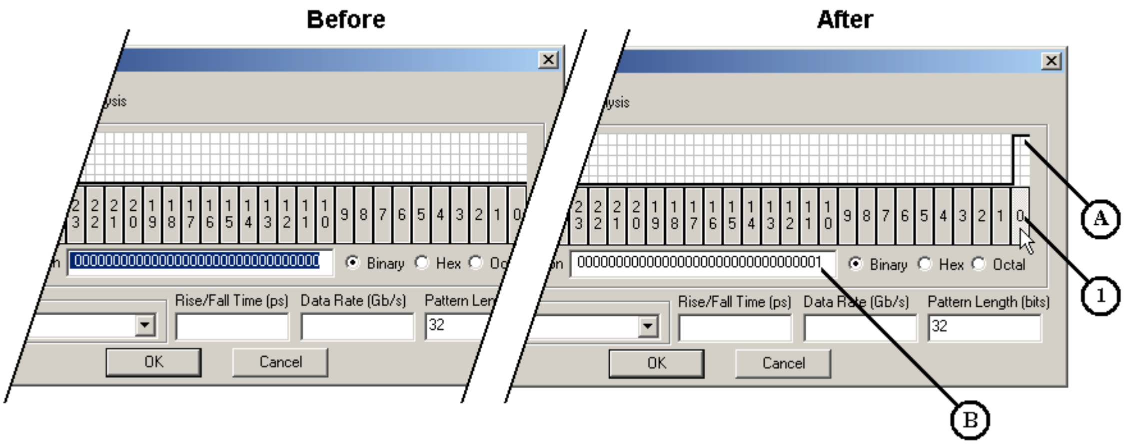
1. Select **Bit Pattern** then **Design**... from the **Tools** menu to open the *Digital Pattern Generator* dialog box shown in Figure 6-6.

Figure 6-6 Digital Pattern Generator



2. Click on the “**0**” key as shown in Figure 6-7.

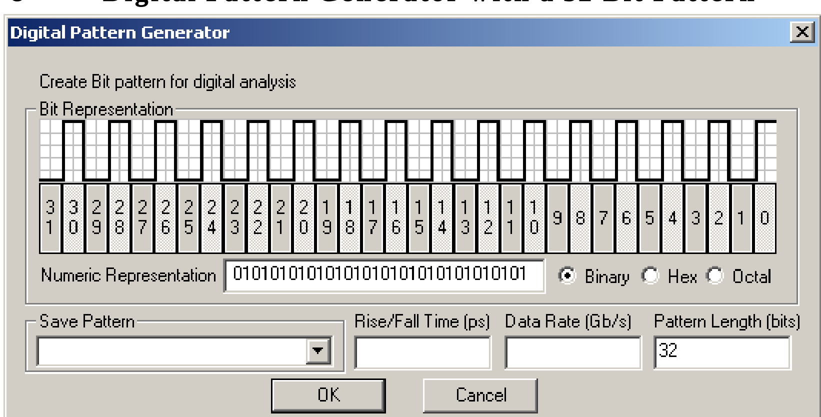
Figure 6-7 Selecting the “0” Piano Key



1. **Selecting the “0” key (piano key) in the Bit Representation area:**
   - **A.** Causes Bit 0 to go high (turn on)
   - **B.** Generates a “1” in the first digit of the binary numeric representation

3. Select the remaining even-numbered *piano* keys so that you have a series of alternating bits as shown in Figure 6-8.

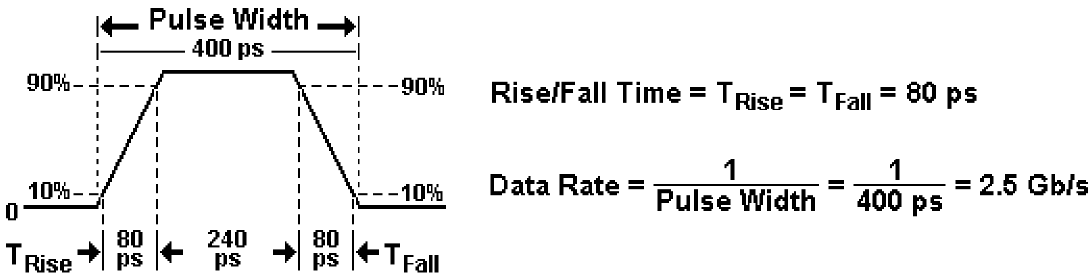
Figure 6-8 Digital Pattern Generator with a 32-Bit Pattern



Note the alternating bits going high (turning on) and the alternating 1’s and 0’s in the **Numeric Representation** area.

4. Enter values in the **Rise/Fall Time** and the **Data Rate** boxes. **Rise/Fall Time** is entered in picoseconds (ps) and **Data Rate** is entered in Gigabits per second (Gb/s). See Figure 6-9 and Figure 6-10.

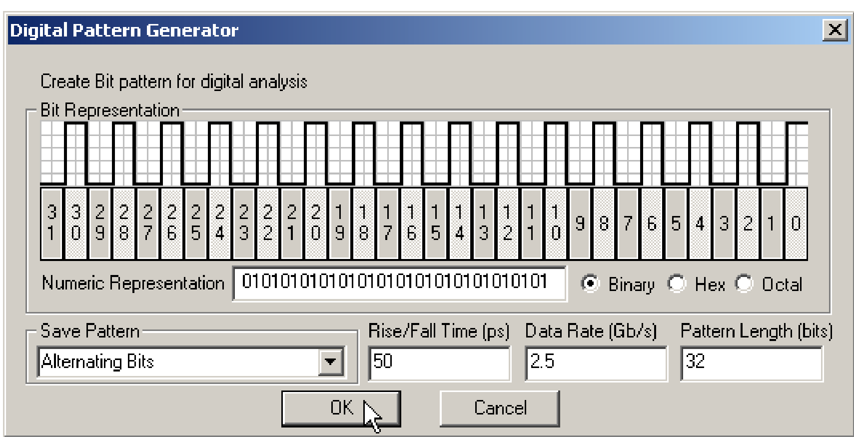
Figure 6-9 Rise/Fall Time and Data Rate



5. Check the value in the **Pattern Length (bits)** box. The default value is 32. The allowable range is between 8 and 32 bits. You may change this if your pattern contains fewer bits. If this value is changed to a value less than 32, any remaining bits are ignored. For example if you enter 10 as the pattern length value, then bits 0 through 9 are used and bits 10 through 31 are ignored.

6. Enter a name for the digital pattern in the **Save Pattern** box. In this example, `Alternating Bits` is entered. Click the **OK** button to save the pattern to be used later. See Figure 6-10.

Figure 6-10 Save the Digital Pattern



Viewing Data using Eye Diagrams

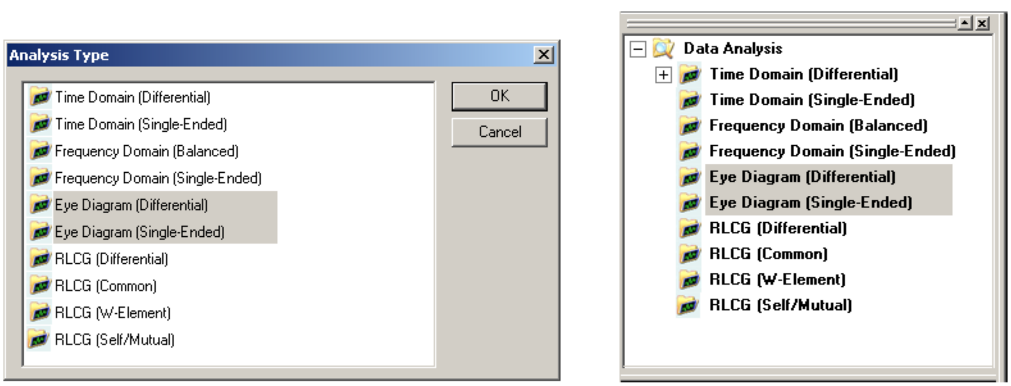
This section guides you with opening measured data in the eye diagram plot mode and viewing the data in the way that best suits your requirements. There are 8 output plots for balanced mode and 12 output plots for single-ended mode. Only transmission paths are displayed; no reflection paths are displayed.

Opening a Eye Diagram Plot Window

The eye diagram may be viewed in either balanced or single-ended mode. You may open the eye diagram plot window in one of four ways.

- From the Startup Wizard immediately before selecting the **Measure** button where you must select the analysis type - see (A) of Figure 6-11
- From the Startup Wizard Welcome Screen by selecting **Load Measurement** where you must select the analysis type - see (A) of Figure 6-11
- From the **Open** selection in the **File** menu or the **Open** icon in the **Toolbar** where you must select the analysis type - see (A) of Figure 6-11
- From the **Browser** when data is already being viewed in another analysis type by selecting one of the Eye Diagram choices - see (B) of Figure 6-11

Figure 6-11 Opening the Eye Diagram Plot Window

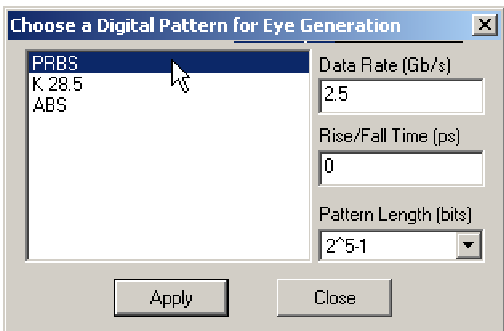


(A) (B)

NOTE When using the **Browser** method to open a window, first select the bit pattern to use with the data (described in the rest of this section), then select the parameters that you want to view from the **Parameter Bar** or the **Data** menu once the blank plot window is displayed.

The *Choose a Digital Pattern for Eye Generation* dialog box is then displayed (see Figure 6-12). You may select a digital pattern to apply from the list of digital patterns. The software is shipped with the **PRBS** (the Pseudo-Random Bit Sequence) selection, the **K 28.5** selection (an industry standard), and the **Arbitrary Bitstream** selection (a *random-like* bit stream). Any bit patterns that you have created and saved using the process described in Designing a Bit Pattern for Eye Diagrams beginning on page 194 are also displayed in the *Choose a Digital Pattern for Eye Generation* dialog box.

Figure 6-12 Choose a Digital Pattern Dialog Box



NOTE If the bit pattern has already been selected for the data, the *Choose a Digital Pattern for Eye Generation* dialog box is not displayed and the data will is displayed using that bit pattern information. To change the bit pattern, select **Bit Pattern** then **Apply**... from the **Tools** menu to display the dialog box shown in Figure 6-12. Then, select the desired bit pattern from the list on the left, enter the desired information (Rise/Fall Time, Data Rate, Pattern Length, and Number of Patterns) as described below, and click **OK**. Then, delete the current plot window and reopen a new plot window using the same data. The new bit pattern is then applied.

After clicking a digital pattern in the *Choose a Digital Pattern for Eye Generation* dialog box list, review the digital pattern parameter entries on the right side of the dialog box and enter the desired parameter values.

- **Rise/Fall Time (pS)** is the time that it takes a signal to transition from a low to a high (10% to 90%) condition (or the time that it takes a signal to transition from a high to a low (90% to 10%) condition).
- **Data Rate (Gb/S)** is the speed that data is transferred over a circuit or a communications line.
- **Pattern Length (bits)** is the number of bits in the digital pattern used to create the eye diagram. This value is the limiting factor in creating unique digital patterns. The value of the **Pattern Length** may be changed for PRBS and ABS. Other defined bit patterns have the **Pattern Length** defined as part of the pattern and may not be changed except by changing the bit pattern definition. Refer to Table 6-3 for the pattern lengths allowed for PRBS and ABS.

Table 6-3 PRBS and ABS Pattern Length Selections and Number of Bits

PRBS Pattern Length Selections		ABS Pattern Length Selections	
Selection	**Number of Bits**	**Selection**	**Number of Bits**
2^5-1	(31 bits)	- -	- -
2^7-1	(127 bits)	2^7	(128 – 2 = 126 bits)
2^9-1	(511 bits)	2^9	(512 – 2 = 510 bits)
2^10-1	(1023 bits)	2^10	(1024 – 2 = 1022 bits)
2^11-1	(2047 bits)	2^11	(2048 – 2 = 2046 bits)
2^13-1	(8191 bits)	2^13	(8192 – 2 = 8190 bits)
2^15-1	(32767 bits)	2^15	(32768 – 2 = 32766 bits)

Then, click **OK** to view the eye diagram in the plot window. **Cancel** closes the dialog box without making any changes.

IMPORTANT **Arbitrary Bitstream**

Though Arbitrary Bitstream (ABS) is a random-like bit stream used to generate eye diagrams, it is different than the Pseudo-Random Binary Sequence (PRBS) standard. The ABS pattern was defined to provide a large random pattern of bits that would quickly converge the eye diagram to show worst case tolerances.

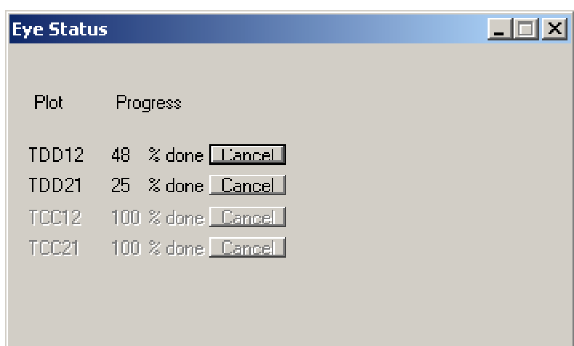
ABS develops a random sequence of bits for the virtual pattern generator using the **Pattern Length** entry. **Pattern Length** is entered to define the number of bits within a specific pattern.

If the Pattern Length = 2^7 bits, there are 128 unique patterns available if you count in binary from 0 to 127.

```
0000000
0000001
0000010
.
.
.
1111101
1111110
1111111
```

ABS then removes patterns 0000000 and 1111111 as invalid patterns so that you now have a total of $2^{\text{Pattern Length}} - 2 = 2^7 - 2 = 126$ unique patterns to choose from. Then ABS uses a random number generator to choose the first pattern to put into the virtual pattern generator and continues to pick new random patterns up to the number of patterns that you have defined within the interface (maximum is 126).

Using both of these values, a random number generator selects unique bit patterns until the appropriate number of patterns are identified. Each of these unique bit patterns are then used to create the eye diagram, one bit pattern at a time.



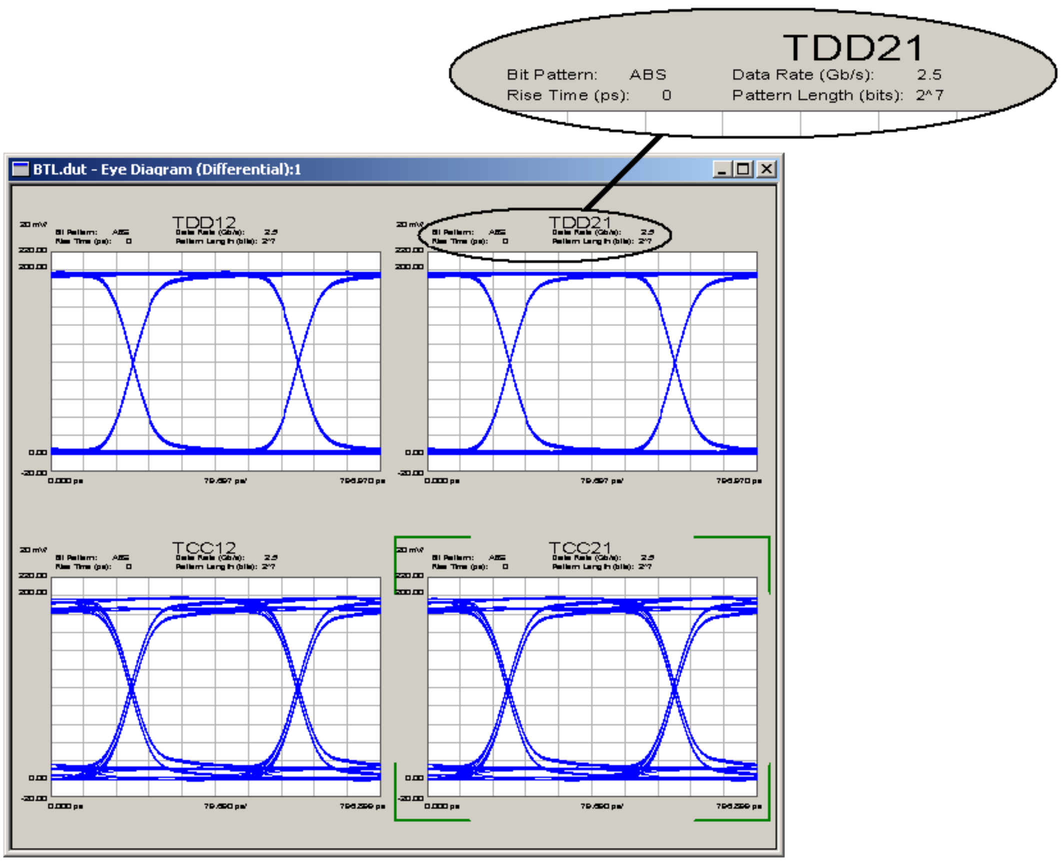
As the eye diagram is created, the *Eye Status* box is displayed. This box shows the status of the eye diagrams as they are being generated. When a parameter is complete, it is grayed. When all parameters are complete, this box is removed and the plots are complete. You can cancel a parameter by selecting the **Cancel** button for that plot. Selecting all **Cancel** buttons that have not been grayed ends the generation of the eye diagram

Viewing All Parameters

In all cases, except when you open the plot window using the **Browser**, all eye diagram parameter plots are displayed. There are four parameter plots for differential measurements and 12 parameter plots for single-ended measurements. Each of the plots are labeled. The bit map information used to create the eye diagram is displayed above the plot as shown in the inset of Figure 6-13.

As noted above, when you open the plot window from the Browser, an empty plot window is displayed. View all plots by selecting **All** from the **Parameter Bar** or from the **Data** menu.

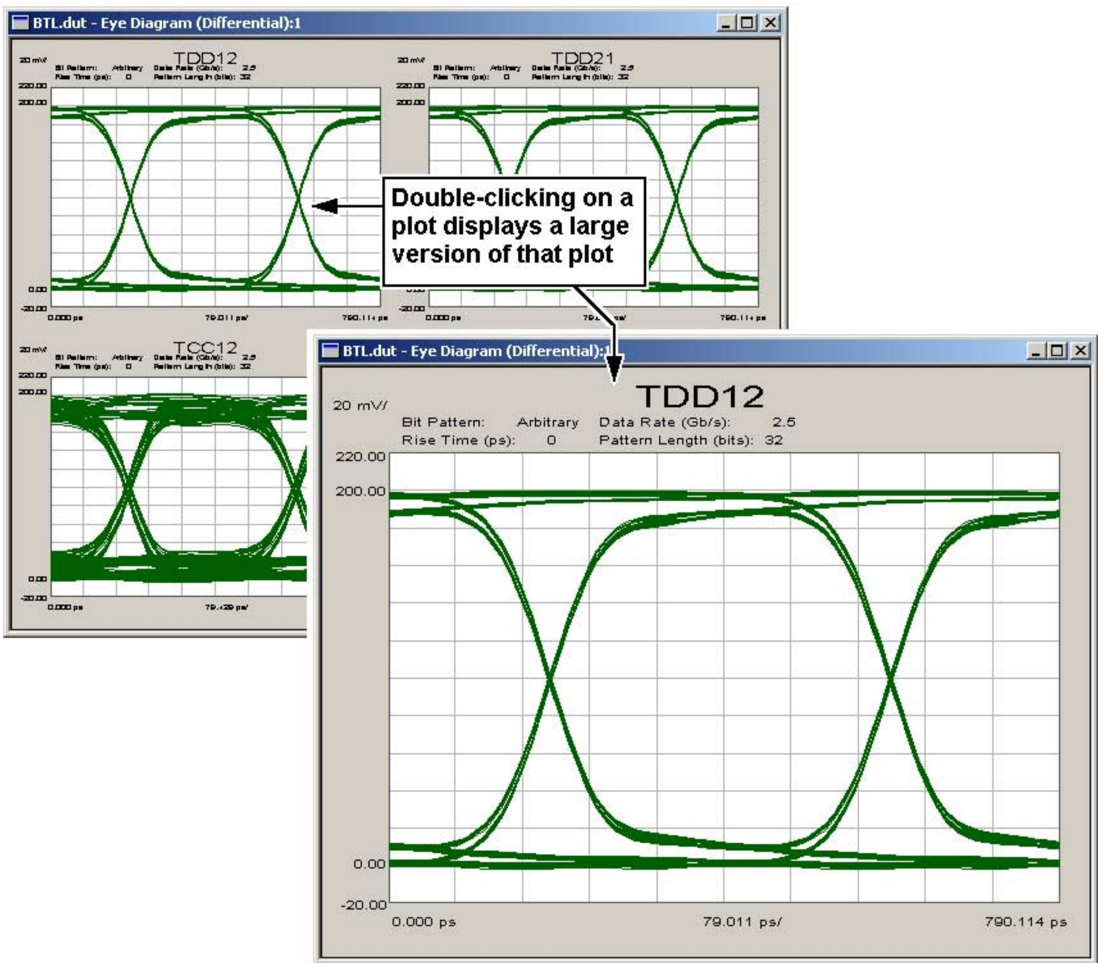
Figure 6-13 Balanced Eye Diagram Plots with Informational Label



Viewing a Single Eye Diagram

Viewing a single plot gives better resolution. To display a single plot, from the window with all plots (or with multiple plots if you have a custom plot window displayed), double-click on the plot that you would like to view. See Figure 6-14.

Figure 6-14 Opening a Single Plot

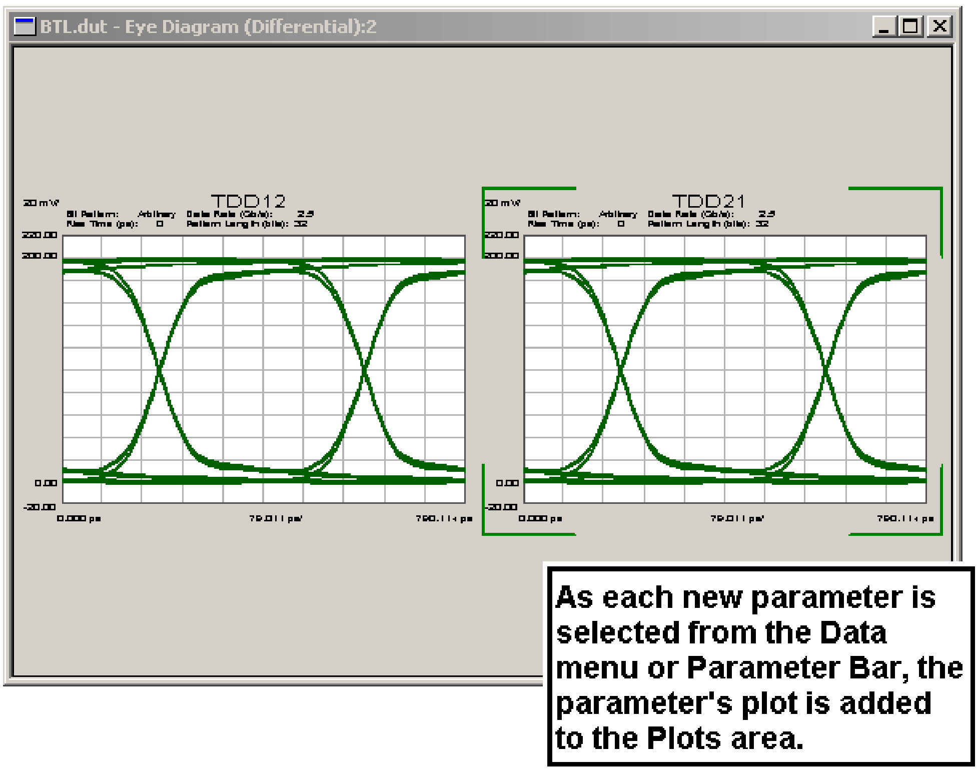


Double-click on the single plot to return to the original view with multiple plots.

Creating a Custom Eye Diagram Plots Window

You can also create a plot window with just the plots you desire. For example, you may want your plot window to show just the two TDDxx plots. To create this custom window, open the measured data file in any analysis type. Then, in the **Browser**, select the data type that you want to display the plots. In this example, select **Eye Diagram (Differential)**. A blank plots window is displayed. From the **Parameter Bar** (or the **Data** menu), click the desired parameters (TDD12 and TDD21 in this example). As each parameter is selected, a new plot is added to the plots window. See Figure 6-15.

Figure 6-15 Custom Eye Diagram Plots Window with Two Plots



7 Analyzing Transmission Line Parameters

There are four modes in which RLCG parameters may be displayed in PLTS:

- Differential
- Common
- W-Element
- Self/Mutual

PLTS exports RLCG data in the **W-Element** mode for use by HSPICE or Advanced Design System (ADS - an integrated design software and test instrumentation solution from Agilent Technologies). The W-Element mode uses the 4-port S-parameters of the symmetrical, coupled transmission line to compute the R (resistance), L (inductance), C (capacitance), and G (conductance) parameters. Each is displayed in a 2-by-2 matrix. There is an R-value for each line and coupling values for R. The same is true for L, C, and G. These parameters can then be used in HSPICE or ADS as a model of the measured transmission line.

There are two modes, **Differential** and **Common**, of the RLCG parameters that treat the coupled line as a 2-port device instead of a 4-port device. These modes simply use the four pure differential-mode parameters or four pure common-mode parameters as a 2-port S-parameter device. This is saying we have a line driven differentially (or in common) and what are the RLCG parameters, impedance, and propagation constant for this line. In this case, the RLCG parameters are a single value, not a 2-by-2 array. There is no self or coupling since it is treated as a single line. Neither of these modes is directly usable in HSPICE or ADS but these modes can give insight to an experienced user.

The fourth mode is the **Self/Mutual** mode. It is a slight deviation from the W-Element mode. The only difference in this mode is the way that the coupling between parameters is defined. The conversion is described in "CPTL RLCG Extraction Procedure" on page 214.

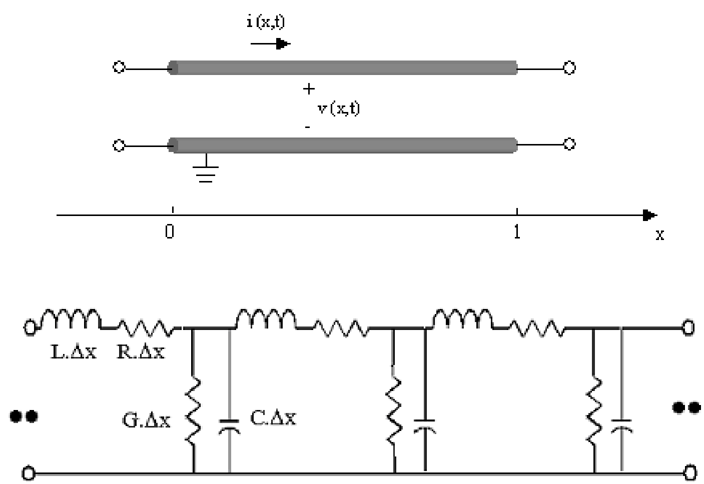
An important issue that is not clearly understood is that the measurements must be for only the line to be modeled. Connectors and single-ended launches to connect to the actual coupled line must be removed using de-embedding or calibration standards in the medial. The simplest way to measure coupled lines is by probing the lines. When the lines are probed, no connectors or launches need be removed.

NOTE When extracting the RLCG parameters for a symmetrical coupled line, the measurement must include only the coupled transmission line. It should not include any connectors, or single-ended launches in the measurement. If any of these are included, the parameters will not accurately model the transmission line. Refer to "Considerations When Extracting RLCG Parameters" on page 222 for more information. Note that Figure 7-11 shows the connector and launches that need to be removed.

Transmission Line Parameters

Transmission lines are distributed devices. However, SPICE type simulators work with lumped elements, not distributed elements. To approximate the distributed behavior of a transmission line, RLCG type models are commonly used. The single transmission line shown in Figure 7-1 can be modeled by a network consisting of a series resistance and inductance with parallel capacitance and conductance.

Figure 7-1 RLCG Model for Single Transmission Line



The different terms included in the model describe the following physical phenomena:

R – Resistive loss of the conductor (transmission line trace). Determined by the conductance of the metal, width, height, and length of the conductor.

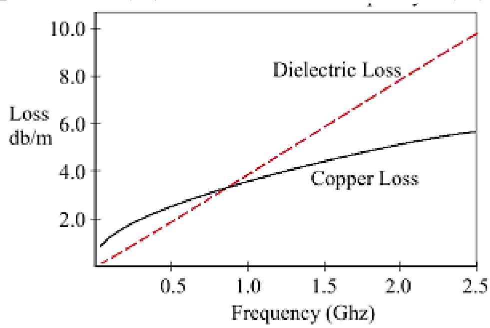
L – Inductive part of the circuit resulting from the layout of the conductors. Determined by the dimensions of the conductor, permeability of the metal, and layout.

C – Capacitive part of the circuit resulting from the layout of the conductors. Determined by the permittivity and thickness of the board material and the area of the conductor.

G – Shunt loss of the dielectric. Determined by the layout of the conductors, permittivity, loss tangent and thickness of the board material.

RLCG modes are frequency-based models. See Figure 7-2.

Figure 7-2 Copper Loss (R) and Dielectric Loss (G)



Equation Set 1 describes the most adopted frequency dependencies of RLCG parameters.

Equation Set 1

$$R = R_{DC} + R_{SKIN}\sqrt{f}$$
$$L = constant$$
$$G = G_{DC} + G_{AC} \cdot \sqrt{f}$$
$$C = constant$$

NOTE These parameters are called fitted parameters.

The values for RLCG are typically specified in per unit length, where the unit of length is in meters. Therefore for a given length of line the value for each of the parameters is easily determined. To best approximate the distributed behavior of the transmission line multiple sections of RLCG circuits are used. The value of the parameters, R for example, is determined by dividing the R value for the given length of line by the number of sections. Since R (and) L values add in series and C and G values add in parallel, a multi-section model for simulation can easily be constructed. For example:

If the value of R for a given line is 2.4 ohms per meter,
and the length of line needed is 100 cm,
then the total resistance needed for the 100 cm line is 0.24 ohms
If 12 sections are used to model the line, then each R is 0.02 ohms.

The same calculation can be made for each of the parameters.

Extracting Fitted RLCG Parameters from S-Parameters

Telegrapher's equations are used to solve for the RLCG values. The Telegrapher's equations described in "Coupled-Transmission Line Models" on page 211 for the 2-coupled line model. Telegrapher's equations deal with the voltage and current as shown in Figure 7-1. However, PLTS measures S-parameters, which are ratios of power reflected from and transmitted thru to the incident power. For a single transmission line, the impedance (Z) and propagation constant (γ) can be derived from the measured 2-port S-parameters of the line. Equation Set 2 defines the S-parameters in terms of Z, Z_0 (characteristic impedance of the measurement system), γ, and *l* (the length of the line).

Equation Set 2

$$(S) = \frac{1}{D_S}\begin{pmatrix} (Z^2 - Z_0^{\ 2})\sinh \gamma l & 2 \times Z \times Z_0 \\ 2 \times Z \times Z_0 & (Z^2 - Z_0^{\ 2})\sinh \gamma l \end{pmatrix}$$

where

$$D_s = 2 \times Z \times Z_0 \cosh \gamma l + (Z^2 + Z_0^2) \sinh \gamma l$$

Using Equation Set 2 and transforming to [ABCD} parameter, we can solve for γ and Z as functions of S-parameters as shown in Equation Set 3 and Equation Set 4:

Equation Set 3

$$e^{-\gamma l} = \left\{\frac{1 - S_{11}^{\ 2} + S_{21}^{\ 2}}{2S_{21}} \pm K\right\}^{-1}$$

where

$$K = \left\{\frac{(S_{11}^{\ 2} - S_{21}^{\ 2} + 1)^2 - (2S_{11})^2}{2S_{21}^{\ 2}}\right\}^{1/2}$$

Equation Set 4

$$Z^2 = Z_0^{\ 2}\frac{(1 + S_{11})^2 - S_{21}^{\ 2}}{(1 - S_{11})^2 - S_{21}^{\ 2}}$$

Once γ and Z are known, from the standard transmission line relationships, values for R, L, C, and G can be determined as shown in Equation Set 5 through Equation Set 10 below:

Equation Set 5

$$\gamma = \sqrt{(R + j\omega L)(G + j\omega C)} = \alpha + jB$$

Equation Set 6

$$Z = \sqrt{\frac{(R + j\omega L)}{(G + j\omega C)}}$$

Then,

Equation Set 7

$$R = Re\{\gamma Z\}$$

Equation Set 8

$$L = Im\{\gamma Z\}/\omega$$

Equation Set 9

$$G = Re\{\gamma/Z\}$$

Equation Set 10

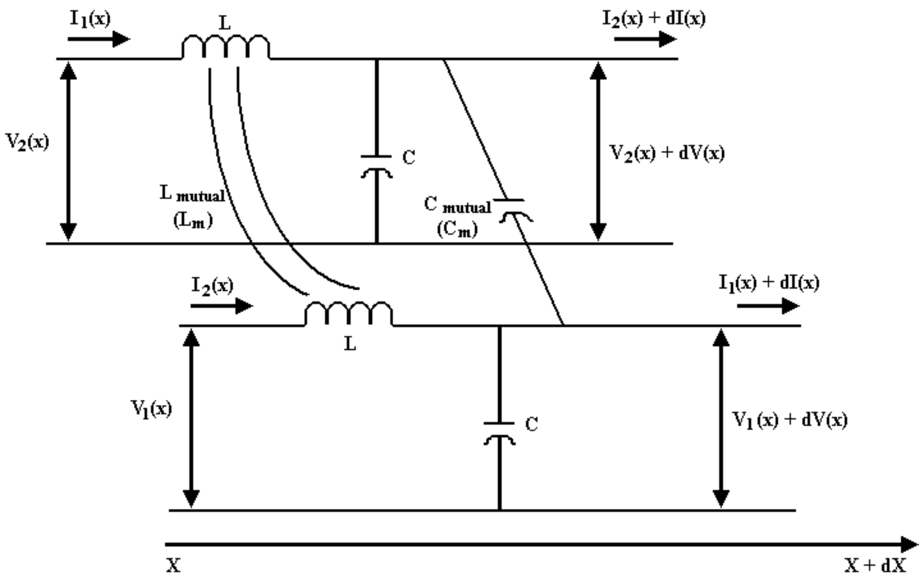
$$C = Im\{\gamma/Z\}/\omega$$

In the case of a pair of coupled transmission lines, each RLCG parameter is actually a 2-by-2 matrix. For symmetrical uniform coupled transmission lines, the matrices are real and symmetrical. The latter is described in more detail in “Coupled-Transmission Line Models” on page 211.

Coupled-Transmission Line Models

Start with an ideal lossless symmetrical coupled-transmission line (CPTL):

Figure 7-3 **Lossless Coupled Line Model**



The Telegraphers set of equations are described in Equation Set 11 and Equation Set 12:

Equation Set 11

$$-\frac{\partial V_1}{\partial x} = L\frac{\partial I_1}{\partial t} + L_m\frac{\partial I_2}{\partial t}$$

$$-\frac{\partial V_2}{\partial x} = L_m\frac{\partial I_1}{\partial t} + L\frac{\partial I_2}{\partial t}$$

Equation Set 12

$$-\frac{\partial I_1}{\partial x} = C\frac{\partial V_1}{\partial t} + C_m\frac{\partial (V_1 - V_2)}{\partial t}$$

$$-\frac{\partial I_2}{\partial x} = C_m\frac{\partial (V_2 - V_1)}{\partial t} + C\frac{\partial V_2}{\partial t}$$

These equations represent the closest form to the physical behavior of CPTL, since they describe each line by its own self parameters (L and C) and the different mutual couplings (Lm and Cm). Obviously, these equations can be extended for the lossy case, where the conductor and dielectric losses would be taken into account.

NOTE *These parameters are called self-parameters.*

By rearranging Equation Set 11 and Equation Set 12, a second set of parameters can be defined as shown in Equation Set 13 and Equation Set 14:

Equation Set 13

$$-\frac{\partial V_1}{\partial x} = L_{11}\frac{\partial I_1}{\partial t} + L_{12}\frac{\partial I_2}{\partial t}$$

$$-\frac{\partial V_2}{\partial x} = L_{21}\frac{\partial I_1}{\partial t} + L_{22}\frac{\partial I_2}{\partial t}$$

Equation Set 14

$$-\frac{\partial I_1}{\partial x} = C_{11}\frac{\partial V_1}{\partial t} + C_{12}\frac{\partial V_2}{\partial t}$$

$$-\frac{\partial I_2}{\partial x} = C_{21}\frac{\partial V_1}{\partial t} + C_{22}\frac{\partial V_2}{\partial t}$$

In the general case, RLCG parameters are grouped in 2-by-2 real matrices, each term being frequency-dependent. In the case of symmetrical coupled-lines, these matrices are symmetrical. See Equation Set 15.

Equation Set 15

$$R = \begin{pmatrix} R_{11} & R_{12} \\ R_{12} & R_{11} \end{pmatrix} \qquad G = \begin{pmatrix} G_{11} & G_{12} \\ G_{12} & G_{11} \end{pmatrix}$$

$$L = \begin{pmatrix} L_{11} & L_{12} \\ L_{12} & L_{11} \end{pmatrix} \qquad C = \begin{pmatrix} C_{11} & C_{12} \\ C_{12} & C_{11} \end{pmatrix}$$

NOTE *These parameters are called spice-parameters.*

Most Spice-type simulators use this type of model description with different variations in the implementation. This aspect is described in more details in Chapter 8, “Importing and Exporting Data.”.

The third model representation is called the *Differential-Common Modes Equivalent Model*. This model was created because the RLCG extraction algorithm deals only with Single-Ended Transmission Lines (SETL). See “Extracting Fitted RLCG Parameters from S-Parameters” on page 209. Therefore, each quadrant from the mixed-mode S-parameters, in particular the Diff-Diff and Com-Com, are treated as two separate SETL, with predefined normalized impedance.

