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: N1930-90003

Printed in USA

October 2003

Supersedes N1930-90002

© Copyright 2002, 2003 Agilent Technologies Inc., all rights reserved.

Notice

The information contained in this document is subject to change without notice.

Agilent Technologies makes no warranty of any kind with regard to this material, including but not limited to, the implied warranties of merchantability and fitness for a particular purpose. Agilent Technologies shall not be liable for errors contained herein or for incidental or consequential damages in connection with the furnishing, performance, or use of this material.

LIMITED WARRANTY

The material contained in this document is provided “as is,” and is subject to being changed, without notice, in future editions. Further, to the maximum extent permitted by applicable law, Agilent disclaims all warranties, either express or implied with regard to this manual and any information contained herein, including but not limited to the implied warranties of merchantability and fitness for a particular purpose. Agilent shall not be liable for errors or for incidental or consequential damages in connection with the furnishing, use, or performance of this document or any information contained herein. Should Agilent and the user have a separate written agreement with warranty terms covering the material in this document that conflict with these terms, the warranty terms in the separate agreement will control.

Assistance

Product maintenance agreements and other customer assistance agreements are available for Agilent Technologies products. For any assistance, refer to “Contacting Agilent” on page 552.

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 Chapter 3, Installing the Physical Layer Test System Software, for the licensing procedure.

Software Compatibility

This user's guide is compatible with Physical Layer Test System software revisions 2.000 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

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.

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

Contents

I Installation

Part I guides you through the initial steps of installing the physical layer test system.

Chapter 1, “Installing the VNA-Based Physical Layer Test System Hardware”

Provides you a step-by-step installation procedure for the VNA-based PLTS hardware.

Chapter 2, “Installing the TDR-Based Physical Layer Test System Hardware”

Provides you a step-by-step installation procedure for the TDR-based PLTS hardware.

Chapter 3, “Installing the Physical Layer Test System Software”

Provides you a step-by-step installation procedure for the PLTS software. This chapter also explains the various areas of the main software screen.

1 Installing the VNA-Based Physical Layer Test System Hardware

The Physical Layer Test System (PLTS) consists of the following items:

- Personal computer (PC)
- VNA-based system (Network analyzer and S-parameter test set)
- PLTS software

The installation procedure in this chapter will lead you through setting up the hardware (the PC and the VNA-based system). After that is complete, you will refer to Chapter 3, Installing the Physical Layer Test System Software, to install the software.

NOTE If you have the TDR-based PLTS system, refer to Chapter 2, "Installing the TDR-Based Physical Layer Test System Hardware," on page 43 for instructions on setting up that system.

This installation procedure will lead you through a series of steps to set up your PLTS hardware. The following is a list of the installation steps:

- Step 1. Set Up the Personal Computer
- Step 2. Verify your System Shipment
- Step 3. Set Up the Network Analyzer
- Step 4. Attach the Test Set to the Network Analyzer (N4420B or N4464A/B Test Set Only)
- Step 5. Install the S-Parameter Test Set on a Bench Top or in an Equipment Rack
- Step 6. Make the Interconnections between the S-Parameter Test Set and the Network Analyzer
- Step 7. Set Up the General Purpose Interface Bus (GPIB)
- Step 8. Power up the Physical Layer Test System

NOTE These installation instructions were written specifically for customers who have just received their PLTS. If you have already been using our S-parameter test set and its corresponding network analyzer, you have probably completed most of these installation steps. Review these installation steps to ensure that your system is currently set up as recommended. Then, begin the software installation process by starting at Chapter 3, "Installing the Physical Layer Test System Software," on page 51.

Step 1. Set Up the Personal Computer

1. Make sure that your PC meets the following minimum system controller requirements:

Table 1-1 Minimum PC Requirements by PLTS Modes of Operation

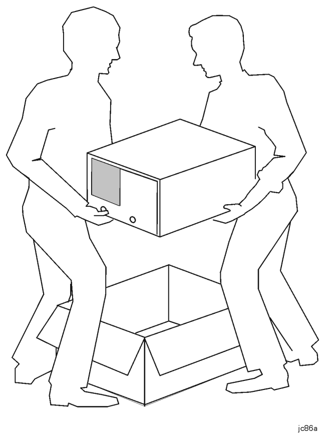
PC Requirement	Measurement Controller Mode	Off-Line Analysis Mode
	In the lab, controlling test equipment and making quick analysis of the results	In your office, performing “What if...” analysis, characterization, cross-domain analysis, filtering, waveform math, and eye diagram simulation
CPU	400 MHz Pentium®[a] II or greater	1 GHz Pentium III compatible PC
Main Memory	256 MB[b]	512 MB
Virtual Memory[c]	512 MB	768 MB
GPIB Interface	Agilent 82357A USB/GPIB Interface for Windows or supported GPIB card (any National Instruments or Agilent 82340/41 or 82350 GPIB card)	No GPIB connection is required to utilize PLTS in the off-line mode. Saved (stored) measurement files can be recalled at any time for analysis.
Operating Systems	Windows 2000 or Windows XP[d]	
Screen Resolution	1024 × 768	

a. Pentium® is a U.S. registered trademark of Intel Corporation.
b. 512 MB of Main Memory is recommended for the Measurement Controller Mode when the measurement is measuring 16,000 points with the PNA B-model network analyzer.
c. As a general rule for optimum PC performance when using PLTS, virtual memory should be 1.5 to 2 times the size of the main memory.
d. Earlier versions of Windows are no longer supported by PLTS.

2. Using the PC documentation, make sure that the PC is operating properly.
3. Make sure the GPIB card is installed in the PC and that it is operating properly.
4. Make sure the PC is located near where you will position the Physical Layer Test System (PLTS). Later in this process, you will connect the GPIB card to the PLTS using a GPIB cable.

Step 2. Verify your System Shipment

1. Unpack your system from the containers in which it was shipped.



WARNING **The test system hardware is heavy. Use proper lifting techniques. The network analyzer can weigh between 53 lb. (24 kg) and 64 lb. (29 kg). The test set can weigh as much as 20 lb. (9 kg).**

2. Carefully inspect the system hardware to make sure that it was not damaged during shipment.

NOTE If your test system was damaged during shipment, contact Agilent Technologies. Refer to "Contacting Agilent" on page 552.

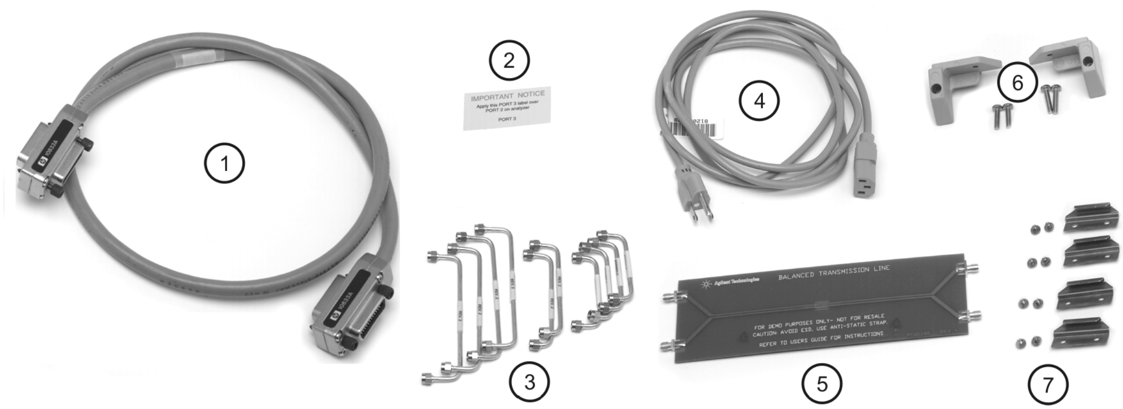
3. Use Table 1-2 to verify that your test set is compatible with your network analyzer and its installed options. "Step 3. Set Up the Network Analyzer" on page 10 has additional network analyzer option information if you have a question. If the installed options are not compatible, contact us before proceeding. Refer to "Contacting Agilent" on page 552.

Table 1-2 Physical Layer Test System Configurations

Test Set Model Number	System Frequency Range	Supported Network Analyzer			
		Model Number	Options[a]		
			Required	Compatible	Incompatible
N4415A	30 kHz to 6.0 GHz	8753ES	006[b], 014	002, 004, 010, 1D5	011, 075, H16
N4416A	300 kHz to 6.0 GHz	E8356A[c]/7A/8A	015	010, 1D5	
N4417A[d]	300 kHz to 9.0 GHz	E8356A[c]/7A[e]/8A	015	010	
		E8801A[c]/2A[e]/3A	014	010, 1E1, 1E5	
N4418A	50 MHz to 20 GHz	8720ES	H32 or H42	010, 012, 400	007, 085, 089
		8722ES[f]	H32 or H44	010, 012, 400	007, 085, 089
N4419A[g]/B	10 MHz to 20 GHz	E8362A[g]/B	014	010, 022, 711, UNL	
N4420B	10 MHz to 40 GHz	E8363A[g]/B	014	010, 022, 711, UNL	
N4421A[g]/B	10 MHz to 50 GHz	E8364A[g]/B	014	010, 022, 711, UNL	

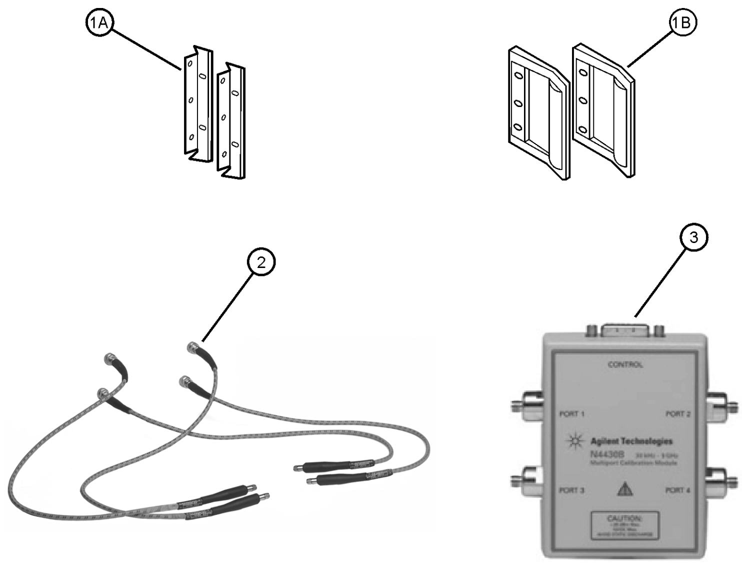
a. This table lists only the most specifically relevant options. For compatibility with options not listed here, contact the factory.
b. Option 006 required only for operation above 3 GHz.
c. Using this network analyzer, the maximum operating frequency is limited to 3 GHz.
d. E8356A family requires N4417A Option 103; E8801A family requires N4417A Option 104.
e. Using this network analyzer, the maximum operating frequency is limited to 6 GHz.
f. When an 8722ES is used with an N4418A, the N4418A requires Option 302. The system's maximum operating frequency is limited to 20.0 GHz.
g. "A" models of this test set or network analyzer have a start frequency of 45 MHz.

4. Check the accessories that were shipped with your system. Your network analyzer accessories will be checked during the network analyzer setup.



Item Nr	Part Number	Part Description
1	8120-3445	GPIB Cable (3 feet)
2	N/A	"Port 3" Label
3	Varies by Test Set Model and Option	Semirigid interconnect cables (refer to Step 6. Make the Interconnections between the S-Parameter Test Set and the Network Analyzer for the appropriate number of cables)
4	Unique to country	AC Power Cord (for the test set)
5	AD00658	Balanced Transmission Line PC Board Device (Sample DUT)
6	Left: Z5823-20239 Right: Z5823-20240	2 Rear Locking Feet (N4420B, N4421A/B only) (With 4 Screws - 2 each - longer: 0515-0686 and shorter: HW00235)
7	1600-1423	4 Lock Links (N4420B, N4421A/B only) with 8 screws (0515-1499)

5. If you ordered any of the following options, check the parts. Option 1CP is shipped in a separate container.



Option Number	Item Number	Part Description
1CP	1A 1B	Rack mount flange kit (For use with handles) Handles (set of 2)
B20	2	Precision 50-ohm cables (4)
060	3	4-Port Electronic Calibration Module/Kit (Not available as an option for the N1951A/53A/53B/55B/57A/57B PLTS)
		Other Calibration Kits
N/A	Not Shown	85033E 3.5mm Calibration Kit (0 Hz (dc) – 9 GHz) [1] 85052D 3.5mm Economy Calibration Kit (0 Hz (dc) – 26.5 GHz) [1] 85056A 2.4mm Precision Calibration Kit (0 Hz (dc) – 50 GHz) [1] 85050C 7 mm Precision Calibration Kit (0 Hz (dc) – 18 GHz) [2]

[1] Kit for SOLT Calibration; [2] Kit for TRL Calibration

Step 3. Set Up the Network Analyzer

1. Using Table 1-3, verify that your network analyzer options are compatible with the physical layer test system. Incompatible options are shaded.

Table 1-3 Common Hardware Option Number Descriptions for Network Analyzers

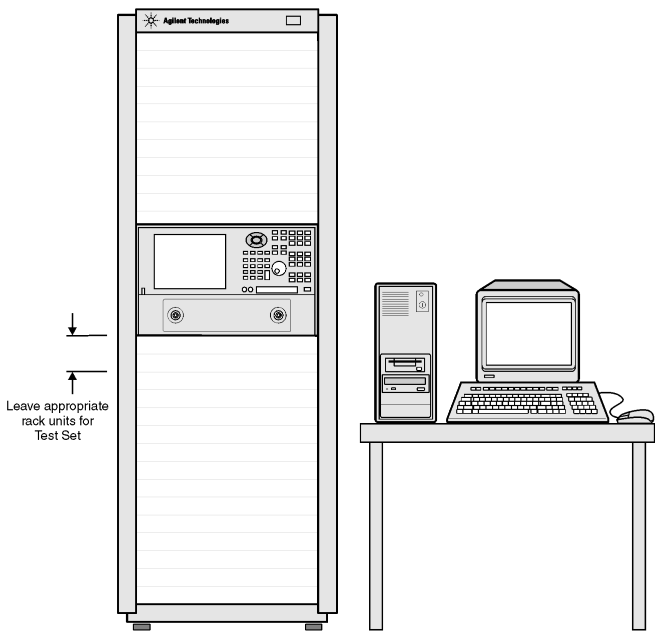
8753ES Network Analyzer Options			
002	Harmonic-Measurement Upgrade	004	Step Attenuator Upgrade
006	6 GHz Upgrade for Standard Units	010	Time Domain Capability
011	Receiver Configuration	014	Configurable Test Set
075	75 Ohm Impedance	1D5	High Stability Frequency Reference
H16	Low Noise Floor		
8720ES and 8722ES Network Analyzer Options			
007	Mechanical Transfer Switch	010	Time Domain Capability
012	Direct Sampler Access	085	High-Power Test System
089	Frequency Offset Mode	1D5	High Stability Frequency Reference
400	Four-Sampler Test Set	H32	Front panel access to A and B samplers and Port 1 and Port 2 switch and coupler
H42	8719/8720 only: Front panel access to all samplers and Port 1 and Port 2 switch and coupler (installs options 400 & 012)	H44	8722 only: Front panel access to R1, R2, A, and B samplers, and Port 1 and Port 2 switch and coupler ports (installs options 400 & 012)
E8356A, E8357A, and E8358A Network Analyzer Options[a]			
010	Time Domain Capability	015	Configurable Test Set
E8801A, E8802A, and E8803A Network Analyzer Options[a]			
010	Time Domain Capability	014	Configurable Test Set
1E1	Extended Power Range	1E5	High Stability Timebase
E8362A/B, E8363A/B, and E8364A/B Network Analyzer Options[a]			
010	Time Domain Capability	014	Configurable Test Set
016[b]	Add Receiver Attenuators	022	Extended Memory

Table 1-3 Common Hardware Option Number Descriptions for Network Analyzers

080[b]	Frequency Offset	081[b]	External Reference Switch
083[b]	Frequency Converter Measurement Application	711	Standard Power Range
UNL	Extended Power Range with Bias Tees		

a. PNA network analyzers should have firmware revision A.03.53 or later. Contact the factory for information regarding PLTS support of earlier firmware revisions.
b. This option has not been tested and is not specified with the Physical Layer Test System.

2. Using the network analyzer's Installation and Quick Start Guide, set up the network analyzer.
3. If you are installing your network analyzer in an equipment rack, be sure to leave at least 2 rack units of space *below* the analyzer to install the test set.



NOTE For the N4420B or N4421A/B test set, connect the E8363A/B or E8364A/B network analyzer to the test set before placing in the rack as a single unit on one set of rails. Refer to "Step 4. Attach the Network Analyzer to the Test Set" on page 12 for instructions.

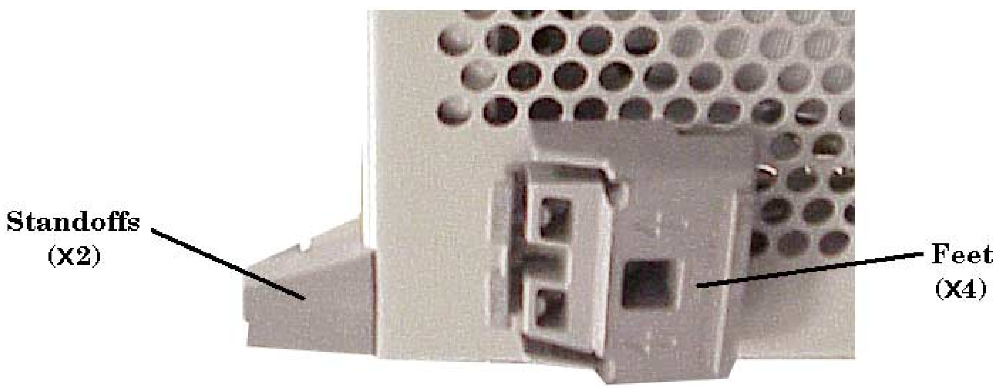
Step 4. Attach the Network Analyzer to the Test Set
(N4420B or N4421A/B Test Set Only)

If your test set is *not* an N4420B, an N4421A, or an N4421B, continue with “Step 5. Install the Test System on a Bench Top or in an Equipment Rack” on page 15.

The E8363A/B and E8364A/B network analyzers are attached to theN4420B and N4421A/B test sets using lock links at the front and locking feet at the rear. This hardware is supplied with the test set. Other network analyzers are ***not*** attached to test sets (N4415A, N4416A, N4417A, N4418A, and N4419A/B) using this hardware.

Preparing the Network Analyzer

1. Remove the four feet from the bottom of the network analyzer.

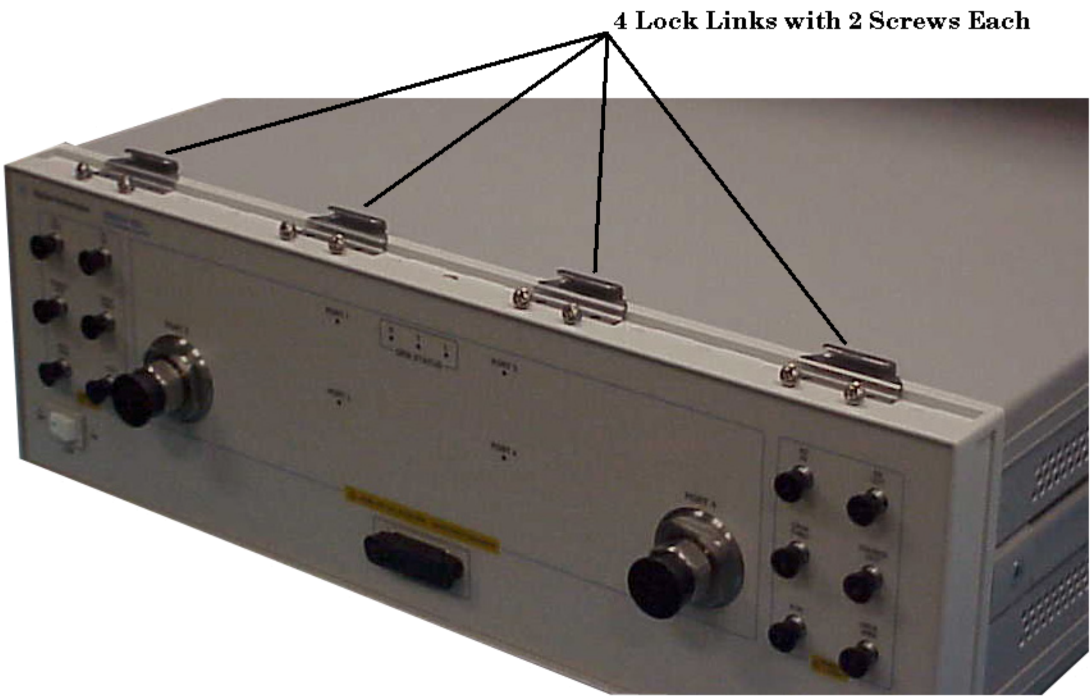


2. Remove the screws from the two lower rear panel standoffs.
3. Install the two rear locking feet where the standoffs were removed. Use part number Z5823-20239 on the left side of the analyzer and part number Z5823-20240 on the right side. Use the two longer screws to secure the feet to the analyzer.

Preparing the Test Set

4. Remove the trim strip from the top of the front frame.
5. Install the four lock links to the top of the front frame using eight screws.

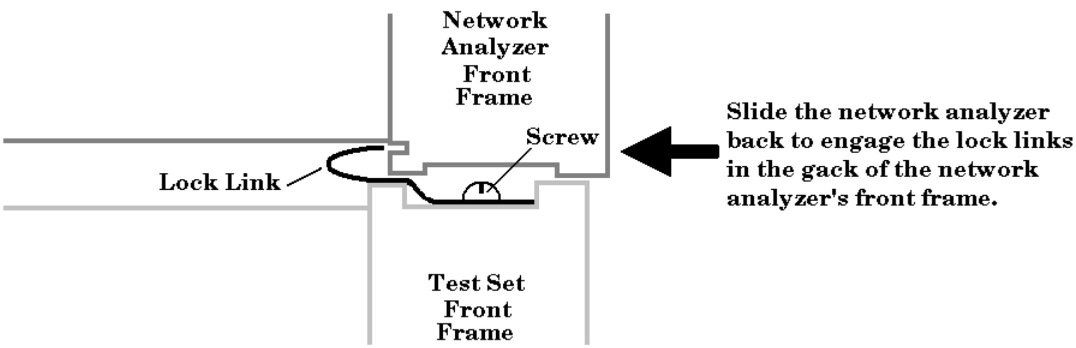
Lock Link Installation



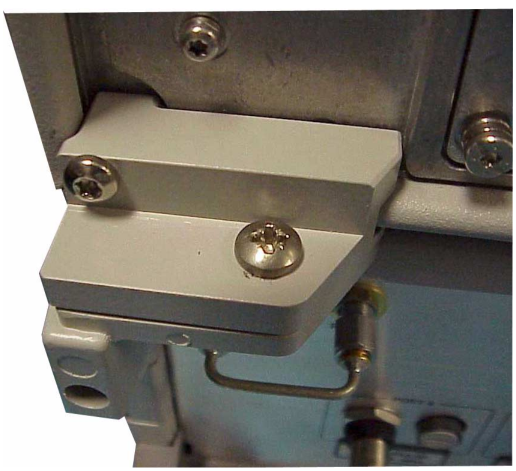
Attaching the Network Analyzer to the Test Set

6. Place the network analyzer on top of the test set ensuring that the front frame of the network analyzer is positioned slightly forward of the lock links that are attached to the the test set. Then slide the network analyzer back so the lock links engage the front frame of the analyzer.

Making the Lock Link Connection



7. Secure the network analyzer's lower locking feet to the test set's upper locking feet by inserting the shorter two screws between the two pairs of locking feet, one on each side of the instrument as shown below.



If the screw holes between the network analyzer's lower locking feet are not aligned with the screw holes in the test set's upper locking feet, loosen the screws securing the feet to the instruments slightly to align.

8. Tighten all screws.

Step 5. Install the Test System on a Bench Top or in an Equipment Rack

The test system can be installed on a bench top or in an equipment rack.

In all installations, consider the following ventilation requirements when deciding where to set up your test system.

CAUTION **Ventilation Requirements:**
When installing the product in a cabinet, the convection into and out of the product must not be restricted. The ambient temperature (outside the cabinet) must be less than the maximum operating temperature of the instrument by 4 °C for every 100 watts dissipated in the cabinet. If the total power dissipated in the cabinet is greater than 800 watts, then forced convection must be used.

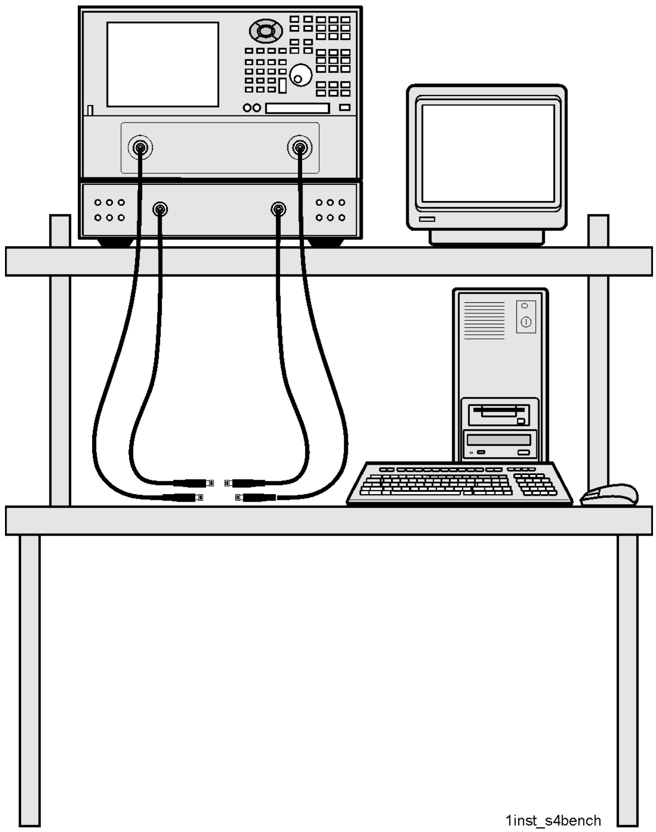
Refer to the section that applies to your installation.

- **For bench top installation**, continue on page 16.
- **For equipment rack installation**, continue on page 17.

To Install on a Bench Top

1. Place the test set and the network analyzer on a bench top. In the example illustration shown below, the system is placed on a riser as an alternative to the bench top. Make sure that there is at least four inches of clearance on the sides and back of the system for adequate ventilation.

   The front panel test cables are shown only as a reminder to make sure they can easily reach the test surface of the bench.



CAUTION Consider the ventilation requirement described on page 15 when selecting the location of your system.

2. Continue with "Step 6. Make the Interconnections between the S-Parameter Test Set and the Network Analyzer" on page 21.

To Install in an Equipment Rack

You may install the PLTS in an equipment rack in one of following two ways:

Removing Feet from VNA	Leaving Feet attached to VNA
1. Install one set of rails into the equipment rack 2. Remove feet from Test Set and VNA 3. Attach mount flanges and the handles to Test Set 4. Attach mount flanges and the handles to VNA 5. Insert test set on rails in equipment rack and screw to rack 6. Place VNA on top of test set and screw VNA into rack 7. Bend Semi rigid interconnect cables to fit between the test set connector and the VNA connector 8. Connect semirigid between test set and VNA	1. Install one set of rails into the equipment rack 2. Remove feet from Test Set only 3. Attach mount flanges and the handles to Test Set 4. Attach mount flanges and the handles to VNA 5. Insert test set on rails in equipment rack and screw to rack 6. Place VNA on top of test set 7. Connect semirigid between test set and VNA

When you install the test set in an equipment rack, you will install rails in the rack to support the weight of the test set, attach the handles and the rack mount flanges to the test set, and secure the test set to the equipment rack.

1. Ensure that the front handle kit, the rack mount flange kit, and the rack mount rail set are complete.

Handle Kit Contents	Flange Kit Contents
• 4 Screws • 2 Side Trim Strip • 2 Handles • Installation Instructions	• 4 Screws (Long) • 4 Screws with Washers • 4 Nuts with Metal Clips • 2 Flanges • Installation Instructions

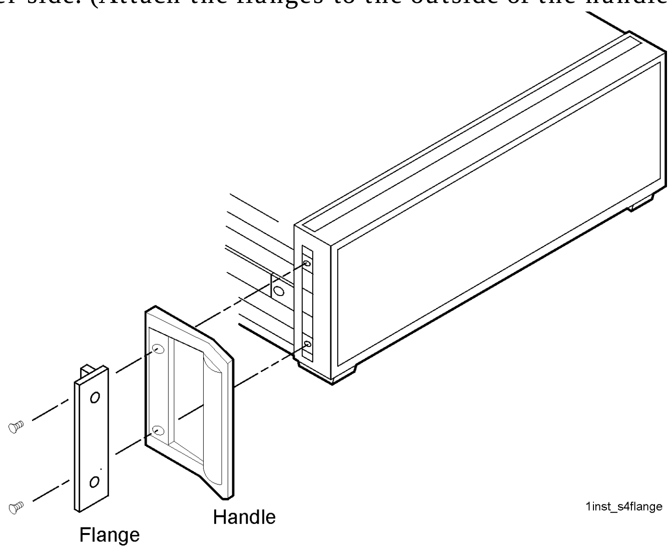
NOTE If any items are damaged or missing from a kit, contact us (refer to “Contacting Agilent” on page 552) to order a replacement kit. Items within these kits are not individually available.

2. Install the rails into the equipment rack using the instructions provided. Consider that the test set is two rack units high (3.5 inches). Mount the test set immediately below the network analyzer.

NOTE For the N4420B, N4421A, or N4421B test set, connect the network analyzer to the test set before placing in the rack as a single unit on one set of rails.

---

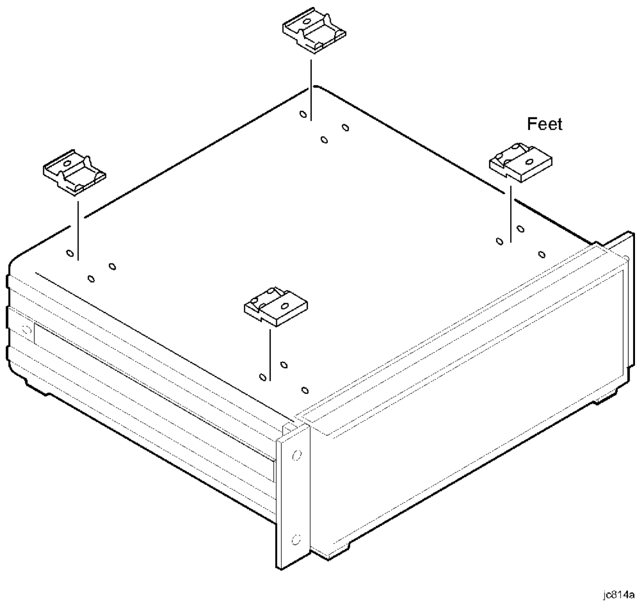
3. Attach the cabinet mount flanges and the handles to the sides of the front panel, using two long screws per side. (Attach the flanges to the outside of the handles.)



WARNING **If an instrument handle is damaged, you should replace it immediately. Damaged handles can break while you are moving or lifting the instrument and cause personal injury or damage to the instrument.**

---

4. Remove the feet before cabinet mounting the analyzer using the directions imprinted on the feet.

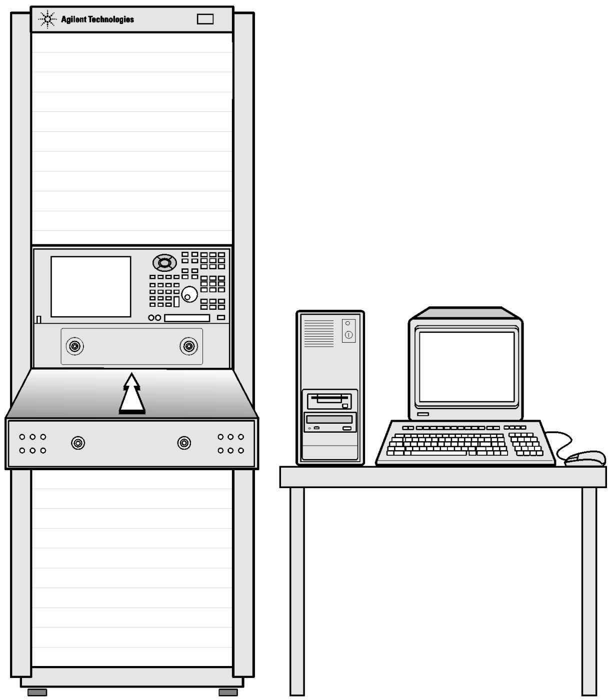


5. Ensure there is adequate clearance between the system cabinet and the sides and back of the test system for adequate ventilation.

CAUTION Consider the ventilation requirements described in "Step 5. Install the Test System on a Bench Top or in an Equipment Rack" on page 15 when selecting the location of your system.

---

6. Lift the test set and slide it onto the rails that you installed earlier from the front of the equipment rack. Secure the test set to the equipment rack using the screws with washers and metal-clipped nuts provided in the flange kit.



1inst_s4rackts

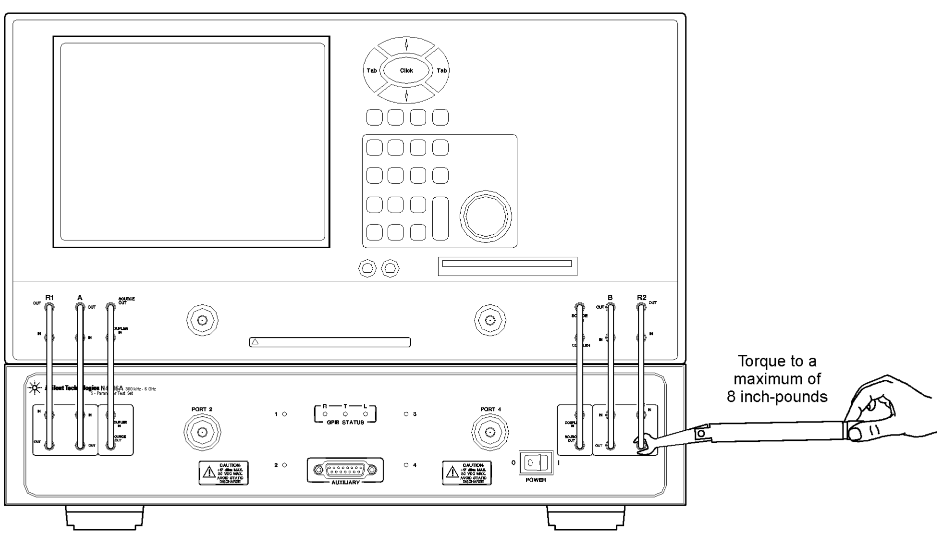
7. Continue with "Step 6. Make the Interconnections between the S-Parameter Test Set and the Network Analyzer" on page 21.

Step 6. Make the Interconnections between the S-Parameter Test Set and the Network Analyzer

1. Locate your system or test set and network analyzer listed below. Refer to the page indicated for information describing the interconnections between the test set and the network analyzer.

Test System Model Number	Test Set Model Number	Network Analyzer Model Number	Refer to:
N1957A/B	N4421A/B	E8364A/B	page 23
N1955B	N4420B	E8363A/B	page 25
N1953A/B	N4419A/B	E8362A/B	page 27
N1951A	N4418A	8720ES, 8722ES	page 29
N1948A	N4417A	E8356A, E8357A, E8358A	page 31
N1947A	N4417A	E8801A, E8802A, E8803A	page 33
N/A	N4416A	E8356A, E8357A, E8358A	page 35
N/A	N4415A	8753ES	page 37

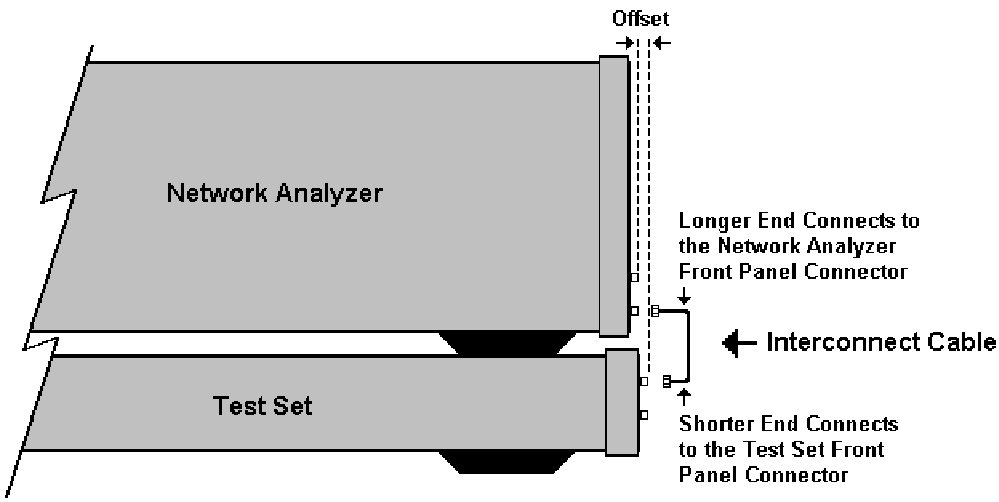
2. Using the illustration and table located on the page referenced above in step 1, connect the interconnect cables between the test set and the network analyzer. Torque the semirigid cable connectors to 8 inch-pounds.



1inst_s5icon

CAUTION When connecting the interconnect cables described in the remaining pages of this section, be careful to install the interconnect cables correctly. The longer end of the interconnect cable connects to the network analyzer front panel connector. Refer to Figure 1-1 for the correct orientation.

Figure 1-1 Interconnect Cable Orientation



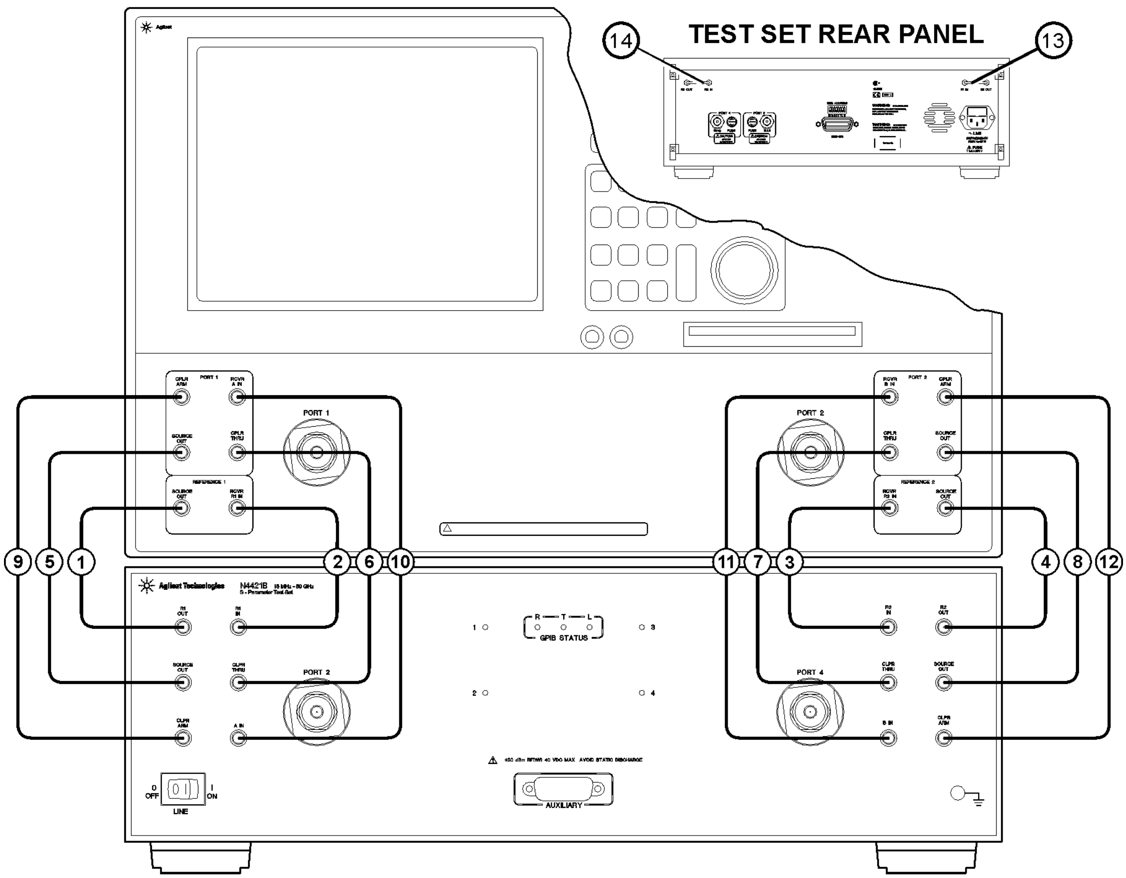
Damage to the interconnect cable can result from improper connection of the cable.

TIP If the test set and the network analyzer are rack mounted, the screws securing the rack mount flanges to the instrument rack may be loosened slightly to allow for minor repositioning of the instruments. Don't forget to retighten the screws when you are done.

3. Continue with "Step 7. Set Up the General Purpose Interface Bus (GPIB)" on page 39.

N1957A/B Test System Interconnections

(or N4421A/B Test Set with E8364A/B Network Analyzer)



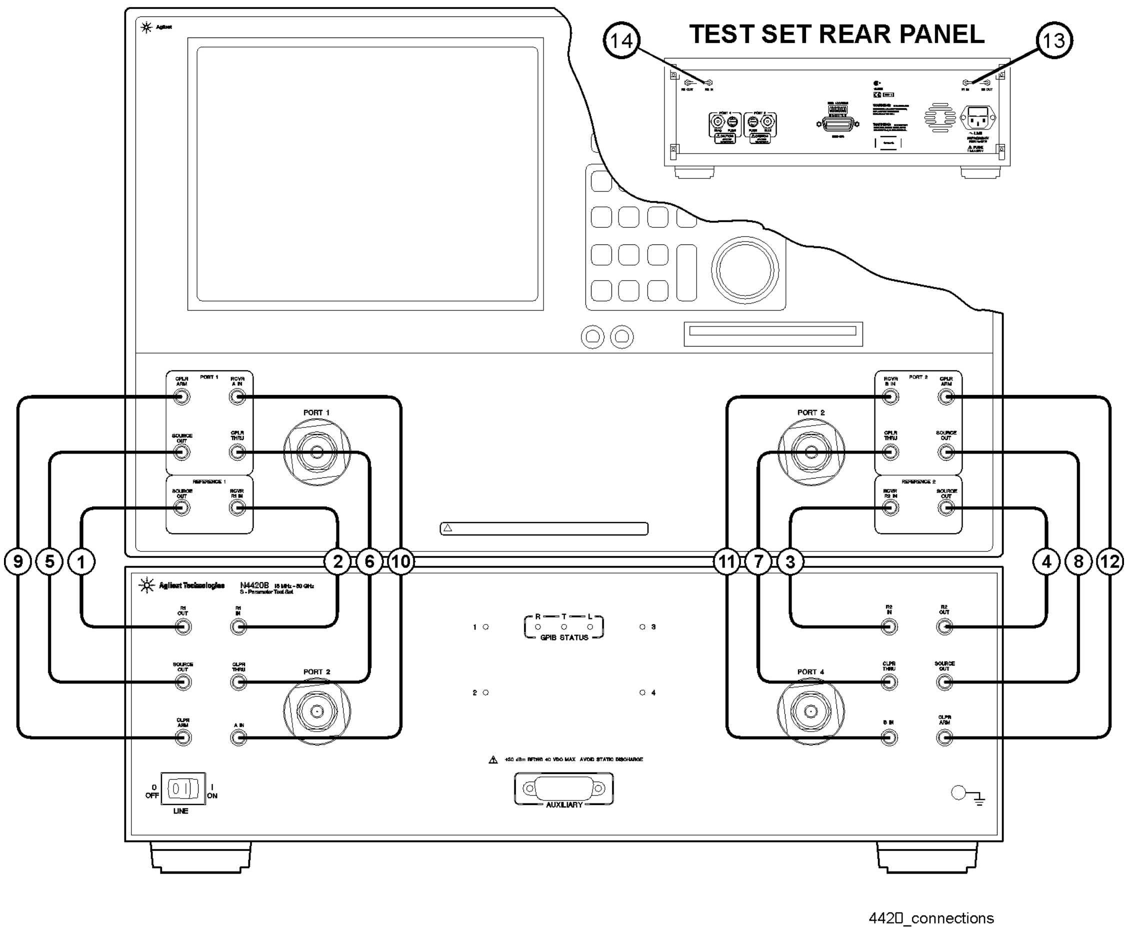
4421_connections

CAUTION Damage to the interconnect cable can result from improper orientation of the cable. Refer to page 22 for detailed information regarding the correct cable orientation.

Call Out Sequence	Cable Part Number	From E8364A/B	To N4421A/B
1	Z5623-20215	REF 1 SOURCE OUT	REF 1 R1 OUT
2	Z5623-20215	REF 1 RCVR R1 IN	REF 1 RCVR R1 IN
3	Z5623-20215	REF 2 RCVR R2 IN	REF 2 RCVR R2 IN
4	Z5623-20215	REF 2 SOURCE OUT	REF 2 R2 OUT
5	Z5623-20216	PORT 1 SOURCE OUT	PORT 1 SOURCE OUT
6	Z5623-20216	PORT 1 CPLR THRU	PORT 1 CPLR THRU
7	Z5623-20216	PORT 2 CPLR THRU	PORT 2 CPLR THRU
8	Z5623-20216	PORT 2 SOURCE OUT	PORT 2 SOURCE OUT
9	Z5623-20217	PORT 1 CPLR ARM	PORT 1 CPLR ARM
10	Z5623-20217	PORT 1 RCVR A IN	PORT 1 RCVR A IN
11	Z5623-20217	PORT 2 RCVR B IN	PORT 2 RCVR B IN
12	Z5623-20217	PORT 2 CPLR ARM	PORT 2 CPLR ARM
13	E8364-20059	REF 1 on rear panel of the test set	
14	E8364-20059	REF 2 on rear panel of the test set	

N1955B Test System Interconnections

(or N4420B Test Set with E8363A/B Network Analyzer)

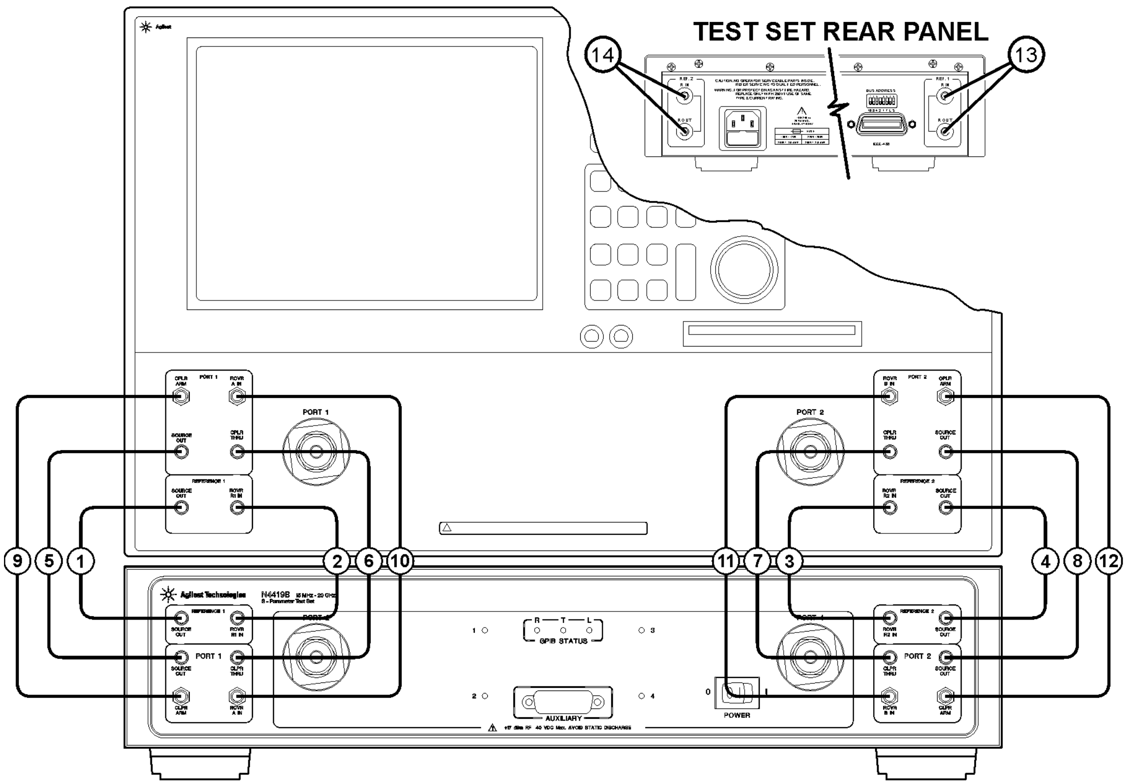


CAUTION Damage to the interconnect cable can result from improper orientation of the cable. Refer to page 22 for detailed information regarding the correct cable orientation.

Call Out Sequence	Cable Part Number	From E8363A/B	To N4420B
1	Z5623-20215	REF 1 SOURCE OUT	REF 1 R1 OUT
2	Z5623-20215	REF 1 RCVR R1 IN	REF 1 RCVR R1 IN
3	Z5623-20215	REF 2 RCVR R2 IN	REF 2 RCVR R2 IN
4	Z5623-20215	REF 2 SOURCE OUT	REF 2 R2 OUT
5	Z5623-20216	PORT 1 SOURCE OUT	PORT 1 SOURCE OUT
6	Z5623-20216	PORT 1 CPLR THRU	PORT 1 CPLR THRU
7	Z5623-20216	PORT 2 CPLR THRU	PORT 2 CPLR THRU
8	Z5623-20216	PORT 2 SOURCE OUT	PORT 2 SOURCE OUT
9	Z5623-20217	PORT 1 CPLR ARM	PORT 1 CPLR ARM
10	Z5623-20217	PORT 1 RCVR A IN	PORT 1 RCVR A IN
11	Z5623-20217	PORT 2 RCVR B IN	PORT 2 RCVR B IN
12	Z5623-20217	PORT 2 CPLR ARM	PORT 2 CPLR ARM
13	E8364-20059	REF 1 on rear panel of the test set	
14	E8364-20059	REF 2 on rear panel of the test set	

N1953A/B Test System Interconnections

(or N4419A/B Test Set with E8362A/B Network Analyzer)



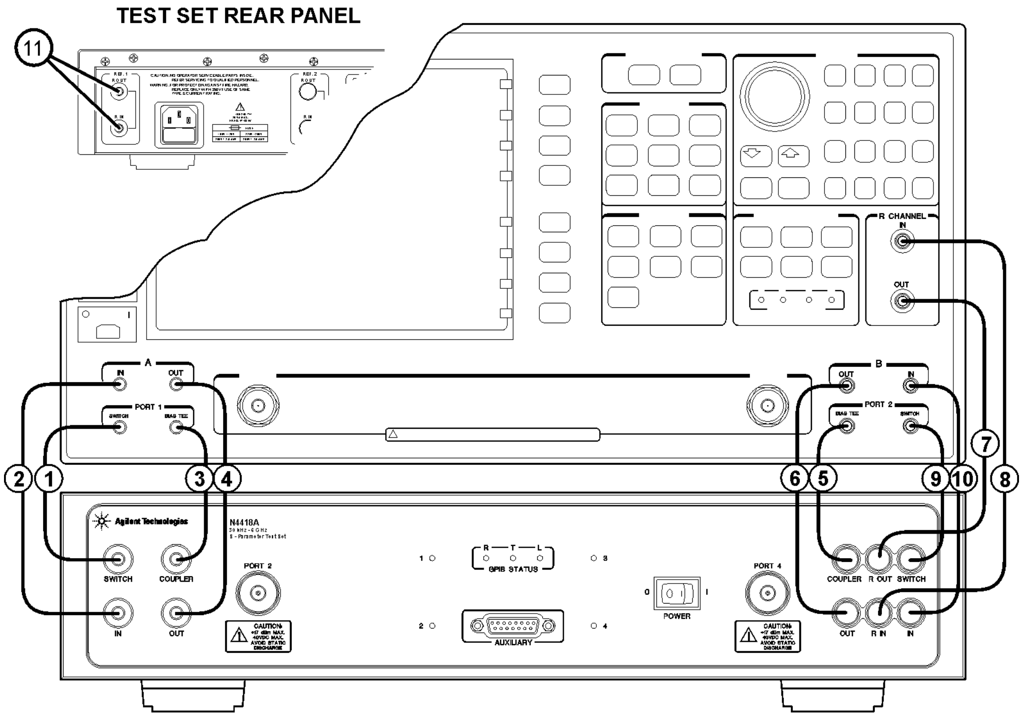
4419_connections

CAUTION	Damage to the interconnect cable can result from improper orientation of the cable. Refer to page 22 for detailed information regarding the correct cable orientation.

Call Out Sequence	Cable Part Number	From E8362A/B	To N4418A
1	AD00756-1	REF 1 SOURCE OUT	REF 1 SOURCE OUT
2	AD00756-1	REF 1 RCVR R1 IN	REF 1 RCVR R1 IN
3	AD00756-1	REF 2 RCVR R2 IN	REF 2 RCVR R2 IN
4	AD00756-1	REF 2 SOURCE OUT	REF 2 SOURCE OUT
5	AD00756-2	PORT 1 SOURCE OUT	PORT 1 SOURCE OUT
6	AD00756-2	PORT 1 CPLR THRU	PORT 1 CPLR THRU
7	AD00756-2	PORT 2 CPLR THRU	PORT 2 CPLR THRU
8	AD00756-2	PORT 2 SOURCE OUT	PORT 2 SOURCE OUT
9	AD00756-3	PORT 1 CPLR ARM	PORT 1 CPLR ARM
10	AD00756-3	PORT 1 RCVR A IN	PORT 1 RCVR A IN
11	AD00756-3	PORT 2 RCVR B IN	PORT 2 RCVR B IN
12	AD00756-3	PORT 2 CPLR ARM	PORT 2 CPLR ARM
13	AD00756-4	REF 1 on rear panel of the test set	
14	AD00756-4	REF 2 on rear panel of the test set	

N1951A Test System Interconnections

(or N4418A Test Set with 8720ES or 8722ES Network Analyzer)

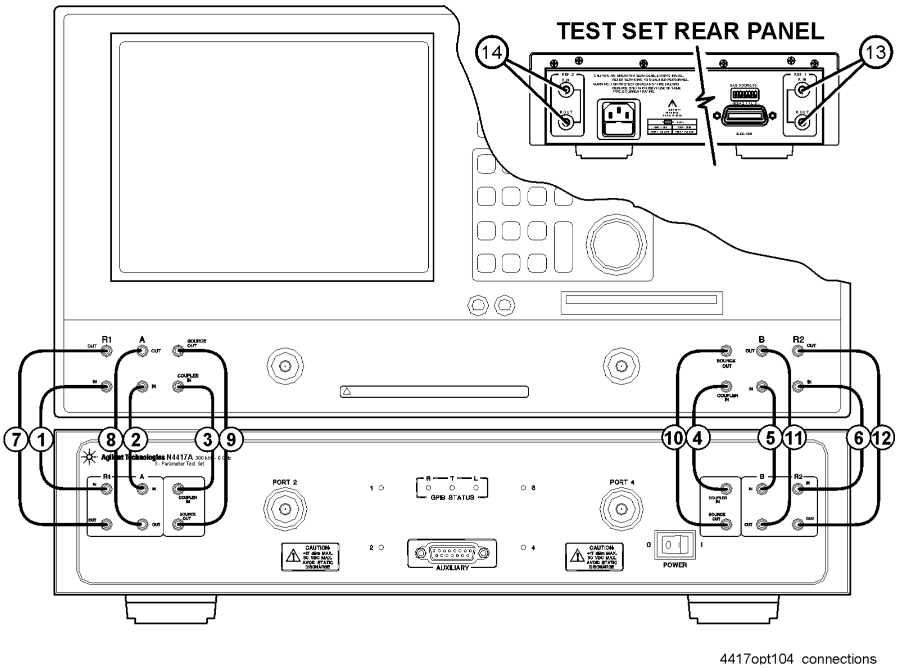


CAUTION Damage to the interconnect cable can result from improper orientation of the cable. Refer to page 22 for detailed information regarding the correct cable orientation.

Call Out Sequence	Cable Part Number	From 8720ES/8722ES	To N4418A
1	AD00599-2	PORT 1 SWITCH	PORT 1 SWITCH
2	AD00599-1	A IN	A IN
3	AD00599-2	PORT 1 BIAS TEE	PORT 1 COUPLER
4	AD00599-1	A OUT	A OUT
5	AD00599-2	PORT 2 BIAS TEE	PORT 2 COUPLER
6	AD00599-1	B OUT	B OUT
7	AD00599-4	R CHANNEL OUT	R OUT
8	AD00599-3	R CHANNEL IN	R IN
9	AD00599-2	PORT 2 SWITCH	PORT 2 SWITCH
10	AD00599-1	B IN	B IN
11	AD00599-5	REF 1 on rear panel of the test set	

N1948A Test System Interconnections

(or N4417A Option 104 Test Set with E8356A, E8357A, or E8358A Network Analyzer)

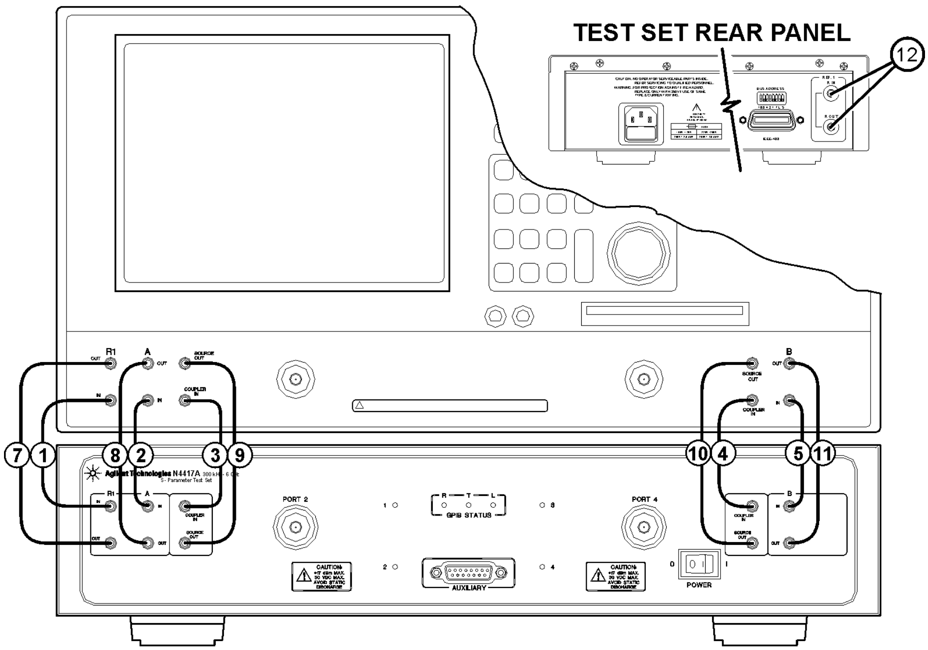


CAUTION Damage to the interconnect cable can result from improper orientation of the cable. Refer to page 22 for detailed information regarding the correct cable orientation.

Installation Sequence	**Cable Part Number**	**From E8356/E8357/E8358**	**To N4417A**
1	AD00747-1-2	R1 IN	R1 IN
2	AD00747-1-2	A IN	A IN
3	AD00747-1-2	COUPLER IN	COUPLER IN
4	AD00747-1-2	COUPLER IN	COUPLER IN
5	AD00747-1-2	B IN	B IN
6	AD00747-1-2	R2 IN	R2 IN
7	AD00747-1-1	R1 OUT	R1 OUT
8	AD00747-1-1	A OUT	A OUT
9	AD00747-1-1	SOURCE OUT	SOURCE OUT
10	AD00747-1-1	SOURCE OUT	SOURCE OUT
11	AD00747-1-1	B OUT	B OUT
12	AD00747-1-1	R2 OUT	R2 OUT
13	AD00747-1-3	REF 1 on rear panel of the test set	
14	AD00747-1-3	REF 2 on rear panel of the test set	

N1947A Test System Interconnections

(or N4417A Option 103 Test Set with E8801A, E8802A, or E8803A Network Analyzer)

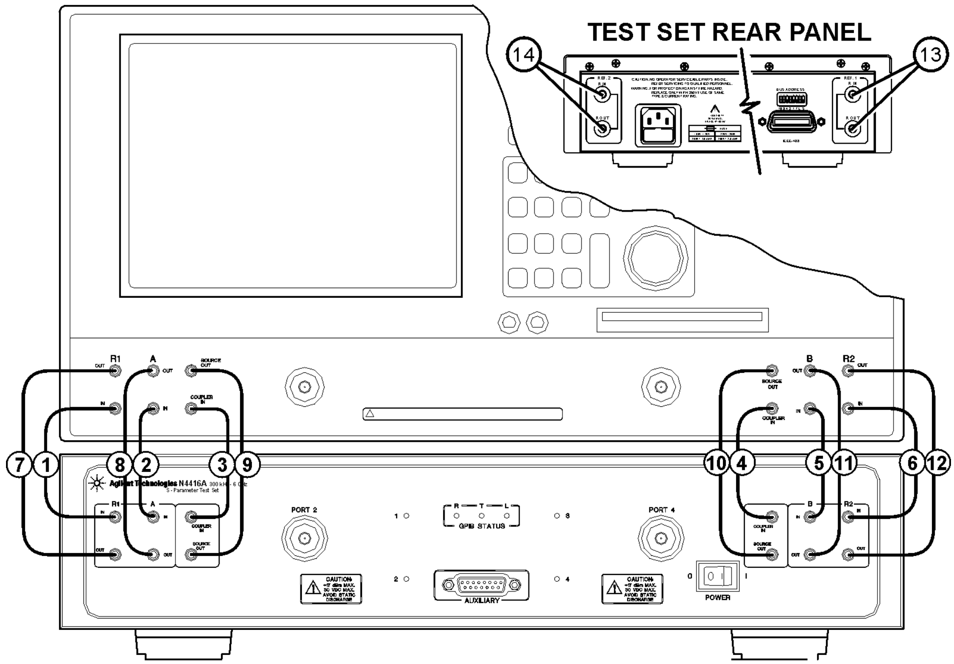


4417opt103_connections

CAUTION	Damage to the interconnect cable can result from improper orientation of the cable. Refer to page 22 for detailed information regarding the correct cable orientation.

Installation Sequence	Cable Part Number	From E8801/E8802/E8803	To N4417A
1	AD00747-2-2	R1 IN	R1 IN
2	AD00747-2-2	A IN	A IN
3	AD00747-2-2	COUPLER IN	COUPLER IN
4	AD00747-2-2	COUPLER IN	COUPLER IN
5	AD00747-2-2	B IN	B IN
6	AD00747-2-1	R1 OUT	R1 OUT
7	AD00747-2-1	A OUT	A OUT
8	AD00747-2-1	SOURCE OUT	SOURCE OUT
9	AD00747-2-1	SOURCE OUT	SOURCE OUT
10	AD00747-2-1	B OUT	B OUT
11	AD00747-2-3	REF 1 on rear panel of the test set	

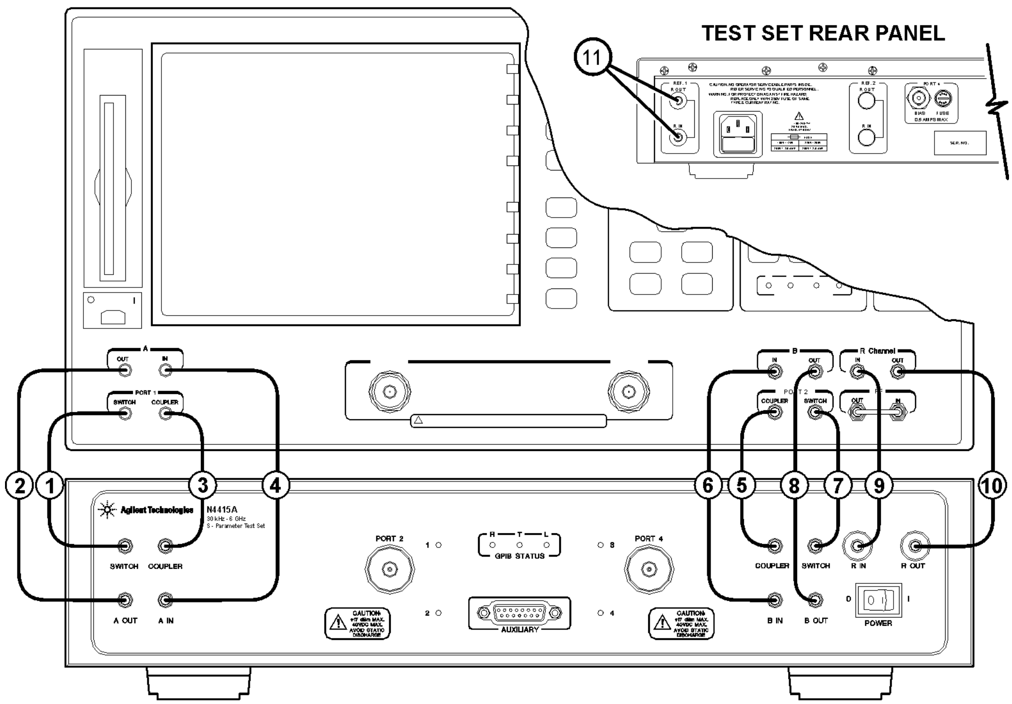
N4416A Test Set with the E8356A, E8357A, or E8358A Network Analyzer



CAUTION Damage to the interconnect cable can result from improper orientation of the cable. Refer to page 22 for detailed information regarding the correct cable orientation.

Installation Sequence	Cable Part Number	From E8356/E8357/E8358	To N4416A
1	AD00653-2	R1 IN	R1 IN
2	AD00653-2	A IN	A IN
3	AD00653-2	COUPLER IN	COUPLER IN
4	AD00653-2	COUPLER IN	COUPLER IN
5	AD00653-2	B IN	B IN
6	AD00653-2	R2 IN	R2 IN
7	AD00653-1	R1 OUT	R1 OUT
8	AD00653-1	A OUT	A OUT
9	AD00653-1	SOURCE OUT	SOURCE OUT
10	AD00653-1	SOURCE OUT	SOURCE OUT
11	AD00653-1	B OUT	B OUT
12	AD00653-1	R2 OUT	R2 OUT
13	AD00653-3	REF 1 on rear panel of the test set	
14	AD00653-3	REF 2 on rear panel of the test set	

N4415A Test Set with the 8753ES Network Analyzer



4415_frtpnl_connections

CAUTION Damage to the interconnect cable can result from improper orientation of the cable. Refer to page 22 for detailed information regarding the correct cable orientation.

Installation Sequence	Cable Part Number	From 8753ES	To N4415A
1	AD00632-2	Port 1 Switch	Switch
2	AD00632-1	A OUT	A OUT
3	AD00632-2	Port 1 Coupler	Coupler
4	AD00632-1	A IN	A IN
5	AD00632-2	Port 2 Coupler	Coupler
6	AD00632-1	B IN	B IN
7	AD00632-2	Port 2 Switch	Switch
8	AD00632-1	B OUT	B OUT
9	AD00632-3	R Channel In	R IN
10	AD00632-3	R Channel Out	R OUT
11	AD00632-4	REF 1 on rear panel of the test set	

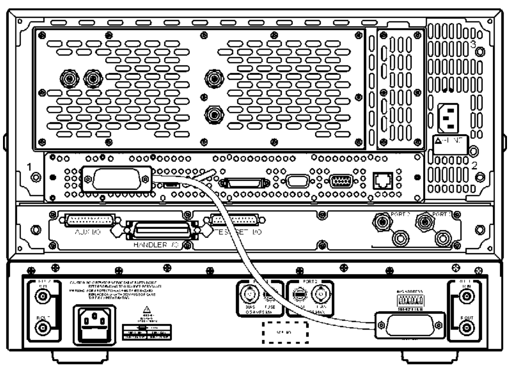
Step 7. Set Up the General Purpose Interface Bus (GPIB)

The PC uses the General Purpose Interface Bus (GPIB) to communicate with the test system hardware. The PLTS software will locate and identify your test system equipment automatically. Each test system device must have a unique GPIB address.

NOTE There are 32 GPIB addresses, numbered 0 to 31. However, there may be the occasion that you need to change the GPIB address for test equipment. GPIB addresses are set either using rear panel switches or using the equipment firmware. Refer to "Setting Up the General Purpose Interface Bus Manually" on page 597 for more information.

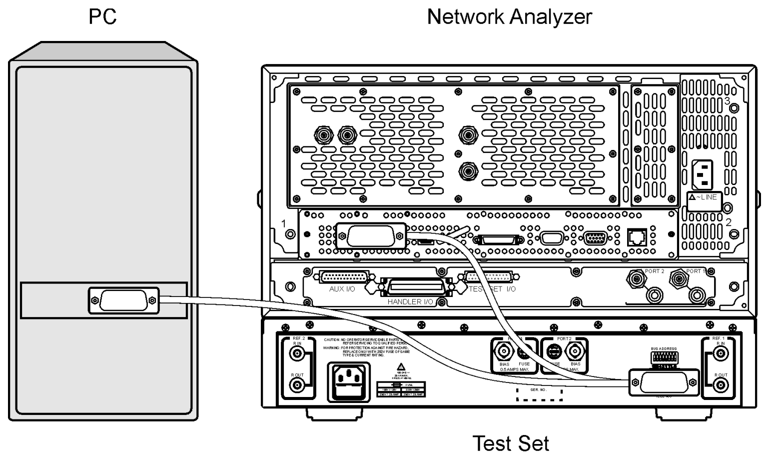
1. Connect a GPIB cable from the rear panel GPIB connector on the network analyzer to the rear panel GPIB connector on the test set.

Network Analyzer



Test Set

2. Connect a second GPIB cable from the PC GPIB card's connector to the GPIB connector on either end of the first cable that was connected.



Step 8. Power up the S-Parameter Test Set

1. Ensure the available ac power supply meets the Power Source Requirements and the operating environment meets the Operating Environment Requirements listed below.

Power Source Requirements			
Input Voltage Range	100 – 120 Vac	- or -	220 – 250 Vac
Frequency Range	47 – 62 Hz / 400 Hz		
Power	40 VA maximum.		
Operating Environment Requirements			
Operating Environment	Indoor use		
Altitude	Operating: Storage:	0 to 2.0 km (6,560 ft.) 0 to 15.24 km (50,000 ft.)	
Temperature	0 °C to 40 °C		
Maximum Relative Humidity	80% for temperatures up to 31 °C; decreasing linearly to 50% for a temperature of 40 °C		

Refer to Chapter 15, “Specifications and Characteristics,” for the complete specifications.

2. Verify that the ac power cable is not damaged, and that the power-source outlet provides a protective earth contact.

CAUTION Always use the three-prong ac power cord supplied with this product. Failure to ensure adequate earth grounding by not using this cord may cause product damage.

3. Turn off the PC and the network analyzer.
4. Connect the ac power cable from the power-source outlet to the ac input on the rear panel of the test set.
5. Turn on the PC, the network analyzer, and the test set by pressing the ON/OFF button on the front panel of each device.

6. If your network analyzer is an N8362A/B, N8363A/B, or N8364A/B, you will need to perform the Phase-Lock IF Gain Adjustment after it has been connected to the test set. This routine adjusts the R Channel receivers ALC gain to ensure phase lock over the entire frequency range. Refer to **Phase-Lock IF Gain Adjustment** in the network analyzer's online help system for details. Use the following steps to perform this adjustment:

   a. On the PNA, from the **System** menu, click **Service**, then **Adjustments**, then **IF Gain Adjustment**.

      If you are unable to find these selections on your N836XA PNA, your analyzer firmware is a revision prior to 3.0. Follow the instructions in "IF Gain Adjustment" on page 602 to complete this adjustment.

   b. Select any special test set options installed.

   c. No connections to the test ports are required.

   d. Click **Begin Adj**. The adjustment takes about a minute to complete.

      The advanced screen is for factory personnel only.

NOTE This adjustment must be performed before using the network analyzer each time the system is assembled or disassembled.

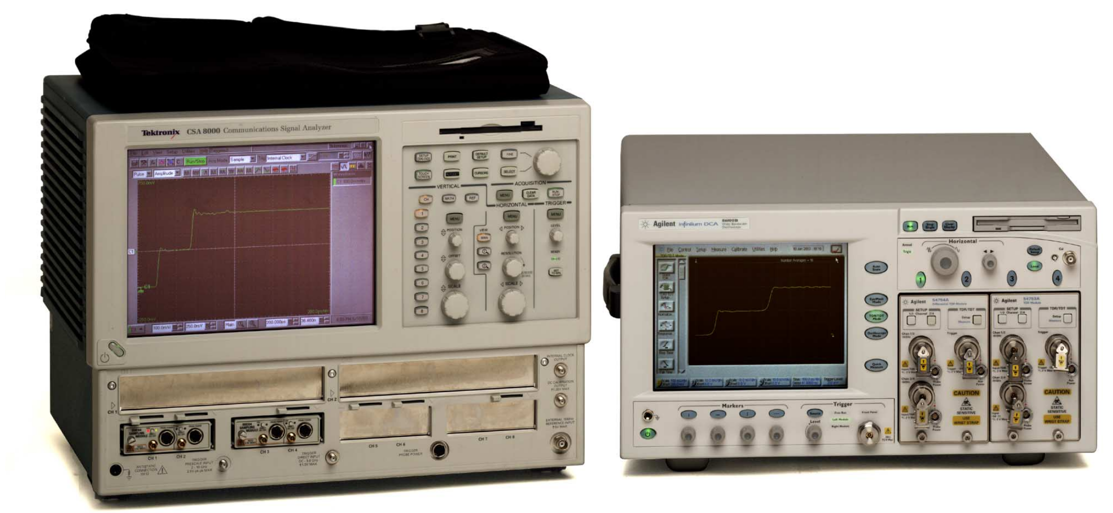
7. The PLTS hardware installation is complete. Continue with the PLTS software installation by referring to Chapter 3, "Installing the Physical Layer Test System Software," on page 51.

2 Installing the TDR-Based Physical Layer Test System Hardware

To make time domain measurements using the Physical Layer Test System (PLTS) software and a TDR system, you need the following equipment:

- Personal computer (PC)
- PLTS software
- TDR-based PLTS hardware (one of the following systems)
  - — Agilent 86100A/B Infiniium DCA Wide-Bandwidth Oscilloscope equipped with two 54754A Differential 18 GHz TDR/TDT Plug-in Modules using Firmware Revision 03.06 or greater
  - — Tektronix CSA8000 Communications Signal Analyzer equipped with two 80E04 Dual Channel, 20 GHz TDR Sampling Modules using Firmware Revision 1.3.3 or greater
  - — Tektronix TDS8000 Digital Sampling Oscilloscope equipped with two 80E04 Dual Channel, 20 GHz TDR Sampling Modules using Firmware Revision 1.3.3 or greater

Figure 2-1 TDR-based Physical Layer Test System Hardware: Tektronix CSA8000 (left) and Agilent 86100A/B DCA (right)



This installation procedure leads you through setting up the hardware (the PC and the TDR-based Physical Layer Test System). After you complete this installation, you will refer to Chapter 3, Installing the Physical Layer Test System Software, to install the software.

NOTE If you have the VNA-based Physical Layer Test System, refer to Chapter 1, "Installing the VNA-Based Physical Layer Test System Hardware," on page 3 for instructions on setting up that system.

The following is a list of the installation steps to set up your TDR system hardware:

- Step 1. Set Up the Personal Computer
- Step 2. Set Up the TDR System
- Step 3. Set Up the GPIB
- Step 4. Power up the TDR System

NOTE These installation instructions were written specifically for customers who have just received their TDR system with their PLTS software. If you have already been using your TDR system, you have probably completed most of these installation steps. Briefly review installation these steps to ensure that your system is currently set up as recommended. Then, begin the software installation process by starting at Chapter 3, "Installing the Physical Layer Test System Software," on page 51.

Step 1. Set Up the Personal Computer

1. Make sure that your PC meets the following minimum system controller requirements:

Table 2-1 Minimum PC Requirements by PLTS Modes of Operation

PC Requirement	Measurement Controller Mode	Off-Line Analysis Mode
	In the lab, controlling test equipment and making quick analysis of the results	In your office, performing "What if..." analysis, characterization, cross-domain analysis, filtering, waveform math, and eye diagram simulation
CPU	400 MHz Pentium II or greater	1 GHz Pentium III compatible PC
Main Memory	256 MB[a]	512 MB
Virtual Memory[b]	512 MB	768 MB
GPIB Interface	Agilent 82357A USB/GPIB Interface for Windows or supported GPIB card (any National Instruments or Agilent 82340/41 or 82350 GPIB card)	No GPIB connection is required to utilize PLTS in the off-line mode. Saved (stored) measurement files can be recalled at any time for analysis.
Operating Systems	Windows 2000 or Windows XP[c]	
Screen Resolution	1024×768	

a. 512 MB of Main Memory is recommended for the Measurement Controller Mode when the measurement is measuring 16,000 points with the PNA B-model network analyzer.
b. As a general rule for optimum PC performance when using PLTS, virtual memory should be 1.5 to 2 times the size of the main memory.
c. Earlier versions of Windows are no longer supported by PLTS.

NOTE Memory, both main and virtual, is critical to using PLTS effectively. There is no substitution for not having enough memory. As more applications are added to the PC, more memory is used. If your PC needs more memory, we suggest you take the time to remove unused programs.

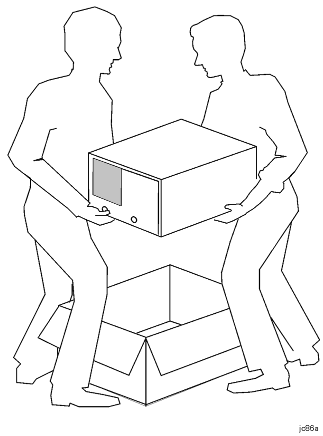
2. Using the PC documentation, make sure that the PC is operating properly.

3. Make sure the GPIB card is installed in the PC and that it is operating properly.

4. Make sure the PC is located near where you will position the oscilloscope-based TDR system. Later in this process, you will connect the GPIB card to the TDR system using a GPIB cable.

Step 2. Set up the TDR System

1. Unpack your system from the containers in which it was shipped.



WARNING **The TDR system hardware can be heavy. Use proper lifting techniques. Refer to the TDR system's documentation for information regarding the equipment weight.**

2. Carefully inspect the system to make sure that it was not damaged during shipment.

NOTE If your TDR system was damaged during shipment, refer to the system's documentation to contact the manufacturer. If the manufacturer is Agilent, refer to "Contacting Agilent" on page 552.

3. Using the TDR system's documentation, set up the system as instructed, ensuring that the system's permanent location is near the PC that was set up in Step 1 on page 46.

CAUTION Both the Agilent and Tektronix TDR systems, although not required, may be connected to a computer network. Connecting to a computer network may present security risks to your TDR system. For the Agilent TDR system, refer to "Embedded Operating System Risk" on page iv for additional information. For the Tektronix TDR systems, refer to your system documentation for additional information.

4. Ensure that your TDR system is one of the following:

   - Agilent 86100A/B Infiniium Digital Communications Analysis Wide-Bandwidth Oscilloscope with:

     — Firmware revision 03.06 (or later)
     — 1 or 2 Agilent 54754A 18 GHz Differential TDR/TDT Plug-In Modules installed

   - Tektronix CSA8000 Communications Signal Analyzer Oscilloscope with:

     — Firmware revision 1.3.3 (check with your Tektronix representative for firmware)
     — 1 or 2 Tektronix 80E04 Dual Channel, 20 GHz TDR Sampling Modules installed in slot 1/2 and/or slot 3/4 only (no support for channels 5, 6, 7, or 8)

   - Tektronix TDS8000 Digital Sampling Oscilloscope with:

     — Firmware revision 1.3.3 (check with your Tektronix representative for firmware)
     — 1 or 2 Tektronix 80E04 Dual Channel, 20 GHz TDR Sampling Modules installed in slot 1/2 and/or slot 3/4 only (no support for channels 5, 6, 7, or 8)

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.

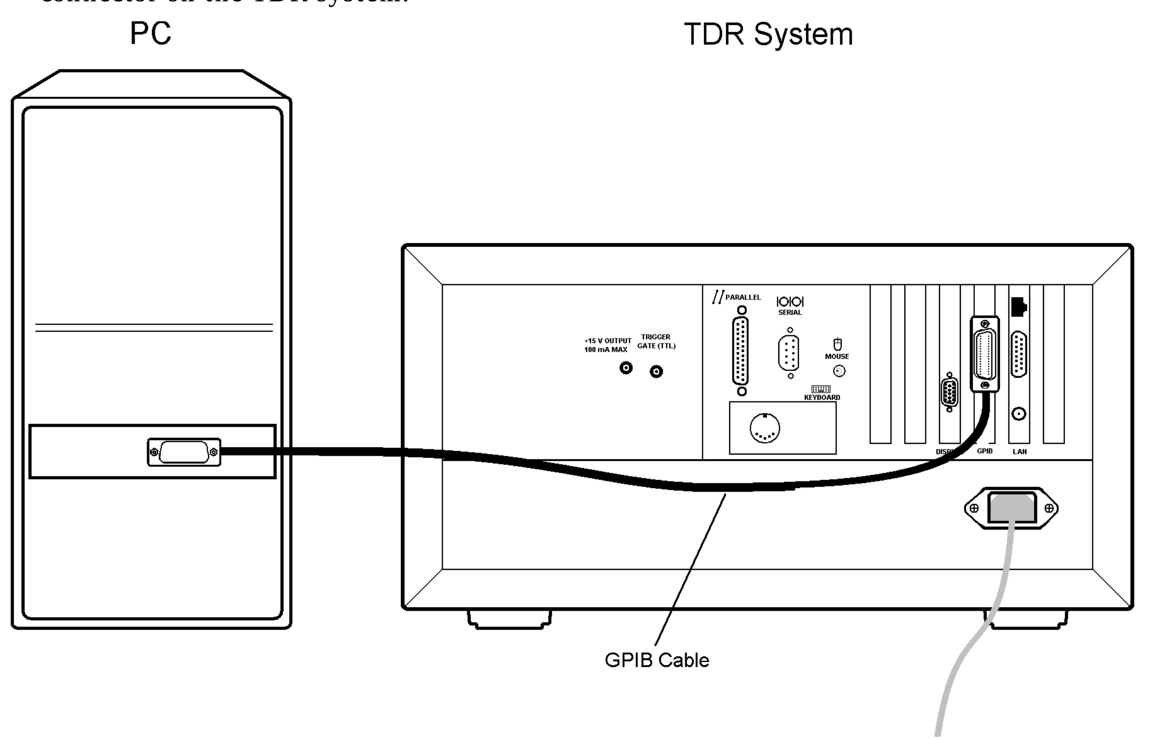
Step 3. Set Up the General Purpose Interface Bus (GPIB)

The PC uses the General Purpose Interface Bus (GPIB) to communicate with the test system hardware. The PLTS software will locate and identify your test system equipment automatically. Each test system device must have a unique GPIB address.

Check the GPIB address of your Agilent TDR system by selecting **Remote Interface** from the **Utilities** menu on the TDR display. To check the GPIB address of your Tektronix TDR system, refer to the programming manual for information.

NOTE There are 32 GPIB addresses, numbered 0 to 31. However, there may be the occasion that you need to change the GPIB address for test equipment. GPIB addresses are set using the equipment firmware. Refer to “Setting Up the General Purpose Interface Bus Manually” on page 597 for more information.

1. Connect a GPIB cable from the PC GPIB card's connector to the rear-panel GPIB connector on the TDR system.



Step 4. Power up the TDR System

If you have not previously powered on your TDR system, start with step 1. If you have already powered on your TDR system, just review steps 1, 2, and 3 before continuing with step 4.

1. Ensure the available ac power supply meets the power source requirements and the operating environment meets the operating environment requirements for the TDR system. Refer to the TDR system documentation for the environmental specifications.
2. Verify that the ac power cable is not damaged, and that the power-source outlet provides a protective earth contact.

CAUTION Always use the three-prong ac power cord supplied with this product. Failure to ensure adequate earth grounding by not using this cord may cause product damage.

3. Connect the ac power cable from the power-source outlet to the ac input on the rear panel of the test set.
4. Turn on the PC and the TDR system.
5. The TDR system hardware installation is complete. Continue with the PLTS software installation by referring to Chapter 3, “Installing the Physical Layer Test System Software,” on page 51.

3 Installing the Physical Layer Test System Software

This chapter guides you through installing the Physical Layer Test System (PLTS) software. The software can be used 1) in the lab to control test equipment making measurements and make a quick analysis of the results, and it can be used 2) in your office to perform “What if...” analysis, characterization, cross-domain analysis, filtering, waveform math, and eye diagram simulation.

- If you are making measurements in the lab, you should have already installed one of the following hardware systems:
  - — VNA-based system (Network analyzer and S-parameter test set) described in Chapter 1, “Installing the VNA-Based Physical Layer Test System Hardware”
  - — TDR-based system (Agilent TDR or Tektronix TDR) described in Chapter 2, “Installing the TDR-Based Physical Layer Test System Hardware”

NOTE Steps 1 and 2 of this chapter were performed during the hardware installation procedures described in Chapter 1 and Chapter 2. You may skip these steps if you already performed them.

- If you are performing analysis in your office, you will only need a PC to use the software.

This software installation chapter will lead you through a series of steps to install the PLTS software. The following is a list of the installation steps:

- Step 1. Set Up the Personal Computer
  (Skip this step if you already performed it during hardware installation.)
- Step 2. Verify your System Shipment
  (Skip this step if you already performed it during hardware installation.)
- Step 3. Install the Physical Layer Test System Software
- Step 4. License the Physical Layer Test System Software
- Step 5. Start the Physical Layer Test System Software
- Step 6. Familiarize Yourself with the PLTS Software Screen

Step 1. Set Up the Personal Computer

This procedure describes the steps that need to be performed if you plan to use PLTS in your office, performing "What if..." analysis, characterization, cross-domain analysis, filtering, waveform math, and eye diagram simulation and you ***do not*** plan to use this software to control test equipment and make measurements.

NOTE If you plan to control test equipment with the software, refer to "Set Up the Personal Computer" step in the appropriate chapter, either Chapter 1, "Installing the VNA-Based Physical Layer Test System Hardware," or Chapter 2, "Installing the TDR-Based Physical Layer Test System Hardware."

1. Make sure that your PC meets the following minimum system controller requirements:

Table 3-1 Minimum PC Requirements for Off-Line Analysis Mode of Operation

CPU	1 GHz Pentium III compatible PC
Main Memory	512 MB
Virtual Memory[a]	768 MB
GPIB Interface	No GPIB connection is required to utilize PLTS in the off-line mode. Saved (stored) measurement files can be recalled at any time for analysis.
Operating Systems	Windows 2000 or Windows XP[b]
Screen Resolution	1024 × 768

a. As a general rule for optimum PC performance when using PLTS, virtual memory should be 1.5 to 2 times the size of the main memory.
b. Earlier versions of Windows are no longer supported by PLTS.

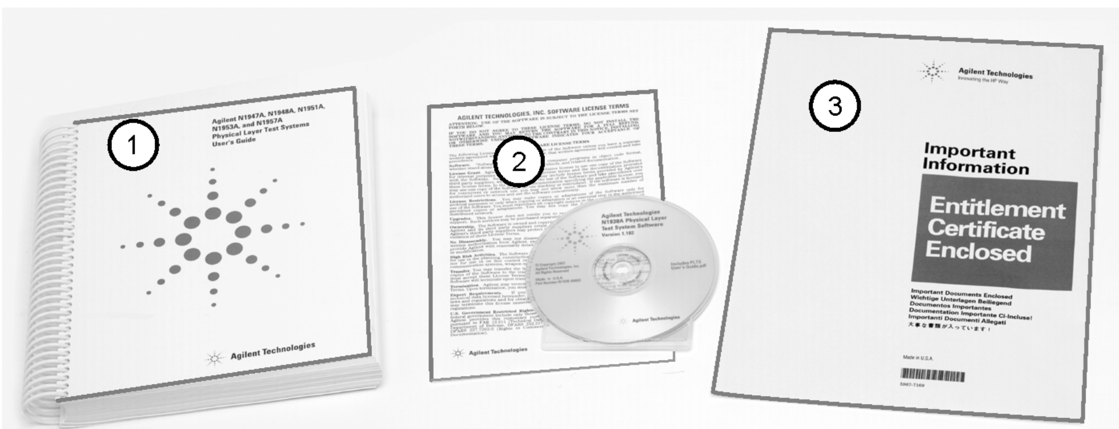
2. Using the PC documentation, make sure that the PC is operating properly.

Step 2. Verify your System Shipment

NOTE This procedure describes the steps that need to be performed if you only purchased the N1930A PLTS Software.

If you purchased a PLTS with the software, refer to “Verify your System Shipment” step in the appropriate chapter, either Chapter 1, “Installing the VNA-Based Physical Layer Test System Hardware.”

1. Unpack and carefully inspect the package to make sure that it was not damaged during shipment.



Item Nr	Part Nr	Part Description
1	N1930-90003	User’s Guide (this document)
2	N1930A	Physical Layer Test System Software CD-ROM (in envelope with Agilent Software License Terms printed on exterior)
3	N/A	Software Entitlement Certificate (in envelope)
Not Shown	N/A	Agilent 86100 firmware upgrade on LM-120 and CD-ROM

NOTE If the contents of your package were damaged during shipment, contact Agilent Technologies. Refer to “Contacting Agilent” on page 552.

Step 3. Install the Physical Layer Test System Software

If you are upgrading your computer with a new version of PLTS, refer to "Installation of PLTS as a Software Upgrade" on page 66.

In addition to loading PLTS files, PLTS installs three other software packages that it requires. As part of the installation, PLTS also installs:

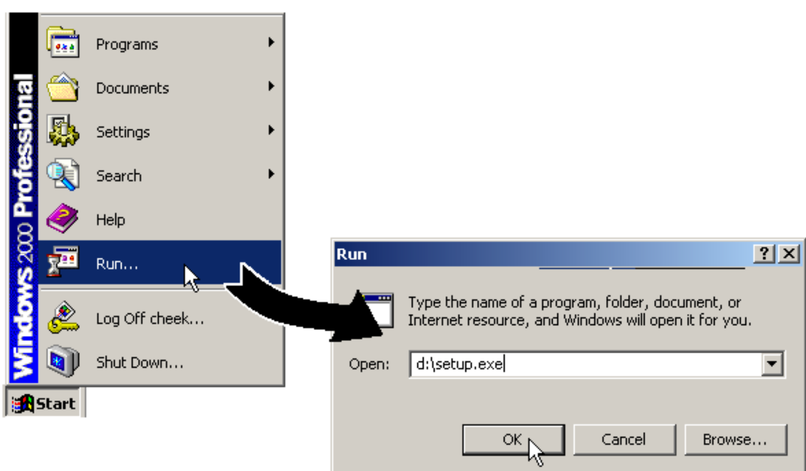
- Microsoft .NET Framework
- Agilent T&M Programmers Toolkit Redistributable Package
- Agilent IO Libraries

NOTE If you already have of these software packages installed on your PC, you will be asked if you want to upgrade to this version. If this version is newer than your version, please upgrade your software.

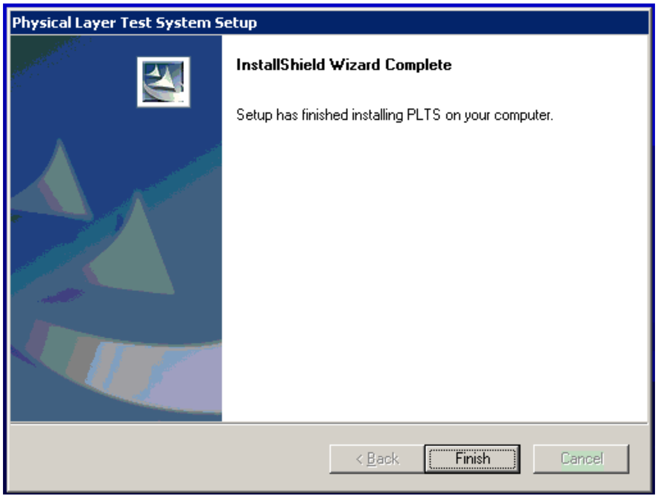
However, if your version is newer than the version PLTS wants to install, please decline to load the PLTS version.

1. Insert the PLTS software CD-ROM in your PC. The software should begin the installation process automatically.

   If the software does not start the automatic installation process, select **Start > Run...** in Windows as shown below. With the *Run* dialog box displayed, enter your CD-ROM drive letter followed by "\:setup.exe". For example: `D\:setup.exe.` Then click **OK** and follow the onscreen instructions

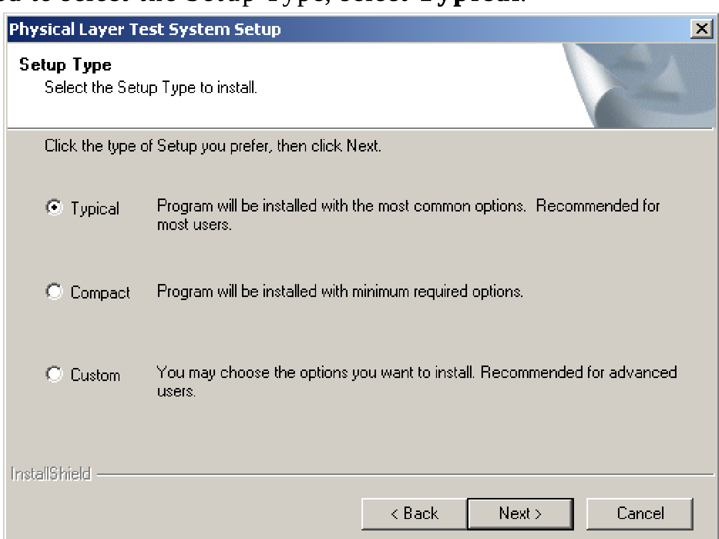


2. After the InstallShield Wizard extracts the PLTS files, the InstallShield Wizard welcome screen is displayed.



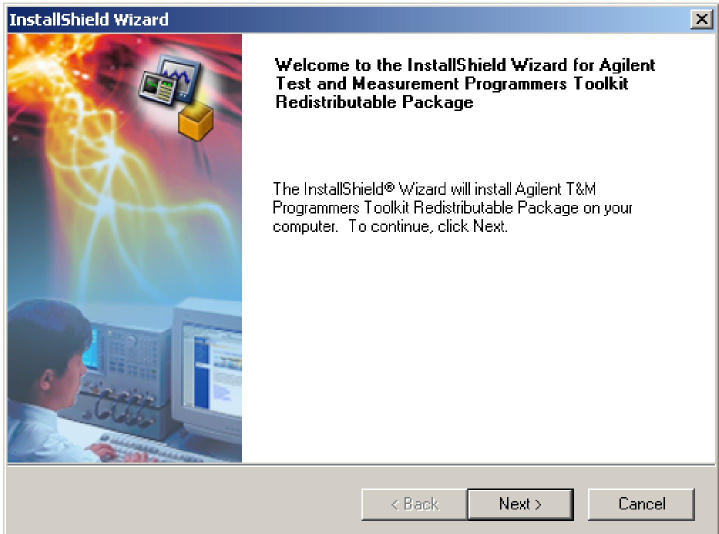
3. Before installing PLTS, a license agreement is displayed, read the agreement. You will be asked if you accept the terms of the license agreement. If you click No, the PLTS setup will close. To install PLTS, you must accept this agreement by clicking Yes.

4. When asked to select the Setup Type, select **Typical**.

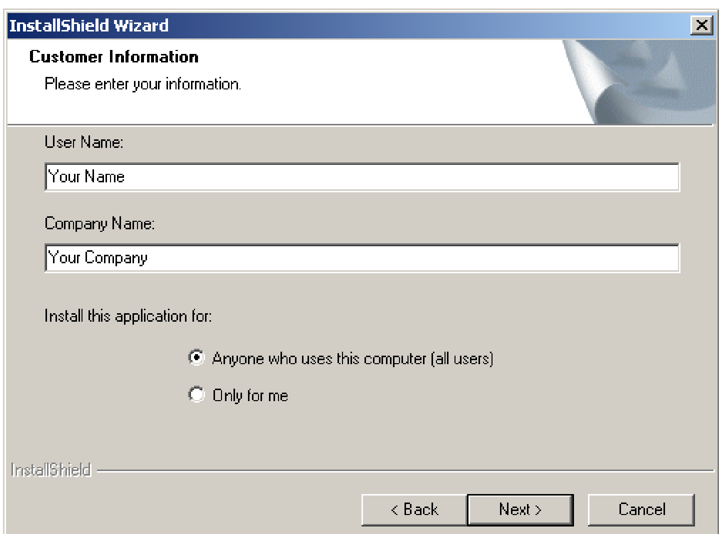


After checking the settings, the installation procedure begins installing PLTS program files.

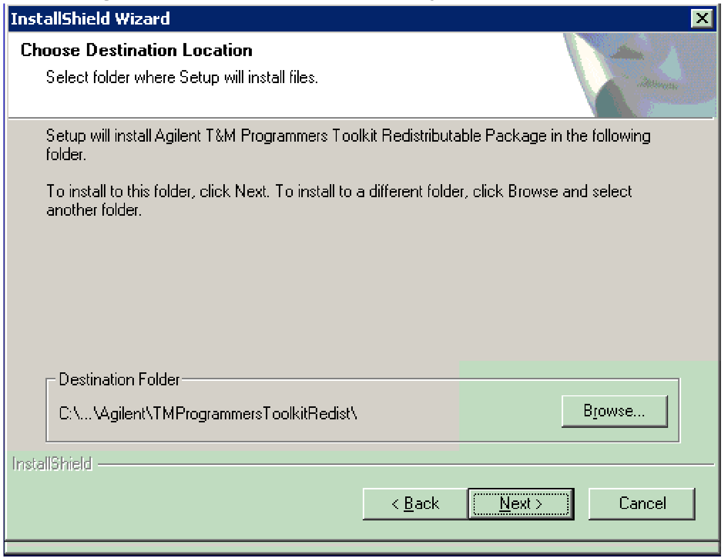
5. When the *Agilent Test and Measurement Programmers Toolkit Redistributable Package Welcome* screen is displayed, click **Next >** to be guided through the installation process.



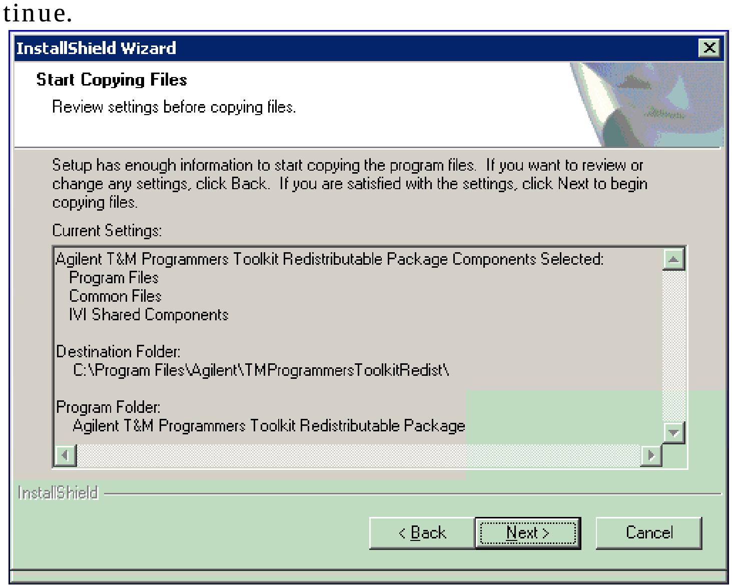
6. You will be asked to enter your name and your company's name. We recommend that you select to install this application for anyone who uses the computer.



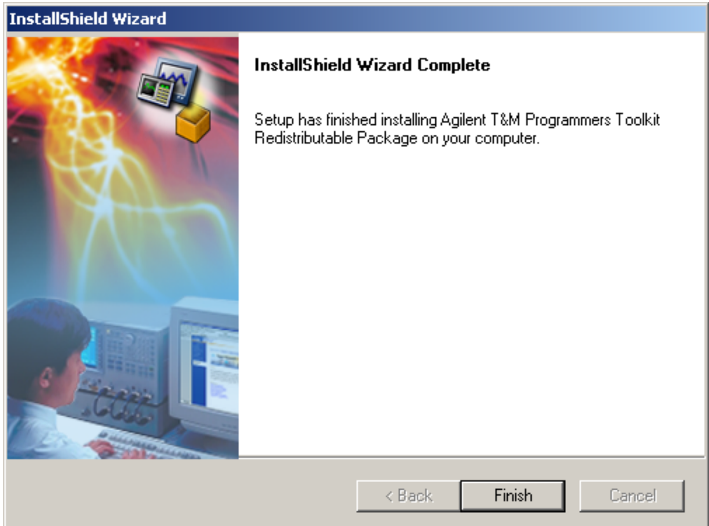
7. When asked to choose a destination location for Agilent T&M Programmers Toolkit Redistributable Package, use the default directory. Click **Next >** to continue.



8. The installation wizard asks you to review the settings. Use the default settings. Click **Next >** to continue.

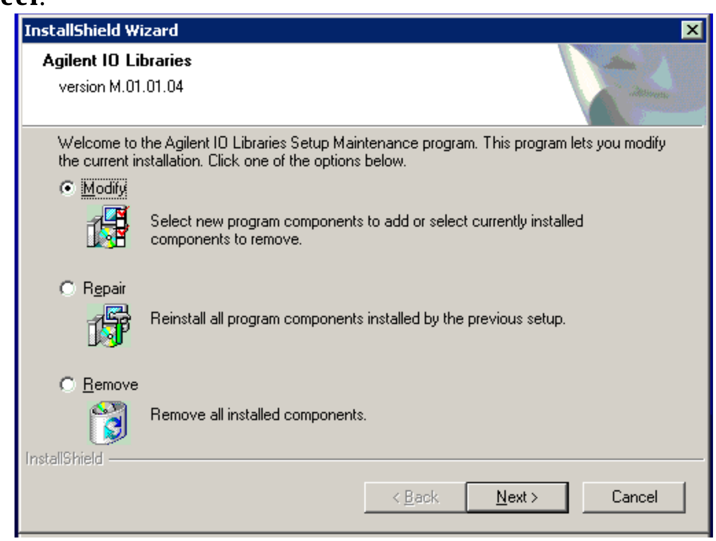


9. The installation process guides you through the T&M Programmers Toolkit Redistributable Package installation process until the installation has finished.

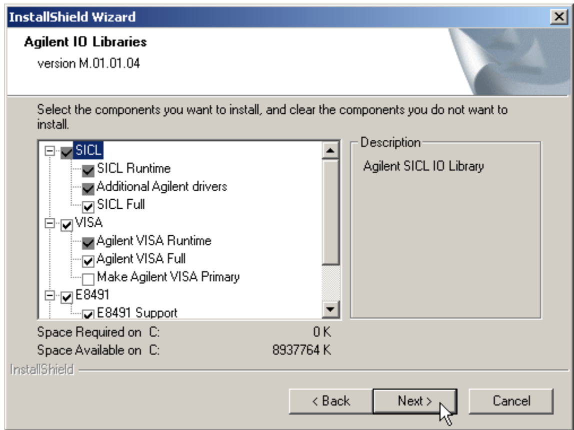


When you click **Finish**, the InstallShield Wizard extracts the files needed to load the Agilent IO Libraries.

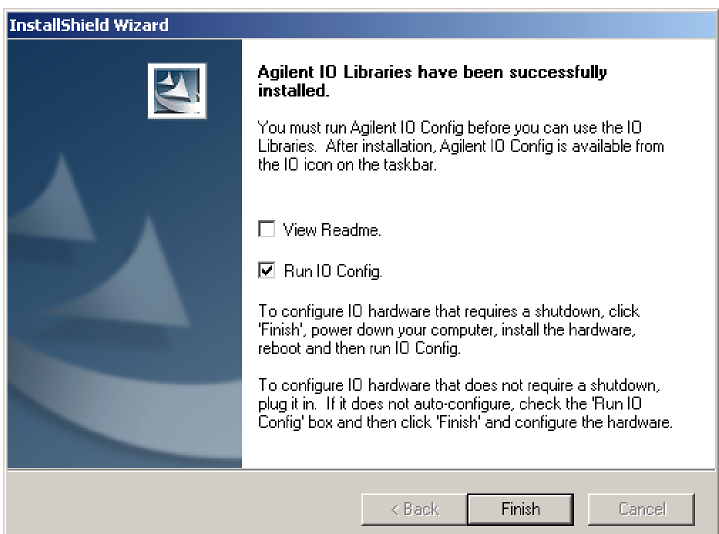
10. You will be asked if you want to upgrade Agilent IO Libraries to this version. If this version is newer than your version, please upgrade your software.However, if your version is newer than the version PLTS wants to install, please decline to load the PLTS version by selecting **Cancel**.



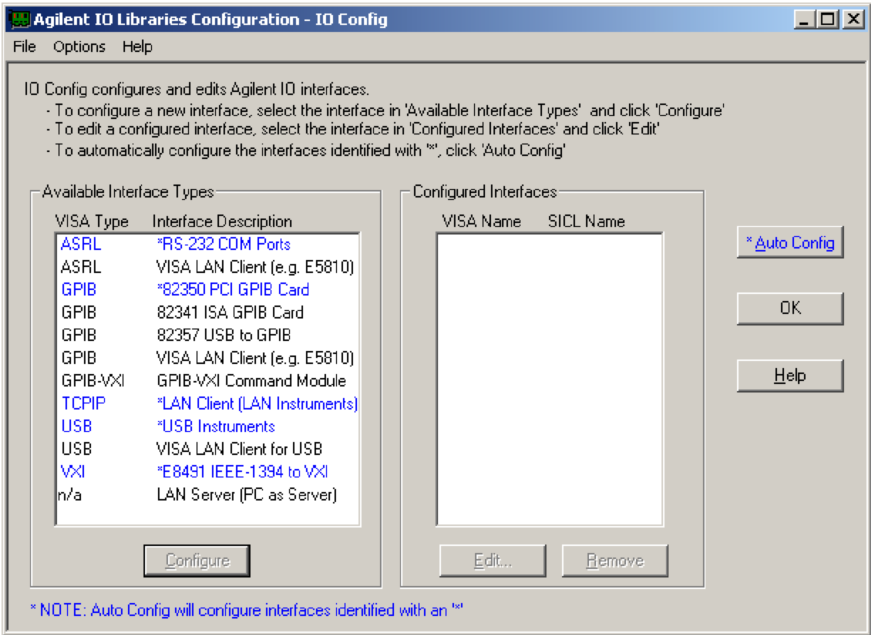
11. If the following Agilent IO Libraries screen is displayed, leave the selections unchanged and select the **Next >** button.



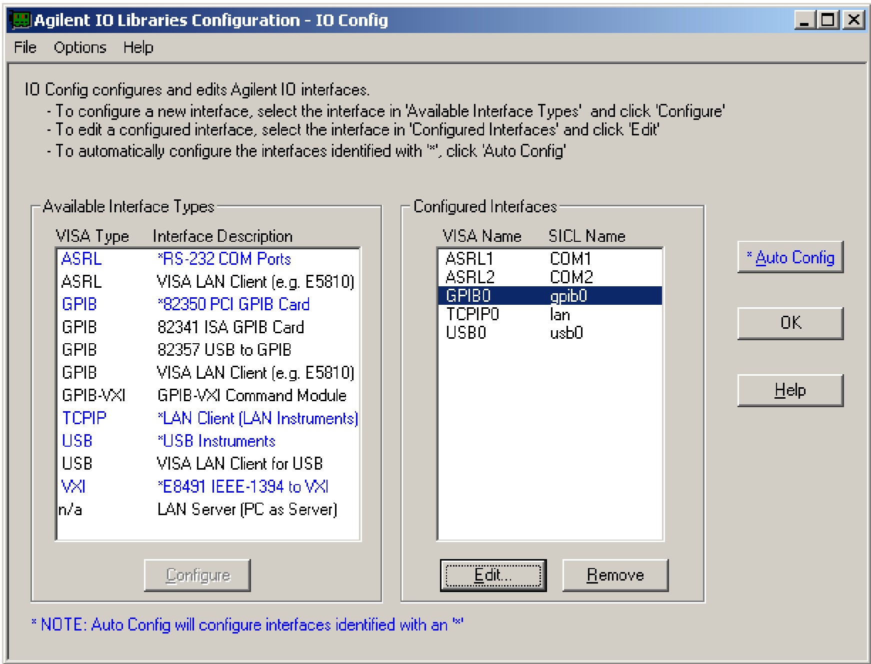
12. When the InstallShield Wizard notifies you that the Agilent IO Libraries has been installed, select the **Run IO Config** check box and click **Finish** to configure the IO.



13. The *Agilent IO Libraries Configuration - IO Config* screen is displayed. Select the **Auto Config** button.

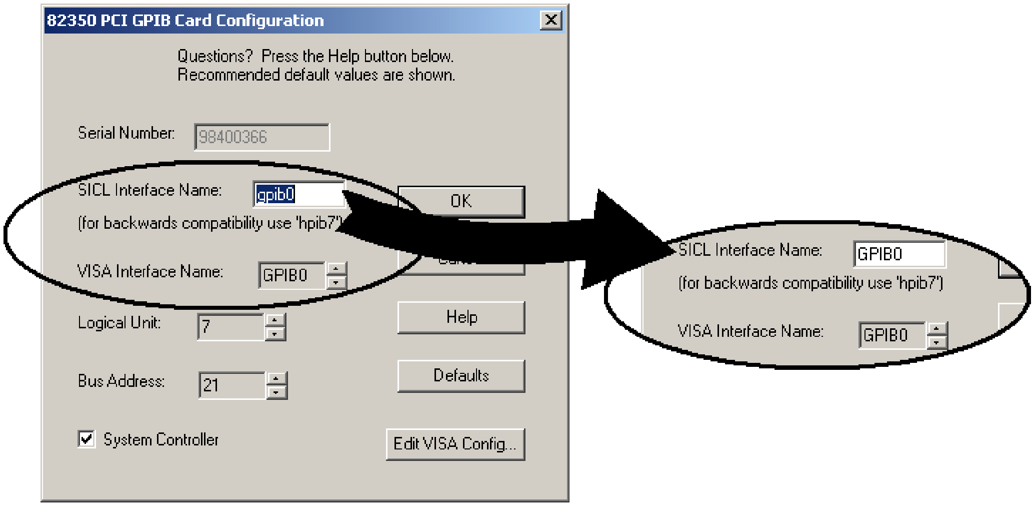


14. When the Auto Config process is run, the **Configured Interfaces** list is displayed. Check the GPIB entry. For GPIB entry, the **SICL Name** must match the **VISA Name** exactly including the case of the letters.

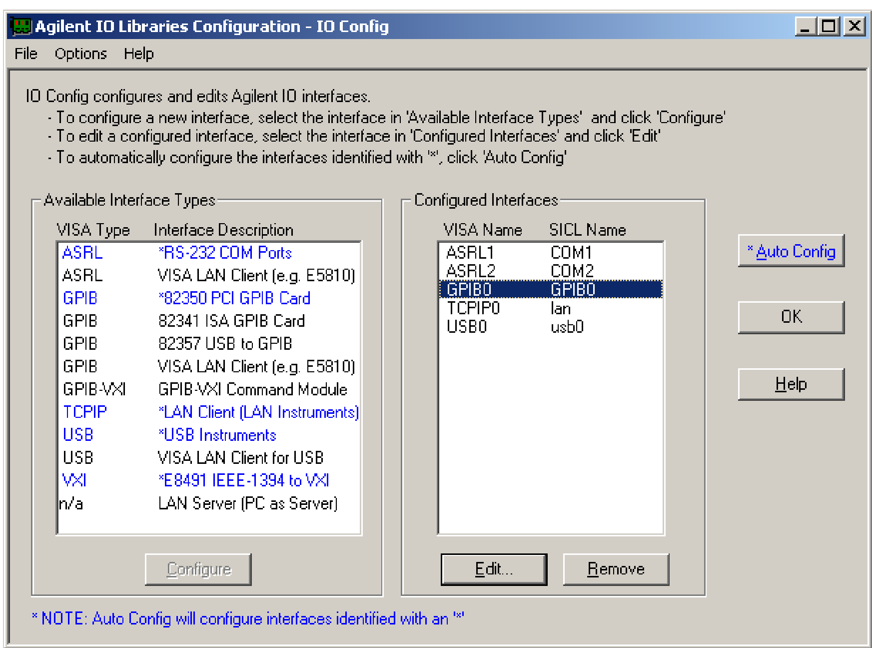


Since the “gpib0” SICL name does not match the “GPIB0” VISA name, select the GPIB entry and click the **Edit**... button.

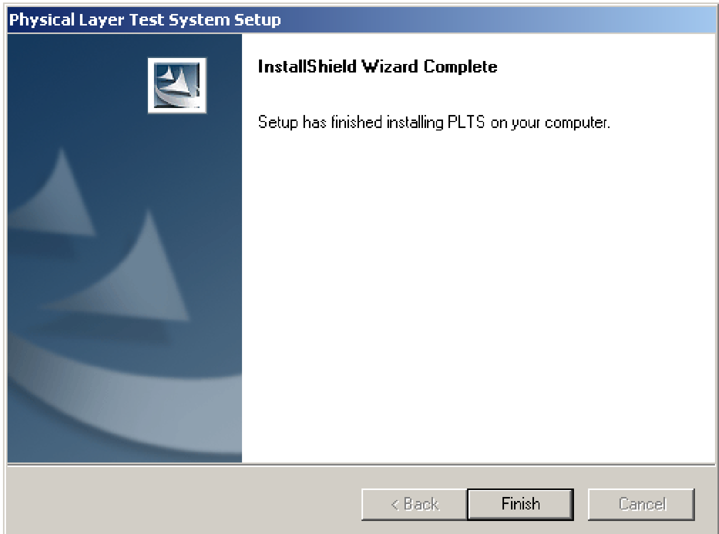
15. With the *GPIB Card Configuration* dialog box for your GPIB card displayed, change the **SICL Interface Name** to match the **VISA Interface Name** exactly. Then click **OK** to return to the *Agilent IO Libraries Configuration - IO Config* screen.



