

User Manual



IConnect® and MeasureXtractor™ TDR and VNA Software

071-1835-00

This document supports version 3.6.

www.tektronix.com

Copyright © Tektronix, Inc. All rights reserved. Licensed software products are owned by Tektronix or its suppliers and are protected by United States copyright laws and international treaty provisions.

Parts of this software Copyright ©2000-2004 National Instruments Corporation. All Rights Reserved.

Use, duplication, or disclosure by the Government is subject to restrictions as set forth in subparagraph (c)(1)(ii) of the Rights in Technical Data and Computer Software clause at DFARS 252.227-7013, or subparagraphs (c)(1) and (2) of the Commercial Computer Software - Restricted Rights clause at FAR 52.227-19, as applicable.

Tektronix products are covered by U.S. and foreign patents, issued and pending. Information in this publication supercedes that in all previously published material. Specifications and price change privileges reserved.

TEKTRONIX and TEK are registered trademarks of Tektronix, Inc.

Contacting Tektronix

Tektronix, Inc.
14200 SW Karl Braun Drive or P.O. Box 500
Beaverton, OR 97077 USA

For product information, sales, service, and technical support:

- In North America, call 1-800-833-9200.
- Worldwide, visit www.tektronix.com to find contacts in your area.

Warranty 9(b)

Tektronix warrants that the media on which this software product is furnished and the encoding of the programs on the media will be free from defects in materials and workmanship for a period of three (3) months from the date of shipment. If any such medium or encoding proves defective during the warranty period, Tektronix will provide a replacement in exchange for the defective medium. Except as to the media on which this software product is furnished, this software product is provided "as is" without warranty of any kind, either express or implied. Tektronix does not warrant that the functions contained in this software product will meet Customer's requirements or that the operation of the programs will be uninterrupted or error-free.

In order to obtain service under this warranty, Customer must notify Tektronix of the defect before the expiration of the warranty period. If Tektronix is unable to provide a replacement that is free from defects in materials and workmanship within a reasonable time thereafter, Customer may terminate the license for this software product and return this software product and any associated materials for credit or refund.

THIS WARRANTY IS GIVEN BY TEKTRONIX WITH RESPECT TO THE PRODUCT IN LIEU OF ANY OTHER WARRANTIES, EXPRESS OR IMPLIED. TEKTRONIX AND ITS VENDORS DISCLAIM ANY IMPLIED WARRANTIES OF MERCHANTABILITY OR FITNESS FOR A PARTICULAR PURPOSE. TEKTRONIX' RESPONSIBILITY TO REPLACE DEFECTIVE MEDIA OR REFUND CUSTOMER'S PAYMENT IS THE SOLE AND EXCLUSIVE REMEDY PROVIDED TO THE CUSTOMER FOR BREACH OF THIS WARRANTY. TEKTRONIX AND ITS VENDORS WILL NOT BE LIABLE FOR ANY INDIRECT, SPECIAL, INCIDENTAL, OR CONSEQUENTIAL DAMAGES IRRESPECTIVE OF WHETHER TEKTRONIX OR THE VENDOR HAS ADVANCE NOTICE OF THE POSSIBILITY OF SUCH DAMAGES.

Contents

	Page
Chapter 1 Welcome to IConnect	1-1
IConnect Overview	1-2
What You Can Do With IConnect	1-4
IConnect Installation Notes.....	1-6
IConnect Shortcuts	1-8
Contact TDA Systems	1-9
Chapter 2 Getting Started	2-1
Preparing to Take Measurements	2-4
True Impedance Profile (Z-Line) Computation Tutorial	2-8
Eye Diagram Tutorial	2-19
S-parameter Computation Tutorial	2-37
Lumped LC Computation Tutorial	2-54
Single and Coupled Line Modeling Tutorial.....	2-63
Lossy Line Modeling Tutorial.....	2-80
MeasureXtractor™ Modeling Tutorial.....	2-97
Chapter 3 IConnect Measure-Model-Simulate Methodology	3-1
The Value of Measure-Model-Simulate	3-2
Measure-Model-Simulate within IConnect.....	3-3
Chapter 4 Measurement	4-1
Measurement Overview.....	4-2
GPIB Installation.....	4-3
Running IConnect Locally on the Tektronix TDS/CSA8XXX	4-4
Oscilloscope Setup	4-5
Configuring IConnect to Communicate with your Instrument	4-6
Waveform Acquisition	4-9
Saving and Restoring the Instrument Setup.....	4-11
Measurement Tips.....	4-13
Chapter 5 Models	5-1
Model Extraction Overview	5-3
Model Extraction Tips	5-4
Simple Source	5-5
PWL Source	5-6
Lumped.....	5-10
Lumped Coupled	5-11
Single Line.....	5-13
Symmetric Coupled Lines.....	5-19
Lossy Line	5-27
Symmetric Coupled Lossy Lines	5-36
Data Driven Models	5-44
Data Driven Single Line.....	5-52
Data Driven Coupled Lines.....	5-54
Generic 2-port.....	5-56
Termination.....	5-58
Composite Model.....	5-59
Eye Diagram Computation and Mask Testing	5-65

Partitioning Waveforms in IConnect	5-80
Chapter 6 Simulation	6-1
Verification Overview	6-2
Simulator Installation	6-3
Configuring IConnect to Communicate with your Simulator	6-4
Model Verification with Multiple Simulators	6-6
Model Verification Tips	6-8
Chapter 7 Managing and Viewing Waveforms	7-1
Using the Waveform Viewers	7-2
Waveform Selection	7-4
Waveform File Operations	7-6
Waveform Viewing Operations	7-8
Using Labels in the Waveform Viewers	7-11
Time Domain Waveform Viewer	7-14
Frequency Domain Waveform Viewer	7-16
Importing and Exporting Waveform Data	7-17
Chapter 8 Waveform Computations	8-1
Waveform Computations Overview	8-2
Waveform Filtering	8-4
Impedance Profile Computation	8-6
Step Spectrum Computation	8-9
S-Parameter Computation	8-11
Capacitance and Inductance Computation	8-14
Flip Waveform	8-19
Waveform Math	8-20
Chapter 9 Utilities	9-1
Screen Capture	9-2
E-mail Utilities	9-4
Chapter 10 User Options	10-1
General Options	10-2
Computation Options	10-3
Simulation Options	10-5
Waveform Viewer Options	10-7
Appendix 1 Troubleshooting	A-1
Appendix 2 References	A-5
Index	I-1

Environmental Considerations

This section provides information about the environmental impact of the product.

Product End-of-Life Handling

Observe the following guidelines when recycling an instrument or component:

Equipment Recycling. Production of this equipment required the extraction and use of natural resources. The equipment may contain substances that could be harmful to the environment or human health if improperly handled at the product's end of life. In order to avoid release of such substances into the environment and to reduce the use of natural resources, we encourage you to recycle this product in an appropriate system that will ensure that most of the materials are reused or recycled appropriately.



The symbol shown to the left indicates that this product complies with the European Union's requirements according to Directive 2002/96/EC on waste electrical and electronic equipment (WEEE). For information about recycling options, check the Support/Service section of the Tektronix Web site (www.tektronix.com).



Chapter 1 Welcome to IConnect

IConnect[®] and MeasureXtractor[™] is TDR and VNA software for high-speed interconnect characterization. It is an integrated measurement, modeling and model validation tool for high-speed interconnects in printed and flexible circuit boards, packages, ATE sockets, connectors, cables and cable assemblies. IConnect provides the data that you need – accurate interconnect measurements and equivalent circuit models that predict reflections, ringing and crosstalk in high-speed interconnects.

Contents

1.1	IConnect Overview.....	1-2
1.2	What You Can Do With IConnect.....	1-4
1.3	IConnect Installation Notes.....	1-6
1.4	IConnect Shortcuts.....	1-8
1.5	Contact TDA Systems	1-9

1.1 IConnect Overview

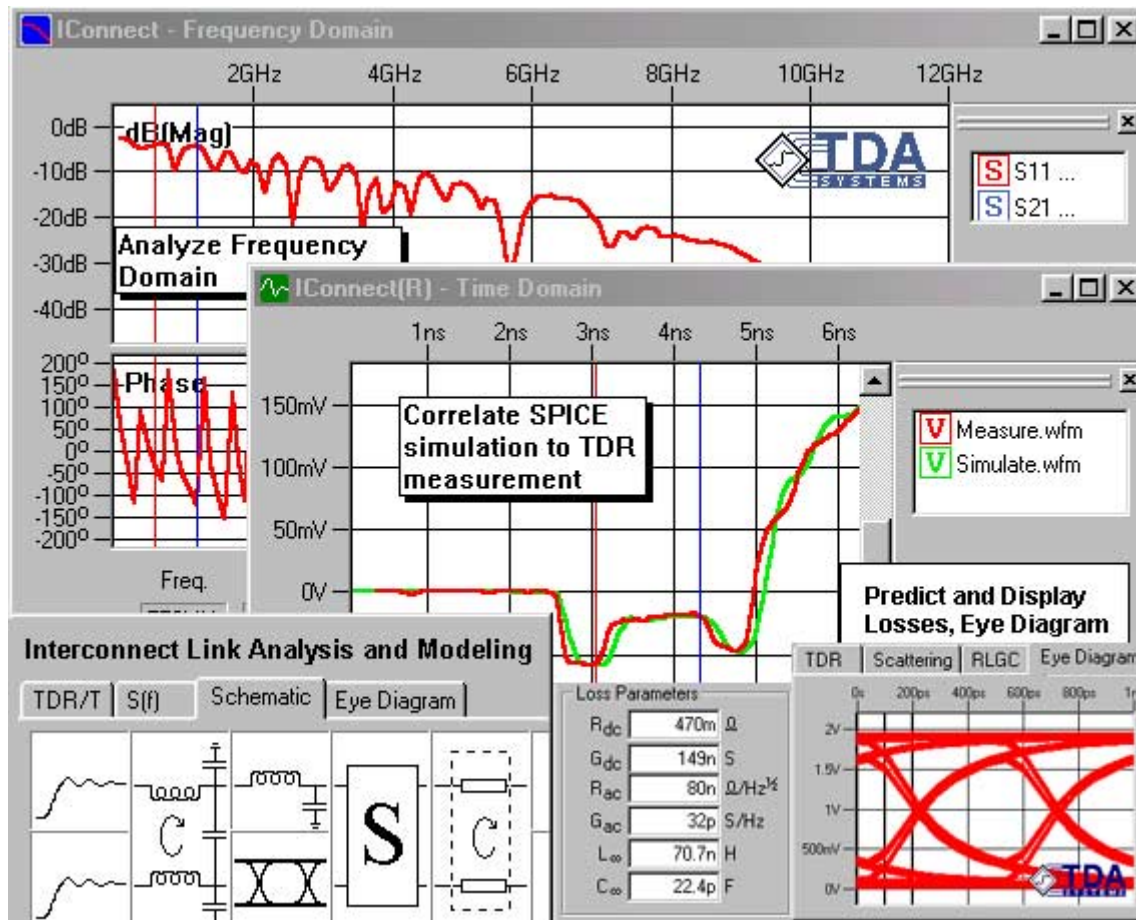


Figure 1-1: IConnect data analysis and model validation windows. IConnect's integrated measurement, model extraction, and model validation allow you to efficiently obtain accurate models and predict eye diagrams.

IConnect and MeasureXtractor TDR and VNA software from TDA Systems is the efficient, easy-to-use and cost-effective solution for measurement-based performance evaluation of gigabit interconnect links and devices, including signal integrity analysis, impedance, S-parameter and eye diagram tests, and fault isolation. IConnect provides an integrated simulate-and-compare link between SPICE/IBIS simulators and TDR/T measurements, and allows the designer to quickly extract and validate gigabit interconnect models, and to **predict eye-diagram degradation, jitter, losses, crosstalk, reflections and ringing** in PCBs and flex boards, packages, sockets, connectors, cable assemblies, and at the input die capacitance. IConnect provides simple and efficient algorithms for **computing single ended and differential S-parameters, insertion and return loss** from TDR/T measurement. The IConnect true impedance profile improves the oscilloscope resolution and accuracy, and helps **locate failures** more easily.

The new IConnect Interconnect Link Simulator allows the designer to easily analyze interconnects by **using TDR/T or S-parameter data directly in simulations** without the extra step of model extraction, to perform concurrent time and frequency domain analysis of the interconnect link performance, and to quickly perform equalizer and emphasis circuit design and analysis. The IConnect MeasureXtractor™ automatic model extraction tool then converts TDR/T or S-parameters into an **exact interconnect model**, compatible with any SPICE or IBIS simulators. These models

then allow the designer to quickly perform system level analysis of the interconnect link with transmitter and receiver. IConnect makes it easy to analyze gigabit interconnect links in various serial and parallel link standards, in computer and communications backplanes and connectors, cables and cable assemblies, disk drive flex interconnects, ATE sockets, fixtures and backplanes, BGAs and chip-scale packages, and at the die input. With IConnect, modeling and analysis tasks are accomplished quickly, resulting in faster design time, lowering design costs. IConnect® runs on Windows XP/2000/NT 4.0/95/98/2000 and interfaces to a TDR oscilloscope via GPIB or local oscilloscope interfaces. The IConnect modeling approach results in faster time to market and lower design costs.

IConnect also enables more accurate impedance measurements and improves the TDR oscilloscope resolution by removing multiple reflections from TDR waveforms. Smaller discontinuities can be resolved more easily, enabling the designer to identify the exact locations of open/short failures in multisegment interconnects. Impedance can be displayed versus time or physical distance on the x-axis, which can be important for failure analysis applications.

Measurement-based Modeling

The IConnect software package is built upon the concept of measurement-based modeling. This approach to modeling begins with measurement of the characteristics of a device, and a model is extracted from the measured data. After extraction, the model response is verified by comparison against the data from which the model was extracted. IConnect enhances productivity by simplifying these measurement, modeling, and verification tasks, and by integrating them into a single comprehensive software package.

Measurement

IConnect contains a seamless interface for data transfer from several supported measurement instruments. Supported instruments include Tektronix and Agilent (formerly Hewlett-Packard) oscilloscopes and the Agilent 8510 network analyzer operating in time-domain mode. Your computer's hard drive is also considered to be a measurement instrument, and may be used to store measured waveforms, transfer data from other formats, or even create new waveforms. Online documentation and application information are available to assist the user in determining the necessary measurements to extract a particular model, and telephone and e-mail support are available if desired. TDA Systems will also assist IConnect users in setting up their own measurement systems and making accurate measurements.

Modeling

IConnect contains many different models which are applicable to typical interconnect structures. Each of these models is based on parameters which can be extracted directly from time-domain measurements using the algorithms available from within IConnect. The models are saved as SPICE files for simple direct evaluation using SPICE-based simulators. Online documentation and application information are available to enable our customers to become accustomed to what type of model is appropriate to represent a particular interconnect structure.

Verification

IConnect supports direct and simple verification of a model against measured data by directly linking with a simulator to evaluate the model and compare its response against measured data. The currently supported simulators are PSpice from Cadence, HSpice from Avant!, and Berkeley SPICE 3F from the University of California. A large amount of flexibility is available so that models can be verified against a variety of excitation and termination conditions without having to leave the IConnect framework.

1.2 What You Can Do With IConnect

IConnect is the efficient, easy-to-use and cost-effective solution for measurement-based performance evaluation of gigabit interconnect links and devices, including signal integrity analysis, impedance, S-parameter and eye diagram tests, and fault isolation.

Signal Integrity

IConnect® TDR and VNA software: efficient and easy signal integrity analysis for gigabit interconnects

- Easy SPICE/IBIS modeling for PCB, flex board, connector, cable, package, socket
- Simple differential and single ended S-parameters, frequency dependent losses
- Integrated simulate-and-compare link between SPICE/IBIS simulators and TDR/T
- More accurate impedance measurements
- Cost-effective eye diagram testing including crosstalk

IConnect® Link Simulator: efficient and easy interconnect link simulation

- Easily analyze interconnects by using TDR/T or S-parameters directly in simulations
- Quickly predict eye diagram, jitter, losses, crosstalk, reflections, ringing
- Concurrent time and frequency domain interconnect link analysis
- Easily perform equalizer and emphasis design and analysis
- Efficiently validate analytical and field solver models

IConnect® MeasureXtractor™: automatic tool for exact model extraction

- Automatically convert TDR/T or S-parameter data into exact frequency dependent models
- Easily ensure SPICE / IBIS simulation accuracy by using exact frequency dependent models
- Easily perform system level analysis of the interconnect link with transmitter and receiver

Failure Analysis

- Resolving smaller discontinuities
- Locating exact position of open, short and “soft” faults
- Easy data manipulation and analysis

IConnect utilizes a standard Windows XP/2000/NT4.0/98/95 interface. IConnect can also run locally on the Tektronix TDS/CSA8000 and TDS/CSA8200 oscilloscopes, so you will not need to buy an additional lab computer.

We can provide you with a complete TDR solution for your interconnect signal integrity and failure analysis, including IConnect TDR and VNA software, an Agilent or Tektronix TDR instrument, recommendations on probing, cabling and fixturing solutions, and high-quality TDR applications support. To ensure the success of our customers, we offer optional on-site product training. We are confident that IConnect TDR and VNA software and TDA Systems interconnect signal integrity analysis expertise will enable you to significantly improve your interconnect signal integrity analy-

sis, impedance, S-parameter and eye diagram tests, design and model validation, and fault isolation results.

1.3 IConnect Installation Notes

Minimum System Requirements

- 400 MHz Pentium™ Processor
- 256 MB of RAM (1 GB of RAM may be necessary for large MeasureXtractor runs)
- 40 MB Hard Drive free space
- National Instruments GPIB board, version 2.1 (not required for Tektronix TDS8XXX or CSA8XXX local VISA interface)
- Windows 95, 98, NT 4.0, 2000, or XP
- 1024 x 768 monitor resolution

Installation Overview

The following components are installed as outlined in this section:

- IConnect (Includes Berkeley SPICE 3F)
- Sentinel hardware key drivers

To get started using IConnect®, we suggest working through the online tutorials, which are accessible via the **Help/Getting Started...** menu command. The [“True Impedance Profile \(Z-Line\) Computation Tutorial” on page 2-8](#) is a standalone section for those using IConnect® to improve impedance measurement accuracy. The [“S-parameter Computation Tutorial” on page 2-37](#) shows you how to obtain differential, single-ended and mixed mode S-parameter data, insertion and return loss. [“Eye Diagram Tutorial” on page 2-19](#) guides you through the steps necessary to perform eye diagram electrical standard compliance testing and eye mask testing. The [“MeasureXtractor™ Modeling Tutorial” on page 2-97](#) demonstrates how to generate a behavioral SPICE model with the least amount of effort. The [“Lossy Line Modeling Tutorial” on page 2-80](#) guides you through extraction and verification of a single-ended or differential lossy transmission line model, and [“Single and Coupled Line Modeling Tutorial” on page 2-63](#) demonstrates techniques for using impedance profile to model electrically short or lossless structures. The [“Lumped LC Computation Tutorial” on page 2-54](#) illustrates the JEDEC standard-based techniques for modeling electrically short interconnects, such as packages or sockets. Finally, the “Failure Analysis Using TDR” tutorial, accessible via the **Help/Getting Started...** menu, is available for failure analysts who use IConnect as a tool to assist in electrical fault location.

ICONNECT/HARDWARE KEY DRIVER INSTALLATION

1. Place the IConnect CD in the CD-ROM drive. The IConnect setup program should automatically start. If it does not, go to your CD-ROM drive using Windows Explorer and double-click the Setup.exe icon. Note that if you have filename extensions hidden in windows explorer, it is the file named “Setup” whose icon is a computer and floppy disks.
2. Follow the setup instructions to install IConnect. As part of the setup wizard, the Sentinel hardware key drivers are also installed.
3. Plug the parallel-port hardware key into your computer’s parallel port, or the USB hardware key into your USB port, prior to starting IConnect. If you have another peripheral device attached to your parallel port, plug it into the back of the hardware key.

Note: You are given the choice of installing the hardware key drivers either automatically or interactively. Interactive installation allows configuration as well. However, automatic installation should be effective in most cases.

ADOBE ACROBAT READER

If you wish to read or print either the Berkeley SPICE manual or the PDF version of the IConnect written manual, you will need to install Adobe Acrobat Reader if it is not already present on your system. You may accomplish this by navigating to the directory **PDF Manual** of your IConnect installation CD, and double clicking on the file **Acrobat Reader 6.01 Install.exe**. Please note, however, that Acrobat is NOT necessary to view the IConnect online help.

MANUAL SENTINEL HARDWARE KEY DRIVER INSTALLATION/REMOVAL/CONFIGURATION

Although the IConnect setup wizard will automatically install the Sentinel hardware key drivers, it may at some point be desirable to manually install, uninstall, or reconfigure the drivers. You may be required to have Administrator privileges to install the drivers.

From the TDA Systems folder on the **Start** menu, (or the folder in which you chose to install IConnect) choose **Sentinel Drivers Configuration**. This program allows you to either install, uninstall, or configure the drivers as necessary. Click the **Add/Repair/Remove Installation** button to add features, repair a broken installation, or remove the installed drivers. Click the **Configure Driver** button to configure the Sentinel hardware key drivers. Help on configuring the drivers is available by clicking the **Help** button inside of the **Configure Driver** dialog.

1.4 IConnect Shortcuts

This keyboard shortcut listing has been created to assist you with many of the short cuts available to IConnect Software users.

General	
Open IConnect model and/or waveform files	Drag files over IConnect window and drop them
Copy active window to clipboard as bitmap	Ctrl+Shift+A
Copy main window to clipboard as bitmap	Ctrl+Shift+M

Waveforms	
Select All	Ctrl+A
Waveform Context Menu (single or multiply selected waveforms)	Right click on a waveform
Nudge Waveform	Arrow Key (If waveform leg end highlighted in blue, hit tab first
Nudge Waveform Faster	Shift+ Arrow
Move Waveform	Drag down while holding left mouse button down
Constrain Motion to Horizontal or Vertical	Shift + Drag
Select Single Waveform	Single Left click
Add Waveform to Selection	Ctrl+left click
Waveform Properties (single or multiple selection)	Right click, choose Properties
Single Waveform Properties	Left double click
Delete Selected Waveforms	Delete

Labels	
Move	Drag while holding left mouse button down
Select	Left click
Multiple Selection	Ctrl+left click
Edit Text	Click on selected label
Special Text	%name is the name of the associated waveform if any
Delete Selected Labels	Delete
Label properties	Right click on label
New Label	Right click in waveform viewer

Waveform Viewer	
Context Menu	Right click in open area
Save or Save As	File menu or waveform context menu
Zoom	Left click in an open area and drag a rectangle around what you want to see OR Hold down Shift+Ctrl in any area and drag a rectangle
Zoom to 100%	Context menu or view menu- scales plot to remove scrollbars
Autoscale	Context menu or view menu- scales plot to fit all waveforms
Zoom In	Context or view menu- zooms in by 2x
Zoom Out	Context or view menu- zooms out by 2x

1.5 Contact TDA Systems



TDA Systems is committed to providing convenient, quality technical support for its products. If you have any questions or suggestions regarding IConnect or measurement-based modeling, please feel free to contact us by telephone, E-mail, or internet at:

(503) 246-2272 (phone)

(503) 246-2282 (fax)

iconnect-support@tektronix.com

http://www.tek.com/products/oscilloscopes/sampling/interconnect_analysis/index.html

We look forward to hearing from you!



Chapter 2 Getting Started

This chapter contains seven tutorials to help get you started using IConnect[®] and MeasureXtractor[™] TDR and VNA software.

[“S-parameter Computation Tutorial” on page 2-37](#) will lead you through using your TDR oscilloscope to take accurate single-ended, differential, and mixed model S-Parameter measurements.

[“True Impedance Profile \(Z-Line\) Computation Tutorial” on page 2-8](#) is a standalone tutorial for users interested in obtaining true impedance readouts for analysis, but with no interest in developing SPICE models. Users intending to create SPICE models should start with the [“Single and Coupled Line Modeling Tutorial” on page 2-63](#), which outlines how to convert true impedance information into a validated single-ended or differential SPICE model.

The [“Lossy Line Modeling Tutorial” on page 2-80](#) explains how to extract an accurate model for a lossy transmission line or differential pair in IConnect, and the [“MeasureXtractor[™] Modeling Tutorial” on page 2-97](#) introduces TDA Systems’ MeasureXtractor[™] automated modeling technology with one-click model extraction capability.

Finally, the [“Eye Diagram Tutorial” on page 2-19](#) shows how to compute an eye diagram based on TDT measurements without extracting a model, and the [“Lumped LC Computation Tutorial” on page 2-54](#) shows how to quickly and accurately extract lumped element models for electrically small devices.

Please [contact TDA Systems](#) if you have any questions about the tutorials.

Contents

2.1	Preparing to Take Measurements	2-4
2.2	True Impedance Profile (Z-Line) Computation Tutorial.....	2-8
2.2.1	What is True Impedance Profile?.....	2-9
2.2.2	True Impedance Profile in Failure Analysis	2-10
2.2.3	Measurements for Impedance Profile Analysis.....	2-11
2.2.4	DUT connections	2-12
2.2.5	Setting up the Impedance Profile.....	2-13
2.2.6	Computing the Impedance Profile.....	2-14
2.2.7	Using Cursor Readout	2-15
2.2.8	Using the Zoom Tool.....	2-16
2.2.9	Using Impedance Averaging.....	2-17
2.2.10	Using Differential Impedance Profile.....	2-18
2.3	Eye Diagram Tutorial	2-19
2.3.1	Waveform Acquisition and Setup.....	2-20
2.3.2	Choosing the Proper Termination	2-22
2.3.3	Selecting a Reference.....	2-23
2.3.4	DUT Connections.....	2-24
2.3.5	Loading Data.....	2-25
2.3.6	Display the Eye	2-27
2.3.7	Eye Diagram Options.....	2-28
2.3.8	Display the Final Eye	2-30
2.3.9	Eye Diagram Zooming	2-31
2.3.10	Eye Mask Testing	2-32
2.3.11	Measure Jitter	2-33
2.3.12	Correlation	2-34
2.3.13	System Eye Diagram	2-36
2.4	S-parameter Computation Tutorial.....	2-37
2.4.1	What are S-Parameters?	2-38
2.4.2	Waveform Acquisition Setup.....	2-39
2.4.3	S-parameter Data Acquisition	2-41
2.4.4	DUT Connections.....	2-42
2.4.5	Differential, Common, and Mixed Modes.....	2-43
2.4.6	Compute S-parameters.....	2-45
2.4.7	S-Parameter Options	2-46
2.4.8	Calibration.....	2-47
2.4.9	Choosing the Data Display	2-48
2.4.10	Differential Example.....	2-49
2.4.11	Exporting Touchstone.....	2-51
2.4.12	Correlation	2-53
2.5	Lumped LC Computation Tutorial	2-54
2.5.1	Waveform Acquisition and Setup.....	2-55
2.5.2	Measurement Setup Tips.....	2-56
2.5.3	Self Inductance	2-57
2.5.4	Self Capacitance.....	2-58
2.5.5	Mutual Inductance.....	2-59
2.5.6	Mutual Capacitance	2-60
2.5.7	I/O Buffer Input Capacitance.....	2-61
2.5.8	Simulate	2-62
2.6	Single and Coupled Line Modeling Tutorial.....	2-63
2.6.1	Waveform Acquisition and Setup.....	2-64
2.6.2	DUT Connections.....	2-65

2.6.3	Loading Data.....	2-66
2.6.4	Computing Z-Line	2-68
2.6.5	Windowing the Data.....	2-69
2.6.6	Partitioning Details	2-70
2.6.7	Other Practical Details	2-71
2.6.8	Symmetric Coupled Model Details	2-72
2.6.9	Saving the Model	2-74
2.6.10	Simulating	2-76
2.6.11	PWL Source.....	2-77
2.6.12	Analyzing the Results	2-78
2.6.13	Risetime Filtering	2-79
2.7	Lossy Line Modeling Tutorial	2-80
2.7.1	Waveform Acquisition and Setup.....	2-81
2.7.2	Choosing the Extract Termination Type.....	2-82
2.7.3	DUT Connections.....	2-83
2.7.4	Loading Data.....	2-84
2.7.5	Model Extraction	2-86
2.7.6	Model Accuracy	2-87
2.7.7	Other Practical Details	2-88
2.7.8	Choosing the Output Format.....	2-89
2.7.9	Saving the Model	2-90
2.7.10	Simulating	2-92
2.7.11	PWL Source.....	2-93
2.7.12	Analyzing the Results	2-94
2.7.13	Eye Diagram	2-96
2.8	MeasureXtractor™ Modeling Tutorial	2-97
2.8.1	Waveform Acquisition and Setup.....	2-98
2.8.2	Choosing the Extraction Termination Type.....	2-99
2.8.3	DUT Connections.....	2-100
2.8.4	Loading Data.....	2-101
2.8.5	Model Extraction	2-103
2.8.6	Model Accuracy	2-104
2.8.7	Saving the Model	2-105
2.8.8	Simulating	2-107
2.8.9	PWL TDR Source	2-109
2.8.10	Analyzing the Results	2-110

2.1 Preparing to Take Measurements

Good measurement practices

It is important to follow good measurement practices when working with your 18-20 Ghz TDR oscilloscope, such as:

- Let the instrument warm up for 20-30 minutes before performing measurements
- Use good quality low loss cables and good probes with small signal-to-ground spacing
- Use a lot of averaging in the TDR oscilloscope to reduce noise. 128 averages will give an extra 20dB of dynamic range; use at least that many if your instrument averages quickly.
- De-skew the channels in the TDR oscilloscope while doing differential measurements (for coupled models). Follow your oscilloscope de-skewing procedure before beginning acquiring differential data.

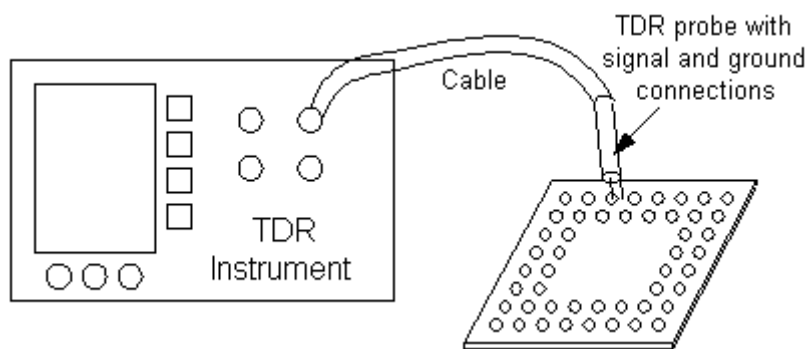


Figure 2-1: Package and TDR.

How to connect to the measurement instrument

You need to make sure that you have a means to acquire the data and communicate with the instrument. You can acquire the data either via GPIB from a computer with a National Instruments GPIB card connected to the instrument, or through a local interface with the IConnect software running on the PC-based instrument itself. You also need to make sure that the software settings match the physical settings of the oscilloscope. These settings can be made visible by clicking on **View/Acquisition Window** in IConnect as shown in Figure 2-2 below.

Acquiring data

In the Acquisition window **Instrument** drop-down list, select the oscilloscope you intend to use, or select the Virtual Scope to follow this tutorial and acquire the data from the hard drive.

If you have chosen **Virtual Scope**, you can click on the **Instrument Settings** tab to select the directory from which the data will be acquired, as shown in Figure 2-3.

If you have chosen an oscilloscope from the **Instrument** drop-down, the **Instrument Settings** tab shown in Figure 2-4, allows you to set the software parameters. You need to make sure that these parameters, such as the instrument GPIB Address, match the physical settings of the oscilloscope and of your GPIB card, and that the instrument is ready to communicate via GPIB or local interface. Typically, the GPIB board index is set to 0, and the address must match the GPIB address of your instrument. To change these settings properly, it may be necessary to consult your instrument or GPIB board manual.

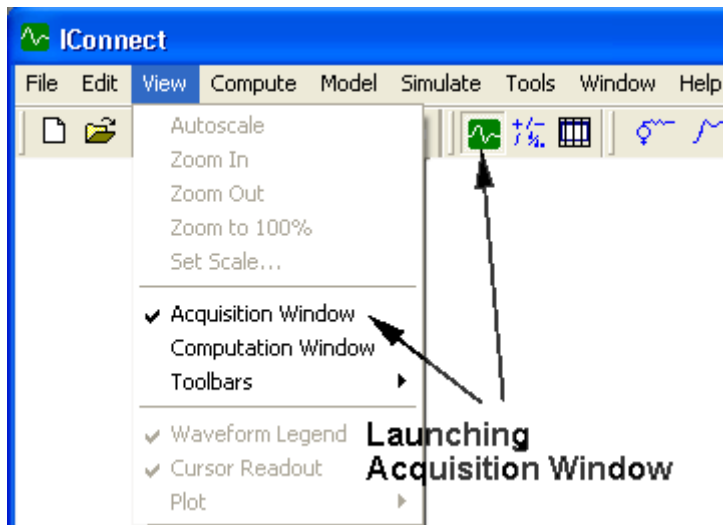


Figure 2-2: Displaying the Acquisition window.

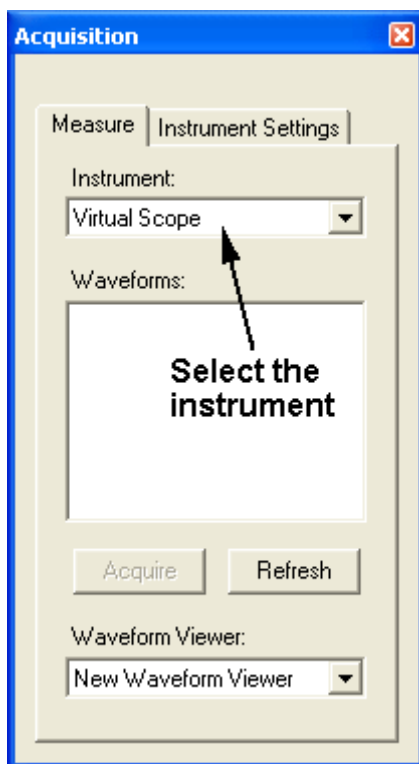


Figure 2-3: Acquisition Window (Measure tab) and Parameters (Instrument Settings tab).

Acquisition Window

Keep in mind that if you are extracting a model, you must acquire all the required waveforms - for example, reference, reflection and transmission - without changing the timebase on the TDR oscilloscope. The first waveform transition, corresponding to the interface from the sampling head to the cable, must be windowed out. When you look at an open reference waveform, you see essentially a two-step staircase. The first step in the staircase must be windowed out, as shown in Figure 2-5, and the measurement must be zoomed in on the DUT, as shown in Figure 2-6.

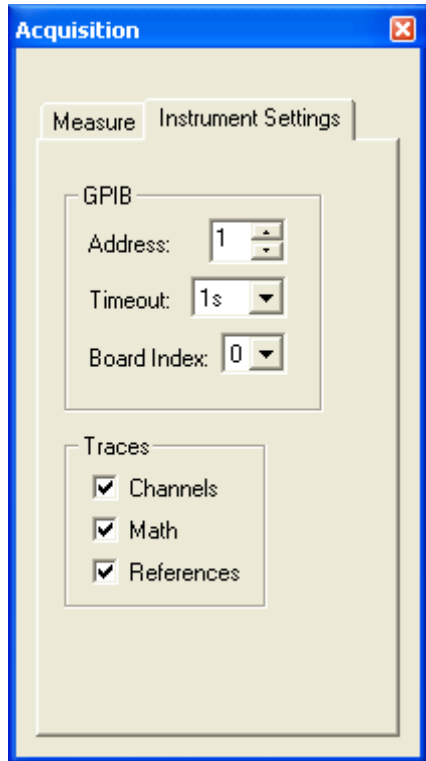


Figure 2-4: Updating the software Instrument Settings to match the physical settings of the oscilloscope.

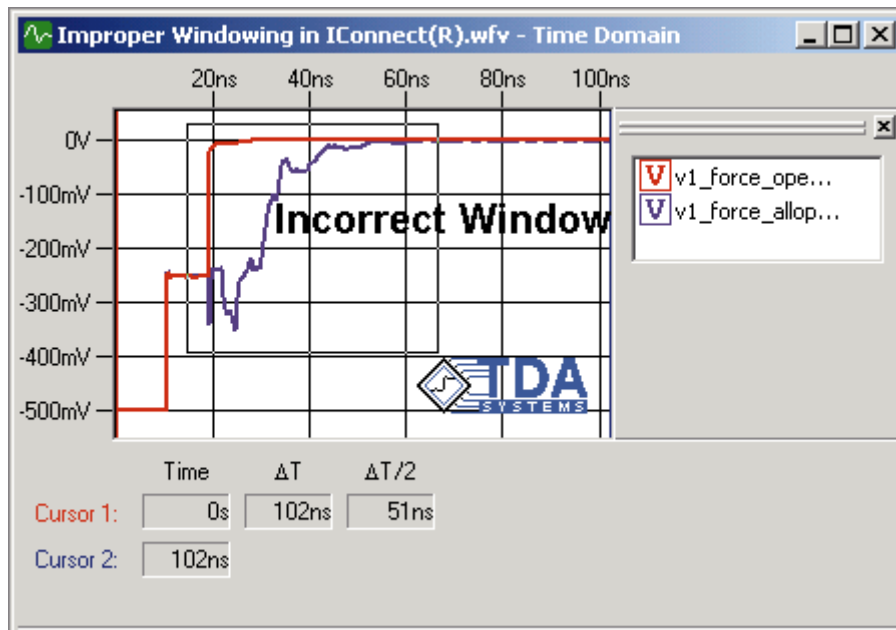


Figure 2-5: Incorrect Acquisition Window. You do not want to capture the first transition in your TDR waveforms

For more measurement tips and suggestions on good measurement practices, see ["Measurement Tips"](#) on page 4-13.

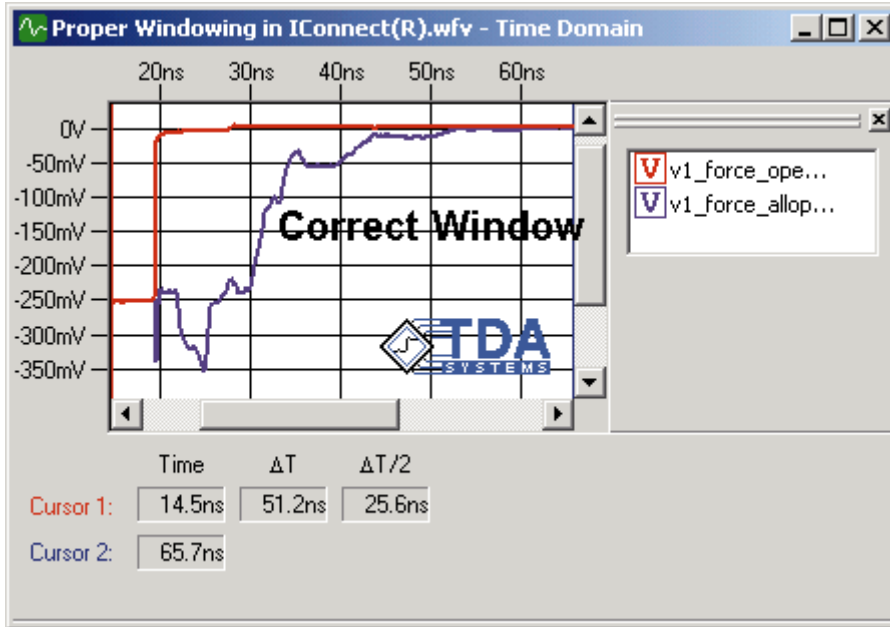


Figure 2-6: Correct Acquisition Window.

2.2 True Impedance Profile (Z-Line) Computation Tutorial

Welcome to the IConnect True Impedance Profile (Z-Line) Computation Tutorial. This tutorial guides you through the steps necessary to compute the single ended and differential impedance profile of a device under test (DUT). The IConnect True Impedance Profile computation employs the powerful “peeling algorithm,” and allows you to substantially improve the impedance measurement accuracy for the devices with impedance discontinuities. The tutorial leads you from a set of raw measurements to the true impedance profile, and allows you to evaluate the impedance of your device accurately, quickly and efficiently.

2.2.1 What is True Impedance Profile?

The impedance profile is the characteristic impedance of a PCB trace, package lead, or other transmission-line-type structure, as a function of distance. Because transmission lines with impedance discontinuities tend to have multiple reflections that occur between the discontinuities which superpose on each other, a post processing algorithm such as that contained in IConnect is necessary to determine the true impedance profile of the device. The impedance display function in TDR oscilloscopes does not take these multiple reflections into account, and therefore the TDR oscilloscope impedance reading for a multi-segment interconnect may have significant error.

Although the impedance profile is used within IConnect as a starting point for signal integrity modeling, it is also useful in its own right as an analysis tool to observe the type and position of discontinuities along a line.

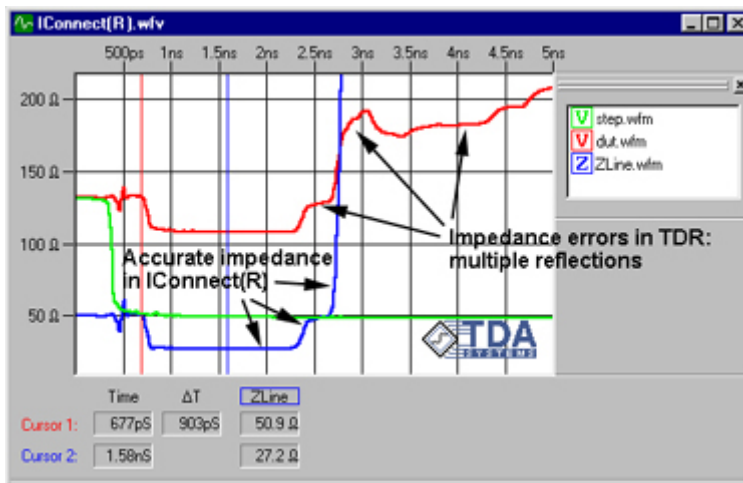


Figure 2-7: True impedance profile example.

2.2.2 True Impedance Profile in Failure Analysis

Using a true impedance profile, and not a raw TDR waveform as acquired by a TDR oscilloscope, is critical to accurately pinpoint open and short failures in packages, boards, connectors and cables. In this example, we see how the raw TDR waveform (green trace in the picture), due to the presence of the multiple reflections in the trace, gives us a confusing indication of where the failure may have occurred. The true impedance profile (red waveform), on the other hand, pinpoints the failure location precisely and unequivocally.

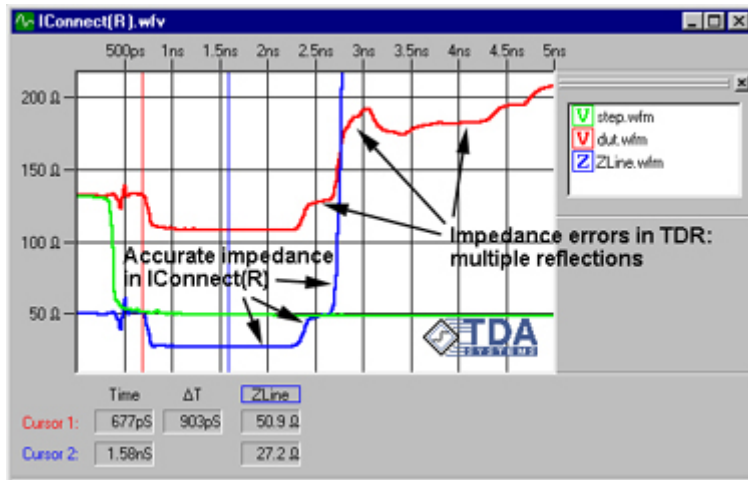


Figure 2-8: True impedance profile is a very useful tool in failure analysis applications.

2.2.3 Measurements for Impedance Profile Analysis

You should begin by reviewing [“Preparing to Take Measurements” on page 2-4](#) if necessary. It discusses good measurement practices as well as how to get your scope communicating with IConnect.

To follow our example, select Virtual Scope in the drop-down list at the top of the acquisition window. The virtual scope is a tool for loading waveforms from your hard drive using the same interface as that used for oscilloscopes.



Figure 2-9: IConnect virtual scope for reading waveforms from disk

Press the **Instrument Settings** tab at the top of the acquisition window to set the Virtual Scope options, as shown in Figure 2-9. Set the directory to **Tutorials\Single Line**, under the IConnect installation directory, e.g. **C:\Program Files\TDA Systems\IConnect**. Set the filter to ***.wfm**, and press the **Measure** tab to begin taking measurements. Select the waveforms **DUT.wfm** and **Short.wfm** from the acquisition window. Press the **Acquire** button, which opens up a new waveform viewer containing the two waveforms.

2.2.4 DUT connections

For single line modeling, TDR waveforms must be acquired with only channel M1 connected to the DUT. The termination on the far end of the DUT is not important, and can be a short, an open, matched (50Ω) or any other impedance value.

In differential or common mode, both transmission and reference measurements must be differential (that is, the negative switching channel is subtracted from the positive switching channel) or common mode (the two positive switching channels are added together).

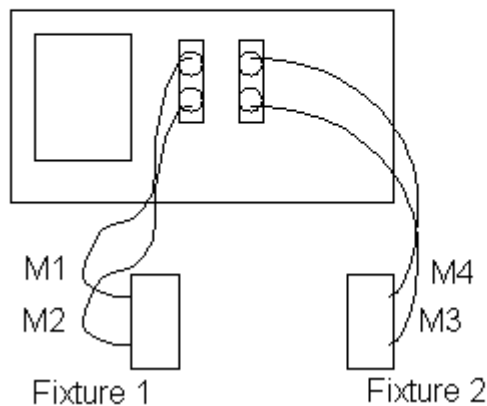
For a symmetric interconnect, we can use odd and even mode waveforms, acquired on the positive switching channel with the stimulus being in differential (for odd mode data) or common mode (for even mode data).

It is important to de-skew the TDR oscilloscope channels while doing differential measurements. Follow your oscilloscope de-skewing procedure before acquiring differential data.

Reference measurement:

Differential Stimulus is ON on channels M1, M2.

Reflection is measured on the same channels M1-M2



Reflection and transmission measurements:

Differential Stimulus is ON on channels M1, M2.

Transmission is measured on channels M3-M4

Reflection is measured on the channels M1-M2

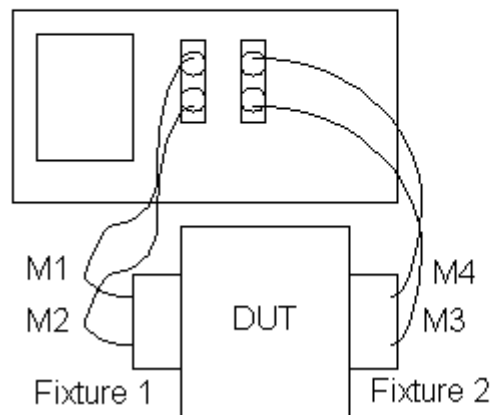


Figure 2-10: Measurement setup for differential reference, reflection, and transmission.

2.2.5 Setting up the Impedance Profile

Begin computing the profile by making the computation window visible by pressing the button with the mathematical symbols from the auxiliary toolbar. The computation window is easily identified by the dropdown list box marked **Compute** at its top.

The computation window may be docked on either side of the main IConnect window, or it may be allowed to float inside or outside the main window. Go to the **Compute** drop-down list box on the computation window and select **Z-Line** for impedance profile.

Select the name of the waveform viewer containing the waveforms from the **Waveform Viewer** drop-down list box. In the **Waveforms** group of list boxes, select the response (reflection) from the **DUT** list box, and select TDR input step (short or open can provide an equally good quality reference) from the **Step** list box.

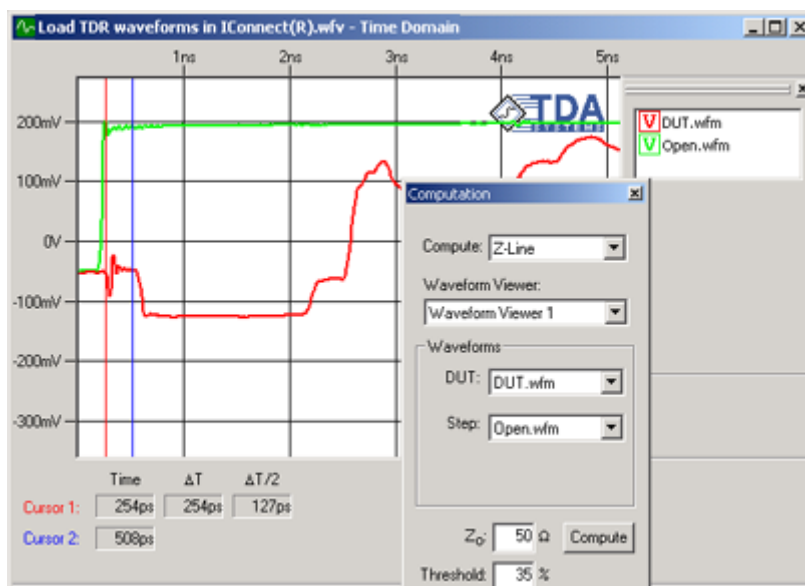


Figure 2-11: Selecting the waveforms for the impedance profile computation.