The new set of RLCG parameters extracted for the differential and common modes can be represented in a frequency-dependent matrix format, as shown in Equation Set 16.

Equation Set 16

$$R = \begin{pmatrix} R_{dd} & 0 \\ 0 & R_{cc} \end{pmatrix} \qquad G = \begin{pmatrix} G_{dd} & 0 \\ 0 & G_{cc} \end{pmatrix}$$

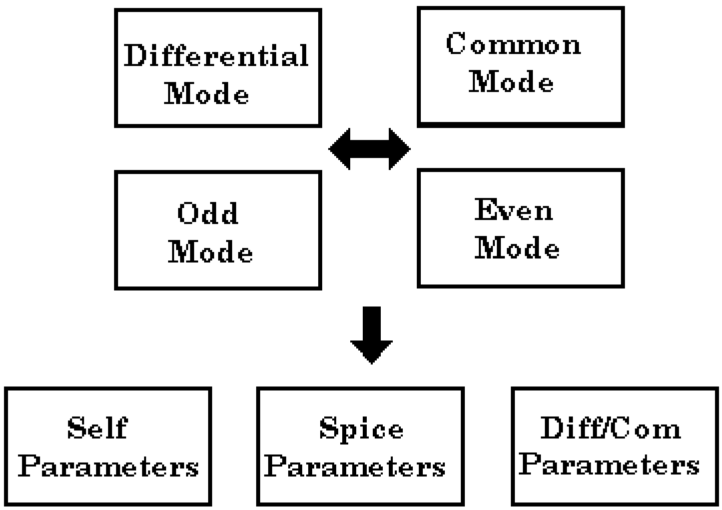
$$L = \begin{pmatrix} L_{dd} & 0 \\ 0 & L_{cc} \end{pmatrix} \qquad C = \begin{pmatrix} C_{dd} & 0 \\ 0 & C_{cc} \end{pmatrix}$$

NOTE *These parameters are called Diff/Com parameters.*

CPTL RLCG Extraction Procedure

PLTS starts by extracting the RLCG parameters for both the *Diff-Diff* mode and the *Com-Com* mode. In the case of a symmetrical CPTL, mode-conversion should be negligible. Then we offer the following options to visualize. See Figure 7-4.

Figure 7-4 RLCG Extraction Block Diagram



This section describes the formulas for the different transformations.

Equation Set 17 and Equation Set 18 relate the Odd and Even modes to the Differential and Common modes of propagation:

Equation Set 17

$$Z_{cc} = \frac{Z_e}{2}$$

$$\gamma_{cc} = \gamma_e$$

Equation Set 18

$$Z_{dd} = 2\,Z_o$$

$$\gamma_{dd} = \gamma_o$$

Using the propagation constant and the characteristic impedance for the Odd/Even modes, the Spice parameters are derived in Equation Set 19 and Equation Set 20.

Equation Set 19

$$R_{11} = \mathrm{Re}\{\gamma_e Z_e + \gamma_o Z_o\}$$
$$R_{12} = \mathrm{Re}\{\gamma_e Z_e - \gamma_o Z_o\}$$
$$L_{11} = \mathrm{Im}\{\gamma_e Z_e + \gamma_o Z_o\}/\omega$$
$$L_{12} = \mathrm{Im}\{\gamma_e Z_e - \gamma_o Z_o\}/\omega$$

Equation Set 20

$$G_{11} = \mathrm{Re}\{\gamma_e / Z_e + \gamma_o / Z_o\}$$
$$G_{12} = \mathrm{Re}\{\gamma_e / Z_e - \gamma_o / Z_o\}$$
$$C_{11} = \mathrm{Im}\{\gamma_e / Z_e + \gamma_o / Z_o\}/\omega$$
$$C_{12} = \mathrm{Im}\{\gamma_e / Z_e - \gamma_o / Z_o\}/\omega$$

Finally, Self parameters can be derived as shown in Equation Set 21 and Equation Set 22.

Equation Set 21

$$R_S = R_{11}$$
$$R_m = R_{12}$$
$$L_S = L_{11}$$
$$L_m = L_{12}$$

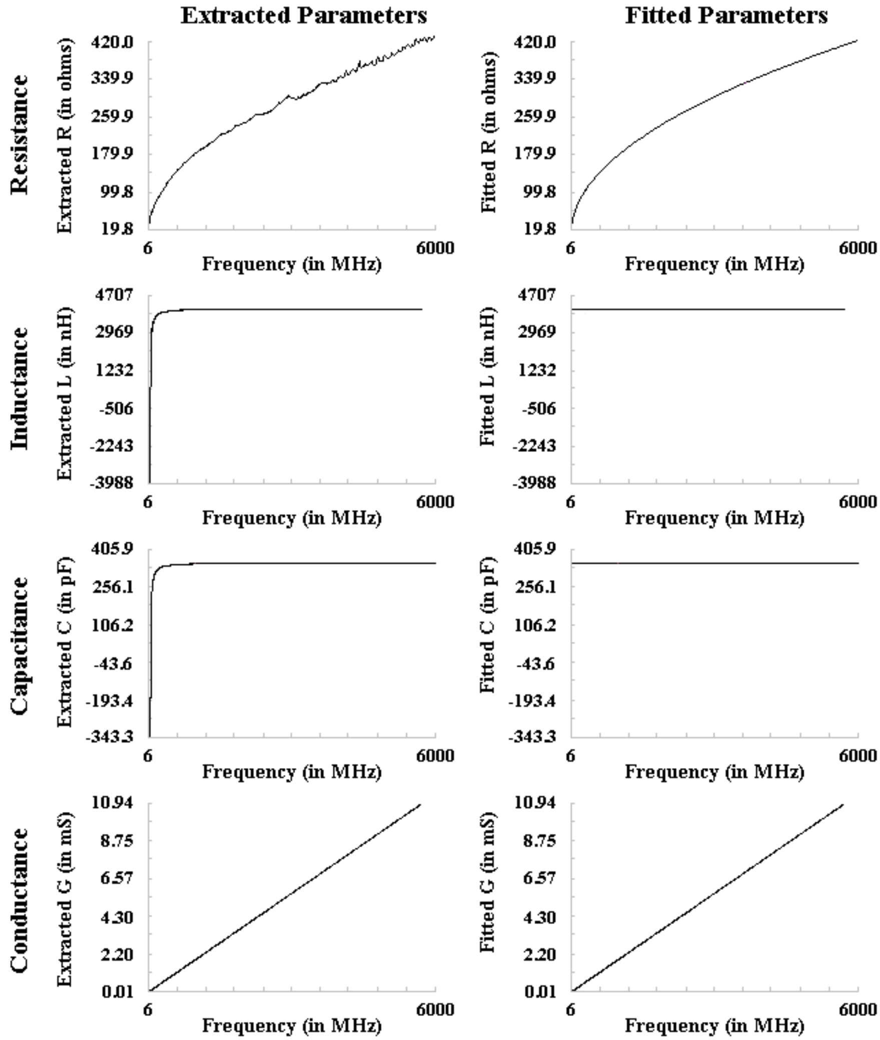
Equation Set 22

$$G_S = G_{11} + G_{12}$$
$$G_m = -G_{12}$$
$$C_S = C_{11} + C_{12}$$
$$C_m = -C_{12}$$

RLCG Output Plots

Figure 7-5 illustrates the difference between the extracted parameters and the fitted curve.

Figure 7-5 Extracted and Fitted Parameters



This plot format can be applied to Diff/Com, Spice or Self parameters.

Figure 7-6 illustrates the propagation constant and Figure 7-7 illustrates the characteristic impedance in real-imaginary format. Since these two parameters are complex numbers, you have the choice of plotting these parameters in other formats, like Magnitude/Phase, and dB/Phase versus linear or log of the frequency.

Figure 7-6 Propagation Constant versus Frequency

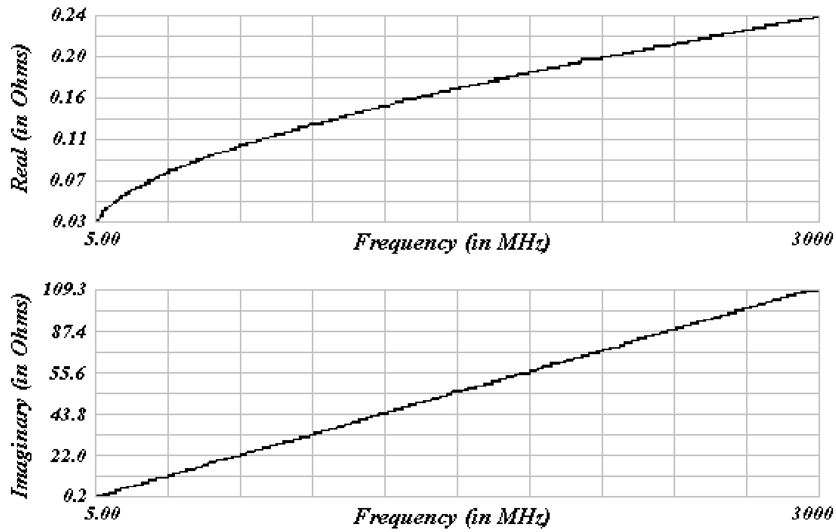
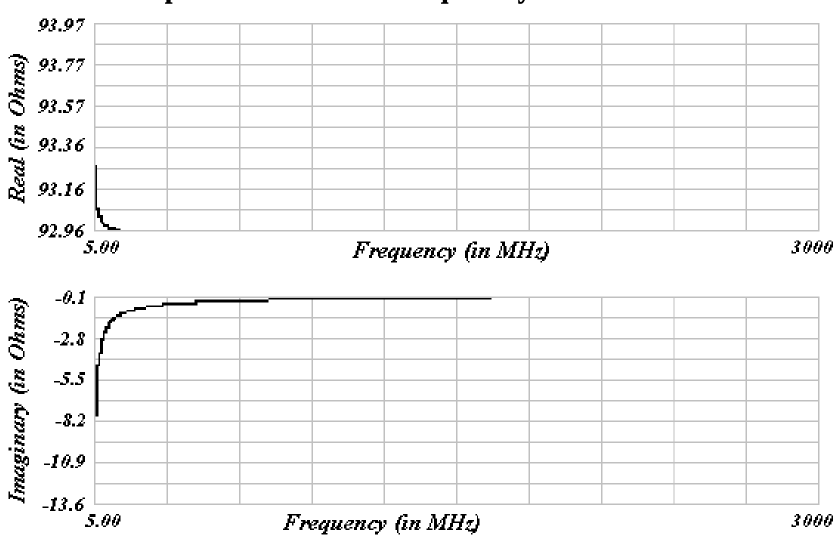


Figure 7-7 Impedance versus Frequency



Export Data Formats

The export data section is intended to link PLTS to the main Electronic Design Automation (EDA) software tools used by R&D engineers in the Signal Integrity field. These EDA simulators are:

- Agilent Advanced Design System (ADS)
- Avanti HSPICE

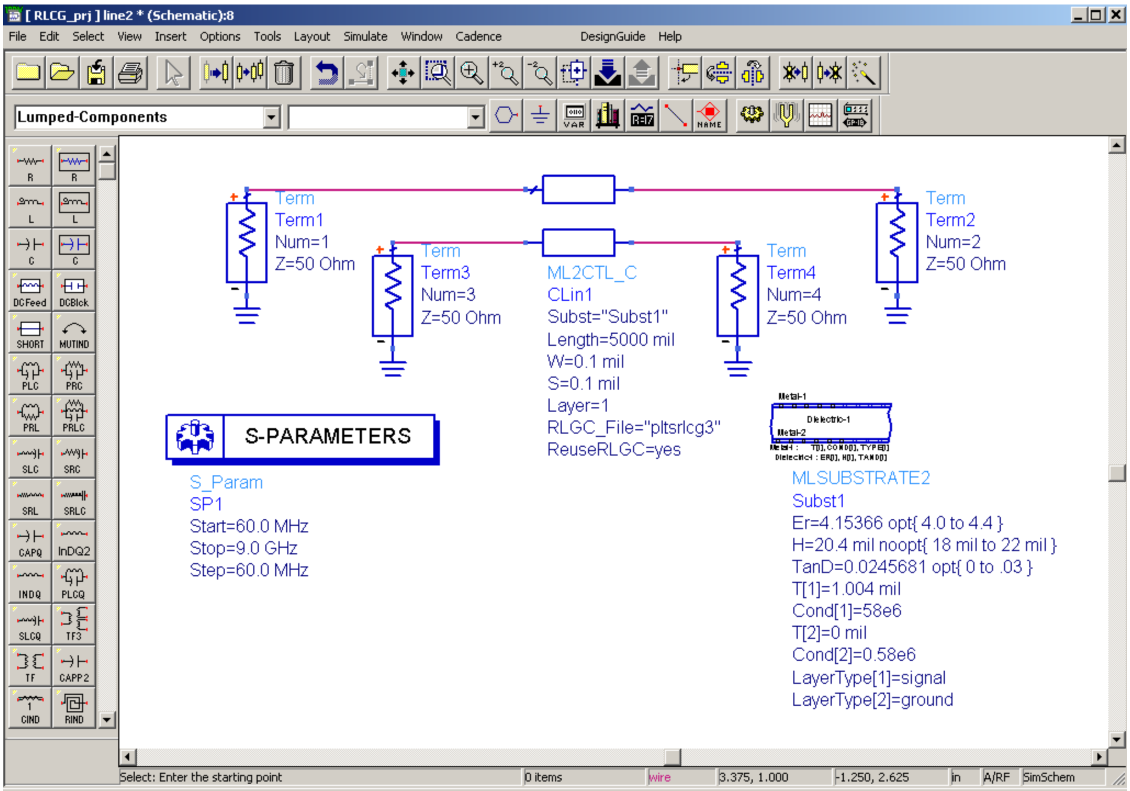
For ADS, there are two main export formats defined. The first format exports fitted parameters in an ASCII file to be used with the ML2CTL_C model found in the Multilayer TL Library. The syntax is described in the following example:

```
BEGIN DSCR(RLGC)
! C[i][j]/eps0 l[i][j]/mu0 Rdc[i][j] Rhf[i][j]/sqrt(f_GHz)
G[i][j]/omega*eps0
% C(real) L(real) Rdc(real) Rhf(real) G(real)
8.911534 0.326182 1.345030 13.7199 0.133030
-1.013052 0.067352 -0.000279 0.838288 -0.005866
8.911534 0.326182 1.345030 13.7199 0.133030
END
```

NOTE The ADS format has the capacitance values divided by ε_0, the inductance values divided by μ_0, and the high frequency resistance (R_s) divided by the square root of frequency (in GHz).

To use this file in ADS, insert the ML2CTL_C component (found in the Tlines-Multilayer Pallet) and have the “RLCG_File” parameter point to the exported file. You will also need to set the “ReuseRLCG” parameter to “yes” and set the “Length” parameter to the length of the line you wish to simulate. Figure 7-8 is an ADS schematic using the RLCG parameters to simulate the S-parameters of the modeled line.

Figure 7-8 ADS Schematic Using the ML2CTL_C Component



The second format exports extracted parameters versus frequency in an ASCII file. Two files are created, one with the self parameters (i.e. R11, C11, etc.) and one with the mutual parameters (i.e. R12, C12, etc.). The file format is shown in Figure 7-9 and Figure 7-10.

Figure 7-9 Exported Extracted Parameter File Format for Self Parameters

```
BEGIN DSCRDATA
%INDEX Freqs R11 L11 C11 G11
1, 6, 2.66401879116, 4.27585633246E-007, 8.23882000485 4E-011, 4.43976201528 9E-005
2, 12, 3.06090516578, 4.20664887436E-007, 8.10560436923 4E-011, 8.87952304632 5E-005
3, 18, 2.98776516048, 4.24233863773 3E-007, 8.17200728402 6E-011, 0.000133192840 7736
4, 24, 3.59335057255, 4.23348841566 9E-007, 8.15545708542 9E-011, 0.00017759045 1084
5, 30, 3.75264079220, 4.22491733119 6E-007, 8.13801691357 9E-011, 0.00022198806 13944
6, 36, 4.08618771111, 4.21718199037 7E-007, 8.12278337490 9E-011, 0.00026638567 17047
7, 42, 4.25560425287, 4.21662418608 7E-007, 8.12146412040 4E-011, 0.00031078328 20151
8, 48, 4.39088429085, 4.21269845625 5E-007, 8.11369751950 1E-011, 0.00035518089 23255
9, 54, 4.80588650011, 4.20749695517 3E-007, 8.10317893467 1E-011, 0.00039957850 26358
10, 60, 4.91226085022, 4.20336137330 8E-007, 8.09547734522 6E-011, 0.00044397611 29462
.
.
.
.
|
1495, 8970, 42.49338404989, 4.092890149054E-007, 7.87396068187 2E-011, 0.066374427423 84
1496, 8976, 42.38159372248, 4.09289314206 6E-007, 7.87395417388 6E-011, 0.066418825034 15
1497, 8982, 42.32048825007, 4.09290889852 1E-007, 7.87395462594 2E-011, 0.066463222644 46
1498, 8988, 41.67401476677, 4.09289810869 7E-007, 7.87392268091 8E-011, 0.066507620254 77
1499, 8994, 41.10014201313, 4.09291172930 9E-007, 7.87392295084 9E-011, 0.066552017865 08
1500, 9000, 40.26217750834, 4.09296241515 5E-007, 7.87398853000 6E-011, 0.066596415475 39
END
```

Figure 7-10 Exported Extracted Parameter File Format for Mutual Parameters

```
BEGIN DSCRDATA
%INDEX Freqs R12 L12 C12 G12
1, 6, 0.35556715478, 8.70694864901 4E-008, -9.60804097331 8E-012, -1.95408990758E-006
2, 12, 0.37713958302, 8.56392542122 4E-008, -9.45684151497 9E-012, -3.90817390964 8E-006
3, 18, 0.32124892501, 8.67333738673E-008, -9.46143708282 7E-012, -5.86225791171 7E-006
4, 24, 0.42018406929, 8.64750810103 3E-008, -9.45761745266 5E-012, -7.81634191378 5E-006
5, 30, 0.35282987035, 8.64442670185E-008, -9.40878164012 4E-012, -9.77042591585 3E-006
6, 36, 0.41402798975, 8.63378429481 7E-008, -9.38088588656 3E-012, -1.17245099179 2E-005
7, 42, 0.41090926359, 8.63644588782 9E-008, -9.37181765076 7E-012, -1.36785939199 9E-005
8, 48, 0.42575559378, 8.63159614037 1E-008, -9.35652579649E-012, -1.56326779220 6E-005
9, 54, 0.51094252063, 8.62871232793 7E-008, -9.32897533811 1E-012, -1.75867619241 3E-005
10, 60, 0.50611430653, 8.6161443332E-008, -9.32821601948 3E-012, -1.95408459261 9E-005
.
.
.
.|
1495, 8970, 6.44695845120, 8.52568461543 5E-008, -8.80320658165E-012, -0.002921355586998
1496, 8976, 6.17257628863, 8.52588138713 7E-008, -8.80282091051 6E-012, -0.002923309673
1497, 8982, 6.05775653757, 8.52637805412 8E-008, -8.80190024692 4E-012, -0.002925263757002
1498, 8988, 5.81947087962, 8.52652936115 9E-008, -8.80151940995 9E-012, -0.002927217841004
1499, 8994, 5.63914201175, 8.52696058037 6E-008, -8.80071968247 2E-012, -0.002929171925006
1500, 9000, 5.46693156328, 8.52756217056 6E-008, -8.79980795697 1E-012, -0.002931126009008
END
```

For Avanti HSPICE, the following format exports fitted parameters in an ASCII file to be used with the W-element TL model. The syntax is described in the following example:

```
* RLCG parameters for a 2-conductor lossy
* frequency-dependent line

* N (number of signal conductors)
*****************************************
2

* Lo
4.098919e-007
8.463660e-008  4.098919e-007

* Co
7.890361e-011
-8.969661e-012  7.890361e-011

* Ro
1.34503
-0.000278525  1.34503

* Go
9.842513e-012
-5.905520e-012  9.842513e-012

* Rs
0.000419683
2.65087e-005  0.000419683

* Gd
7.399602e-012
-3.256807e-013  7.399602e-012
```

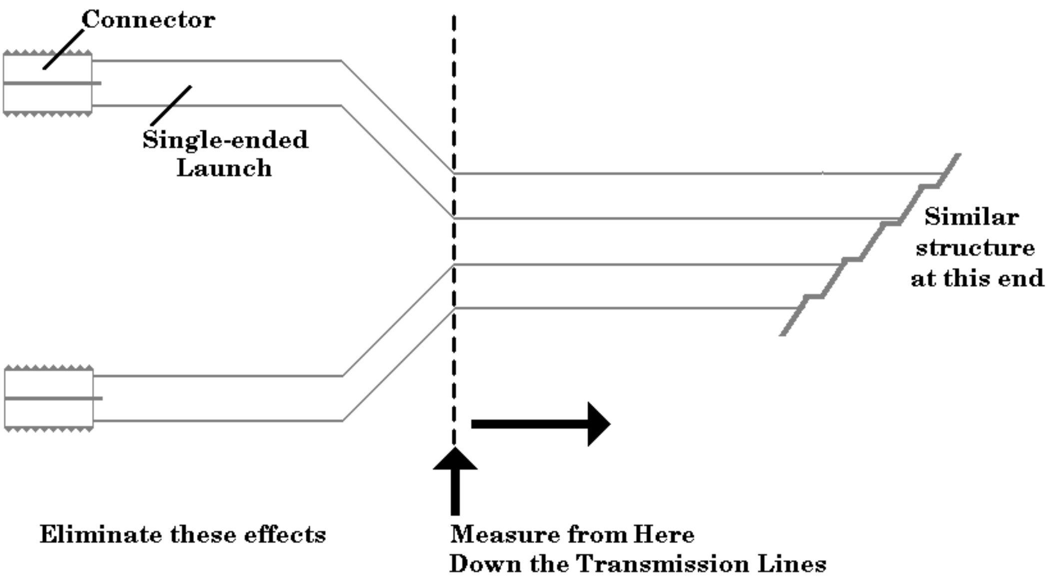
To use this file in HSPICE, use the W-element and have the “RLCGfile” parameter point to the file exported from PLTS. A simple netlist using the W-element and the exported parameters to do an S-parameter simulation of the modeled line is shown in the following example:

```
Single Ended TDR simulation of TL
.OPTIONS LIST NODE POST
.OP
.tran 0.01ns 5ns
.NET V(OUT) VIN ROUT=50 RIN=50
* .PLOT AC S11(DB) S22(DB) S21(DB)
VIN 3 0 DC=0 AC=1 pulse(0v 0.4v 0ns 100ps)
W1 In 1 0 OUT 2 0 N=2 L=0.127 RLGCfile=probednew.rlgc
R1 1 0 50
R2 2 0 50
R3 3 IN 50
R4 OUT 0 50
.END
```

Considerations When Extracting RLCG Parameters

When extracting the RLCG parameters for a symmetrical coupled line, the measurement must include only the coupled transmission line. It should not include any connectors, or single-ended launches in the measurement. If any of these are included, the parameters will not accurately model the transmission line. Figure 7-11 shows the connector and launches that need to be removed.

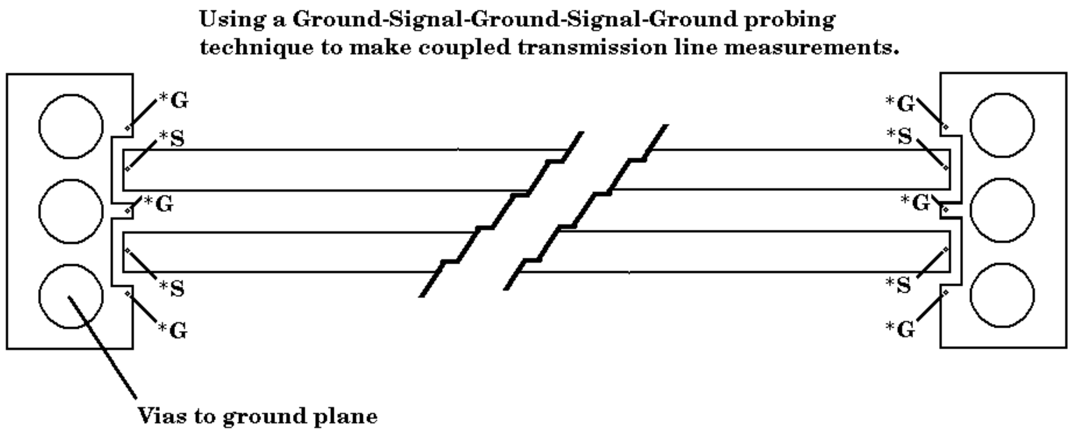
Figure 7-11 Typical Device with Connectors to be Tested



The effects to the left of the dotted line need to be removed. These can be removed one of two ways. The first is to characterize the launch structure (to the left of the dotted line) and then de-embed it from the measurement. This is not easily done. The other way is to create calibration standards on the board that include the connector and launch and use them to calibrate with. However, the parasitics of the standards need to be characterized and entered into the calibration kit definition.

The easiest way to characterize the transmission line is to do a probed measurement. By performing a probe calibration there are no connectors or launches to remove. Figure 7-12 shows a typical probed measurement.

Figure 7-12 Typical Probed Measurement of a Transmission Line



The Parameters for Each RLCG Mode

The data (parameters) available for each of the four RLCG modes varies due to the model assumptions. The individual parameter selections are based on the specific RLCG data analysis type. The following lists each data analysis type and its associated parameters.

RLCG (Differential):	Rd, Ld, Cd, Gd, Zor, Zoi, Ad, Bd
RLCG (Common):	Rc, Lc, Cc, Gc, Zor, Zoi, Ac, Bc
RLCG (W-Element):	R11, L11, C11, G11, R12, L12, C12, G12
RLCG (Self/Mutual):	Rs, Ls, Cs, Gs, Rm, Lm, Cm, Gm

where,

A represents the Attenuation Constant (α)	**B** represents the Phase Constant (β)
C represents Capacitance	**G** represents Conductance
L represents Inductance	**R** represents Resistance
Z represents Impedance	

Viewing Transmission Line Data

This section guides you with opening measured data and viewing the data using transmission line (RLCG) parameter extraction. There are eight transmission line parameters for each transmission line mode. You may elect to view any number of these parameters.

Opening a Transmission Line Plot Window

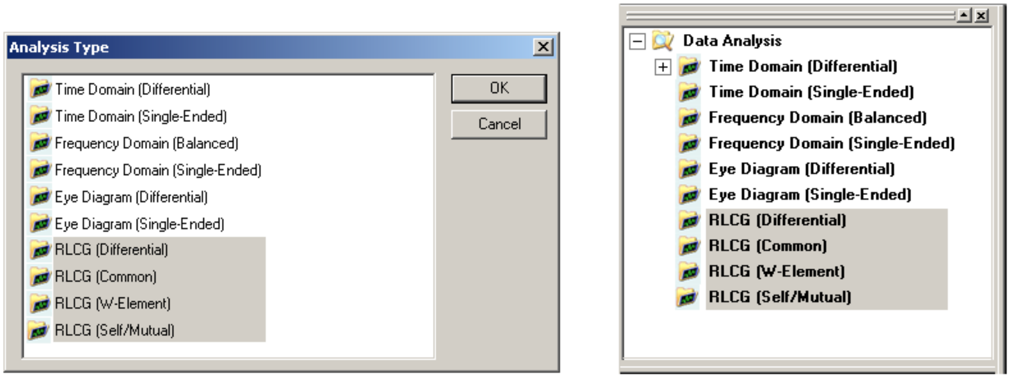
The transmission line parameters may be viewed in any of four modes (differential, common, W-Element, and self/mutual).

You may open the transmission line plot window in one of four ways.

- From the Startup Wizard immediately before selecting the **Measure** button where you must select the analysis type - see (A) of Figure 7-13
- From the Startup Wizard Welcome Screen by selecting **Load Measurement** where you must select the analysis type - see (A) of Figure 7-13
- From the **Open** selection in the **File** menu or the **Open** icon in the **Toolbar** where you must select the analysis type - see (A) of Figure 7-13
- From the **Browser** when data is already being viewed in another analysis type by selecting one of the RLCG choices - see (B) of Figure 7-13

NOTE Using the **Browser** method to open a window requires that you select the parameters that you want to view from the **Parameter Bar** or the **RLCG** menu once the blank plot window is displayed after the T-Line parameters are defined in the next step.

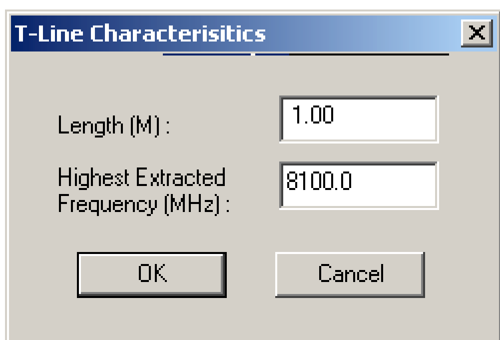
Figure 7-13 Opening the Transmission Line Plot Window



(A) (B)

The *T-Line Characteristics* dialog box (Figure 7-14) is then displayed. Enter the length of the transmission line (in meters) and the highest measured frequency (in megahertz) and then click **OK**.

Figure 7-14 T-Line Characteristics Dialog Box



Length (M) can be used to scale extracted values in units/meter.

Highest Extracted Frequency (MHz) defaults to the stop frequency value. However, this can be set at a lower frequency to better fit your parameters.

The highest extracted frequency is 90% of the maximum measured frequency. For example, the **Highest Extracted Frequency (MHz)** is 45 GHz for a 50 GHz measurement. In the case shown in Figure 7-14, the **Highest Extracted Frequency (MHz)** is 8100 MHz for the 9 GHz measurement. This allows for some guard band of the data, extra bandwidth for use in time to frequency conversions, and to allow some extra frequency range to get good data and allow for time domain roll off.

The individual parameter selections are based on the specific RLCG data analysis type. The following lists each data analysis type and its associated parameters.

RLCG (Differential):	Rd, Ld, Cd, Gd, Zor, Zoi, Ad, Bd
RLCG (Common):	Rc, Lc, Cc, Gc, Zor, Zoi, Ac, Bc
RLCG (W-Element):	R11, L11, C11, G11, R12, L12, C12, G12
RLCG (Self/Mutual):	Rs, Ls, Cs, Gs, Rm, Lm, Cm, Gm

where, **A** represents the Attenuation Constant (α) **B** represents the Phase Constant (β)

C represents Capacitance **G** represents Conductance

L represents Inductance **R** represents Resistance

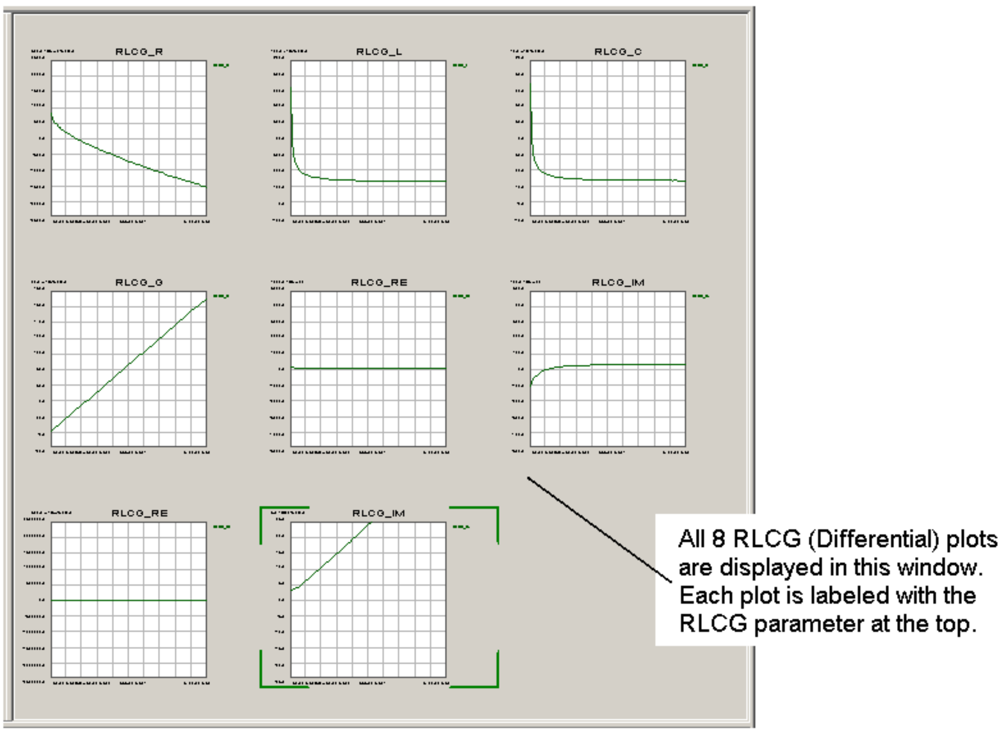
Z represents Impedance

Viewing All Parameters

In all cases, except when you open the plot window using the **Browser**, all eight of the RLCG parameter plots are displayed. Each of the plots is labeled. See Figure 7-15.

As mentioned previously, when you open the plot window from the **Browser**, an empty plot window is displayed. View all eight plots by selecting **All** from the **Parameter Bar** or from the **RLCG** menu with **New Plot** selected.

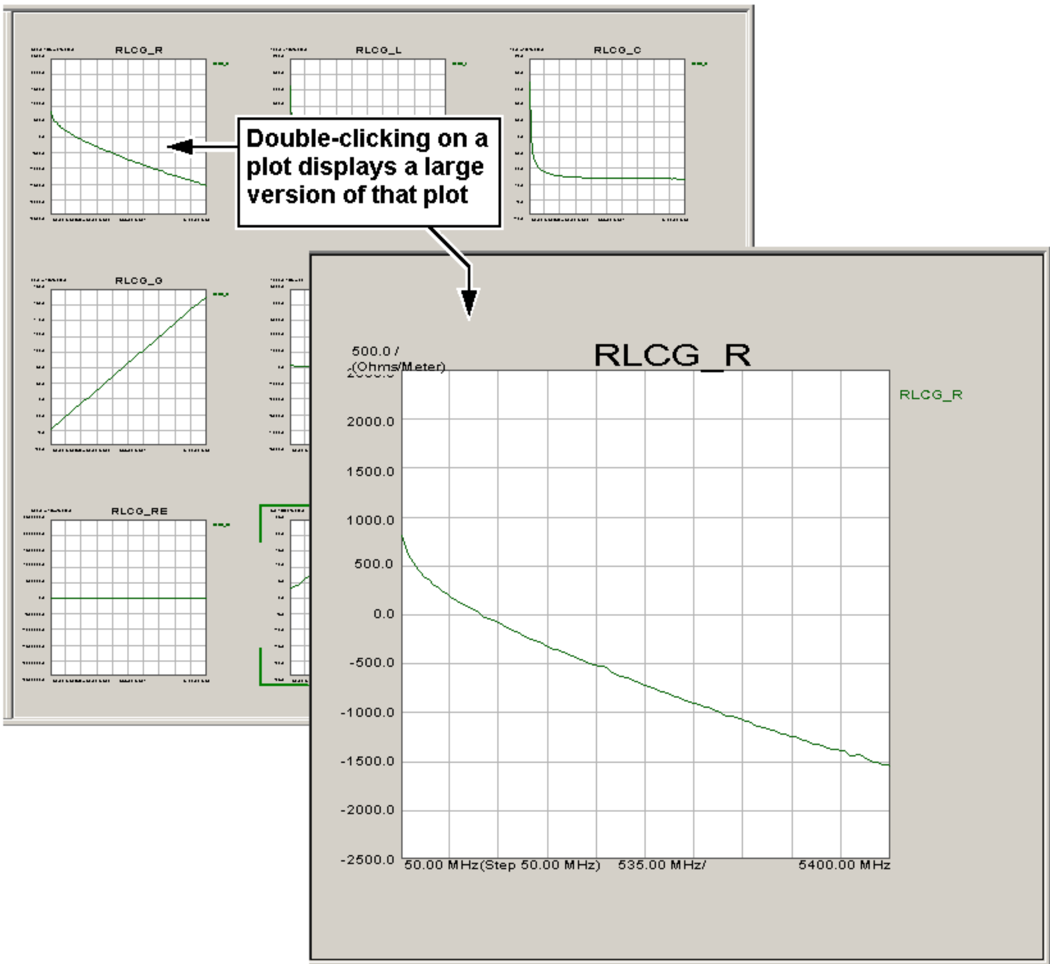
Figure 7-15 All 8 RLCG Plots



Viewing a Single RLCG Parameter

Viewing a single plot gives better resolution. To display a single plot, from the window with all eight plots (or with multiple plots if you have a custom plot window displayed), double-click on the plot that you would like to view. See Figure 7-16.