16. In the *Agilent IO Libraries Configuration - IO Config* screen, verify that the SICL name matches the VISA name for the GPIB card. Then click **OK**.

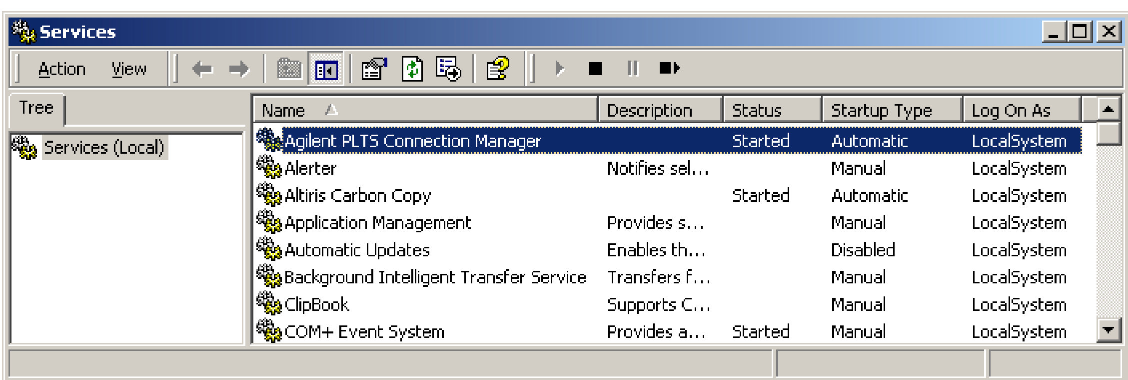


17. When the InstallShield Wizard is complete, click the **Finish** button. The PLTS software installation is complete.

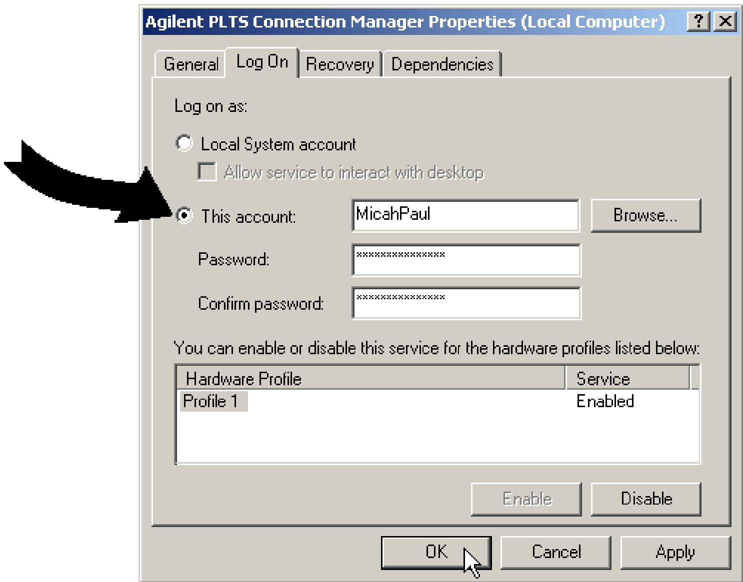


18. PLTS communicates with instruments using a Microsoft Service, the PLTS Connection Manager. The log on for this service needs to be changed to from a local system account to a specific account.

    To make this change, select **Start > Settings > Control Panel**. From the *Control Panel*, select **Administrative Tools** and then **Services**. Double-click the **Agilent PLTS Connection Manager** selection.



19. In the *Agilent PLTS Connection Manager Properties* dialog box window under the **Log On** tab, select the **This account** selection. Enter your user name with administrative privileges. Enter and confirm the password. Then click **OK** to save the change.



20. Continue with “Step 4. License the Physical Layer Test System Software” on page 77.

Installation of PLTS as a Software Upgrade

Use this procedure to upgrade your PLTS software. This procedure will first guide you through removing the older PLTS version and then guide you through installing the new PLTS software.

In addition to loading PLTS files, PLTS installs three other software packages that it requires. As part of the installation, PLTS also installs:

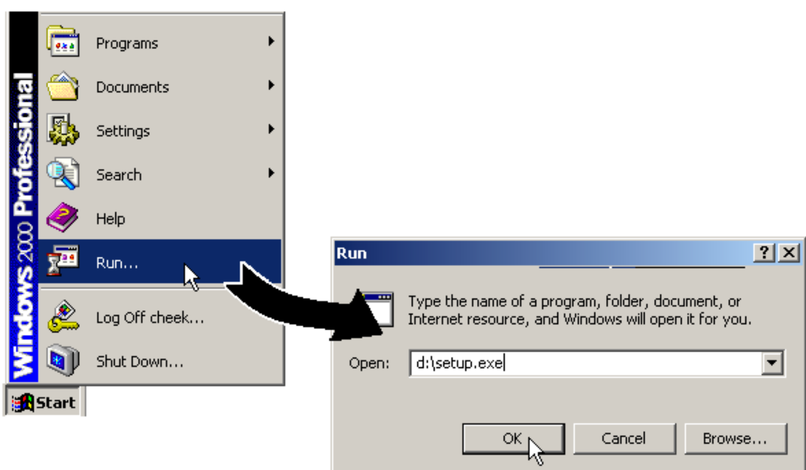
- Microsoft .NET Framework
- Agilent T&M Programmers Toolkit Redistributable Package
- Agilent IO Libraries

NOTE If you already have of these software packages installed on your PC, you will be asked if you want to upgrade to this version. If this version is newer than your version, please upgrade your software.

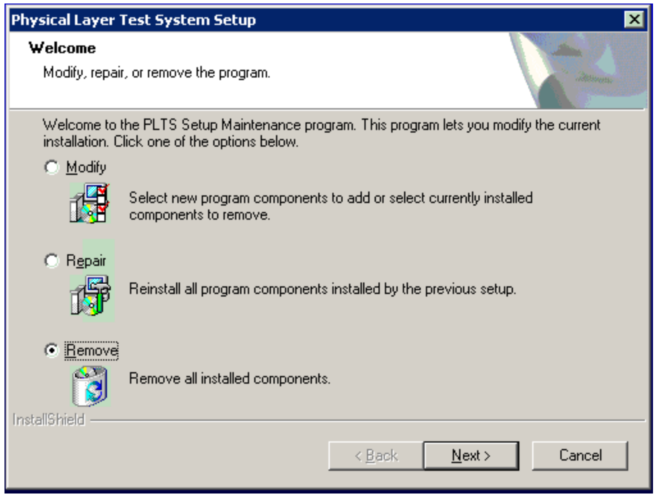
However, if your version is newer than the version PLTS wants to install, please decline to load the PLTS version.

1. Insert the PLTS software CD-ROM in your PC. The software should begin the installation process automatically.

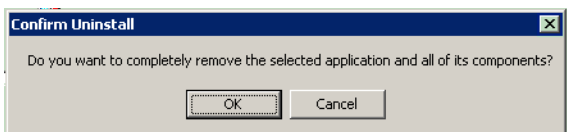
   If the software does not start the automatic installation process, select **Start > Run...** in Windows as shown below. With the *Run* dialog box displayed, enter your CD-ROM drive letter followed by "\:setup.exe". For example: `D\:setup.exe.` Then click **OK** and follow the onscreen instructions



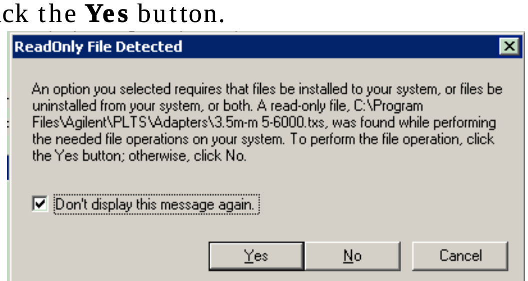
2. When asked select the **Remove** selection and click the **Next >** button.



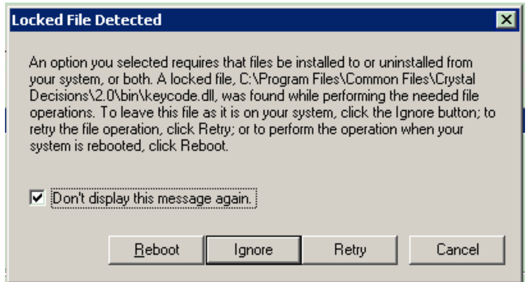
3. When asked to confirm the removal, click **OK** to completely remove the PLTS application and its components.



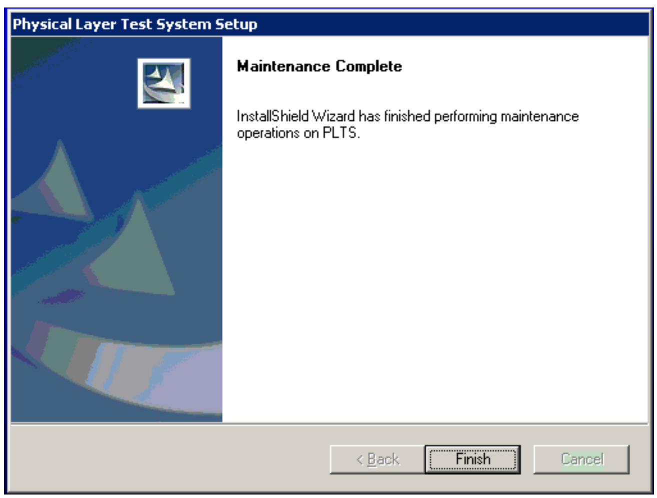
4. During the PLTS software removal process, several read-only files are detected. Click the **Yes** button each time this is encountered or select the **Don't display this message again** check box and then click the **Yes** button.



5. During the PLTS software removal process, several locked files are detected. Click the **Ignore** button each time this is encountered or select the **Don’t display this message again** check box and then click the **Ignore** button.

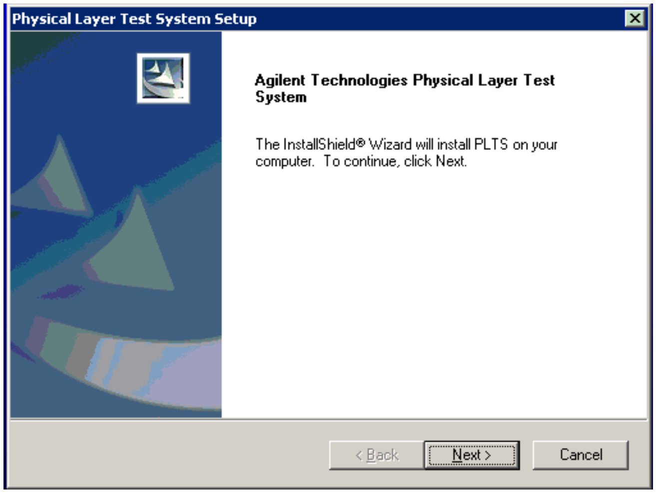


6. The PLTS software removal process will finish. Click the **Finish** button to complete the process.



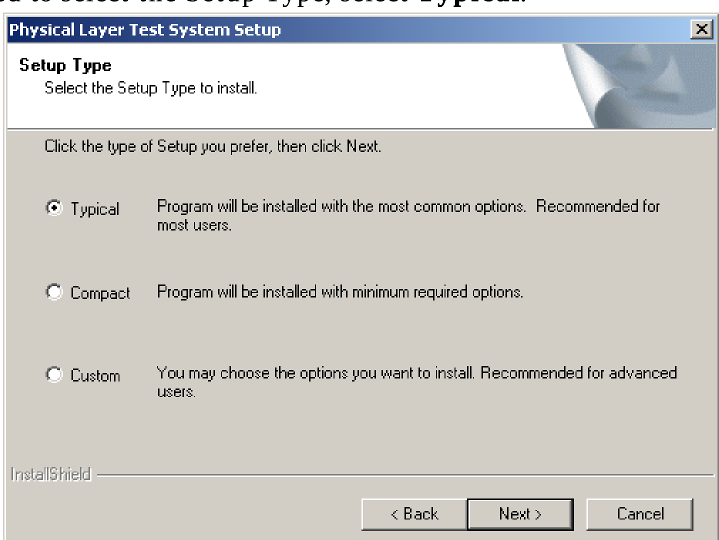
7. Repeat step 1 to start the installation of the new PLTS software.

8. After the InstallShield Wizard extracts the PLTS files, the InstallShield Wizard welcome screen is displayed. Click the **Next >** button to start the installation.



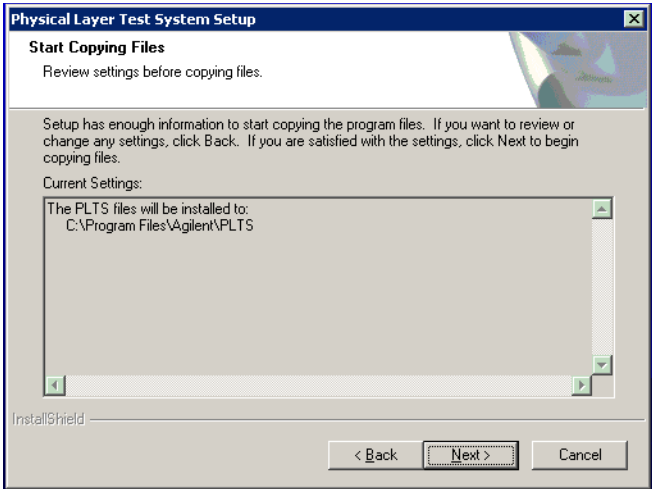
9. Before installing PLTS, a license agreement is displayed, read the agreement. You will be asked if you accept the terms of the license agreement. If you click **No**, the PLTS setup will close. To install PLTS, you must accept this agreement by clicking **Yes**.

10. When asked to select the Setup Type, select **Typical**.

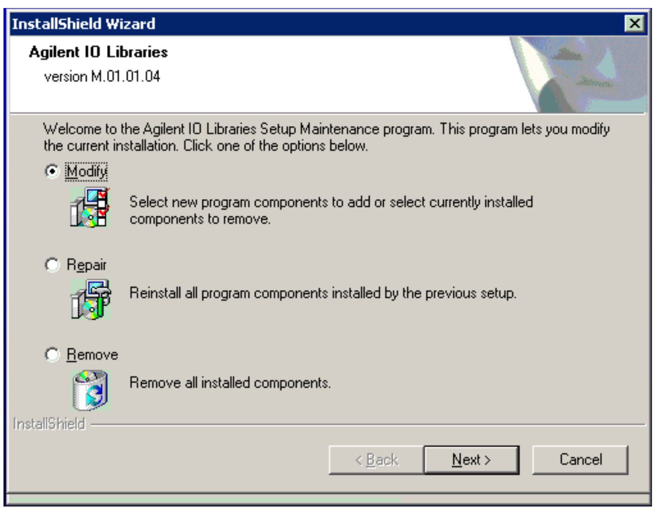


After checking the settings, the installation procedure begins installing PLTS program files.

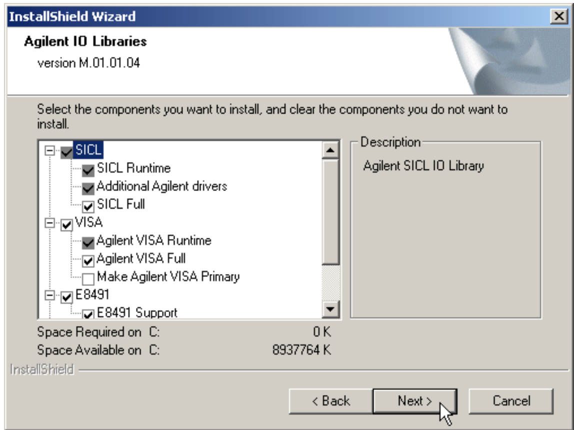
11. Review the settings and then click the **Next >** button.



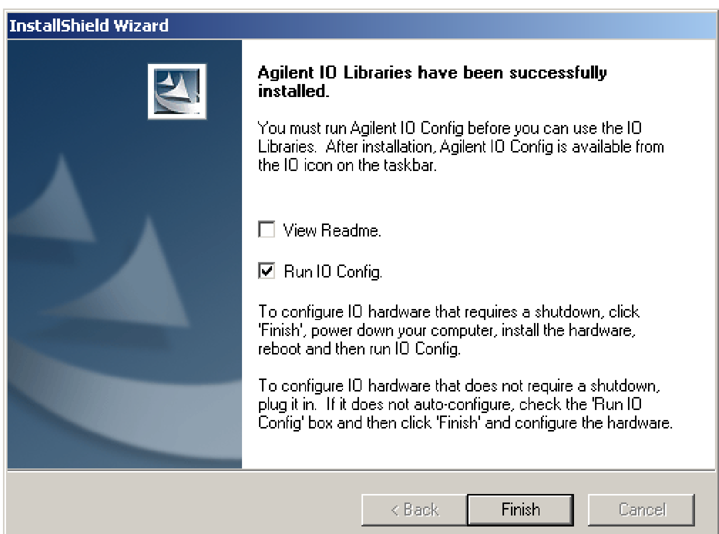
12. When asked to modify the Agilent IO Libraries installation, leave the selection set to **Modify** and click the **Next >** button.



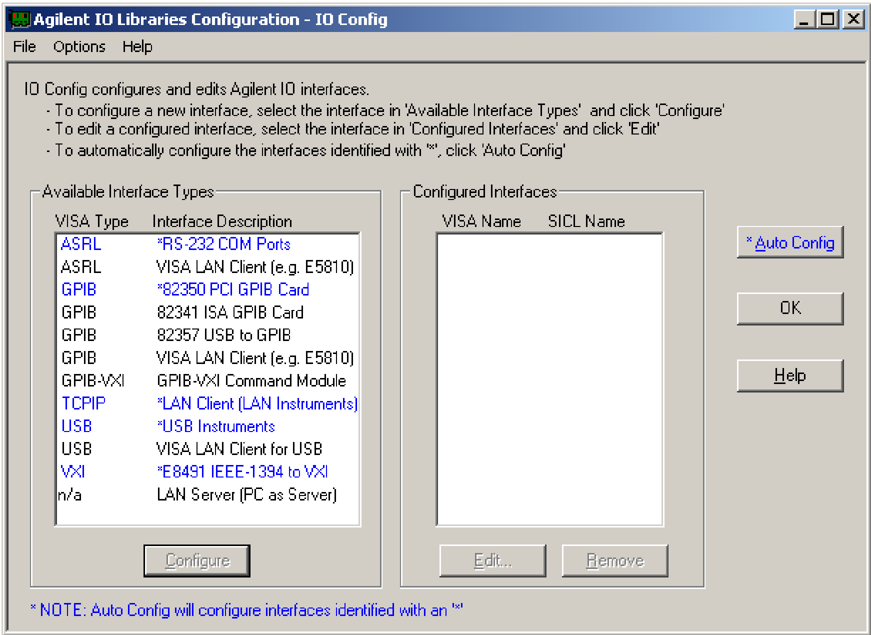
13. If the following Agilent IO Libraries screen is displayed, leave the selections unchanged and select the **Next >** button.



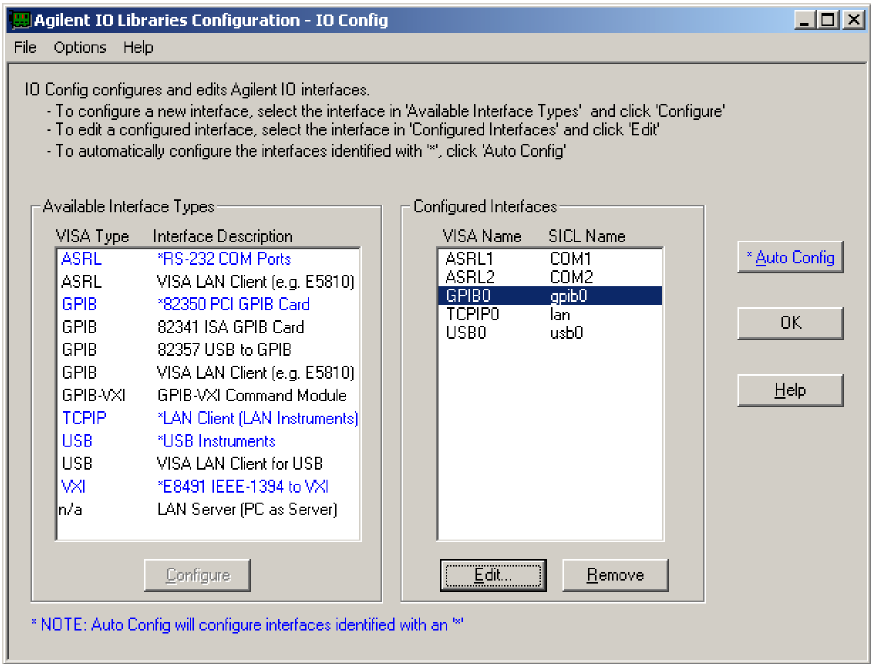
14. When the InstallShield Wizard notifies you that the Agilent IO Libraries has been installed, select the **Run IO Config** check box and click **Finish** to configure the IO.



15. The *Agilent IO Libraries Configuration - IO Config* screen is displayed. Select the **Auto Config** button.

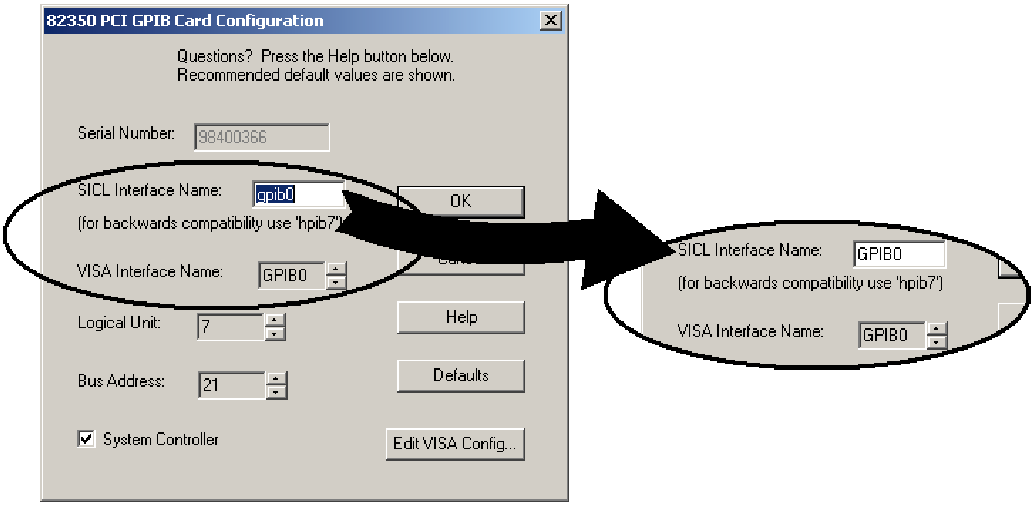


16. When the Auto Config process is run, the **Configured Interfaces** list is displayed. Check the GPIB entry. For GPIB entry, the **SICL Name** must match the **VISA Name** exactly including the case of the letters.

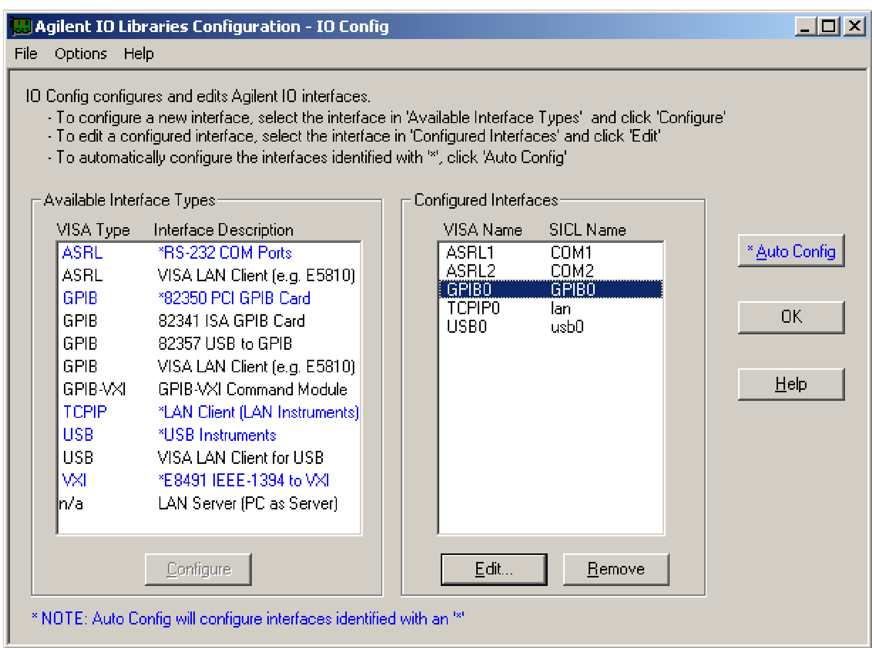


Since the “gpib0” SICL name does not match the “GPIB0” VISA name, select the GPIB entry and click the **Edit**... button.

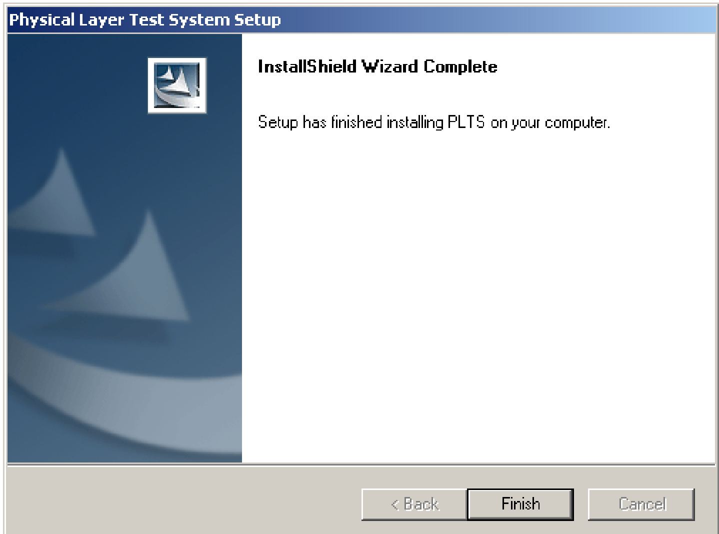
17. With the *GPIB Card Configuration* dialog box for your GPIB card displayed, change the **SICL Interface Name** to match the **VISA Interface Name** exactly. Then click **OK** to return to the *Agilent IO Libraries Configuration - IO Config* screen.



18. In the *Agilent IO Libraries Configuration - IO Config* screen, verify that the SICL name matches the VISA name for the GPIB card. Then click **OK**.

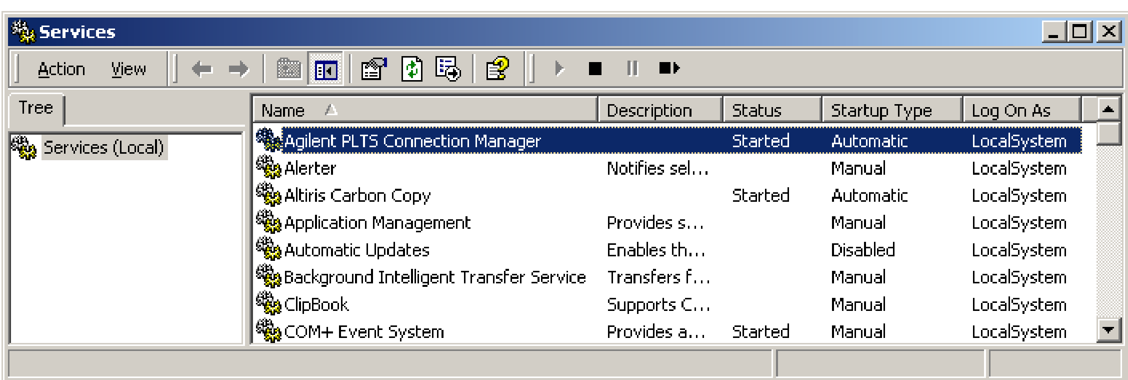


19. When the InstallShield Wizard is complete, click the **Finish** button. The PLTS software installation is complete.

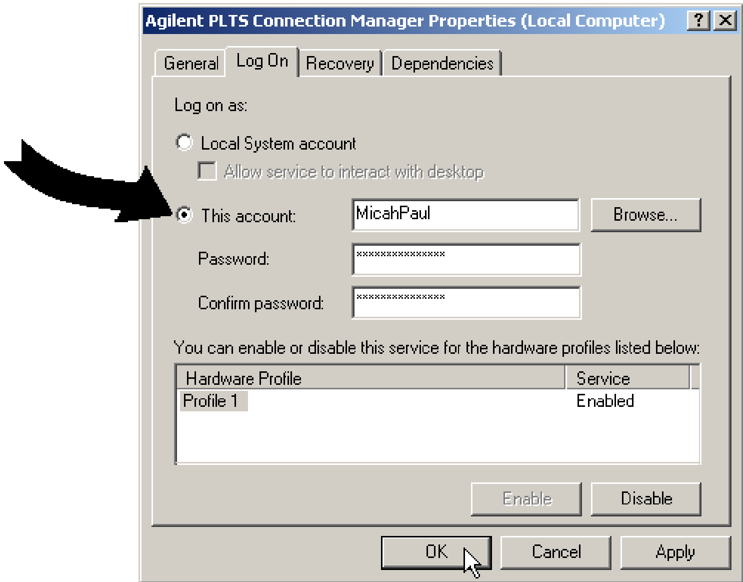


20. PLTS communicates with instruments using a Microsoft Service, the PLTS Connection Manager. The log on for this service needs to be changed to from a local system account to a specific account.

    To make this change, select **Start > Settings > Control Panel**. From the *Control Panel*, select **Administrative Tools** and then **Services**. Double-click the **Agilent PLTS Connection Manager** selection.



21. In the *Agilent PLTS Connection Manager Properties* dialog box window under the **Log On** tab, select the **This account** selection. Enter your user name with administrative privileges. Enter and confirm the password. Then click **OK** to save the change.



22. Continue with “Step 4. License the Physical Layer Test System Software” on page 77.

Step 4. License the Physical Layer Test System Software

PLTS is sold with one of two type of licenses, either node-locked or network-server floating. The type of license that you have was decided when the software was purchased.

A **node-locked license** entitles you to use the software on only one personal computer and the software enforces that restriction.

A **network-server floating license** entitles you to install the software license on only one server and check the license out to any personal computer on the server network that has PLTS installed. The software enforces the restriction that a license can checked out to one user at a time unless multiple PLTS network-server floating licenses are owned.

Stop! Determine the type of license that you own. Do not proceed until you have determined the type of license you own.

- If you own a node-locked license, continue with "Setting Up the Node-locked License" on page 77.
- If you own a network-server floating license, continue with "Setting Up the Network-Server Floating License" on page 79.

Setting Up the Node-locked License

This procedure instructs you how to license your Physical Layer Test System Software which is required before it can be used. The software is node-locked which means that license entitles you to use the software on only one personal computer (PC) and the software enforces that restriction.

CAUTION The software is node-locked which means that license entitles you to use the software on only one personal computer and the software enforces that restriction. Make sure that you license the software to the correct PC.

During this procedure, you will identify the Host ID of that PC. The host ID is a unique identifier for that computer. Then, you will use the host ID to license the software to that unique identifier of your PC. Therefore, it is very important to affirm that you have loaded and are licensing the software to the correct PC.

1. In Windows, select **Start > Programs** and locate the **Command Prompt** selection on your PC. Open the *Command Prompt* window by clicking the **Command Prompt** selection.

2. In the *Command Prompt* window, change the directory to the **License** subdirectory in the PLTS subdirectory using the following command:

   cd x:\dddd\ssss\PLTS\License

where,		
	x	is the letter of the drive the software was installed on.
	dddd	is the directory name the software was installed on.
	ssss	is any subdirectories the software was installed on.

   The default directory that the software is loaded on is: C:\Program Files\Agilent. Therefore, enter the following command to change to the default directory:

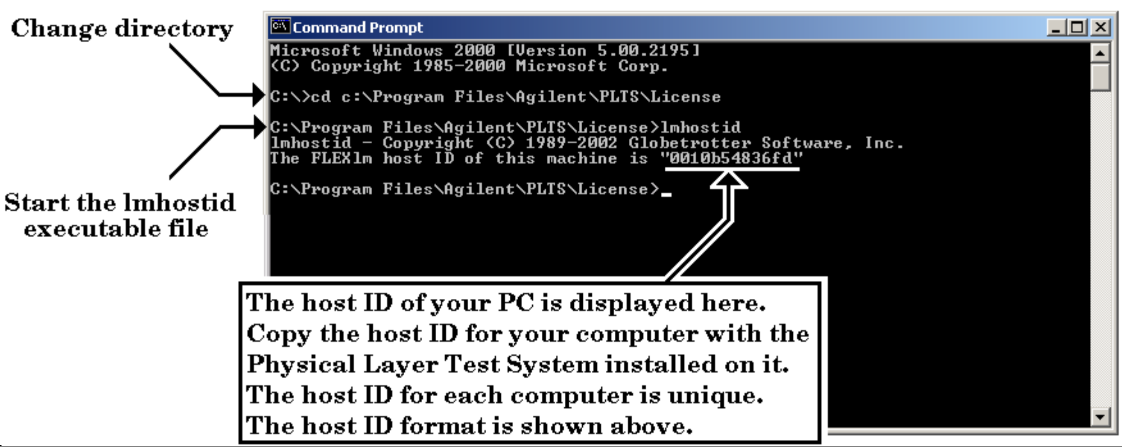
   ```
   cd c:\Program Files\Agilent\PLTS\License
   ```

   Refer to the illustration in step 4 on page 78.

3. Run the lmhostid executable file that will display the host ID for the PC that you have installed the software on by entering the following command as shown in the following figure:

   ```
   lmhostid
   ```

4. The host ID of the computer is displayed in window opened by the **lmhostid.exe** file. Write the host ID for your computer in the space provided below. You will need the host ID to license the software.



Record the host ID for your computer here:

5. Locate the **Software Entitlement Certificate** that was shipped with your software.

   The certificate lists the **Order Number** and the **Certificate Number** for your software. These two numbers will also be used to license your software.

The certificate also lists the web site where you must register your software.

6. Go to the Agilent web site listed on the **Software Entitlement Certificate**. Follow the instructions at the web site to receive your license file for the software. You will need to provide the following information:

   - **Order Number**
   - **Certificate Number**
   - **Host ID for your computer that will run the PLTS Software**

   The web site will also ask you provide your e-mail address. The license file will be e-mailed to you promptly.

7. Once you receive the E-mail with the attached licence file, save the file to the “License” directory (the same directory that you used earlier to identify the host ID).

8. Make a back up of the license file and store in a safe location.

   This back up file is very important! You may need this back up file if you encounter problems with your computer or if the licence file is lost or erased.

9. Continue with “Step 5. Start the Physical Layer Test System Software” on page 85.

Setting Up the Network-Server Floating License

This procedure instructs you how to set up PLTS with a *Network License Server/Floating* type of license. This type of license allows you use PLTS on one of many PCs (or clients), utilizing a single license on a common server. The server issues the license to any of the clients that attempt to run PLTS. If the license is already being used by another client, notification is provided that the license is in use. If you have multiple licenses of this type, multiple clients can be issued a license at the same time, up to the number of licenses installed.

CAUTION During this procedure, you will identify the Host ID of the server. The host ID is a unique identifier for a computer. Then, you will use the host ID to obtain a software license for that unique identifier of your server. Therefore, it is very important to affirm that you have loaded and are licensing the software to the correct server.

To use PLTS, the client PC must be on the same network as the license server.

NOTE With this type of license, a PC can be both a server and a client. This allows the PC to use the PLTS software or issue the license for other PCs on the network to use PLTS.

PLTS supports servers operating the Windows 2000 or Windows XP operating system.

Preparing the License Server

To install Network License Server/Floating type of license after you have installed PLTS on your PC (one of the client machines):

1. Copy the following files from your PC that has PLTS installed to the server:

agilent.exe	lmgrd.exe	lmhostid.exe	lmtools.exe	lmgrd.dl

You will find these files in the license directory of the PLTS software. If you used the default path to set up PLTS on your PC, that path is:.

C:/Program Files/Agilent/PLTS/License

While you could save these files to any directory on your server, we recommend that you create a similar directory structure on your server.

Obtaining the License

2. On your license server, in Windows, select **Start > Programs** and locate the **Command Prompt** selection. Open the ***Command Prompt*** window by clicking the **Command Prompt** selection.
3. In the *Command Prompt* window, change the directory to the **License** directory that you created to save the five files (the servers PLTS License directory):

   cd x:\dddd\License

where,	x	is the letter of the drive where you created a directory for the 5 files.
	dddd	is all directory names between the drive letter and the License directory you created.

4. In the *Command Prompt* window, run the lmhostid executable file to display the host ID for the license server by typing:

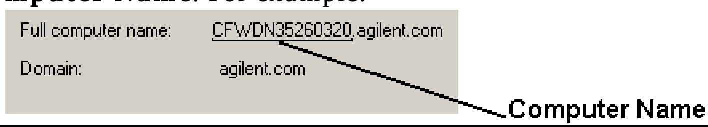
   ```
   lmhostid
   ```

5. The host ID of the server is displayed in the *Command Prompt* window. Write the host ID for your computer in the space provided below. You will need the host ID to license the software.

Record the host ID for your server here:

CAUTION This software license is locked to a specific server that you provide information for when obtaining the license. Make sure that you obtain and provide the Host ID for the correct server when obtaining your license.

6. You also need the network computer name of the server to redeem license. To locate the network name of your server: From Windows *Control Panel*, double-click the **System** icon to open the *System Properties* dialog box. On the **Network Identification** tab, locate the **Full Computer Name** entry. The network computer name of the server is the first portion of the **Full Computer Name**. For example:



Record the network computer name of your server here:

7. Locate the **Software Entitlement Certificate** that was shipped with your software.

   The certificate lists the **Order Number** and the **Certificate Number** for your software. These two numbers will also be needed to license your software.

   The certificate also lists the web site where you must register your software.

8. Go to the Agilent web site listed on the **Software Entitlement Certificate**. Follow the instructions at the web site to receive your license file for the software. You will need to provide the following information:

   - **Order Number**
   - **Certificate Number**
   - **Host ID for your PLTS license server**
   - **Network Computer Name for your PLTS license server**

   The web site will also ask you provide your e-mail address. The license file will be e-mailed to you promptly.

Setting Up the License Server

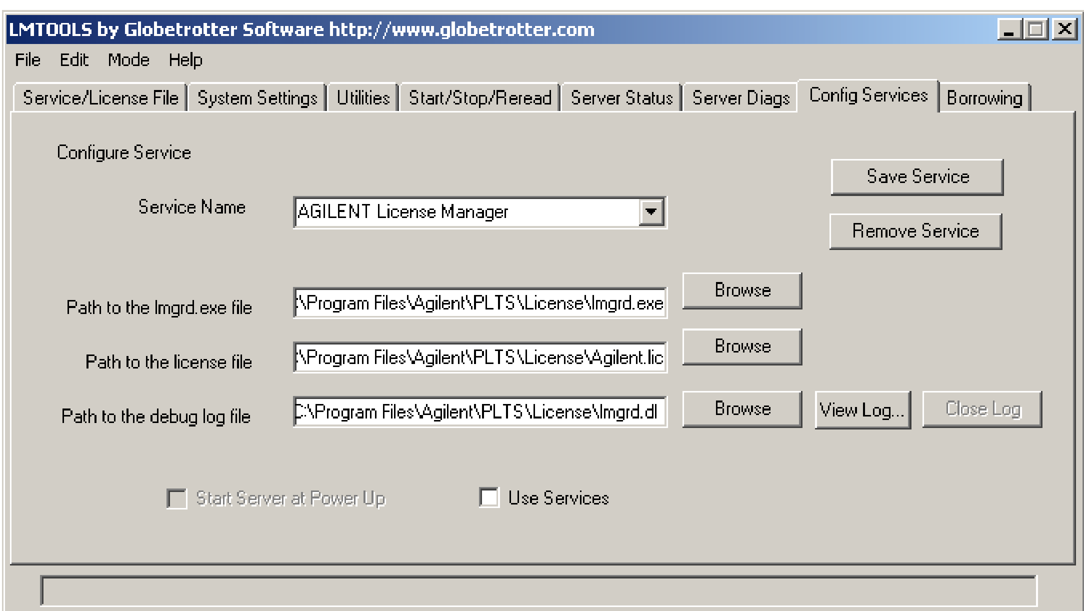
9. Once you receive the E-mail with the attached licence file, save the license file to the server's PLTS License directory (the same directory that you created on your server to save the five files to earlier).

10. Make a back up of the license file and store in a safe location.

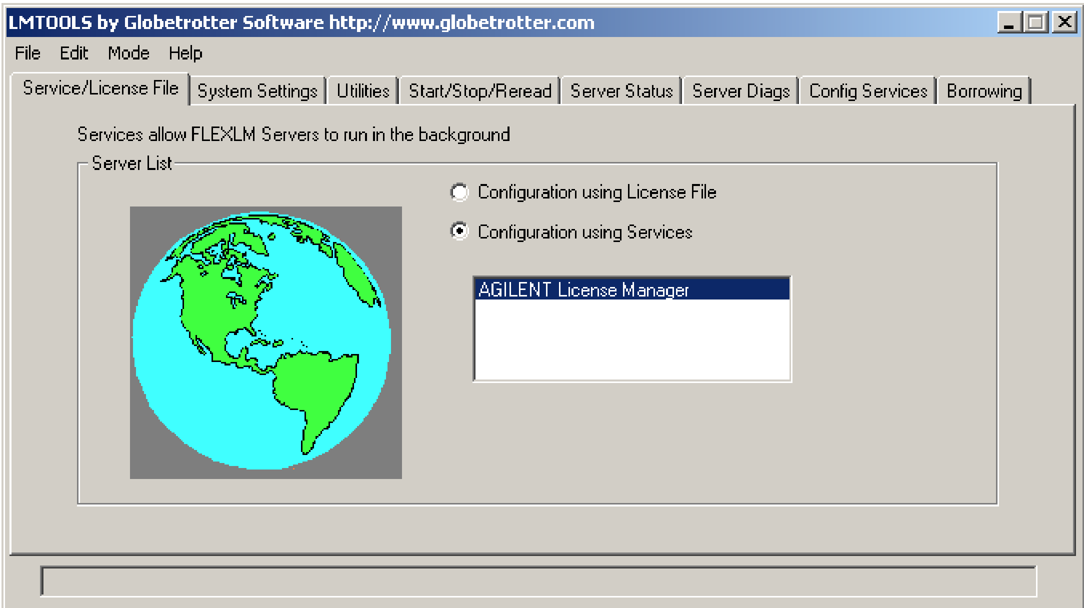
    This back up file is very important! You may need this back up file if you encounter problems with your computer or if the licence file is lost or erased.

11. In the server's PLTS License directory, double click the **lmtools.exe** file to open the *LMTools* dialog box. Select the **Config Services** tab. On this tab:

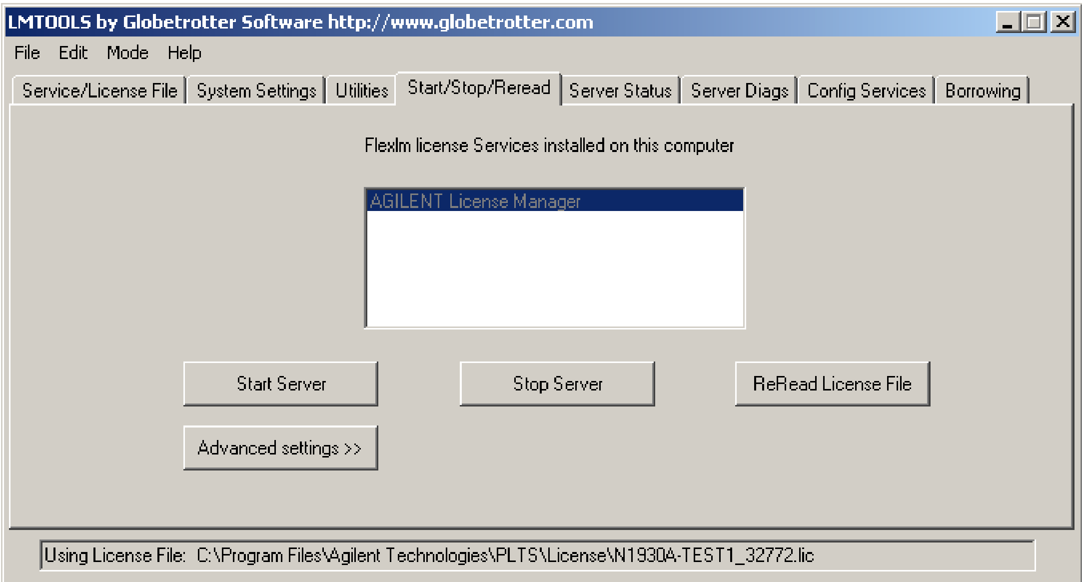
    a. In the **Service Name** list, select or enter **AGILENT License Manager**.
    b. In the **Path to the LMgrd.exe file** text box, enter the path to the server's PLTS license directory followed by the **lmgrd.exe** file name.
    c. In the **Path to the license file** text box, enter the path to the server's PLTS license directory followed by the file name of the license that you copied to the directory.
    d. In the **Path to the debug log file** text box, enter the path to the server's PLTS license directory followed by the **lmgrd.dl** file name.
    e. Click the **Use Services** check box and then click the **Start Server at Power Up** check box to make the PLTS license available for PLTS on restart of the server.
    f. Click the **Save Service** button to save the changes.



12. In the *LMTools* dialog box, select the **Service/License File** tab. On this tab, highlight **AGILENT License Manager** and click the **Configuration using Services** selection.



13. In the *LMTools* dialog box, select the **Start/Stop/Reread** tab. On this tab, highlight **AGILENT License Manager** and click the **Start Server** button.

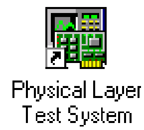


This step must be repeated each time the server is powered up if you did not click the **Start Server at Power Up** check box earlier in this procedure.

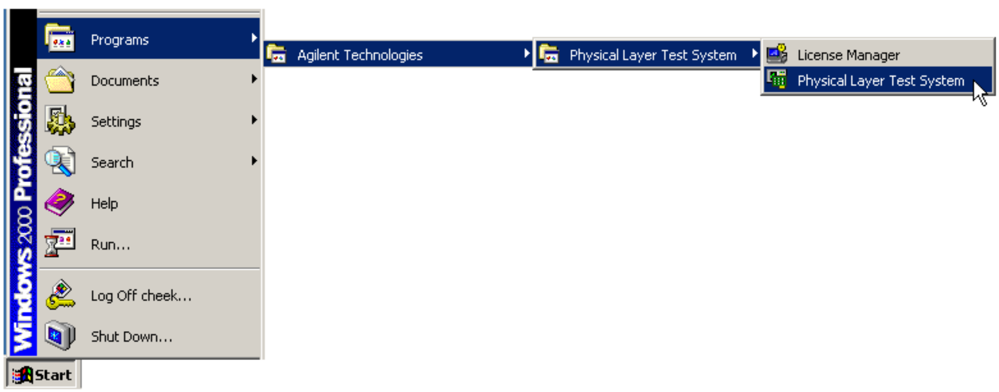
14. Continue with “Step 5. Start the Physical Layer Test System Software” on page 85.

Step 5. Start the Physical Layer Test System Software

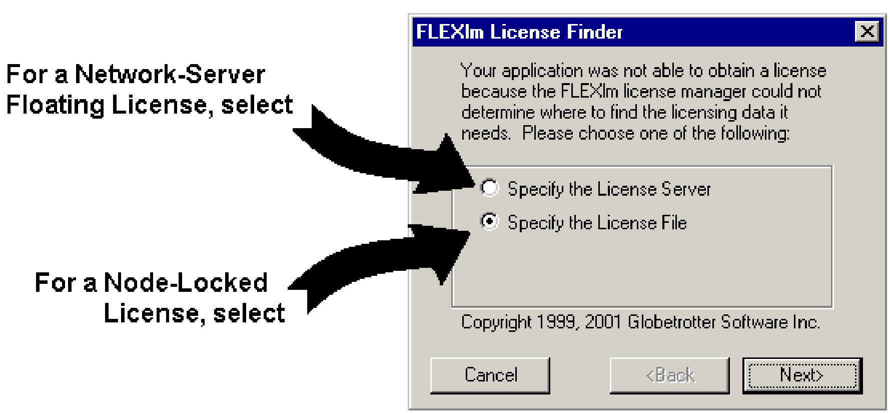
1. Start the PLTS software by double-clicking the **Physical Layer Test System** icon on the PC desktop.



You may also run the program by selecting **Start**, **Programs, Agilent Technologies**, **Physical Layer Test System**, **Physical Layer Test System**.



2. The first time you start the software, you will be asked to identify the license type of the software.



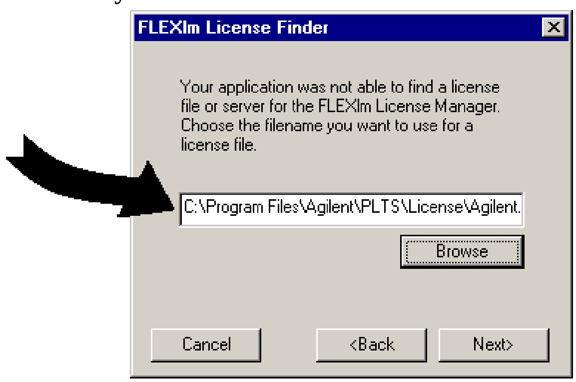
When the dialog box is displayed:

- **If you own a node-locked license:**
  Select **Specify the License File** and then click the **Next >** button.
- **If you own a network-server floating license:**
  Select **Specify the License Server** and then click the **Next >** button.

3. Based on the type of license you own:

- **If you own a node-locked license:**

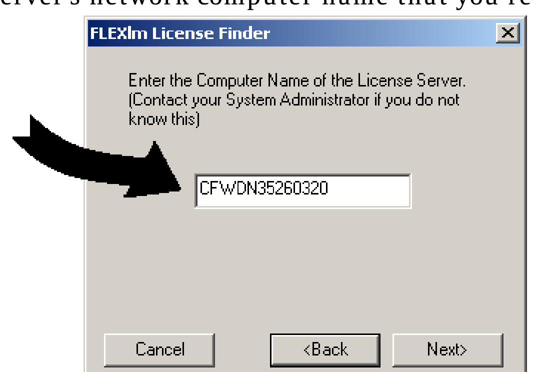
  When this *FLEXlm License Finder* dialog box is displayed asking you to choose the license filename, click the **Browse** button and select the license file in the License directory.



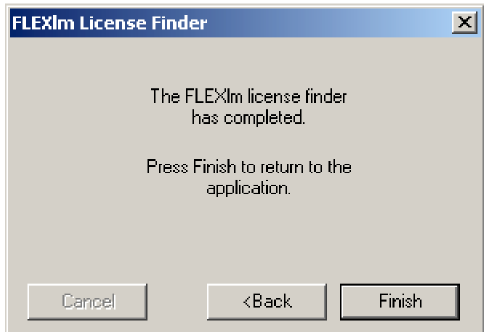
  Once the directory and the license file is identified, the directory path is displayed in the FLEXlm License Finder dialog box. Select the **Next>** button.

- **If you own a network-server floating license:**

  The client PC needs to be configured to look for the PLTS license on the Server. Enter the server's network computer name that you recorded earlier.



4. When the license is found and identified, select the Finish button to continue starting the software.



NOTE Make a back-up copy of the license file if you have not already done so.

---

5. After the licensing is completed, the software scans the GPIB bus to find the PLTS systems that are available to use.

   The PLTS software looks for the following VNA-based and TDR-based PLTS systems:

VNA-Based Systems:

N1957B (E8364B & N4421B)
N1955B (E8363B & N4420B)
N1953B (E8362B & N4419B)
N1951A (8720ES & N4418A)
N1948A (E8358A & N4417A)
N1947A (E8803A & N4417A)

TDR-Based Systems:

Agilent 86100A/B
Tektronix CSA8000
Tektronix TDS8000

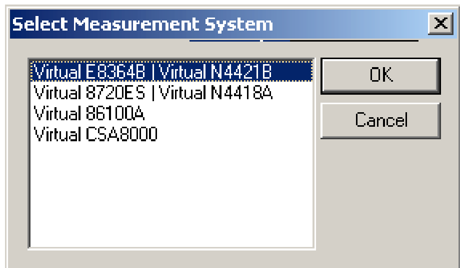
- If the software recognizes one of the systems, it sets your software to make a measurement using the measurement wizard. You should review "Step 6. Familiarize Yourself with the PLTS Software Screen" on page 91.

  Then, based on your PLTS system hardware, continue with either:

  — Chapter 4, "Setting Up and Making Measurements using the VNA-Based PLTS," on page 105
  — Chapter 6, "Setting Up, Calibrating, and Making Measurements using the TDR-Based PLTS," on page 183

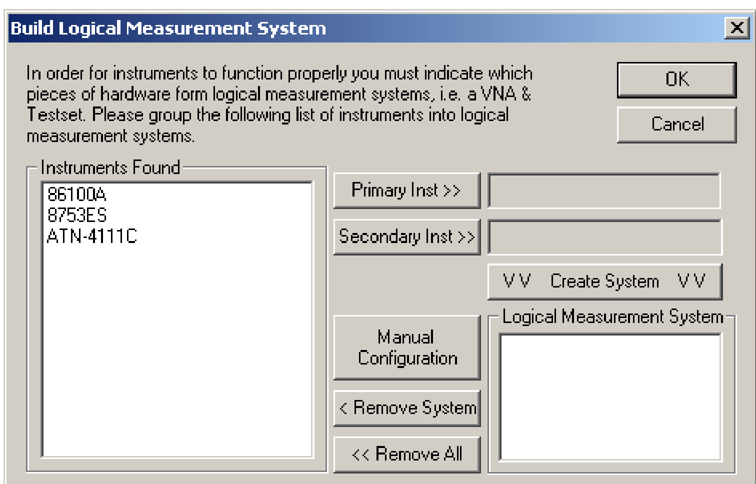
- If the software does not recognize one of the systems, it notifies you: "No instruments were detected. PLTS will operate in analysis-only mode." This may be because:

You want to perform analysis only and do not have any PLTS system hardware connected to the GPIB. In this case, select the virtual instrument that be matches your PLTS system or if known, the system that was used to measure the data. Continue at "Step 6. Familiarize Yourself with the PLTS Software Screen" on page 91.



— PLTS does not recognize your system hardware or PLTS finds more than one system connected to your GPIB. Continue with the next step.

6. If the PLTS software does not recognize your system hardware or if it finds more than one system on the GPIB, it displays the *Build Logical Measurement System* dialog box. This dialog box allows you to identify your PLTS system hardware for the PLTS software.



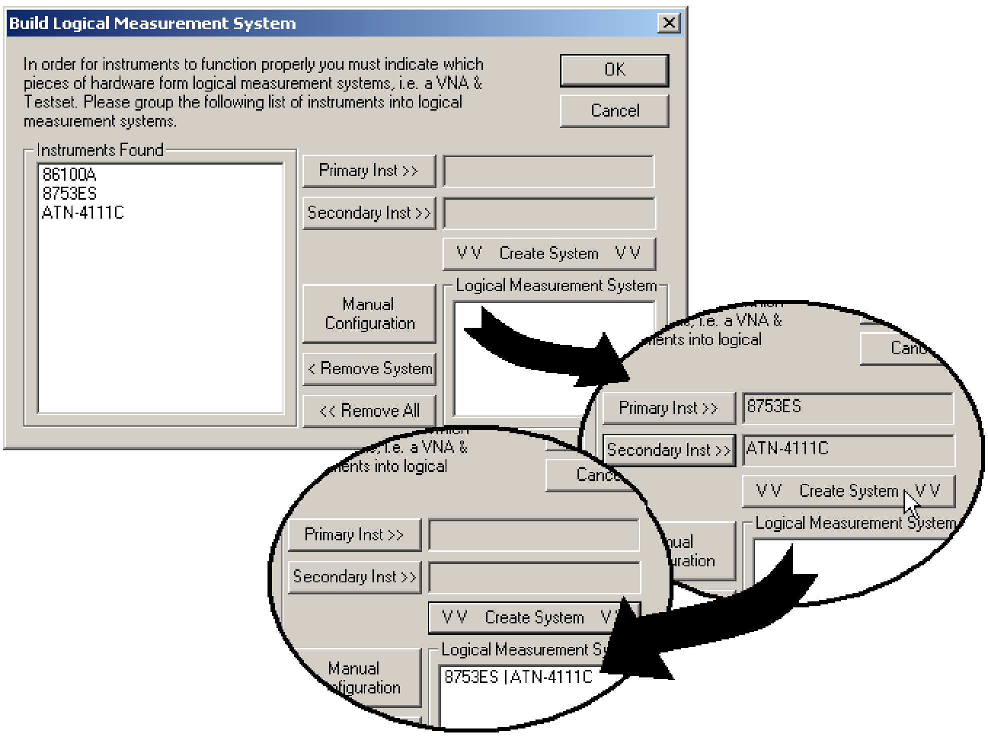
- To identify your VNA-based system (VNA with a test set) for the software:

  a. In the **Instruments Found** list, click the model number of your network analyzer to highlight it. Then click the **Primary Inst >>** button which moves the model number to the text box at the right of the button.

b. In the **Instruments Found** list, click the model number of your test set to highlight it. Then click the **Secondary Inst >>** button which moves the model number to the text box at the right of that button.

c. Click the **Create System** button which moves the combination of the devices to the **Logical Measurement System** box.

d. Click the **OK** button to save the system so the software identifies the PLTS system.



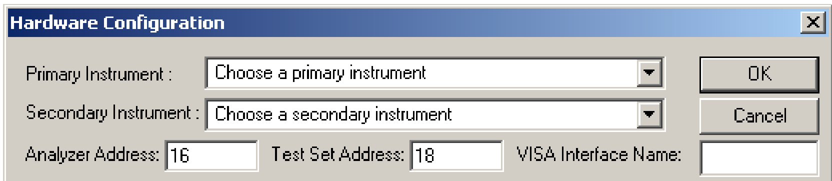
- To identify your TDR-based (or single-instrument) system for the software:

a. In the **Instruments Found** list, click the model number of your network analyzer to highlight it. Then click the **Primary Inst >>** button which moves the model number to the text box at the right of the button.

b. Click the **Create System** button which moves the device to the **Logical Measurement System** box. (Note that this step may be omitted for single-instrument systems, such as the TDR systems.)

c. Click the **OK** button to save the system so the software identifies the PLTS system.

NOTE If you can not get your system identifies, you can manually configure your system by selecting the **Manual Configuration** button to display the Hardware Configuration dialog box.



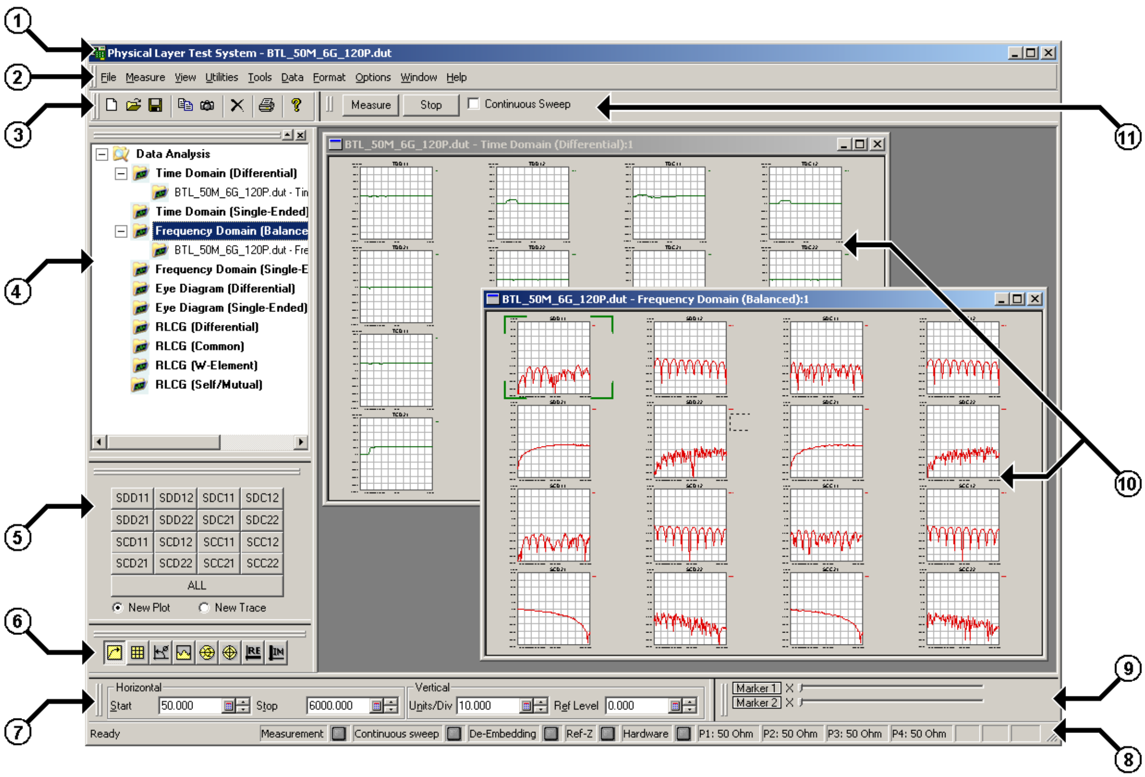
Select the primary instrument (typically at VNA or a TDR) from the **Primary Instrument** list. If necessary, select a secondary instrument (typically at test set) from the **Secondary Instrument** list.

Enter the address for the primary instrument in the **Analyzer Address** text box and the secondary instrument in the **Test Set Address** text box. Also, enter the **VISA Interface Name**.

Step 6. Familiarize Yourself with the PLTS Software Screen

This section describes the features of the Physical Layer Test System software. Refer to Figure 3-1, Main Window, for the locations of each of the software's main features.

Figure 3-1 Main Window



1. **Title Bar**
2. **Menu Bar**
3. **Toolbar**
4. **Browser**
5. **Parameter Bar**
6. **Format Bar**
7. **Scaling Bar**
8. **Status Bar**
9. **Marker Bar**
10. **Plots Area**
11. **Measure Bar**

Each of these areas are described in the remainder of this section.

The software was designed with an informal flow down the left edge of the window in mind. It is an informal flow because with the flexibility of the software, this is an easy way, not the

only way to setup many of the features. For example, you may open a device file from areas 2 and 3. Then you will select a data analysis that shows the open file in area 4. You will need to select parameters to display from area 5. If you need to change the format of the display, you change the format in area 6. Then you can adjust the scaling of the plots in area 7.

1. Title Bar

The title bar (Figure 3-2) displays the title of the software. In the brackets, it also displays the file name of the data in the active plot window if the data has been saved. The data analysis type of the active plot window is also displayed. At the right side of the title bar, buttons are also provided quickly perform tasks at the program level.

Figure 3-2 Title Bar

Clicking:	**Performs:**
	Minimizes the program window to the Windows task bar.
	Toggles the program window to full screen mode.
	Toggles the program window to partial screen mode.
	Closes the program.

2. Menu Bar

The menu bar (shown in Figure 3-3) lists each of the drop down menu names currently available. When you click a menu name, the menu is displayed. The menu names and menus change to reflect appropriate choices for the data analysis type of the active plot window. Refer to Chapter 14, “Menu Reference,” for detailed information about each of the menu bar selections.

When the active plot window is maximized, completely filling the plot area, the menu bar also displays three buttons to control the plot window.

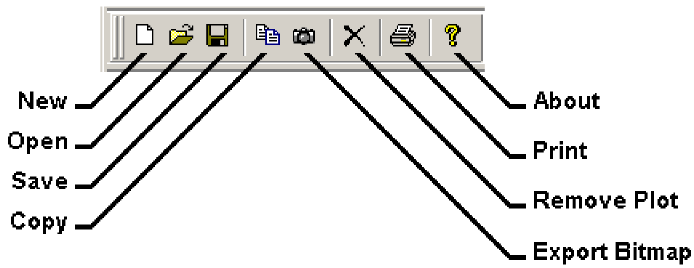
Figure 3-3 Menu Bar

Clicking:	Performs:
	Minimizes the plot window within the plot windows display area.
	Toggles the plot window to partially fill the plot windows display area.
	Closes the plot window.

3. Toolbar

The toolbar (Figure 3-4) provides buttons for beginning a new measurement, opening existing files, saving current data, copying current data, exporting data, removing plots, and printing.

Figure 3-4 Toolbar



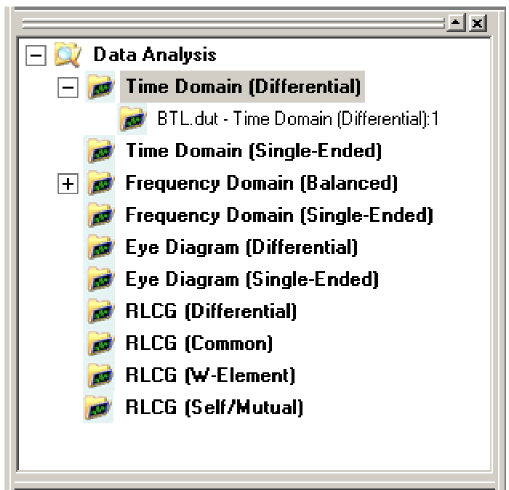
Clicking:		Performs:
	**New**	Starts the setup wizard so that a new measurement may be made.
	**Open**	Opens the *Open* dialog box so that an existing file may be opened.
	**Save**	Opens the *Save A*s dialog box so that a file may be saved.
	**Copy**	Copies the active plot to the clipboard.
	**Export Bitmap**	Saves the active plot as a bitmap file (bmp).
	**Remove Plot**	Deletes the active plots.
	**Print**	Prints the active plots.
	**About**	Opens the *About PLTS* information.

4. Browser

The browser (Figure 3-5) provides access to each measurement and data analysis type. Selecting a new Data Analysis type from the browser opens up a corresponding plot window. Each Data Analysis type may view multiple sets of data.

- Clicking a bold label opens a new window in the plots area with that data analysis type.
- Clicking a non-bold label displays that plot and makes it the active plot in the plots area.
- Clicking the minus sign (–)hides the data selections for a given data analysis type.
- Clicking the plus sign (+) shows the data selections.

Figure 3-5 Browser



5. Parameter Bar

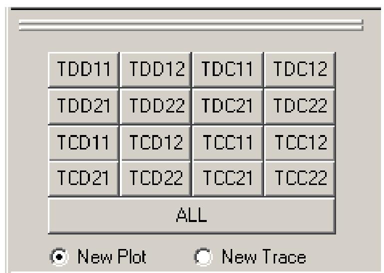
The parameter bar (Figure 3-6) allows you to open a new plot, or add a new trace to an existing plot. Figure 3-6 shows the parameter bar for the balanced time domain plots. Each analysis type has a unique parameter bar specific to that type of analysis. Refer to Figure 14-24, “Parameter Bars for Each Data Analysis Type” on page 435 to see all parameter bars.

Select **New Plot** and then click a parameter to display a new plot.

- or -

Select **New Trace** and then click a parameter to overlay a new trace to an existing plot. Clicking the parameter a second time removes the added trace from the plot.

Figure 3-6 Parameter Bar for Balanced Time Domain Plots



6. Format Bar

The format bar (Figure 3-7) is only available when the active plot window is a time domain or frequency domain selection. The format bars allow you to format the parameters for the active time domain or frequency domain plot window. To display the format bar, select **Format Bar** from the **View** menu. It is hidden with the same selection.

The time domain format bar differs from the frequency domain format bar. It has different format selections and it is divided into three areas. Therefore, the time domain format bar has three selections while the frequency domain format bar has one selection. Each format bar matches its corresponding **Format** menu (selected in the **Menu** bar.) The format bar and the title of each button for both time and frequency domain are shown below.

Figure 3-7 Format Bars

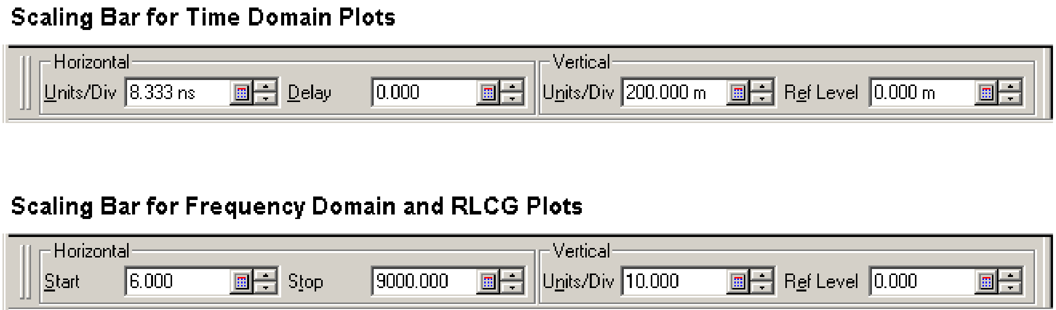
Time Domain | Frequency Domain

Time Domain	Frequency Domain
For detailed information about each of the selections, refer to "Selecting Time Domain Display Formats" on page 272.	For detailed information about each of the selections, refer to "Selecting Frequency Domain Display Formats" on page 233.
Impulse	Log Mag (default selection)
Step (default selection)	Linear Mag
Volts (default selection)	Phase
Real	Group Delay
Log Mag	Smith Chart
Impedance	Polar Chart
ns (default selection)	Real
cm	Imaginary

7. Scaling Bar

The scaling bar (see Figure 3-8) allows you to change the scale on both the horizontal and vertical axis.

Figure 3-8 Scaling Bars



For time domain, the horizontal axis entries are units/division and delay. The vertical axis units are units/division and reference level.

For frequency domain and RLCG, the horizontal axis entries are the start frequency and the stop frequency. The vertical axis units are the units/division and the reference level.

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

---

8. Status Bar

The status bar (Figure 3-9) provides a visual indication of when conditions are happening within the software and the impedance associated with each system channel.

When any of the conditions listed below are active, an indicator adjacent to the condition title is displayed in a bright color. If the condition is not active, the color of the indicator is gray. The conditions are:

Measurement is bright when a measurement is being performed by the software.

Continuous sweep is bright when the measurement is being continuously swept.

De-Embedding is bright when the active plot has de-embedding data applied.

Ref-Z is bright when the active plot has reference impedance applied.

Hardware is bright when the hardware has been identified by the software.

Figure 3-9 Status Bar

9. Marker Bar

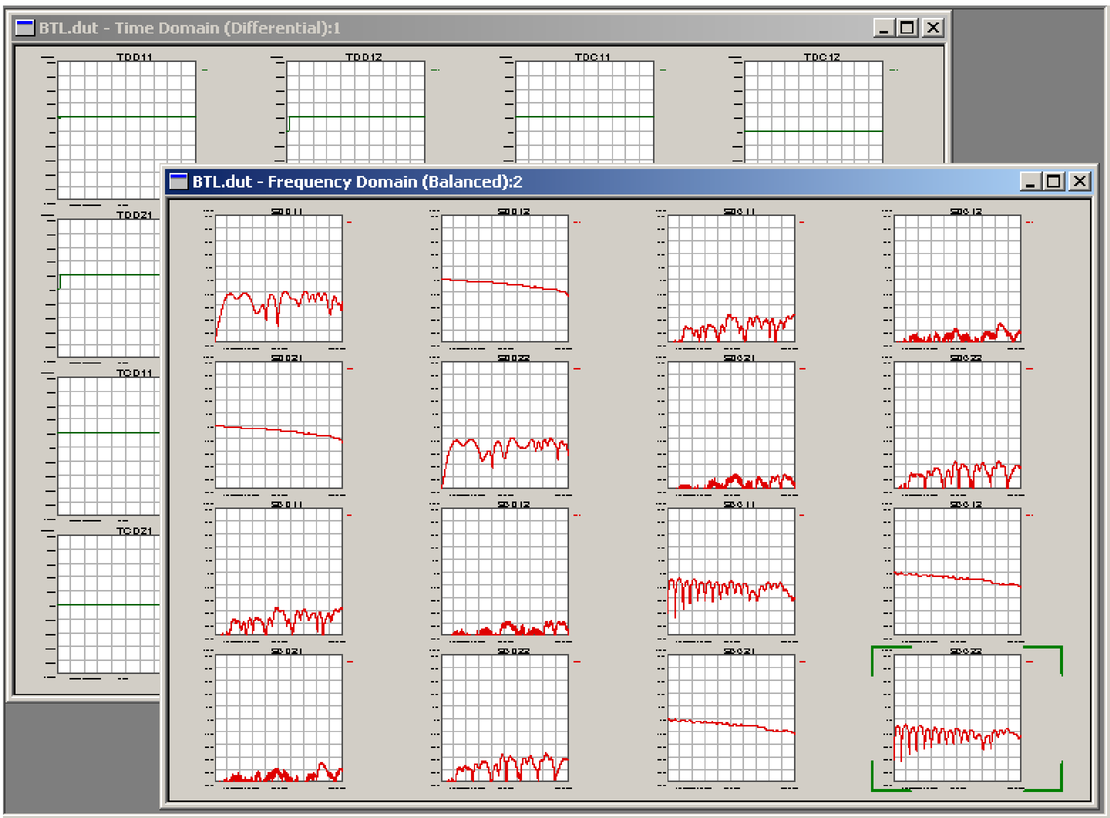
The marker bar (Figure 3-10) allows you to add and display up to two markers to the active plot. The X-Y values of each cursor and the delta (the value of their difference) is displayed at the right edge of the plot. The eye diagram marker bar and the frequency domain polar and Smith chart marker bars differ slightly. Refer to “Markers” on page 365 for complete marker bar information.

Figure 3-10 Marker Bar

10. Plots Area

The plot windows (Figure 3-11) display the measurement results in the data analysis type selected. You select the data analysis type when you make a measurement, open existing data files, or make a selection in the browser. The **Parameter Bar** or the **Data** menu on the **Menu** bar allows you to display plots for all of the parameters or just the parameters you want to view. Multiple plot windows can be displayed in the plot window display area at the same time as shown in the figure.

Figure 3-11 Plot Windows in the Plots Area



11. Measure Bar

The measure bar allows you to make measurements on your DUT without using the wizard. With a valid calibration file loaded, you may measure a single sweep or measure continuous sweeps.

Figure 3-12 Measure Bar

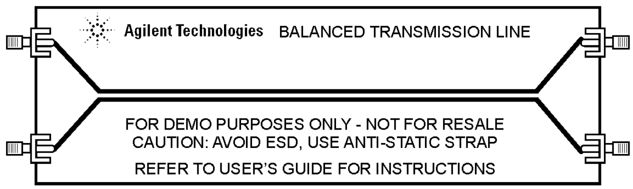
II Calibration and Measurements

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

4 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 4-1.

Figure 4-1 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 4-2. 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 4-2 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.

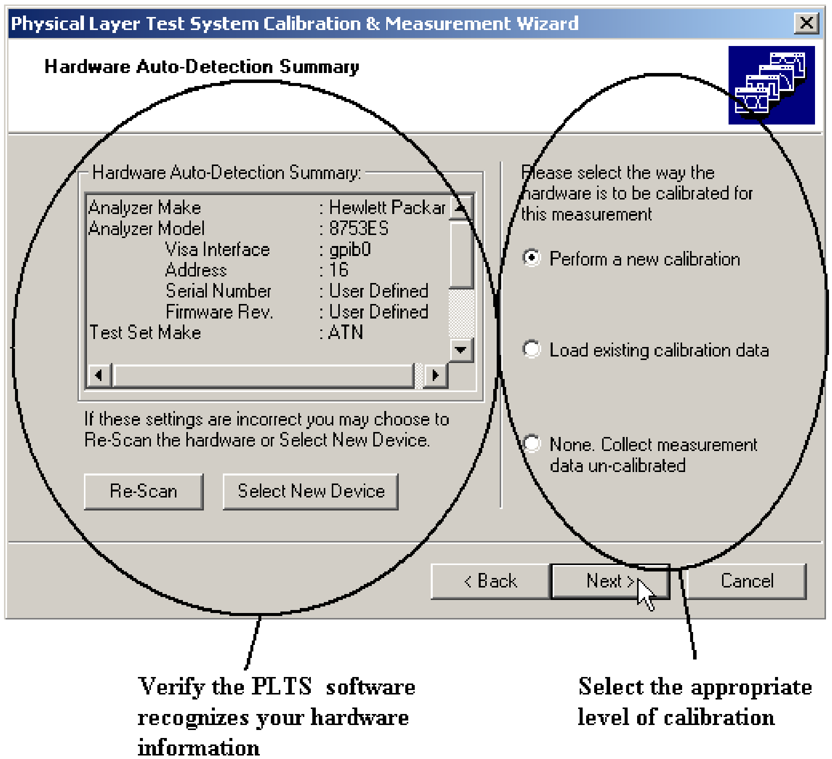
   This step uses the right portion of the *Hardware Auto-Detection Summary* dialog box to select the appropriate level of calibration to be performed. You may perform a new calibration, reuse existing calibration data, or perform measurements without calibration.

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 107, the *Hardware Auto-Detection Summary* dialog box is displayed. See Figure 4-3.

Figure 4-3 Hardware Auto-Detection Summary Dialog Box



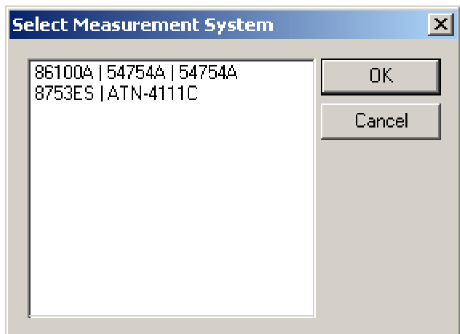
Verifying the Software Recognizes the PLTS Hardware

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 4-4. 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 4-2.

Figure 4-4 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 4-3), 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 5, “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 4-5 on page 111), and then proceed directly to the measurement screen. This option is ***not recommended*** for qualitative data collection.

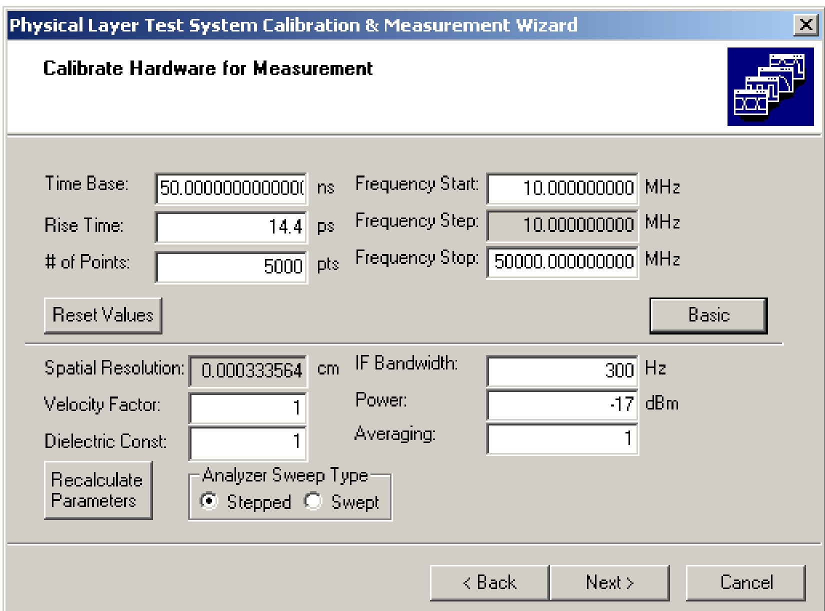
Select the **Perform a new calibration** choice for this example exercise and then click the **Next >** button.

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 4-5. 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. 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 4-5 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 4-1 for a description of each parameter.

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

The **Advanced** button shown in Figure 4-5 toggles between **Advanced** and **Basic**. When in **Advanced**, the entire dialog box is displayed. When in **Basic**, just the upper portion of Figure

4-5 (the top six entries of Table 4-1) is displayed.

Table 4-1 Measurement and Calibration Parameter Entry Descriptions

Parameter	Description
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.
Rise Time	sets the transition time (in picoseconds) 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.
Frequency Step	sets the step size between points. This value is locked to (is the same as) the Frequency Start value.
Frequency Stop	sets the stop frequency of the sweep.
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 measurement bandwidth.
Velocity Factor	is the numerical value related to the speed of energy through transmission lines with different dielectrics.
Dielectric Constant	is not used directly. It is used to calculate the velocity factor, which affects the way the data is displayed on the plots.
IF Bandwidth	allows you to change the IF bandwidth. Narrow IF bandwidths allow you to view low-level signal, however, require more data samples per point and thus slow 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 the traces of consecutive sweeps until the total number of sweeps is equal to the averaging factor, for a fully averaged trace for 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.

How to Perform a Calibration

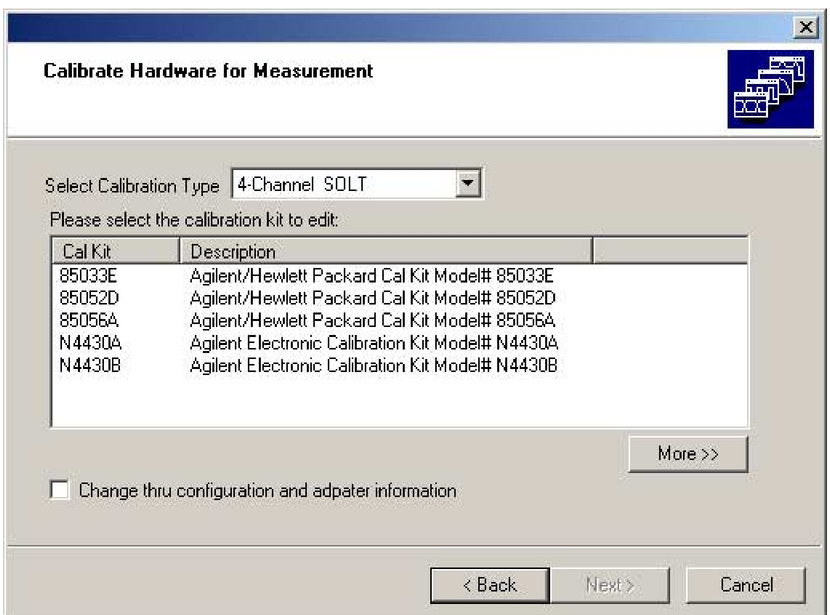
After you review and change the calibration and measurement parameters as needed and click **Next >**, the second of three *Calibrate Hardware for Measurement* dialog boxes is displayed as shown in Figure 4-6. This *Calibrate Hardware for Measurement* dialog box is used to select your calibration kit. However, the calibration procedure is described in Chapter 5, "Performing Error Correction on the VNA-Based PLTS."

To perform the calibration, continue with "Selecting a Calibration Type" on page 127. 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 114.

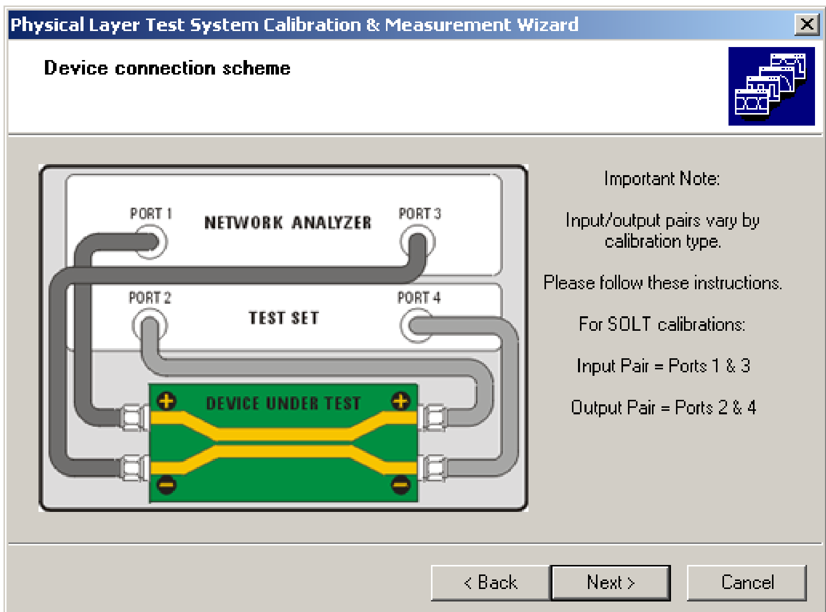
Figure 4-6 Select Calibration Type Dialog Box



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 4-7. 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.

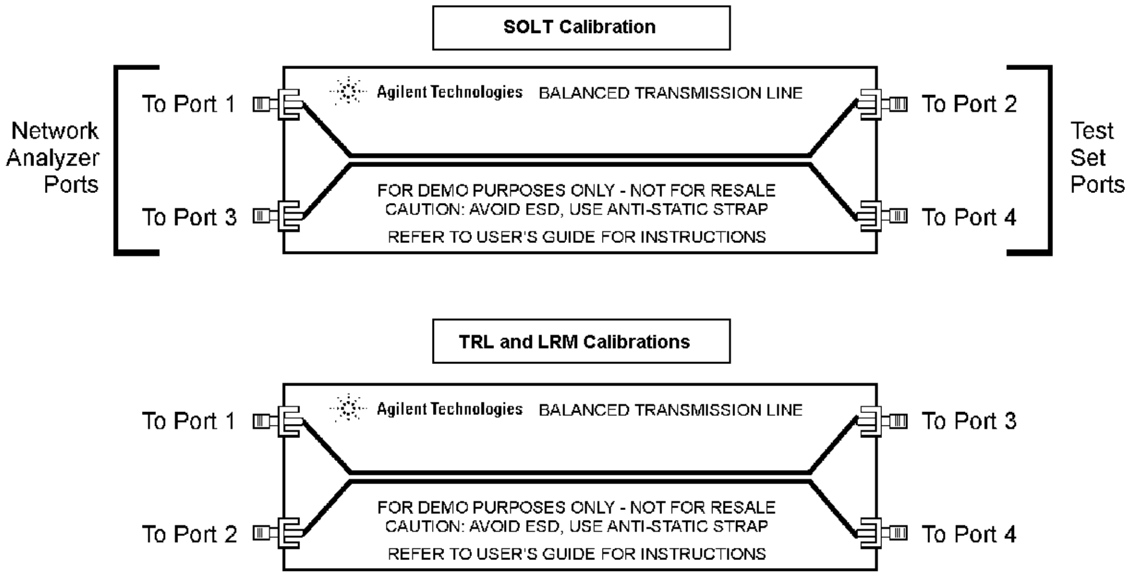
Figure 4-7 Device Connection Scheme Window



SOLT calibration data requires a different connection scheme than either TRL or LRM calibration data. See Figure 4-8.

NOTE Figure 4-8 shows the difference in the DUT connection between SOLT calibrations and TRL/LRM calibrations.

Figure 4-8Device Connection Differences by Calibration Type



Notice the difference in the ports 2 and 3 connections for the calibration methods.

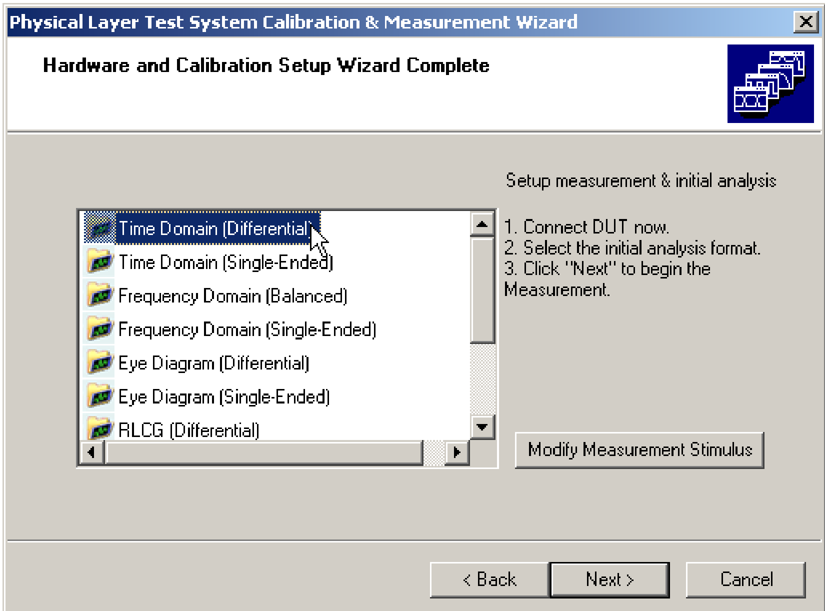
2. The *Hardware and Calibration Setup Wizard Complete* window is displayed. See Figure 4-9.

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

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

Figure 4-9 Hardware and Calibration Setup Wizard Complete Window



The **Modify Measurement Stimulus** button opens the *Measurement Stimulus* dialog box. See Figure 4-10. This dialog box allows you to change the many 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 111.

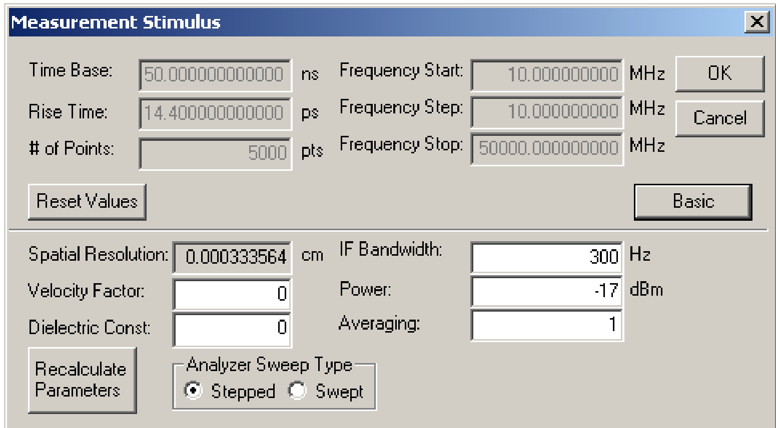
For the purpose of this example, leave the settings unchanged and click the **OK** button.

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 389 for help.

Figure 4-10 Stimulus Dialog Box

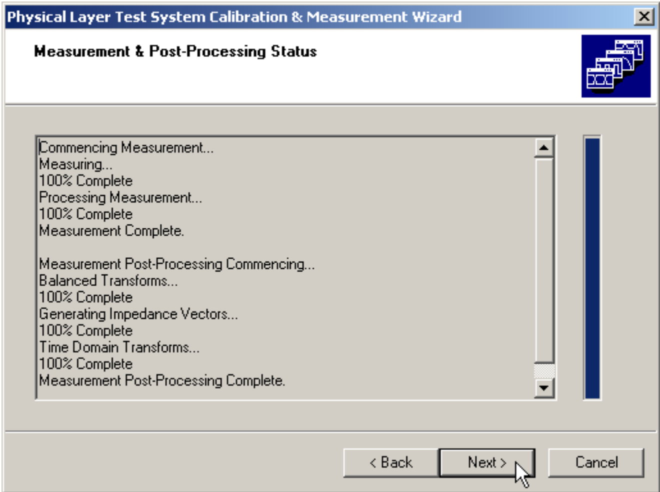


3. Once you exit the *Measurement Stimulus* dialog box, the *Hardware and Calibration Setup Wizard Complete* window is displayed again (see Figure 4-9), select the **Next >** button to start the measurement.

4. The software displays the wizard's *Measurement & Post-Processing Status* window and starts the measurement and the measurement post-processing. See Figure 4-11. 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.

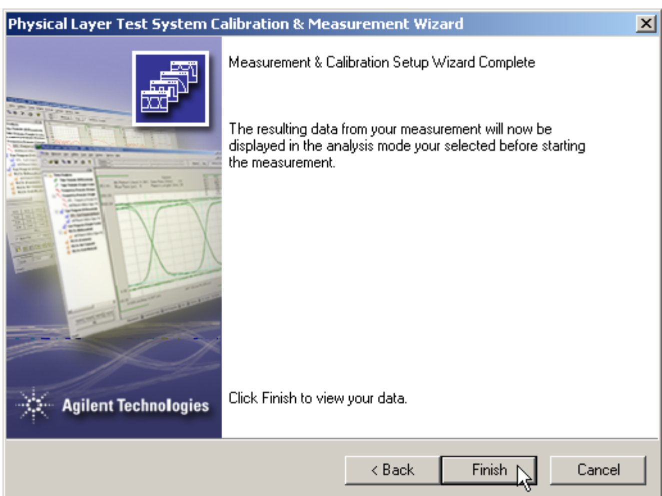
   Once the post-processing is complete, click the **Next >** button to display the *Measurement and Calibration Setup Wizard Complete* window.

Figure 4-11 Measurement & Post-Processing Status Window



5. After selecting the **Next >** button, the *Measurement and Calibration Setup Wizard Complete* window is displayed. See Figure 4-11. Click the **Finish** button to display the measurement results in the main PLTS window.

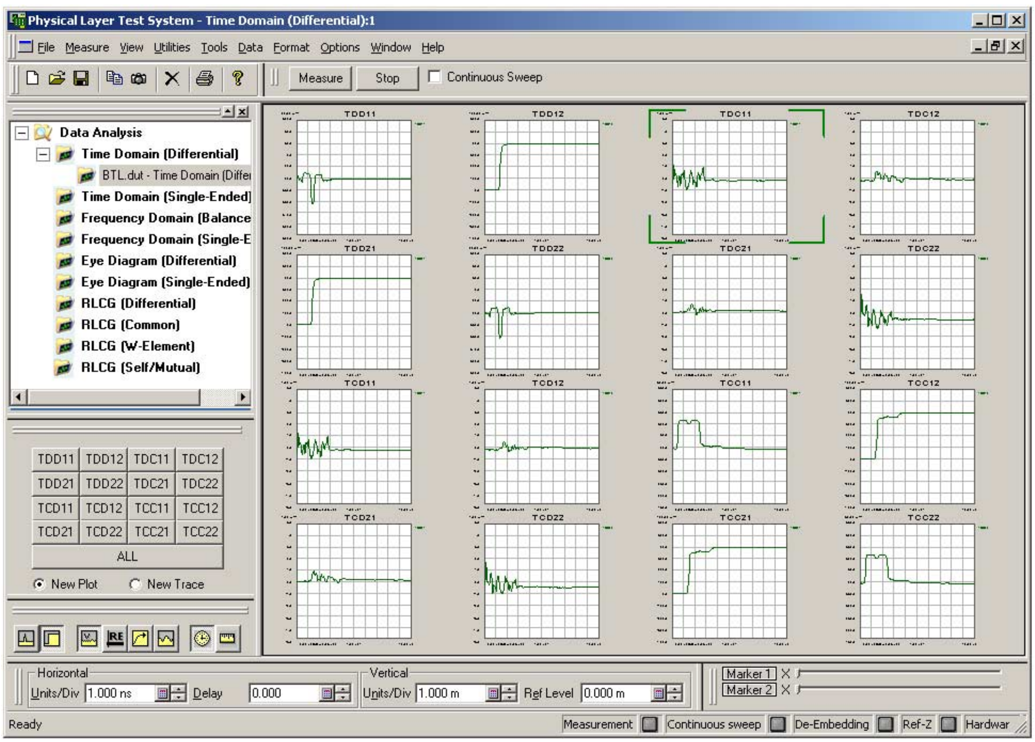
Figure 4-12 Measurement & Calibration Setup Wizard Complete Window



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 4-13. This shows the results in the **Time Domain (Differential)** format, the analysis format selected earlier.

Figure 4-13 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 7, “Analyzing Data in the Frequency Domain,” on page 221 for information.
- **Time Domain.** The Time Domain analysis may be analyzed in Differential or Single-Ended mode. Refer to Chapter 8, “Analyzing Data in the Time Domain,” on page 241 for information.

- **Eye Diagram.** The Eye Diagram analysis may be analyzed in Differential or Single-Ended mode. Refer to Chapter 9, “Analyzing Data using Eye Diagrams,” on page 279 for information.
- **Transmission Line (RLCG).** The RLCG analysis may be analyzed in Differential, Common, W-Element, and Self/Mutual modes. Refer to Chapter 10, “Analyzing Transmission Line Parameters,” on page 297 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” on page 419.

5 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 125.
  - 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 4, skip to "Selecting a Calibration Type" on page 127.
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 4-5 on page 111.

   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.

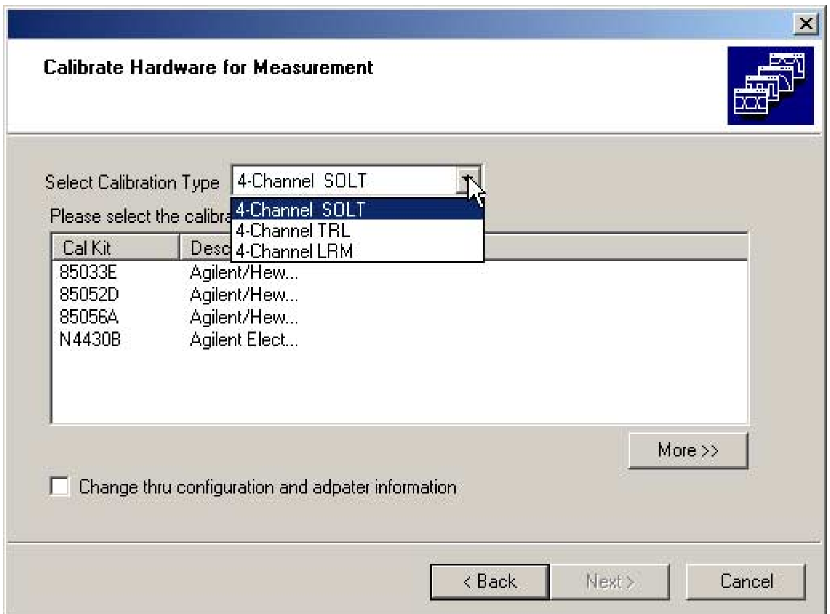
   You can change any of the time domain or frequency domain parameters, and then by selecting the **Recalibrate Parameters** button, the remaining parameters are recalibrated and displayed. Once all of the parameters are set to your satisfaction, select **OK** to return to the *Calibrate Hardware for Measurement* 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 5-2.

Selecting a Calibration Type

The dialog box shown in Figure 5-2 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 5-1 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 5-1, PLTS Port Configurations for a Balanced Line by Calibration Type, for the

configuration of each port measuring a balanced line.

Table 5-1 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 129.
- If you selected **4-Channel TRL** calibration, refer to "Performing a TRL Calibration" on page 148.
- If you selected **4-Channel LRM** calibration, refer to "Performing an LRM Calibration" on page 162.

Performing an SOLT Calibration

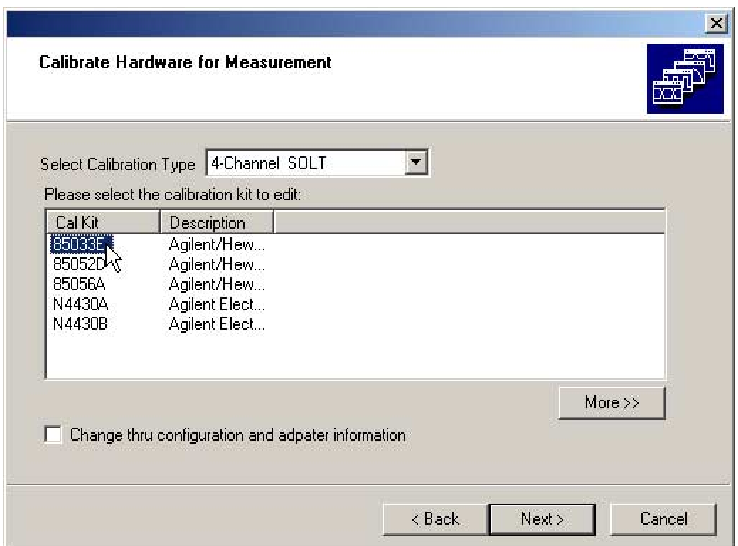
You have selected the **4-Channel SOLT** calibration type. You will first select a calibration kit and then you will perform a calibration (either mechanical or electronic, depending on your calibration kit). After you finish the calibration, you will be ready to make a measurement.

Selecting a SOLT Calibration Kit

The dialog box shown in Figure 5-2 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 132 for details.

Figure 5-2 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 5-2 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 132.)
- Characterize your own thru adapters to meet or exceed your frequency requirements (refer to "Characterizing Adapters" on page 176) and select that newly-created thru (adapter) file to create a new **Thru Standards** definition (refer to "Defining a SOLT Calibration Kit" on page 132).

1. Select your calibration kit. Refer to Table 5-2 for the appropriate calibration kit.

Table 5-2 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.

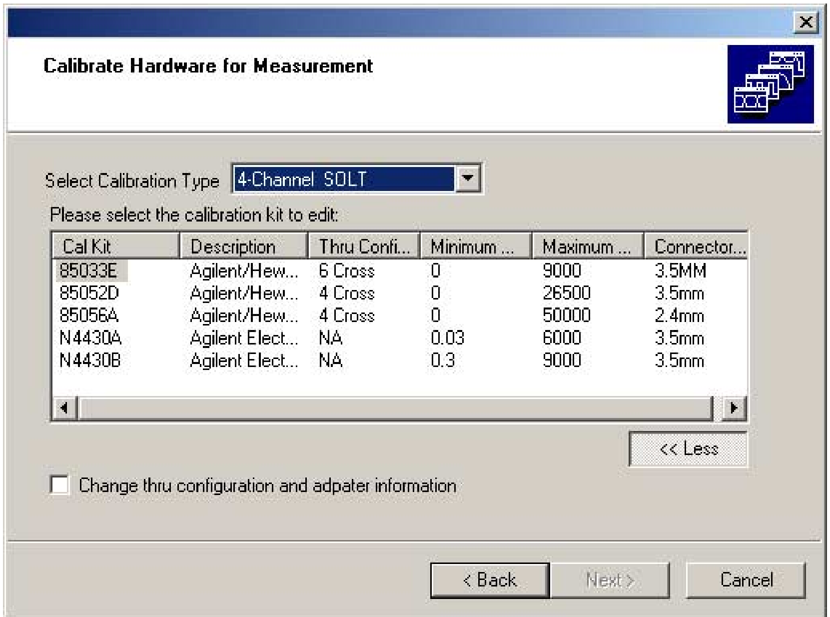
The *Nominal Frequency Range* columns display the minimum and maximum frequencies that the calibration kit will operate. These frequencies are limited by the physical calibration kit. They can *not* be changed.

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.

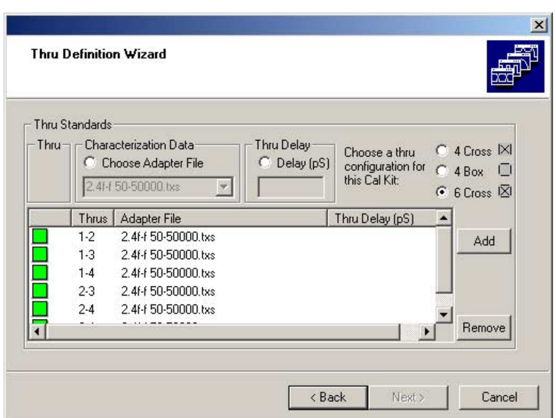
See Figure 5-3.

Figure 5-3 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 5-4 Thru Definition Wizard



Refer to step 10 on page 134 (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 5-3. Clicking the **Cancel >** button will exit the start up wizard without making any changes to the calibration kit definition.

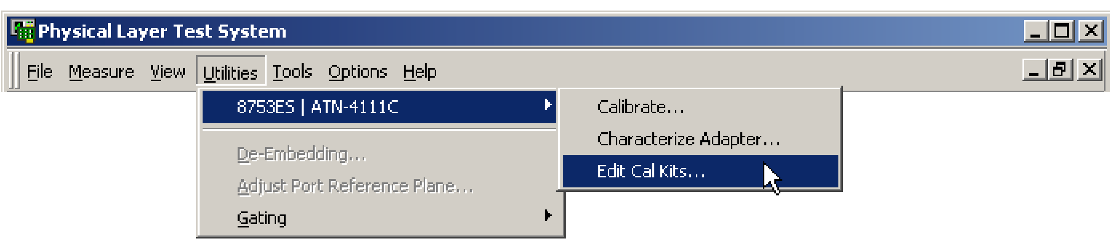
4. Once the calibration kit is selected, click the **Next >** button to display the calibration window.
   - If you selected a mechanical calibration kit, refer to "Performing a Mechanical SOLT Calibration" on page 137 for additional information.
   - If you selected an electronic calibration kit, refer to "Performing an Electronic Calibration" on page 144 for additional information.

Defining a SOLT Calibration Kit

A SOLT calibration kit may be defined if the calibration kits shown in Figure 5-2 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" on page 441 for additional details.

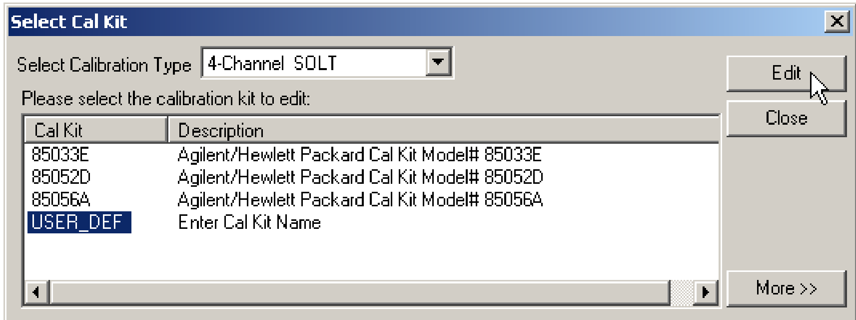
1. Exit the startup wizard by clicking the close button in the upper right corner of the wizard:
   
2. From the **Utilities** menu, select **Calibration**, **Edit CalKit** as shown in Figure 5-5.

Figure 5-5 Selecting Edit CalKit



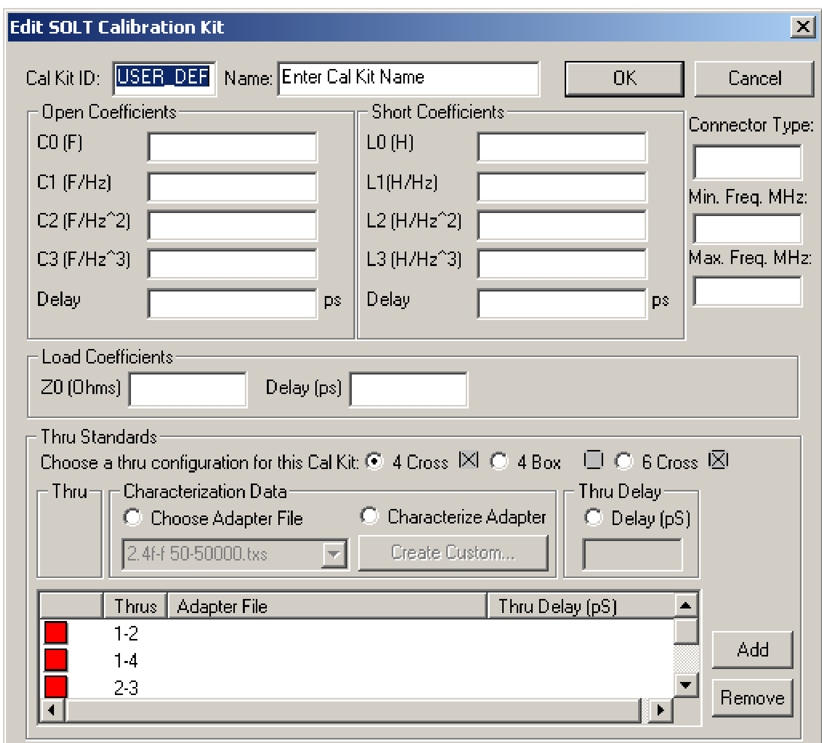
3. In the *Select Cal Kit* dialog box, select **4-Channel SOLT** from the **Select Calibration Type** list, then select the **USER_DEF** selection and click **Edit** to start the process. See Figure 5-6.

Figure 5-6 Select Cal Kit Dialog Box



4. The calibration kit definition is performed in the *Edit SOLT Calibration Kit* dialog box shown in Figure 5-7.

Figure 5-7 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 5-6, **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 5-2.

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 5-2.

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`**

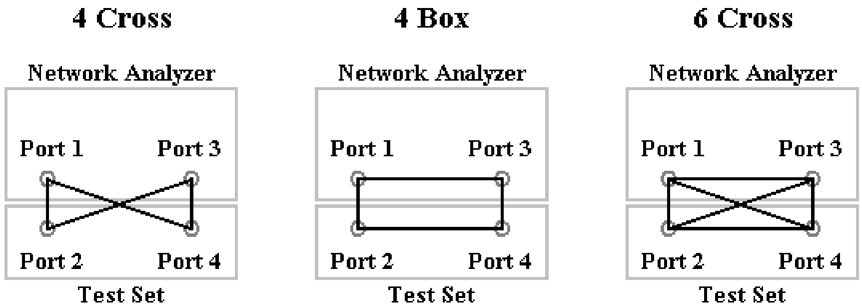
   Enter the **Delay** in picoseconds. This value may also be found in the calibration kit documentation if you are using a standard from a calibration kit.

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 5-9 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 5-8 shows the paths that will be calibrated using a thru for each selection.

Figure 5-8 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 176 for information on the **Custom Adapter Characterization Wizard**.

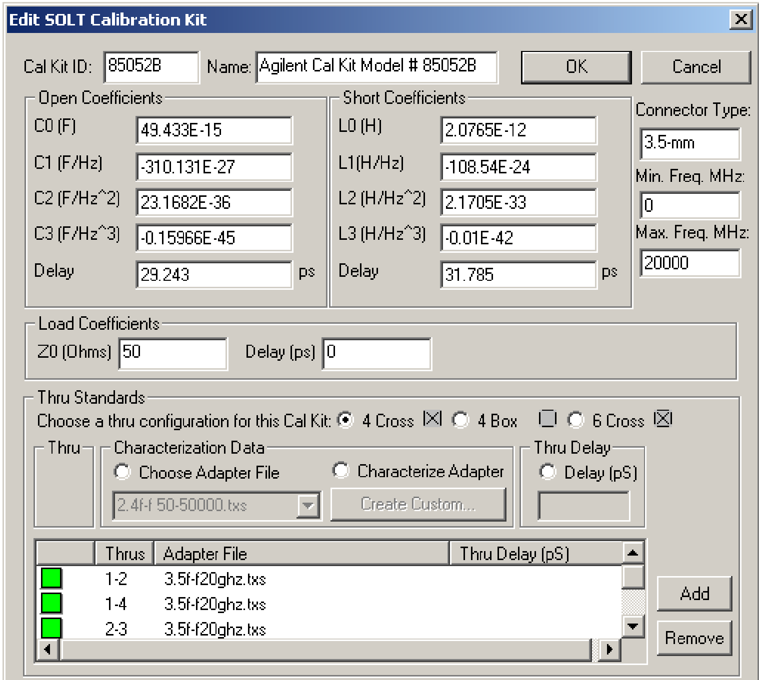
- **Delay (ps)** allows you to insert just the delay (in picoseconds) of the thru adapter, based on the electronic length of the device. If you select **Delay (ps)**, enter the time delay of the thru in the text area. If your thru is part of an Agilent (or Hewlett-Packard) calibration kit, this delay 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 5-9 Completed Edit SOLT Calibration Kit Dialog Box

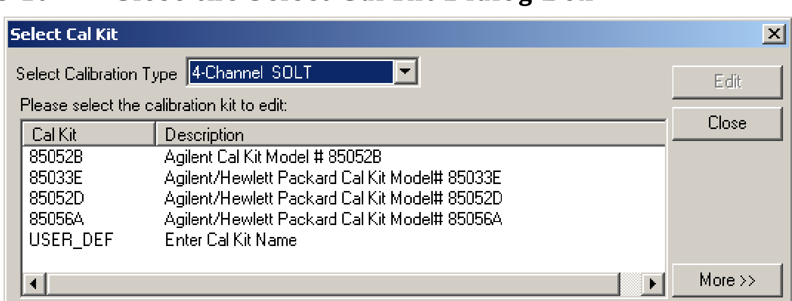


11. Once the SOLT calibration kit data has been entered into the dialog box, select **OK** to save the calibration kit data and exit the *Edit SOLT Calibration Kit* dialog box.

    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 5-10.

Figure 5-10 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 4 on page 105. If you do not need assistance, return to "Selecting a SOLT Calibration Kit" on page 129.

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.

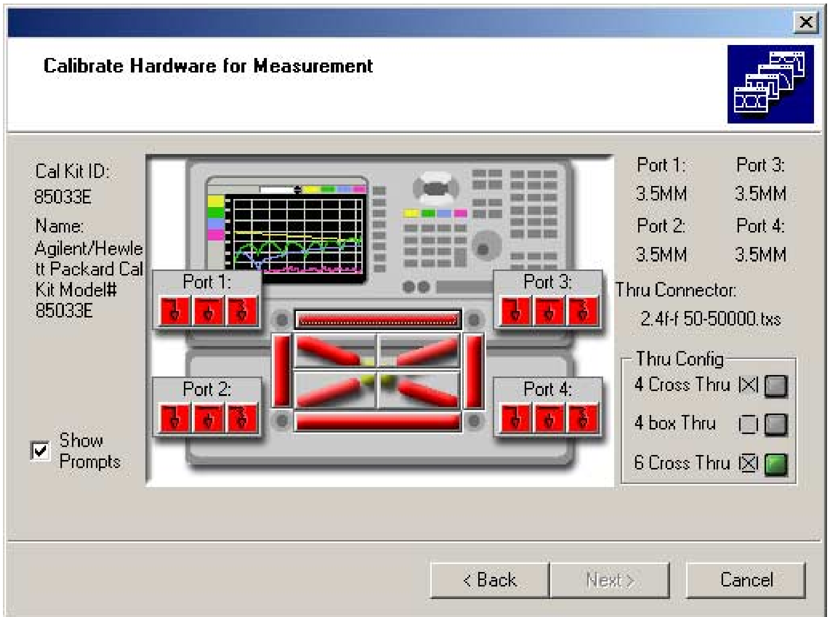
The 4-port (channel) Short/Open/Load/Thru (SOLT) calibration type is one of the most comprehensive calibrations. This calibration effectively removes the directivity error, crosstalk, source match error, load match error, frequency response reflection tracking error, and frequency response transmission 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 5-11 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 141). In the center of the screen, the selected thru configuration is displayed (see "The Thru Component" on page 138).