2.2.6 Computing the Impedance Profile

At the bottom of the computation window, set Z_0 to 50Ω and **Threshold**: to 35% if necessary. These are default values which do not need to be changed under normal operating conditions. Z_0 corresponds to the reference impedance value at the left-most point in the acquisition window, which should be someplace in the middle of your 50Ω cable connecting the DUT to the oscilloscope. If it is not, read the impedance value for that right most point on the scope first, and then enter it into the Z_0 box in IConnect. Threshold is a noise and measurement error reduction parameter which will usually be set between 25% and 75%. If your measurements are noisy or your fixture has large discontinuities, you may need to increase this parameter. Refer to “[Impedance Profile Computation](#)” on page 8-6 of the IConnect reference manual for further information on threshold.

Once all computation parameters have been set as shown in the picture, press the **Compute** button, which computes the impedance profile and places it in the waveform viewer.

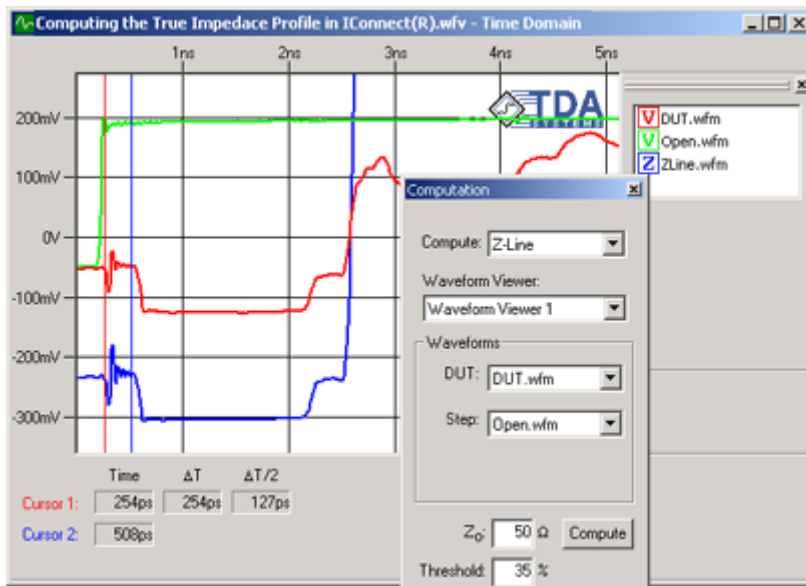


Figure 2-12: After the impedance profile is computed, it is displayed in the waveform viewer.

2.2.7 Using Cursor Readout

Right-click on the reflection waveform (or on its name in the waveform legend window) and select **Cursor Readout**. Move the cursors with the mouse to view waveform data. You can also inspect the computed capacitance and inductance values between the cursors.

If you prefer to see the horizontal scale and readouts in distance units rather than time units, you may do so by right-clicking in an empty area of the waveform viewer and choosing **Plot/Versus Distance** from the resultant context menu.

To set the dielectric constant, choose the **Tools/Options...** command from the main menu, select the **Waveform Viewer** tab, and set ϵ_r to the appropriate value.

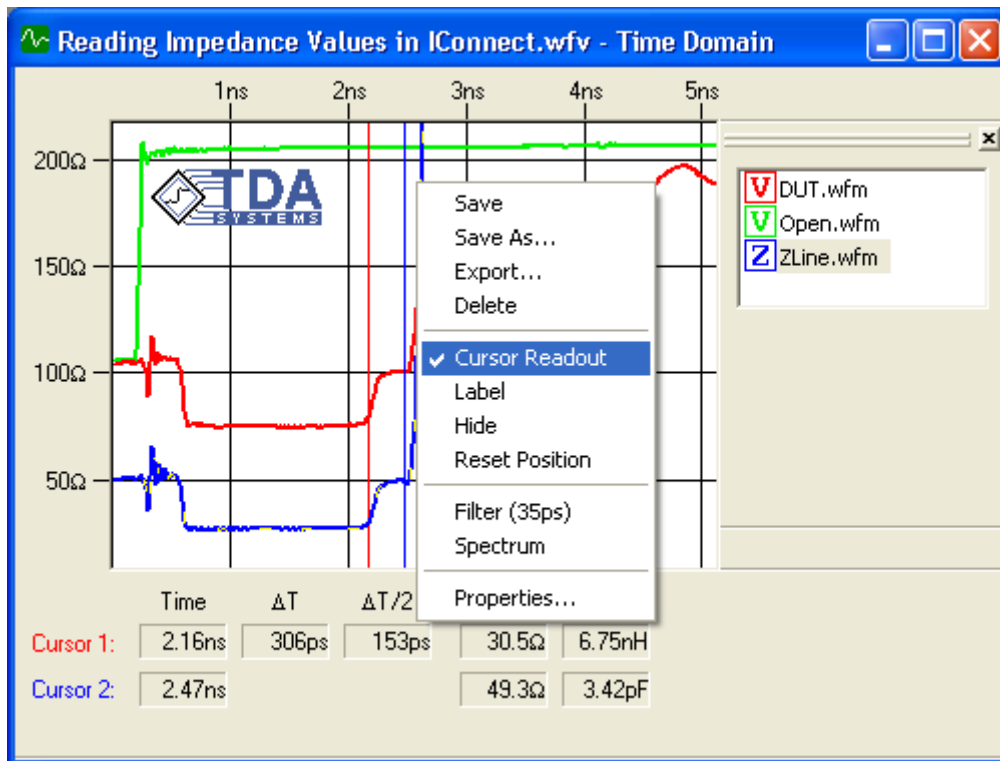


Figure 2-13: Displaying cursor readout for the computed impedance profile waveform.

2.2.8 Using the Zoom Tool

To inspect an area of the waveform more closely, such as the connector-induced glitch at 500 ps, hold the left mouse button down on an empty area of the display and drag a rectangle over the area to which you would like to zoom, as shown here. Alternatively, you can right-click in an empty area of the display, and select a desired zoom level or other action from the Context Menu.

Make sure that the rectangle you draw is large enough to cause the zoom to occur. When you are finished inspecting the glitch, right-click in the window and choose **View/Zoom to 100%** to return to the original view.

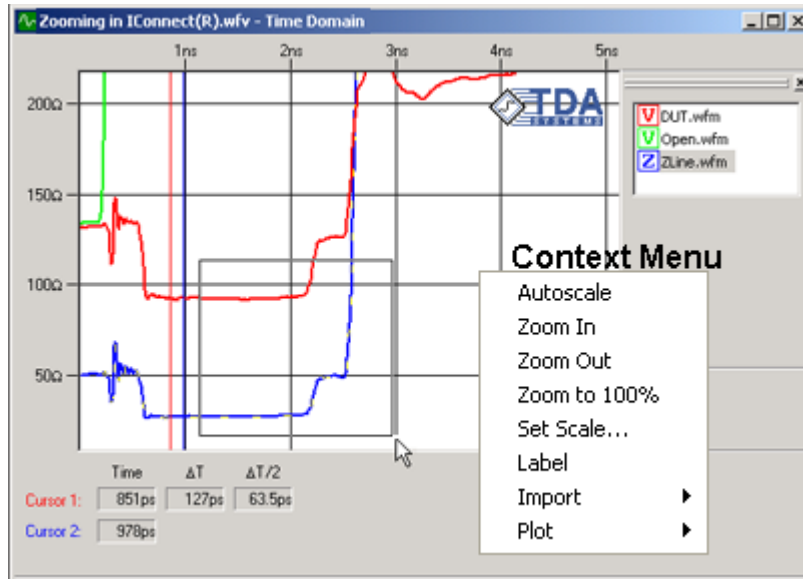


Figure 2-14: You can zoom in by dragging a zoom rectangle, and zoom out via the context menu.

2.2.9 Using Impedance Averaging

The easiest way to read average, minimum, and maximum impedance values in IConnect is by using the Single Line Modeling window, which can be opened from either the **Model** menu or the **Model** toolbar.

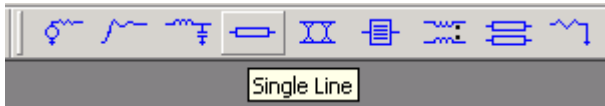


Figure 2-15: Creating a Single Line model from the model toolbar.

Then, load a pre-computed impedance profile into the single-line modeling window, use partitions to select the segment of interest, and select either **Min**, **Max**, or **Mean** value computation to get the corresponding impedance value for the given segment. For more information on single-line modeling window, see the [“Single and Coupled Line Modeling Tutorial”](#) on page 2-63.

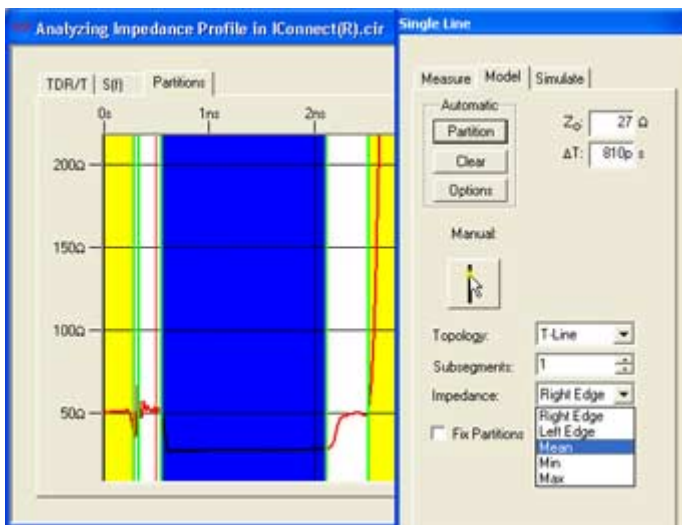


Figure 2-16: Using the Single Line model viewer to measure average impedance.

2.2.10 Using Differential Impedance Profile

If you are interested in acquiring a differential impedance profile, you need to follow these additional steps: In the case where the two lines in the differential pair are symmetric, you stimulate the differential pair with a differential stimulus in the TDR oscilloscope, and acquire the waveform on the positive switching channel (Ch1). No math in the oscilloscope is required (this is the odd mode impedance). You acquire a reference open or a short waveform on the same channel, and use it to compute the odd mode impedance profile. Then, you multiply it by 2 to get the differential impedance. Do not use the negative channel waveforms, as they do not give you correct impedance profile.

Alternatively, when you compute the impedance profile, change the characteristic impedance value (Z_0 , right above Threshold parameter in the impedance computation window) to 100Ω from 50Ω , and you will get the differential impedance right away, bypassing the odd mode impedance step.

For the case of differential lines with mild asymmetry, proceed as follows: When the TDR is in differential mode, and the oscilloscope displays voltage, subtract the negative switching channel (Ch2) from the positive switching channel Ch1 ($DUT=Ch1-Ch2$). Then, acquire a reference on Ch1 and multiply it by 2 (or, acquire reference on both channels simultaneously and subtract $Ch1Ref-Ch2Ref$ again). Now, use the **DUT** waveform and the computed reference waveform to compute the impedance profile and multiply it by 2, or use characteristic impedance $Z_0 = 100\Omega$ and simply compute the impedance profile. This lengthened process allows you to account for mild non-symmetry in the lines. In any case, never use the negative switching channel by itself - IConnect does not understand it and does not treat it properly.

2.3 Eye Diagram Tutorial

Welcome to the IConnect Eye Diagram Tutorial. This tutorial guides you through the steps necessary to compute an interconnect's eye diagram from a TDR oscilloscope transmission measurement. The eye diagram is a common figure of merit for many interconnect standards and signaling schemes. The tutorial leads you from a set of raw measurements to the eye diagram for your device under test (DUT), and allows you to evaluate the impedance of your device accurately, quickly and efficiently.

What is an Eye Diagram?

The Eye Diagram is a method for visualizing a digital data stream, in which each consecutive clock cycle is overlaid on top of the first cycle in the data stream. The digital data pattern may be switching from 1 to 0, from 0 to 1, or stay at a 1 or 0 level. As a result, this continuously changing data stream, observed within a single cycle, produces a display resembling a human eye, as shown in Figure 2-17.

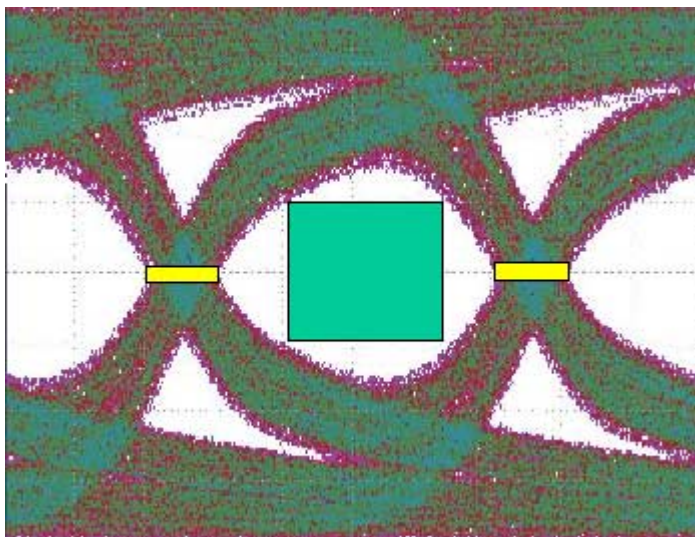


Figure 2-17: Eye Diagram.

The eye diagram was initially used primarily in communications standards, but it has recently propagated to signaling schemes in other industries, such as computer (Infiniband, Serial ATA, PCI Xpress), automotive, and consumer electronics.

2.3.1 Waveform Acquisition and Setup

You should begin by reviewing “Preparing to Take Measurements” on page 2-4 if necessary.

In addition to standard best practices measurement techniques discussed in “Preparing to Take Measurements” on page 2-4, for the Eye Diagram measurement it is particularly important to select a window that is long enough that it allows the DUT waveforms to completely settle to its steady state DC level. We are about to start making high-frequency analysis of the DUT, and we need to allow the signal to settle to make this analysis accurately.

We also need to make sure that a sufficient number of data points is selected in the oscilloscope acquisition window. This is particularly critical for analyzing long interconnects, such as cables. The sample interval should be at most one third the risetime of the clock signal you will be using to generate your eye. In the **S(f)** tab, the S-parameter frequency should be displayed to at least $1.5/t_r$, where t_r is the risetime of the clock signal you will use to generate your eye. The highest displayed frequency (f_{upper}) is related to the time domain time step (t_{step}), window length (t_{window}) and the number of points (N) as shown in this equation:

$$f_{upper} = \frac{1}{2t_{step}} = \frac{N}{2t_{window}}$$

To follow our example, select **Virtual Scope** in the drop-down list at the top of the acquisition window. The virtual scope is a tool for loading waveforms from your hard drive using the same interface as that used for oscilloscopes. Press the **Instrument Settings** tab at the top of the acquisition window to set the Virtual Scope options. Set the directory to **Tutorials\Lossy Line**, under the IConnect installation directory, e.g. **C:\Program Files\TDA Systems\IConnect**. Set the filter to ***.wfm**, and press the **Measure** tab to begin taking measurements.

Continue by clicking on the **Symmetric Coupled Lossy Line** model button on the model toolbar to create an empty lossy line model, as shown in Figure 2-18. The Lossy Line button may be identified by its picture of a coupled eye diagram.

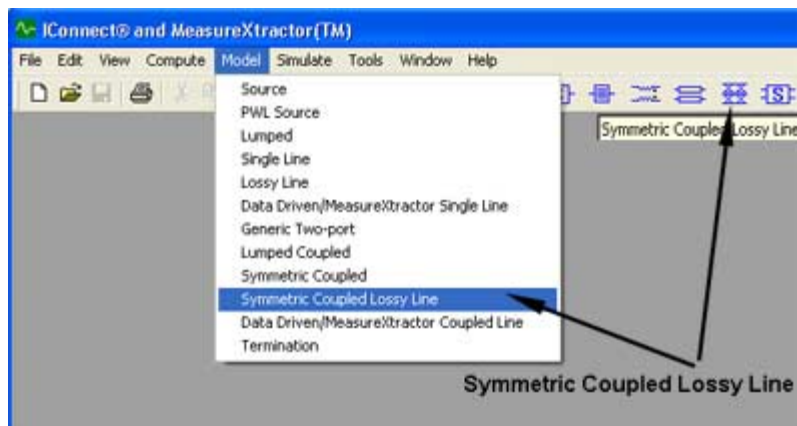


Figure 2-18: You can initiate a lossy line model from either the Model menu or from the Model toolbar.

Note that losses are a characteristic of the material of the interconnect. Therefore, to get the best results, it is important to extract the losses on a transmission line of controlled impedance. It is

possible to extract the losses based on structure with changing impedance, but if changes are very big, it will be difficult to extract losses accurately.

2.3.2 Choosing the Proper Termination

On the right-hand side of the window is a smaller window with tabs labeled **Measure**, **Model**, and **Simulate**. The window comes up initially with the **Measure** tab selected. Since in this tutorial we will be using the matched load approach, i.e., measuring a DUT which is connected to a 50Ω oscilloscope input at the far end, begin by selecting **Matched** from the **Type** dropdown box, as shown in Figure 2-19.

If you do not have the access to the far end of the DUT, which is required to perform the transmis-

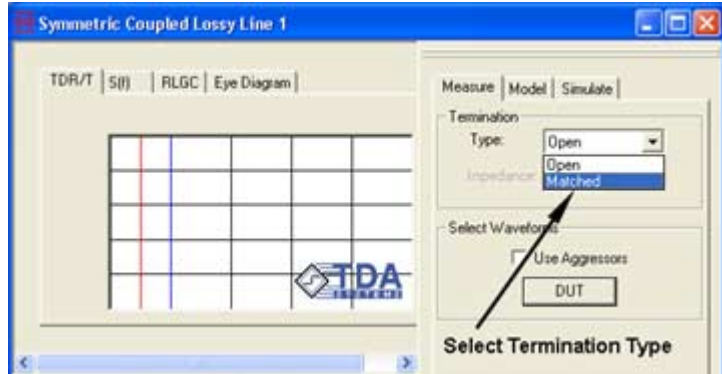


Figure 2-19: Select Termination Type.

sion measurement, you can still measure the eye diagram, but you have to first go through a complete model creation process for your interconnect. For more information on lossy line modeling in IConnect, refer to [“Lossy Line” on page 5-27](#).

2.3.3 Selecting a Reference

To compute an eye diagram, you need to obtain a reference and a DUT waveform. The reference waveform de-embeds the cable, probe, and fixture from the measurement, and establishes a measurement reference plane. The reference waveform can be an open reference, acquired at the end of the cable, fixture, or probe interfacing your TDR instrument to the DUT.

Alternatively, one can use a through measurement as a reference. Using through as a reference can be advantageous when a good reflected reference open can not be achieved, or to better comply with certain measurement standard procedures, such as Serial ATA, Infiniband, or PCI Xpress, which provide through structures on their fixtures for de-embedding the insertion loss of a fixture.

2.3.4 DUT Connections

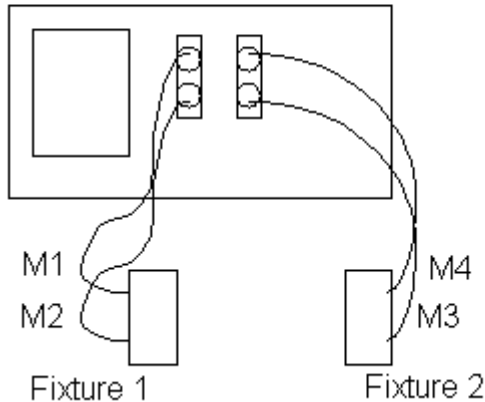
When using the **Matched** extraction method, TDR waveforms are acquired with 50Ω cables connected to both DUT ports, which ensures a 50Ω termination at the far end of the DUT, as shown in Figure 2-20. The cables, probes, and fixtures used to connect the DUT to both channels of the TDR oscilloscope are assumed by IConnect to be of the same length. In differential mode, both transmission, reference, and aggressor waveforms must be differential (that is, negative switching channel subtracted from the positive switching channel).

For a symmetric interconnect, we can use odd mode waveforms, acquired on the positive switching channel with the stimulus in differential mode.

Reference measurement:

Differential Stimulus is ON on channels M1, M2.

Reflection is measured on the same channels M1-M2



Reflection and transmission measurements:

Differential Stimulus is ON on channels M1, M2.

Transmission is measured on channels M3-M4

Reflection is measured on the channels M1-M2

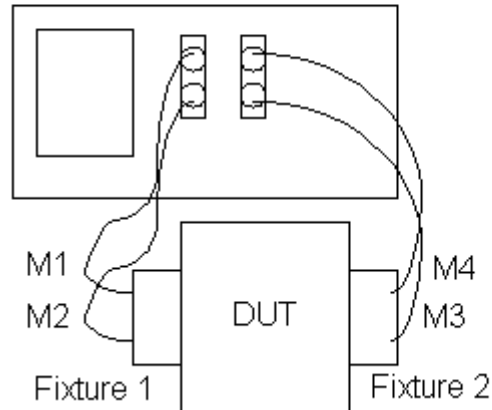


Figure 2-20: Measurement connection scheme.

2.3.5 Loading Data

With **Termination Type** set to **Matched** in the Measure tab, proceed to data acquisition. Now we can acquire the three required waveforms for the eye diagram prediction: Reference, Reflection, and Transmission. First, make sure to select the appropriate instrument in the Acquisition window. Click the **DUT** button in the **Select Waveforms** group box, and load Reference and Odd Transmission waveforms as shown in Figure 2-21. You can acquire the appropriate waveform from the instrument - or load the previously acquired waveforms from disk. Read the waveform descrip-

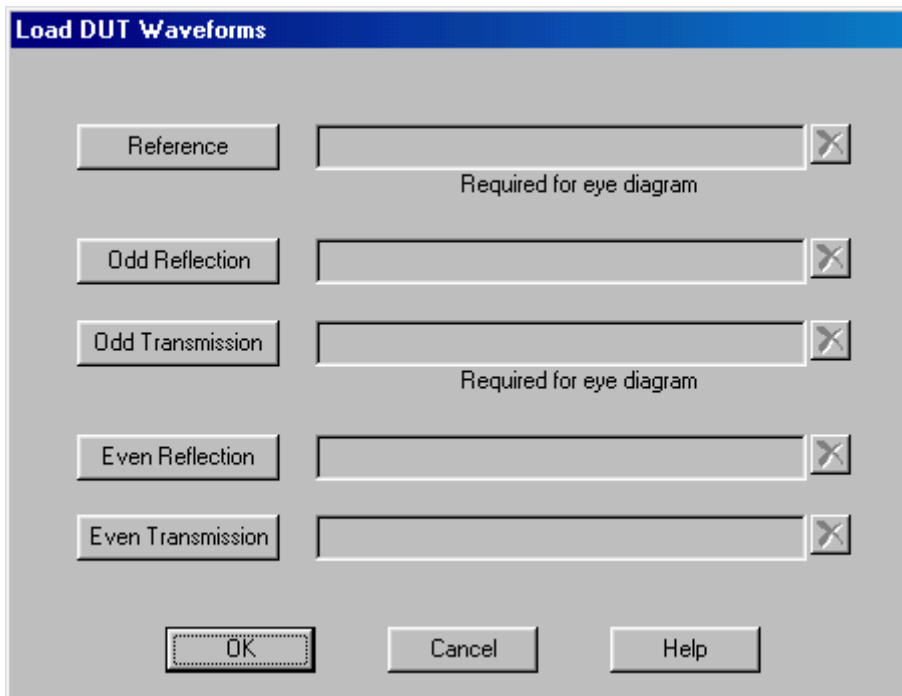


Figure 2-21: Loading waveforms into the Lossy Line model.

tions in the scope interface window, shown in Figure 2-22, to capture a proper waveform from the instrument. The two tutorial waveforms, named **Reference.wfm** and **Transmission.wfm**, can be found in the **Tutorials\Eye Diagram with Crosstalk** directory under the IConnect installation directory.

Note that the Reflection waveform is not required to generate the eye. Additionally, we can use the waveforms in this directory to illustrate computation of the effect of crosstalk from the adjacent lines to the DUT line of the eye.

View the Insertion Loss

When the waveform is loaded into the loss line TDR/T viewer, the frequency-domain scattering parameter S_{21} (insertion loss) will be automatically computed and displayed in the **S(f)** waveform viewer. To view it, click the **S(f)** tab in the lossy line window. Under the **S(f)** tab, the S-parameter frequency should be displayed to at least $1.5/t_r$, where t_r is the risetime of the clock signal you will use to generate your eye. If it is not displayed up to that frequency, you will need to remeasure your data with increased time resolution (decreased time step).

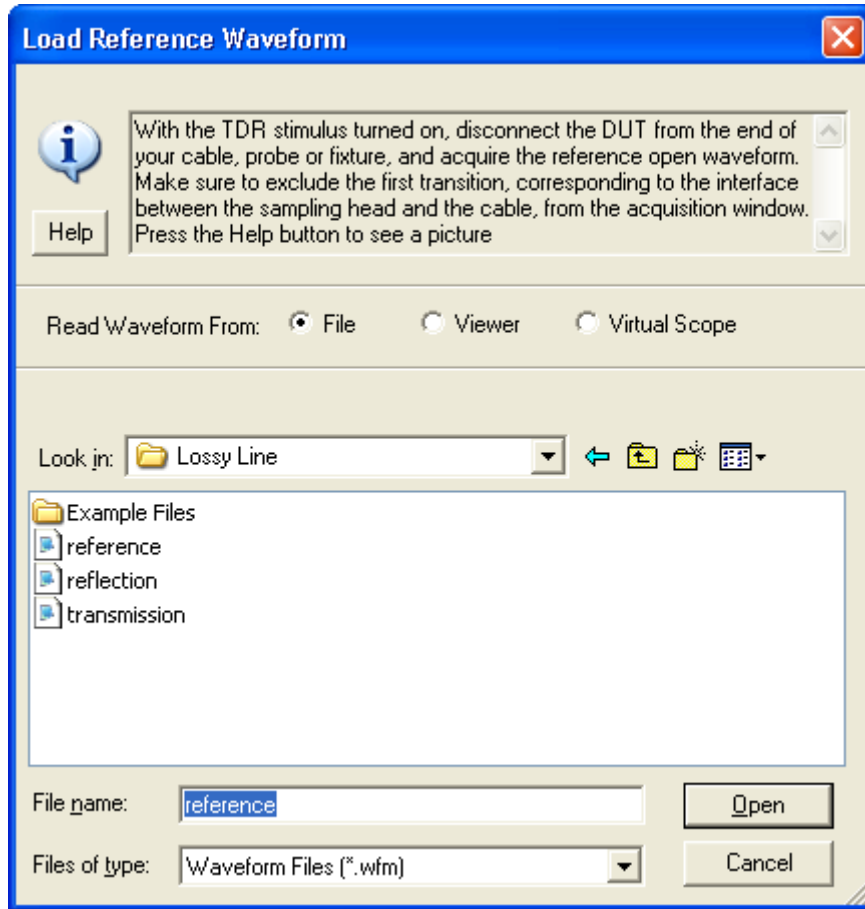


Figure 2-22: Instrument interface window.

2.3.6 Display the Eye

Now we can evaluate the deterministic jitter contributed by your device. To generate an eye diagram, click on the **Simulate** tab on the right side of the lossy line model window. In the **Eye Diagram** group box, click the **Display** button in order to generate and view an eye diagram based on the default eye diagram parameters.

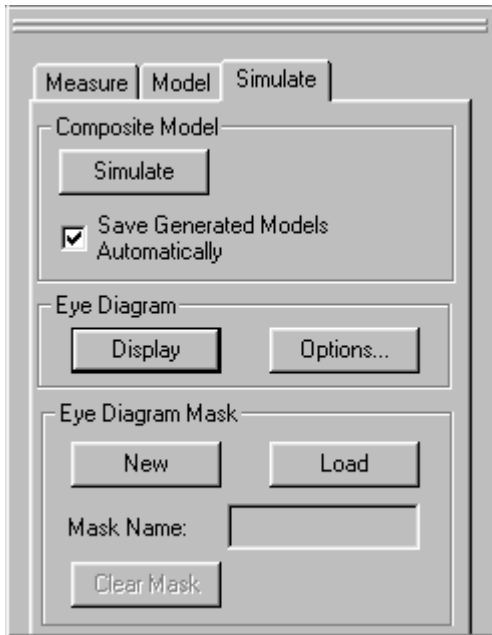


Figure 2-23: Display the eye diagram by pressing the Display button in the Eye Diagram box.

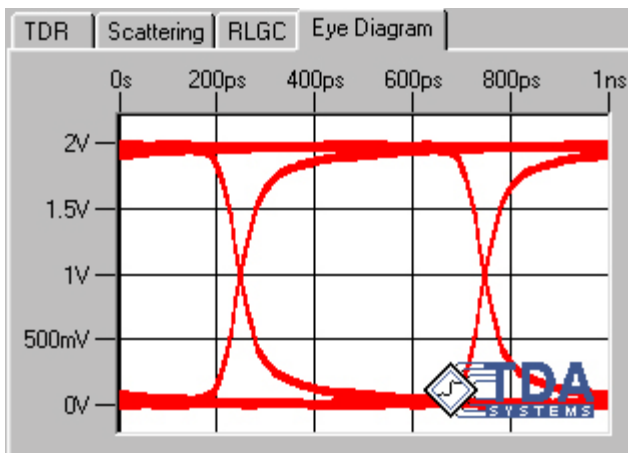


Figure 2-24: Eye diagram display.

2.3.7 Eye Diagram Options

If you would like to customize the eye diagram parameters, click the **Options...** button to display the **Eye Diagram and Mask Options** dialog, which allows you to set the eye parameters and define the eye mask.

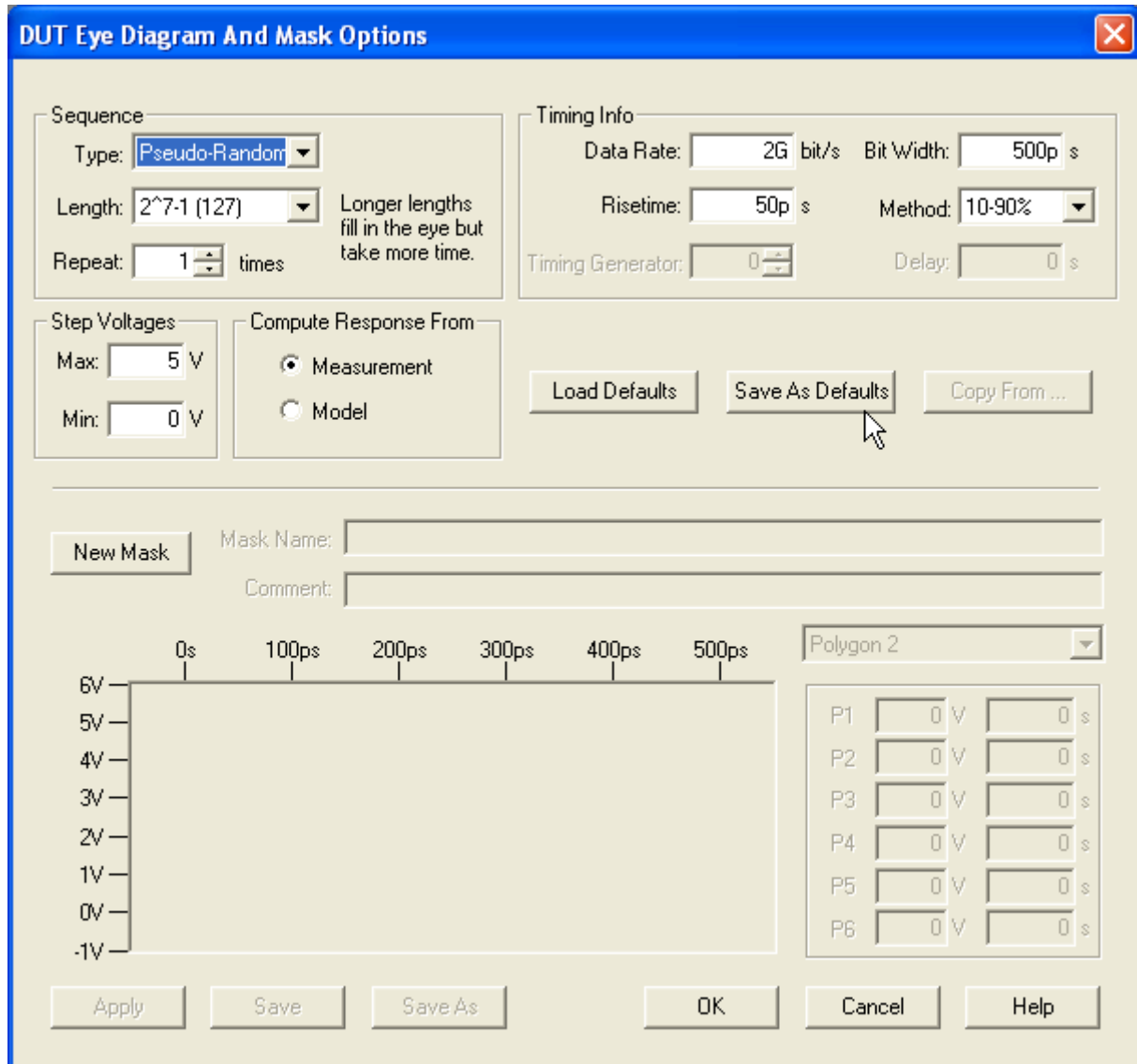


Figure 2-25: Eye Diagram and Mask Options dialog.

Sequence

Select the desired sequence for your application, or choose **Custom** and enter your own sequence.

Timing Info

Data rate or **bit width** determine the speed at which the eye diagram is displayed; bit width = 1/ data rate (gigabit). **Risetime** cannot be more than 50% of period and typically is much less.]==

Step Voltages

Select the desired voltages for your application.

Compute Response From

Set to **Measurement**. The measurement-based eye is an exact conversion of the transmission into the eye, whereas a model-based eye accuracy will depend on the accuracy of the model.

2.3.8 Display the Final Eye

Once you have customized the options, click **Display** again to recompute the eye diagram as shown in Figure 2-26. If the eye diagram appears “jagged” and “broken,” the most likely cause is that the time step is not small enough to produce frequency domain data up to the required frequency. Re-acquire the data with a shorter acquisition window or larger number of points in the window.

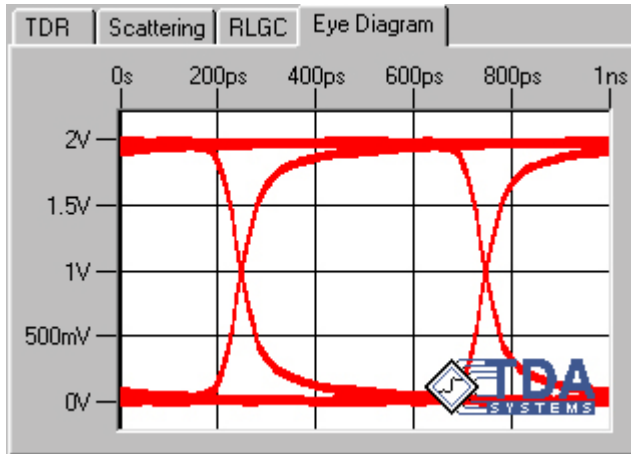


Figure 2-26: Display the eye diagram.

2.3.9 Eye Diagram Zooming

When zooming in on an eye generated with long data sequences, you will be asked to choose the zoom method as shown in Figure 2-27. Choosing **Continue** is the fastest method, but you will be

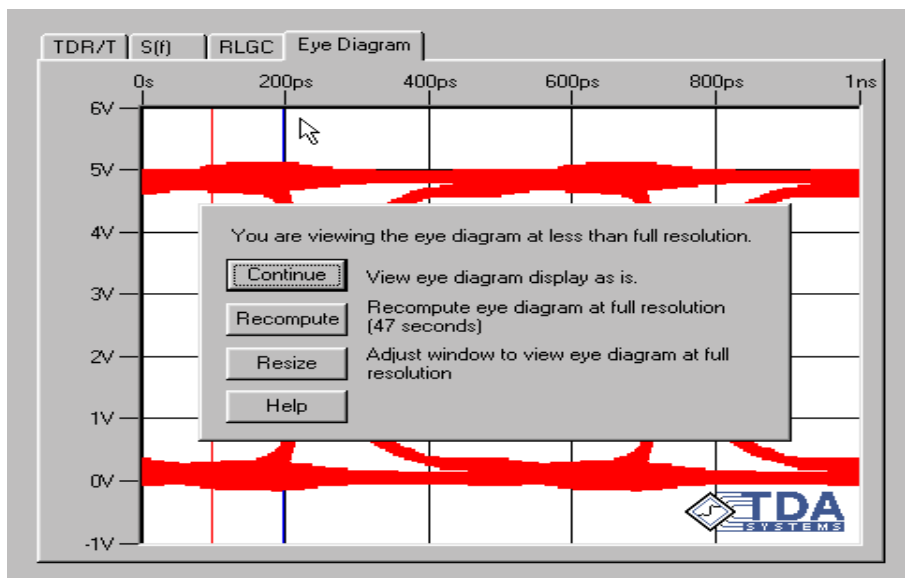


Figure 2-27: Choose the zoom method.

viewing the zoom area at less than full resolution. Choosing **Recompute** is the slowest but most accurate method. Choosing **Resize** will return you to the eye diagram you originally viewed.

When recomputing the eye diagram, if the computation takes too long, you can use the **Cancel** button to cancel the computation.

2.3.10 Eye Mask Testing

You can set up your mask to perform electrical compliance testing with such standards as PCI Xpress, SATA, Infiniband, XAUI, etc.

As shown in Figure 2-28, you can select the appropriate polygon (2 is the main central polygon, 1

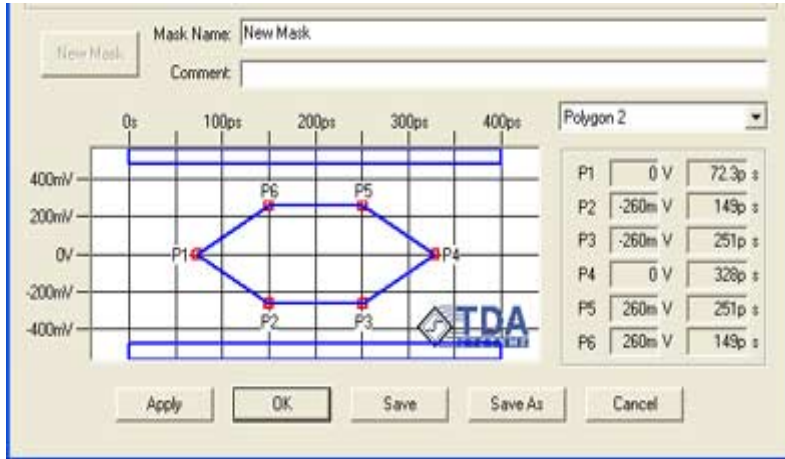


Figure 2-28: Select a polygon.

and 3 are the upper and lower polygons). The vertices of the eye mask can be moved by hand. Click **Apply** to apply the eye mask to existing eye, or **Save** to save the eye mask information into a file. Eye mask information can also be saved as part of a lossy line model.

2.3.11 Measure Jitter

Click **Enable Eye Measurements** at the bottom of the eye window shown in Figure 2-29,, and

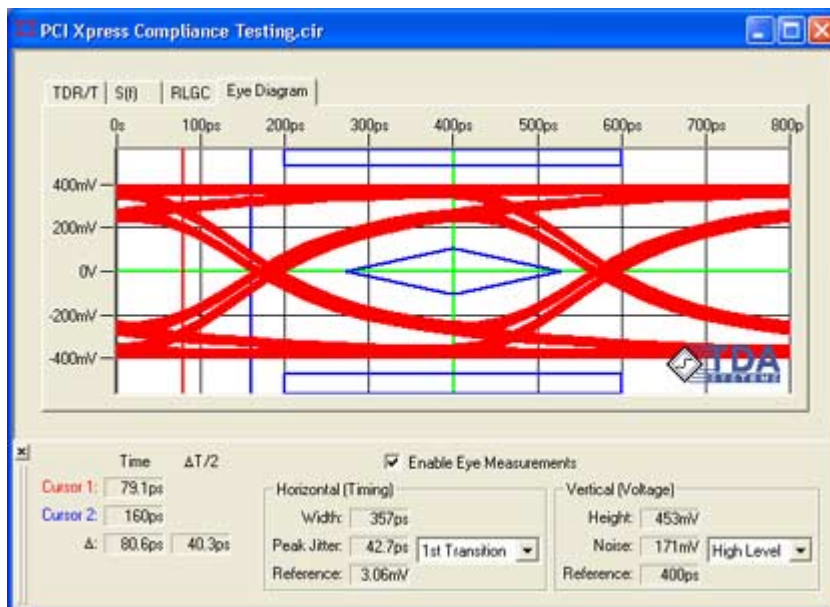


Figure 2-29: Display the eye measurements.

you will be able to read eye width and height, as well as peak-to-peak jitter and amplitude noise. Amplitude noise can be read at the high or low levels. Note that interconnects generate only deterministic jitter, and there is no meaning for "RMS" jitter.

2.3.12 Correlation

In the eye diagram measurement for an interconnect, we characterize the deterministic jitter due to crosstalk and losses in the passive, linear interconnect system. As a result, TDR-based eye diagrams are the exact representation of the eye diagram degradation for the given interconnect. This is true as long as the measurement is performed with sufficient accuracy and care to characterize a given interconnect to a frequency that is at least $1.5/t_r$, where t_r is the risetime of the clock signal you will use to generate your eye.

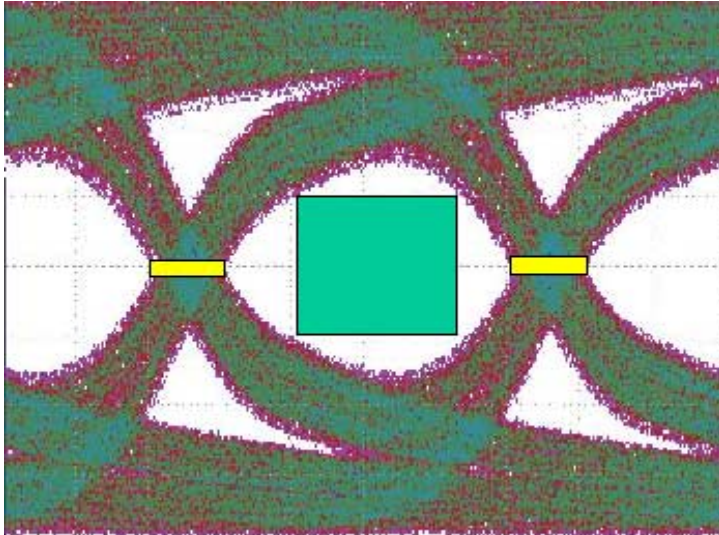


Figure 2-30: Directly measured eye diagram response.

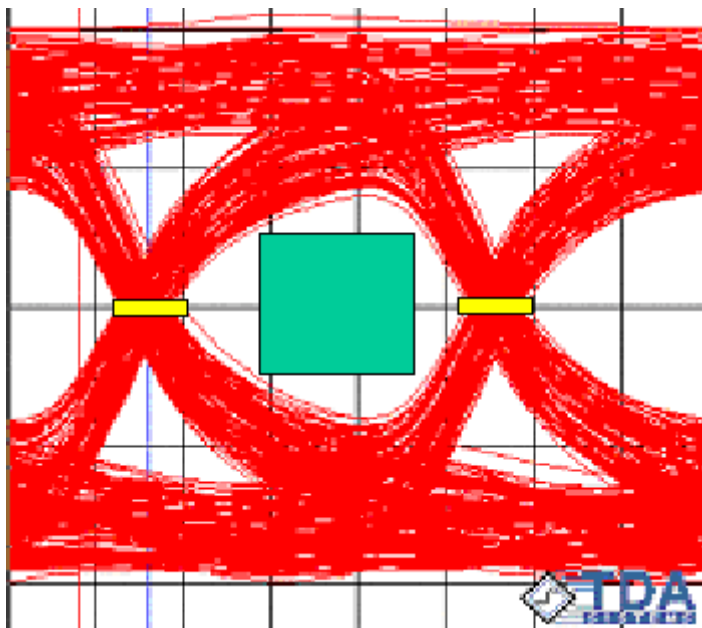


Figure 2-31: Eye diagram response as computed by IConnect from TDT measurements.

Eye Diagram Accuracy

Despite the simplicity of the TDR transmission eye diagram measurement process, we would like to address the accuracy questions. We will compare eye diagram measurements in IConnect using a TDR oscilloscope transmission measurement versus eye diagram measurements using a system based on a bit-error-rate tester or a pattern generator and a sampling oscilloscope.

Since all we are characterizing is the deterministic jitter due to crosstalk and losses in the passive, linear interconnect system, a TDR-based eye diagram for the interconnect is effectively the exact representation of the eye diagram degradation for the given interconnect. As such, TDR-based eye diagram computation is correct by design.

To confirm this, we provide several examples of correlation of TDR-based eye diagram measurements to directly measured eye diagrams. As we stated before, as long as the TDR transmission measurement is performed with sufficient accuracy and care to characterize a given interconnect to a frequency that is at least $1.5/t_r$, where t_r is the risetime of the clock signal you will use to generate your eye, the eye diagram will be correct.

2.3.13 System Eye Diagram

The eye diagram measured in IConnect shows signal degradation due to interconnect losses and crosstalk. In order to obtain a complete picture for the digital system eye diagram, the designer needs to extract the lossy coupled models for the interconnects in IConnect TDR and VNA software, and simulate propagation of the signal from a real digital I/O driver into these interconnect models in a simulator that has eye diagram display capability.

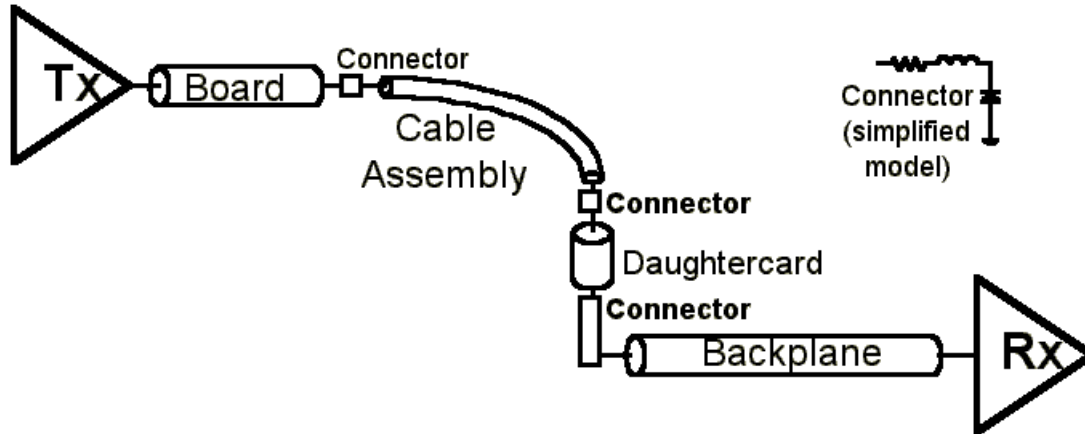


Figure 2-32: System used to measure and predict the eye diagram.

2.4 S-parameter Computation Tutorial

Welcome to the IConnect S-parameter Computation Tutorial. This tutorial guides you through the steps necessary to obtain the single ended or differential S-parameters and insertion and return loss for an interconnect from Time Domain Reflection (TDR) and Transmission (TDT) measurements. Frequency-domain scattering parameters are a useful way to visualize the electrical properties of a system. This tutorial leads you from a set of raw measurements to a complete set of S-parameters for your device under test (DUT), and allows you to evaluate the insertion and return loss of your device accurately, quickly, and efficiently.

2.4.1 What are S-Parameters?

S-Parameters and TDR/T Data

Frequency-domain scattering parameters (S-parameters) are a useful way to visualize the electrical properties of a system. They are a dual representation of TDR and TDT data in the frequency domain. Although S-parameters have historically been measured in the frequency domain using a network analyzer, the same information is contained in TDR and TDT measurements.

Differential and mixed mode S-parameters can be obtained from corresponding TDR oscilloscope data. From a practical standpoint, TDR/T-based S-parameters are much simpler and quicker to obtain, while providing more than sufficient dynamic range. In addition, if a designer already has a TDR oscilloscope, he or she can save money by not buying a network analyzer, and using the TDR oscilloscope with IConnect TDR and VNA software instead.

2.4.2 Waveform Acquisition Setup

Begin by reviewing “[Preparing to Take Measurements](#)” on page 2-4 if necessary.

Acquisition Window

In addition to the best practices outlined in “[Preparing to Take Measurements](#)” on page 2-4, for the S-parameter measurement it is particularly important to select a window that is long enough that it allows the DUT waveforms to completely settle to its steady state DC level. We are about to start making high-frequency analysis of the DUT, and we need to allow the signal to settle to make this analysis accurately.

We also need to make sure that a sufficient number of data points is selected in the oscilloscope acquisition window. This is particularly critical for analyzing long interconnects, such as cables. The upper frequency limit of your measurement is determined as

$$f_{upper} = \frac{1}{2t_{step}} = \frac{N}{2t_{window}} \quad \text{and} \quad f_{step} = \frac{1}{t_{window}}$$

where N is the number of points in the acquisition window, t_{step} is the time step, and t_{window} is the acquisition window length. Thus, to get S-parameter data to the desired upper frequency f_{upper} you will need to make sure that the ratio between the number of points and window length is sufficient.

To get the desired frequency step, you will need to make sure that the window length is sufficient. When working with long cables, it is generally a good idea to keep the number of points at the maximum number allowed for a given instrument. Furthermore, to increase the dynamic range of the measurements, it is important to keep number of averages and number of points to the maximum, and the window as short as possible, as the equation below indicates:

$$SNR(N, N_{avg}) = SNR(N_0, N_{avg}) \cdot \sqrt{\frac{N}{N_0}} \cdot \sqrt{\frac{N_{avg}}{N_{avg0}}}$$

Make sure to use heavy averaging in your instrument. Using 128 averages will give an extra 20dB or dynamic range; use at least that many if your instrument averages quickly. Keeping the acquisition window as short possible maximizes the effective power.