Figure 7-16 Opening a Single Plot

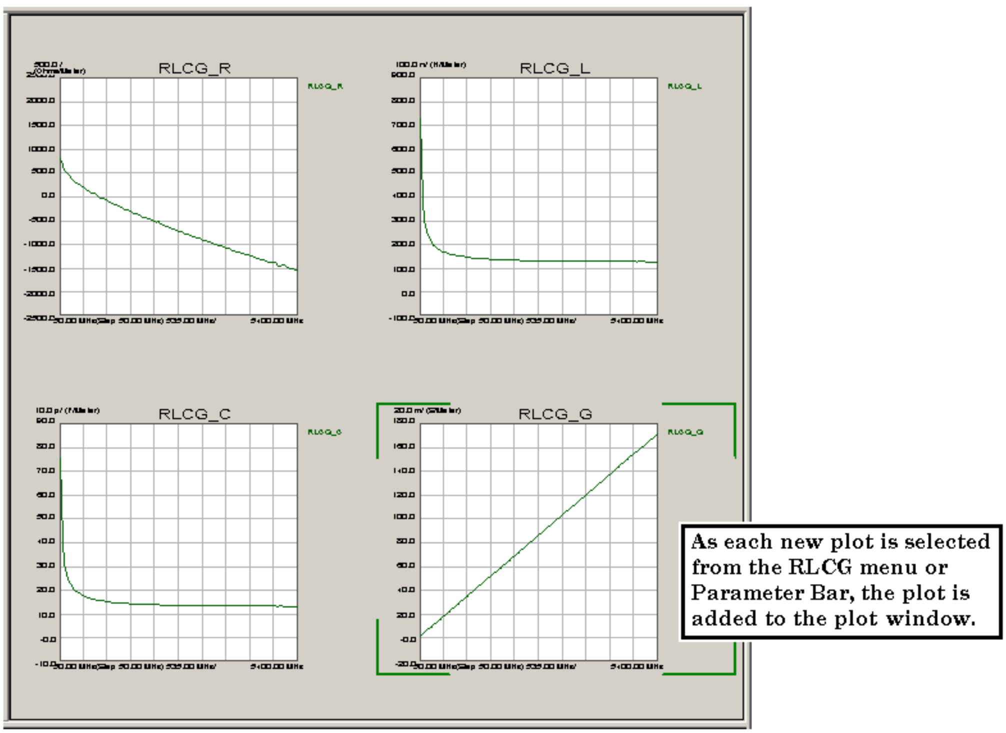


Double-click on the single plot to return to the original view with multiple plots.

Creating a Custom RLCG Plot Window

You can also create a plot window with just the plots you desire. For example, you may want your plot window to show just the four RLCG plots of RLCG (Differential). To create this custom window, open the measured data file in any analysis type. Then, in the **Browser**, select the data type that you want to display the plots. In this example, select **RLCG (Differential)**. A blank plots window is displayed. With **New Plot** selected in the **Parameter Bar** (or the **RLCG** menu), click the desired parameters (R, L, C, and G in this example). As each parameter is selected, a new plot is added to the plots window. See Figure 7-17.

Figure 7-17 Custom Plots Window with Four Plots

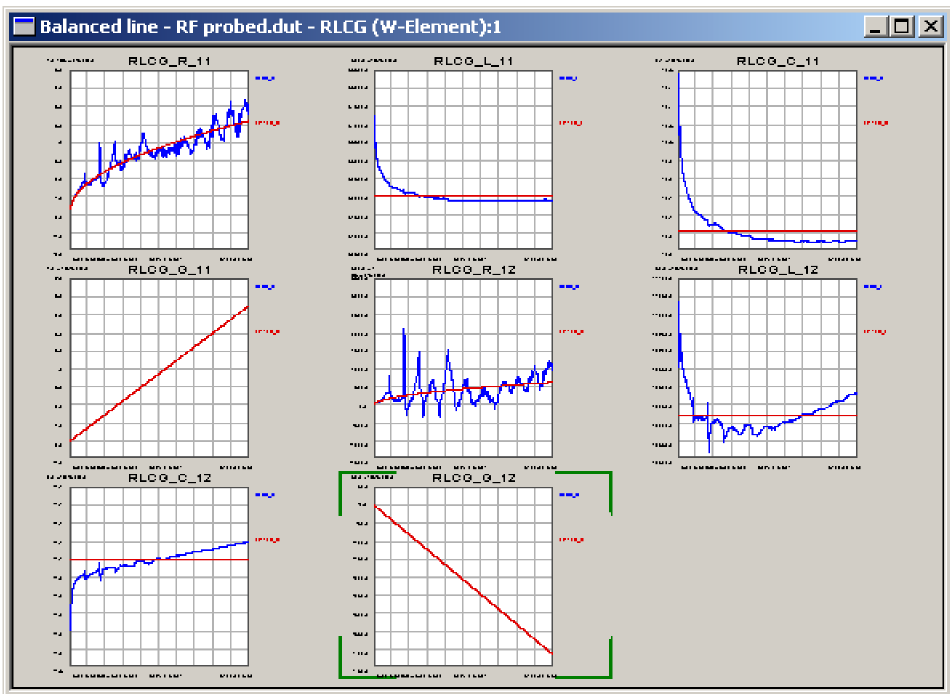


Viewing Fitted and Smoothed RLCG W-Element Traces

When data is viewed in RLCG (W-Element), two traces are displayed in each of the eight plots. The two default traces in each plot are the *Extracted* data (represented by the blue trace) and *Fitted* data (represented by the red trace).

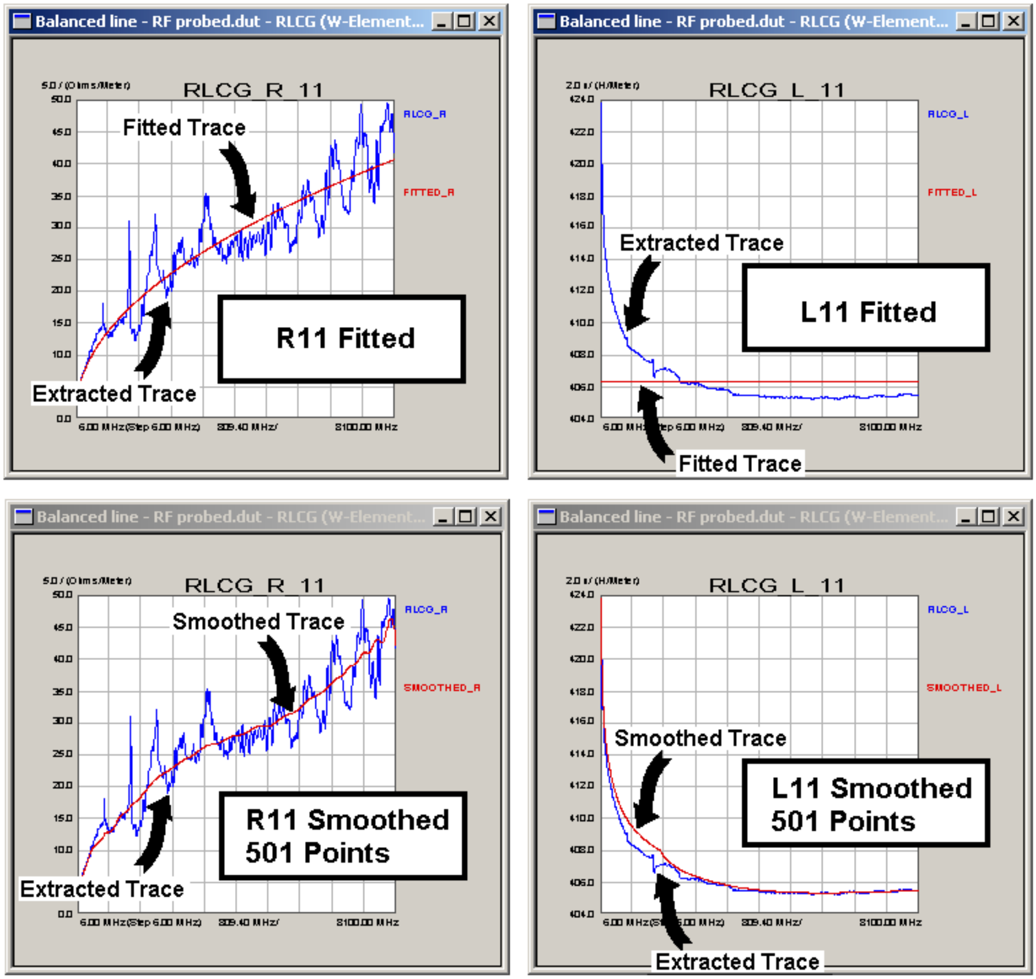
However, both traces may not be readily apparent in every plot initially. Both R (R11 and R12) and both G (G11 and G12) plots may appear to have only the red Fitted traces. The R plots can easily be resolved by autoscaling both of plots (see “Autoscale” on page 238). For the sloping linear G plots, the red Fitted trace is lying exactly on top of the blue Extracted trace. See Figure 7-18.

Figure 7-18 RLCG (W-Element) Display



In W-Element, viewing the Extracted data with the Fitted data is the default status. However, you have the option of viewing the Extracted data with Smoothed data as well. Figure 7-19 shows W-element plots for R11 and L11 extracted data traces. In addition to the Extracted data traces, the upper two plots also show the default Fitted data traces. In the lower two plots, the Extracted data traces are shown with Smoothed data traces.

Figure 7-19 Comparison of Fitted Data and Smoothed Data for R11 and L11



Extracted Data

Extracted data is the data that has been derived (or extracted) from the measured frequency domain values. The blue extracted data trace is always displayed in the W-element. This data can also be exported using the RLCG export feature. See "RLCG" on page 256.

Fitted Data

Fitted data is used to show the general trend of the extracted data using a minimal set of data. The trend is defined by the W-Element model definitions in HSPICE. It is the traditional lossy transmission model common in most simulators. The fitted data set is computed from the extracted data and used to define each of the traces. Fitted data is displayed with a red trace.

- **Resistance (R)**: R has two values for each trace R_0 and R_S. R varies as the square root of frequency, so R at a given frequency point can be calculated with the formula: $R = R_0 + R_S \times \sqrt{frequency}$ where R_0 is the R value at 0 Hertz and R_S is the parameter for the frequency variation.
- **Inductance (L)**: Each L has one value, which is the average of the extracted data values for L and is constant versus frequency.
- **Capacitance (C)**: Like L, each C has one value, which is the average of the extracted data values for C and is constant versus frequency.
- **Conductance (G)**: G has two values for each trace G_0 and G_D, G is linear as a function of frequency, so G at a given frequency point can be calculated with the formula: $G = G_0 + G_D \times frequency$ where G_0 is the G value at 0 Hertz and G_D is the slope value.

From these values, each of the fitted traces are calculated and displayed. These values can also be exported using the RLCG export feature. See “RLCG” on page 256.

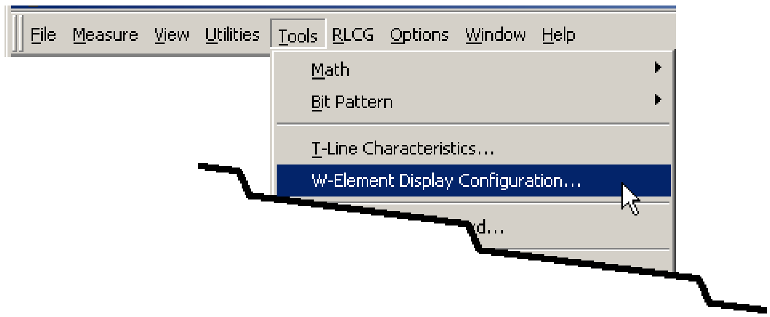
Smoothed Data

Smoothed data is extracted data that has had a smoothing mask applied making the result display the general trends of the extracted data. As additional points are added to the smoothing mask, more of the extended data’s peaks and dips are rounded off (or smoothed). Smoothing uses a running average approach at each data point to smooth the data. For example, if the integer “5” is entered as the smoothing value, at data point *N*, PLTS averages the five values (*N*–2, *N*–1, *N*, *N*+1, *N*+2) for data point *N*. This averaging is used for every data point. Smoothed data is displayed as a red trace. This data can also be exported using the RLCG export feature. See “RLCG” on page 256.

To View the W-Element Data with the Smoothed Data Option

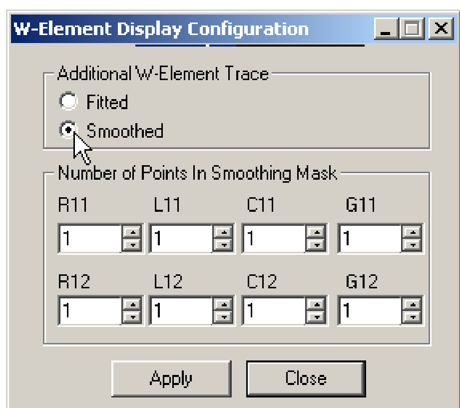
1. From the Tools menu, select W-Element Display Configuration.

Figure 7-20



2. With the **W-Element Display Configuration** dialog box displayed, select **Smoothed** in the **Additional W-Element Trace** area to view the smoothed data.

Figure 7-21 W-Element Display Configuration Dialog Box

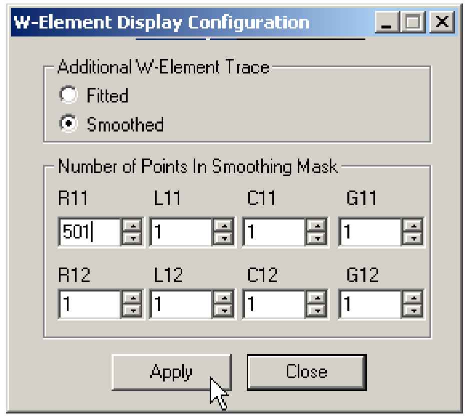


3. In the **Number of Points in Smoothing Mask** area, enter the number of points for the smoothing mask for the desired parameter. Continue entering the smoothing values for all desired parameters.

   In this example, the value `501` was entered as the value in the **R11** box.

   The number of point values must be odd integers.

Figure 7-22 Enter the Number of Masking Points and Apply the Change

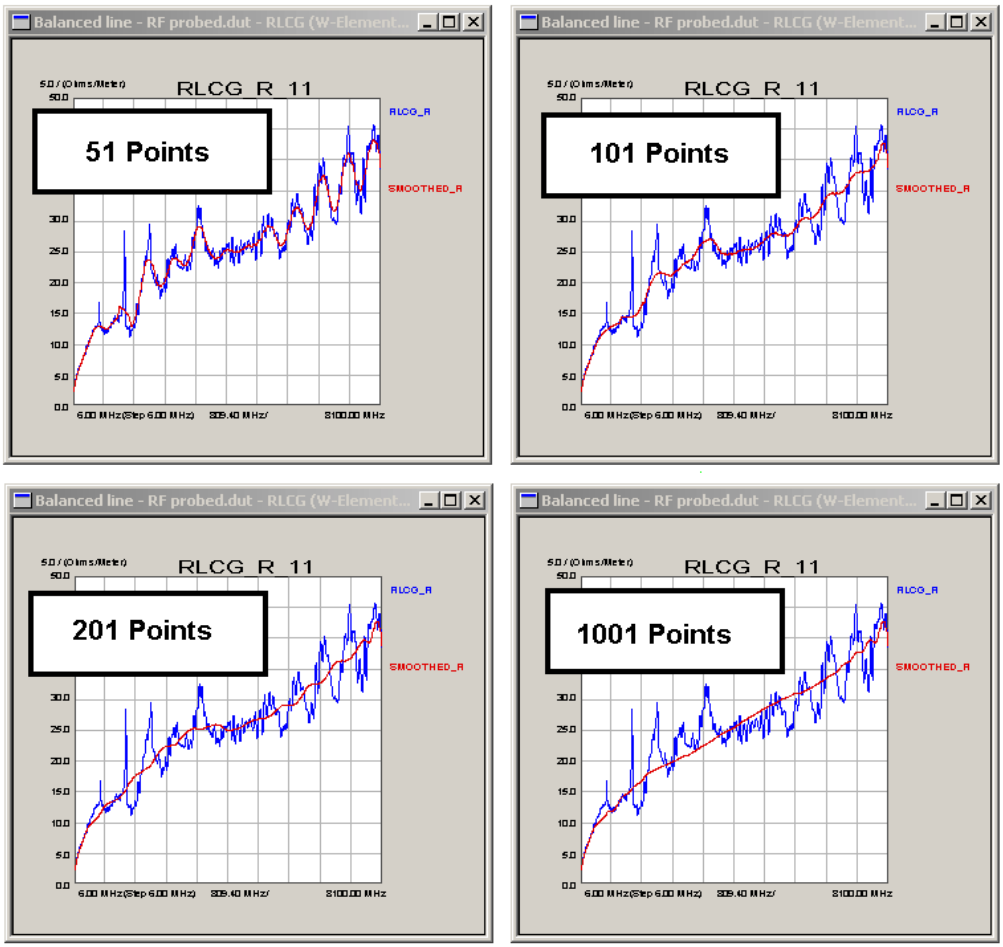


4. Choose the **Apply** button to apply to changes.

   This dialog box remains displayed so that changes to the point values may be optimized or you may return to viewing the Fitted traces. The **Close** button closes the dialog box.

TIP As additional points are added to the smoothing mask, more of the extended data's peaks and dips are rounded off (or smoothed). A change in the number of points at the lower end has a more dramatic smoothing effect than a similar change at the higher end. As an example, the change from 51 to 201 points has a greater effect to the trace than from 201 to 1001 points.

Figure 7-23The Effect of the Number of Points in the Smoothing Mask

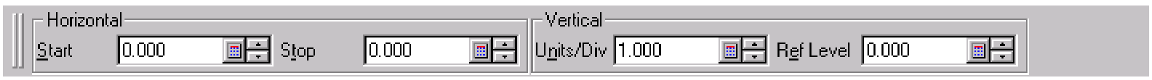


Refine the number of points so the smoothing traces are displayed to meet your requirements.

Setting the Scale

The PLTS software allows you to change the horizontal and vertical scale of the plots using the **Scaling Bar**.

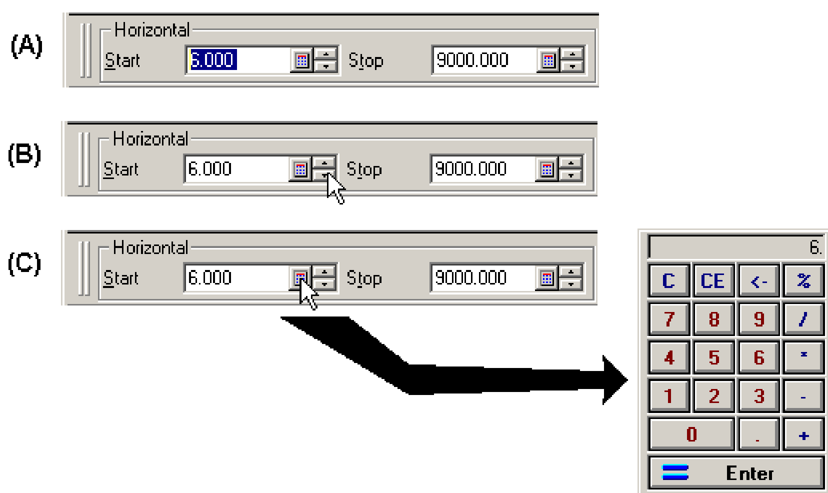
Figure 7-24 Scaling Bar



Change the **Scaling Bar** values by:

- Clicking and dragging within a scaling bar entry box to highlight the current value and then typing the new value. See (A) of Figure 7-25.
- Selecting the up/down arrow buttons to the right of each entry. See (B) of Figure 7-25.
- Selecting the calculator icon to the right of each entry to display a keypad. Click the keypad's numeric buttons to enter a new value and click the **Enter** button to save the new value. See (C) of Figure 7-25. The scaling calculator icon varies slightly between scaling entries meet the requirements of the specific entry.

Figure 7-25 Entering a Scale Value



The horizontal scale is changed by changing the start and stop frequencies in megahertz (MHz). Note that you can not extend the start and stop frequencies beyond the start and stop

frequencies used in the measurement.

The vertical scale units are changed using the same method as used for the horizontal units. The units vary to be appropriate for each plot. For example, when the plot is inductance, the units are in Henrys while when a resistive plots is displayed, the units are ohms.

Quick Scale Features

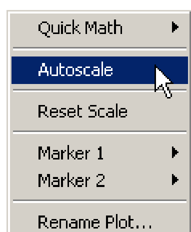
PLTS has three features that make scaling changes quickly and easily. The three features are:

- **Autoscale**
- **Reset Scale**
- **Copy Plot Format** used with **Paste Plot Format** (see "Copying and Pasting Plot Formats" on page 320)

Autoscale

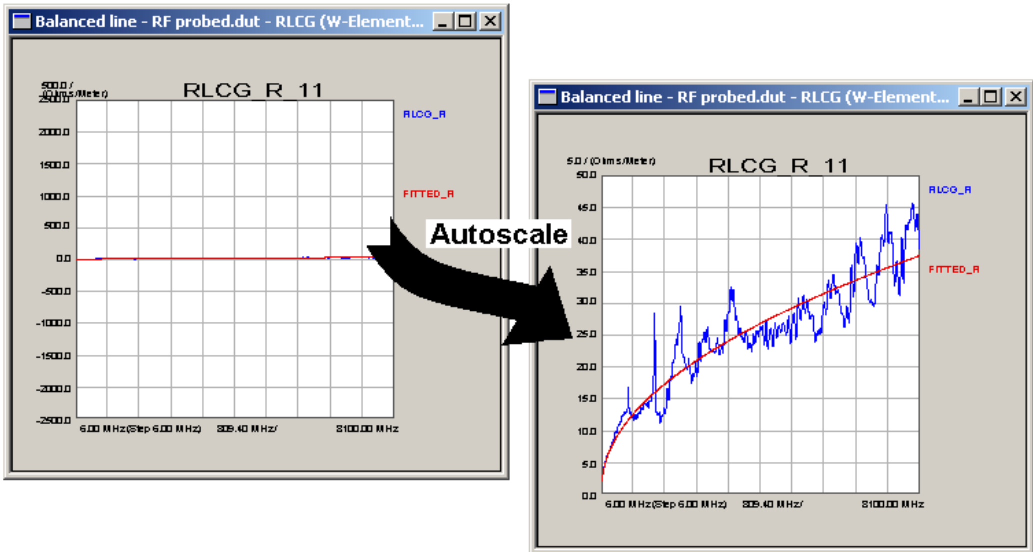
Autoscale changes the vertical scale of the active plot to allow the trace to occupy approximately 80% of the vertical axis of the display. It places the display such that the graticule values are numbers that are easy to work with.

Figure 7-26 Autoscale



To autoscale a plot, select the plot, then right click on the plot to display the quick menu displayed in Figure 7-26. Click **Autoscale** to change the vertical scale of the plot. Figure 7-27 shows an RLCG W-Element Resistance plot that has **Autoscale** applied to it.

Figure 7-27 A Plot that has been Autoscaled

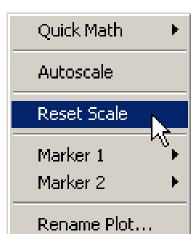


Reset Scale

Reset Scale resets the vertical and horizontal scale of the active plot to the default settings. This is useful when you are adjusting the scale and the trace is moved off screen and can no longer be seen.

To reset the scale of a plot, select the plot, then right click on the plot to display the quick menu displayed in Figure 7-28. Click **Reset Scale** to reset the plot to the default settings.

Figure 7-28 Reset Scale



Exporting Transmission Line Data

The PLTS can export the transmission line data to the main Electronic Design Automation (EDA) software tools used by R&D engineers in the field of signal integrity. These EDA simulators are:

- HSPICE
- Advanced Design System (ADS)

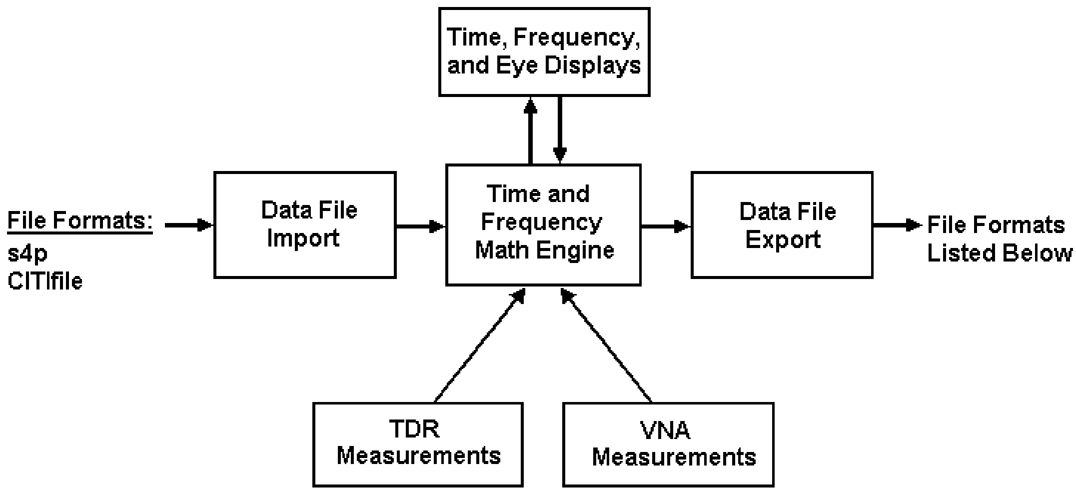
Refer to "Export Data Formats" on page 218 for detailed information on the file formats for exported data.

Refer to "RLCG" on page 256 for detailed information on exporting RLCG data from PLTS.

8 Importing and Exporting Data

In addition to making measurements and reviewing the data files within the software, PLTS has both import and export capabilities. While PLTS makes quick and accurate measurements and provides an excellent platform to view measured data, you will want to import data from other sources that you can view and compare. You will also want to export data for use with modeling and simulation tools. PLTS can be used with a variety of modeling and simulation tools and can import and export in multiple formats. Figure 8-1 shows how the data file import and data file export capabilities blend with the whole PLTS software product.

Figure 8-1 PLTS Data File Import and Export Block Diagram



PLTS exports data files in the following formats:

- Plot to Clipboard
- Plot to Image File
  - Windows Bitmap (.bmp)
  - JPEG Bitmap (.jpg)
  - Targa Bitmap (.tga)
- Frequency Domain
  - Real/Imaginary {comma delimited (.csv) or tab delimited (.txt)}
  - Log Mag {comma delimited (.csv) or tab delimited (.txt)}
  - Linear {comma delimited (.csv) or tab delimited (.txt)}
  - Phase {comma delimited (.csv) or tab delimited (.txt)}
- CITIfile (.cit)
- Touchstone
  - Magnitude, angle (.s4p)
  - dB, angle (.s4p)

- Real, Imaginary (.s4p)
- Time Domain (one or all parameters) {comma delimited (.csv) or tab delimited (.txt)}
- TDA MeasureXtractor
- RLCG
  - HSPICE
    - ❑ W-Element
    - ❑ W-Element Tabular
  - Advanced Design System (ADS)
    - ❑ ML2CTL Model
    - ❑ MDIF

Importing Data Files

Select **Import** from the **File** menu to import a single-ended measurement data file. Then select either **CITIFile** to import a file in CITIfile format or **Touchstone** (S4P) to import a file in Touchstone format. Then select from one of the port selections (either **Port Configuration = 1 & 3 -> 2 & 4** or **Port Configuration = 1 & 2 -> 3 & 4**) based on the calibration type used with the original measurement. Only 4-port data may be imported.

CAUTION PLTS will not import files with Power Sweep data. Attempting to import files with Power Sweep data could cause PLTS to close without notice.

NOTE A CITIfile can include balanced data information, however, only CITIfiles that include 4-port single-ended data may be imported.

Figure 8-2 File Menu with Import Expanded

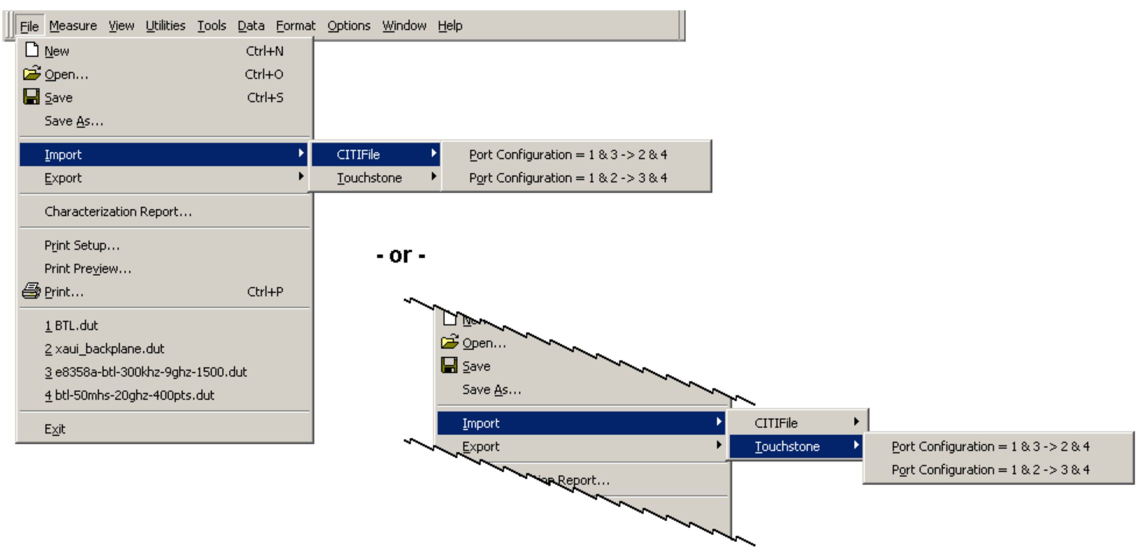
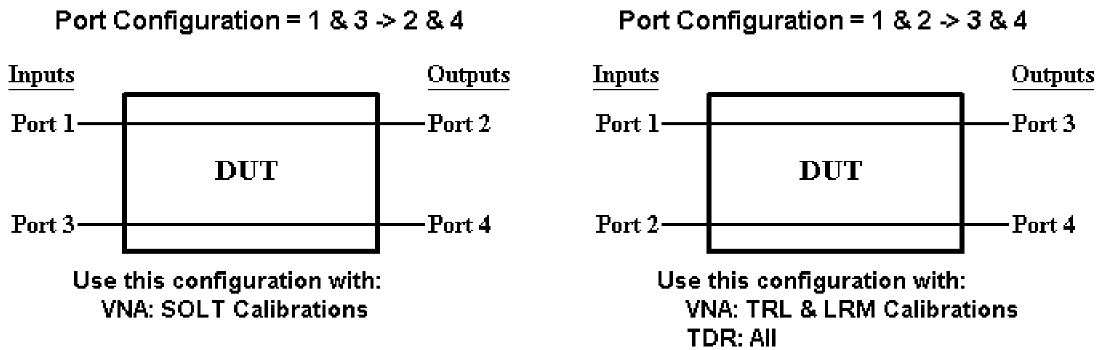


Figure 8-3 Balanced Transform Port Configuration Diagram



CITIFile

CITIFile imports data previously saved in CITIfile (*.cit) format. Files imported in this fashion can be used for comparison with other data sets using trace memory and math functions. Refer to Figure 8-3 and choose from one of the following port selections. This format is described in "CITIfile" on page 259.

Port Configuration = 1 & 3 -> 2 & 4 is used to import single-ended measurement data taken with a VNA-based system calibrated using the SOLT calibration.

Port Configuration = 1 & 2 -> 3 & 4 is used to import single-ended measurement data taken with a VNA-based system calibrated using a TRL or LRM calibration or for all TDR measurements.

Touchstone

Touchstone imports data previously saved in Touchstone (*.S4P) format. Refer to Figure 8-3 and choose from one of the following port selections. This format is described in "S4P (Touchstone)" on page 264.

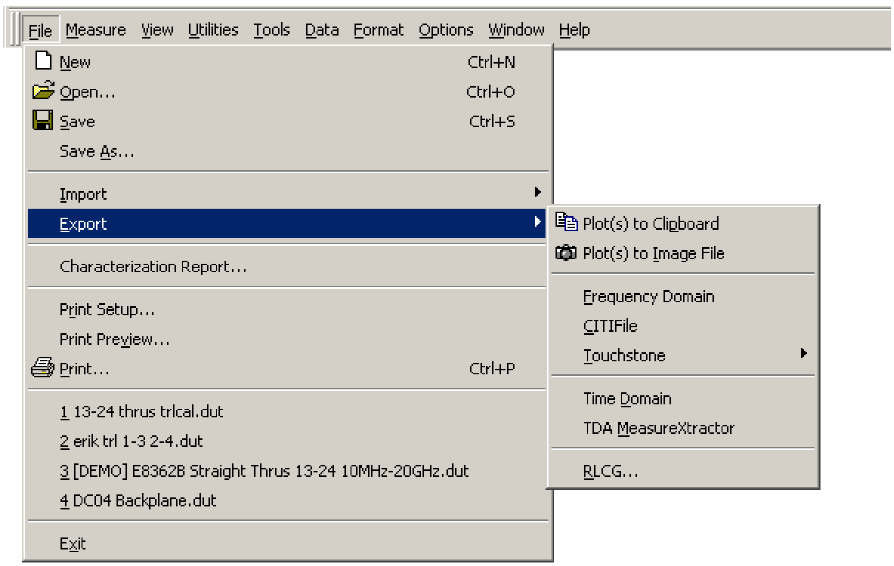
Port Configuration = 1 & 3 -> 2 & 4 is used to import single-ended measurement data taken with a VNA-based system calibrated using the SOLT calibration.

Port Configuration = 1 & 2 -> 3 & 4 is used to import single-ended measurement data taken with a VNA-based system calibrated using a TRL or LRM calibration or for all TDR measurements.

Exporting Data Files

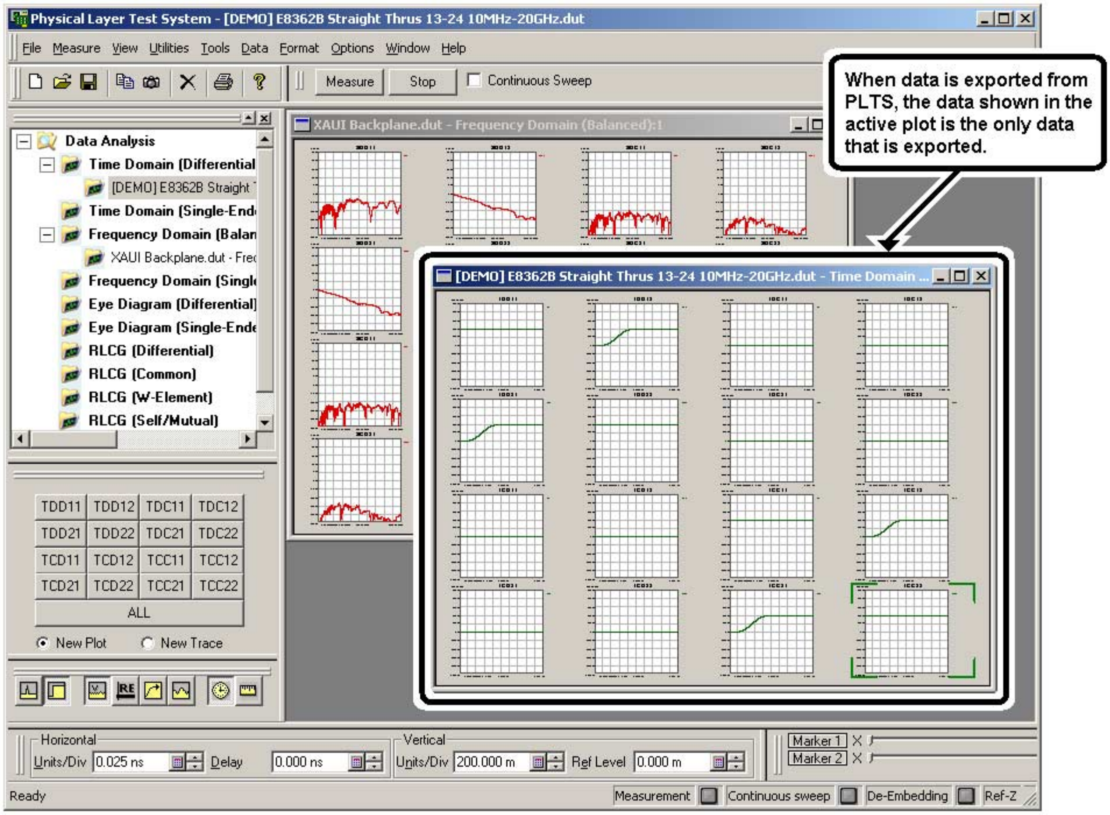
Select **Export** from the **File** menu to export a file. Then select from the following choices to select a specific format: **Plots to Clipboard**, **Plots to Image File**, **Frequency Domain**, **CITIFile**, **Touchstone**, **Time Domain**, **TDA MeasureXtractor**, and **RLCG**.

Figure 8-4 File Menu with Export Expanded



Select **Export** from the **File** menu to export a file. Then select from the following choices to select a specific format: **Plots to Clipboard**, **Plots to Image File**, **Frequency Domain**, **CITIFile**, **Touchstone**, **Time Domain**, **TDA MeasureXtractor**, and **RLCG**.

Figure 8-5 Exported Data: Active Plot Data Only



Plot(s) to Clipboard

Plot(s) to Clipboard exports the active plot window image to the Windows clipboard. It does not copy or export data.

From the **File** menu, select **Export**, then **Plot(s) to Clipboard** to export the active plot window image to Windows clipboard. The contents of the clipboard can then be pasted into other Windows programs.

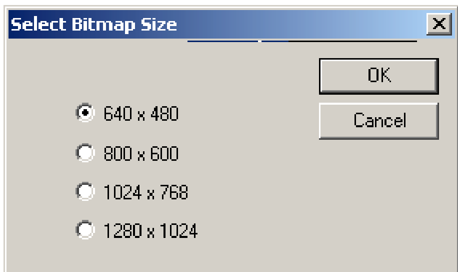
This feature is the same as the **Copy** icon on the **Toolbar**.

Plot(s) to Image File

Plot(s) to Image File exports the contents of the active plot window to an image file.

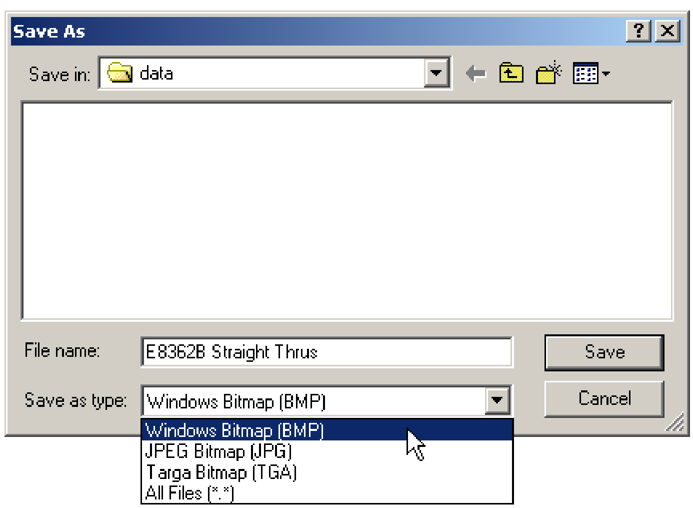
The *Select Bitmap Size* dialog box is displayed giving you the choice of bitmap sizes. Choose from 640 x 460, 800 x 600, 1024 x 768, or 1280 x 1024 pixels. See Figure 8-6.