Figure 5-11 The SOLT Calibration Interface

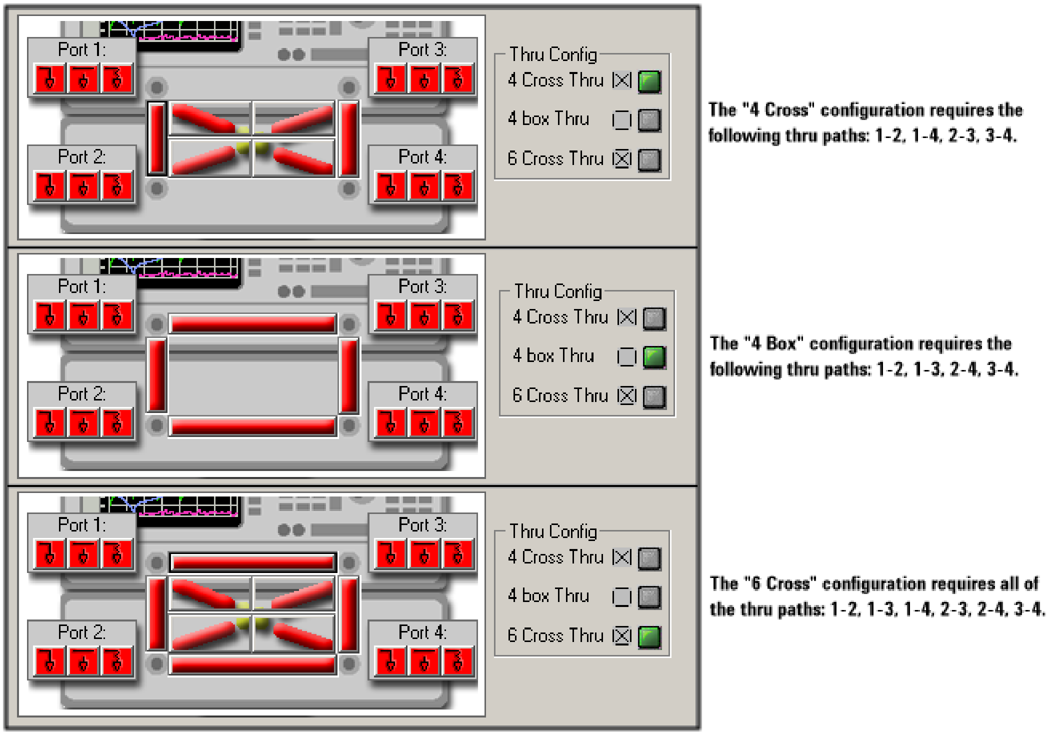


Note that all of the icons and the thru configuration paths are displayed in red at the start of the calibration. As each calibration standard is connected and measured, the color of its corresponding icon will change to blue. Once the measurement is complete, the color of the corresponding icon will 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 132.

Figure 5-12 "Thru Config" Options



The selection in the **Thru Config** area of the *Calibrate Hardware for Measurement* window (shown in Figure 5-11 on page 138) dictates the calibration pattern of the thru calibration. Figure 5-12 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).

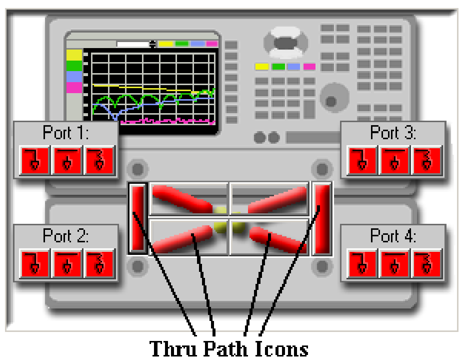
To Perform the Thru Calibration Component Use the following steps to perform the Thru calibration:

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.

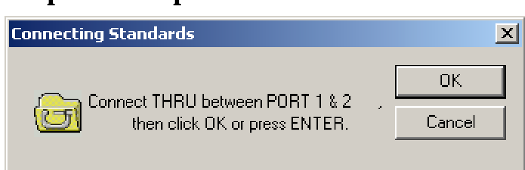
2. With the *Calibrate Hardware for Measurement* window displayed, select one of the Thru path icons on the window.

Figure 5-13 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 5-14.

Figure 5-14 Example Prompt for Thru

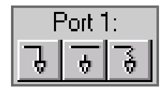


4. Continue to select the Thru icon following the instructions as each prompt is displayed. The Thru component of the calibration is complete once the color of all of the Thru paths

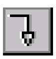
have changed to green.

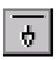
5. Continue with "The Short/Open/Load (SOL) Component" to complete the calibration.

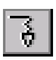
The Short/Open/Load (SOL) Component Performing the SOL portion of the calibration consists of connecting each of the standards to each of the ports for a reference measurement. Any sequence may be used, and multiple connections may be made simultaneously for convenience (for example: connect the short 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 5-11 on page 138) 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 5-15. 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.

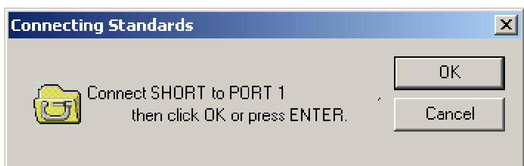
2. Select a short, open, or load standard from the calibration kit and connect it to one of the test ports. Note: You may save time by connecting more than one standard (for example: connect the short to port 1, load to port 2, and open to port 3, then continue).
3. Click the corresponding icon to perform 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 143.

---

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 5-15.

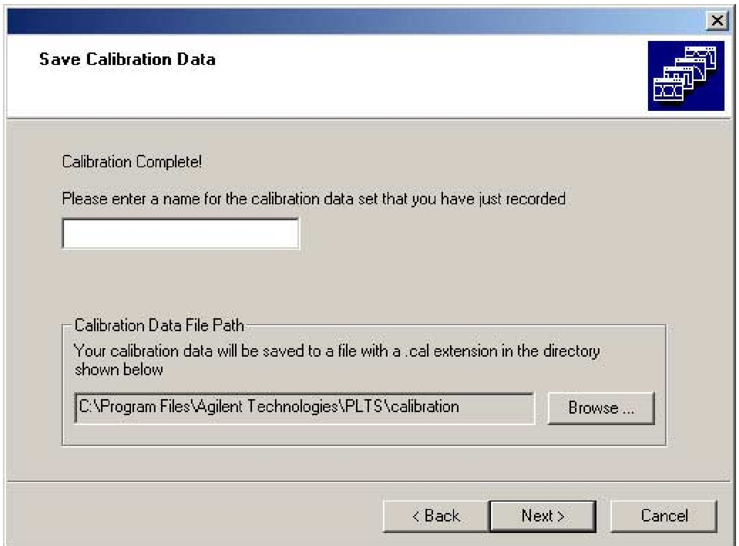
Figure 5-15 Example Prompt for Shorts, Opens, and Loads



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 143.

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 5-16.

Figure 5-16 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:

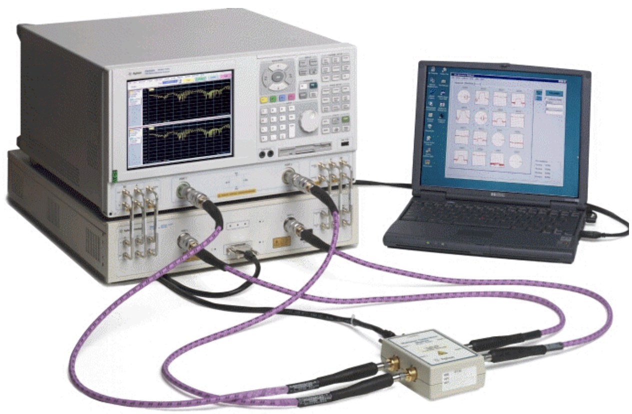
   - As part of the example measurement of Chapter 4, return to "How to Make a Measurement" on page 114.
   - By entering the startup wizard at startup or by selecting **New** from the **File** menu, the software sends you to the Startup Wizard's *Setup & Calibration Complete!* window where you can make 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 5-17 for a typical equipment setup.

Figure 5-17 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.

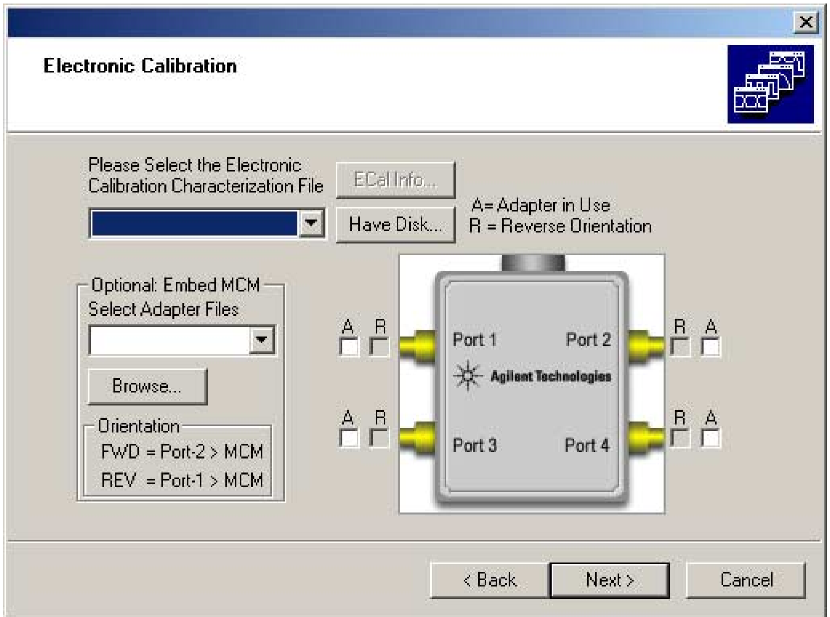
Using standard alignment precautions, the test set may have the ECal module connected and disconnected in any power state. After calibration, the ECal module may remain connected or may be disconnected from the test set Auxiliary connector with no effect on the calibration.

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 132, the *Electronic Calibration* is displayed. See Figure 5-18.

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

Figure 5-18 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.

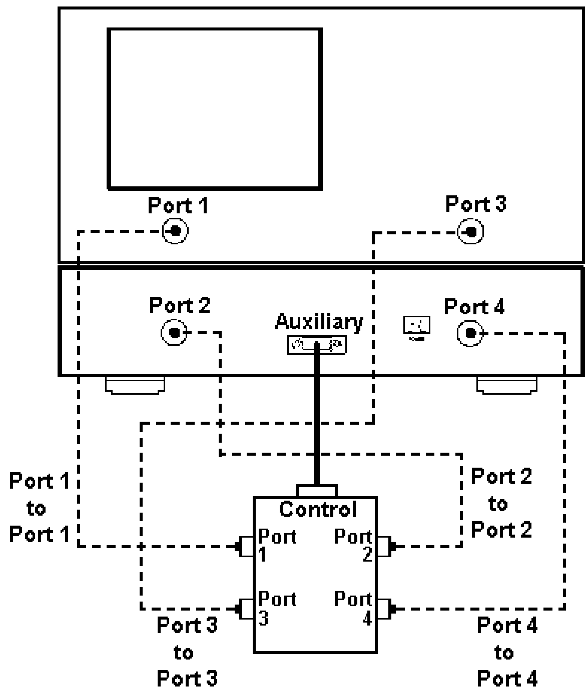
3. Connect the other end of the test cables and adapters to the ECal module so that the connections identified in Table 5-3 are made. If adapters are required to make the connections from the test cable/adapter setup to the ECal module, use the adapters in the ECal module kit.

Table 5-3 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 5-19 for test cable connections to the ECal module.

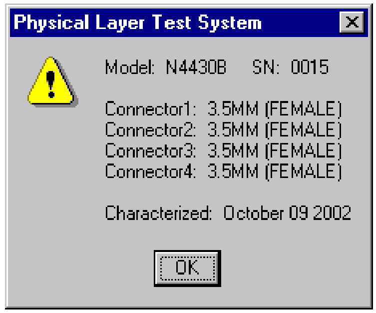
Figure 5-19 Connecting Test Cables to the ECal Module



4. Select the electronic calibration characterization file for your ECal module from the list. Refer to Figure 5-18. 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 **Have Disk** ... 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 176 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 143.

Performing a TRL Calibration

You have selected the 4-Channel TRL (THRU - REFLECT - LINE) calibration type. TRL 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 TRL 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 TRL 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 T, R, and L standards are less stringent and the standards are more easily fabricated.

The TRL calibration kit contains the following:

- zero length THRU
- “flush” short for the REFLECT standard (0 second offset)
- 50-ohm terminations with known delay for the LINE

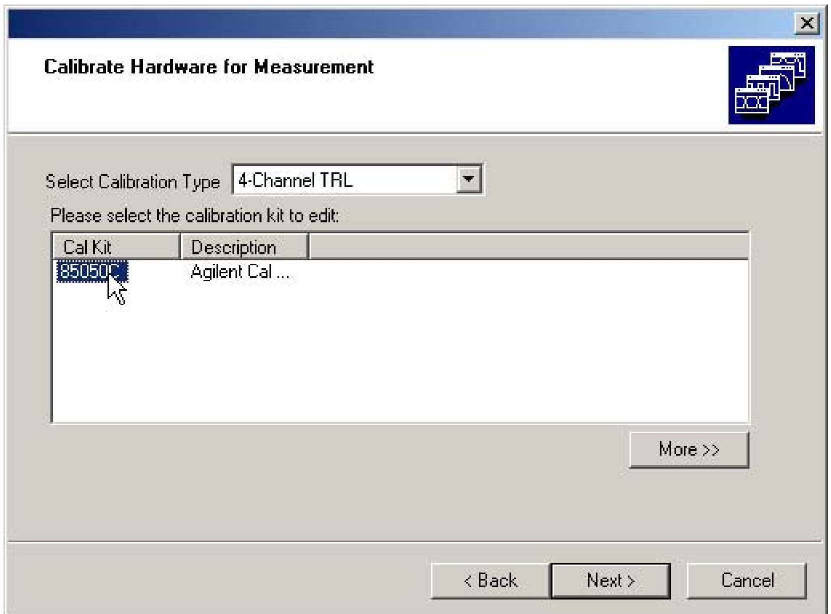
NOTE TRL calibration is not supported for physical layer test systems that use the 8753ES or the 8720ES-series network analyzers.

Start with “Selecting a TRL Calibration Kit”.

Selecting a TRL Calibration Kit

The dialog box shown in Figure 5-20 lists the defined TRL 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 5-20 Select TRL Calibration Kit Dialog Box



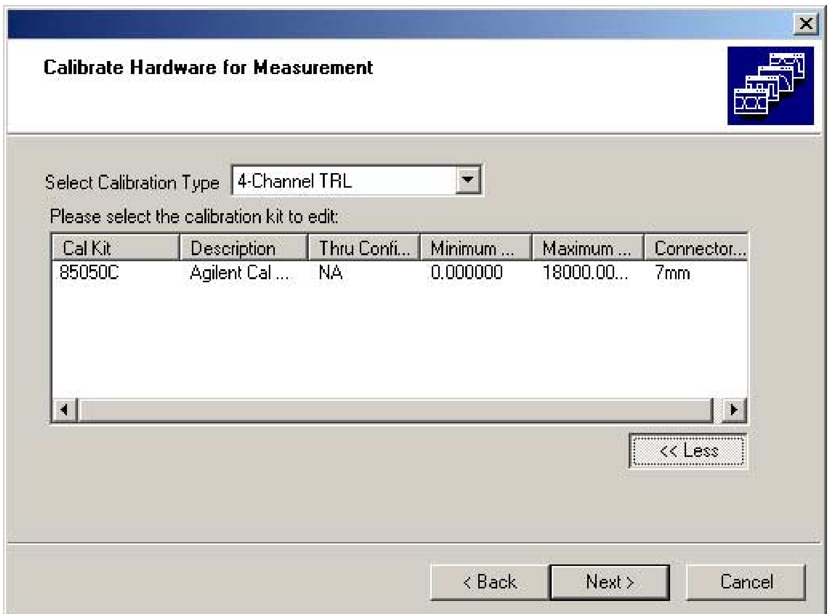
NOTE No TRL calibration kits are defined in the software when you receive it.

- 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 150.
- 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 127.
  2. Make sure that the start and stop frequencies that were defined for the TRL calibration kit ("Defining a TRL Calibration Kit" on page 150) match or exceed the start and stop frequencies that were defined in the

measurement parameters ("Setting Up the Calibration and Measurement Parameters" on page 111).

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 5-21.

Figure 5-21 Select Calibration Kit Dialog Box with More Button Selected



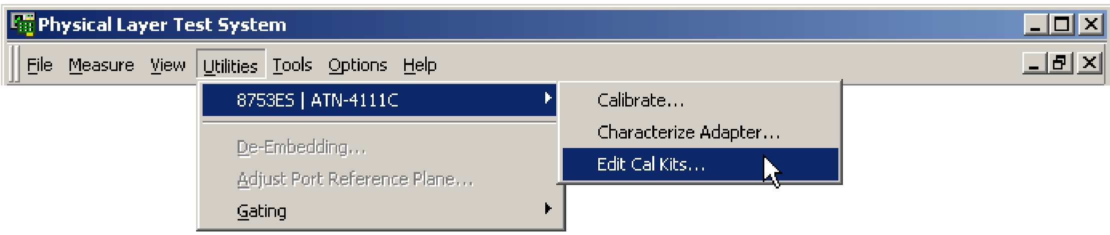
3. Once the calibration kit is selected, click the **Next >** button to display the calibration window and continue with "Performing a TRL Calibration" on page 154.

Defining a TRL Calibration Kit

The dialog box shown in Figure 5-23 shows only the **USER_DEF** selection. This indicates that a TRL calibration kit must be defined before a TRL calibration may be performed.

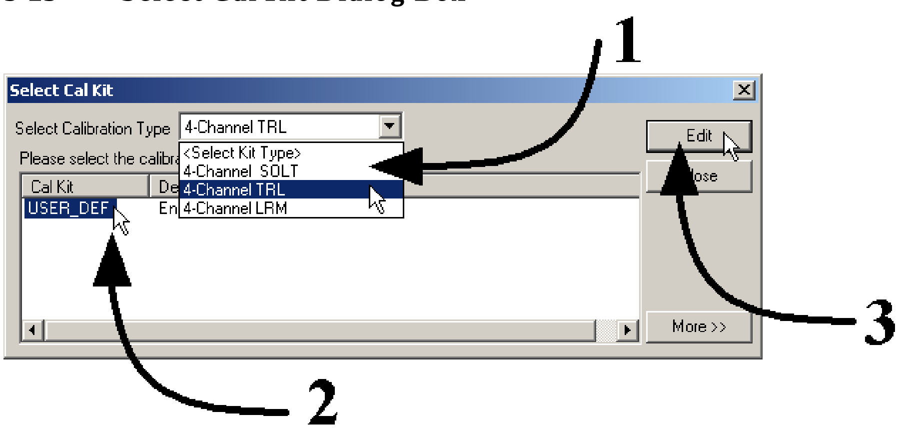
1. From the **Utilities** menu, select your PLTS system name and then select **Edit Cal Kits**... as shown in Figure 5-22.

Figure 5-22 Selecting Edit Cal Kits…



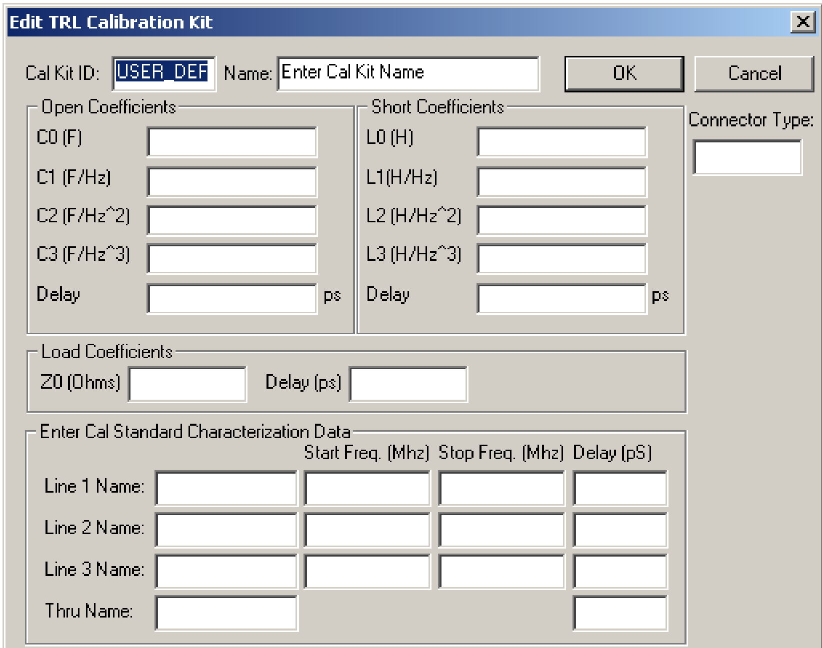
2. In the *Select Cal Kit* dialog box, select **4-Channel TRL** from the **Select Calibration Type** list, then select the **USER_DEF** selection and click **Edit** to start the definition process. See Figure 5-23.

Figure 5-23 Select Cal Kit Dialog Box



3. The TRL calibration kit is defined using the *Edit TRL Calibration Kit* dialog box. See Figure 5-24.

Figure 5-24 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 5-25, **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 5-21.

5. Enter the delay value of the device you are using as the Reflect in either the **Open Coefficients** area or the **Short Coefficients** area. Enter the delay value in only one area for your Reflective device. If you are using a standard from a calibration kit, the delay value may be found in the calibration kit documentation.

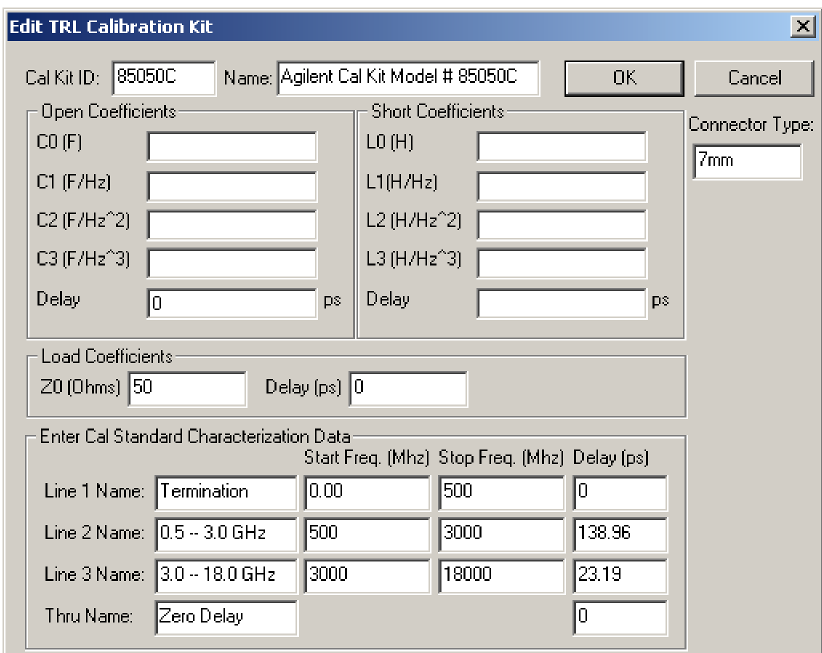
   - If the Reflect device is an open, enter the **Delay** (in picoseconds) in the **Open Coefficients** area.
   - If the Reflect device is an short, enter the **Delay** (in picoseconds) in the **Short Coefficients** area.

   Figure 5-25 shows the data that is used in this example.

6. 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. Figure 5-25 shows the data that is used in this example.

7. Complete of the *Edit TRL Calibration Kit* dialog box. For the **Enter Cal Standard Characterization Data** 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 three 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 5-25 shows the data that is used in this example.

Figure 5-25 Completed Edit TRL Calibration Kit Dialog Box

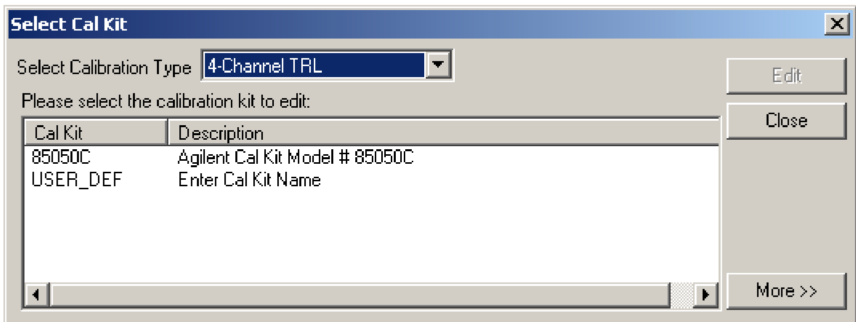


8. Once the TRL calibration kit data has been entered into the dialog box, select the **OK** button to save the calibration kit data and exit the *Edit TRL Calibration Kit* dialog box.

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

9. Once the TRL calibration kit data has been saved, select **Close** to close the *Select Cal Kit* dialog box. See Figure 5-26.

Figure 5-26 Close the Select Cal Kit Dialog Box



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

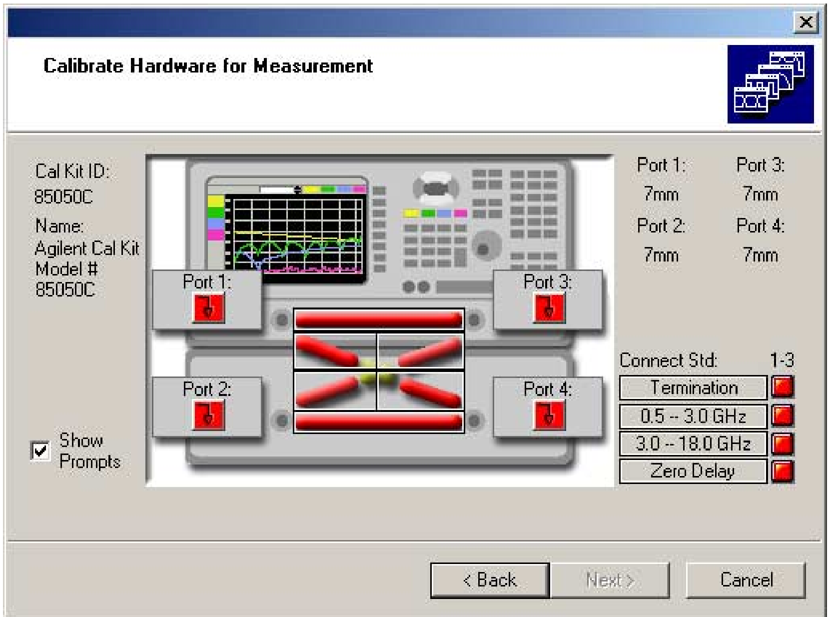
- If you need help completing the startup wizard, return to Chapter 4 on page 105.
- If you do not need assistance, continue with "Selecting a TRL Calibration Kit" on page 149.

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 5-27 Initial TRL Calibration Display



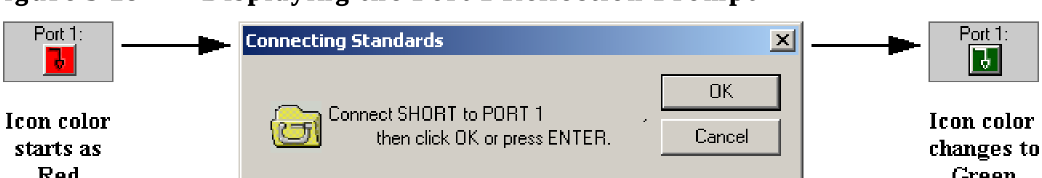
The TRL calibration is performed in two parts. First the Reflect portion of the calibration is done. Then the Thru-Line portion of the calibration is done.

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 5-28, connect the reflective device (in this case, defined as a short) from your calibration kit to Port 1 and click **OK**.

Figure 5-28 Displaying the Port 1 Reflection Prompt



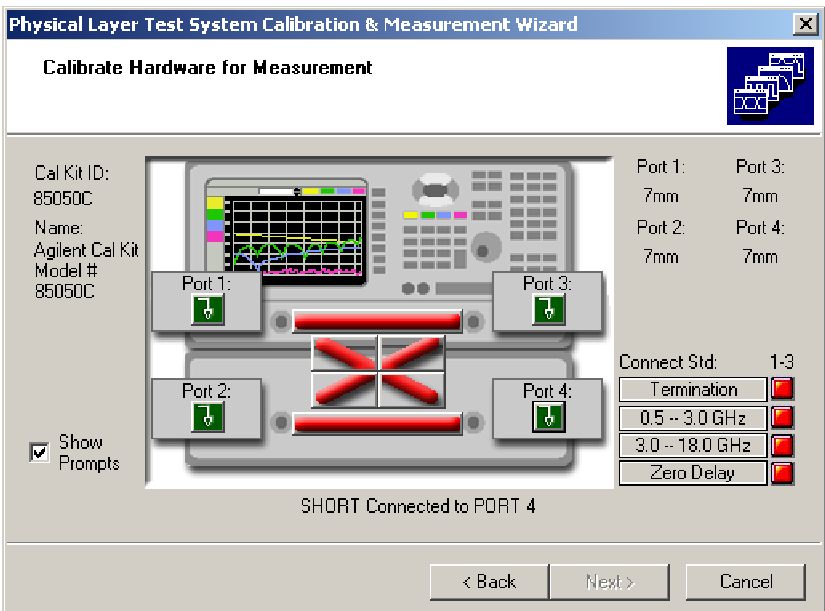
As the portion of the calibration is complete, the Port 1 icon color changes from red to green. Actually, as the measurement is being made, the icon is blue, however, the measurement is so quick you may not even see the icon turn blue before turning to green.

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 5-29.

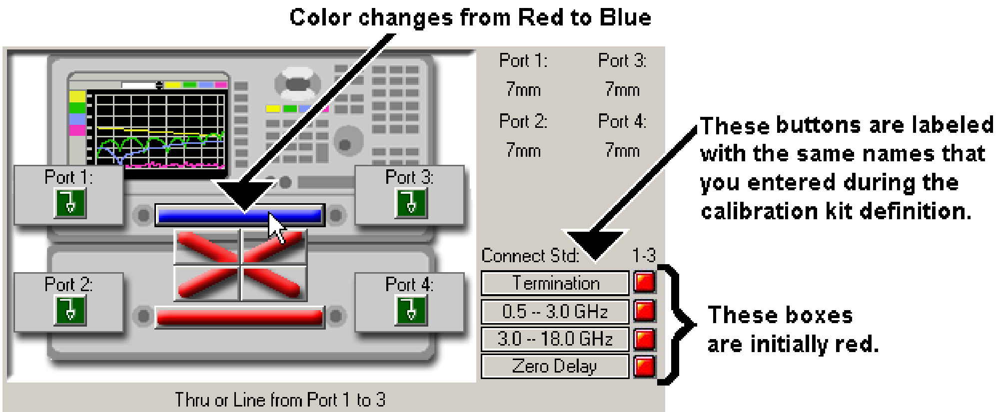
Figure 5-29 Reflection Portion of Calibration Complete



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 5-30.

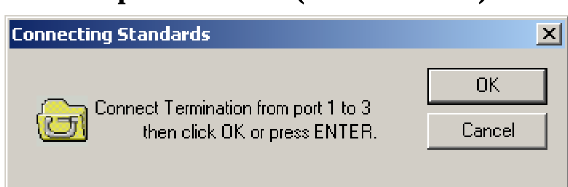
Figure 5-30 Start Thru-Line Calibration Port 1 - Port 3



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 5-25 on page 153. 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 5-31 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.

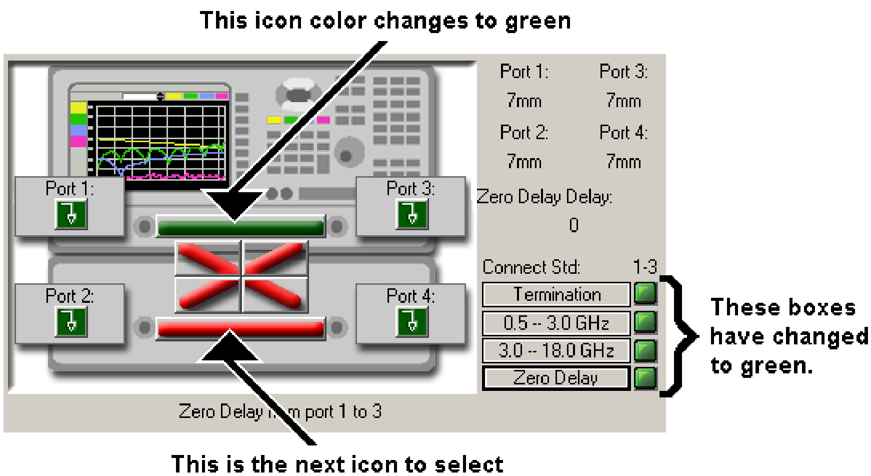
4. Repeat step 2 and 3 for Lines 2 (**0.5 -- 3.0GHz**), Line 3 (**3.0 - 18GHz**), and the Thru (**ZeroDelay**).

   Once these measurements are taken, the calibration between ports 1 and 3 is complete. See Figure 5-32. On the display, the color of:

   - Each box to the right of the buttons have changed from red to green.

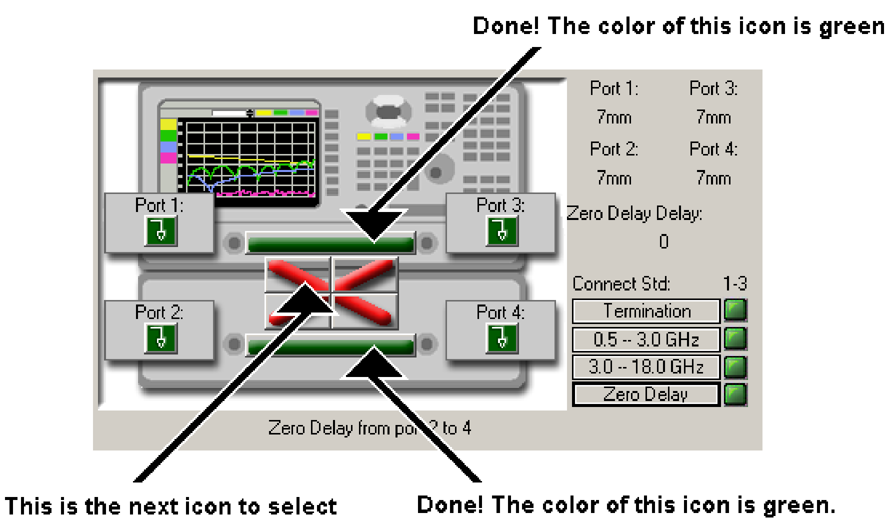
- The line between ports 1 and 3 changes from blue to green indicating that this path is complete.

Figure 5-32 Port 1 - Port 3 Line Portion of Calibration Complete



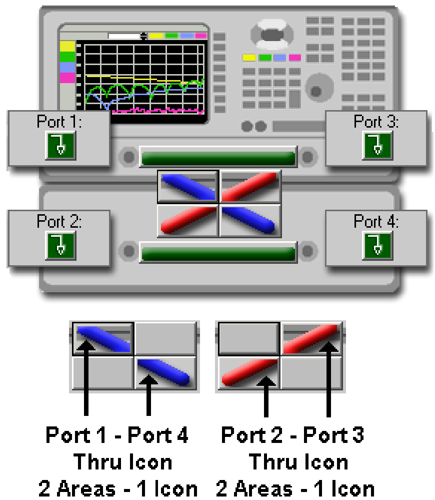
5. Repeat steps 1 through 4 for the Port 2 to Port 4 path.

Figure 5-33 Line Portion of Calibration Complete



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

Figure 5-34 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.

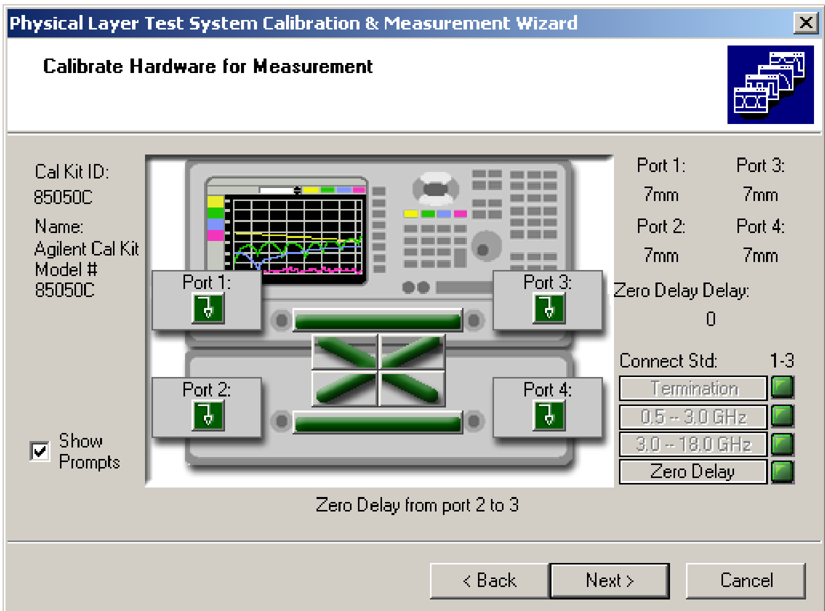
8. After the thru is connected between ports 1 and 4 as indicated on the prompt and click the **OK** button.

   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 5-35.

Figure 5-35 TRL Calibration Complete

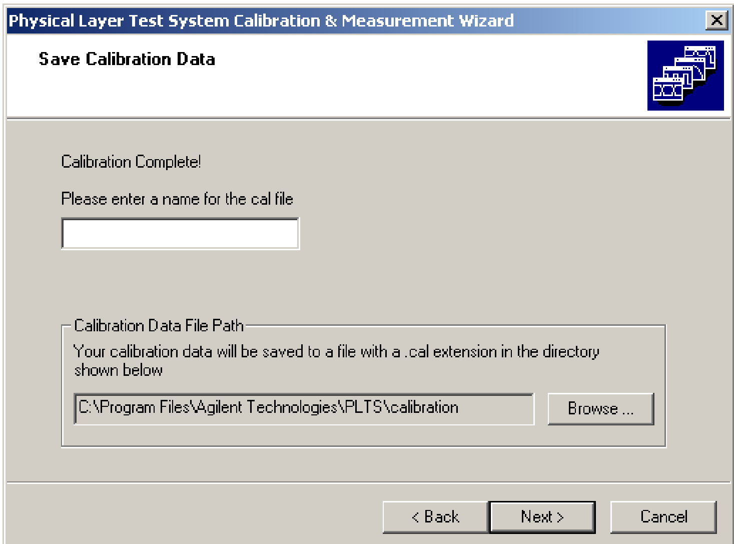


11. Click the **Next >** button.

Continue at “To Save the TRL Calibration” on page 161.

To Save the TRL Calibration Once you have completed the TRL calibration, the *Save Calibration Data* dialog box is displayed as shown in Figure 5-36.

Figure 5-36 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 4, return to "How to Make a Measurement" on page 114.
   - 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).

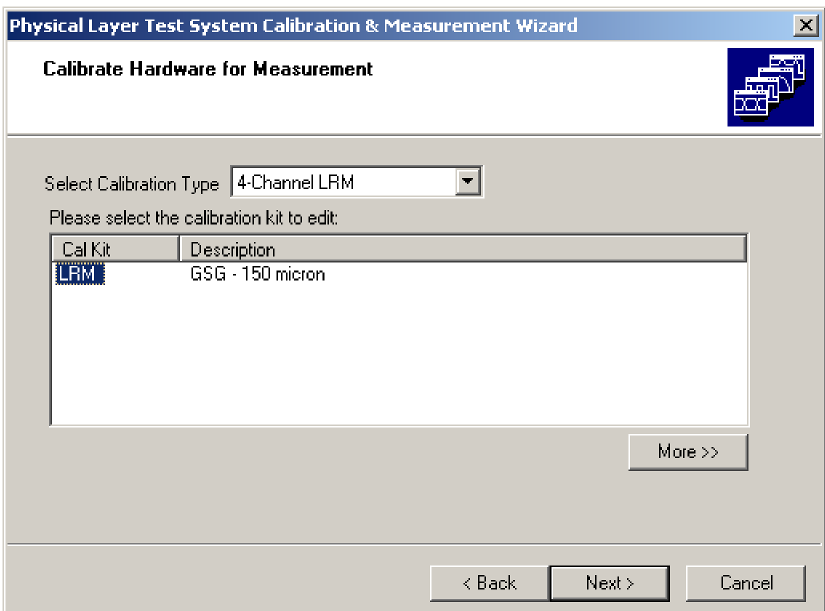
NOTE LRM calibration is not supported for physical layer test systems that use the 8753ES or the 872XES network analyzers.

Start with “Selecting an LRM Calibration Kit”.

Selecting an LRM Calibration Kit

The dialog box shown in Figure 5-37 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 5-37 Select LRM Calibration Kit Dialog Box



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

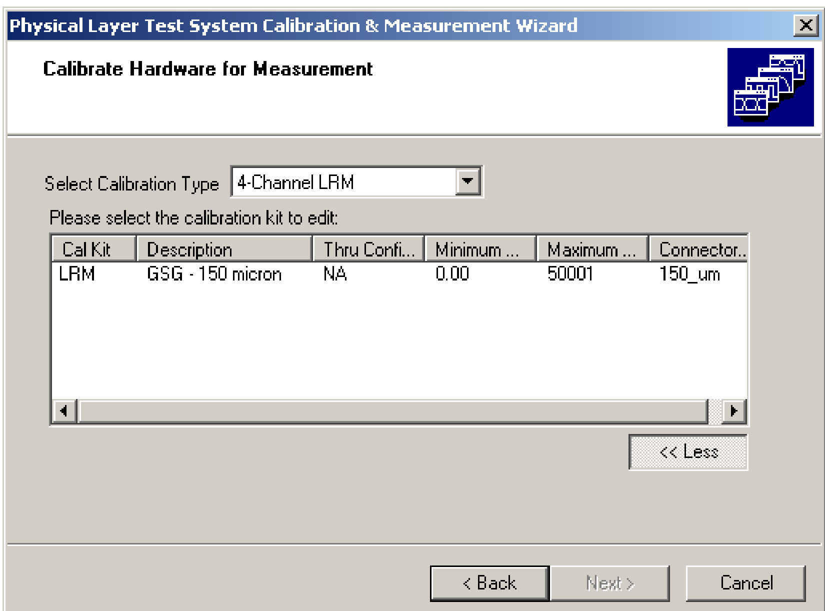
- If you *have not* defined your LRM calibration kit, select the **Cancel** button and define your LRM calibration kit using the procedure described in “Defining an LRM Calibration Kit” on page 165.
- If you *have* defined a LRM calibration kit but it is not displayed in the dialog box:

1. Make sure that you have selected **4-Channel LRM** from the **Select Calibration Type** list. See “Selecting a Calibration Type” on page 127.
2. Make sure that the start and stop frequencies that were defined for the LRM calibration kit (“Defining an LRM Calibration Kit” on page 165) match or exceed the start and stop frequencies that were defined in the measurement parameters (“Setting Up the Calibration and Measurement Parameters” on page 111).

---

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 5-38.

Figure 5-38 Select Calibration Kit Dialog Box with More Button Selected



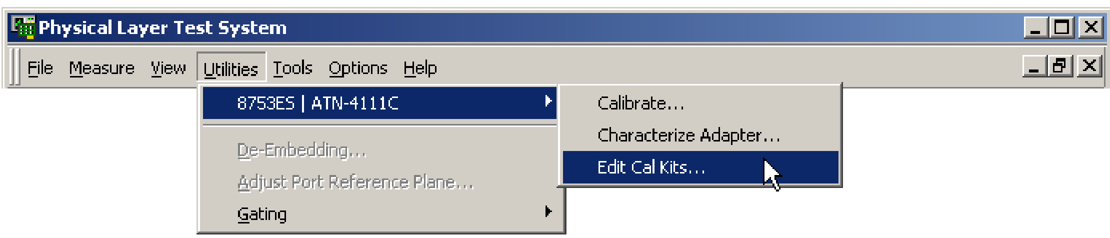
3. Once the calibration kit is selected in the **Cal Kit** column, click the **Next >** button to display the calibration window and continue with “Performing an LRM Calibration” on page 168.

Defining an LRM Calibration Kit

The dialog box shown in Figure 5-40 shows only the **USER_DEF** selection. This indicates that a LRM calibration kit must be defined before a LRM calibration may be performed.

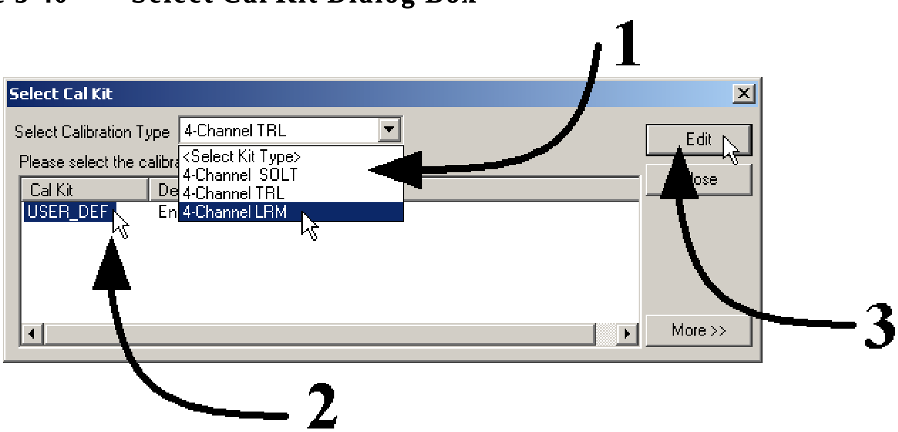
1. From the **Utilities** menu, select your PLTS system name and then select **Edit Cal Kits**... as shown in Figure 5-39.

Figure 5-39 Selecting Edit Cal Kits...



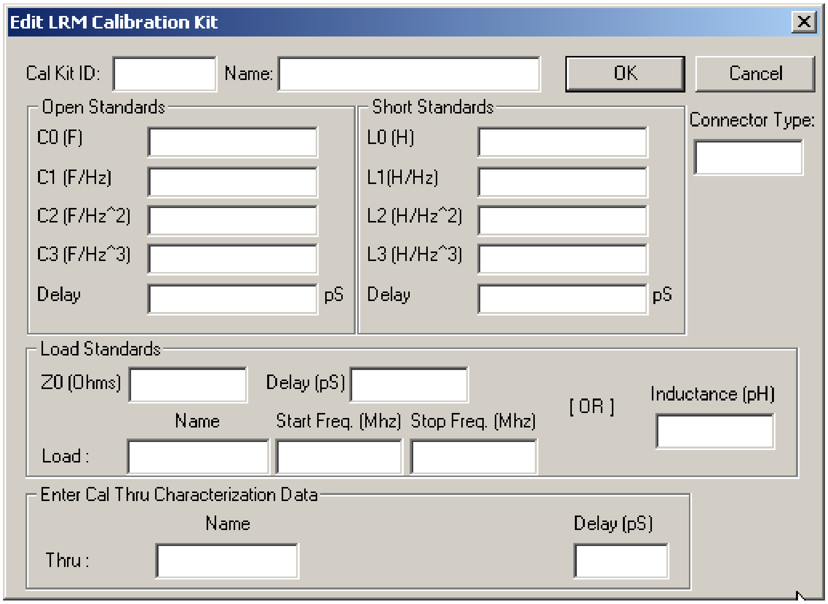
2. In the *Select Cal Kit* dialog box, select **4-Channel LRM** from the **Select Calibration Type** list, then select the **USER_DEF** selection and click **Edit** to start the definition process. See Figure 5-40.

Figure 5-40 Select Cal Kit Dialog Box



3. The LRM calibration kit is defined using the *Edit LRM Calibration Kit* dialog box. See Figure 5-41.

Figure 5-41 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 5-42, **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 5-38.

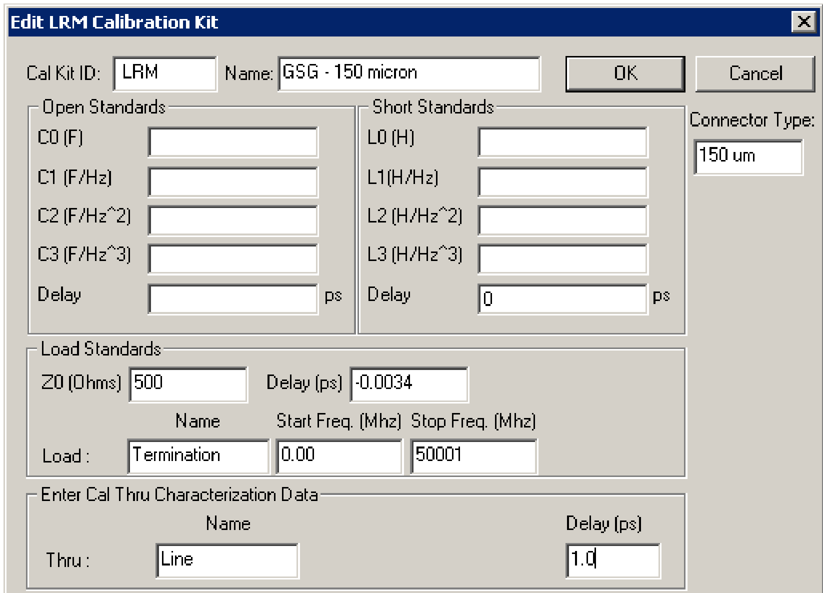
5. For the Reflect: Enter the delay value (in picoseconds) of the short that you are using in the **Short Coefficients** area. Figure 5-42 shows the data that is used in this example.

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 5-38 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 5-42 shows the data that is used in this example.

Figure 5-42 Completed Edit LRM Calibration Kit Dialog Box

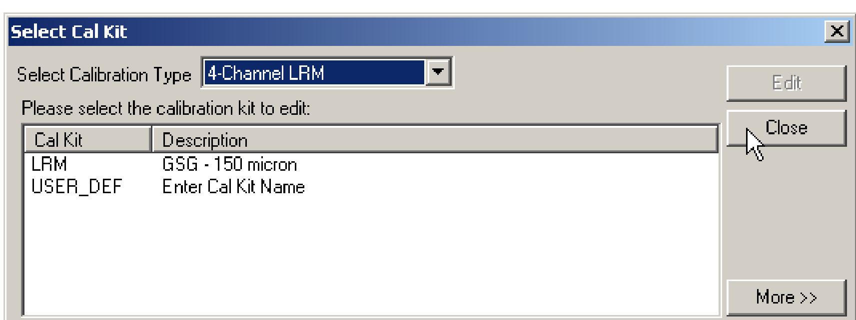


8. Once the LRM calibration kit data has been entered into the dialog box, select the **OK** button to save the calibration kit data and exit the *Edit LRM Calibration Kit* dialog box.

   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 5-43.

Figure 5-43 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.

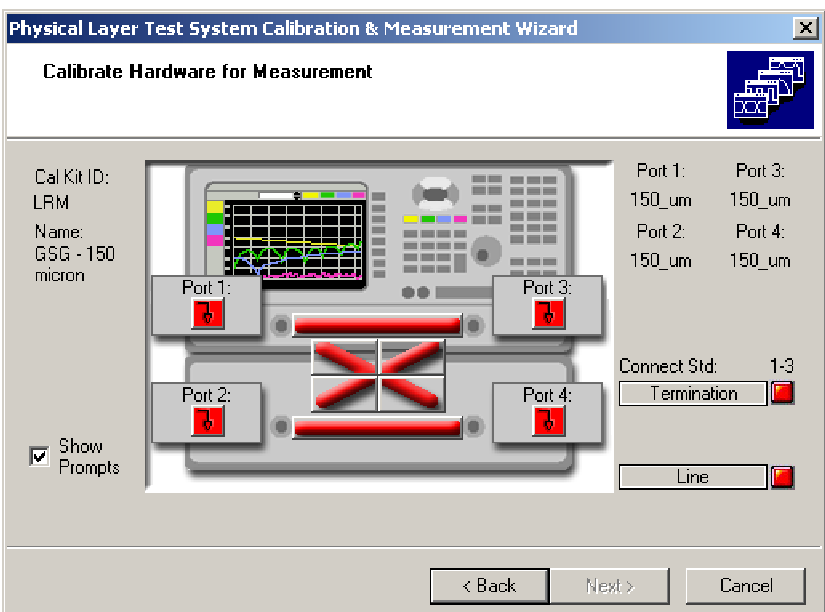
    - If you need help completing the startup wizard, return to Chapter 4 on page 105.
    - If you do not need assistance, continue with "Selecting an LRM Calibration Kit" on page 163.

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 5-44 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 5-45, connect the reflective device (in this case, defined as a short) from your calibration kit to Port 1 and click **OK**.

   Figure 5-45 Displaying the Port 1 Reflection Prompt

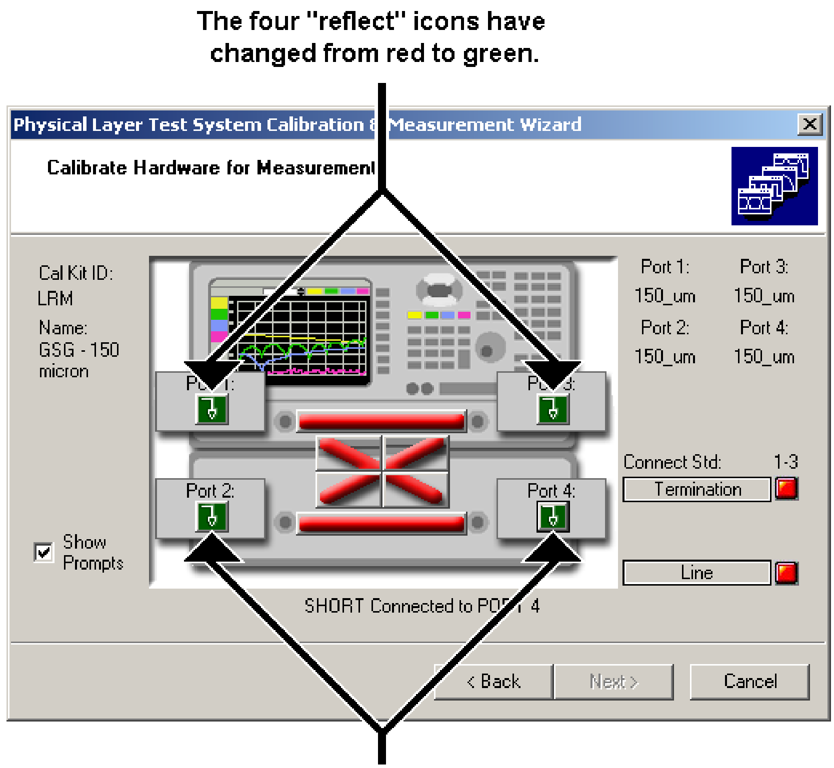
   As the portion of the calibration is complete, the Port 1 icon color changes from red to green. Actually, as the measurement is being made, the icon is blue, however, the measurement is so quick you may not even see the icon turn blue before turning to green.

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 5-46.

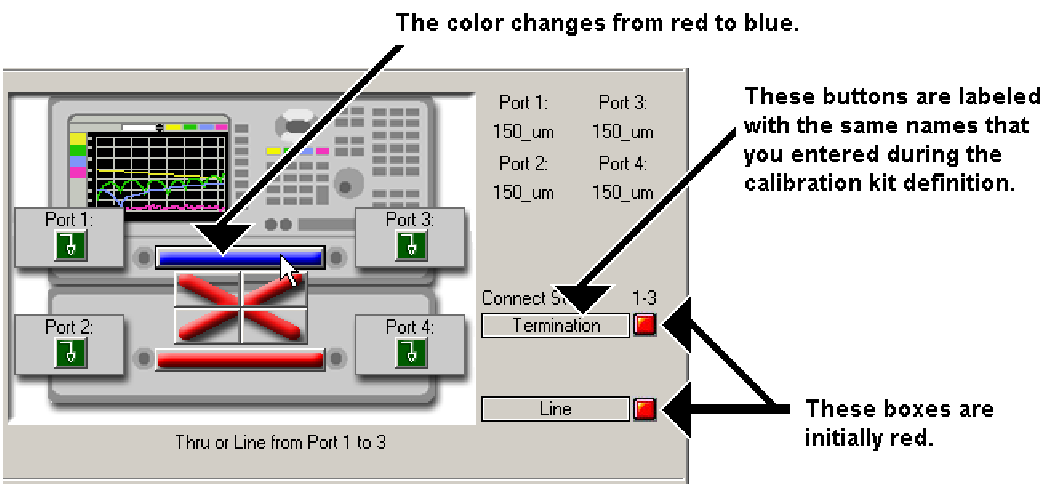
Figure 5-46 Reflection Portion of Calibration Complete



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 5-47.

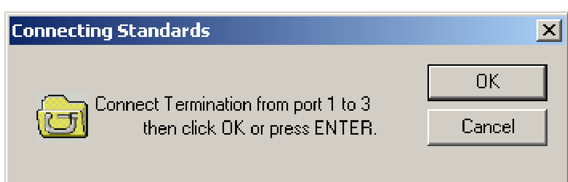
Figure 5-47 Start Thru - Line Calibration Port 1 - Port 3



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 5-42 on page 167. 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 5-48 Prompt for Load (Termination) Calibration



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

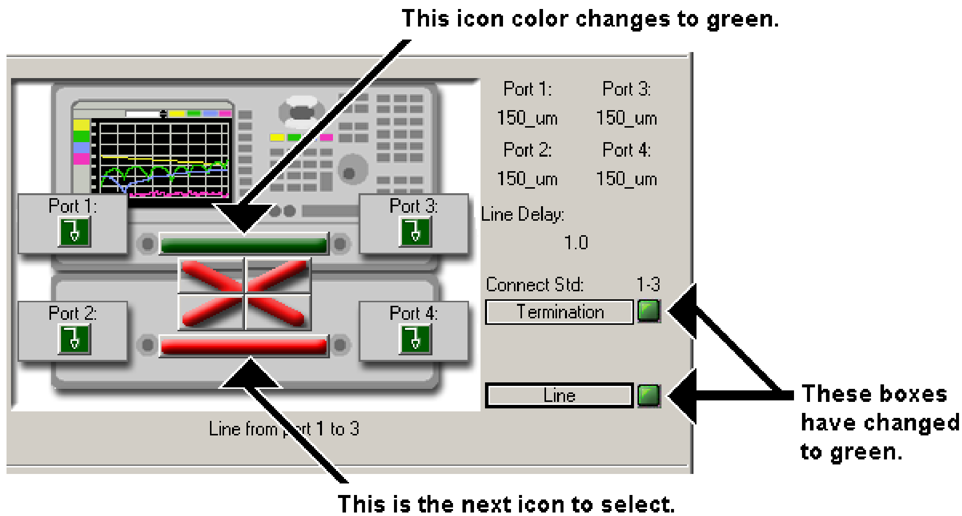
   Once this measurement is made, the color of the box at the right of the **Termination** button changes from red to green.

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 5-49. On the display, the color of:

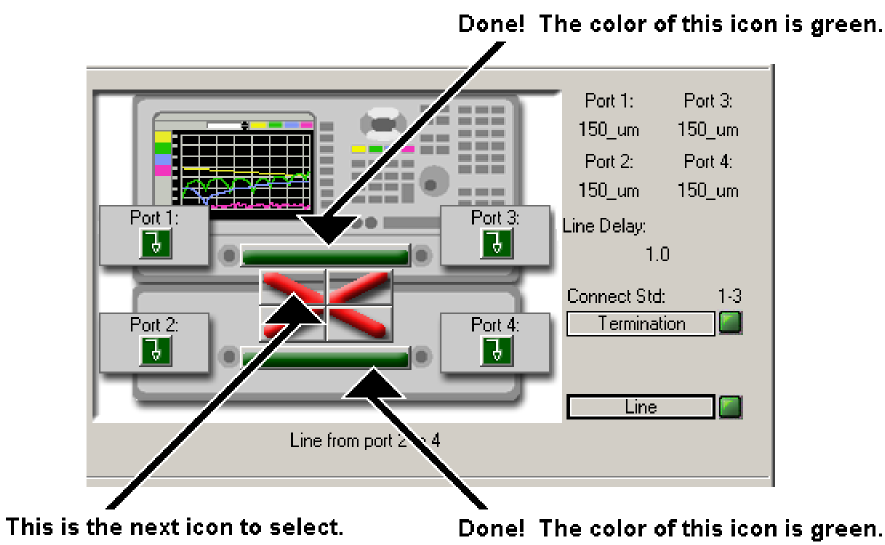
- Each box to the right of the buttons have changed from red to green.
- The line between ports 1 and 3 changes from blue to green indicating that this path is complete.

Figure 5-49 Port 1 - Port 3 Line Portion of Calibration Complete



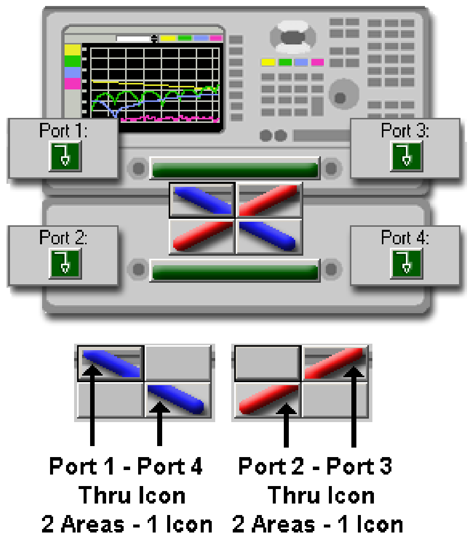
5. Repeat steps 1 through 4 for the Port 2 to Port 4 path.

Figure 5-50 Line Portion of Calibration Complete



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

Figure 5-51 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.

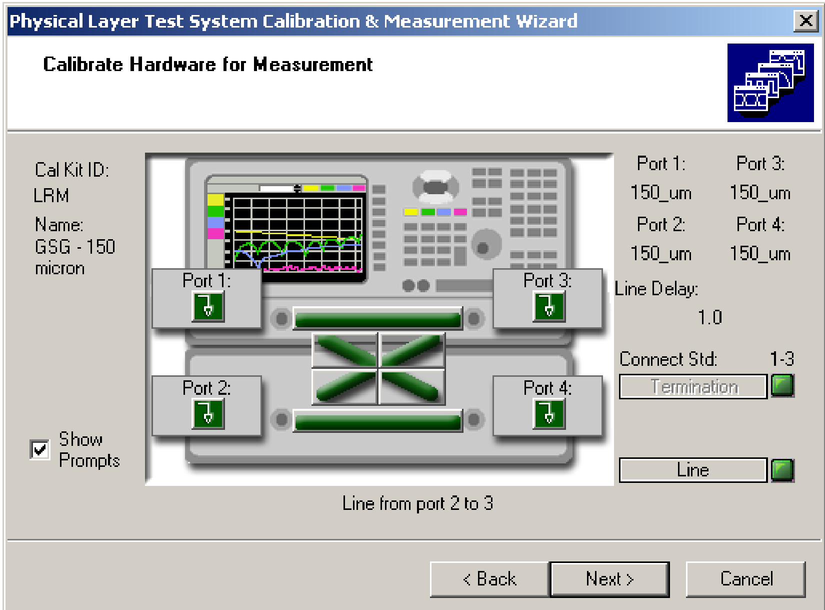
   The calibration is performed and the color of the box to the right of the thru button and the thru icon are changed to green.

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 5-52.

    The color of all icons has changed to green.

Figure 5-52 LRM Calibration Complete

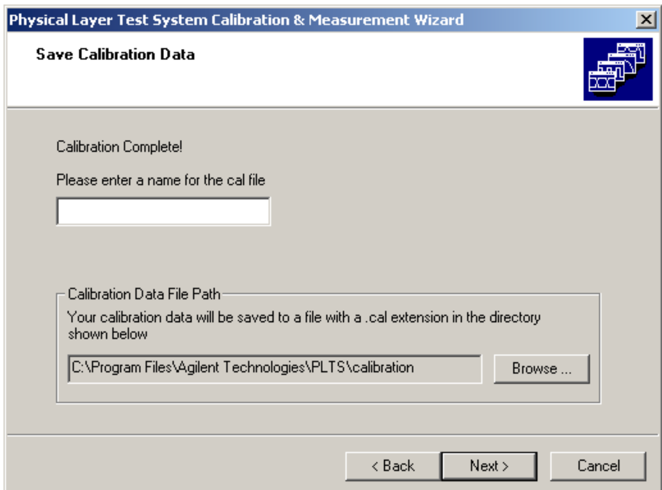


11. Click the **Next >** button.

Continue at “To Save the LRM Calibration” on page 175.

To Save the LRM Calibration Once you have completed the LRM calibration, the *Save Calibration Data* dialog box is displayed as shown in Figure 5-53.

Figure 5-53 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 4, return to "How to Make a Measurement" on page 114.
   - 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.

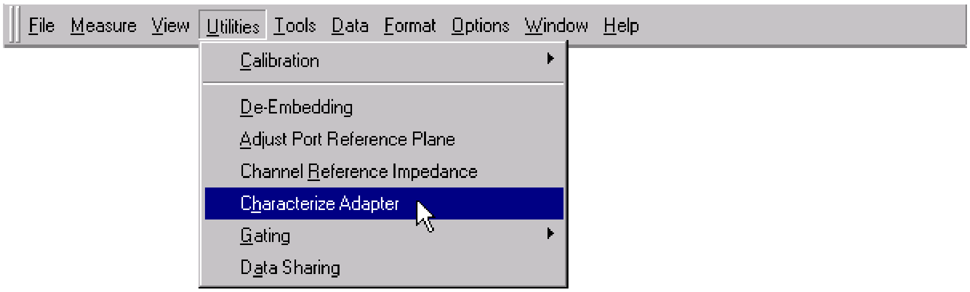
Follow these steps to characterize your adapters:

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

---

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

Figure 5-54 Characterize Adapter Selection

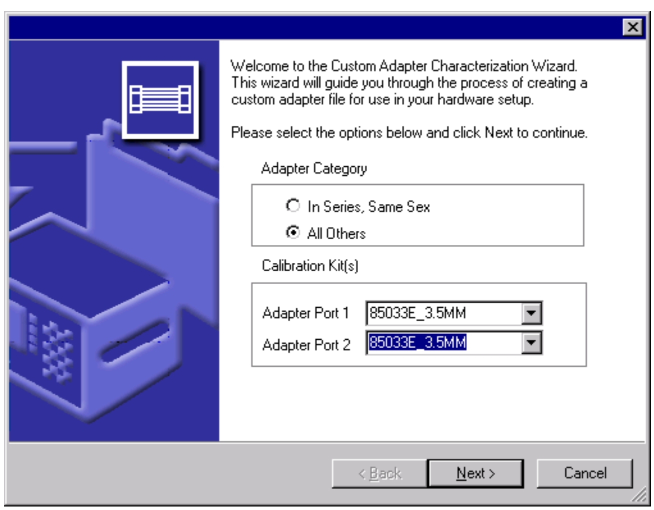


2. In the *Custom Adapter Characterization Wizard* dialog box, select the appropriate adapter category and the calibration kit to be used. See Figure 5-55. Press **Next >** to continue to the next window.

   The two adapter characterization categories are:

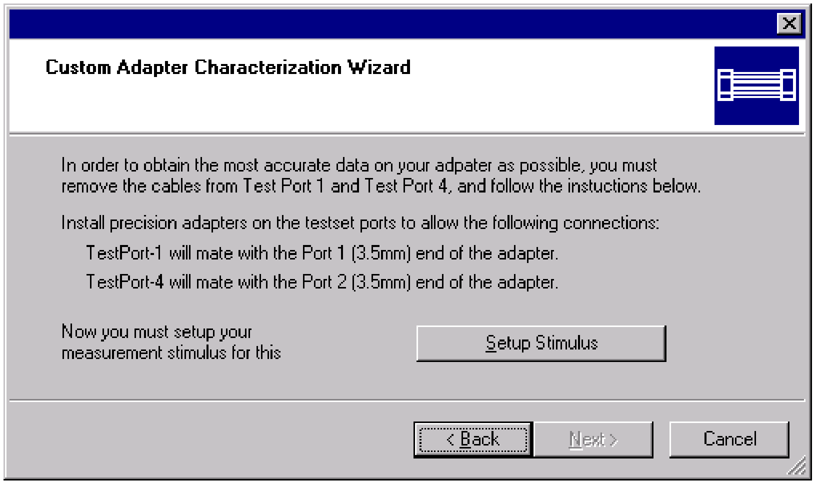
   - **"In-Series, Same-Sex"** uses a single port (Port 1) and applies only to adapters having the same type and gender on both ends (3.5 mm male-to-male, for example). Select the calibration kit to be used on Port 1 from the pull-down menu.
   - **"All Others"** uses two ports (Ports 1 and 2) and allows 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 5-55 Custom Adapter Characterization Wizard



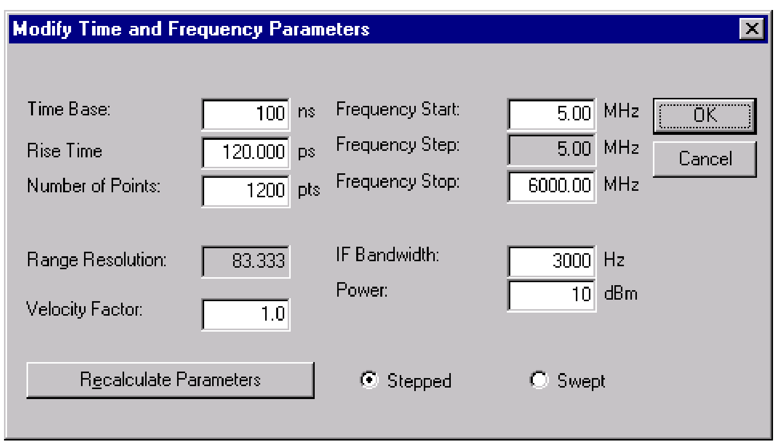
3. Make the connections listed on the Custom Adapter Characterization Wizard Setup Instructions window (Figure 5-56) and then click the **Setup Stimulus** button.

Figure 5-56 Custom Adapter Characterization Wizard Setup Instructions



4. Make any required changes to the stimulus to match your actual measurement conditions. When you make any change, click the **Recalculate Parameters** button to update all parameters. Refer to Figure 5-57. When you are satisfied with the parameters, click **OK**.

Figure 5-57 Stimulus Parameter Dialog Box



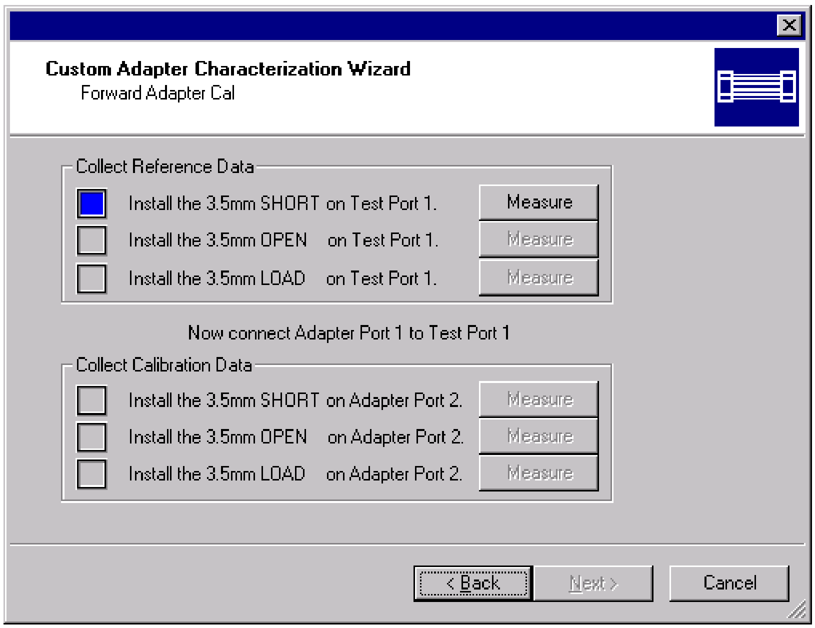
Refer to "Measurement and Calibration Parameter Entry Descriptions" on page 112 for

definitions of each parameter.

5. Following the order shown on your *Forward Orientation Adapter Calibration* dialog box, install the calibration standards as listed and click the **Measure** button.

   Depending on your previous selections, your window may differ slightly from the window shown in Figure 5-58.

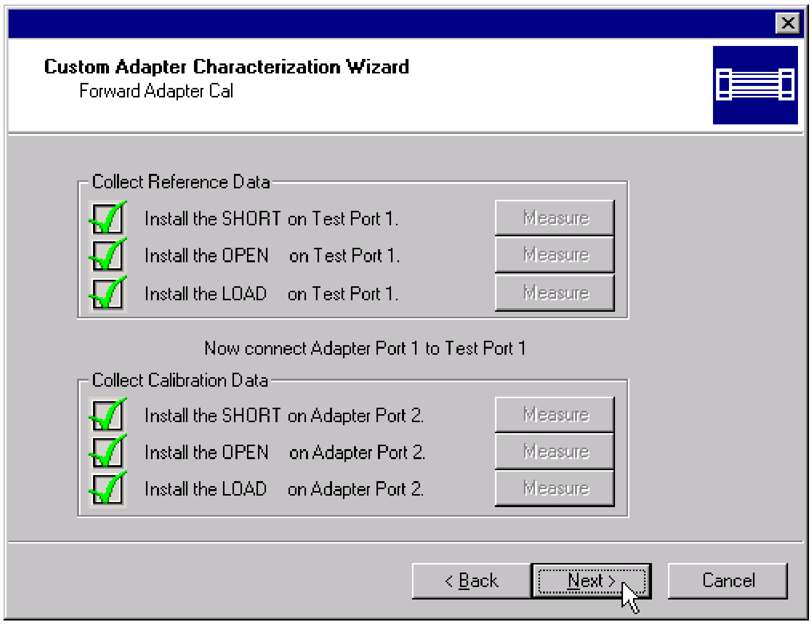
Figure 5-58 Forward Orientation Adapter Calibration Window



NOTE The **Next >** button remains inactive until all data choices are measured.

6. As you make each measurement, a check mark is displayed in the check box to the left of the measurement. See Figure 5-59. Once all measurements are made, click **Next >** to continue.

Figure 5-59 Completed Forward Orientation Adapter Calibration Window

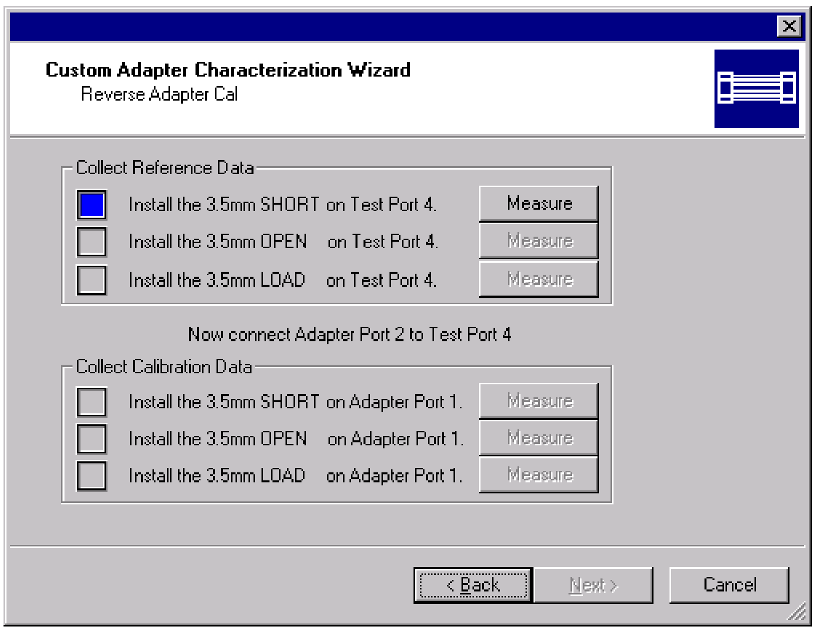


7. Following the order shown on your *Reverse Orientation Adapter Calibration* window, install the calibration standards as listed and click the **Measure** button. Depending on your previous selections, your window may differ slightly from the window shown in Figure 5-60.

   As you make each measurement, a check mark is displayed in the check box to the left of the measurement. Once all measurements are made, click **Next >** to continue.

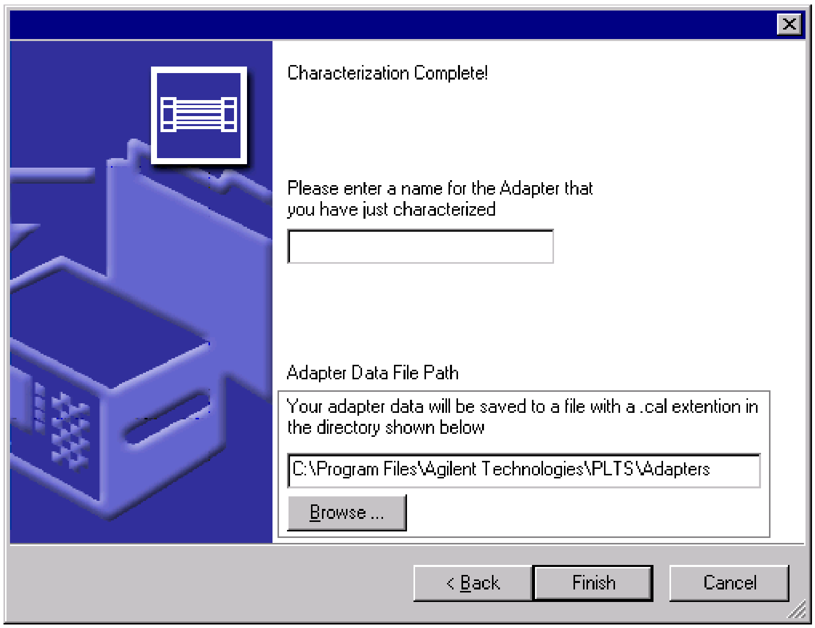
NOTE The **Next >** button remains inactive until all data choices are measured.

Figure 5-60 Reverse Orientation Adapter Calibration Window



8. When the characterization is complete, save the adapter file using the dialog box shown in Figure 5-61. 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 5-61 Save Adapter Characterization File Dialog Box



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.

6 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.

NOTE PLTS does not support TDR-based measurements using probing techniques.

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

- For the Agilent 86100A/B, we recommend you use:
  - — Four 3.5 mm (male-female) cables (such as the Agilent N4418A-B20)
    -or-
  - — Four 3.5 mm (male-male) cables 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
    -or-
  - — Four 3.5 mm (male-female) cables (such as the Agilent N4418A-B20)

You also need the following calibration standards to perform the TDR calibration. As Table 6-1 shows, different standards are required depending on the TDR-based PLTS hardware and the desired calibration type.

Table 6-1 Required Calibration Standards

TDR System		Calibration For Single Ended Measurements		Calibration for Differential Measurements	
	Module Calibration	Reference Plane Cal Only	Normalization plus Reference Plane Cal	Reference Plane Cal Only	Normalization plus Reference Plane Cal
Agilent 86100A/B	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

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
   - Time Domain 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.

- **Frequency Domain.** The Frequency Domain analysis may be analyzed in Balanced or Single-Ended mode.
- **Time Domain.** The Time Domain analysis may be analyzed in Differential 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

When PLTS is first started, the Startup Wizard is displayed. The *Startup Wizard Welcome* screen is displayed in Figure 6-1. 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 6-1 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.

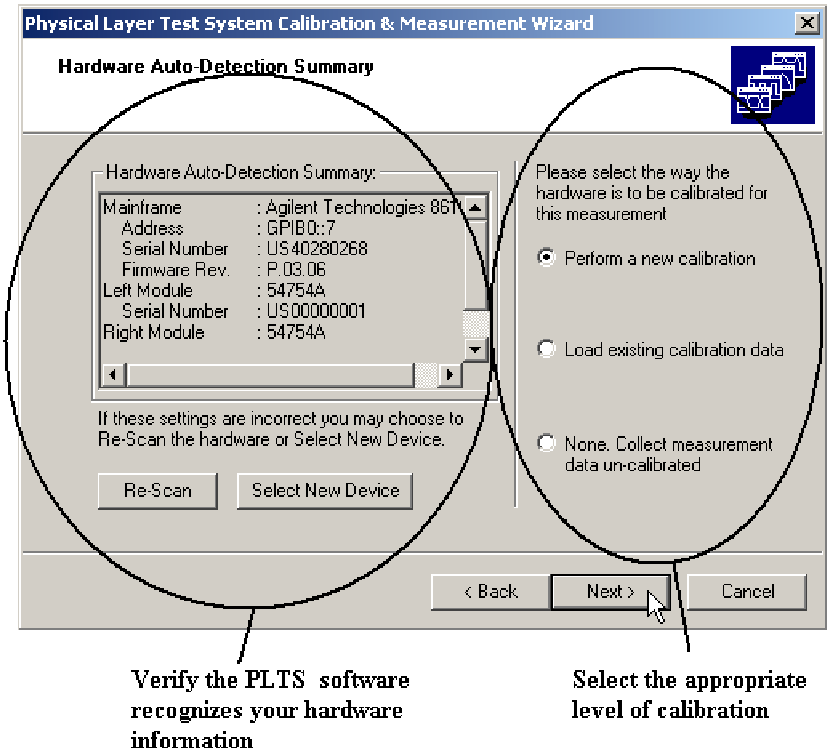
   This step uses the *Setup TDR* dialog box to display the default TDR calibration and measurement settings, based on the PLTS hardware recognized by the software. You may modify these default TDR calibration and measurement settings as required.

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.

When you selected **New Measurement** and then clicked the **Next >** button described on page 186, the *Hardware Auto-Detection Summary* dialog box is displayed. See Figure 6-2.

Figure 6-2 Hardware Auto-Detection Summary Dialog Box



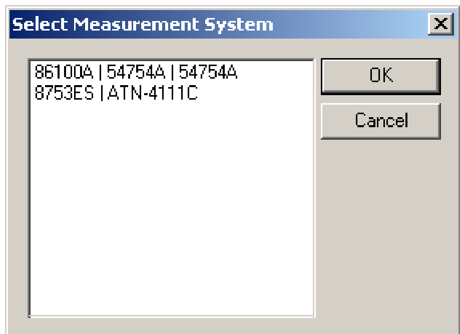
To Verify the Software Recognizes the PLTS Hardware

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 6-3. 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 6-1.

Figure 6-3 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 6-2), select your calibration strategy:

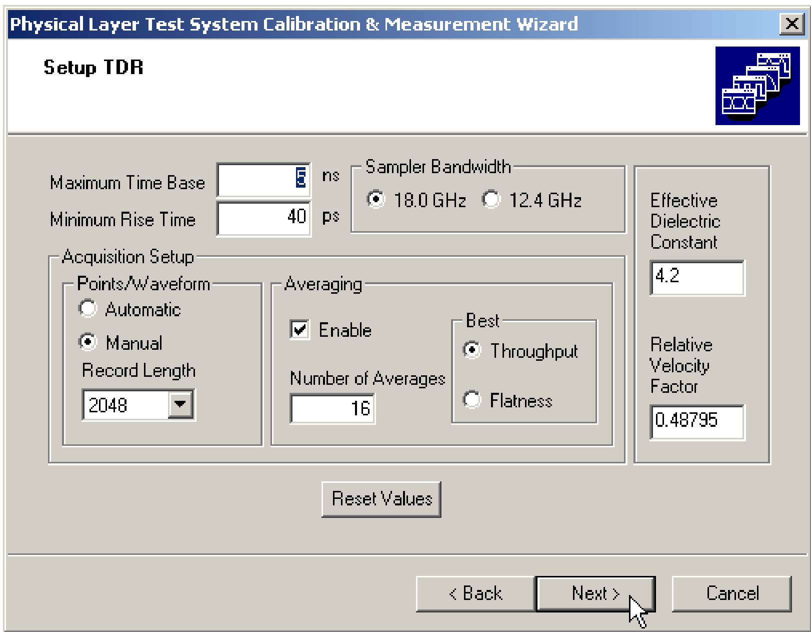
- **Perform a new calibration** continues with the Startup Wizard performing a new calibration before making a measurement.
- **Load existing calibration data** allows you to select a previously saved calibration (*.cal) file to be used for the new measurement. See Chapter 7, “Performing Error Correction on the TDR-Based PLTS,” for guidance on calibration intervals, etc.
- **None. Collect measurement data uncalibrated** allows you skip the calibration, select measurement parameters (see Figure 6-4 on page 190), 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 6-4. This screen allows you to set the TDR parameters for the calibration and measurement. Each of these parameters is described in Table 6-2, "TDR Parameter Descriptions."

Figure 6-4 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 6-2 TDR Parameter Descriptions

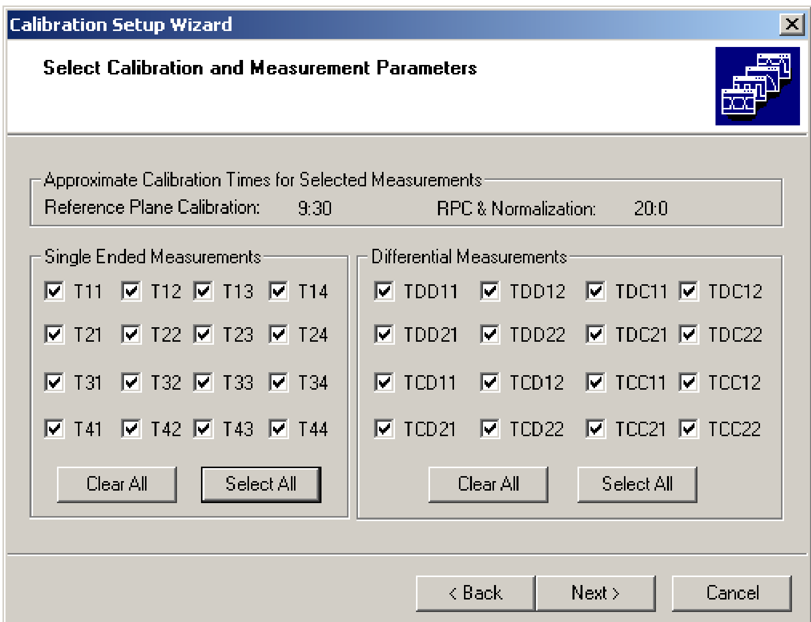
Parameter	**Description**
Maximum Time Base	Sets the desired measurement range in nanoseconds.
Minimum Rise Time	Sets the rise time of normalized measurements in picoseconds. This is a function of normalization and is available to only the Agilent 86100-based TDR system.
Sampler Bandwidth	Sets the Agilent 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 number of points are either: – 16, 32, 64, 128, 256, 512, 1024, 2048, and 4096 -or- – 20, 50, 100, 250, 500, 1000, 2000, and 4000
Averaging	**Enable** turn the trace averaging on and off. **Number of Averages** sets the number of sweeps to be averaged. **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. It has a slower measurement time. This is available to only the Agilent 86100-based TDR system.
Effective Dielectric Constant	Specifies the dielectric constant for your transmission medium.
Relative Velocity Factor	Specifies the relative velocity for your transmission medium.

To Select the Calibration and Measurement Parameters

The *Select Calibration and Measurement Parameters* dialog box is displayed. See Figure 6-5. The left side of the dialog box lists all 16 of the single-ended measurements that the TDR system can make. The right side of the dialog box lists all 16 of the differential measurements that the TDR system can make.

NOTE If you are using equipment other than the prescribed Agilent and Tektronix equipment, a subset of the these measurements may be displayed as active while other measurements may be inactive (grayed).

Figure 6-5 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.

Table 6-3, Approximate Calibration Times for Single Ended Parameters and Table 6-4, Approximate Calibration Times for Differential Parameters provide a guide for estimating

calibration times for each selected parameter. Module calibration times are not listed.

Table 6-3 Approximate Calibration Times for Single Ended Parameters

	Standards				Thrus					
	Approximate Cal Time per column (in min:sec): Ref Plane Cal Only: 0:20 Norm + Ref Plane Cal: 1:00				Approximate Cal Time[a] per column (in min:sec): Ref Plane Cal Only: 0:40 Norm + Ref Plane Cal: 1:00					
	Port1	Port2	Port3	Port4	Port 1–2	Port 1–3	Port 1–4	Port 2–3	Port 2–4	Port 3–4
T11	X									
T12					X					
T13						X				
T14							X			
T21					X					
T22		X								
T23								X		
T24									X	
T31						X				
T32								X		
T33			X							
T34										X
T41							X			
T42									X	
T43										X
T44				X						

a. If the calibration for a parameter is performed, no additional calibration or time is required for any other parameters listed within that column. For example, if you performed the Port 1-2 Thru calibration for T12, selecting T21 does not require an additional calibration and the calibration time remains the same.

Table 6-4 Approximate Calibration Times for Differential Parameters

	Standards				Thrus
	Approximate Cal Time[a] per column (in min:sec): Ref Plane Cal Only: 0:20 Norm + Ref Plane Cal: 1:00				Approximate Cal Time[a] per column (in min:sec): Ref Plane Cal Only: 0:40 Norm + Ref Plane Cal: 1:00
	Left Module		Right Module		
	Differ -ential Mode Cal	Common Mode Cal	Differ -ential Mode Cal	Common Mode Cal	Ports 1–3 & 2–4
TDD11	X				
TDD12					X
TDD21					X
TDD22			X		
TDC11		X			
TDC12					X
TDC21					X
TDC22				X	
TCD11	X				
TCD12					X
TCD21					X
TCD22			X		
TCC11		X			
TCC12					X
TCC21					X
TCC22				X	

a. If the calibration for a parameter is performed, no additional calibration or time is required for any other parameters listed within that column. For example, if you performed the Port 1 standards calibration for TDD11, selecting TCD11 does not require an additional calibration and the calibration time remains the same.

Leave all of the 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 does not 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 module calibration.

   This step uses the *Calibrate TDR and Modules* wizard window to perform the module calibration.

5. Connecting the calibration standards.

   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 6-6.

Start the calibration:

To Select a Calibration Kit and Define the Calibration File Name

The PLTS software has five default TDR calibration kits and their information stored in it.

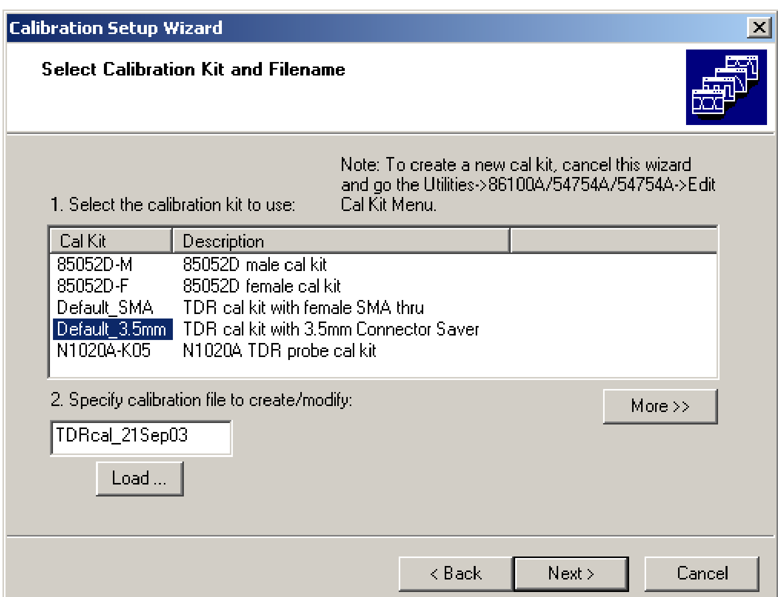
Table 6-5 Pre-Defined TDR Calibration Kits

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

   PLTS is shipped with several calibration kits already defined for the TDR calibration. All of these calibration kits are listed in the **1. Select the calibration kit to use:** list. PLTS is set to the **Default 3.5mm** selection.

Figure 6-6 Select Calibration Kit and Filename Wizard Screen



The **1. 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 6-7. The button label toggles between **More >>** and **<< Less**.

Figure 6-7 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 **1. 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 199.

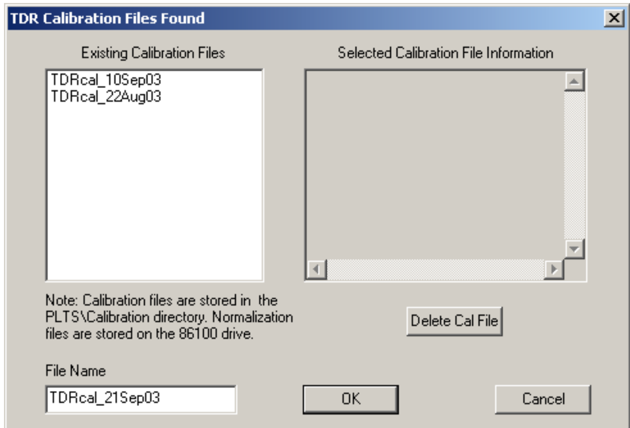
2. In the **2. Specify calibration file to create/modify:** text box of the *Select Calibration Kit and Filename* wizard screen, specify the file that you will use to save the calibration data.

   The text box displays a default calibration file name based on the current date. It uses the string “TDRcal_” followed by the current date. In this example, “TDRcal_21Sep03” is displayed.

   You may enter another file name if you like. 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**

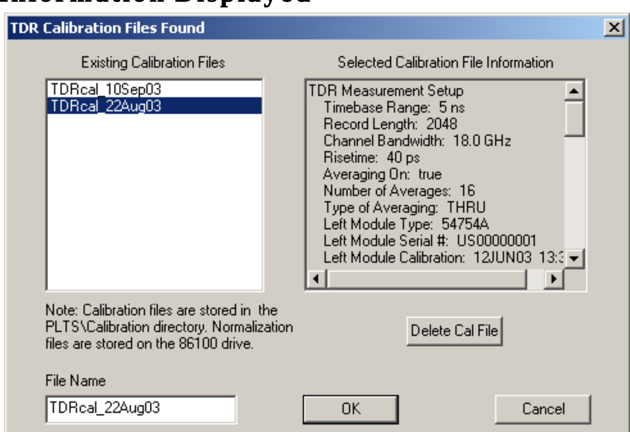
   If you want to overwrite a previously saved calibration file, you may select the **Load**... button to display the *TDR Calibration Files Found* dialog box. This dialog box displays each TDR calibration file that has been saved in the **Existing Calibration Files** list. See Figure 6-8.

Figure 6-8 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 6-9.

Figure 6-9 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.

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

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

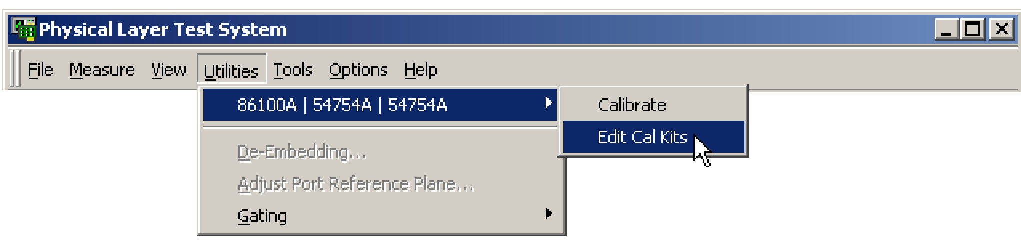
To Define a Calibration Kit

If none of the previously defined calibration kits in the **1. 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:
   

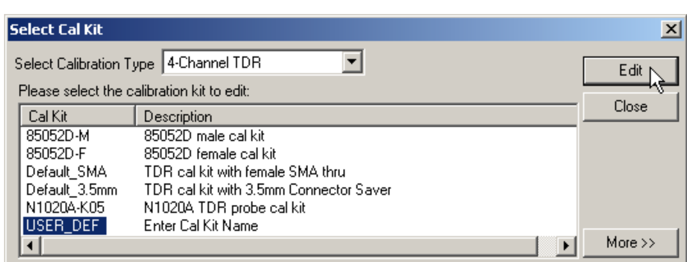
2. From the **Utilities** menu, select the TDR System Model (in this case, **86100A|54754A|54754A**), the click **Edit Cal Kits** as shown in Figure 6-10.

Figure 6-10 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 6-11.

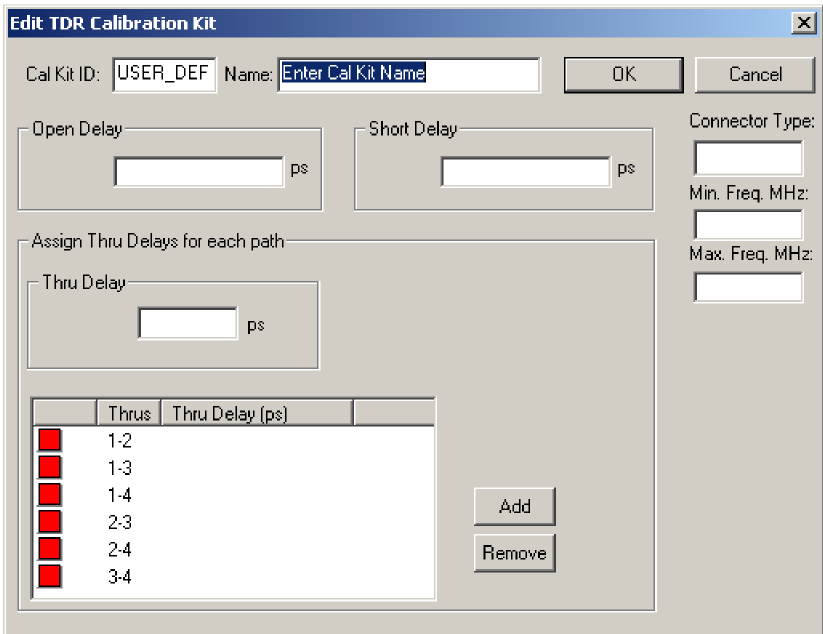
Figure 6-11 Select Cal Kit Dialog Box



4. The calibration kit definition is performed in the *Edit TDR Calibration Kit* dialog box shown in Figure 6-12. 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 6-6.

Figure 6-12 Blank Edit TDR Calibration Kit Dialog Box



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

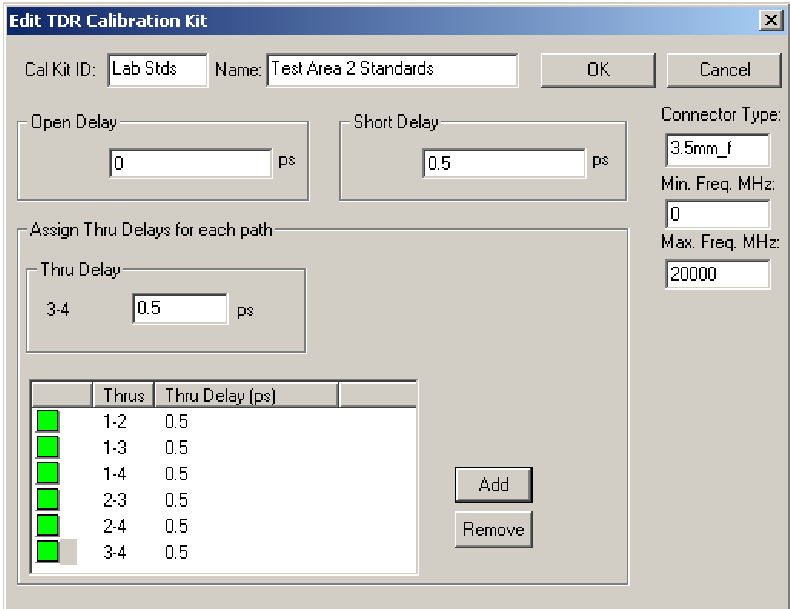
   The software compares the minimum and maximum frequency entries against the measurement setup frequencies when determining which calibration kits to make available for selection for the calibration as in Figure 6-6.

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 6-13 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 6-13 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.

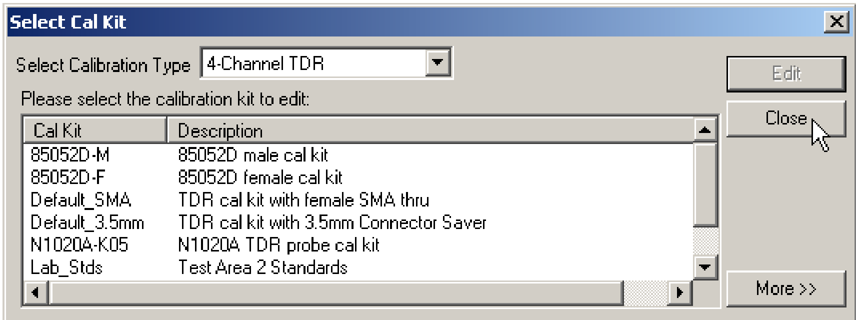
   The **Remove** button disassociates the delay value with the thru path allowing you to change the delay value for the path.

   All or 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.

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 6-14 Select Cal Kit Dialog Box with Newly Defined Cal Kit

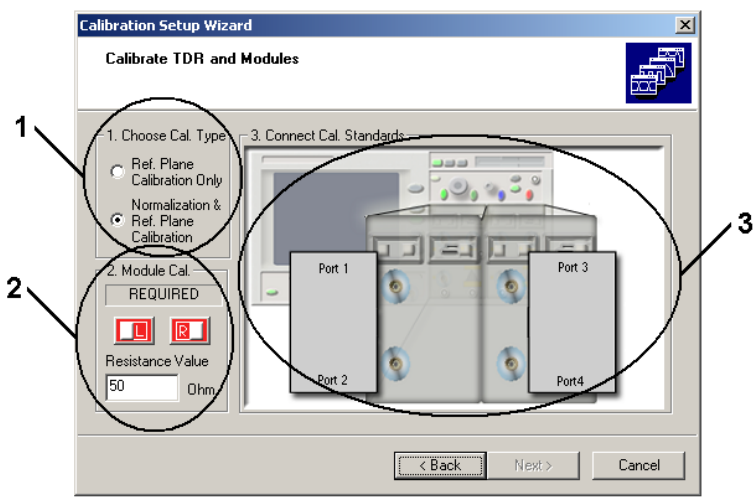


Return to the *Startup Wizard* by selecting **New** from the **File** menu to make your measurement. Refer back to "Starting the Startup Wizard" on page 186.

To Choose the Calibration Type

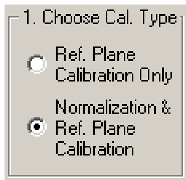
After selecting the calibration kit, verifying the data file, and selecting the **Next >** button, the *Calibrate TDR and Modules* wizard window is displayed. See Figure 6-15. This wizard window is used for the entire calibration.

Figure 6-15 Calibrate TDR and Modules Wizard Screen: Initial Display



However, to select the calibration type, you will only use the **1. Choose Cal. Type** area which is labeled "**1**" in Figure 6-15. This area is shown in Figure 6-16. The **1. Choose Cal. Type** area allows you to select between the two calibration types listed.

Figure 6-16 1. 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 calibration
- Removes errors, such as cable loss, mismatch, and reflections
- Has all of the features as the reference plane calibration, except the following:
  - — Is available only to the Agilent 86100-based TDR system using the 86100's built-in normalization feature
  - — Requires the module calibration

Choose the Calibration Type

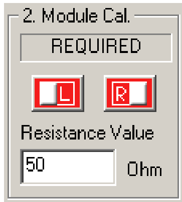
1. In the **1. Choose Cal. Type** area (see Figure 6-16), select either **Ref. Plane Calibration Only** or **Normalization & Ref. Plane Calibration**.

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 **2. Module Cal.** area (see Figure 6-17) has a text box that indicates the status of the module calibration. It displays: REQUIRED, RECOMMENDED, or VALID

Figure 6-17 2. Module Cal. Area



The module calibration is not required if you selected if you selected **Ref. Plane Calibration Only** in the **1. 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:

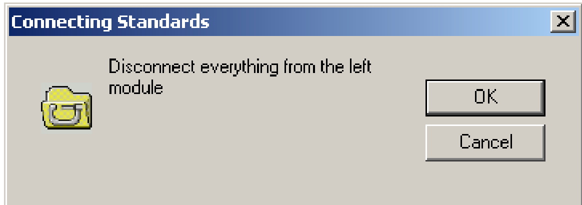
- **Red**, the module calibration is required.
- **Yellow**, the module calibration is recommended.
- **Green**, the module calibration is valid.

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 6-18 shows the first prompt that is displayed.

Figure 6-18 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. 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 RECOMMEDED, now has changed to VALID.

To Connect the Calibration Standards

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 **3. Connect Cal. Standards** area of the *Calibrate TDR and Modules* wizard screen shown in Figure 6-19.

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 and the signals arrive at the input of the DUT at exactly the same time.

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 6-19 **Calibrate TDR and Modules Wizard Screen**

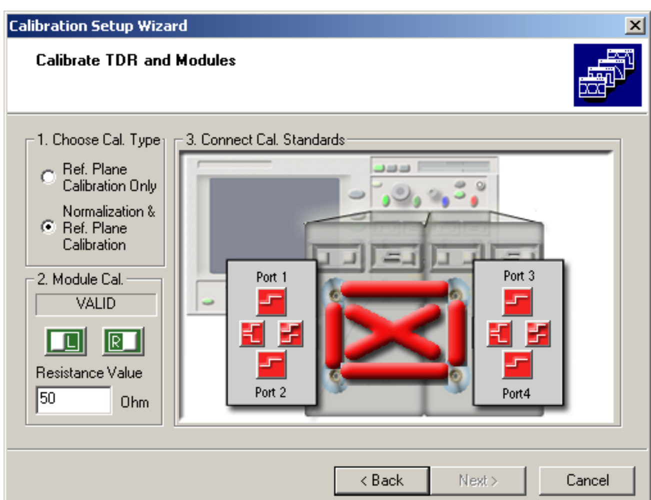


Figure 6-19 shows the *Calibrate TDR and Modules* wizard screen when all calibration and measurement parameters selected. The **3. Connect Cal Standards** area displays several icons. All available icons are displayed in this illustration.

Both modules in Figure 6-19 are shown with four icons. Two of these four icons are the same. There are only three unique icons:

1 2 3

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 192. Table 6-6 shows when each of these icons are displayed and which calibrations are required based on the selected parameters.

Table 6-6 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)
	**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.

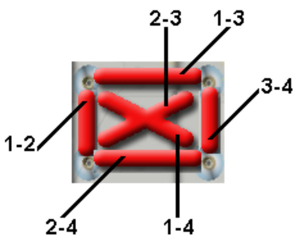
The color of the icons displayed in the **3. Connect Cal Standards** area is red if the calibration has not performed. While the calibration is being performed, the icon color is blue. When the calibration is complete, the icon color turns to green.

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 6-7 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 192. Table 6-7 also shows when each of these icons are displayed and which calibrations are required based on the selected parameters.

Table 6-7 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**).

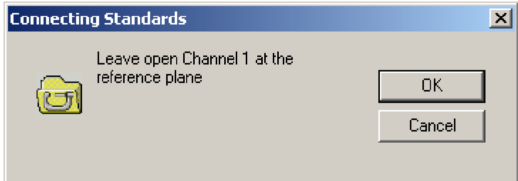
Selecting an icon in the **3. Connect Cal Standards** area starts the calibration. The order that the calibration is performed is completely arbitrary. You may select the icons in any order.

In this example, we will be calibrating all of the icons. See Figure 6-19 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 6-20. These prompts are used throughout the TDR calibration.

Figure 6-20 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.

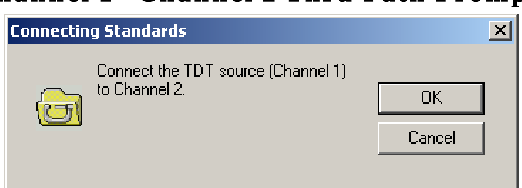
Perform the Thru Path Calibration

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 **3. 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 6-7, if necessary) and follow the PLTS software prompts to perform the thru path calibration between these channels. Figure 6-21 shows the prompt that is displayed.

Figure 6-21 Channel 1 - Channel 2 Thru Path Prompt



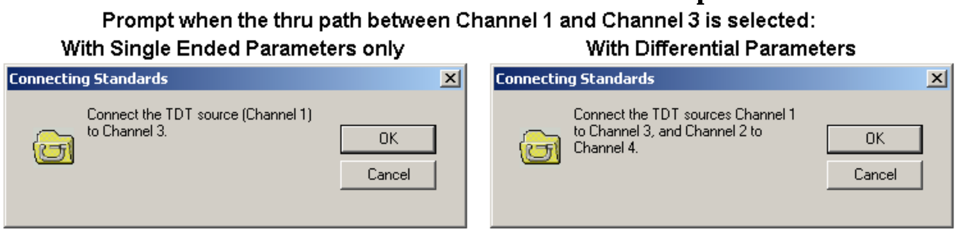
NOTE Follow the PLTS software prompts on your PC rather than following any prompts that is displayed on your TDR to ensure your calibration is correct.

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 6-22 when the thru icon between channels 1 and 3 (or between channels 2 and 4) is selected.

Figure 6-22 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 212.

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 initially as well as later.

3. Modifying the measurement stimulus and measured parameters

   This optional step allows you to make last minute changes to the measurement parameters and the list of parameters to be measured. The caveat is only changes that will not require a recalibration may be made.

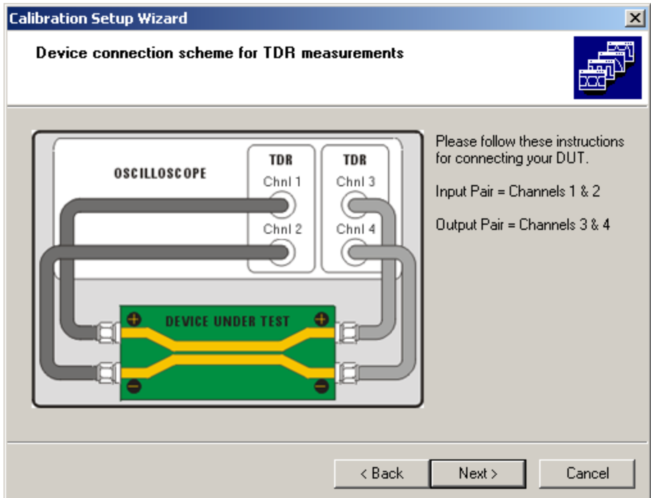
4. Running the measurement

   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 6-23. 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 6-23 Connecting the DUT to the TDR System



To Select the Initial Displayed Format of the Measurement

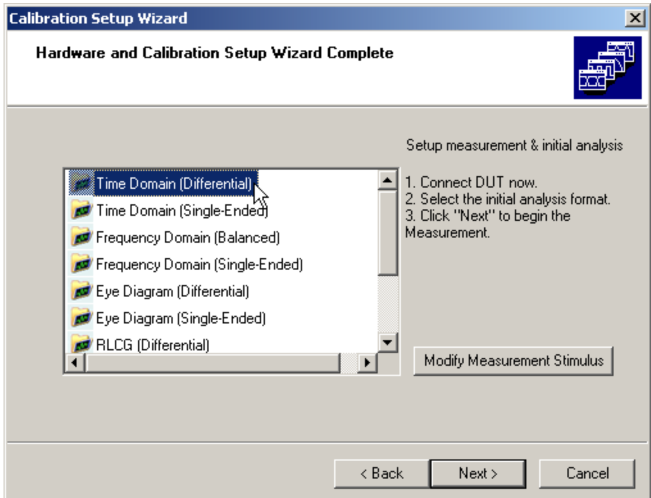
1. The *Hardware and Calibration Setup Wizard Complete* window is displayed. See Figure 6-24.

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

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

Figure 6-24 Hardware and Calibration Setup Wizard Complete

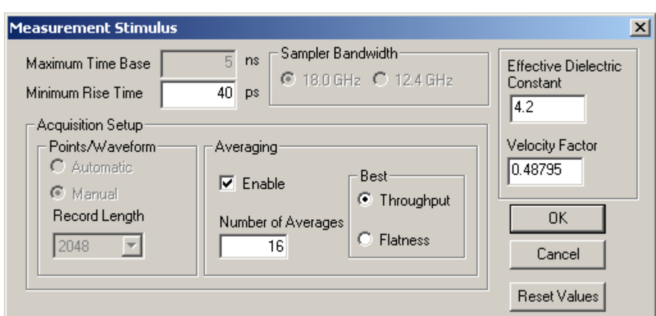


To Modify the Measurement Stimulus and Measured Parameters

1. The **Modify Measurement Stimulus** button opens the *Measurement Stimulus* dialog box. See Figure 6-25. This dialog box allows you to change the many 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 190.

Figure 6-25 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".

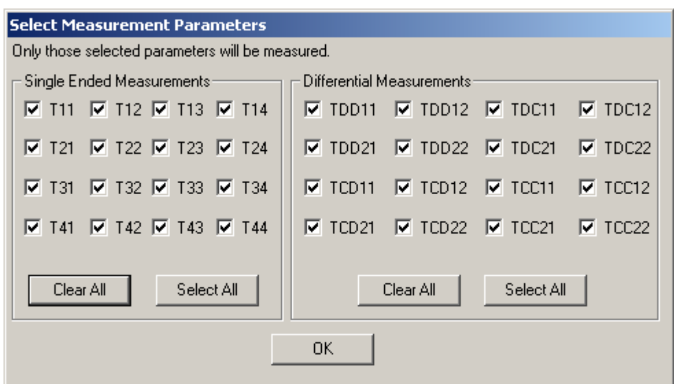
The **Cancel** button resets any changes that were made after opening this dialog box.

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 389 for help.

---

2. Once you exit the *Measurement Stimulus* dialog box, the *Select Measurement Parameters* dialog box is displayed. See Figure 6-26. 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 192, 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 6-26 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.

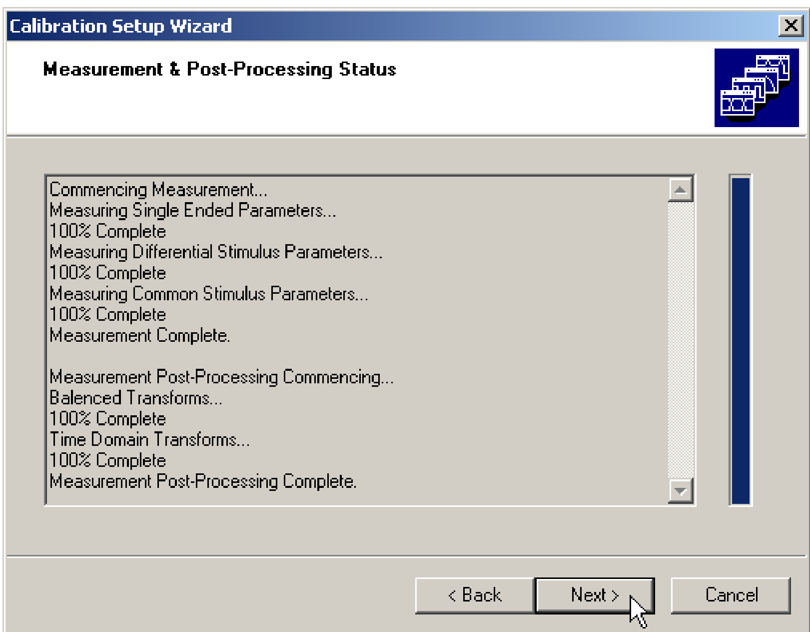
3. With the *Hardware and Calibration Setup Wizard Complete* window is displayed (see Figure 6-24), select the **Next >** button to start the measurement.