When dealing with differential measurements, you need to make sure that your signals arrive at the DUT at the same time. You can follow the procedure in your oscilloscope manual, and having observed the difference in delays between the two channels in differential mode, adjust for half of that difference by using delta TDR function in the scope.

Tutorial Waveform Data

To follow our example, select **Virtual Scope** in the drop-down list at the top of the acquisition window. The Virtual Scope is a tool for loading waveforms from your hard drive using the same interface as that used for oscilloscopes.

Press the **Instrument Settings** tab at the top of the acquisition window to set the Virtual Scope options. Set the directory to **Tutorials\Lossy Line** under the IConnect installation directory, e.g. C:\Program Files\TDA Systems\IConnect\. Set the filter to ***.wfm**, and press the **Measure** tab to begin taking measurements. Select the waveforms **Reflection.wfm**, **Transmission.wfm**, and

Getting Started

Open.wfm from the acquisition window. Press the **Acquire** button, which opens up a new waveform viewer containing the two waveforms.

2.4.3 S-parameter Data Acquisition

To compute S-parameters, you need to obtain a reference and a DUT waveform.

The reference waveform de-embeds the cable, probe, and fixture from the measurement, and establishes a measurement reference plane. The reference waveform can be an open reference, acquired at the end of the cable, fixture, or probe interfacing your TDR instrument to the DUT. Alternatively, one can use a through measurement as a reference. Using through as a reference can be advantageous when a good reflected reference open cannot be achieved, or to better comply with certain measurement standard procedures, such as Serial ATA, Infiniband or PCI Xpress, which provide through structures on their fixtures for de-embedding the insertion loss of a fixture.

A DUT waveform can be any waveform measured from the DUT: forward reflection (TDR), forward transmission from channel 1 to channel 2 (TDT), backward reflection or backward transmission from channel 2 to channel 1 (in which case the signal is going to be launched from the second output of the TDR sampling head). The corresponding S-parameters will be S_{11} (forward TDR),

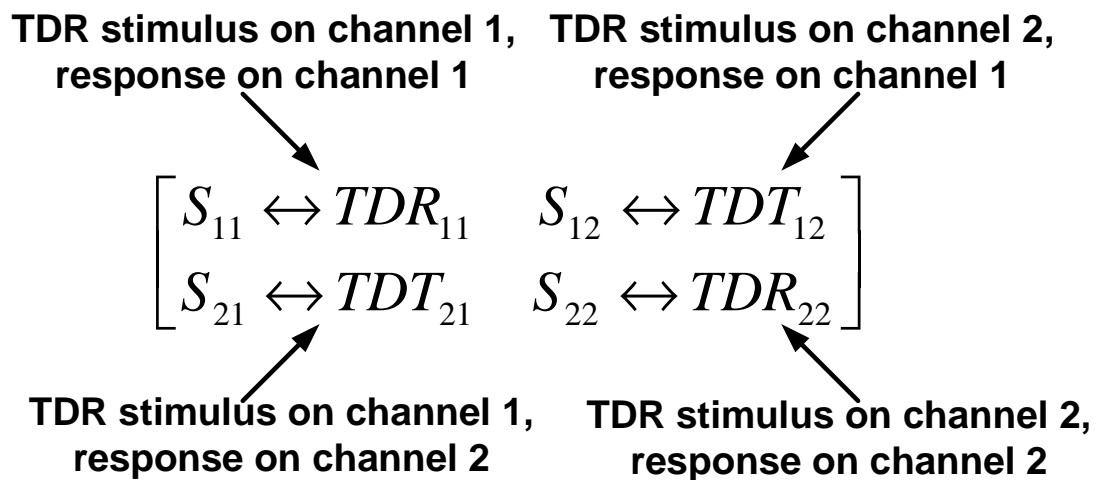


Figure 2-33: Mapping between scattering parameters and TDR/T measurements.

S_{21} (forward TDT), S_{22} (backward TDR) and S_{12} (backward TDT). When measuring frequency domain crosstalk, just like in time domain, you would be measuring TDT to the adjacent channel, resulting in S_{21} into an adjacent channel. Please note that even if you are taking TDR (return loss) measurement, the far end of the device must be terminated with 50Ω .

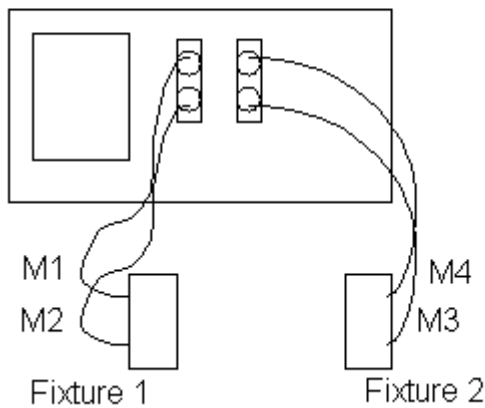
2.4.4 DUT Connections

The TDR waveform in the S-parameter computation must be acquired with the 50Ω cables connected to both DUT ports, thus ensuring a 50Ω termination at the far end of the DUT. The cables, probes or fixtures used to connect the DUT to both channels of the TDR oscilloscope are assumed by the IConnect to be of the same length. In differential mode, both transmission and reference must be differential (that is, negative switching channel subtracted from the positive switching channel).

Reference measurement:

Differential Stimulus is ON on channels M1, M2.

Reflection is measured on the same channels M1-M2



Reflection and transmission measurements:

Differential Stimulus is ON on channels M1, M2.

Transmission is measured on channels M3-M4

Reflection is measured on the channels M1-M2

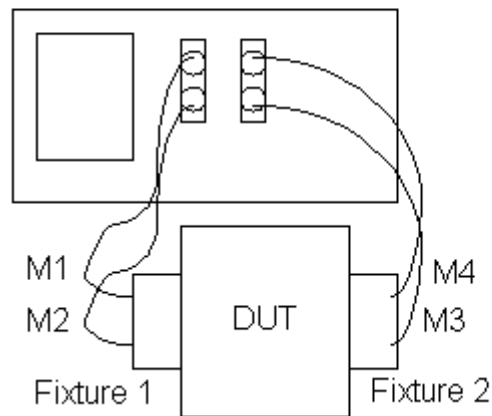


Figure 2-34: Measurement termination scheme.

For a symmetric interconnect, we can use odd mode waveforms, acquired on the positive switching channel with the stimulus being in differential mode. Please note that even if you are only taking a TDR (return loss) measurement, the far end of the device must be terminated with 50Ω on each port. 100Ω differential termination is acceptable for differential response measurements.

2.4.5 Differential, Common, and Mixed Modes

For differential, common mode and mixed mode S-parameters, the same applies - a reference and a DUT waveform are required. Before beginning the measurements, however, make sure to follow the procedure for your TDR instrument to align the TDR sources at the DUT measurement reference plane. For differential stimulus waveforms, the oscilloscope must have the two stimulus signals switching in opposite directions. For common mode stimulus waveforms, the oscilloscope must have the stimuli switching in the same direction. For differential response, subtract one response channel from the other. For the common mode response, add the two response channels together.

Differential S-parameters

In the case where the two lines in the differential pair are symmetric, you stimulate the differential pair with a differential stimulus in the TDR oscilloscope (stimuli channels switching in the opposite direction), and acquire the waveform on the positive switching channel (Ch1). If you use a reference open waveform, you acquire it on the same channel, and use it to compute the differential S_{11} (return loss).

For insertion loss look at the transmission waveform on the positive switching channel (most likely Ch3). If you are using a transmission without the DUT in place as a reference, you will be acquir-

$$\begin{array}{c}
 \text{Differential TDR stimulus, differential response} \\
 \left[\begin{array}{cccc}
 S_{11dd} \leftrightarrow TDR_{11dd} & S_{12dd} \leftrightarrow TDT_{12dd} & & \\
 S_{21dd} \leftrightarrow TDT_{21dd} & S_{22dd} \leftrightarrow TDR_{22dd} & S_{11dc} \leftrightarrow TDR_{11dc} & S_{12dc} \leftrightarrow TDT_{12dc} \\
 & & S_{21dc} \leftrightarrow TDR_{21dc} & S_{22dc} \leftrightarrow TDR_{22dc} \\
 & S_{11cd} \leftrightarrow TDR_{11cd} & S_{12cd} \leftrightarrow TDT_{12cd} & S_{11cc} \leftrightarrow TDR_{11cc} & S_{12cc} \leftrightarrow TDT_{12cc} \\
 & S_{21cd} \leftrightarrow TDT_{21cd} & S_{22cd} \leftrightarrow TDT_{22cd} & S_{21cc} \leftrightarrow TDT_{21cc} & S_{22cc} \leftrightarrow TDR_{dd}
 \end{array} \right]
 \end{array}$$

Figure 2-35: Differential-Differential block of mixed mode S-Parameters

ing it on the same channel.

For a case of differential lines with mild asymmetry, proceed as follows: when the TDR is in differential mode, and the oscilloscope displays voltage, subtract the negative switching channel (Ch2) from the positive switching channel Ch1 (DUT = Ch1 - Ch2). Then, acquire reference on both channels simultaneously and subtract Ch1Ref - Ch2Ref again. Now, use the DUT waveform and the computed reference waveform to compute differential S_{11} . Again, for insertion loss, use the difference between positive and negative switching transmission channels (most likely Ch3 - Ch4). This latter procedure should always be used when acquiring data for frequency domain crosstalk computation.

Common Mode S-parameters

The same discussion as for differential S-parameters applies, except the stimulus used must be a common mode stimulus (stimuli channels switching in the same direction). In addition, for traces with asymmetry, you will need to add the appropriate channels instead of subtracting them (reference is produced from Ch1Ref+Ch2Ref, common mode S_{11} or return loss from Ch1+Ch2, and common mode S_{21} or insertion loss from Ch3+Ch4). See Figure 2-36.

Mixed Mode S-parameters

Mode conversion parameters are much less important, but if you need to obtain them, proceed using the same process as for common mode S parameters, except for the following. For differen-

$$\begin{array}{c}
 \text{Common mode TDR stimulus, common mode response} \\
 \left[\begin{array}{cccc}
 S_{11dd} \leftrightarrow TDR_{11dd} & S_{12dd} \leftrightarrow TDT_{12dd} & S_{11dc} \leftrightarrow TDR_{11dc} & S_{12dc} \leftrightarrow TDT_{12dc} \\
 S_{21dd} \leftrightarrow TDT_{21dd} & S_{22dd} \leftrightarrow TDR_{22dd} & S_{21dc} \leftrightarrow TDR_{21dc} & S_{22dc} \leftrightarrow TDR_{22dc} \\
 S_{11cd} \leftrightarrow TDR_{11cd} & S_{12cd} \leftrightarrow TDT_{12cd} & S_{11cc} \leftrightarrow TDR_{11cc} & S_{12cc} \leftrightarrow TDT_{12cc} \\
 S_{21cd} \leftrightarrow TDT_{21cd} & S_{22cd} \leftrightarrow TDR_{22cd} & S_{21cc} \leftrightarrow TDT_{21cc} & S_{22cc} \leftrightarrow TDR_{22cc}
 \end{array} \right]
 \end{array}$$

Figure 2-36: Common-Common block of mixed mode S-Parameters

tial-to-common mode conversion the stimulus is differential, whereas the acquisition is performed only using the sum of the channels (reference is produced from Ch1ref-Ch2ref, differential-to-common mode conversion S_{11} or return loss from Ch1+Ch2, and differential-to-common mode conversion S_{21} or insertion loss from Ch3+Ch4). For common mode to differential mode conversion, the stimulus is common mode, whereas the acquisition is performed using only the difference between the channels.

Normally, the differential-to-common mode differences should be 0 across the acquisition window; when there is skew, however, they will be non-zero and will produce non-zero frequency dependent mode conversion.

$$\begin{array}{c}
 \text{Mixed mode} \\
 \left[\begin{array}{cccc}
 S_{11dd} \leftrightarrow TDR_{11dd} & S_{12dd} \leftrightarrow TDT_{12dd} & S_{11dc} \leftrightarrow TDR_{11dc} & S_{12dc} \leftrightarrow TDT_{12dc} \\
 S_{21dd} \leftrightarrow TDT_{21dd} & S_{22dd} \leftrightarrow TDR_{22dd} & S_{21dc} \leftrightarrow TDT_{21dc} & S_{22dc} \leftrightarrow TDR_{22dc} \\
 S_{11cd} \leftrightarrow TDR_{11cd} & S_{12cd} \leftrightarrow TDT_{12cd} & S_{11cc} \leftrightarrow TDR_{11cc} & S_{12cc} \leftrightarrow TDT_{12cc} \\
 S_{21cd} \leftrightarrow TDT_{21cd} & S_{22cd} \leftrightarrow TDR_{22cd} & S_{21cc} \leftrightarrow TDT_{21cc} & S_{22cc} \leftrightarrow TDR_{22cc}
 \end{array} \right]
 \end{array}$$

Figure 2-37: Mode Conversion block of mixed mode S-Parameters

2.4.6 Compute S-parameters

If the **Compute** window is not visible, select **S-Parameter** from the **Compute** menu. The scattering parameter is computed from the computation window by choosing S-Parameter from the **Compute** list box as shown in Figure 2-38, or by clicking on the separate 'S' button on the toolbar.

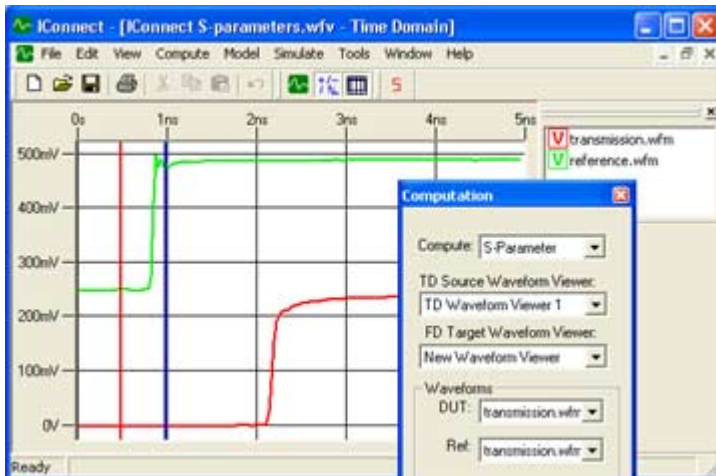


Figure 2-38: Computing S-parameter.

From the drop-down list box titled **Ref.**, choose your reference waveform.

From the list box titled **DUT**: choose the measured response of the DUT to the input step. Click the **Compute** button, and a new frequency-domain waveform viewer will be created with the scattering response in it.

2.4.7 S-Parameter Options

Additional options are available when computing S-Parameters, as shown in Figure 2-39. First,

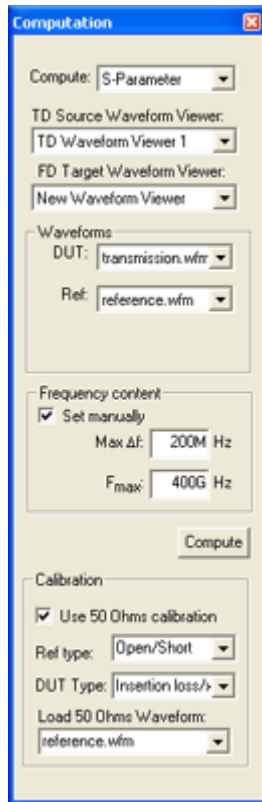


Figure 2-39: S-Parameter computation window.

the user can let IConnect determine the highest frequency content of the data, or the user can check the **Set Manually** box and set the desired Δf and F_{upper} by hand. Furthermore, the user can choose to perform a 50Ω calibration, using a precision 50Ω standard at the DUT reference plane. The user also has a choice between open, short, or thru reference. Please keep in mind that the 50Ω calibration waveform is ignored for insertion loss/crosstalk measurements if a thru reference is used.

2.4.8 Calibration

Calibration methods are often used to remove the effects of the fixturing used to measure the DUT. The calibration implemented in IConnect is a short version of a standard S-O-L-T (Short-Open-Load-Thru calibration), which provides sufficient accuracy for digital signal integrity measurements and digital design electrical compliance tests. It is critical to remember, however, that the accuracy benefits that calibration provides only manifest themselves when high-quality calibration standards are used, and when the measurement repeatability is ensured. Measurement repeatability is always key to improving accuracy.

It should be noted that for most signal integrity and digital design tasks, good measurements could be obtained even without calibration, if quality cables and connectors are used.

2.4.9 Choosing the Data Display

Please note that since we used a forward TDT (transmission) waveform of the DUT, the resulting S-parameter is S_{21} (insertion loss). If we used a TDR (reflection waveform), we would obtain return loss; if we used TDT into the adjacent line, we would obtain crosstalk in frequency domain.

Right-click in the frequency domain waveform viewer, select **Plot**, and choose the desired format for the S-parameter display as shown in Figure 2-40. The most common display in signal integrity

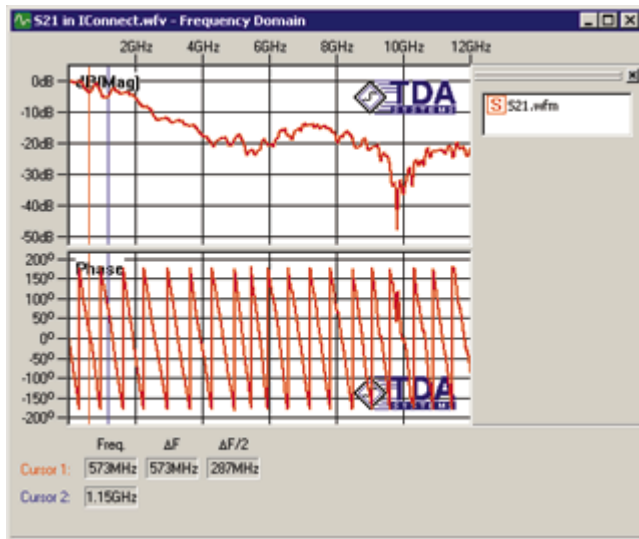


Figure 2-40: Single-ended S_{21} data computed from TDT measurement.

is dB(Mag)/Phase. Right-click on the displayed waveform, and select **Cursor Readout** to view the values.

2.4.10 Differential Example

Figure 2-41 is an example of differential return loss (S_{11dd} parameter) in IConnect. Right-click in the frequency domain waveform viewer, select **Plot**, and choose the desired format for the S-parameter display. The most common display in signal integrity is dB(Mag)/Phase. Right-click on the displayed waveform, and turn on cursor readout to view the values.

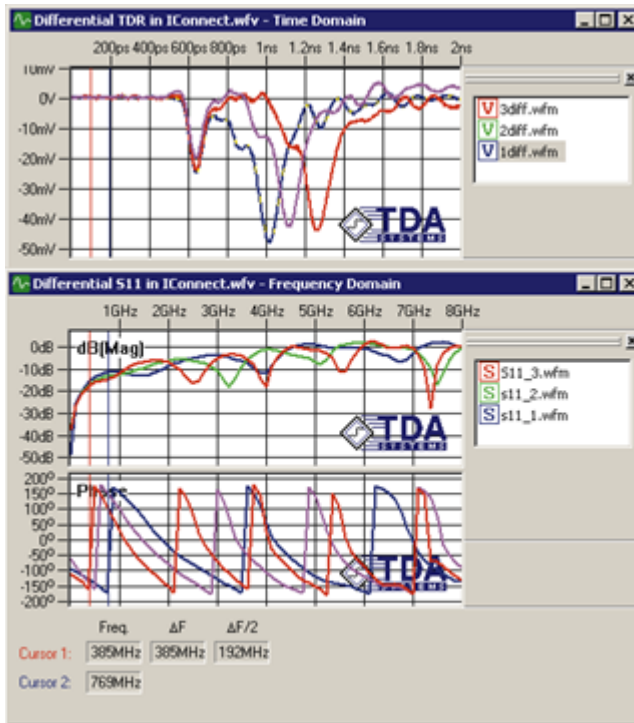


Figure 2-41: Differential S-Parameter example.

S-Parameter Window

Please note that the S-parameters can also be computed by simply opening any of the modeling windows (for example, lossy line modeling window, as shown in Figure 2-42), loading the appropriate data, and clicking on the **S(f)** tab as shown in Figure 2-42.

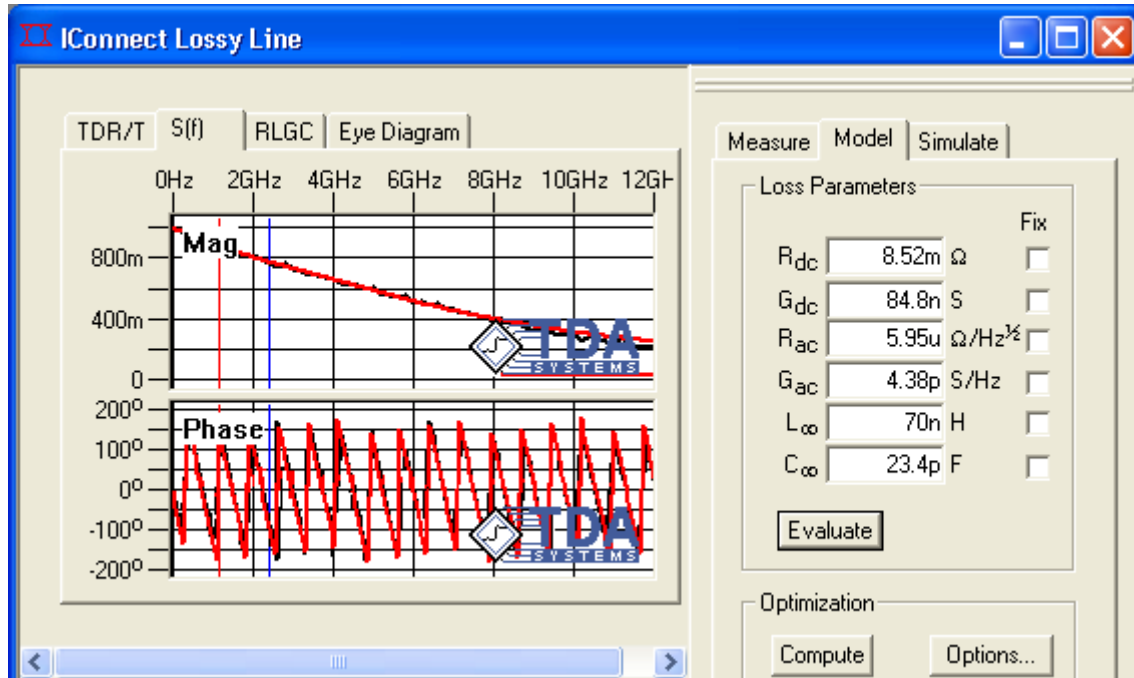


Figure 2-42: The S(f) tab allows you to view your measurements and model response in the frequency domain.

2.4.11 Exporting Touchstone

You can export the data in your comma-separated values (csv) or Touchstone format, Touchstone being the standard S-parameter interchange format, using the **File/Export** menu as shown in Figure 2-43. You can export 1-port, 2-port, 4-port single-ended and 4-port differential S-parameters.

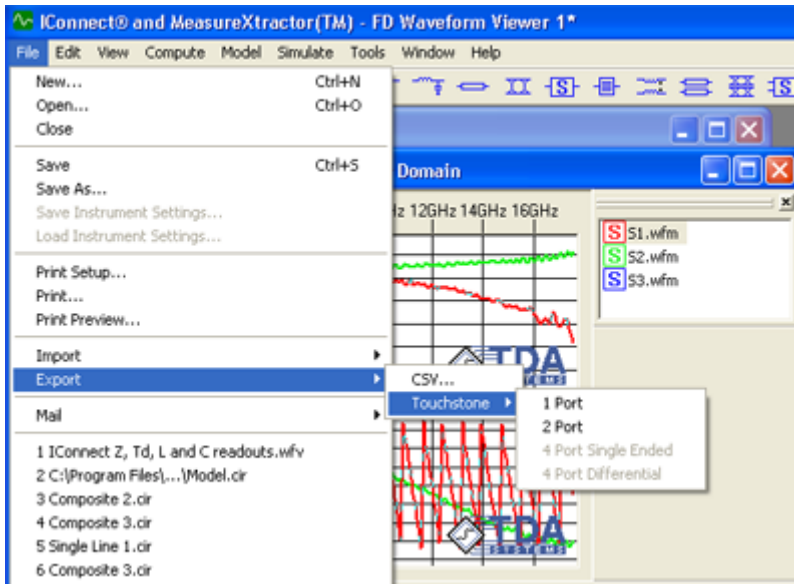


Figure 2-43: Exporting Touchstone data.

Exporting S2P and S4P

You will need at least 3 waveforms in your frequency domain viewer to export a 2-port Touchstone file if your DUT is reciprocal; i.e., $S_{21} = S_{12}$. You will need 4 waveforms if it is non-reciprocal.

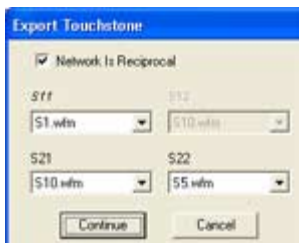


Figure 2-44: Export a 2-port Touchstone file.

You will need at least 10 waveforms to export a 4-port Touchstone file if your DUT is reciprocal, and 16 waveforms if it is not.

Most interconnect networks are reciprocal.



Figure 2-45: Export a 4-port Touchstone file.

2.4.12 Correlation

Figure 2-46 provides an excellent illustration of correlation between a TDR-based insertion loss computed in IConnect and a VNA insertion loss up to 12 GHz.

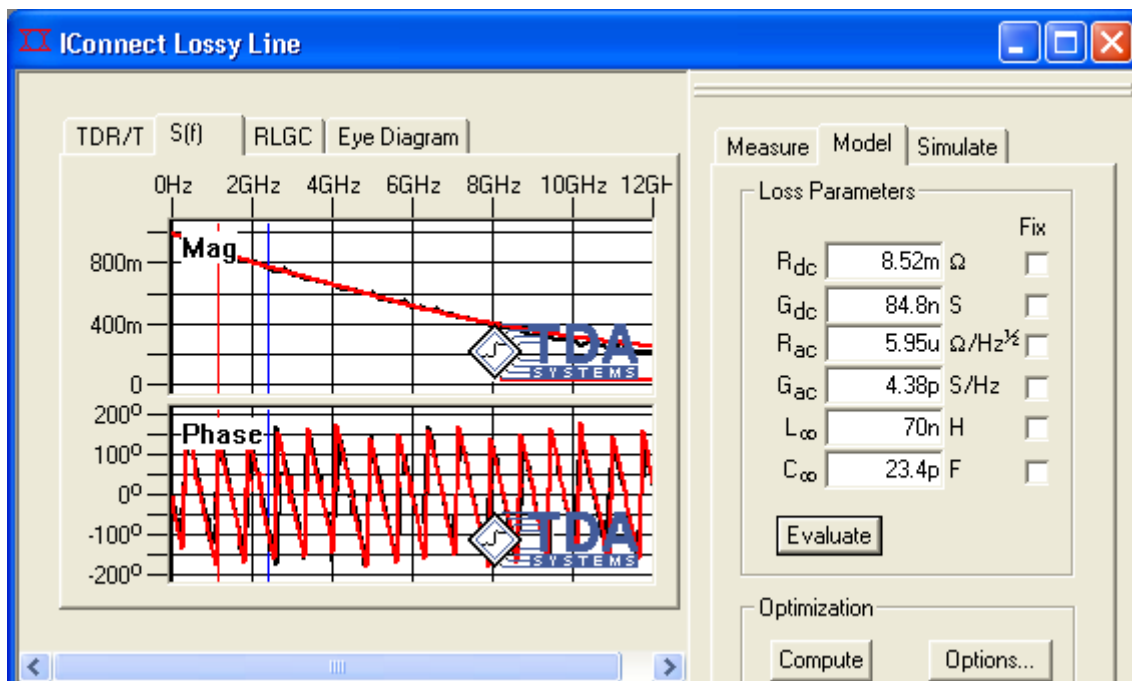


Figure 2-46: Correlation example.

2.5 Lumped LC Computation Tutorial

Welcome to the IConnect Lumped LC Computation Tutorial. This tutorial guides you through the steps necessary to obtain accurate measurements for inductance and capacitance in IConnect. This is a relative measurement technique, which is based on two comparative measurements with and without the Device Under Test (DUT) in place, and is based on the JEDEC document JEP-123 and the IBIS standard for measuring L and C. The tutorial leads you from a set of raw measurements to accurately measured L and C values, and allows you to evaluate the model and verify its accuracy.

This technique can be used to measure the inductance or capacitance of a package, connector, via, or an active device I/O buffer capacitance.

Validity of L and C Models

The JEDEC and IBIS standard techniques for L and C computation described here are only valid if your DUT is electrically short compared to the rise time of the signals propagating through it (i.e., $t_{dut} \ll t_{rise}$). If the device is not electrically short, the DUT cannot be described using a simple lumped or lumped coupled model, whether obtained through the techniques described here or any other time or frequency domain measurement technique. A distributed model, based on the impedance profile computed in IConnect, must be used instead, or the DUT must be split into shorter subsegments, each such subsegment shorter than the signal rise time. For practical purposes, the electrical length of the DUT must be 2 to 3 times shorter than the rise time of the signals.

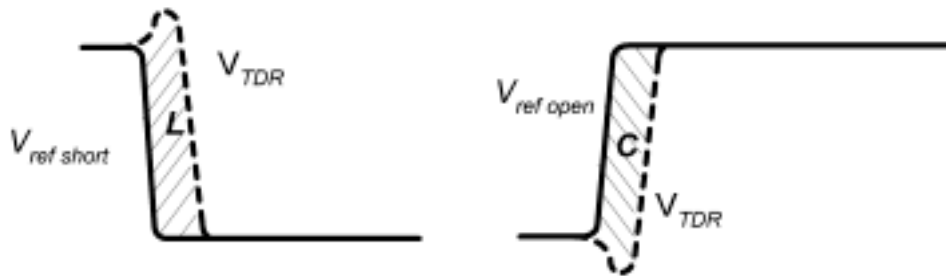


Figure 2-47: Computation of L and C

2.5.1 Waveform Acquisition and Setup

Begin by reviewing “[Preparing to Take Measurements](#)” on page 2-4 if necessary. For LC measurements, it is particularly important to select a window that is long enough that it allows the DUT waveforms to completely settle to its steady state DC level. Since we are performing a steady-state analysis of the DUT, we need to allow the signal to settle to make this analysis accurately.

The asymptotic capacitance and inductance computation are accessed via the computation window, from the Compute menu, as shown in Figure 2-48. The computation to be performed is chosen from the computation drop-down, which causes the drop-down list boxes and their labels in the Waveforms group to adjust for data entry. It may be necessary to choose a waveform viewer from which to choose the input waveforms for the computation. This viewer will also receive the resulting waveform once it is computed.

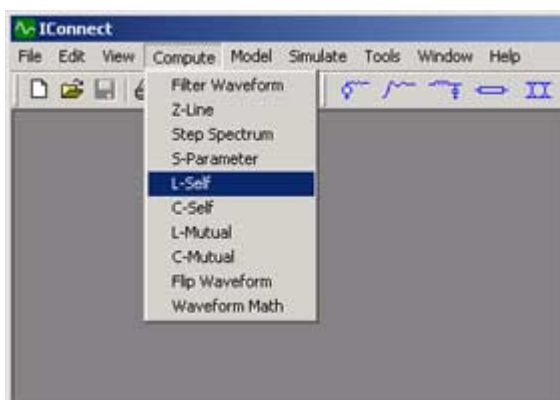


Figure 2-48: Choosing self inductance computation from the main menu.

To follow our example, select **Virtual Scope** in the drop-down list at the top of the acquisition window. The virtual scope is a tool for loading waveforms from your hard drive using the same interface as that used for oscilloscopes. Press the **Instrument Settings** tab at the top of the acquisition window to set the Virtual Scope options. Set the directory to **Tutorials\I/O Buffer Capacitance**, under the IConnect installation directory, e.g. **C:\Program Files\TDA Systems\IConnect**. Set the filter to ***.wfm**, and press the **Measure** tab to begin taking measurements.

Select the waveforms **Fixture_open.wfm** and **DUT_open.wfm** from the acquisition window. These are the waveforms with the ATE fixture and socket for the given device with and without the actual device in the socket. Press the **Acquire** button, which opens up a new waveform viewer containing the two waveforms.

2.5.2 Measurement Setup Tips

Wait for steady state

Each of these methods is based on the DC relationships between charge and voltage (capacitance) and flux and current (inductance). For this reason, it is necessary that all waveforms have reached a steady state value in order to get accurate results.

Use the correct reference waveform

Parasitic measurement requires information on what the ideal response would be - open circuit for self capacitance, or short circuit for self inductance. Mutual measurements only require input information to determine the initial and final voltage and values.

Pay attention to quiescent nearby leads or lines

Also pay attention to the far ends of the lines under test. For capacitance measurements, the lead(s) in question should be open-circuited at the far end, and adjacent leads short-circuited to ground, and vice versa for inductance measurements.

2.5.3 Self Inductance

In the computation window, choose **L-Self** as your computation, as shown in Figure 2-49. In the **DUT**: waveform chooser, select the measurement of your DUT short-circuited at the far end. For the **Short**: waveform chooser, select the reference waveform that corresponds to your measurement system shorted where the DUT would normally be placed. It may be necessary to enter the characteristic impedance and step amplitude for your measurement system prior to performing the computation, or change the step amplitude to the correct value.

Once these parameters have been chosen, pressing the **Compute** button performs the computation.

The resultant waveform will be placed in the chosen waveform viewer and the inductance is measured as the final asymptotic level that the inductance waveform reaches. If the cursors are used to make the measurement, they read directly in units of Henries.

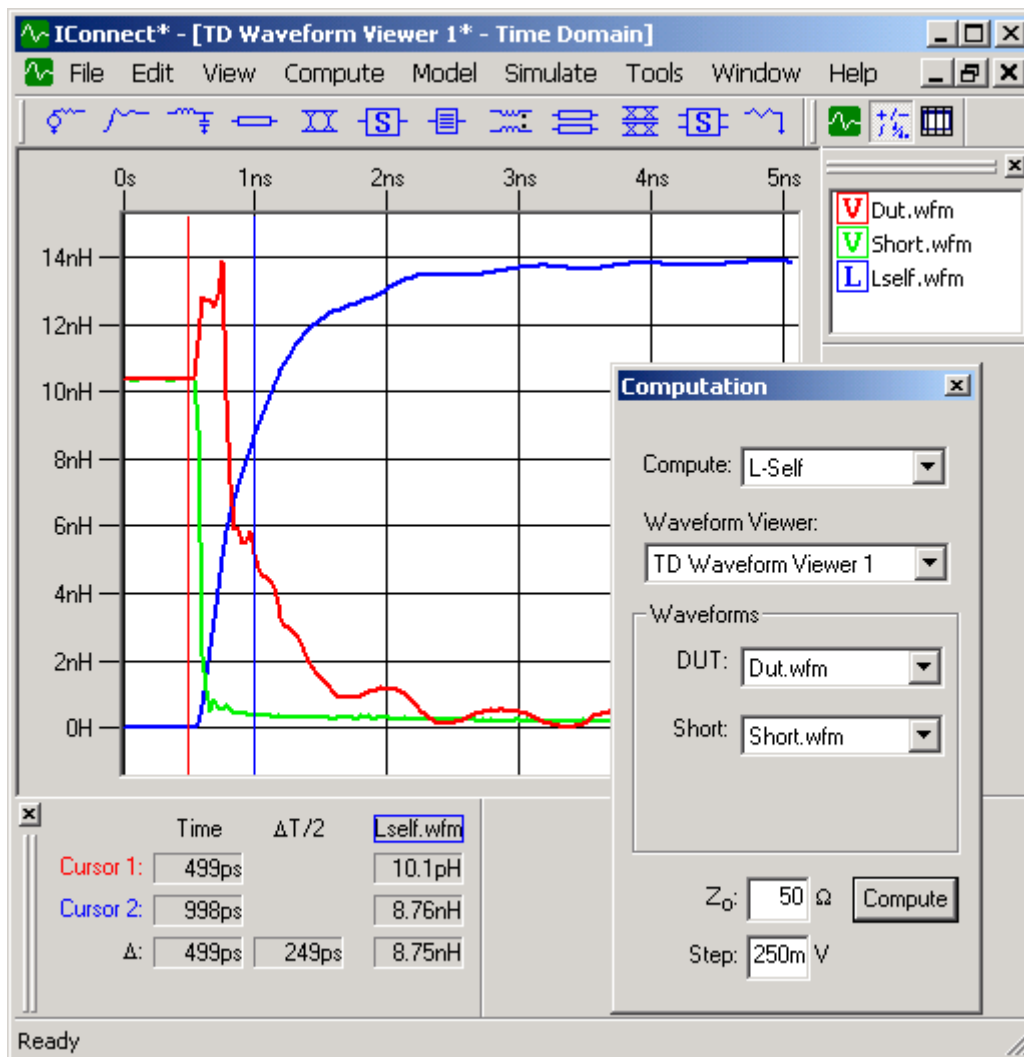


Figure 2-49: Displaying a self inductance measurement.

2.5.4 Self Capacitance

In the computation window, choose C-Self as your computation, as shown in Figure 2-50. In the **DUT**: waveform chooser, select the measurement of the DUT open-circuited at the far end. In the **Open**: waveform option, select the reference waveform, which corresponds to the measurement system open-circuited where the DUT would normally be placed.

It may be necessary to enter the characteristic impedance and step amplitude for your measurement system prior to performing the computation, or change the step amplitude to the correct value. Once these parameters have been chosen, pressing the **Compute** button performs the computation.

The resultant waveform will be placed in the chosen waveform viewer, and the capacitance is typically measured as the final asymptotic level that the capacitance waveform reaches. If the cursors are used to make the measurement, they read directly in units of Farads.

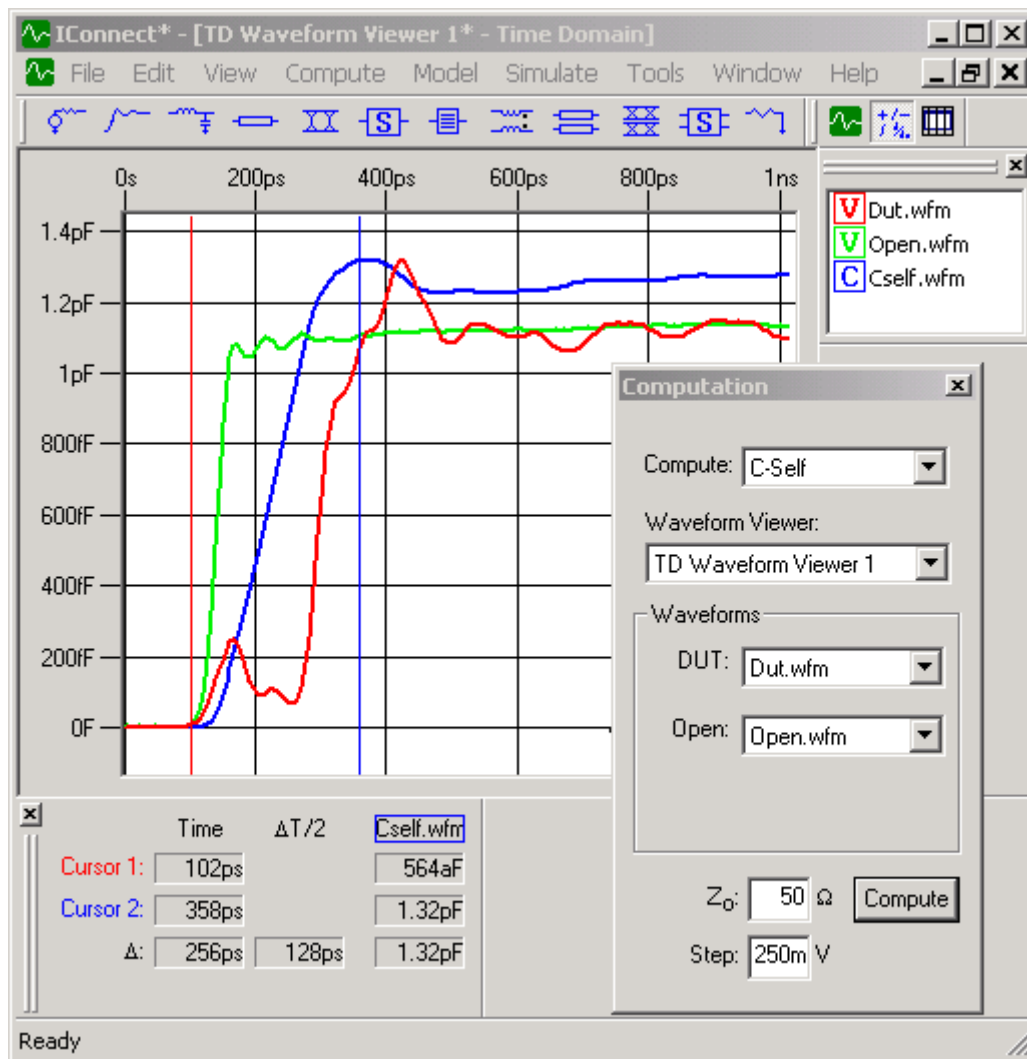


Figure 2-50: Displaying a self capacitance measurement.

2.5.5 Mutual Inductance

In the computation window, choose **L-Mutual** as your computation, as shown in Figure 2-51. In the **DUT**: waveform option, select the measurement of the near-side crosstalk on the quiescent line with adjacent leads open at the near end, short-circuited at the far end. In the **Short**: waveform chooser, select the waveform that corresponds to your measurement system shorted where the DUT would normally be placed. In the **Background**: waveform option, select the background waveform, which is acquired on the channel used to measure the DUT waveform when the DUT is disconnected.

It may be necessary to enter the characteristic impedance and step amplitude for your measurement system prior to performing the computation.

Once the above parameters have been chosen, pressing the **Compute** button performs the computation. The resultant waveform will be placed in the chosen waveform viewer, and the inductance is typically measured as the final asymptotic level that the inductance waveform reaches. If the cursors are used to make the measurement, they read directly in units of Henries.

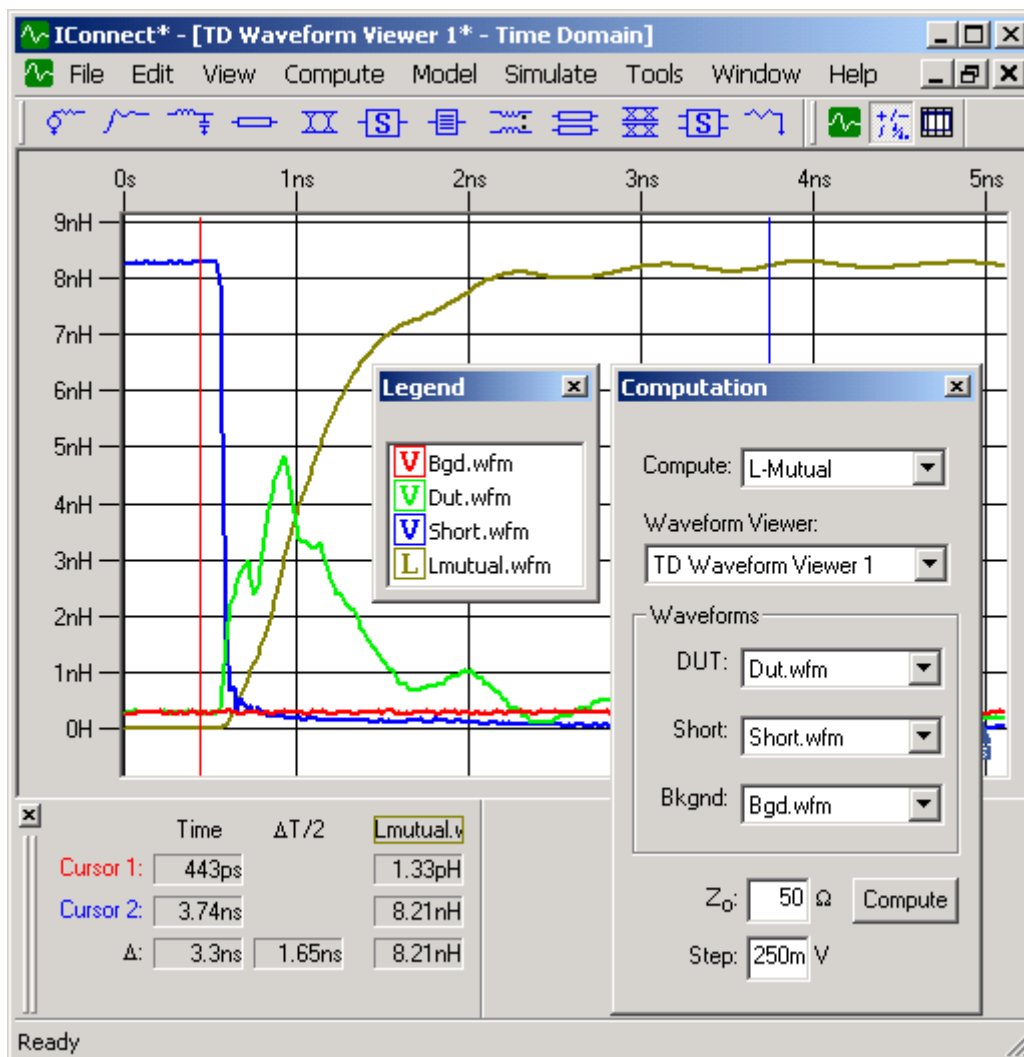


Figure 2-51: Displaying a mutual inductance measurement.

2.5.6 Mutual Capacitance

In the computation window, choose **C-Mutual** as your computation, as shown in Figure 2-52. In the **DUT**: waveform option, select the measurement of the near-side crosstalk on the quiescent line with adjacent leads shorted at the near end, open-circuited at the far end.

In the **Open**: waveform choice, select the waveform which corresponds to your measurement system open-circuited, where the DUT would normally be placed. In the **Background**: waveform option, select the background waveform, which is acquired on the channel used to measure the DUT waveform when the DUT is disconnected.

It may be necessary to enter the characteristic impedance and step amplitude for your measurement system prior to performing the computation. Once the above parameters have been chosen, pressing the **Compute** button performs the computation. The resultant waveform will be placed in the chosen waveform viewer, and the capacitance is typically measured as the final asymptotic level that the waveform reaches. If the cursors are used to make the measurement, they read directly in units of Farads.

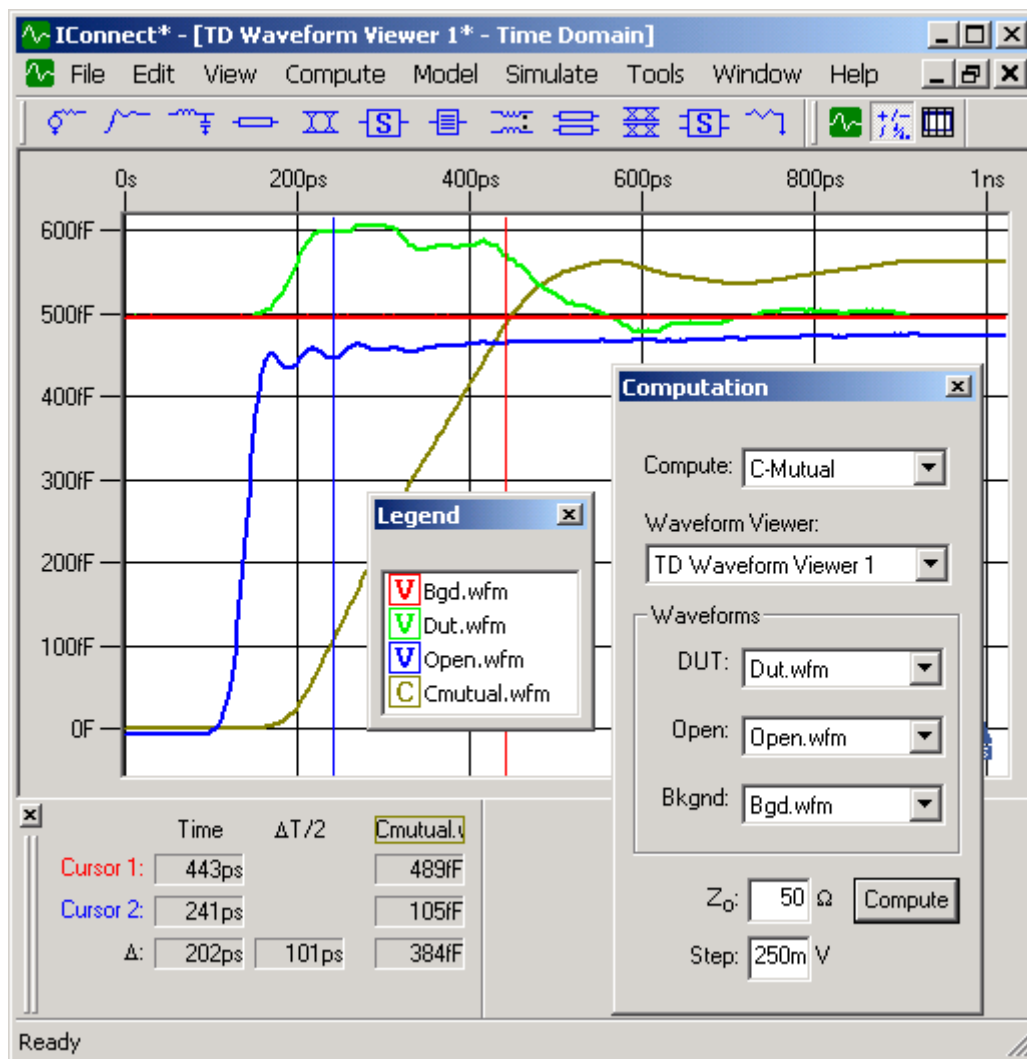


Figure 2-52: Displaying a mutual capacitance measurement.