Figure 8-6 Select Bitmap Size Dialog Box



When you export the image file using the *Save As* dialog box, you may choose from Windows Bitmap (BMP), JPEG Bitmap (JPG), or Targa Bitmap (TGA) formats from the **Save as type:** list. Select **Save** to export the image file. See Figure 8-7.

Figure 8-7 Select Format when Saving the Image File



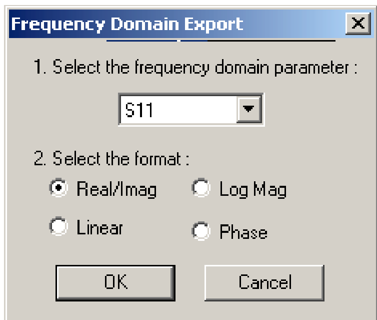
This feature is the same as the **Export Bitmap** icon (the camera icon) in the **Toolbar**.

Frequency Domain

Frequency Domain exports the data of a single S-parameter (either single-ended or balanced) or the data of all 32 single-ended and balanced parameters using one of four frequency domain formats: Real/Imaginary, Log Magnitude, Linear Magnitude, or Phase.

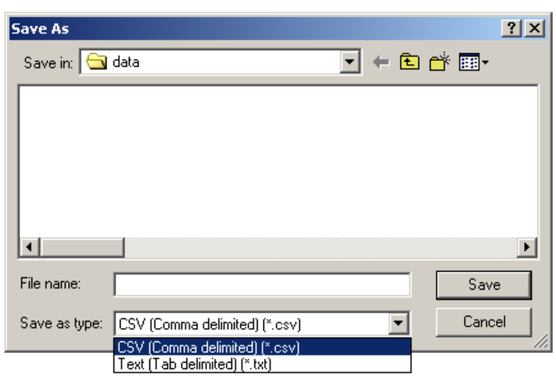
1. From the **File** menu, select **Export**, **Frequency Domain** to display the *Frequency Domain Export* dialog box.

Figure 8-8 Frequency Domain Export Dialog Box



2. In the **Select the frequency domain parameter** list, select any one of the 16 single-ended parameters or any one of the 16 balanced parameters to export. You can also export all 32 of the parameters by selecting **All** at the bottom of the list.
3. In the **Select the format** area, select one of the four formats to save the data as: **Real/Imaginary**, **Log Magnitude**, **Linear Magnitude**, or **Phase**.
4. Select the **OK** button to open the *Save As* dialog box.

Figure 8-9 Save As Dialog Box



5. In the *Save As* dialog box, select the directory that you want to save the data in.
6. In the **File name:** text box, enter a file name for the data to be saved.
7. In the **Save as type:** list, select the file format of the data to be saved.

   You may select either **CSV** (comma-separated variable with .csv extension) or **Text** (tab delimited with .txt extension).
8. Click the **Save** button to save the data.

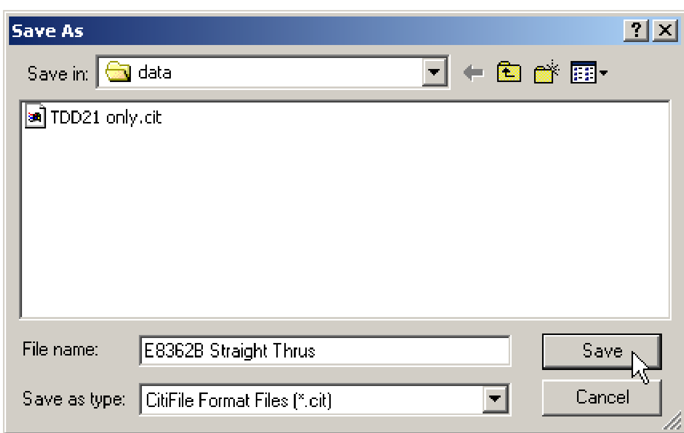
   When a single parameter is selected, the parameters data is saved to the file that you named with the extension that you selected. When all parameters are selected, each parameter is saved to a separate file; thus 32 files are created. Each file is saved with the name you entered. However, each file name has a parameter labeled appended to the name. For example, you entered **`device4`** as the file name, the S11 parameter file is named: **`device4_S11`**

CITIFile

CITIFile exports the current frequency domain data in CITIfile format (*.cit). This format is described in “CITIfile” on page 259.

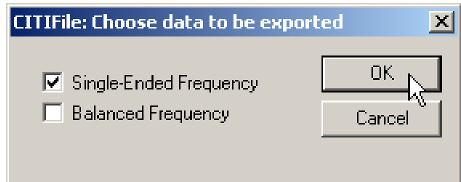
1. From the **File** menu, select **Export**, **CITIFile** to display the *Save As* dialog box. Enter a file name in the **File name:** area and click the **Save** button. See Figure 8-10.

Figure 8-10 CITIFile Save As Dialog Box



2. The *CITIFile: Choose data to be exported* dialog box is displayed. Select **Single-Ended Frequency** and/or **Balanced Frequency** and then click the **OK** button to export the data. See Figure 8-11.

Figure 8-11 CITIFile: Choose data to be exported Dialog Box



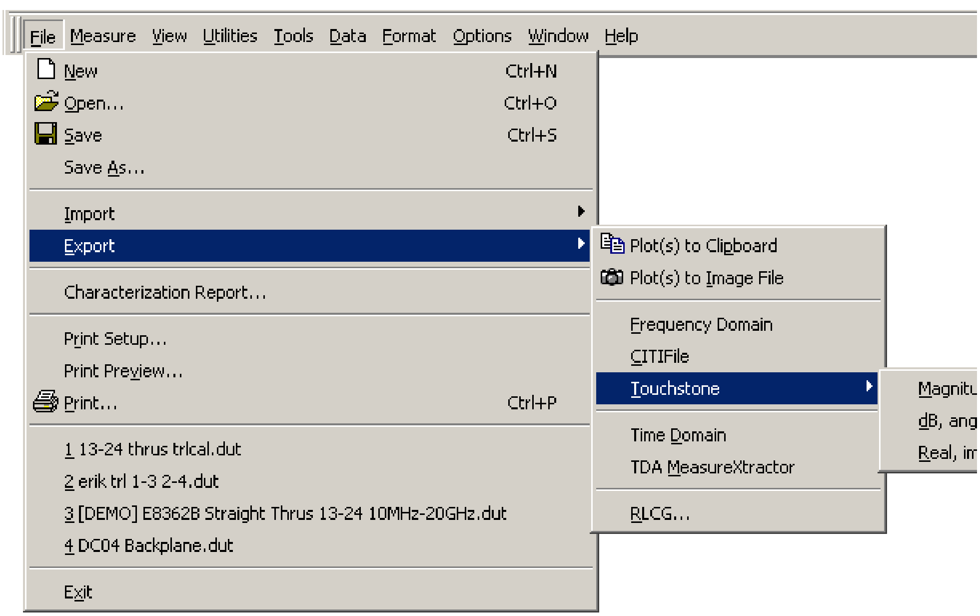
Touchstone

Touchstone exports the current data in the S4P format which also has the following data format choices in which the data may be saved:

- **Magnitude, angle**
- **dB, angle** (power, angle)
- **Real, imaginary**

This format is described in "S4P (Touchstone)" on page 264.

Figure 8-12 File Menu with Export and Touchstone Expanded

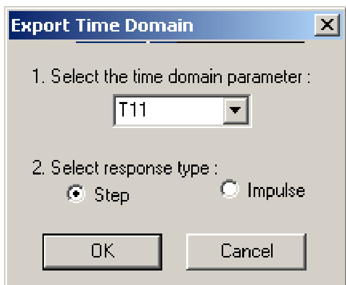


Time Domain

Time Domain exports the data of a single time domain parameter (either single-ended or differential) or the data of all 32 single-ended and differential parameters using either the Step or Impulse response type.

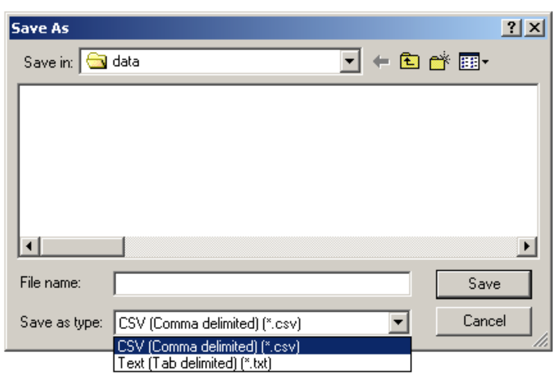
1. From the **File** menu, select **Export**, **Time Domain** to display the *Export Time Domain* dialog box.

Figure 8-13 Export Time Domain Dialog Box



2. In the **Select the time domain parameter** list, select any one of the 16 single-ended parameters or any one of the 16 differential parameters to export. You can also export all 32 of the parameters by selecting **All** at the bottom of the list.
3. In the **Select response type** area, select either: **Step** or **Impulse**
4. Select the **OK** button to open the *Save As* dialog box.

Figure 8-14 Save As Dialog Box



5. In the *Save As* dialog box, select the directory that you want to save the data in.

6. In the **File name:** text box, enter a file name for the data to be saved.
7. In the **Save as type:** list, select the file format of the data to be saved.

   You may select either **CSV** (comma-separated variable with .csv extension) or **Text** (tab delimited with .txt extension).
8. Click the **Save** button to save the data.

   When a single parameter is selected, the parameters data is saved to the file that you named with the extension that you selected. When all parameters are selected, each parameter is saved to a separate file; thus 32 files are created. Each file is saved with the name you entered. However, each file name has a parameter labeled appended to the name. For example, you entered **`device4`** as the file name, the T11 parameter file is named: **`device4_T11`**

TDA Systems MeasureXtractor

TDA MeasureXtractor directly export 4-port S-parameters in Touchstone format for import into TDA Systems' MeasureXtractor. This is a powerful capability that allows you to describe the exact frequency dependent behavior of your passive device using an S-parameter block inside of MeasureXtractor. MeasureXtractor also allows you to create a SPICE compatible model for further circuit simulation.

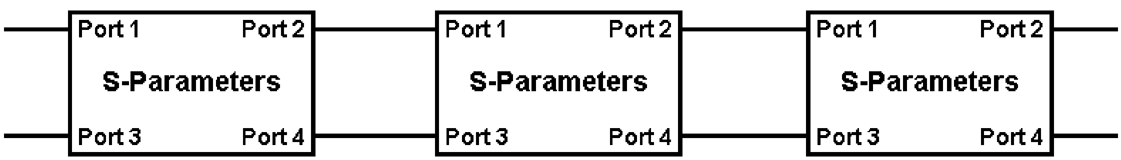
Access the PLTS → TDA MeasureXtractor export feature by selecting **Export** then **TDA MeasureXtractor** from the **File** menu.

File Format and Port Conventions

The format of the 4-port S-parameter Touchstone file is magnitude (in dB) and phase (in degrees) information at each frequency point. Port designations in the resultant Touchstone file are as they appear in the DUT file.

- If you are only using a single S-parameter defined data block in your MeasureXtractor schematic, your results are specific to what you attach to each of the ports.
- If you are using cascaded S-parameter data blocks in your schematic they must have ports 1 and 3 as the input ports and ports 2 and 4 as the output ports (see Figure 8-15). Since TDR measurements typically have ports 1 and 2 as the input ports and 3 and 4 as outputs, a port swap between port 2 and port 3 will need to be performed. You can also read TDR waveforms directly into TDA MeasureXtractor for data driven models.

Figure 8-15 Requirement when S-Parameter Blocks are Cascaded



Recommendations for a Good MeasureXtractor-compatible File

- For TDR-based measurements set the number of points to 2000 or above and make all 16 single-ended measurements (differential measurements are not required). A complete 4-port measurement set is required to export directly to MeasureXtractor. Alternately, you can export selected S-parameters directly from PLTS into a Touchstone file from the **File** menu by selecting **Export**, **Touchstone**, and **Magnitude, angle**.
- When using a VNA, perform your measurements with a 20 MHz start frequency and a 20 GHz stop frequency. Since Start and Step frequency settings are coupled, this will ensure several things including getting about 1000 points of data (so as to avoid undersampling) and an adequate extrapolation down to dc. Not following these

precautions can lead to difficulty with successfully extracting a circuit in MeasureXtractor.

- Maintain good fixturing, calibration practices, and a low IF bandwidth when using a VNA to ensure good reciprocity and prevent your passive device from turning active at any frequency point. The MeasureXtractor software requires good reciprocity (e.g. S12 = S21, S34 = S43 to better than about 2%) for its algorithms to function and utilizes an input checker that will reject files with poor reciprocity. You can visually check for reciprocity or perform math functions within PLTS before exporting to MeasureXtractor as an early indicator.
- Use only short-to-medium length devices. Using devices that are less than 20 wavelengths, or no more than 100 rise times long, will keep extraction time and simulation time of the resultant S-parameter block in MeasureXtractor reasonable.
- Obtain dc response information on your device if possible. This will enhance the accuracy of the model created inside of MeasureXtractor.

Recommendations for Using the Exported File in MeasureXtractor

In TDA Systems' software:

- Change the **Max Frequency** field under **Tools** > **Options** > **Waveform Viewer** to match the maximum frequency of your Touchstone file.
- Follow the guidelines listed in the TDA Systems' documentation for rise time, truncating, and dc values.
- To avoid difficulties in performing the extraction, make sure your device takes up a significant portion of the total time domain window (> 33%). If your device only takes up 10% or less of the time domain window, you can eliminate the lowest frequency data points to shorten the time span by directly editing the Touchstone file.

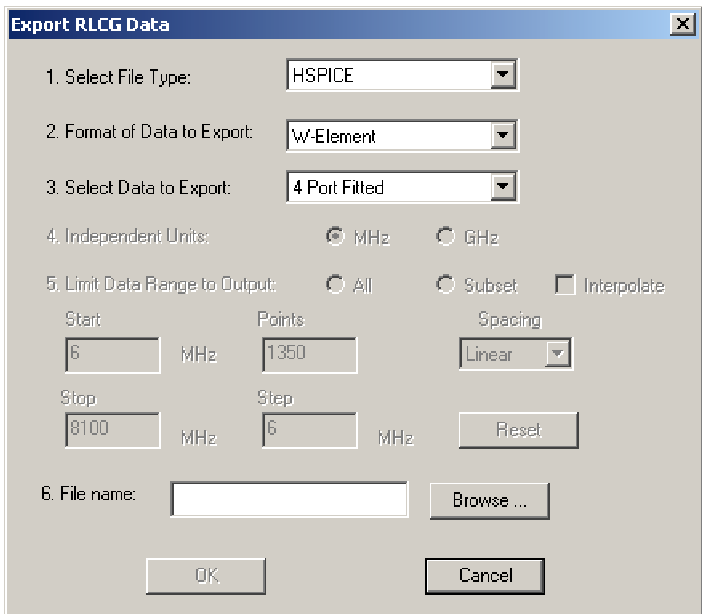
RLCG

RLCG data may be exported from PLTS to be used by other applications. PLTS exports RLCG data in the W-Element mode only. PLTS provides a variety of formats in which you may export data. Refer to "RLCG Data Formats" on page 268 for details on each of the formats.

To export RLCG W-Element data:

1. From the **File** menu, select **Export**, **RLCG**... to display the *Export RLCG Data* dialog box.

Figure 8-16 Dialog Box



2. From the **Select File Type:** list, select either the **HSPICE** or **ADS** choice to match the modeling application with which you will use the exported data.

   The *Export RLCG Data* dialog box exports the current data in one of two RLCG formats:

   - **HSPICE** selection exports the RLCG data in a format compatible with HSPICE (a Simulation Program with Integrated Circuit Emphasis) software. Refer to "Exporting Transmission Line Data" on page 240.
   - **ADS** selection exports the RLCG data in a format compatible with the Agilent Advanced Design System (ADS) software format.

3. From the **Format of Data to Export** list, select the format in which you want to export the RLCG data.

   - If you selected **HSPICE** in step 2, you may select either:
     — **W-Element** which will allow export of 4 Port Fitted data
     — **W-Element Tabular** which will allow export of 4 Port Extracted or 4 Port Smoothed data
   - If you selected **ADS** in step 2, you may select either:
     — **ML2CTL Model** which will allow export of 4 Port Fitted data
     — **MDIF** which will allow export of 4 Port Extracted or 4 Port Smoothed data

4. If you selected either **W-Element Tabular** or **MDIF** in step 3, select the data to export from the **Select Data to Export** list.

   You may select to export either **4 Port Extracted** or **4 Port Smoothed** data.

   The **W-Element Tabular** and **MDIF** selections also allow you to limit the data range that is exported. See "Advanced W-Element Export Features" on page 258 before continuing to step 5.

   If you selected either **W-Element** or **ML2CTL Model** in step 3, **4 Port Fitted** data is the only data type that may be exported. Continue with step 5.

5. In the **File name** box, enter a file name to export the W-Element data. The file extension is applied automatically and varies by selected format. The **Browse**... button may be used to locate a specific file.

6. Click the **OK** button to export the W-Element data to the specified file.

   The **Cancel** button closes the dialog box without exporting W-Element data.

Advanced W-Element Export Features

The **W-Element Tabular** and **MDIF** selections allow you to change the data units that are exported. These selections also allow you to define a subset of the data to export.

Independent Units: In the **Independent Units** area, you may select to export the W-Element data in either **MHz** units or **GHz** units.

Limit Data Range to Output: In the **Limit Data Range to Output** area, you have the option of exporting a subset (or a reduced set) of the data. You may define the subset in a variety of ways.

- **All** is the default setting. This setting will export all of the W-Element data as defined above.
- **Subset** will export a subset of the W-Element data. The **Start**, **Points**, **Spacing**, **Stop**, and **Step** entries are used to define the subset.
  - **Start** is the start frequency of the W-Element data. The initial entry is the lowest frequency of the data. This value may be increased to a higher start frequency.
  - **Stop** is the stop frequency of the W-Element data. The initial entry is the highest frequency of the data. This value may be decreased to a lower start frequency.
  - **Points** is the number of data points measured from the start frequency to the stop frequency. This value is set to the actual number of points in the W-Element data. This value may be increased or decreased.
  - **Step** is the frequency between each data point when **Linear** is selected as the **Spacing** value. This value may be changed to increase or decrease the frequency stepped between the data points.
  - **Spacing** allows you to change from exporting W-Element data in the **Linear** mode (default). This allows you to specify the number of points to export data in logarithmic (**Log**) mode or the points per decade (**Dec**) or octave (**Oct**).
  - **Interpolate** informs you that changes have been made to the settings that require the data to be interpolated.Unchecking this box when **Linear** is selected as the **Spacing** value reverts the data output to the data points actually measured, within the frequency range specified.
  - **Reset** button returns the settings to the original values of the W-Element data.

File Formats

This section describes the S4P (Touchstone) and the CITIfile file formats. The Touchstone file format begins on page 264.

CITIfile

CITIfile is a standardized data format, used for exchanging data between different computers and instruments. CITIfile is an abbreviation for "Common Instrumentation Transfer and Interchange file".

This standard has been a group effort between instrument designers and designers of computer-aided design programs. As much as possible, CITIfile meets current needs for data transfer, and it was designed to be expandable so it can meet future needs.

CITIfile defines how the data inside an ASCII package is formatted. Since it is not tied to any particular disk or transfer format, it can be used with any operating system (BASIC, DOS, UNIX, etc.), with any disk format (LIF, DOS, HFS, etc.), or with any transfer mechanism (disk, LAN, GPIB, etc.).

By careful implementation of the standard, instruments and software packages using CITIfile are able to load and work with data created on another instrument or computer. It is possible, for example, for a network analyzer to directly load and display data measured on a scalar analyzer, or for a software package running on a computer to read data measured on the network analyzer.

NOTE For many data processing applications, the S4P file may provide a more convenient format.

Data Formats

There are two main types of data formats: binary and ASCII. CITIfile uses the ASCII text format. While this format does take up more bytes of space than a binary format, ASCII data is a transportable, standard type of format which is supported by all operating systems. In addition, the ASCII format is accepted by most text editors. This allows files to be created, examined, and edited easily, making CITIfile easier to test and debug.

File and Operating System Formats

CITIfile was designed to be independent of the data storage mechanism, and therefore may be implemented for any file system. However transfer between file systems may sometimes be

necessary. Any commercially available software that has the ability to transfer ASCII files between systems may be used to transfer CITIfile data.

Definition of CITIfile Terms

This section will define the following terms: *package, header, data array, keyword*

Package A typical package is divided into two parts: The first part, the header, is made up of keywords and setup information. The second part, the data, usually consists of one or more arrays of data. Example 1 shows the basic structure of a CITIfile package:

Example 1, A CITIfile Package

The "header" part	CITIFILE A.01.00
	NAME MEMORY
	VAR FREQ MAG 3
	DATA S RI
The "data" part	BEGIN
	-3.54545E-2, -1.38601E-3
	0.23491E-3, -1.39883QE-3
	2.00382E-3, -1.40022E-3
	END

When stored in a disk file there may be more than one CITIfile package.

The CITIfile Header The header section contains information about the data that will follow. It may also include information about the setup of the instrument that measured the data. For example, the header may include information such as:

- CITIfile version number
- Network analyzer model number
- Firmware revision currently installed in the analyzer
- Type of Data
- Data Format
- Measurement parameters
- Start and stop frequencies
- Number of sample points

The CITIfile header shown in Example 1 has just the bare minimum of information necessary; no instrument setup information was included.

An Array of Data An array is numeric data that is arranged with one data element per line.

In the Smith chart and polar formats, the data is in real and imaginary pairs. In all other formats, the data is still in pairs, but the second term of the pair is 0E0. All information is true formatted data in the same format as on the analyzer display (dB, SWR, etc.).

A CITIfile package may contain more than one array of data. Arrays of data start after the BEGIN keyword, and the END keyword will follow the last data element in an array. A CITIfile package does not necessarily need to include data arrays; for instance, CITIfile could be used to store the current state of an instrument. In that case the keywords VAR, DATA, BEGIN, and END would not be required.

CITIfile Keywords

Table 8-1 CITIfile Keywords

Keyword	Explanation and Examples
CITIFILE	CITIFILE A.01.01 identifies the file as a CITIfile, and indicates the revision level of the file. The CITIfile keyword and revision code must precede any other keywords. The CITIfile keyword at the beginning of the package assures the device reading the file that the data that follows is in the CITIfile format. The revision number allows for future extensions of the CITIfile standard. The revision code shown here following the CITIfile keyword indicates that the machine writing this file is using the A.01.01 version of CITIfile as defined here. Any future extensions of CITIfile will increment the revision code.
NAME	NAME CAL_SET allows the current CITIfile “package” to be named. The name of the package should be a single word with no embedded spaces. A list of standard package names follows:
Label	Definition.
RAW_DATA	Uncorrected data.
DATA	Data that has been error corrected.When only a single data array exists, it should be named DATA.
FORMATTED	Corrected and formatted data.
MEMORY	Data trace stored for comparison purposes.
CAL_SET	Coefficients used for error correction.
CAL_KIT	Description of the standards used.
DELAY_TABLE	Delay coefficients for calibration.

Table 8-1 CITIfile Keywords

Keyword	Explanation and Examples
VAR	VAR FREQ MAG 201 defines the name of the independent variable (FREQ), the format of values in a VAR_LIST_BEGIN table (MAG, if used), and the number of data points (201).Typical names for the independent variable are FREQ (in Hz), TIME (in seconds), and POWER (in dBm). For the VAR_LIST_BEGIN table, only the "MAG" format is supported at this point. # #NA POWER1 1.0E1 allows variables specific to a particular type of device to be defined. The pound sign (#) tells the device reading the file that the following variable is for a particular device. The "NA" shown here indicates that the information is for a Network Analyzer. This convention allows new devices to be defined without fear of conflict with keywords for previously defined devices. The device identifier (i.e. NA) may be any number of characters.
SEG_LIST_BEGIN	SEG_LIST_BEGIN indicates that a list of segments for the independent variable follow. Format for the segments is: [segment type] [start] [stop] [number of points]. The current implementation only supports a single segment. If there is more than one segment, the VAR_LIST_BEGIN construct is used. CITIfile revision A.01.00 supports only the SEG (linear segment) segment type.
SEG_LIST_END	SEG_LIST_END defines the end of a list of independent variable segments.
VAR_LIST_BEGIN	VAR_LIST_BEGIN indicates that a list of the values for the independent variable (declared in the VAR statement) follow. Only the MAG format is supported in revision A.01.00.
VAR_LIST_END	VAR_LIST_END defines the end of a list of values for the independent variable.
DATA	DATA S[1,1] RI defines the name of an array of data that will be read later in the current CITIfile package, and the format that the data will be in. Multiple arrays of data are supported by using standard array indexing. Versions A.01.00 and A.01.01 of CITIfile only support the RI (real and imaginary) format, and a maximum of two array indexes. Commonly used array names include the following: "S" for "S parameter" Example: S[2,1] "E" for "Error term" Example: E[1] "USER" for "User parameter" Example: USER[1] "VOLTAGE" Example: VOLTAGE[1] "VOLTAGE_RATIO" for a ratio of Example: VOLTAGE_RATIO[1,0] two voltages (A/R).

Table 8-1 CITIfile Keywords

Keyword	Explanation and Examples
CONSTANT	CONSTANT [name] [value] allows for the recording of values which don't change when the independent variable changes. CONSTANTs are part of the main CITIfile definition. Users must not define their own CONSTANTs. Use the #KEYWORD device specification to create your own KEYWORD instead. The #NA device specification is an example of this. No constants were defined for revision A.01.00 of CITIfile. CITIfile revision A.01.01 defined the following constant: CONSTANT TIME [year] [month] [day] [hour] [min] [secs] Example: CONSTANT TIME 1999 02 26 17 33 53.25

- The COMMENT statement is not absolutely required, but is highly recommended to aid readability.
- The year should always be the full four digits ("1999" is correct, but "99" is not). This is to avoid problems with the year 2000, when the shortened version of the year will be "00."
- The hour value should be in 24-hour "military" time.
- When writing a CITIfile and the fractional seconds value is zero, then the "seconds" value may be printed either with or without a decimal point: either "47.0" or "47" would be acceptable. When reading a CITIfile, the seconds value should always be read as if it were a floating point number.

S4P (Touchstone)

These files contain S-parameters that describe frequency-dependent linear networks for 4-port components. The S4P data file format is also known as Touchstone format.

An *.s4p* file can be used to model the behavior of a linear device using S-parameters. The file contains the S-parameters, the component with a link to the file is placed within the schematic.

This section describes:

- An overview of the S4P file
- The basic S4P format
- The basic S4P format applied to S-parameters, plus example

S4P data files are ASCII text files in which data appears line by line, one line per data point, in increasing order of frequency. Each line of data consists of a frequency value and one or more pairs of values for the magnitude and phase of each S-Parameter at that frequency. Values are separated by one or more spaces, tabs or commands. Comments are preceded by an exclamation mark (!). Comments can appear on separate lines, or after the data on any line or lines. Extra spaces are ignored. PLTS uses the following format for 4-port touchstone file identification: filename.s4p

Basic S4P File Format

The following example shows the general format for component data files. It consists of:

- An option line
- Data lines
- Comments

The Option Line

The option line, specifying the frequency units and the normalizing impedance, precedes the data lines.

option line – *# <frequency unit> <parameter> <format> <R n>*

<data line>

...

<data line>

where,

= The delimiter that tells the program you are specifying these parameters

frequency units = The units desired (GHz, MHz, KHz, or Hz)

parameter = The parameter desired ("S" for S4P components)

format = The format desired (DB for dB-angle, MA for magnitude-angle, and RI for real-imaginary)

R n = The reference resistance in ohms, where n is a positive number of ohms of the real impedance to which the parameters are normalized)

In summary, the option line should read: `# [HZ/KHZ/MHZ/GHZ] [S] [MA/DB/RI] [R n]`

Where square brackets [...] indicate optional information; .../.../.../ indicates that you select one of the choices; and, n is replaced by a positive number.

The default option line for component data files is:

```
# GHZ S MA R 50
```

An example of an Option Line for a file with Frequency in GHz, S-parameters in real-imaginary format, normalized to 100 ohms:

```
# GHz S RI R 100
```

Data Line

Data lines contain the data of interest. For 4-port data files, the network parameters appear in the file in a matrix form, each row starting on a separate line. A maximum of four network parameters (with 2 real numbers for each) appear on any line. The remaining network parameters are continued on as many additional lines as are needed.

Data Line Formats When you type the data below the option line, the columns need not line up precisely like those shown. The syntax for data is as follows:

4-Port Components Magnitude-Angle format:

Freq	Mag	Ang	Mag	Ang	Mag	Ang	Mag	Ang
f	*S11*	*S11*	*S12*	*S12*	*S13*	*S13*	*S14*	*S14*
	S21	*S21*	*S22*	*S22*	*S23*	*S23*	*S24*	*S24*
	S31	*S31*	*S32*	*S32*	*S33*	*S33*	*S34*	*S34*
	S41	*S41*	*S42*	*S42*	*S43*	*S43*	*S44*	*S44*

where *f* = Frequency,
Mag = Magnitude of S-parameter, and
Ang = Angle of S-parameter

Comments

You can document your data files by preceding a comment with the exclamation mark (!) on any line. A comment can be the only entry on a line or can follow the data on any line.

S4P Format Examples

Here are formatting reference examples for S-parameter files.

S4P files can have MA, RI, or DB formats. The following examples show the format of each format style.

The MA file format is:

```
# frequency_unit S MA R impedance ! 1st row
```

freq	*magS11*	*angS11*	*magS12*	*angS12*	*magS13*	*angS13*	*magS14*	*angS14*
	magS21	*angS21*	*magS22*	*angS22*	*magS23*	*angS23*	*magS24*	*angS24*
	magS31	*angS31*	*magS32*	*angS32*	*magS33*	*angS33*	*magS34*	*angS34*
	magS41	*angS41*	*magS42*	*angS42*	*mag43*	*angS43*	*magS44*	*angS44*

where *freq* = Frequency,
mag = Magnitude of S-parameter, and
ang = Angle of S-parameter

The RI file format is:

```
# frequency_unit S RI R impedance ! 1st row
```

freq	*realS11*	*imagS11*	*realS12*	*imagS12*	*realS13*	*imagS13*	*realS14*	*imagS14*
	realS21	*imagS21*	*realS22*	*imagS22*	*realS23*	*imagS23*	*realS24*	*imagS24*
	realS31	*imagS31*	*realS32*	*imagS32*	*realS33*	*imagS33*	*realS34*	*imagS34*
	realS41	*imagS41*	*realS42*	*imagS42*	*real43*	*imagS43*	*realS44*	*imagS44*

where *freq* = Frequency,
real = Real portion of S-parameter value, and
imag = Imaginary portion of S-parameter value

The DB file format is:

```
# frequency_unit S DB R impedance ! 1st row
```

freq	*dBS11*	*angS11*	*dBS12*	*angS12*	*dBS13*	*angS13*	*dBS14*	*angS14*
	dBS21	*angS21*	*dBS22*	*angS22*	*dBS23*	*angS23*	*dBS24*	*angS24*
	dBS31	*angS31*	*dBS32*	*angS32*	*dBS33*	*angS33*	*dBS34*	*angS34*
	dBS41	*angS41*	*dBS42*	*angS42*	*db43*	*angS43*	*dBS44*	*angS44*

where *freq* = Frequency,
dB = dB value of S-parameter, and
Ang = Angle of S-parameter

4-Port S4P (Touchstone) File Example

The following example shows an S4P file in Magnitude format with the frequencies displayed in GHz, and having a 50-ohm reference resistance. This examples shows actual data at three frequency points.

```
# GHZ S MA R 50
5.00000  0.60262  161.240   0.40611  -42.2029  0.42918  -66.5876  0.53640  -79.3473
         0.40611  -42.2029  0.60262  161.240   0.53640  -79.3473  0.42918  -66.5876
         0.42918  -66.5876  0.53640  -79.3473  0.60262  161.240   0.40611  -42.2029
         0.53640  -79.3473  0.42918  -66.5876  0.40611  -42.2029  0.60262  161.240
6.00000  0.57701  150.379   0.40942  -44.3428  0.41011  -81.2449  0.57554  -95.7731
         0.40942  -44.3428  0.57701  150.379   0.57554  -95.7731  0.41011  -81.2449
         0.41011  -81.2449  0.57554  -95.7731  0.57701  150.379   0.40942  -44.3428
         0.57554  -95.7731  0.41011  -81.2449  0.40942  -44.3428  0.57701  150.379
7.00000  0.50641  136.693   0.45378  -46.4151  0.37845  -99.0918  0.62802  -114.196
         0.45378  -46.4151  0.50641  136.693   0.62802  -114.196  0.37845  -99.0918
         0.37845  -99.0918  0.62802  -114.196  0.50641  136.693   0.45378  -46.4151
         0.62802  -114.196  0.37845  -99.0918  0.45378  -46.4151  0.50641  136.693
```

RLCG Data Formats

When RLCG data is opened in the W-Element, it may be exported for use in other modeling applications, such as HSPICE or Agilent ADS. You may choose to export either extracted, fitted, or smoothed data. This section provides some information on the exported files. Refer to "RLCG" on page 256 for the export procedure.

- Fitted data may be export using the HSPICE W-Element or the ADS ML2CTL Model format. Exporting data in these formats creates a file that is very small in size (under 1 kB for all fitted traces.) See "HSPICE W-Element/ADS ML2CTL Model Formats" below for more information on this format.
- Extracted data or smoothed data may be exported using the HSPICE W-Element Tabular or the ADS MDIF format. Exporting data in these formats creates files that are large in size (over 250 kB for all extracted or smoothed traces with a likelihood for much larger files.) These files save the R, L, C, and G data at each frequency point which means when there are more frequency points in the data, larger files are created. See "HSPICE W-Element Tabular/ADS MDIF Formats" on page 270 for more information on this format.

  However, PLTS has the ability to export a subset of this large extracted or smoothed data set. You may define a subset of this extracted or smoothed data to export. Exporting a subset, depending on the subset that you define, can result in significantly smaller files sizes. See "Subset" on page 272 for more information on the subset format.

HSPICE W-Element/ADS ML2CTL Model Formats

W-Element/ML2CTL Model use just a few values to describe the RLCG W-Element plots. Only Fitted data may be exported using the W-Element/ML2CTL Model selections (see "Fitted Data" on page 233).

For example, a complete set of exported values for the HSPICE W-Element Fitted format includes the following parameters:

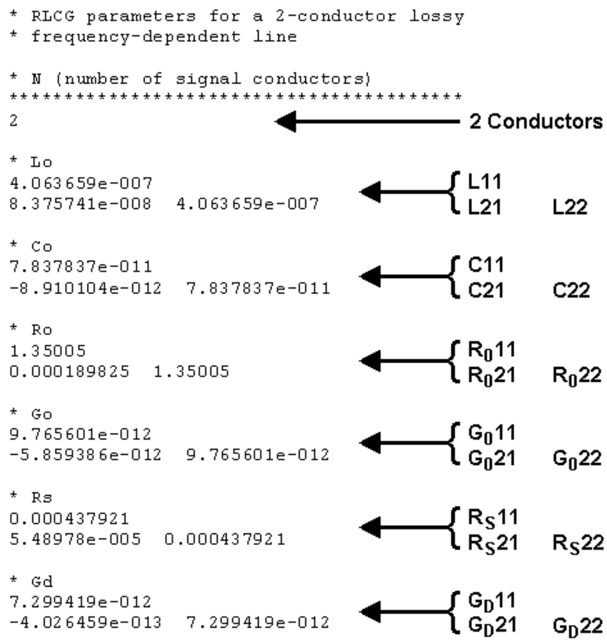
$R_0$11, $R_0$12, $R_0$22, R_s11, R_s12, R_s22, L11, L12, L22, C11, C12, C22, $G_0$11, $G_0$12, $G_0$22, G_D11, G_D12, and G_D22

As shown above, each R, L, C, and G has three values; one for each of the differential lines (i.e. L11 and L22) and one for the coupling between the lines (i.e. L12).

This data is imported into HSPICE or ADS where the modeling software uses these values in fixed equations of a given form to reconstruct (or "*fit*" the data as closely as possible to) the original extracted data plots. These values are used to reconstruct the original extracted R, L, C, and G plots based on a set of rules for each trace. 8-17 shows a file after it was exported

from PLTS using the HSPICE W-Element Fitted format.

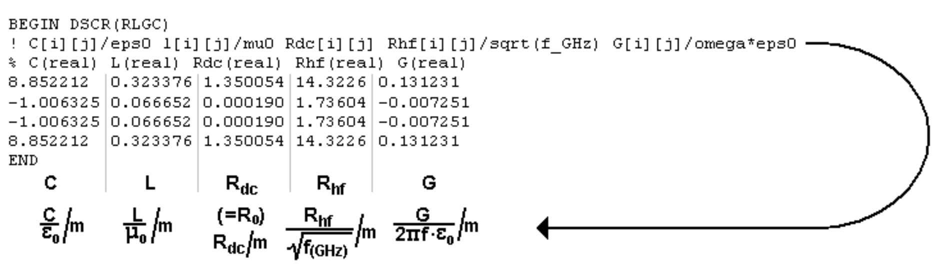
Figure 8-17 HSPICE W-Element Fitted Format



```
* RLCG parameters for a 2-conductor lossy
* frequency-dependent line

* N (number of signal conductors)
*****************************************
2

* Lo
4.063659e-007
8.375741e-008   4.063659e-007

* Co
7.837837e-011
-8.910104e-012   7.837837e-011

* Ro
1.35005
0.000189825   1.35005

* Go
9.765601e-012
-5.859386e-012   9.765601e-012

* Rs
0.000437921
5.48978e-005   0.000437921

* Gd
7.299419e-012
-4.026459e-013   7.299419e-012
```


8-18 shows a file after it was exported from PLTS using the ADS ML2CTL Model Fitted format. It uses a similar set of data as the HSPICE W-Element Fitted format above.