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 6-27. 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 results in the main PLTS window.

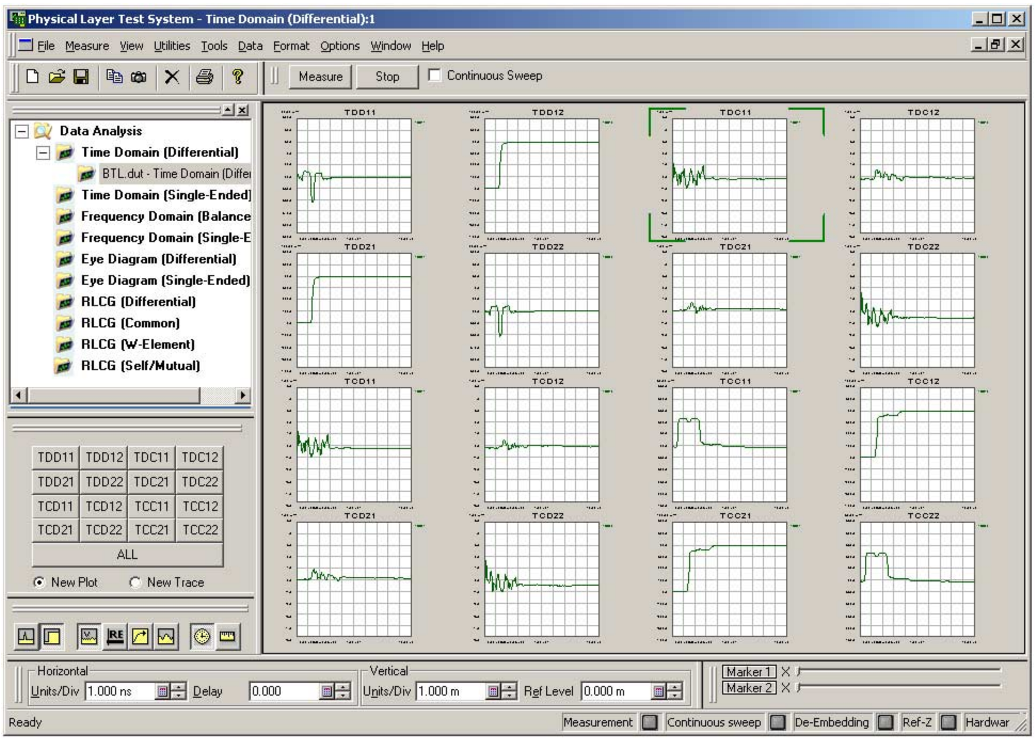
Figure 6-27 Measurement & Post-Processing Status Window



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 6-28. This shows the results in the **Time Domain (Differential)** format, the analysis format selected earlier.

Figure 6-28 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 7, “Analyzing Data in the Frequency Domain,” on page 221 for information.
- **Time Domain.** The Time Domain analysis may be analyzed in Differential or Single-Ended mode. Refer to Chapter 8, “Analyzing Data in the Time Domain,” on page 241 for information.

- **Eye Diagram.** The Eye Diagram analysis may be analyzed in Differential or Single-Ended mode. Refer to Chapter 9, “Analyzing Data using Eye Diagrams,” on page 279 for information.
- **Transmission Line (RLCG).** The RLCG analysis may be analyzed in Differential, Common, W-Element, and Self/Mutual modes. Refer to Chapter 10, “Analyzing Transmission Line Parameters,” on page 297 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” on page 419.

III Data Analysis, Exporting, and Utilities

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

Chapter 7, "Analyzing Data in the Frequency Domain"

Provides information about analyzing measured data in the frequency domain.

Chapter 8, "Analyzing Data in the Time Domain"

Provides information about analyzing measured data in the time domain.

Chapter 9, "Analyzing Data using Eye Diagrams"

Provides information about analyzing measured data using the Eye diagram.

Chapter 10, "Analyzing Transmission Line Parameters"

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

Chapter 11, "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 12, "Removing Unwanted Effects from the Measurement"

Provides information about using measurement enhancement features such as gating, phase skew, and de-embedding.

Chapter 13, "Using Analysis Tools and Utilities"

Provides information about using the PLTS tools, such as markers, trace math, overlaying traces, zooming, 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.

7 Analyzing Data in the Frequency Domain

Physical layer test systems combine a vector network analyzer and a test set to make measurements by sweeping a signal in the frequency domain. The test system software can then mathematically transform these frequency domain measurements into other analysis types such as time domain, eye diagrams, and transmission line (RLCG). However, there is information to be gained from frequency domain measurements that these other analysis types do not provide.

Frequency measurements 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 224. *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 226.

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 7-2).

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 7-1 S-Parameter Notation

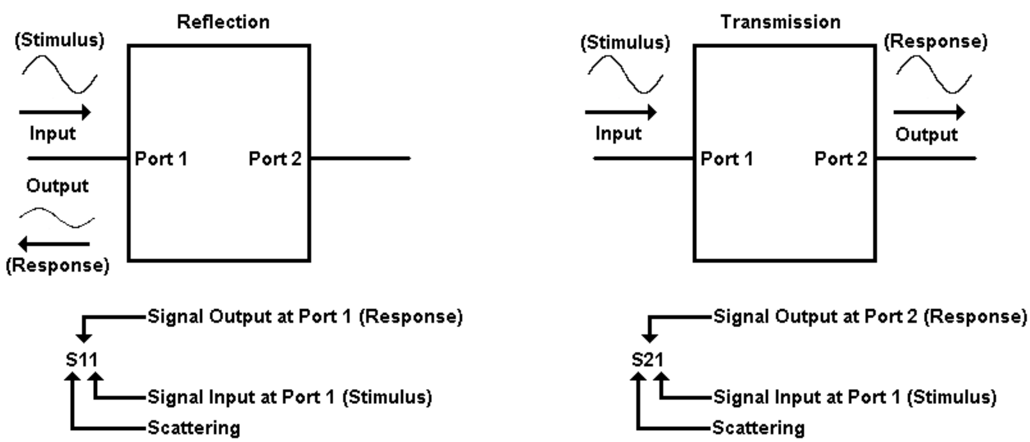
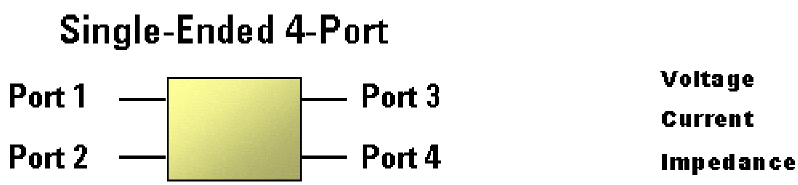


Figure 7-2 **Single-Ended S-Parameter Review**

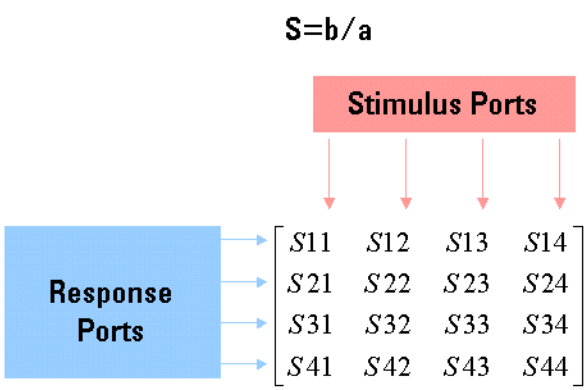


	Normalized Power Waves
stimulus	$an = \frac{1}{2 \cdot \sqrt{\mathrm{Re}\{Zn\}}}(Vn + In \cdot Zn)$
response	$bn = \frac{1}{2 \cdot \sqrt{\mathrm{Re}\{Zn\}}}(Vn - In \cdot Zn)$

S=b/a

Figure 7-3 shows the naming convention for a single-ended 4-port S-matrix, showing the ratio of all possible combinations of response/stimulus.

Figure 7-3 **Single-Ended S-Matrix**



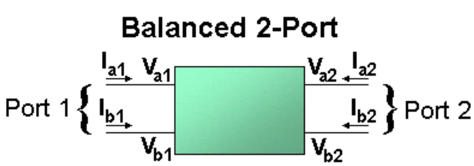
Mixed Mode (Balanced) S-Parameters

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 7-4 Mixed Mode S-Parameter Basics



	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	$Z_{Dn} = \frac{V_D+}{I_D+}$	$Z_{Cn} = \frac{V_C+}{I_C+}$

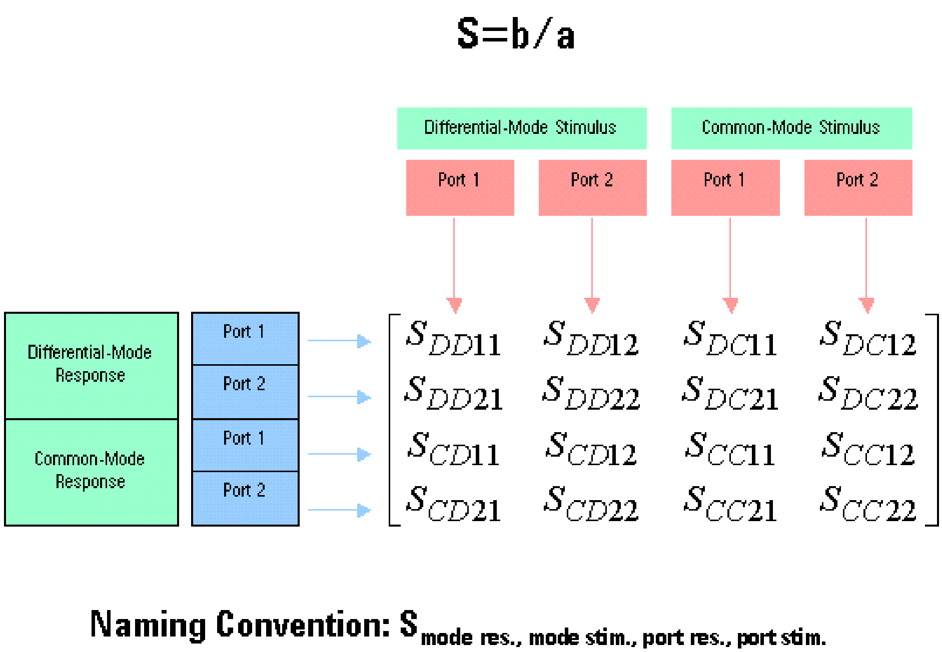
	Normalized Power Waves	
	Differential-Mode	**Common-Mode**
Stimulus	$b_{dn} = \frac{1}{2 \cdot \sqrt{Re(Zdn)}}(Vdn + Idn \cdot Zdn)$	$b_{cn} = \frac{1}{2 \cdot \sqrt{Re(Zcn)}}(Vcn + Icn \cdot Zcn)$
Response	$b_{dn} = \frac{1}{2 \cdot \sqrt{Re(Zdn)}}(Vdn - Idn \cdot Zdn)$	$b_{cn} = \frac{1}{2 \cdot \sqrt{Re(Zcn)}}(Vcn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

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 7-5 shows the naming convention for a mixed mode S-matrix, showing the ratio of all possible combinations of response/stimulus.

Figure 7-5 Mixed-Mode (Balanced) S-Matrix



Viewing Data in the Frequency Domain

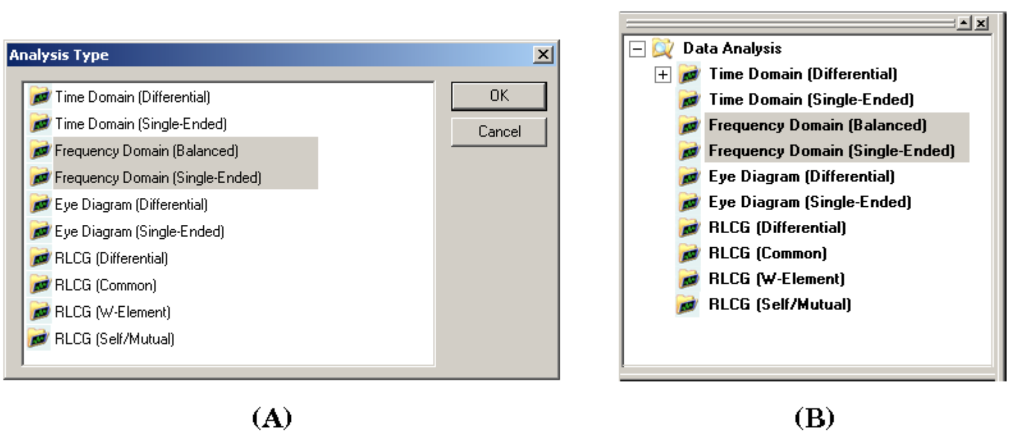
This section guides you with opening measured data in the frequency domain and viewing the data in the way that best suits your requirements. As discussed earlier, there are 16 4-port S-parameters for both single-ended and balanced devices. See Figure 7-3 and Figure 7-5. You may elect to view any number 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 7-6
- From the Startup Wizard Welcome Screen by selecting **Load Measurement** where you must select the analysis type - see (A) of Figure 7-6
- 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 7-6
- 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 7-6

Figure 7-6 Opening the Frequency Domain Plot Window



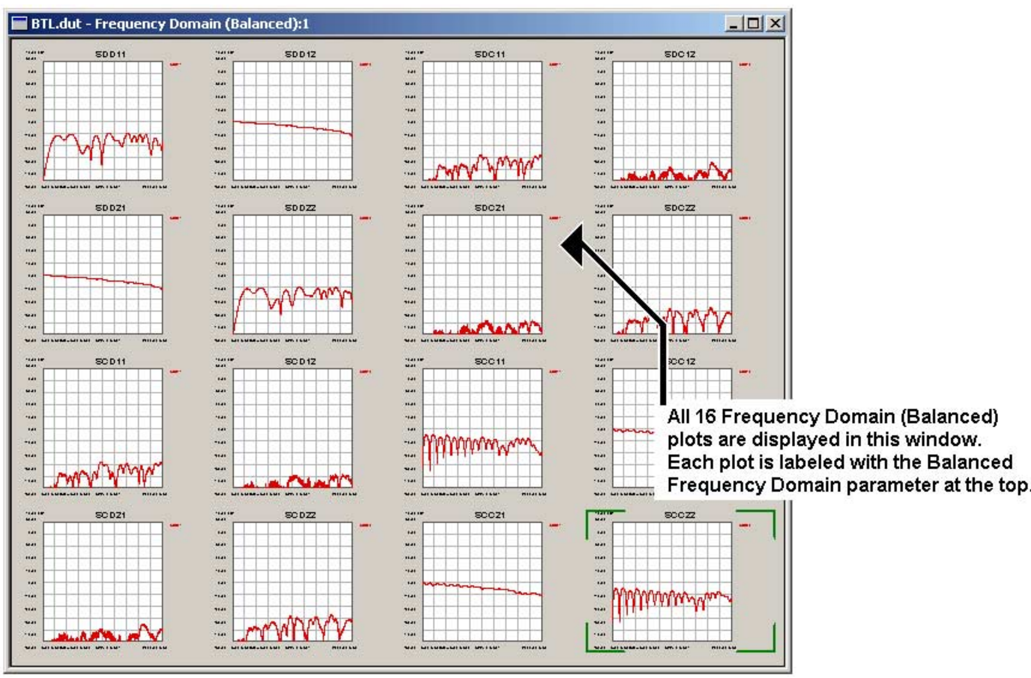
NOTE Using the **Browser** method to open a window requires that you select the parameters that you want to view from the **Parameter Bar** or the **Data** menu when the blank plot window is displayed.

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 7-3 and Figure 7-5. Each plot is labeled with its parameter at the top. See Figure 7-7.

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.

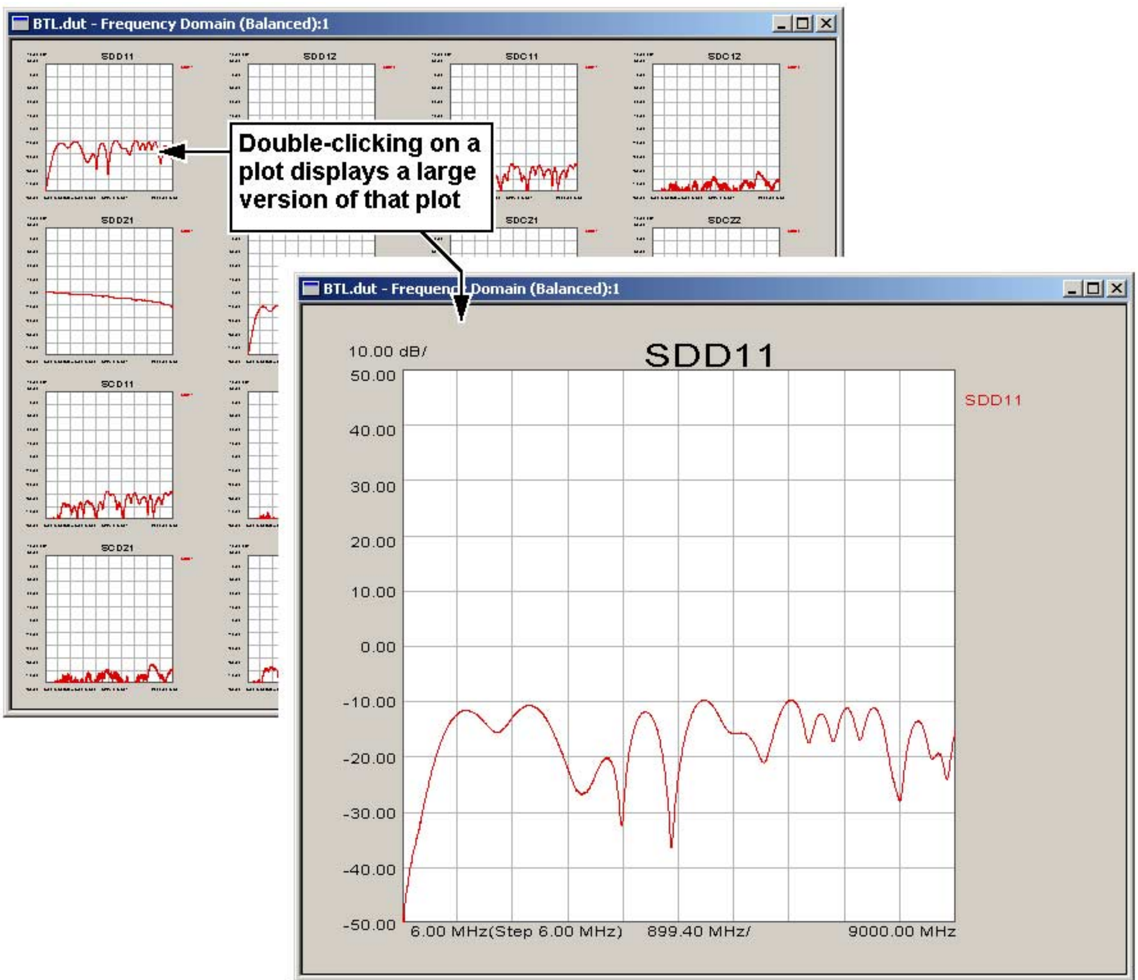
Figure 7-7 All 16 Balanced Frequency Domain Plots



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 7-8.

Figure 7-8 Opening a Single Plot

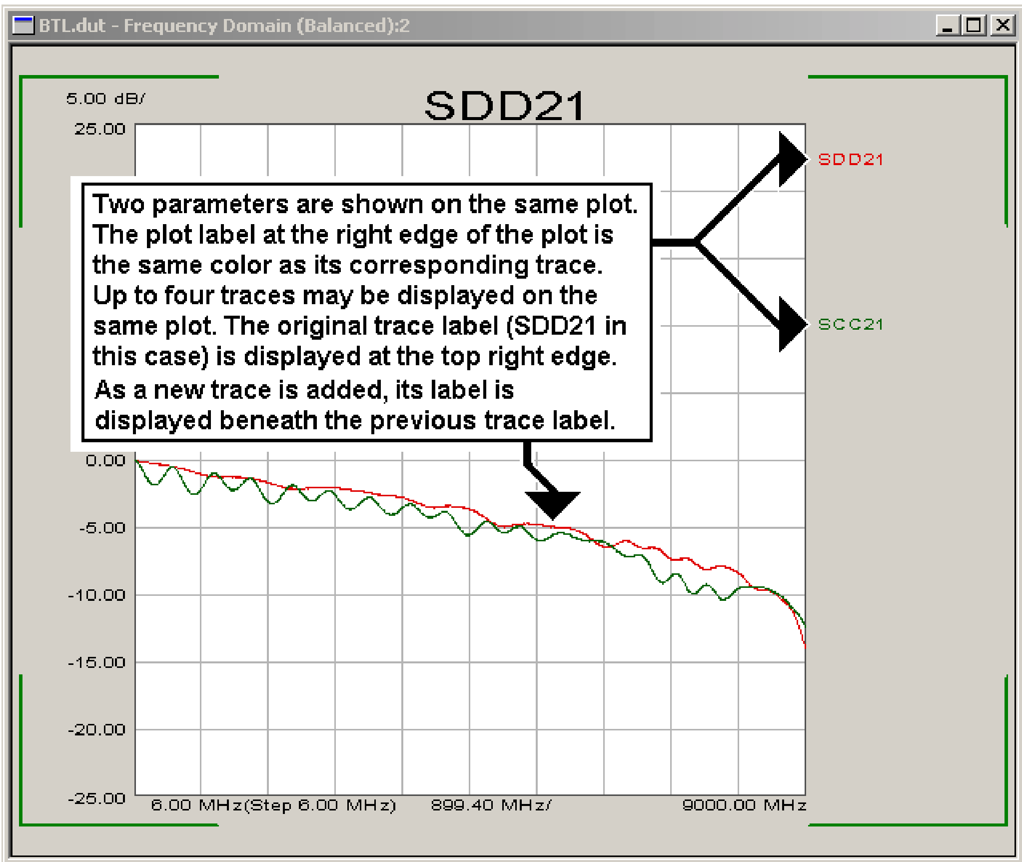


Double-click on the single plot to return to the original view with multiple plots.

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 7-8. 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 7-9 A Single Plot with Multiple Traces

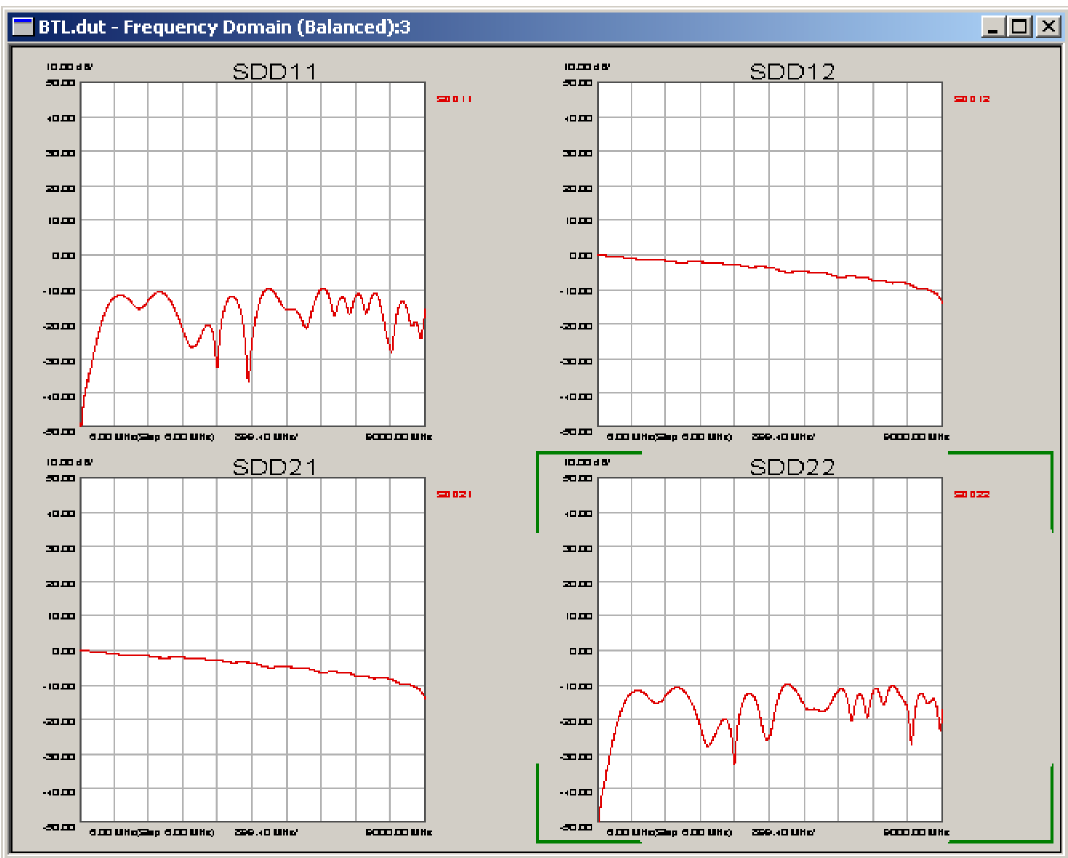


It could contain as many as 4 of the parameters. All trace parameters are labeled at the right edge of the plot. The color of the trace labels are the same color as the traces they represent.

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 7-10.

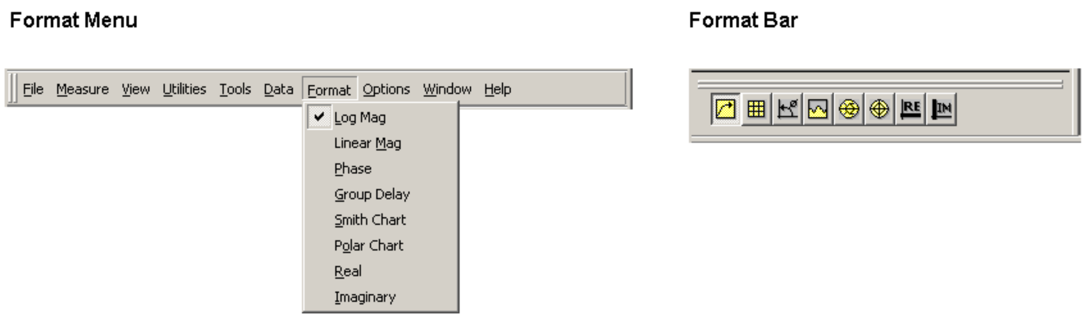
Figure 7-10 Custom S-Parameter Plots Window with Four Plots



Selecting Frequency Domain Display Formats

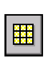
A data format is the way the physical layer test system presents the measurement graphically. Select a data format appropriate to the information you want to learn about the device. You may use either the **Format Bar** or the **Format** menu to select the format.

Figure 7-11 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 370 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).

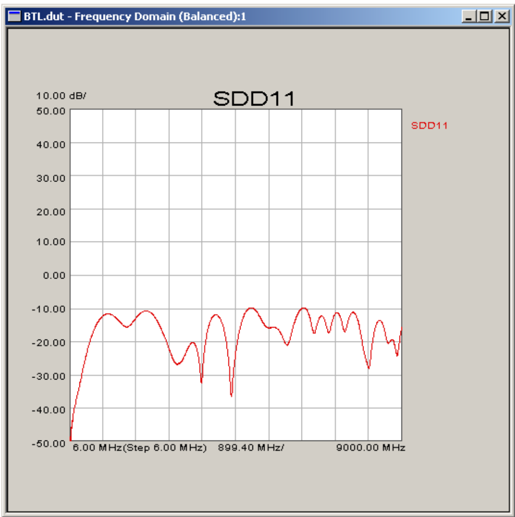
When in this format, the cursor bar allows you to choose the cursor value in either Mag + Phase or Inductance style. Refer to "Frequency Domain Polar and Smith Chart Markers" on page 370 for additional information.

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

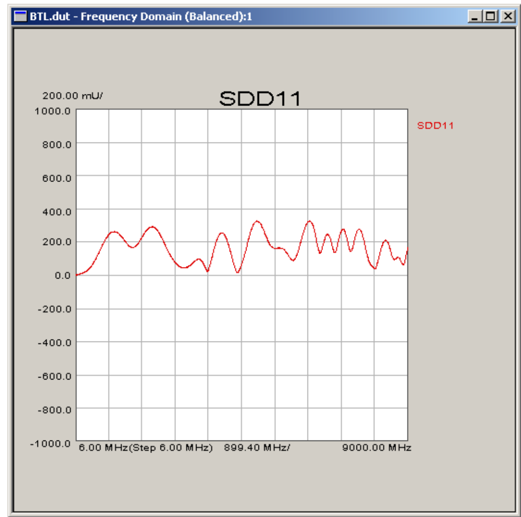
Imaginary displays only the imaginary (reactive) portion of the measured complex data. Typical measurements are impedance for designing matching circuits.

Table 7-1 Frequency Domain Formats

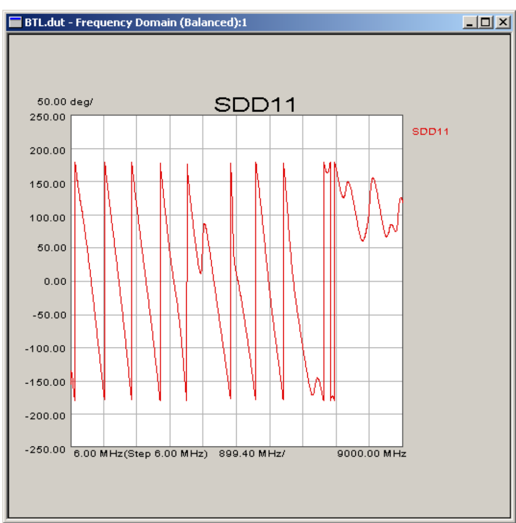
Log Mag



Linear Mag



Phase



Group Delay

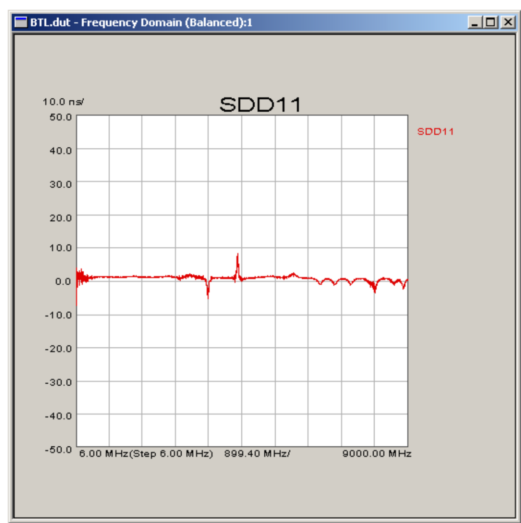
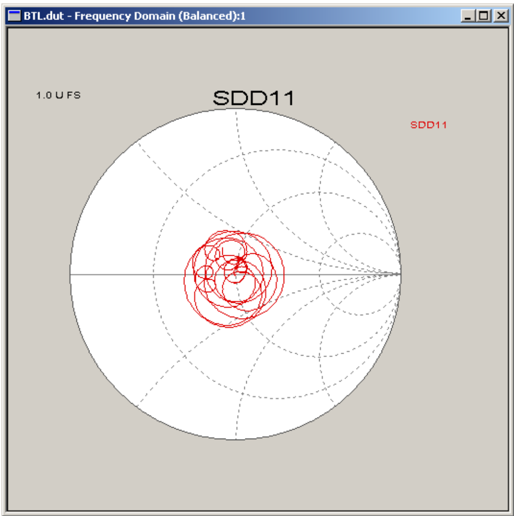
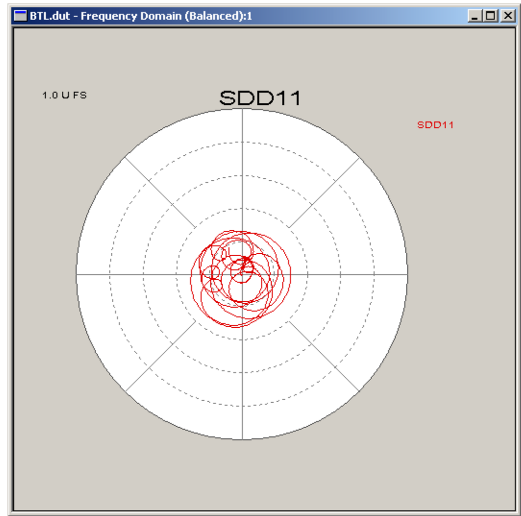


Table 7-1 Frequency Domain Formats

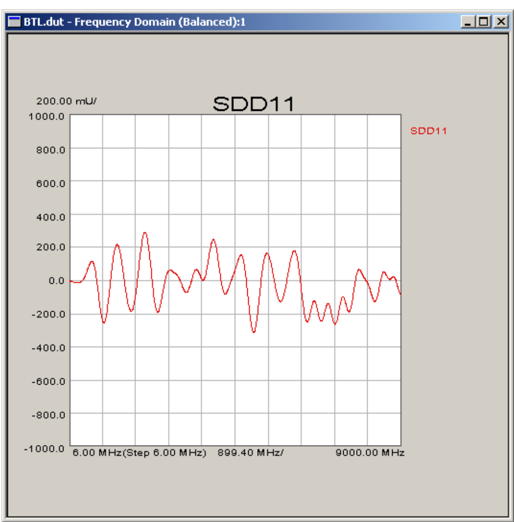
Smith Chart



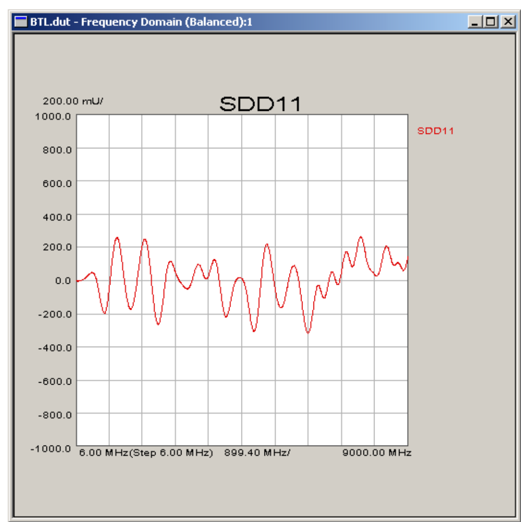
Polar Chart



Real



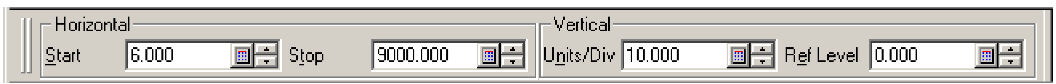
Imaginary



Setting the Scale

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

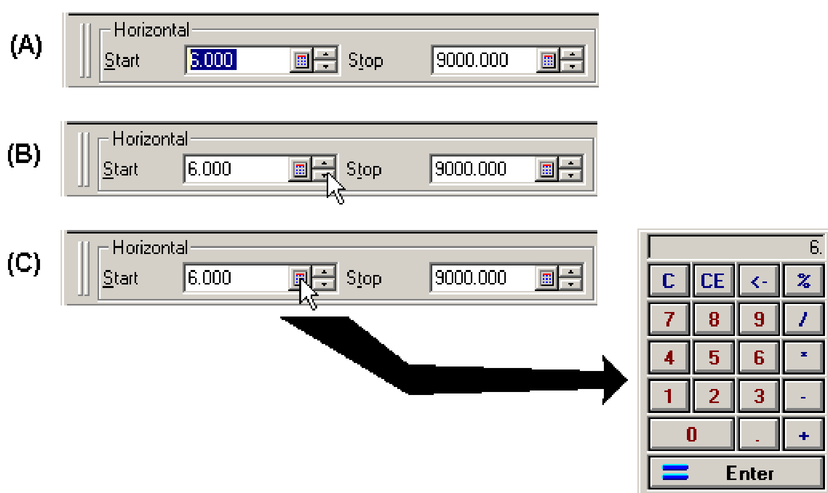
Figure 7-12 Scaling Bar



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 7-13.
- Selecting the up/down arrow buttons to the right of each entry. See (B) of Figure 7-13.
- 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 7-13.

Figure 7-13 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

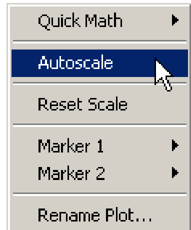
PLTS has three features that make scaling changes quickly and easily. The three features are:

- **Autoscale**
- **Reset Scale**
- **Copy Plot Format** used with **Paste Plot Format** (see “Copying and Pasting Plot Formats” on page 394)

Autoscale

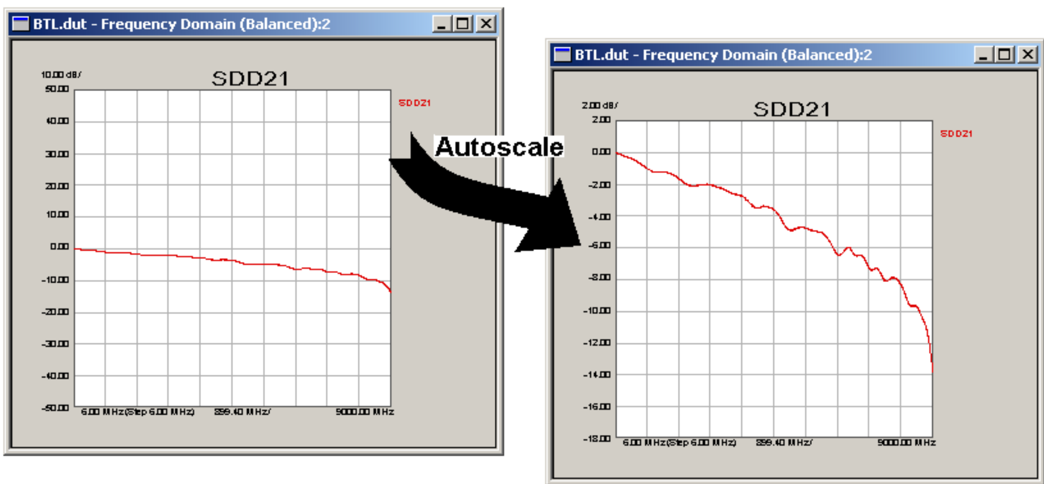
Autoscale changes the vertical scale of the active plot to allow the trace to occupy approximately 80% of the vertical axis of the display. It places the display such that the graticule values are numbers that are easy to work with.

Figure 7-14 Autoscale



To autoscale a plot, select the plot, then right click on the plot to display the quick menu displayed in Figure 7-16. Click **Autoscale** to change the vertical scale of the plot. Figure 7-15 shows a frequency domain plot that has **Autoscale** applied to it.

Figure 7-15 A Plot that has been Autoscaled

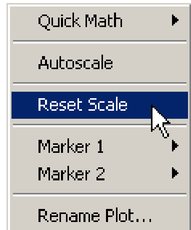


Reset Scale

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 7-16. Click **Reset Scale** to reset the plot to the default settings.

Figure 7-16 Reset Scale



8 Analyzing Data in the Time Domain

The Agilent physical layer test system or (PLTS) combined with Agilent's vector network analyzers and their companion multiport test sets perform measurements in the frequency domain by sweeping a RF signal and measuring the RF responses of a device under test or (DUT). The Agilent PLTS software can also mathematically transform these frequency domain data S parameters into their time domain counterparts and display them in either their step, impedance or their impulse/response modes.

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. And the PLTS software accomplishes it's frequency domain transformation to time domain by utilizing this 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.

An alternative method of obtaining time-domain characterization of a device is to make the measurement directly in the time domain by synthesizing a step waveform, applying it to the device, and observing the response on an oscilloscope. 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 the physical layer test system 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 224.

- 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 8-1 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 8-1 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

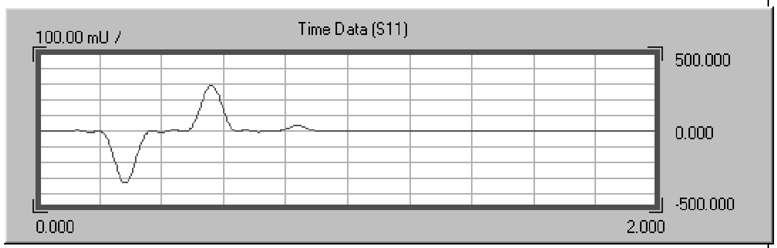
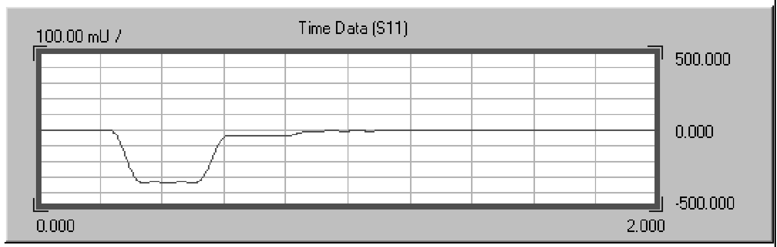
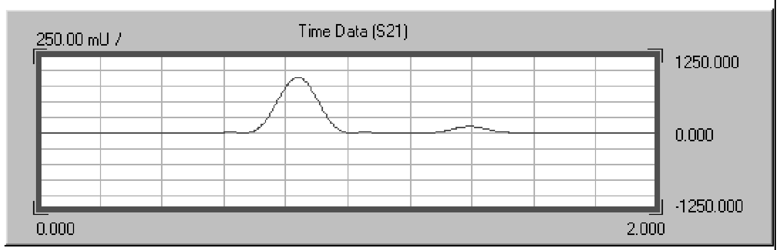
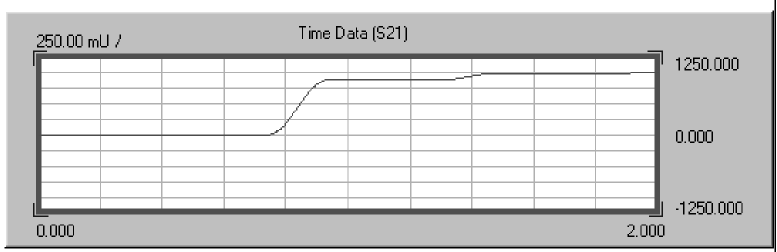
Table 8-1 Relationship of Frequency Domain Parameters to Time Domain Equivalents

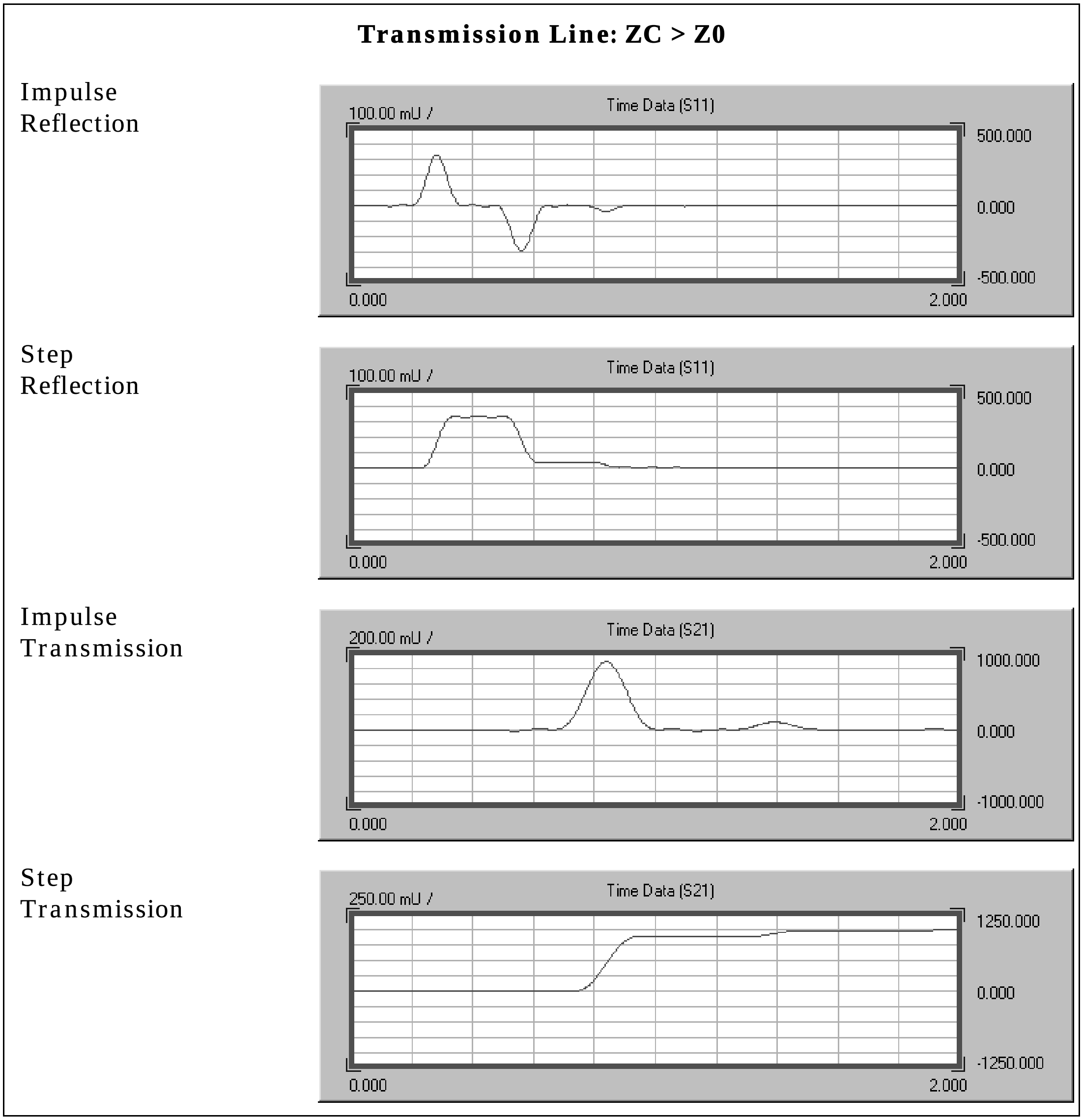
Mode	Direction	Type	Parameter
Differential-to-Common	Forward	TDT	SCD21
Differential-to-Common	Reverse	TDR	SCD22
Common-to-Differential	Forward	TDR	SDC11
Common-to-Differential	Reverse	TDT	SDC12
Common-to-Differential	Forward	TDT	SDC21
Common-to-Differential	Reverse	TDR	SDC22
Differential-to-Single-ended	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 8-2 shows various circuit elements and associated time-domain signatures.

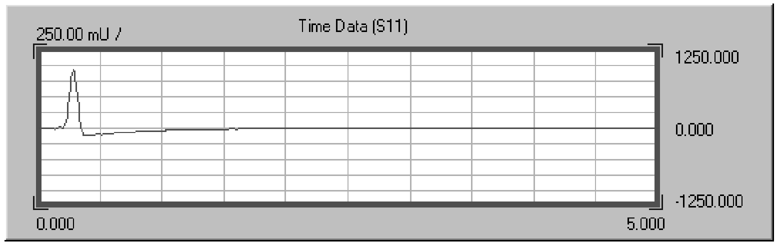
Table 8-2 **Time Domain Signatures**

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

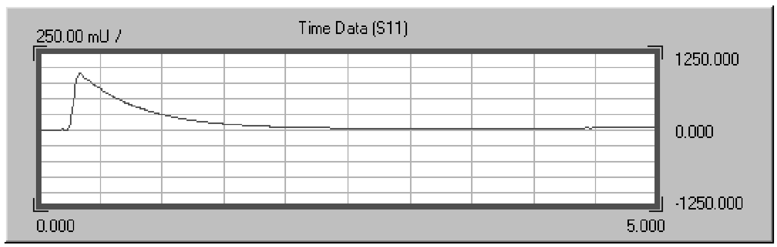

Transmission Line: ZC > Z0
Impulse Reflection
100.00 mU /
Time Data (S11)
500.000
0.000
-500.000
0.000
2.000
Step Reflection
100.00 mU /
Time Data (S11)
500.000
0.000
-500.000
0.000
2.000
Impulse Transmission
200.00 mU /
Time Data (S21)
1000.000
0.000
-1000.000
0.000
2.000
Step Transmission
250.00 mU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

Series Inductor

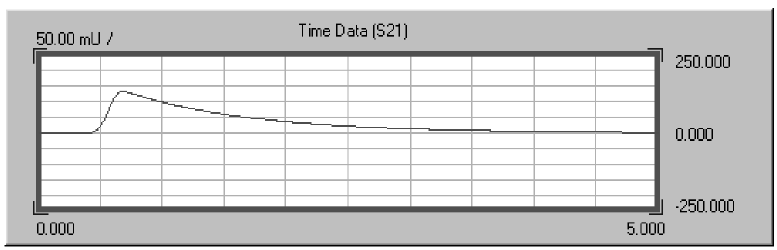
Impulse Reflection



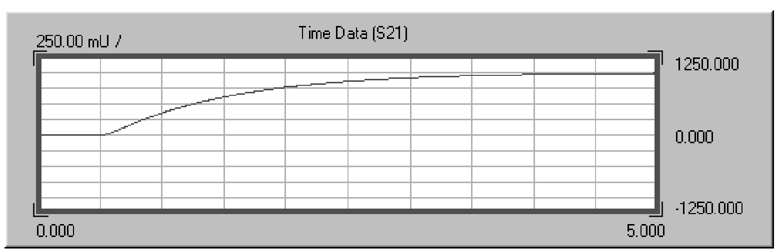
Step Reflection



Impulse Transmission

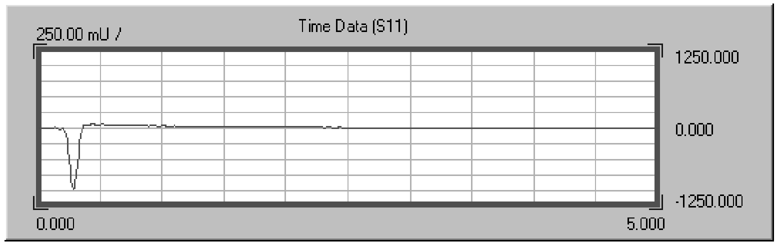


Step Transmission

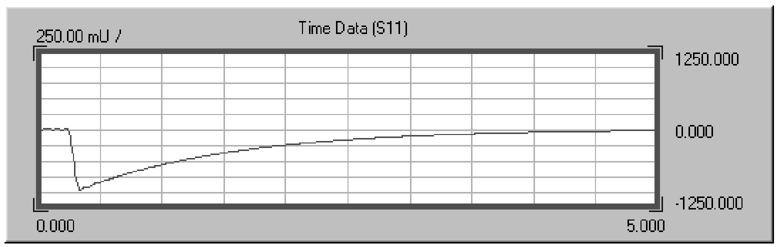


Shunt Capacitor

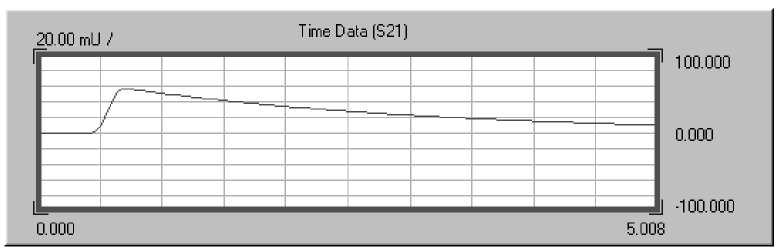
Impulse Reflection



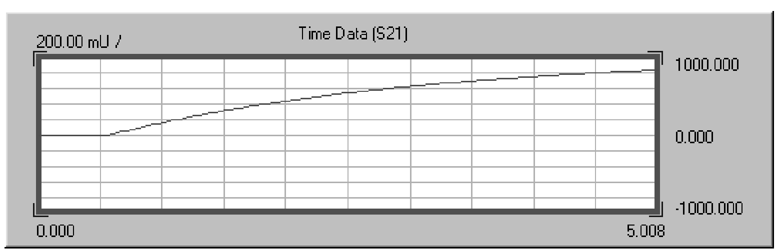
Step Reflection



Impulse Transmission



Step Transmission



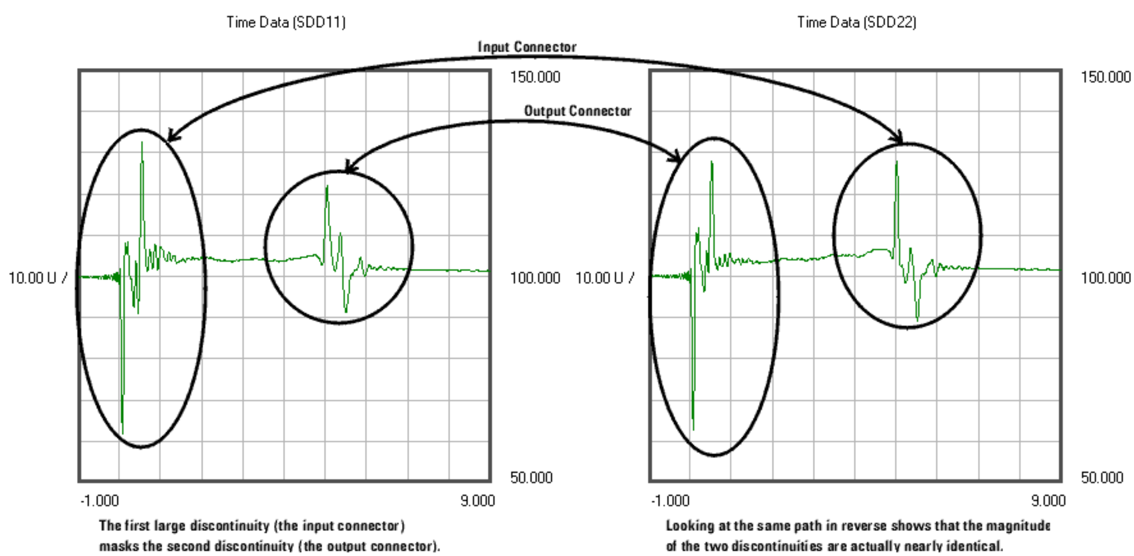
Practical Considerations

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 8-1 Masking Effects



Masking effects can be seen in Figure 8-1. 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 the PLTS system extends the ability of the instrument to accurately characterize devices that have several discontinuities or high loss.

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 Sidelobes limit the dynamic range of the Time Domain measurement by hiding low level response within the sidelobes of the higher level responses. The effects of sidelobes can be improved by windowing.

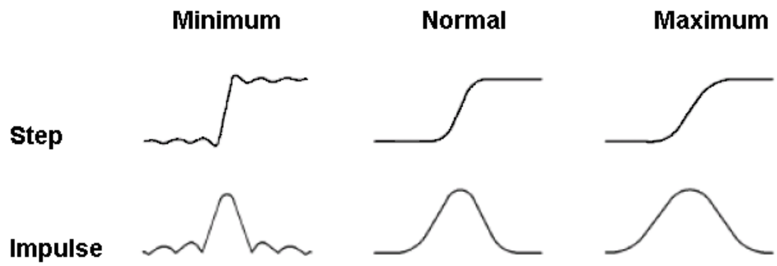
Windowing improves the dynamic range of the Time Domain measurement by modifying (filtering) the Frequency Domain data prior to conversion to the Time Domain to produce an impulse stimulus with lower sidelobes. This greatly enhances the effectiveness in viewing Time Domain responses that are very different in magnitude. The sidelobe 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 a the tradeoff with increased rise time.

Three windows are available: **Minimum**, **Normal**, and **Maximum**. The sidelobe levels of the Time Domain stimulus depend only on the Window that is selected. See Figure 8-3.

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. It is turned on only when the Time Domain response is viewed. Figure 8-2 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 8-2 **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:

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

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)

MF = Multiplication factor constant for the filter coefficient

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

Table 8-3 shows the equivalent rise times for a maximum frequency of 20 GHz for each window setting.

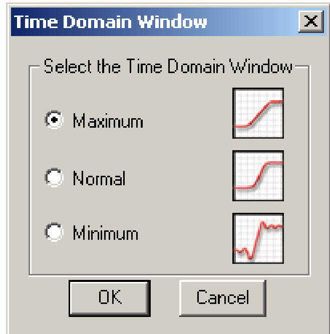
Table 8-3 **Equivalent Rise Times for a Maximum Frequency of 20 GHz**

Window Setting	Maximum Frequency	Filter Coefficient	Equivalent Rise Time
Minimum	20 GHz	0.54	27 ps
Normal	20 GHz	0.71	35.5 ps
Maximum	20 GHz	0.91	45.5 ps

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 8-3 Time Domain Window Dialog Box



- **Maximum** gives the minimum sidelobes and this provides the greatest dynamic range. The filter coefficient value is 0.91. This is the default setting.
- **Normal** gives reduced sidelobes and is normally the most useful. The filter coefficient value is 0.71.
- **Minimum** is essentially no window and therefore give the highest sidelobes. The filter coefficient value is 0.54.

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**. Then delete the current plot window and reopen a new plot window using the same data. 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 245, 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.

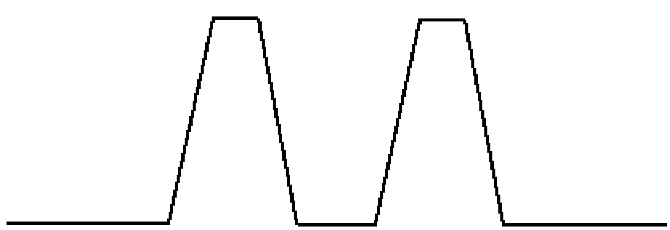
Spatial Resolution

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 8-4.

Figure 8-4 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 8-5, the response in the time domain is dramtically 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 8-5 Example 1: Displayed Response

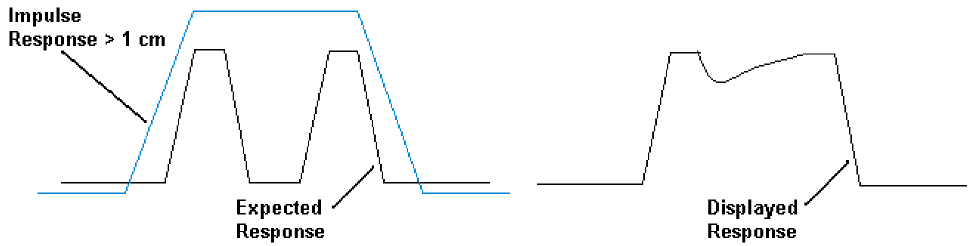
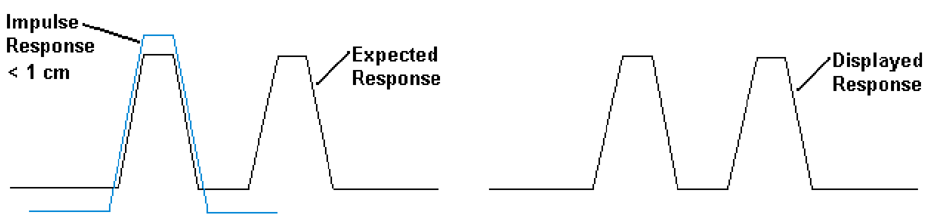


Figure 8-6 shows the same device using an IR with a pulse-width that is less than 1 cm. The response you see in the time domain is much more like what is expected. It looks like the picture on the right side of the illustration. This is because the IR is narrower than the two adjacent discontinuities, and the IR is able to capture only the power level of the individual discontinuities.

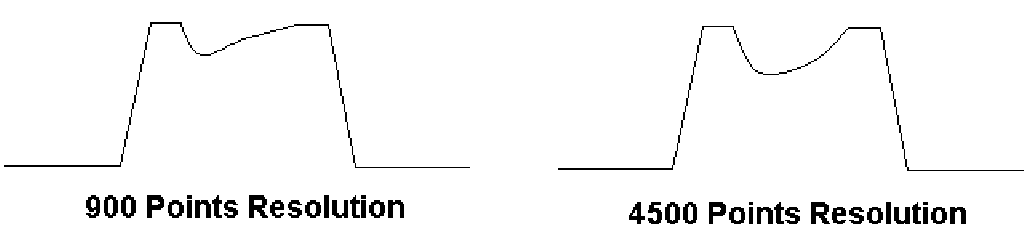
Figure 8-6 Example 2: Displayed Response



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 8-7 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 8-7 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 8-8) and a few real life examples of this process.

Example 9-1 Standard Transmission Measurement

Parameter = SDD21 (Transmission)

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

At Point 1,

Tstart = 0 ns

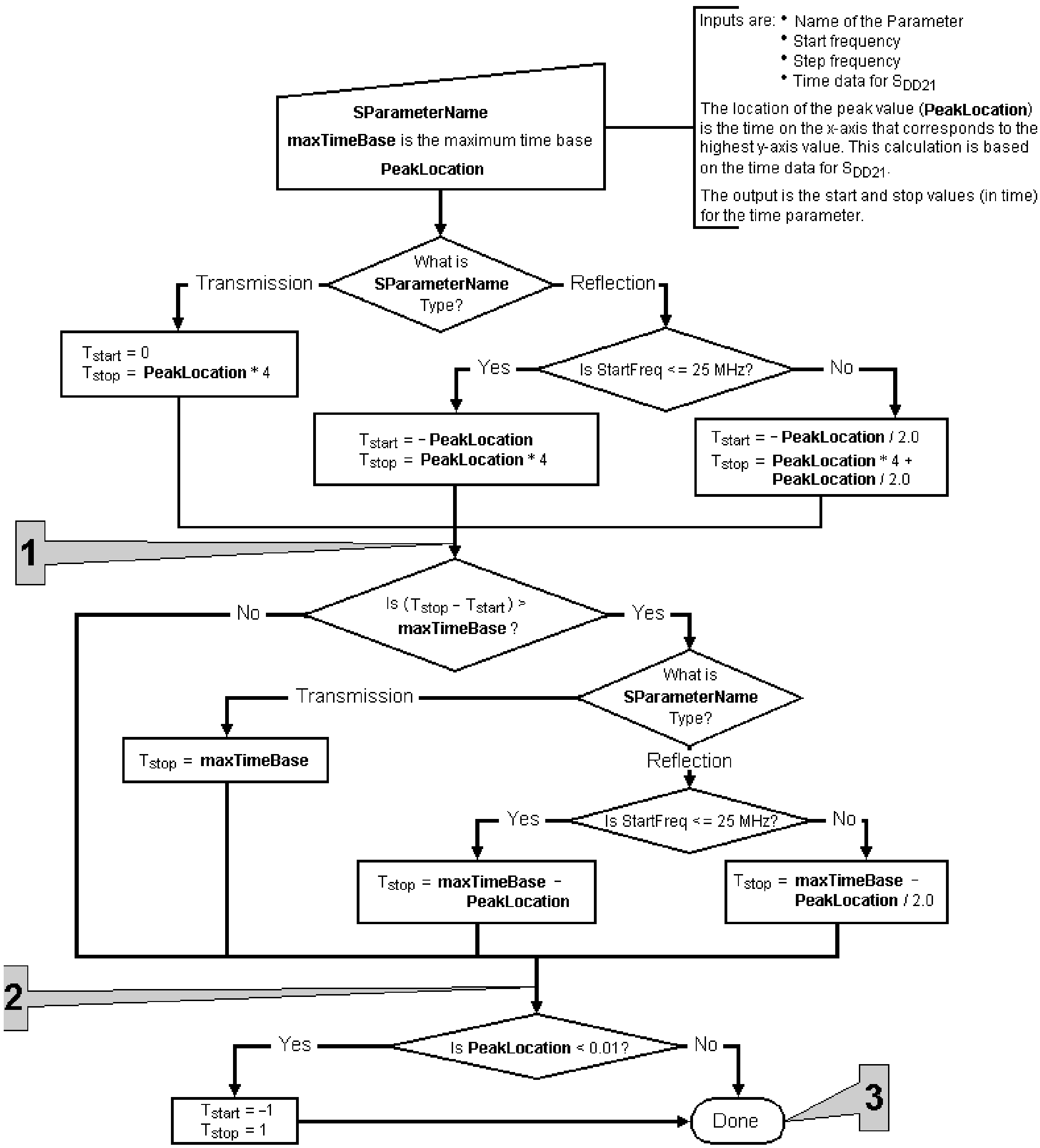
Tstop = 8 ns

The result is:

Tstart = 0 ns

Tstop = 8 ns

Figure 8-8 Automated Start and Stop Algorithm Flow Diagram



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

Example 9-2 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 9-3 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

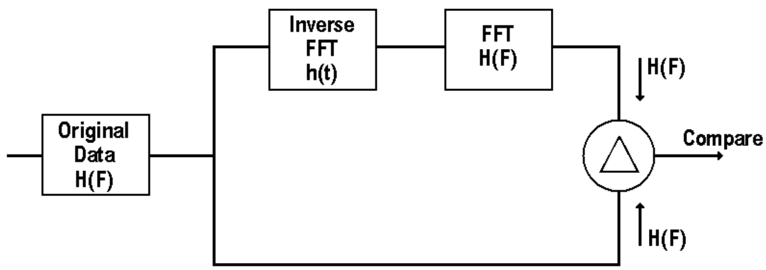
Tstop = 1 ns

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.

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 8-9. 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 8-9 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.

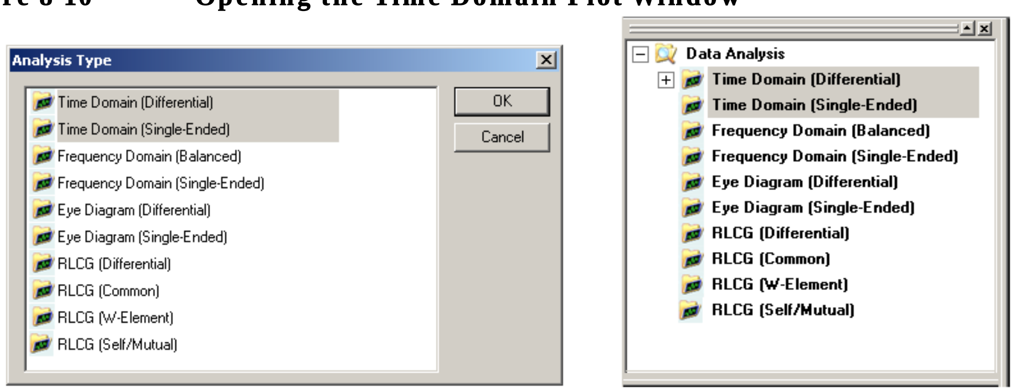
Opening a Time Domain Plot Window

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 8-10
- From the Startup Wizard Welcome Screen by selecting **Load Measurement** where you must select the analysis type - see (A) of Figure 8-10
- 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 8-10
- 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 8-10

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 8-10 Opening the Time Domain Plot Window



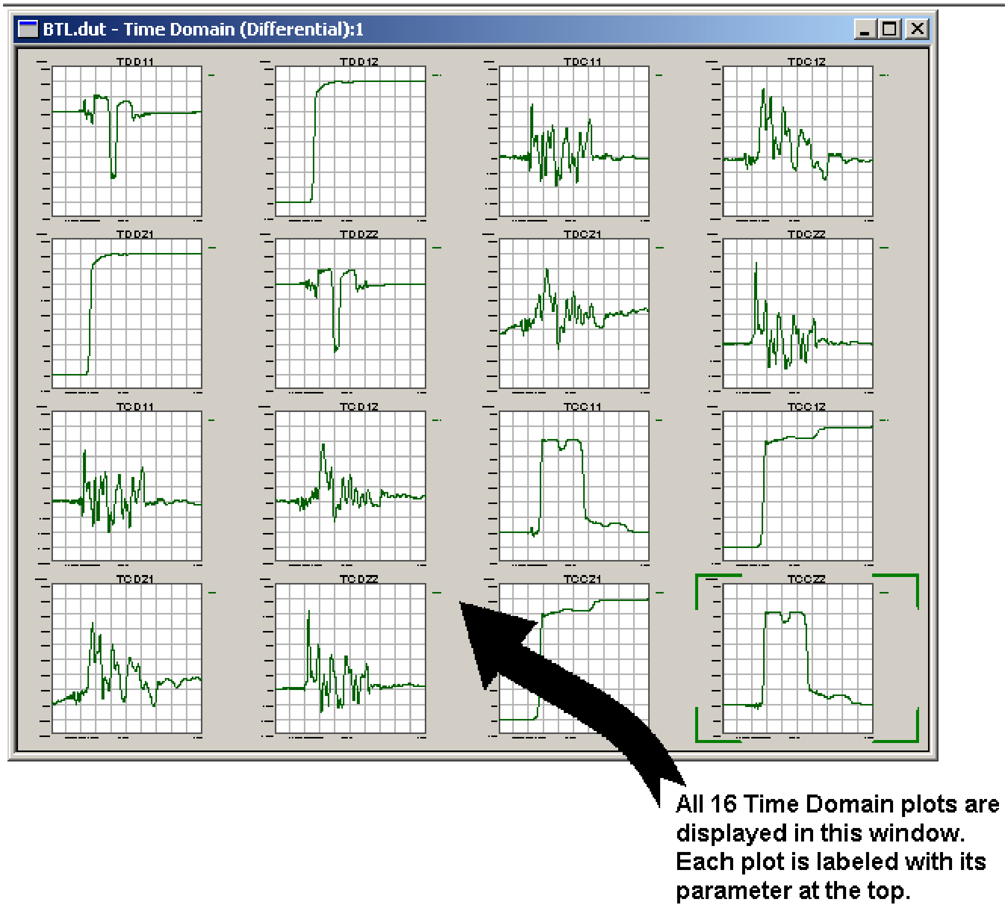
(A) (B)

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 8-11.

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.

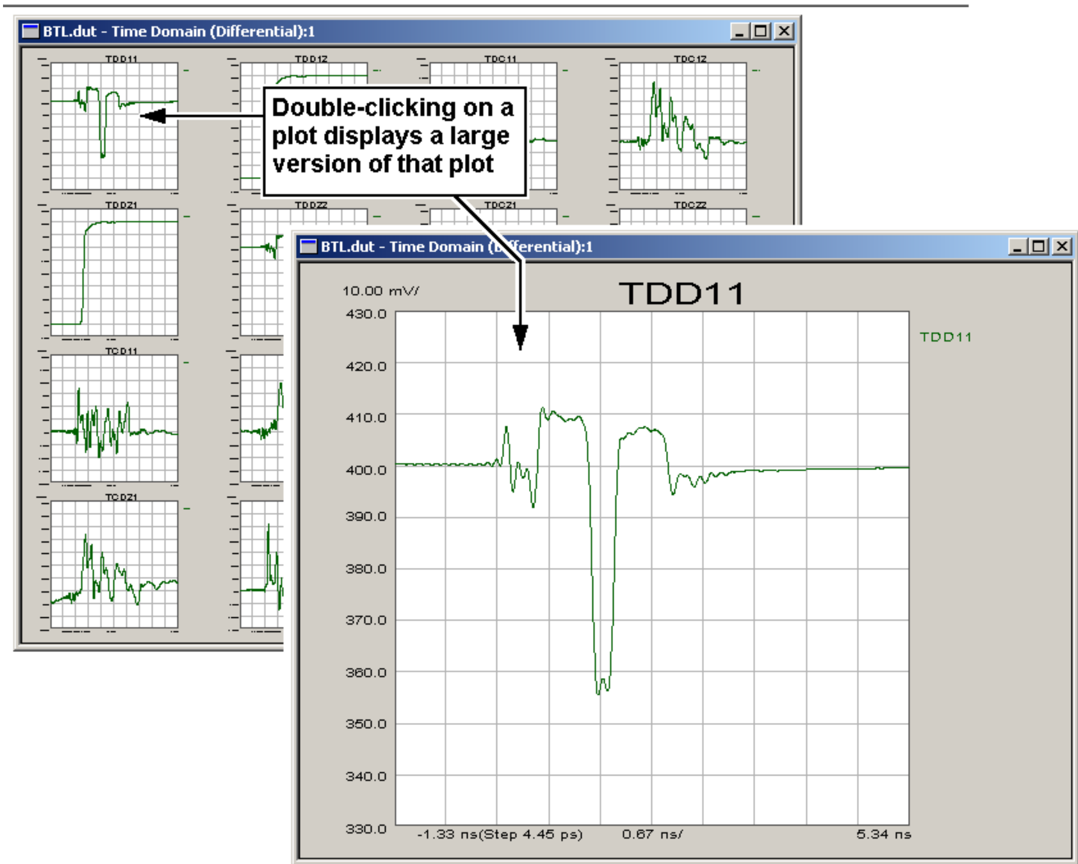
Figure 8-11 All 16 Differential Time Domain Plots



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 8-12.

Figure 8-12 Opening a Single Plot

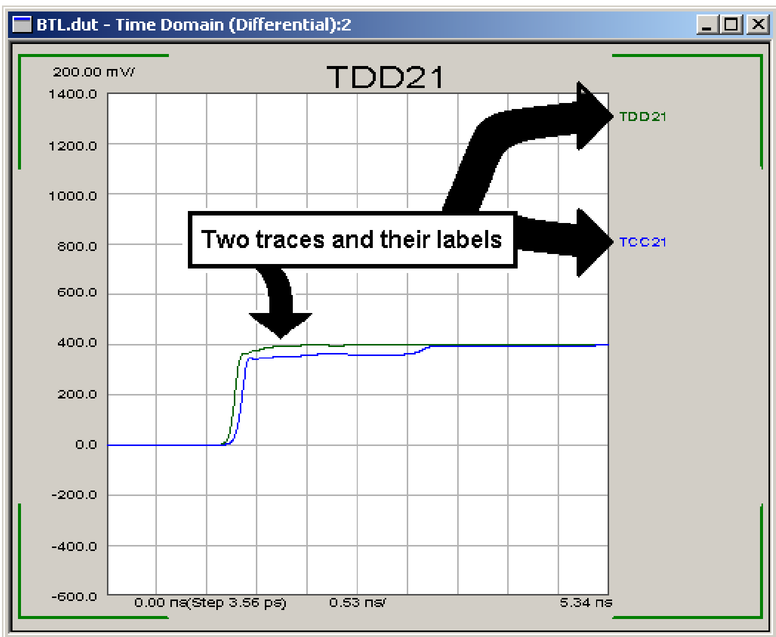


Double-click on the single plot to return back to the original view with multiple plots.

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 8-12. 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 8-13 A Single Plot with Multiple Traces

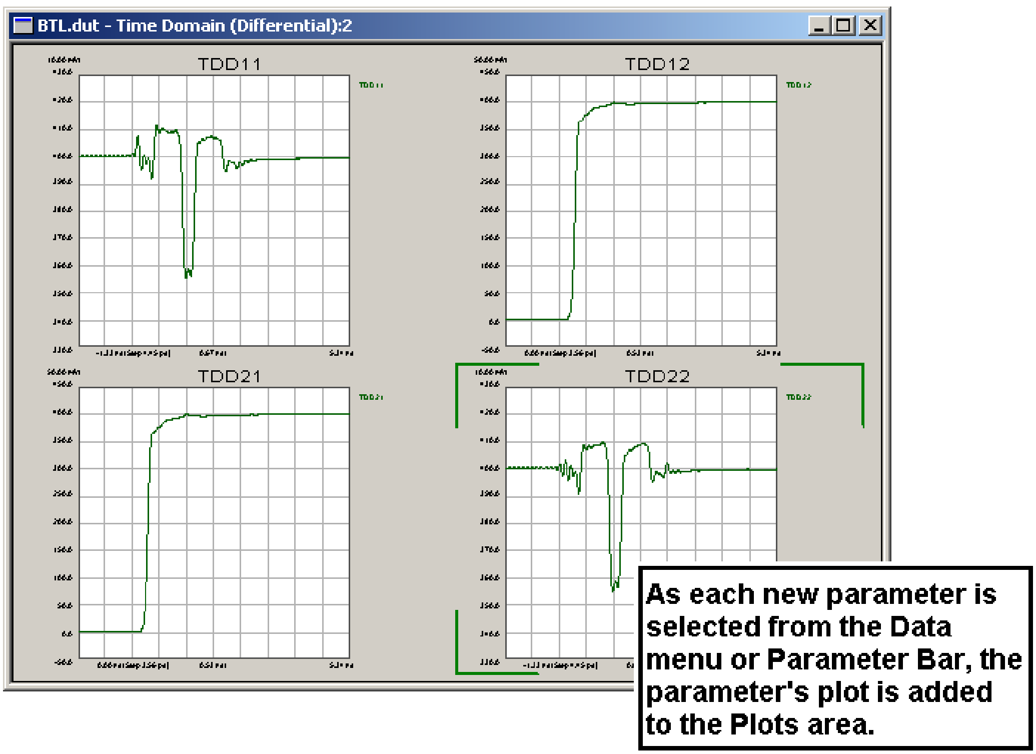


It could contain as many as 4 of the parameters. All trace parameters are labeled at the right edge of the plot. The color of the trace labels are the same color as the traces they represent.

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 8-14.

Figure 8-14 Custom Time Domain Plots Window with Four Plots



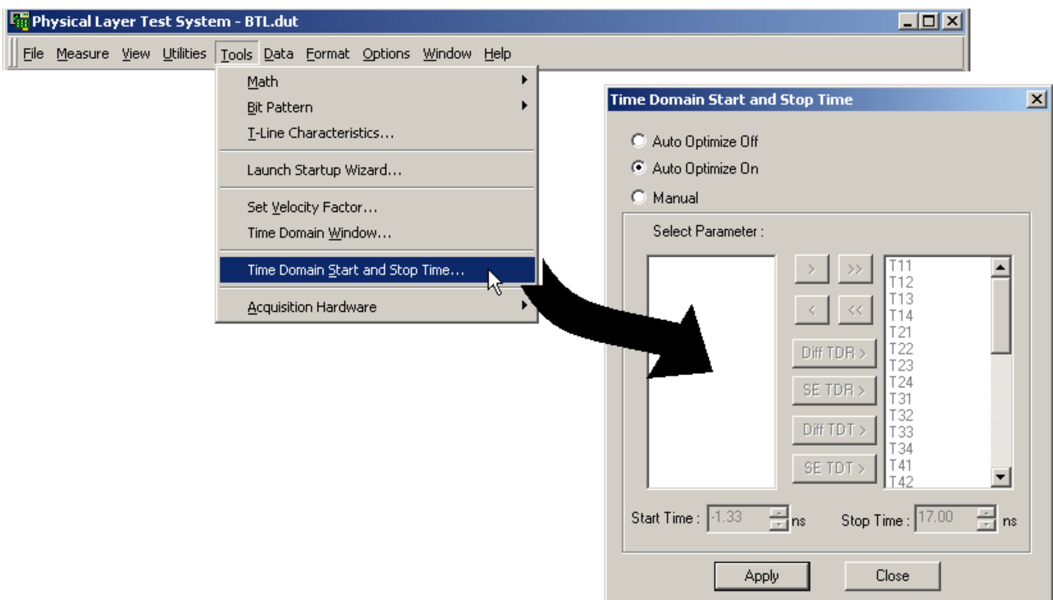
Optimizing the Time Domain Time Scale for Viewing

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.

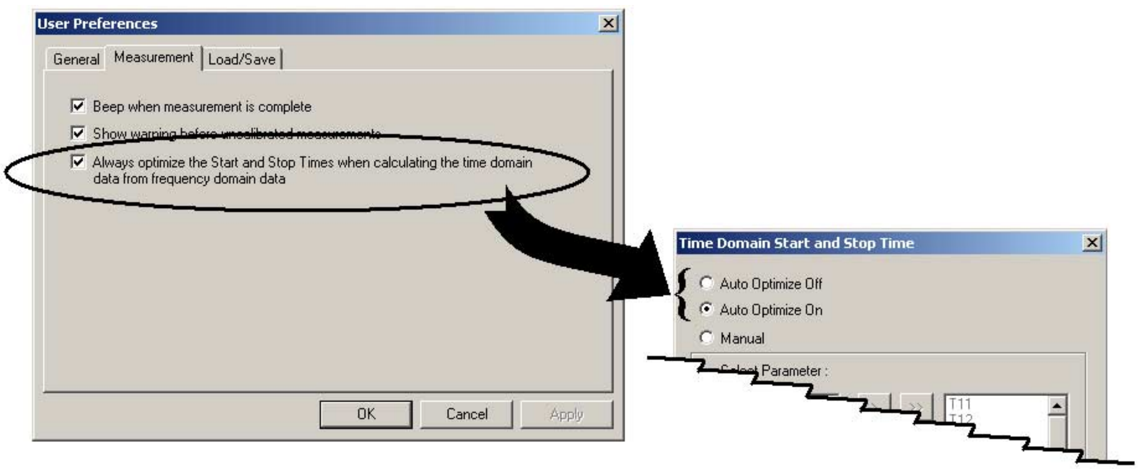
Figure 8-15 Opening the Time Domain Start and Stop Time Dialog Box



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 8-16. Refer to "User Preferences" on page 464 for additional information.

Figure 8-16 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.

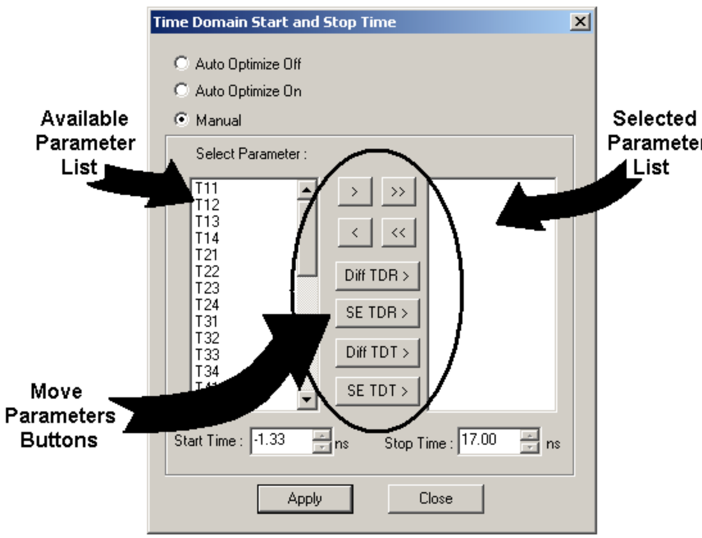
If **Auto Optimize On** is selected, the time domain data is optimized in all parameters for viewing and the start time and stop time are changed to show the time domain time range such that there is not a lot of unusable information displayed.

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 8-17 Time Domain Start and Stop Time Dialog Box in Manual Mode

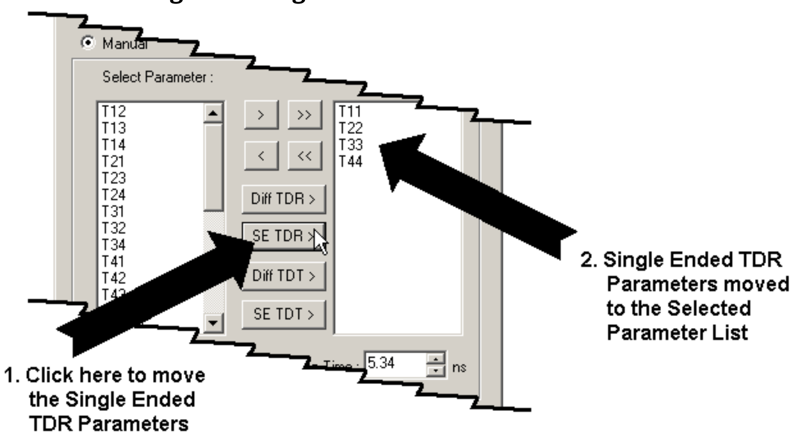


Button Label	Description
>	Moves the highlighted parameters in the available parameters list to the right into the selected parameters list
<	Moves the highlighted parameters in the selected parameters list to the left into the available parameters list
>>	Moves all parameters in the available parameters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

Button Label	Description
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.

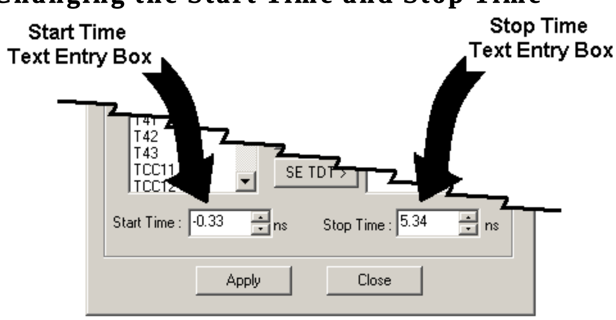
For example, to change the start and stop time of the single ended TDR parameter plots, you must move the parameters to the **Selected Parameter List**. See Figure 8-18.

Figure 8-18 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 8-19.

Figure 8-19 Changing the Start Time and Stop Time



The times can be changed two ways:

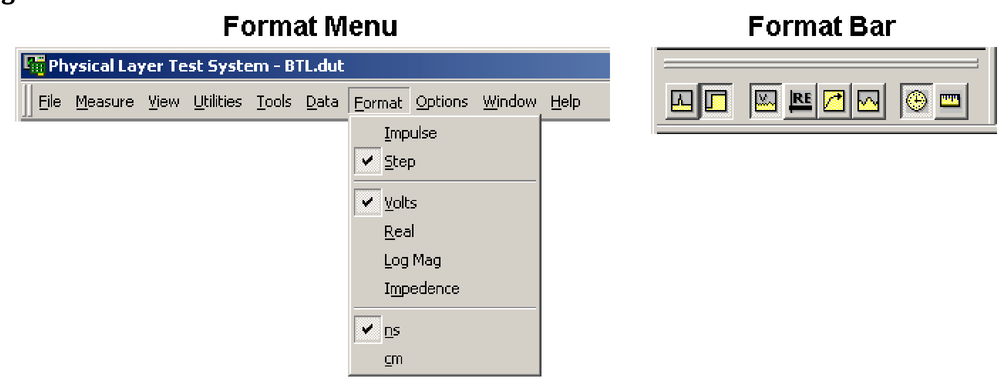
1. By typing in a new value for one or both of the times and then clicking the **Apply** button.
2. By dragging over the value to change with the mouse, and clicking the spinner to the new value. This method changes the value with clicking the **Apply** button.

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 8-19 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

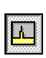
A data format is the way the physical layer test system presents the measurement graphically. Select a data format appropriate to the information you want to learn about the device. You may use either the **Format Bar** or the **Format** menu to select the format.

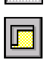
Figure 8-20 Format Menu and Format Bar for Time Domain



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.

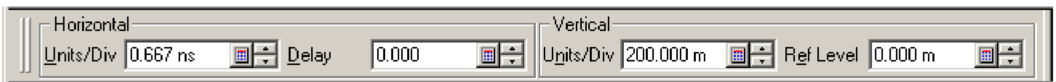
Time Domain Format Bar

	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.

Setting the Scale

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

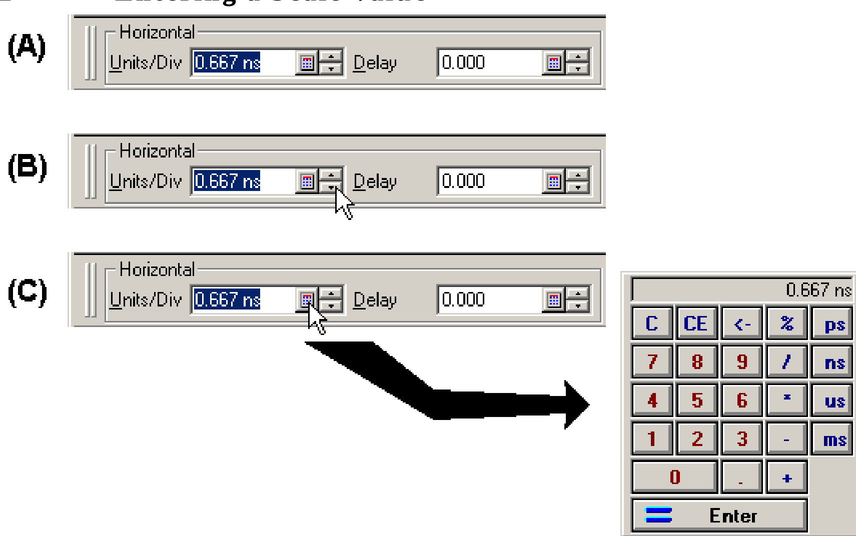
Figure 8-21 Time Domain Scaling Bar



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 8-22.
- Selecting the up/down arrow buttons to the right of each entry. See (B) of Figure 8-22.
- 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 8-22. The scaling calculator icon varies slightly between scaling entries meet the requirements of the specific entry.

Figure 8-22 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:

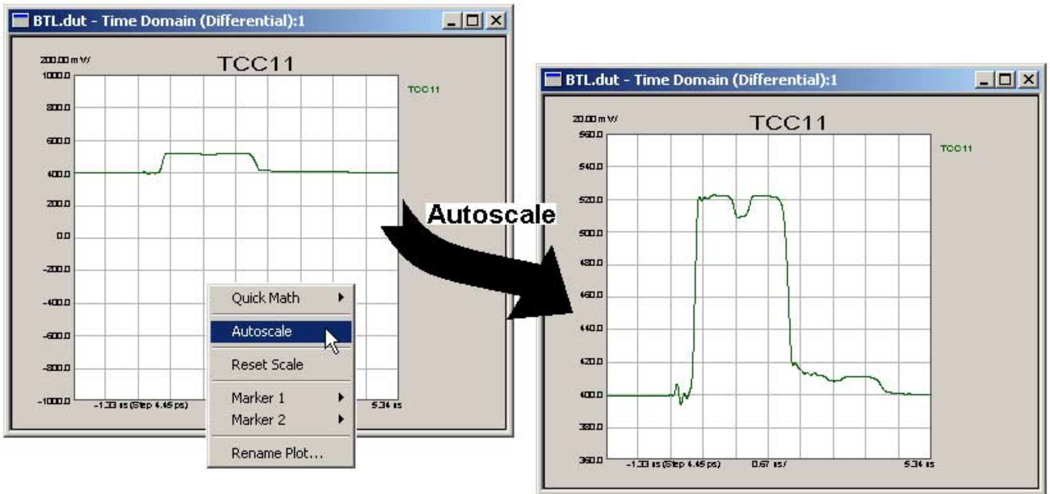
- **Autoscale**
- **Reset Scale**
- **Copy Plot Format** used with **Paste Plot Format** (see "Copying and Pasting Plot Formats" on page 394)

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 8-23. Click **Autoscale** to change the vertical scale of the plot. The figure shows a time domain plot that has **Autoscale** applied to it.

Figure 8-23 Autoscale

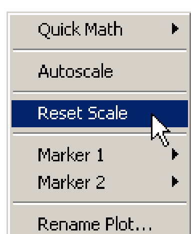


Reset Scale

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 8-24. Click **Reset Scale** to reset the plot to the default settings.

Figure 8-24 Reset Scale



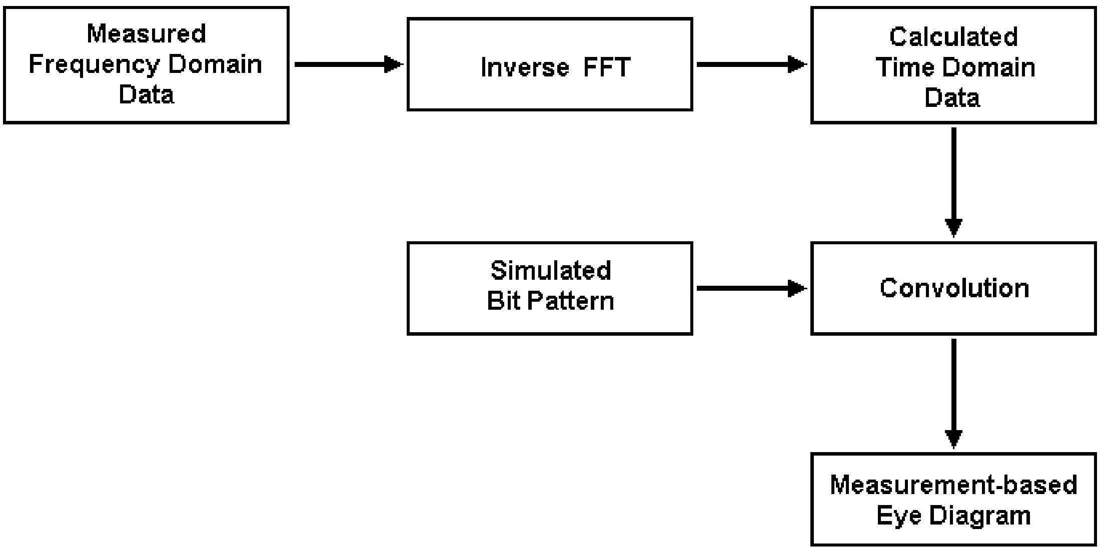
Gating

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 352.

9 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 9-1 for a simplified block diagram of the eye diagram creation process.

Figure 9-1 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.

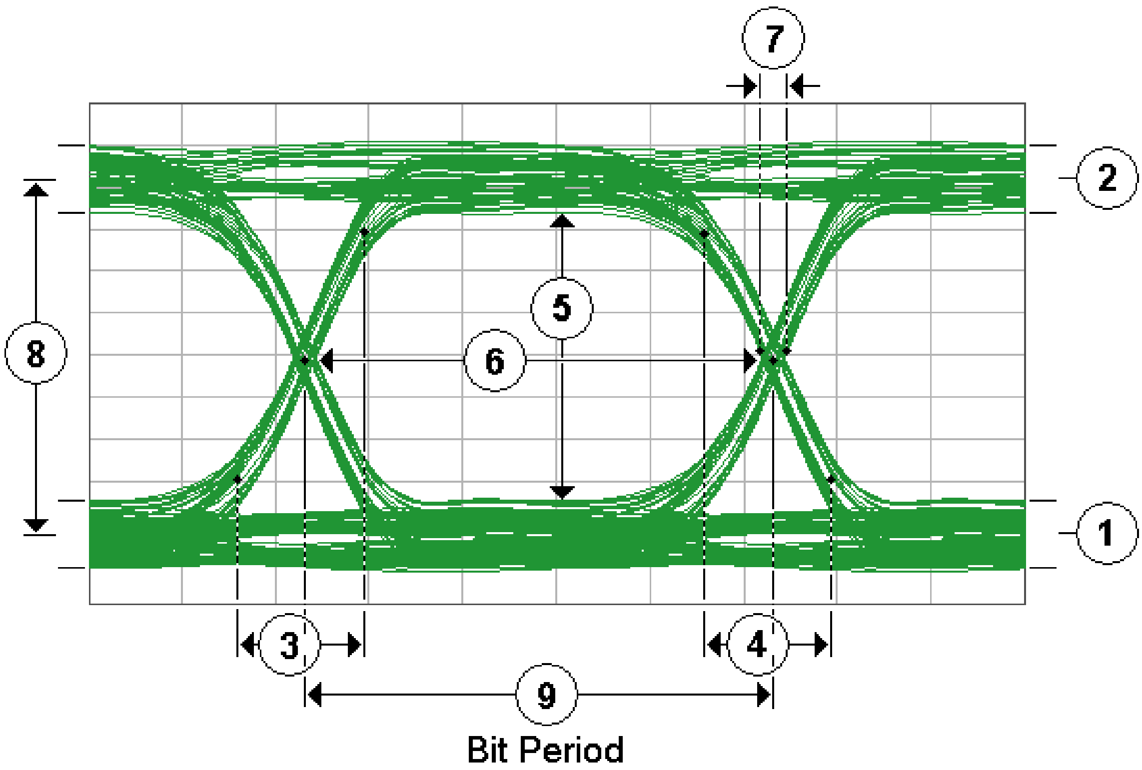
The software allows you to specify a simulated bit pattern (sequence) of between 8 and 32 bits that is convolved with the calculated time domain data to generate the eye diagram. You may select from three simulated bit patterns types: 1) a pre-defined bit pattern, 2) a bit pattern that you have defined, or 3) an arbitrary bit stream (a random-like bit pattern).

The Eye Diagram

The eye diagram shown in Figure 9-2 identifies key eye diagram definitions. These definitions are listed below this illustration. The eye diagram of the TCC21 parameter shown in Figure 9-2 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 9-2 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 enhances the displayed Time Domain plots. 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:

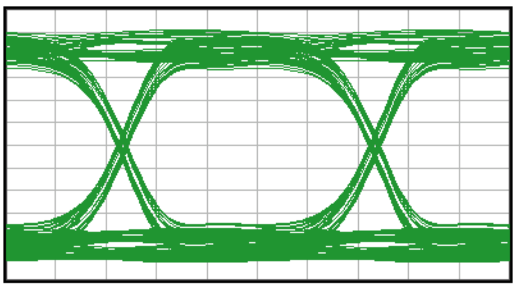
- Finite Impulse Width limits the ability to resolve between two closely-spaced responses. The effects of the finite impulse width cannot be improved with increasing the frequency span of the measurement.
- Impulse Sidelobes limit the dynamic range of the Time Domain measurement by hiding low level response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

sidelobes can be improved by windowing.

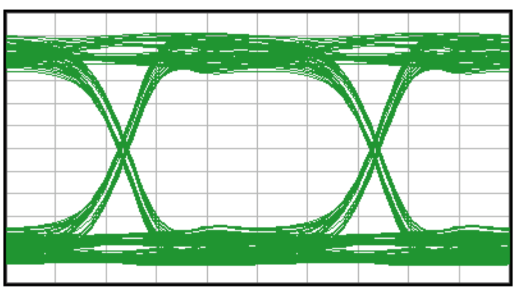
Windowing improves the dynamic range of the Time Domain measurement by modifying (filtering) the Frequency Domain data prior to conversion to the Time Domain to produce an impulse stimulus with lower sidelobes. This greatly enhances the effectiveness in viewing Time Domain responses that are very different in magnitude. The sidelobe 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 a the tradeoff with increased rise time.

Three windows are available: **Maximum**, **Normal**, and **Minimum**. The sidelobe levels of the Time Domain stimulus depend only on the Window that is selected. Figure 9-3 shows the same eye diagram that was displayed in Figure 9-2 with each of the three window settings.

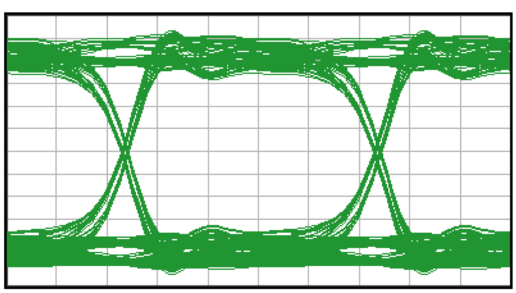
Figure 9-3 The Effect of the Time Domain Windowing Selection



Time Domain Window: Maximum



Time Domain Window: Normal



Time Domain Window: Minimum

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. It is turned on only when the Time Domain response is viewed.

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:

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

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)

FM = Multiplication factor constant for the filter coefficient

As an example: System rise time = 27 ps = (1000/ 20 GHz) × 0.54 (see Table 9-1)

Table 9-1 shows the equivalent rise times for a maximum frequency of 20 GHz for each window setting.

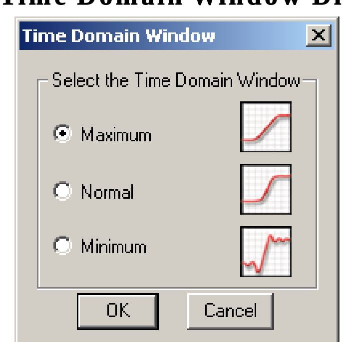
Table 9-1 Equivalent Rise Times for a Maximum Frequency of 20 GHz

Window Setting	Maximum Frequency	Filter Coefficient	Equivalent Rise Time
Minimum	20 GHz	0.54	27 ps
Normal	20 GHz	0.71	35.5 ps
Maximum	20 GHz	0.91	45.5 ps

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 9-4 Time Domain Window Dialog Box



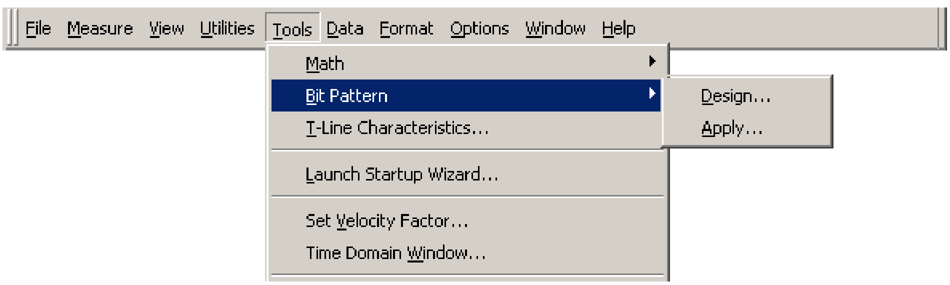
- **Maximum** gives the minimum sidelobes and this provides the greatest dynamic range. The filter coefficient value is 0.91. This is the default setting.
- **Normal** gives reduced sidelobes and is normally the most useful. The filter coefficient value is 0.71.
- **Minimum** is essentially no window and therefore give the highest sidelobes. The filter coefficient value is 0.54.

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**. Then delete the current plot window and reopen a new plot window using the same data. The new windowing choice is then applied to the data.

Designing a Bit Pattern for Eye Diagrams

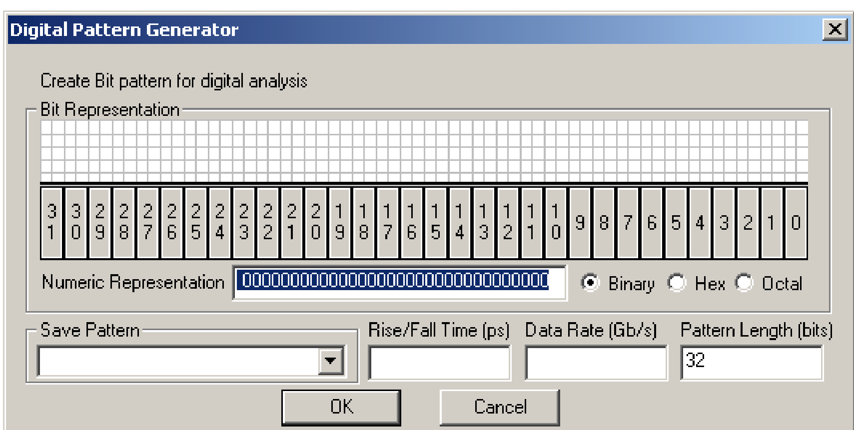
The **Bit Pattern** feature in the **Tools** menu allows you to design a digital bit pattern and to apply a bit pattern that has been saved to eye diagram plots. Only the industry standard K 28.5 bit pattern has been created and included in the software. However, the PLTS software allows you to create bit patterns, from 8 to 32 bits very easily.

Figure 9-5 Tools Menu with Bit Pattern Expanded



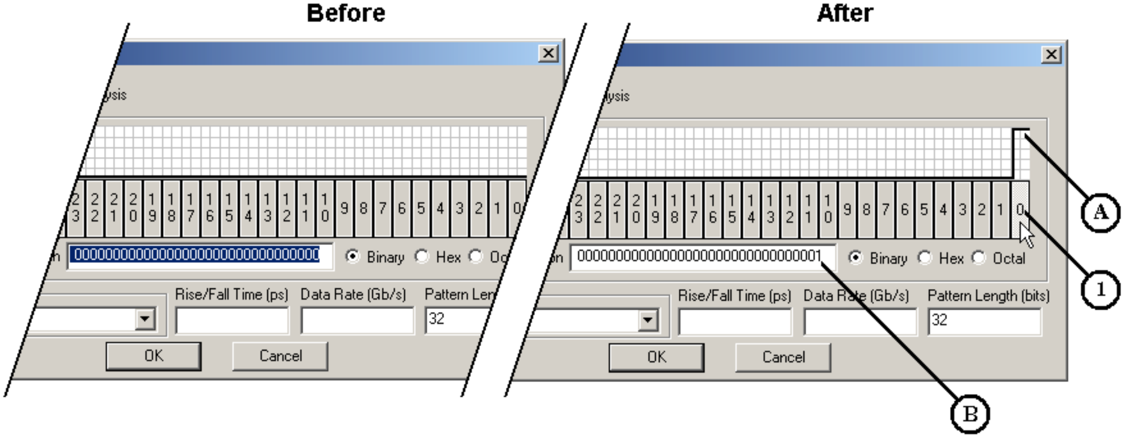
1. Select **Bit Pattern** then **Design**... from the **Tools** menu to open the *Digital Pattern Generator* dialog box shown in Figure 9-6.

Figure 9-6 Digital Pattern Generator



2. Click on the "**0**" key as shown in Figure 9-7.

Figure 9-7 Selecting the "0" Piano Key



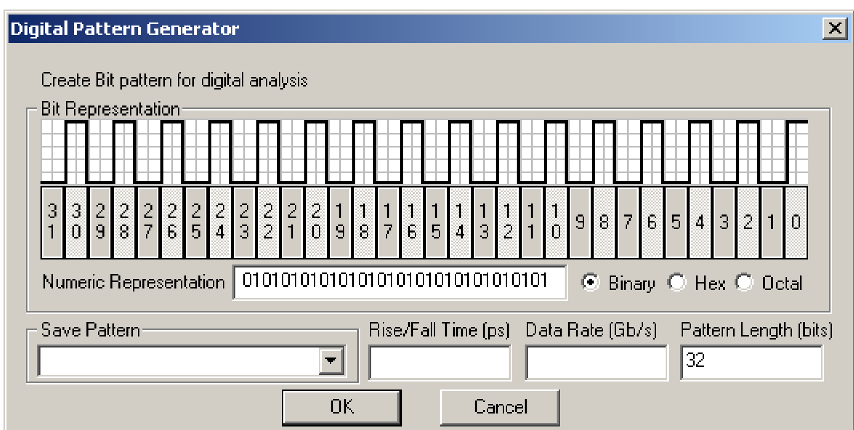
1. **Selecting the "0" key (piano key) in the Bit Representation area:**

   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 9-8.

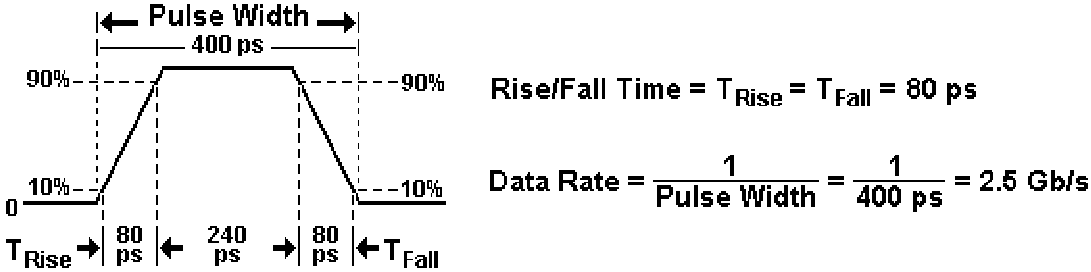
Figure 9-8 Digital Pattern Generator with a 32-Bit Pattern



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 9-9 and Figure 9-10.

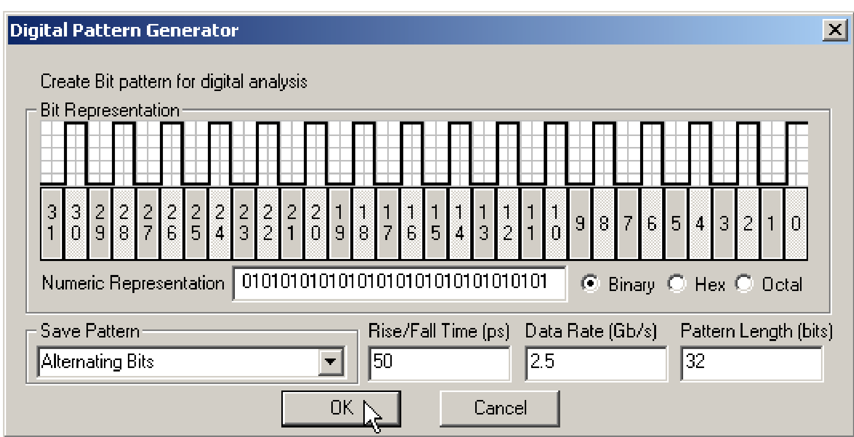
Figure 9-9 Rise/Fall Time and Data Rate



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 9-10.

Figure 9-10 Save the Digital Pattern



Viewing Data using Eye Diagrams

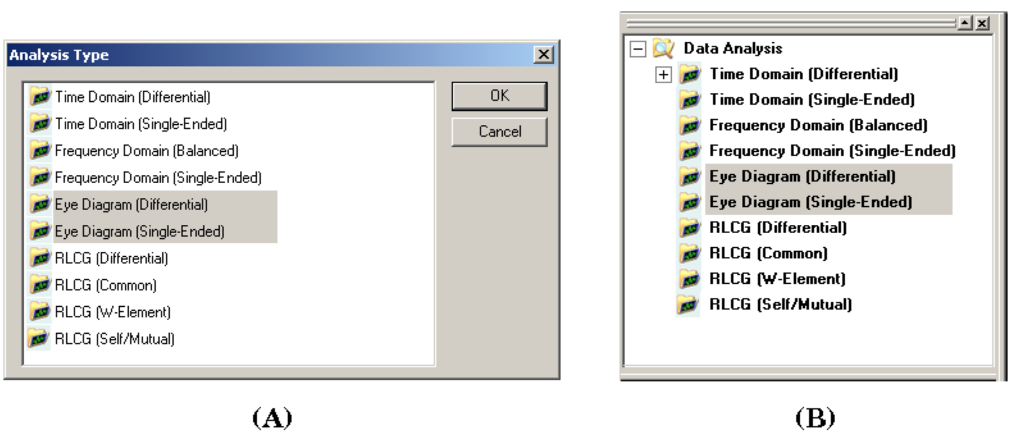
This section guides you with opening measured data in the eye diagram plot mode and viewing the data in the way that best suits your requirements. There are 8 output plots for balanced mode and 12 output plots for single-ended mode. Only transmission paths are displayed; no reflection paths are displayed.

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 9-11
- From the Startup Wizard Welcome Screen by selecting **Load Measurement** where you must select the analysis type - see (A) of Figure 9-11
- 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 9-11
- 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 9-11

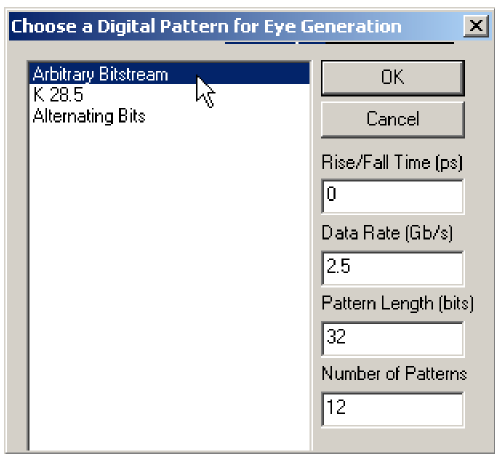
Figure 9-11 Opening the Eye Diagram Plot Window



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 9-12). You may select a digital pattern to apply from the list of digital patterns. The software is shipped with 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 286 are also displayed in the *Choose a Digital Pattern for Eye Generation* dialog box.

Figure 9-12 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 9-12. 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** allowed is between 8 and 32. Where **B** is the pattern length entered, the number of unique bit patterns is: $2^B - 2$. If B = 32 (the maximum number of allowable bits), then 4.29×10^9 unique bit patterns are generated.
- **Number of Patterns** (active only when **Arbitrary Bitstream** is selected) is used to set the number of unique bit patterns used in creating the eye diagram. This value must be equal or less than the number of unique bit patterns determined by the **Pattern Length** entry.

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** and the **Number of Patterns** entries. First, **Pattern Length** is entered (from 8 to 32 bits) defining the number of bits within a specific pattern. Second, **Number of Patterns** is defined and randomly selected which will have a maximum value of $2^{\text{Pattern Length}} - 2$. This is easiest to understand with an example:

If the Pattern Length = 8 bits, there are 256 unique patterns available if you count in binary from 0 to 255.

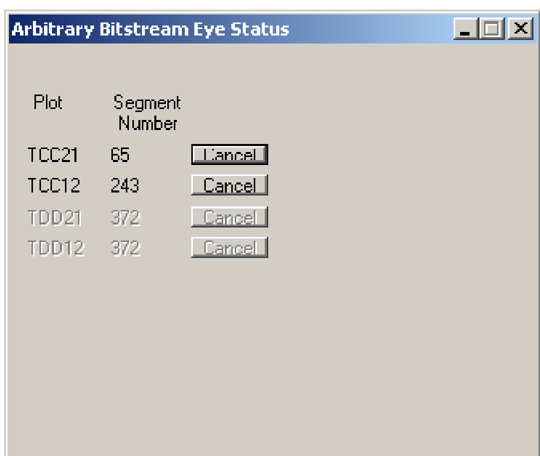
```
00000000
00000001
00000010
```

```
.
.
.
11111101
11111110
11111111
```

ABS then removes patterns 00000000 and 11111111 as invalid patterns so that you now have a total of $2^{\text{Pattern Length}} - 2 = 2^8 - 2 = 254$ 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 254).