2.5.7 I/O Buffer Input Capacitance

As shown in Figure 2-53, In the **DUT**: waveform chooser, select the **DUT_open.wfm**, the waveform of the fixture with the packaged part inside the socket. In the **Open**: waveform option, select **fixture_open.wfm**, the waveform of the fixture without the part inside the socket.

Follow the same procedure as for general C-Self for computation of input die capacitance. In the computation window, choose **C-Self** as your computation.

It may be necessary to enter the characteristic impedance and step amplitude for your measurement system prior to performing the computation, or change the step amplitude to the correct value.

Once these parameters have been chosen, pressing the **Compute** button performs the computation. The resultant waveform will be placed in the chosen waveform viewer, and the capacitance is typically measured as the final asymptotic level that the capacitance waveform reaches. If the cursors are used to make the measurement, they read directly in units of Farads. In this case, we are reading 4.6pF.

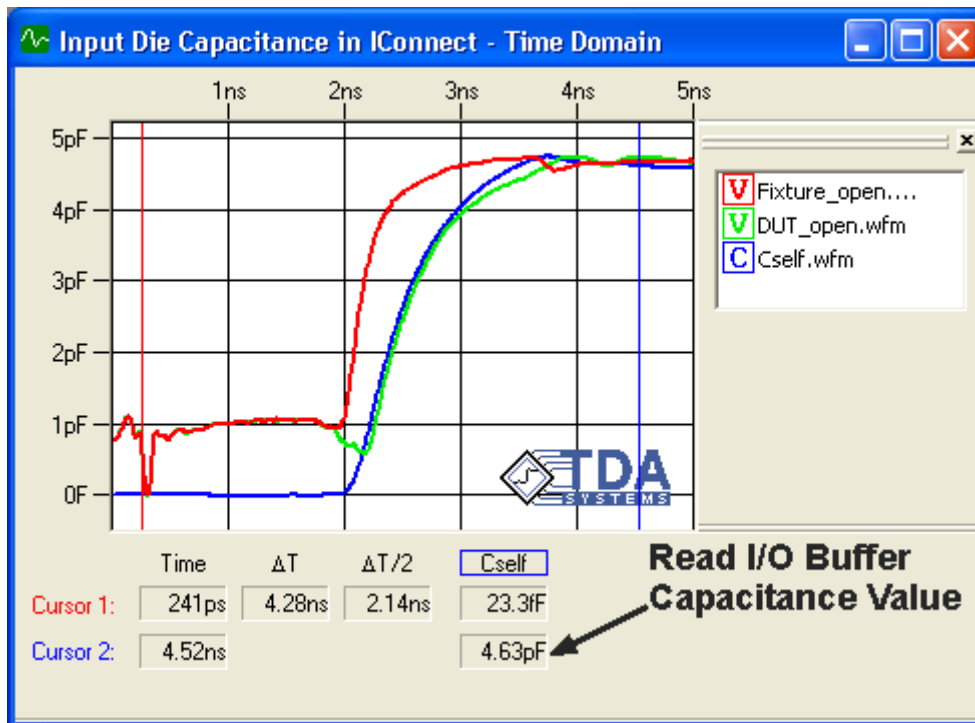


Figure 2-53: Extracting I/O buffer capacitance.

2.5.8 Simulate

Even though the computational methods described above are standardized by JEDEC and the IBIS Accuracy Standard, it is important to verify the accuracy of the computed values by running a simulation and comparing it against a measurement. Use either a Lumped or Lumped Coupled model, as shown in Figure 2-54, for the device model, depending on whether you are required or not required to compute mutual parameters and crosstalk. Set up a new Composite model viewer, add a TDR source, device model and termination, and run a simulation to validate the model. For more information on running a simulation and validating a model in IConnect, refer to [“Simulation” on page 6-1](#).

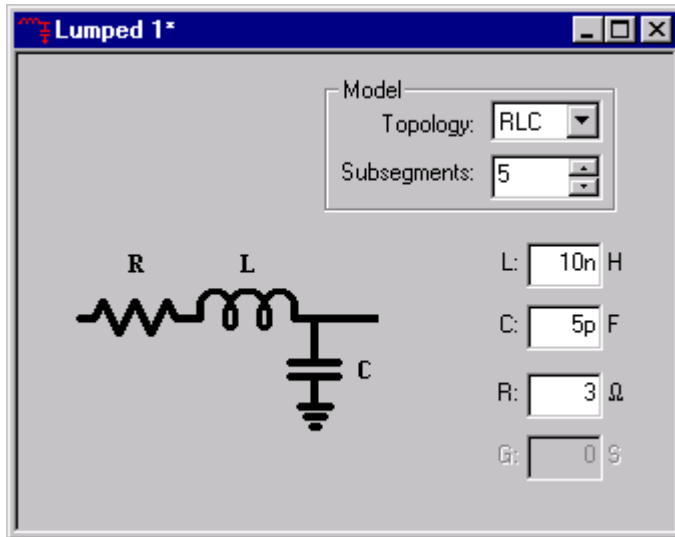


Figure 2-54: IConnect lumped model editor.

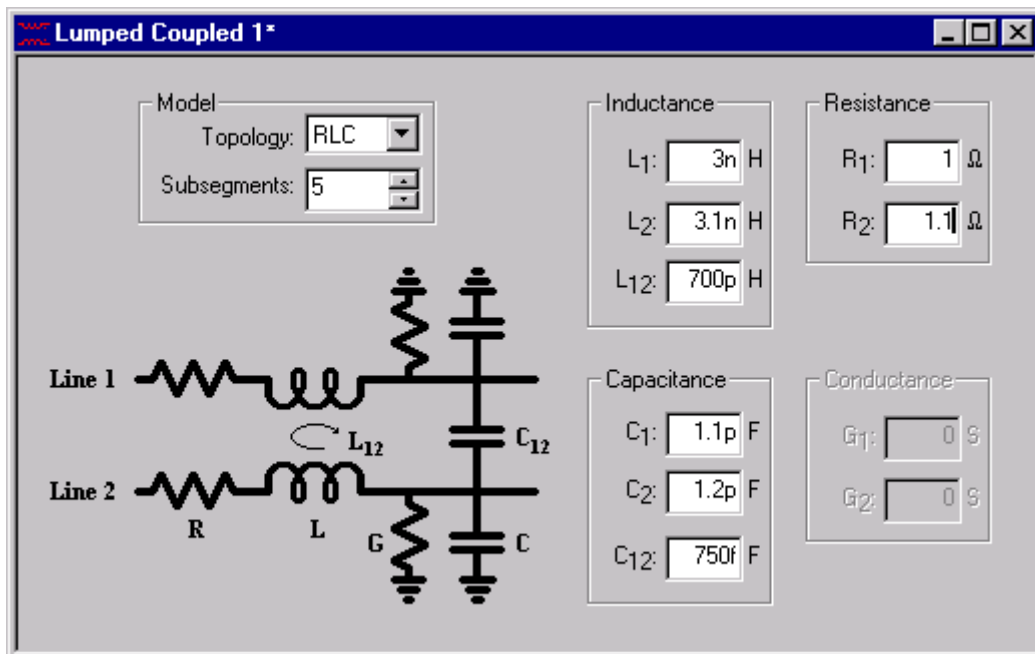


Figure 2-55: IConnect lumped coupled model editor.

2.6 Single and Coupled Line Modeling Tutorial

Welcome to the IConnect Single and Coupled Line Modeling Tutorial. This tutorial guides you through the steps necessary to obtain single and coupled lossless transmission line model in IConnect. The single and coupled transmission line model allows you to simulate the reflections, ringing and crosstalk in your interconnects, and is useful for short transmission line segments, electrically long packages and connectors, single-ended, differential, and coupled.

This tutorial leads you from a set of raw measurements to a complete SPICE model, and allows you to evaluate the model and verify its accuracy. Should you have additional questions about this tutorial, please consult [“Single Line” on page 5-13](#) or [“Symmetric Coupled Lines” on page 5-19](#) in the IConnect reference manual, or contact us to get more information through our technical support.

Single-Ended and Differential TDR

Single-ended and differential TDR modeling techniques allow you to extract accurate transmission line impedance for the traces on the board, differential and coupled transmission line models, as well as self and mutual inductances and capacitances of vias and other lumped components in the system. The same techniques are applicable for characterization of package, socket, connector and cable interconnects that are electrically long. The interconnect is electrically long when the rise time of the signals propagating through these interconnects is much longer than the electrical length of the interconnect.

2.6.1 Waveform Acquisition and Setup

You should begin by reviewing “[Preparing to Take Measurements](#)” on page 2-4 if necessary. It contains useful information on setting up your instrument to communicate with IConnect, as well as suggestions for taking good measurements.

A Single Line window can be created either from the **Model/Single Line** menu item, or by using the **Single Line** button on the Model Toolbar. The Symmetric Coupled Line Model can be launched from the **Model/Symmetric Coupled** menu item, or by using the **Symmetric Coupled** button on the Model Toolbar.

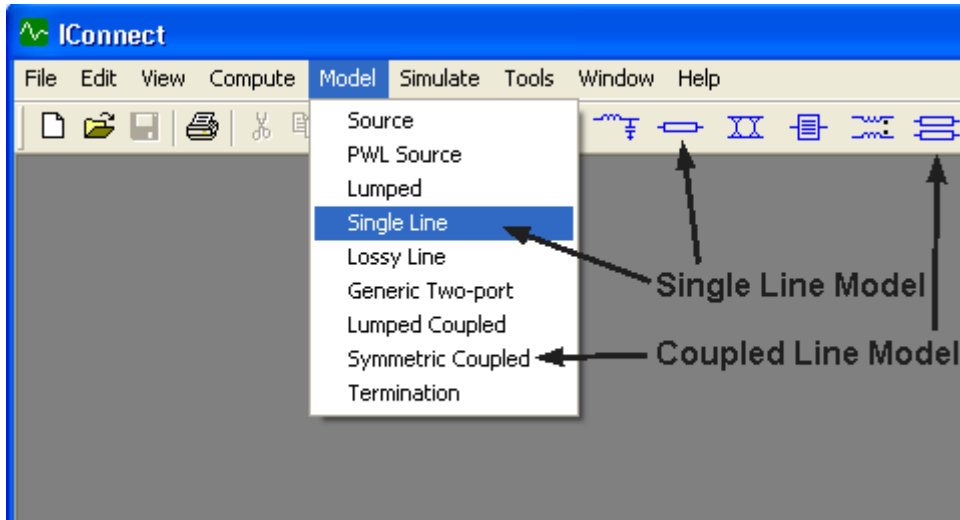


Figure 2-56: Creating a Single Line or Symmetric Coupled Lines model.

2.6.2 DUT Connections

For single line modeling, TDR waveforms must be acquired with only channel M1 connected to the DUT. The termination on the far end of the DUT is not important, and can be a short, an open, matched (50Ω) or any other impedance value.

In differential or common mode, both transmission and reference measurements must be differential (that is, the negative switching channel is subtracted from the positive switching channel) or common mode (the two positive switching channels are added together).

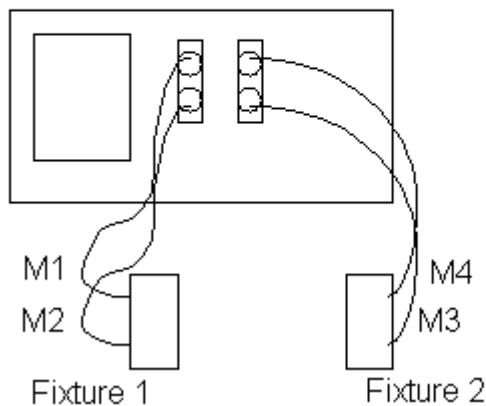
For a symmetric interconnect, we can use odd and even mode waveforms, acquired on the positive switching channel with the stimulus being in differential (for odd mode data) or common mode (for even mode data).

It is important to de-skew the TDR oscilloscope channels while doing differential measurements. Follow your oscilloscope de-skewing procedure before acquiring differential data.

Reference measurement:

Differential Stimulus is ON on channels M1, M2.

Reflection is measured on the same channels M1-M2



Reflection and transmission measurements:

Differential Stimulus is ON on channels M1, M2.

Transmission is measured on channels M3-M4

Reflection is measured on the channels M1-M2

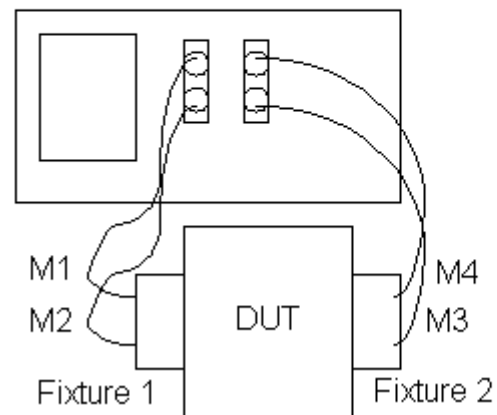


Figure 2-57: Measurement setup for differential reference, reflection, and transmission.

2.6.3 Loading Data

Two waveforms will be required for single line modeling - a reference open waveform and a reflection waveform. In the case of a symmetric coupled line model, the two reflection waveforms must be acquired -- in odd mode and even mode configurations.

Read the waveform descriptions in the scope interface window, shown in Figure 2-58 to capture a

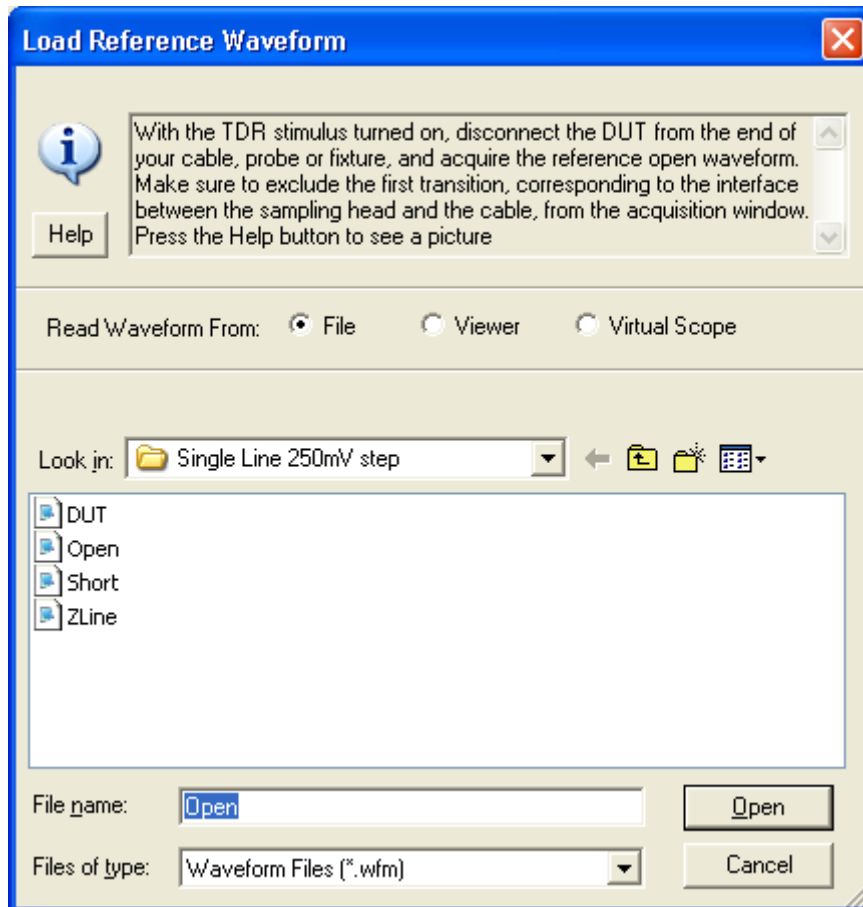


Figure 2-58: Scope interface window.

proper waveform from the instrument. The tutorial waveforms, named **Open.wfm** and **DUT.wfm**, can be found in the **Tutorials\Single Line** directory under the **IConnect** installation directory.

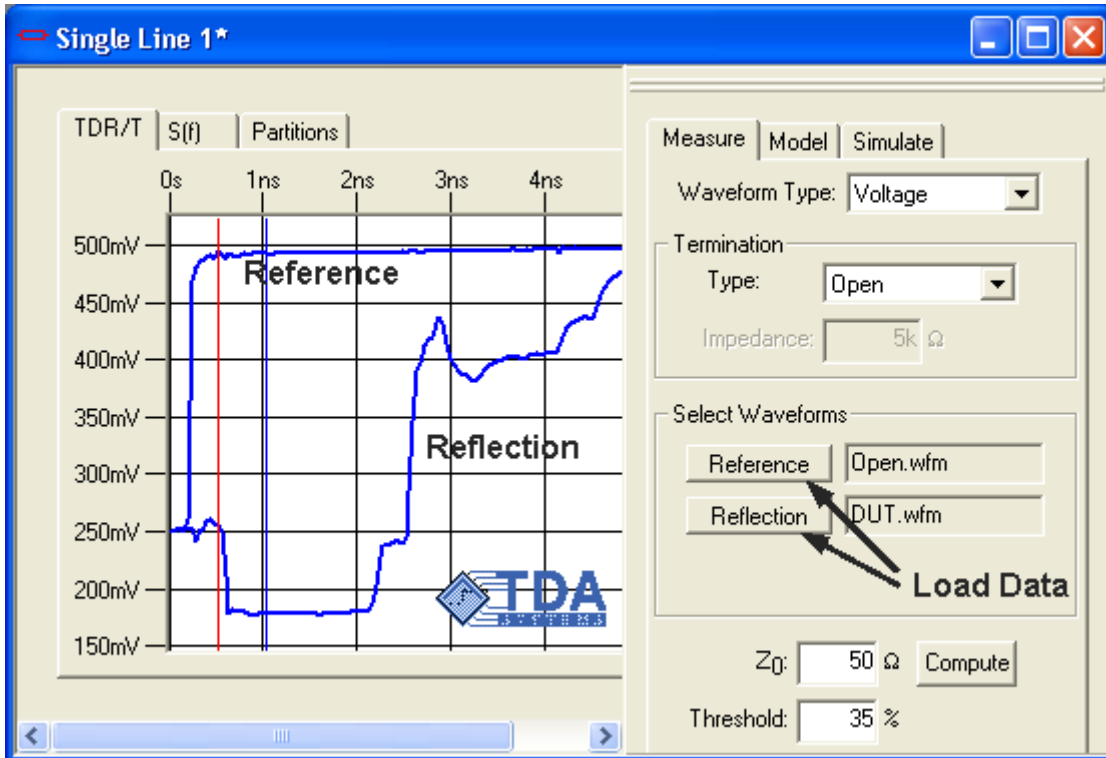


Figure 2-59: Loading reference and reflection waveforms into the single line model.

2.6.4 Computing Z-Line

Now make sure that the type of termination selected in the **Termination Type** drop-down box matches the actual termination at the end of your DUT. If your interconnect DUT terminates into a high-impedance I/O buffer, choose **Open** termination type, since a simple model for such buffer consists of an input buffer capacitance and an open termination.

Click **Compute** and the impedance profile (Z-line) is computed and displayed in the **Partitions** window. In the case of the symmetric coupled lines model, you will have two impedance profiles - Odd and Even - displayed simultaneously. To learn more about impedance profile computation, please refer to the ["True Impedance Profile \(Z-Line\) Computation Tutorial"](#) on page 2-8 or ["Impedance Profile Computation"](#) on page 8-6.

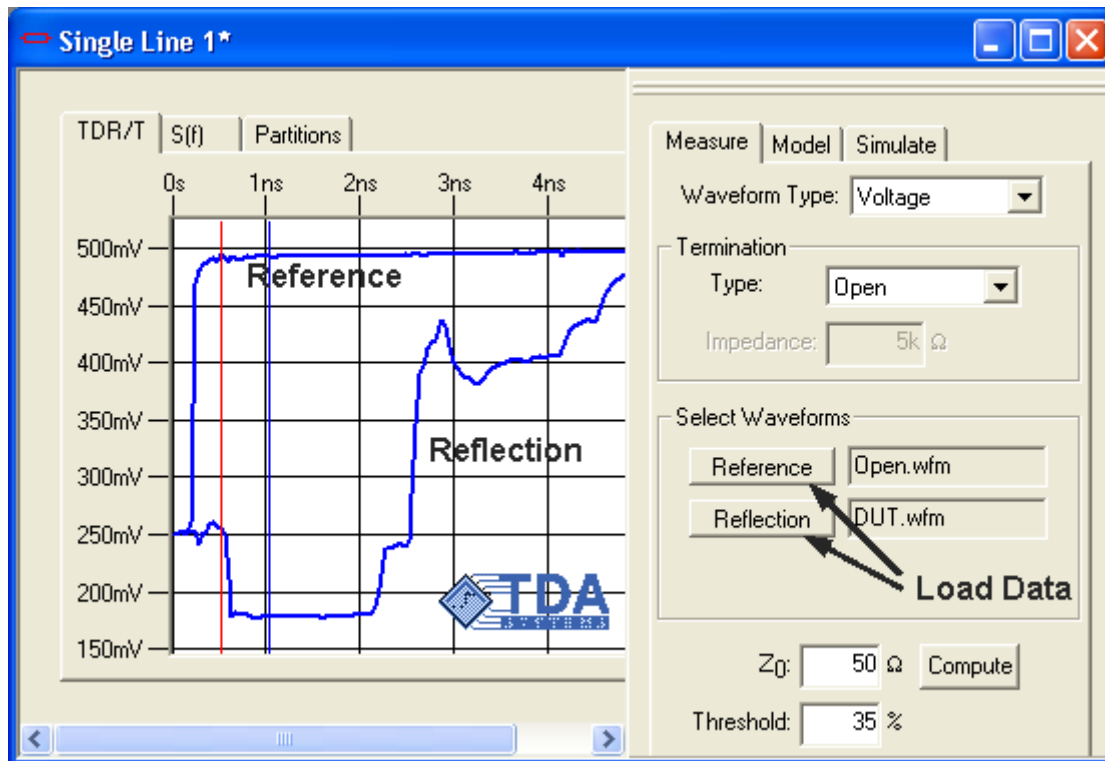


Figure 2-60: Compute the impedance profile by pressing the Compute button.

2.6.5 Windowing the Data

Click on the **Model** tab. Select the green cursor on the left, and move it to the beginning of the waveform response, which occurs around 250ps. Select the green cursor on the right, and move it to the end of the waveform response, which occurs around 2.5ns. By placing these two cursors, you are windowing out the portions of the waveform (covered by yellow) that correspond to the measurement setup rather than to the DUT itself.

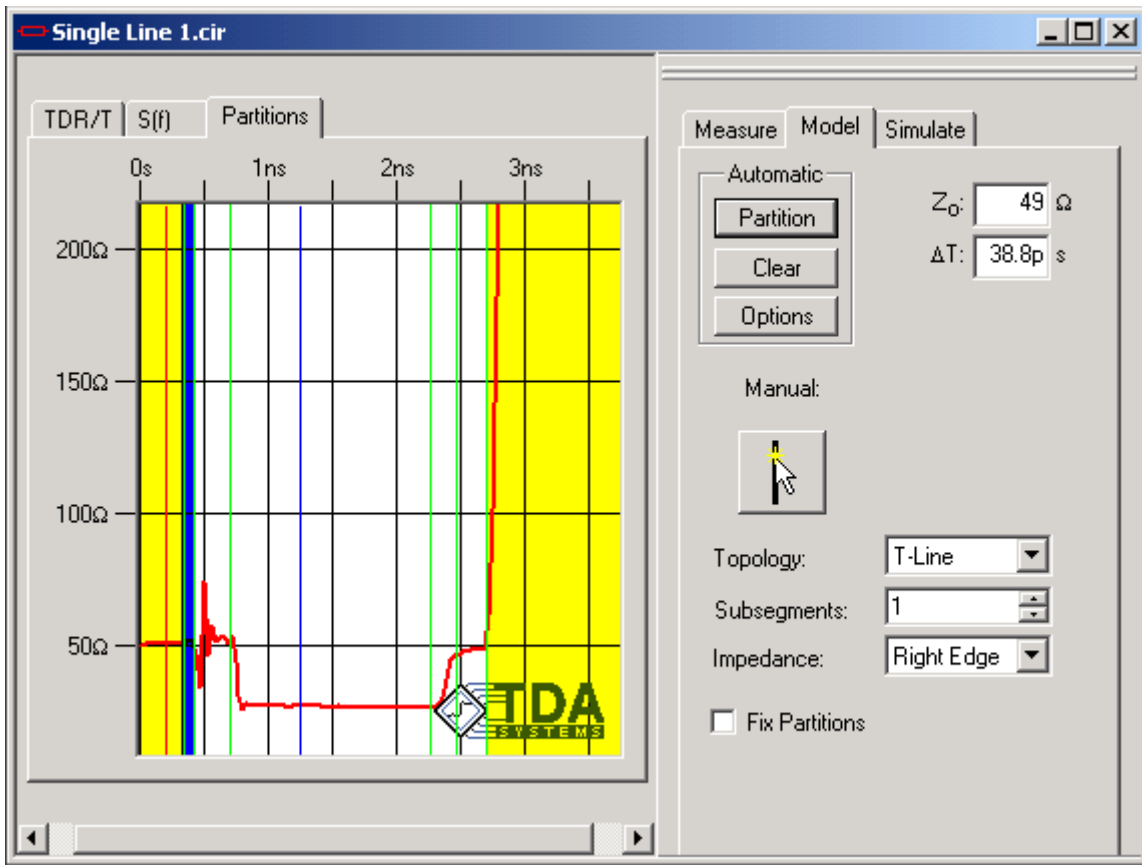


Figure 2-61: Windowing out the part of the response we don't want to model.

2.6.6 Partitioning Details

Even though IConnect will automatically partition the waveform, it is important to know how to place these partitions - to know where one line begins and the other one ends - in case you need to adjust the partitions placed by IConnect. When the TDR step propagates through the interconnect, the TDR waveform stays at a steady level. As soon as it sees the next transmission line with a different impedance in its path, it begins to change. Therefore, we place our partitions where the waveform begins to change.

The same rule applies to transmission lines and lumped elements - capacitances and inductances - since a capacitance or an inductance is just a very short piece of transmission line. Typically, you partition at the peaks of lumped elements, and select **Mean** value computation from the **Impedance** drop down box.

You can zoom in the **Partitions** window by holding down Control-Shift while dragging out the zoom area with your mouse.

2.6.7 Other Practical Details

As a general rule, partitions can be created at the right edge of a segment of fairly constant impedance, and the impedance value assigned using the **Right Edge** criteria. This approach results in accurate models for many types of structures. However, your choice of partition size and type should be based on your own judgment to achieve a simple model which is accurate enough for your needs.

Remember, a transmission line of delay T_d requires a risetime less than $2T_d$ to resolve it. In other words, any segment with $T_d < T_r/2$ need not be modeled, where T_r is the risetime of the target system in which the model will be used. Each of the partitions uses **T-Line** topology with a single segment. Note that additional RLGC topologies are available for other modeling applications.

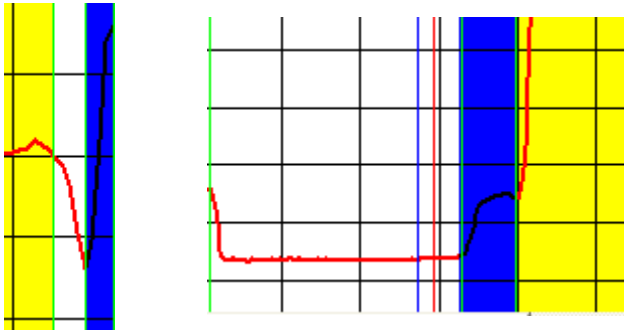


Figure 2-62: Single line partitioning examples.

2.6.8 Symmetric Coupled Model Details

The files for this symmetric coupled line modeling example are found in the directory **Tutorials/ Coupled Lines** under the IConnect installation directory. Open the Symmetric Coupled Line modeling window, and load the required even (even.wfm), odd (odd.wfm) and reference (short.wfm) waveforms using the appropriate buttons under the **Measure** tab. Click the **Compute** button, and use the same partitioning procedure as that for the Single Line model.

For our symmetric coupled line example, it may be instructive to note that the physical structure from which the measurements were taken was a set of two coupled lines, which diverged to SMA connectors at either end. From this knowledge, we would expect to see, qualitatively speaking, a coupled section in the middle of the model, and two short uncoupled sections at either end.

The question remains, however, as to the degree of coupling and the lengths of the coupled and uncoupled segments. The leftmost partition can be seen to be uncoupled, since the even- and odd-mode responses are the same. To inform IConnect of this fact, select the leftmost partition and uncheck the **Coupled** checkbox. Its edge is where the even- and odd-mode impedance profiles begin to diverge.

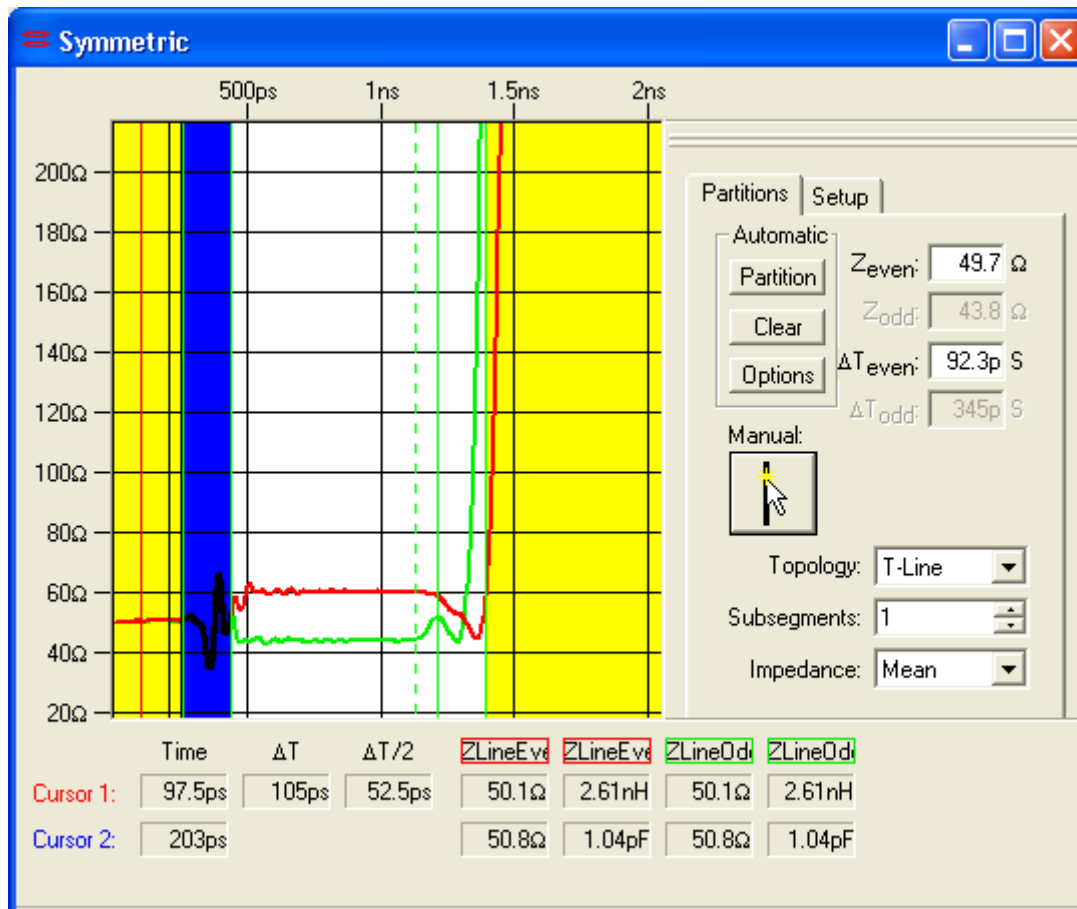


Figure 2-63: Partitioning the first, uncoupled segment.

The second partition, highlighted in blue, is obviously coupled, since the even- and odd-mode impedances are different from each other. If this partition is highlighted and the **Coupled** check-

box is checked, a second dotted odd-mode partition divider should appear within the selected partition.



Figure 2-64: Partitioning the second, coupled segment.

The purpose of this odd-mode divider is to mark where the odd impedance changes. The edge of the even divider should be at the right edge of the uniform impedance segment in the even-mode (red) waveform, and the edge of the odd (dotted) divider should be at the edge of the uniform segment in the odd-mode (green) waveform. If the odd-mode divider turns red when you move it, you have moved it to a position in which the model parameters preclude being able to build a passive model, which could result in simulation stability problems. The divider must be moved, or the parameters manually adjusted, before the model can be saved.

2.6.9 Saving the Model

We are returning to the Single Line modeling example.

Save the Single Line model to a meaningful name, such as **Single Line.cir**, in the **Single Line** directory. This may be done via the **File/Save** command, from the standard toolbar, by using the <Ctrl+S> keyboard shortcut, or by right-clicking in the model window and choosing **Save** from the resultant context menu. Make sure to select the appropriate SPICE output format when saving the model.

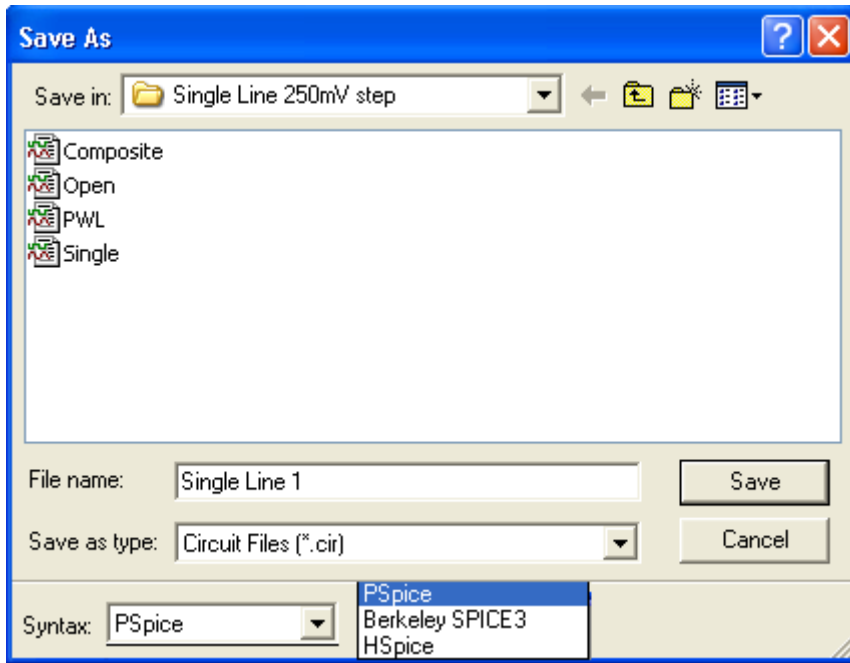


Figure 2-65: Saving the single line model.

Viewing the Model

Once the model is saved, the easiest way to view the saved model is to right click in the modeling window and click **View Netlist**.

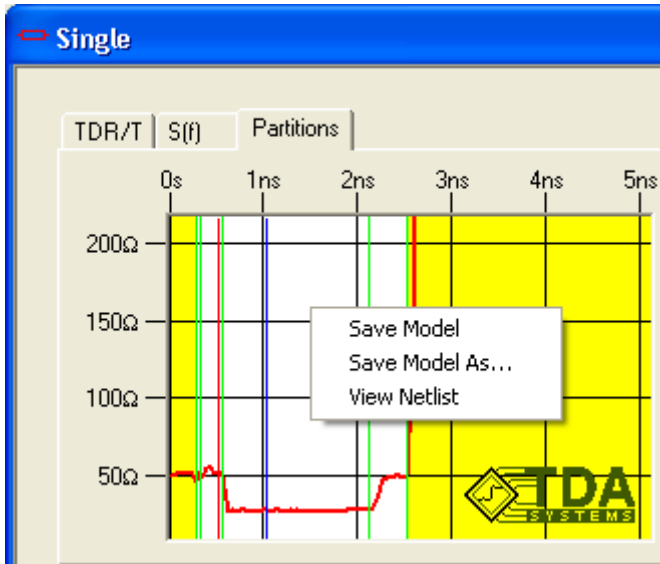


Figure 2-66: Use the context menu to quickly view the model netlist.

2.6.10 Simulating

Click on the **Simulate** tab, and then click on **Simulate** to automatically generate the model and simulate it in your simulator of choice. You can choose a link to your preferred simulator in **Tools/Options/Simulator Options**. Make sure that your selected file format matches your simulator selection.

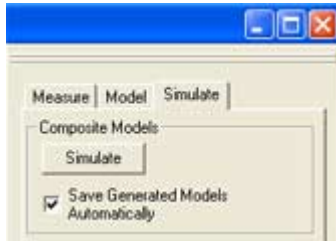


Figure 2-67: Click the **Simulate** button to generate a composite model and launch the simulation.

Now the Composite model, including the piecewise linear TDR source, single line model and termination, is generated automatically. You can double-click on any of the composite model elements to view the included models. In the case of a symmetric coupled lines model, two composite models will be generated, to validate the accuracy for both even and odd modes of propagation.

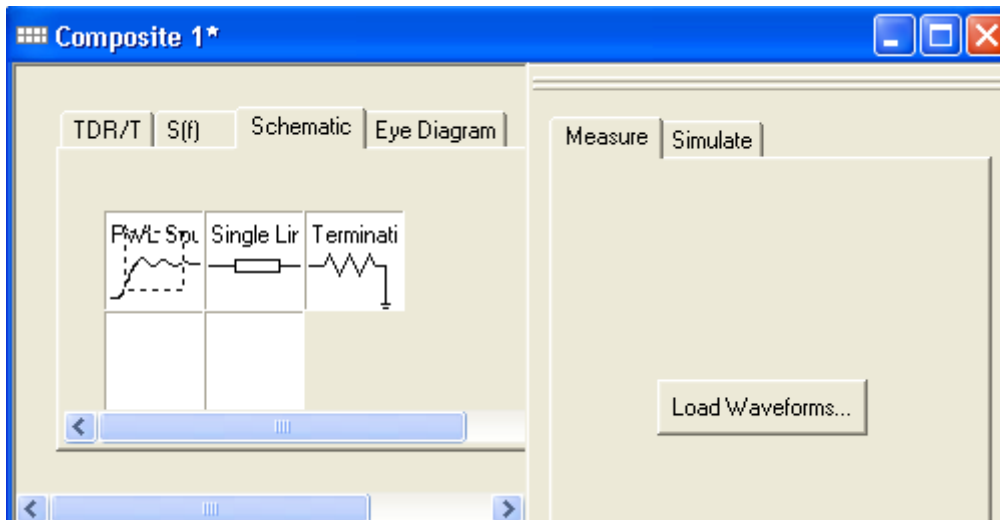


Figure 2-68: Composite model with the DUT submodel, a source to drive it, and a resistance to terminate it.

2.6.11 PWL Source

If you believe that you need to increase the precision with which IConnect models the TDR source, decrease the **Max Error** settings and click **Partition** to repartition the model. Make sure to save it before continuing. To reverse the polarity of the TDR source, simply change the sign in the **Scale** box, in the **Model** group box.

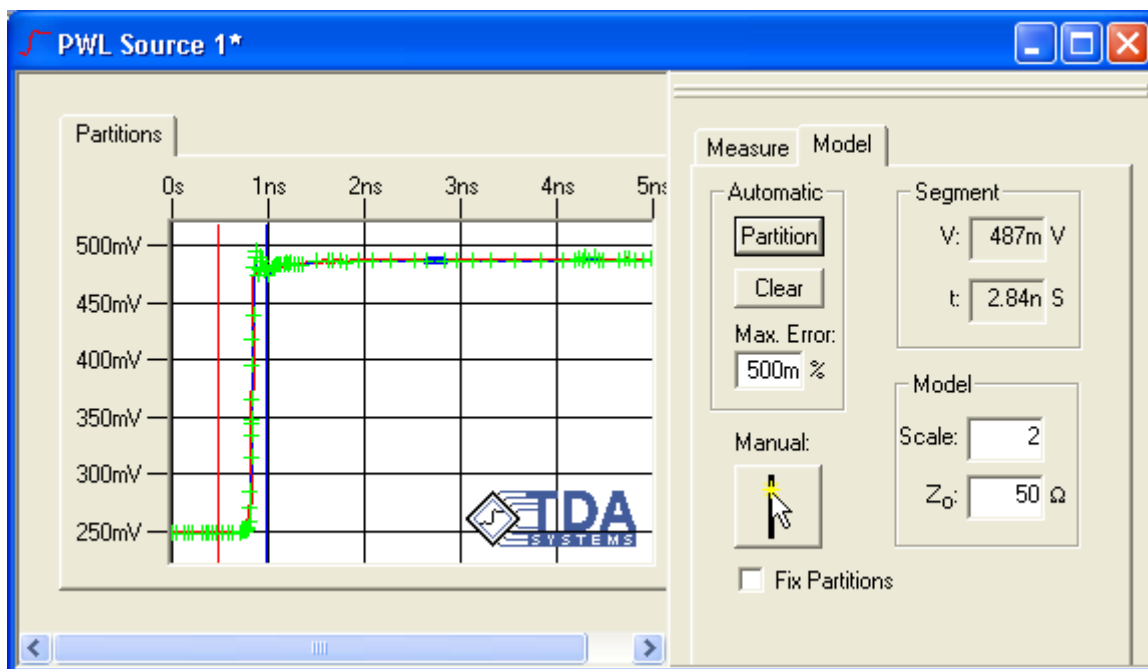


Figure 2-69: You can edit the automatically generated PWL source if you wish.

2.6.12 Analyzing the Results

After the simulation is complete, the simulated waveforms are placed in the composite model waveform viewer. Click on the **TDR/T** or **S(f)** tabs to observe the correlation. As you can observe, even without modeling the losses, the correlation is very good. However, to be absolutely certain of the accuracy of the model, you are advised to achieve correlation for both reflection and transmission waveforms. If the match is satisfactory, then the modeling session is done. If it is not, it may be necessary to return to the various model editors, alter the models, and resimulate.

If the waveforms are offset on the screen, drag one of them with the mouse to overlay on top of the other. In the case of a symmetric coupled line model, the match for both even and odd mode waveforms must be ensured.

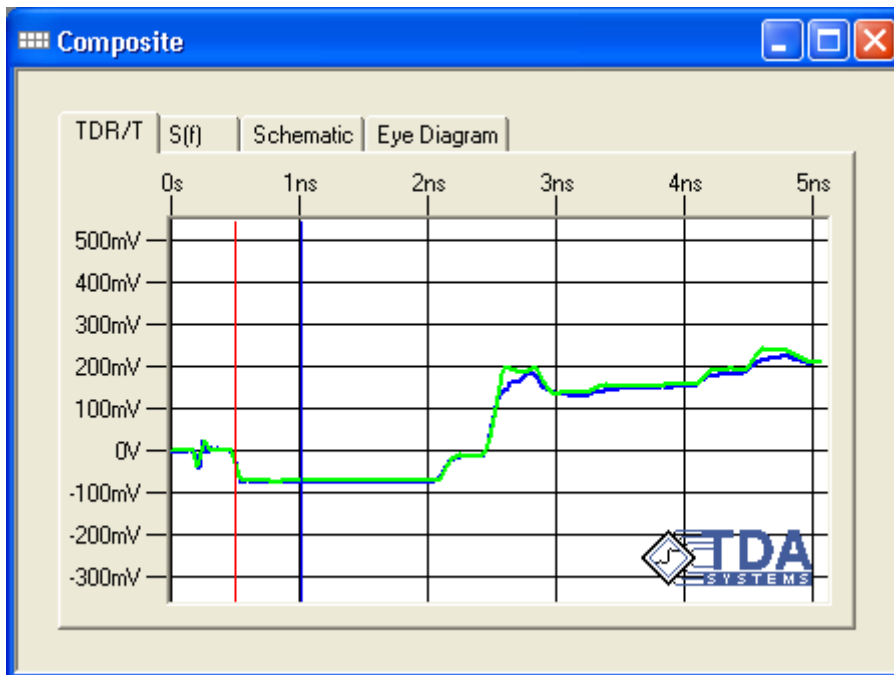


Figure 2-70: Viewing correlation in the Composite model viewer.

2.6.13 Risetime Filtering

In most cases the model you are creating will be used in a system whose risetime is slower than the 30-40ps risetime of the TDR measurement head. In that case, it makes sense to filter the waveforms prior to comparison to see how both measurement and simulation would compare at a lower risetime. The lower risetime will correspond better to the fast corner of the actual drivers used in the system with the given interconnect under test.

To filter multiple waveforms, choose **Filter Waveform** from the **Compute** drop-down list. Type the desired risetime in the text box labeled **Risetime** at the bottom of the Computation Window. Select the waveforms to be filtered, right click on a selected waveform, and choose **Filter** from the menu.

The correlation at the actual device rise time is more important than the correlation at the TDR rise time.

2.7 Lossy Line Modeling Tutorial

Welcome to the IConnect Lossy Line Modeling Tutorial. This tutorial guides you through the steps necessary to obtain a lossy transmission line model and eye diagram in IConnect. The lossy transmission line model allows you to simulate the frequency dependent skin effect and dielectric losses for your interconnects, and is useful for single line and balanced differential line modeling (the latter in the odd mode configuration), as well as predicting the eye diagram degradation due to losses. The symmetric coupled lossy lines model allows you to simulate the skin effect and dielectric loss, together with the intersymbol interference (ISI) and crosstalk effects.

It also allows you to predict the eye diagram degradation due to losses and ISI in the interconnects. This tutorial leads you from a set of raw measurements to a complete SPICE model, and allows you to evaluate the model and verify its accuracy. Should you have additional questions about this tutorial, please refer to [“Lossy Line” on page 5-27](#), or contact us to get more information through our technical support.

2.7.1 Waveform Acquisition and Setup

You should begin by reviewing “[Preparing to Take Measurements](#)” on page 2-4 if necessary.

In addition to the best measurement practices described in “[Preparing to Take Measurements](#)” on page 2-4, it is particularly important for lossy line and eye diagram measurement to select a window that is long enough that it allows the DUT waveforms to completely settle to their steady state DC levels. We are about to make a high-frequency analysis of the DUT, and we need to allow the signal to settle to make this analysis accurately.

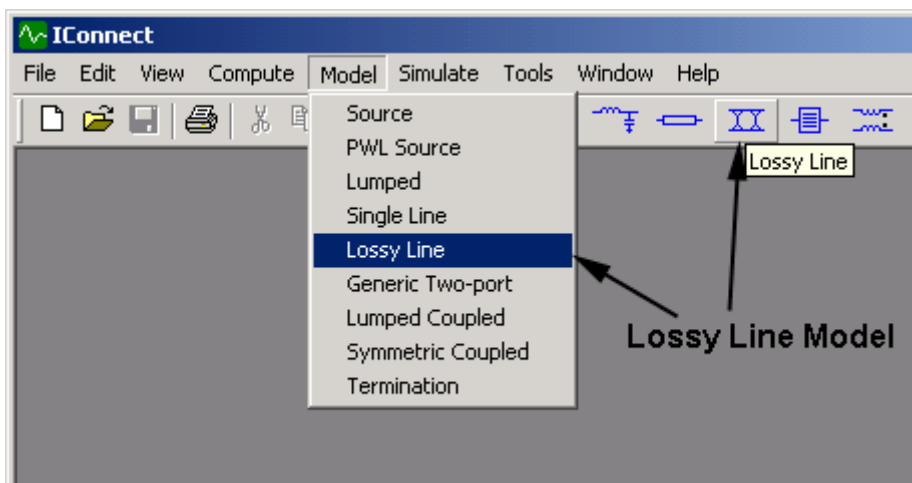


Figure 2-71: The lossy line model can be created via the Model menu or Model toolbar.

A Lossy Line model can be created either from the **Model/Lossy Line** menu item, or by using the **Lossy Line** button on the Model Toolbar. A Symmetric Coupled Lossy Lines model can be launched from the **Model/Symmetric Coupled Lossy Line**, or using the **Symmetric Coupled Lossy Line** button on the Model toolbar.

Note that losses are a characteristic of the material of the interconnect. Therefore, to get the best results, it is important to extract the losses on a transmission line of controlled impedance. It is possible to extract the losses based on structure with changing impedance, but if changes are very large, it will be difficult to extract losses accurately.

2.7.2 Choosing the Extract Termination Type

You can select the **Matched** load in the **Measure** tab on the right of the IConnect window. This termination type will provide lossy transmission line data based on the TDT (Time Domain Transmission) measurement, with the far end of the device terminated in 50Ω . You can also use the **Open** termination type, based on TDR (reflection) waveforms, in case TDT is difficult to perform (e.g., far end of the device is inaccessible). However, the transmission measurement is generally more accurate.