Figure 8-18 ADS ML2CTL Model Fitted Format



```
BEGIN DSCR(RLGC)
! C[i][j]/eps0 l[i][j]/mu0 Rdc[i][j] Rhf[i][j]/sqrt(f_GHz) G[i][j]/omega*eps0
% C(real) L(real) Rdc(real) Rhf(real) G(real)
8.852212   0.323376 1.350054 14.3226 0.131231
-1.006325 0.066652 0.000190 1.73604 -0.007251
-1.006325 0.066652 0.000190 1.73604 -0.007251
8.852212   0.323376 1.350054 14.3226 0.131231
END
```

HSPICE W-Element Tabular/ADS MDIF Formats

W-Element Tabular/MDIF export all of the values in a tabular for. For each frequency point the R, L, C, and G values are saved. Extracted and Smoothed data may be exported using the W-Element Tabular/MDIF selections (see "Extracted Data" on page 232 and "Smoothed Data" on page 233).

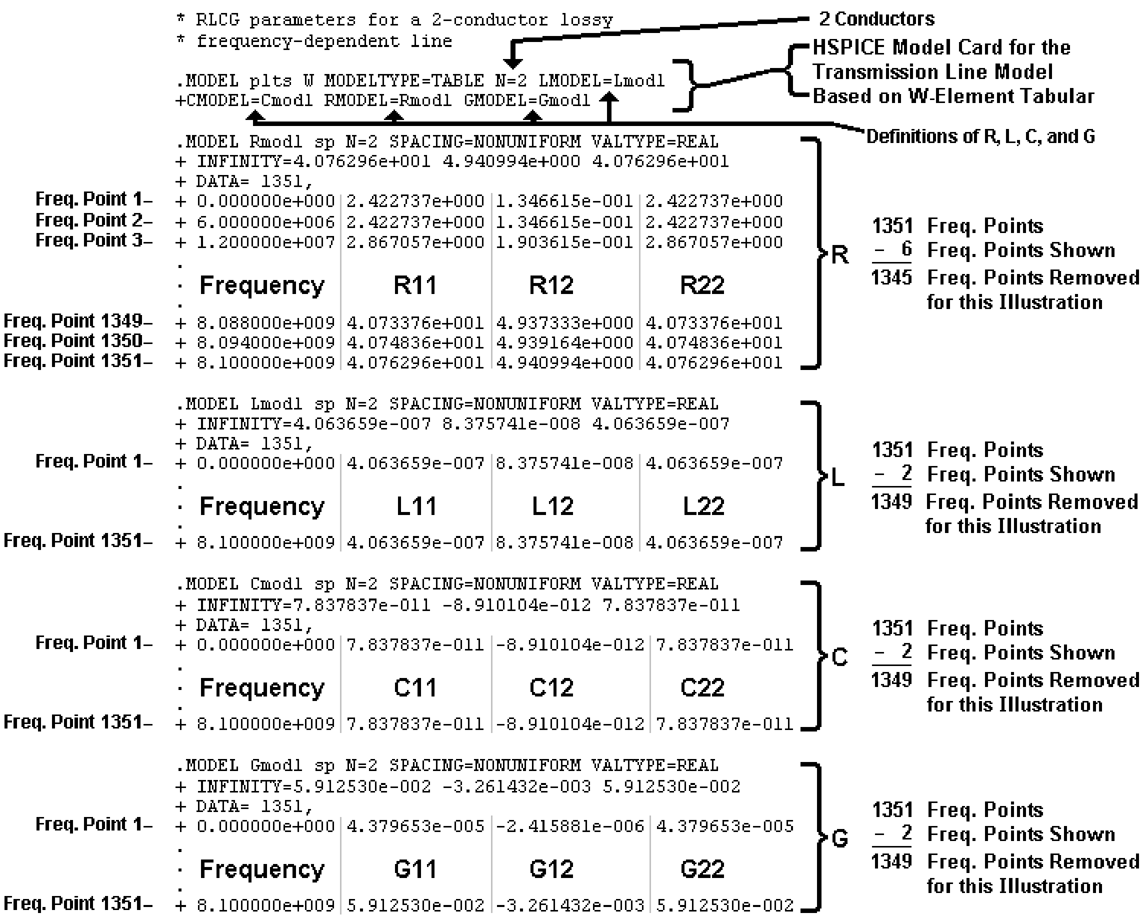
Instead of trying to fit the data to resemble the extracted data, W-Element Tabular/MDIF allows the exported data to resemble the actual extracted traces.

- In W-Element Tabular format, if you have 1000 frequency points, you will have 3000 values for R (1000 for R11, 1000 for R12, and 1000 for R22), 3000 values for L, 3000 values for C, and 3000 values for G. In this format, R21/L21/C21/G21 values are defined as symmetrical to R12/L12/C12/G12 and are assigned the same values.
- In MDIF format, if you have 1000 frequency points, you will have two files (one for R11/L11/C11/G11 and one for R12/L12/C12/G12). Each file has 1000 values for R, 1000 values for L, 1000 values for C, and 1000 values for G. In this format, R22/L22/C22/G22 values are defined as symmetrical to R11/L11/C11/G11 and are assigned the same values. Likewise, R21/L21/C21/G21 values are defined as symmetrical to R12/L12/C12/G12 and are assigned the same values.

Unlike Fitted data where each value is computed for each frequency from the parameters, the data values for Extracted and Smoothed data have an associated frequency.

8-19 shows a file after it was exported from PLTS using the HSPICE W-Element Tabular Extracted format. Smoothed data would be in the same format.

Figure 8-19 HSPICE W-Element Tabular Extracted Format



8-20 shows a file after it was exported from PLTS using the ADS MDIF Extracted format. Smoothed data would be in the same format. This file is just one of two files that are created with this export type. In addition to the “11” file, a similar file for the “12” data is also created.

Figure 8-20 ADS MDIF Extracted Format

```
BEGIN DSCRDATA
%INDEX Freqs R11 L11 C11 G11
1, 6.00000000000, 2.42273703886, 4.06365876003E-007, 7.837837174798E-011, 4.379652514476E-005
2, 12.00000000000, 2.86705708812, 4.06365876003E-007, 7.837837174798E-011, 8.759304052391E-005
3, 18.00000000000, 3.20799585321, 4.06365876003E-007, 7.837837174798E-011, 0.0001313895559031
.
.
.
1348, 8088.00000000000, 40.73375603346, 4.06365876003E-007, 7.837837174798E-011, 0.05903770274087
1349, 8094.00000000000, 40.74836152348, 4.06365876003E-007, 7.837837174798E-011, 0.05908149925625
1350, 8100.00000000000, 40.76296160105, 4.06365876003E-007, 7.837837174798E-011, 0.05912529577162
END
```

Index | Frequencies | R11 | L11 | C11 | G11

1350 Freq. Points
– 6 Freq. Points Shown
1344 Freq. Points Removed for this Illustration

Subset

PLTS has the ability to export a subset of a extracted or smoothed data set using the HSPICE W-Element Tabular or the ADS MDIF formats. This feature is ***not*** available using the HSPICE W-Element or the ADS ML2CTL Model formats. Exporting a subset is described in “Advanced W-Element Export Features” on page 258. Depending on the subset that you define, exporting a subset of the data can result in significantly smaller files sizes.

This feature allows you to change the start, stop, and step frequencies and the number of points. It also allows you to export a user-defined number of points per the entire logarithmic scale, by decade, or by octave. The formats for the subsets are the same as described for the W-Element Tabular or the MDIF formats.

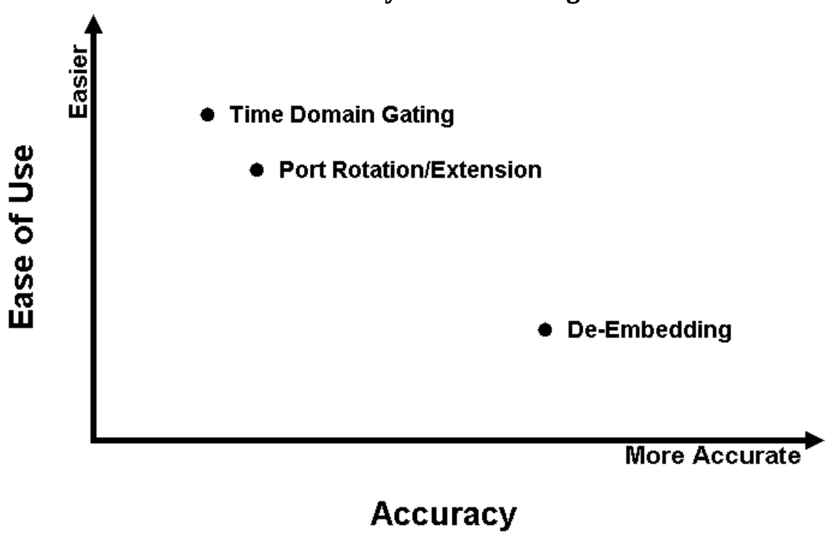
9 Removing Unwanted Effects from the Measurement

All digital interconnect components or channels require an interface to the Coax connections of the PLTS. These interfaces – called *fixtures* or *probes* – typically introduce loss and discontinuities that mask the true performance of your component or channel. You must be aware of fixture and probing effects and account for them in your measurement setup.

There are multiple approaches to removing fixture and probing effects. Calibration to the reference plane is one way to remove effects. Another is to use post-processing techniques such as Gating, Port Rotation/Extension and De-Embedding. These techniques are described below. All of these techniques are applied to fully corrected (calibrated) measurement data. They do not directly effect the calibration. However, you can combine good calibration techniques with one or more of these post-processing fixture removal techniques to obtain the best measurement accuracy.

CAUTION If you have not yet saved your measured data, save it before removing any of these unwanted effects by selecting **Save** from the **File** menu.

Figure 9-1 Ease Versus Accuracy of Removing Unwanted Effects



- **Gating** provides the ability to remove the effect of a particular circuit element mathematically from time-domain plots. The gated section removes a section of the plot that you define, replacing it with an ideal transmission line having the same electrical delay as the removed section. 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.

- **Port Rotation/Extension** (part of the Reference Plane Adjustment) mathematically extends the measurement plane to the DUT, but assumes the fixture looks like a perfect transmission line: a flat magnitude response, a linear phase response, and a constant impedance. Port rotation/extension is done after a coaxial calibration has been performed at the end of the test cables. If the fixture is very well designed, this technique may be sufficient.
- **De-embedding** (part of the Reference Plane Adjustment) affords a very accurate technique that removes the effect of added loss, phase shift, and mismatch due to adding adapters, probes, and fixtures to your DUT. It combines the errors determined from a coaxial calibration with the errors in the fixture to obtain a single error coefficient array that corrects for everything up to the measurement plane of the DUT. De-embedding uses an accurate linear model of the fixture, or measured S-parameter data of the fixture. This modeled or measured data is then mathematically removed from the DUT measurement data in post-processing.

Gating

The controls for gating the time domain response provide the ability to remove the effect of a particular circuit element mathematically. You define the start- and stop-points of the gate. The 'Ideal' gate that you create replaces the gated section with an ideal transmission line having the same electrical delay as the section that it replaced.

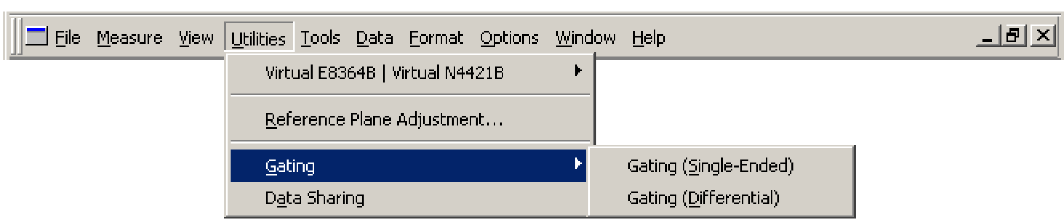
Up to 10 gates can be added. The gates are numbered sequentially from the left edge of the plot to the right edge. For example, the first gate (Gate 1) was added. Then a second gate is added to the left of the Gate 1. The gates are then renumbered such that the gate on the left (the second gate that was added) becomes Gate 1, and the original gate is changed to Gate 2. Either deleting or moving a gate can cause the gates to be renumbered.

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.

To Add a Gate

1. Click **Gating** from the **Utilities** menu. Then click either **Gating (Single-Ended)** or **Gating (Balanced)** depending on whether your Time Domain plot is single-ended or differential.

Figure 9-2 Utilities Menu with Gating Expanded



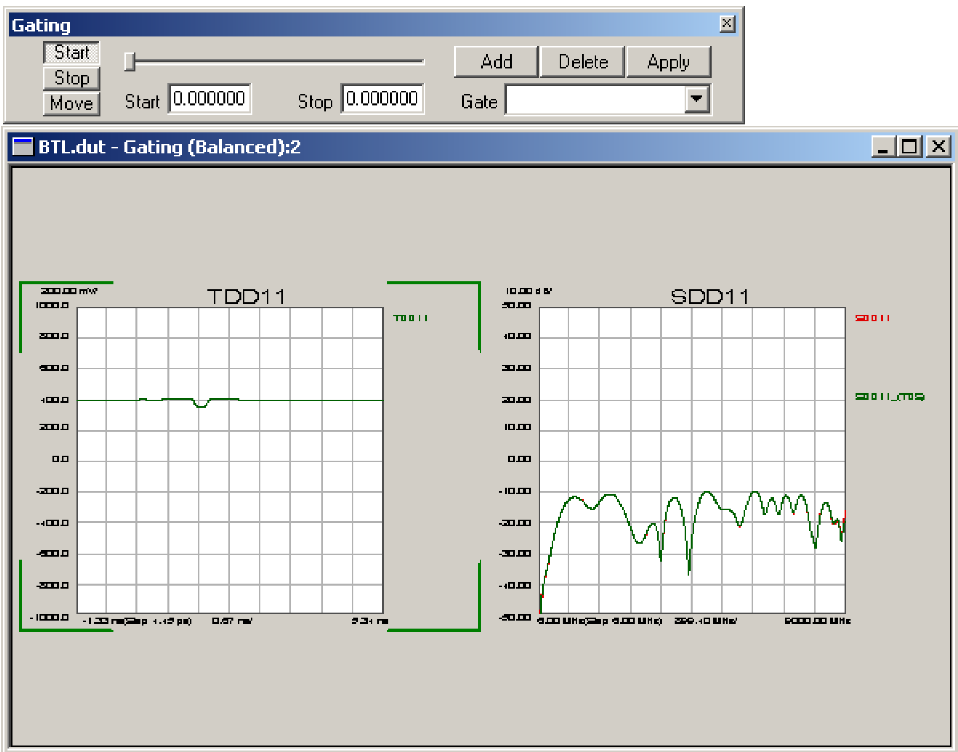
The **Gating Bar** (shown in Figure 9-3) and an empty plot window are displayed when gating is selected from the **Utilities** menu.

Figure 9-3 Gating Bar

2. From the **Parameter Bar** (or from the **Data** menu), select the parameter that will have gates added to it.

   When you select the parameter, two plots are displayed in the plots window. The left plot is the parameter you selected displayed in Time Domain mode. The right plot is that same parameter displayed in Frequency Domain Mode. See Figure 9-4. When adding a gate to the time domain trace, you will also be able to see how adding the gate affects the parameter in the Frequency Domain by watching the right plot.

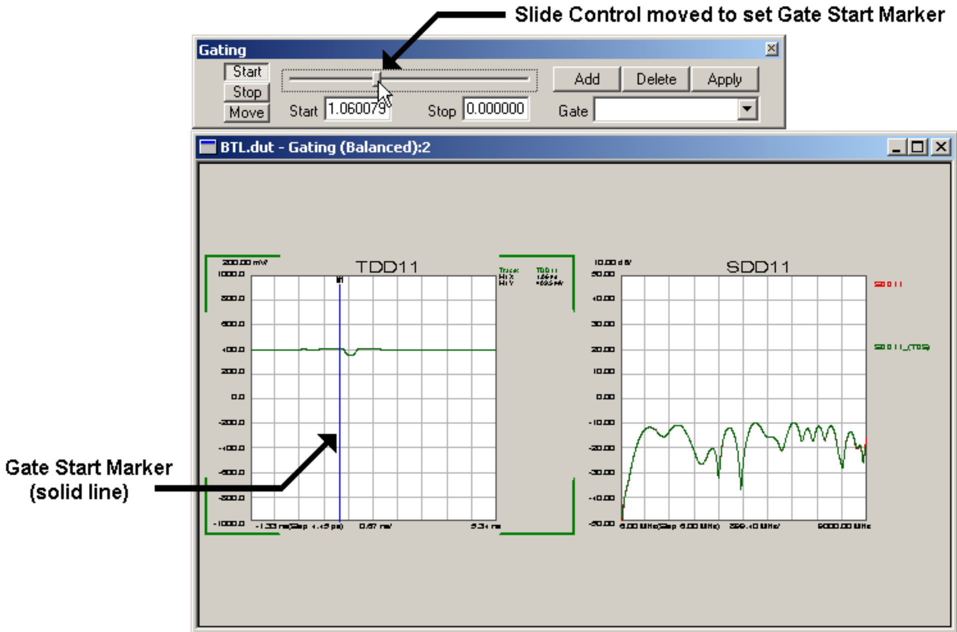
Figure 9-4 Gating Display: Time Domain Plot and Frequency Domain Plot



3. Click the **Start** button.

4. While watching the gate start marker, a solid blue vertical line on your time domain plot, move the slide control to the right to set the start position of the gate. As the slide control is positioned, the time (or distance) of the start gating marker is displayed in the **Start** box. See Figure 9-5.

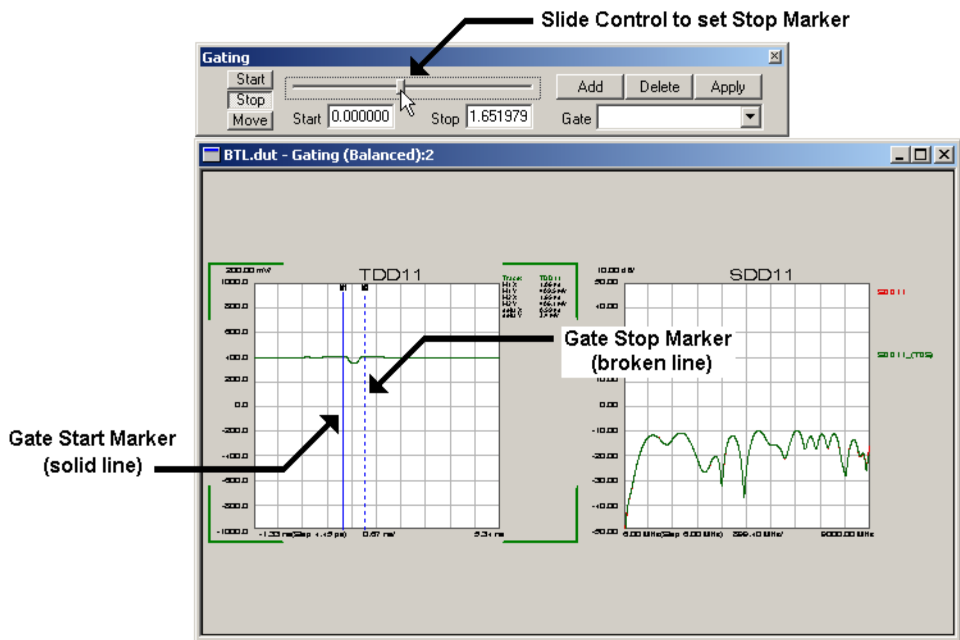
Figure 9-5 Setting the Gating Start Position



5. Click **Stop** button.

6. While watching the gate stop marker, a dashed blue vertical line on your time domain plot, move the slide control to the right to set the stop position of the gate. See Figure 9-6.

Figure 9-6 Setting the Gating Stop Position

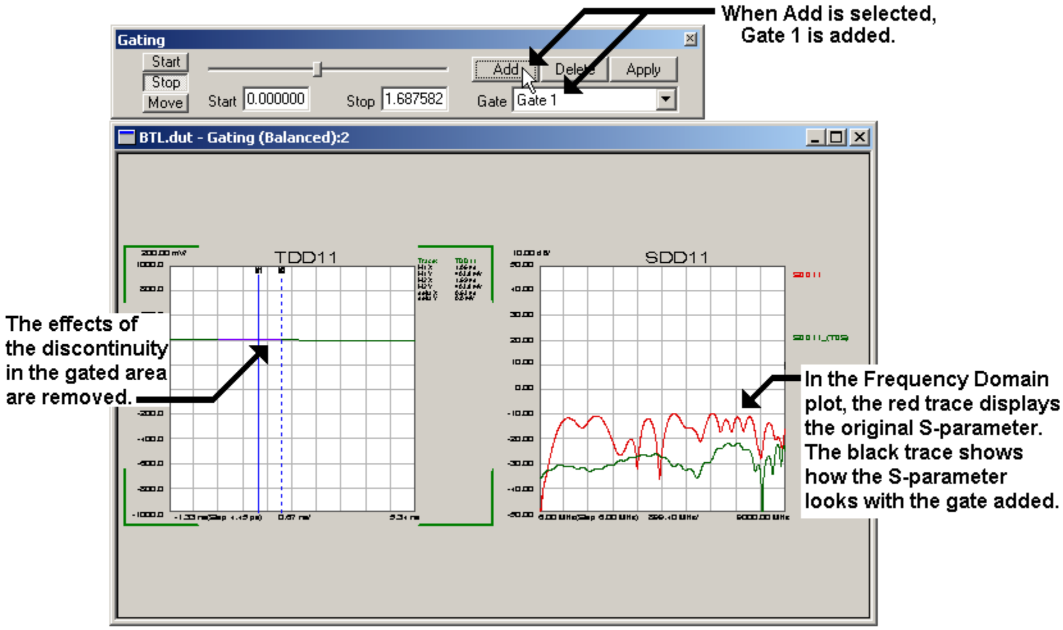


7. Click the **Add** button to add the gate.

   When **Add** is selected:

   - A gate identifier is added to the **Gate** box to identify the number of the gate that was just added. See Figure 9-7.
   - In the Time Domain plot, a straight blue line is added between the start and stop points to reflect that the measured data is being replaced mathematically with an *ideal* transmission line.
   - In the Frequency Domain plot, the red trace shows how the original S-parameter looks. The black trace shows how the S-parameter looks when the effects of the gate are taken into account.

Figure 9-7 The Effects of Adding a Gate



8. Repeat Steps 3 through 7 to add additional gates.

   The maximum number of gates that can be added is 10.

To Move a Gate

A gate can be moved by selecting the gate from the **Gate** list and clicking the **Move** button, and moving the slide control left or right to the desired location. Then select the **Apply** button to have the move take affect. The range between the start and stop values of the gate remains the same, just the start and stop values are changed.

To Delete a Gate

Selecting a gate from the **Gate** list and clicking the **Delete** button will delete the gate. The gate numbers are moved down in number when a gate is deleted. For example, when Gate 4 is deleted, Gate 5 becomes Gate 4, and Gate 6 becomes Gate 5, and so on.

Reference Plane Adjustment

Adapters, fixtures, or probes that are used to measure your DUT introduce errors such as additional loss, phase shift (skew), and mismatch (impedance discontinuities). Once you have made your measurements, you often need to remove the effects of these devices.

The **Reference Plane Adjustment** dialog box (Figure 9-8) allows you to manipulate the measured data in a variety of ways to remove these effects from the data. The manipulation of the measurement data is done mathematically and may be saved for future use. From the **Reference Plane Adjustment** dialog box, you can apply the following adjustments to the each of the four ports individually:

2-port De-embedding or **4-port De-embedding**

Adapters, test fixtures, or other non-DUT structures introduce unwanted effects and error into measurement results. These unwanted effects typically include additional loss, phase shift, mismatch, discontinuities, and time delay. De-embedding is the process of mathematically removing previously characterized performance of the non-DUT structure from the measurement in post-processing.

NOTE De-embedding is a powerful utility, but requires advanced S-parameter characterization of the non-DUT structure through measurement or modeling. While measuring the structure can provide the most accurate characterization, it may be difficult or impossible to perform the measurement due to connectivity issues. In these cases, S-parameter models may offer the best alternative.

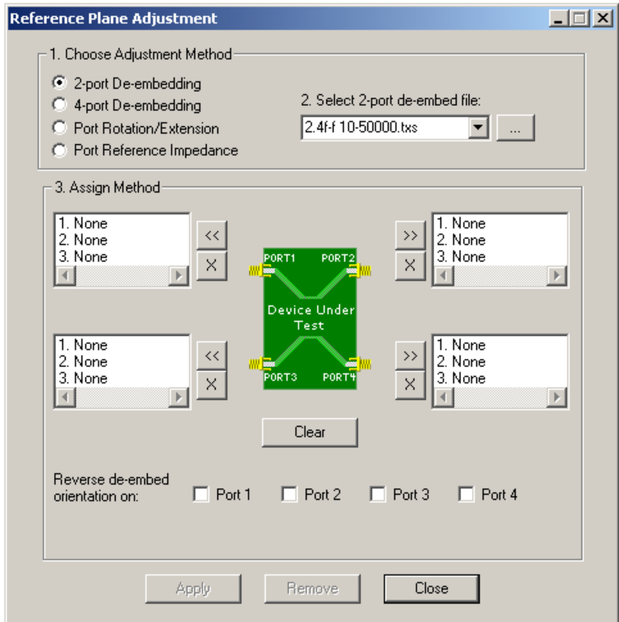
Port Rotation (Extension)

Port rotation allows you to move the calibrated reference plane from the end of the test cable toward the DUT in an attempt to eliminate the effects of phase shift introduced by the addition of adapters, fixtures, and probes. Port rotation does not correct for additional loss and mismatch introduced by these items.

Port Reference Impedance

By default, all four ports are set to 50 ohms impedance. This adjustment mathematically transforms the measured data to show the performance of the DUT would change in a non-50 ohm measurement system.

Figure 9-8 Reference Plane Adjustment Dialog Box



The **Reference Plane Adjustment** dialog box is associated with the active plot window. It remains displayed so you can continue to make changes until you select the **Close** button or minimize it.

To Use Port Reference Adjustment

There a three main steps in using the port reference adjustment.

1. Choose the adjustment method and set the associated options.
2. Assign the adjustment method to each applicable port.
3. Apply the adjustments to the active file.

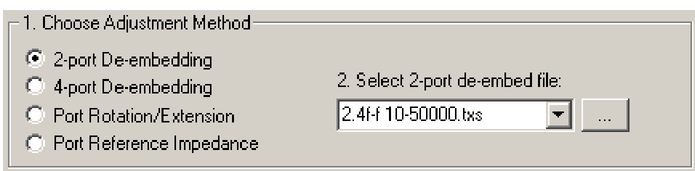
To Choose the Adjustment Method and its Options

The first step in using the port reference adjustment is choosing the adjustment method and it's options.

There are four adjustments: 2-port de-embedding, 4-port de-embedding, port rotation/extension, and port reference impedance. Each adjustment has its own options that are displayed when the adjustment is select from the list.

1. Choose an adjustment method from the **Choose Adjustment Method** area. You have the following four adjustment methods to choose from.
   - 2-port De-embedding (may not be used with 4-port De-embedding)
   - 4-port De-embedding (may not be used with 2-port De-embedding)
   - Port Rotation/Extension
   - Port Reference Impedance

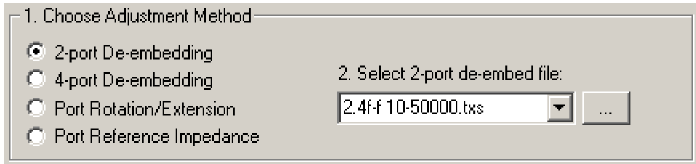
Figure 9-9 Reference Plane Adjustment Dialog Box Choose Adjustment Method Area



2. Choose the options specific to the adjustment method from the right side of the **Choose Adjustment Method** area.
   - For **2-port De-embedding**, select a 2-port de-embed file. See Figure 9-10.

     When **2-port De-embedding** is selected, a list is displayed showing the .txs files (files created by the PLTS adapter characterization tool) and the .s2p files in the *Adapters* folder where PLTS was installed. The default location is: **C:/Program Files/Agilent/PLTS/Adapters**

Figure 9-10 2-Port De-embedding Adjustment Method Options

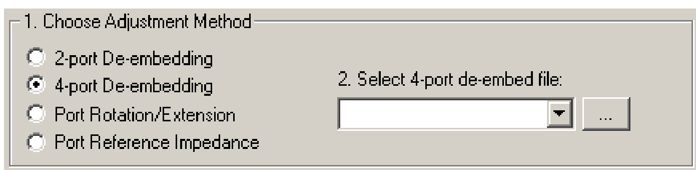


The browse button (...) allows you to locate and use .txs and .s2p files stored elsewhere on your computer.

- For **4-port De-embedding**, select a 4-port de-embed file. See Figure 9-11.

  When **4-port De-embedding** is selected, a list is displayed showing the .s4p files, .cit files, and the .dut files in the *deembedding* folder where PLTS was installed. The default location is: **C:/Program Files/Agilent/PLTS/deembedding**

Figure 9-11 4-Port De-embedding Adjustment Method Options

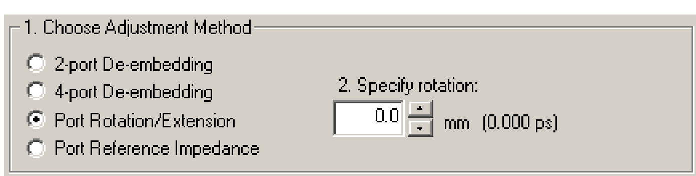


The browse button (...) allows you to locate and use .s4p, .cit, and .dut files stored elsewhere on your computer.

- For **Port Rotation/Extension**, specify the rotation length in mm. See Figure 9-12.

  When **Port Rotation/Extension** is selected, a **Specify Rotation** box is displayed. This box allows you to enter the rotation in millimeters (mm). The equivalent time in picoseconds (ps) is also displayed. You may enter real numbers between -50000.0 mm and 50000.0 mm by 0.1 mm increments using the spinners. The spinners increase or decrease the rotation value of the selected units. However, you can enter values to 11 places to the right of the decimal using the keyboard.

Figure 9-12 Port Rotation/Extension Adjustment Method Options



- For **Port Reference Impedance**, specify the impedance in ohms. See Figure 9-13.

  When **Port Reference Impedance** is selected, a **Specify port reference impedance** box is displayed. This box allows you to enter the impedance in ohms. You may enter real numbers between 0.0 ohms and 1000.0 ohms by 0.1 ohm increments using the spinners. The spinners increase or decrease the impedance value of the selected units. However, you can enter values to 11 places to the right of the decimal using the keyboard.

Figure 9-13 Port Reference Impedance Adjustment Method Options

To Assign the Chosen Adjustment Method to the Ports

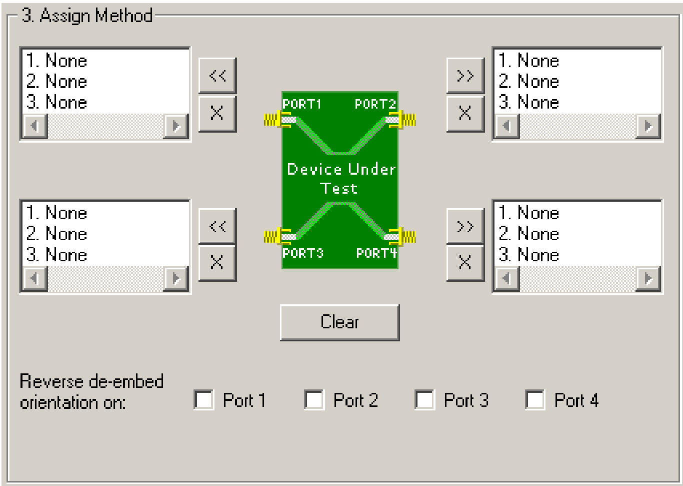
3. With an adjustment method chosen and the options selected or specified, click the port's assignment button (<< >>) to assign the adjustment.

   When either 2-port or 4-port de-embedding is assigned to a port, a label is displayed in the port's adjustment list. It is labeled with "1." followed by the de-embedding file name you chose in step 2 on page 283. For 4-port de-embedding, when the adjustment is assigned to (or removed from) one of the input ports, it will be assigned to (or removed from) both input ports. The output ports are handled the same way.

   When Port Rotation/Extension is assigned to a port, a label is displayed in the port's adjustment list. It is labeled with "`2.` ***x.x*** `mm rotation`" where ***x.x*** is the rotation that was set in step 2 on page 283. Each port may be assigned the same or unique values.

   When Port Reference Impedance is assigned to a port, a label is displayed in the port's adjustment list. It is labeled with "`3.` ***xx.x*** `Ohm impedance`" where ***xx.x*** is the impedance that was set in step 2 on page 283. Each port may be assigned the same or unique values.

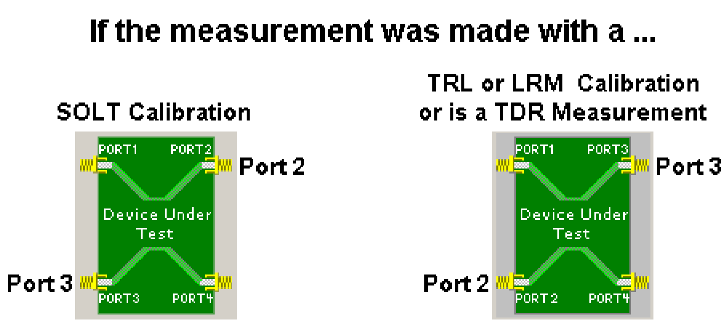
Figure 9-14 Reference Plane Adjustment Dialog Box Assign Method Area



You may remove an adjustment method from a port by selecting the adjustment method name in the port's adjustment list box and then clicking the removal button (×).

Clicking the **Clear** button removes all of the adjustment methods from all ports. It also clears the reverse orientation check box settings. However, the **Clear** button will not remove any adjustments that are currently applied to the active .dut file.

CAUTION Port 2 and Port 3 change their position in the **Assign Method** area depending on the calibration type of associated plot window.



Make sure you assign the adjustment to the appropriate port.

---

4. Repeat step 3 until the adjustment method is assigned all of the appropriate ports.

5. If the adjustment method is either 2-port or 4-port de-embedding, you may also select the **Reverse de-embed orientation on** selection for the appropriate ports.

   - 2-port de-embedding allows you to select any or all of four ports.
   - 4-port de-embedding allows you to select:

     — One or both **Port 1-3** or **Port 2-4** for an SOLT calibration

     — One or both **Port 1-2** or **Port 3-4** for a TRL or LRM calibration or for a TDR measurement

6. If you wish to add more adjustments to the ports, repeat steps 1 through 5.

To Apply the Reference Plane Adjustments

7. Click the **Apply** button to apply the adjustments to the active data window. The data displayed on the views will then be updated.

NOTE Before clicking **Apply**, make sure the .dut file that you want the adjustments applied to is the active plot window.

When the **De-embedding** indicator on status bar is highlighted, it means that the current data has been modified with the de-embedding information shown in the ports listed with de-embedding. When the **Port Rotation** indicator on status bar is highlighted, it means that the current data has been modified because at least one port has been rotated.

The state of the adjustments are maintained in the .dut file which means that when the .dut file is loaded again (after saving the adjusted .dut), the appropriate status bar indicators on the status bar will be highlighted, and the dialog box will open with the list boxes and reverse orientation check boxes initialized appropriately.

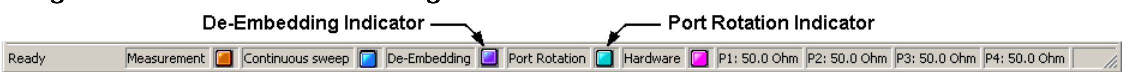
After the adjustments are applied, the **Apply** button is disabled, and the **Remove** button is enabled. The list box entries are not cleared.

8. Click the **Remove** button to remove the current adjustments from the active plot window.

NOTE Before clicking **Remove**, make sure the .dut file that you want the adjustments removed from is the active plot window.

This resets the active plot window to display the original corrected measurement data. The port adjustment list boxes are not cleared. If the De-embedding or Port Rotation status bar indicators were highlighted, they are now dimmed.

Figure 9-15 De-Embedding and Port Rotation Indicators



9. Click the Close button to close the *Reference Plane Adjustment* dialog box.

10 Using Analysis Tools and Utilities

This chapter describes several tools that are very useful in performing your analysis. This chapter includes:

- Markers
- Click and Drag Zooming
- Math
- Data Sharing
- Characterization Report Generator
- Copying and Pasting Plot Formats
- Renaming Plots
- Printing
- File Converter

Markers

PLTS uses two different types of markers. The marker type used is dependent on the analysis type. For time domain, frequency domain, and RLCG analysis, refer to Time-Domain, Frequency-Domain, and RLCG Markers. For eye diagram analysis, refer to "Eye Diagram Markers" on page 299.