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 *Arbitrary Bitstream 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



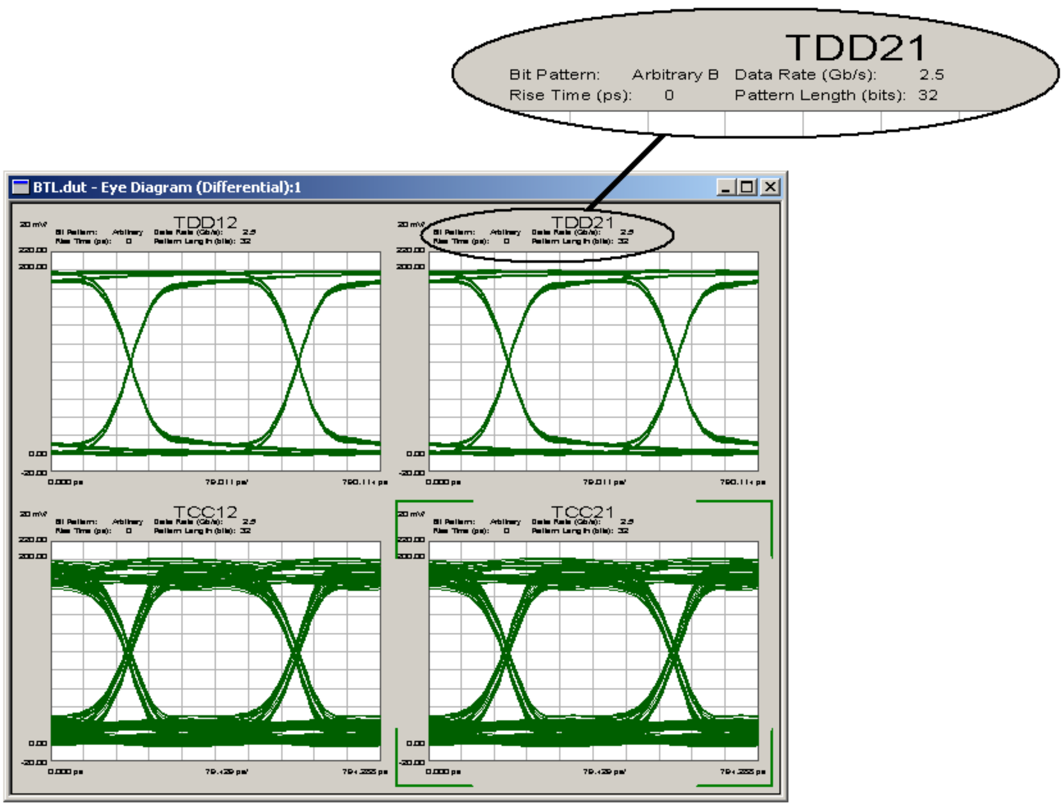
Using this procedure, the arbitrary bit stream is reproduced when the **Pattern Length** entry and the **Number of Patterns** entry are the same values. As the **Number of Patterns** value becomes larger, the time to complete this process also increases.

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 9-13.

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.

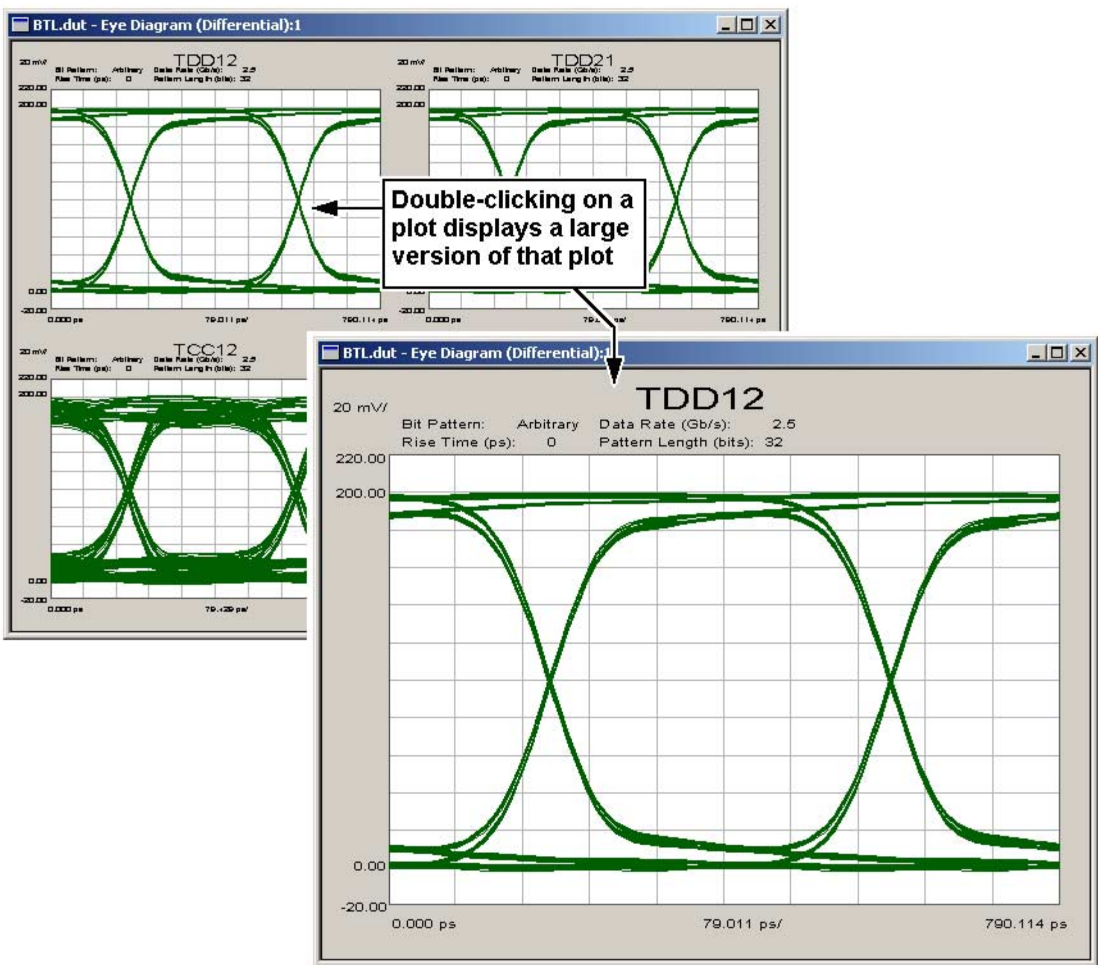
Figure 9-13 Balanced Eye Diagram Plots with Information Bar



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 9-14.

Figure 9-14 Opening a Single Plot

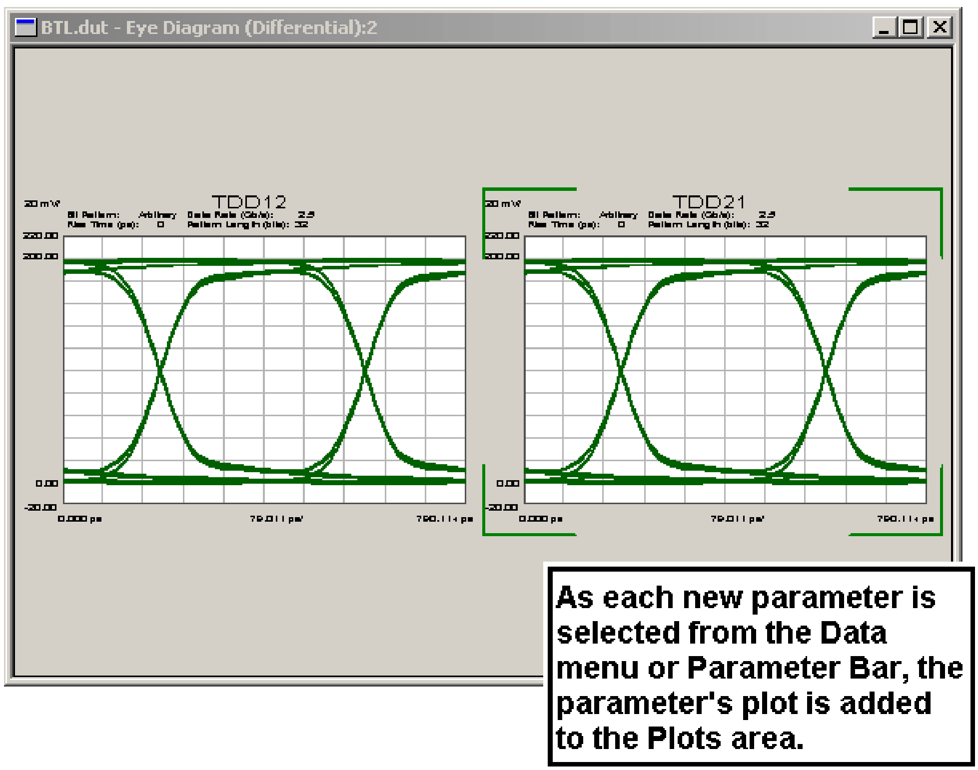


Double-click on the single plot to return to the original view with multiple plots.

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 9-15.

Figure 9-15 Custom Eye Diagram Plots Window with Two Plots



10 Analyzing Transmission Line Parameters

There are four formats in which RLCG parameters may be displayed in PLTS:

- W-Element
- Differential
- Common
- Self/Mutual

Only the **W-Element** format may be exported for use by HSPICE or ADS. The W-Element format 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 formats, **Differential** and **Common**, of the RLCG parameters that treat the coupled line as a 2-port device instead of a 4-port device. These formats 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 formats is directly usable in HSPICE or ADS but these formats can give insight to an experienced user.

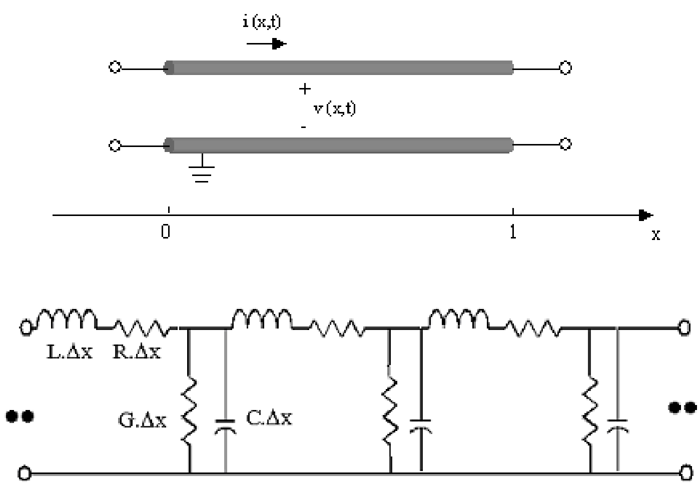
The fourth format is the **Self/Mutual** format. It is a slight deviation from the W-Element format. The only difference in this format is the way that the coupling between parameters is defined. The conversion is described in "CPTL RLCG Extraction Procedure" on page 306.

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.

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 10-1 can be modeled by a network consisting of a series resistance and inductance with parallel capacitance and conductance.

Figure 10-1 RLCG Model for Single Transmission Line

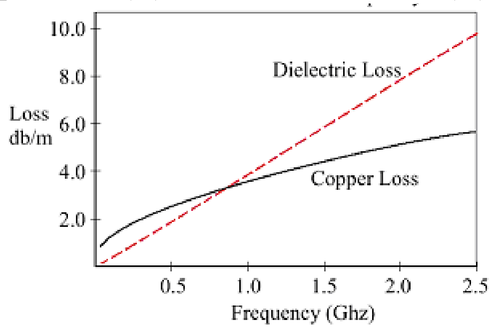


The different terms included in the model describe the following physical phenomena:

- **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 10-2.

Figure 10-2 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 303 for the 2-coupled line model. Telegrapher's equations deal with the voltage and current as shown in Figure 10-1. However, we measure 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

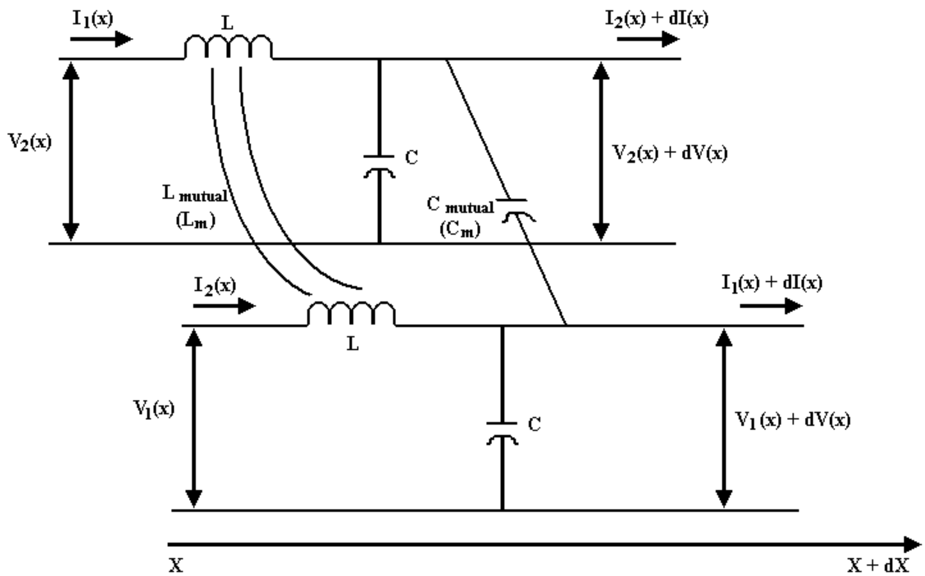
$$C = Im\{\gamma/Z\}/\omega$$

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 the following section.

Coupled-Transmission Line Models

Start with an ideal lossless symmetrical coupled-transmission line (CPTL):

Figure 10-3 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 extented 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 11, “Importing and Exporting Data.”.

The third model representation that will be introduced with PLTS is called Differential-Common Modes Equivalent Model. We created this model mainly for the following reason:

> *The current RLCG extraction algorithm deals only with single-ended transmission lines (SETL). See the “Extracting fitted RLCG parameters from S- parameters” section above. So, we treat each quadrant from the mixed-mode S-parameters, in particular the Diff-Diff and Com-Com, 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

$$R = \begin{pmatrix} R_{dd} & 0 \\ 0 & R_{cc} \end{pmatrix} \qquad G = \begin{pmatrix} G_{dd} & 0 \\ 0 & G_{cc} \end{pmatrix}$$

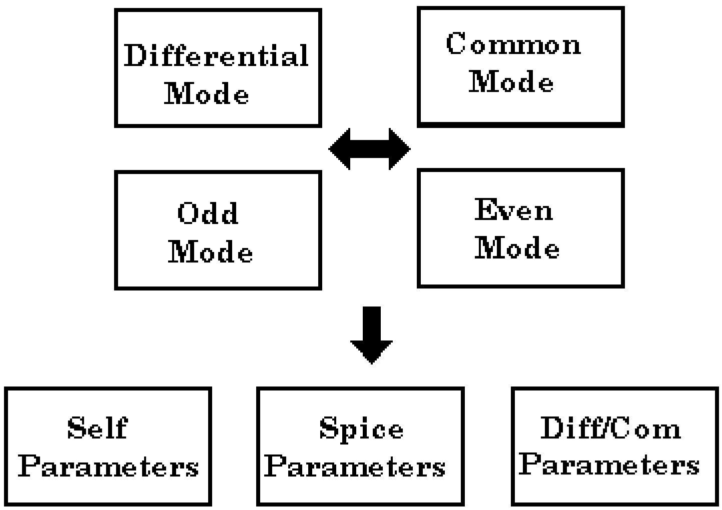
$$L = \begin{pmatrix} L_{dd} & 0 \\ 0 & L_{cc} \end{pmatrix} \qquad C = \begin{pmatrix} C_{dd} & 0 \\ 0 & C_{cc} \end{pmatrix}$$

NOTE *These parameters are called Diff/Com parameters.*

CPTL RLCG Extraction Procedure

As previously described, we start by extracting the RLCG parameters for each of these two modes: Diff-Diff and Com-Com. In the case of a symmetrical CPTL, mode-conversion should be negligible. Then we offer the following options to visualize. See Figure 10-4.

Figure 10-4 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

$$R_S = R_{11}$$
$$R_m = R_{12}$$
$$L_S = L_{11}$$
$$L_m = L_{12}$$

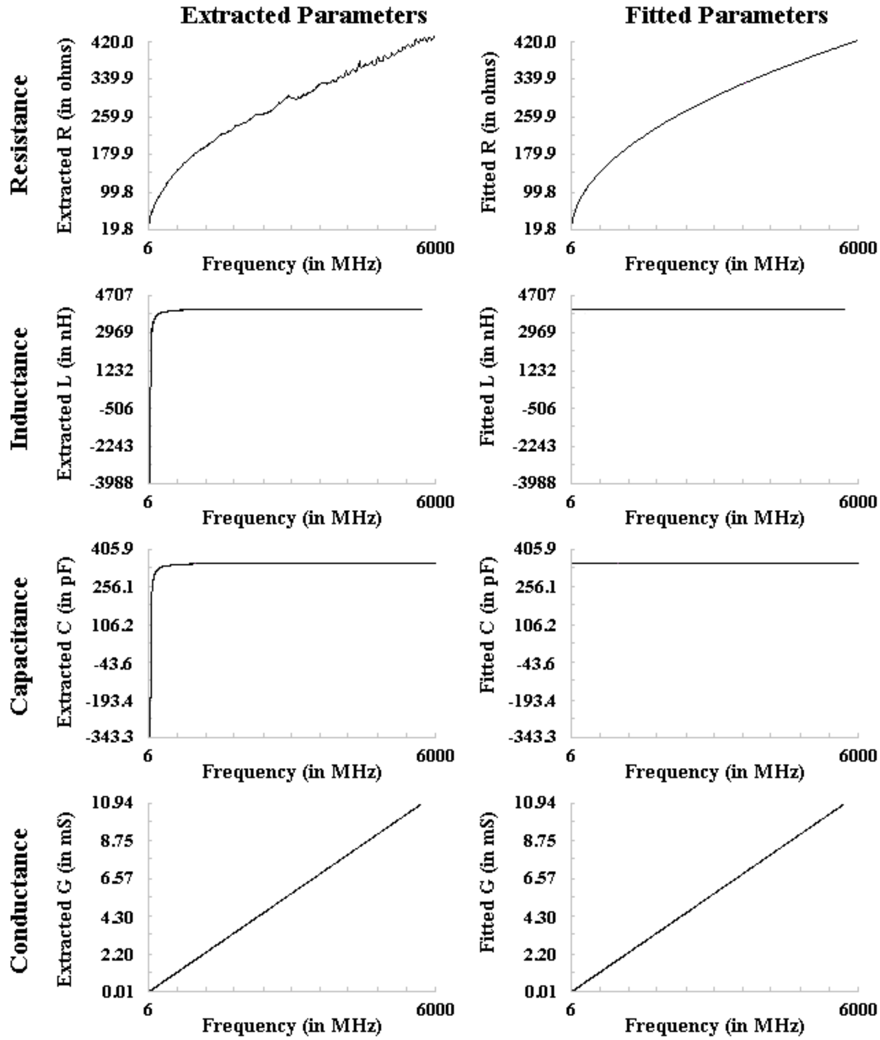
Equation Set 22

$$G_S = G_{11} + G_{12}$$
$$G_m = -G_{12}$$
$$C_S = C_{11} + C_{12}$$
$$C_m = -C_{12}$$

RLCG Output Plots

Figure 10-5 illustrates the difference between the extracted parameters and the fitted curve.

Figure 10-5 Extracted and Fitted Parameters



This plot format should be applied to either Diff/Com, Spice or Self parameters.

Figure 10-6 illustrates the propagation constant and Figure 10-7 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, dB/Phase versus linear or log of the frequency. Version 1.0 displays the familiar real and imaginary part of the impedance and the alpha and beta of the propagation constant.

Figure 10-6 Propagation Constant versus Frequency

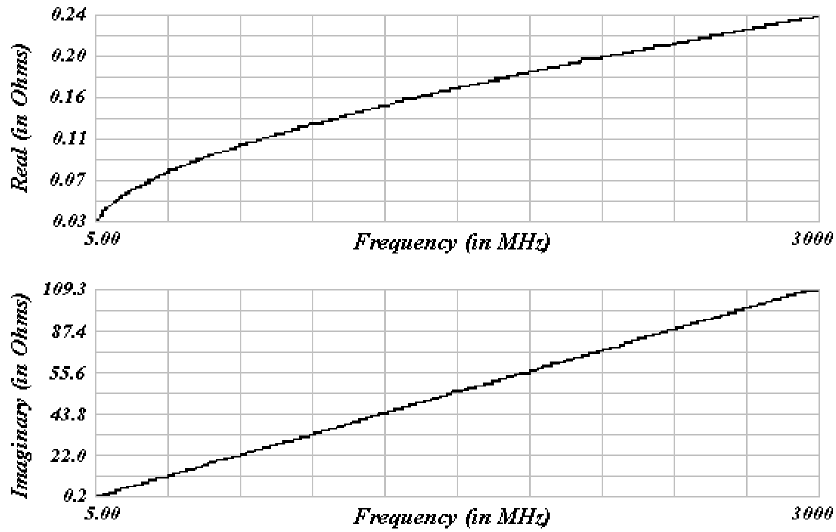
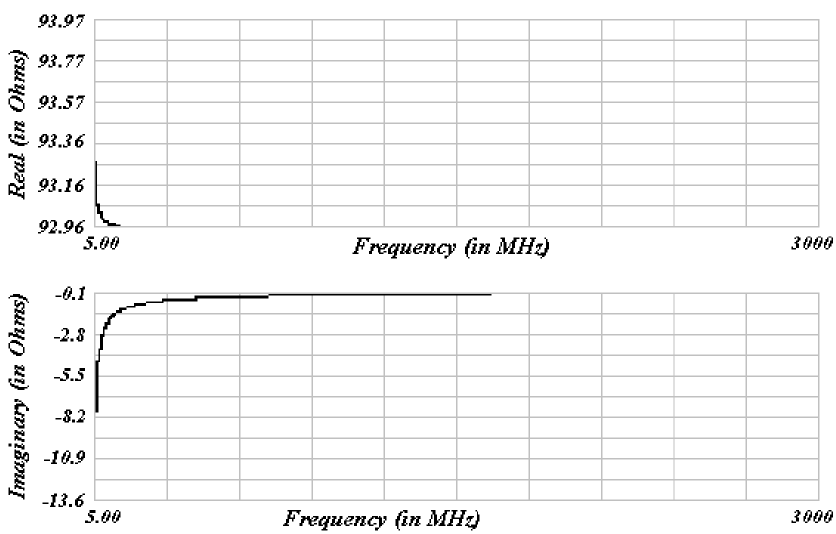


Figure 10-7 Impedance versus Frequency



Export Data Formats

Export data section is intended to link PLTS to the main EDA tools used by R&D engineers in the field of Signal Integrity. These EDA simulators are:

- Agilent ADS
- Avanti HSPICE
- IBM PowerSpice (not supported in Release 1)

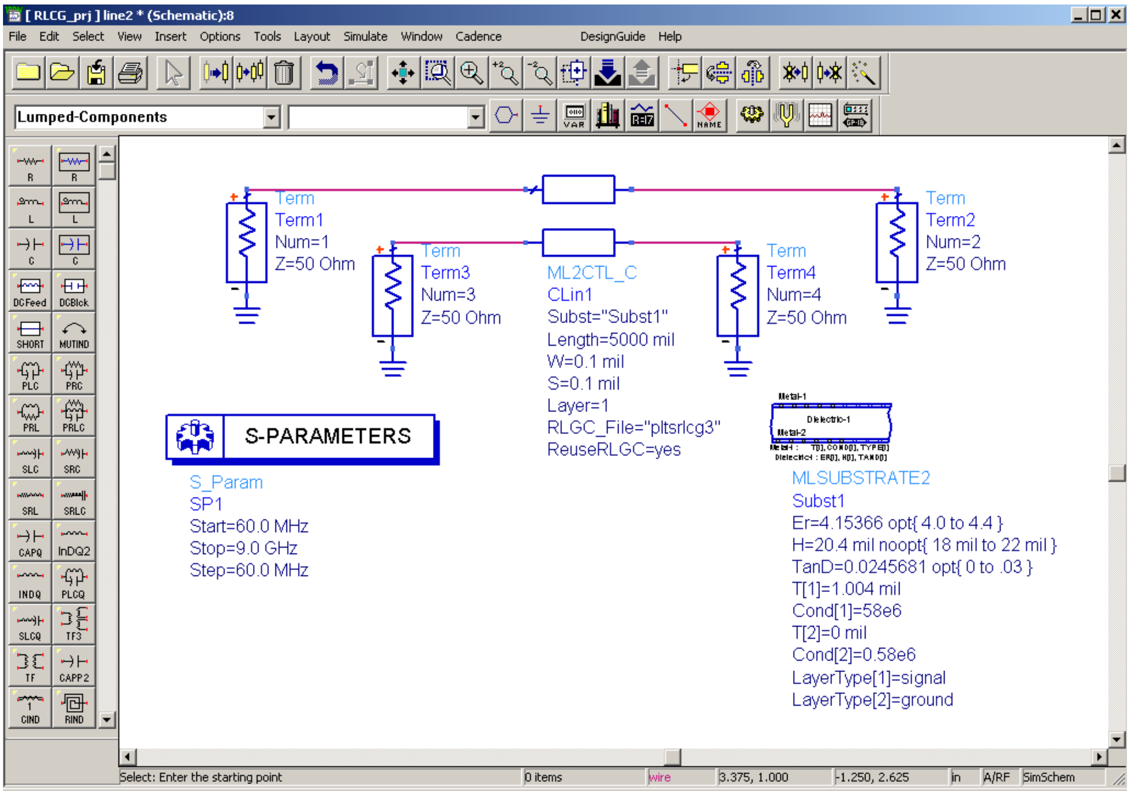
For Agilent ADS, there are two main export formats defined so far. The first format exports fitted parameters in an ASCII file to be used with the ML2CTL_C model found in the Multilayer TL Library. The syntax is described in the following example:

```
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 epsilon zero, the inductance values divided by mu zero, and the high frequency resistance (Rs) 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 10-8 is an ADS schematic using the RLCG parameters to simulate the S-parameters of the modeled line.

Figure 10-8 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 10-9 and Figure 10-10.

Figure 10-9 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, 4.4397620152 89E-005
2, 12, 3.06090516578, 4.20664887436E-007, 8.10560436923 4E-011, 8.87952304632 5E-005
3, 18, 2.98776516048, 4.24233863 7733E-007, 8.17200728402 6E-011, 0.00013319284 07736
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, 7.873960681872E-011, 0.06637442742384
1496, 8976, 42.38159372248, 4.092893142066E-007, 7.873954173886E-011, 0.06641882503415
1497, 8982, 42.32048825007, 4.092908898521E-007, 7.873954625942E-011, 0.06646322264446
1498, 8988, 41.67401476677, 4.092898108697E-007, 7.873922680918E-011, 0.06650762025477
1499, 8994, 41.10014201313, 4.092911729309E-007, 7.873922950849E-011, 0.06655201786508
1500, 9000, 40.26217750834, 4.092962415155E-007, 7.873988530006E-011, 0.06659641547539
END
```

Figure 10-10 Exported Extracted Parameter File Format for Mutual Parameters

```
BEGIN DSCRDATA
%INDEX Freqs R12 L12 C12 G12
1, 6, 0.35556715478, 8.706948649014E-008, -9.608040973318E-012, -1.95408990758E-006
2, 12, 0.37713958302, 8.563925421224E-008, -9.456841514979E-012, -3.908173909648E-006
3, 18, 0.32124892501, 8.67333738673E-008, -9.461437082827E-012, -5.862257911717E-006
4, 24, 0.42018406929, 8.647508101033E-008, -9.457617452665E-012, -7.816341913785E-006
5, 30, 0.35282987035, 8.64442670185E-008, -9.408781640124E-012, -9.770425915853E-006
6, 36, 0.41402798975, 8.633784294817E-008, -9.380885886563E-012, -1.172450991792E-005
7, 42, 0.41090926359, 8.636445887829E-008, -9.371817650767E-012, -1.367859391999E-005
8, 48, 0.42575559378, 8.631596140371E-008, -9.35652579649E-012, -1.563267792206E-005
9, 54, 0.51094252063, 8.628712327937E-008, -9.328975338111E-012, -1.758676192413E-005
10, 60, 0.50611430653, 8.6161443332E-008, -9.328216019483E-012, -1.954084592619E-005
.
.
.
.|
1495, 8970, 6.44695845120, 8.525684615435E-008, -8.80320658165E-012, -0.002921355586998
1496, 8976, 6.17257628863, 8.525881387137E-008, -8.802820910516E-012, -0.002923309673
1497, 8982, 6.05775653757, 8.526378054128E-008, -8.801900246924E-012, -0.002925263757002
1498, 8988, 5.81947087962, 8.526529361159E-008, -8.801519409959E-012, -0.002927217841004
1499, 8994, 5.63914201175, 8.526960580376E-008, -8.800719682472E-012, -0.002929171925006
1500, 9000, 5.46693156328, 8.527562170566E-008, -8.799807956971E-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
```

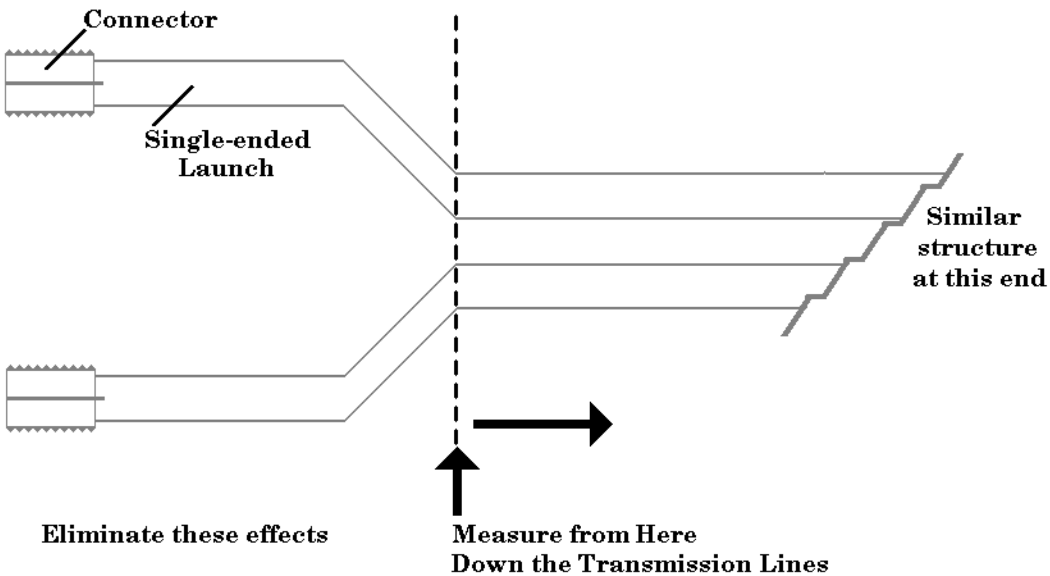
To use this file in HSPICE use the W element and have the “RLCGfile” parameter point to the file exported from PLTS. A simple netlist using the W element and the exported parameters to do an S-parameter simulation of the modeled line is shown in the following example:

```
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 two-coupled 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 10-11 shows the connector and launches that need to be removed.

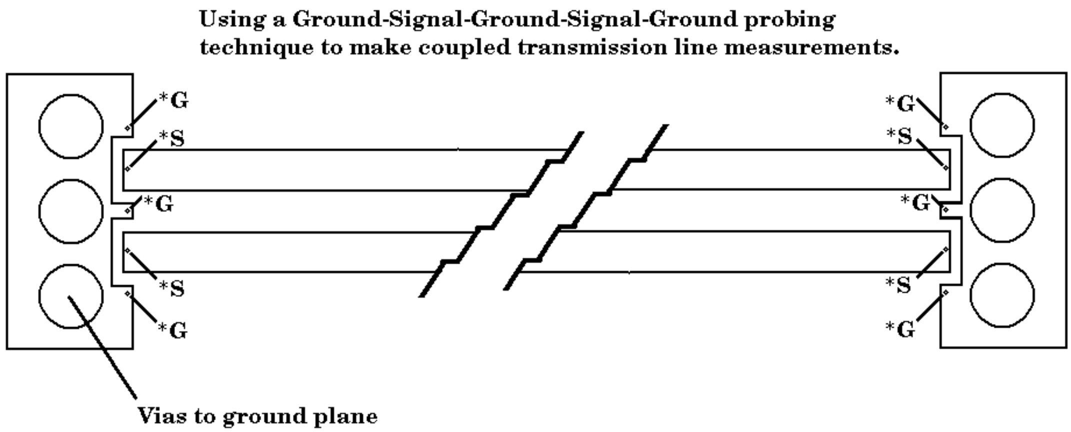
Figure 10-11 Typical Connectorized Device to be Tested



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 10-12 shows a typical probed measurement.

Figure 10-12 Typical Probed Measurement of a Transmission Line



The Parameters for Each RLCG Format

The data (parameters) available for each of the four RLCG formats 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 format. You may elect to view any number of these parameters.

Opening a Transmission Line Plot Window

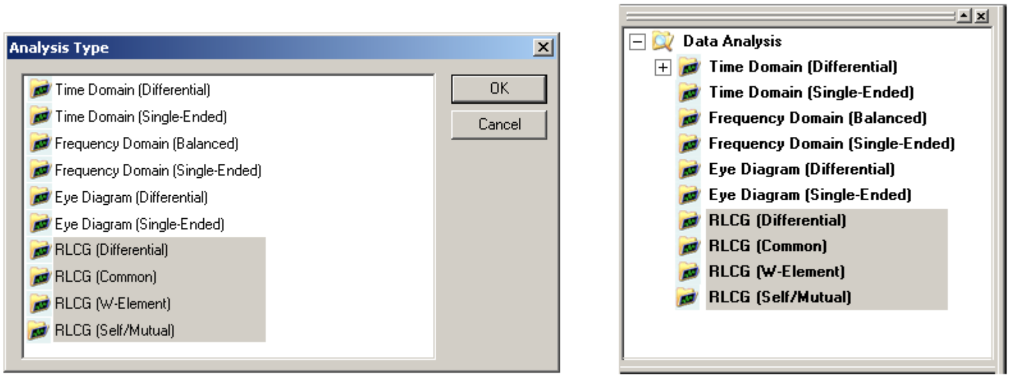
The transmission line parameters may be viewed in any of four formats (differential, common, W-Element, and self/mutual).

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 10-13
- From the Startup Wizard Welcome Screen by selecting **Load Measurement** where you must select the analysis type - see (A) of Figure 10-13
- 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 10-13
- From the **Browser** when data is already being viewed in another analysis type by selecting one of the RLCG choices - see (B) of Figure 10-13

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.

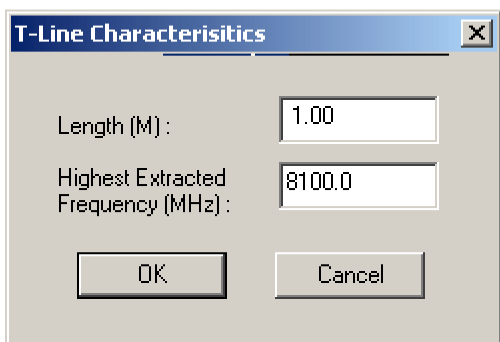
Figure 10-13 Opening the Transmission Line Plot Window



(A) (B)

The *T-Line Characteristics* dialog box (Figure 10-14) is then displayed. Enter the length of the transmission line (in meters) and the highest measured frequency (in megahertz) and then click **OK**.

Figure 10-14 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. 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, 45 GHz for a 50 GHz measurement or in the case shown in Figure 10-14, 8100 MHz for a 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 food 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

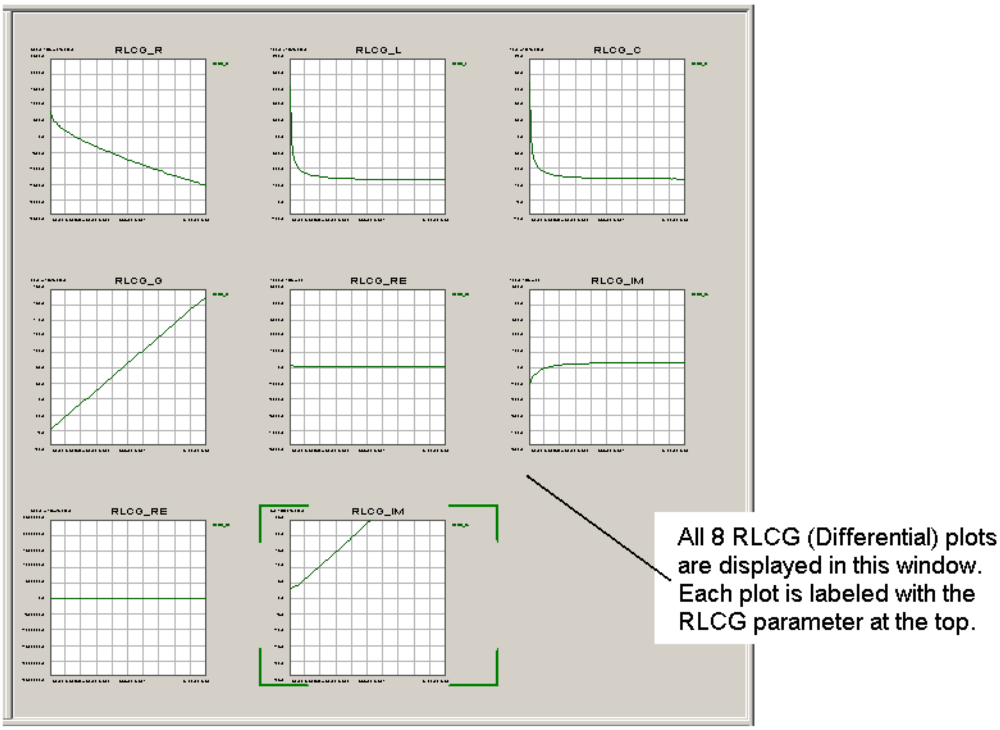
where, **A** represents the Attenuation Constant (α)
B represents the Phase Constant (β)
C represents Capacitance
G represents Conductance
L represents Inductance
R represents Resistance
Z represents Impedance

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 are labeled. See Figure 10-15.

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.

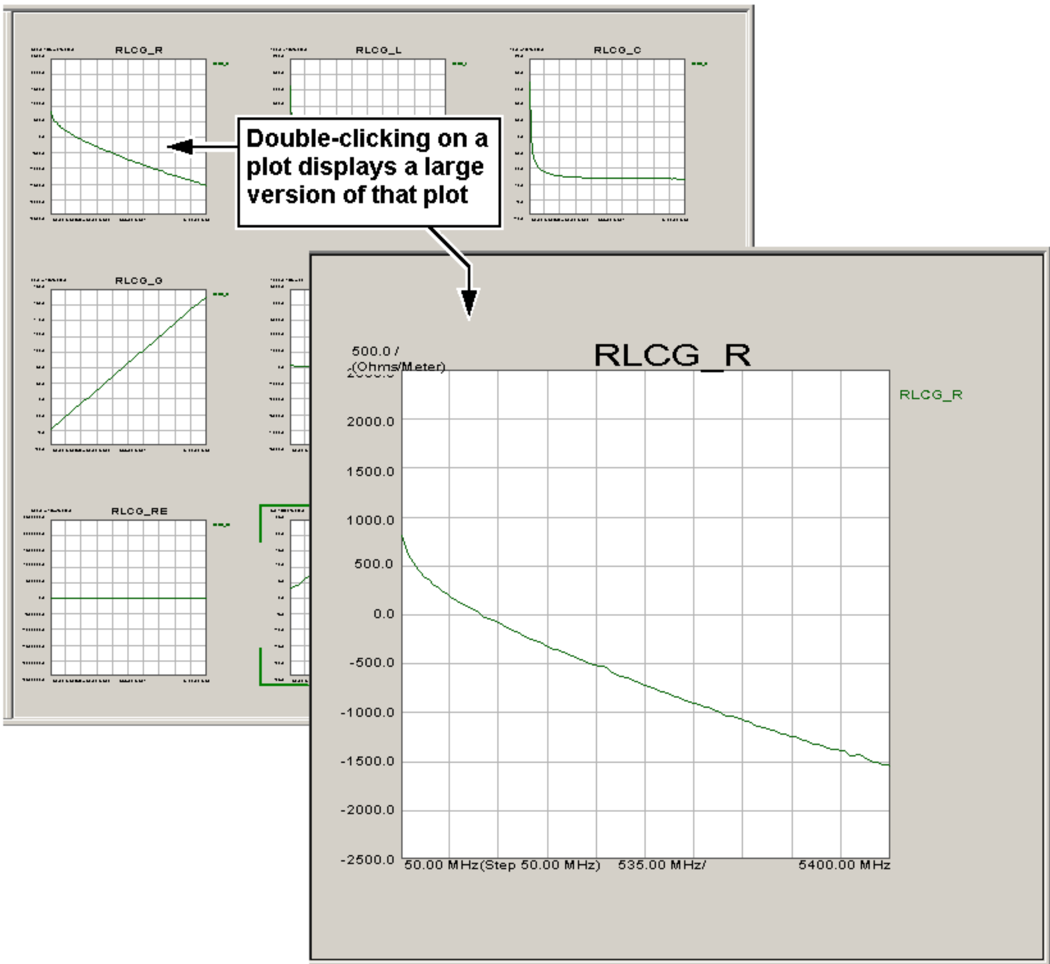
Figure 10-15 All 8 RLCG Plots



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 10-16.

Figure 10-16 Opening a Single Plot

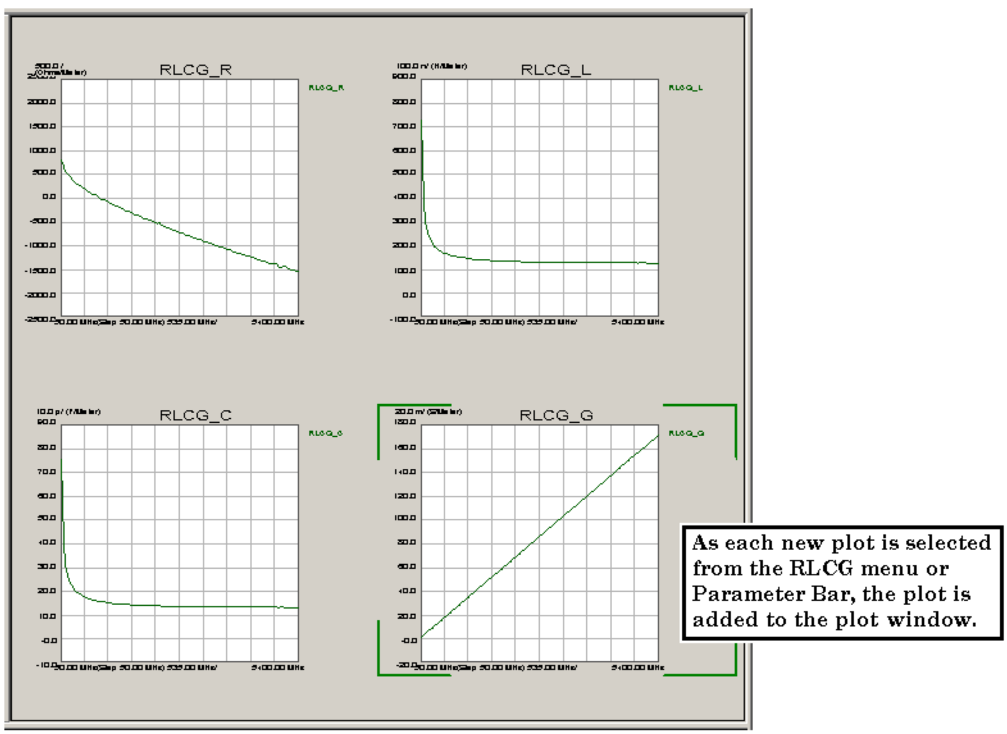


Double-click on the single plot to return to the original view with multiple plots.

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 10-17.

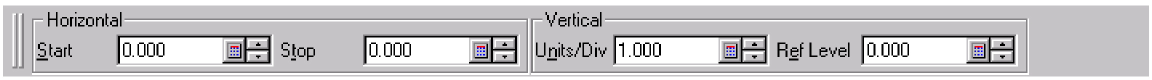
Figure 10-17 Custom Plots Window with Four Plots



Setting the Scale

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

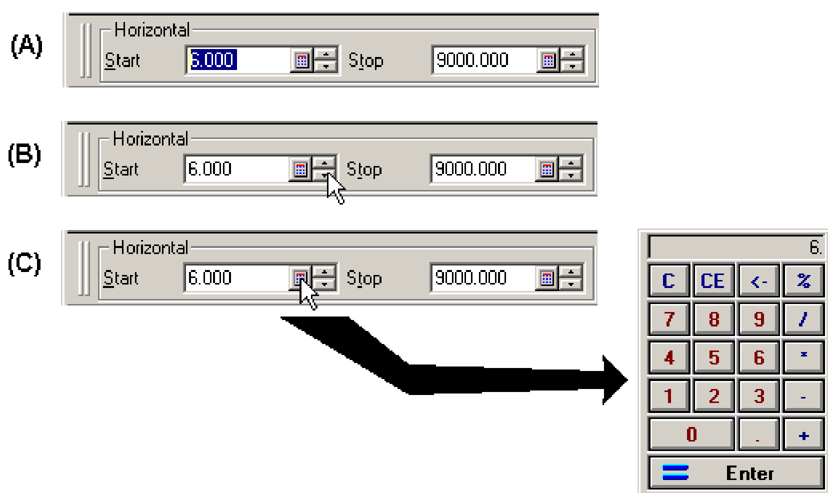
Figure 10-18 Scaling Bar



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 10-19.
- Selecting the up/down arrow buttons to the right of each entry. See (B) of Figure 10-19.
- 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 10-19. The scaling calculator icon varies slightly between scaling entries meet the requirements of the specific entry.

Figure 10-19 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.

Exporting Transmission Line Data

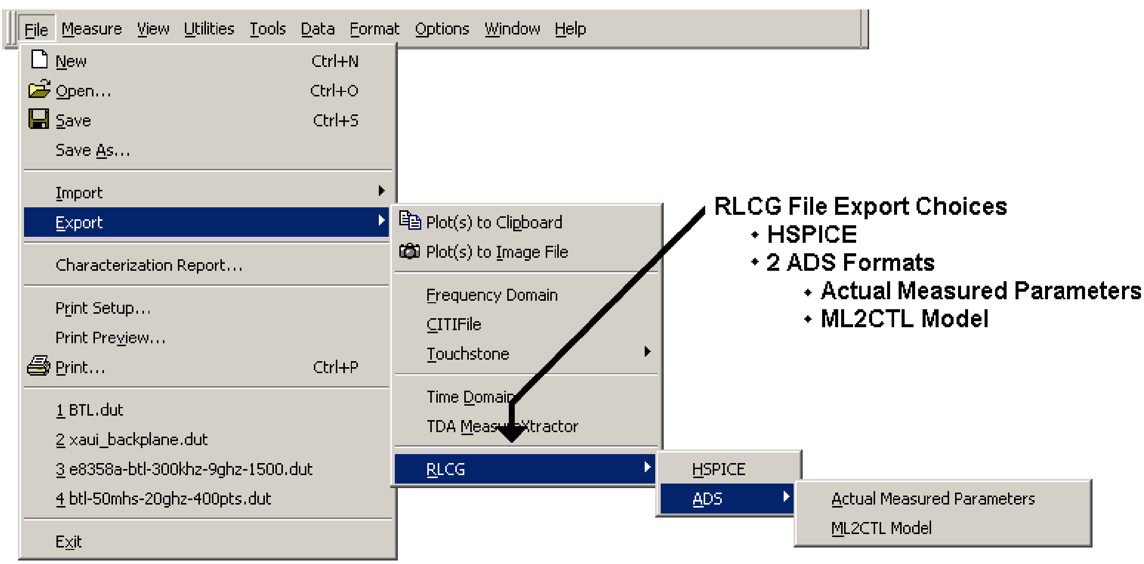
The PLTS can export the transmission line data to the main electronic design automation (EDA) software tools used by R&D engineers in the field of signal integrity. These EDA simulators are:

- HSPICE
- ADS (Advanced Design System - Integrated Design Software and Test Instrumentation solution from Agilent Technologies)

Refer to "Export Data Formats" on page 310 for detailed information on the file formats for exported data.

Selecting the **Export** in the **File** menu to access the three RLCG export menu selections. A brief description and the syntax of each selection is listed in this section.

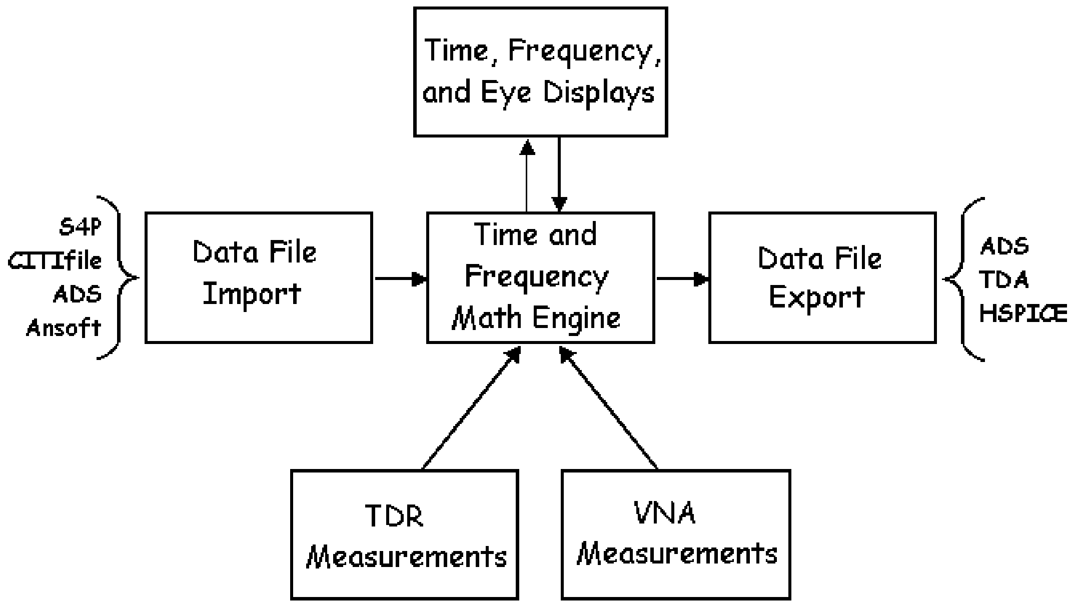
Figure 10-20 RLCG Export Menu



11 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 and you will 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 11-1 shows how the data file import and data file export capabilities blend with the whole PLTS software product.

Figure 11-1 PLTS Data File Import and Export Block Diagram



File Formats

This section describes the S4P (Touchstone) and the CITIfile file formats. The Touchstone file format begins on page 334.

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 11-1 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 11-1 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 11-1 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: 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 described by frequency-dependent linear network parameters for 4-port components. The S4P data file format is the type of file format that also known as Touchstone format.

An *.s4p* file can be used with an S4P component to model the behavior of a linear model using S-parameters. The file contains the S-parameters, the component 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 set of units desired (GHz, MHz, KHz, 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 files with Frequency in GHz, S-parameters in real-imaginary format, normalized 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
```

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.

Figure 11-2 File Menu with Import and CITIFile Expanded

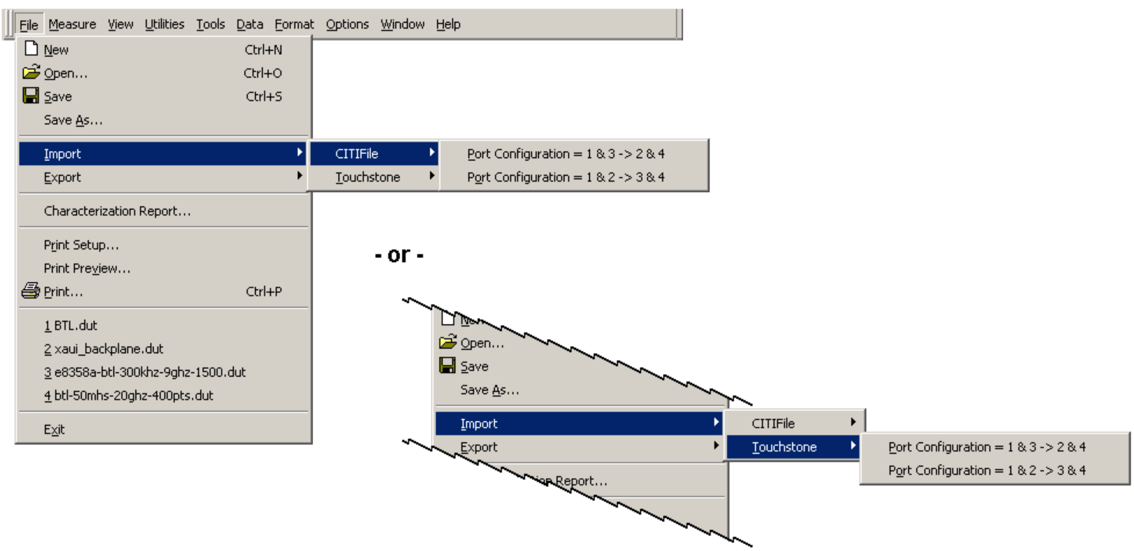
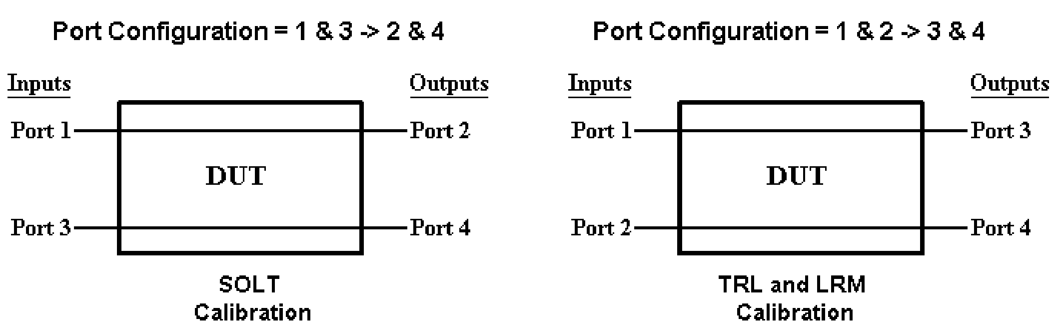


Figure 11-3 Balanced Transform Port Configuration Diagram



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 11-3 and choose from one of the following port selections.

Port Configuration = 1 & 3 -> 2 & 4 is used to import single-ended measurement data that has taken with the system calibrated using the SOLT calibration.

Port Configuration = 1 & 2 -> 3 & 4 is used to import single-ended measurement data that has taken with the system calibrated using the TRL calibration.

Touchstone

Touchstone imports data previously saved in Touchstone (*.S4P) format. Refer to Figure 11-3 and choose from one of the following port selections.

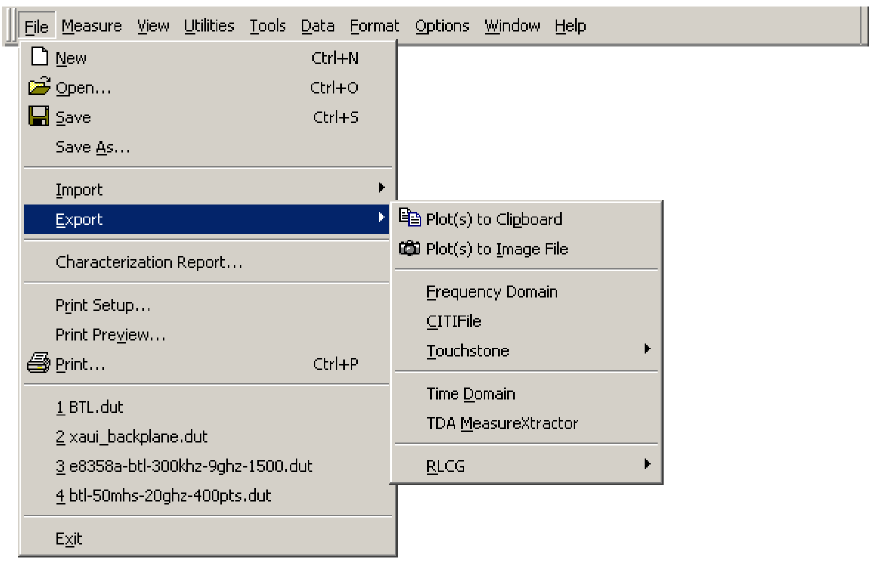
Port Configuration = 1 & 3 -> 2 & 4 is used to import single-ended measurement data that has taken with the system calibrated using the SOLT calibration.

Port Configuration = 1 & 2 -> 3 & 4 is used to import single-ended measurement data that has taken with the system calibrated using the TRL calibration.

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 11-4 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**.

Plots to Clipboard

Plot(s) to Clipboard exports the active plots window 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 plot images to Windows clipboard. The contents of the clipboard can then be pasted into other Windows programs.

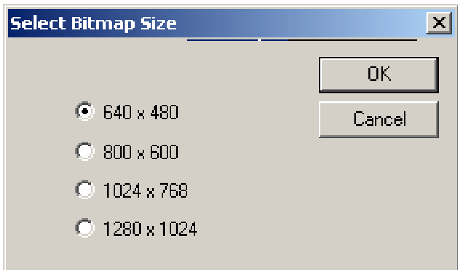
This feature is the same as the **Copy** icon on the **Toolbar**.

Plots 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 11-5.

Figure 11-5 Select Bitmap Size Dialog Box



When you export it, you may choose from Windows Bitmap (*.BMP), JPEG Bitmap (*.JPG), or Targa Bitmap (*.TGA) formats. Select **OK** to export the image.

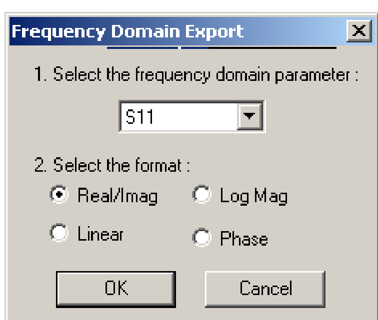
This feature is the same as the **Export Bitmap** icon (the camera icon) in the **Toolbar**.

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 using one of four frequency domain formats: Real/Imaginary, Log Magnitude, Linear Magnitude, or Phase.

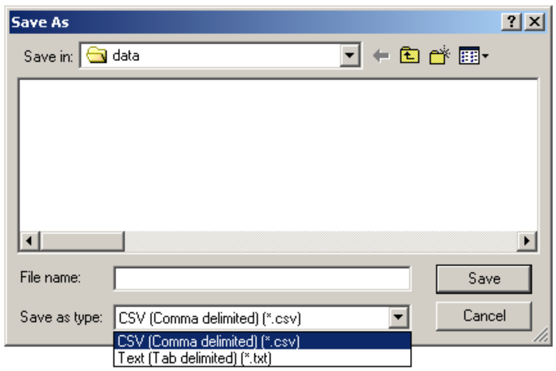
1. From the **File** menu, select **Export**, **Frequency Domain** to display the *Frequency Domain Export* dialog box.

Figure 11-6 Frequency Domain Export Dialog Box



2. In the **1. 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 **2. 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 11-7 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 to save the data to.

   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

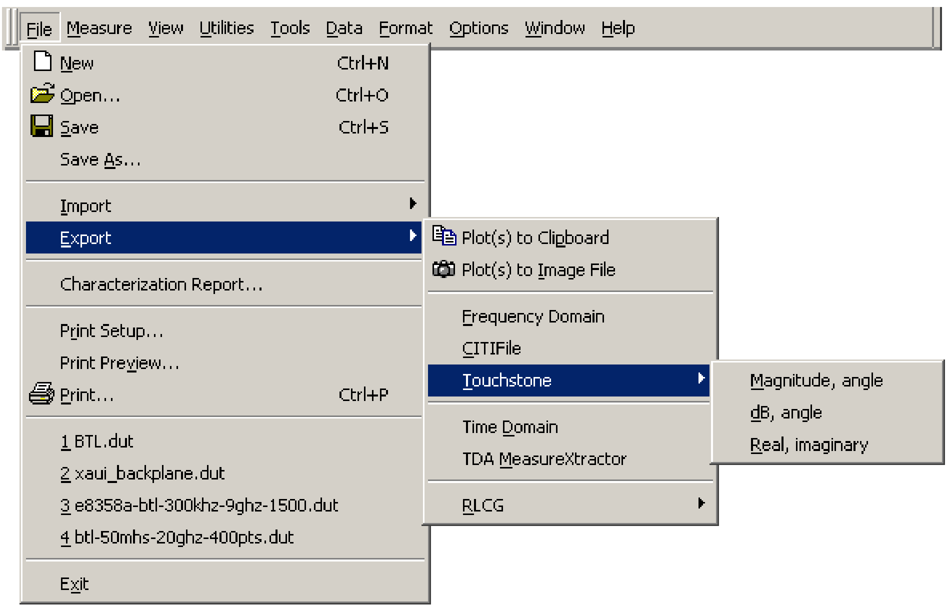
CITIFile exports the current data in CITIfile format (*.cit).

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**

Figure 11-8 File Menu with Export and Touchstone Expanded

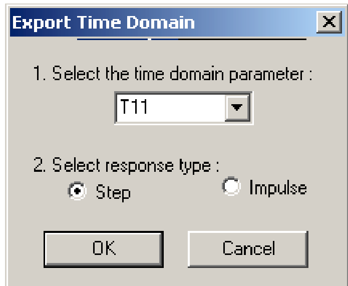


Time Domain

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 using either the Step or Impulse response type.

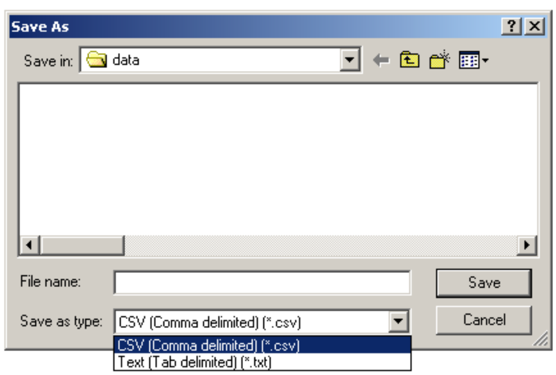
1. From the **File** menu, select **Export**, **Time Domain** to display the *Export Time Domain* dialog box.

Figure 11-9 Export Time Domain Dialog Box



2. In the **1. 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 **2. Select response type** area, select either: **Step** or **Impulse**
4. Select the **OK** button to open the *Save As* dialog box.

Figure 11-10 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 to save the data to.

   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 MeasureXtractor

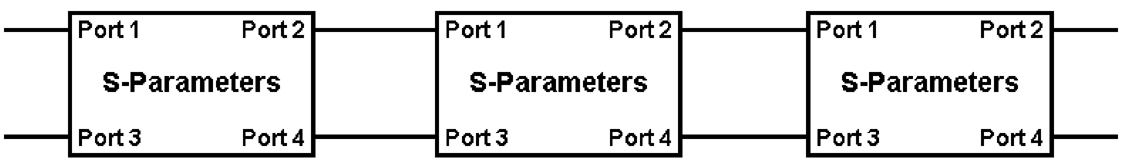
TDA MeasureXtractor directly export 4-port S-parameters in Touchstone format for import into TDA Systems' MeasureXtractor. This is a powerful capability that allows you to describe the exact frequency dependent behavior of your passive device using an S-parameter block inside of MeasureXtractor. MeasureXtractor also allows you to create a SPICE compatible model for further circuit simulation. You can 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 11-11). 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 IConnect MeasureXtractor for data driven models.

Figure 11-11 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 measurement starting at as low a frequency as possible. 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 will 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 earlier 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:

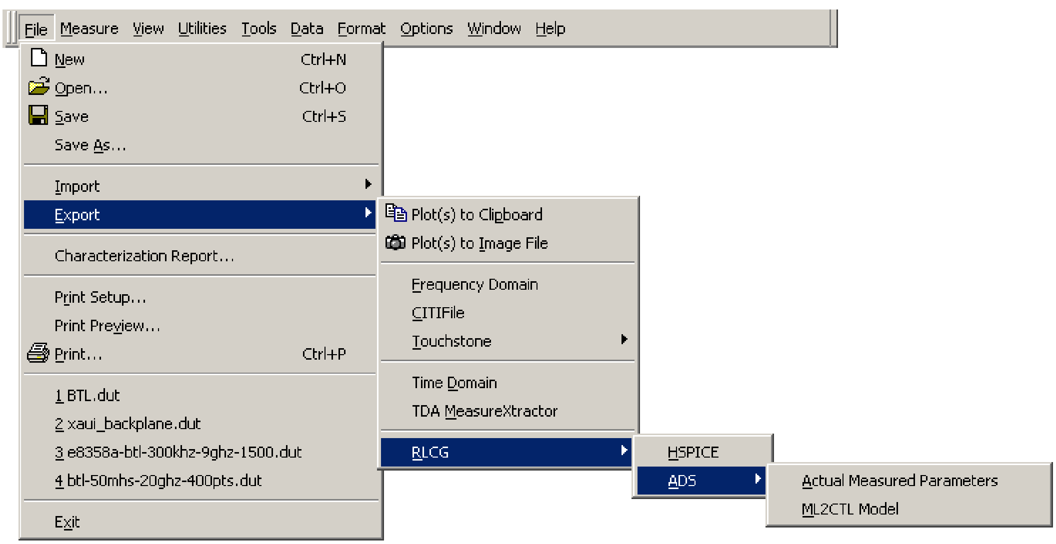
- Change the **Max Frequency** field under **Tools** > **Options** > **Waveform Viewer** to match the maximum frequency of your Touchstone file.
- Follow the guidelines listed in the TDA Systems' documentation for rise time, truncating, and dc values.

RLCG

RLCG exports the current data in one of two RLCG formats:

- **HSPICE** 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 325.
- **ADS** exports the RLCG data in a format compatible with the Agilent Advanced Design System (ADS) software format which also has the two following data format choices in which the data may be saved:
    - ❑ **Actual Measured Parameters** exports extracted parameters versus frequency in an ASCII file for ADS. Refer to "Exporting Transmission Line Data" on page 325.
    - ❑ **ML2CTL** exports fitted parameters in an ASCII file to be used with the ADS MultiLayer 2 Coupled Transmission Lines (ML2CTL) model. Refer to "Exporting Transmission Line Data" on page 325.

Figure 11-12 File Menu with Export, RLCG, and ADS Expanded

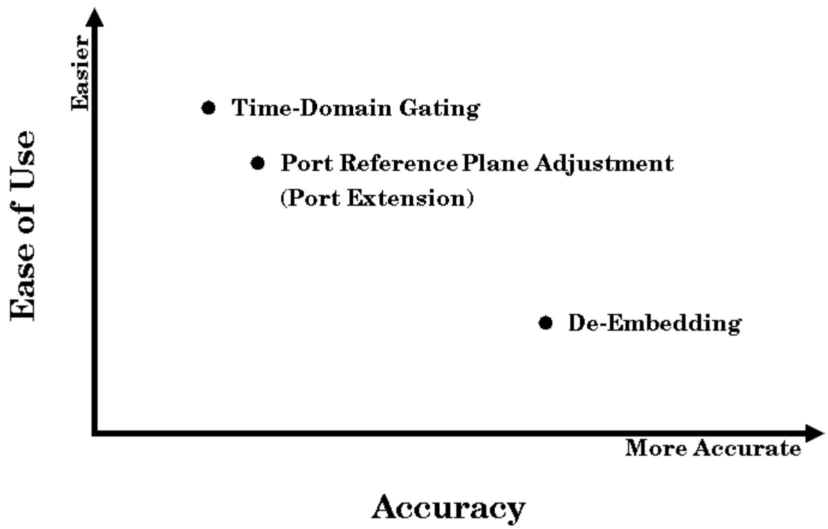


12 Removing Unwanted Effects from the Measurement

To reduce your measurement, analysis, and redesign cycles, the physical layer test system offers three techniques to help you define/redefine your device under test (DUT).

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 12-1 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 Reference Plane Adjustment** (or port reference plane rotation) 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 constant impedance. Port reference plane adjustments are usually 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** 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. Deembedding 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. An 'Ideal' gate replaces the gated section with an ideal transmission line having the same electrical delay as the section that it replaced.

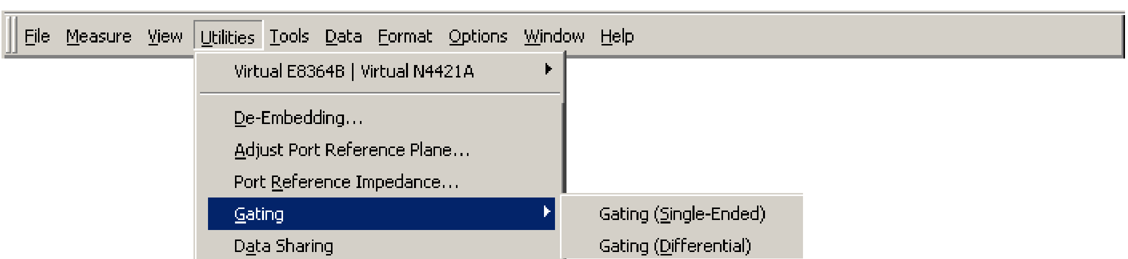
Up to 10 gates can be added. The gates are numbered sequentially from the left edge of the plot to the right edge. For example, the first gate (Gate 1) was added. Then a second gate is added to the left of the Gate 1. The gates are then renumbered such that the gate on the left (the second gate that 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 12-2 Utilities Menu with Gating Expanded



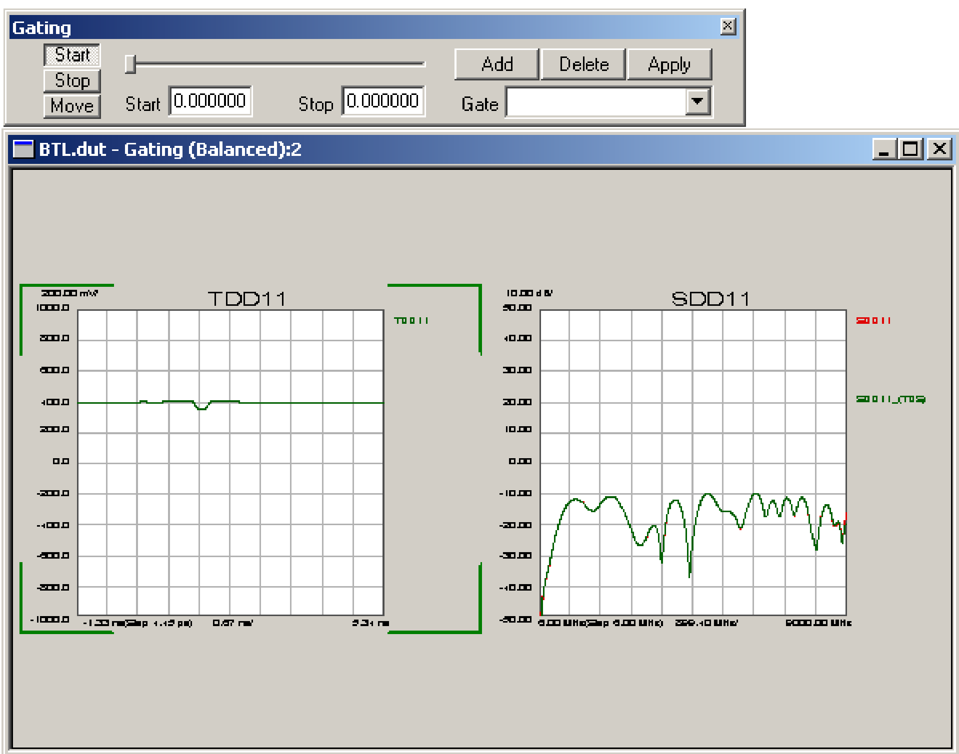
The **Gating Bar** (shown in Figure 12-3) and an empty plot window are displayed when gating is selected from the **Utilities** menu.

Figure 12-3 Gating Bar

2. From the **Parameter Bar** (or from the **Data** menu), select the parameter that will have gates added to it.

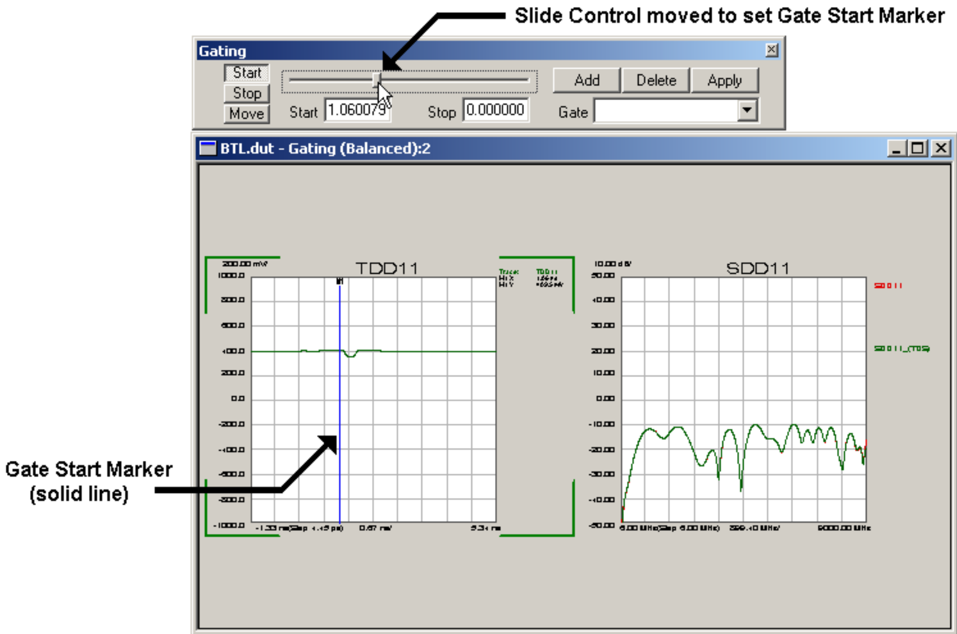
   When you select the parameter, two plots are displayed in the plots window. The left plot is the parameter you selected displayed in Time Domain mode. The right plot is that same parameter displayed in Frequency Domain Mode. See Figure 12-4. When adding a gate to the time domain trace, you will also be able to see how adding the gate affects the parameter in the Frequency Domain by watching the right plot.

Figure 12-4 Gating Display: Time Domain Plot and Frequency Domain Plot



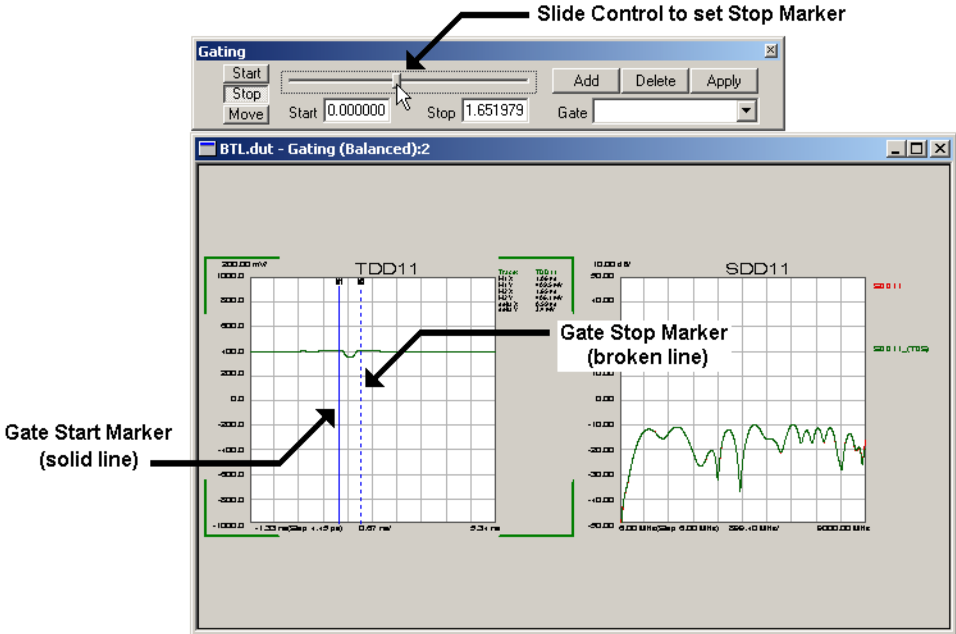
3. Click the **Start** button.
4. While watching the gate start marker, a solid blue vertical line on your time domain plot, move the slide control to the right to set the start position of the gate. As the slide control is positioned, the time (or distance) of the start gating marker is displayed in the **Start** box. See Figure 12-5.

Figure 12-5 Setting the Gating Start Position



5. Click **Stop** button.
6. While watching the gate stop marker, a broken 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 12-6.

Figure 12-6 Setting the Gating Stop Position

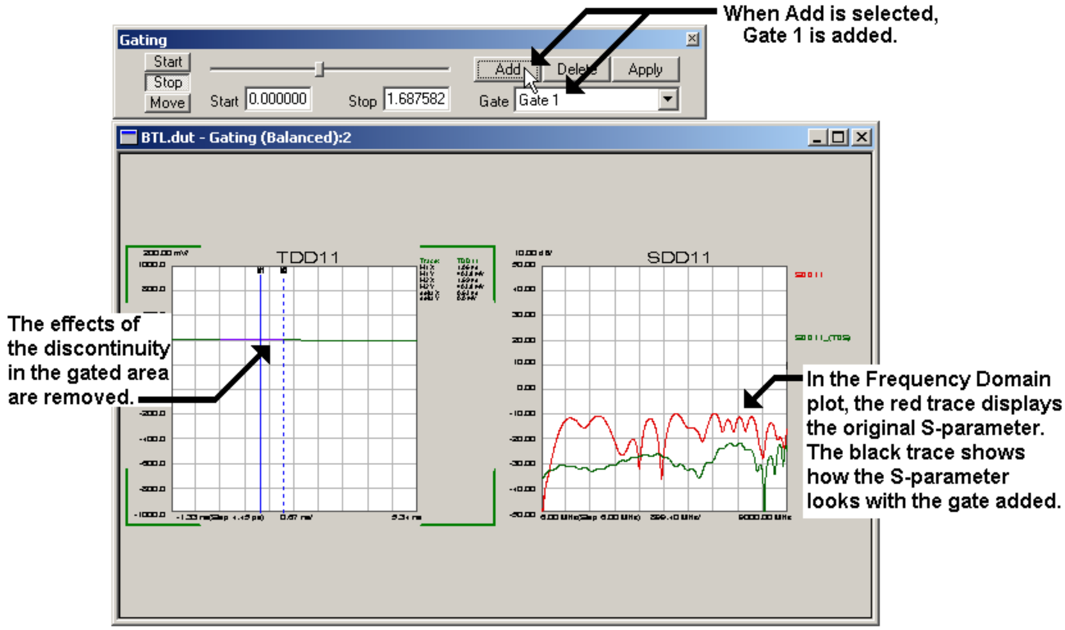


7. Click the **Add** button to add the gate.

   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 12-7.
   - 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 12-7 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

A gate can be deleted by selecting the gate from the **Gate** list and clicking the **Delete** button. 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.

Port Reference Plane Adjustment

Adding adapters, fixtures, and probes introduces additional loss, phase shift, and mismatch that can add error to the measurement of your DUT. The port reference plane adjustment feature 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 reference plane adjustment does not correct for additional loss and mismatch introduced by these items.

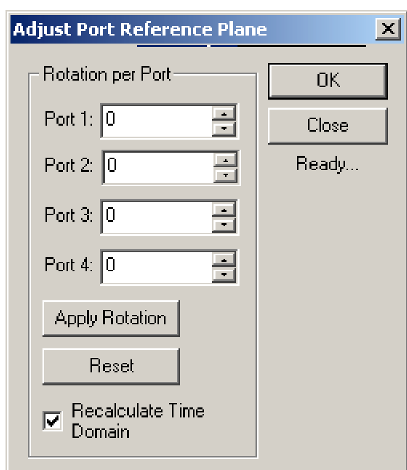
Port reference plane adjustment can be adjusted using either of the two methods described in this section. Refer to To Adjust Port Reference Plane located below or "To Rotate the Reference Plane Using the De-Embedding Dialog Box" on page 358.

To Adjust Port Reference Plane

The easiest way to adjust the port reference plane is by using the following method:

1. Select **Adjust Port Reference Plane** from the **Utilities** menu to open the *Adjust Port Reference Plane* dialog box where you can change the reference plane rotation settings and recalculate the displayed data.

Figure 12-8 Adjust Port Reference Plane Dialog Box



2. Enter the new reference plane values for the desired ports by using the arrows or entering the values in the port box directly.

   A positive value rotates the phase towards the DUT (effectively removing a length of 50Ω

line) and a negative number rotates the phase away from the DUT (effectively adding a length of 50Ω line).

3. Click **Apply Rotation** to recalculate the data and display the new plots.

   Note that when reference plane rotation is applied, the **De-Embedding** indicator color changes to a bright color. See Figure 12-10 on page 359.

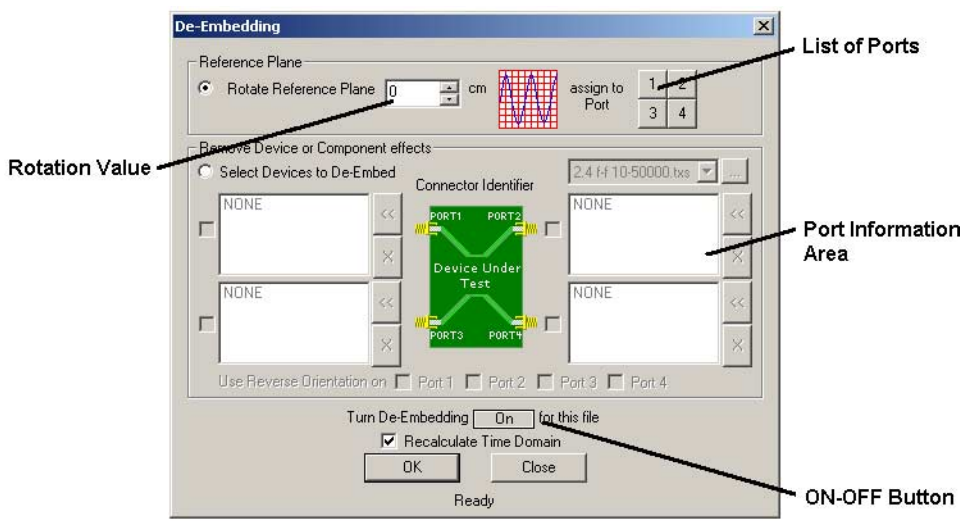
   Reset returns the values to zero, recalculates the data, and displays original plots.

To Rotate the Reference Plane Using the De-Embedding Dialog Box

1. Select **De-Embedding** from the **Utilities** menu to open the *De-Embedding* dialog box.

   Use the *De-Embedding* dialog box to configure DUT reference planes by de-embedding probes, fixtures, or adapters from the measured data.

Figure 12-9 De-Embedding Dialog Box



2. In the **Reference Plane** area, click **Rotate Reference Plane**.

3. Enter a reference plane rotation value (in centimeters-in-air).

   You can use the spinner or click in the **cm** box and enter a value directly.

   A positive value rotates the phase towards the DUT (effectively removing a length of 50Ω line) and a negative number rotates the phase away from the DUT (effectively adding a length of 50Ω line).

4. Select the port to assign the rotation value to a port by selecting from the port numbers.

   As you select a port, that rotation value is shown below in the information area for that port.

5. Repeat steps 3 and 4 until you have assigned rotation values to all appropriate ports.

6. Turn the port reference plane adjustment on by clicking the **ON/OFF** button (shown in Figure 12-9) until the label reads: **De-embedding is currently ON for this file**

7. Click **OK** to accept the configuration and exit the dialog box.

   Cancel ignores any changes and exits the dialog box.

The **De-Embedding** indicator on status bar means that De-embedding is being applied. See Figure 12-10.

Figure 12-10 De-Embedding Indicator on the Status Bar

De-Embedding

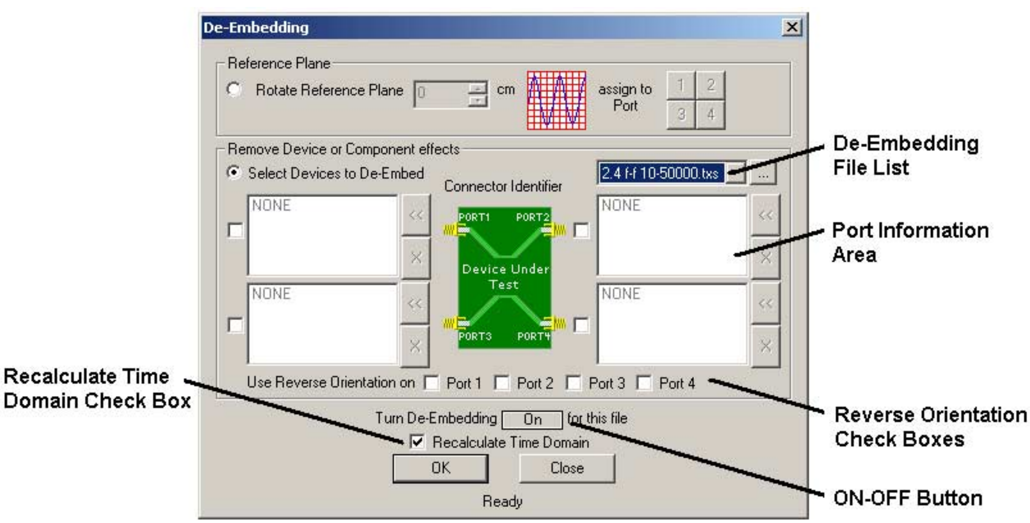
Adapters, test fixtures, or other non-DUT structures introduce unwanted effects and error into measurement results. These unwanted effects typically include additional loss, phase shift, mismatch, discontinuities, and time delay. De-embedding is the process of mathematically removing previously characterized 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.

After an S-parameter file of the non-DUT structure is created in CitiFile format, it is user-modified and saved as an adapter characterization file (with a ".txs" filename extension). See "Converting a CitiFile to a PLTS Adapter File" on page 600. At that point, it is ready to be used by following the instructions below.

1. Select **De-Embedding** from the **Utilities** menu to open the *De-Embedding* dialog box.

Figure 12-11 De-Embedding Dialog Box



2. Click **Select Device to De-Embed**.
3. Select a de-embedding file (.txs) that you want to use from the de-embedding file list.
4. Select the check box associated with the port to which the file applies.
5. Click the associated **<<** button to add the file to the port. (Clicking the ✕ button will remove the file from the port.)
6. Repeat steps 3 through 5 until you have assigned de-embedding files to all appropriate ports.
7. Select all ports that use reverse adapter orientation.

   Reverse orientation is noted by check marks for each port specified in the **Use Reverse Orientation** check boxes. Reverse orientation refers to when port 1 of the adapter is connected to the DUT. Refer to "Characterizing Adapters" on page 176 for further information and instructions on performing adapter characterization.
8. Turn the de-embedding on by clicking the **ON/OFF** button (shown in Figure 12-11) until the label reads: **De-embedding is currently ON for this file**
9. Click **OK** to accept the configuration and exit the dialog box.

   Cancel ignores any changes and exits the dialog box.

The **De-Embedding** indicator on status bar means that de-embedding is being applied. See Figure 12-12.

Figure 12-12 De-Embedding Indicator on the Status Bar

13 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 373.

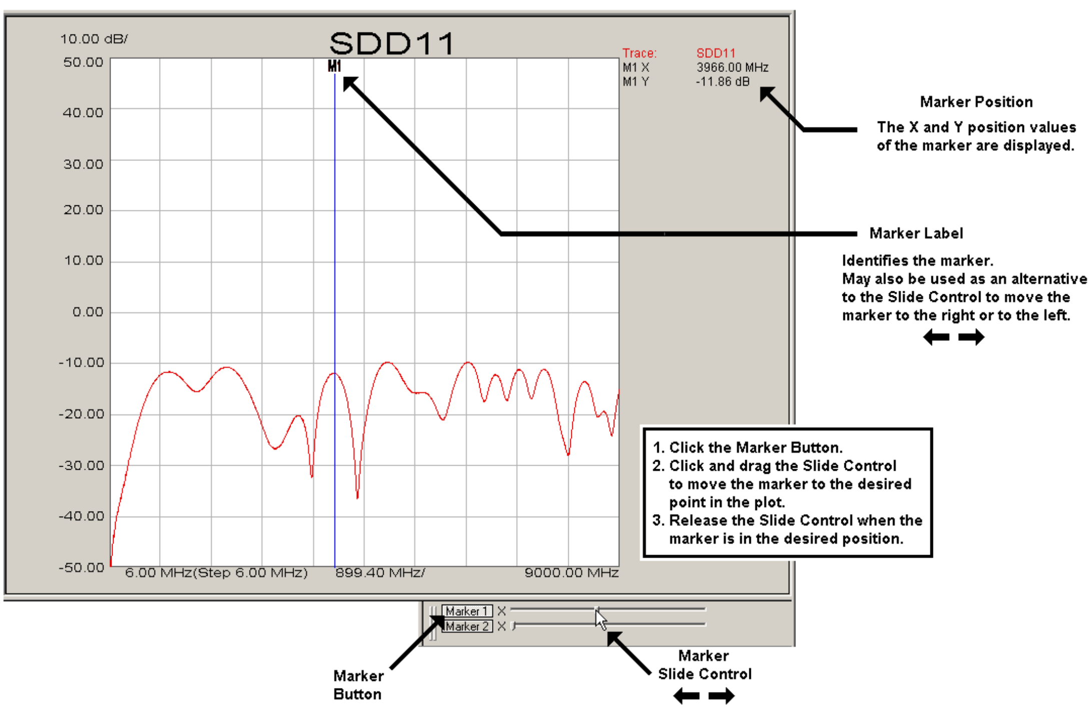
Time-Domain, Frequency-Domain, and RLCG Markers

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 set on each plot.

To use the markers:

1. With a plot displayed, click the **Marker 1** button.

Figure 13-1 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 blue 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.
4. Note that 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.

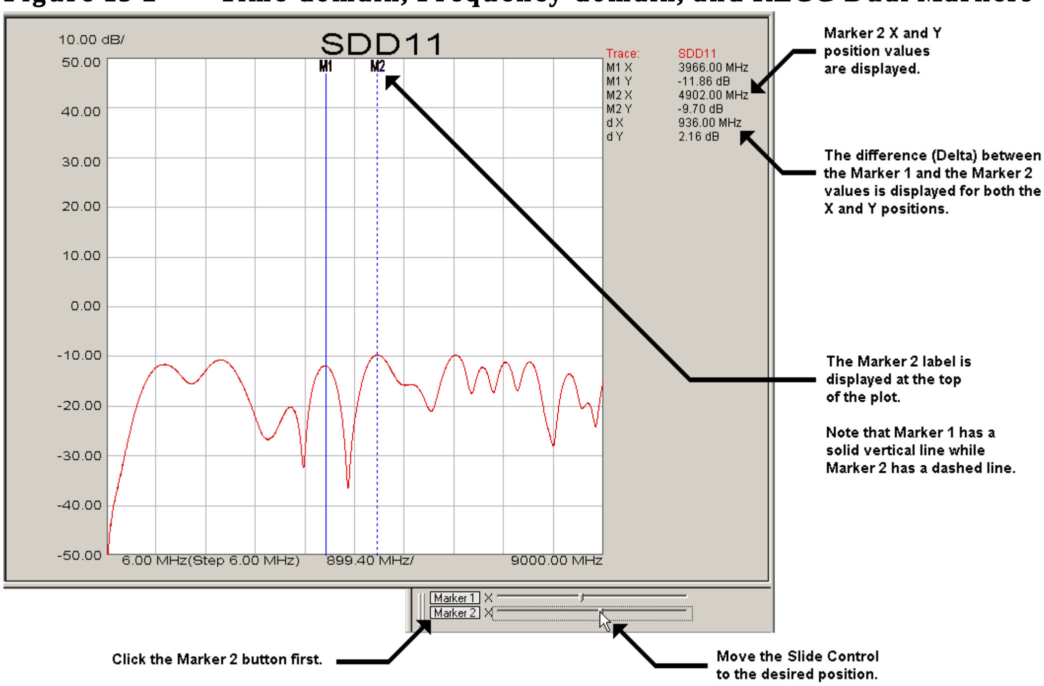
TIP **Moving the Marker One Measurement Point at a Time**

The marker may be moved one measurement point at a time by pressing the keyboards arrows (← & →). Each keyboard press, changes the X and Y values.

---

5. To display the second marker, click the **Marker 2** button.

Figure 13-2 Time-domain, Frequency-domain, and RLCG Dual Markers



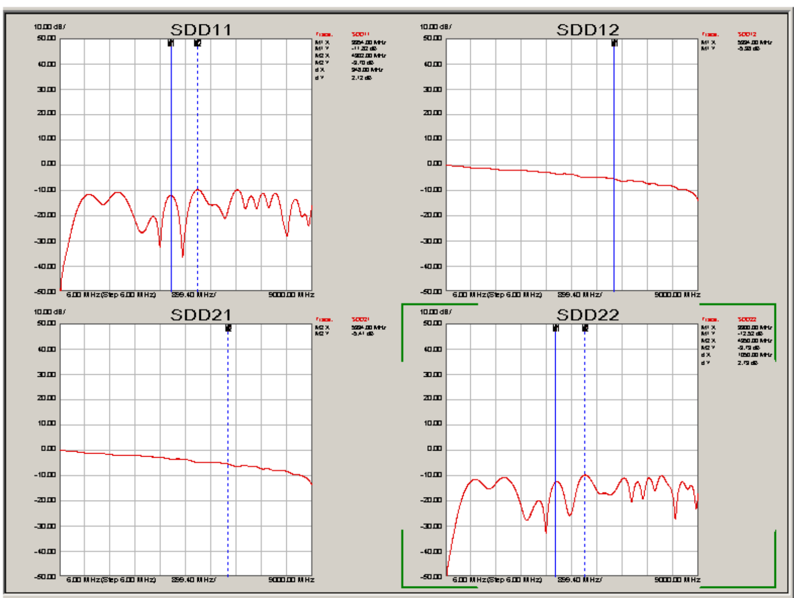
6. Click and drag the **Marker 2 Slide Control** to the right, releasing it at the desired position of the plot.

   As the **Slide Control** is moved, the marker is displayed. **Marker 2** is a broken blue vertical line with the label **M2** at the top.

7. Note that the X and Y position values for Marker 2 are displayed near the upper right corner of the plot. When both markers as used, the difference (delta) between the position of Marker 1 and Marker 2 are also displayed immediately below the Marker 2 information.

8. Note that markers may be used with multiple plots. As your needs arise, you may have up to two markers in each plot. The marker information is displayed at the upper right corner of each plot.

Figure 13-3 Markers with Multiple Plots



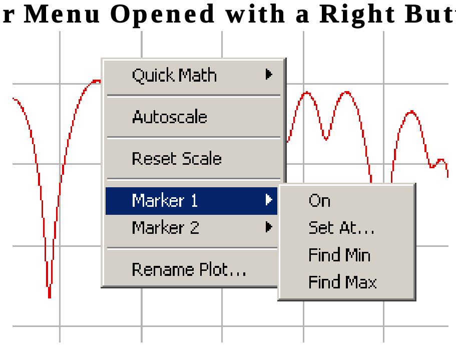
9. Also note that markers may be used with multiple traces. You may use either one or two markers in a plot that displays as many as four traces. The marker information is for each trace is displayed along the right edge of the plot.

Figure 13-4 Markers with Multiple Traces

Quick Markers

By clicking the right mouse button with the cursor over the plot, you can access the markers (either **Marker 1** or **Marker 2**) quickly. See Figure 13-5. Both **Marker 1** and **Marker 2** have four choices available: **On**, **Set At**..., **Find Min**, and **Find Max**.

Figure 13-5 Marker Menu Opened with a Right Button Mouse Click



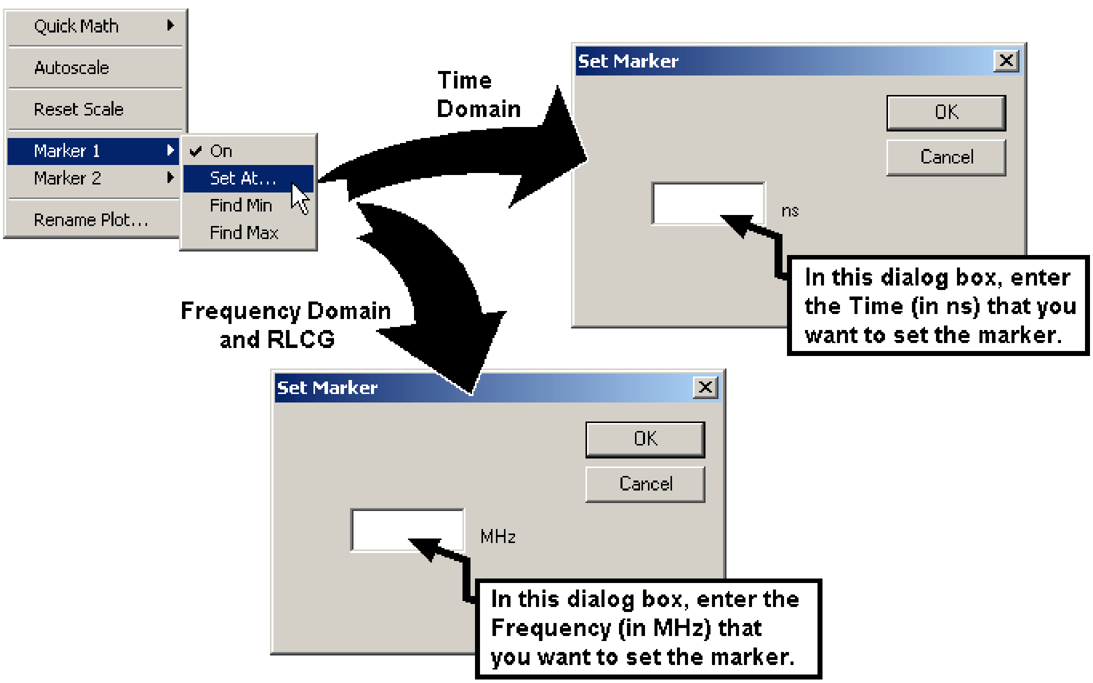
On Turns the associated marker on or off. When the marker is turned on, a check mark is displayed to the left side of the **On** selection. The marker, the vertical blue line is displayed with a label at the top of the plot.

- Marker 1 is a solid blue line with the **M1** label at the top of the plot.
- Marker 2 is a broken blue 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 13-6.

Figure 13-6 Set Marker Dialog Boxes



Find Min Places the associated marker at the point of the minimum displayed value. If multiple traces are displayed, this selection finds the lowest value of all of the traces.

Find Max Places the associated marker at the point of the maximum displayed value. If multiple traces are displayed, this selection finds the highest value of all of the traces.

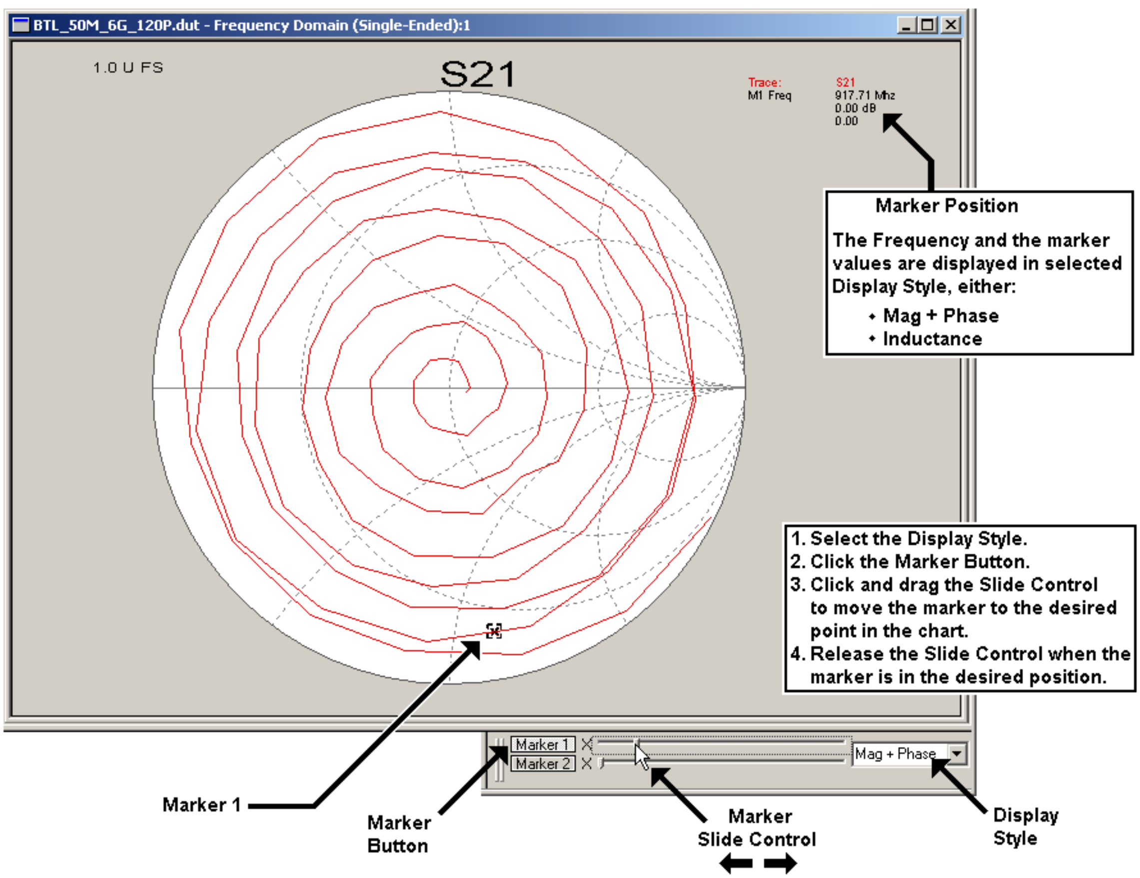
Frequency Domain Polar and Smith Chart Markers

Frequency domain plots that are displayed in either Polar or Smith Chart formats use markers that are displayed differently than the markers shown in "Frequency Domain Polar and Smith Chart Markers" on page 370. When a marker is used in a Smith or Polar Chart format, it is essentially a point that follows the displayed trace around the chart. Up to two markers may be set on each plot.

To use the markers:

1. With the plot displayed on the Polar or Smith Chart, select the display style on the Marker Bar. You may select either **Mag + Phase** or **Inductance**. See Figure 13-7.

Figure 13-7 Single Marker for Frequency Domain Polar and Smith Charts

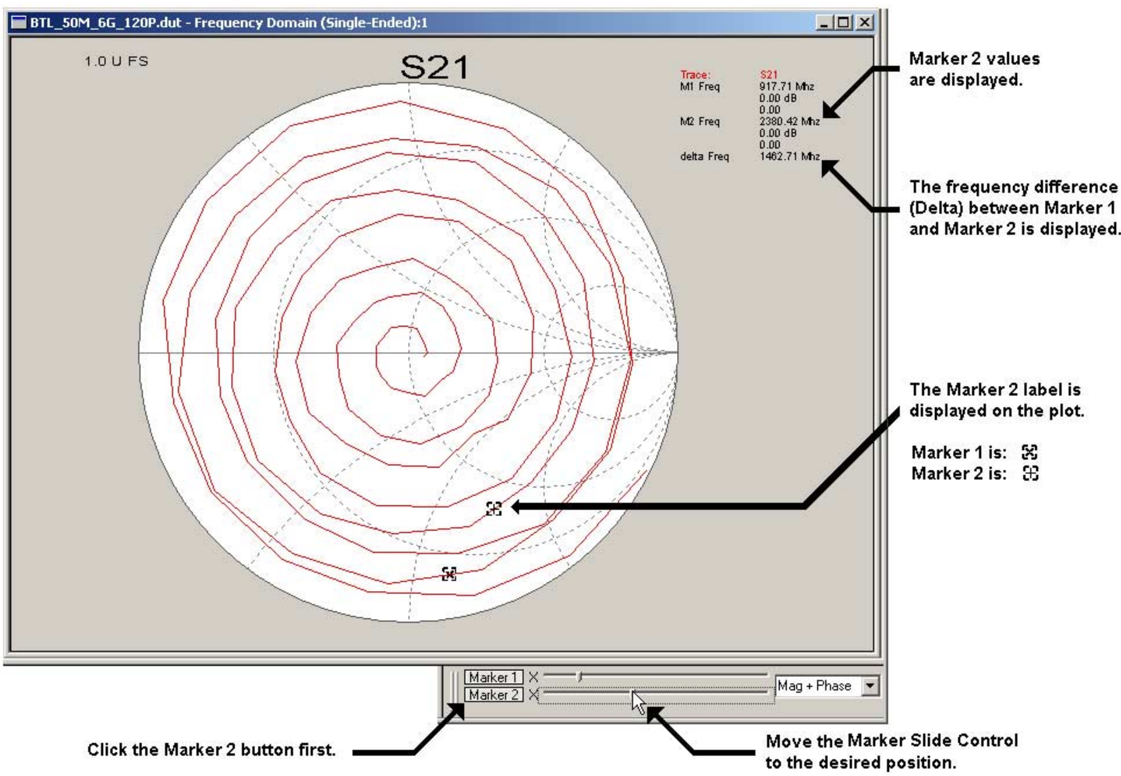


The **Mag + Phase** style displays the magnitude and the phase at the marker frequency. **Inductance** displays the magnitude and inductance at the marker frequency.

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.
5. Note that the values for the markers position are displayed near the upper right corner of the plot.
6. To display the second marker, click the **Marker 2** button. See Figure 13-8.

Figure 13-8 Polar and Smith Chart Dual Markers



7. Click and drag the **Marker 2 Slide Control** to the right, releasing it at the desired position of the plot.

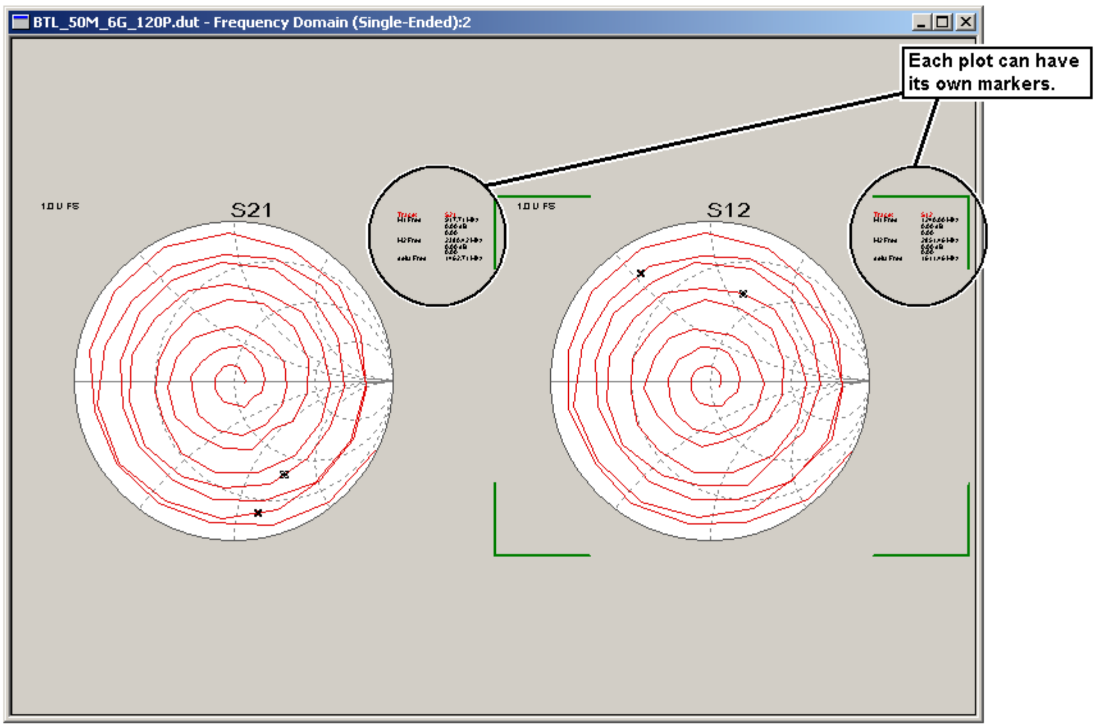
   As the **Slide Control** is moved, the marker is displayed. See Figure 13-8 and note the

difference between the two marker symbols.

8. Note that the values for Marker 2 are displayed near the upper right corner of the plot. When both markers as used, the frequency difference (delta) between Marker 1 and Marker 2 is also displayed immediately below the Marker 2 information.

9. Note that markers may be used with multiple plots. As your needs arise, you may have up to two markers in each plot. The marker information is displayed at the upper right corner of each plot. See Figure 13-9.

Figure 13-9 Markers with Multiple Polar and Smith Chart Plots



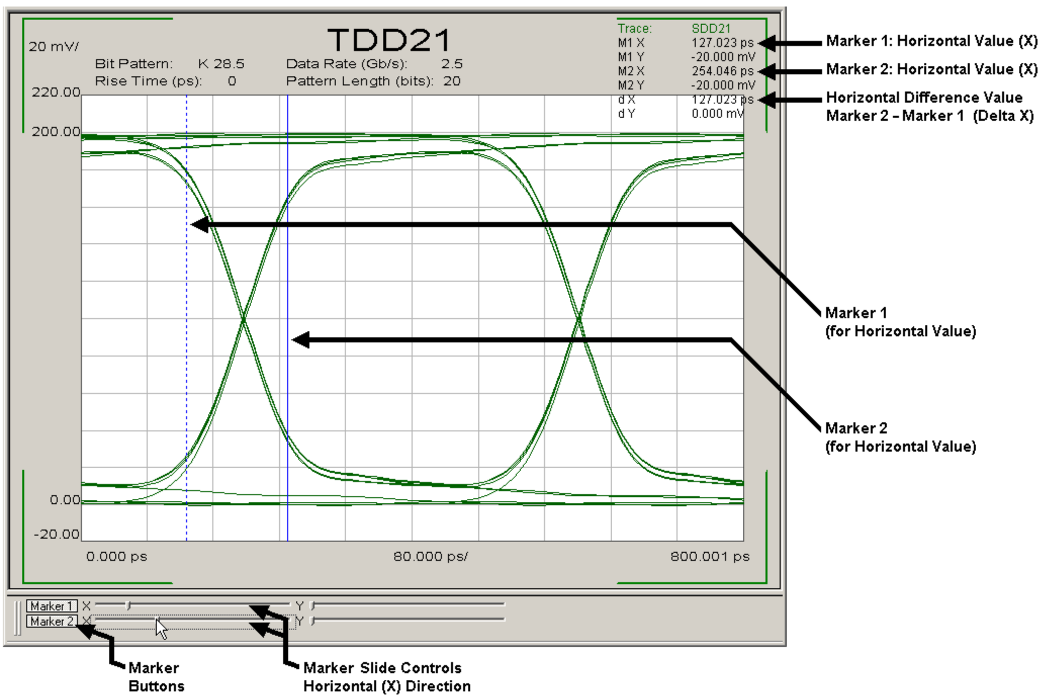
Eye Diagram Markers

Eye diagram analysis uses markers that are displayed both vertically and horizontally on a plot. As the vertical marker intersects the eye, the value of the horizontal (X) position is displayed. As the horizontal marker intersects the eye, the value of the vertical (Y) position is displayed. Up to two vertical and two horizontal markers may be set on each plot.

To use the markers:

1. With a plot displayed, click the **Marker 1** button.

Figure 13-10 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 to the right, a broken blue vertical line (the marker) is displayed.

3. Release the **X Slide Control** at the desired position of the plot.

4. Note that the value of the X position of the marker is displayed near the upper right corner of the plot.

5. To display the second marker, click the **Marker 2** button and repeat steps 2 through 4 for the second marker.

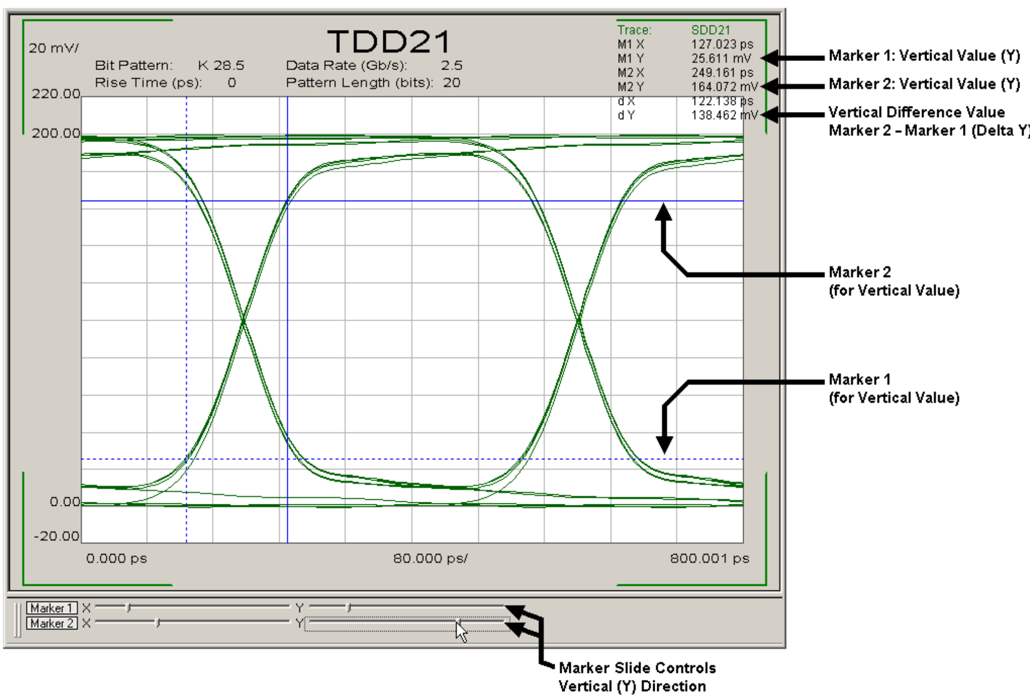
   Marker 2, a solid blue vertical line, is displayed.

6. Also note that with the two markers displayed, the difference (delta) between the two markers is displayed.

7. To display the vertical (Y) position, click the **Marker 1** button and drag the **Marker 1 Y Slide Control** to the right, releasing the slide control at the desired position on the plot. See Figure 13-11.

   As the **Y Slide Control** is moved to the right, a broken blue horizontal line (the marker) is displayed.

Figure 13-11 Eye Diagram Markers: Vertical (Y) Position



8. Repeat step 7 for Marker 2.

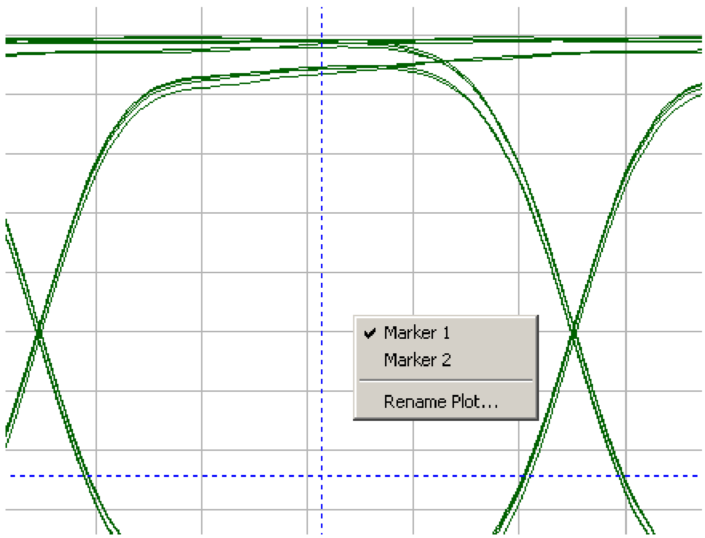
   Marker 2 is a solid horizontal line.