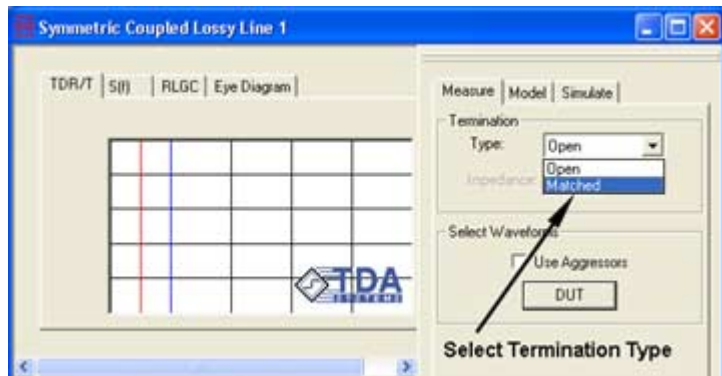


Figure 2-72: Choosing a termination type for the Lossy Line model.

2.7.3 DUT Connections

For single line modeling, TDR waveforms must be acquired with only channel M1 connected to the DUT. The termination on the far end of the DUT is not important, and can be a short, an open, matched (50Ω) or any other impedance value.

In differential or common mode, both transmission and reference measurements must be differential (that is, the negative switching channel is subtracted from the positive switching channel) or common mode (the two positive switching channels are added together).

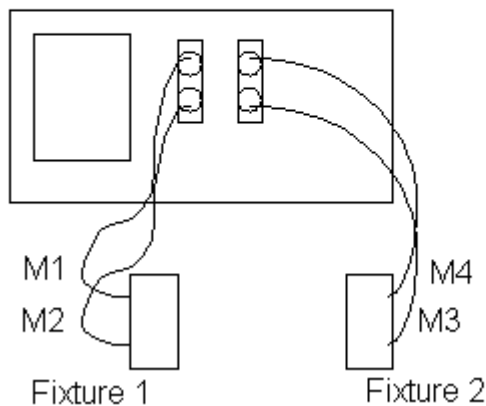
For a symmetric interconnect, we can use odd and even mode waveforms, acquired on the positive switching channel with the stimulus being in differential (for odd mode data) or common mode (for even mode data).

It is important to de-skew the TDR oscilloscope channels while doing differential measurements. Follow your oscilloscope de-skewing procedure before acquiring differential data.

Reference measurement:

Differential Stimulus is ON on channels M1, M2.

Reflection is measured on the same channels M1-M2



Reflection and transmission measurements:

Differential Stimulus is ON on channels M1, M2.

Transmission is measured on channels M3-M4

Reflection is measured on the channels M1-M2

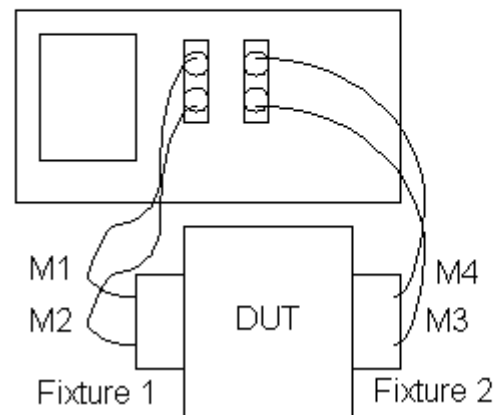


Figure 2-73: Measurement setup for differential reference, reflection, and transmission.

2.7.4 Loading Data

Now we can acquire the three required waveforms for matched-load lossy line modeling: Reference, Reflection, and Transmission. First, make sure to select the appropriate instrument in the Acquisition window. Click the button for each waveform in the **Measure** tab, and acquire the appropriate waveform from the instrument.

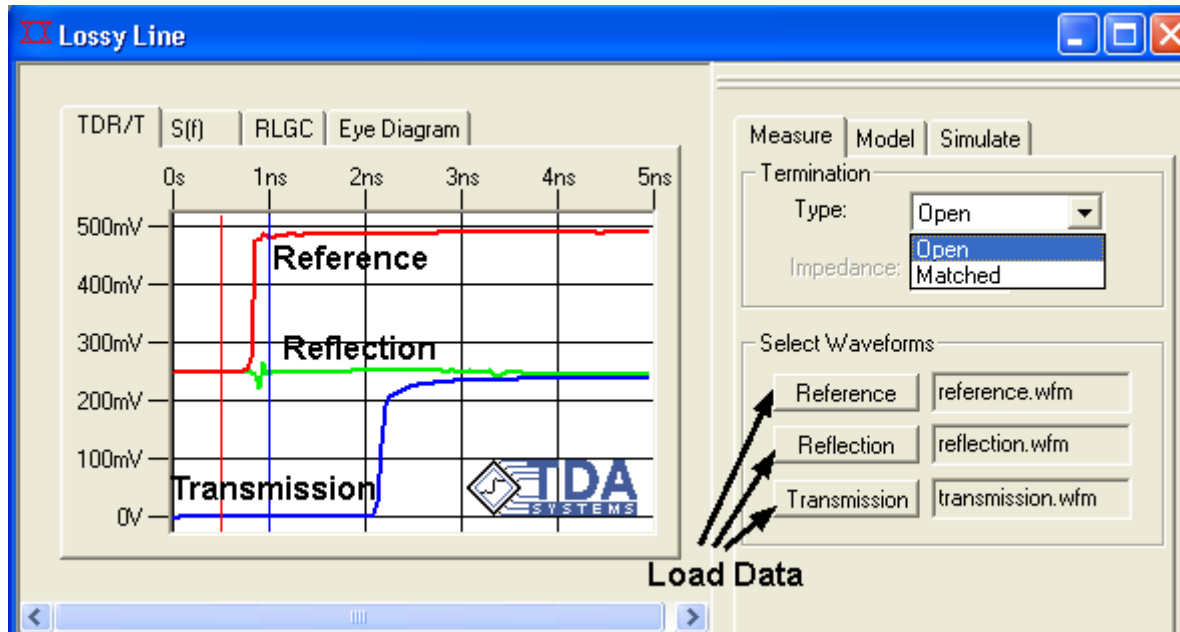


Figure 2-74: Each waveform is loaded by clicking on its respective button.

For symmetric lossy coupled lines, click on the **Even Mode** radio button, and acquire the two additional waveforms in the even mode.

TDR/T waveforms in the **Matched** termination extraction must be acquired with 50Ω cables connected to both DUT ports, thus ensuring a 50Ω termination at the far end of the DUT. The cables, probes and fixtures used to connect the DUT to both channels of the TDR oscilloscope are assumed by IConnect to be of the same length.

Three waveforms will be required for single lossy line modeling - a reference open waveform, and TDR and TDT waveforms. In the case of a symmetric coupled lossy lines model, these three waveforms must be acquired in odd mode configuration. In addition to the three odd mode waveforms, two waveforms for the even mode measurements will be required. Read the waveform descriptions in the scope interface window, shown in Figure 2-75, to capture a proper waveform from the instrument. The three tutorial waveforms, named **Reference.wfm**, **Reflection.wfm**, and **Transmission.wfm**, can be found in the **Tutorials\Lossy Line** directory under the IConnect Installation directory.

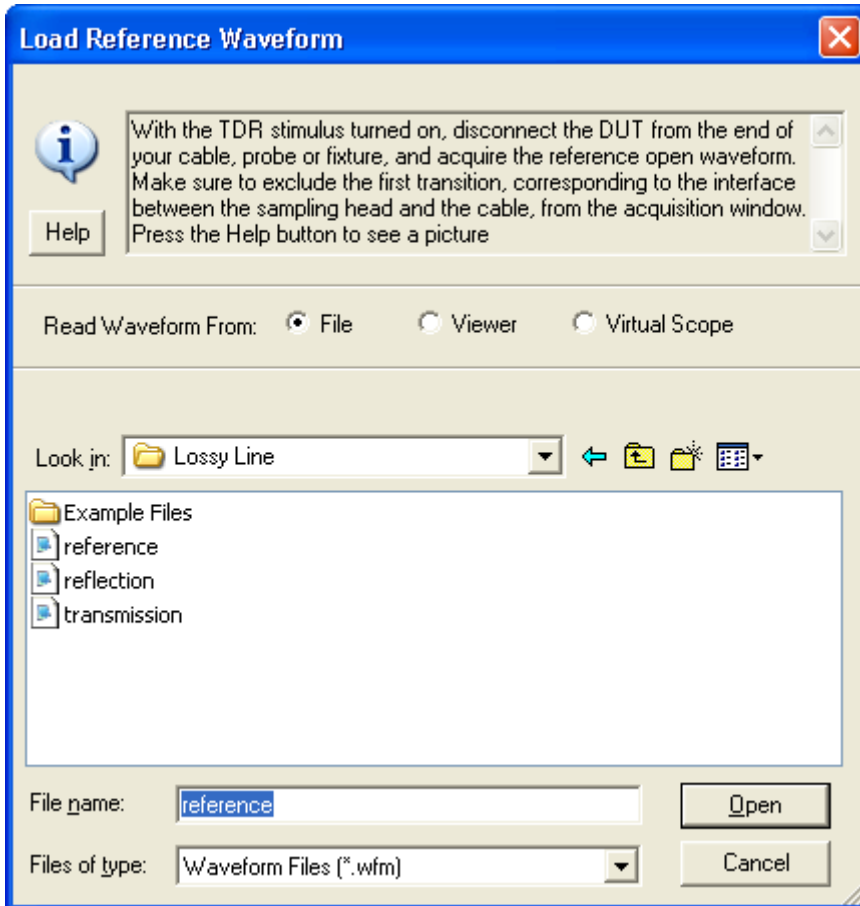


Figure 2-75: Instrument interface window.

2.7.5 Model Extraction

Once the data is loaded, click on the **Model** tab. Click **Compute** in the **Optimization** group; the lossy line model is being extracted. If you know any of the parameters, you can fix them during the optimization by checking the **Fix** box for the given parameter. In fact, it may be preferred to extract the DC resistance of the interconnect beforehand, using a DC meter, and fix it during optimization.

During the optimization, the text of the **Compute** button changes to **Cancel**, and the number of iterations performed is continuously displayed. After the maximum number of iterations reached, or the desired resolution is achieved, the computation stops, and the results of a simulation based on the newly extracted model are displayed in the **TDR/T** tab.

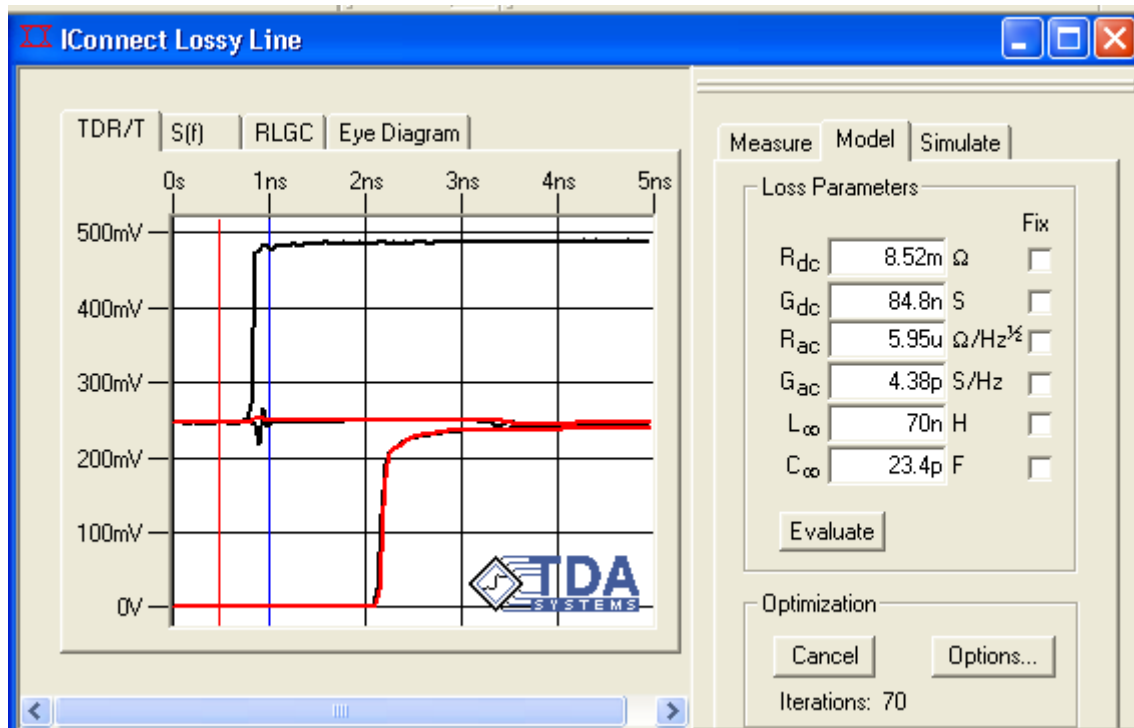


Figure 2-76: The model is optimized by clicking on the Optimize button (labeled Cancel because an extraction is running).

2.7.6 Model Accuracy

The correlation between the simulated (red) and measured data (black) in time domain is the key to accuracy of the extracted model. You can, however, also observe the correlation in frequency domain, by pressing the **S(f)** tab. You can extract a better accuracy model, with correlation in both the time and frequency domains, by increasing the precision and the number of iterations in the optimization **Options**, or by changing the values manually and pressing the **Evaluate** button. Keep in mind that in digital design, time domain correlation is much more important than frequency domain correlation.

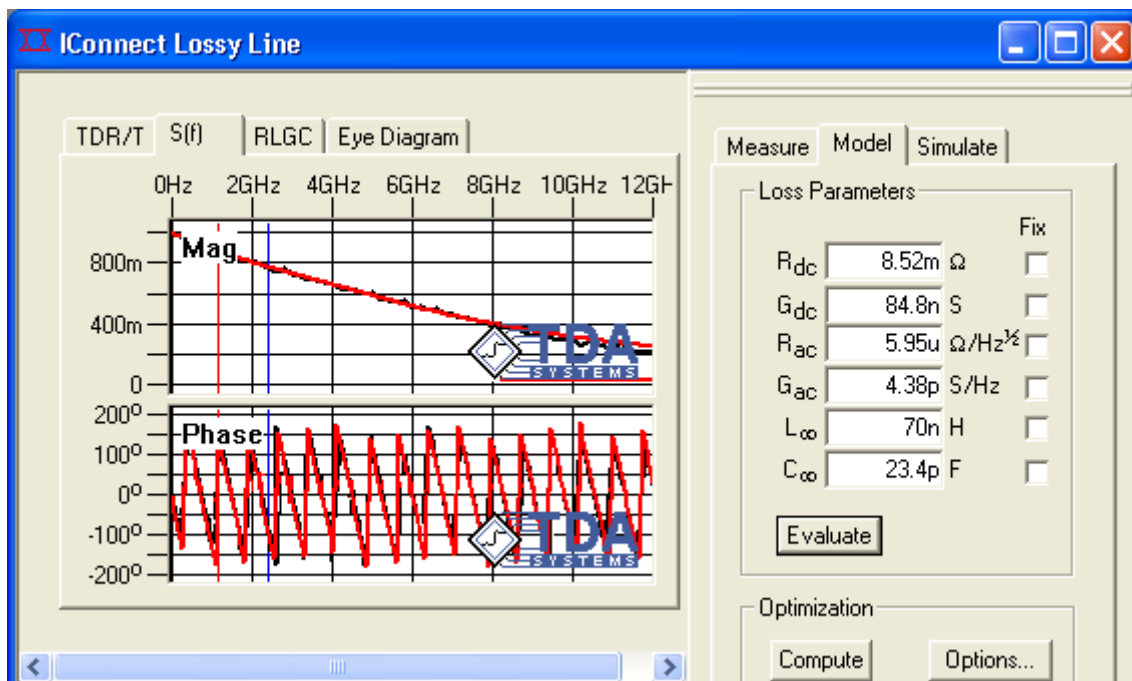


Figure 2-77: Lossy line model correlation in the frequency domain.

2.7.7 Other Practical Details

You may also repeatedly click the **Optimize** button. The values may change slightly, but the new values will still give you an accurate model within the specified **Precision** parameter, accessible under the optimization **Options** button.

You may play what-if scenarios by changing any of the parameters manually, and clicking the **Evaluate** button. The new values will be evaluated and the new simulated waveform will be shown in red. Note that if the red waveform does not change substantially as you change the values in the window, this implies that your changes are not significant enough, or that the given loss mechanism is too small to matter.

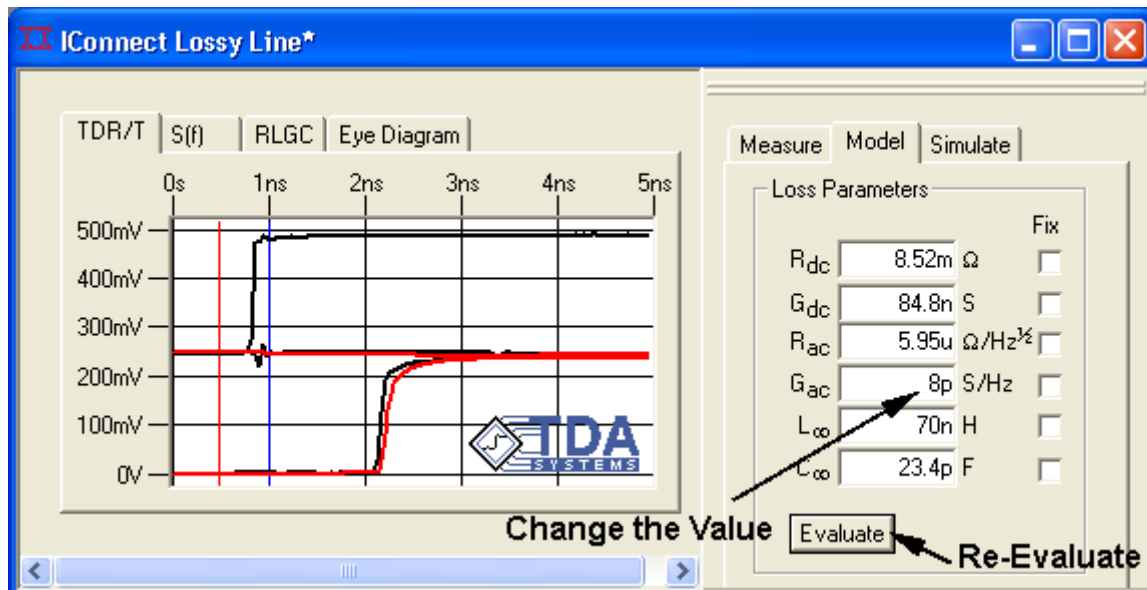


Figure 2-78: Lossy line response to altering its optimized parameter values.

2.7.8 Choosing the Output Format

When saving the model to a file, you can select between the Hspice W-element model and a non-Hspice lumped behavioral model. When saving to a lumped model, make sure to enter realistic rise time number to decrease the required number of subsegments for the model to be accurate to that rise time. Alternatively, you can decrease the number of subsegments manually. You need to have at least 3 segments per length of TDR rise time.

You can use the scaling parameter (“**Scale parameters by**”) to generate a per-unit length model, or to easily scale the length of the model. In addition, you can save the model in a table vs. frequency format, by clicking on the **RLGC** tab and exporting the frequency dependent R, L, G and C values.

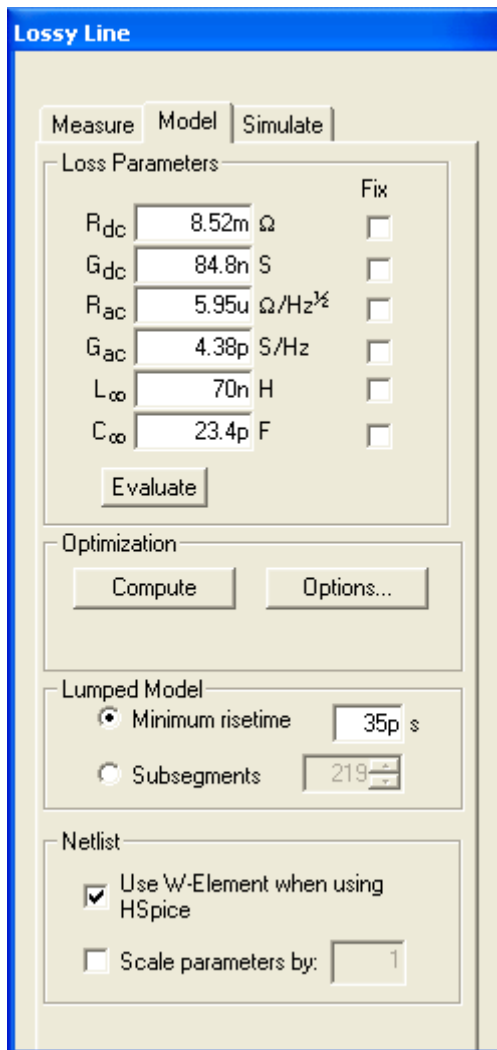


Figure 2-79: Values in the Lumped Model and Netlist boxes affect how the Lossy Line model is written to disk.

2.7.9 Saving the Model

Save the model: make sure you select HSpice while saving it.

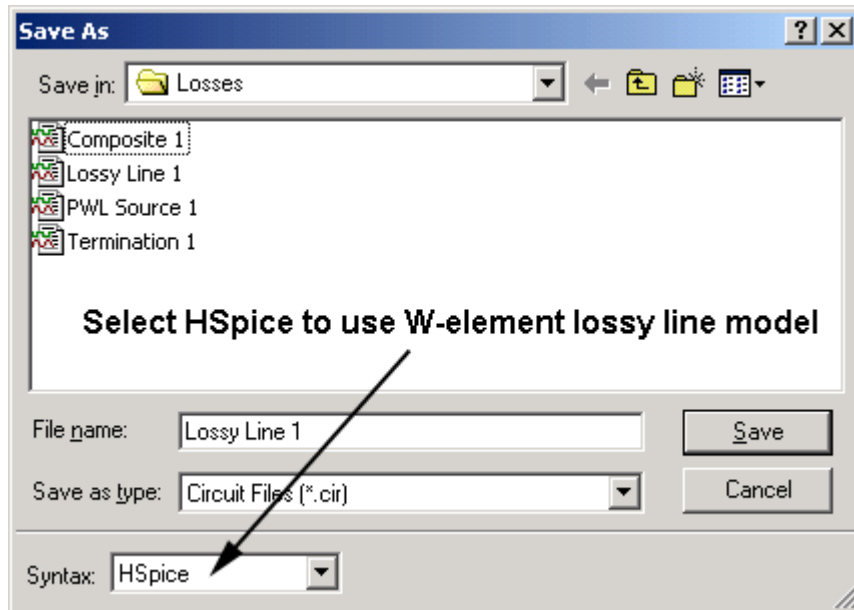


Figure 2-80: Choosing the simulator syntax when saving the model.

Right-click just outside the modeling window and click View netlist to observe the model that has been produced. You should see a standard HSpice W-element model, the accepted advanced model for lossy line simulation:

```
W1 N=1 port1 gnd_ port2 gnd_ RLGCMODEL=Lossy_Line_Hspice_Model l=1.00
```

```
.MODEL Lossy_Line_Hspice_Model W MODELTYPE=RLGC N=1 Lo=7e-008 Co=2.486e-011  
Ro=0 Go=0 Rs=5.95e-006 Gd=5.84e-012
```

If Hspice is not the simulator you are using with IConnect, you can resave the model and choose a non-HSpice syntax.

Viewing the Model

Once the model is saved, the easiest way to view the saved model is to right-click in the modeling window and select **View Netlist**.

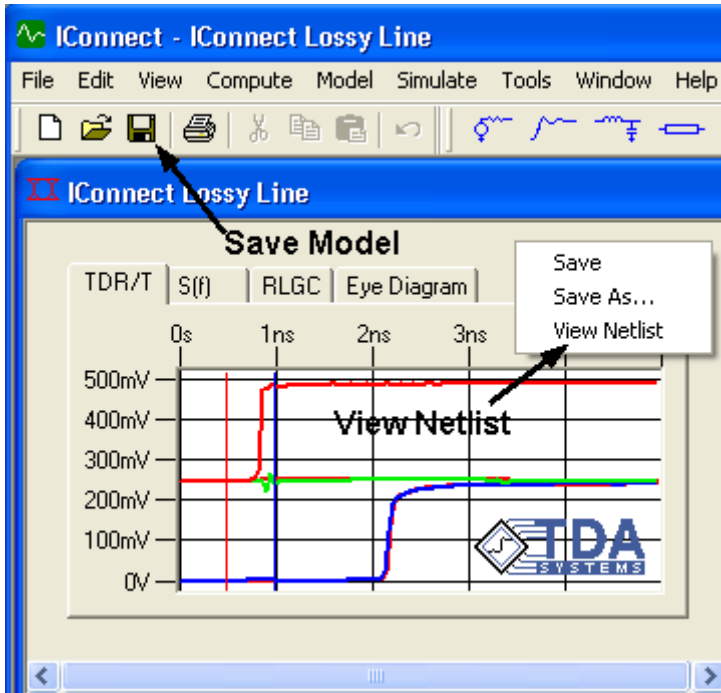


Figure 2-81: Once the model has been saved, you can view its netlist.

2.7.10 Simulating

Click on **Simulate** tab, and then click on **Simulate** to automatically generate the model and simulate it in your simulator of choice. You can choose a link to your preferred simulator in **Tools/Options/Simulator Options**. Make sure that your selected file format matches your simulator selection.

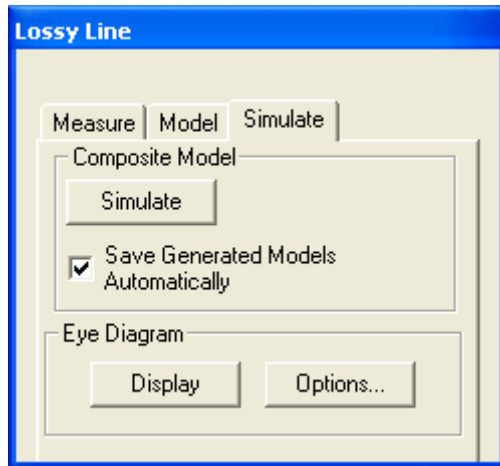


Figure 2-82: Clicking on the simulate button generates a composite model if necessary and launches the simulation.

Now the composite model, including the piece-wise linear TDR source, lossy line model, and termination are generated. You can double-click on any of the composite model elements to view the included models.

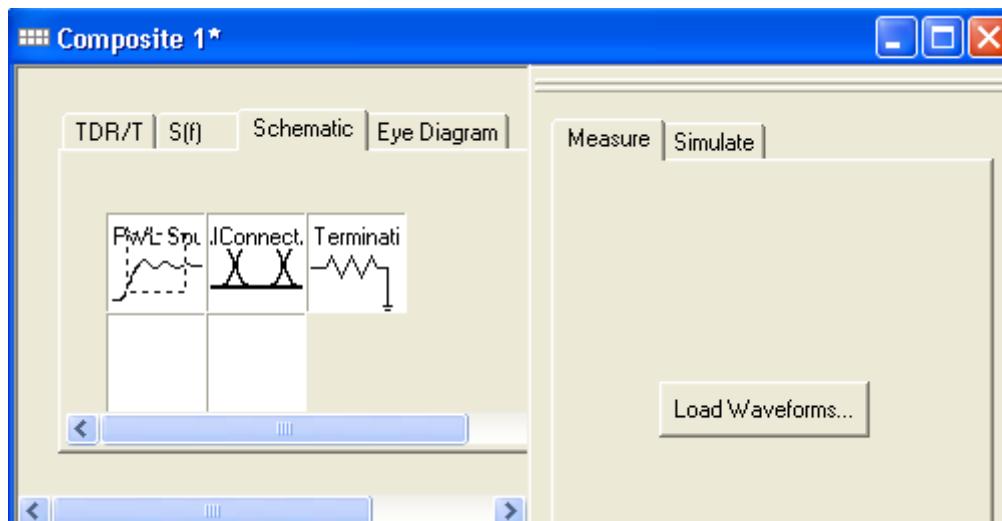


Figure 2-83: Composite model is generated automatically to validate the lossy line model.

2.7.11 PWL Source

If you believe that you need to increase the precision with which IConnect models the TDR source, decrease the **Max Error** settings and click **Partition** to redo the model. Make sure to save it before continuing. To reverse the polarity of the TDR source, simply change the sign in the **Scale** box under **Model**.

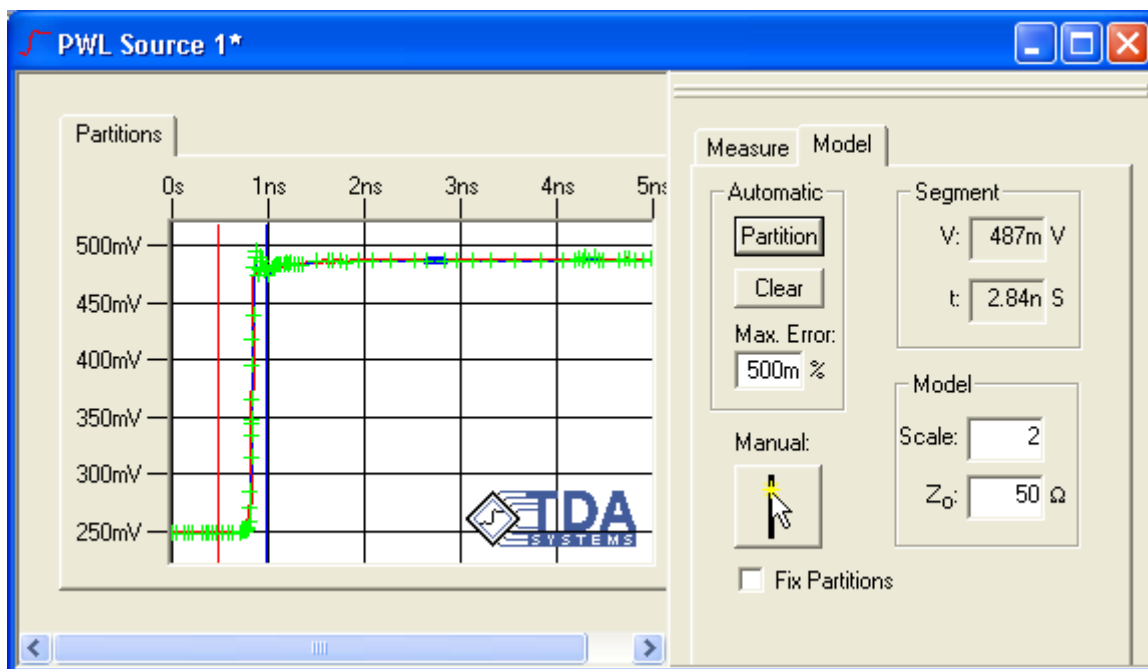


Figure 2-84: You can edit the automatically generated PWL Source if you wish.

2.7.12 Analyzing the Results

As you can observe, the correlation is perfect, except for the launch - the SMA connector in this case - which was not de-embedded during the measurement. We can easily model the connector, however, using the Z-line technique and Single-line modeling in IConnect.

After the simulation is complete, the simulated waveforms are placed in the composite model waveform viewer. Click on **TDR/T** or **S(f)** tabs to observe the correlation.

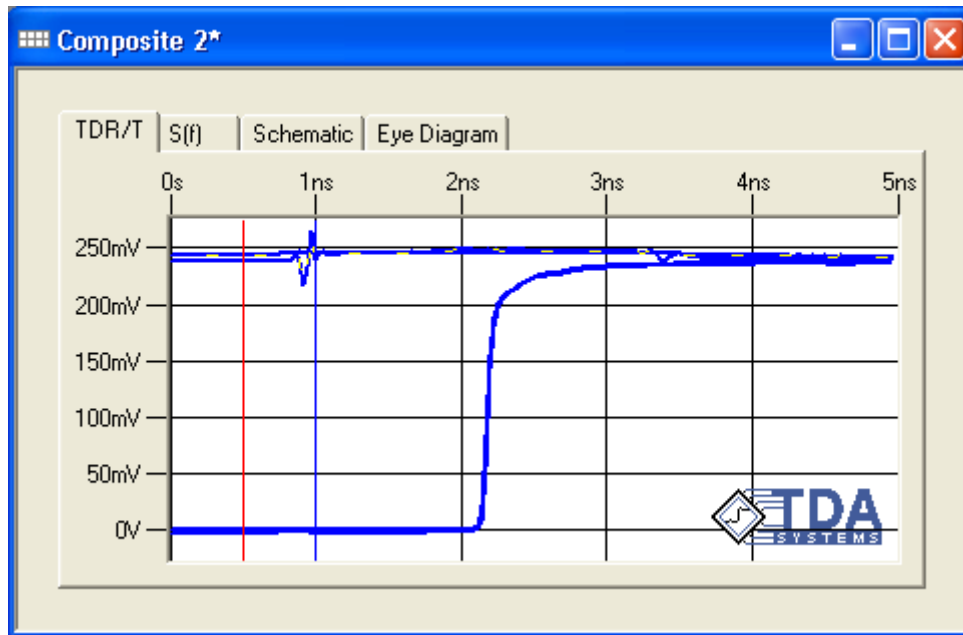


Figure 2-85: Near-perfect correlation between measurement and simulation for Lossy Line example.

The connector model can then be added to the composite model for the most accurate modeling of the transmission line losses. The transmission line length may need to be scaled back using **Scale Parameters By:** in the **Model** tab in the IConnect lossy line modeling window to accommodate the additional introduced length due to the connector.

A high-speed backplane or cable assembly connector can be modeled in the same fashion.

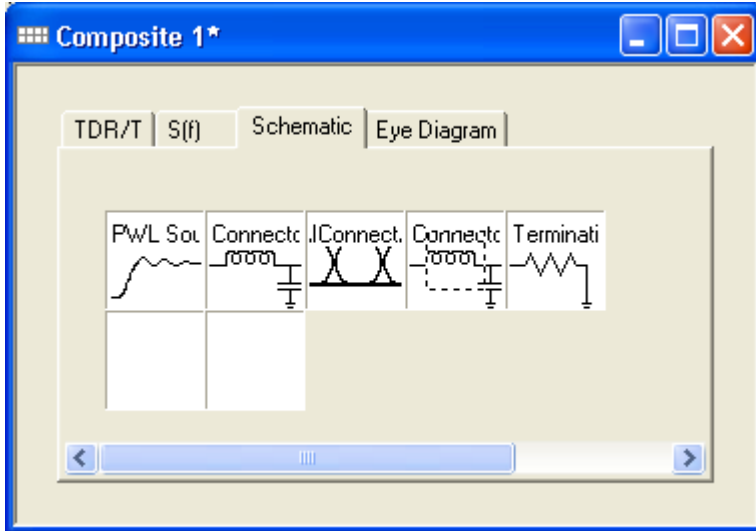


Figure 2-86: Composite model containing both the lossy line model and the end connector models.

2.7.13 Eye Diagram

Now a single click on the button generates the eye diagram. Note that if you select the eye diagram based on measurement, it should correlate well to the eye diagram based on the model, as well as to any eye diagram data measured with a pattern generator that you may have. The eye diagram based on measurement does not impose any restrictions on what kind of Device Under Test (DUT) this eye diagram is computed from.

You can essentially grab a huge communications board with multiple daughter cards, take a TDT measurement on it, and predict the eye diagram immediately! For more information on generating the eye diagram, see ["Eye Diagram Tutorial" on page 2-19](#).

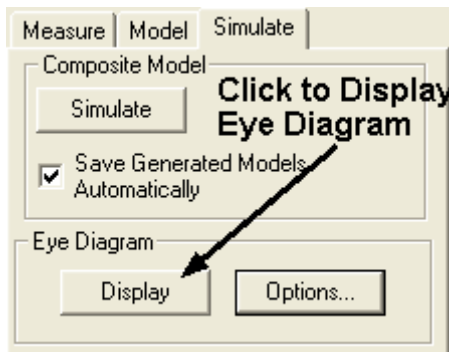


Figure 2-87: Clicking display generates and displays an eye diagram.

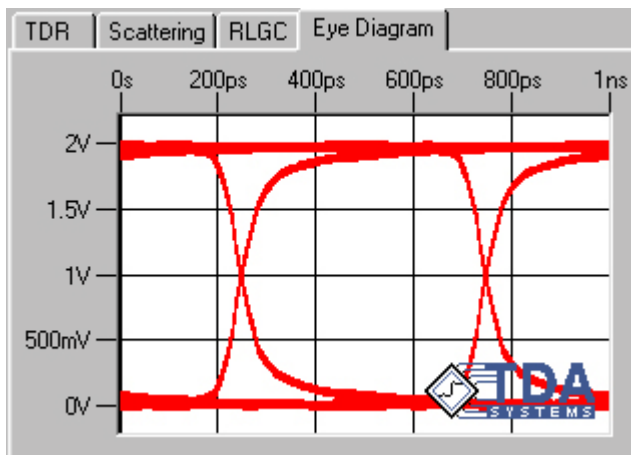


Figure 2-88: IConnect eye diagram display.

2.8 MeasureXtractor™ Modeling Tutorial

Welcome to the IConnect MeasureXtractor™ Modeling Tutorial. This tutorial guides you through the steps necessary to obtain an exact frequency-dependent model for your interconnect in IConnect MeasureXtractor™. MeasureXtractor™ data-driven models allow you to simulate any interconnect structure, and will include both time dependent (reflection, ringing, crosstalk) and frequency dependent (frequency loss) behavior of the interconnect. The extraction can be based on either time domain reflectometry and transmission (TDR/T) measurements or frequency dependent S-parameter data.

The data driven model, in conjunction with IConnect Link Simulator, allows you to include your TDR/T and S-parameter data directly into simulations, without creating a model first.

The MeasureXtractor™ modeling technique can be used for any interconnect structure. The size of the model is proportional to the interconnect size and complexity. The passivity, stability and causality of the models are guaranteed. This Tutorial leads you from a set of raw measurements to a complete SPICE model, and allows you to evaluate the model and verify its accuracy.

2.8.1 Waveform Acquisition and Setup

You should begin by reviewing “[Preparing to Take Measurements](#)” on page 2-4 if necessary. In addition to the best practices given in that section, it is particularly important when taking MeasureXtractor measurements to select a window that is long enough that it allows the DUT waveforms to completely settle to their steady state DC levels. We are performing high-frequency analysis of the DUT, and we need to allow the signal to settle to make this analysis accurately.

MeasureXtractor data driven models can be started either from the **Data Driven/MeasureXtractor Single Line** or **Data Driven/MeasureXtractor Coupled Line** menu items under the Model menu, or by using the S-parameter button on the Modeling Toolbar.

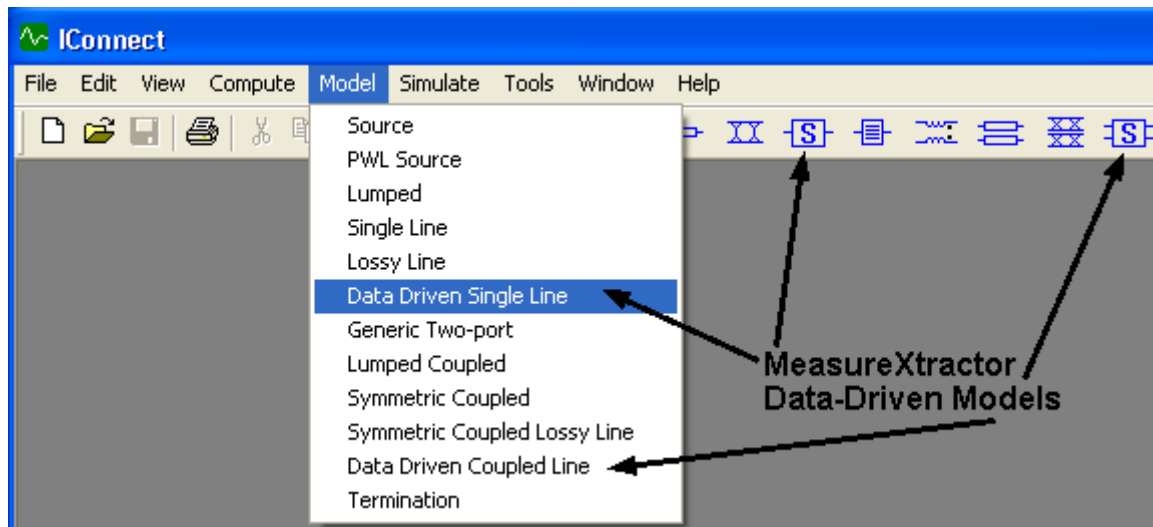


Figure 2-89: Creating a Data Driven model.

2.8.2 Choosing the Extraction Termination Type

You can choose to use Time or Frequency domain data for MeasureXtractor data driven modeling by clicking on the **Time** or **Frequency** radio button as shown in Figure 2-90. If you choose Time domain, you can acquire the data directly from a TDR oscilloscope or TDR-capable network analyzer. If you choose Frequency domain, you would need to acquire and store your data as frequency-dependent Touchstone file format beforehand, with port 1 (and 3) as inputs, and 2 (and 4) as outputs. For more information on Touchstone formats, refer to [“Importing and Exporting S-Parameter Data in Touchstone Format”](#) on page 7-21 of the user manual.

Set DC Values, **Truncate**, and **Recompute** buttons allow you to modify the time domain data computed from your S-parameters for more accurate extraction. The minimum (fastest) rise time for the computation is set by IConnect.

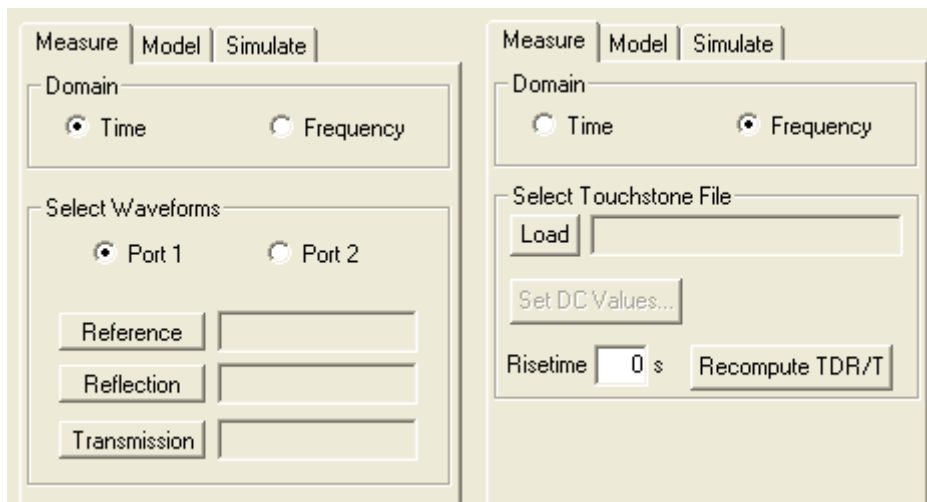


Figure 2-90: Choosing either Time or Frequency Domain for loading data.

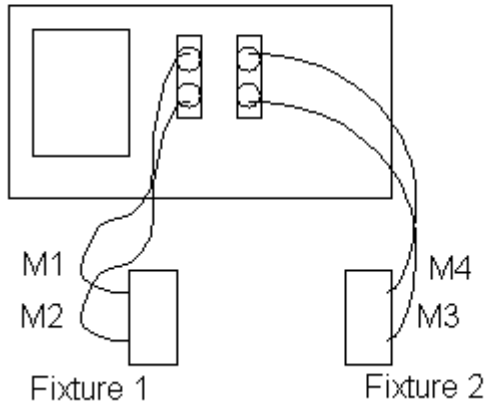
2.8.3 DUT Connections

All the waveforms must be acquired with 50Ω probes, fixtures, or cables connected to all the DUT ports, thus ensuring a 50Ω termination at every DUT port. The cables, probes and fixtures used to connect the DUT to both channels of the TDR oscilloscope are assumed by IConnect to be of the same length. IConnect also assumes that you have deskewed the scope channels. While the cables, probes and fixtures are connected to all DUT ports, we will only be acquiring one measurement at a time. Figure 2-91 shows the termination schemes used to make the measurements. However, the measurements themselves are single ended, although the figure refers to differential measurements. The reference waveforms must be acquired at each port, with the cables, probes, and fixtures disconnected from the DUT and left open-ended.

Reference measurement:

Differential Stimulus is ON on channels M1, M2.

Reflection is measured on the same channels M1-M2



Reflection and transmission measurements:

Differential Stimulus is ON on channels M1, M2.

Transmission is measured on channels M3-M4

Reflection is measured on the channels M1-M2

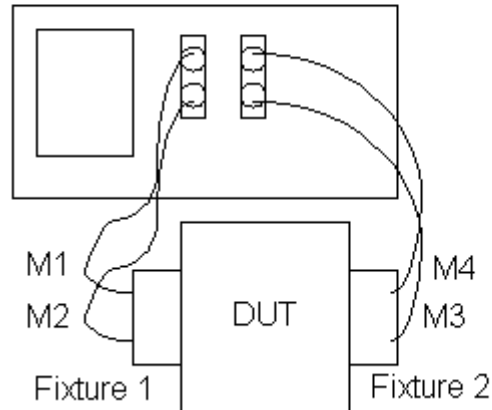


Figure 2-91: Reference measurements and DUT measurements. Note that although the figure refers to differential measurements, all MeasureXtractor measurements are single ended.

2.8.4 Loading Data

Now we can acquire the required waveforms for MeasureXtractor data driven modeling. For each port, we need to acquire a reference waveform, a reflection waveform, and transmission waveforms from this port to all other ports.

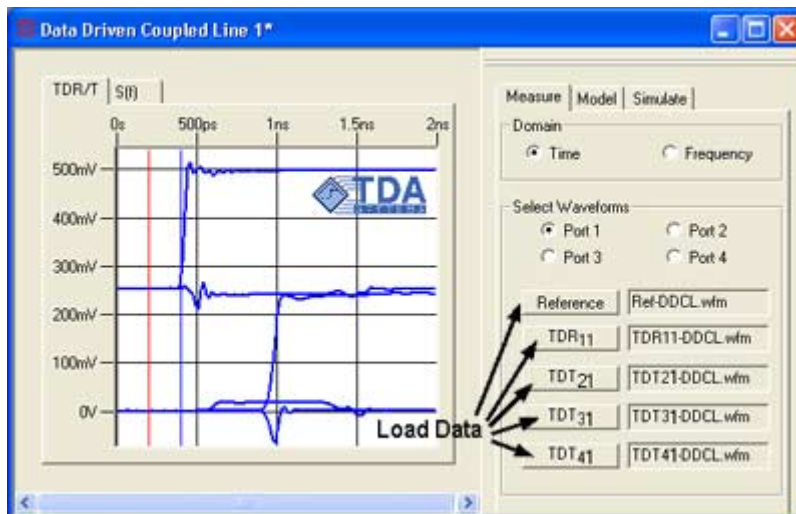


Figure 2-92: Loading waveform data into the Data Driven/MeasureXtractor Coupled Line model.

First, make sure to select the appropriate instrument in the Acquisition window. Then, sequentially click on the **Select Waveform** radio buttons, and then click on the button for each waveform in the **Measure** tab to load the data. Since for passive structures such as interconnects the transmission from port X to port Y equals reverse transmission from port Y to port X, these measurements will only be acquired once. For example, transmission from port 1 to port 2 (TDT_{21}) is required, but transmission from port 2 to port 1 (TDT_{12}) is not. All the waveforms must be acquired with the 50Ω cables connected to both DUT ports, thus ensuring a 50Ω termination at the far end of the DUT. The cables, probes and fixtures used to connect the DUT to both channels of the TDR oscilloscope are assumed by the IConnect to be of the same length.

A total of 5 waveforms is required for the Data Driven/MeasureXtractor Single Line models, and 14 waveforms for the Data Driven/MeasureXtractor Coupled Line models. Read the waveform descriptions, shown in Figure 2-93, in the scope interface window to capture a proper waveform from the instrument. The tutorial waveforms can be found in **Tutorials** directory under the IConnect installation directory.

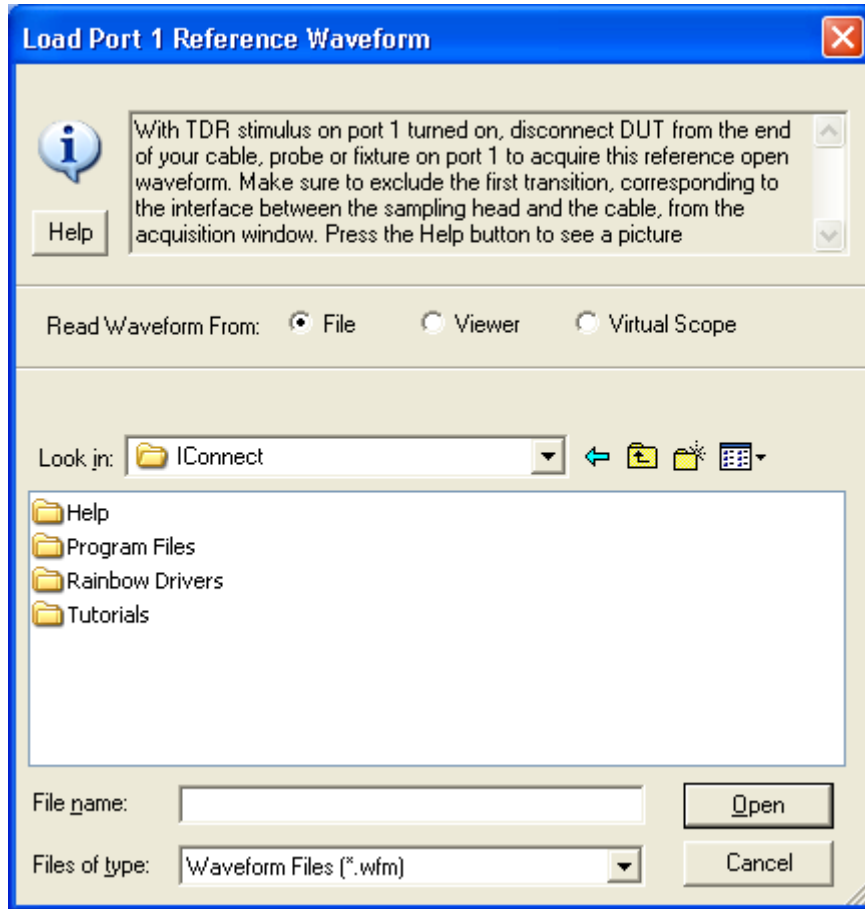


Figure 2-93: Reading the tutorial waveforms into IConnect.