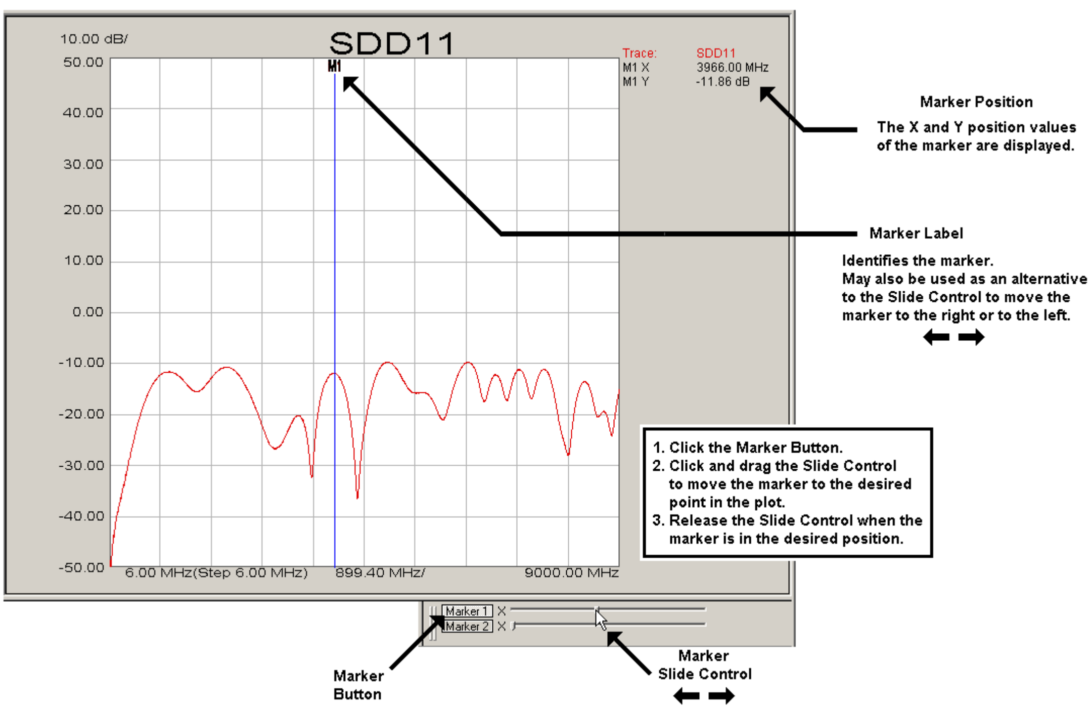
Time-Domain, Frequency-Domain, and RLCG Markers

Time domain, frequency domain, and RLCG analysis use markers that are displayed vertically on a plot. As the vertical marker intersects a trace on the plot, the values of the horizontal and vertical position at the intersection are displayed. Up to two markers may be used on each plot.

To use the markers:

1. With a plot displayed, click the **Marker 1** button.

Figure 10-1 Single Marker for Time Domain, Frequency Domain, and RLCG



2. Click and drag the **Marker 1 Slide Control** to the right.

   As the **Slide Control** is moved to the right, a solid black vertical line (the marker) is displayed with an **M1** label at the top of the line.

TIP Once the vertical marker line is displayed, you may adjust the marker by clicking and dragging the marker line itself instead of the **Slide Control**.

3. Release the **Slide Control** at the desired position of the plot.

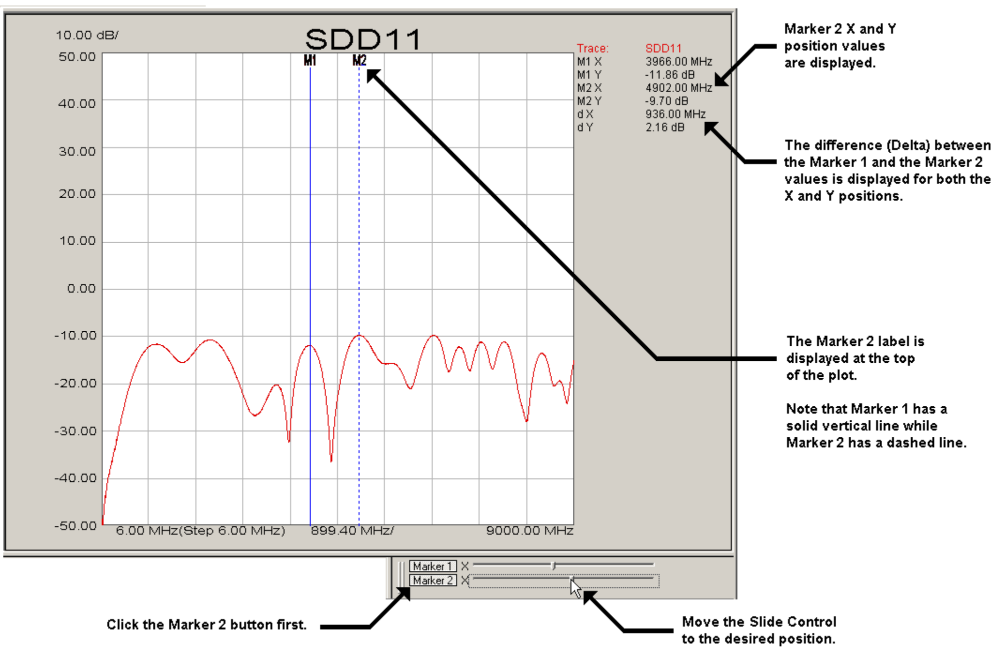
   The values of the X and Y position for the point that the marker intersects the trace are displayed near the upper right corner of the plot.

TIP **Moving the Marker One Measurement Point at a Time**

A marker may be moved one measurement point at a time by pressing the keyboards arrows (← & →) if the marker button (**Marker 1** or **Marker 2**) is selected. Each keyboard press changes the X and Y values.

4. To display the second marker, click the **Marker 2** button.

Figure 10-2 Time-domain, Frequency-domain, and RLCG Dual Markers



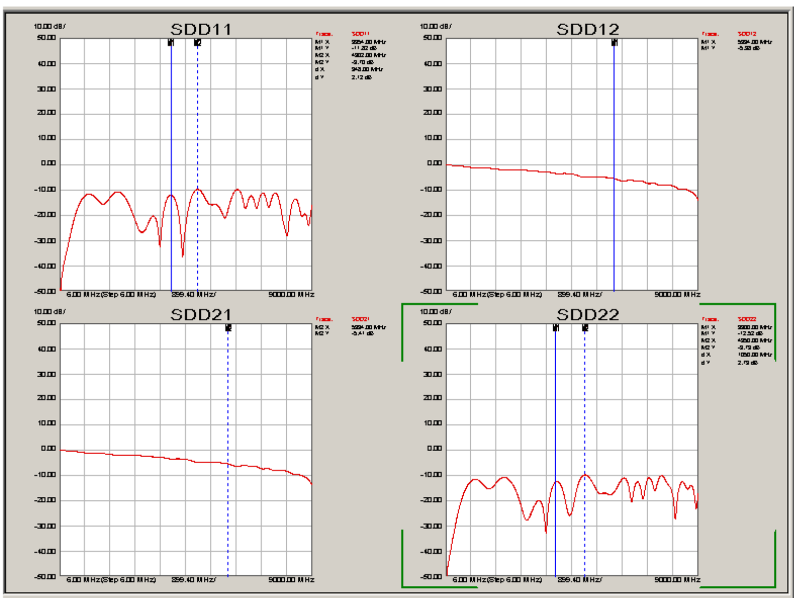
5. Click and drag the **Marker 2 Slide Control** to the right, releasing it at the desired position of the plot.

   As the **Slide Control** is moved, the marker is displayed. **Marker 2** is a dashed black vertical line with the label **M2** at the top.

The X and Y values for markers are displayed near the upper right corner of the plot. When both markers are used, the difference (delta) between the markers is also displayed.

Markers may be used with multiple plots but are unique within individual plots. You may have up to two markers in each plot. The marker information is displayed at the upper right corner of each plot.

Figure 10-3 Markers with Multiple Plots



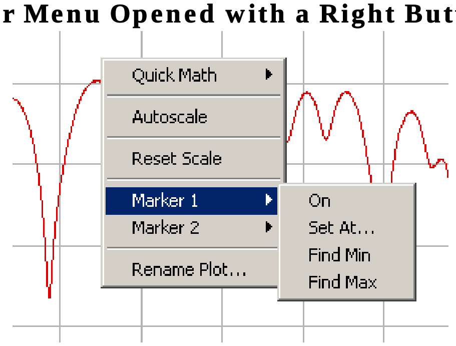
Markers may be used with multiple traces. You may use either one or two markers in a plot that displays as many as four traces. The marker information for each trace is displayed along the right edge of the plot.

Figure 10-4 Markers with Multiple Traces

Quick Markers

Markers may be accessed quickly. Click the right mouse button over the plot to display the shortcut menu shown in Figure 10-5. Both **Marker 1** and **Marker 2** have four choices available: **On**, **Set At**..., **Find Min**, and **Find Max**.

Figure 10-5 Marker Menu Opened with a Right Button Mouse Click



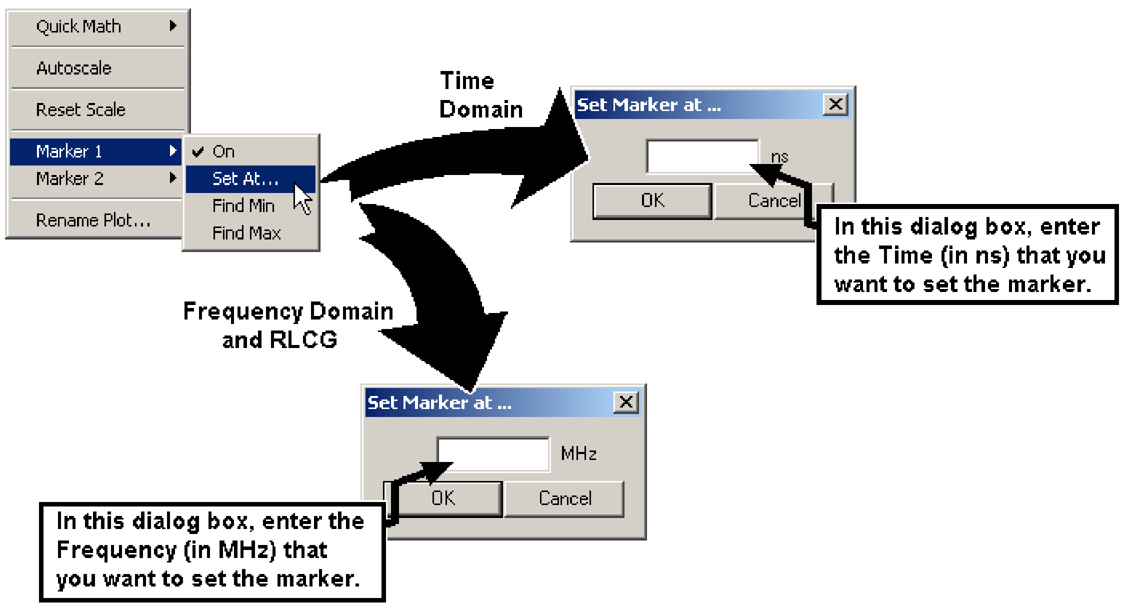
On Turns the associated marker on or off. When the marker is turned on, a check mark is shown on the shortcut menu to the left side of the **On** selection. The marker is displayed with a label at the top of the plot.

- Marker 1 is a solid black line with the **M1** label at the top of the plot.
- Marker 2 is a dashed black line with the **M2** label at the top of the plot.

Adjust the marker by clicking and dragging the marker line to the left or to the right.

Set At... Places the associated marker at a specified time (with a time domain plot) or at a specified frequency (with a frequency domain or RLCG plot). The *Set Marker* dialog box allows you to enter the time or frequency to set the marker. See Figure 10-6.

Figure 10-6 Set Marker Dialog Boxes



Find Min Places the marker at the minimum measured value. If multiple traces are displayed, this selection finds the minimum measured value of all of the traces.

Find Max Places the marker at the maximum measured value. If multiple traces are displayed, this selection finds the maximum measured value of all of the traces.

Frequency Domain Polar and Smith Chart Markers

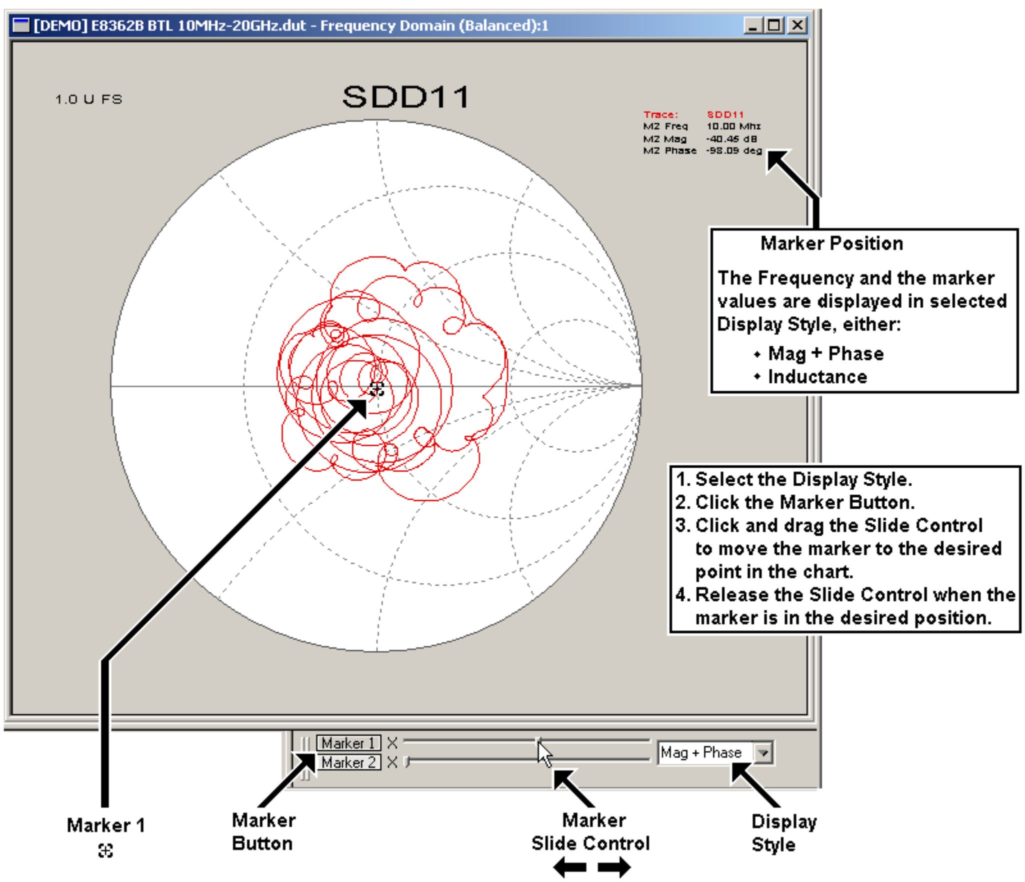
Polar or Smith Chart formats display markers that are displayed differently than other frequency domain formats. When a marker is used in a Polar or Smith Chart format, it is essentially a point that follows the displayed trace around the chart. Up to two markers may be used on each plot.

To use the markers:

1. With the plot displayed on the Polar or Smith Chart, select the display style, either **Mag + Phase** or **Inductance**, on the Marker Bar. See Figure 10-7.

   Mag + Phase displays the magnitude and the phase at the marker frequency. **Inductance** displays the magnitude and inductance at the marker frequency.

Figure 10-7 Single Marker for Frequency Domain Polar and Smith Charts

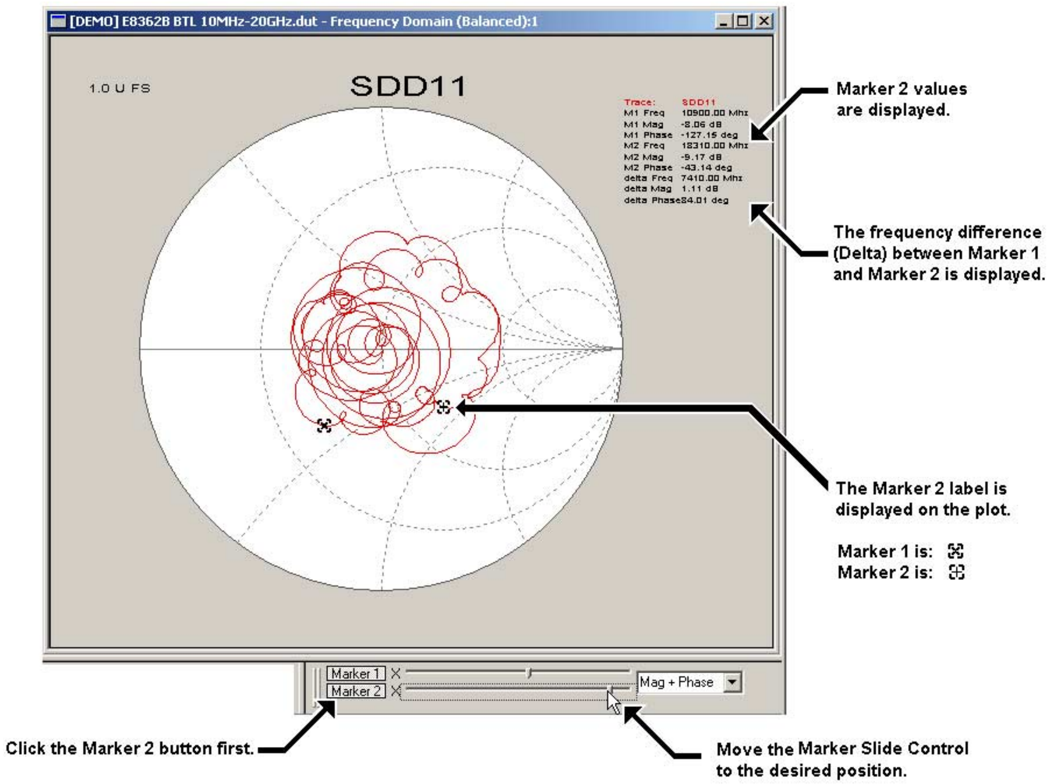


2. Click the **Marker 1** button.
3. Click and drag the **Marker 1 Slide Control** to the right.

   As the **Slide Control** is moved to the right, a marker is moved along the trace.
4. Release the **Slide Control** at the desired position of the plot.

   The marker values are displayed near the upper right corner of the plot.
5. Click the **Marker 2** button to display the second marker. See Figure 10-8.

Figure 10-8 Polar and Smith Chart Dual Markers



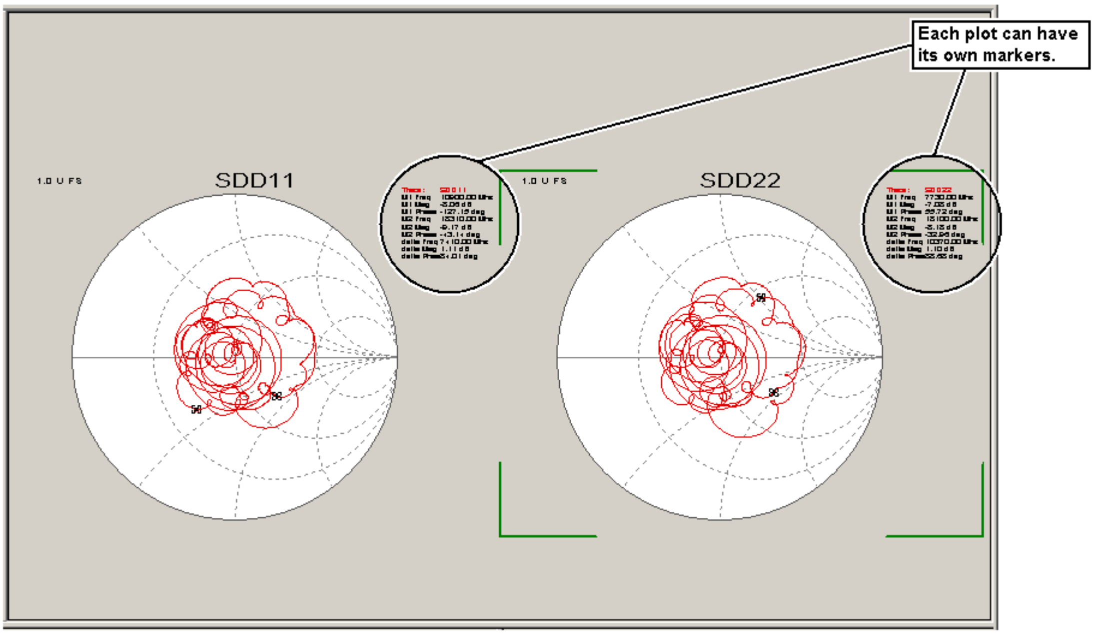
6. Click and drag the **Marker 2 Slide Control** to the right, releasing it at the desired plot position.

   The Marker 2 values are displayed near the upper right corner of the plot.

   When two markers are used, the frequency difference (delta) between the markers is also displayed. See Figure 10-8.

Markers may be used with multiple plots. You may use one or two markers in a plot. The marker information is displayed at the upper right corner of each plot. See Figure 10-9.

Figure 10-9 Markers with Multiple Polar and Smith Chart Plots



Eye Diagram Markers

Eye diagrams may use up to two markers. They are labeled **Marker 1** and **Marker 2**. Each of these markers have two components, a horizontal (X) position and a vertical (Y) position marker.

The horizontal position markers are displayed as vertical lines that extend from the top to the bottom of the eye diagram. See Figure 10-10.

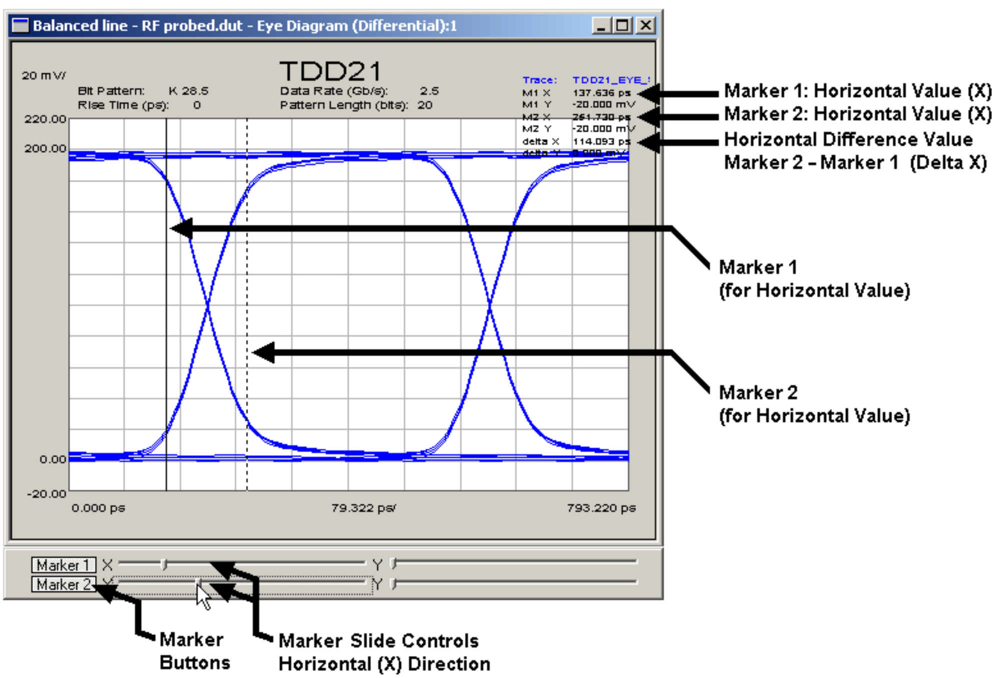
The vertical position markers are displayed as horizontal lines that extend from the left side to the right side of the eye diagram. See Figure 10-11.

To use the markers:

Horizontal (X) Markers

1. With a plot displayed, click the **Marker 1** button.

Figure 10-10 Eye Diagram Markers: Horizontal (X) Position



2. Click and drag the Marker 1 **X Slide Control** to the right.

   As the **X Slide Control** is moved, the marker, a solid black vertical line is displayed.

3. Release the **X Slide Control** at the desired position of the plot.

   The marker's horizontal (X) position is displayed near the upper right corner of the plot.

4. Click the **Marker 2** button and repeat steps 2 and 3 to display the second marker, a dashed black vertical line.

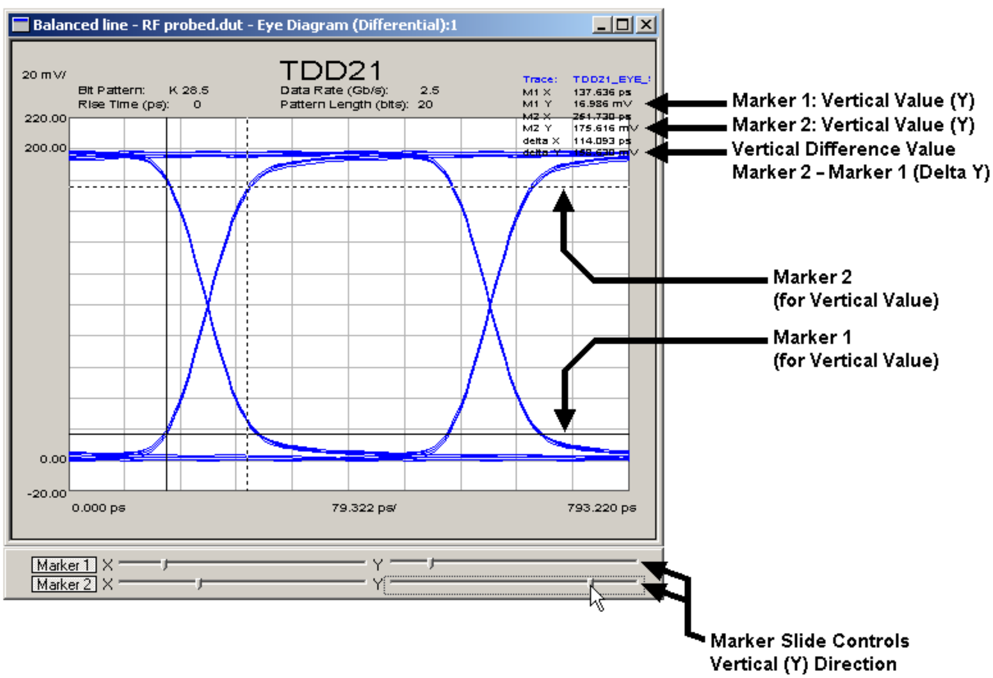
   The difference (delta) between the two markers is displayed near the upper right corner of the plot. See Figure 10-10.

Vertical (Y) Markers

5. To display the Marker 1 vertical (Y) marker, a solid black horizontal line, click the **Marker 1** button and drag the Marker 1 **Y Slide Control**, releasing the slide control when the marker is at the desired position on the plot. See Figure 10-11.

   As the **Y Slide Control** is moved, the marker, a solid black horizontal line is displayed.

Figure 10-11 Eye Diagram Markers: Vertical (Y) Position



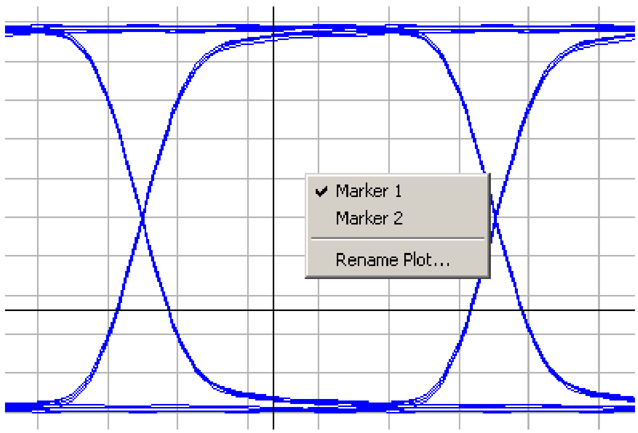
6. Repeat step 5 for Marker 2. Marker 2 is a dashed horizontal line.

   With two markers displayed, the difference (delta) between the two markers is displayed.

Quick Markers

By clicking the right mouse button with the cursor over the eye diagram plot, you can access the marker choices: **Marker 1** and **Marker 2**. See Figure 10-12. These selections turn both the horizontal and vertical markers on or off. The values displayed are computed from the screen coordinates of the plot.

Figure 10-12 Eye Diagram Marker Menu Opened with a Right Button Mouse Click



Marker 1 This selection turns both the horizontal and vertical markers of Marker 1 on or off. When the marker is turned on, a check mark is displayed to the left of **Marker 1**.

Marker 2 This selection turns both the horizontal and vertical markers of Marker 2 on or off. When the marker is turned on, a check mark is displayed to the left of **Marker 2**.

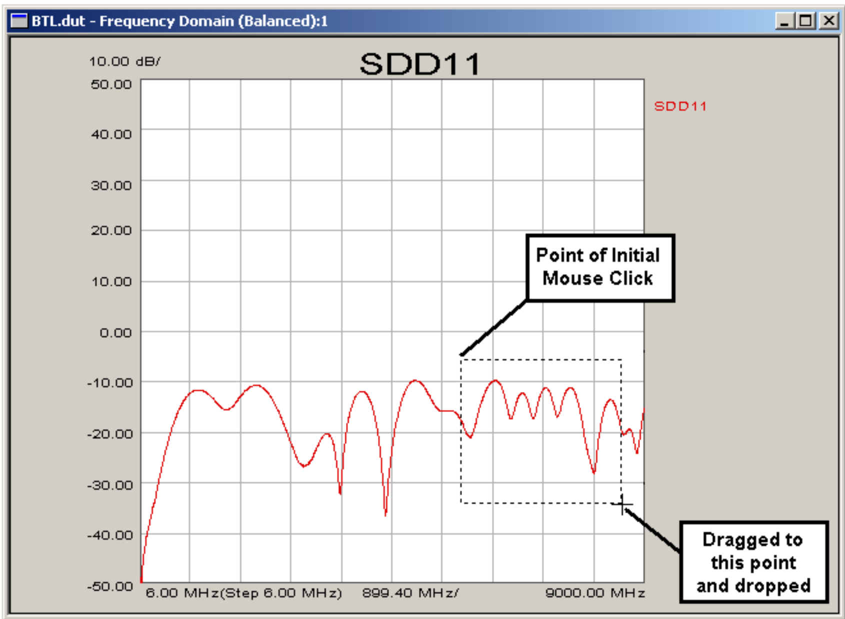
Click and Drag Zoom

NOTE The zoom feature is not available for Eye Diagrams, Polar or Smith Chart plots.

To zoom in on an area of the active plot:

1. Click within the plot and drag the mouse, creating a rectangle as the mouse is moved. See Figure 10-13.

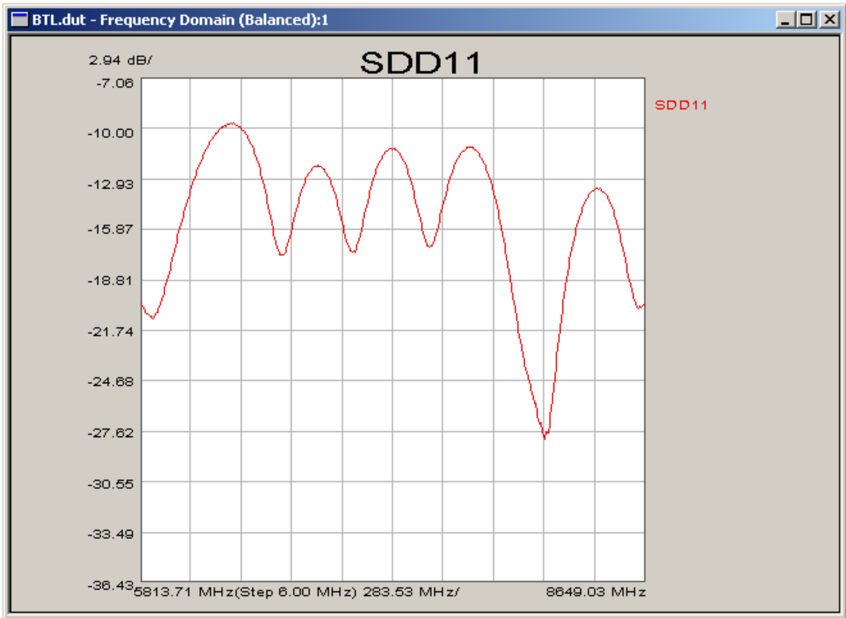
Figure 10-13 Defining Zoom Area



The rectangle is created by dashed lines from the point of the mouse click to the point that the mouse was dragged.

2. Release the mouse button once the rectangle encloses the plot area of interest.

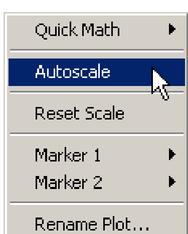
Figure 10-14 New Plot after Zoom



The rectangular area of the plot that you have defined is now displayed, replacing the original plot. The displayed plot now has new X-axis start and stop frequencies and new Y-axis scale.

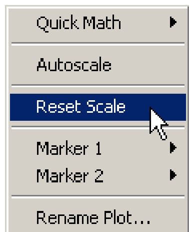
3. If the new plot does not center the traces in the Y-axis, clicking the right mouse button with the cursor over the plot and selecting **Autoscale** sets the Y-axis scale so that the data is better displayed.

Figure 10-15 Autoscale Selection



4. To return the plot to the original horizontal and vertical settings, click the right mouse button with the cursor over the plot and select **Reset Scale**.

Figure 10-16 Reset Scale Selection



Math

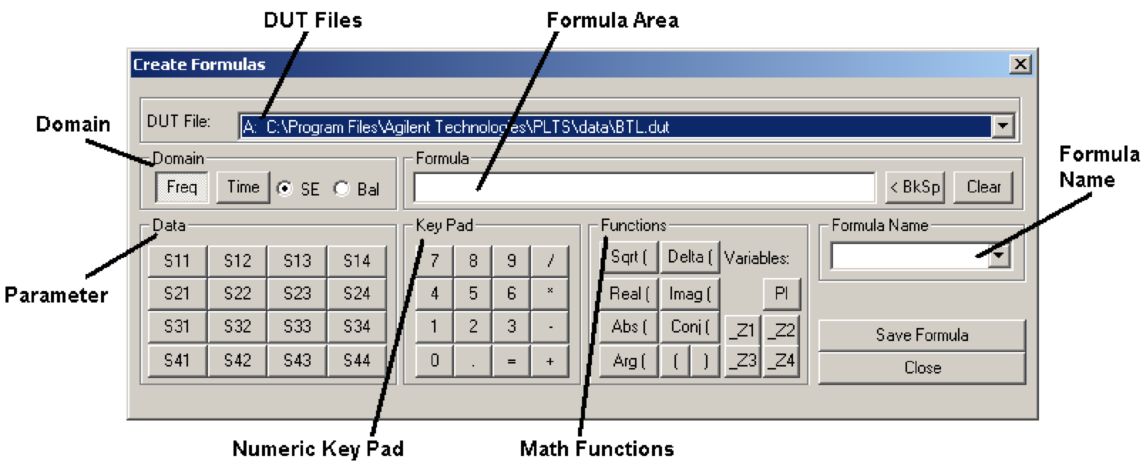
Math may be performed on traces that are displayed within a plot. For example, you may divide the data of one trace by the data of a second trace to derive the vectorial difference.

Math may be performed on a trace using two methods. You may create a formula and apply the formula to a displayed trace (see "Creating a Math Formula" on page 305) or you may perform a quick math function on a displayed trace (see "Using Quick Math" on page 311).

Creating a Math Formula

Math formulas may be created, saved, and applied to opened traces at a later time. Formulas are created using the *Create Formulas* dialog box. Figure 10-17 shows the *Create Formulas* dialog box that is used to create and save math formulas. This dialog box is divided into several areas.

Figure 10-17 Create Formulas Dialog Box



DUT Files Identifies the desired **Data** area selection to apply the math operation. It allows you to select the file of the data used in the math operation from a list of open files. Data from multiple (up to three) files may be used in a formula.

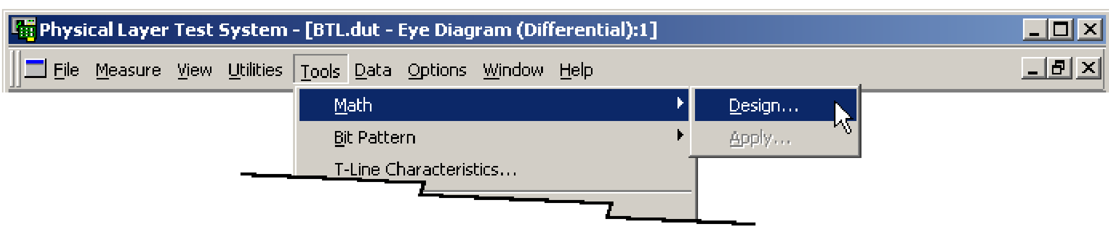
Domain Identifies the desired **Data** area selection to apply the math operation. It allows you to select between Frequency Domain and Time Domain. When in frequency domain, you must select between single-ended and balanced data. When in time domain, you must select between single-ended and differential data.

Formula	Displays the math formula that you are creating.
Data	Used to build the formula. It allows you to select parameters based on the selections in the Domain area.
Key Pad	Used to build the formula. It allows you to enter numeric entries and operators.
Functions	Used to build the formula. It allows you to enter math symbols and delimiters as well as some frequency domain formats.
Formula Name	Allows you to enter a name to save the formula and recall it for future use.

To create a math formula:

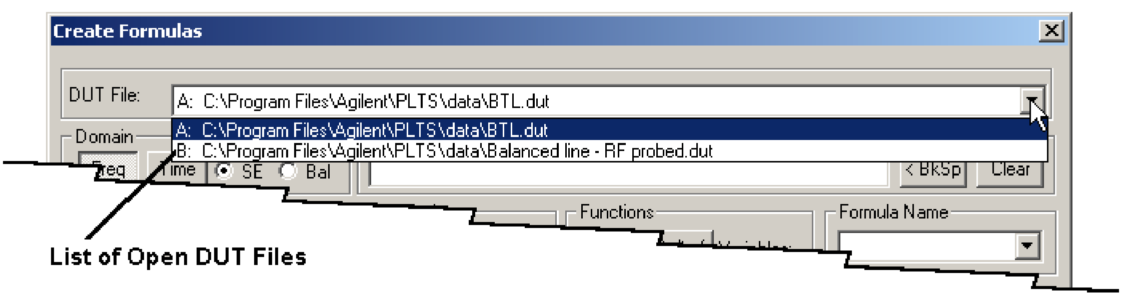
1. Select **Math**, **Design**... from the **Tools** menu to open the *Create Formulas* dialog box as shown in Figure 10-18.