9. Note that 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 13-12. These selections turn both the Horizontal and Vertical markers on or off.

Figure 13-12 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

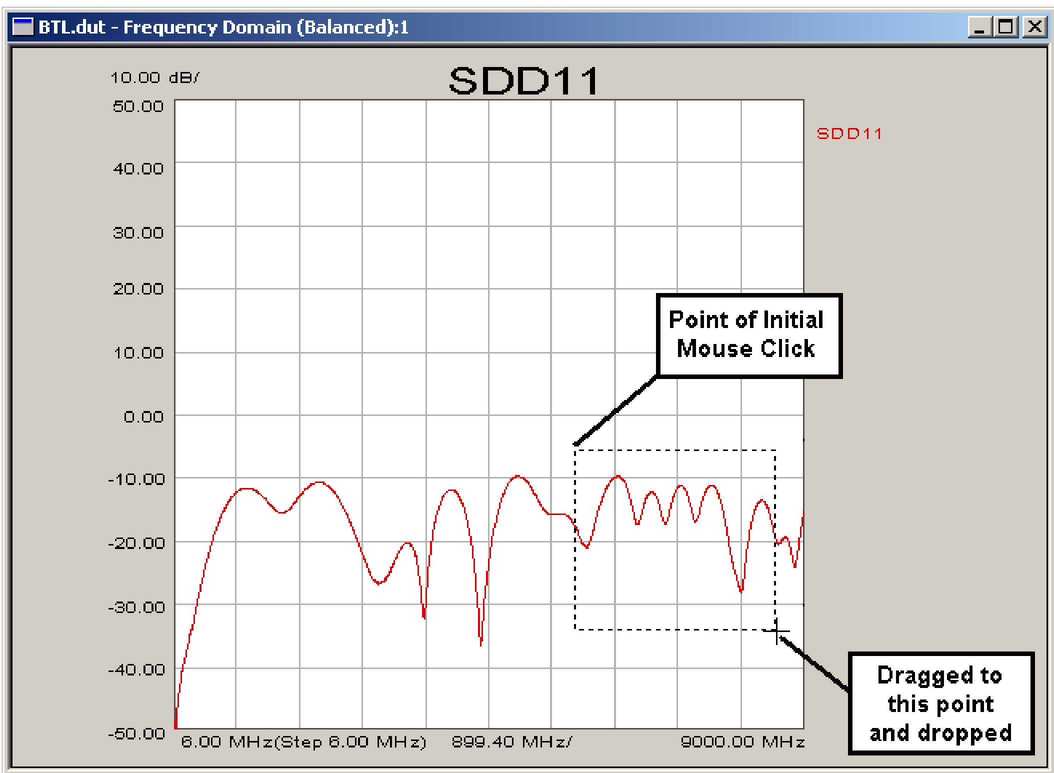
While you are looking at a plot and you want to zoom in on a particular area of interest, you can zoom using by clicking and dragging across the area of interest within the plot.

NOTE The zoom feature is not available for eye diagram plots.

To zoom in on an area:

1. With a plot displayed, click within the plot and drag the mouse, creating a rectangle as the mouse is moved. See Figure 13-13.

Figure 13-13 Defining Zoom Area

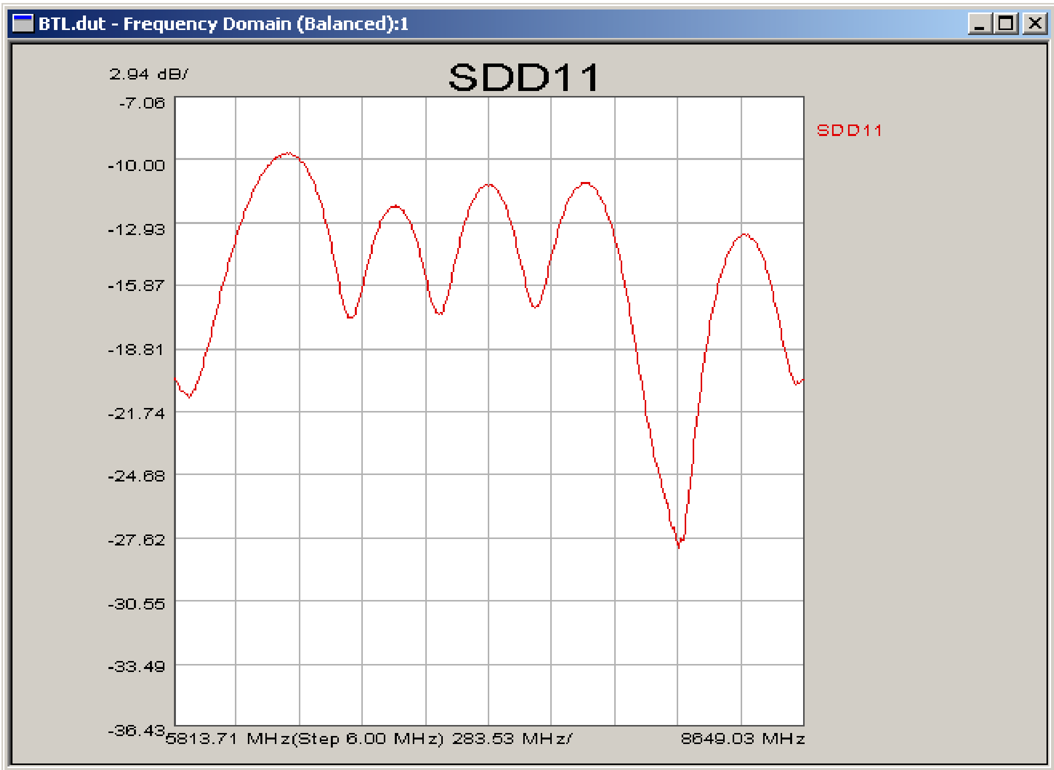


The rectangle is created by dashed lines from the point of the mouse click to the position

that the mouse has been dragged.

2. Once the rectangle encloses the area of the plot that you want to zoom in to, release the mouse button.

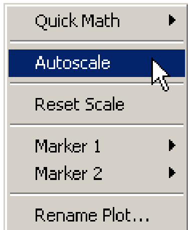
Figure 13-14 New Plot after Zoom



The rectangular area of the plot that you have defined with the mouse is now displayed, replacing the original plot. Note the new X-axis start and stop frequencies and the new Y-axis scale.

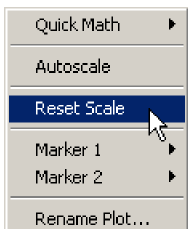
3. If the new plot does not center the traces in the Y-axis, clicking the right mouse button with the cursor over the plot and selecting **Autoscale** sets the Y-axis scale so that the data is better displayed.

Figure 13-15 Autoscale Selection



4. When you are have finished examining the new plot, you can click the right mouse button with the cursor over the plot and select **Reset Scale** to return the plot to the default horizontal and vertical settings.

Figure 13-16 Auto Zoom Selection



Math

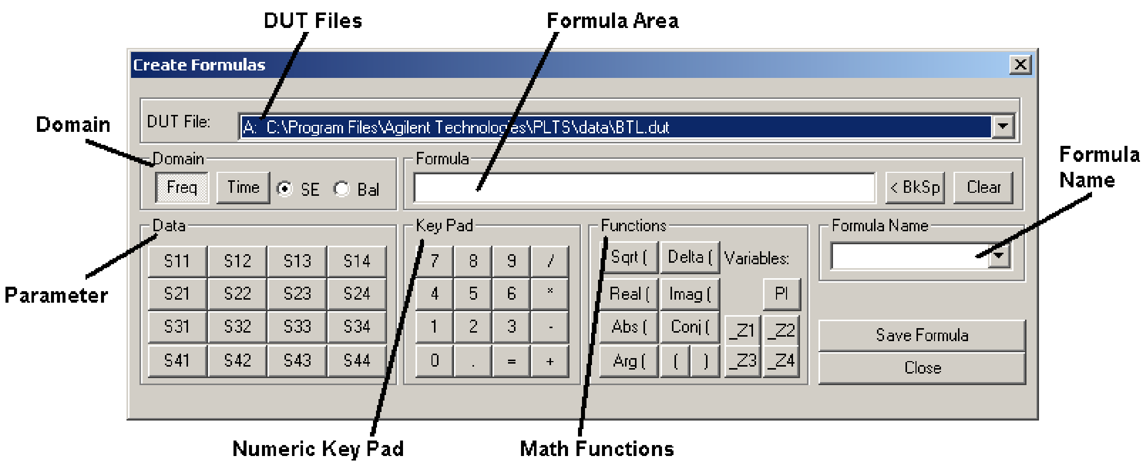
Math may be performed on traces that are displayed within a plot. For example, you may divide the data of one trace by the data of a second trace for a comparison of gain or attenuation between the two traces. Another commonly-used math function is subtracting the data of one trace from the data of a second tract for a measure of directivity.

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 379) or you may perform a quick math function on a displayed trace (see “Using Quick Math” on page 385).

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 13-17 shows the *Create Formulas* dialog box that is used to create and save math formulas. This dialog box is divided into several areas.

Figure 13-17 Create Formulas Dialog Box



DUT Files Used to identify 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 files may be used in a formula.

Domain Used to identify 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.

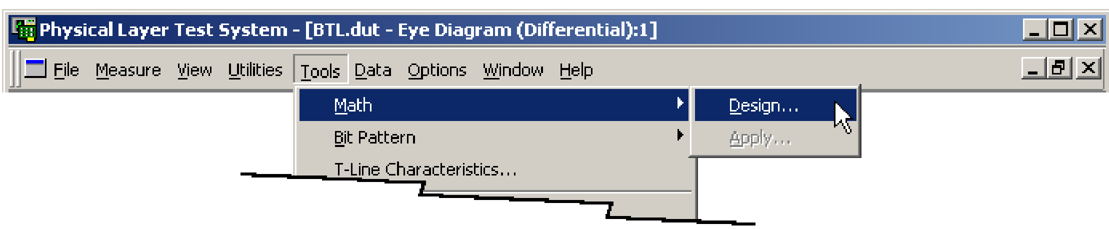
Functions Used to build the formula. It allows you to enter math symbols and delimiters as well as some frequency domain formats.

Formula Name allows you to enter a name to save the formula and recall it for future use.

To create a math formula:

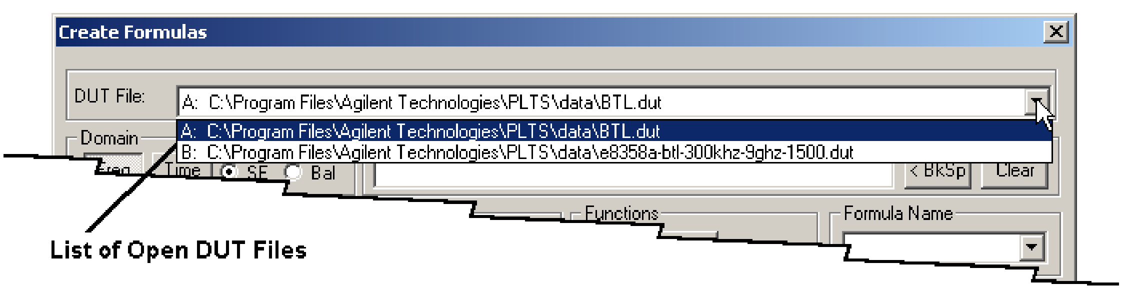
1. Select **Math**, **Design**... from the **Tools** menu to open the *Create Formulas* dialog box as shown in Figure 13-18.

Figure 13-18 Opening the Create Formulas Dialog Box



2. In the DUT File list, select the file that the first **Data** area parameter belongs.

Figure 13-19 DUT File Selection List



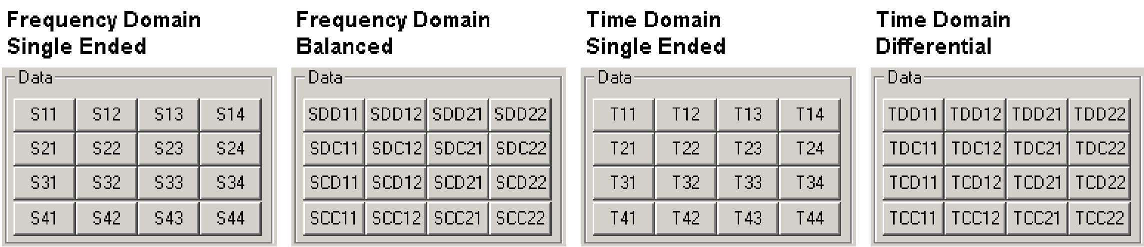
3. In the **Domain** area, select the domain and the mode of the first **Data** area parameter.

   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.

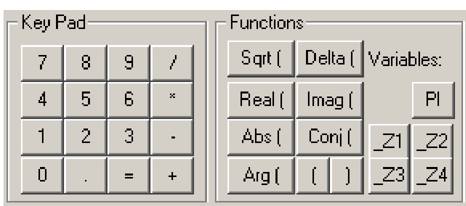
   Each **Domain** area combination displays a different set of **Data** area parameters. Refer to Figure 13-20 for the **Data** area selections for each of the four combinations.

Figure 13-20 Data Area Parameters for Each Domain Area Combination



4. Start building your formula using the appropriate **Data** area parameters shown in Figure 13-20 and the **Key Pad** and **Functions** area buttons shown in Figure 13-21.

Figure 13-21 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 (/).

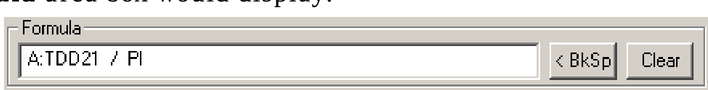
The **Functions** area buttons are:

Table 13-1 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

5. As you make selections from the **Data**, the **Key Pad**, and the **Functions** areas, these are displayed in the **Formula** area.

   Just as an example, if you to divide TDD21 by π,
   you would select these buttons: **TDD21**, **/**, **PI**
   and the **Formula** area box would display:



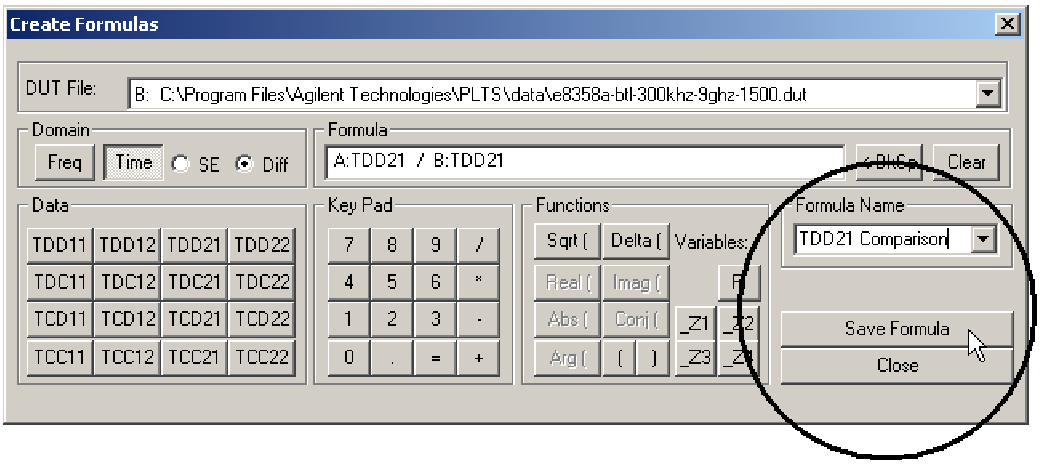
   Use the **< BkSp** button to back step the cursor one position at a time.
   Use the **Clear** button to remove the entire entry from the **Formula** area box.

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 13-22.

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 13-22 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.

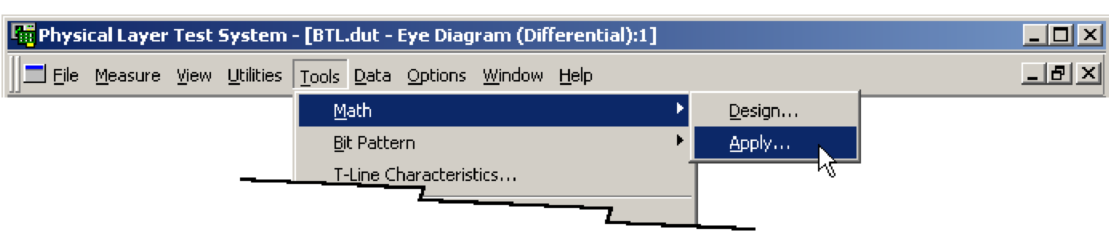
Applying a Math Formula

Once math formulas have been created and saved, they may be applied to opened data files.

To create a math formula:

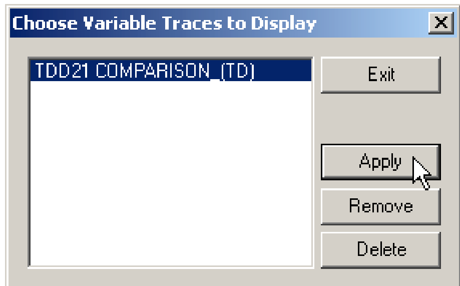
1. Select **Math**, **Apply**... from the **Tools** menu to open the *Choose Variable Traces to Display* dialog box as shown in Figure 13-23.

Figure 13-23 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 13-24.

Figure 13-24 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 to apply the formula to the opened data files.

   A plot is displayed showing the data file plot after the formula has been applied.

4. When you have finished reviewing the plot, click the **Remove** button to remove the data file plot after the formula has been applied.

5. If you want to delete a formula from the list of formulas, select the formula and click the **Delete** button.

   The formula is deleted and is no longer available.

6. Click **Exit** to return to the main PLTS window.

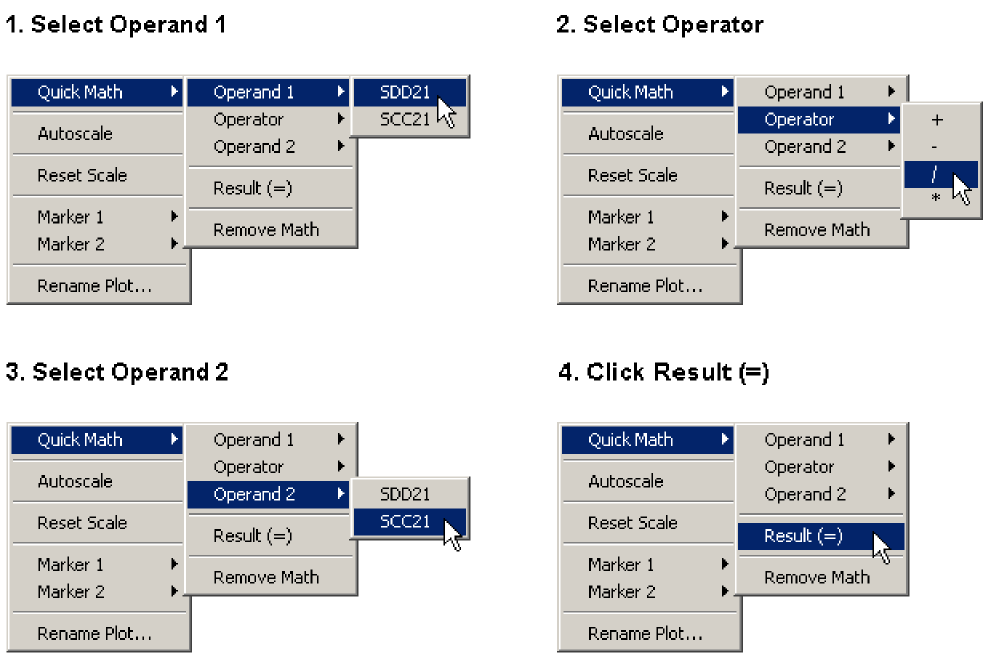
Using Quick Math

You can use Quick Math to perform simple math operations on two traces in the same plot. A plot allows up to four traces to be displayed. As quick math is performed, the resultant plots are displayed on the plot. Since two data traces are required on the plot, you may perform one or two quick math functions on these data traces so that either three or four traces are displayed on the plot. If three traces are displayed on the plot, only one quick math function may be performed.

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 13-25.

Figure 13-25 Four Steps for Quick Math



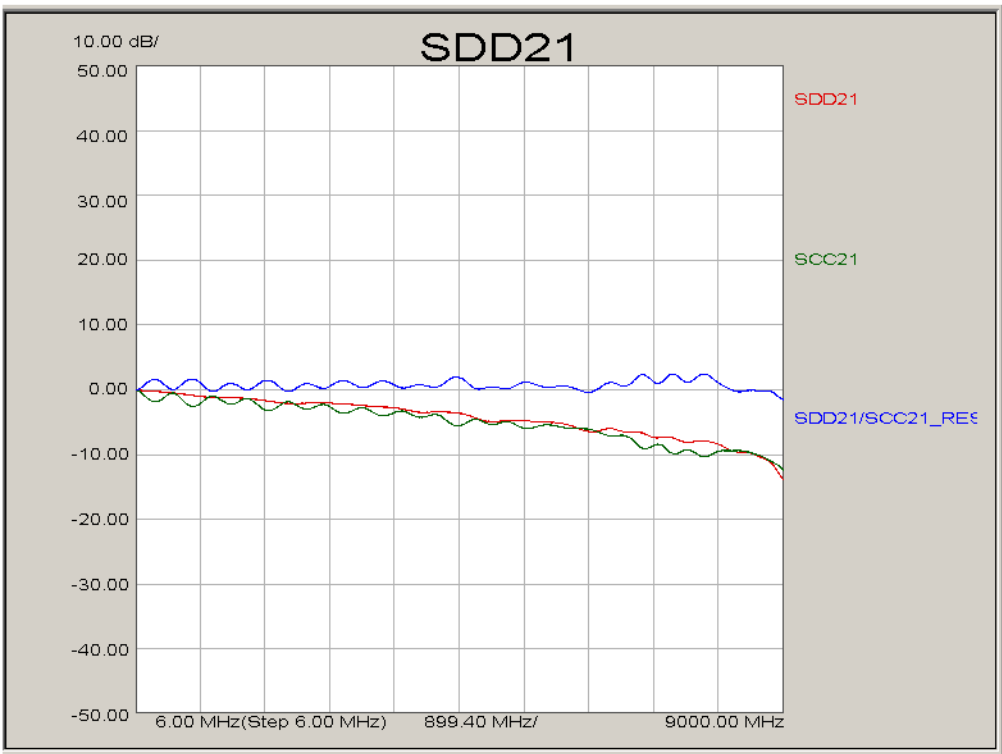
2. Click the right mouse button with the cursor over the plot. Then click **Quick Math** and **Operator**. Click the desired math operation.

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 13-26 Two Traces with Quick Math Result Displayed (SDD21/SCC21)



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 multiple traces from different files on the same plot. You may place traces from a maximum of three files 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.

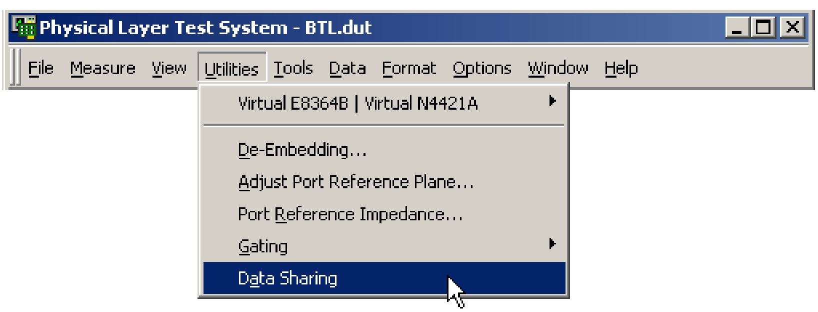
The data sharing feature is only available for Time Domain and Frequency Domain parameters only. You may only use data sharing between file of the same type. For example, if you have a “Differential Time Domain” parameter, you may only share with other files in the “Differential Time Domain” format.

The measurement parameters of the shared files do not need be the same. For example, in Frequency domain, the start and stop frequencies of the two files do *not* need to be same.

To use the data sharing feature, with the two files opened:

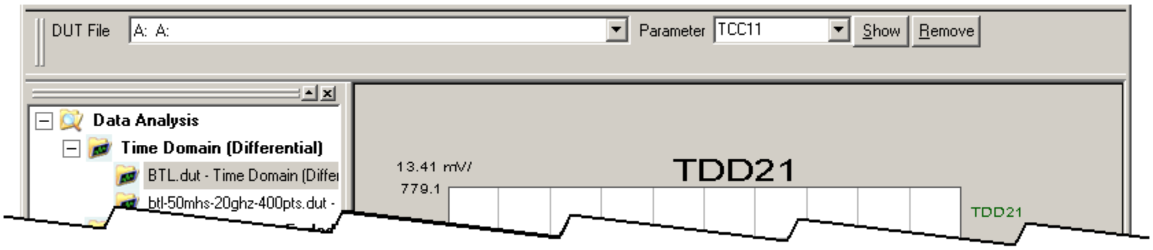
1. Display the parameter that you are interested in comparing against a parameter of another file so that it displayed individually for best viewing by double-clicking the plot if it is in a view with multiple plots.
2. From **Utilities** menu, select **Data Sharing** to open the data sharing bar.

Figure 13-27 Opening the Data Sharing Bar



3. From the **DUT File** list shown in Figure 13-28, select the data file what you want to share with the plot used in step 1.

Figure 13-28 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 if you want to data share another plot.
7. When you have finished with the data sharing plot, click the **Remove** button to remove all of the shared traces from the original trace.

Characterization Report Generator

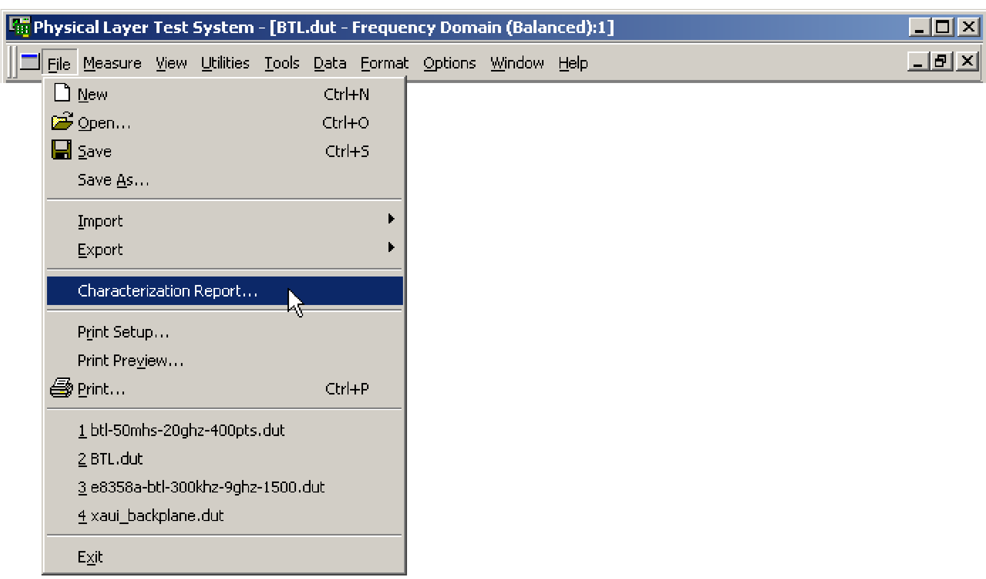
The Characterization Report is a standard report that can be generated to show:

- 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:

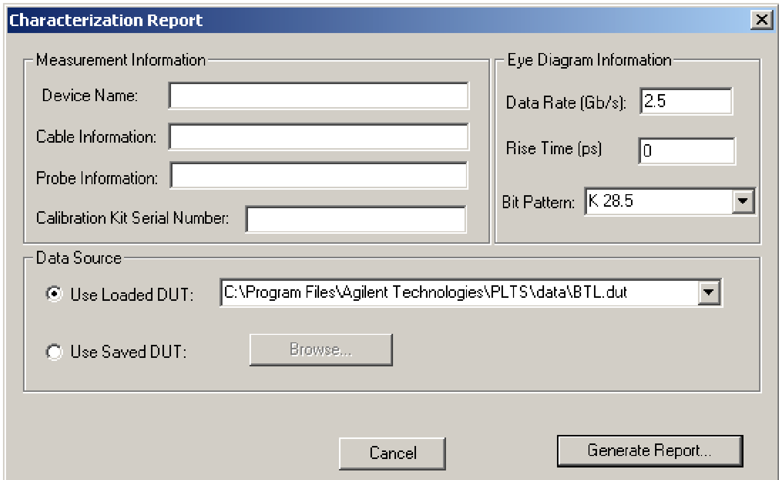
1. Select **Characterization Report**... from the **File** menu to open the *Characterization Report* dialog box. See Figure 13-29.

Figure 13-29 Open Characterization Report



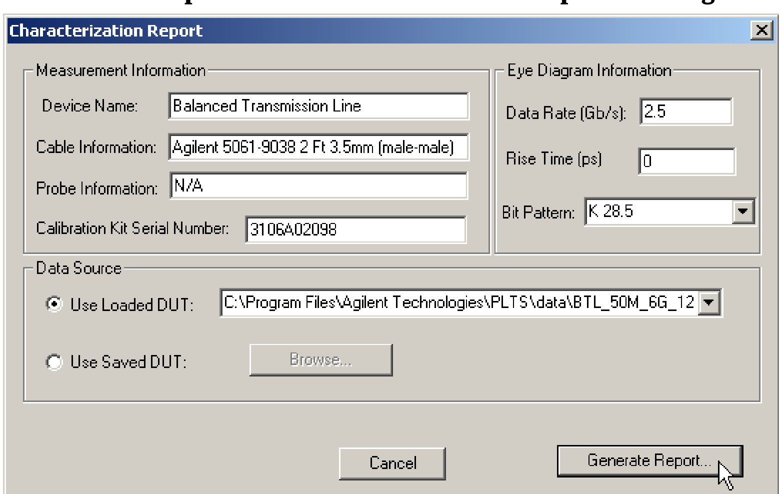
The blank *Characterization Report* dialog box is then displayed as shown in Figure 13-30.

Figure 13-30 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.

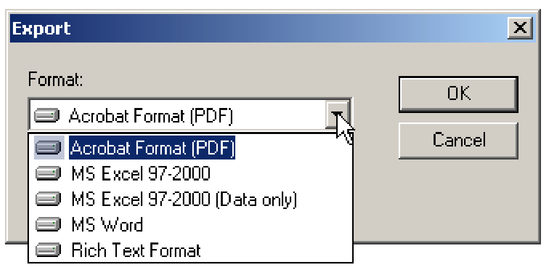
Figure 13-31 Completed Characterization Report Dialog Box



- 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.

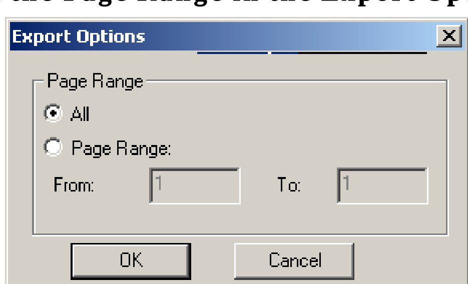
3. In the *Export* dialog box (see Figure 13-32), select the format in which to save the characterization report from the list. Then click the **OK** button.

Figure 13-32 Select the Report Format in the Export Dialog Box



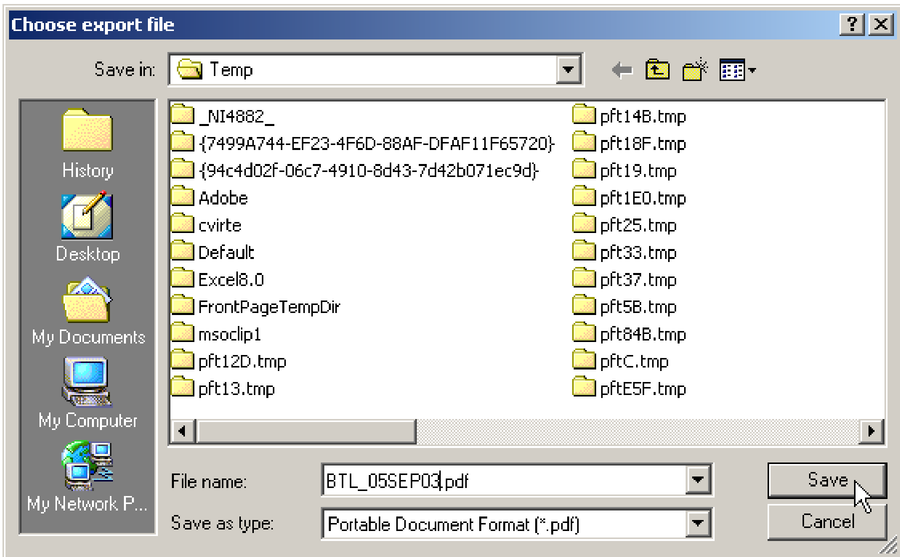
4. In the *Export Options* dialog box (see Figure 13-33), select the page range choice. **All** saves all of the characterization report pages. **Page Range** allows you to enter the range of pages to save. Then click the **OK** button.

Figure 13-33 Select the Page Range in the Export Options Dialog Box



5. In the *Choose export file* dialog box (see Figure 13-34), 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 13-34 Choose Export File Dialog Box

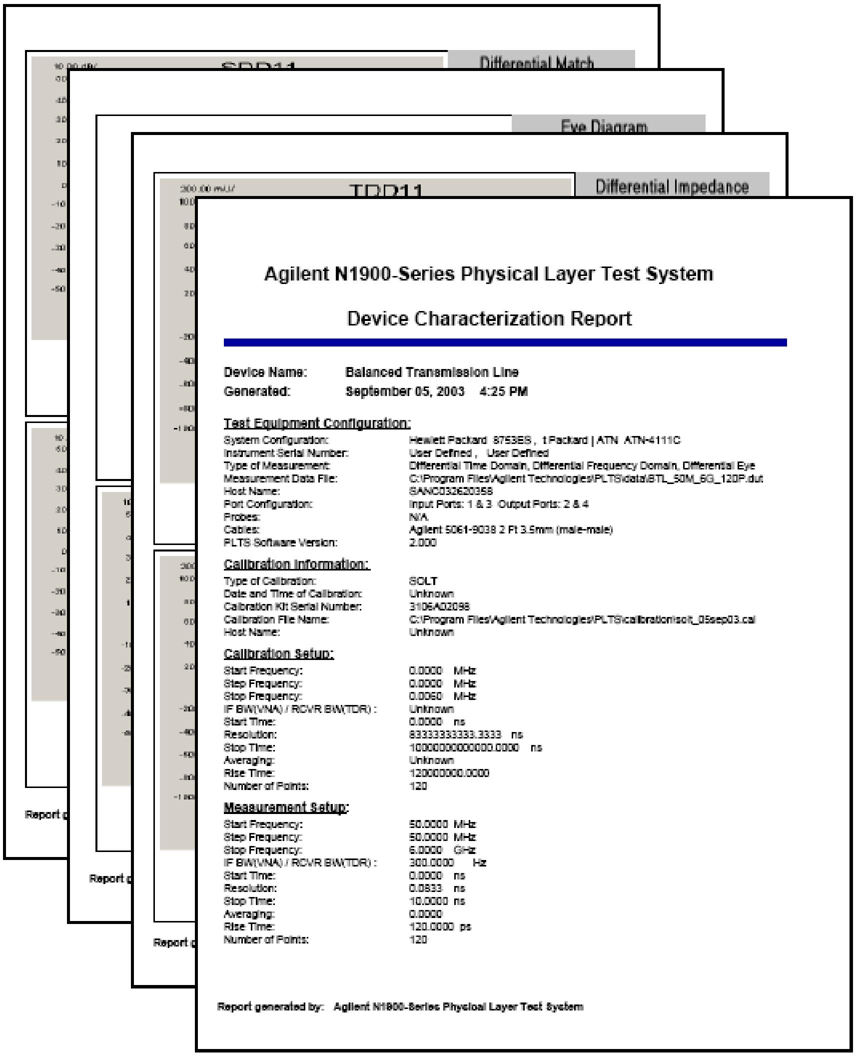


6. Go to the directory to which you saved your characterization report and open the file to review. This is a four-page report as shown in Figure 13-35.

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 13-35 Device Characterization Report



Agilent N1900-Series Physical Layer Test System

Device Characterization Report

Device Name:	**Balanced Transmission Line**
Generated:	**September 05, 2003 4:25 PM**

Test Equipment Configuration:

System Configuration:	Hewlett Packard 8753ES , t Packard | ATN ATN-4111C
Instrument Serial Number:	User Defined , User Defined
Type of Measurement:	Differential Time Domain, Differential Frequency Domain, Differential Eye
Measurement Data File:	C:\Program Files\Agilent Technologies\PLTS\data\BTL_50M_6G_120P.dut
Host Name:	SANC032620358
Port Configuration:	Input Ports: 1 & 3 Output Ports: 2 & 4
Probes:	N/A
Cables:	Agilent 5061-9038 2 Ft 3.5mm (male-male)
PLTS Software Version:	2.000

Calibration Information:

Type of Calibration:	SOLT
Date and Time of Calibration:	Unknown
Calibration Kit Serial Number:	3106A02098
Calibration File Name:	C:\Program Files\Agilent Technologies\PLTS\calibration\solt_05sep03.cal
Host Name:	Unknown

Calibration Setup:

Start Frequency:	0.0000 MHz
Step Frequency:	0.0000 MHz
Stop Frequency:	0.0060 MHz
IF BW(VNA) / RCVR BW(TDR) :	Unknown
Start Time:	0.0000 ns
Resolution:	83333333333.3333 ns
Stop Time:	10000000000000.0000 ns
Averaging:	Unknown
Rise Time:	120000000.0000
Number of Points:	120

Measurement Setup:

Start Frequency:	50.0000 MHz
Step Frequency:	50.0000 MHz
Stop Frequency:	6.0000 GHz
IF BW(VNA) / RCVR BW(TDR) :	300.0000 Hz
Start Time:	0.0000 ns
Resolution:	0.0833 ns
Stop Time:	10.0000 ns
Averaging:	0.0000
Rise Time:	120.0000 ps
Number of Points:	120

Report generated by: Agilent N1900-Series Physical Layer Test System

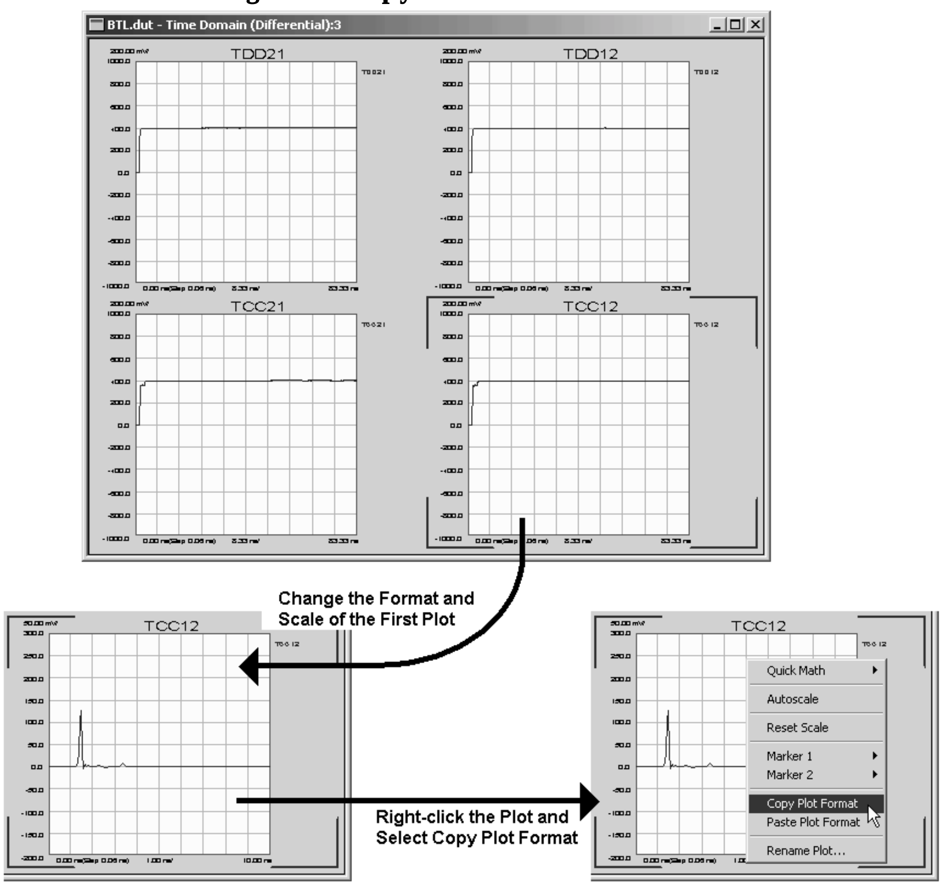
Copying and Pasting Plot Formats

The plots aren't always displayed in the format or scale that you would like to see them. Now, you can change the plots quickly using the quick format function. This is very useful when you are displaying several plots within the plot window.

To perform a quick plot format function:

1. With your data plots displayed, select the first plot to change. Make any changes in the format or scale that enables you to display the plot as you would like it displayed. See Figure 13-36. In this example, the format was changed from step to impulse and the vertical scale was changed.

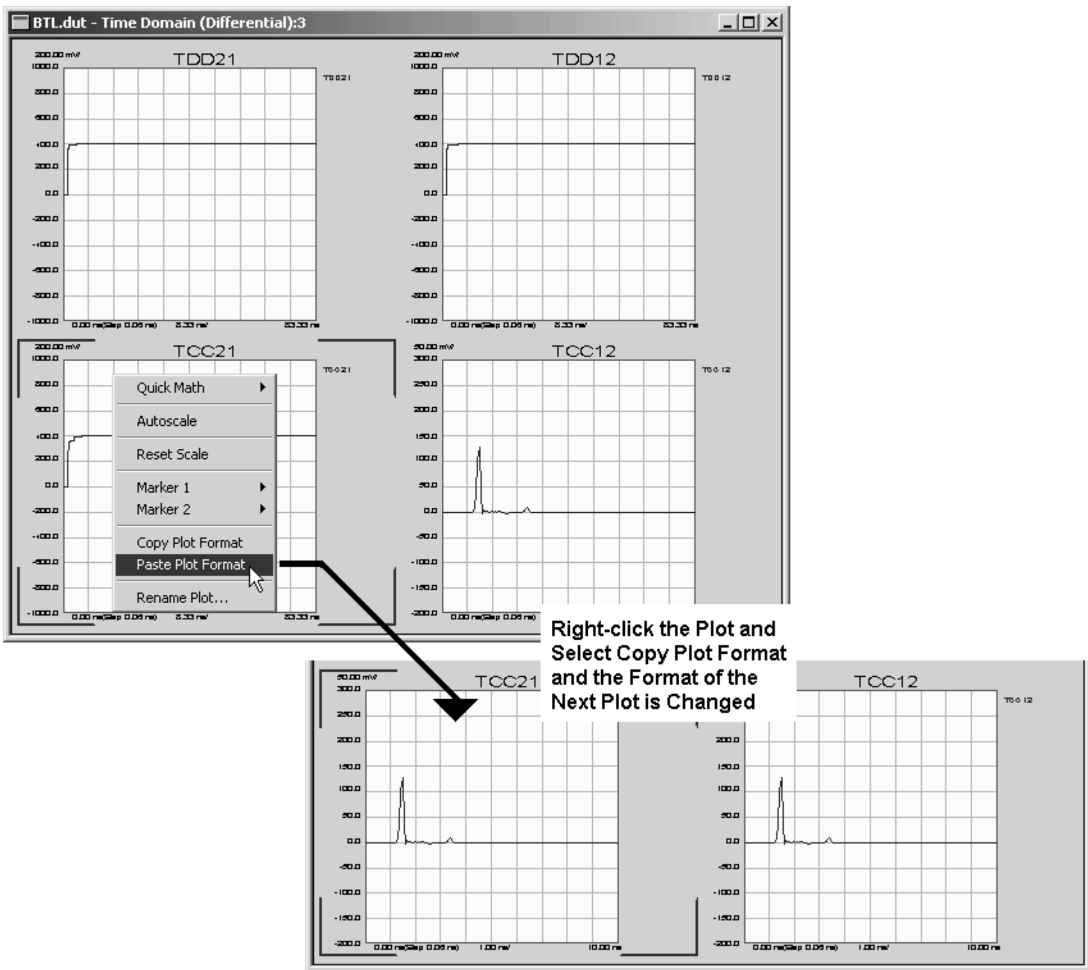
Figure 13-36 Change and Copy the Plot Format and Scale



2. Click the right mouse button with the cursor over the plot. Then click **Copy Plot Format**.
3. Select the next plot to be displayed with these settings by clicking the plot. See Figure 13-37.
4. Click the right mouse button with the cursor over the plot. Then click **Paste Plot Format**.

   The plot is now displayed with the new format and scale settings.

Figure 13-37 Select a Plot and Paste the Format



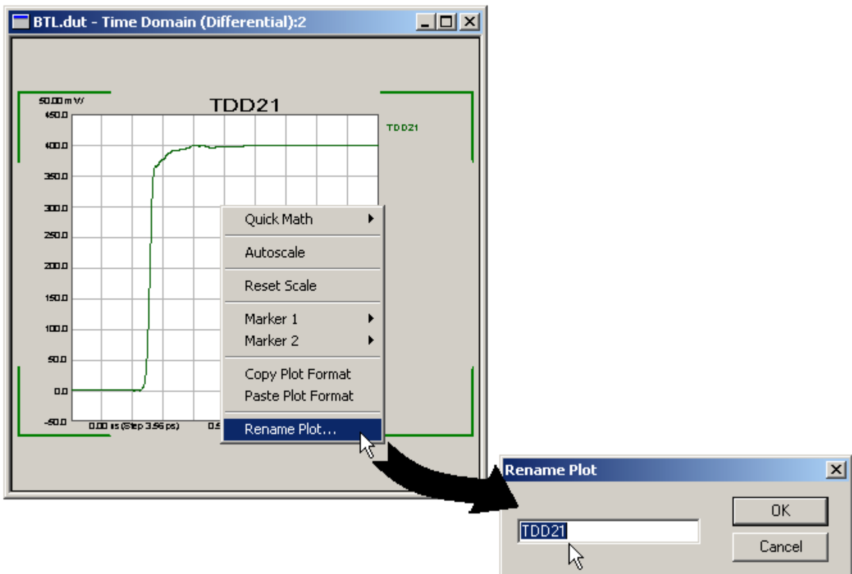
5. Repeat steps 3 and 4 to change as many plots to the new format and scale that you like.

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. To rename a plot:

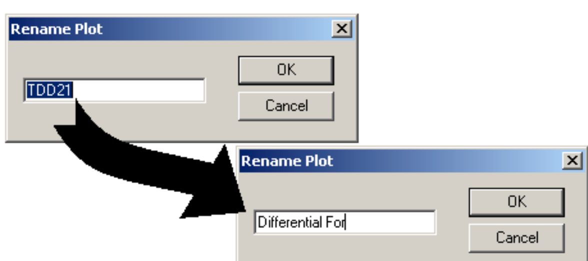
1. Right click on the plot to display the quick menu as shown in Figure 13-38. From the quick menu, select **Rename Plot**... to open the *Rename Plot...* dialog box.

Figure 13-38 Open the Rename Plot Dialog box



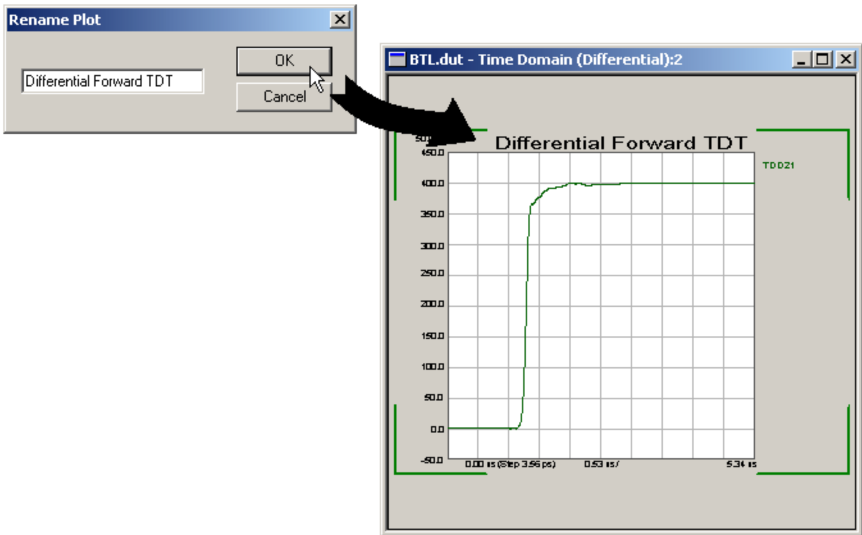
2. Highlight the original label and type in the new label. See Figure 13-39.

Figure 13-39 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 13-40.

Figure 13-40 Saving the Label

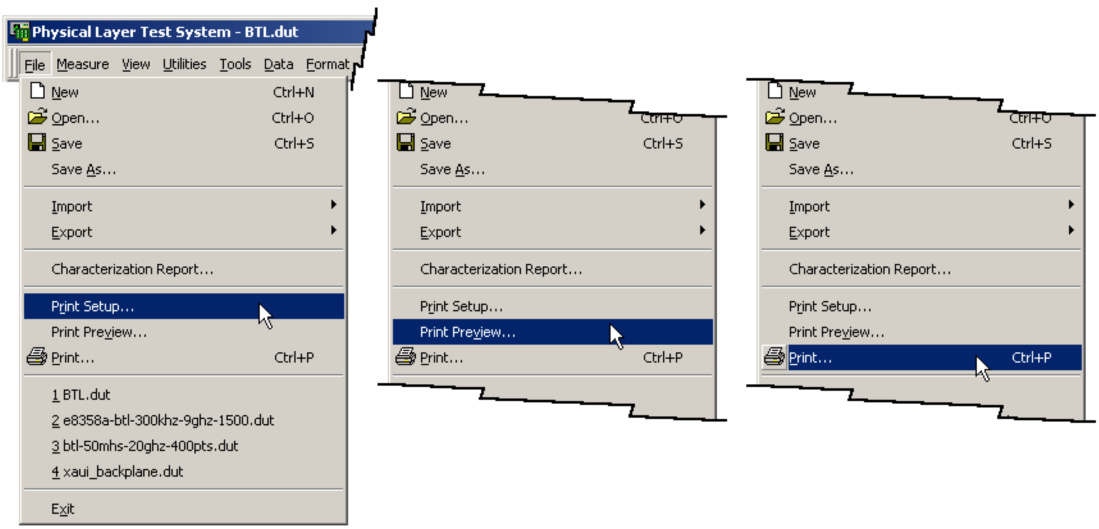


Printing

The Print feature in the **File** menu has three options. Refer to Figure 13-41. Each of the print options is described in this section.

- For Print Setup..., refer to page 399.
- For Print Preview..., refer to page 403.
- For Print..., refer to page 405.

Figure 13-41 Three Print Options from File Menu



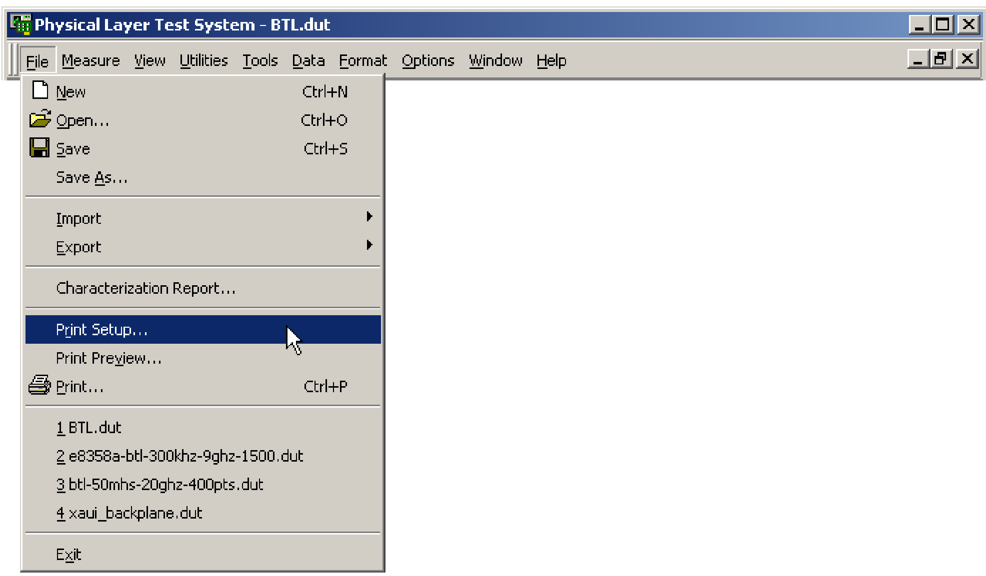
Print Setup...

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 13-42.

Figure 13-42 Opening the Print Setup Dialog Box

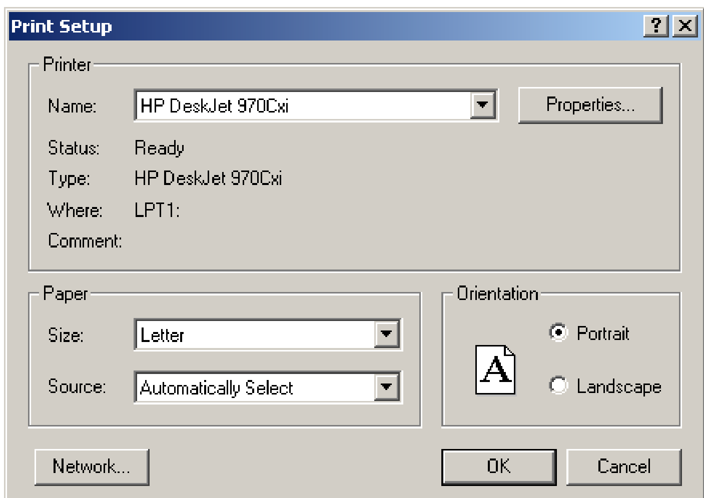


2. Confirm the settings in the *Print Setup* dialog box. See Figure 13-43. Change any incorrect settings for your printer.

   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 13-43 Print Setup Dialog Box



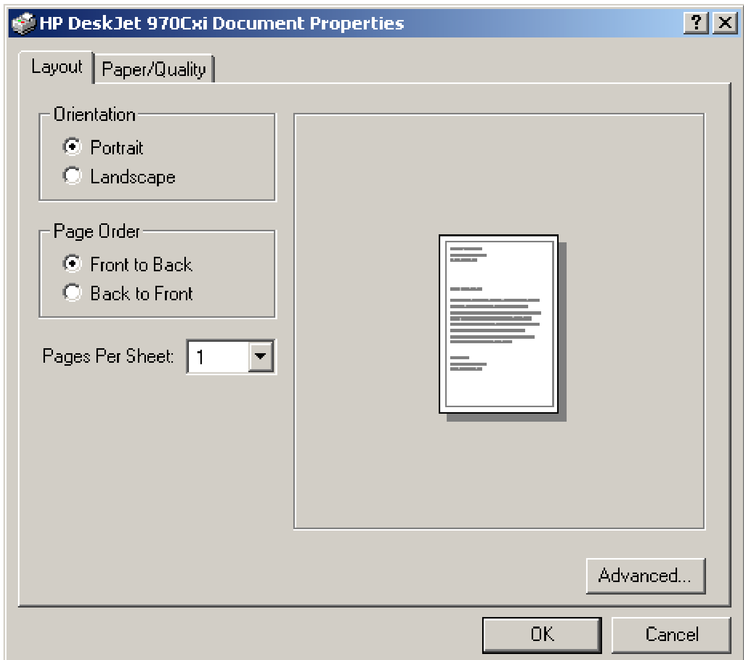
More advanced settings are available by selecting the **Properties**... button or the **Network**... button.

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.

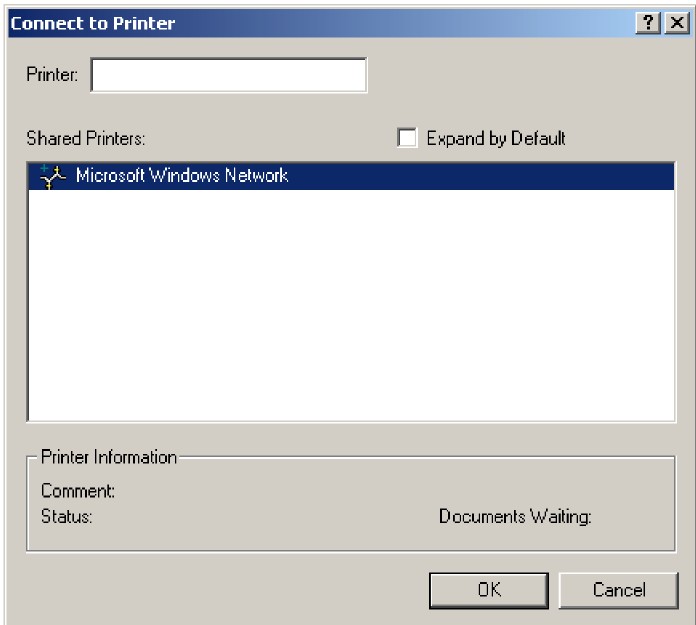
3. If you select the **Properties**... button, the *Document Properties* dialog box is displayed. See Figure 13-44. Each printer has its own unique *Document Properties* dialog box. Refer to your printer documentation for the specific information available for your printer selections.

Figure 13-44 Example Document Properties Dialog Box (Layout Tab)



4. If you select the **Network**... button, the *Connect to Printer* dialog box is displayed. See Figure 13-45.

Figure 13-45 Connect to Printer Dialog Box

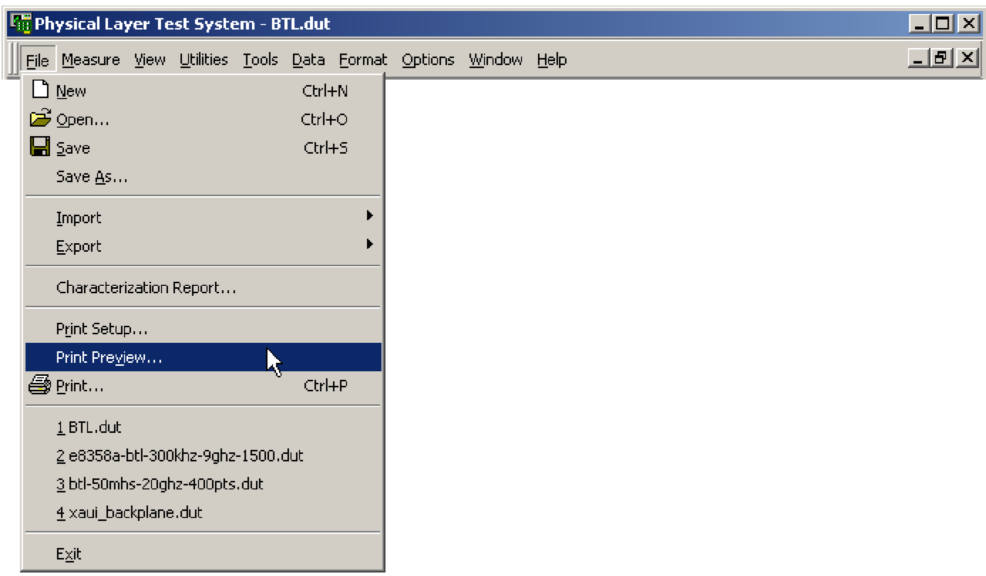


Print Preview…

The **Print Preview**… option allows you to preview your active plot screen before printing the display. To preview your active plot screen before printing:

1. Select **Print Preview**… from the **File** menu to open the preview screen of the active plot. See Figure 13-46.

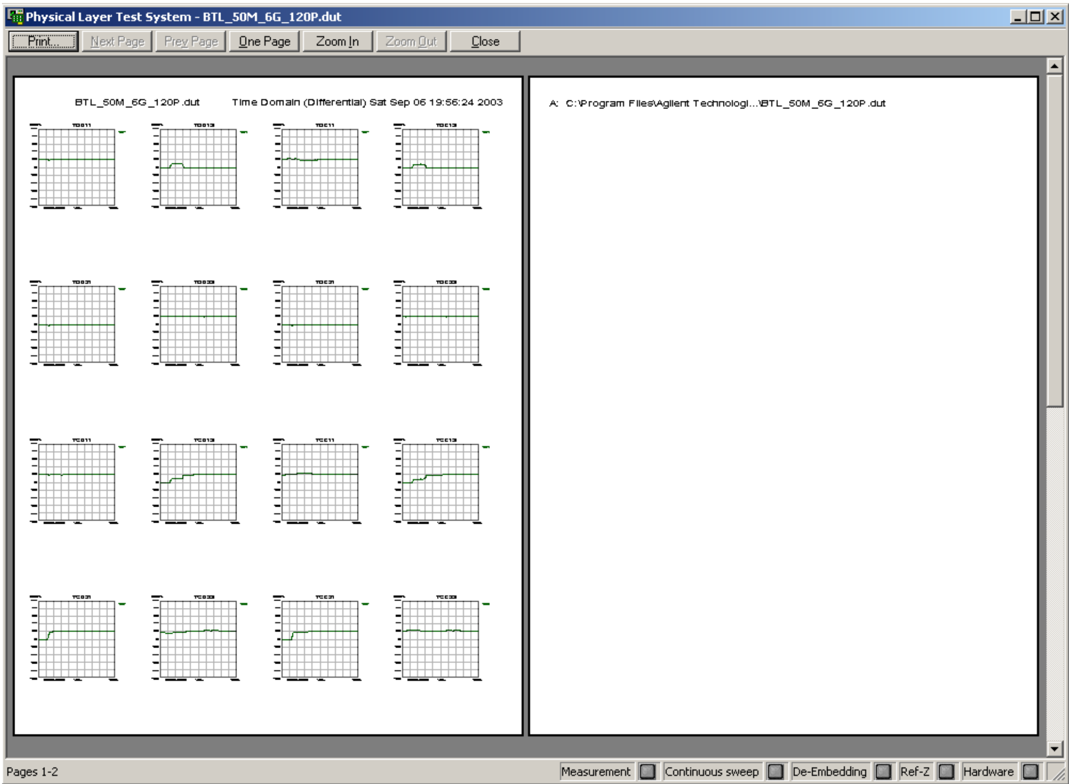
Figure 13-46 Opening the Print Preview Dialog Box



2. Examine the displayed active plot as needed prior to printing. See Figure 13-47.

   The screen showing the active plot that will be printed is displayed for your review.

Figure 13-47 Preview Screen of the Active Plot



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 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.

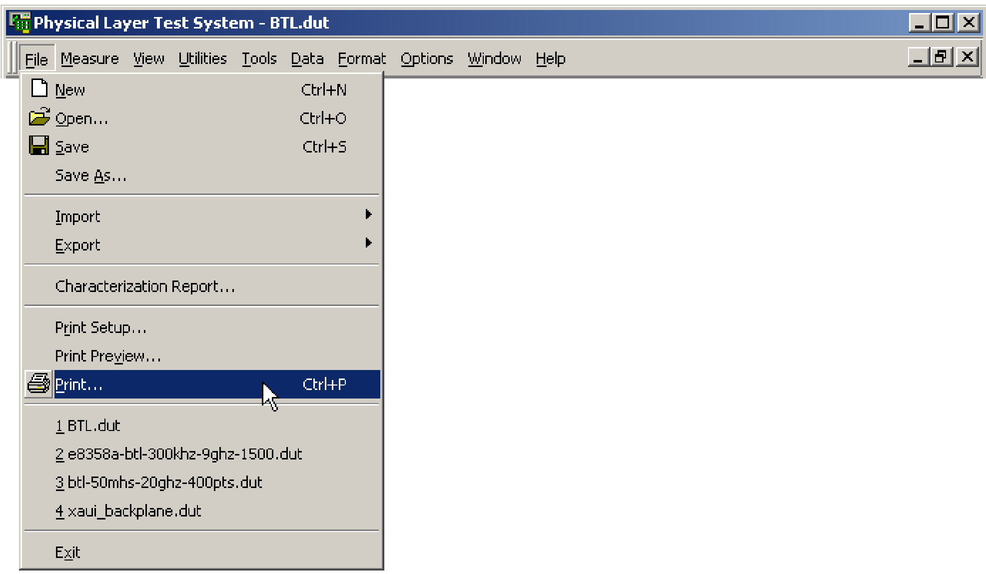
Close: Closes the preview screen without printing.

Print...

The active plot may be printed to the printer defined in the *Print Setup* dialog box. To print the active plot:

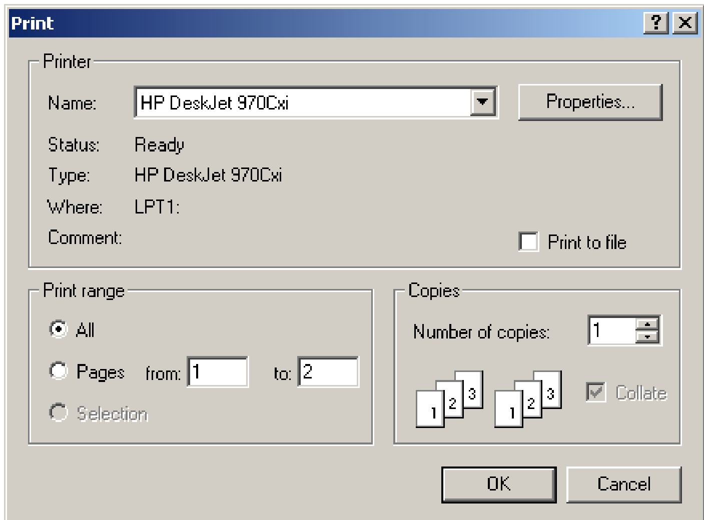
1. Select **Print**... from the **File** menu to open the *Print* dialog box. See Figure 13-46.

Figure 13-48 Opening the Print Dialog Box



2. Make the *Print* dialog box selections that meet your requirements. See Figure 13-49.

Figure 13-49 Print...



In the **Printer** area:

- Select another printer from the **Name** list if the displayed printer is not appropriate.
- Select the **Properties**... button to display the *Document Properties* dialog box unique to the selected printer. This dialog box allows you to make several advanced choices. See Figure 13-44 on page 401. Refer to your printer documentation for specific information available for your printer selection.
- Select the **Print to file** check box, to save the active plot 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 screen.

   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 page 432.
- Selecting the Print button in the print preview display. See page 403.

File Converter

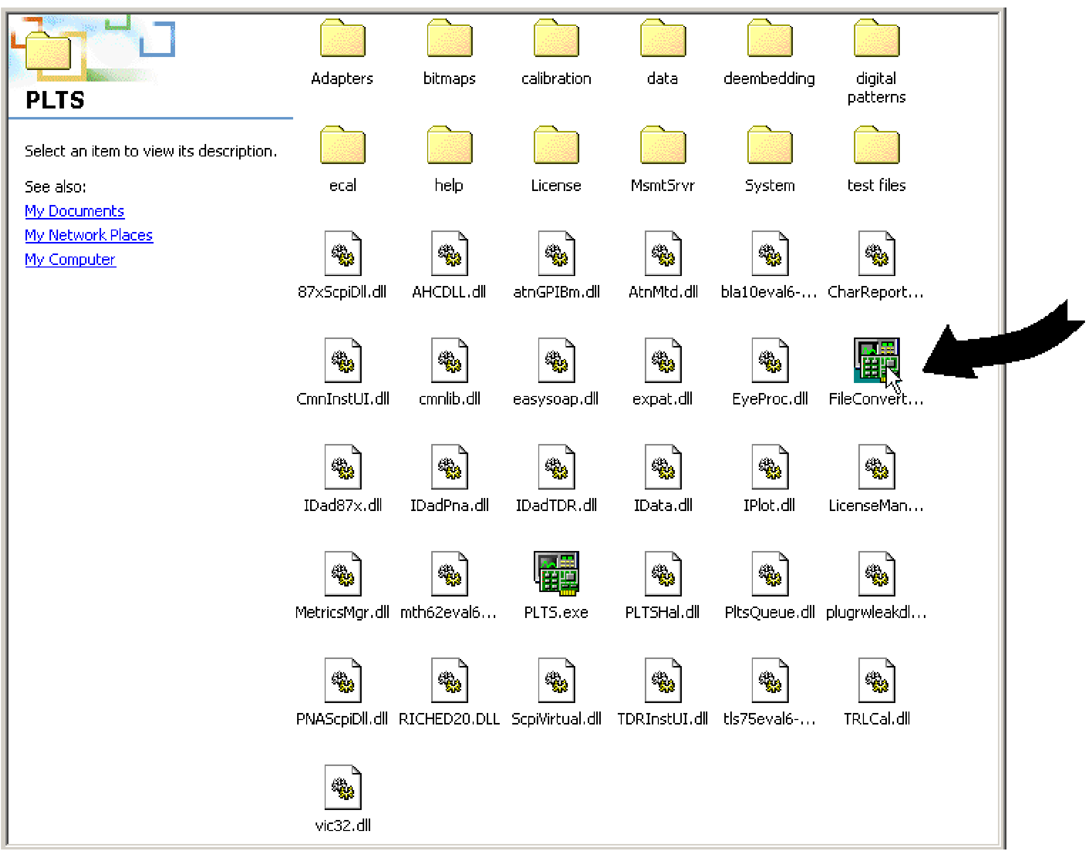
The File Converter was installed with PLTS, it is a separate utility. The File Converter converts files in .dut, .cit, and .s2p formats to files in .txs format. These .txs files can be used during the calibration as files for thru adapters.

It is easiest to copy a Thru adapter file into the Adapters directory of PLTS. If you used the default directory on your C drive, the path is: C:/Program Files/Agilent/PLTS/Adapters

To begin the file conversion:

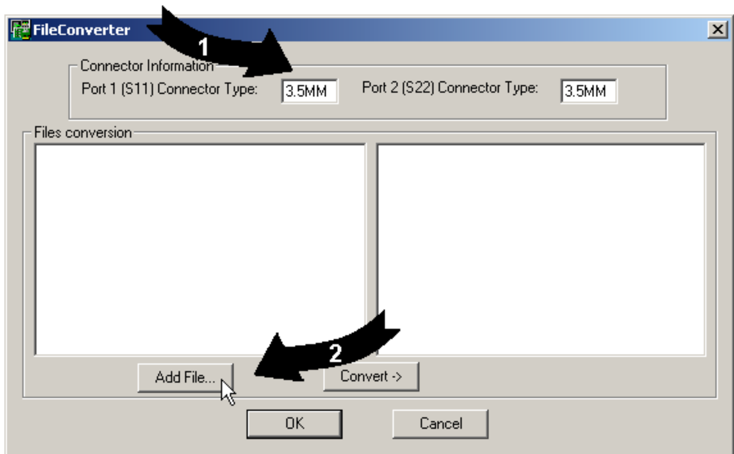
1. From the PLTS directory mentioned above, double-click the FileConverter.exe file icon to open the File Converter.

Figure 13-50 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 13-51 File Converter Dialog Box

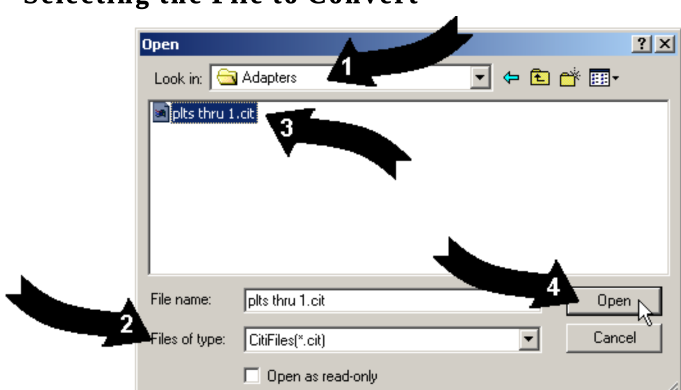


3. In the *Open* dialog box, change to the directory where you copied the files to be converted. In this case: C:/Program Files/Agilent/PLTS/Adapters

   Select the type of file (.dut, .cit, or .s2p file) to be converted in the **File of type:** box. Select the file that you want to convert. When you have selected your file, click the **Open** button.

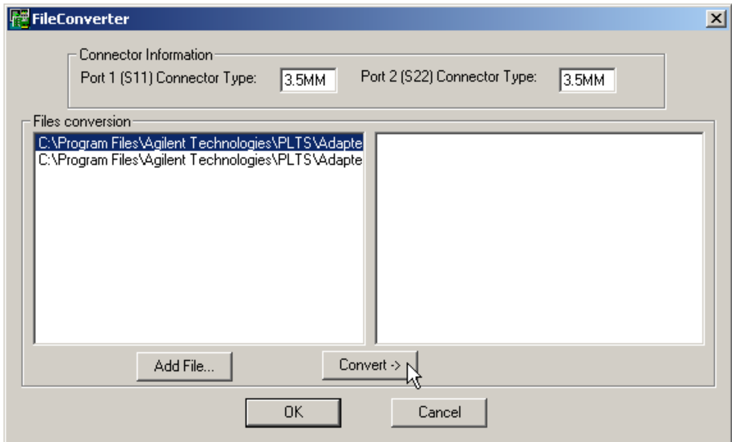
   Repeat if you want to convert multiple files.

Figure 13-52 Selecting the File to Convert



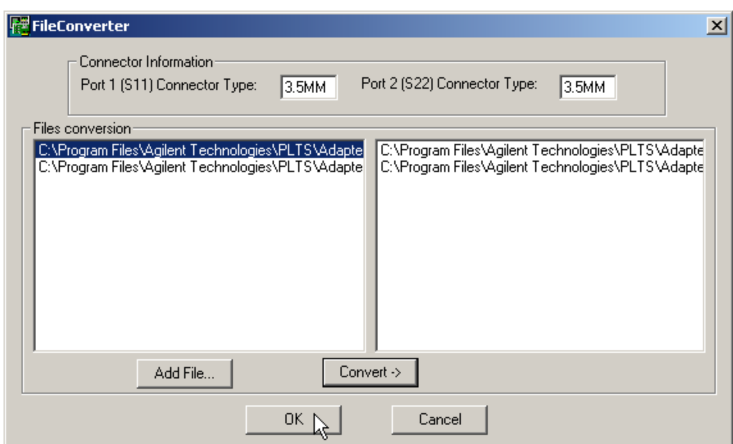
4. With the files in the left list of the **Files conversion** area, click the **Convert ->** button.

Figure 13-53 Convert the Files



5. The converted files are listed in the right list of the **Files conversion** area. Click the **OK** button to save the files into the directory where they were found.
   In this case: C:/Program Files/Agilent/PLTS/Adapters

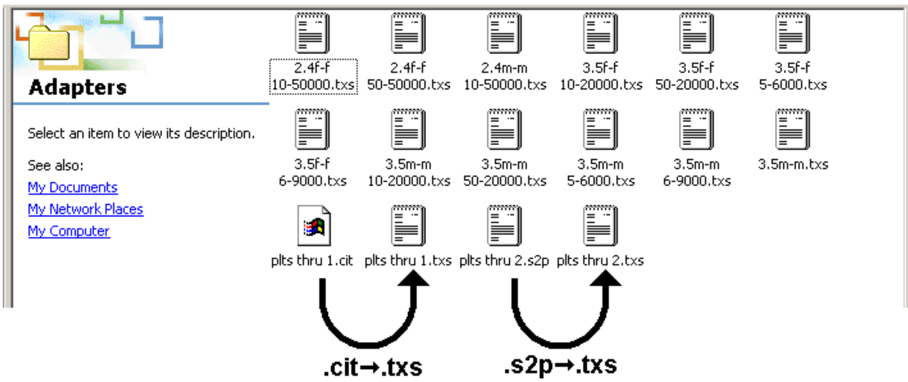
Figure 13-54 Save the Converted Files



Then close the *FileConverter* dialog box.

6. Open the Adapters directory to find the newly converted files. In this case, two files were converted.

Figure 13-55 Adapter Directory with Newly Converted Files



The new .txs files can now be used to define a calibration kit.

IV Reference

Part IV provides reference information related to operating the physical layer test system.

Chapter 14, "Menu Reference"

Provides you with descriptions of each menu bar selection in the test system software.

Chapter 15, "Specifications and Characteristics"

Provides you with the specifications and characteristics of the physical layer test system.

Chapter 16, "Test Set Front Panel and Rear Panel"

Provides you with specific information regarding each test set front panel and rear panel connector, switch, fuse, and LED indicator.

Chapter 17, "Troubleshooting and Maintenance"

Provides you with information about troubleshooting the test system, contacting Agilent for assistance, care of test cables and coaxial connectors, and electrostatic discharge.

Chapter 18, "Safety and Regulatory Information"

Provides you with information that will allow you to operate the test system safely. This chapter also lists the appropriate information required by regulatory agencies.

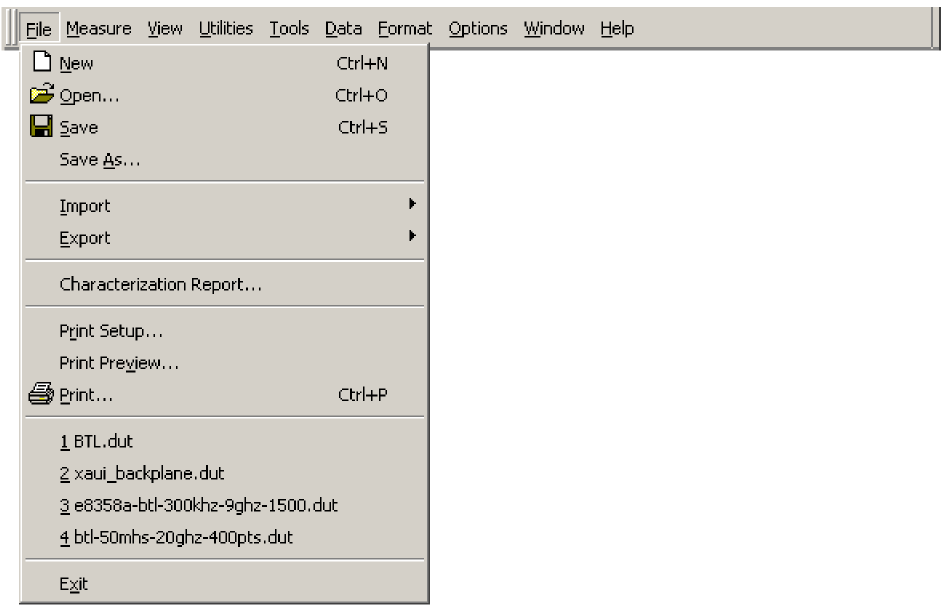
14 Menu Reference

The following menus and each of their selections are described in detail in this chapter. Many of these menus are displayed only under specific measurement conditions. Refer to each specific menu on the pages listed for selections and conditions.

File Menu

The **File** menu provides access to many standard software functions such as creating, opening, and saving data, importing and exporting files, creating a characterization report, printing, and exiting the software.

Figure 14-1 File Menu



New

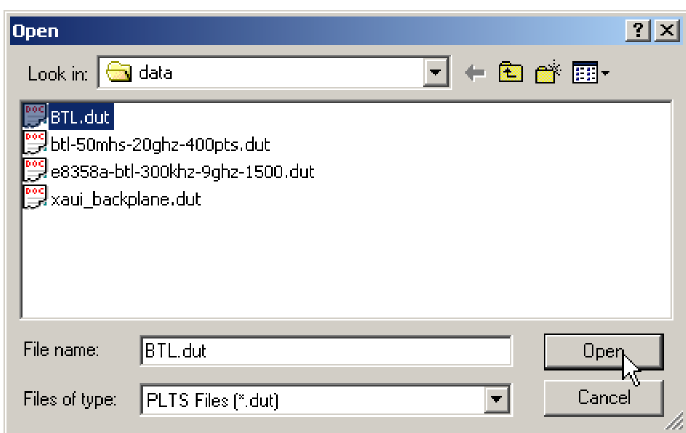
Select **New** from the **File** menu to start a new measurement process and open the *Startup Wizard* window.

Figure 14-2 Startup Wizard Window

Open

Select **Open** from the **File** menu to load a previously saved file.

Figure 14-3 Open Dialog Box

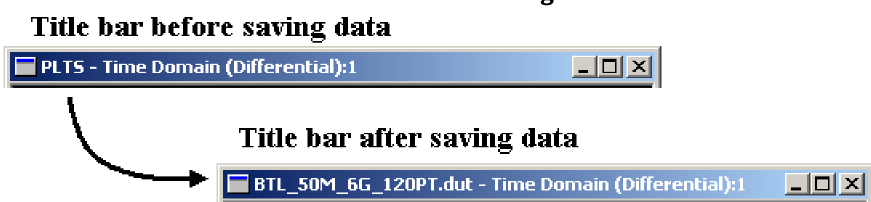


Save

Select **Save** from the **File** menu to save the current measurement or calibration data. New data or imported data may be saved.

Until measurement data is saved, the plot window title bar is labeled PLTS along with the analysis type of the plot window and the sequential plow window number. Once the data is saved, the PLTS label is replaced with the name of the saved file. Figure 14-4 shows the plot window title bar before the data was saved and how it changes to match the file name (BTL_50M_6G_120PT.dut) after it is saved.

Figure 14-4 Plot Window Title Bar Changes After Data is Saved

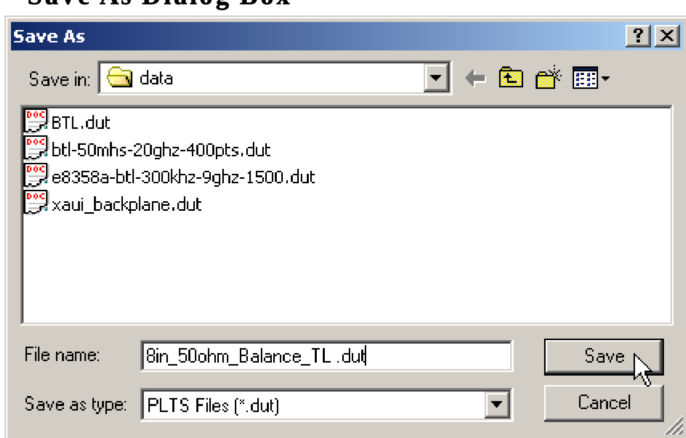


Only one unsaved plot window may be open at a time. If you have one unsaved plot window, you must either save the data or delete the window before you can open another plot window with unsaved data.

Save As...

Select **Save As**... from the **File** menu to save new measurement or calibration data or existing data as a new file.

Figure 14-5 Save As Dialog Box



Import

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** 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.

Figure 14-6 File Menu with Import and CITIFile Expanded

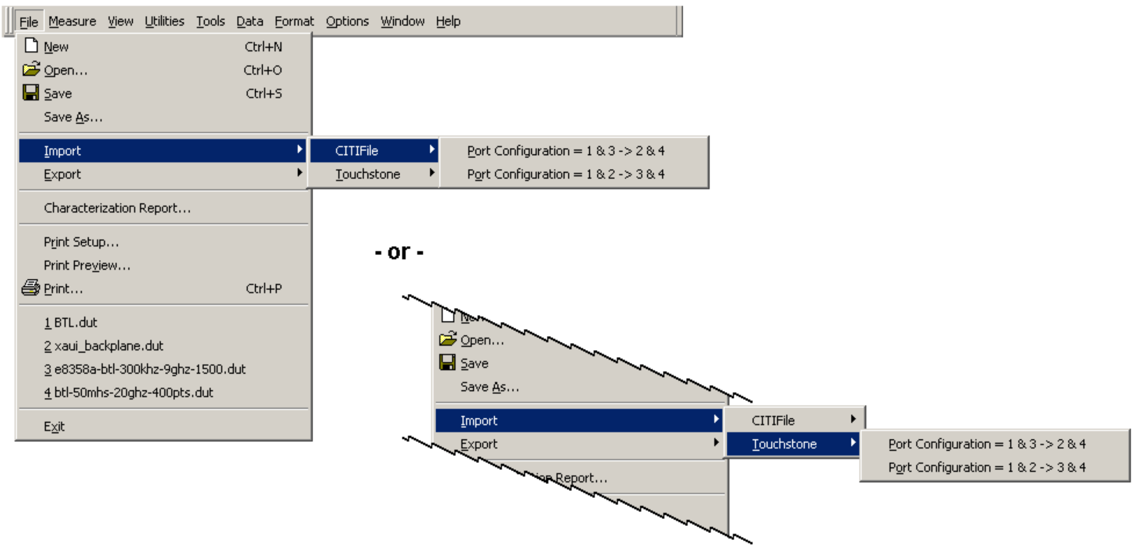
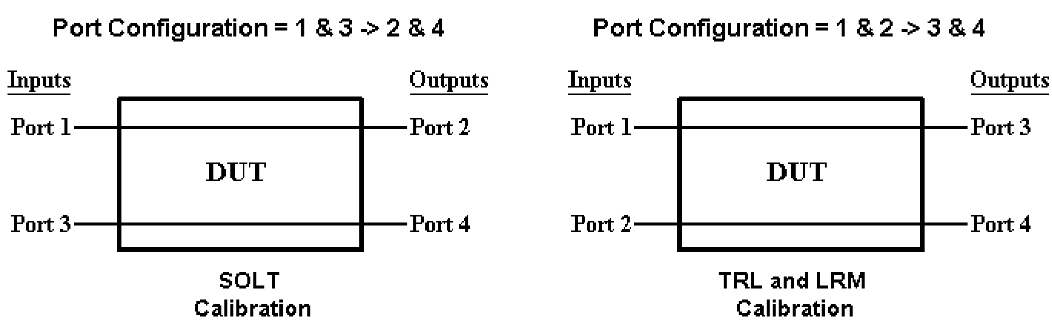


Figure 14-7 Balanced Transform Port Configuration Diagram



CITIFile

CITIFile imports data previously saved in CITIfile (*.cit) format. CITIfiles imported in this

fashion can be used for comparison with other data sets using trace memory and math functions. Refer to Figure 14-7 and choose from one of the following port selections.

Port Configuration = 1 & 3 -> 2 & 4 is used to import single-ended measurement data that has taken with the system calibrated using the SOLT calibration.

Port Configuration = 1 & 2 -> 3 & 4 is used to import single-ended measurement data that has taken with the system calibrated using the TRL calibration.

Touchstone

Touchstone imports data previously saved in Touchstone (*.S4P) format. Refer to Figure 14-7 and choose from one of the following port selections.

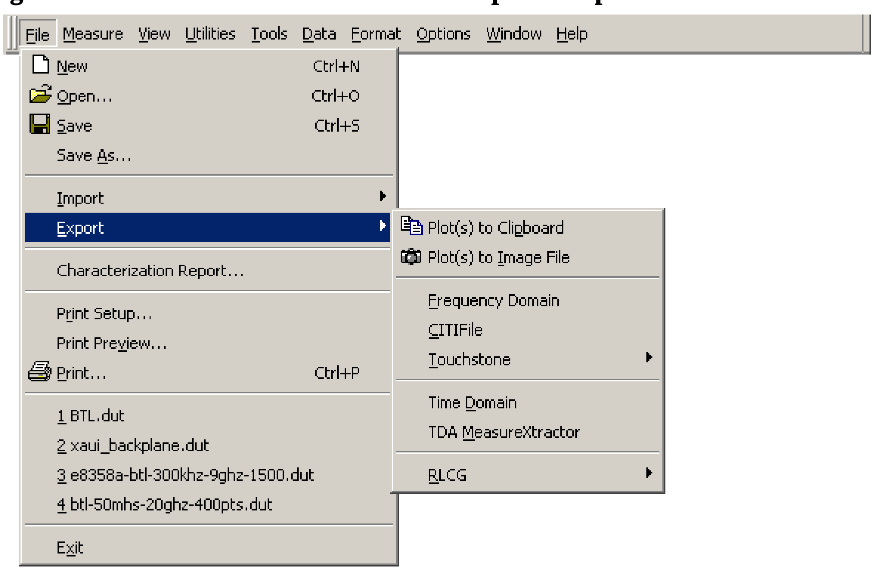
Port Configuration = 1 & 3 -> 2 & 4 is used to import single-ended measurement data that has taken with the system calibrated using the SOLT calibration.

Port Configuration = 1 & 2 -> 3 & 4 is used to import single-ended measurement data that has taken with the system calibrated using the TRL calibration.

Export

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 14-8 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**.

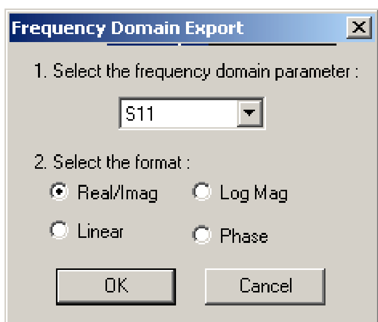
Plots to Clipboard exports the contents of the current plot window to the Windows clipboard.

Plots to Image File exports the contents of the current plot window as an image file. When you export it, you may choose from Windows Bitmap (*.BMP), JPEG Bitmap (*.JPG), or Targa Bitmap (*.TGA) formats.

Frequency Domain exports the data of a single S-parameter (either single-ended or balanced) in one of four frequency domain formats: Real/Imaginary, Log Magnitude, Linear Magnitude, or Phase. All four formats are exported in tabular format as a text file, each with a header. The measured data is bounded by a `BEGIN` and an `END` message. The Real/Imaginary data of each measured point is exported as comma-separated values.

Use the *Frequency Domain Export* dialog box shown in to select the S-parameter (all single-ended and balanced S-parameters are listed). Then click **OK** to export the data.

Figure 14-9 Frequency Domain Export Dialog Box

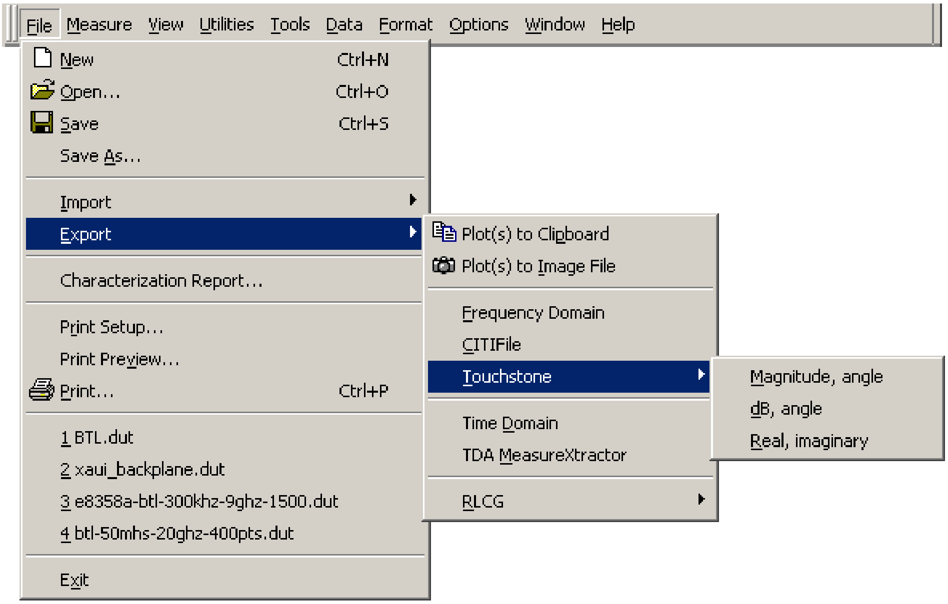


CITIFile exports the current data in CITIfile format (*.cit).

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**

Figure 14-10 File Menu with Export and Touchstone Expanded



Time Domain exports the current time domain data in text format (*.txt). It saves a name, the parameter and format information in a header. In the body, each X and Y measurement coordinate is displayed separated by a comma. These coordinates are bounded by a BEGIN and END message.

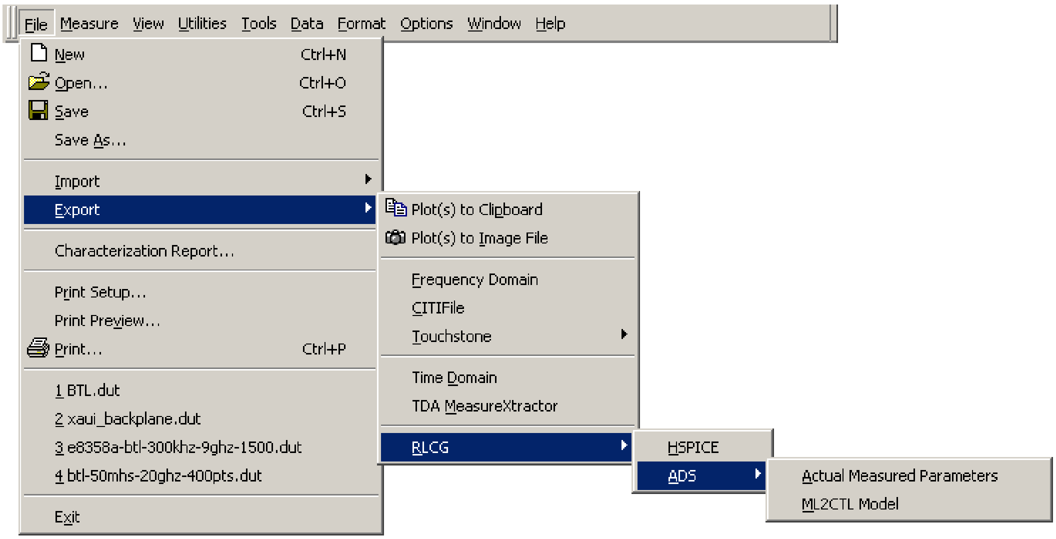
TDA MeasureXtractor exports measured data into the TDA MeasureXtractor format. Refer to "TDA MeasureXtractor" on page 346.

RLCG exports the current data in one of two RLCG formats:

- **HSPICE** 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 325.
- **ADS** exports the RLCG data in a format compatible with the Agilent Advanced Design System (ADS) software format which also has the two following data format choices in which the data may be saved:
  - ❑ **Actual Measured Parameters** exports extracted parameters versus frequency in an ASCII file for ADS. Refer to "Exporting Transmission Line Data" on page 325.
  - ❑ **ML2CTL** exports fitted parameters in an ASCII file to be used with the ADS MultiLayer 2 Coupled Transmission Lines (ML2CTL) model. Refer to "Exporting

Transmission Line Data" on page 325.

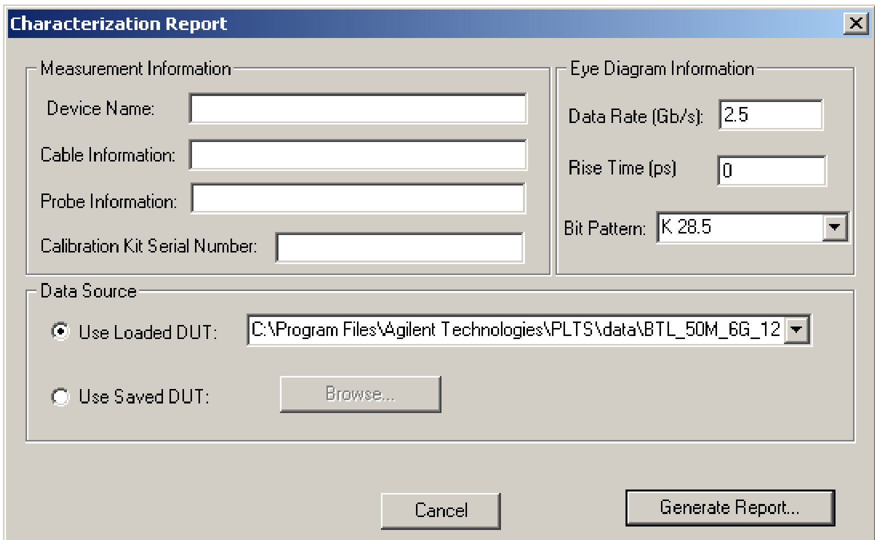
Figure 14-11 File Menu with Export, RLCG, and ADS Expanded



Characterization Report...

Click the **File** menu and then click **Characterization Report**... to start creating the characterization report. The *Characterization Report* dialog box is displayed.

Figure 14-12 Characterization Report Dialog Box

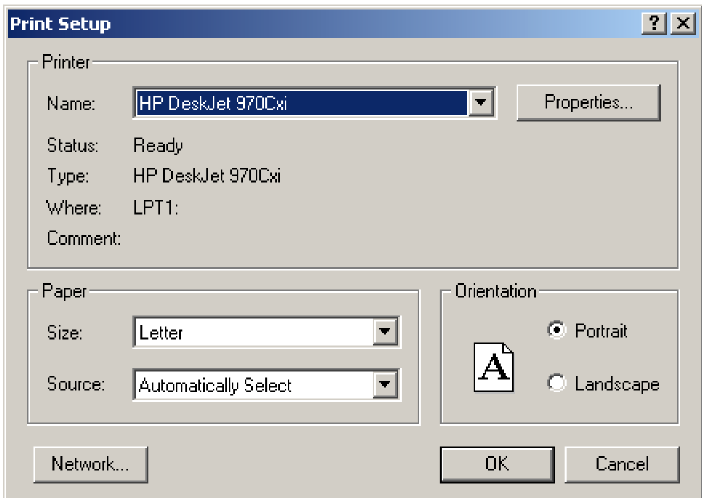


Refer to “Characterization Report Generator” on page 389 for more information.

Print Setup…

Click the **File** menu and then click **Print Setup**… to review the printer settings. The *Print Setup* dialog box is displayed. The *Print Setup* dialog box allows you to select the destination printer and its properties, the paper size and printer paper source, additional network printers, and choose the orientation of the paper when printed.

Figure 14-13 Print Setup Dialog Box



If you are connected to other networks, click **Network...** to find additional printers.

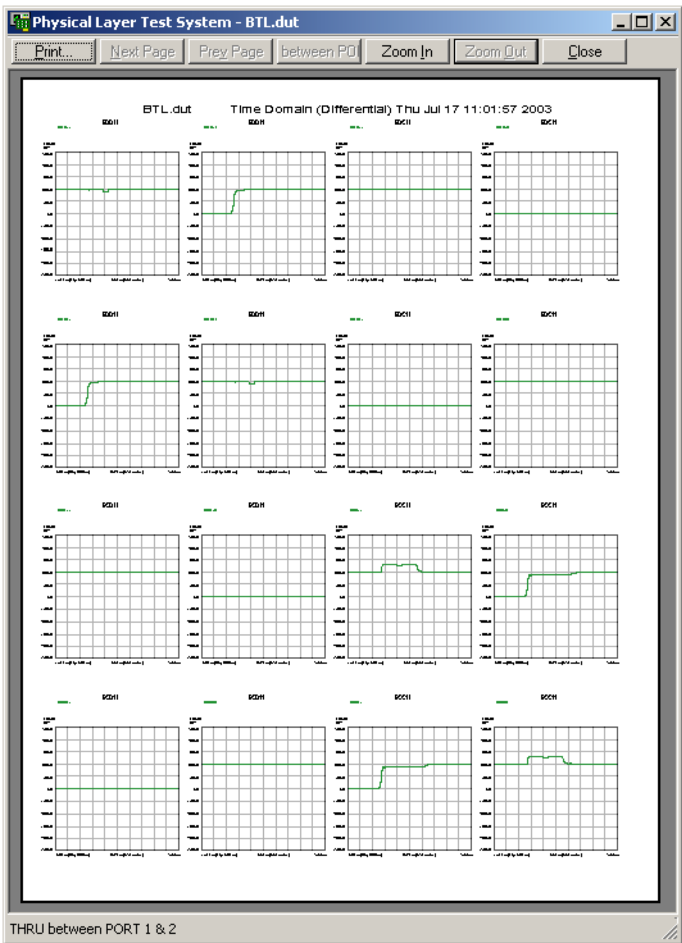
Click **OK** to print the selected plots and return to the program.

Click **Cancel** to close the dialog box and return to the program without printing.

Print Preview...

Click the **File** menu and then click **Print Preview**... to review the active plot window prior to printing. The data is displayed in the preview window. The preview window allows you to move between multiple pages of data, zooming in and zooming out on displayed data, printing the data, and closing the window.

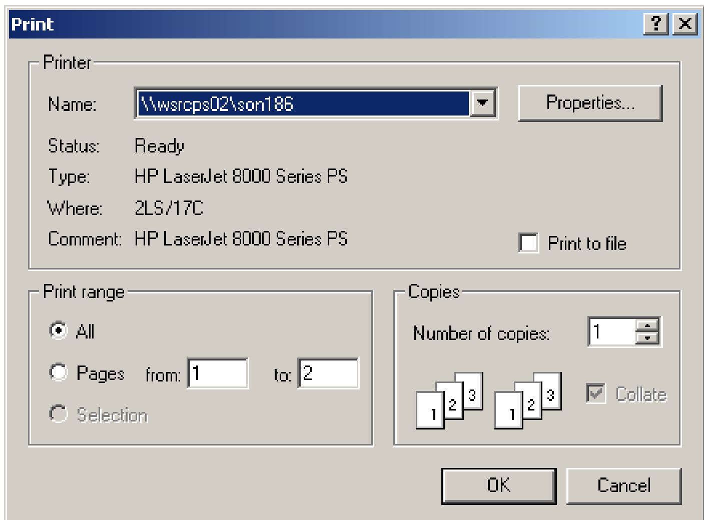
Figure 14-14 Displayed Data when Print Preview... is Selected



Print...

Click **Print**... from the **File** menu to print the selected plots and display the *Print* dialog box. The *Print* dialog box allows you to select the destination printer and its properties, the range of pages to be printed, and the number of copies you wish to print.

Figure 14-15 Print Dialog Box



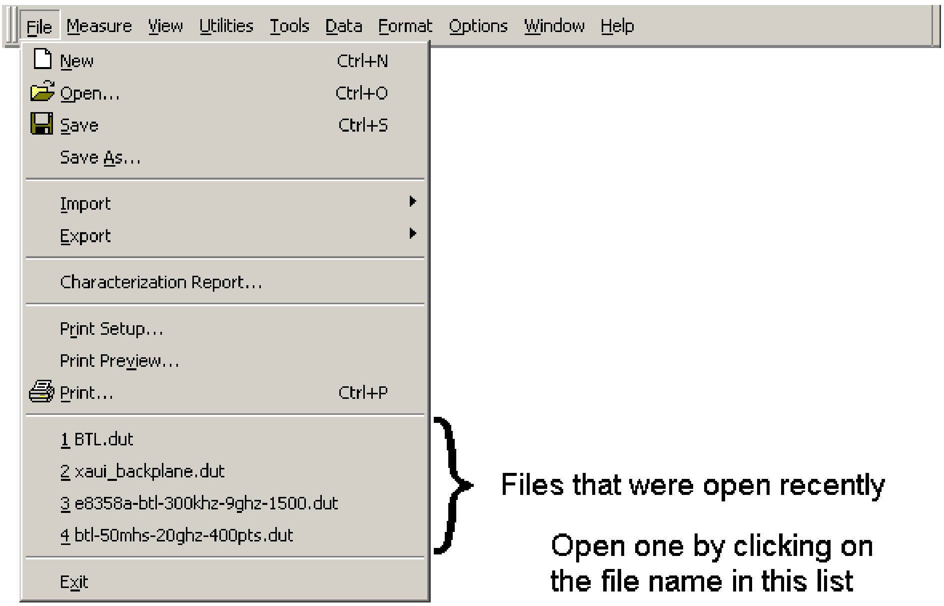
Click **OK** to print the selected plots and return to the program.

Click **Cancel** to close the dialog box and return to the program without printing.

Recent Files

Open any of the four most recently accessed files by clicking the name of the file from this list. Only the four most recently accessed files are displayed.

Figure 14-16 Opening Recently Accessed Files



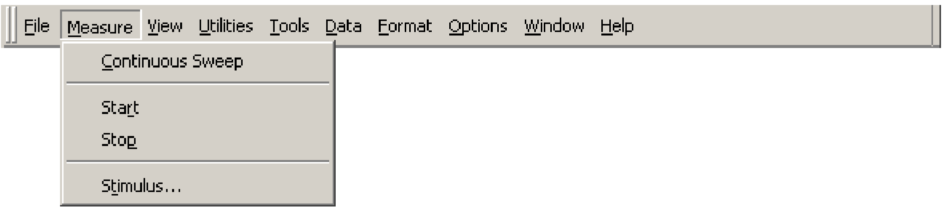
Exit

Click **Exit** from the **File** menu to quit this program.

Measure Menu

The Measure menu allows you to start a measurement, start and stop a continuous sweep measurement, and change the stimulus.

Figure 14-17 Measure Menu



Continuous Sweep

Continuous Sweep sets the system to make measurements as the selected frequency span is swept continuously. After each measurement, the displayed data is updated.

To perform a continuous sweep measurement, select **Continuous Sweep** so that it has a check mark on its left side on the Measure menu. With **Continuous Sweep** active, the **Start** and **Stop** menu selections control continuous sweep. Select **Start** to begin a continuous measurement and select **Stop** to stop the measurement.

The **Continuous sweep** indicator on status bar means that system is currently in the continuous sweep mode. See Figure 14-18.

Figure 14-18 Continuous Sweep Indicator on the Status Bar

Start

Start begins a measurement when it is selected. When **Continuous Sweep** has a check mark, **Start** begins a continuous sweep measurement; otherwise it begins a single sweep measurement.

Stop

Stop stops a continuous sweep measurement.

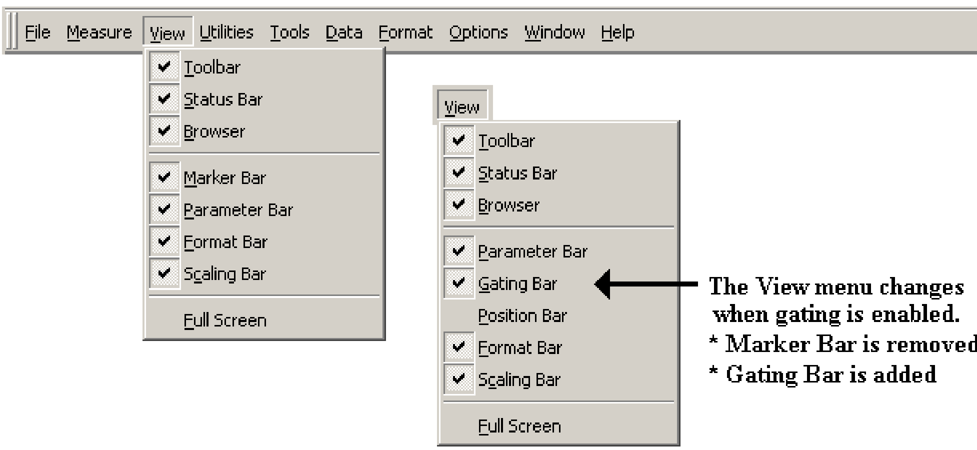
Stimulus...

Stimulus... opens the *Modify Time and Frequency Parameters* dialog box so that you can change the parameters. Refer to the stimulus information on "Setting Up the Calibration and Measurement Parameters" on page 111 for detailed information regarding changing the stimulus parameters.

View Menu

The **View** menu opens and closes the various tool bars.

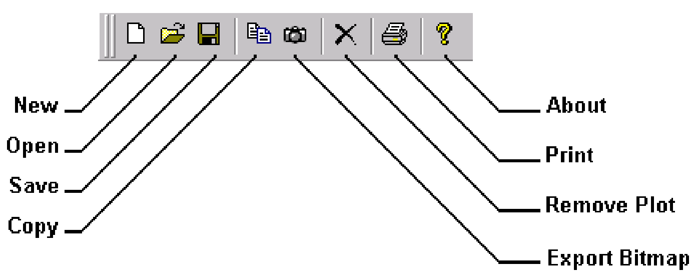
Figure 14-19 View Menu



Toolbar

The **Toolbar** provides quick access to several File menu features as well as two others. The features that are available on the **Toolbar** are **New**, **Open**, **Save**, **Copy**, **Export Bitmap**, **Remove Plot**, **Print**, and **About**.

Figure 14-20 Toolbar



Status Bar

The **Status Bar** provides a graphic display of when several features are active. Each feature has a label on the left and an indicator just to the right side of the label. When a feature is active, the indicator is changed to a bright color. When the feature in not active, the indicator is gray. The impedance of each test system channel is displayed at the right side of this bar.

Figure 14-21 Status Bar

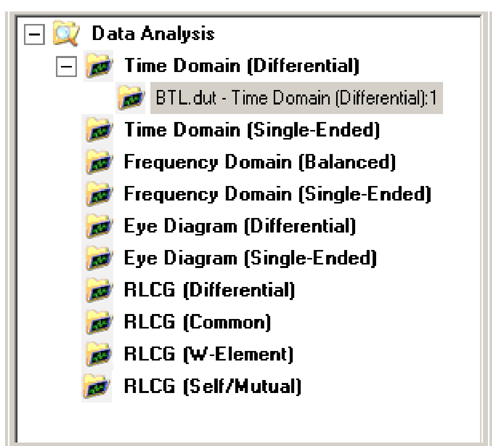
Measurement	is bright when a measurement is being performed
Continuous sweep	is bright when a continuous sweep measurement is being performed
De-Embedding	is bright when de-embedding is applied to the data
Ref-Z	is bright when port reference impedance is applied to the data
Hardware	is bright when the software recognizes the GPIB-connected hardware

Browser

The **Browser** allows selection of data analysis type and the selection of plots to view. Selecting a data analysis type (a bold selection) opens a blank plot window for that analysis type. Selecting an existing plot (a non-bold selection) makes that plot active and displays it in the front of the plot window.

Data analysis types that have a plot opened have either a “+” or a “–” to the left of the label. Select the “+” to display all plots of that type or the “–” to collapse and hide all plots.

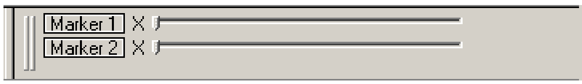
Figure 14-22 Browser



Marker Bar

The **Marker Bar** allows you to add up to 2 markers to a plot. Simply select the plot, select the marker button, and click and drag the horizontal scroll bar in the window to move the marker in the plot. The marker X and Y values are displayed to the right of the plot. The frequency domain Smith Chart and Polar Chart formats allow you to choose between magnitude/phase and impedance styles. This is *not* available when time-domain gating is enabled.

Figure 14-23 Marker Bar



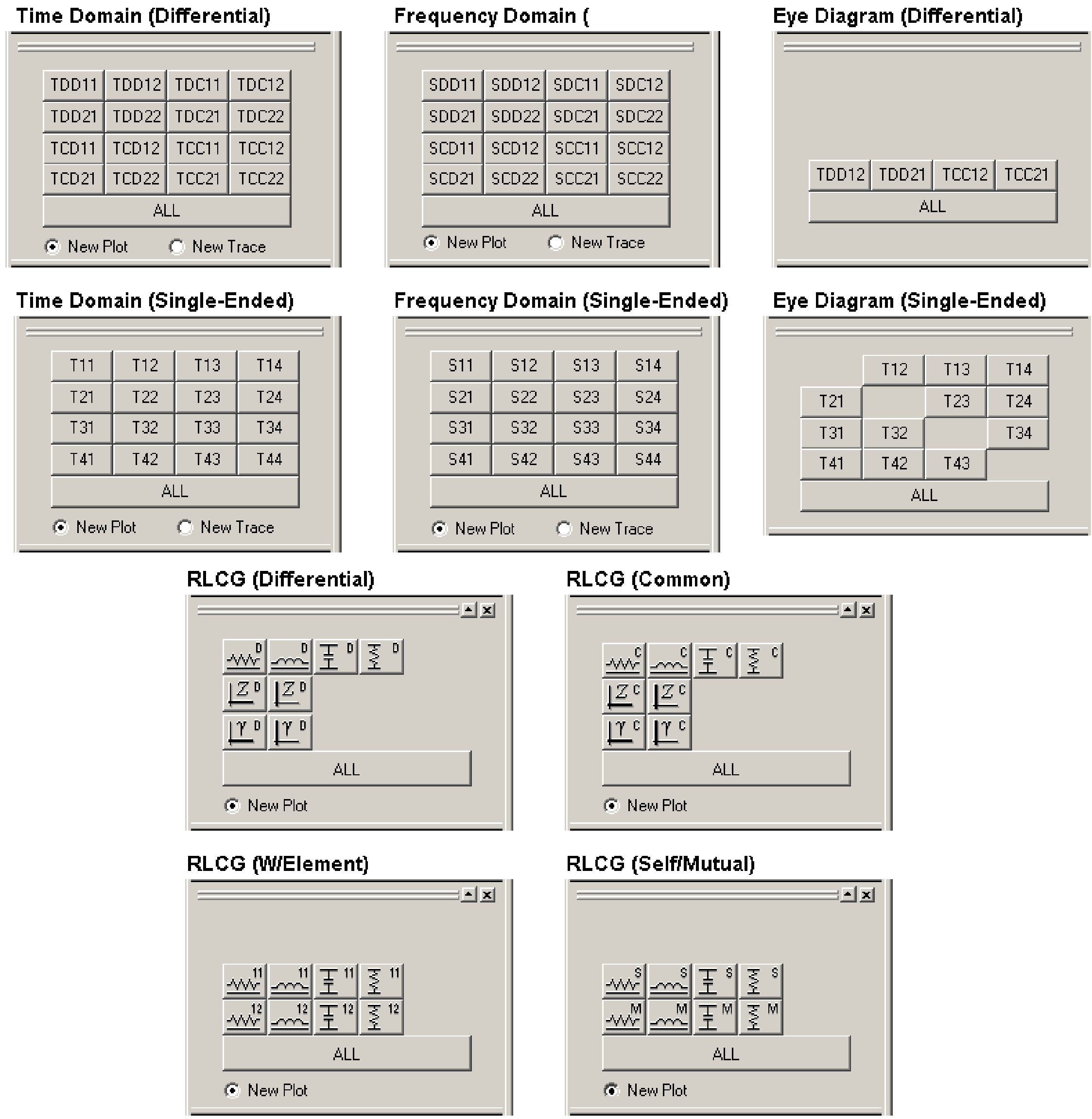
Refer to “Markers” on page 365 for Marker information.

Parameter Bar

The **Parameter Bar** displays each individual parameter for each specific data analysis type as well as the capability to display all of the parameters. See Figure 14-24 on page 435.