2.8.5 Model Extraction

Once the data is loaded, click on **Model** tab. Press the **Extract** button; the model is extracted as shown in Figure 2-94. After the model is extracted, the information about the number of poles in the model and maximum error is displayed.

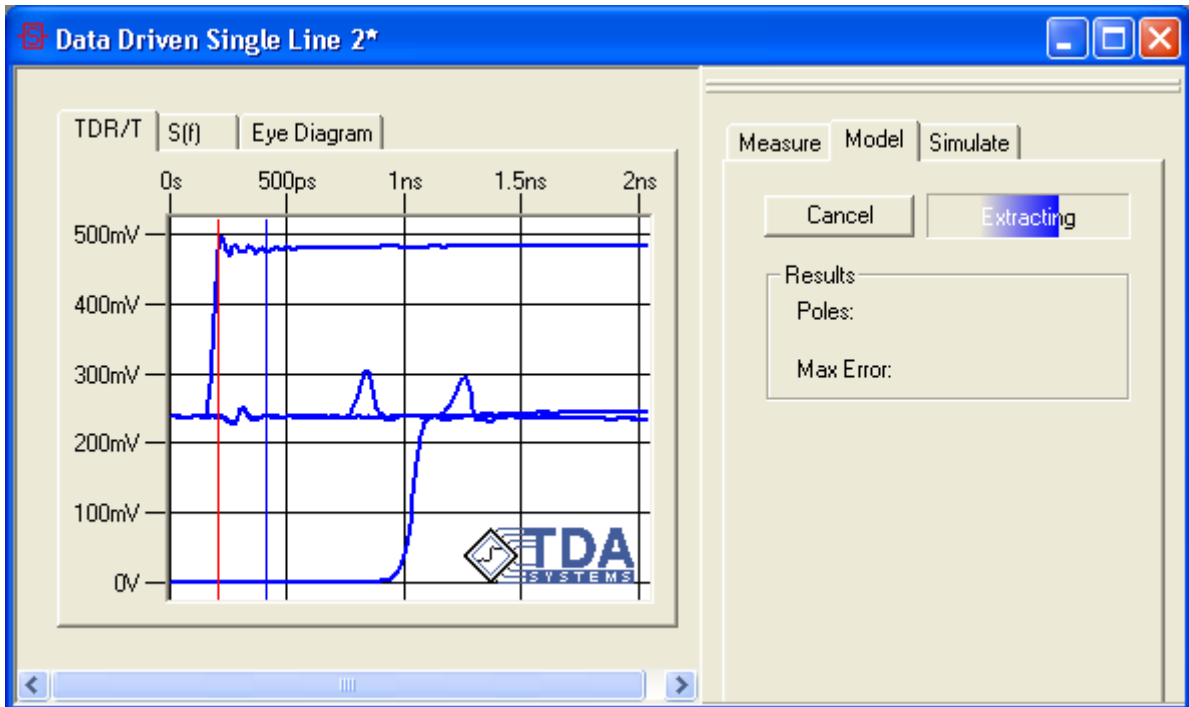


Figure 2-94: MeasureXtractor in the process of extracting a two-port model.

2.8.6 Model Accuracy

The correlation between the simulated (red) and measured (blue) data in the time domain and frequency domain is exact. Both time domain amplitude and delay, and frequency domain magnitude and phase match exactly, as shown in Figure 2-95.

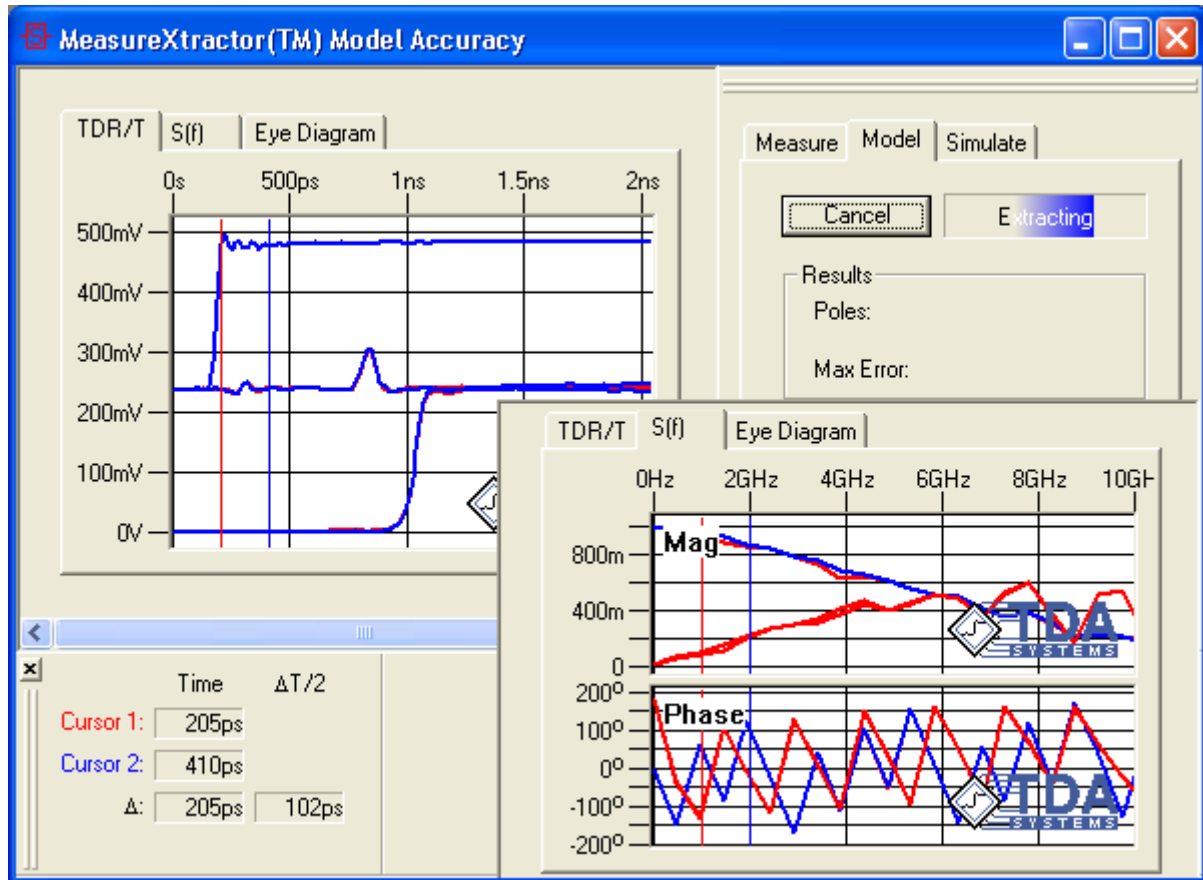


Figure 2-95: Resulting measurement to model correlation.

2.8.7 Saving the Model

Save the model to a meaningful name, such as Data Driven.cir, in the Data Driven directory. This may be done via the **File/Save** command, from the standard toolbar, by using the <Ctrl+S> keyboard shortcut, or by right clicking in the model window and choosing **Save** from the resultant context menu. Make sure to select the appropriate SPICE output format while saving the model, as shown in Figure 2-96.

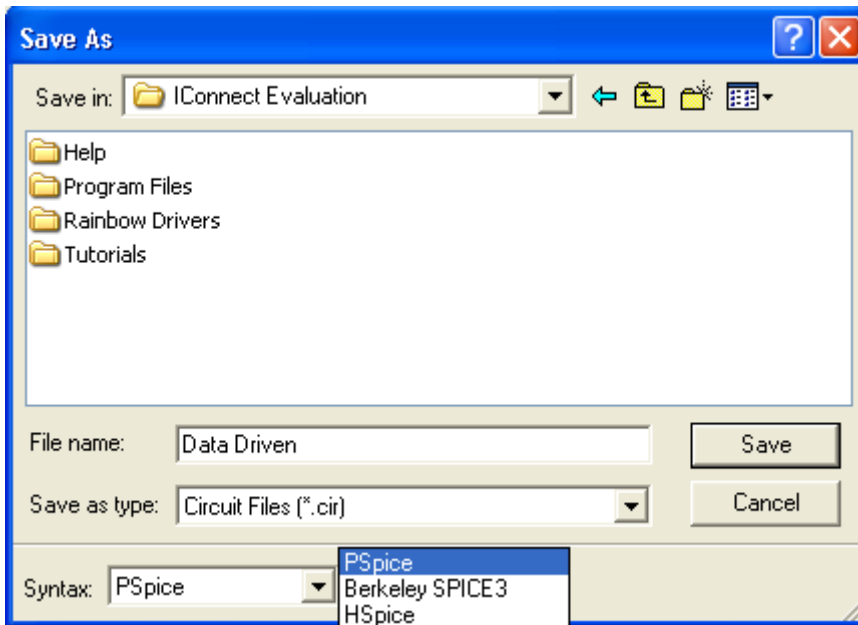


Figure 2-96: Saving the model.

Viewing the Model

Once the model is saved, the easiest way to view it is to right click in the modeling window, and click View Netlist, as shown in Figure 2-97.

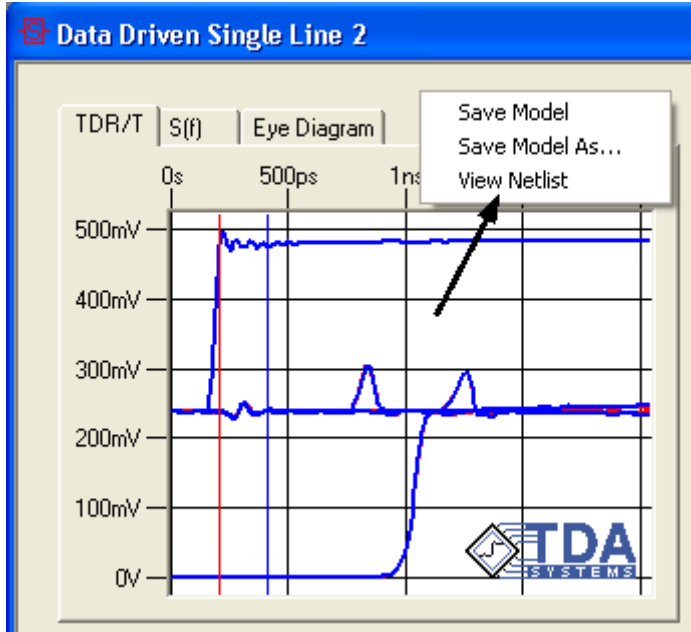


Figure 2-97: You can optionally view the SPICE Netlist.

2.8.8 Simulating

Click on the **Simulate** tab, then press the **Simulate** button to automatically generate the model and simulate it in your simulator of choice, as shown in Figure 2-98. You can choose a link to your

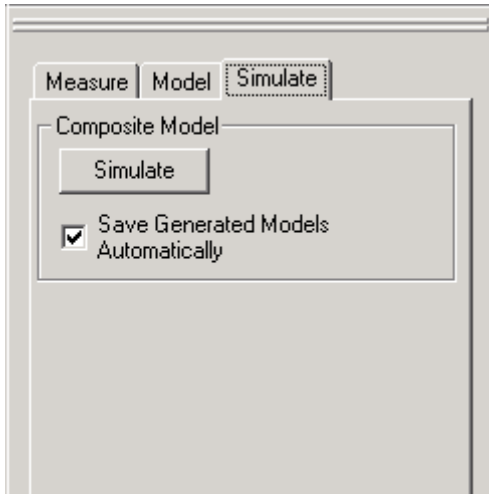


Figure 2-98: Data-Driven Simulate tab.

preferred simulator by choosing the **Tools/Options** menu and clicking the **Simulation** tab. Make sure that your selected file format matches your simulator selection.

Now the two Composite models, including the piecewise linear TDR source, data driven model, and termination are generated, as shown in Figure 2-99. You can double-click on any of the com-

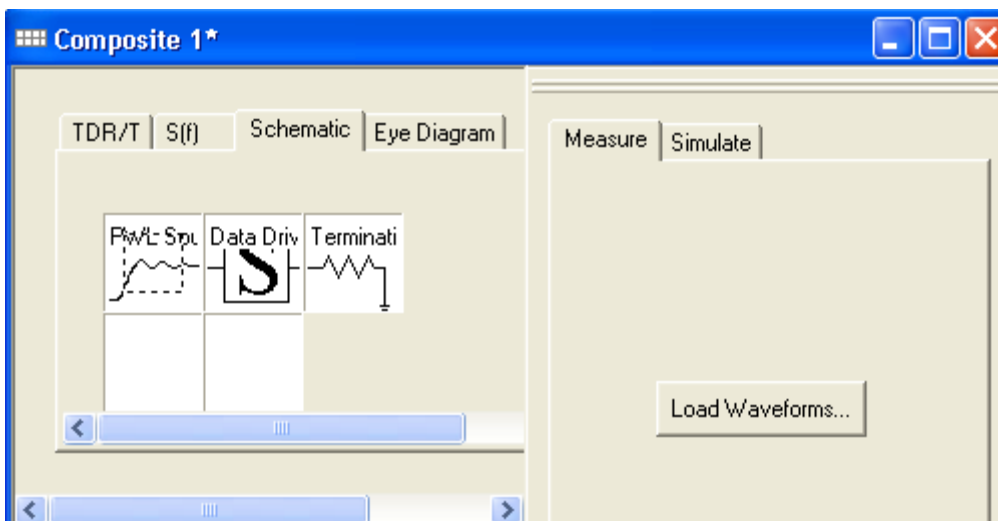


Figure 2-99: Automatically-generated Composite model.

posite model elements to view the included models. The two composite models correspond to the interconnect driven from both ports.

Getting Started

Note: For Data Driven/MeasureXtractor Coupled Line, there will be 4 composite models generated.

2.8.9 PWL TDR Source

If you believe that you need to increase the precision with which IConnect models the TDR source, decrease the Max Error settings and click Partition to repartition the PWL Source model, as shown in Figure 2-100. Make sure to save it before continuing. To reverse the polarity of the TDR source, simply change the sign in the **Scale** box, in the **Model** group box.

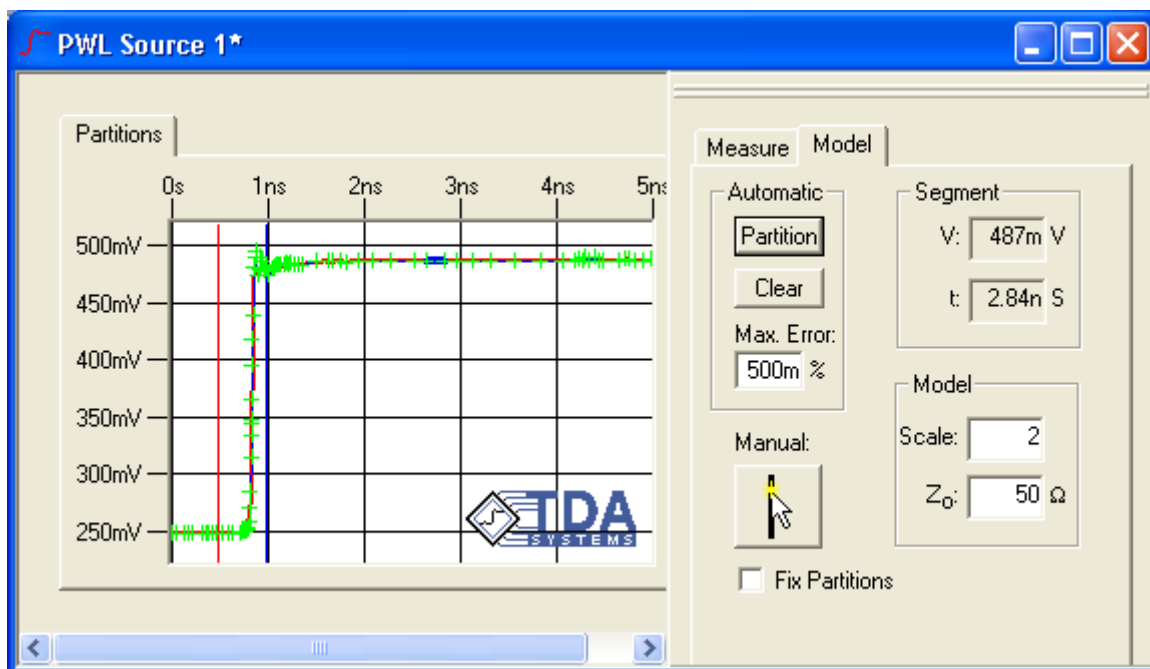


Figure 2-100: Automatically-generated PWL Source model.

2.8.10 Analyzing the Results

After the simulation is complete, the simulated waveforms are placed in the Composite model waveform viewer. Click on the **TDR/T** or **S(f)** tabs to observe the correlation, as shown in Figure 2-101. The correlation is exact in both time and frequency domain.

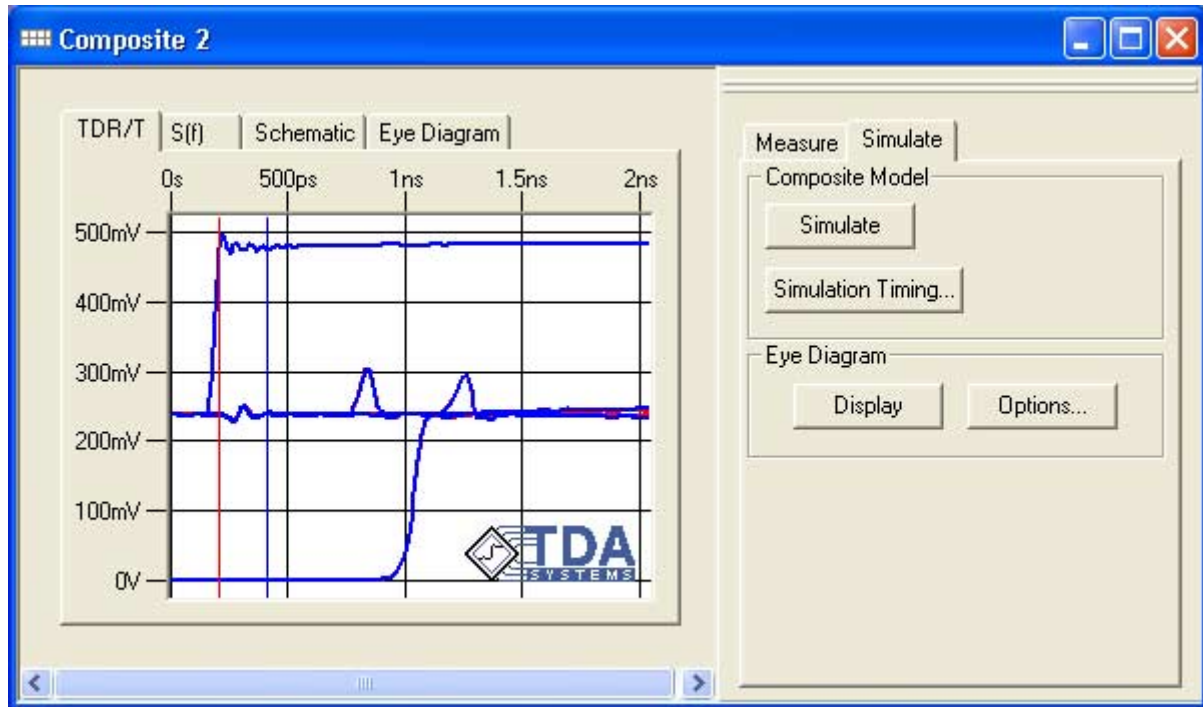


Figure 2-101: Nearly exact correlation between the measured data and the data-driven model extracted by MeasureXtractor.

When importing Touchstone files, please note that IConnect assumes ports 1 and 3 to be the input ports, and ports 2 and 4 to be the output ports. The model will still be extracted and valid if a differ-



Figure 2-102: Touchstone input and output ports.

ent port numbering system was used in the imported Touchstone file, but the validation simulation results may be incorrect.



Chapter 3 IConnect Measure-Model-Simulate Methodology

IConnect[®] and MeasureXtractor[™] TDR and VNA software simplifies the process of creating accurate, validated models from measurement data. It is built around the Measure-Model-Simulate methodology, in which you begin with a set of measurements, create a model from those measurements, and run simulations to verify that your model accurately predicts the measured behavior. IConnect makes it simple to do all three of these steps without ever leaving the application, and in many cases all three can be accomplished within a single window!

Contents

3.1	The Value of Measure-Model-Simulate	3-2
3.2	Measure-Model-Simulate within IConnect.....	3-3

3.1 The Value of Measure-Model-Simulate

Models you can trust

The primary benefit of traversing the Measure-Model-Simulate loop is that you emerge with a model that you can trust because it has been correlated against measurement. ICConnect dramatically simplifies the validation process with its integrated link to multiple simulators. Many times engineers are asked to rely on models with a dubious validation history. With ICConnect, since you easily do the validation yourself, you are willing to rely on the extracted models.

Measurement interpretation

The measurement process allows insight into your design that would be difficult to achieve otherwise. The many capabilities of ICConnect simplify the task of interpreting your measurements. Frequent use of TDR measurements provides an intuitive understanding of your digital interconnection system. Furthermore, the dual time- and frequency-domain displays help you become conversant in both domains, and allow you to use your TDR as a network analyzer. And conversion from frequency-domain measurements into their TDR equivalents allows you to use a network analyzer as a TDR.

Models for existing devices

You can quickly create models for existing and even legacy devices for which you do not have geometry or materials information. And because your models are validated against measurement, they may be more accurate than those available from the part vendor.

3.2 Measure-Model-Simulate within IConnect

IConnect has an improved user interface to make it even more convenient to traverse the measure-model-simulate loop. Many of the IConnect models now have Measure, Model, and Simulate tabs right in the model extraction window itself.

The **Measure** tab helps you to bring in the necessary waveforms to extract the model by showing you exactly which waveforms need to be loaded to extract the model, and performing sanity checks to help weed out measurement errors. The waveforms are displayed in the TDR/T and S(f) waveform viewing tabs, which are integrated time-domain and frequency-domain waveform viewers with all the waveform management capabilities to which IConnect users have become accustomed.

The **Model** tab provides you with the ability to extract the parameters which are relevant to the particular model. Because each model has different parameters and extraction methods, each model tab will typically be different from the others. In many cases the model tab will be associated with a model viewing tab on the left that graphically represents the current parameter values, and in many cases allows you to graphically make changes to the model parameters. Some models may be optimized as well.

The **Simulate** tab gives you one-click capability to validate the model you have just extracted. It generates the necessary SPICE models, allows you to save them in the directory of your choosing, and launches the simulation for you, returning the results to the model's **TDR/T** viewing tab for validation. This is accomplished without your having to edit a SPICE netlist or open a single file. The simulate tab also generates and displays eye diagrams for lossy models.

For more information

More information on how to make the most out of IConnect's integrated support for the Measure-Model-Simulate methodology is available in various chapters in this manual. A number of tutorials which provide real-life application examples are available in ["Getting Started" chapter on page 2-1](#). The ["Models" chapter on page 5-1](#) describes how the Measure, Model, and Simulate tabs of each of the models works. The ["Measurement" chapter on page 4-1](#) and ["Preparing to Take Measurements" on page 2-4](#) tell you how to set up your measurement instrument to work transparently with IConnect, while the ["Simulation" chapter on page 6-1](#) tells how to integrate your simulator into IConnect to be used in model validation.



Chapter 4 Measurement

IConnect[®] and MeasureXtractor[™] TDR and VNA software is an integrated waveform acquisition engine for your TDR measurement instrument. This chapter will help you interface your instrument with IConnect, and will get you started acquiring data. It also provides information to help you take accurate measurements.

Contents

4.1	Measurement Overview	4-2
4.2	GPIB Installation	4-3
4.3	Running IConnect Locally on the Tektronix TDS/CSA8XXX.....	4-4
4.4	Oscilloscope Setup.....	4-5
4.5	Configuring IConnect to Communicate with your Instrument	4-6
4.6	Waveform Acquisition	4-9
4.7	Saving and Restoring the Instrument Setup	4-11
4.8	Measurement Tips	4-13

4.1 Measurement Overview

Measurement is performed as a cooperative effort between the IConnect software, the GPIB card, and your TDR measurement instrument. Before measurements are made from within the IConnect framework, it is necessary to properly set up your instrument, as well as to install the GPIB interface. For questions regarding how to take accurate and effective measurements, you may refer to [“Measurement Tips” on page 4-13](#), or to the references on measurement and modeling techniques given in [“References” on page A-5](#), or you may contact TDA Systems.

If you are simply evaluating the IConnect software, the measurement instrument and GPIB interface may be unnecessary, since waveforms are available on disk to be used for evaluation purposes.

4.2 GPIB Installation

The GPIB card and driver installation should be performed according to the manufacturer's instructions. IConnect is written and tested using GPIB hardware and software from [National Instruments](#). After installing the hardware and software, be certain to run the manufacturer-supplied diagnostics to ensure correct operation.

When configuring the board, note its board index if it is anything other than GPIB0, as it will be necessary to specifically configure IConnect if you intend to access your instrument through any board other than GPIB0. This is only an issue for computers with multiple GPIB cards installed.

If you encounter problems, consult "[Troubleshooting](#)" on [page A-1](#) for more advice on setting up your GPIB card.

While TDA Systems is happy to assist you with GPIB-related questions, the card manufacturer may be of more assistance.

4.3 Running IConnect Locally on the Tektronix TDS/CSA8XXX

Because the Tektronix TDS/CSA8000 and TDS/CSA8200 run on an open, Microsoft Windows-based platform, you have the ability to install IConnect directly on the oscilloscope. This approach makes it unnecessary to maintain a computer in addition to the TDS/CSA8XXX, and further makes it unnecessary to purchase and install a GPIB card.

In order for IConnect to communicate correctly with the TDS/CSA8XXX on which it is installed, the TekVisa software from Tektronix must be present. If you purchased your TDS8000 prior to 2002 and have not upgraded your software since then, you may need to upgrade your scope software. If you suspect TekVisa is not present and find that IConnect cannot acquire waveforms when running locally on your scope, you should upgrade your scope software. The upgrade is available from the [Tektronix web site](#).

Once you have installed IConnect, and installed TekVisa if necessary, you are ready to make measurements. Simply select the **Tektronix TDS/CSA8XXX Local** option from the **Instrument** dropdown box in the **Acquisition** window. As with the other instruments, you must still select appropriate parameters such as GPIB address and timeout values.

Note: If you wish to run IConnect on a separate computer connected to a TDS/CSA8XXX, you may still do this, as before, using a GPIB controller card from National Instruments connected to the TDS/CSA8XXX GPIB port. This requires that you choose the **Tektronix TDS/CSA8XXX** option from the **Instrument** dropdown box in the **Acquisition** window.

In order to use IConnect effectively and efficiently, TDA Systems recommends that if you are planning to operate IConnect locally on the TDS/CSA8XXX, that you use a second monitor in addition to the LCD display provided on the instrument front panel. This allows you to view the TDS/CSA8XXX system software on the local instrument display while running IConnect and other applications on the secondary, larger monitor. Your TDS/CSA8XXX manual will guide you through the process.

4.4 Oscilloscope Setup

In order to benefit from IConnect's efficient, integrated measurement interface, it is necessary to correlate the software settings with the physical settings of the oscilloscope. To change these settings properly, it may be necessary to consult your oscilloscope manuals.

Online

First, it is necessary to make sure that your oscilloscope is online. Most scopes have the ability to be placed into local mode, in which the scope ignores commands transmitted over the GPIB bus. However, this precludes use by an external program such as IConnect.

Talk/Listen

It is also necessary to have your oscilloscope in talk/listen mode so that communication between the scope and the GPIB controller may flow in both directions. Some oscilloscopes are always in talk/listen mode, so that once the scope is online, this setting need not be made.

GPIB Address

Finally, the GPIB address of your scope must correspond to the setting in IConnect. Any address between 0 and 31 is valid, although 0 is discouraged since it will typically conflict with the address of the GPIB controller. IConnect comes with default values of 1 for the Tektronix instruments and 7 for the Agilent instruments. However, these values may be changed within IConnect to correlate with your particular measurement setup.

Other Instrument-specific GPIB parameters

No other GPIB parameters need be adjusted on your scope. They are either set via software from IConnect, or they are taken into account such that their values are not relevant.

Agilent 8510 time domain setup

If you are using an Agilent 8510, it must be operating in time-domain display mode, using a step stimulus, in addition to setting the GPIB address. Your 8510 user manual can guide you through these processes.

4.5 Configuring IConnect to Communicate with your Instrument

All necessary configuration of IConnect for communication with your measurement instrument is performed via the **Instrument Settings** page of the Acquisition Window, which is accessed by pressing the **Instrument Settings** tab as shown in Figure 4-1. The Acquisition Window is made visible using the **View/Acquisition Window** menu command or the waveform button on the auxiliary toolbar. The settings you make will depend on what type of instrument you are using.

Oscilloscope

If you are using an Agilent (HP) or Tektronix oscilloscope, you may need to configure its GPIB address into IConnect. The default settings of 1 for the Tektronix scope and 7 for the Agilent scope typically will match the setting from the factory.

You may also need to configure IConnect to match the GPIB address of your GPIB card. The default value of 0 is standard for a typical one-card installation.

You may set the timeout value, which determines how long the GPIB card will wait for your instrument to respond before flagging an error. The default value of one second will be sufficient for a typical installation.

Note: Some instruments, the Agilent (HP) 54120 in particular, require a longer timeout setting if heavy averaging is performed. Longer averaging requires a longer timeout value.

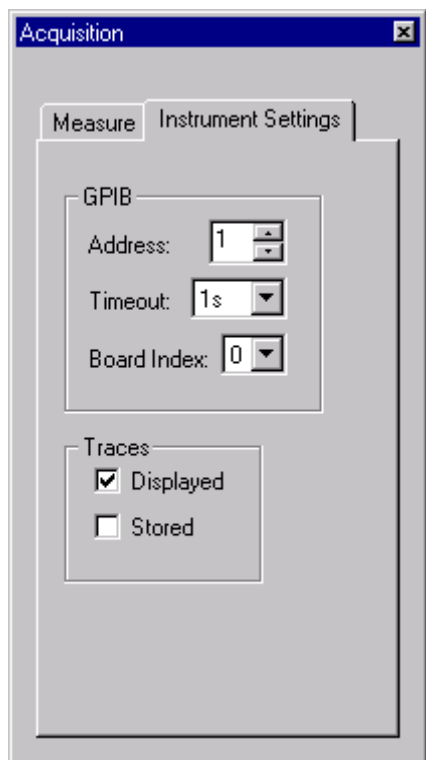


Figure 4-1: Setting the GPIB communication parameters.

Virtual Scope

If you are using your disk drive as a virtual scope, the necessary configuration consists of choosing a directory from which to acquire waveforms, and choosing an appropriate filename filter so that only files with certain names are displayed for acquisition, as shown in Figure 4-2. Typically the filter is set to *.wfm, although depending on your personal file naming preferences, it may be set to anything which the operating system recognizes as a file filter. Filters often include the wild-card characters ?, which matches a single character, and *, which matches 0 or more characters.

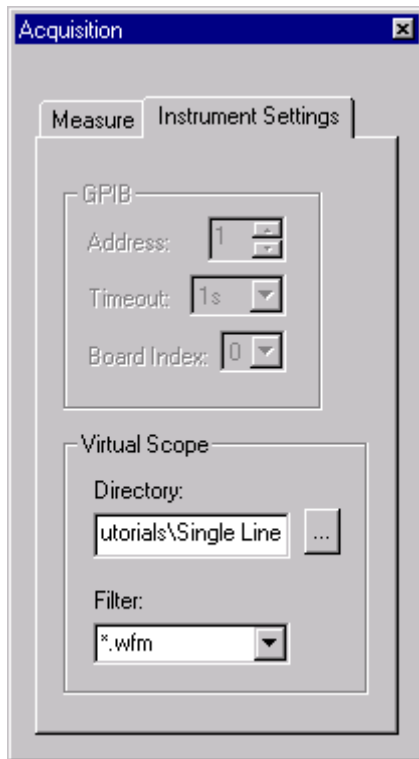


Figure 4-2: Setting the virtual scope options.

For example, suppose you have a directory containing the following files:

Tek Demo Board Reflection.wfm
Tek Demo Board Transmission.wfm
HP Demo Board1 Reflection.wfm
HP Demo Board1 Transmission.wfm
HP Demo Board2 Reflection.wfm
HP Demo Board2 Transmission.wfm
HP Demo Board3 Reflection.wfm
HP Demo Board3 Transmission.wfm
Demo Board Comparision.doc

The following table shows which file names will be displayed in response to various filters:

Measurement

Filter	Displayed Waveforms
*.wfm	Tek Demo Board Reflection.wfm Tek Demo Board Transmission.wfm HP Demo Board1 Reflection.wfm HP Demo Board1 Transmission.wfm HP Demo Board2 Reflection.wfm HP Demo Board2 Transmission.wfm HP Demo Board3 Reflection.wfm HP Demo Board3 Transmission.wfm
Tek*.wfm	Tek Demo Board Reflection.wfm Tek Demo Board Transmission.wfm
HP*Reflection.wfm	HP Demo Board1 Reflection.wfm HP Demo Board2 Reflection.wfm HP Demo Board3 Reflection.wfm
HP*2*.wfm	HP Demo Board2 Reflection.wfm HP Demo Board2 Transmission.wfm
*DemoBoard? *.wfm	HP Demo Board1 Reflection.wfm HP Demo Board1 Transmission.wfm HP Demo Board2 Reflection.wfm HP Demo Board2 Transmission.wfm HP Demo Board3 Reflection.wfm HP Demo Board3 Transmission.wfm

4.6 Waveform Acquisition

Having installed the GPIB interface and set up your oscilloscope, waveform acquisition from within IConnect is a simple matter. If none of the peripheral hardware has been installed, waveforms may be acquired from disk for evaluative purposes by using the Virtual Scope.

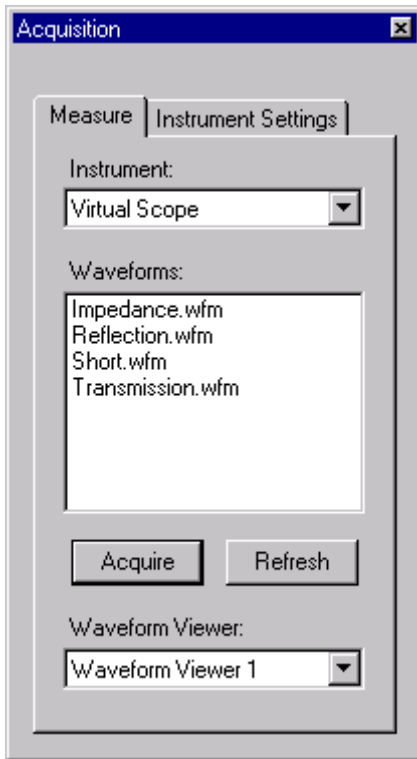


Figure 4-3: Waveform Acquisition window.

Acquisition Window

All acquisition takes place from the waveform acquisition window, which is a dockable window associated with the main IConnect window. If the window is not visible, you may toggle its visibility using the **View/Acquisition Window** menu command. For more convenient access, you may use the **View/Toolbars/Auxiliary Toolbars** menu command to enable visibility of the auxiliary toolbar, at which point the visibility of the acquisition window may be toggled using its toolbar button. If you have recently changed instrument settings, you may need to press the **Measure** tab of the acquisition window in order to take measurements.

Instrument Selection

The appropriate measurement instrument may be selected using the drop-down list box at the top of the acquisition window. As soon as the instrument is selected, IConnect attempts to fill the waveform selection list box with the available waveforms. If no waveforms appear, or an error message appears, it may be necessary to change the instrument options using the **Setup** tab of the acquisition window, as shown in Figure 4-4. These options may include general GPIB options such as GPIB address, as well as instrument-specific options that are only applicable in the context of a particular instrument.

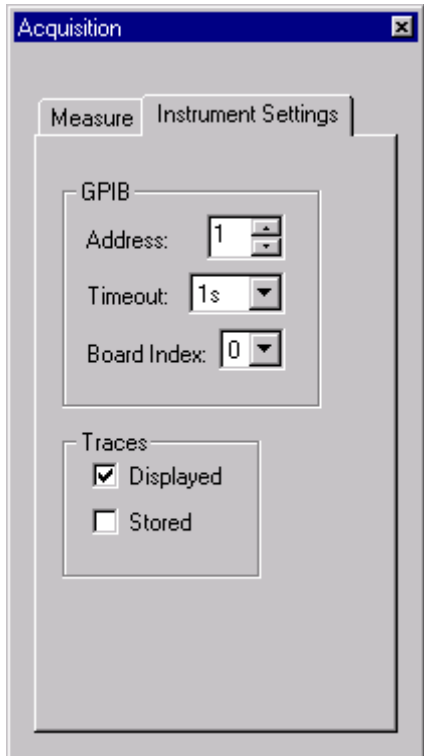


Figure 4-4: Instrument Settings page of the Acquisition Window.

Waveform Viewer Selection

You may choose which waveform viewer you would like to receive the newly acquired waveform or waveforms. This selection is made using the waveform viewer selection drop-down list box near the bottom of the acquisition window. The list box contains the names of all currently open waveform viewers, as well as the **New Waveform Viewer** option. If you choose the name of an existing waveform viewer, the acquired waveform or waveforms will be placed in that viewer. If you choose **New Waveform Viewer**, a new viewer will be created to receive the waveforms.

Waveform Transfer

Once the appropriate instrument is selected and the associated options are chosen from within IConnect, the waveforms available for transfer are automatically listed in the waveform selection list box. Double click on the waveform name to transfer a waveform into the selected waveform viewer, or select the waveform and press the **Acquire** button. The list box supports multiple selection, so that multiple waveforms may be selected and then all are acquired when the **Acquire** button is pressed.

Note: If changes are made in the instrument that change which waveforms are available from the instrument for acquisition, it will be necessary to press the **Refresh** button to update the waveform selection list box.

4.7 Saving and Restoring the Instrument Setup

Using IConnect, you may easily save the entire state of your measurement instrument to a file, and restore it at a later time. This capability is very convenient for users in a situation in which many people are using the same instrument, and adjusting the settings to meet their individual needs.

Saving Instrument Settings

To save the instrument settings, choose **Save Instrument Settings** from the **File** menu. A window will come up, asking you for a file name to which the settings should be saved.

Note: If you use more than one TDR instrument, you should include a reference to the name of the instrument in the filename that you choose. This will allow you, when the time comes, to load the correct settings back into the correct instrument. IConnect cannot determine from the contents of the settings file which type of instrument it came from.

Once you have chosen a filename, press the **OK** button, and the settings will be saved to a file. The location of the file will also be remembered by IConnect for easy retrieval at a later time.

Restoring Instrument Settings

Instrument settings which have been saved to a file may be loaded in at a later time to restore the instrument to its previous state. In order to restore the settings from a file, choose **Load Instrument Settings** from the **File** menu. A window will come up which contains the names of instrument settings files you have saved in the past. When you select a file and press **OK**, or simply double-click on the filename, IConnect will attempt to load the instrument settings from the file.

Note: Attempting to load a settings file saved by one type of instrument into another type of instrument will not work, and may cause unpredictable results on that instrument.

If you wish to see the settings files without their path names prepended, check the box labeled **Hide Paths**.

If the settings file you would like to load is not listed in the window (for example, if someone provided you with a settings file from another instrument of the same type), click on the **Browse...** button, and a standard window will allow you to find the file on the disk. Once you have found it and clicked **OK**, the settings will be loaded into the instrument immediately. The name of the file will also be remembered for later use.

Note: Because the horizontal position of the TDR step and response vary due to temperature and other factors, restoring the scope state may not place the waveform at the same point on the screen as before. This can be solved by adjusting the horizontal delay.

Loading a File Containing General Instrument Commands

Although the save and restore capability of IConnect is designed primarily to be used to save and restore the entire state of the instrument, it may also be used to send any set of commands to the instrument. To accomplish this, place the commands you would like to send to the instrument in a text file. Then choose **Load Settings...** from the **File** menu. Click the **Browse...** button, and find the file in the resulting window. Select it and press **OK**, and the commands will be sent to the instrument. If the instrument does not respond as you expected, ensure that there is a valid GPIB

Measurement

connection by reading a waveform into IConnect. If the connection exists, you should verify that the command syntax is correct.

4.8 Measurement Tips

Taking accurate measurements is unquestionably the most important element in measurement-based modeling. In the absence of accurate measurements, skilled modeling and thorough verification are a waste of time. The following guidelines are helpful to remember.

Warm up the instrument

A Time Domain Reflectometry (TDR) oscilloscope is a high performance, high accuracy instrument. To fully utilize the accuracy and precision of the instrument, you should allow the instrument 20-30 minutes before performing any measurements, to allow its temperature to stabilize. Your oscilloscope manual can give more exact specifications on warm-up times for your particular instrument. Similarly, to prevent any waveform drift from affecting the measurements, it is best to keep your lab temperature stable during your measurement sessions.

Make a clean connection to the Device Under Test (DUT)

A clean interface to the DUT is critical to achieve high quality measurements. High-quality microwave grade cables and probes will allow you to deliver full incident rise time to the DUT, allowing you to achieve higher resolution. In fixturing applications, a fixture with a low-inductance, low-resistance ground return path will provide the cleanest measurement interface. Since the minimum discontinuity that can be resolved by a TDR step must be separated from the discontinuity next to it by about 1/2 the observed rise time, delivering a faster rise time to the DUT allows you to resolve smaller DUT sizes. TDR oscilloscopes in the industry are typically instruments with bandwidths of 20 GHz and beyond (risetimes of 45 ps and below), and great care must be taken in measurement applications requiring the full bandwidth. For more information on probing, fixturing and cabling, download our **Probing Quick Guide** at the [TDA Systems web site](#).

Window out the initial transition

In order to take useful TDR measurements for model extraction in IConnect, it is necessary to window out what is called the initial transition. What this really means is setting the timebase and delay on your oscilloscope such that the correct portion of the waveform is shown on the screen. As an example, assume that you are using a 50 Ω coax cable to connect to your DUT. With the DUT disconnected, zoom out on the scope until you can see a staircase waveform with three distinct levels, with the middle level halfway between the first and third. The initial transition is between the first and second levels, and should be windowed out. The transition of interest is the transition from the second to the third level. You will want to zero in on this transition, and ensure that the initial transition does not show on the screen. All IConnect algorithms are designed to operate on data acquired in this fashion. If the initial transition is visible in your measurements, you will not be able to extract meaningful models in IConnect.

Capture a sufficient time record

In TDR and transmission (TDR/T) measurement, it is important to capture all the transitions that characterize the DUT. In many cases this includes the low-frequency asymptote, which means the waveform must settle to an appropriate level (0V for short termination, full incident step amplitude for an open, and exactly half the incident step amplitude for a matched, or 50 Ω termination). If your measurement window stops before the waveform has settled, you may clip the waveform at a level that can distort any modeling work to follow. For more information on TDR-based interconnect modeling and basics of TDR, check out our **TDR Interconnect Modeling Quick Guide** at the [TDA Systems web site](#).



Chapter 5 Models

IConnect[®] and MeasureXtractor[™] TDR and VNA software helps you extract many different types of interconnect models. This chapter contains information on each of the different models that IConnect makes available to you as a modeler, as well as step-by-step instructions for easy, accurate extraction.

Contents

5.1	Model Extraction Overview	5-3
5.2	Model Extraction Tips.....	5-4
5.3	Simple Source	5-5
5.4	PWL Source.....	5-6
5.4.1	PWL Source Measure Tab.....	5-7
5.4.2	PWL Source Model Tab.....	5-8
5.5	Lumped	5-10
5.6	Lumped Coupled.....	5-11
5.7	Single Line.....	5-13
5.7.1	Single Line Measure Tab	5-14
5.7.2	Single Line Model Tab	5-16
5.7.3	Single Line Simulate Tab	5-18
5.8	Symmetric Coupled Lines.....	5-19
5.8.1	Symmetric Coupled Lines Measure Tab.....	5-21
5.8.2	Symmetric Coupled Lines Model Tab	5-23
5.8.3	Symmetric Coupled Lines Simulate Tab.....	5-26
5.9	Lossy Line	5-27
5.9.1	Lossy Line Measure Tab.....	5-29

5.9.2	Lossy Line Model Tab.....	5-31
5.9.3	Lossy Line Simulate Tab.....	5-34
5.10	Symmetric Coupled Lossy Lines	5-36
5.10.1	Symmetric Coupled Lossy Lines Measure Tab	5-37
5.10.2	Symmetric Coupled Lossy Lines Model Tab.....	5-39
5.10.3	Symmetric Coupled Lossy Lines Simulate Tab	5-42
5.11	Data Driven Models.....	5-44
5.11.1	Data Driven Measure Tabs	5-46
5.11.2	Data Driven Model Tabs	5-50
5.11.3	Data Driven Simulate Tabs	5-51
5.12	Data Driven Single Line.....	5-52
5.13	Data Driven Coupled Lines	5-54
5.14	Generic 2-port	5-56
5.15	Termination	5-58
5.16	Composite Model.....	5-59
5.16.1	Composite Model Measure Tab.....	5-60
5.16.2	Composite Model Schematic Viewer	5-61
5.16.3	Composite Model Simulate Tab.....	5-63
5.17	Eye Diagram Computation and Mask Testing.....	5-65
5.17.1	Waveforms for Eye Diagram Computation	5-66
5.17.2	Eye Diagram Options.....	5-68
5.17.3	Adding Crosstalk to an Eye Diagram.....	5-71
5.17.4	Eye Measurements and Analysis.....	5-74
5.17.5	Mask Testing.....	5-77
5.18	Partitioning Waveforms in IConnect.....	5-80

5.1 Model Extraction Overview

Model extraction in IConnect is a cooperative effort between IConnect's underlying algorithms and the different model viewers available for creating, editing, and saving the various types of models. Each of the models is accessible both from the commands listed under the **Model** menu item, and from the model toolbar whose visibility can be toggled from the **View/Toolbars/Model Toolbar** menu command.

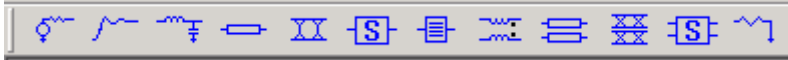


Figure 5-1: Model toolbar.

Many of the model viewers have the ability to take measurements, extract a model, and validate it on your simulator, without having to open another window. For more complex modeling tasks, individual models can be cascaded using the schematic editor in the composite model viewer.

Model files are typically saved with .cir extensions, and may be loaded from the main menu using the **File/Open** command. More typically, they are loaded from the composite model viewer.

This chapter outlines the specifics of how to accurately and easily extract each type of model available to IConnect users. There are also a number of tutorials which walk you through the creation of each type of model in [“Getting Started” on page 2-1](#).

5.2 Model Extraction Tips

Following are a few general guidelines for accurate, efficient modeling.

Become accustomed to working with models directly from the composite model viewer.

This will save you the intermediate step of writing a model to disk and later having to reload it into the composite viewer for verification.

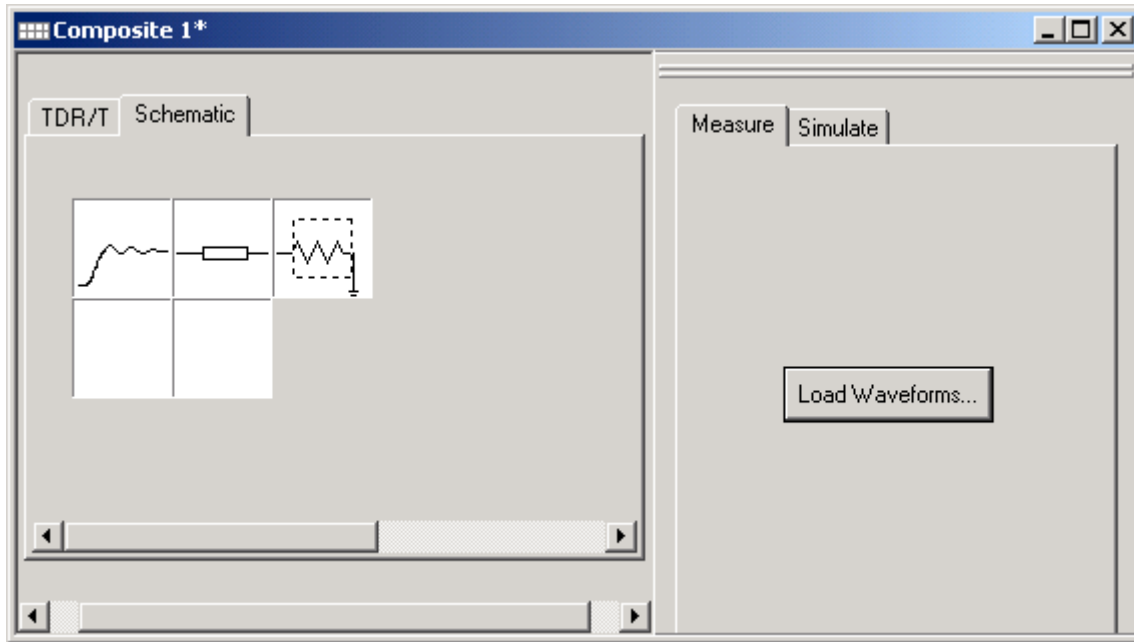


Figure 5-2: Composite model viewer.