Figure 10-18 Opening the Create Formulas Dialog Box



2. In the DUT File list, select the file that the first **Data** area parameter belongs.

Figure 10-19 DUT File Selection List



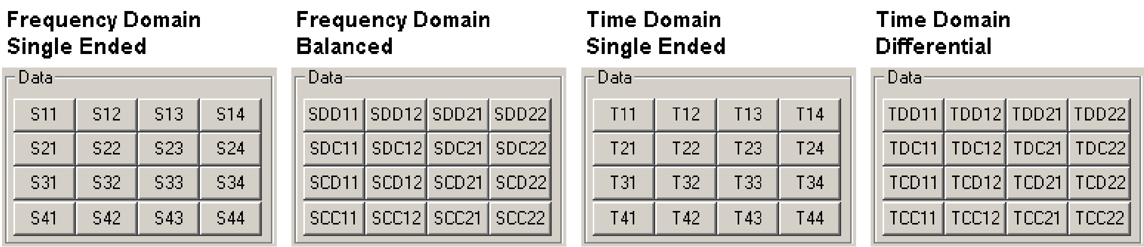
3. In the **Domain** area, select the domain and the mode of the first **Data** area parameter.

   The **Domain** area allows you to select between Frequency Domain (**Freq**) and Time Domain (**Time**).

   - In frequency domain, select between single-ended (**SE**) data and balanced (**Bal**) data.
   - In time domain, select between single-ended (**SE**) data and differential (**Diff**) data.

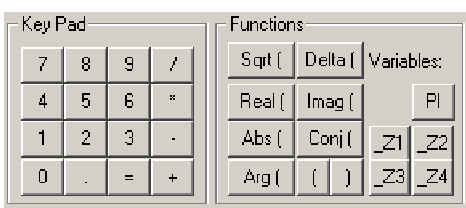
   Each **Domain** area combination displays a different set of **Data** area parameters. Refer to Figure 10-20 for the **Data** area selections for each of the four available combinations.

Figure 10-20 Data Area Parameters for Each Domain Area Combination



4. Start building your formula using the appropriate **Data** area parameters shown in Figure 10-20 and the **Key Pad** and **Functions** area buttons shown in Figure 10-21.

Figure 10-21 Key Pad and Functions Areas



The **Key Pad** area buttons are just the ten numerals(0 through 9) and several basic math symbols/operators: decimal (.), equals (=), add (+), subtract (-), multiply (*), and divide (/).

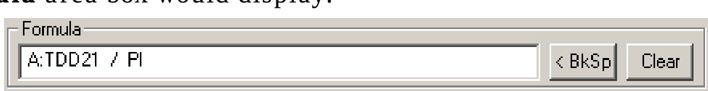
The **Functions** area buttons are:

Table 10-1 Functions Area Buttons

Sqrt (	Use the square root of the following term.	**Delta (**	Use the difference (Δ) of the following terms.
Real (	Use the real portion of the following complex term. This choice is not available when **Time** is selected in the Domain area.	**Imag (**	Use the imaginary portion of the following complex term. This choice is not available when **Time** is selected in the Domain area.
Abs (	Use the absolute value of the following term. This choice is not available when **Time** is selected in the Domain area.	**Conj (**	Use the standard conjugate function for complex numbers. This choice is not available when **Time** is selected in the Domain area.
Arg (	Use the standard argument function for complex numbers. This choice is not available when **Time** is selected in the Domain area.	**(** and **)**	The “open parenthesis” and the “closed parenthesis” delimiters
PI	pi (π) - the constant of 3.14159…	**_Z1** through **_Z4**	The impedance vectors for Port 1, Port 2, Port 3, and Port 4. These choices are not available when **Time** is selected in the Domain area.

5. As you make selections from the **Data**, the **Key Pad**, and the **Functions** areas, these are displayed in the **Formula** area.

   For example, to divide TDD21 by π, select these buttons: **TDD21**, **/**, **PI** and the **Formula** area box would display:



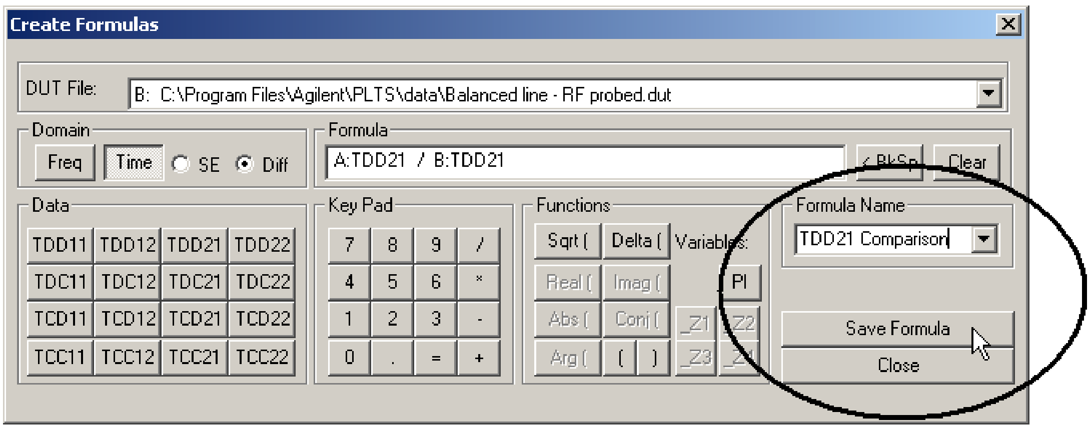
   Use the **< BkSp** button to back step the cursor one position at a time.
   Use the **Clear** button to remove the entire entry from the **Formula** area box.

6. Repeat steps 2 through 5 until your formula is complete.

7. Once your formula is complete, enter a name for the formula in the **Formula Name** box and click the **Save** button. See Figure 10-22.

   If you want to save the formula with the same name as a previously saved formula, select the formula name from the drop down list and click the **Save** button to delete the previously saved formula and save the new formula.

Figure 10-22 Save the Formula



8. When you have finished saving your math formula, click **Close** to close the *Create Formulas* dialog box. Continue to Applying a Math Formula to use the formula with opened data files.

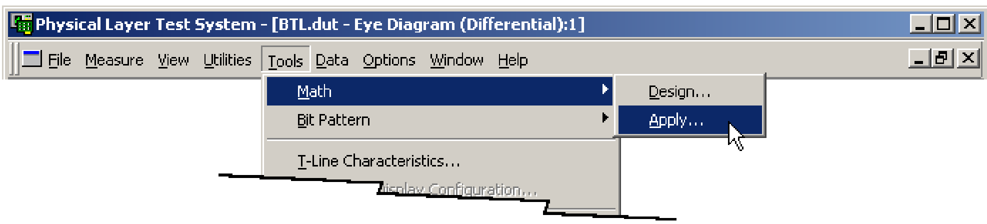
Applying a Math Formula

Once math formulas have been created and saved, they may be applied to opened data files.

To apply a math formula:

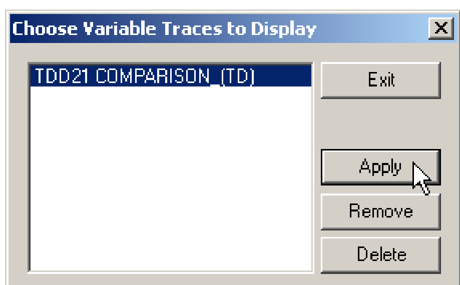
1. Select **Math**, **Apply**… from the **Tools** menu to open the *Choose Variable Traces to Display* dialog box as shown in Figure 10-23.

Figure 10-23 Opening the Apply Formulas Dialog Box



2. With the *Choose Variable Traces to Display* dialog box opened, select a formula from the list of formulas. See Figure 10-24.

Figure 10-24 Choose Variable Traces to Display Dialog Box



Note that **(TD)** was appended to the right of the formula name. This informs you that this is a time domain formula. Had this been a frequency domain formula, **(FD)** would have been appended to the formula name.

3. Click the **Apply** button. The resultant trace is added to the active plot.
4. Click the **Remove** button to remove the trace.
5. If you want to delete a formula from the list of formulas, select the formula and click the **Delete** button.
6. Click **Exit** to return to the main PLTS window.

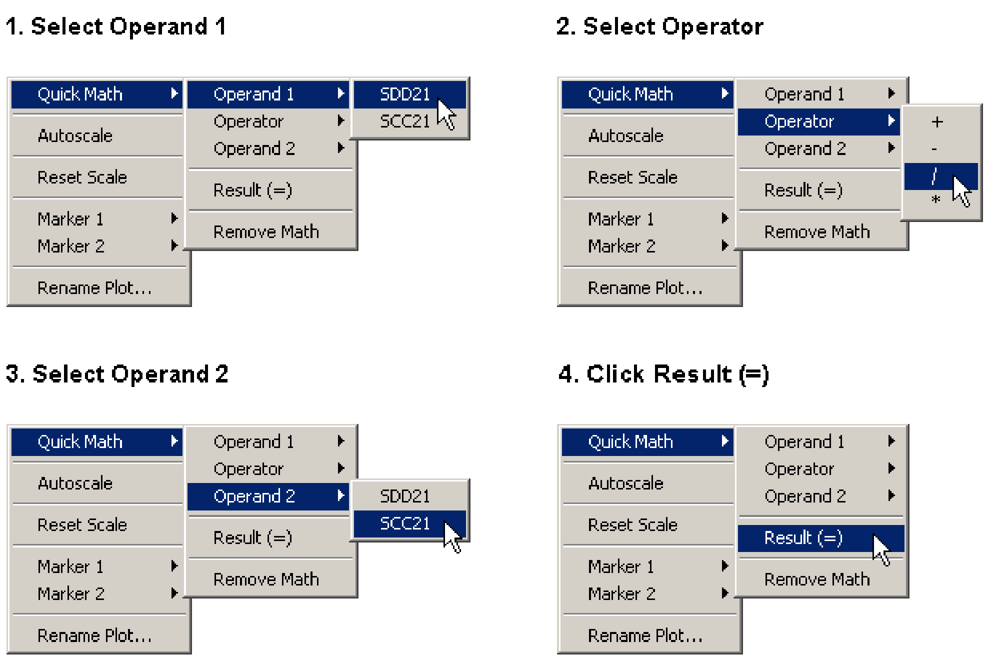
Using Quick Math

You can use Quick Math to perform simple math operations on two traces in the same plot. As quick math is performed, the resultant trace is displayed on the plot. A plot allows up to four measurement and/or math traces.

To perform a quick math function:

1. With two (or three) data traces displayed on the active plot, click the right mouse button with the cursor over the plot. Then click **Quick Math** and **Operand 1**. Click the label identifying the first trace of the math operation. Refer to Figure 10-25.

Figure 10-25 Four Steps for Quick Math



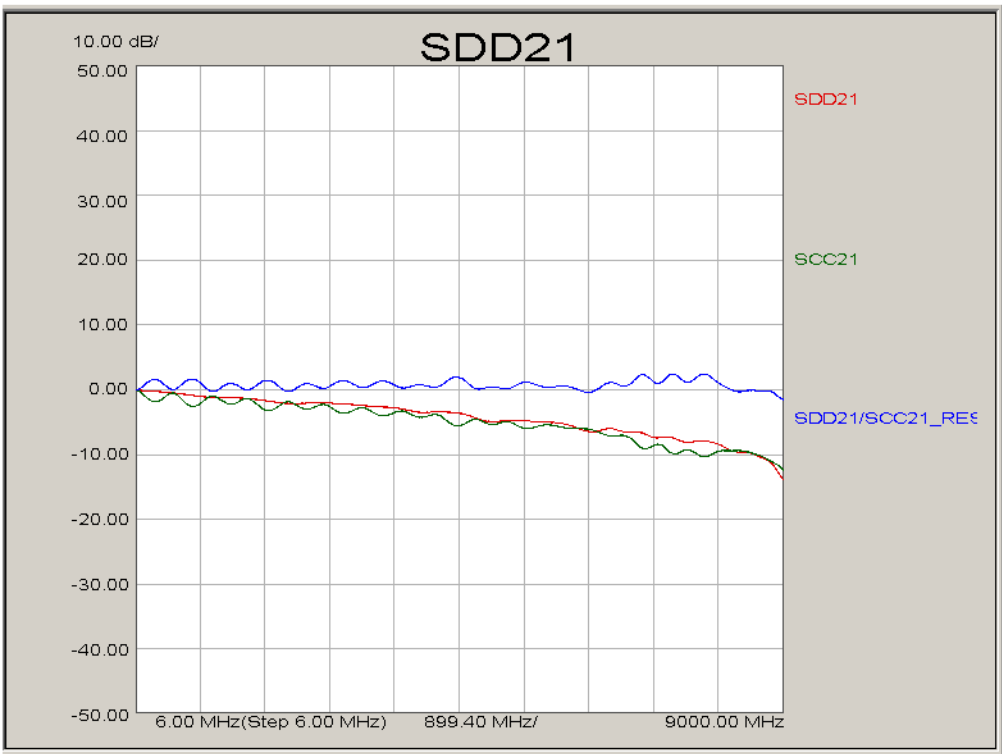
2. Click the right mouse button with the cursor over the plot. Then click **Quick Math** and **Operator**. Click the desired math operation.

3. Click the right mouse button with the cursor over the plot. Then click **Quick Math** and **Operand 2**. Click the label identifying the second trace of the math function.

4. Click the right mouse button with the cursor over the plot. Then click **Quick Math** and **Result (=)** to display the result of the math operation.

   A new trace is displayed showing the results of the math operation. The trace label is displayed along the right side of the plot.

Figure 10-26 Two Traces with Quick Math Result Displayed (SDD21/SCC21)



5. Remove all Quick Math result traces by clicking the right mouse button with the cursor over the plot, then click **Quick Math** and **Remove Math**.

Data Sharing

Data sharing allows you to place a maximum of four traces from different files with different measurement setup parameters on a single plot. This is very helpful when comparing a parameter from the current measurement with the same parameter from a previous measurement that you made.

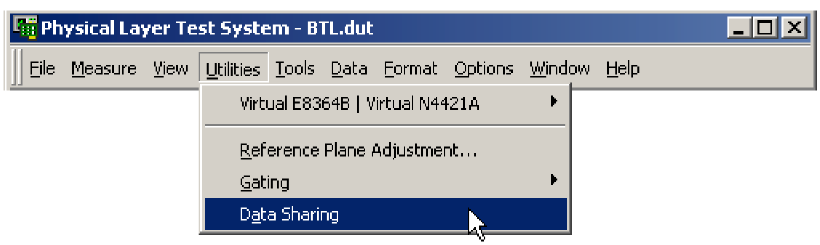
The data sharing feature is only available for Time Domain and Frequency Domain parameters. You may only use data sharing between parameters of the same type. For example, if you have a “Differential Time Domain” parameter, you may only share with other parameters in the “Differential Time Domain” format.

The measurement parameters of the shared files do not need be the same. For example, in Frequency domain, you can compare SDD11 from one file with SCD11 from another file.

To use the data sharing feature, with the two files opened:

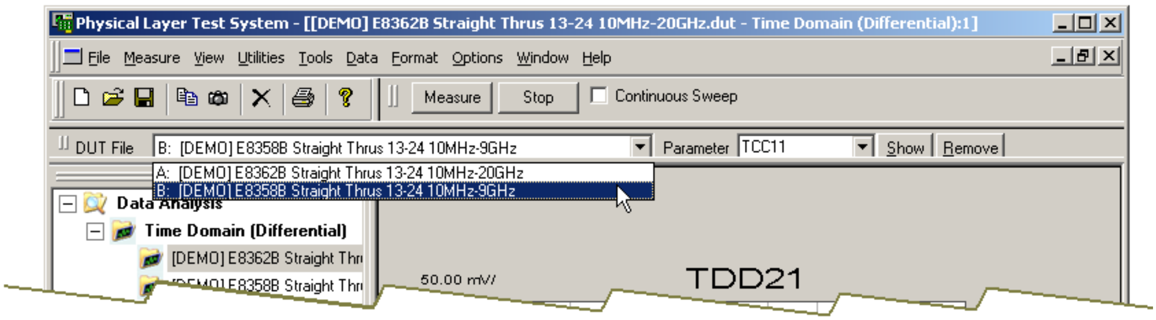
1. Double-click the plot to select the parameter that you are interested in comparing against a parameter of another file.
2. From **Utilities** menu, select **Data Sharing** to open the data sharing bar.

Figure 10-27 Opening the Data Sharing Bar



3. From the **DUT File** list shown in Figure 10-28, select the data file that you want to share with the plot used in step 1.

Figure 10-28 Data Sharing Bar



4. From the **Parameter** list, select the parameter to share with the original plot in step 1.
5. Click the **Show** button to display the selected parameter with the original plot in step 1.

   As files are shared with the original plot, a label noting the parameter of the original plot is displayed to the right of the plot. The parameters of shared files are also have labels displayed to the right of the plot, however, these parameter labels have the shared file appended to the label.
6. Repeat steps 3 through 5 to add another trace to the plot.
7. To remove a trace from the plot, select the DUT file containing the parameter from the **DUT File** list, select the parameter from the **Parameter** list, then click the **Remove** button.
8. Repeat step 7 to remove another trace from the plot.

Characterization Report Generator

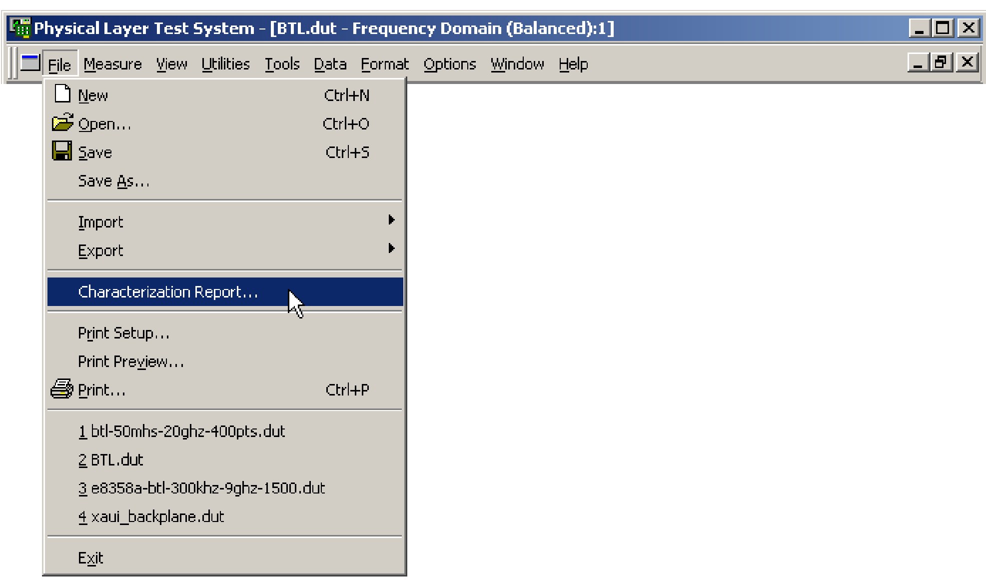
The Characterization Report is a standard report that can be generated to show:

- Device, Test Equipment, Measurement, and Calibration information
- In the time domain, plots and information for:
  - — Differential Impedance (TDD11)
  - — Common Mode Impedance (TCC11)
  - — Eye Diagram
- In the frequency domain, plots and information for:
  - — Differential Loss (SDD21)
  - — Differential Match (SDD11)
  - — Common to Differential (SCD21)

To generate the characterization report:

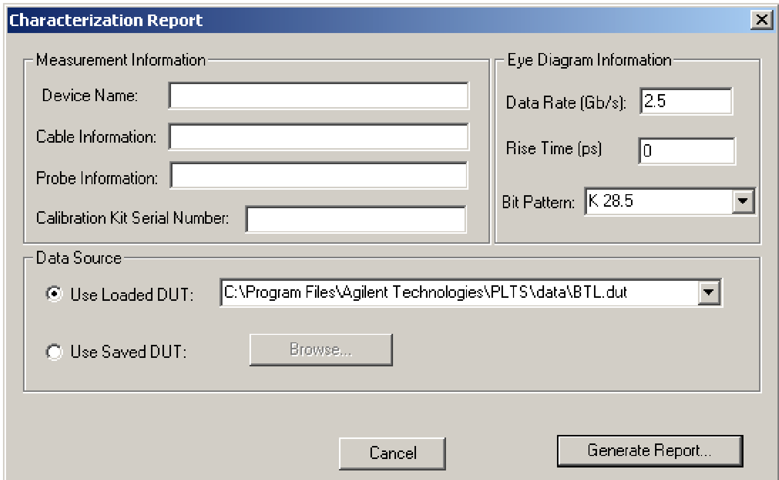
1. Select **Characterization Report**... from the **File** menu to open the *Characterization Report* dialog box. See Figure 10-29.

Figure 10-29 Open Characterization Report



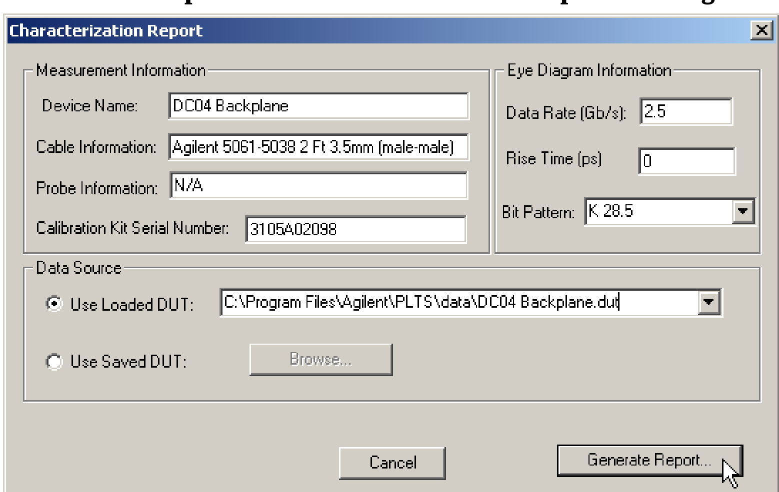
The blank *Characterization Report* dialog box is then displayed as shown in Figure 10-30.

Figure 10-30 Blank Characterization Report Dialog Box



2. Complete the *Characterization Report* dialog box entries. The dialog box has three areas: **Measurement Information**, **Eye Diagram Information**, and **Data Source**. Each area is described below. When the *Characterization Report* dialog box is complete, select the **Generate Report** button to continue.

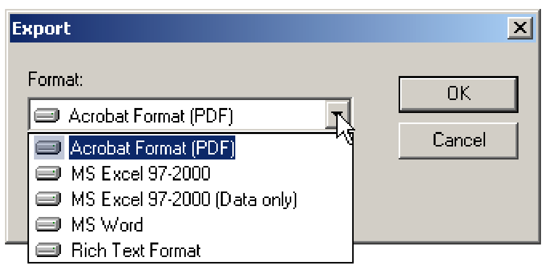
Figure 10-31 Completed Characterization Report Dialog Box



- The **Measurement Information** area allows you to input the following information: **Device Name** (a descriptive name for the DUT), **Cable Information** (any information about the cables used during the test), **Probe Information** (information about any probes used during the test), and **Calibration Kit Serial Number** (affixed to the calibration kit box).
- The **Eye Diagram Information** area allows you input the **Data Rate (Gb/s)**, **Rise Time (ps)**, and the **Bit Pattern**. This information is used to generate the eye diagram for the characterization report.
- The **Data Source** area allows you select the source of the data used to create your characterization report. **Use Loaded DUT** allows you to select any data file that is currently opened by selecting the file from the list. **Use Saved DUT** allows you to select any data file that is saved by clicking the **Browse**... button to locate and select the file.

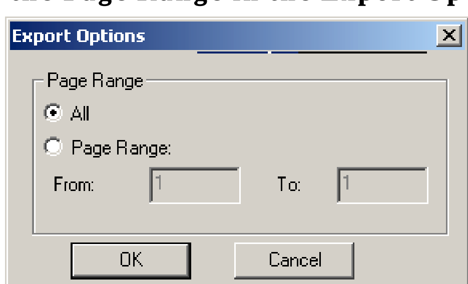
3. In the *Export* dialog box (see Figure 10-32), select the format in which to save the characterization report from the list. Then click the **OK** button.

Figure 10-32 Select the Report Format in the Export Dialog Box



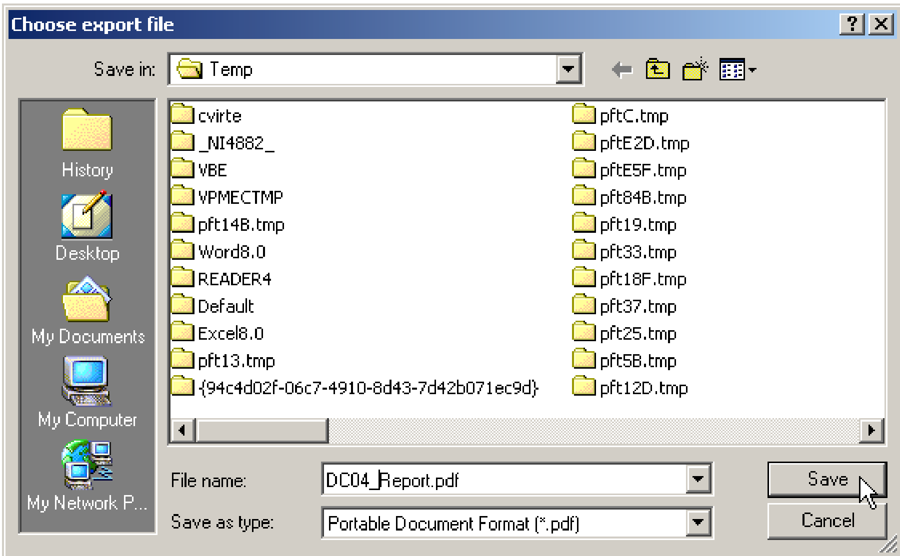
4. In the *Export Options* dialog box (see Figure 10-33), select the page range choice. **All** saves all of the characterization report pages. **Page Range** allows you to enter the range of pages to save. Then click the **OK** button.

Figure 10-33 Select the Page Range in the Export Options Dialog Box



5. In the *Choose export file* dialog box (see Figure 10-34), choose the directory to save the characterization report. You may also change the file name or format from the inputs that were previously entered. Then click the **Save** button to save the characterization report.

Figure 10-34 Choose Export File Dialog Box



CAUTION If you want to save to a file that is already open, close it before you attempt to save it.

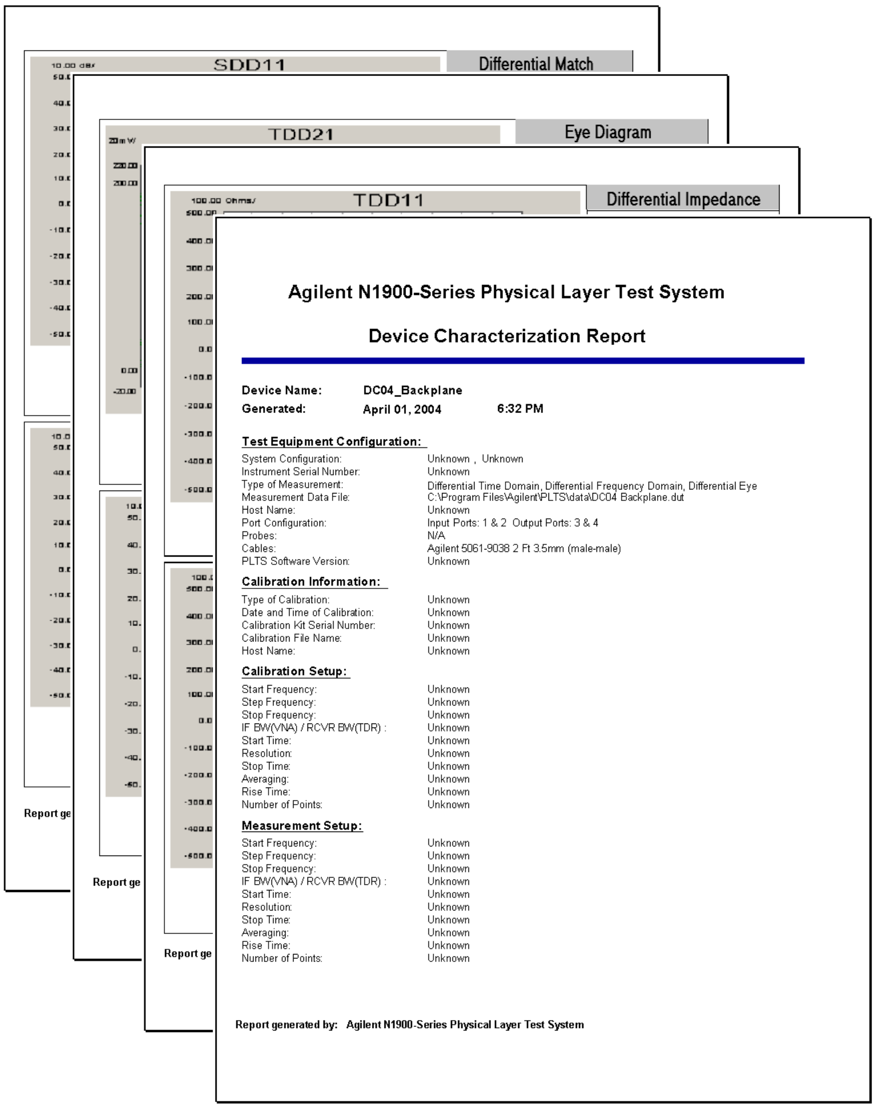
6. Go to the directory to which you saved your characterization report and open the file to review. This is a four-page report as shown in Figure 10-35.

   The following information is available on the Device Characterization Report:

Page 1	Device Name and Report Time Stamp Test Equipment Configuration Calibration Information Calibration Setup and Measurement Setup Information
Page 2	Differential Impedance (TDD11) Plot and Information Common Mode Impedance (TCC11) Plot and Information

Page 3	Eye Diagram Plot and Information Differential Loss (SDD21) Plot and Information
Page 4	Differential Match (SDD11) Plot and Information Common to Differential (SCD21) Plot and Information

Figure 10-35 Device Characterization Report



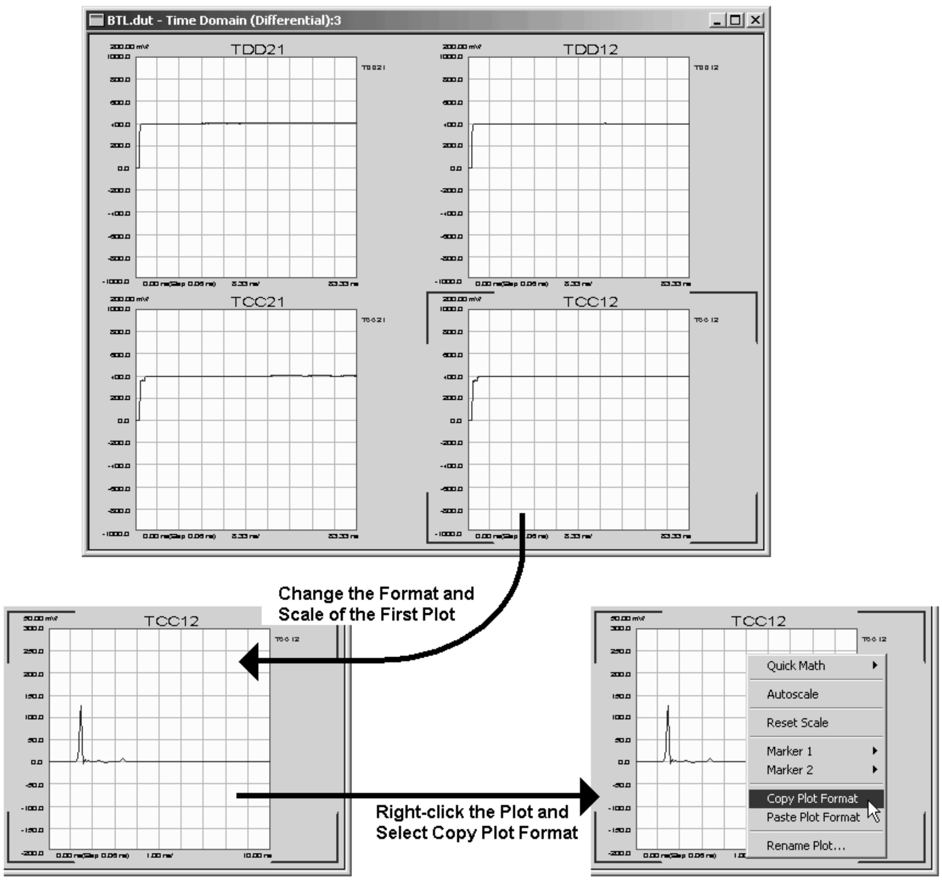
Copying and Pasting Plot Formats

You can change the plot formats quickly using the Copy/Paste Plot Format functions. These functions are very useful when you are displaying several plots within the plot window.

To perform Copy/Paste Plot Format functions:

1. With your data plots displayed, make any changes in the format or scale of the plot to be copied. See Figure 10-36. In this example, the format of TCC12 was changed from step to impulse and the vertical scale was changed.

Figure 10-36 Change and Copy the Plot Format and Scale

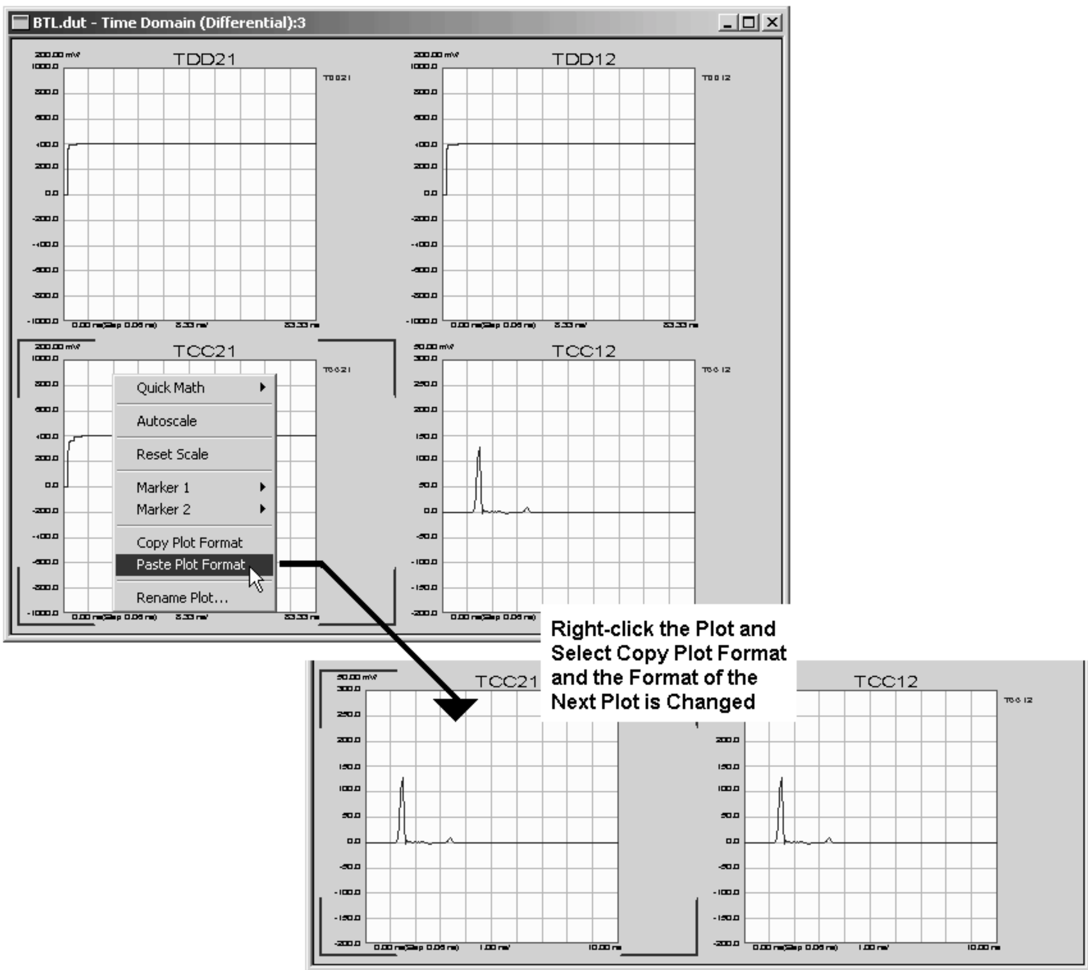


2. Click the right mouse button with the cursor over the plot to be copied, TCC12 in this example. Then click **Copy Plot Format**.

3. First, click the plot to be changed and right-mouse click the plot. Then click **Paste Plot Format**. See Figure 10-37.

   The plot is now displayed with the new format and scale.