The time domain, frequency domain, and RLCG data analysis types also have the option to allow you to display multiple plots on the same plot or separate plots. For Time Domain, Frequency Domain, and Eye Diagram, refer to “Data Menu” on page 457 for detailed information. For RLCG, refer to “RLCG Menu” on page 462 for detailed information.

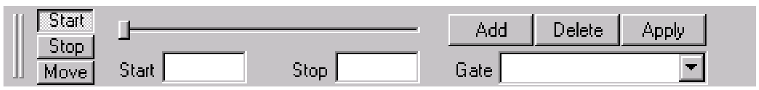
Figure 14-24 Parameter Bars for Each Data Analysis Type



Gating Bar

The **Gating Bar** allows you to add up to 10 gates to a time domain plot. Select **Gating** from the **Utilities** menu to display this bar. After the time domain plot is displayed, slide the horizontal control to set the gate's stop and start position and then press **Add** to add the gate to the plot. Gates may also be deleted and moved from this window. See "Gating" on page 352. This is only available when time domain Gating is enabled.

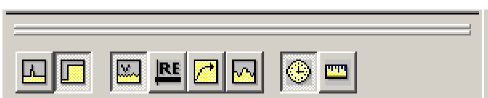
Figure 14-25 Gating Bar



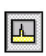
Format Bar

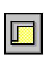
The **Format Bar** displays the plot using the format selected from the bar. **Format Bar** is only available for Time Domain and Frequency Domain plots. As shown below, the Time Domain Format Bar differs from the Frequency Domain Format Bar. Refer to "Format Menu" on page 460 for a detailed description of each format item.

Time Domain Format Bar



For detailed information about each of the selections, refer to "Selecting Time Domain Display Formats" on page 272.

 Impulse

 Step

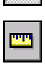
 Volts

 Real

 Log Mag

 Impedance

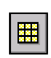
 ns (nanoseconds)

 cm (centimeters)

Frequency Domain Format Bar

For detailed information about each of the selections, refer to "Selecting Frequency Domain Display Formats" on page 233.

 Log Mag

 Linear Mag

 Phase

 Group Delay

 Smith Chart

 Polar Chart

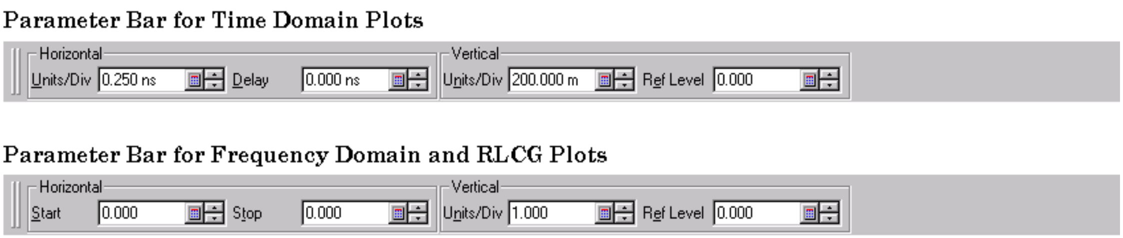
 Real

 Imaginary

Scaling Bar

Scaling Bar allows you to change the scale of the active plot. There are two different scaling bars available. The analysis type determines the scaling bar that is displayed. Eye diagrams do not have an associated scaling bar.

Figure 14-26 Scaling Bar



Full Screen

Full Screen enlarges the **Plot** area to full screen by:

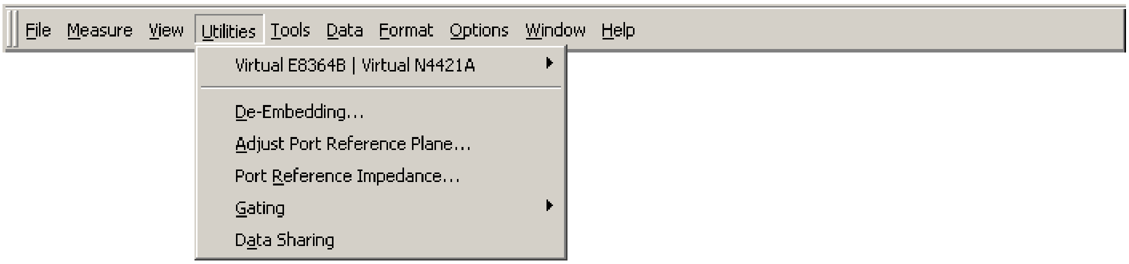
- Removing the **Browser**. The **Browser** may be turned on and off by selecting the **View** menu.
- Making the Parameter Bar, Scaling Bar, Format Bar, and Marker Bar free floating over the PLTS window. These tool bars may be turned on and off through the **View** menu.

Clinking the **Full Screen** selection again, turns of the Full Screen feature, returning the Plot area, Browser, and the tool bars back to normal.

Utilities Menu

The **Utilities** menu provides access to calibration resources and several enhancement tools that you can use to provide a realistic analysis result.

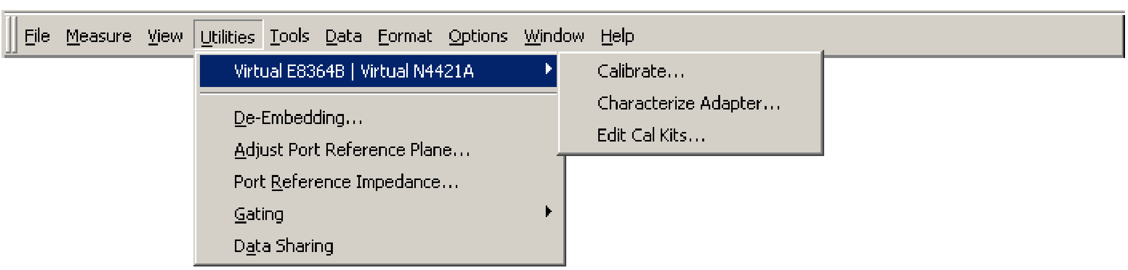
Figure 14-27 Utilities Menu



Calibration

The **Calibration** selection allows you to start a calibration, characterize an adapter used for calibration, or edit the definition of mechanical calibration kits.

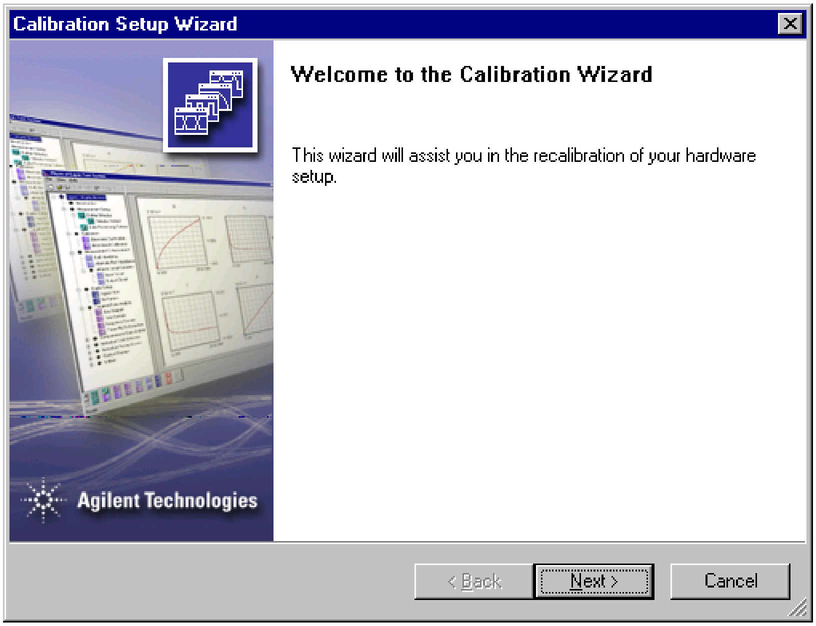
Figure 14-28 Utilities Menu with Calibration Expanded



Calibrate

Selecting **Calibrate** opens the wizard so that you may begin your calibration. Refer to Chapter 5, “Performing Error Correction on the VNA-Based PLTS,” for detailed calibration information.

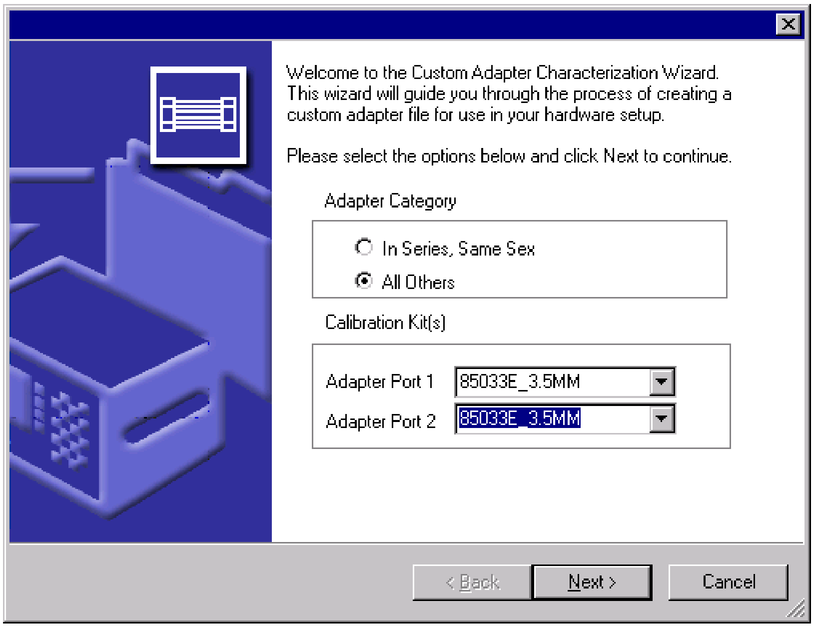
Figure 14-29 Calibration Wizard



Characterize Adapter

Adapters used in measurements and calibration must be characterized to ensure accurate results. The Physical Layer Test System software has a wizard that steps you through this adapter characterization process. Select **Characterize Adapter** from the **Utilities** menu to start the *Custom Adapter Characterization Wizard*.

Figure 14-30 Custom Adapter Characterization Wizard



A short/open/load calibration is performed directly at the network analyzer front panel test port with out any cables. Then the calibration is repeated with the adapter inserted. The resulting adapter S-parameters are saved in CITIfile format, which can later be de-embedded from the device measurement.

To allow for best interpolation of adapters used in broadband measurements, characterize the adapter over the entire frequency range of the system with as many points as possible.

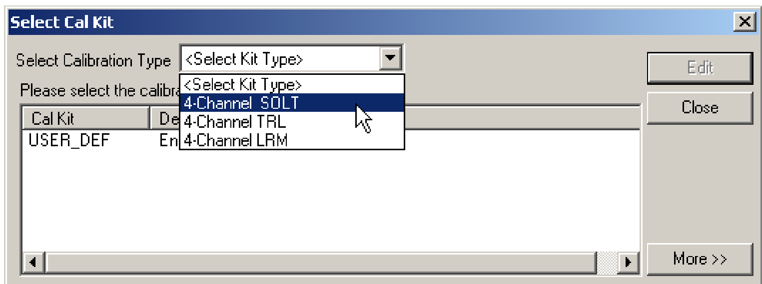
Your adapters must have an orientation, forward and reverse directions. Mark the connectors on the adapter as ports 1 and 2. Forward orientation has the lower-numbered adapter port connected to the test-set port.

Refer to “Characterizing Adapters” on page 176 for detailed instructions on performing adapter characterization.

Edit Cal Kit

The **Edit Cal Kit** feature gives you flexibility to make changes to your calibration kit definition or to add new calibration kits. First, select the calibration type from the **Select Calibration Type** list. Then, from the **Cal Kit** column, select your calibration kit model number to edit an existing kit or select **USER_DEF** to define a new calibration kit.

Figure 14-31 Select Cal Kit Dialog Box



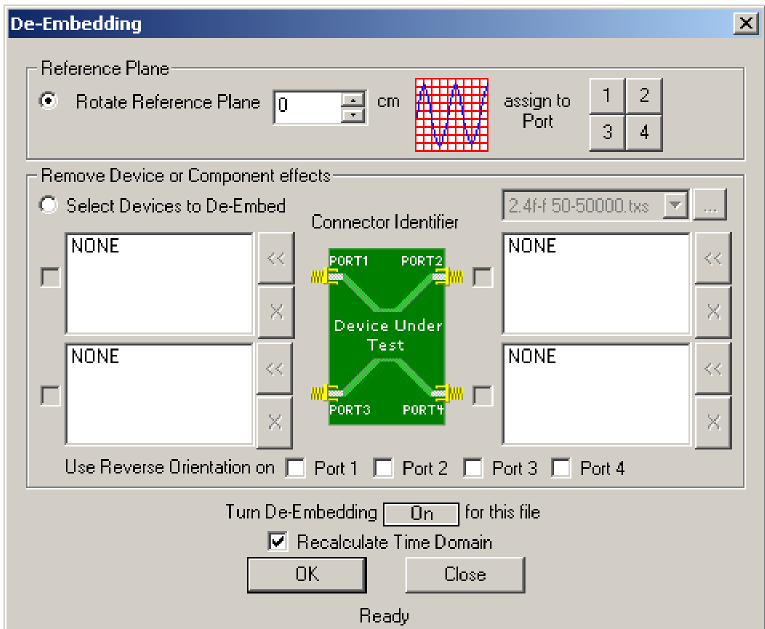
An *Edit Calibration Kit* dialog box is displayed allowing you to change the calibration kit parameters. The **USER_DEF** selection provides a blank dialog box that you can enter new parameter values. Refer to the list below for an illustration showing each of the *Edit Calibration Kit* dialog boxes and the location of detailed instructions for completing the calibration kit definitions.

Calibration Type	To define the calibration kit, refer to the following section for detailed instructions:
SOLT	"Defining a SOLT Calibration Kit" on page 132
TRL	"Defining a TRL Calibration Kit" on page 150
LRM	"Defining an LRM Calibration Kit" on page 165

De-Embedding

Select **De-Embedding** from the **Utilities** menu to open the *De-Embedding* dialog box. Use the *De-Embedding* dialog box to configure DUT reference planes by de-embedding probes, fixtures, or adapters from the measured data.

Figure 14-32 De-Embedding Dialog Box



Select the reference plane rotation using one of the following methods:

- **Reference Plane**

  You can use the **Adjust Port Reference Plane** selection in the **Utilities** menu (see “Adjust Port Reference Plane” on page 444) or you can use this method of adjusting the reference plane.

  1. Click **Rotate Reference Plane**.
  2. Enter a reference plane rotation value (in centimeters-in-air).

     A positive value rotates the phase towards the DUT (effectively removing a length of 50Ω line) and a negative number rotates the phase away from the DUT (effectively adding a length of 50Ω line).
  3. Select the port to assign the rotation value.

4. Repeat steps 2 and 3 until you have assigned rotation values to all appropriate ports.
5. Make sure that De-embedding is on. If the **OFF** button is displayed, click it to turn de-embedding on.

- **Remove Device or Component Effects**

1. Click **Select Device to De-Embed**.
2. Select a de-embedding file (.txs) that you want to use from the drop-down box.
3. Select the check box associated with the port to which the file applies.
4. Click the associated **<<** button to add the file to the port. (Clicking the ✕ button will remove the file from the port.)
5. Repeat steps 2 through 4 until you have assigned de-embedding files to all appropriate ports.
6. Select all ports that use reverse orientation.

   Reverse orientation is noted by check marks for each port specified in the **Use Reverse Orientation** check boxes.
7. Make sure that de-embedding is on. If the **OFF** button is displayed, click it to turn de-embedding on.

Click **OK** to accept the configuration and exit the dialog box or **Cancel** to ignore any changes and exit the dialog box.

De-embedding files hold information about the ports on the fixture that you are de-embedding, and match the connectors during de-embed setup. If you try to specify a fixture connection that cannot be made, the software warns you.

NOTE Port connectors are defined by the cal kits assigned to them in the *Calibration Kit Selection* dialog box.

The **De-Embedding** indicator on status bar means that is being applied. See Figure 14-33.

Figure 14-33 De-Embedding Indicator on the Status Bar

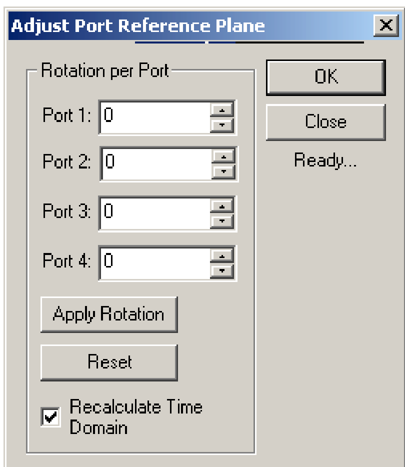
Adjust Port Reference Plane

Select **Adjust Port Reference Plane** from the **Utilities** menu to open the *Adjust Port Reference Plane* dialog box where you can change the reference plane rotation settings and recalculate the displayed data.

Enter the new reference plane rotation values for the desired ports by using the arrows or entering the values in the port box directly. A positive value rotates the phase towards the DUT (effectively removing a length of 50Ω line) and a negative number rotates the phase away from the DUT (effectively adding a length of 50Ω line).

Click **Apply Rotation** to recalculate the data and display the new plots. **Reset** returns the values to zero, recalculates the data, and displays original plots.

Figure 14-34 Adjust Port Reference Plane Dialog Box

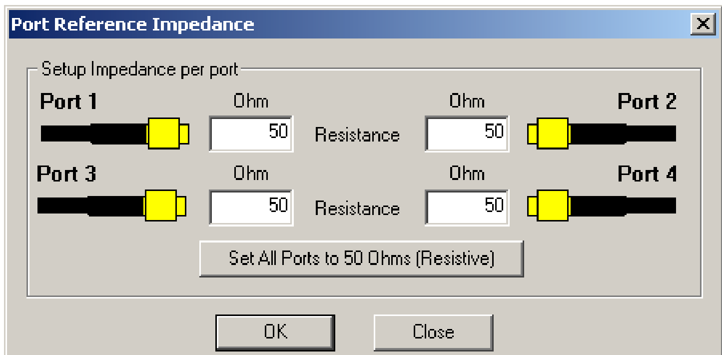


Note that when reference plane is applied, the **De-Embedding** indicator color changes to a bright color. See Figure 14-33.

Port Reference Impedance

Select **Port Reference Impedance** from the **Utilities** menu to open the *Port Reference Impedance* dialog box. By default, all four ports are set to a resistive 50Ω impedance. Enter new reference impedances in the port boxes as desired.

Figure 14-35 Port Reference Impedance Dialog Box



This feature does not affect the impedance presented to the DUT by the system hardware. Rather, it mathematically transforms the measured data to show how the performance of a linear device would change in a non-50Ω measurement system. Impedance transforms can be specified either before or after measurement. Only the current data set is affected.

The reference impedance can only be specified as a resistive circuit.

Differential reference impedances are additive. For example, specifying a reference impedance of 125Ω on port 1 and 125Ω on port 3 corresponds to a differential-mode reference impedance of 250Ω on balanced port one. Likewise, the common-mode reference impedances are the parallel equivalent impedance.

Click **Set All Ports to 50 ohms (Resistive)** to return to the default configuration.

Click **OK** to accept the configuration and exit the dialog box or **Cancel** to ignore any changes and exit the dialog box.

The **Ref-Z** indicator on status bar means that the current data has been modified for a non-50Ω reference impedance for at least one port. See Figure 14-36.

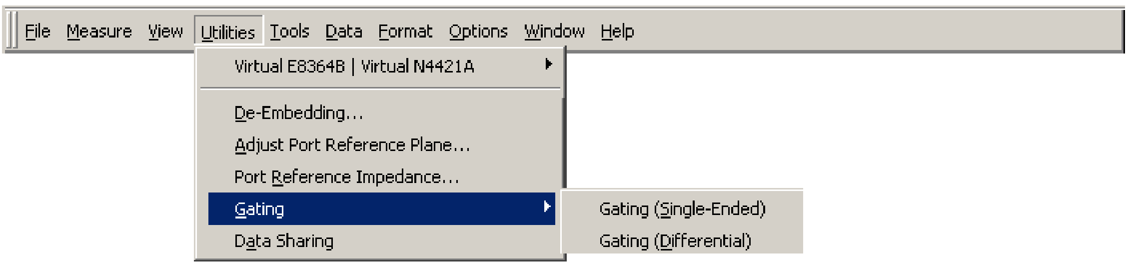
Figure 14-36 Port Reference Impedance (Ref-Z) Indicator on the Status Bar

Gating

Gating the time domain response provides the ability to mathematically remove the effect of a particular circuit element. 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. Refer to "Gating" on page 352 for additional information.

Click **Gating** from the **Utilities** menu. Then click either **Gating (Single Ended)** or **Gating (Differential)** depending on whether your Time Domain plot is single-ended or differential.

Figure 14-37 Utilities Menu with Gating Expanded



Gating (Single Ended)

Select **Gating (Single Ended)** when you are planning to gate a single-ended Time Domain plot.

Gating (Differential)

Select **Gating (Differential)** when you are planning to gate a differential Time Domain plot.

Data Sharing

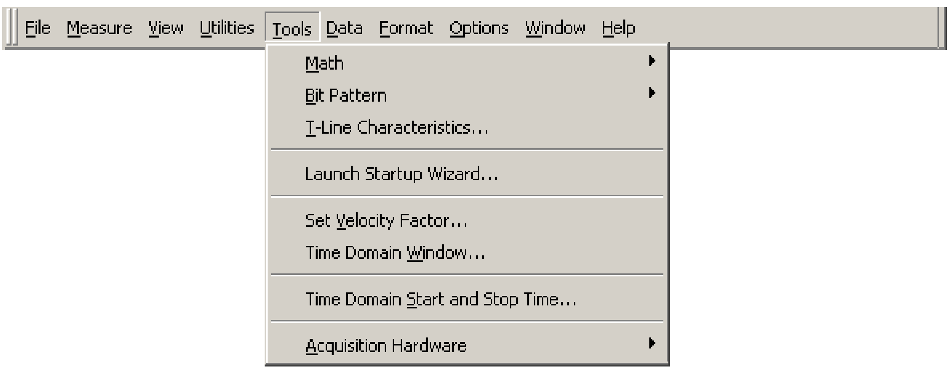
Data Sharing is used to overlay the plot of one measurement over the plot of another so that differences and similarities between the two plots can easily be viewed.

Refer to "Data Sharing" on page 387 for additional information.

Tools Menu

The **Tools** menu allows access to the Math, Bit Pattern, and T-Line Characteristics features. It also allows you to launch the startup wizard, set the velocity factor, change the time domain windowing, set the time domain start and stop times, and scan or select new data acquisition hardware.

Figure 14-38 Tools Menu



Math

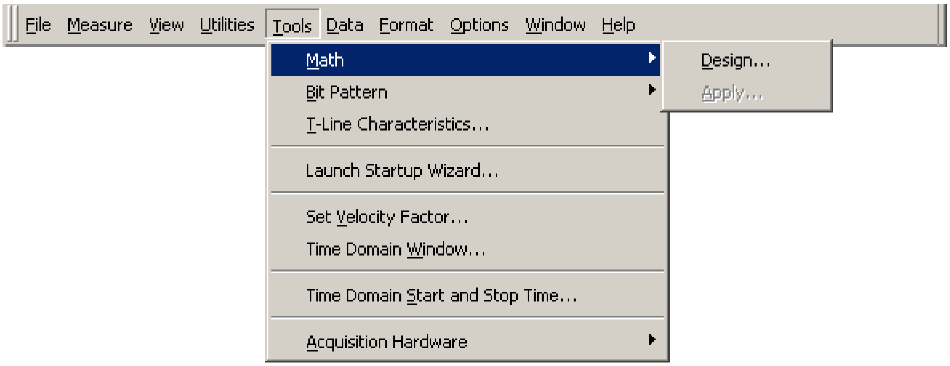
The **Math** feature allows you to design and save a mathematical formula and to apply that formula to compare measured parameters. For example, you could use the math feature to determine noise immunity on a balanced line by calculating the common mode rejection ratio (CMRR) using the following equation:

$$CMRR = (SDD21)/(SCC21)$$

You could also characterize the loss of a single-ended frequency domain measurement using the following equation:

$$Loss = \frac{1}{\sqrt{2}}(S21 + jS31)$$

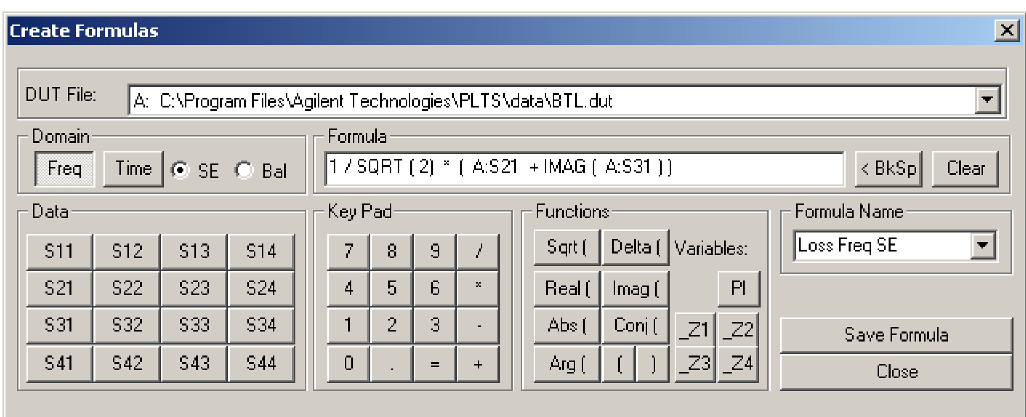
Figure 14-39 Tools Menu with Math Expanded



Design

To create a formula for use with the measured data, select **Design** from the **Math** choice of the **Tools** menu. Select the domain, either **Freq** or **Time** and either **SE** (single-ended), **Diff** (differential for time domain) or **Bal** (balanced for frequency domain) in the **Domain** area. Click the **Formula** box and begin entering the equation using the buttons in the **Data**, **Key Pad**, and **Functions** areas. Enter your equation from left to right. When you have finished, enter a name in the **Formula Name** box and click the **Save Formula** button. When you have finished inputting equations, click the **Close** button. For more details, refer to "Creating a Math Formula" on page 379.

Figure 14-40 Create Formulas Dialog Box

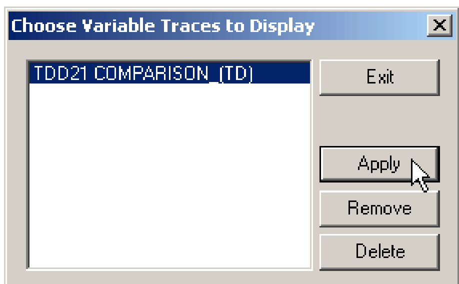


Apply

To apply a formula to the active data, display the *Choose Variable Traces to Display* dialog box by selecting **Apply** from the **Math** choice of the **Tools** menu. Then select the formula name from those displayed in the list. Click the **Apply** button to apply the formula to the active data and display a trace showing the data with the formula applied.

Remove removes the selected formula from the active data and removes the trace shown when **Apply** was clicked. **Delete** removes the selected variable from the list in the dialog box. **Exit** closes the dialog box.

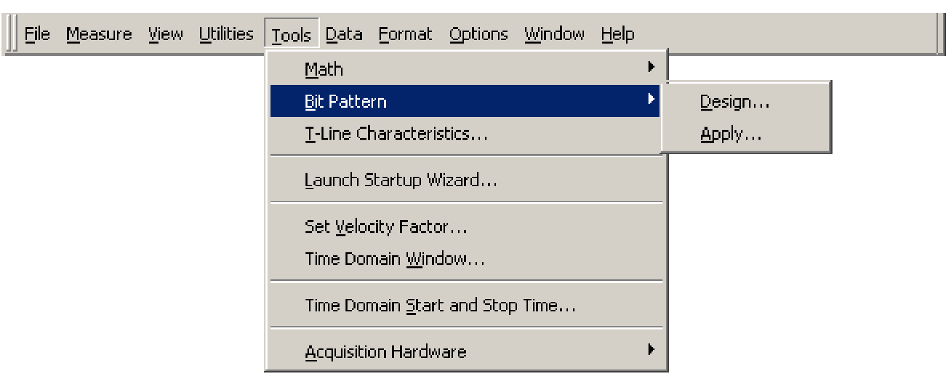
Figure 14-41 Choose Variable Traces to Display Dialog Box



Bit Pattern

The **Bit Pattern** feature allows you to design a digital pattern and save the pattern. Then you can apply the digital pattern to eye diagram plots.

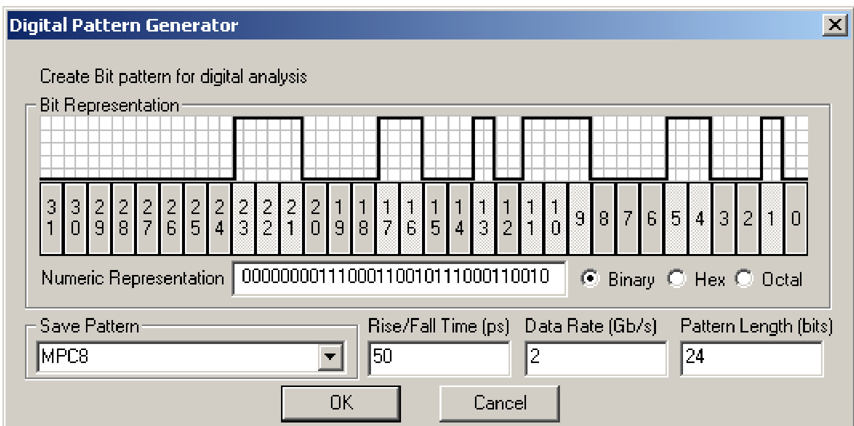
Figure 14-42 Tools Menu with Bit Pattern Expanded



Design

Select **Design** from the **Bit Pattern** choice in the **Tools** menu to design a digital pattern using the *Digital Pattern Generator*. The *Digital Pattern Generator* allows you to create a pattern of between 8 and 32 bits. You may create the pattern in one of two ways, either clicking the numbered keys (0 to 31) or by typing the numeric value in either Binary (base 2), Octal (base 8), or Hexadecimal (base 16) formats. As you enter the pattern inputs, the pattern is displayed in the upper portion of the **Bit Representation** area.

Figure 14-43 Digital Pattern Generator

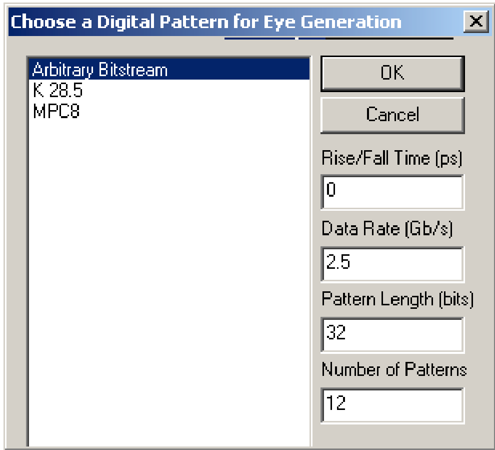


After the pattern has been created and is displayed correctly, save the pattern by entering a pattern name in the **Save Pattern** box and values in the **Rise/Fall Time (pS)**, **Data Rate (Gb/s)**, and **Pattern Length** boxes. Then select **OK** to save the digital pattern.

Apply

Apply a digital pattern to an eye diagram using the *Choose a Digital Pattern for Eye Generation* dialog box. This dialog box also allows you to change the values for the rise/fall time, data rate, pattern length, and number of patterns (arbitrary bitstream only). A digital pattern must be applied to view data using the eye diagram data analysis type.

Figure 14-44 Choose a Digital Pattern for Eye Generation Dialog Box



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 condition (or the time that it takes a signal to transition from a high to a low condition). Refer to Figure 9-9 on page 288 for additional information regarding transition time.
- **Data Rate (Gb/s)** is the speed that data is transferred over a circuit or a communications line. Refer to Figure 9-9 on page 288 for additional information regarding data rate.
- **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 number of bits allowed between 8 and 32. Where **B** is the number of bits entered, the number of unique bit patterns is: $2^B - 2$. If B = 32 (the maximum number of allowable bits), the then 4.29×10^9 unique bit patterns are generated.
- **Number of Patterns** (active only when **Arbitrary Bitstream** is selected) is used to indicate the number of unique bit patterns to use in creating the eye diagram. This value must be equal or less than the number of unique bit patterns determined by the pattern length entry.

Then, click **OK** to view the eye diagram in the plot window. **Cancel** closes the dialog box without making any changes.

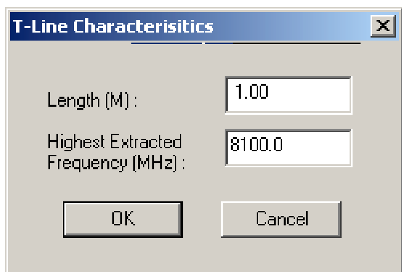
For additional information regarding Arbitrary Bitstream, refer to “Arbitrary Bitstream” on page 291.

NOTE You may change the digital pattern using the dialog box shown in Figure 14-44 by selecting **Bit Pattern** then **Apply**… from the **Tools** menu.

T-Line Characteristics

T-Line Characteristics displays the RLCG data analysis *T-Line Characteristics* dialog box. This is the same dialog box that is displayed when any RLCG analysis is selected. Enter the length of the transmission line (in meters) and the highest frequency (in megahertz) in this dialog box to change the existing transmission line characteristics.

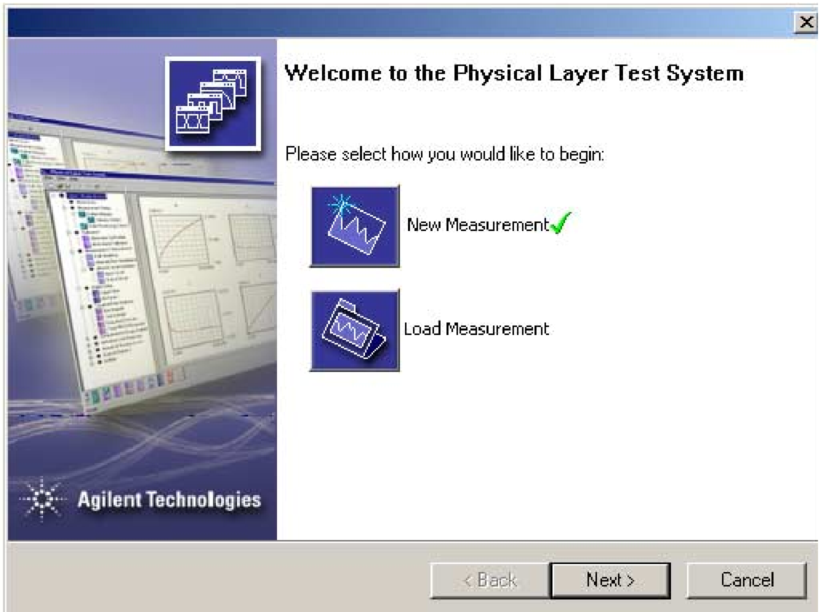
Figure 14-45 T-Line Characteristics



Launch Startup Wizard

When **Launch Startup Wizard** is selected, this menu choice starts the **Physical Layer Test System Startup Wizard**. See Figure 14-46.

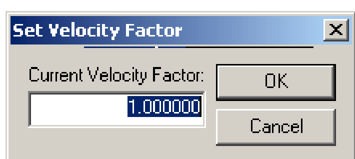
Figure 14-46 Startup Wizard



Set Velocity Factor

When **Set Velocity Factor** is selected from the **Tools** menu, the dialog box shown in Figure 14-47 is displayed. Enter the new velocity factor in the **Current Velocity Factor** box and click **OK**. The maximum allowable value is 1.000000.

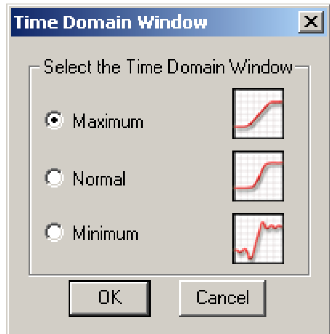
Figure 14-47 Set Velocity Factor Dialog Box



Time Domain Window...

When **Time Domain Window**... is selected from the **Tool** menu, the *Time Domain Window* dialog box is displayed (see Figure 14-48). This dialog box allows you to set the Time Domain Window setting to one of three levels, **Maximum** (the default value), **Normal**, and **Minimum**. See “Time Domain Windowing” on page 251 for additional information.

Figure 14-48 Time Domain Window Dialog Box



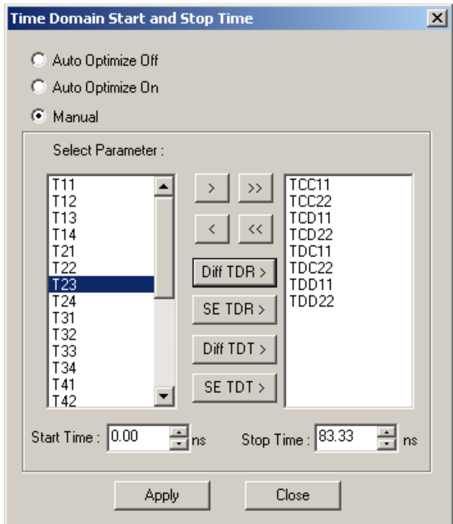
- **Maximum** gives the minimum sidelobes and this provides the greatest dynamic range. This is the default setting.
- **Normal** gives reduced sidelobes and is normally the most useful.
- **Minimum** is essentially no window and therefore give the highest sidelobes.

OK closes the dialog box. **Cancel** closes the dialog box without making changes.

Time Domain Start and Stop Time...

When **Time Domain Start and Stop Time**... is selected, you can change the start and stop time values for your time domain plots that were converted from frequency domain parameters (S-parameters). The *Time Domain Start and Stop Time* dialog box is opened. See Figure 14-49. You can have these plots displayed in their full time range, automatically optimized, or you may change the start and stop time values manually. Refer to "Optimizing the Time Domain Time Scale for Viewing" on page 267 for additional information.

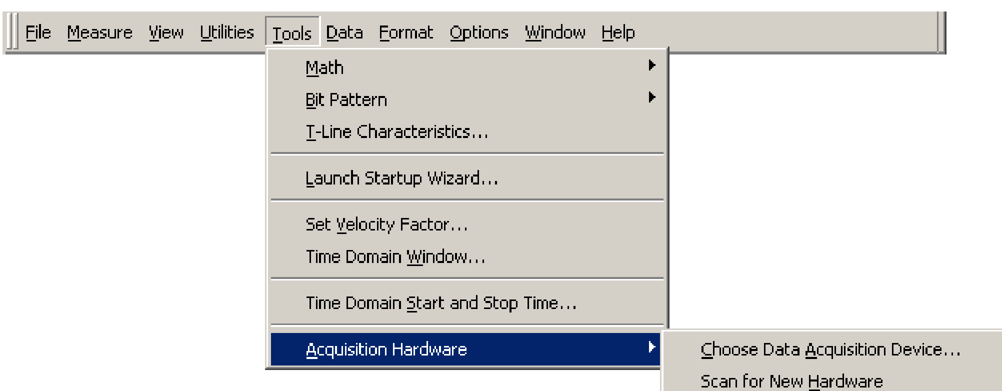
Figure 14-49 Time Domain Start and Stop Time Dialog Box



Acquisition Hardware

Acquisition Hardware allows you to select **Choose Data Acquisition Device**… or **Scan for New Hardware**.

Figure 14-50 Tools Menu with Acquisition Hardware Expanded

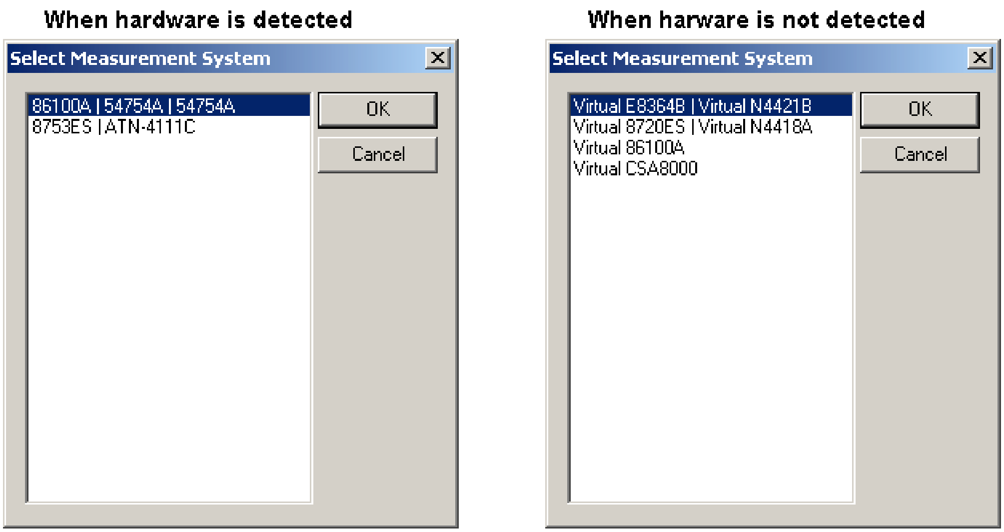


Choose Data Acquisition Device…

The **Choose Data Acquisition Device**… selection displays all of the hardware that is

detected in the *Select Measurement System* dialog box. See Figure 14-51. For example, the left dialog box shown in shows both an 86100A TDR-based PLTS, as well as an 8753ES VNA-based PLTS. When no hardware is detected, the software displays virtual PLTS hardware systems (two VNA-based and two TDR-based systems). A virtual PLTS hardware system allows you to use the software just like you would while using actual PLTS hardware with the exception of making a measurement.

Figure 14-51 Select Measurement System Dialog Box



Select your hardware and click the **OK** button to change your hardware.

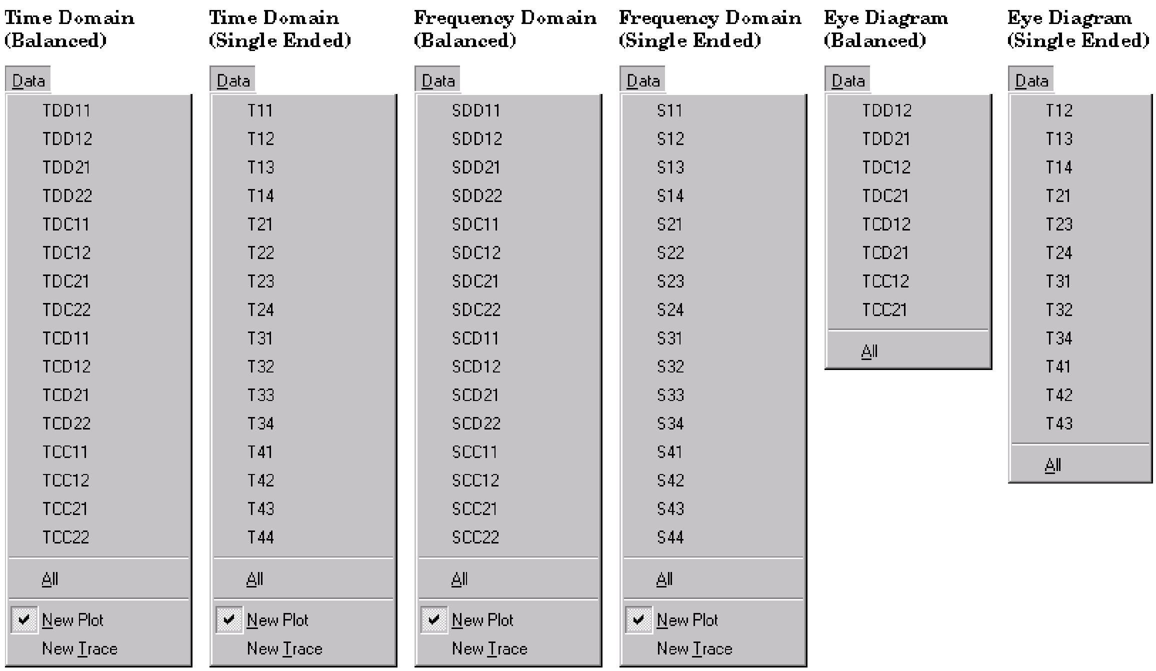
Scan for New Hardware

The **Scan for New Hardware** selection causes the PLTS software to check the GPIB bus for connected hardware. After the bus has been scanned, the *Select Measurement System* dialog box that is shown in Figure 14-51 is displayed.

Data Menu

The **Data** menu displays each individual parameter for each specific data analysis type as well as the capability to display all of the parameters. The time domain and frequency domain data analysis types also have the option to allow you to display multiple plots on the same plot or separate plots. Figure 14-52 shows each of the data menus and their selections. See also "Parameter Bar" on page 434.

Figure 14-52 Data Menu



Individual Parameter Selections

The individual parameter selections are based on the specific data analysis type. The following lists each data analysis type and its associated parameters:

- **Time Domain (Differential)**

  TDD11, TDD12, TDD21, TDD22, TDC11, TDC12, TDC21, TDC22, TCD11, TCD12, TCD21, TCD22, TCC11, TCC12, TCC21, TCC22

- **Time Domain (Single Ended)**

  T11, T12, T13, T14, T21, T22, T23, T24, T31, T32, T33, T34, T41, T42, T43, T44

- **Frequency Domain (Balanced)**

  SDD11, SDD12, SDD21, SDD22, SDC11, SDC12, SDC21, SDC22, SCD11, SCD12, SCD21, SCD22, SCC11, SCC12, SCC21, SCC22

- **Frequency Domain (Single Ended)**

  S11, S12, S13, S14, S21, S22, S23, S24, S31, S32, S33, S34, S41, S42, S43, S44

- **Eye Diagram (Differential)**

  TDD12, TDD21, TDC12, TDC21, TCD12, TCD21, TCC12, TCC21

- **Eye Diagram (Single Ended)**

  T12, T13, T14, T21, T23, T24, T31, T32, T34, T41, T42, T43

All

Selecting **All** displays:

- All 16 of the time domain parameters if the active plot window is a time domain window.
  - If **New Plot** is chosen, selecting **All** displays all 16 of the parameters on individual plots in one window.
  - If **New Trace** is chosen, selecting **All** displays all 16 of the parameters on a single plot.
- All 16 of the frequency domain parameters if the active plot window is a frequency domain window.
  - If **New Plot** is chosen, selecting **All** displays all 16 of the parameters on individual plots in one window.
  - If **New Trace** is chosen, selecting **All** displays all 16 of the parameters on a single plot.
- 4 of the eye diagram parameters if the active plot window is a differential eye diagram window.
- All 12 of the eye diagram parameters if the active plot window is a single-ended eye diagram window.

New Plot

When **New Plot** is chosen, selecting any of the time or frequency domain parameters will display a new plot with that parameter within the active plots window. If **New Plot** is chosen, selecting **All** displays all 16 of the time or frequency domain parameters on individual plots in one window.

New Trace

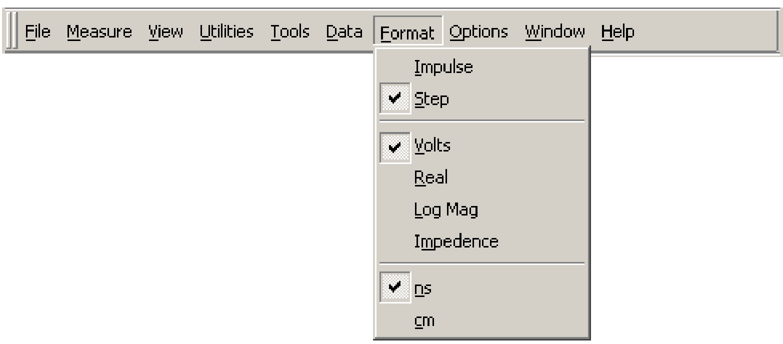
When **New Trace** is chosen, selecting any of the time or frequency domain parameters will display a new trace within the active plot. If **New Trace** is chosen, selecting **All** displays all 16 of the time or frequency domain parameters on a single plot.

Format Menu

There are two versions of the **Format** menu, one for active displays in the time domain and another for active displays in the frequency mode. The **Format** menu is not displayed for displays in other modes. See also "Format Bar" on page 436.

Time Domain Format Menu

Figure 14-53 Format Menu for Time Domain Measurements



Impulse Sets the active time domain plot to show the response with an impulse stimulus.

Step Sets the active time domain plot to show the response with a step-voltage stimulus. This is the default setting.

Volts Sets the active time domain plot's vertical axis to Volts mode. This is the default setting.

Real Sets the active time domain plot's vertical axis to Real mode.

Log Mag Sets the active time domain plot's vertical axis to Log Mag mode.

Impedance Sets the active time domain plot's vertical axis to impedance mode. Active only for reflection plots with a step stimulus.

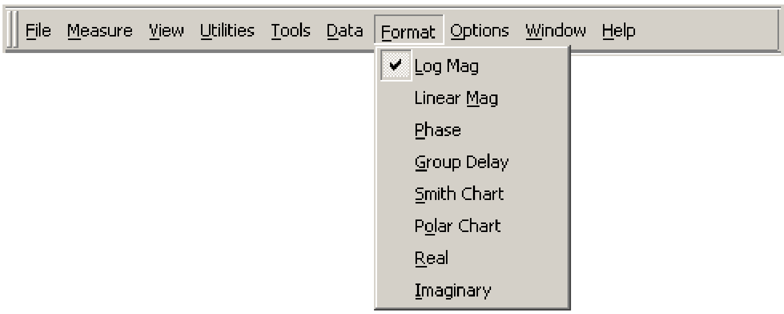
ns Sets the active time domain plot's horizontal axis to nanoseconds (ns). This is the default setting.

cm Sets the active time domain plot's horizontal axis to centimeters (cm).

Refer to "Selecting Time Domain Display Formats" on page 272 for more information.

Frequency Domain Format Menu

Figure 14-54 Format Menu for Frequency Domain Measurements



Log Mag Displays the active frequency domain plot in Log Magnitude format. This is the default setting.

Linear Mag Displays the active frequency domain plot in Linear Magnitude format.

Phase Displays the active frequency domain plot in Phase format.

Group Delay Displays the active frequency domain plot in Group Delay format.

Smith Chart Displays the active frequency domain plot in Smith Chart format.

Polar Chart Displays the active frequency domain plot in Polar Chart format.

Real Displays the active frequency domain plot in Real format.

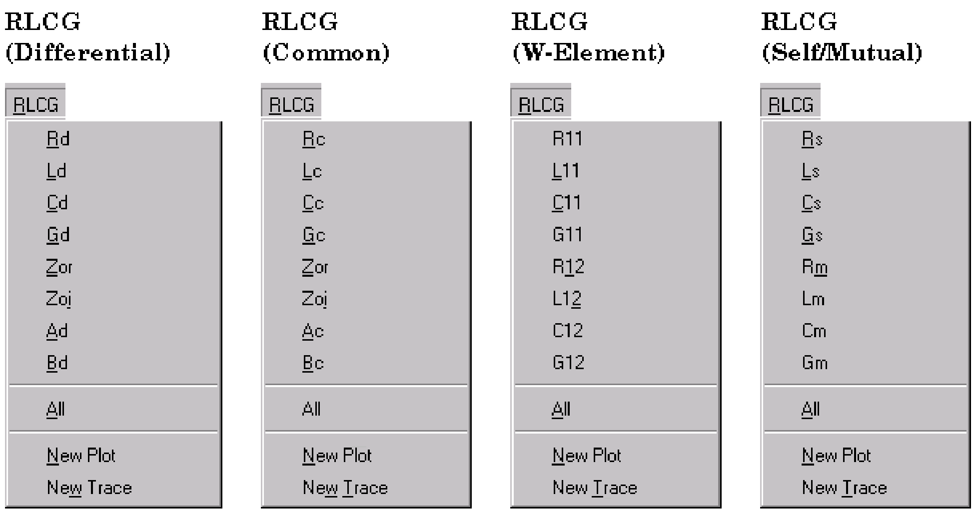
Imaginary Displays the active frequency domain plot in Imaginary format.

Refer to “Selecting Frequency Domain Display Formats” on page 233 for more information.

RLCG Menu

When any of the RLCG data analysis types are selected, the **RLCG** menu is displayed. Each of the four RLCG data analysis types has its own menu. Refer to Figure 14-55.

Figure 14-55 RLCG Menus



Individual Parameter Selections

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	

All

If **New Plot** is chosen, selecting **All** displays all eight parameters on individual plots in one window. If **New Trace** is chosen, selecting **All** displays all eight parameters on a single plot.

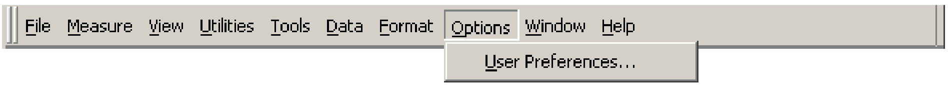
New Plot

When **New Plot** is chosen, selecting any of the RLCG parameters will display a new plot with that parameter within the active plots window. If **New Plot** is chosen, selecting **All** displays all eight parameters on individual plots in one window.

Options Menu

The **Options** menu provides access to the *User Preferences* dialog box

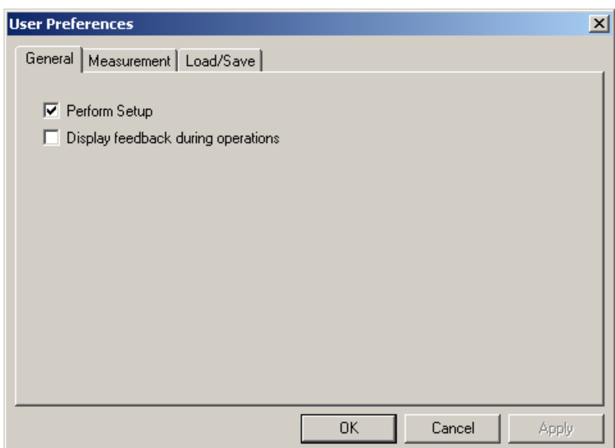
Figure 14-56 Options Menu



User Preferences

The *User Preferences* dialog box allows you to customize your Physical Layer Test System software. This dialog box has the following three tabs to choose from: **General**, **Measurement**, and **Load/Save**.

Figure 14-57 User Preferences Dialog Box



General

The **General** tab (see Figure 14-57) has the following choices:

- **Perform Setup** - when selected, the PLTS software, when it is next started, will operate as if it has just been installed and go through a complete setup routine. At the conclusion of this routine, the **Perform Setup** option is automatically turned off.
- **Display feedback during operations** - controls the display of informational and

progress windows for various operations, such as the file conversion summary window and the measurement post-processing progress window. When selected, these windows are displayed; otherwise, these windows are hidden. This option's default setting is ON.

Measurement

The **Measurement** tab (see Figure 14-57) has the following choices:

- **Beep when measurement is complete -** when checked, the PC emits a beep to indicate that the measurement is complete.
- **Show warning before uncalibrated measurements -** when checked, a message is displayed confirming that you are making an uncalibrated measurement.
- **Always optimize the Start and Stop Times when calculating the time domain data from frequency domain data -** when checked, the time domain start and stop times are optimized for best viewing. This selection affects the *Time Domain Start and Stop Time* dialog box default optimize selection. See "Automated Start and Stop Settings In Time Domain" on page 257.

Load/Save

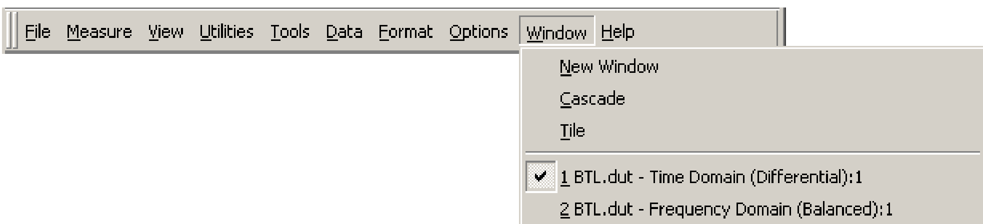
The **Load/Save** tab (see Figure 14-57) has the following choices:

- **Save data in BMS Format -** when checked, data is saved in Balanced Measurement System format.

Window Menu

The **Window** menu provides an efficient method of working with analysis windows. It gives you the ability to add new analysis windows, arrange open analysis windows for optimum viewing, arrange the minimized analysis window icons, and select/display individual analysis windows.

Figure 14-58 Window Menu



New Window

New Window adds a new analysis window to the display area. The new window will display the analysis type that is currently selected.

Cascade

Cascade arranges all of the open plot windows to optimum and equal size with the active window arranged on top so that is fully displayed. Any window may be accessed by a single click.

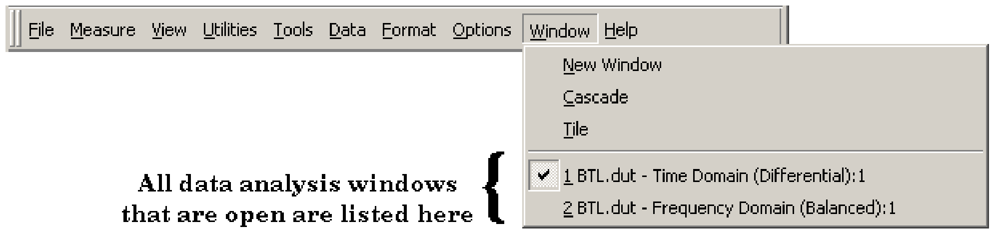
Tile

Tile displays all open plot windows completely in the plot window area. Each plot window is reduced in size to accommodate new plot windows.

List of Open Analysis Windows

The **Window** menu also lists the open analysis windows below the standard **Window** menu selections. Each open analysis window is listed in the order they were opened with the most recent window listed at the bottom. The active analysis window is shown with a check mark displayed at its left side. Selecting a window from the list displays the window in the display area.

Figure 14-59 List of Open Data Analysis Windows



Help Menu

The **Help** menu provides access to information about the software.

Figure 14-60 Help Menu

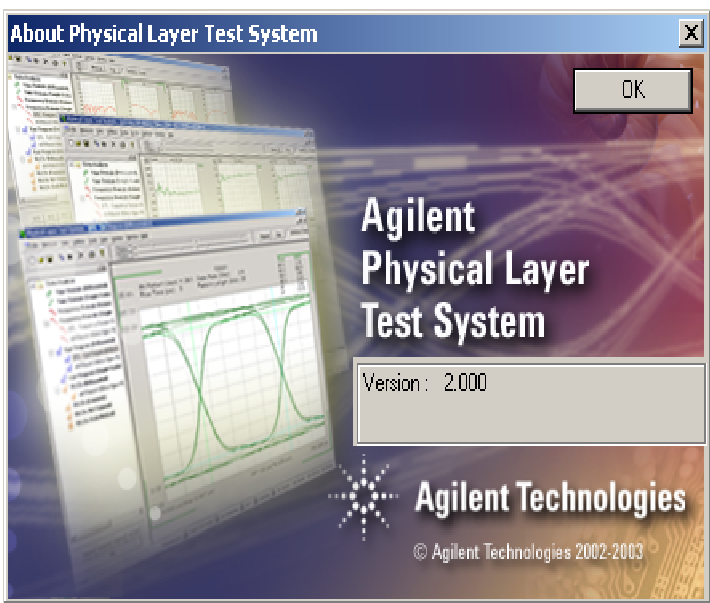
Help…

The **Help**… selection displays the user's guide (this manual) in pdf format.

About PLTS…

The *About PLTS*… window displays the software version information.

Figure 14-61 About PLTS… Window



15 Specifications and Characteristics

Definitions

All specifications and characteristics apply over a 25 °C ±5 °C range (unless otherwise stated) and 90 minutes after the instrument has been turned on.

Specification (spec.)	Warranted performance. Specifications include guard bands to account for the expected statistical performance distribution, measurement uncertainties, and changes in performance due to environmental conditions.
Characteristic (char.)	A performance parameter that the product is expected to meet before it leaves the factory, but that is not verified in the field and is not covered by the product warranty. A characteristic includes the same guard bands as a specification.
Typical (typ.)	Expected performance of an average unit which does not include guard bands. It is not covered by the product warranty.
Nominal (nom.)	A general, descriptive term that does not imply a level of performance. It is not covered by the product warranty.
Calibration	The process of measuring known standards to characterize the system's systematic (repeatable) errors.
Corrected (residual)	Indicates performance after error correction (calibration). It is determined by the quality of calibration standards and how well “known” they are, plus system repeatability, stability, and noise.
Uncorrected (raw)	Indicates instrument performance without error correction. The uncorrected performance affects the stability of a calibration.

N1947A and N1948A Electrical Specifications and Characteristics

The following specifications are applicable for a system in the following configurations:

Network Analyzer:	Agilent E8803A Option 014 (N1947A System)
	Agilent E8358A Option 015 (N1948A System)
Test Set:	Agilent N4417A Option 103 (N1947A System)
	Agilent N4417A Option 104 (N1948A System)
Calibration Kit:	Agilent 85052C Precision 3.5 mm
Test Port Cables:	Agilent N4417A Option B20
Calibration Technique:	Four-Port SOLT

System Dynamic Range

The test port transmission measurements are valid at 10 Hz IF bandwidth with four-port error correction and +10 dBm maximum output power. The dynamic range is the difference between the rms noise floor and the maximum output power.

Table 15-1 System Dynamic Range

Frequency Range	Specification
300 kHz to 1.3 GHz	120 dB[a]
1.3 GHz to 3.0 GHz	120 dB
3.0 GHz to 6.0 GHz	108 dB
6.0 GHz to 9.0 GHz	103 dB

a. May be limited to 100 dB at particular frequencies below 750 MHz due to spurious receiver residuals.

Measurement Port

Residual uncertainties for corrected data using four-port error correction. These apply for 25 °C with less than 1 °C variation from calibration.

Table 15-2 Measurement Port Characteristics

Description	Characteristic			
	300 kHz to 1.3 GHz	1.3 GHz to 3.0 GHz	3.0 GHz to 6.0 GHz	6.0 GHz to 9.0 GHz
Directivity	50 dB	47 dB	42 dB	40 dB
Source Match	42 dB	42 dB	38 dB	35 dB
Load Match	50 dB	47 dB	42 dB	40 dB
Reflection Tracking	± 0.006 dB	± 0.007 dB	± 0.009 dB	± 0.015 dB
Transmission Tracking	± 0.012 dB	± 0.015 dB	± 0.040 dB	± 0.060 dB

Measurement Uncertainties

The following graphics show the worst case transmission and reflection magnitude and phase uncertainty for the N1947A and N1948A systems.

Figure 15-1 3.5 mm Transmission Magnitude and Phase Uncertainty

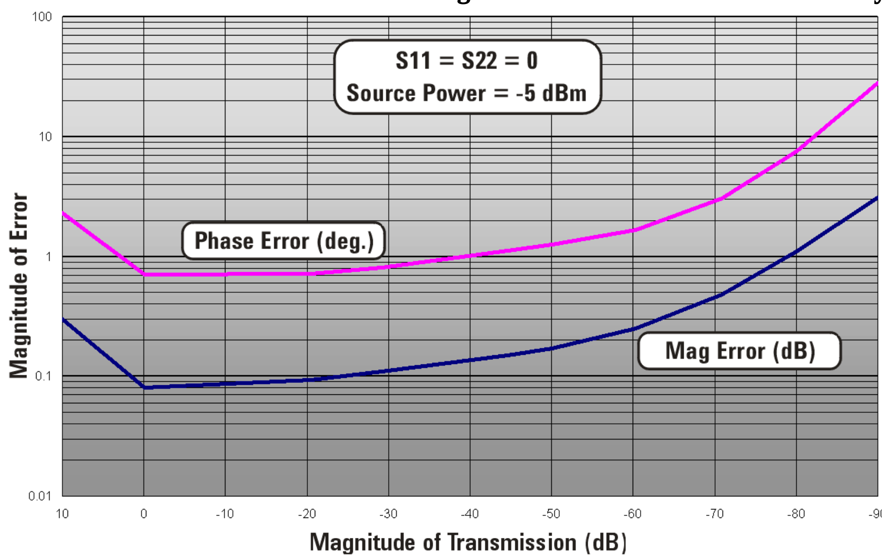
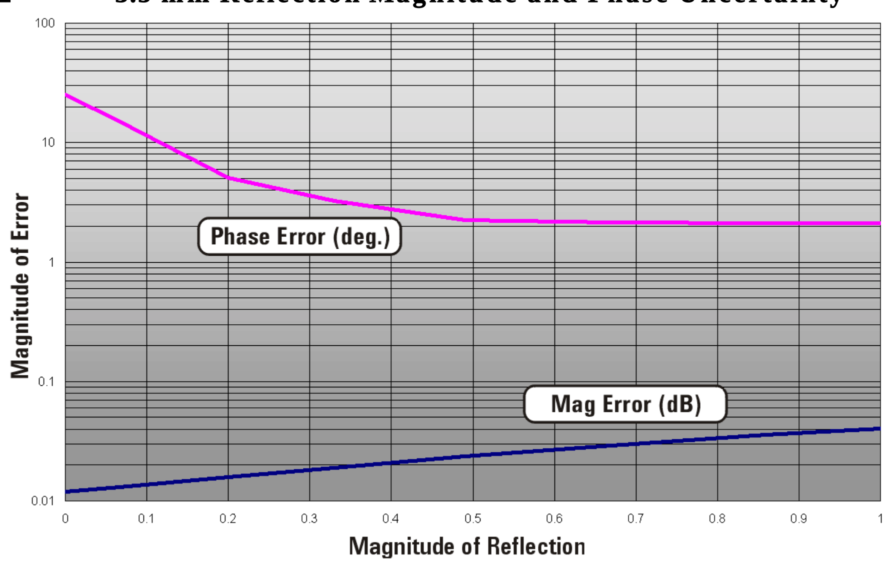


Figure 15-2 3.5 mm Reflection Magnitude and Phase Uncertainty



Test Set Performance

Table 15-3 Test Set Performance

Description	Specification	Supplemental Information
Frequency Range	300 kHz to 9.0 GHz	
Impedance		50 Ohms (nom.)
Insertion Loss		
Source Out to Coupler In	4.5 dB maximum	
Port 2 to A In and Port 4 to B In	8.5 dB maximum	
A In to A Out and B In to B Out	8.0 dB maximum	
Isolation (port to port and A to B)	≥ 105 dB	
Maximum Operating Level	+20 dBm	
Damage Level		+30 dBm (typ.)
Test Port Connectors		50 Ohms (nom.) Type-N Connectors

Power Supply

The power supply requirements for the test sets are listed below.

Table 15-4 Test Set Power Supply Specifications

Description	Specification
Input Voltage Range	100 to 120 Volts 220 to 250 Volts
Frequency Range	47 to 62 Hertz
Power	40 VA

N1951A Electrical Specifications and Characteristics

The following specifications are applicable for a system in the following configurations:

Network Analyzer:	Agilent 8720ES Option H32
Test Set:	Agilent N4418A
Calibration Kit:	Agilent 85052C Precision 3.5 mm
Test Port Cables:	Agilent N4418A Option B20
Calibration Technique:	Four-Port SOLT

System Dynamic Range

The test port transmission measurements are valid at 10 Hz IF bandwidth with four-port error correction and +5 dBm maximum output power. The dynamic range is the difference between the rms noise floor and the maximum output power.

Table 15-5 System Dynamic Range

Frequency Range	Specification
50 MHz to 840 MHz	77 dB
840 MHz to 20.0 GHz	90 dB

Measurement Port

Residual uncertainties for corrected data using four-port error correction. These apply for 25 °C with less than 1 °C variation from calibration.

Table 15-6 Measurement Port Characteristics

Description	Characteristic		
	50 MHz to 2.0 GHz	2.0 GHz to 8.0 GHz	8.0 GHz to 20.0 GHz
Directivity	48 dB	48 dB	43 dB
Source Match	41 dB	41 dB	38 dB
Load Match	48 dB	48 dB	43 dB
Reflection Tracking	± 0.005 dB	± 0.005 dB	± 0.008 dB
Transmission Tracking	± 0.014 dB	± 0.014 dB	± 0.035 dB

Measurement Uncertainties

The following graphics show the worst case transmission and reflection magnitude and phase uncertainty for the N1951A system.

Figure 15-3 3.5 mm Transmission Magnitude and Phase Uncertainty

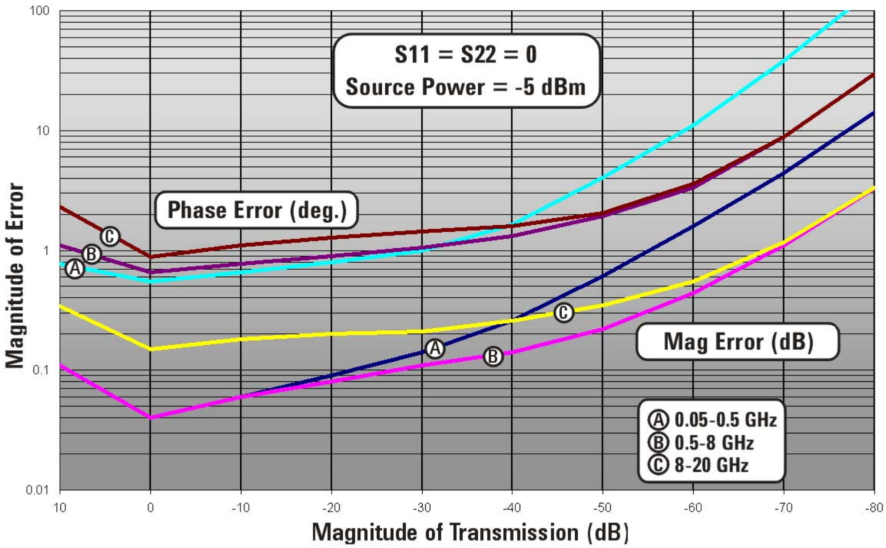
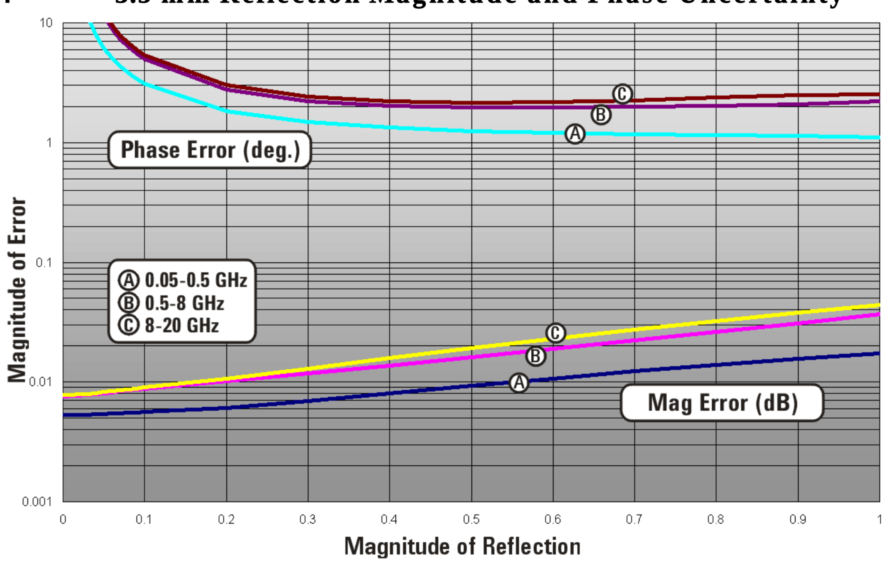


Figure 15-4 3.5 mm Reflection Magnitude and Phase Uncertainty



Test Set Performance

Table 15-7 Test Set Performance

Description	Specification	Supplemental Information
Frequency Range	50 MHz to 20.0 GHz	
Impedance		50 Ohms (nom.)
Insertion Loss	8 to 10 dB	
Isolation (port to port)	≥ 85 dB	
Maximum Operating Level	+20 dBm	
Damage Level		+30 dBm (typ.)
DC Bias Range (Option UNK only)		40 VDC, 500 mA
Test Port Connectors		50 Ohms (nom.) 3.5 mm (m) Connectors

Power Supply

The power supply requirements for the test sets are listed below.

Table 15-8 Test Set Power Supply Specifications

Description	Specification
Input Voltage Range	100 to 120 Volts 220 to 250 Volts
Frequency Range	47 to 62 Hertz
Power	40 VA

N1953A/B Electrical Specifications and Characteristics

The following specifications are applicable for a system in the following configurations:

System:	N1953A	N1953B
Network Analyzer:	Agilent E8362A Options 014 and UNL	Agilent E8362B Options 014 and UNL
Test Set:	Agilent N4419A	Agilent N4419B
Calibration Kit:	Agilent 85052D 3.5 mm	Agilent 85052D 3.5 mm
Test Port Cables:	Agilent N4419A Option B20	Agilent N4419B Option B20
Calibration Technique:	Four-Port SOLT	Four-Port SOLT

System Dynamic Range

The test port transmission measurements are valid at 10 Hz IF bandwidth with four-port error correction and –5 dBm default maximum output power. The dynamic range is the difference between rms noise floor and the output power.

Table 15-9 System Dynamic Range

Frequency Range	Specification	Supplemental Information
10 MHz to 45 MHz[a]		60 dB (char.)
45 MHz to 500 MHz	70 dB	
500 MHz to 2.0 GHz	100 dB	
2.0 GHz to 10.0 GHz	100 dB	
10.0 GHz to 20.0 GHz	85 dB	

a. This frequency range is not applicable to the N1957A.

Measurement Port

Residual uncertainties for corrected data using four-port error correction. These apply for 25 °C with less than 1 °C variation from calibration.

Table 15-10 Measurement Port Characteristics

Description	Characteristic		
	45 MHz to 2.0 GHz	**2.0 GHz to 10.0 GHz**	**10.0 GHz to 20.0 GHz**
Directivity	56 dB	42 dB	40 dB
Source Match	42 dB	36 dB	33 dB
Load Match	56 dB	42 dB	40 dB
Reflection Tracking	± 0.0025 dB	± 0.009 dB	± 0.013 dB
Transmission Tracking	± 0.020 dB	± 0.032 dB	± 0.050 dB

Measurement Uncertainties

The following graphics show the worst case transmission and reflection magnitude and phase uncertainty for the system.

Figure 15-5 3.5 mm Transmission Magnitude and Phase Uncertainty

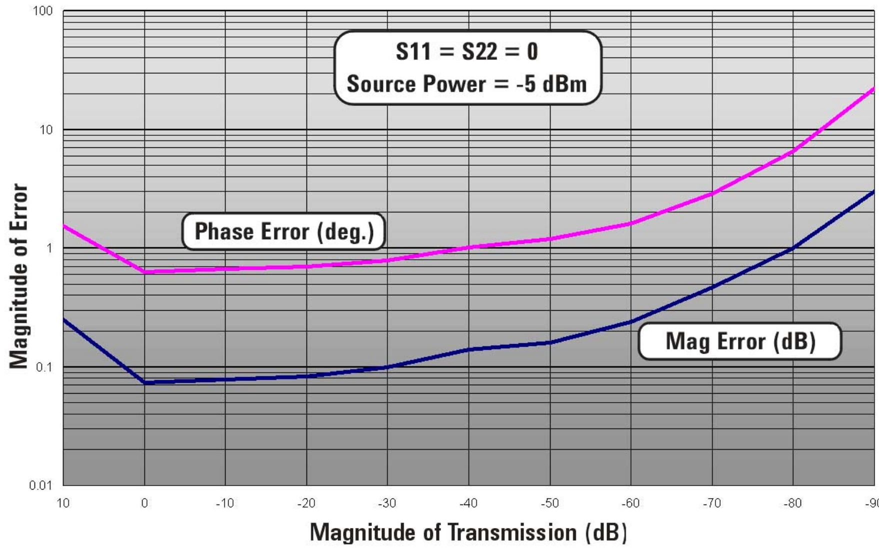
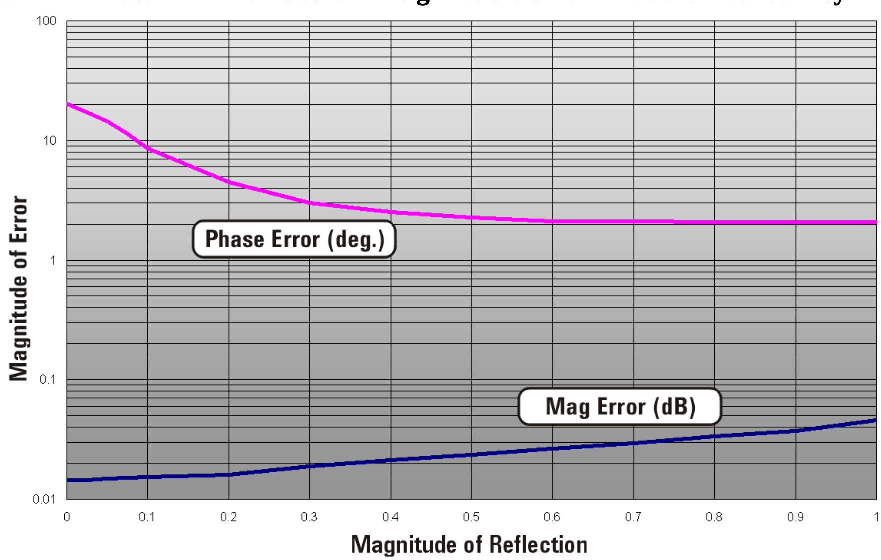


Figure 15-6 3.5 mm Reflection Magnitude and Phase Uncertainty



Test Set Performance

Table 15-11 Test Set Performance

Description	Characteristics	Supplemental Information
Frequency Range	45 MHz to 20.0 GHz [a] 10 MHz to 20.0 GHz[b]	
Impedance		50 Ohms (nom.)
Insertion Loss		
Source Out to Coupler Thru	5.0 dB maximum	
Port 2 to Rcvr A In and Port 4 to Rcvr B In		
45 MHz to 1.0 GHz		18 to 45 dB (typical)
1.0 GHz to 20.0 GHz		18 to 25 dB (typical)
Rcvr A In to Cplr Arm and Rcvr B In to Cplr Arm	8.0 dB maximum	
Isolation (port to port)		
45 MHz to 200 MHz	≥ 70 dB	
200 MHz to 20 GHz	≥ 90 dB	
Maximum Operating Level	+20 dBm	
Damage Level		+30 dBm (typical)
Test Port Connectors		50 Ohms (nom.) 3.5 mm Connectors

a. N1957A only.
b. N1957B only.

Power Supply

The power supply requirements for the test sets are listed below.

Table 15-12 Test Set Power Supply Specifications

Description	Specification
Input Voltage Range	100 to 240 Volts
Frequency Range	47 to 63 Hertz
Power	40 VA

N1955B Electrical Specifications and Characteristics

The following specifications are applicable for a system in the following configurations:

Network Analyzer:	Agilent E8363B Options 014 and UNL
Test Set:	Agilent N4420B
Calibration Kit:	Agilent 85056A 2.4 mm
Test Cables:	Agilent N4420B Option B20
Calibration Technique:	Four-Port SOLT

System Dynamic Range

The test port transmission measurements are valid at 10 Hz IF bandwidth with four-port error correction and –17 dBm default maximum output power. The dynamic range is the difference between rms noise floor and the output power.

Table 15-13 System Dynamic Range

Frequency Range	Specification	Supplemental Information
10 MHz to 45 MHz		55 dB (char.)
45 MHz to 500 MHz	55 dB	
500 MHz to 10.0 GHz	70 dB	
10.0 GHz to 20.0 GHz	70 dB	
20.0 GHz to 40.0 GHz	55 dB	

Measurement Port

Residual uncertainties for corrected data using four-port error correction. These apply for 25 °C with less than 1 °C variation from calibration.

Table 15-14 Measurement Port Characteristics

Description	Characteristic			
	45 MHz to 0.5 GHz	0.5 GHz to 10.0 GHz	10.0 GHz to 20.0 GHz	20.0 GHz to 40.0 GHz
Directivity	43 dB	39.5 dB	39 dB	33 dB
Source Match	38 dB	34 dB	34 dB	27 dB
Load Match	43 dB	39.5 dB	39 dB	33 dB
Reflection Tracking	± 0.001 dB	± 0.002 dB	± 0.008 dB	± 0.026 dB
Transmission Tracking	± 0.015 dB	± 0.020 dB	± 0.040 dB	± 0.20 dB

Measurement Uncertainties

The following graphics show the worst case transmission and reflection magnitude and phase uncertainty for the system.

Figure 15-7 2.4 mm Transmission Magnitude and Phase Uncertainty

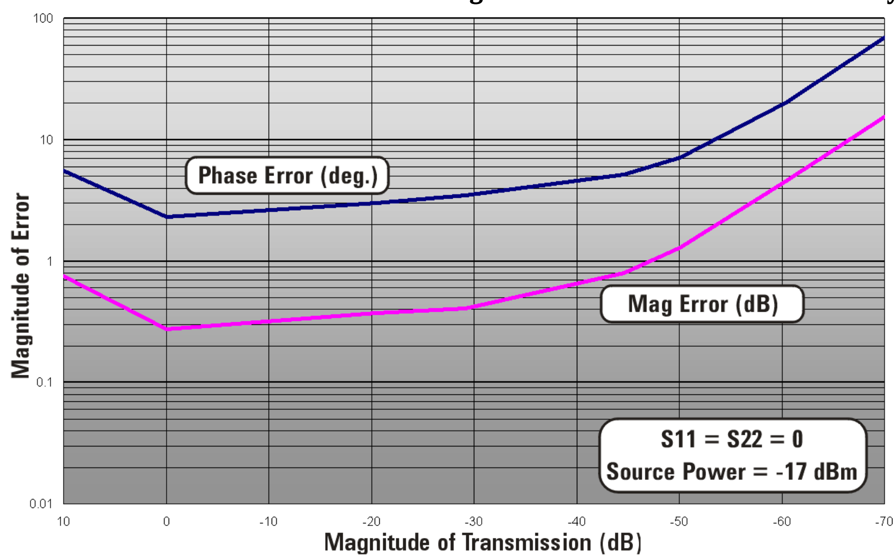
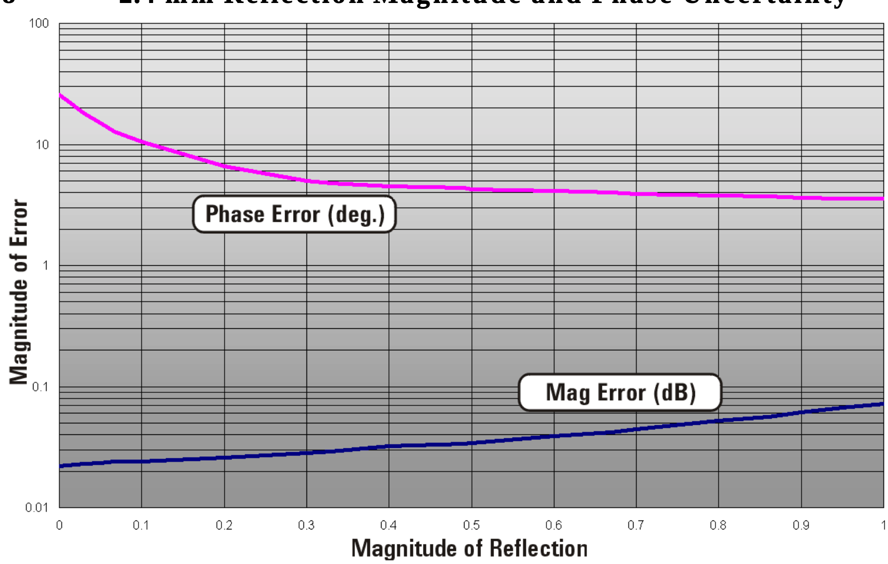


Figure 15-8 2.4 mm Reflection Magnitude and Phase Uncertainty



Test Set Performance

Table 15-15 Test Set Performance

Description	Characteristics	Supplemental Information
Frequency Range	10 MHz to 40.0 GHz	
Transition Time (10 to 90%, TR=.72/BW)	18 ps	
Impedance		50 Ohms (nom.)
Insertion Loss		
Source Out to Coupler Thru	12.0 dB maximum	
Port 2 to Rcvr A In and Port 4 to Rcvr B In		
45 MHz to 1.0 GHz		18 to 45 dB (typical)
1.0 GHz to 40.0 GHz		16 to 26 dB (typical)
Rcvr A In to Cplr Arm and Rcvr B In to Cplr Arm	15.0 dB maximum	
Isolation (port to port)		
45 MHz to 200 MHz	≥ 70 dB	
200 MHz to 40 GHz	≥ 90 dB	
Maximum Operating Level	+20 dBm	
Damage Level		+30 dBm (typical)
Test Port Connectors		50 Ohms (nom.) 2.4 mm (m) Connectors

Power Supply

The power supply requirements for the test sets are listed below.

Table 15-16 Test Set Power Supply Specifications

Description	**Specification**
Input Voltage Range	100 to 240 Volts
Frequency Range	47 to 63 Hertz
Power	40 VA

N1957A/B Electrical Specifications and Characteristics

The following specifications are applicable for the systems in the following configurations:

System:	N1957A	N1957B
Network Analyzer:	Agilent E8364A Options 014 and UNL	Agilent E8364B Options 014 and UNL
Test Set:	Agilent N4421A	Agilent N4421B
Calibration Kit:	Agilent 85056A 2.4 mm	Agilent 85056A 2.4 mm
Test Cables:	Agilent N4421A Option B20	Agilent N4421B Option B20
Calibration Technique:	Four-Port SOLT	Four-Port SOLT

System Dynamic Range

The test port transmission measurements are valid at 10 Hz IF bandwidth with four-port error correction and –17 dBm default maximum output power. The dynamic range is the difference between rms noise floor and the output power.

Table 15-17 System Dynamic Range

Frequency Range	Specification	Supplemental Information
10 MHz to 45 MHz[a]		55 dB (char.)
45 MHz to 500 MHz	55 dB	
500 MHz to 10.0 GHz	70 dB	
10.0 GHz to 20.0 GHz	70 dB	
20.0 GHz to 50.0 GHz	55 dB	

a. This frequency range is not applicable to the N1957A.

Measurement Port

Residual uncertainties for corrected data using four-port error correction. These apply for 25 °C with less than 1 °C variation from calibration.

Table 15-18 Measurement Port Characteristics

Description	Characteristic			
	45 MHz to 0.5 GHz	**0.5 GHz to 10.0 GHz**	**10.0 GHz to 20.0 GHz**	**20.0 GHz to 50.0 GHz**
Directivity	43 dB	39.5 dB	39 dB	33 dB
Source Match	38 dB	34 dB	34 dB	27 dB
Load Match	43 dB	39.5 dB	39 dB	33 dB
Reflection Tracking	± 0.001 dB	± 0.002 dB	± 0.008 dB	± 0.026 dB
Transmission Tracking	± 0.015 dB	± 0.020 dB	± 0.040 dB	± 0.20 dB

Measurement Uncertainties

The following graphics show the worst case transmission and reflection magnitude and phase uncertainty for the system.

Figure 15-9 2.4 mm Transmission Magnitude and Phase Uncertainty

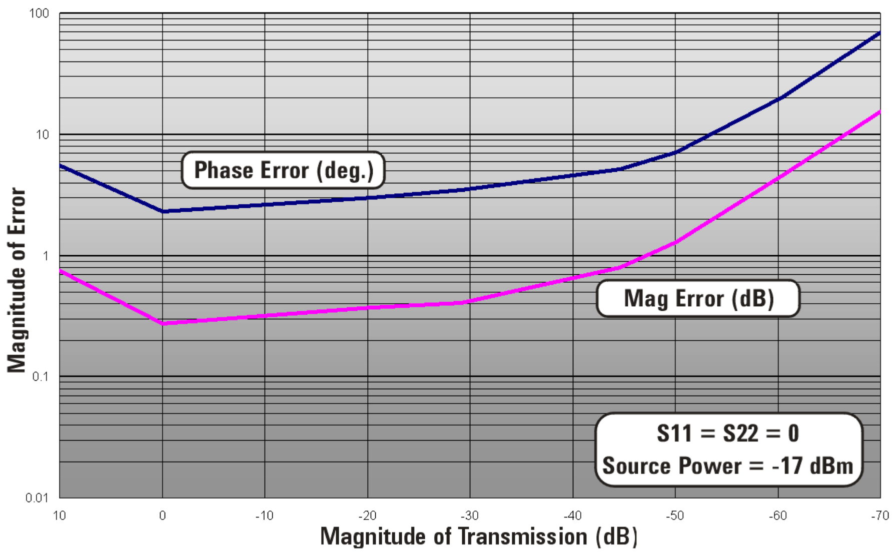
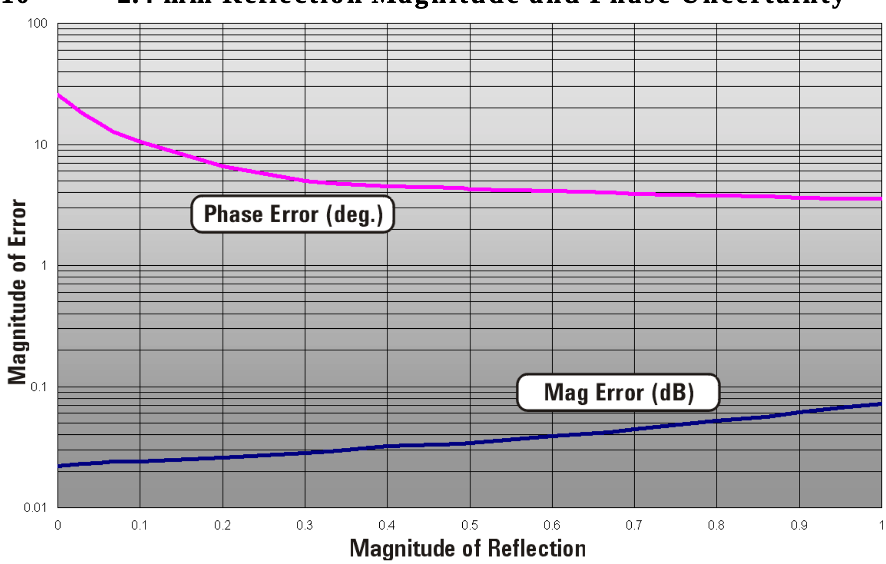


Figure 15-10 2.4 mm Reflection Magnitude and Phase Uncertainty



Test Set Performance

Table 15-19 Test Set Performance

Description	Characteristics	Supplemental Information
Frequency Range	45 MHz to 50.0 GHz[a] 10 MHz to 50.0 GHz[b]	
Transition Time (10 to 90%, TR=.72/BW)	14 ps	
Impedance		50 Ohms (nom.)
Insertion Loss		
Source Out to Coupler Thru	12.0 dB maximum	
Port 2 to Rcvr A In and Port 4 to Rcvr B In		
45 MHz to 1.0 GHz		18 to 45 dB (typical)
1.0 GHz to 50.0 GHz		16 to 26 dB (typical)
Rcvr A In to Cplr Arm and Rcvr B In to Cplr Arm	15.0 dB maximum	
Isolation (port to port)		
45 MHz to 200 MHz	≥ 70 dB	
200 MHz to 50 GHz	≥ 90 dB	
Maximum Operating Level	+20 dBm	
Damage Level		+30 dBm (typical)
Test Port Connectors		50 Ohms (nom.) 2.4 mm (m) Connectors

a. N1957A only.
b. N1957B only.

Power Supply

The power supply requirements for the test sets are listed below.

Table 15-20 Test Set Power Supply Specifications

Description	Specification
Input Voltage Range	100 to 240 Volts
Frequency Range	47 to 63 Hertz
Power	40 VA

General Characteristics

The test set environmental operating conditions and physical characteristics are displayed on the following pages.

Environmental Operating Conditions

The environmental operating conditions for the test set are listed below.

Table 15-21 Test Set Environmental Operating Conditions

Description	Conditions
Operating Environment	Indoor use
Altitude	
Operating:	0 to 2.0 km (6,560 ft)
Storage:	0 to 15.24 km (50,000 ft)
Temperature	0 °C to 40 °C
Maximum Relative Humidity	80% for temperatures up to 31 °C decreasing linearly to 50% for a temperature of 40 °C

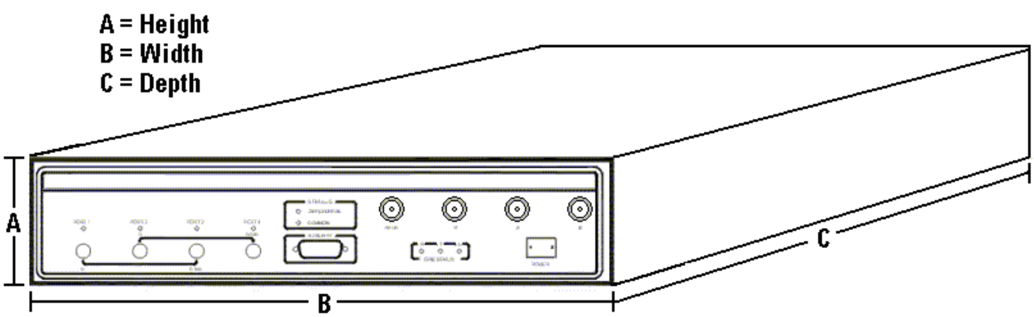
This product is designed for use in INSTALLATION CATEGORY II and POLLUTION DEGREE 2, per IED 61010-1 and 664, respectively.

Physical Characteristics

The weight and dimensions for the test sets are listed below.

Weight and Dimensions

Table 15-22 Test Set Weight and Dimensions



Model Number	**Weight**	**Dimensions**		
		Height (A)	**Width (B)**	**Depth (C)**
N4415A, N4416A, N4417A, N4418A, and N4419A/B	9.0 kilograms (19.9 pounds)	3.0 in (7.62 cm)	16.75 in (42.55 cm)	19.25 in (48.90 cm)
N4420B, and N4421A/B	9.0 kilograms (19.9 pounds)	5.5 in (13.97 cm)	16.75 in (42.55 cm)	16.75 in (42.55 cm)

16 Test Set Front Panel and Rear Panel

This page intentionally left blank.

This chapter provides a graphical overview of the test sets used as part of the physical layer test system. This chapter also illustrates the front and rear panels of the S-parameter test sets separately. The features of each front and rear panel (such as connectors, switches, LEDs, and fuses) are identified and briefly described.

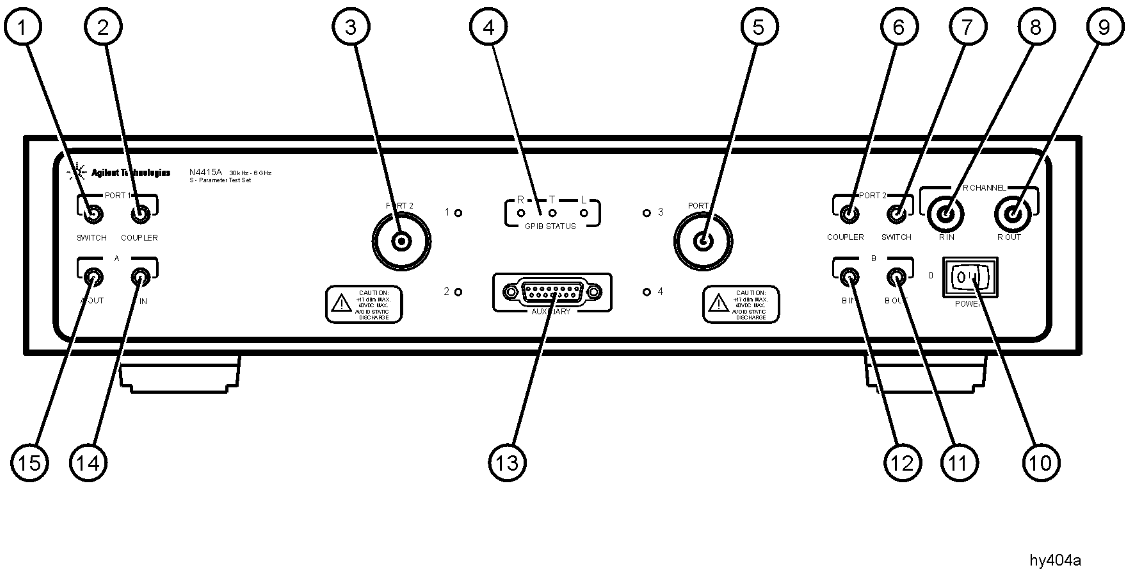
The individual network analyzer features are described in the network analyzer documentation. They are not be described in this document!

The front and rear panel of each S-parameter test set model is illustrated and described. Refer to the page number listed below for your test set model number.

For Model Number:	Refer to:
N4415A	page 500
N4416A	page 504
N4417A	page 508
N4418A	page 512
N4419A/B	page 516
N4420B	page 520
N4421A /B	page 524

N4415A

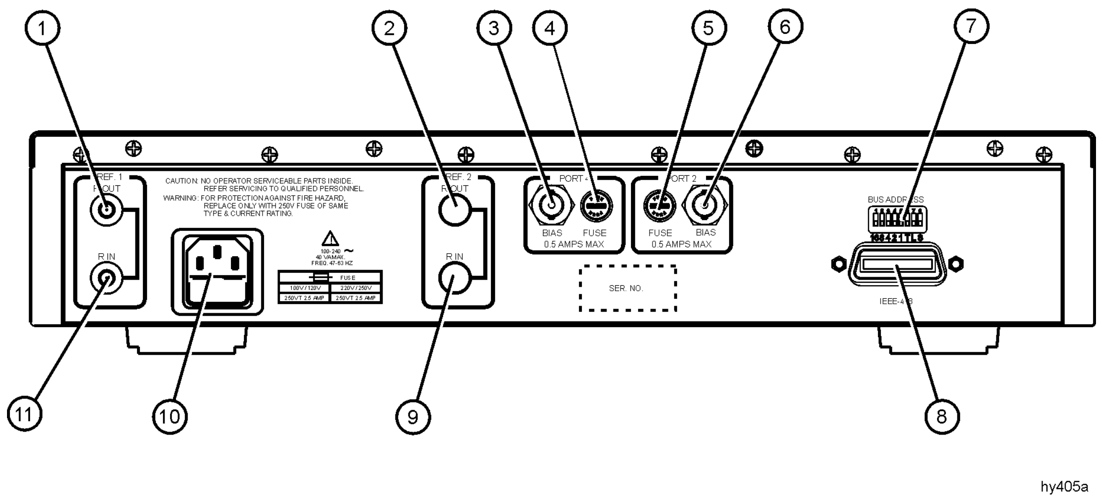
N4415A Front Panel



ID Number	Front Panel Feature	Feature Description
1	SWITCH	SMA (f) connector that is connected to the network analyzer PORT 1 SWITCH connector using a semirigid cable.
2	COUPLER	SMA (f) connector that is connected to the network analyzer PORT 1 COUPLER connector using a semirigid cable.
3	PORT 2	APC-7 connector that is connected to the DUT or fixture. (+17 dBm maximum operating level)
4	GPIB STATUS	Three LEDs (R, T, and L) that display the GPIB status of the test set when it is communicating with the network analyzer. R = Remote Operation, T = Talk mode, L = Listen mode.
5	PORT 4	APC-7 connector that is connected to the DUT or fixture. (+17 dBm maximum operating level)

ID Number	Front Panel Feature	Feature Description
6	COUPLER	SMA (f) connector that is connected to the network analyzer PORT 2 COUPLER connector using a semirigid cable.
7	SWITCH	SMA (f) connector that is connected to the network analyzer PORT 2 SWITCH connector using a semirigid cable.
8	R IN	SMA (f) connector that is connected to the network analyzer R Channel IN connector using a semirigid cable.
9	R OUT	SMA (f) connector that is connected to the network analyzer R Channel OUT connector using a semirigid cable.
10	POWER	ON/OFF switch that disconnects the mains circuits from the mains supply before other parts of the test set. The front panel POWER switch disconnects the mains circuits from the mains supply after the EMC filters and before other parts of the instrument.
11	B OUT	SMA (f) connector that is connected to the network analyzer B OUT connector using a semirigid cable.
12	B IN	SMA (f) connector that is connected to the network analyzer B IN connector using a semirigid cable.
13	AUXILIARY	15-pin ribbon (f) connector that may be connected to the Agilent N4430A/B ECal module to provide ECal capability.
14	A IN	SMA (f) connector that is connected to the network analyzer A IN connector using a semirigid cable.
15	A OUT	SMA (f) connector that is connected to the network analyzer A OUT connector using a semirigid cable.

N4415A Rear Panel

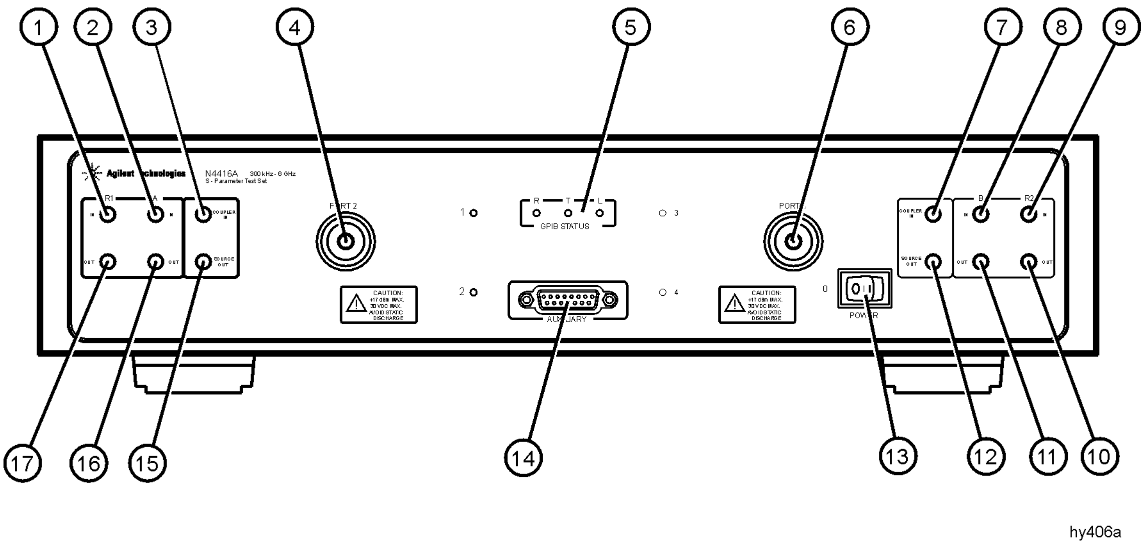


ID Number	Rear Panel Feature	Feature Description
1	REF 1 R OUT	SMA (f) connector, used as an output reference signal
2	REF 2 R OUT	Not Used
3	PORT 4 BIAS	BNC (f) connector. The bias port is used to supply a dc voltage to an active DUT, such as an amplifier or a transistor.
4	PORT 4 FUSE	Bias Fuse, 0.5 A, 250 V (Agilent part number 2110-0012)
5	PORT 2 FUSE	Bias Fuse, 0.5 A, 250 V (Agilent part number 2110-0012)
6	PORT 2 BIAS	BNC (f) connector. The bias port is used to supply a dc voltage to an active DUT, such as an amplifier or a transistor.
7	BUS ADDRESS	Switch that is used to set the GPIB address. Refer to “Step 7. Set Up the General Purpose Interface Bus (GPIB)” on page 39 for further information.
8	IEEE-488	24-pin IEEE-488/PCB (f) connector. The GPIB is the communication bus with the PC and the network analyzer.
9	REF 2 R IN	Not Used

ID Number	Rear Panel Feature	Feature Description
10	Power Cord Connector	Connector, 100-120 Vac or 220-250Vac input and Fuse, T 2.5 A 250 V (Agilent part number 2110-0681)
11	REF 1 R IN	SMA (f) connector, used as an input reference signal

N4416A

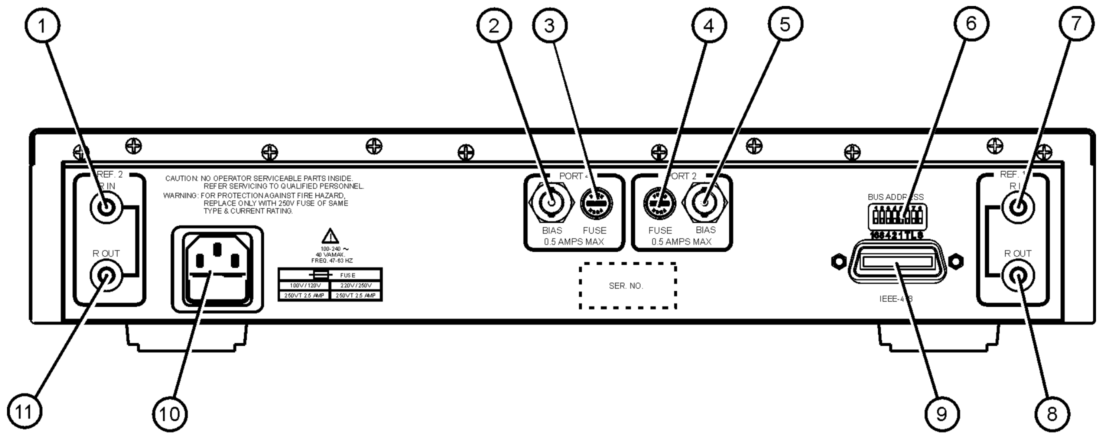
N4416A Front Panel



ID Number	Front Panel Feature	Feature Description
1	R1 IN	SMA (f) connector that is connected to the network analyzer R1 IN connector using a semirigid cable.
2	A IN	SMA (f) connector that is connected to the network analyzer A IN connector using a semirigid cable.
3	COUPLER IN	SMA (f) connector that is connected to the network analyzer COUPLER IN connector using a semirigid cable.
4	PORT 2	APC-7 connector that is connected to the DUT or fixture. (+17 dBm maximum operating level)
5	GPIB STATUS	Three LEDs (R, T, and L) that display the GPIB status of the test set when it is communicating with the network analyzer. R = Remote Operation, T = Talk mode, L = Listen mode.

ID Number	Front Panel Feature	Feature Description
6	PORT 4	APC-7 connector that is connected to the DUT or fixture. (+17 dBm maximum operating level)
7	COUPLER IN	SMA (f) connector that is connected to the network analyzer COUPLER IN connector using a semirigid cable.
8	B IN	SMA (f) connector that is connected to the network analyzer B IN connector using a semirigid cable.
9	R2 IN	SMA (f) connector that is connected to the network analyzer R2 IN connector using a semirigid cable.
10	R2 OUT	SMA (f) connector that is connected to the network analyzer R2 OUT connector using a semirigid cable.
11	B OUT	SMA (f) connector that is connected to the network analyzer B OUT connector using a semirigid cable.
12	SOURCE OUT	SMA (f) connector that is connected to the network analyzer SOURCE OUT connector using a semirigid cable.
13	POWER	ON/OFF switch that disconnects the mains circuits from the mains supply before other parts of the test set. The front panel POWER switch disconnects the mains circuits from the mains supply after the EMC filters and before other parts of the instrument.
14	AUXILIARY	15-pin ribbon (f) connector that may be connected to the Agilent N4430A/B ECal module to provide ECal capability.
15	SOURCE OUT	SMA (f) connector that is connected to the network analyzer SOURCE OUT connector using a semirigid cable.
16	A OUT	SMA (f) connector that is connected to the network analyzer A OUT connector using a semirigid cable.
17	R1 OUT	SMA (f) connector that is connected to the network analyzer R1 OUT connector using a semirigid cable.

N4416A Rear Panel



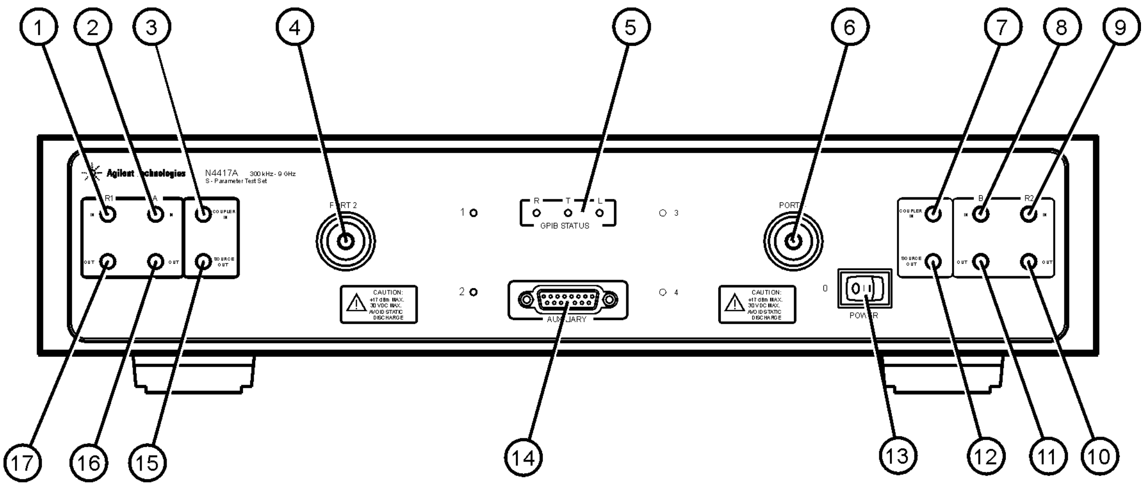
hy407a

ID Number	Rear Panel Feature	Feature Description
1	REF 2 R IN	SMA (f) connector, used as an input reference signal
2	PORT 4 BIAS	BNC (f) connector. The bias port is used to supply a dc voltage to an active DUT, such as an amplifier or a transistor.
3	PORT 4 FUSE	Bias Fuse, 0.5 A, 250 V (Agilent part number 2110-0012)
4	PORT 2 FUSE	Bias Fuse, 0.5 A, 250 V (Agilent part number 2110-0012)
5	PORT 2 BIAS	BNC (f) connector. The bias port is used to supply a dc voltage to an active DUT, such as an amplifier or a transistor.
6	BUS ADDRESS	Switch that is used to set the GPIB address. Refer to “Step 7. Set Up the General Purpose Interface Bus (GPIB)” on page 39 for further information.
7	REF 1 R IN	SMA (f) connector, used as an input reference signal
8	REF 1 R OUT	SMA (f) connector, used as an output reference signal
9	IEEE-488	24-pin IEEE-488/PCB (f) connector. The GPIB is the communication bus with the PC and the network analyzer.

ID Number	Rear Panel Feature	Feature Description
10	Power Cord Connector	Connector, 100-120 Vac or 220-250Vac input and Fuse, T 2.5 A 250 V (Agilent part number 2110-0681)
11	REF 2 R OUT	SMA (f) connector, used as an output reference signal

N4417A

N4417A Front Panel

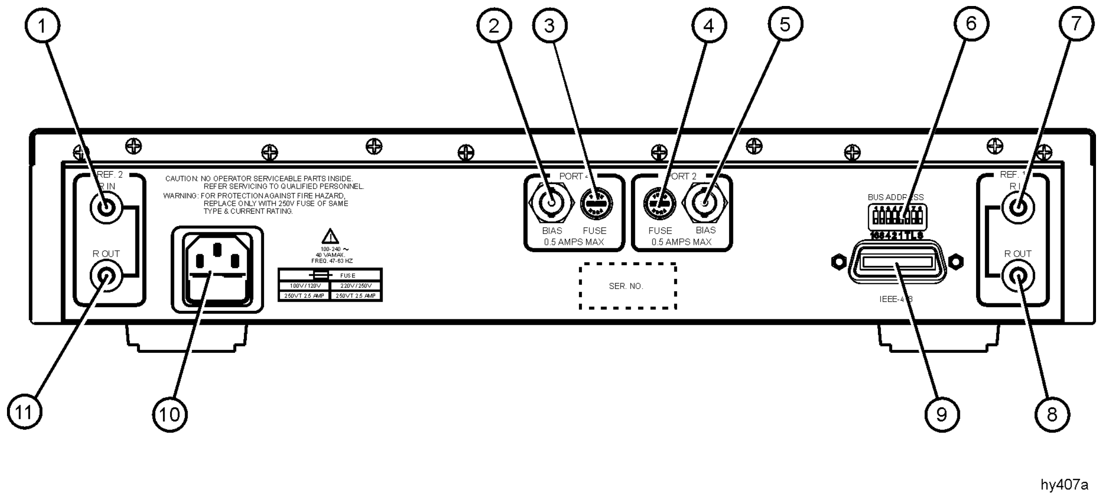


4417_frtpnl

ID Number	Front Panel Feature	Feature Description
1	R1 IN	SMA (f) connector that is connected to the network analyzer R1 IN connector using a semirigid cable.
2	A IN	SMA (f) connector that is connected to the network analyzer A IN connector using a semirigid cable.
3	COUPLER IN	SMA (f) connector that is connected to the network analyzer COUPLER IN connector using a semirigid cable.
4	PORT 2	APC-7 connector that is connected to the DUT or fixture. (+17 dBm maximum operating level)
5	GPIB STATUS	Three LEDs (R, T, and L) that display the GPIB status of the test set when it is communicating with the network analyzer. R = Remote Operation, T = Talk mode, L = Listen mode.

ID Number	Front Panel Feature	Feature Description
6	PORT 4	APC-7 connector that is connected to the DUT or fixture. (+17 dBm maximum operating level)
7	COUPLER IN	SMA (f) connector that is connected to the network analyzer COUPLER IN connector using a semirigid cable.
8	B IN	SMA (f) connector that is connected to the network analyzer B IN connector using a semirigid cable.
9	R2 IN	SMA (f) connector that is connected to the network analyzer R2 IN connector using a semirigid cable. This connector is installed on Option 104 only. It is not installed on Option 103.
10	R2 OUT	SMA (f) connector that is connected to the network analyzer R2 OUT connector using a semirigid cable. This connector is installed on Option 104 only. It is not installed on Option 103.
11	B OUT	SMA (f) connector that is connected to the network analyzer B OUT connector using a semirigid cable.
12	SOURCE OUT	SMA (f) connector that is connected to the network analyzer SOURCE OUT connector using a semirigid cable.
13	POWER	ON/OFF switch that disconnects the mains circuits from the mains supply before other parts of the test set. The front panel POWER switch disconnects the mains circuits from the mains supply after the EMC filters and before other parts of the instrument.
14	AUXILIARY	15-pin ribbon (f) connector that may be connected to the Agilent N4430A/B ECal module to provide ECal capability.
15	SOURCE OUT	SMA (f) connector that is connected to the network analyzer SOURCE OUT connector using a semirigid cable.
16	A OUT	SMA (f) connector that is connected to the network analyzer A OUT connector using a semirigid cable.
17	R1 OUT	SMA (f) connector that is connected to the network analyzer R1 OUT connector using a semirigid cable.