Know which models are most effective for which modeling tasks. Some assistance is available in the reference material, but a certain amount of expertise must be gained through experience and through physical understanding of the problem at hand. TDA Systems is also available to give advice on modeling issues.

Ensure that your measured data is accurate. Your models can only be as accurate as your measurements, so taking the time to do it right will pay off later. Ensuring that shorts are shorts, opens are opens, and performing other sanity checks on measured data will save time and headache in troubleshooting later.

Keep models as simple as possible. They are faster to simulate, easier to troubleshoot, and easier to explain to a colleague or customer. One effective way to create simple models is to filter your measured data prior to model extraction, with care of course, to ensure that you are indeed modeling for the risetime at which your circuit will be operating, and not modeling an order of magnitude higher than necessary. Also, be certain that you are modeling your DUT, and not the parasitics associated with your measurement system.

Do you have more modeling tips? Please send them along to TDA Systems.

5.3 Simple Source

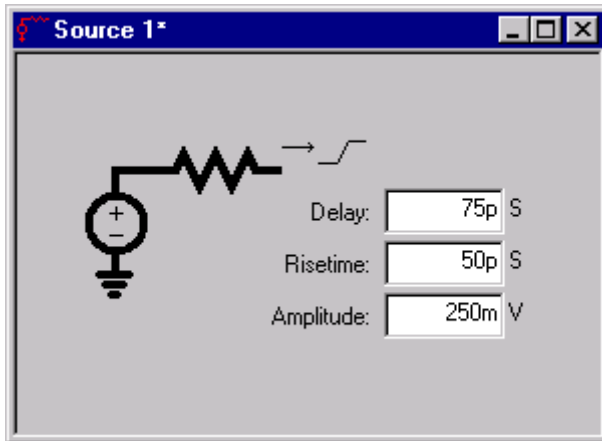


Figure 5-3: Simple Source model editor.

The source model is an excitation model for launching an incident step into a DUT model, and is accessible either from the **Model/Source** menu command or from the model tool bar. The model parameters are step delay, step risetime, and step amplitude. The delay is the time elapsed from $t=0$ before the ramp begins, and the risetime is the 0%-100% risetime of the ramp. The amplitude is that of the incident voltage step injected into the DUT model, so in model verification applications it should be the same as the step launched into the DUT during measurement, e.g. ~200mV for HP TDRs, ~250mV for Tektronix TDRs.

The source model offers a quick, but less accurate, way to simulate a step input. The accuracy is reduced because the sharp corners at the transitions are likely to inject more high-frequency content into your DUT model than your real-world drivers will be injecting into your DUT. Simulation results should be interpreted accordingly.

5.4 PWL Source

The PWL (piecewise-linear) source model is an excitation model for launching an incident step into a DUT model, and is accessible either from the **Model/PWL Source** menu command or from the Model tool bar. The model consists of a set of time-voltage pairs, which describe a time-domain voltage waveform. The data points from which the waveform is built are defined via a graphical user interface, and are built from a measured voltage waveform in order to accurately represent the excitation conditions present during measurement.

5.4.1 PWL Source Measure Tab

The first step toward defining a PWL Source model is to load a voltage waveform by pressing the **Measure** tab, and then pressing the **Reference** button to bring up the waveform selection dialog. You may select a waveform from either your hard drive, your currently open instrument, or from an open waveform viewer.

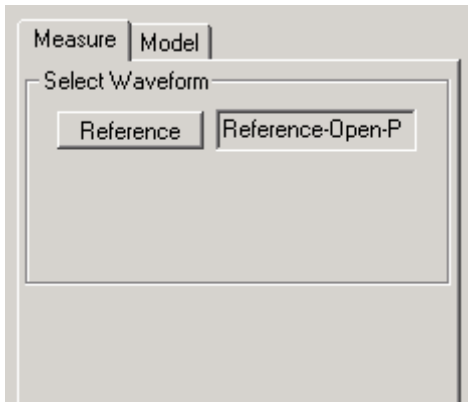


Figure 5-4: Piecewise-linear source Measure tab.

5.4.2 PWL Source Model Tab

When a waveform is initially loaded, the model consists of two data points, one at $t=0$ and one at $t=t_{\text{final}}$. Additional data points are added to the model using the **Model** tab to create the new partitions. There should be enough partitions to accurately represent the measured waveform. (A

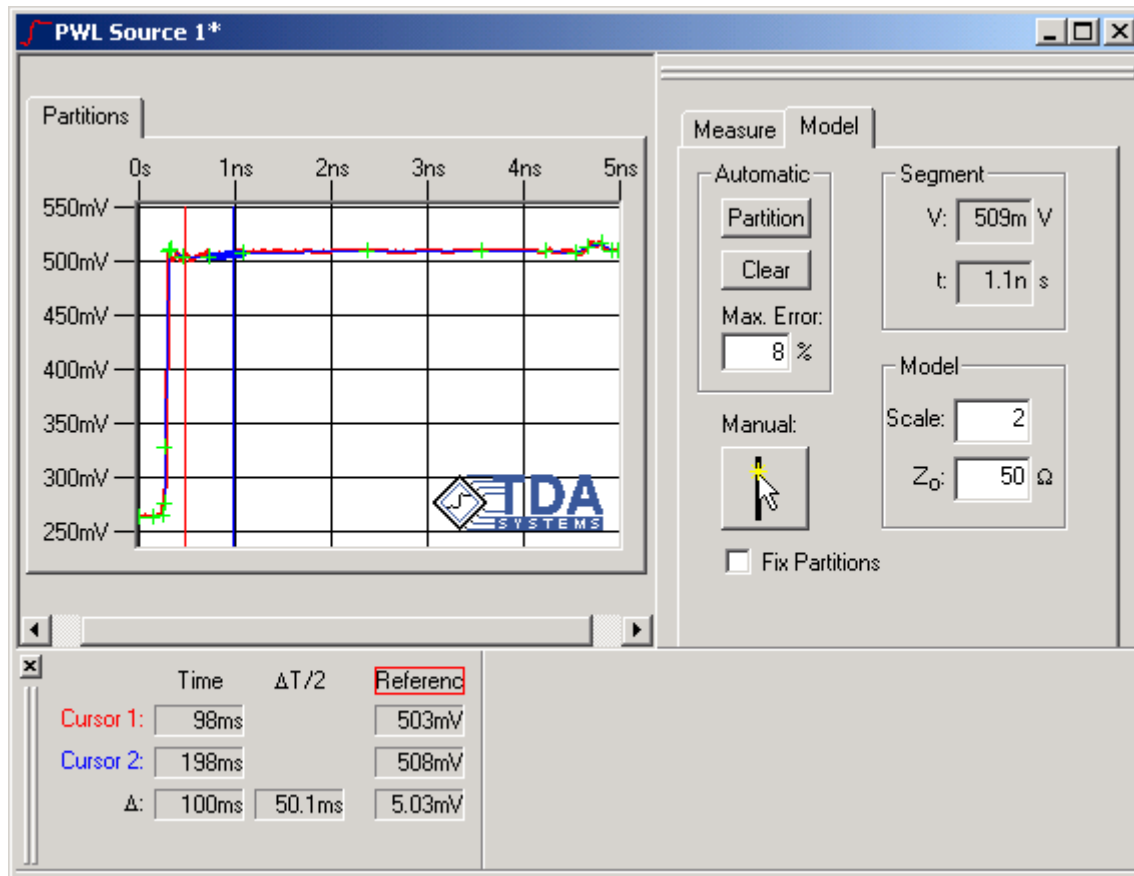


Figure 5-5: The PWL source model tab is used to partition the PWL source waveform.

common rule of thumb is three to five partitions in the transition.) Creation and manipulation of partitions is discussed in “Partitioning Waveforms in IConnect” on page 5-80.

Note: The more partitions created, the slower the subsequent simulation will be. For this reason, create as few as are necessary to achieve the desired accuracy.

Automatic partitioning

As a starting point, you will probably wish to use the automatic partitioning capability to partition the waveform. The parameter that controls the error allowed in the process of autopartitioning is the **Max. Error:** box in the partition window. To understand this parameter, think of the line segment connecting the 10% and 90% points of the edge of the step as being the hypotenuse of a right triangle. The area of this triangle defines the reference error. Now, consider the integral of the difference between the actual waveform and the secant approximation introduced by the piecewise linear representation of the waveform. The cumulative area between any two data points is the actual error. If the **Max. Error:** parameter is set to 5%, for example, the error intro-

duced by the PWL representation is bounded such that the actual error is never more than 5% of the reference error. When you close IConnect, your maximum error setting is retained until you run it again.

The remaining parameters for the model are the scale factor and Z_0 . Z_0 is the characteristic impedance of your measurement system, typically 50Ω , and becomes the output impedance of the modeled TDR source. The scale factor is the amount by which the waveform should be scaled to achieve the proper amplitude step at the launch point into the DUT model. Nominally this should be a value of 2.0 if the step waveform loaded via the measure tab is a positive-going step, or -2.0 if the step waveform is a negative-going step. The factor of two is necessary because a 500mV voltage step is reduced to 250mV as it travels down the launch cable.

5.5 Lumped

Note: Information on how to compute lumped L and C values, and tutorial information on how to incorporate them into IConnect's Measure-Model-Simulate methodology is found in [“Capacitance and Inductance Computation” on page 8-14](#) and [“Lumped LC Computation Tutorial” on page 2-54](#).

The lumped model is built up from RLGC, RLC, RC, or LC sections. The LC sections may be of pi, T, LC, or CL topology. The model may be used as a valid low-frequency approximation to an ideal single transmission line, or as a model for electrically small structures such as packages and connectors which are well represented by such a topology. The model is accessible either from the

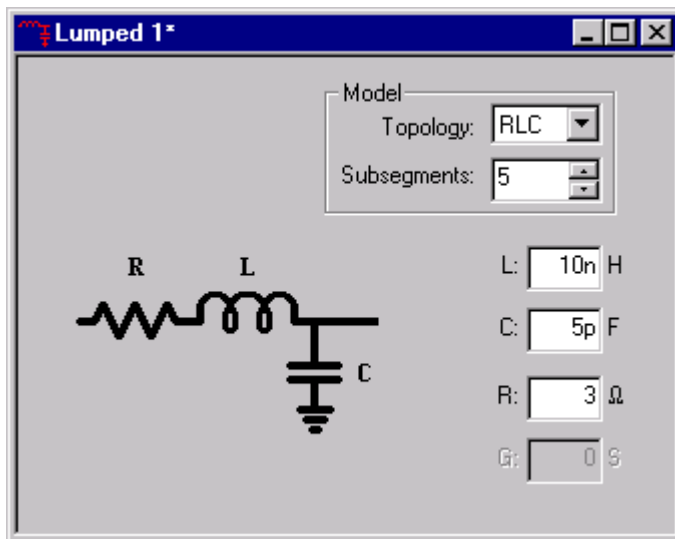


Figure 5-6: Lumped model editor.

Model/Lumped menu command or from the model toolbar. The potential parameters to the model are the inductance L, the capacitance C, the resistance R, and the conductance G. A picture shows graphically which topology has been chosen, and how the R, L, G and C parameters are distributed in that topology. If a more distributed model is desired, you may increase the number of subsegments, in which case each of the values shown in the picture is divided by that number and the subsegments are cascaded. Although this will not typically change the transmission characteristics of your model, it will generally improve the reflection characteristics to look more like those of a distributed transmission line.

Example: Assume that we have entered RLGC topology with 5 subsegments and $L=10\text{nH}$, $C=5\text{pF}$, $R=0.5\Omega$, and $G=5\text{mS}$. This results in a model with 5 cascaded RLGC subsegments, each having $L=2\text{nH}$, $C=1\text{pF}$, $R=0.1\Omega$, and $G=1\text{mS}$.

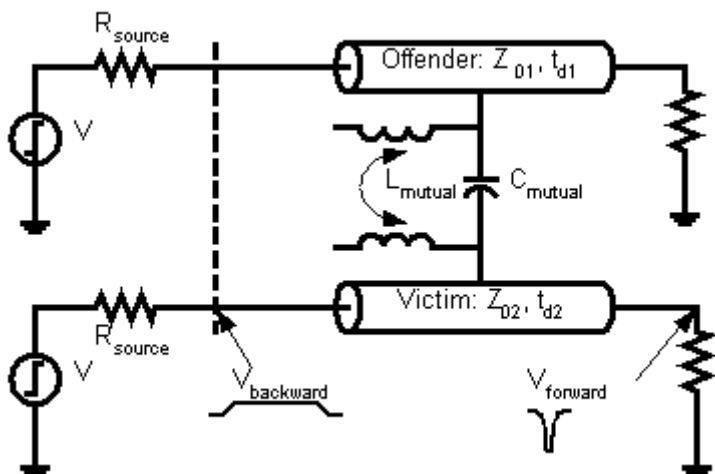
5.6 Lumped Coupled

Note: Information on how to compute lumped L and C values, and tutorial information on how to incorporate them into IConnect's Measure-Model-Simulate methodology is found in [“Capacitance and Inductance Computation”](#) on page 8-14 and [“Lumped LC Computation Tutorial”](#) on page 2-54.

Background

Crosstalk occurs due to energy coupling between adjacent lines. The signal propagating on the offender line transfers part of its energy onto the victim line.

Crosstalk effects can be separated into capacitive and inductive crosstalk. Capacitive crosstalk is due to displacement current through the capacitance between the lines, similar to capacitance between two parallel plates. Inductive crosstalk is due to induction current from the offender line into the victim line.



Forward crosstalk (far end of the victim) is proportional to signal risetime on the offender line, proportional to the line length. It is positive for capacitive coupling and negative for inductive coupling. Backward crosstalk is not a function of risetime, and is a sum of capacitive and inductive coupling.

Further references:

[1] H.W.Johnson, M. Graham, *High-Speed Digital Design, A Handbook of Black Magic*, Prentice Hall, 1993. [2] A. Feller, H.R. Kaupp, J.J. Digiacomio, "Crosstalk and Reflections in High-Speed Digital Systems", *AFIPS Conference Proceedings*, Vol. 27, Part 1, Fall 1965 Joint Computer Conference, pp. 511-525.

Coupled lines modeling

Inductive and capacitive crosstalk may be analyzed separately by providing a current path exclusively for either type of crosstalk. This can be achieved similarly to the self capacitance and inductance measurements. By leaving the far end of the victim line open, only a capacitive path for the current is provided. Alternately, by shorting the far end of the victim to ground, only an inductive path is obtained. In both cases, it is preferred, but not necessary, to terminate the aggressor line with a matched impedance. This prevents the signal on the aggressor line from being reflected and generating further crosstalk, which can complicate the measurement.

Crosstalk can be computed directly from near-end crosstalk observation. For inductive crosstalk, the value computed using the L-mutual computation from the induced waveform, reference short and the background noise waveform directly represents the mutual inductance between the lines. For capacitive crosstalk, the computed value is the mutual capacitance between the aggressor and victim.

Lumped Coupled Model Extraction in IConnect

The IConnect lumped coupled model is built up from RLCG, RLC, RC, or LC sections. The LC sections may be of pi, T, LC, or CL topology. Because it models two structures with inductive and capacitive coupling between them, it may be used as a valid low-frequency approximation to an ideal set of two coupled transmission lines, or as a model for electrically small coupled structures such as packages and which are well represented by such a model.

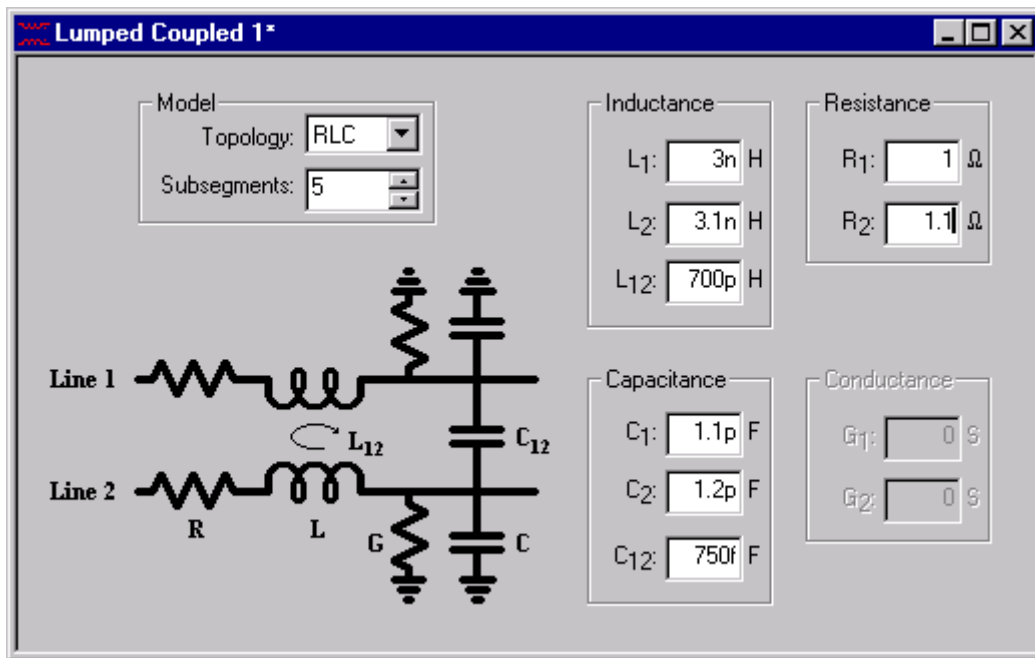


Figure 5-7: Lumped Coupled model editor.

The model is accessible either from the **Model/Lumped Coupled** menu command or from the model toolbar. The potential parameters to the model are the self and mutual inductance L_1 , L_2 , and L_{12} , and the self and mutual capacitance C_1 , C_2 , and C_{12} , the resistance R , and the conductance G . Although the picture in the modeling window shows the RLGC topology, the various LC topologies available are equivalent to the analogous topologies shown in the Lumped (uncoupled) model. If a more distributed model is desired, you may increase the number of subsegments, in which case each of the values shown in the picture is divided by that number and the subsegments are cascaded.

Example: As an example, assume that after extracting values for a medium-sized package we have entered $L_1=L_2=6\text{nH}$, $C_1=C_2=1\text{pF}$, $L_{12}=3\text{nH}$, $C_{12}=500\text{fF}$, LC topology, 5 subsegments. This results in a model with 5 cascaded LC subsegments, each having $L_1=L_2=1.2\text{nH}$, $C_1=C_2=200\text{fF}$, $L_{12}=600\text{pH}$, $C_{12}=100\text{fF}$.

5.7 Single Line

The Single Line model consists of nonuniform cascaded RLGC and/or transmission line sections, and is useful for characterizing structures that are well represented as a single conducting path. Examples would be microstrip traces and some package leads. The single line model is accessible either from the Model/Single Line menu command or from the model tool bar. The graphical user interface of the single line model editor determines the parameters and topology of each of the sections, although R and G values must be determined manually if desired.

Note: A helpful tutorial for modeling with the Single Line model is located in [“Single and Coupled Line Modeling Tutorial”](#) on page 2-63.

5.7.1 Single Line Measure Tab

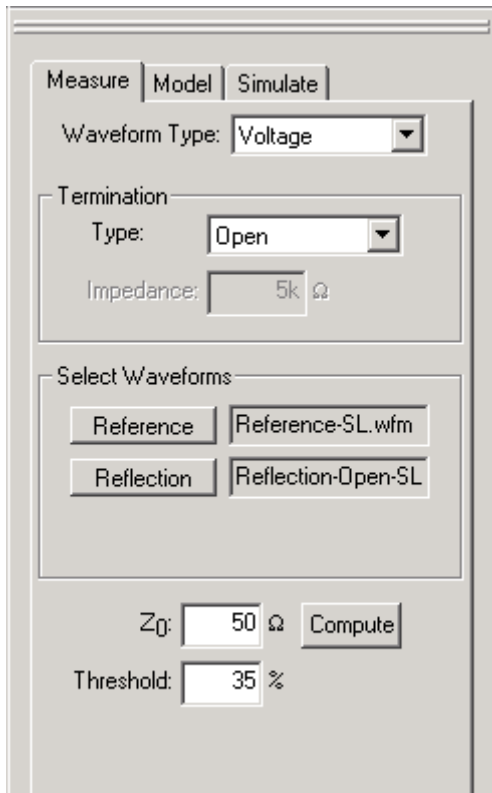


Figure 5-8: Single Line Measure tab.

Waveform Type

The first step toward extracting a Single Line model is to load an impedance waveform from disk by tabbing to the **Measure** tab of the partition window. If you prefer to begin with measured voltage waveforms, choose “Voltage” from the **Waveform Type**: drop-down list.

Note: Using the “Impedance” waveform type is not recommended, since it will not allow IConnect to automatically build the SPICE netlists for validating your model. It is available primarily for backward compatibility with prior versions of IConnect.

Termination Type

You must now tell IConnect how you have the far end of your DUT terminated by making the appropriate selection from the **Type**: dropdown list box in the **Termination** box. If the DUT is unterminated at the far end, choose “Open”. If it is matched, choose “Matched”. IConnect will need this information when it is time to automatically verify your model.

Waveform Selection

Continue by selecting a reference waveform and a reflection waveform as described in [“Impedance Profile Computation” on page 8-6](#). If you have selected matched termination type and would like to make a TDT measurement to use in validating your model, you can select an optional transmission waveform as well.

Compute

Once you have selected your reference and reflection waveforms, press the **Compute** button, and the impedance profile of the line will be computed and placed in the **Partitions** viewing tab, and the **Model** tab will automatically be activated.

Note: Depending on the level of noise in your waveform and potential discontinuities present at your signal launch, it may be necessary to adjust the threshold parameter from its default value if there is oscillation present in your computed waveform. For more information on computing the impedance profile, see ["Impedance Profile Computation" on page 8-6](#).

If you are instead beginning with an existing impedance profile waveform, choose "Impedance" from the **Waveform Type:** drop-down list. Press the **Z_{Line}** button and select the impedance waveform either from the disk, from your current instrument, or from an open waveform viewer. Once a valid waveform is selected, the **Model** tab will automatically be activated.

5.7.2 Single Line Model Tab

The sections of the Single Line model are created by partitioning the DUT's impedance profile waveform into logical sections. IConnect assists in determining the parameters of each of the sections to speed the creation of the model. The sections are edited graphically using the **Partitions** viewing tab, or edited textually using the **Model** tab.

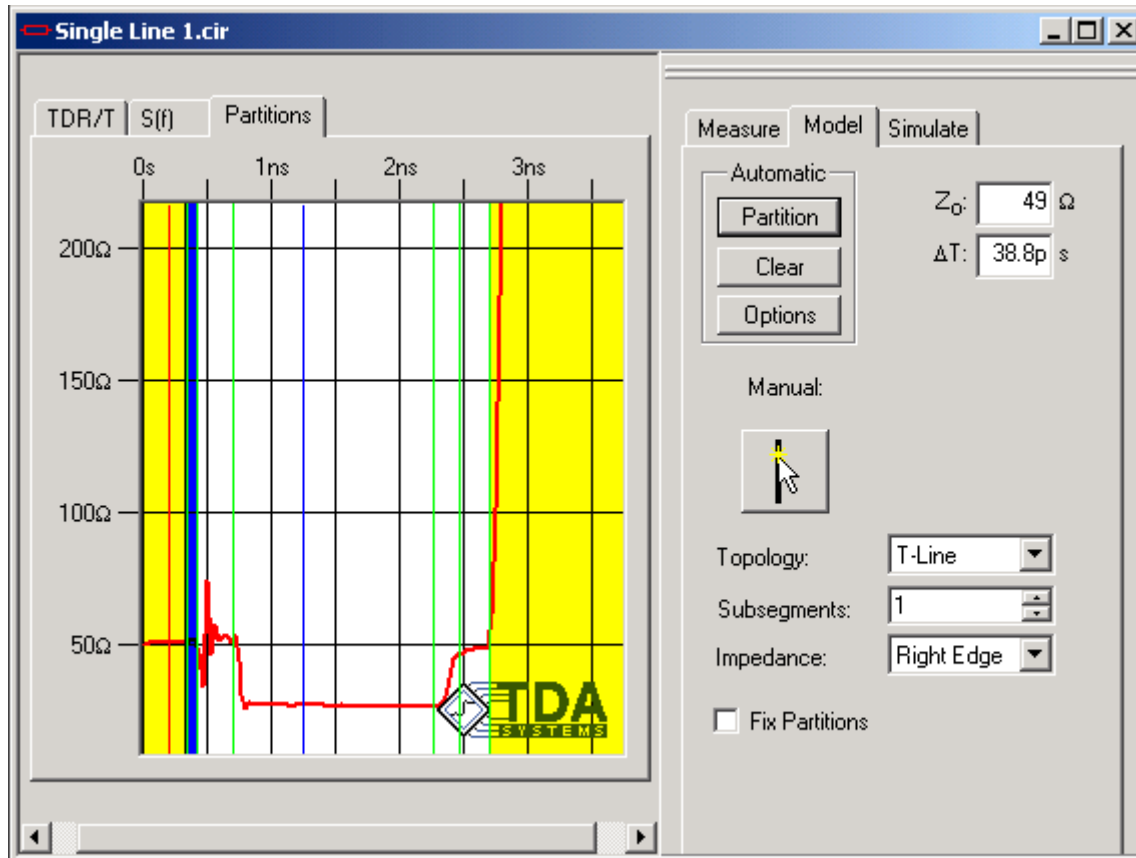


Figure 5-9: Single Line model editor.

When the impedance waveform is initially loaded into the **Partitions** viewing tab, the model consists of a single active partition, with a yellow inactive partition at each end of the waveform. The inactive partitions signify that the parts of the waveform they cover are not to be included in the model, since they represent either the response of the measurement system, in the case of the left-hand inactive partition, and/or termination circuitry, in the case of the right-hand inactive partition. Additional partitions are added to the model automatically by clicking the button labeled **Partition** in the **Automatic** box, or manually using the manual partition creation button. It is up to the discretion of the modeler how many partitions are necessary and what the type of each ought to be. Creation and manipulation of partitions is discussed in [“Partitioning Waveforms in IConnect” on page 5-80](#).

Note: The more partitions created, the slower the subsequent simulation will be. Thus, as few as are necessary to achieve the desired accuracy ought to be created.

Automatic partition creation

The automatic partitioning capability in IConnect can supply a suggested partitioning scheme to get you started in many modeling situations. It should be stressed, however, that this capability does not take precedence over your ability as a modeler, and in many cases manual adjustment by someone with knowledge of the physics of the specific circuit will result in a more accurate model.

There are two parameters involved in impedance profile partitioning: minimum feature height and minimum feature width. Both are set by pressing the **Options** button in the partition window. The minimum feature width, set in units of seconds, can be thought of as the narrowest feature that you need to resolve in your model, and can be set as large as half of your design's target risetime, but not lower than the TDR risetime itself. The minimum feature height, given in ohms, can be thought of as the necessary impedance difference that must be present in order to create a new partition. This value should be small enough to catch the impedance variations that you need to model, but large enough that it does not model measurement noise.

The usefulness of the resultant model, and as such the usefulness of the autopartitioning capability, depends on properly setting the autopartition parameters. If too many partitions are being created, you will need to increase the minimum feature width or height or both. If important waveform features are being skipped, you may need to decrease these parameters. The best settings will depend on how clean your measurements are, how fast a risetime you are targeting your models at, and the resolution of your measured data. When you close IConnect, your current autopartition settings are saved until the next time you start it.

Partition parameters

Each partition is a segment of the final single line model, and as such each has its own electrical parameters. If the topology of the partition is anything other than **TLine**, then the R, L, C, G, and number of subsegments parameters have the same meaning as described for the lumped model. If the topology is chosen to be **TLine**, then the section is modeled by a single section of lossless ideal transmission line described by characteristic impedance Z_0 and delay T_d . It should be noted that Z_0 , T_d , R, L, G, and C are all editable parameters, and that the values suggested by IConnect are meant to be used as a starting point for a good model. (In many cases very little subsequent tweaking will be required to improve the model fit.)

One parameter specific to the impedance-based models is referred to on the single-line partition bar as **Impedance**. The various options in the drop-down list box, **Right Edge**, **Left Edge**, **Mean**, **Min**, and **Max**, all direct IConnect as to what algorithm to use when calculating the impedance value for the selected section from the individual waveform data points. Even in the case in which the topology of the section is not **Tline**, the impedance is still used as an intermediate value to obtain the L and C parameters.

Note: IConnect will not automatically suggest R or G values. It is the responsibility of the user to supply these values if they are desired.

5.7.3 Single Line Simulate Tab

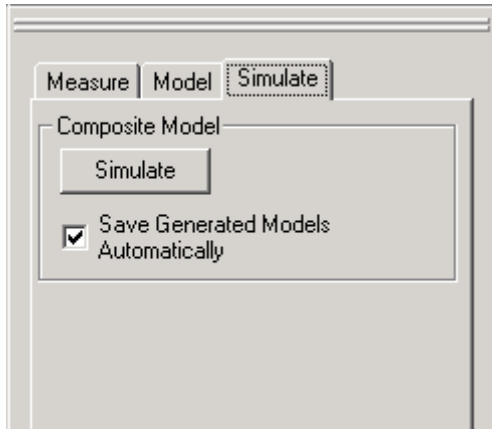


Figure 5-10: Single Line Simulate tab.

Simulation

The **Simulate** tab is used to set up a complete SPICE simulation and compare the results against your measurements, in order to validate your model. After you have extracted the model, you can press the **Simulate** button in the **Composite Model** group on the **Simulate** tab. IConnect will automatically generate a composite model for you, complete with source and termination models, from your measurements and extracted model parameters. IConnect will launch the simulation for you, and the results will be placed in a the **TDR/T** viewing tab alongside your measured waveforms for comparison.

If the **Save Generated Models Automatically** box is checked, as many models as possible will be saved to default names. If the box is unchecked, you will be asked in succession to save each of the submodel files and the composite model file prior to your initial simulation run.

Note: If files are saved automatically, they will overwrite pre-existing files with the same name. For this reason, users should be careful when working on more than one model in the same directory when automatic saving is being used.

5.8 Symmetric Coupled Lines

Note: A helpful tutorial for modeling with the Symmetric Coupled Lines model is located in [“Single and Coupled Line Modeling Tutorial”](#) on page 2-63.

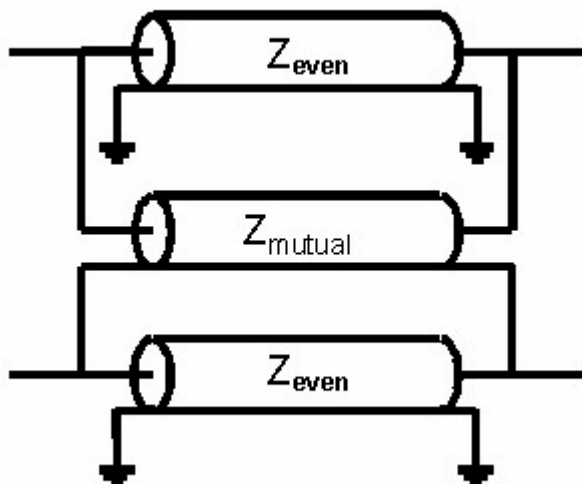
The symmetric coupled lines model consists of nonuniform cascaded coupled RLGC and/or transmission line sections, and is useful for characterizing structures that are well represented as dual symmetric conducting paths. Examples are coupled microstrip traces and some coupled package leads or connector lines. The symmetric coupled model is accessible either from the **Model/Symmetric Coupled** menu command or from the model tool bar. The graphical user interface of the symmetric coupled lines model editor assists the user in determining the parameters and topology of each of the sections, although R and G values must be determined manually if desired.

Even/Odd Mode Modeling

A common approach to modeling two symmetric coupled transmission lines is via the even and odd modes, each of which is described by its own characteristic impedance and propagation constant. The even mode is the response of the system when excited by identical signals at each of the two inputs, while the odd mode is the response of the system when excited by opposite signals at the two inputs. Since any pair of signals at the input to the system can be decomposed into a pair of even signals and a pair of odd signals, an accurate model of the even and odd responses of the system completely describes its response to any arbitrary pair of signals.

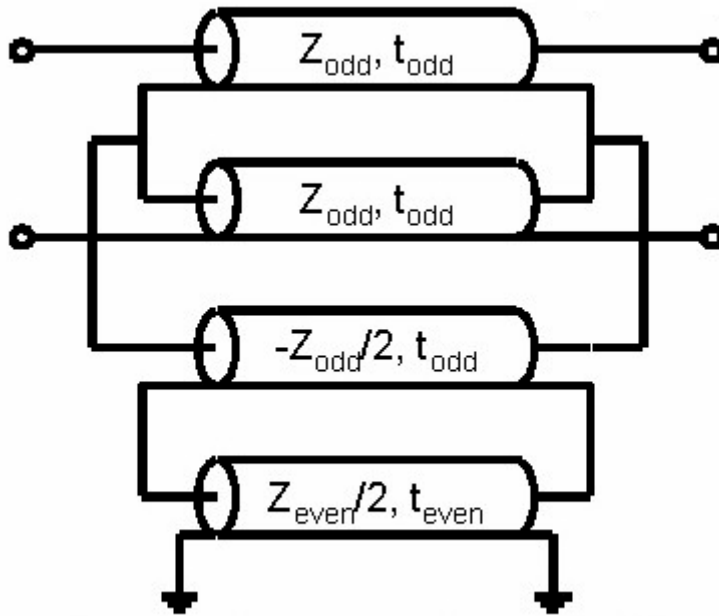
Through use of the even- and odd-mode impedances and delays, IConnect takes symmetric coupled lines modeling a step further by allowing the user to model a response in multiple sections. Since each section has its own electrical parameters, circuits may be characterized in which impedance and coupling vary with position. IConnect allows both distributed and lumped segments in the symmetric coupled lines model, as well as coupled and uncoupled segments.

A simple distributed model which enforces that $T_{\text{even}} = T_{\text{odd}}$ is shown below. Since it accurately recreates the even- and odd-mode impedances, this model is sufficient for circuits in which the timing difference between the even and odd modes is negligible, or for circuits which see only one mode or the other (a typical example would be differential circuits).



A more complex model which accurately characterizes both impedances and both delays is shown below. A passive system is constrained by $Z_{\text{even}} \geq Z_{\text{odd}}$, $T_{\text{even}} \geq T_{\text{odd}}$, and

$Z_{\text{even}}T_{\text{odd}} \geq Z_{\text{odd}}T_{\text{even}}$. Because of the negative impedance associated with one of the transmission lines in the model, IConnect enforces these conditions to ensure the passivity of the model.



Lumped segments are also employed in the symmetric coupled lines model. The lumped segments are of the same topology as those available in the Lumped Coupled model, where the component values are computed according to

$$L = \frac{1}{2}(t_{\text{even}}Z_{\text{even}} + t_{\text{odd}}Z_{\text{odd}})$$

$$L_m = \frac{1}{2}(t_{\text{even}}Z_{\text{even}} - t_{\text{odd}}Z_{\text{odd}})$$

$$C = \frac{t_{\text{even}}}{Z_{\text{even}}}$$

$$C_m = \frac{1}{2}\left(\frac{t_{\text{odd}}}{Z_{\text{odd}}} - \frac{t_{\text{even}}}{Z_{\text{even}}}\right)$$

5.8.1 Symmetric Coupled Lines Measure Tab

The first step toward extracting a model is to compute even- and odd-mode impedance waveforms. This is done in the **Measure** tab.

In order to take even- and odd-mode measurements for your Symmetric Coupled Lossy Line extraction, you will need to set up your TDR to take differential measurements. Your TDR manual will have information on how to do this.

The measurements may be taken with the differential pair either open circuited or terminated at the far end. For more information on how to set up your measurement, refer to ["Single and Coupled Line Modeling Tutorial"](#) on page 2-63.

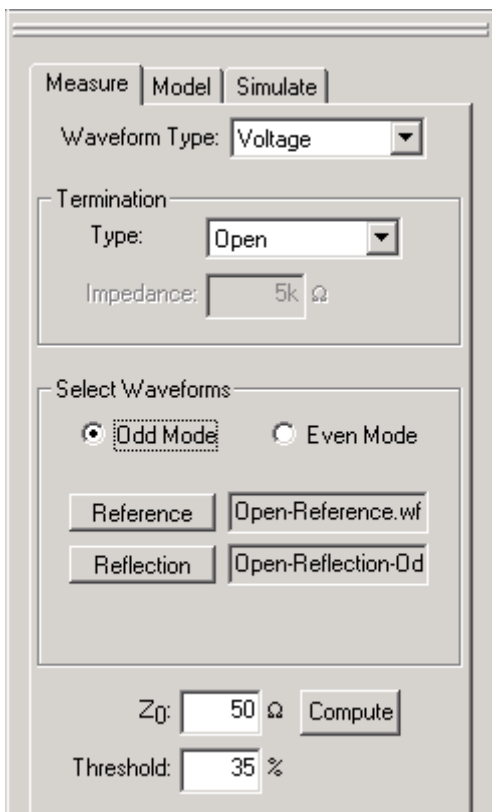


Figure 5-11: Symmetric Coupled Lines Measure tab.

Waveform Type

The **Waveform Type** dropdown list allows you to select which type of waveform you want to start with. We recommend starting with **Voltage** waveforms and computing your impedance waveforms via the ZLine algorithm directly from the **Measure** tab. If you choose **Voltage**, you can load voltage waveforms from anywhere, including live waveforms directly from your TDR. Choosing type **Impedance** allows you to load impedance waveforms previously computed with the ZLine algorithm elsewhere in IConnect.

Note: Waveform Type Impedance functionality has been maintained for backward compatibility, and is not recommended for use since it does not allow you to take advantage of IConnect's automated Measure/Model/Simulate capability.

Termination

In the Termination group box, you can tell IConnect what is happening at the far end of your DUT. If the far end is open-circuited, choose Open. If it's matched, choose Matched.

Select Waveforms

You must begin by selecting a reference waveform. This waveform should be an open circuit (or short circuit) step measured with the DUT removed from the cable or fixture. The step should be measured at the positive-going channel of your differential setup.

The even-mode reflection is the TDR waveform measured at the positive-going channel of the differential setup with both TDR channels launching positive steps. Odd-mode reflection is the TDR waveform measured at the positive-going channel of the differential setup with the negative-going channel launching a negative step.

You can also measure optional transmission waveforms in each case. Care must be taken when measuring differential transmission, since it requires adjustment of timing skew between sampling heads.

For more information on measurement setup for even and odd measurements, see ["Single and Coupled Line Modeling Tutorial" on page 2-63](#).

If you are using **Waveform Type: Impedance**, you can simply load in even and odd impedance waveforms you computed previously.

Compute

Once you have loaded the necessary voltage waveforms, press the Compute button to compute the even and odd impedance profiles of the coupled lines using IConnect's Z-Line algorithm. The impedance profile waveforms will be placed in the Partitions viewing window, and the Model tab on the right will automatically be activated.

Note: You can find more information on the Z-Line algorithm in ["Impedance Profile Computation" on page 8-6](#).

Z_0 and Threshold

Z_0 and Threshold are parameters for the Z-Line algorithm. Z_0 is roughly the impedance of your launch cable or trace, and Threshold is the Z-Line noise reduction parameter. For more specific information on these parameters, see ["Impedance Profile Computation" on page 8-6](#).

5.8.2 Symmetric Coupled Lines Model Tab

Sections of the Symmetric Coupled Lines model are created by partitioning the DUT's even (common) and odd (differential) impedance profile waveforms into logical sections. IConnect assists in determining the parameters of each of the sections to speed the creation of the model. The sections are edited graphically using the **Partitions** viewing tab, or edited textually using the **Model** tab.

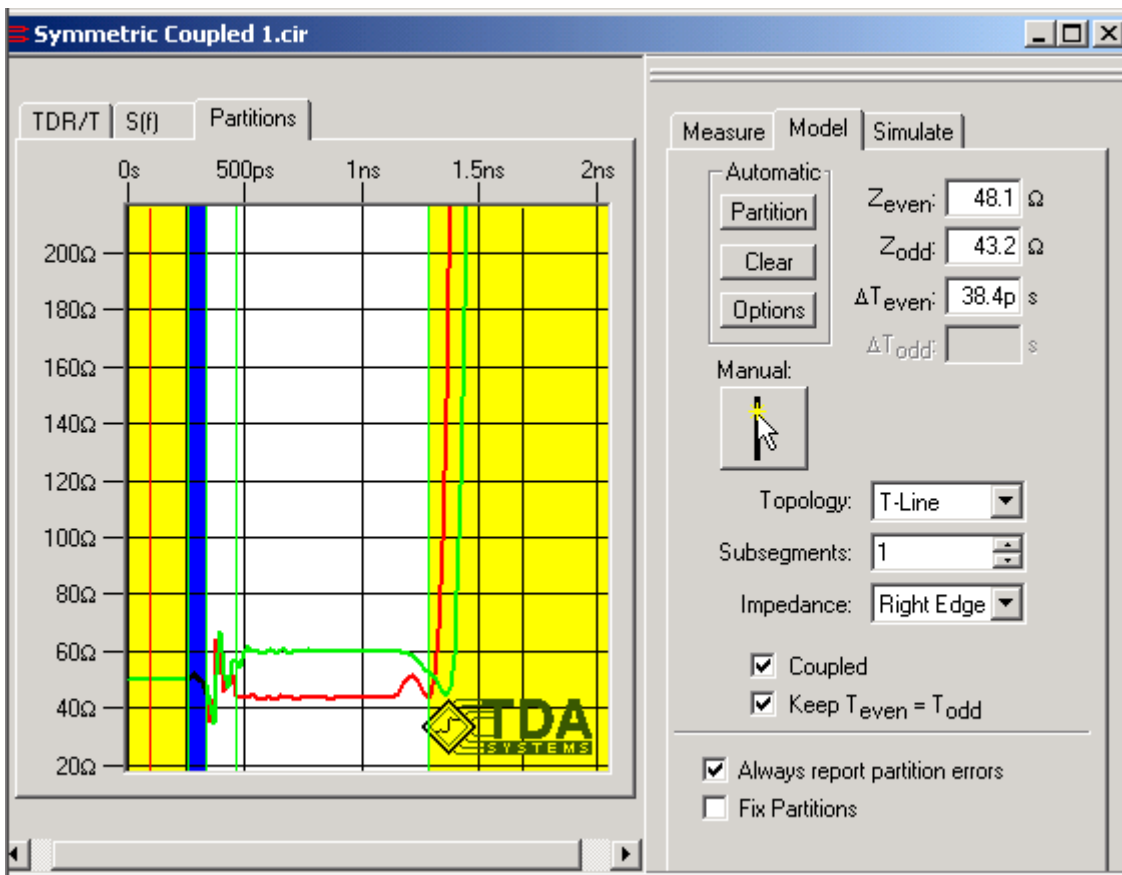


Figure 5-12: Symmetric Coupled Lines Model tab.

When the waveforms are initially loaded, the model consists of a single active partition, with two yellow inactive partitions end of the waveform. The inactive partitions signify that the parts of the waveforms they cover are not to be included in the model, since they represent either the response of the measurement system and/or termination circuitry. Additional partitions are added to the model automatically by clicking the button labeled **Partition**, or manually using the partition creation button. It is left to the discretion of the modeler how many partitions are necessary and what the type of each ought to be. Creation and manipulation of partitions is discussed in [“Partitioning Waveforms in IConnect” on page 5-80](#).

Note: The more partitions created, the slower the subsequent simulation will be. Thus, as few as are necessary to achieve the desired accuracy ought to be created.

Automatic partition creation

The automatic partitioning capability in IConnect can supply a suggested partitioning scheme to get you started in many modeling situations. It should be stressed, however, that this capability does not take precedence over your ability as a modeler, and in many cases manual adjustment by someone with knowledge of the physics of the specific circuit will result in a more accurate model.

There are two parameters involved in impedance profile partitioning: minimum feature height and minimum feature width. Both apply to the even mode waveform, and are set by pressing the **Options** button in the partition window. The minimum feature width, set in units of seconds, can be thought of as the narrowest feature that you need to resolve in your model, and can be set as large as half of your design's target risetime, but not lower than the TDR risetime itself. The minimum feature height, given in ohms, can be thought of as necessary impedance difference that must be present in order to create a new partition. This value should be small enough to catch the impedance variations that you need to model, but large enough that it does not model measurement noise. Additional information in automatic partitioning of impedance waveforms may be found in ["Single Line" on page 5-13](#).

Partition parameters

Each partition is a segment of the final coupled lines model, and as such each has its own electrical parameters. If the topology of the partition is anything other than **TLine**, then the R , L , L_m , C , C_m , G , and number of subsegments parameters have the same meaning as described for the lumped coupled model. If the topology is chosen to be **TLine**, then the section is characterized by its even-mode impedance and delay (Z_e, T_e) and its odd-mode impedance and delay (Z_o, T_o). The details of the model internals have already been described at the beginning of this section. It should be noted that Z_e , T_e , Z_o , T_o , R , L , L_m , G , C , and C_m are all editable parameters, and that the values suggested by IConnect are meant to be used as a starting point for a good model. (In many cases very little subsequent tweaking will be required to improve the model fit.)

Each partition may be either uncoupled or coupled. This simplifies modeling of line pairs which may run separately for a distance and together for a distance. This property is chosen by using the check box marked **Coupled** on the **Partition** tab. For coupled segments, the user has the option of choosing whether or not to keep the even and odd mode delays equal, via the **Keep $T_{\text{even}} = T_{\text{odd}}$** checkbox. Because it simplifies the modeling process, this option is useful for applications in which the delay difference between the modes is negligible. If a segment is chosen to have T_{even} and T_{odd} varying, a dashed cursor appears within the segment. This cursor may be used to choose the odd-mode delay for the section. The odd-mode delay is taken to be the distance from the left-hand even-mode cursor for the section to the odd-mode cursor for the section. Note that the odd-mode delay may not be larger than the even-mode delay, nor may it be less than or equal to zero. Finally, it should be noted that there are certain combinations of even and odd Z and t which cause C_m to go negative. Although positioning of the cursors is allowed which might lead to such combinations, the cursor color is changed to red and the resulting model may not be saved, since it could have stability problems in the subsequent simulation.

Note: If you would like to be apprised of partition errors as you work, check **Always report partition errors** in the partition tab.

One parameter specific to the impedance-based models is referred to on the partition tab as **Impedance**. The various options in the drop-down list box, **Right Edge**, **Left Edge**, **Mean**, **Min**, and **Max**, all direct IConnect as to what algorithm to use when calculating the impedance value for the selected section from the individual waveform data points. Even in the case in which the topol-

ogy of the section is not **TLine**, the impedance is still used as an intermediate value to obtain the L and C parameters.

Note: IConnect will not automatically suggest R or G values. It is the responsibility of the user to supply these values if they are desired.

5.8.3 Symmetric Coupled Lines Simulate Tab

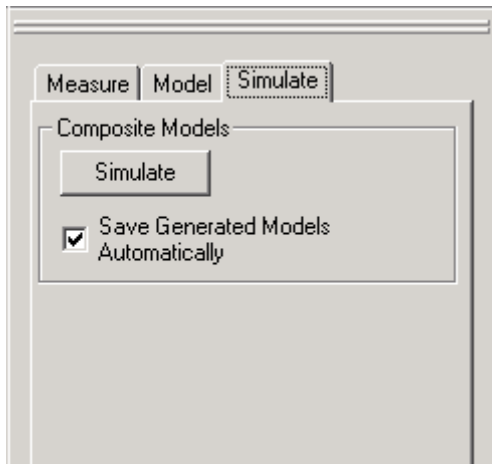


Figure 5-13: Symmetric Coupled Lines Simulate tab.

Simulation

The **Simulate** tab is used to set up a complete SPICE simulation and compare the results against your measurements, in order to validate your model. After you have extracted the model, you can press the **Simulate** button in the **Composite Model** group on the **Simulate** tab. IConnect will automatically generate two composite models for you: one for the even mode simulation, and one for the odd mode simulation. The models are generated complete with source and termination models based on information you provided in the Symmetric Coupled Lines **Measure** tab. IConnect will launch the simulation for you, and the results will be placed in a the **TDR/T** viewing tab alongside your measured waveforms for comparison.

If the **Save Generated Models Automatically** box is checked, as many models as possible will be saved to default names. If the box is unchecked, you will be asked in succession to save each of the submodel files and the composite model file prior to your initial simulation run.

Note: If files are saved automatically, they will overwrite pre-existing files with the same name. For this reason, users should be careful when working on more than one model in the same directory when automatic saving is being used.

If you decide to tweak your model after the initial extraction, you can continue pressing the **Simulate** button after each tweak to evaluate the results.

Even-mode and Odd-mode Verification

IConnect generates and simulates both even and odd modes for you automatically. The reason is that any signal carried by two coupled transmission lines can be reduced to its even and odd components. If both of these components are accurately represented by your model, then any combination of them will also be accurately represented.

Since you know that the signal on your differential pair will be primarily differential, it is more important to accurately model odd-mode impedance and delay. However, accurately modeling of both even and odd responses means your model will not have to rely on the assumption that any common mode signal will be negligible.

5.9 Lossy Line

Note: “[Lossy Line Modeling Tutorial](#)” on [page 2-80](#) is a tutorial which takes you step by step through the process of extracting and validating a lossy line.