Figure 10-37 Select a Plot and Paste the Format



4. Repeat step 3 to change as many plots to the new format and scale that you like.

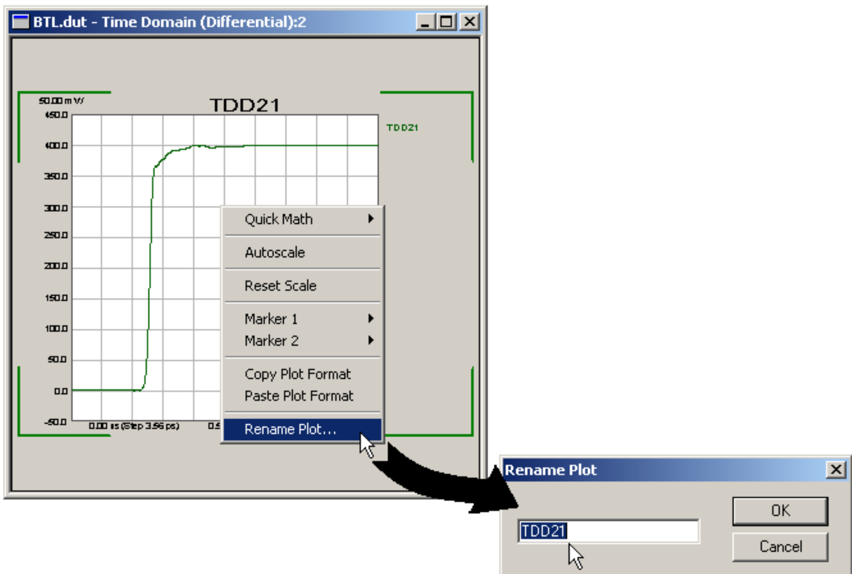
Renaming Plots

Plots are labeled with the parameter that they display. These labels can be changed to text of your choice. Labels are limited to 22 characters in length.

To rename a plot:

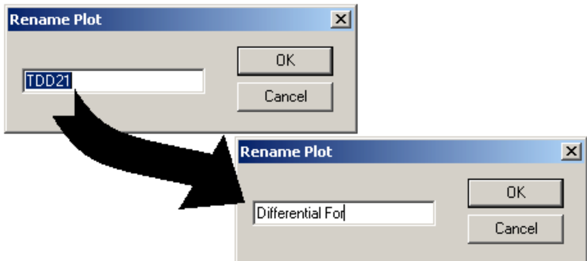
1. Right click on the plot to display the shortcut menu as shown in Figure 10-38. From the shortcut menu, select **Rename Plot**... to open the *Rename Plot...* dialog box.

Figure 10-38 Open the Rename Plot Dialog box



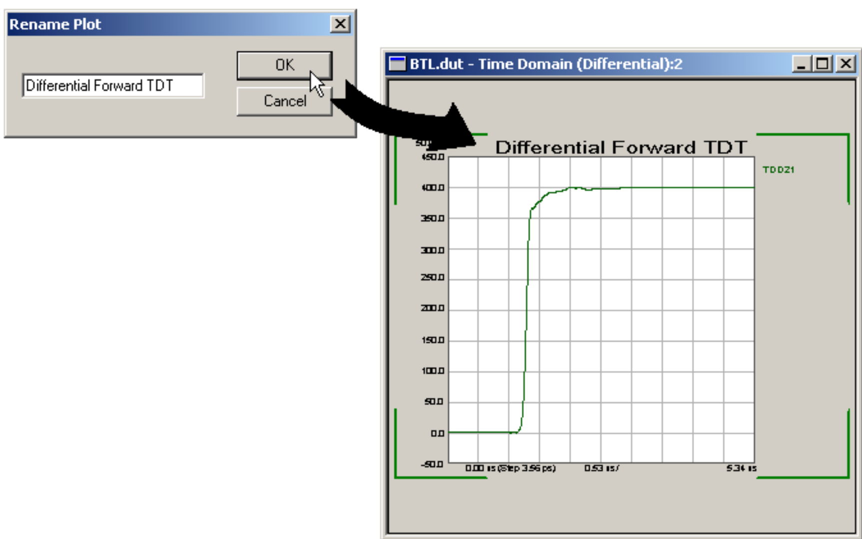
2. Highlight the original label and type in the new label. See Figure 10-39.

Figure 10-39 Enter the New Label



3. When you have finished entering the new label, click the **OK** button. The new label is displayed above the plot. See Figure 10-40.

Figure 10-40 Saving the Label

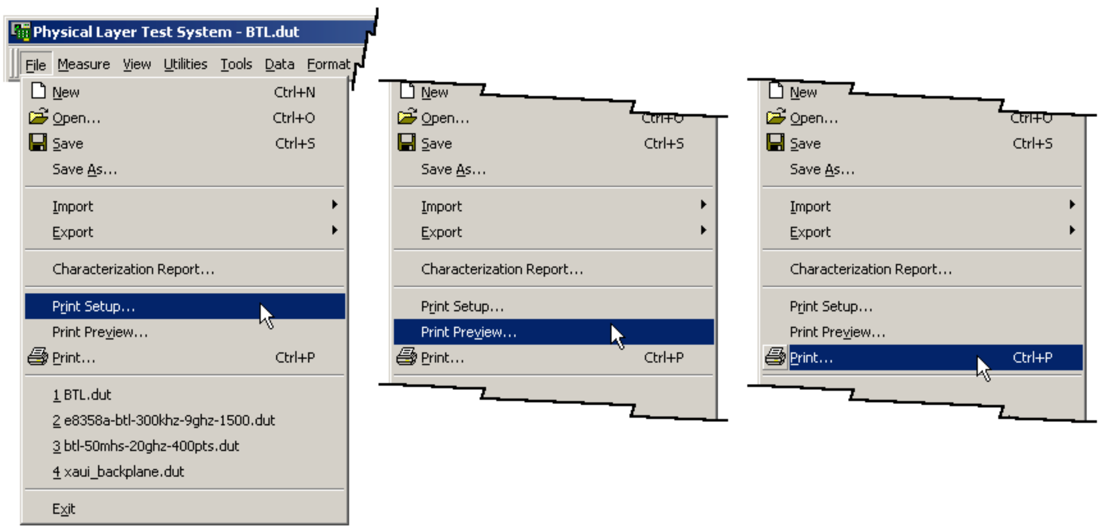


Printing

The Print feature in the **File** menu has three options. Refer to Figure 10-41. Each of the print options is described in this section.

- For Print Setup..., refer to page 325.
- For Print Preview..., refer to page 327.
- For Print..., refer to page 329.

Figure 10-41 Three Print Options from File Menu



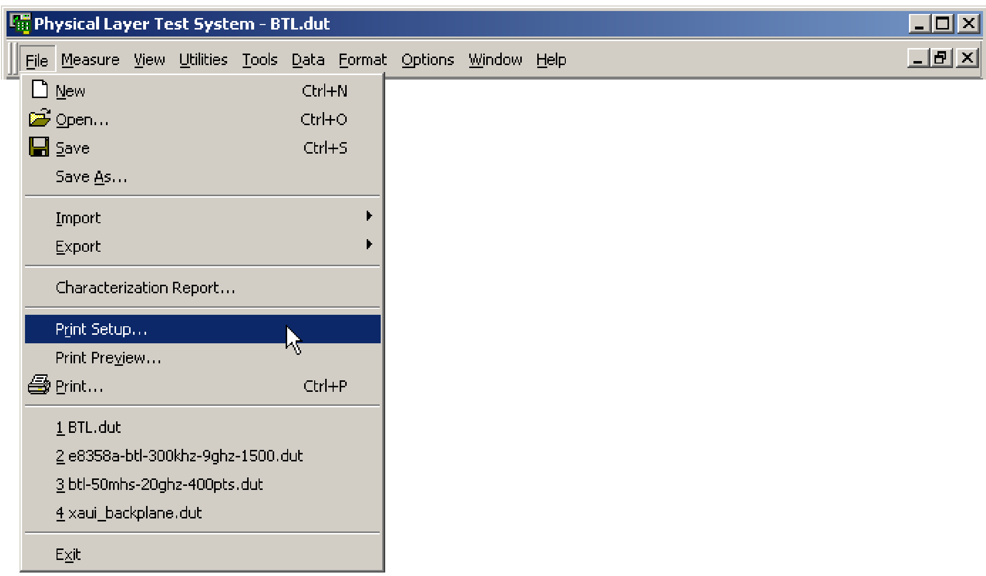
Print Setup...

The *Print Setup* dialog box allows you to set your printer settings to print the displayed PLTS information in the way you choose.

To set the settings for the printer:

1. Select **Print Setup**... from the **File** menu to open the *Print Setup* dialog box. See Figure 10-42.

Figure 10-42 Opening the Print Setup Dialog Box

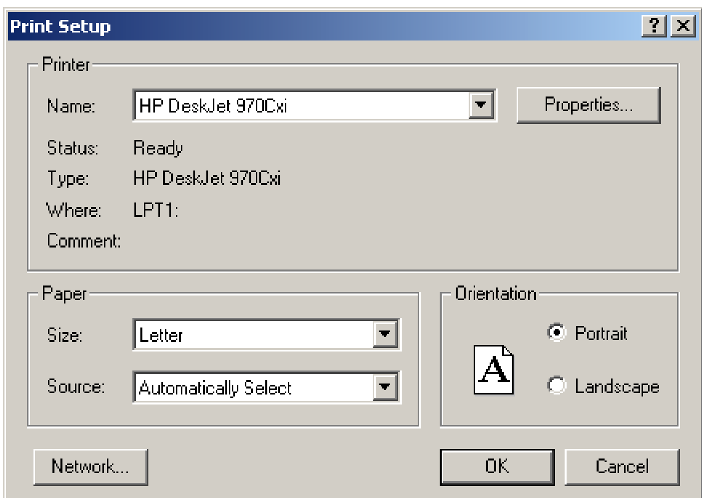


2. Confirm the settings in the *Print Setup* dialog box. See Figure 10-43. Change any incorrect settings for your printer.

   The *Print Setup* dialog box allows you to change the following basic printer functions:

   - The printer: Select a new printer from the **Name** list in the **Printer** area.
   - The paper size: Select the paper size from the **Size** list in the **Paper** area.
   - The paper source: Select the paper source from the **Source** list in the **Paper** area.
   - The paper orientation: Select the orientation, either **Portrait** or **Landscape** in the **Orientation** area.

Figure 10-43 Print Setup Dialog Box



More advanced settings are available by selecting the **Properties**... button or the **Network**... button.

The **OK** button saves the changes that were made, exits the *Print Setup* dialog box, and returns to the PLTS window.

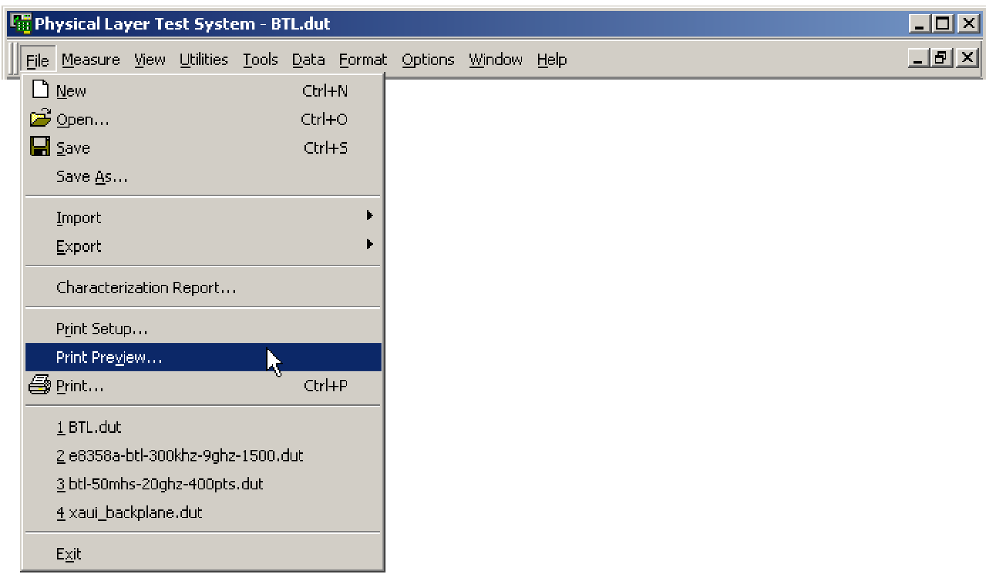
The **Cancel** button cancels any changes that were made, exits the *Print Setup* dialog box, and returns to the PLTS window.

Print Preview…

The **Print Preview**… option allows you to preview your active plot window before printing the display. To preview your active plot window before printing:

1. Select **Print Preview**… from the **File** menu to open the preview screen of the active plot window. See Figure 10-44.

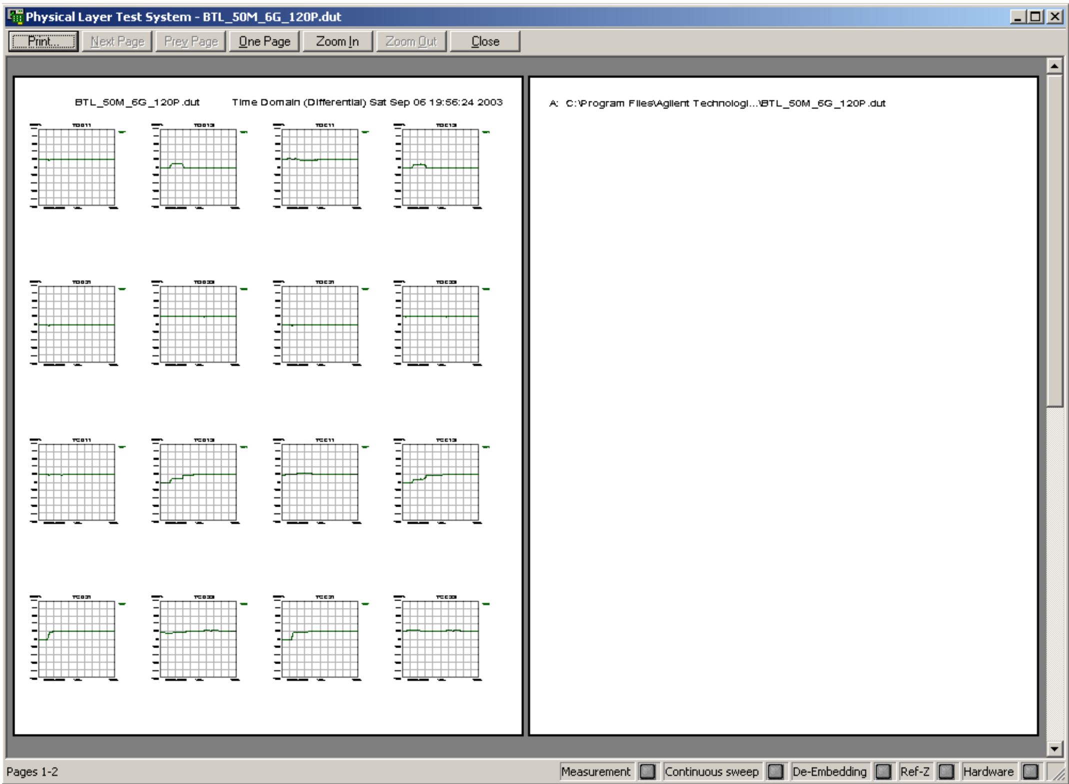
Figure 10-44 Opening the Print Preview Dialog Box



2. The active plot window is displayed for your review. See Figure 10-45.

   The second page of the preview screen displays a list of the open DUT files.

Figure 10-45 Preview Screen of the Active Plot Window



In addition to the area to be printed, the screen displays the following seven buttons across the top of the screen.

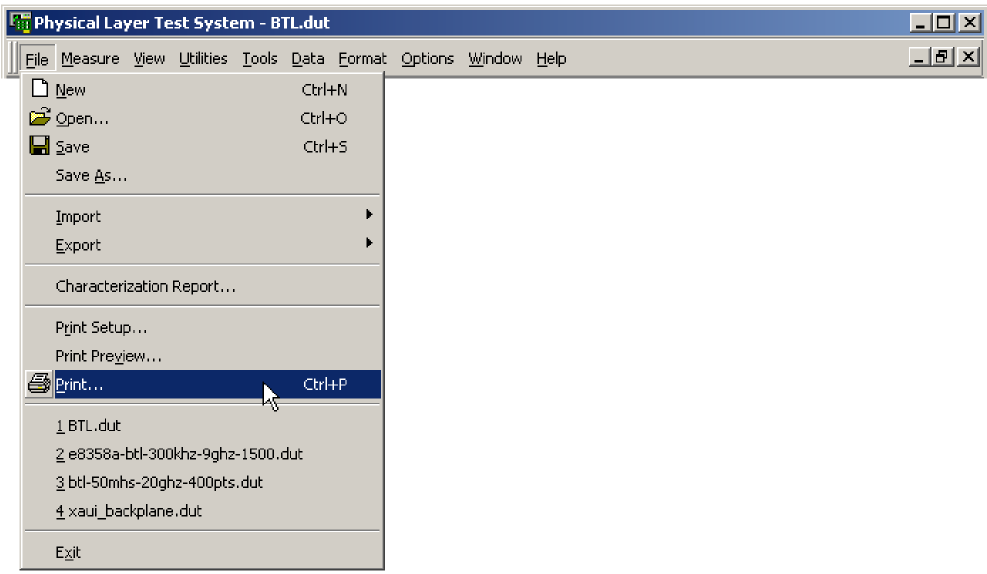
Print:	Prints the active plot window in the preview screen to the printer.
Next Page:	Active when **One Page** is selected. Moves the display to the next page of the display.
Prev Page:	Active when **One Page** is selected. Moves the display to the previous page of the display.
One Page or **Two Page**:	Toggles between the two selections. When only one page of the display is shown, **Two Page** is displayed. When both of the pages of the display are shown, **One Page** is displayed.
Zoom In:	Allows you to zoom in on the displayed active plot.
Zoom Out:	Allows you to zoom out from the displayed active plot.
Close:	Closes the preview screen without printing.

Print...

The active plot window may be printed to the printer defined in the *Print Setup* dialog box. To print the active plot window:

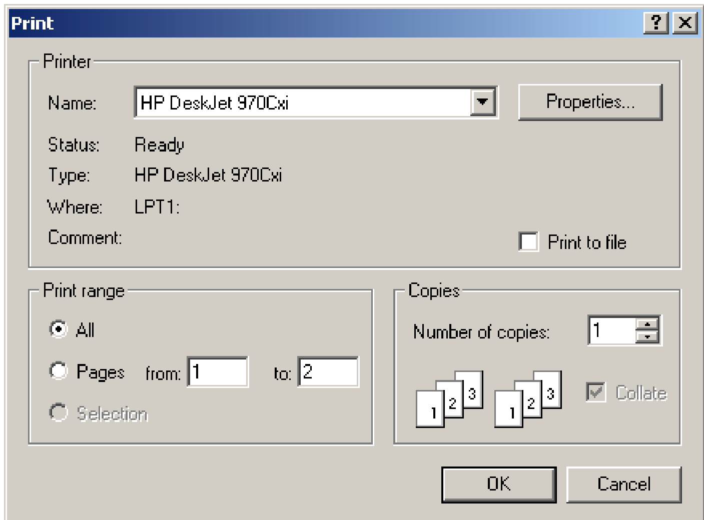
1. Select **Print**... from the **File** menu to open the *Print* dialog box. See Figure 10-44.

Figure 10-46 Opening the Print Dialog Box



2. Make the *Print* dialog box selections that meet your requirements. See Figure 10-47.

Figure 10-47 Print...



In the **Printer** area:

- Select another printer from the **Name** list if the displayed printer is not appropriate.
- Select the **Print to file** check box, to save the active plot window as a print (.prn) file rather than printing to a printer.

In the **Print range** area:

- Select **All** to print all pages.
- Select **Pages** to print specific pages. Then enter the first page to print in the **from:** entry box and enter the last page to print in the **to:** entry box.

In the **Copies** area:

- Enter the **Number of copies** to change the number of copies that you want to print. You may enter a number in the entry box or you may click the spinners (the up arrow and the down arrow at the right edge of the entry box) to change the number of copies.
- If the **Number of copies** entry is greater than “1”, you can toggle **Collate** check box on and off. If the check box has a check, the print copies will be collated.

3. When the *Print* dialog box entries are correct, click the **OK** button to print the active plot window.

   The **Cancel** button exits the dialog box without printing and returns to the main PLTS window.

NOTE The display may also be printed by:

- Selecting the printer button on the Toolbar. See “Toolbar” under the “View Menu” section in the “Menu Reference” chapter of the *PLTS Installation and Reference Guide*.
- Selecting the **Print** button in the print preview display. See page 327.

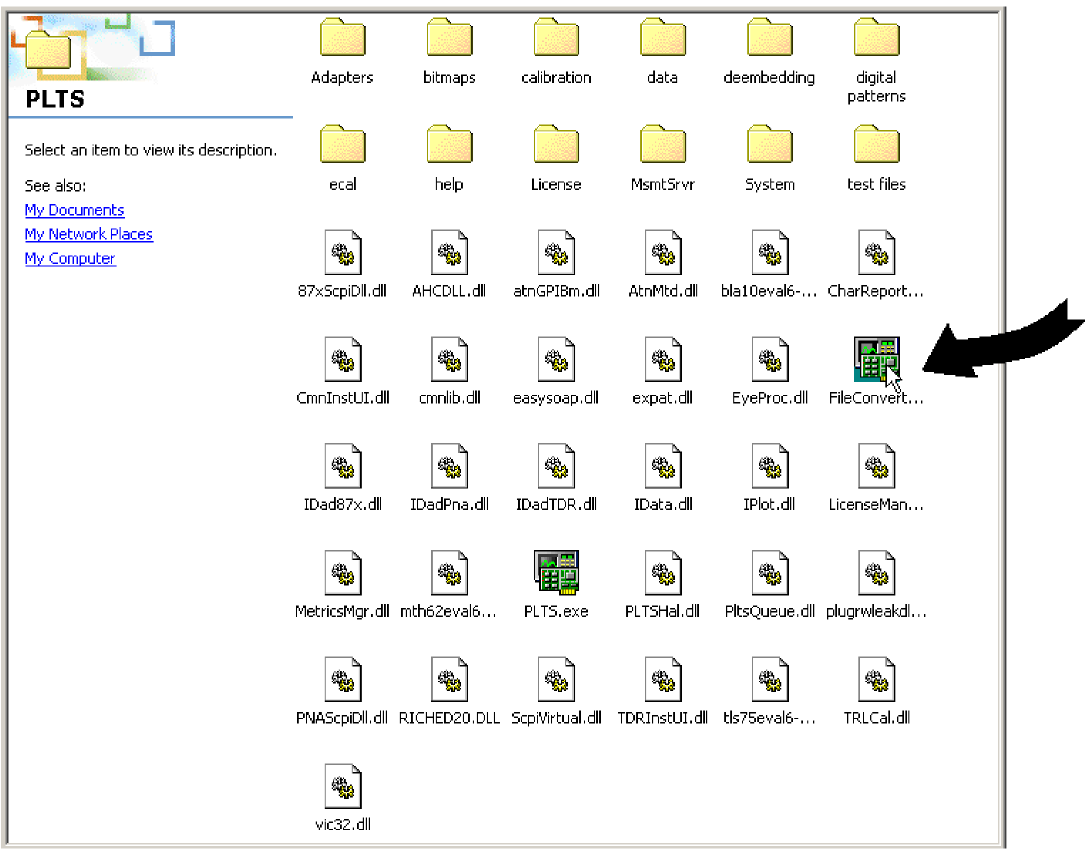
File Converter

The File Converter converts files in .dut, .cit, and .s2p formats to files in .txs format. These .txs files can be used as thru adapter files for calibration and as deembedding files. The File Converter was installed with PLTS, it is a separate utility.

To begin the file conversion:

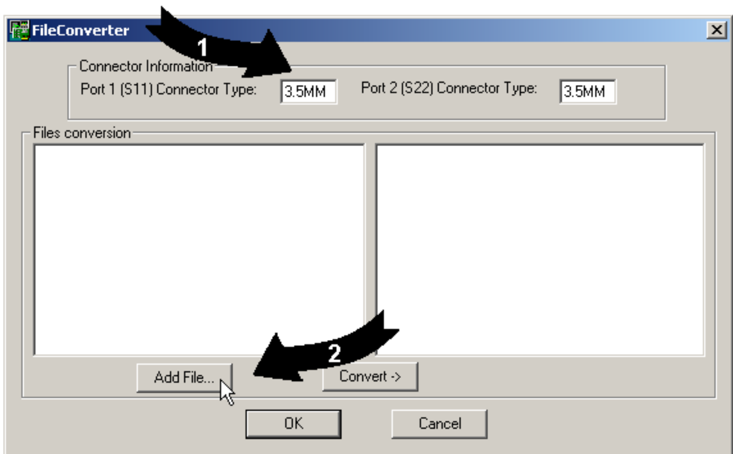
1. From the PLTS directory mentioned above, double-click the FileConverter.exe file icon to open the File Converter.

Figure 10-48 FileConverter.exe Executable File



2. Enter the connector type of the connector in the text boxes and then click the **Add File**… button to open the *Open* dialog box.

Figure 10-49 File Converter Dialog Box

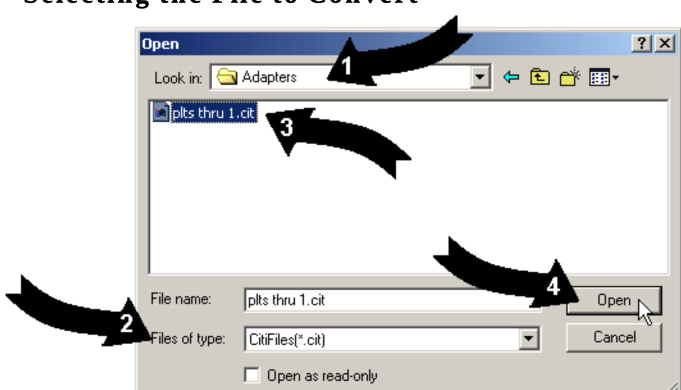


3. In the *Open* dialog box, change to the directory where the files to be converted are located. In this case: C:/Program Files/Agilent/PLTS/Adapters

   Select the type of file (.dut, .cit, or .s2p file) to be converted in the **Files of type:** box. Select the file that you want to convert. Click the **Open** button.

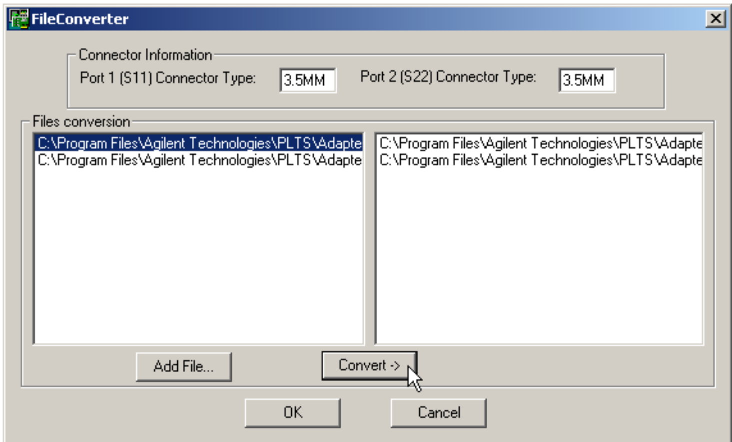
   Repeat if you want to convert multiple files.

Figure 10-50 Selecting the File to Convert



4. With the files in the left list of the **Files conversion** area, click the **Convert ->** button to convert and save the files into the original directory.

Figure 10-51 Convert and Save the Files

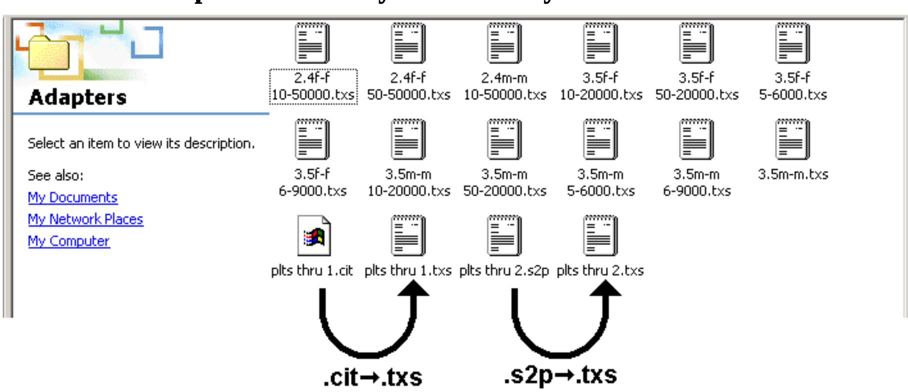


The converted files are listed in the right list of the **Files conversion** area.
In this case: C:/Program Files/Agilent/PLTS/Adapters

5. Click the **OK** button to close the *FileConverter* dialog box.

6. Open the Adapters directory to find the newly converted files. In this case, two files were converted.

Figure 10-52 Adapter Directory with Newly Converted Files



The new .txs files can now be used to define a calibration kit. We recommend you move the newly-converted .txs files into the Adapters directory of PLTS.

III Appendix

Appendix is a collection of supplementary information that you may find useful.

Appendix A, “Procedures”

Provides procedures that may be needed but are not commonly used while using the physical layer test system.

A Procedures

The procedures in this appendix may be required while using the physical layer test system; however, they are not used commonly. The following is a list of the procedures.

- Setting Up the General Purpose Interface Bus Manually is used when you want to set up your GPIB in a manual mode.
- Using the Network Analyzer to Make 2-Port Measurements is used when you want to make 2-port measurements using only the network analyzer.
- Converting a CitiFile to a PLTS Adapter File is used when you want to convert a citifile (".cit") containing measured data to an adapter file (".txs") for de-embedding.
- IF Gain Adjustment is used to set up the E8362A, E8363A, or E8364A PNA network analyzer, with firmware revision less than 3.0, to be used in the PLTS system.

Setting Up the General Purpose Interface Bus Manually

The Physical Layer Test System software will locate and identify your test system equipment automatically. However, there may be the occasion that you need to set up the GPIB address for equipment manually.

The PC uses the General Purpose Interface Bus (GPIB) to communicate with the test system hardware. Each test system device must have a unique GPIB address. There are 32 GPIB addresses, numbered 0 to 31. GPIB addresses for test equipment are set either using switches on the rear panel (as in the case of the test sets for the VNA-based systems and some network analyzers (8720ES and 8753ES)) or using the equipment firmware (as in the case of the PNA network analyzers and Agilent TDR system).

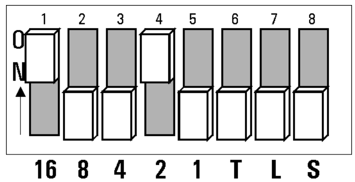
In the case of GPIB addresses that are set using the rear-panel switches, the GPIB addresses are five-bit binary numbers that are set with the switches labeled “16” through “1” (“1” is the least significant digit). The five address switches allow for 32 GPIB addresses, numbered from 0 to 31. The test set has a default address of “18” that is set at the factory. While this address is a unique address in most cases, the address may be changed (if required) to avoid conflicts with other equipment on the same bus.

There are three other switches labeled “T”, “L”, and “S” which correspond to Talk, Listen, and Status. The factory default for these switches is off.

CAUTION If your PC is using an Agilent 82357A USB/GPIB Interface for Windows or any Agilent GPIB card, you must perform the following task to allow the software to recognize the Agilent GPIB device:

In Windows, select **Start**, **Programs**, **Agilent IO Libraries**, then **IO Config**. Click “gpib0” in the **Configured Interfaces** list, then click **Edit**. Change the **SICL Interface Name** to “hpib7” and click **OK**. Then click **OK** to confirm the change. Restarting the software should allow the software to recognize the Agilent GPIB card.

1. Make sure the GPIB address switch on the rear panel of the test set is set to the correct address. The illustration below shows the factory default setting of “18”, which is set by turning on switches “16” and “2”. Make sure the “T”, “L”, and “S” switches are set as shown (off).



2. Make sure that the network analyzer GPIB address is set to a unique address. The factory default address of the network analyzer is “16”. Refer to the network analyzer documentation for information about how to check and set the GPIB address of the analyzer.
3. If your equipment does not have a switch on the rear panel to change the GPIB address, refer to the equipment documentation to changed the address.
4. Make sure that the GPIB cables are connected from the PC to the test equipment. Refer to the hardware installation chapter for your PLTS system in the *PLTS Installation and Reference Guide*.

NOTE After changing any GPIB address setting, cycle the ac power on all system equipment, including the PC, to establish the new GPIB address.

Using the Network Analyzer to Make 2-Port Measurements

You can make two port measurements using the front panel control of the network analyzer without having to disconnect the network analyzer from the test set using the following procedure.

1. Turn off the PLTS software.
2. Toggle the power on the test set to reset the switches.
3. Leave the power of the test set in the ON position.
4. Use the network analyzer from the front panel.

Converting a CitiFile to a PLTS Adapter File

The Physical Layer Test System uses adapter files to de-embed measurement data. PLTS adapter files (".txs" files) can be created by adding a two-line comment near the top of an S-parameter citifile (".cit" file) as shown below. You can use an MS Windows-based ASCII text editor such as Notepad to perform this procedure. The following shows a citifile for a thru device with 3.5mm connector at each end. (The measurement data within this example file has been replaced with a vertical ellipsis only as a space savings for this example.)

```
CITIFILE A.01.00
NAME S-Parameters
VAR FREQ MAG 400
DATA S[1,1] RI
DATA S[1,2] RI
DATA S[2,1] RI
DATA S[2,2] RI
SEG_LIST_BEGIN
SEG 50000000 20000000000 400
SEG_LIST_END
BEGIN
.
.
.
END
```

Using the two comment lines shown below, identified with "#TXS", the type of device and the connector type of its ports are identified.

```
#TXS TYPE=THRU
#TXS PORT1=3.5MM   PORT2=3.5MM
```

These two comment lines are inserted be the first and second line of the citifile as shown below. This is then saved as a ".txs" file in the PLTS deembedding directory.

```
CITIFILE A.01.00
#TXS TYPE=THRU
#TXS PORT1=3.5MM   PORT2=3.5MM
NAME S-Parameters
VAR FREQ MAG 400
DATA S[1,1] RI
DATA S[1,2] RI
DATA S[2,1] RI
DATA S[2,2] RI
```

```
SEG_LIST_BEGIN
SEG 50000000 20000000000 400
SEG_LIST_END
BEGIN
.
.
.
END
```

IF Gain Adjustment

This procedure is for E8362A, E8363A, and E8364A PNA network analyzers that have firmware revisions less than Revision 3.0.

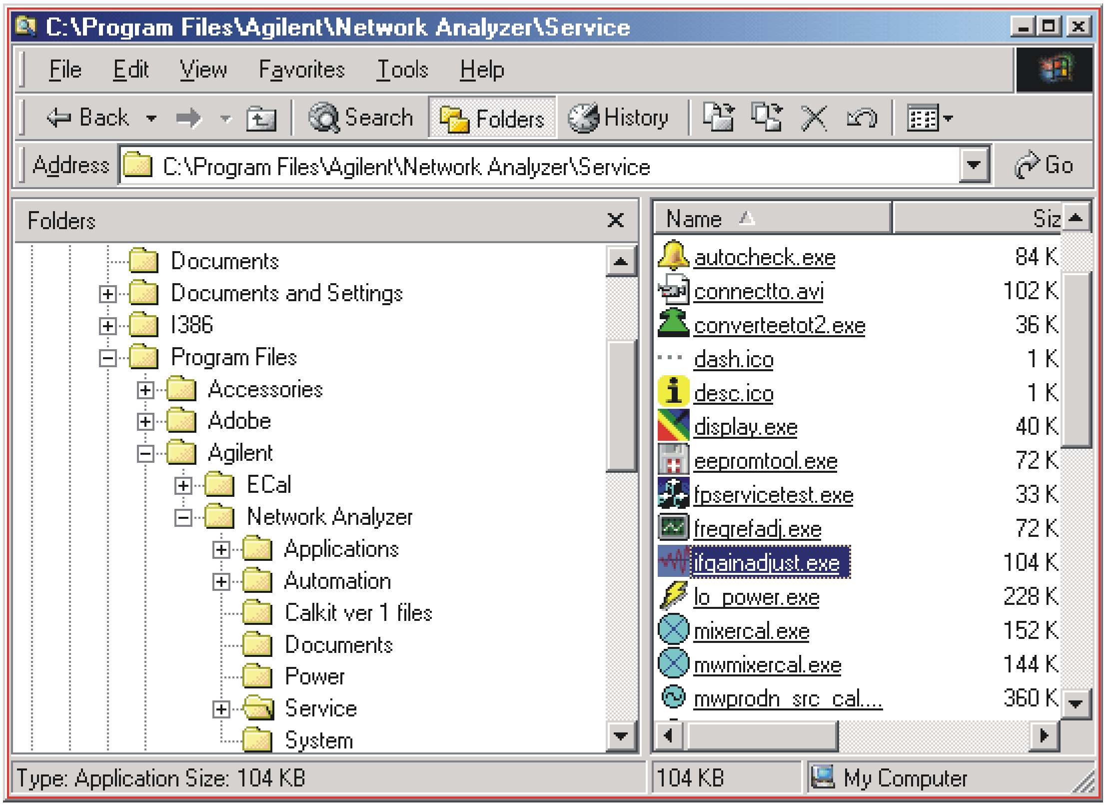
It is recommended that the Service IF Gain Adjustment test be run *before* using the test set. This routine adjusts the R Channel receivers ALC gain to ensure phase lock over the entire frequency range of the PNA Series Network Analyzer. Connect the test set to the analyzer before adjusting the IF gain.

NOTE When the analyzer is removed from the test set for service, or for other applications that do not require the test set, the IF gain adjustment must be run again to return the R Channel receiver ALC gain back to normal.

Adjustment Test

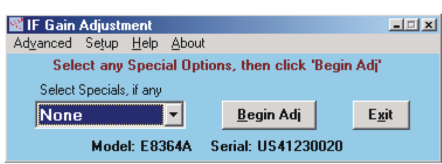
1. Close the PNA Series Network Analyzer window.
2. Open “My Computer”, located on the desk top, double click on “Hard Disk (c:).” (You may also use Explorer). Refer to Figure A-1.
3. Open the following folder path: Programs Files/Agilent/Network Analyzer/Service

Figure A-1 File Path on PNA Network Analyzer Window



4. Double click “ifgainadjust.exe.”
5. Minimize the PNA Network Analyzer window when it appears. You should see the IF Gain Adjust window.

Figure A-2 IF Gain Adjustment Window



6. Select the test set in the “Select Specials, if any” pull down menu and click on “Begin Adj.”

Symbols

Numerics

A

B

C