N4417A Rear Panel

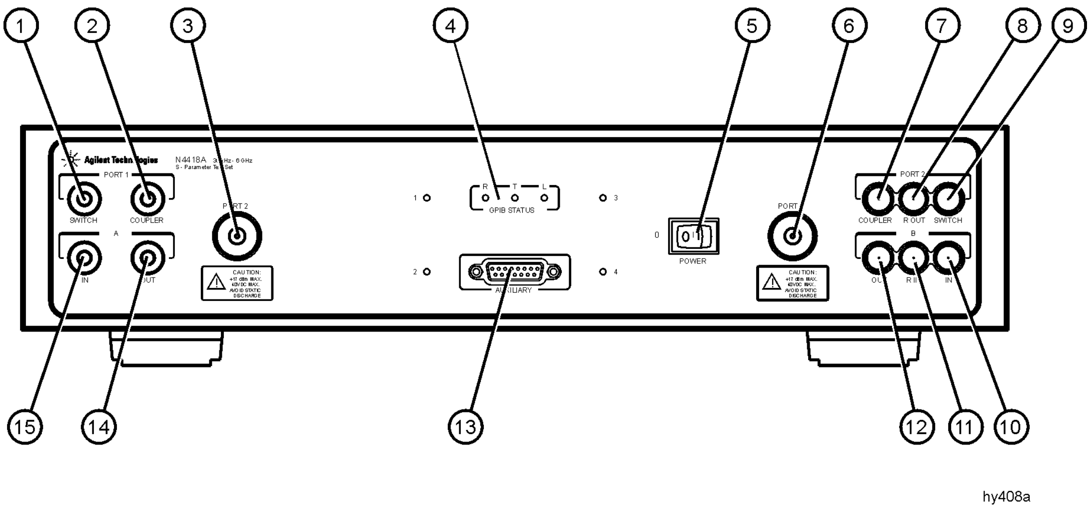


ID Number	Rear Panel Feature	Feature Description
1	REF 2 R IN	SMA (f) connector, used as an input reference signal. This connector is installed on Option 104 only. It is not installed on Option 103.
2	PORT 4 BIAS	BNC (f) connector. The bias port is used to supply a dc voltage to an active DUT, such as an amplifier or a transistor.
3	PORT 4 FUSE	Bias Fuse, 0.5 A, 250 V (Agilent part number 2110-0012)
4	PORT 2 FUSE	Bias Fuse, 0.5 A, 250 V (Agilent part number 2110-0012)
5	PORT 2 BIAS	BNC (f) connector. The bias port is used to supply a dc voltage to an active DUT, such as an amplifier or a transistor.
6	BUS ADDRESS	Switch that is used to set the GPIB address. Refer to "Step 7. Set Up the General Purpose Interface Bus (GPIB)" on page 39 for further information.
7	REF 1 R IN	SMA (f) connector, used as an input reference signal
8	REF 1 R OUT	SMA (f) connector, used as an output reference signal

ID Number	Rear Panel Feature	Feature Description
9	IEEE-488	24-pin IEEE-488/PCB (f) connector. The GPIB is the communication bus with the PC and the network analyzer.
10	Power Cord Connector	Connector, 100-120 Vac or 220-250Vac input and Fuse, T 2.5 A 250 V (Agilent part number 2110-0681)
11	REF 2 R OUT	SMA (f) connector, used as an output reference signal. This connector is installed on Option 104 only. It is not installed on Option 103.

N4418A

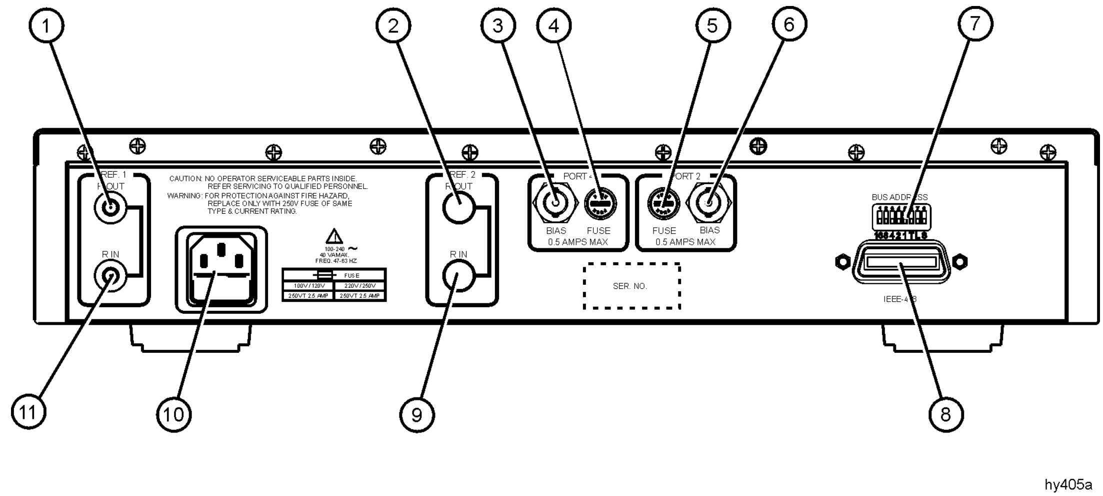
N4418A Front Panel



ID Number	Front Panel Feature	Feature Description
1	SWITCH	SMA (f) connector that is connected to the network analyzer PORT 1 SWITCH connector using a semirigid cable.
2	COUPLER	SMA (f) connector that is connected to the network analyzer PORT 1 BIAS TEE connector using a semirigid cable.
3	PORT 2	APC-3.5 (m) connector with 20 mm nut that is connected to the DUT or fixture. (+17 dBm maximum operating level)
4	GPIB STATUS	Three LEDs (R, T, and L) that display the GPIB status of the test set when it is communicating with the network analyzer. R = Remote Operation, T = Talk mode, L = Listen mode.

ID Number	Front Panel Feature	Feature Description
5	POWER	ON/OFF switch that disconnects the mains circuits from the mains supply before other parts of the test set. The front panel POWER switch disconnects the mains circuits from the mains supply after the EMC filters and before other parts of the instrument.
6	PORT 4	APC-3.5 (m) connector with 20 mm nut that is connected to the DUT or fixture. (+17 dBm maximum operating level)
7	COUPLER	SMA (f) connector that is connected to the network analyzer PORT 2 BIAS TEE connector using a semirigid cable.
8	R OUT	SMA (f) connector that is connected to the network analyzer R CHANNEL OUT connector using a semirigid cable.
9	SWITCH	SMA (f) connector that is connected to the network analyzer PORT 2 SWITCH connector using a semirigid cable.
10	IN	SMA (f) connector that is connected to the network analyzer B IN connector using a semirigid cable.
11	R IN	SMA (f) connector that is connected to the network analyzer R CHANNEL IN connector using a semirigid cable.
12	OUT	SMA (f) connector that is connected to the network analyzer B OUT connector using a semirigid cable.
13	AUXILIARY	15-pin ribbon (f) connector that may be connected to the Agilent N4430A/B ECal module to provide ECal capability.
14	OUT	SMA (f) connector that is connected to the network analyzer A OUT connector using a semirigid cable.
15	IN	SMA (f) connector that is connected to the network analyzer A IN connector using a semirigid cable.

N4418A Rear Panel

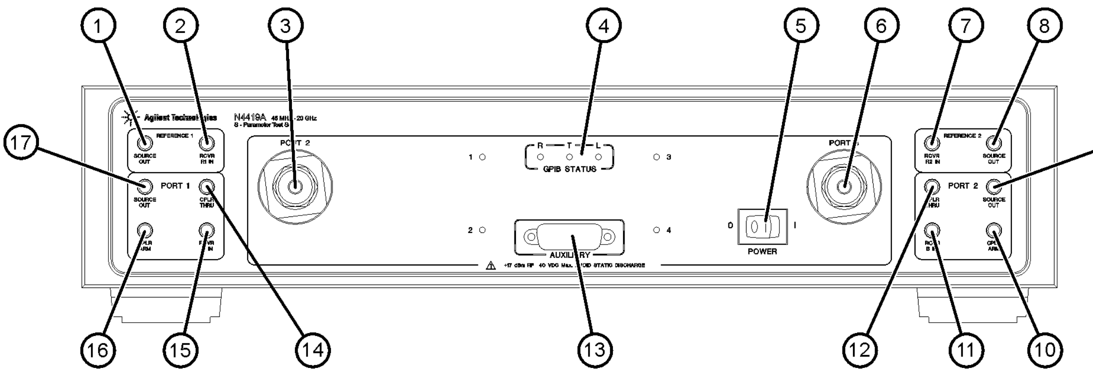


ID Number	Rear Panel Feature	Feature Description
1	REF 1 R OUT	SMA (f) connector, used as an output reference signal
2	REF 2 R OUT	SMA (f) connector, used as an output reference signal
3	PORT 4 BIAS	BNC (f) connector. The bias port is used to supply a dc voltage to an active DUT, such as an amplifier or a transistor.
4	PORT 4 FUSE	Bias Fuse, 0.5 A, 250 V (Agilent part number 2110-0012)
5	PORT 2 FUSE	Bias Fuse, 0.5 A, 250 V (Agilent part number 2110-0012)
6	PORT 2 BIAS	BNC (f) connector. The bias port is used to supply a dc voltage to an active DUT, such as an amplifier or a transistor.
7	BUS ADDRESS	Switch that is used to set the GPIB address. Refer to “Step 7. Set Up the General Purpose Interface Bus (GPIB)” on page 39 for further information.
8	IEEE-488	24-pin IEEE-488/PCB (f) connector. The GPIB is the communication bus with the PC and the network analyzer.
9	REF 2 R IN	SMA (f) connector, used as an input reference signal.

ID Number	Rear Panel Feature	Feature Description
10	Power Cord Connector	Connector, 100-120 Vac or 220-250Vac input and Fuse, T 2.5 A 250 V (Agilent part number 2110-0681)
11	REF 1 R IN	SMA (f) connector, used as an input reference signal

N4419A/B

N4419A/B Front Panels

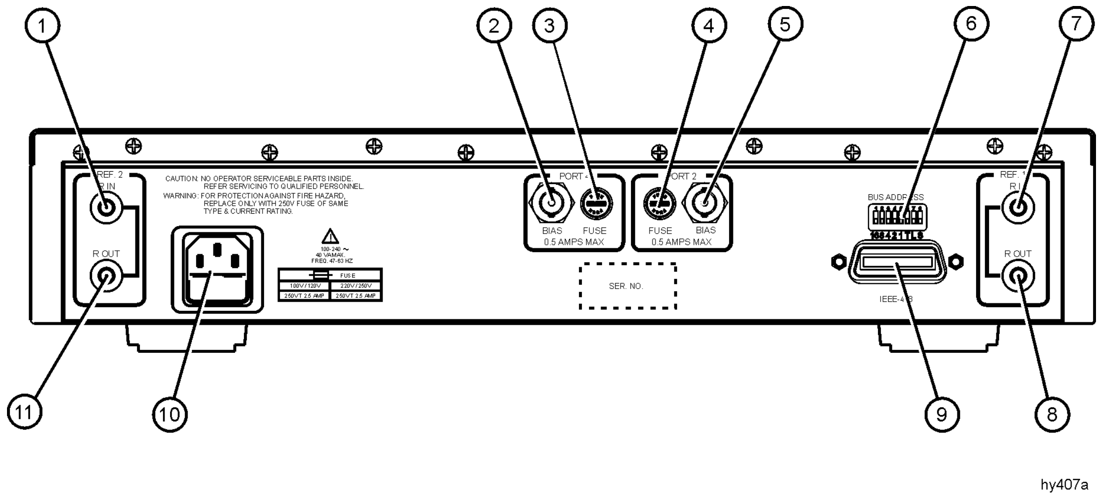


4419frnpnl

ID Number	Front Panel Feature	Feature Description
1	REFERENCE 1 SOURCE OUT	SMA (f) connector that is connected to the network analyzer REFERENCE 1 SOURCE OUT connector using a semirigid cable.
2	REFERENCE 1 RCVR R1 IN	SMA (f) connector that is connected to the network analyzer REFERENCE 1 RCVR R1 IN connector using a semirigid cable.
3	PORT 2	*PORT 2 - APC-3.5 (m) connector with 20 mm nut that is connected to the DUT or fixture. (+17 dBm maximum operating level)*
4	GPIB STATUS	Three LEDs (R, T, and L) that display the GPIB status of the test set when it is communicating with the network analyzer. R = Remote Operation, T = Talk mode, L = Listen mode.
5	POWER	ON/OFF switch that disconnects the mains circuits from the mains supply before other parts of the test set. The front panel POWER switch disconnects the mains circuits from the mains supply after the EMC filters and before other parts of the instrument.
6	PORT 4	*PORT 4 - APC-3.5 (m) connector with 20 mm nut that is connected to the DUT or fixture. (+17 dBm maximum operating level)*

ID Number	Front Panel Feature	Feature Description
7	REFERENCE 2 RCVR R2 IN	SMA (f) connector that is connected to the network analyzer REFERENCE 2 RCVR R2 IN connector using a semirigid cable.
8	REFERENCE 2 SOURCE OUT	SMA (f) connector that is connected to the network analyzer REFERENCE 2 SOURCE OUT connector using a semirigid cable.
9	PORT 2 SOURCE OUT	SMA (f) connector that is connected to the network analyzer PORT 2 SOURCE OUT connector using a semirigid cable.
10	PORT 2 CPLR ARM	SMA (f) connector that is connected to the network analyzer PORT 2 CPLR ARM connector using a semirigid cable.
11	PORT 2 RCVR B IN	SMA (f) connector that is connected to the network analyzer PORT 2 RCVR B IN connector using a semirigid cable.
12	PORT 2 CPLR THRU	SMA (f) connector that is connected to the network analyzer PORT 2 CPLR THRU connector using a semirigid cable.
13	AUXILIARY	15-pin ribbon (f) connector that may be connected to the Agilent N4430A/B ECal module to provide ECal capability.
14	PORT 1 CPLR THRU	SMA (f) connector that is connected to the network analyzer PORT 1 CPLR THRU connector using a semirigid cable.
15	PORT 1 RCVR A IN	SMA (f) connector that is connected to the network analyzer PORT 1 RCVR A IN connector using a semirigid cable.
16	PORT 1 CPLR ARM	SMA (f) connector that is connected to the network analyzer PORT 1 CPLR ARM connector using a semirigid cable.
17	PORT 1 SOURCE OUT	SMA (f) connector that is connected to the network analyzer PORT 1 SOURCE OUT connector using a semirigid cable.

N4419A/B Rear Panel

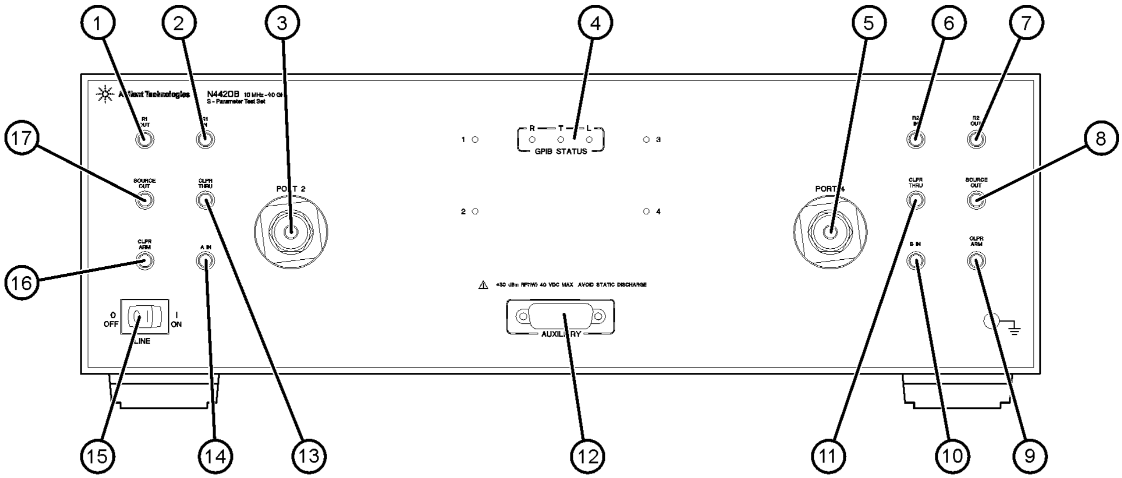


ID Number	Rear Panel Feature	Feature Description
1	REF 2 R IN	SMA (f) connector, used as an input reference signal
2	PORT 4 BIAS	BNC (f) connector. The bias port is used to supply a dc voltage to an active DUT, such as an amplifier or a transistor.
3	PORT 4 FUSE	Bias Fuse, 0.5 A, 250 V (Agilent part number 2110-0012)
4	PORT 2 FUSE	Bias Fuse, 0.5 A, 250 V (Agilent part number 2110-0012)
5	PORT 2 BIAS	BNC (f) connector. The bias port is used to supply a dc voltage to an active DUT, such as an amplifier or a transistor.
6	BUS ADDRESS	Switch that is used to set the GPIB address. Refer to "Step 7. Set Up the General Purpose Interface Bus (GPIB)" on page 39 for further information.
7	REF 1 R IN	SMA (f) connector, used as an input reference signal
8	REF 1 R OUT	SMA (f) connector, used as an output reference signal
9	IEEE-488	24-pin IEEE-488/PCB (f) connector. The GPIB is the communication bus with the PC and the network analyzer.

ID Number	Rear Panel Feature	Feature Description
10	Power Cord Connector	Connector, 100-120 Vac or 220-250Vac input and Fuse, T 2.5 A 250 V (Agilent part number 2110-0681)
11	REF 2 R OUT	SMA (f) connector, used as an output reference signal

N4420B

N4420B Front Panel

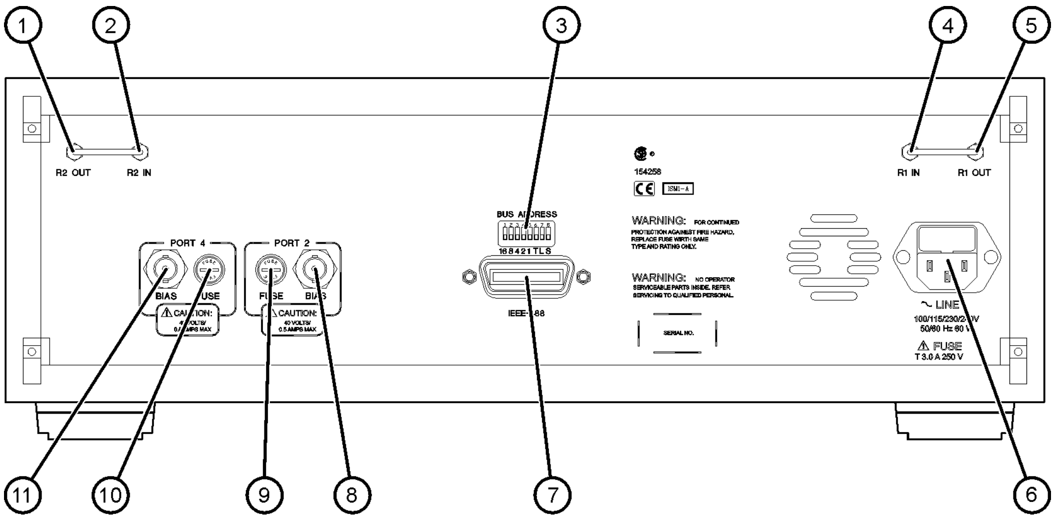


n4420_frtpnl

ID Number	Front Panel Feature	Feature Description
1	R1 OUT	2.4 mm (f) connector that connects to the network analyzer REFERENCE 1 OUT connector using a semirigid cable.
2	R1 IN	2.4 mm (f) connector that connects to the network analyzer REFERENCE 1 RCVR R1 connector using a semirigid cable.
3	PORT 2	2.4 mm bulkhead test port connector that is connect to the DUT or fixture. (+17 dBm maximum operating level)
4	GPIB STATUS	Three LEDs (R, T, and L) that display the GPIB status of the test set when it is communicating with the network analyzer. R = Remote Operation, T = Talk mode, L = Listen mode.
5	PORT 4	2.4 mm bulkhead test port connector that is connect to the DUT or fixture. (+17 dBm maximum operating level)

ID Number	Front Panel Feature	Feature Description
6	R2 IN	2.4 mm (f) connector that connects to the network analyzer REFERENCE 2 RCVR R2 connector using a semirigid cable.
7	R2 OUT	2.4 mm (f) connector that connects to the network analyzer REFERENCE 2 OUT connector using a semirigid cable.
8	SOURCE OUT	2.4 mm (f) connector that connects to the network analyzer PORT 2 SOURCE OUT connector using a semirigid cable.
9	CPLR ARM	2.4 mm (f) connector that connects to the network analyzer PORT 2 CPLR ARM connector using a semirigid cable.
10	B IN	2.4 mm (f) connector that connects to the network analyzer PORT 2 B IN connector using a semirigid cable.
11	CPLR THRU	2.4 mm (f) connector that connects to the network analyzer PORT 2 CPLR THRU connector using a semirigid cable.
12	AUXILIARY	15-pin ribbon (f) connector. Not currently used.
13	CPLR THRU	2.4 mm (f) connector that connects to the network analyzer PORT 1 CPLR THRU connector using a semirigid cable.
14	A IN	2.4 mm (f) connector that connects to the network analyzer PORT 1 A IN connector using a semirigid cable.
15	LINE	ON/OFF switch that disconnects the mains circuits from the mains supply before other parts of the test set. The front panel POWER switch disconnects the mains circuits from the mains supply after the EMC filters and before other parts of the instrument.
16	CPLR ARM	2.4 mm (f) connector that connects to the network analyzer PORT 1 CPLR ARM connector using a semirigid cable.
17	SOURCE OUT	2.4 mm (f) connector that connects to the network analyzer PORT 1 SOURCE OUT connector using a semirigid cable.

N4420B Rear Panel



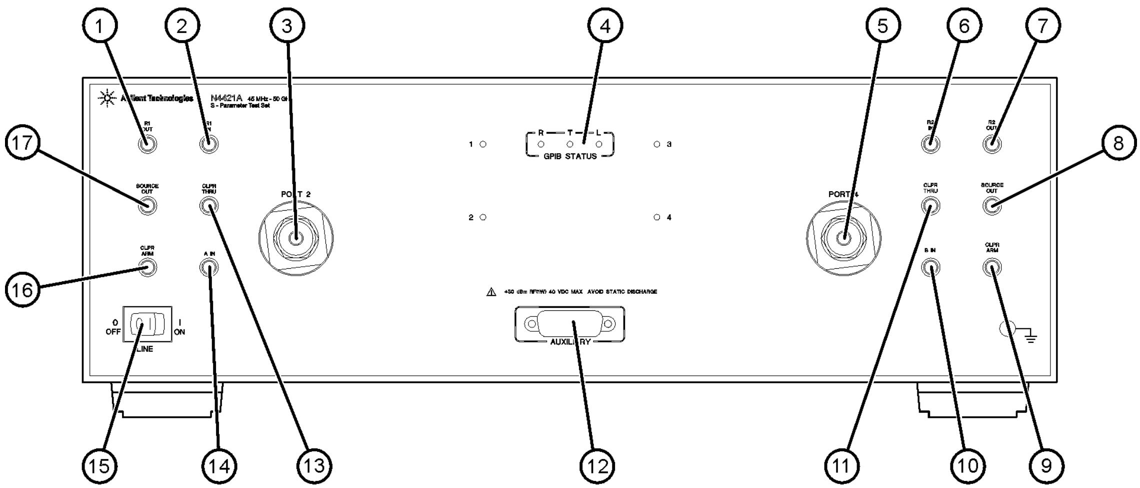
4421_rearpnl

ID Number	Rear Panel Feature	Feature Description
1	REF 2 R OUT	2.4 mm (f) connector, used as an output reference signal
2	REF 2 R IN	2.4 mm (f) connector, used as an input reference signal.
3	BUS ADDRESS	Switch that is used to set the GPIB address. Refer to “Step 7. Set Up the General Purpose Interface Bus (GPIB)” on page 39 for further information.
4	REF 1 R IN	2.4 mm (f) connector, used as an input reference signal
5	REF 1 R OUT	2.4 mm (f) connector, used as an output reference signal
6	Power Cord Connector	Connector, 100-120 Vac or 220-250 Vac input and Fuse, T 2.5 A 250 V (Agilent part number 2110-0681)
7	IEEE-488	24-pin IEEE-488/PCB (f) connector. The GPIB is the communication bus with the PC and the network analyzer.
8	PORT 2 BIAS	BNC (f) connector. The bias port is used to supply a dc voltage to an active DUT, such as an amplifier or a transistor.

ID Number	Rear Panel Feature	Feature Description
9	PORT 2 FUSE	Bias Fuse, 0.5 A, 250 V (Agilent part number 2110-0012)
10	PORT 4 FUSE	Bias Fuse, 0.5 A, 250 V (Agilent part number 2110-0012)
11	PORT 4 BIAS	BNC (f) connector. The bias port is used to supply a dc voltage to an active DUT, such as an amplifier or a transistor.

N4421A/B

N4421A/B Front Panel

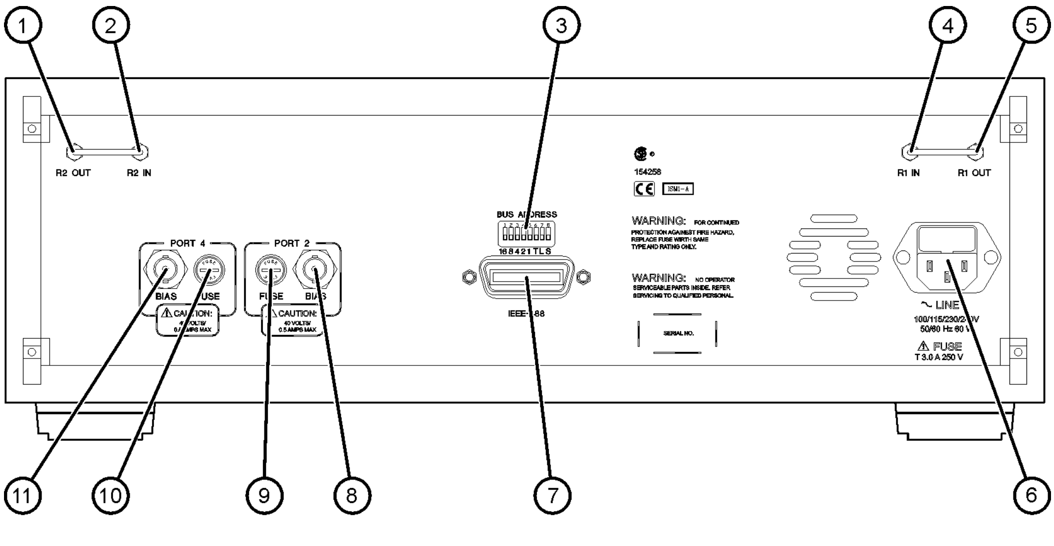


n4421_frtpnl

ID Number	Front Panel Feature	Feature Description
1	R1 OUT	2.4 mm (f) connector that connects to the network analyzer REFERENCE 1 OUT connector using a semirigid cable.
2	R1 IN	2.4 mm (f) connector that connects to the network analyzer REFERENCE 1 RCVR R1 connector using a semirigid cable.
3	PORT 2	2.4 mm bulkhead test port connector that is connect to the DUT or fixture. (+17 dBm maximum operating level)
4	GPIB STATUS	Three LEDs (R, T, and L) that display the GPIB status of the test set when it is communicating with the network analyzer. R = Remote Operation, T = Talk mode, L = Listen mode.
5	PORT 4	2.4 mm bulkhead test port connector that is connect to the DUT or fixture. (+17 dBm maximum operating level)

ID Number	Front Panel Feature	Feature Description
6	R2 IN	2.4 mm (f) connector that connects to the network analyzer REFERENCE 2 RCVR R2 connector using a semirigid cable.
7	R2 OUT	2.4 mm (f) connector that connects to the network analyzer REFERENCE 2 OUT connector using a semirigid cable.
8	SOURCE OUT	2.4 mm (f) connector that connects to the network analyzer PORT 2 SOURCE OUT connector using a semirigid cable.
9	CPLR ARM	2.4 mm (f) connector that connects to the network analyzer PORT 2 CPLR ARM connector using a semirigid cable.
10	B IN	2.4 mm (f) connector that connects to the network analyzer PORT 2 B IN connector using a semirigid cable.
11	CPLR THRU	2.4 mm (f) connector that connects to the network analyzer PORT 2 CPLR THRU connector using a semirigid cable.
12	AUXILIARY	15-pin ribbon (f) connector. Not currently used.
13	CPLR THRU	2.4 mm (f) connector that connects to the network analyzer PORT 1 CPLR THRU connector using a semirigid cable.
14	A IN	2.4 mm (f) connector that connects to the network analyzer PORT 1 A IN connector using a semirigid cable.
15	LINE	ON/OFF switch that disconnects the mains circuits from the mains supply before other parts of the test set. The front panel POWER switch disconnects the mains circuits from the mains supply after the EMC filters and before other parts of the instrument.
16	CPLR ARM	2.4 mm (f) connector that connects to the network analyzer PORT 1 CPLR ARM connector using a semirigid cable.
17	SOURCE OUT	2.4 mm (f) connector that connects to the network analyzer PORT 1 SOURCE OUT connector using a semirigid cable.

N4421A/B Rear Panel



4421_rearpnl

ID Number	Rear Panel Feature	Feature Description
1	REF 2 R OUT	2.4 mm (f) connector, used as an output reference signal
2	REF 2 R IN	2.4 mm (f) connector, used as an input reference signal.
3	BUS ADDRESS	Switch that is used to set the GPIB address. Refer to “Step 7. Set Up the General Purpose Interface Bus (GPIB)” on page 39 for further information.
4	REF 1 R IN	2.4 mm (f) connector, used as an input reference signal
5	REF 1 R OUT	2.4 mm (f) connector, used as an output reference signal
6	Power Cord Connector	Connector, 100-120 Vac or 220-250 Vac input and Fuse, T 2.5 A 250 V (Agilent part number 2110-0681)
7	IEEE-488	24-pin IEEE-488/PCB (f) connector. The GPIB is the communication bus with the PC and the network analyzer.
8	PORT 2 BIAS	BNC (f) connector. The bias port is used to supply a dc voltage to an active DUT, such as an amplifier or a transistor.

ID Number	Rear Panel Feature	Feature Description
9	PORT 2 FUSE	Bias Fuse, 0.5 A, 250 V (Agilent part number 2110-0012)
10	PORT 4 FUSE	Bias Fuse, 0.5 A, 250 V (Agilent part number 2110-0012)
11	PORT 4 BIAS	BNC (f) connector. The bias port is used to supply a dc voltage to an active DUT, such as an amplifier or a transistor.

17 Troubleshooting and Maintenance

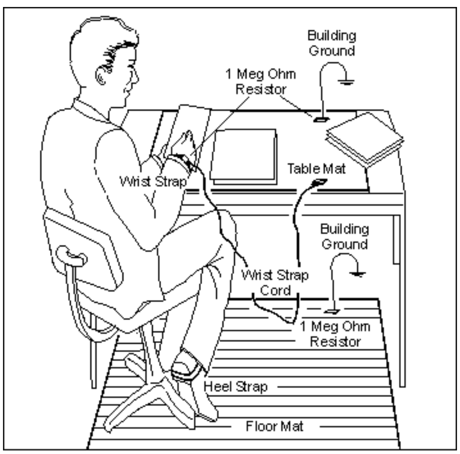
Electrostatic Discharge

Although protected internally, test systems are sensitive to electrostatic discharge (ESD). Static discharges too small to feel can damage or degrade the test equipment or your devices.

Use standard precautions to protect against ESD before using the test system for calibration or measurement.

Use the following illustration and list of equipment to set up a static-safe workstation.

Figure 17-1 Static-Safe Workstation



fomat46

- static-control table mat and earth ground wire: part number 9300-0797
- wrist-strap cord: part number 9300-0980
- wrist-strap: part number 9300-1367
- heel-straps: part number 9300-1308
- floor mat: not available through Agilent Technologies

Troubleshooting

Use the following table to help troubleshoot your test system.

Table 17-1 Troubleshooting the Test System

Symptom	Cause	Cure
One or more biases not applied.	Bias fuse blown.	Check bias fuses. Replace blow fuse with fuse of the same type and rating. Refer to the test set rear-panel information in Chapter 16.
Control computer can't communicate with the test set.	Accidental change to GPIB switch settings.	Set the GPIB address as needed. Restart the test system. See "Step 7. Set Up the General Purpose Interface Bus (GPIB)" on page 39.
The test set does not come on the first time you use it.	Line fuse not installed, or incorrect line fuse installed.	Install the line fuse. Refer to "Replacing the Test Set Line Fuse" on page 534.
Excessive ripple in data.	Load termination damaged by excessive RF power.	Contact Agilent Technologies. See "Contacting Agilent" on page 552 for more information.
	Loose connections between VNA and test set and/or between test set and DUT.	Check and torque the connectors.
	Poor test cable repeatability.	Replace test cables as needed. You can replace a single cable, without replacing the entire set.
High loss on one path with poor raw data match (as seen during analyzer sweep) or inability to make a good calibration.	Possible signal channel damage.	Contact Agilent Technologies. See "Contacting Agilent" on page 552 for more information.

Additional Troubleshooting Assistance

Situation

During installation, the PLTS software does not recognize the Agilent GPIB card. The GPIB card default address is “gpib0” (though it could be any address “gpib1” thru “gpib7” as well).

Background: The software first looks for the presence of a National Instruments GPIB card by looking in the National Instruments library for addresses “gpib0” through “gpib7”. Because the software does not find the card because it is an Agilent GPIB card that is not supported by National Instruments. Then the software looks for the presence of a Hewlett-Packard HP-IB or Agilent Technologies GPIB card in the SICL library. The program looks for the following six addresses in this file: “hpib”, “hpib7”, “HPIB”, “HPIB7”, “Hpib”, and “Hpib7”. However, the Agilent GPIB address is not found.

Solution: 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.

Maintenance

This section provides basic maintenance information such as cleaning, fuse replacement, cable care, and connector care.

Cleaning

Clean the cabinet, using a dry or damp cloth only.

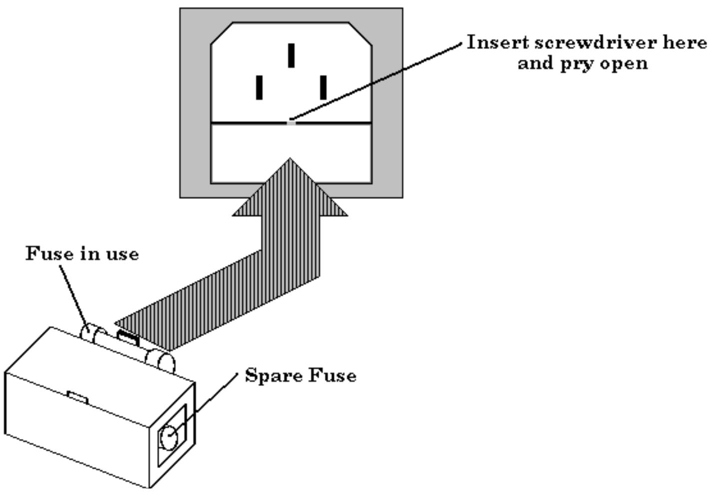
WARNING **To prevent electrical shock, disconnect the Agilent Technologies (N4415A, N4416A, N4417A, N4418A, N4419A/B, N4420B, and N4421A/B) S-parameter test set from mains before cleaning. Use a dry cloth or one slightly dampened with water to clean the external case parts. Do not attempt to clean internally.**

Replacing the Test Set Line Fuse

To replace the line fuse, disconnect the power cord from the rear of the test set, use a small screwdriver to pry open the line fuse holder and slide it open until it reaches its stop. Replace the fuse in use with the spare fuse and slide the fuse holder back into the instrument.

WARNING **For continued protection against fire hazard replace the test set line fuse only with same type and rating (115V and 230V operation: T2.5A 250V). The use of other fuses or material is prohibited.**

Figure 17-2 Line Fuse Replacement



Care of Test Cable Assemblies

Proper use and care of your test cable assemblies will yield positive results including:

- longer life
- higher performance
- better repeatability

Performing the routine inspection and cleaning of the test cable assemblies, especially the connectors, is very important to making the best possible measurements.

Connector Mating

Alignment of the center lines of the connectors of the test cables with the test set and DUT connectors before mating is important. It is possible to start the threads on the connector nuts without good alignment, but this will result in bent pins and damaged inserts. Resistance encountered while turning the connector nut may be due to one of the following:

- The pins are not aligned.
- The coupling nut is cross-threaded.
- The connector (or its mate) has been damaged by excessive torque.

Stop and determine the reason. To proceed without doing so risks the destruction of the assembly and the mating connector.

Holding a connector nut stationary while screwing the socket into the connector will wear away the connector plating and score the connector parts. If the pins lock up, serious damage can be caused.

Connector Torque

Make sure to grasp the body of the connector before applying final torque. If the connector body is allowed to rotate with the nut, the connector plating, outer interface rim, or pin assembly may be damaged. Excess torque applied to the connector will be transferred to the cable assembly. Refer to "Care of RF and Microwave Coaxial Connectors" on page 537 for more information.

Depending on the connector, over-torque can cause damage to connectors in a variety of ways:

- mushroomed outer interface shells
- mushroomed pin shoulders
- recessed or protruding pins
- recessed or protruding dielectrics
- bent pins
- chipped plating
- coupling nut retaining ring damage
- damage to coupling threads

Cable Handling

The test cables have a 1 inch minimum bend radius. Exceeding this radius can result in poor measurements and poor repeatability. Be alert to tight bends where they are not necessarily obvious — like at the end of connector strain relief tubing.

Swept 90° adapters may be used (typically, at the test set front panel) to minimize cable bending. However, if the adapters are used, they must be in place during the calibration.

Cables are often stored in a coiled configuration. Coiled cable “set” (the tendency to stay coiled) can occur if the cables are left coiled. Use large coil diameters (one or two feet) to minimize cable set. Unroll coiled cables prior to use – never just pull out the loops.

Avoid pinching, crushing, or dropping objects on cable assemblies. Dragging a cable over a sharp edge will tend to flatten one side, and it is highly likely that the minimum bend radius will be exceeded.

Care of RF and Microwave Coaxial Connectors

Proper connector care is critical for accurate and repeatable measurements. The following information will help you preserve the precision and extend the life of your connectors - saving both time and money. Prior to making connections to your test system, review the connector care information within this section.

This section is made up of three main subjects:

- **Connector Care Reference**
  which contains information about:
  - ❑ Safety Reminders
  - ❑ Connector Cleaning Supplies
  - ❑ Connector Care Quick Reference
- **Connector Care Concepts**
  which contains information about:
  - ❑ Connector Service Life
  - ❑ Connector Grades and Performance
  - ❑ Adapters as Connector Savers
  - ❑ Connector Mating Plane Surfaces
  - ❑ Gaging Fundamentals
  - ❑ Handling and Storing Connectors
- **Connector Care Procedures**
  which contains information about:
  - ❑ Inspecting Connectors
  - ❑ Cleaning Connectors
  - ❑ Making Connections
  - ❑ Separating Connections
  - ❑ Gaging Connectors
  - ❑ Using a Torque Wrench

Connector Care Reference

This section includes the following information:

- Safety Reminders
- Connector Cleaning Supplies
- Connector Care Quick Reference

Safety Reminders When cleaning connectors:

- Always use protective eyewear when using compressed air or nitrogen.
- Keep isopropyl alcohol away from heat, sparks and flame. Use with adequate ventilation. Avoid contact with eyes, skin and clothing.
- Avoid electrostatic discharge (ESD). Wear a grounded wrist strap (having a 1 MΩ series resistor) when cleaning device, cable or test port connectors.

Connector Cleaning Supplies Products commonly used to clean connectors are listed below. To order these and other connector care products, contact Agilent Technologies.

Table 17-2 Connector Cleaning Supplies

Cleaning Supplies Description	Agilent Part Number
Lint-Free Swabs, small 100 ct.	9301-1243
IPA 99.5% alcohol, 30 ml bottle	8500-5344
Compressed Air, 235 ml can	8500-6659

Connector Care Quick Reference Use the following table as a quick guide for connector care:

Table 17-3 Connector Care Quick Reference

Handling and Storage			
Do	Keep connectors clean	**Do Not**	Touch mating-plane surfaces
	Extend sleeve or connector nut		Set connectors contact-end down
	Use plastic end-caps during storage		
Visual Inspection			
Do	Inspect all connectors carefully	**Do Not**	Use a damaged connector - ever
	Look for metal particles, scratches, and dents		
Connector Cleaning			
Do	Try compressed air first	**Do Not**	Use any abrasives
	Use isopropyl alcohol		Get liquid into plastic support beads
	Clean connector threads		
Gaging Connectors			
Do	Clean and zero the gage before use	**Do Not**	Use an out-of-spec connector
	Use the correct gage type		
	Use correct end of calibration block		
	Gage all connectors before first use		
Making Connections			
Do	Align connectors carefully	**Do Not**	Apply bending force to connection
	Make preliminary connection lightly		Over tighten preliminary connection
	Turn only the connector nut		Twist or screw any connection
	Use a torque wrench for final connect		Tighten past torque wrench “break” point

Connector Care Concepts

This section includes the following concepts:

- Connector Service Life
- Connector Grades and Performance
- Adapters as Connector Savers
- Connector Mating Plane Surfaces
- Gaging Fundamentals
- Handling and Storing Connectors

Connector Service Life Even though calibration standards, cables, and test set connectors are designed and manufactured to the highest standards, all connectors have a limited service life. This means that connectors can become defective due to wear during normal use. For best results, all connectors should be inspected and maintained to maximize their service life.

Visual Inspection should be performed each time a connection is made. Metal particles from connector threads often find their way onto the mating surface when a connection is made or disconnected. See Inspection procedure.

Cleaning the dirt and contamination from the connector mating plane surfaces and threads can extend the service life of the connector and improve the quality of your calibration and measurements. See Cleaning procedure.

Gaging connectors not only provides assurance of proper mechanical tolerances, and thus connector performance, but also indicate situations where the potential for damage to another connector may exist. See “Gaging Fundamentals” on page 543.

Proper connector care and connection techniques yield:

- Longer Service Life
- Higher Performance
- Better Repeatability

Connector Grades and Performance The three connector grades (levels of quality) for the popular connector families are listed below. Some specialized types may not have all three grades.

- **Production grade** connectors are the lowest grade and the least expensive. It is the connector grade most commonly used on the typical device under test (DUT). It has the lowest performance of all connectors due to its loose tolerances. This means that production grade connectors should always be carefully inspected before making a connection to the analyzer. Some production grade connectors are not intended to mate with metrology grade connectors.
- **Instrument grade** is the middle grade of connectors. It is mainly used in and with test instruments, most cables and adapters, and some calibration standards. It provides long life with good performance and tighter tolerances. It may have a dielectric supported interface and therefore may not exhibit the excellent match of a metrology grade connector.
- **Metrology grade** connectors have the highest performance and the highest cost of all connector grades. This grade is used on calibration standards, verification standards, and precision adapters. Because it is a high precision connector, it can withstand many connections and disconnections and, thus, has the longest life of all connector grades. This connector grade has the closest material and geometric specifications. Pin diameter and pin depth are very closely specified. Metrology grade uses an air dielectric interface and a slotless female contact which provide the highest performance and traceability.

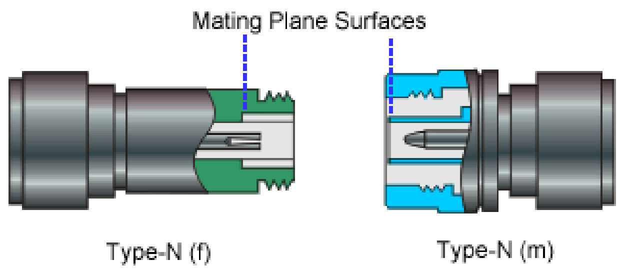
NOTE In general, Metrology grade connectors should not be mated with Production grade connectors.

Adapters as Connector Savers Make sure to use a high quality (Instrument grade or better) adapter when adapting a different connector type to the analyzer test ports. It is a good idea to use an adapter even when the device under test is the same connector type as the analyzer test ports. In both cases, it will help extend service life, and protect the test ports from damage and costly repair.

The adapter must be fully inspected before connecting it to the analyzer test port and inspected and cleaned frequently thereafter. Because calibration standards are connected to the adapter, the adapter should be the highest quality to provide acceptable RF performance and minimize the effects of mismatch.

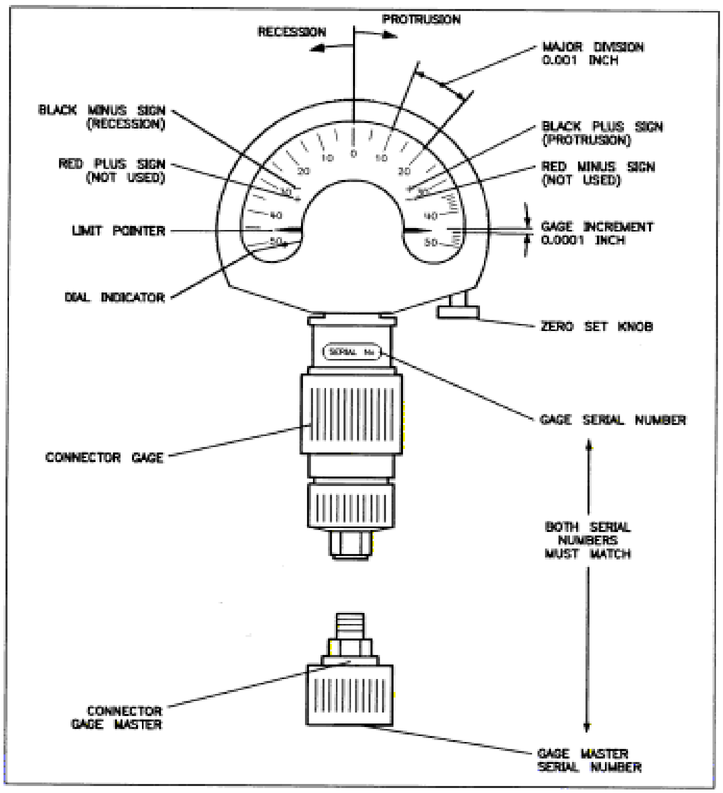
Connector Mating Plane Surfaces An important concept in RF and microwave measurements is the reference plane. For a network analyzer, this is the surface or point that measurements are referenced to at calibration. In connectors, the reference plane is defined as the plane where the mating plane surfaces meet. Good connections require perfectly flat contact between connectors at all points on the mating plane surfaces (see Figure 17-3).

Figure 17-3 Connector Mating Surfaces (Reference Plane)



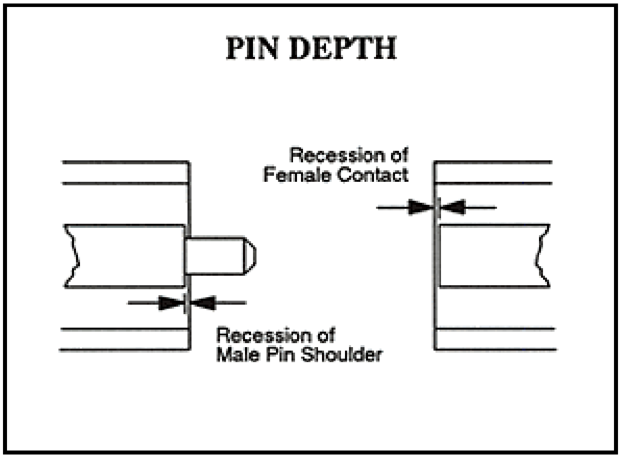
Gaging Fundamentals Connector gages are important tools used to measure center conductor pin depth in connectors. See Figure 17-4. Connector pin depth is generally the distance between the mating plane of the outer conductor and the end of the center conductor, or the shoulder of the center conductor for a stepped male pin.

Figure 17-4 Typical Connector Gage



Recession and Protrusion Pin depth is negative (recession) if the center conductor is recessed below the outer conductor mating plane, usually referred to as the “reference plane”. Pin depth is positive (protrusion) if the center conductor projects forward from the connector reference plane. See Figure 17-5.

Figure 17-5 Connector Pin Depth



1. Recession of female contact
2. Recession of male pin shoulder

Some connectors, like Type-N connectors, have the mating plane of the center conductors offset from the connector reference plane. In this case the zero setting “gage masters” generally offset the nominal distance between the center conductor mating plane and the connector reference plane.

When to Gage Connectors Connectors should be gaged at each of the following events:

- Before using a connector or adapter the first time.
- When visual inspection or electrical performance suggests the connector interface may be out of range.
- After every 100 connections, depending on use.

Connector Gage Accuracy Connector gages (those included with calibration and verification kits), are capable of performing coarse measurements only. This is due to the repeatability uncertainties associated with the measurement. It is important to recognize that test port connectors and calibration standards have mechanical specifications that are extremely precise. Only special gaging processes and electrical testing (performed in a calibration lab) can accurately verify the mechanical characteristics of these devices. The pin depth specifications in the Agilent calibration kit manuals provide a compromise between the pin depth accuracy required, and the accuracy of the gages. The gages shipped with calibration and verification kits allow you to measure connector pin depth and avoid damage from out-of-specification connectors.

NOTE Before gaging any connector, the mechanical specifications provided with that connector or device should be checked.

Handling and Storing Connectors Use the following precautions when handling or storing connectors.

- Install protective end caps when connectors are not in use.
- Never store connectors, airlines, or calibration standards loose in a box. This is a common cause of connector damage.
- Keep connector temperature the same as analyzer. Holding the connector in your hand or cleaning connector with compressed air can significantly change the temperature. Wait for connector temperature to stabilize before using in calibration or measurements.
- Do not touch mating plane surfaces. Natural skin oils and microscopic particles of dirt are difficult to remove from these surfaces.
- Wear a grounded wrist strap and work on a grounded, conductive table mat. This helps protect the analyzer and devices from electrostatic discharge (ESD).
- Wear a grounded wrist strap and work on a grounded, conductive table mat. This helps protect the analyzer and devices from electrostatic discharge (ESD).

Connector Care Procedures

This section includes the following procedures:

- Inspecting Connectors
- Cleaning Connectors
- Making Connections
- Separating Connections
- Gaging Connectors
- Using a Torque Wrench

Inspecting Connectors Use the following procedures when inspecting connectors.

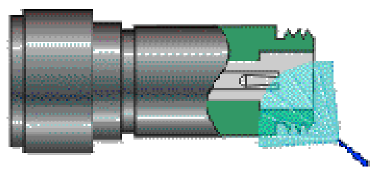
1. Wear a grounded wrist strap (having a 1 MΩ series resistor) when cleaning device, cable or test port connectors.
2. Use a magnifying glass (>10×) and inspect the connector for the following conditions:
    - Badly worn plating or deep scratches
    - Deformed threads
    - Metal particles on threads and mating plane surfaces
    - Bent, broken, or mis-aligned center conductors
    - Poor connector nut rotation

CAUTION A damaged or out-of-specification device can destroy a good connector attached to it even on the first connection. Any connector with an obvious defect should be marked for disposal or sent out for repair.

Cleaning Connectors Use the following procedures when cleaning connectors.

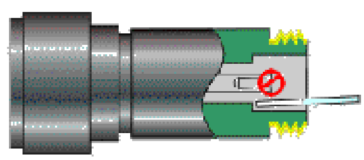
1. Wear a grounded wrist strap (having a 1 MΩ series resistor) when cleaning device, cable or test port connectors.
2. Use clean, low-pressure air to remove loose particles from mating plane surfaces and threads (see Figure 17-6). Inspect connector thoroughly. If additional cleaning is required, continue with the following steps.

Figure 17-6 Removing Loose Particles using Clean, Low Pressure Air



3. Moisten-do not saturate-a lint-free swab with isopropyl alcohol. See Cleaning Supplies for recommended type.
4. Clean contamination and debris from mating plane surfaces and threads. When cleaning interior surfaces, avoid exerting pressure on center conductor and keep swab fibers from getting trapped in the female center conductor. See Figure 17-7.

Figure 17-7 Cleaning Surfaces using a Lint-Free Swab



5. Let alcohol evaporate-then use compressed air to blow surfaces clean.
6. Inspect connector. Make sure no particles or residue remains.
7. If defects are still visible after cleaning, the connector itself may be damaged and should not be used. Determine the cause of damage before making further connections.

Gaging Connectors Use the following procedures when gaging connectors.

1. Wear a grounded wrist strap (having a 1 MΩ series resistor) when cleaning device, cable or test port connectors.
2. Select proper gage for device under test (DUT).
3. Inspect and clean gage, gage master, and DUT.
4. Zero the connector gage.
   a. While holding gage by the barrel, carefully connect gage master to gage. Finger-tighten connector nut only.
   b. Use proper torque wrench to make final connection. If needed, use additional wrench to prevent gage master (body) from turning.
   c. The gage pointer should line up exactly with the zero mark on gage. If not, adjust “zero set” knob until gage pointer reads zero. On gages having a dial lock screw and a movable dial, loosen the dial lock screw and move the dial until the gage pointer reads zero. Gages should be zeroed before each set of measurements to make sure zero setting has not changed.
   d. Remove gage master.
5. Gage the DUT.
   a. While holding gage by the barrel, carefully connect DUT to gage. Finger-tighten connector nut only.
   b. Use proper torque wrench to make final connection and, if needed, use additional wrench to prevent DUT (body) from turning.
   c. Read gage indicator dial for recession or protrusion and compare reading with device specifications.

CAUTION If the gage indicates excessive protrusion or recession, the connector should be marked for disposal or sent out for repair.

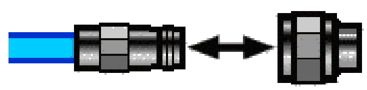
---

6. For maximum accuracy, measure the device a minimum of three times and take an average of the readings. After each measurement, rotate the gage a quarter-turn to reduce measurement variations.
7. If there is doubt about measurement accuracy, be sure the temperatures of the parts have stabilized. Then perform the cleaning, zeroing, and measuring procedure again.

Making Connections Use the following procedures when making connections.

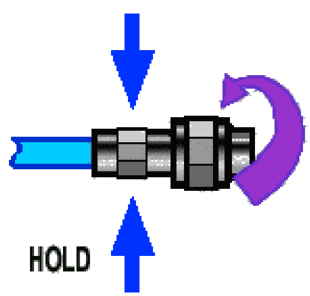
1. Wear a grounded wrist strap (having a 1 MΩ series resistor) when cleaning device, cable or test port connectors.
2. Inspect, clean, and gage connectors. All connectors must be undamaged, clean, and within mechanical specification.
3. Carefully align center axis of both devices. The center conductor pin-from the male connector-must slip concentrically into the contact finger of the female connector. See Figure 17-8.

Figure 17-8 Aligning the Center Axis of Both Connectors



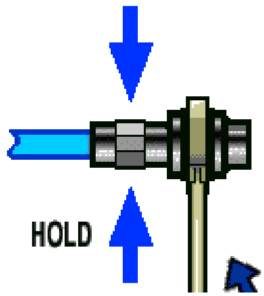
4. Carefully push the connectors straight together so they can engage smoothly. Rotate the connector nut (not the device itself) until finger-tight, being careful not to cross the threads. See Figure 17-9.

Figure 17-9 Pushing the Connectors Together and Rotating the Nut



5. Use a torque wrench to make final connection. Tighten until the “break” point of the torque wrench is reached. Do not push beyond initial break point. Use additional wrench, if needed, to prevent device body from turning. See Figure 17-10.

Figure 17-10 Using a Torque Wrench to Make the Final Connection



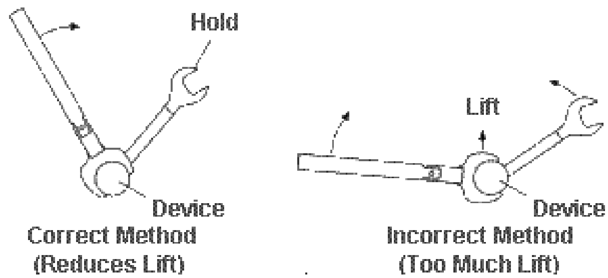
Separating Connections Use the following procedures when separating connections.

1. Support the devices to avoid any twisting, rocking or bending force on either connector.
2. Use an open-end wrench to prevent the device body from turning.
3. Use another open-end wrench to loosen the connector nut.
4. Complete the disconnection by hand, turning only the connector nut.
5. Pull the connectors straight apart.

Using a Torque Wrench Use the following procedures when using a torque wrench.

1. Make sure torque wrench is set to the correct torque setting.
2. Position torque wrench and a second wrench (to hold device or cable) within 90° of each other before applying force. Make sure to support the devices to avoid putting stress on the connectors. See Figure 17-11.

Figure 17-11 Positioning the Wrenches



3. Hold torque wrench lightly at the end of handle-then apply force perpendicular to the torque wrench handle. Tighten until the “break” point of the torque wrench is reached. Do not push beyond initial break point. See Figure 17-12.

Figure 17-12 Torquing with the Torque Wrench

Contacting Agilent

You may use the following table to contact Agilent Technologies for assistance with any Agilent product.

Table 17-4 Contacting Agilent

Online assistance: www.agilent.com/find/assist

United States *(tel)* 1 800 452 4844	**Latin America** *(tel)* (305) 269 7500 *(fax)* (305) 269 7599
New Zealand *(tel)* 0 800 738 378 *(fax)* (+64) 4 495 8950	**Japan** *(tel)* (+81) 426 56 7832 *(fax)* (+81) 426 56 7840
Malaysia *(tel)* 1 800 828 848 *(fax)* 1 800 801 664	**India** *(tel)* 1-600-11-2929 *(fax)* 000-800-650-1101
Taiwan *(tel)* 0800-047-866 *(fax)* (886) 2 25456723	**Hong Kong** *(tel)* 800 930 871 *(fax)* (852) 2506 9233
Canada *(tel)* 1 877 894 4414 *(fax)* (905) 282-6495	**Europe** *(tel)* (+31) 20 547 2323 *(fax)* (+31) 20 547 2390
Australia *(tel)* 1 800 629 485 *(fax)* (+61) 3 9210 5947	**Singapore** *(tel)* 1 800 375 8100 *(fax)* (65) 836 0252
Thailand *(tel) outside Bangkok:* (088) 226 008 *(tel) within Bangkok:* (662) 661 3999 *(fax)* (66) 1 661 3714	**People's Republic of China** *(tel) (preferred):* 800-810-0189 *(tel) (alternate):* 10800-650-0021 *(fax)* 10800-650-0121
Philippines *(tel)* (632) 8426802 *(tel) (PLDT subscriber only):* 1 800 16510170	 *(fax)* (632) 8426809 *(fax) (PLDT subscriber only):* 1 800 16510288

Make sure have the following information readily available when you call:

- the serial number of the test set
- a list of any options or accessories installed in or in use with the test set
- the type of GPIB board in your computer
- any information you can supply about the DUT
- the nature of the problem
- the version number of software

Shipment for Service

If you are sending the instrument to Agilent Technologies for service, ship the test set to the nearest service center for repair, including a description of any failed test and any error message. Ship the instrument using the original or comparable antistatic packaging materials.

Refer to "Contacting Agilent" on page 552 for additional information.

18 Safety and Regulatory Information

Safety Information

Review to the safety information in this section before operating your physical layer test system.

Safety Symbols

The following safety symbols are used throughout this manual. Familiarize yourself with each of the symbols and its meaning before operating the physical layer test system.

CAUTION Caution denotes a hazard. It calls attention to a procedure that, if not correctly performed or adhered to, would result in damage to or destruction of the instrument. Do not proceed beyond a caution note until the indicated conditions are fully understood and met.

WARNING **Warning denotes a hazard. It calls attention to a procedure which, if not correctly performed or adhered to, could result in injury or loss of life. Do not proceed beyond a warning note until the indicated conditions are fully understood and met.**

Instrument Markings

Familiarize yourself with each of the markings and its meaning before operating the physical layer test system.

	The ON symbol. The ON symbol is used to mark the positions of the instrument line switch.
	The OFF symbol. The OFF symbol is used to mark the positions of the instrument line switch.
	The ON symbol. The ON symbol is used to mark the positions of the instrument line switch.
	The OFF symbol. The OFF symbol is used to mark the positions of the instrument line switch.
	The AC symbol. The AC symbol is used to indicate the required nature of the line module input power.
	The instruction documentation symbol. The product is marked with this symbol when it is necessary for the user to refer to the instructions in the documentation.
	The CE mark is a registered trademark of the European Community. (If accompanied by a year, it is when the design was proven.)
	The CSA mark is a registered trademark of the Canadian Standards Association.
ISM1-A	This is a symbol of an Industrial Scientific and Medical Group 1 Class A product.
ICES / NMB-001	This is a marking to indicate product compliance with the Canadian Interference-Causing Equipment Standard (ICES-001).
N10149	The C-Tick mark is a registered trademark of the Australian Spectrum Management Agency.

Safety Considerations

Familiarize yourself with each of the safety considerations before operating the physical layer test system.

NOTE Positioning the Test System for Use

When setting up the physical layer test system for use, position the equipment so that the front panel power switch is easy to reach.

NOTE This instrument has been designed and tested in accordance with the standards listed on the Manufacturer's Declaration of Conformity and has been supplied in a safe condition. This instruction documentation contains information and warnings which must be followed by the user to ensure safe operation and to maintain the instrument in a safe condition.

Safety Earth Ground

WARNING **This is a Safety Class 1 product (provided with a protective earthing ground incorporated in the power cord). The mains plug shall only be inserted in a socket outlet provided with a protective earth contact. Any interruption of the protective conductor, inside or outside the instrument, is likely to make the instrument dangerous. Intentional interruption is prohibited.**

CAUTION Always use the three-prong AC power cord supplied with this product. Failure to ensure adequate earth grounding by not using this cord may cause product damage.

Before Applying Power

CAUTION	Install the instrument so that the ON/OFF switch is readily identifiable and is easily reached by the operator. The ON/OFF switch or the detachable power cord is the instrument disconnecting device. It disconnects the mains circuits from the mains supply before other parts of the instrument. Alternately, an externally installed switch or circuit breaker (which is readily identifiable and is easily reached by the operator) may be used as a disconnecting device.

CAUTION	Before switching on this instrument, make sure that the correct fuse is installed and the supply voltage is in the specified range.

Servicing

WARNING	**No operator serviceable parts inside. Refer servicing to qualified personnel. To prevent electrical shock, do not remove covers.**

WARNING	**These servicing instructions are for use by qualified personnel only. To avoid electrical shock, do not perform any servicing unless you are qualified to do so.**

WARNING	**The opening of covers or removal of parts is likely to expose dangerous voltages. Disconnect the instrument from all voltage sources while it is being opened.**

WARNING	**The power cord is connected to internal capacitors that may remain live for 5 seconds after disconnecting the plug from its power supply.**

WARNING	**For continued protection against fire hazard replace line fuse only with same type and rating (115V and 230V operation: T2.5A 250V). The use of other fuses or material is prohibited.**

General

WARNING **To prevent electrical shock, disconnect the Agilent Technologies (N4415A, N4416A, N4417A, N4418A, N4419A/B, N4420B, and N4421A/B) S-parameter test set from mains before cleaning. Use a dry cloth or one slightly dampened with water to clean the external case parts. Do not attempt to clean internally.**

WARNING **If this product is not used as specified, the protection provided by the equipment could be impaired. This product must be used in a normal condition (in which all means for protection are intact) only.**

CAUTION This product is designed for use in Installation Category II and Pollution Degree 2 per IEC 1010 and 664 respectively.

CAUTION VENTILATION REQUIREMENTS: When installing the product in a cabinet, the convection into and out of the product must not be restricted. The ambient temperature (outside the cabinet) must be less than the maximum operating temperature of the product by 4° C for every 100 watts dissipated in the cabinet. If the total power dissipated in the cabinet is greater that 800 watts, then forced convection must be used.

Regulatory Information

The Agilent Technologies S-Parameter test system complies with the regulatory requirements listed in this section.

Compliance with Canadian EMC Requirements

This ISM device complies with Canadian ICES-001.

Cet appareil ISM est conforme a la norme NMB du Canada.

Compliance with German Noise Requirements

This is to declare that this instrument is in conformance with the German Regulation on Noise Declaration for Machines (Laermangabe nach der Maschinenlaermrerordung –3. GSGV Deutschland).

Acoustic Noise Emission/Geraeuschemission	
LpA <70 dB	LpA <70 dB
Operator position	am Arbeitsplatz
Normal position	normaler Betrieb
per ISO 7779	nach DIN 45635 t. 19

Declaration of Conformity

DECLARATION OF CONFORMITY

According to ISO/IEC Guide 22 and CEN/CENELEC EN 45014

Manufacturer's Name: Agilent Technologies, Inc.

Manufacturer's Address: 40 Shatttuck Road
Andover, MA 01810
USA

Declares that the products

Product Name: Multiport Test Sets & Calibration Modules

Model Number: N4413A, N4414A, N4415A, N4416A, N4417A, N4418A, N4419A, N4421A, N4425A, N4430A, N4430B, N4419B, N4420B, N4421B

Product Options: This declaration covers all options of the above products.

Conform to the following product standards:

EMC: EN 61326:1998

Standard	Limit
EN 55011/A-1999	Group 1, Class A
EN 61000-4-2:1995	4 kV CD, 8 kV AD
EN 61000-4-3:1998+AMD1	3 V/m, 80 - 1000 MHz
EN 61000-4-4:1995	0.5 kV sig., 1 kV power
EN 61000-4-5:1995	0.5 kV L-L, 1 kV L-G
EN 61000-4-6:1996	3 V, 0.15 – 80 MHz
EN 61000-4-11:1994	1 cycle, 100%

Safety: EN 61010-1:1993 +A2:1995

Supplementary Information:
The products herewith comply with the requirements of the Low Voltage Directive 73/23/EEC and the EMC Directive 89/336/EEC and carry the CE-marking accordingly.

Andover, MA, USA [0917/03]

Peter Rienzo/Order Fulfillment Manager

For further information, please contact your local Agilent Technologies sales office, agent or distributor.

Rev. A

V Appendices

Appendices is a collection of supplementary information that you may find useful.

Appendix A, “Glossary”

Defines terms that you may encounter while performing measurements using the physical layer test system.

Appendix B, “Procedures”

Provides procedures that may be needed but are not commonly used while using the physical layer test system.

A Glossary

– Symbols –	
.SnP	.SnP data format creates component data files that describe frequency dependent linear network parameters for n port components. This format is used to import/export S-parameter data. Also see “Touchstone” on page 592.
γ **(Gamma)**	Gamma is the complex propagation constant. $\gamma = \alpha + j\beta$ where α is the attenuation per length and β is related to the wave velocity.
Γ **(Gamma)**	In network analysis, Γ (Gamma) is the ratio of the reflected voltage signal level to the incident signal voltage level (Γ = Vreflected/Vincident). It is the complex reflection coefficient that consists of magnitude (ρ) and phase (Φ) components.
δ **(delta)**	Skin depth. A measure of the depth of current penetration towards the center of a conductor from the perimeter at a particular frequency.
Tan δ	Loss tangent of a material. This is mostly a measure of the ease at which electric fields penetrate (or propagate through) a material. Typical values in electronic materials are in the 0.001 – 0.025 range.
ε_r	ε_r is the relative permittivity of a material (also referred to as dielectric constant), which is mostly a measure of a material’s density. Most insulators used in electronics are in the range of 3 to 10. ε_o is the permittivity of air, which is $8.85e^{-12}$.
μ **(mu)**	μ (mu) is the relative permeability of a material. A measure of how easily a material is magnetized. Most all non-magnetic materials have a value of 1. μ_o is the permeability of a vacuum, which is $4\pi e^{-7}$.
μ**W**	1. Microwave. See “Microwave” on page 580. 2. Microwatts (one-millionth of a watt)
ρ **(rho)**	1. (Material properties) Bulk resistivity of a material (e.g. Cu = $1.7e^{-8}$ ohm-meters). A measure of a materials resistance to current flow. 2. (Network analysis) Magnitude portion of the complex reflection coefficient (Γ). The magnitude of the ratio between the reflected and the incident voltages.

σ (sigma)	1. (Material properties) Bulk conductivity of a material (the inverse of resistivity). 2. (Statistics) Standard deviation, which is the measure of the dispersion or spread of the statistical average of all results for a particular measurement. See “Standard Deviation” on page 589.
τ (tau)	In network analysis, τ (tau) is the magnitude portion of the complex transmission coefficient (T). The magnitude of the ratio between the transmitted and the incident voltages.
T (Tau)	T (Tau) is the ratio of the transmitted voltage signal level to the incident signal voltage level (T = Vtransmitted/Vincident). It is the complex transmission coefficient that consists of magnitude (τ) and phase (ø) components.
ø (phi)	In network analysis, ø (phi) is the phase angle portion of the complex transmission coefficient (the ratio between the transmitted and the incident voltages).
Φ (Phi)	In network analysis, Φ (Phi) is the phase angle portion of the complex reflection coefficient (the ratio between the reflected and the incident voltages).
– Numeric –	
2-Level (PAM2)	A 2-level data signal generates 1 bit per symbol.
4-Level (PAM4)	A 4-level data signal generates 2 bits per symbol consuming half the bandwidth of a 2-level signal.
– A –	
Active Device	An active device is a device that requires a source of energy to add gain to a signal or control a circuit. Examples of active device are LEDs, transistors, and integrated circuits.
American Wire Gauge (AWG)	AWG is the standard for determining wire size. The gauge number varies inversely with the diameter of the wire.
Analog	Analog is a method of transmitting information. Analog is characterized by adding data of continuously varying frequency or amplitude to carrier waves. Digital transmissions depend on the varying between two discrete levels. An analog signal responds to changes in light, sound, heat and pressure.

Arbitrary Bitstream (ABS)	Arbitrary Bitstream is a random-like bit stream used to generate eye diagrams. The "number of bits" and the "number of patterns" user inputs is used to create the ABS. The "number of bits" entry identifies the number of unique bit patterns that are available. The "number of patterns" identifies the number of these unique bit patterns that are used to create the eye diagram. Using both of these values, a random number generator selects unique bit patterns until the appropriate number of bit patterns is identified. Each of these unique bit patterns are then used to create the eye diagram, one bit pattern at a time.
Attenuation	Attenuation is a reduction in signal amplitude. The difference between transmitted and received power due to loss through equipment, lines, or other transmission devices; usually expressed in decibels.
Averaging	Averaging is a waveform acquisition mode in which the instrument acquires waveforms from multiple data acquisitions and then averages the waveforms together, point by point. Averaging significantly reduces noise and improves resolution of the displayed waveform. The noise sources can average to zero over time while the underlying waveform is preserved. The effective resolution of the displayed waveform increases as more acquisitions are averaged together. This improves the stability of both the display and waveform measurements.
– B –	
Backplane	A backplane is a circuit board that has sockets and circuitry and serves as an interconnection between other cards (circuit boards) that are plugged into the sockets. Typically refers to a special, heavy-duty printed or discrete wired circuit board. In terms of PCs, this circuit board is synonymous with the motherboard. Typically the backplane is designed for a longer life than the daughter cards and has improved signal integrity. It is often manufactured out of more expensive PCB materials. Backplanes are either active (also called intelligent) or passive. Active backplanes also contain a microprocessor or circuitry that performs computing functions. Passive backplanes have no computing circuitry.
Balanced Device	A balanced device is composed of two nominally identical halves. Practically speaking, the signals on each side of the device can have any relative amplitude and phase relationship, but they can be decomposed into a differential-mode (anti-phase) component, and a common-mode (in-phase) component.

Balanced Measurement System (BMS)	The BMS is the predecessor to the Physical Layer Test System. The BMS acquires 2-, 3-, or 4-port frequency domain test data using a network analyzer and multiport test set and displays the data in either frequency or time domain. ATN Microwave originally developed this product before being acquired by Agilent Technologies.
Balun	A balun (balanced-unbalanced) is an impedance matching device used to connect a balanced line or device with an unbalanced coaxial line or device.
Bandwidth	In analog terms, bandwidth is the difference between the maximum and minimum frequency. It is measured in hertz (Hz). In digital terms, bandwidth is the data transmission capacity of an electronic circuit or system. It is measured in bits (or bytes) per second.
Baud	Baud refers to the number of level transitions through a device in a one second period. One baud is one state-transition or level-transition per second. In the past, baud was the accepted units for data transmission rate. However, bits per second (bps) is replacing baud as a more accepted unit of measure.
Bit	Bit is an abbreviation of the term "Binary Digit". A bit is the smallest unit of computer data. It is a single digit number, either a one or a zero. A collection of bits makes up a group called a "byte" or a "word" which is equivalent to one alphanumeric character.
Bit Pattern	A bit pattern is a series of bits that are convolved with a time domain impulse response of a system to create an eye diagram.
Bits Per Second (bps)	Bits per second (bps) is the unit of measure for the rate or speed of data transfer. Refer to “Data Transfer Rate” on page 572.
Bit Rate	Bit rate is the number of bits that are sent through a circuit per second, calculated as inverse of the 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.
Broadband	Broadband is high-speed transmission. Although not a standard, it commonly refers to computer data and telecommunication rates in excess of 1.544 Mbps, the rate of T1 lines. Broadband often refers to Internet access using cable modems and DSL. A broadband network can carry voice, video and data all at the same time.

Bus	A collection of wires in a cable (or copper traces on a circuit board) that serve as a common data path between multiple devices. The bus is used to transmit signals (data, status, and control) between the devices that share the bus. Bus typically refers to a parallel data transmission structure (1 clock, multiple data channels). As designs transition from parallel to serial with data rates in excess of 1 Gbps barrier, the terms "Channel" or "Lane" may become more commonly used than "Bus".
Byte	A byte is a unit of data that is eight bits in length. A byte represents a single character, such as a letter, number, or symbol. "Byte" may be preceded with Kilo (Kilobyte) for 1 thousand bytes, Mega (Megabyte) for 1 million bytes, or Giga (Gigabyte) for 1 billion bytes.
– **C** –	
C (capacitance)	Capacitance (farads) is a measure of stored electric charge.
Calibration	In network analyzer systems, calibration is the process of removing systematic errors from measurements (also known as accuracy enhancement or error correction). See “SOLT Calibration” on page 588 and “Thru Reflect Line (TRL) Calibration” on page 591.
Calibration Kit	A calibration kit contains hardware and software required to perform error correction on a network analyzer for a specific measurement and/or test set.
Characteristic Impedance (Zo)	Characteristic impedance is the impedance looking into the end of an infinitely long lossless transmission line.
Coaxial Cable (Coax)	Coaxial cable is a cable with the inner solid or stranded wire acting as the primary conductor wire that is surrounded by a solid or braided metallic shield serving as the ground with an insulating medium between the two.

Common Instrumentation Transfer and Interchange File (CITIfile)	CITIfile is an ASCII data format that is useful when exchanging data between different computers and instruments. CITIfile is a data storage convention designed to be independent of the operating system, and therefore may be implemented by any file system that has the ability to transfer ASCII files. A typical CITIfile package is divided into two sections, the header and the data. 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. The Data is a numeric array of data that is arranged with one data element per line. The data section may contain more than one array of data. Arrays of data start after the BEGIN keyword, and the END keyword follows the last data element in an array.
Common Mode	Common mode is the in-phase mode of a balanced measurement.
Continuous Sweep Mode	Continuous sweep mode is the vector network analyzer condition where traces are automatically updated each time trigger conditions are met.
Crossing Percentage (Eye)	Crossing percentage (eye) is a measure of the amplitude of an eye diagram crossing points relative to the one level and zero level. Crossing percentage = 100 $(V_{cross} - V_{zero\ level}) / (V_{one\ level} - V_{zero\ level})$
Crossing Point (Eye)	Crossing point (eye) is the point in time, in an eye diagram, where the rising edge of a waveform intersects with the falling edge.
Crossstalk	Crosstalk is the occurrence of a signal at one port of a device being affected by a signal in any other path. Isolation is a description of each signal path (or channel's or lane's) immunity from noise from other channels or outside sources.
– D –	
Data Sharing	Data Sharing is a Physical Layer Test System utility that is used to overlay the plot of one measurement parameter over the plot of the same measured parameter from a different data set so that differences and similarities can be readily seen.

Data Transfer Rate	The number of bits that are sent during a one second period. This is generally associated with high-speed serial data transfer systems. The base unit of measure is bits per second (bps). "bps" may be preceded with K (Kbps) for 1000 bits per second, M (Mbps) for 1,000,000 bits per second, or G (Gbps) for 1,000,000,000 bits per second.
De-embedding	De-embedding eliminates the effects (loss, phase shift, mismatch) of the test fixture, connectors, cables and other equipment. De-embedding combines these effects with the errors determined during a coaxial calibration to account for errors of all of the equipment required to measure the DUT. This characterized data is mathematically removed from the measured data, thus ensuring that only the effects of the DUT are displayed. This technique is very useful for measuring DUTs that require fixtures, such as wafers and packages.
Deterministic Jitter	Deterministic jitter is the deviation of a transition from its ideal time caused by reflections relative to other transitions or events happening on neighboring lines. It is pattern dependent, occurring in a predictable, systematic manner due to the varying data patterns. It adds linearly and is measured in peak-to-peak means.
Differential Mode	Differential mode is the anti-phase mode of a balanced measurement.
Direct Current (DC)	Direct Current is electron flow at zero hertz.
DUT	DUT is an acronym commonly used for the device under test.
Dynamic Range	Dynamic range specifies the amplitude (size) of a signal that can be input into the instrument at a particular vertical scale without overdriving the front end, resulting in an inaccurate acquisition of data.
– E –	
Electromagnetic Interference (EMI)	Any electromagnetic energy created by an outside source that interrupts, degrades, or limits the performance of electronics equipment or systems. It can result from unintentional sources, such as spurious emissions and responses, or it can be induced intentionally, as a form of electronic warfare.

Electronic Calibration (ECal)	Electronic calibration performs a SOLT (Short-Open-Load-Thru) calibration on the Physical Layer Test System using an ECal module controlled by the N1930A software. The ECal module is a state-of-the-art, solid-state device with programmable and highly repeatable impedance states that provide consistent calibrations and eliminate operator errors while bringing convenience and simplicity to your calibration routine. The PLTS uses the N4430A and N4430B ECal modules to calibrate to maximum frequencies of 6 GHz and 9 GHz, respectively.
Ethernet	Ethernet is a network that adheres to the IEEE 802.3 Local Area Network standard.
Extinction Ratio	Extinction ratio is the ratio of the one level and the zero level of an eye diagram. This measurement is made in a section of the eye referred to as the eye window.
Eye Amplitude	Eye amplitude is the difference between the logic 1 level and the logic 0 level histogram mean values of an eye diagram.
Eye Diagram or Pattern	An eye diagram is a waveform display that has a specific number of bits sliced and folded on top of one another to produce an overlaid display. In PLTS, the eye diagram is produced by convolving the time domain impulse response of SDD21 with a repetitive bit pattern, which is wrapped.
Eye Height	Eye height is a measure of the vertical opening of an eye diagram.
Eye Opening Factor	Eye Opening Factor is a measure of the vertical opening of an eye diagram.
Eye Width	Eye width is a measure of the horizontal opening of an eye diagram.
Eye Window	The eye window provides the time boundaries within which signal parameters for eye diagrams are measured.
– F –	
Fall Time	Fall time is the transition time for the falling edge of a pulse to decrease from 90% of its peak value to 10% of its peak value.
Fast Fourier Transform (FFT)	The Fast Fourier Transform is an algorithm for transforming data from the time domain to the frequency domain.

Fixturing	Fixturing is the process of using a test fixture as an interface between your test equipment and your device under test (DUT). See “Test Fixture” on page 591.
Forward Orientation	Forward orientation is the direction an adapter is inserted into the Physical Layer Test System equipment setup. An adapter is "forward oriented" when the end labeled "1" at the PLTS test cable and the end labeled "2" will be connected to the DUT. See “Orientation” on page 582.
FR-4	FR-4 is a common epoxy resin glass laminate that is used as substrate for PC boards.
Frequency	Frequency is the number of periodic oscillations, vibrations, or waves per unit of time, usually expressed in cycles per second, or Hertz (Hz).
Frequency Accuracy	Frequency accuracy is the uncertainty with which the frequency of a signal or spectral component is indicated, either in an absolute sense or relative to another signal or spectral component. Absolute and relative frequency accuracies are specified independently.
Frequency Band	Frequency band is a term that identifies a range of frequencies in the electromagnetic spectrum.
Frequency Range	Frequency range is the range of frequencies over which a device or instrument performance is specified.
Frequency Response	In frequency mode, frequency response is the peak-to-peak variation in the displayed amplitude response over a specified center frequency range. Frequency response is typically specified in terms of dB, relative to the value midway between the extremes.
Frequency Span	Frequency span is represented by the horizontal axis of the display. Generally, frequency span is given as the total span across the full display. Sometimes frequency span (scan width) is represented as a per-division value.
Functions (Math)	Functions (Math) are mathematical operations (such as Add, Subtract, Multiply, Integrate, Versus for XY plots) that can be performed on input waveforms, stored waveform memories, or even other functions.
– G –	
G (conductance)	Conductance is the resistive component in shunt impedance from a signal to ground or across transmission lines.

Gating	Gating enables a network analyzer to remove displayed data between specific intervals. This function is used in the time domain mode to separate device connector and coupling effects.
General Purpose Interface Bus (GPIB)	The GPIB bus provides an easy and reliable interface between GPIB instruments and a computer GPIB interface card using GPIB cables. These cables are available in various lengths. Multiple cables can be daisy-chained together to simplify the system interconnection.
Gigabits per Second (Gbps)	Gigabits per second or one billion bits per second. Refer to “Data Transfer Rate” on page 572.
Gigabyte	Gigabyte is generally defined as either 1,000 or 1,024 megabytes. See “Byte” on page 570.
Gigahertz (GHz)	One billion cycles per second. Refer to “Frequency” on page 574.
Golden Device	A device under test that is measured and its test results are saved for comparison against future measurements as a metric of calibration quality and repeatability.
Graticule	Graticules are the horizontal and vertical grid lines making up the plot area. Graticules allow for easier, more accurate viewing of the waveform data and markers.
Group Delay	A measure of the transit time of a signal through a DUT versus frequency. Group delay can be calculated by differentiating the DUT's insertion-phase response versus frequency. $-\Delta$ Phase / (Δf) (360) where, Δ Phase is the phase difference between two adjacent frequencies, Δf.
– H –	
Hertz (Hz)	One cycle per second. Refer to “Frequency” on page 574.
Horizontal Scale	Horizontal scale is an instrument control that controls the x-axis (time or frequency per division) of displayed waveforms. Horizontal scale is often referred to as sweep speed in some instruments.
HSPICE	HSPICE is a circuit simulation tool based on SPICE (Simulation Program for Integrated Circuits Emphasis). Physical Layer Test System data can be exported in an HSPICE format. HSPICE is a product of Synopsys, Inc.

Hub	A hub is a central connection point for devices in a network. It receives a signal from one device and retransmits the signal to one or more devices. Passive hubs are data conduits that just connect devices, adding nothing to the data passing through them. Active hubs regenerate the data bits in order to maintain a strong signal. Intelligent hubs have additional features that monitor traffic. Switching hubs read the destination address of each packet of information and then forward the information to the proper destination.
– I –	
IEEE 802	IEEE 802 is a series of documents that define the standards for Local and Metropolitan Area Networks. The Institute of Electrical and Electronics Engineers, Inc. (IEEE) publish these standards. The following is a list of subjects that the standards define: IEEE Std 802: Overview and Architecture IEEE Std 802.1: Bridging and Management IEEE Std 802.2: Logical Link Control IEEE Std 802.3: CSMA/CD Access Method IEEE Std 802.5: Token Ring Access Method IEEE Std 802.6: DQDB Access Method IEEE Std 802.7: Broadband LAN IEEE Std 802.10: Security IEEE Std 802.11: Wireless LANs IEEE Std 802.12: Demand Priority Access Method IEEE Std 802.15: Wireless Personal Area Networks IEEE Std 802.16: Broadband Wireless Metropolitan Area Networks
Imaginary	Imaginary is a format that displays the reactive portion of the measured data on a Cartesian format. This is the corollary to the Real format where the resistive portion is displayed.
Impedance	Impedance is the ratio of voltage to current at a port of a circuit, expressed in ohms.
Insertion Loss	Insertion loss is the difference between the power measured before and after the insertion of a device or the attenuation between the input and output of a device.

Inverse Fast Fourier Transform	The Inverse Fast Fourier Transform is an algorithm for transforming data from the frequency domain to the time domain.
Isolation	Isolation is the specification or measure of the immunity that one signal has to being affected by another adjacent signal. Low isolation in digital systems manifests itself as crosstalk or noise on the victim channel.
– J –	
Jitter	Jitter is the measure of the time variances of the rising and falling edges of an eye diagram as these edges affect the crossing points of the eye.
– K –	
K28.5	K28.5 is an industry standard, finite length, specific bit sequence than includes a comma character and 8B/10B encoding.
Kilobits per Second (Kbps)	Kilobits per second or one thousand bits per second. Refer to "Data Transfer Rate" on page 572.
Kilobyte	Kilobyte is generally defined as either 1,000 or 1,024 bytes. See "Byte" on page 570.
Kilohertz (kHz)	One thousand cycles per second. Refer to "Frequency" on page 574.
– L –	
L (inductance)	Inductance (henries) is stored magnetic charge.
Line Reflect Match (LRM) Calibration	LRM calibration is a calibration type that utilizes three simpler, more convenient standards to define the error terms to be removed from the measurement. The measured parameters of the Line, Reflect, and Match standards in a LRM calibration kit provides the same information as a SOLT calibration via a different algorithm. A calibration at the coaxial ports of the test system removes the effects of the system 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 SOLT standards may not be readily available to allow a conventional full 4-port calibration at the desired measurement port of the device.

Line Reflect Match (LRM) Calibration (continued)	LRM calibration is convenient because calibration standards can be fabricated for the specific measurement environment. The characteristic impedance of these fabricated transmission lines can be determined from the physical dimensions and substrate's dielectric constant. The LRM calibration relies on the characteristic impedance of simple transmission lines.
Linear Device	A linear device is a device that only modifies phase or magnitude of frequency components present on the input signal.
Linear Mag	Linear Mag is the display mode in which the vertical deflection is presented in linear function (vertical divisions are uniformly space). This format is used for unit-less measurements, such as reflection coefficient magnitude and for linear measurement units. It is used to display conversion parameters and time domain transform data.
Load	1) A load is a one-port microwave device used to terminate a path in its characteristic impedance. 2) A load is a calibration standard that is an actual line that terminates a path with the path's characteristic impedance. See “SOLT Calibration” on page 588.
Load Match	Load match is a measure of how close the device's terminating load impedance is to the ideal transmission line impedance. Match is usually measured as return loss or standing wave ratio (SWR) of the load.
Log Display	Log display (logarithmic display) is the display mode in which vertical deflection is a logarithmic function of the input signal amplitude. The display calibration is set by selecting the value of the reference level position and scale factor in dB per division.
Log Mag	Log Mag is the display mode in which vertical deflection is a logarithmic function of the input signal amplitude. Log Mag is also called logarithmic display.

Low Voltage Differential Signals (LVDS)	LVDS is a high-speed (gigabits per second), low-noise, low-power, low voltage method of transmitting digital information. The differential signals are transmitted over two traces or over balanced cable using a very low voltage swing between high (binary 1) and low (binary 0). The low swing voltage means that data can switched very quickly which provides the higher data transfer rate. LVDS uses the two traces (cables) for two signals, which are 180 degrees out of phase from each other. Thus, the noise travels at the same level making filtering very effective. This mode of transmission is often used with SCSI hardware.
– M –	
Magnitude	Magnitude is the amplitude of a signal measured in its characteristic impedance without regard to phase.
Markers	Marker lines are used to determine the position or amplitude of the selected point on the display graticule. Marker lines can be positioned on either: • A selected waveform source (input channel, waveform memory, or waveform function) • Independently, anywhere on the display graticule
Mask	A mask is a template consisting of numbered, shaded regions on the instrument display screen. The input waveform must remain within these regions in order to comply with industry standards. The waveforms that intrude these regions are mask violations.
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.
Mask Test	A mask test is a test process used to verify that waveforms generated by a test device conform to industry standards.

Measurement Uncertainty	Measurement uncertainty is the quantified amount of error in a measurement situation. Calibrations are intended to reduce the amount of uncertainty. The following are sources of measurement errors that lead to uncertainty: • Systematic errors (imperfections in calibration standards, connectors, cables, and instrumentation) • Random errors (noise, connector repeatability) • Drift (source and instrumentation)
Megabits per Second (Mbps)	Megabits per second or one million bits per second. Refer to "Data Transfer Rate" on page 572.
Megabyte	Megabyte is generally defined as either 1,000 or 1,024 kilobytes. See "Byte" on page 570.
Megahertz (MHz)	One million cycles per second. Refer to "Frequency" on page 574.
Microprocessor Unit (MPU)	A microprocessor is a computer (central processing unit, CPU) on a single digital semiconductor chip. It performs math and logic operations and executes instructions from memory. A microprocessor requires a power supply, clock and memory to function as a computer.
Microstrip	Microstrip is a planar transmission line that consists of a thin conductive trace (or traces) printed or etched on the top side of an insulating substrate with a parallel ground plane on the other side of the substrate. Microstrips are also used for antennas and antenna arrays.
Microwave	Microwave is the frequency band where radio waves are very short. This band ranges from approximately 1 GHz to 40 GHz, within the UHF, SHF, and EHF frequency bands.
Mixed Mode S-parameters	Mixed-mode S-parameters describe the performance of a device when measuring a balanced device. Each balanced port will support both a common-mode and a differential-mode signal.

Mode Conversion	Mode conversion is a measure of isolation when making balanced S-parameter measurements where there is a: • Differential-mode stimulus with a common-mode response (SCDXX). Mode conversion in this configuration can result in device asymmetry and generation of electromagnetic interference (EMI). • Common-mode stimulus with a differential-mode response (SDCXX). Mode conversion in this configuration can result in device asymmetry and susceptibility to EMI.
– **N** –	
Near End Crosstalk (NEXT)	Signal distortion as a result of signal coupling from one wire pair to another wire pair at various frequencies. When measuring the NEXT, it is usually the resultant voltage excursion expressed as a percentage of the incident voltage swing of the culprit line.
Network Analysis	Network analysis is the characterization of a device, circuit, or system derived by comparing a signal input going into the device to a signal or signals coming out from the device.
Noise	Noise is an unwanted disturbance (voltage or current) superimposed on a useful waveform.
Non-insertable Calibration	A non-insertable calibration is one in which the test port connectors are of the same gender (male-to-male or female-to-female).
Non-linear Device	A non-linear device is a device in which frequency components are added or deleted (not just modified) from the original signal.
Non-Return to Zero (NRZ) Signaling	NRZ signaling is used in differential signaling to describe that swing about some offset voltage do no return to 0 volts.
Normalize	To normalize is to subtract one trace from another to eliminate calibration data errors or to obtain relative information.
– **O** –	
Offset	Offset is used to move or set off a determined amount. Used in instruments for offsetting frequencies, limits, delay, loss, impedance, etc.

One Level	One level is a measure of the mean value of the logical 1 of an eye diagram.
Open	An Open is a calibration standard that is an actual line that terminates a path with an electrical open. See “SOLT Calibration” on page 588.
Orientation	Orientation is the direction that an adapter is inserted between the Physical Layer Test System port's coaxial test cable and the DUT. Each end of the adapter should be labeled: either "1" or "2". Identifying the adapter orientation is important when calibrating the test system. An adapter is "forward oriented" when the end labeled "1" at the test cable and the end labeled "2" will be connected to the DUT. An adapter is "reverse oriented" when the end labeled "2" at the test cable and the end labeled "1" will be connected to the DUT.
– P –	
Passive Device	A passive device is a device that requires only a signal to perform its function. It does not require a source of power for its operation and it provides no gain to a circuit. Examples of passive devices are resistors, inductors, capacitors, cables, and filters.
Pattern Length	Pattern length is the number of bits that are used to create a bit pattern used to create an eye diagram in the Physical Layer Test System. Bit patterns may be between 8 and 32 bits long.
Phase	The fractional part of a cycle through which an oscillation has advanced, measured from an arbitrary starting point; usually measured in radians or degrees. In network analysis, the phase response of the device under test is the change in phase as a function of frequency between the input stimulus and the measured response. In network analysis, the phase response of the device under test is the change in phase as a function of frequency between the input stimulus and the measured response.
Phase Rotation	See “Phase Skew” on page 583.
Phase Shift	Phase shift is the change in phase of a signal between two points of time. Phase shift is expressed in degrees of lead or lag.

Phase Skew	Phase skew is a technique to remove the time delay caused by a fixture. Since the SOLT calibration only calibrates to the end of the test cables, the effects added by the test fixture can be removed mathematically. Phase skew moves the reference plane from the end of the test cables to the connection between the fixture and the DUT by accounting for the electrical length of each fixture path. Other terms for phase skew are phase rotation, port extension, port rotation, and reference plane rotation.
Physical Layer	The physical layer is layer 1, the lowest layer, of the seven-layer Open System Interconnection model. In broad terms, the physical layer is responsible for activating and using physical connections for transfer of electronic bits (zeros and ones) between a device and its transmission medium. The physical layer defines the electrical, mechanical, and handshaking protocols that govern transmission media and signals over the interface connecting a device to the transmission medium. In doing so, the physical layer insulates the data link layer (layer 2) from the physical characteristics of the transmission medium, such as baseband, broadband, or fiber-optic transmission. The physical layer is subdivided into the physical medium-dependent sublayer and the transmission convergence sublayer. Physical layer has three basic mediums: electrical (where SIO is focused), wireless/over air, and optical. All three are used in today's communications systems.
Polar Chart	The Polar chart is a format where each point corresponds to a particular value of both magnitude and phase. Quantities are read vectorally: the magnitude at any point is determined by its displacement from the center (which has zero value), and the phase by the angle counterclockwise from the positive x-axis. Magnitude is scaled in a linear fashion, with the value of the outer circle set to a ratio value of 1. Since there is no frequency axis, frequency information is read from the markers. The default marker readout for the polar format is in linear magnitude and phase.
Port	Port is a network-analysis term for the path that sends/received data to/from the DUT. In logic analysis and oscilloscope terminology a port is more like a channel. A channel in a logic analyzer or an oscilloscope is usually unidirectional (acquisition only) where as a port in a network analyzer is bi-directional. A port is also equivalent to the "pin" of an IC or board tester, which is usually bi-directional as well. So a port is similar to a pin on a package or connector.

Port Extension	See "Phase Skew" on page 583.
Port Rotation	See "Phase Skew" on page 583.
Power Level	Power level (dBm) is the stimulus level at the test port required for the measurement of the device under test.
Probe	A probe is the test device is connected to the Physical Layer Test System (or some other electronic device) that is used to make contact with a DUT to deliver or detect a signal for the purpose of on-circuit measurements. It is often a stylus-like device having multiple conductors for signal and ground paths. The tip is also often spring loaded to apply a constant pressure to the DUT.
Probing	In the frequency domain, probing is the process of using a probe to perform on-circuit measurements. In the time domain, probing typically refers to the technique of either browsing (single probe) or attaching (multiple probes, as in a logic analyzer) to the DUT and attempting to be non-intrusive with respect to the signal. Oscilloscope or logic analyzer-based measurements probing is designed to be as non-intrusive to the active electrical signal as possible while retaining the high bandwidth required to make accurate measurements.
Pseudo-Random Binary Sequence (PRBS)	Pseudo-Random Binary Sequence is a fixed length, somewhat random, digital signal pattern.
Pulse Width	Pulse width is the difference in time between the rising and falling edges of a signal that is transitioning, away from and then back to, its steady state.
– R –	
R (resistance)	Resistance (ohms) is the opposition to the flow of current in a conductor.
Rack Units	A rack unit (U) is the vertical distance (height) between screw holes in an equipment rack. One rack unit (1U) equals 1.75 inches (44.45 mm) of rack space for equipment. 2U provides 3.5 inches of rack space for equipment, and so forth.

Radio Frequency (RF)	RF is the frequency band where radio waves are below the microwave band. The RF band ranges from approximately 50 kHz to 3 GHz, within the LF, MF, HF, VHF, and UHF frequency bands.
Range Resolution	Range resolution is defined as the ability to locate a single response in the time domain. If only one response is present, range resolution is a measure of how closely you can pinpoint the peak of that response. The range resolution is equal to the digital resolution of the display, which is the time domain span divided by the number of points on the display. The range resolution is always much finer than the response resolution.
Real	Real is a format that displays the resistive portion of the measured data on a Cartesian format. This is the corollary to the Imaginary format where the reactive portion is displayed.
Record Length	In frequency domain, record length refers to number of frequency points measured by the network analyzer. A longer record length (number of points) in a VNA implies either more resolution (closer points in the frequency domain) or a wider bandwidth measurement. In time domain, record length refers to the number of time points or instances acquired by the scope or logic analyzer. A longer record length (also occasionally called "points") means greater resolution for any given time duration.
Reference Level	Reference level is an instrument function that allows the user to set the amplitude value at the reference position. On network analyzers, the reference position is also selectable.
Reference Plane	A reference plane is the electrical location at which a network analyzer assumes the system connectors and fixturing ends and the device under test (DUT) begins. The reference plane is set by using calibration standards with known electrical lengths. The closer the reference plane is to the DUT, the better the characterization of the device because of the elimination of test system uncertainties.
Reference Plane Rotation	See “Phase Skew” on page 583.
Reflection	Reflection is the phenomenon in which a traveling wave strikes a discontinuity and returns to the original medium.

Reflection Coefficient	The reflection coefficient is the ratio of the reflected voltage to the incident voltage into a transmission line or circuit. If a transmission line is terminated in its characteristic impedance, the reflection coefficient is zero. If the line is shorted or open, the coefficient is 1.
Reflection Measurements	Reflection measurements characterize the input and output behavior of the device under test. Measured as the ratio of the reflected signal to the incident signal as a function of frequency. Parameters are called return loss, reflection coefficient, impedance match, and standing wave ratio (SWR), all as a function of frequency.
Response	The Physical Layer Test System applies reference signals that are transmitted through the DUT or are reflected from the DUT's input. The transmitted or reflected signal is then detected and compared against the reference signal. A detected signal is called the "response". A reference signal is called the stimulus.
Response Resolution	Response resolution is defined as the ability to resolve two closely spaced responses, or a measure of how close two responses can be to each other and still be distinguished from each other in the time domain.
Reverse Orientation	Reverse orientation is the direction an adapter is inserted into the Physical Layer Test System equipment setup. An adapter is "reverse oriented" when the end labeled "2" at the PLTS test cable and the end labeled "1" will be connected to the DUT. See “Orientation” on page 582.
Rise Time	Rise time is the transition time for the leading edge of a pulse to rise from 10% of its peak value to 90% of its peak value.
RMS	RMS is the root-mean-square (rms) of the voltage values of a waveform. Typically, the rms is taken over the first period of the displayed waveform.

Router (Networking)	A router is an electronic device that links local and wide area networks (LANs and WANs), allowing them to talk to one another even though the networks may be based on different standards. Using routing tables and protocols, routers read the network address in each transmitted frame and decide how to send it based on the most expedient route. Gateway is a generic term for a router. Most routers are specialized computers that are optimized for communications; however, router functions can also be implemented by adding routing software to a file server.
– **S** –	
S-parameters (Scattering Parameters)	A convention used to characterize the way a device modifies signal flow. A four-port device has sixteen S-parameters: four forward transmission parameters, four reverse transmission parameters, four forward reflection parameters, and four reverse reflection parameters.
Scale (horizontal)	Horizontal scale is an instrument control that controls the x-axis or time per division of displayed waveforms. Horizontal scale is often referred to as sweep speed in some instruments.
Scale (vertical)	Vertical scale is an instrument control that controls the y-axis or volts per division for the selected channel. This control allows you to adjust the sensitivity of the instrument.
Serializer-Deserializer (SERDES)	SERDES is a term applied to the serialization (conversion from parallel to serial) and deserialization (conversion from serial to parallel) of data. The serialization portion usually merges clock and data and performs encoding while the deserialization typically performs decoding and clock/data recovery. SERDES are commonly used in high-speed serial links.
Short	A Short is a calibration standard that is an actual line that terminates a path with a precision electrical short. See “SOLT Calibration” on page 588.
Signal-to-Noise Ratio	Signal-to-noise is the ratio of the amplitude of a signal relative to the amplitude of the noise on the signal.
Signal Integrity Engineering	Signal Integrity Engineering is using digital design and analog circuit theory along with accurate models and simulation to design circuits correctly the first time to save time and cost. When problems do occur signal integrity engineering quickly finds the root cause of signal distortions and fixes them.

Single-ended	A singled-ended or unbalanced device, having all of its signals referenced to a common ground potential.
Single-ended Device	A single-ended device has all of its signals referenced to a common ground potential.
Single-Ended Mode (SEM)	Single-ended mode is a method of sending SCSI signals along a cable. Single-ended mode uses one wire for the signal, which is compared to a common ground. The signal is the voltage difference between the two wires. Cable lengths for single-ended mode are restricted to between 6 and 1.5 meters (20 to 5 ft.) with the length decreasing as the data speed increases.
Skew	Skew changes the horizontal position of a waveform on the display independent of any other waveforms on the display. Skew is typically used for overlaying waveforms, or eliminating timing difference caused by different cable and probe lengths. The time base position control moves all of the waveforms on the display at the same time, whereas skew moves individual waveforms.
Skin Effect	Skin effect is the tendency of high-frequency currents to flow close to the surface of the conductor restricting the flow to a small part of the conductor's cross-sectional area. As frequency increases, so does the resistance and thus the loss also increases due to skin effect.
Small Computer System Interface (SCSI)	SCSI (pronounced "skuzzy") is a standardized hardware interface for a computer that acts as an I/O bus that can be used to connect the computer to several peripheral devices, such as printers, disk drives, CD-ROM, CD-R, Zip drives, and scanners.
SOLT Calibration	SOLT is a calibration using four known standards: Short-Open-Load-Thru.

Smith Chart	The Smith chart is the most common way to display complex impedance. The Smith chart is a circular chart with a bisecting horizontal line. The amount of reflection that occurs when characterizing a device depends on the impedance the incident signal sees. Since any impedance can be represented as a real and imaginary part (R+jX or G+jB), these quantities can be plotted on a rectilinear grid (known as the complex impedance plane). All values of reactance and all positive values of resistance from 0 to infinity fall within the outer circle of the Smith chart. Impedances on the chart are always normalized to the characteristic impedance of the test system. A perfect termination (Zo) appears in the center of the chart. A pure open appears at the left end of the bisecting horizontal line (infinity on the x-axis), while a pure short appears at the right end of the same line (zero on the x-axis). Loci of constant resistance now appear as circles, and loci of constant reactance appear as arcs. Inductance (positive reactance) is displayed in the upper half of the Smith chart, while capacitance (negative reactance) is displayed in the lower half.
Source	The source (input channel, function, waveform memory, or constant) used when performing tasks, such as measurements, math, or mask tests.
Standard Deviation	Standard deviation, represented by the Greek letter sigma (σ), is the measure of the dispersion or spread of the statistical average of all results for a particular measurement. In a Gaussian distribution, two sigma, or within ±1σ of the mean is where 68.3 percent of the data points reside. Six sigma, or within ±3σ of the mean is where 99.7 percent of the data points reside.
Start Frequency	Start frequency is the start point of the frequency domain measurement range, or the lowest frequency measured. Together with the stop frequency, they determine the span of the measurement range.
Stimulus	The Physical Layer Test System applies reference signals that are transmitted through the DUT or are reflected from the DUT's input. The transmitted or reflected signal is then detected and compared against the reference signal. A reference signal is called the "stimulus". A detected signal is called the response.

Stop Frequency	Stop frequency is the stop point of the frequency domain measurement range, or the highest frequency measured. Together with the start frequency, they determine the span of the measurement range.
Stripline	A stripline is a planar transmission line structure that consists of a thin conductive trace (or traces) printed or etched within an insulating substrate with parallel ground planes on both sides of the substrate.
Sweep	A sweep is the ability of the source to provide a specified signal level over a specified frequency range in a specified time period.
Sweep Cycle Time	Sweep cycle time is the time required for making a complete sweep and preparing for the next sweep. It can be measured as the time from the start of one sweep to the start of the next sweep.
Sweep Mode	Sweep mode is the way in which a sweep is initiated or selected.
Sweep Type	Sweep type is the method of sweeping the source, e.g., linear, log, or frequency step.
– T –	
T-carrier	T-carrier is a full-duplex digital transmission type that uses four wire cables. One pair is used to transmit; the other pair is used to receive. The cable types were originally twisted pairs, but now include coaxial cable, digital microwave, optical fiber, and other media. Other T-carrier characteristics are: • Symmetry: the same amount of bandwidth is provided in each direction. • Time Division Multiplexing: multiple transmissions can be supported over multiple channels by interleaving the signals over a given carrier frequency. • Unbiasing: all applications and data types are treated the same. Every bit of each transmission is treated the same, regardless of whether it's a voice bit, a data bit, or a video bit.
T1	T1 lines are a standard for broadband digital transmission over telephone lines. T3 lines consist of 24 channels at 1.544 Mbps. These lines are generally used by Internet Service Providers. See “T-carrier” on page 590.

T3	T3 lines consist of 672 channels at 44.736 Mbps. These lines are generally used by Internet Service Providers and are also referred to as DS3 lines. See “T-carrier” on page 590.
Termination	A termination is a load connected to a transmission line or other device.
Test Fixture	A test fixture is a fixture used to hold the DUT, route signals to and from the DUT, and to apply bias voltages and ground paths to the DUT.
Test Set	A test set is the arrangement of hardware (switches, couplers, connectors and cables) that connect a test device input and output to the network analyzer's source and receiver to make S-parameter measurements.
Thru	A Thru (through) is a calibration standard that is an actual through line. See “SOLT Calibration” on page 588.
Thru Reflect Line (TRL) Calibration	TRL calibration is a calibration type that utilizes three simpler, more convenient standards to define the error terms to be removed from the measurement. The measured parameters of the Thru, Reflect, and Line standards in a TRL calibration kit provides the same information as a SOLT calibration via a different algorithm. A calibration at the coaxial ports of the test system removes the effects of the system 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 SOLT standards may not be readily available to allow a conventional full 4-port calibration at the desired measurement port of the device. TRL calibration is convenient because calibration standards can be fabricated for the specific measurement environment. The characteristic impedance of these fabricated transmission lines can be determined from the physical dimensions and substrate's dielectric constant. The TRL calibration relies on the characteristic impedance of simple transmission lines.
Time Domain Network Analysis (TDNA)	TDNA includes both time domain reflectometry (TDR) and time domain transmission (TDT) measurements to characterize a network.

Time Domain Reflectometry (TDR)	TDR gives an intuitive measurement of any discontinuities in a circuit. It measures the location, electrical length, nature of the circuit (resistive, capacitive, inductive), and amount of reflection from discontinuities.
Time Domain Transmission (TDT)	TDT is a measurement technique that measures both attenuation and propagation delay of your device under test.
Topology	Topology is the way that circuits are connected to link the network nodes together. Several network topologies are listed: • Bus topology: A topology in which all nodes, i.e., stations, are connected together by a single bus. • Fully connected topology: A topology in which every node has a direct path to every other node. • Hybrid topology: A combination of any two or more other topologies. • Mesh topology: A topology in which there is a minimum of two nodes with each having a minimum of two (often more) paths. • Ring topology: A topology in which every node has exactly two branches connected to it forming a ring when all connections are made. • Star topology: A topology in which peripheral nodes are connected to a central node. The central node rebroadcasts all transmissions received from any peripheral node to all peripheral nodes on the network. • Tree topology: A topology in which multiple star topologies are connected together when their central nodes are connected to a higher level central node. In turn, this central node may be connected to other higher-level central nodes.
Touchstone	Touchstone data files, also known as .Snp files, are ASCII text files used to import and export S-parameter data. This file format displays the data 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. Comments are preceded by an exclamation mark (!). Comments can appear on separate lines, or after the data on any line or lines.

Trace	A series of data points containing frequency/time and amplitude information on a plot. In the Physical Layer Test System, a plot may have only one trace or it may be defined to have multiple traces.
Transition Time	Transition time is the time duration that a pulse takes to rise from the 10% level to the 90% level when turning on (the rise time) or the time duration that a pulse takes to fall from the 90% level to the 10% level when turning off (the fall time).
Transmission Measurements	The characterization of the transfer function of a device, that is, the ratio of the output signal to the incident signal. Most common measurements include gain, insertion loss, transmission coefficient, insertion phase, and group delay, all measured over frequency.
Twisted Pair	A twisted pair is a cable that is made up of one or more separately insulated twisted-wire pairs which reduces susceptibility to RF noise.
– U –	
Uncorrected Measurements	Uncorrected measurements are measurements made without performing calibration (error correction).
uW	See “Microwave” on page 580.
– V –	
Velocity Factor	Velocity factor is a numerical value related to the speed of energy through transmission lines with different dielectrics (.66 for polyethylene). In making time domain measurements, a velocity factor of 1 = speed of light = 299.7925×10^6 m/s.
Vertical Resolution	Vertical resolution is the degree to which an instrument can differentiate amplitude between two signals.
Via	A via is a hole filled or lined with a conducting material which is used to link two or more conducting layers of a PC board. There are blind vias (a via that connects two or more layers including the top or bottom layer), buried vias (a via that connects two or more layers that does not include the top or bottom layer), and through-hole vias (a via that connects all layers).
– W –	
Waveform	A waveform is a representation of a signal plotting amplitude versus time.

Wavelength	Wavelength is the physical distance that an electromagnetic wave travels during the time it completes one cycle. The distance between points of corresponding phase of two consecutive cycles of a wave. The wavelength (l) is related to the propagation velocity (v) and the frequency (f) by l = v / f.
Windowing	Windowing is a time domain feature that smooths (filters) overshoot and ringing displayed in time domain plots. Overshoot and ringing are caused by the abrupt transitions of start and stop frequencies used in frequency domain measurements.
– **Z** –	
Zero Level	Zero level is a measure of the mean value of the logical 0 of an eye diagram.
Zo	Zo is the characteristic impedance of a transmission line.

B 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 N8362A, N8363A, or N8364A 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).

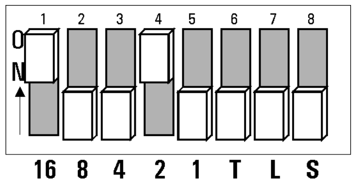
In the case of GPIB addresses that are set using the rear-panel switches, the GPIB addresses are five-bit binary numbers that are set with the switches labeled “16” through “1” (“1” is the least significant digit). The five address switches allow for 32 GPIB addresses, numbered from 0 to 31. The test 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 (either Chapter 1 or Chapter 2).

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 N8362A, N8363A, and N8364A PNA network analyzers that have firmware revisions less than Revision 3.0.

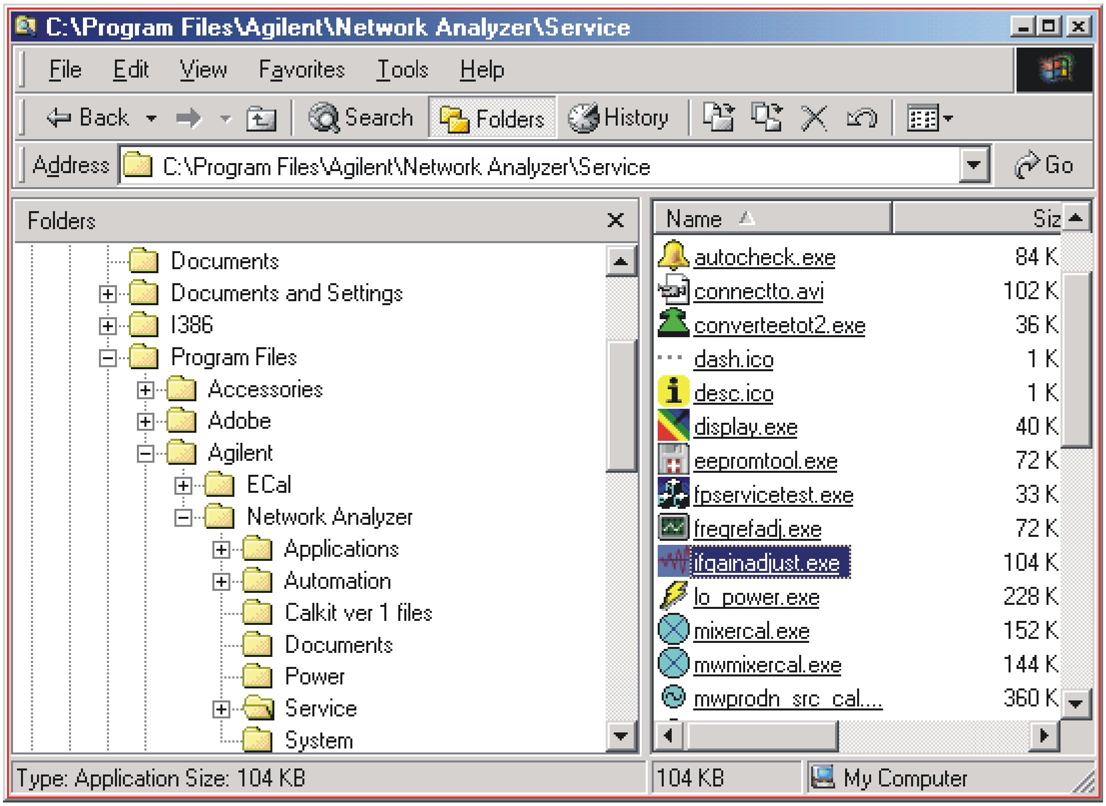
It is recommended that the Service IF Gain Adjustment test be run *before* using the test set. This routine adjusts the R Channel receivers ALC gain to ensure phase lock over the entire frequency range of the PNA Series Network Analyzer. Connect the test set to the analyzer before adjusting the IF gain.

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

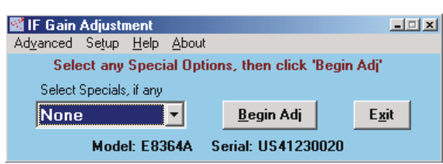
1. Close the PNA Series Network Analyzer window.
2. Open “My Computer”, located on the desk top, double click on “Hard Disk (c:).” (You may also use Explorer). Refer to Figure B-1.
3. Open the following folder path: Programs Files/Agilent/Network Analyzer/Service

Figure B-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 B-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

D

E

F

P

Q

R

S

U

V

W

Z