As digital circuit speeds continue to rise, transmission line loss plays an increasingly important role in signal integrity analysis. Transmission line loss is caused in large part by two factors: skin loss in the conductors, and dielectric loss. The primary results are risetime and amplitude degradation. Propagation delay becomes dependent on risetime, and amplitude degradation becomes dependent on clock frequency and bit sequences. The combination of these effects is eye closure at increased clock frequencies.

Skin loss

Skin loss is caused by the fact that at higher frequencies, electromagnetic fields are not able to penetrate as deep into conductive materials. As a result, current flow is confined to a continuously shallower “skin” on the surface of the metal as frequency increases. The most noticeable result is an increased resistance at higher frequencies as the current flows through an ever-decreasing cross-sectional area. The resistance of a conductor tends to increase with a square-root-like dependency on frequency. The exact relationship between resistance and frequency depends on the geometry of the conductor. Within IConnect, the equation

$$R \approx R_{ac} \sqrt{f}$$

is used to compute skin effect resistance. The approximation is necessary in order to ensure that the resulting transmission line response is causal, and it should be noted that the inductance of the transmission line necessarily becomes weakly frequency dependent as a result. The meanings of R, L, G, and C of a transmission line are discussed at length in the suggested reading material listed below.

Dielectric loss

Dielectric loss is caused when the ideally insulative dielectric conducts current. Conductivity of most dielectrics used in transmission line design tends to increase linearly with frequency in typical digital frequency ranges. Comparing this relationship to the sublinear, square-root-like dependency of conductor resistance on frequency, we can see that at higher frequencies, conductor loss will dominate skin loss. The frequency at which it becomes dominant is dependent on circuit geometry and materials properties. Within IConnect, the equation

$$G \approx G_{ac} f$$

is used to represent the conductance per unit length contributed by the dielectric. The approximation is necessary to ensure that the resulting transmission line response is causal, and it should be noted that the capacitance of the transmission line necessarily becomes weakly frequency dependent as a result.

Suggested reading

D. A. Smolyansky, S. D. Corey, “Characterization of Differential Interconnects from Time Domain Reflectometry Measurements,” - *Microwave Journal*, Vol. 43, No. 3, pp. 68-80 (TDA Systems application note DIFF-1099)

Models

Clayton R. Paul, *Analysis of Multiconductor Transmission Lines*, John Wiley and Sons, New York, 1994.

5.9.1 Lossy Line Measure Tab

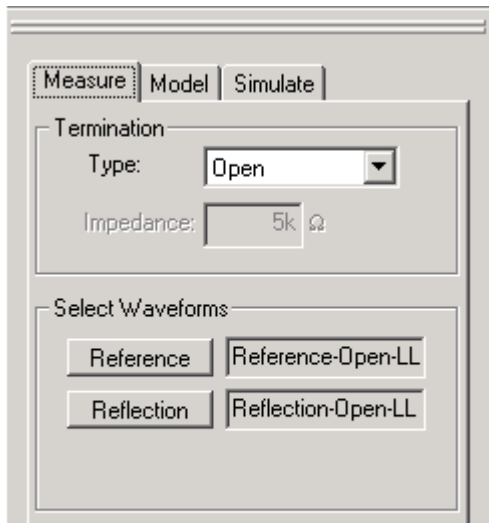


Figure 5-14: Lossy Line Measure tab.

Termination Type

The first step in creating a lossy line model is to load the TDR waveforms. You have the choice of measuring either open-circuit waveforms, or measuring matched reflection and matched transmission waveforms. Which you choose will depend on several factors:

Open-circuit reflection waveforms do not require the ability to fixture one side of the transmission line. Only a clean signal launch at one end of the device is required.

Matched transmission and reflection measurements potentially allow more accurate extraction, provided that care is taken to get accurate measurements, since the loss is very well represented in the transmission waveform. However, taking transmission measurements requires access to the far end of the DUT. It also requires that either the cables being used on the two measurement channels are the same length, or that the two scope channels are deskewed so as to account for any delay difference between the two cables. Your oscilloscope manual will provide details on how to deskew scope channels. If cable delay mismatch is not taken into account, the measured delay of the device will not be accurate.

Once you have decided which extraction method to use, you can inform IConnect by selecting the chosen termination type from the dropdown box labeled **Type** on the **Measure** tab at the right-hand side of the lossy line window.

Waveform Selection

Depending on which method you have chosen, you will have the ability to load in either two or three waveforms. Each waveform is loaded by clicking on its associated button, which allows you to load an active waveform from the current instrument to which you are connected, a previously saved waveform from disk, or a waveform displayed in an open waveform viewer.

Note: The reference waveform refers to an **open-circuit** measurement taken with the DUT removed. This waveform represents the response of the measurement system and fixture in the absence of the DUT.

Models

As the reflection and possibly transmission waveforms are loaded, the scattering parameters will be automatically computed and displayed in the tab labeled **S(f)**.

5.9.2 Lossy Line Model Tab

Model extraction

Having loaded the relevant waveforms into the lossy line modeling window, switch to the tab labeled **Model** at the right-hand side of the lossy line window to extract the model parameters. Within the **Model** tab, you have the option to evaluate the response of the DUT using various model parameters, or to have IConnect optimize the model parameters to fit your measurements.

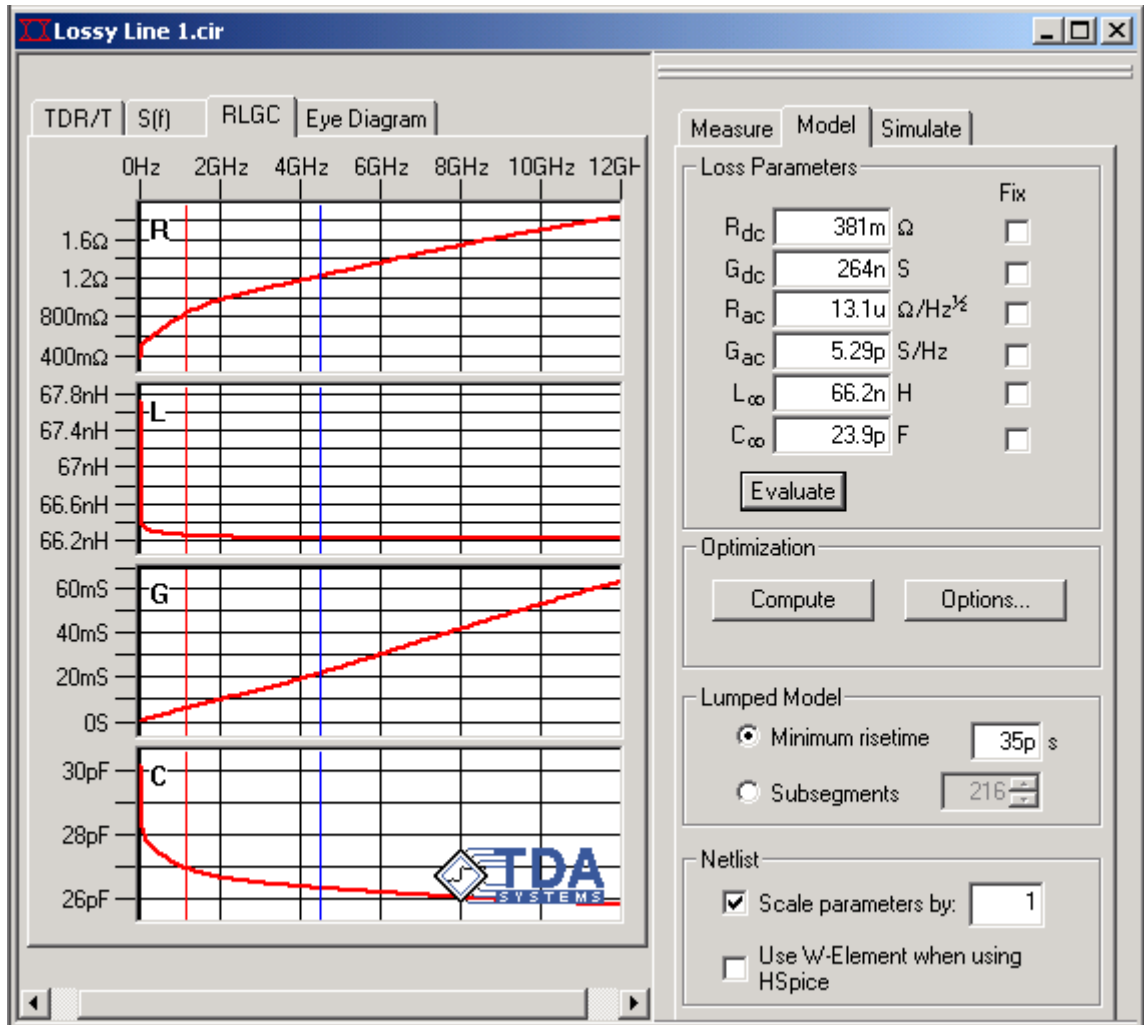


Figure 5-15: When the Compute button is pressed, IConnect optimizes the model parameters and overlays the results on the measured waveforms. Stopping criteria for the optimization may be set by pressing the optimization Options... button.

Evaluate

When you click this button, a model is generated using the parameters currently in the boxes. The response of that model to the measured input step is computed internally within IConnect and displayed in the **TDR/T** viewer tab for comparison with the measured data. The resulting scattering parameters are shown in the **S(f)** tab. The computed R, L, G, and C parameters are shown in the **RLGC** tab.

Optimization

When you click on the **Compute** button in the **Optimization** group, IConnect attempts to optimize the model parameters to fit the measurements you have made. When it has completed the optimization, the results will be displayed as described in the **Evaluate** subsection above. If you press the **Options** button in the **Optimization** group, you will have the option to set two values which are the stopping criteria for the optimization. The **Max. Iterations** value is the maximum number of iterations that IConnect will perform before ending its optimization attempt. The **Precision** is the maximum average percent error that IConnect will allow before the optimization is considered complete. Optimization ends when either the user presses the **Cancel** button, or one of the above two criteria is met.

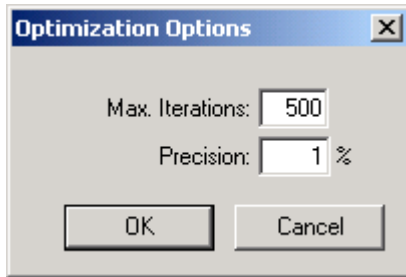


Figure 5-16: The Lossy Line optimization options dialog.

Lumped Model

If you are using lumped-element models, it will be necessary to determine how many segments will be used to represent the circuit. It is often stated that each segment should be no longer electrically than one sixth the risetime of the incident step signal. Using a larger number of subsegments results in a model which is accurate at a faster risetime (higher bandwidth), but the model will be slower to simulate. To specify the number of subsegments, click on the **Subsegments** button and adjust the number of subsegments prior to saving the model.

Note: Specifying the **Minimum Risetime** parameter is the recommended approach, since it causes IConnect to automatically calculate the number of subsegments necessary for an accurate simulation.

Specifying the **Minimum risetime** parameter informs IConnect of the shortest risetime signal with which you intend to excite your model. Click on the **Minimum Risetime** button and enter the risetime value prior to saving the model. IConnect will then calculate the necessary number of subsegments such that the electrical length of each is approximately one sixth the specified risetime. If you want to have greater control over the number of subsegments, you can enter the number explicitly using the **Subsegments** button.

Another issue which may occasionally arise is the ability of a simulator to converge to a solution. Because simulation using distributed models is a relatively new field compared to simulation using lumped elements, it is possible that in some cases simulations using lumped elements may show improved convergence. However, the most common cause for non-convergence in a simulation is bugs in the models of any nonlinear components present in the simulation and not the transmission line models.

Scale Parameters

The IConnect lossy line model gives you the ability to easily scale the length of your model when writing your netlist. This capability is useful since the once the parameters are extracted for a

length of transmission line, they can be scaled to accurately represent transmission lines of varying lengths. To scale the length of your model, check the **Scale Parameters By** box, which will enable the window and allow you to enter a value. If the box is left unchecked, the value in the entry window will be ignored when the model is saved.

Note: The **Scale Parameters By** value is a ratio, and not a percentage.

This property should not be confused with the HSpice length parameter which is specific to the W Element. All W Element models are saved with their length parameter set to 1.0. For more information on the W Element and its associated length parameter, consult your HSpice user manual.

Use W-Element in HSpice

The W-Element is a distributed lossy transmission line model available to HSpice users. If you would like your HSpice lossy line model to be saved as a W-element model, ensure that the **Use W-Element when using HSpice** box is checked. This is the default behavior. If the box is unchecked when the model is saved, the model will be saved as a lumped model, even if it is saved using HSpice syntax. If the model is saved in a simulator syntax other than HSpice, this option is ignored, and the model will always be saved as a lumped-element model.

Note: In some versions of HSpice, setting R_o in the W-Element netlist to a small value will cause HSpice to also set the values of G_d and R_s to zero, resulting in simulated waveforms that do not include skin or dielectric losses. If you see simulation results which appear lossless when they should be lossy and you suspect this is happening, we recommend that you increase the value of the R_{dc} parameter manually in IConnect to a small value, such as $1\text{m}\Omega$, in order to cause HSpice to use the provided values of G_d and R_s .

5.9.3 Lossy Line Simulate Tab

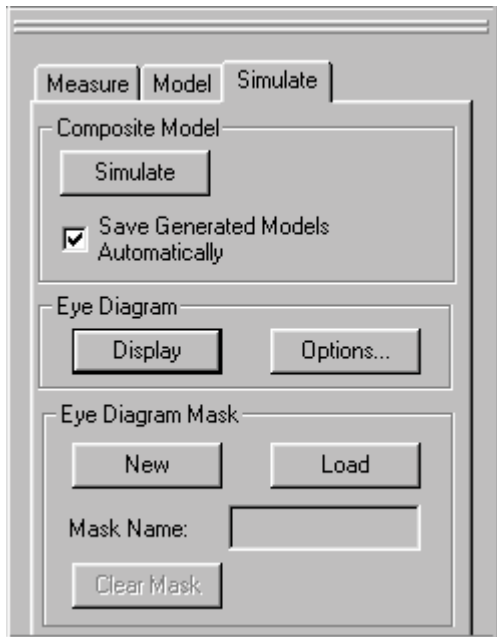


Figure 5-17: Lossy Line Simulate tab.

Simulation

The **Simulate** tab is used to set up a complete SPICE simulation and compare the results against your measurements, in order to validate your model. After you have extracted the model, you can press the **Simulate** button in the **Composite Model** group on the **Simulate** tab. IConnect will automatically generate a composite model for you, complete with source and termination models, from your measurements and extracted model parameters, and will launch the simulation for you. The results will be placed in the **TDR/T** viewing tab alongside your measured waveforms for comparison.

If the **Save Generated Models Automatically** box is checked, as many models as possible will be saved to default names. If the box is unchecked, you will be asked in succession to save each of the submodel files and the composite model file prior to your initial simulation run.

Note: If files are saved automatically, they will overwrite pre-existing files with the same name. For this reason, users should be careful when working on more than one model in the same directory when automatic saving is being used.

If you decide to tweak your model after the initial extraction, you can continue pressing the **Simulate** button after each tweak to evaluate the results.

Eye Diagram

Pressing the **Display** button in the **Eye Diagram** group causes IConnect to display an eye diagram based on the measured data or model parameters and the eye diagram options settings. For information on computing the eye diagram and setting the eye diagram options, see [“Eye Diagram Computation and Mask Testing”](#) on page 5-65.

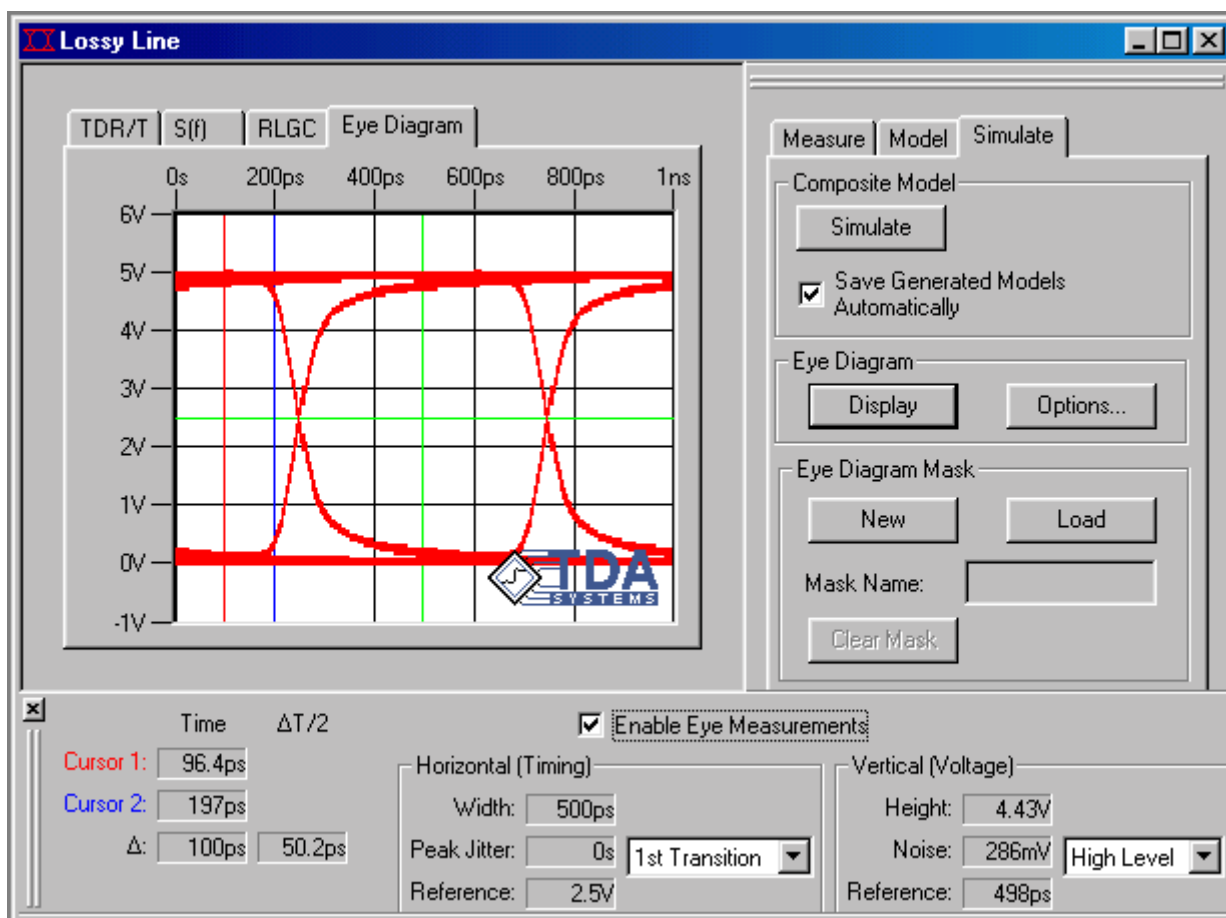


Figure 5-18: When the eye diagram Display button is pressed, IConnect computes an eye diagram based on the current eye diagram options. The options are set by pressing the eye diagram Options... button.

Eye Diagram Mask Testing

You may also perform mask testing on the computed eye diagram. Press the **New** button to create a new mask when none is currently being displayed. If a mask is currently being displayed, the New button will be labeled **Edit Mask**; you may edit the mask by pressing this button. To clear the currently displayed mask, press **Clear Mask**. You may press the **Load** button to load a mask which was previously created and saved in IConnect. For more information on mask creation and manipulation, see [“Eye Diagram Computation and Mask Testing”](#) on page 5-65.

5.10 Symmetric Coupled Lossy Lines

Note: [“Lossy Line Modeling Tutorial” on page 2-80](#) is a tutorial which takes you step by step through the process of extracting and validating uncoupled and coupled lossy line models.

The Symmetric Coupled Lossy Line model allows you to simulate skin effect and dielectric loss, together with intersymbol interference (ISI) and crosstalk effects. It also allows you to predict the eye diagram degradation due to losses and ISI in the interconnects. You may also incorporate eye diagram closure effects due to crosstalk from neighboring line pairs.

Note: Important and useful background information on lossy line modeling may be found in [“Lossy Line” on page 5-27](#). The Symmetric Coupled Lossy Lines model uses the same mathematical basis as the Lossy Line model, except that its RLGC parameters are now 2x2 matrices instead of scalar quantities.

Symmetric Coupled Lossy Lines RLGC Parameters

The Symmetric Coupled Lossy Lines R, L, G, and C parameters are 2x2 matrices of the form

$$\begin{bmatrix} x_{11} & x_{12} \\ x_{12} & x_{11} \end{bmatrix}$$

The matrices are symmetric (off-diagonals x_{12} are equal) because RLGC matrices are always symmetric. The matrix is persymmetric (diagonals x_{11} are equal) because of the symmetry of the differential transmission lines being modeled. As a result, only two matrix entries, x_{11} and x_{12} are necessary to represent each parameter. Due to the definitions of R, L, G, and C, the off-diagonal elements of the G and C matrices are necessarily negative values.

Even and Odd Mode Extraction

Because there are twice as many parameters as there are in a single lossy line model, even and odd measurements are necessary to extract the model.

5.10.1 Symmetric Coupled Lossy Lines Measure Tab

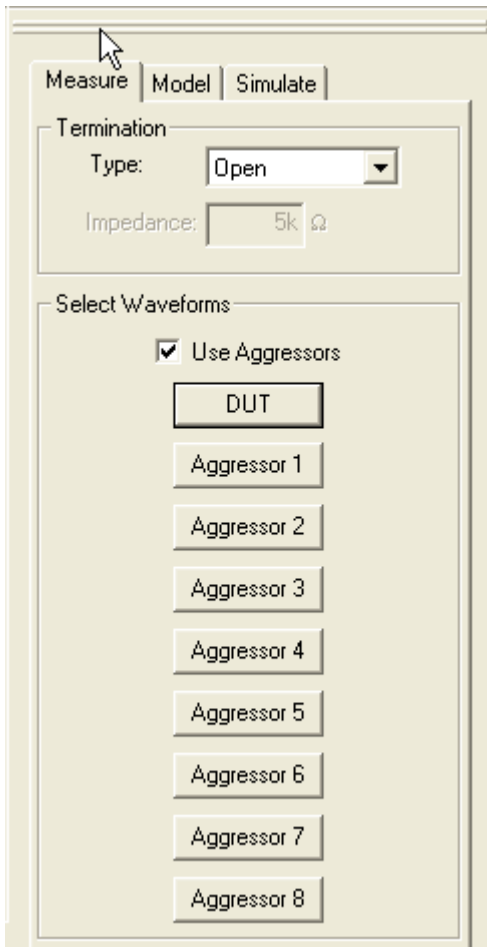


Figure 5-19: Symmetric Coupled Lossy Lines Measure tab with aggressors enabled. Aggressors need only be enabled if you are computing an eye diagram with crosstalk.

Termination Type

The first step in creating a symmetric coupled lossy lines model is to load the TDR waveforms. You have the choice of measuring either open-circuit waveforms, or measuring matched reflection and matched transmission waveforms. Which you choose will depend on several factors:

Open-circuit reflection waveforms do not require the ability to fixture one side of the transmission line. Only a clean signal launch at one end of the device is required. You must only deskew between two channels on a single sampling head, which is straightforward.

Matched transmission and reflection measurements potentially allow more accurate extraction, provided that care is taken to get accurate measurements, since the loss is very well represented in the transmission waveform. However, taking transmission measurements requires access to the far end of the DUT. It also requires that either the cables being used on the two measurement channels are the same length, or that the two scope channels are deskewed so as to account for any delay difference between the two cables. Furthermore, because the symmetric coupled lossy line model requires differential measurements, the deskewing has to be done between separate sampling heads, which is not a trivial task. Your oscilloscope manual will provide details on how to

deskew scope channels. If cable delay mismatch is not taken into account, the measured delay of the device will not be accurate.

Once you have decided which extraction method to use, you can inform IConnect by selecting the chosen termination type from the dropdown box labeled **Type** on the **Measure** tab at the right-hand side of the lossy line window.

Waveform Selection

To select your waveforms, you will need to click on the **DUT** button, which opens a window titled "Load DUT Waveforms". Depending on which method you have chosen, you will have the ability to load in either three or five waveforms. Each waveform is loaded by clicking on its associated button, which allows you to load an active waveform from the current instrument to which you are connected, a previously saved waveform from disk, or a waveform displayed in an open waveform viewer. Once you have loaded all DUT waveforms, click **OK**.

Note: The reference waveform refers to an **open-circuit** measurement taken with the DUT removed. This waveform represents the response of the measurement system and fixture in the absence of the DUT.

The even-mode reflection is the TDR waveform measured at the positive-going channel of the differential setup with both TDR channels launching positive steps. Odd-mode reflection is the TDR waveform measured at the positive-going channel of the differential setup with the negative-going channel launching a negative step. If you have chosen matched termination type, you will need to measure a TDT waveform in each case as well.

As the reflection and possibly transmission waveforms are loaded, the scattering parameters will be automatically computed and displayed in the tab labeled **S(f)**.

Aggressor Waveforms

If you intend to perform an eye diagram analysis which includes crosstalk from nearby line pairs, you will need to load aggressor waveforms for each nearby line pair. Begin by clicking on the **Use Aggressors** button on the **Measure** page. A set of eight buttons will appear, allowing you to incorporate crosstalk effects from up to eight nearby line pairs. Clicking on a button opens a window that allows you to enter the appropriate aggressor waveforms. Detailed information on loading aggressor waveforms is located in ["Waveforms for Eye Diagram Computation"](#) on page 5-66.

5.10.2 Symmetric Coupled Lossy Lines Model Tab

Model extraction

Having loaded the relevant waveforms into the symmetric coupled lossy lines modeling window, switch to the tab labeled **Model** at the right-hand side of the lossy line window to extract the model parameters. Within the **Model** tab, you have the option to evaluate the response of the DUT using various model parameters, or to have IConnect optimize the model parameters to fit your measurements.

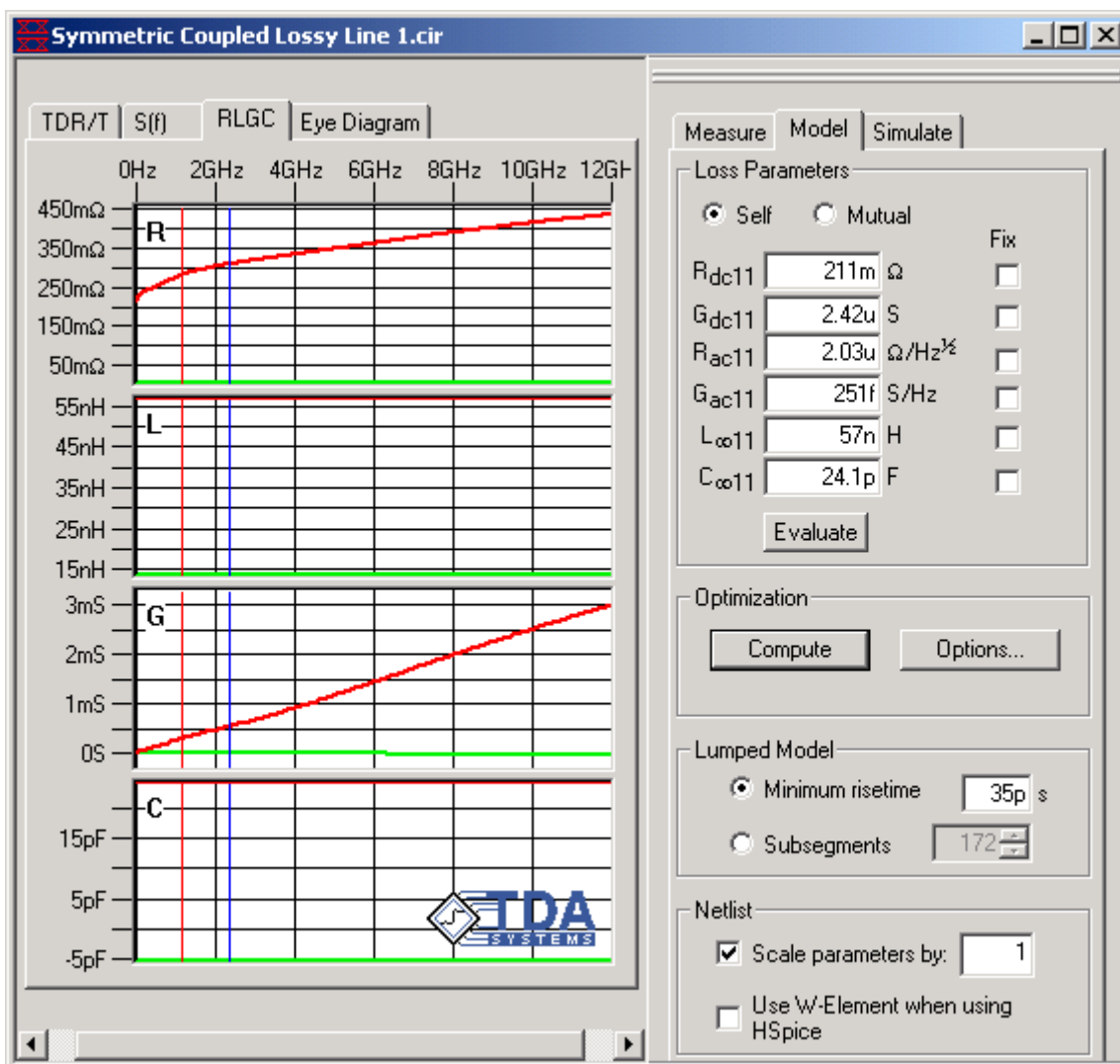


Figure 5-20: When the Compute button is pressed, IConnect optimizes the model parameters and overlays the results on the measured waveforms. Stopping criteria for the optimization may be set by pressing the optimization Options... button.

Self and Mutual Parameters

The self and mutual parameters are the diagonal and off-diagonal elements of the matrices described in “Symmetric Coupled Lossy Lines” on page 5-36. To edit one or the other, first click on the **Self** or **Mutual** radio button to display the correct set of parameters for editing.

Evaluate

When you click this button, a model is generated using the parameters currently in the boxes. The response of that model to the measured input step is computed internally within IConnect and displayed in the **TDR/T** viewer tab for comparison with the measured data. The resulting scattering parameters are shown in the **S(f)** tab. The computed R, L, G, and C parameters are shown in the **RLGC** tab.

Optimization

When you click on the **Compute** button in the **Optimization** group, IConnect attempts to optimize the model parameters to fit the measurements you have made. When it has completed the optimization, the results will be displayed as described in the **Evaluate** subsection above.

Note: You may be able to speed up your optimization by first fixing all mutual parameters at zero and optimizing only the self parameters. Then unfix the mutual parameters and allow them to optimize as well. Typically R_{dc12} and R_{ac12} may be fixed at zero.

If you press the **Options** button in the **Optimization** group, you will have the option to set two values which are the stopping criteria for the optimization. The **Max. Iterations** value is the maximum number of iterations that IConnect will perform before ending its optimization attempt. The **Precision** is the maximum average percent error that IConnect will allow before the optimization is considered complete. Optimization ends when either the user presses the **Cancel** button, or one of the above two criteria is met.

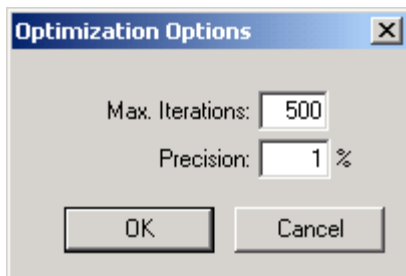


Figure 5-21: The Symmetric Coupled Lossy Line optimization options dialog.

Lumped Model

If you are using lumped-element models, it will be necessary to determine how many segments will be used to represent the circuit. It is often stated that each segment should be no longer electrically than one sixth the risetime of the incident step signal. Using a larger number of subsegments results in a model which is accurate at a faster risetime (higher bandwidth), but the model will be slower to simulate. To specify the number of subsegments, click on the **Subsegments** button and adjust the number of subsegments prior to saving the model.

Note: Specifying the **Minimum Risetime** parameter is the recommended approach, since it causes IConnect to automatically calculate the number of subsegments necessary for an accurate simulation.

Specifying the **Minimum risetime** parameter informs IConnect of the shortest risetime signal with which you intend to excite your model. Click on the **Minimum Risetime** button and enter the risetime value prior to saving the model. IConnect will then calculate the necessary number of subsegments such that the electrical length of each is approximately one sixth the specified risetime.

If you want to have greater control over the number of subsegments, you can enter the number explicitly using the **Subsegments** button.

Another issue which may occasionally arise is the ability of a simulator to converge to a solution. Because simulation using distributed models is a relatively new field compared to simulation using lumped elements, it is possible that in some cases simulations using lumped elements may show improved convergence. However, the most common cause for non-convergence in a simulation is bugs in the models of any nonlinear components present in the simulation and not the transmission line models.

Scale Parameters

The IConnect lossy line model gives you the ability to easily scale the length of your model when writing your netlist. This capability is useful since the once the parameters are extracted for a length of transmission line, they can be scaled to accurately represent transmission lines of varying lengths. To scale the length of your model, check the **Scale Parameters By** box, which will enable the window and allow you to enter a value. If the box is left unchecked, the value in the entry window will be ignored when the model is saved.

Note: The **Scale Parameters By** value is a ratio, and not a percentage.

This property should not be confused with the HSpice length parameter which is specific to the W Element. All W Element models are saved with their length parameter set to 1.0. For more information on the W Element and its associated length parameter, consult your HSpice user manual.

Use W-Element in HSpice

The W-Element is a distributed lossy transmission line model available to HSpice users. If you would like your HSpice lossy line model to be saved as a W-element model, ensure that the **Use W-Element when using HSpice** box is checked. This is the default behavior. If the box is unchecked when the model is saved, the model will be saved as a lumped model, even if it is saved using HSpice syntax. If the model is saved in a simulator syntax other than HSpice, this option is ignored, and the model will always be saved as a lumped-element model.

Note: In some versions of HSpice, setting the R_o matrix in the W-Element netlist to small values will cause HSpice to also set the values of the G_d and R_s matrices to zero, resulting in simulated waveforms that do not include skin or dielectric losses. If you see simulation results which appear lossless when they should be lossy and you suspect this is happening, we recommend that you increase the value of the R_{dc11} parameter manually in IConnect to a small value, such as $1\text{m}\Omega$, in order to cause HSpice to use the provided values of G_d and R_s .

5.10.3 Symmetric Coupled Lossy Lines Simulate Tab

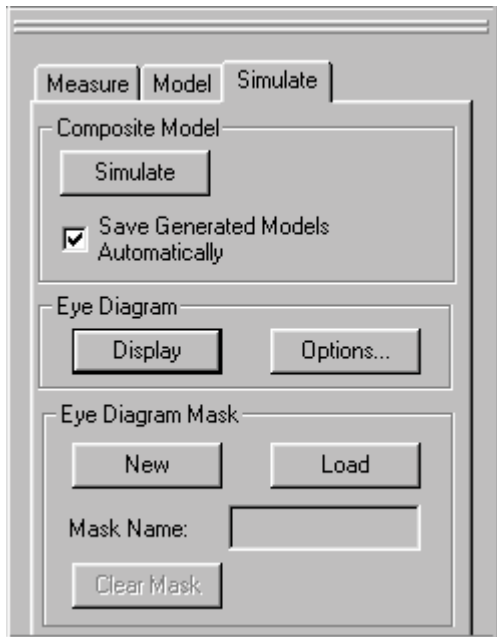


Figure 5-22: Symmetric Coupled Lossy Lines Simulate tab.

Simulation

The **Simulate** tab is used to set up a complete SPICE simulation and compare the results against your measurements, in order to validate your model. After you have extracted the model, you can press the **Simulate** button in the **Composite Model** group on the **Simulate** tab. IConnect will automatically generate two composite models for you: one for the even mode simulation, and one for the odd mode simulation. The models are generated complete with source and termination models based on information you provided in the Symmetric Coupled Lines **Measure** tab. IConnect will launch the simulation for you, and the results will be placed in a the **TDR/T** viewing tab alongside your measured waveforms for comparison.

If the **Save Generated Models Automatically** box is checked, as many models as possible will be saved to default names. If the box is unchecked, you will be asked in succession to save each of the submodel files and the composite model file prior to your initial simulation run.

Note: If files are saved automatically, they will overwrite pre-existing files with the same name. For this reason, users should be careful when working on more than one model in the same directory when automatic saving is being used.

If you decide to tweak your model after the initial extraction, you can continue pressing the **Simulate** button after each tweak to evaluate the results.

Even-mode and Odd-mode Verification

IConnect generates and simulates both even and odd modes for you automatically. The reason is that any signal carried by two coupled transmission lines can be reduced to its even and odd components. If both of these components are accurately represented by your model, then any combination of them will also be accurately represented.

Since you know that the signal on your differential pair will be primarily differential, it is more important to accurately model odd-mode impedance and delay. However, accurately modeling of both even and odd responses means your model will not have to rely on the assumption that any common mode signal will be negligible.

Eye diagram

Pressing the **Display** button in the **Eye Diagram** group causes IConnect to display an eye diagram of the odd mode of your device, based on the measured data or model parameters and the eye diagram options settings. For information on computing the eye diagram and setting the eye diagram options, see [“Eye Diagram Computation and Mask Testing” on page 5-65](#).

Eye Diagram Mask Testing

You may also perform mask testing on the computed eye diagram. Press the **New** button to create a new mask when none is currently being displayed. If a mask is currently being displayed, the New button will be labeled **Edit Mask**; you may edit the mask by pressing this button. To clear the currently displayed mask, press **Clear Mask**. You may press the **Load** button to load a mask which was previously created and saved in IConnect. For more information on mask creation and manipulation, see [“Eye Diagram Computation and Mask Testing” on page 5-65](#).

5.11 Data Driven Models

Note: “[MeasureXtractor™ Modeling Tutorial](#)” on page 2-97 takes you through the measurement and modeling process for the Data Driven Single Line and Data Driven Coupled Lines models.

The data driven models in IConnect allow you to bypass any manual intervention in the modeling process using IConnect’s MeasureXtractor™ modeling technology. The extracted model is a SPICE netlist which accurately represents the circuit being modeled. The passivity of the model is guaranteed by the extraction process. IConnect currently supports automatic model extraction for two-port and four-port models, which correspond nicely to single-line and coupled-line circuits.

When is a data-driven model most appropriate?

Data Driven models are best used for devices which are electrically short or medium-sized. A rough rule of thumb might be that the device should be fewer than 20 wavelengths in length. Longer devices may take too long to extract. Another equivalent rule of thumb might be that the electrical length of the device shouldn’t be more than 100 times the risetime of the signal you will use to excite the model. This is because for waveforms with a large number of data points, the extraction time will increase. Furthermore, simulation time of the resultant model will also increase. If your circuit is large, such as

Note: Electrically large circuits will take longer to extract using the data driven model, and will have larger simulation run times. If your circuit’s electrical length is greater than, say, 100 risetimes, you may do better to use one of IConnect’s distributed models, such as Lossy Line or Coupled Lossy Line.

Data driven model extraction results in a SPICE netlist which very accurately replicates the behavior of the measured device, but this netlist cannot be tweaked, and will not be structurally recognizable. As a result, you should not use the data driven models if you intend to tweak the resulting model.

Time-domain and frequency-domain data

As with all IConnect models, the data driven models may be extracted from TDR data. However, you may also extract your model directly from frequency-domain scattering parameters stored in a Touchstone file. Touchstone is a common file format used to export frequency-domain data from network analyzers and field solvers.

Multiport networks

The data driven models are based on the concept of multiport networks and a system matrix. For this reason, you have to take multiple measurements to be able to characterize the model. An n -port network is generally characterized by an $n \times n$ system matrix, which implies n^2 waveforms. However, because interconnect systems are reciprocal, their system matrices are symmetric, which means the lower half of the matrix is the same as the other half. In measurement terms, this means that the TDT measurement from port 1 to port 2 is the same as that from port 2 to port 1.

As a result, we need only $n(n + 1)/2$ response waveforms to characterize the circuit. If we are using TDR measurements, we will need an additional n reference waveforms to de-embed the effects of the measurement system on the results. This is the open circuit waveform measured at the end of the fixturing system used to excite the model, just like the reference waveforms used in

all other IConnect models. Each of the data driven measure tabs shows you exactly which waveforms you will need to measure in order to extract the model.

5.11.1 Data Driven Measure Tabs

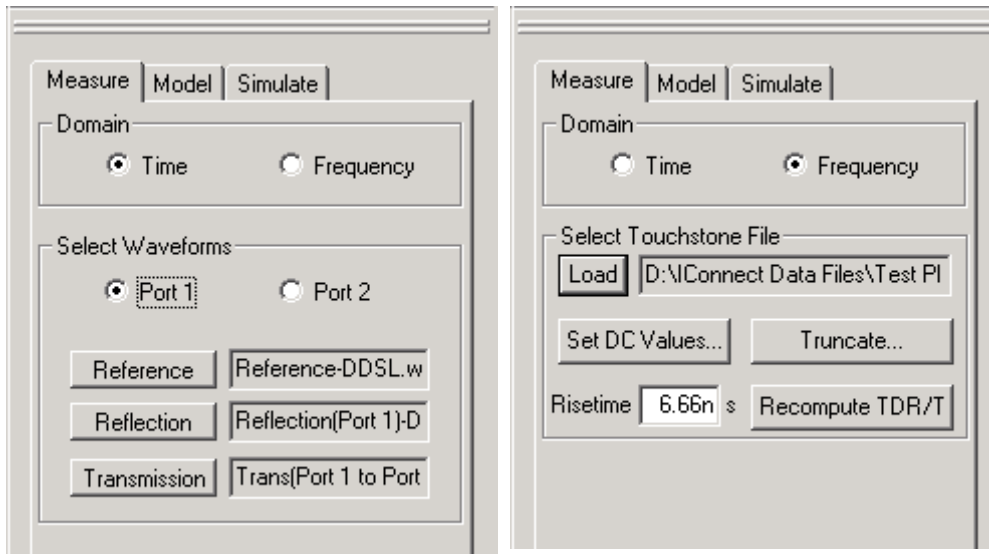


Figure 5-23: The Data Driven Single Line Measure tab shown here can load either time-domain or frequency-domain data. The same is true for the Data Driven Coupled Lines measure tab, not shown.

Domain

Use the **Time** and **Frequency** domain buttons to configure IConnect to load the type of data you will be using. If you are using time-domain data, you will load the data from your instrument or from a waveform file on disk the same way it is loaded into the other IConnect models. If you are using frequency domain files, you will load it from a Touchstone file.

Note: If you are bringing in frequency domain data to extract your data-driven model, be sure to read the sections below on how to get the best results.

Time Domain: Select Waveforms

Begin time-domain waveform selection by clicking on the radio button for the port you want to measure. (IConnect defaults to port 1.) You will need to load a reference waveform and a set of TDR and TDT devices from your DUT, as directed by the Measure tab.

Frequency Domain: Load Touchstone Files

You will need to load a two-port Touchstone file if you are using Data Driven Single Line, or a four-port Touchstone file if you are using Data Driven Coupled Line. Most frequency-domain measurement instruments and most field solvers are capable of exporting Touchstone data.

Note: IConnect expects data which is sampled evenly in frequency. If your data is not, you will have to edit it such that it is.

Frequency Domain: Risetime

Since IConnect will be converting your frequency-domain measurements into equivalent TDR measurements, it will need to know the risetime that you want to target in the transformation. If

you set the target risetime to zero, IConnect will compute the fastest possible risetime and use it. Otherwise you may set the risetime to whatever is appropriate for your modeling task. Ideally it should be not much higher than the risetime the DUT will see in actual usage. If you set a risetime which is deemed too high based on the frequency range of the measurements, IConnect will set it to the fastest possible risetime supported by the data.

Frequency Domain: Set DC Values

Because it is common for frequency-domain measurement instruments and field solvers to not measure or compute the device response at DC, it is usually necessary to extrapolate the measured data down to zero frequency. IConnect will automatically do this on its own when the Touchstone file is loaded, based on the lowest frequency measurements present. However, if you know the DC response based on DC measurements or other means, you may click the **Set DC Values** button, which will bring up a window for entering the DC response. Remember, if you do not accurately know the DC response, you should allow IConnect to do its own extrapolation.

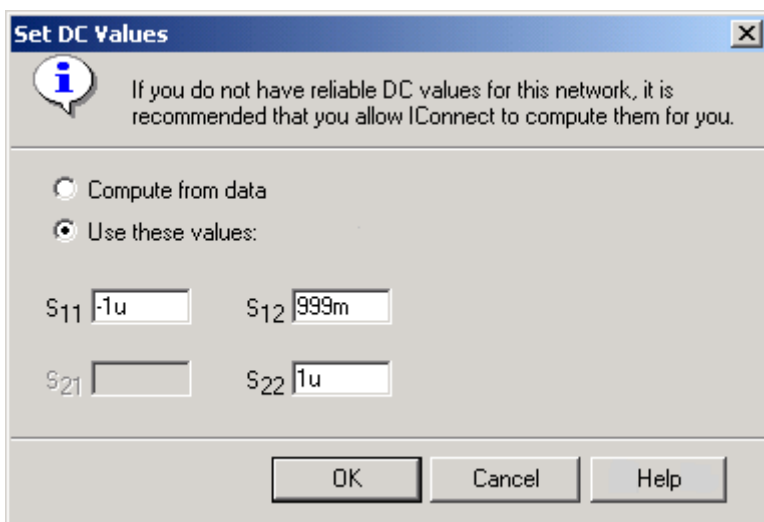


Figure 5-24: Set DC Values dialog.

If you click **OK** from this dialog, the Touchstone file will be reloaded into IConnect. If the **Compute from data** button was clicked, IConnect will perform the extrapolation to DC on its own. If the **Use these values** button was checked, IConnect will not do its own extrapolation, but will rely on the values supplied. Remember, if you do not know the values, you should allow IConnect to perform its own extrapolation.

Note: If you know the DC values, you may do well to enter them in the Touchstone file itself. However, remember that IConnect looks for evenly sampled frequency-domain data, so your sampling scheme may not support inclusion of the DC response in the Touchstone file.

Truncate

When IConnect transforms your frequency-domain measurements into TDR measurements, it will automatically truncate your waveforms if there is a long stretch of steady state behavior after the "interesting" part of the waveforms. However, in certain cases, in particular when IConnect has to extrapolate several data points in the low-frequency portion of the spectrum, it may not successfully be able to truncate the waveforms due to ambiguity in the low-frequency extrapolation. If it is clear to you, based on your physical knowledge of the device under test, that the waveforms can

be accurately truncated, you should do so, since it can greatly speed up the speed of your MeasureXtractor model extraction.

Note: It is very important that you do not over-truncate your waveforms. If you do, your extraction will not be accurate, because inaccuracies will probably be introduced when your data is checked for passivity.

It is recommended that you leave around 10% of the length of your waveform beyond where it reaches steady state. The same is true when taking TDR measurements.

The truncation is performed by clicking the **Truncate...** button, clicking on the **Truncate waveforms To** radio button, and entering the time value to which you would like to truncate the data set. If you have previously truncated your data and have decided you would rather not have it truncated, click the **Use automatically calculated waveform length** radio button, and IConnect will use its own algorithms to determine the truncation length. If you would like to truncate your data to a length longer than your currently selected truncation length, but shorter than IConnect's automatically determined length, you will have to first have the data automatically truncated, and then truncate it to your desired length. This is because IConnect will not allow you to "truncate" your data to a length longer than the current truncation length.

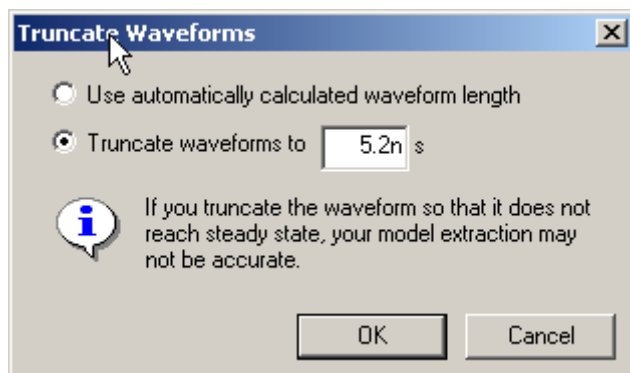


Figure 5-25: You can use the Truncate Waveforms dialog to truncate waveforms beyond where they are truncated by IConnect's automatic truncation capability. This should be done with care, based on your physical knowledge of the DUT.

Obtaining good measurements for MeasureXtractor

As with any numerical algorithm, the MeasureXtractor will perform better if supplied with good data. The MeasureXtractor is designed to be highly resistant to the noise that is inherent in all real-world measurements. Any typical level of noise will not present a problem.

The most important thing you can do to get good results is to ensure that your measurement is passive. Although this may seem trivial when measuring a passive device, the fact is that many interconnect devices are nearly lossless at some frequencies. Small amounts of error in your measurements, contributed by poor fixturing in TDR measurements and poor calibration in VNA measurements, can cause your measured data to be active. If this is the case, IConnect will attempt to passify your data, but may incur error in the process. This is the most common reason for inaccuracy in an extracted model, and can be avoided by carefully taking accurate measurements.

One detail that is often overlooked when taking TDT measurements is the importance of using matched cables, or alternating deskewing the measurement channels. Mismatch in your cables

causes skew between your measurement channels which can contaminate the delay of the signals you measure.

Another helpful tip is to not undersample your data. Undersampling introduces aliasing error which can make your measurements non-passive, leading to error when IConnect passifies your data.

Finally, when measuring frequency-domain data, accuracy will be improved by measuring to as low a frequency as possible. This assists IConnect in accurately extrapolating your data to DC. In the best case scenario, you would measure the DC values yourself.

5.11.2 Data Driven Model Tabs

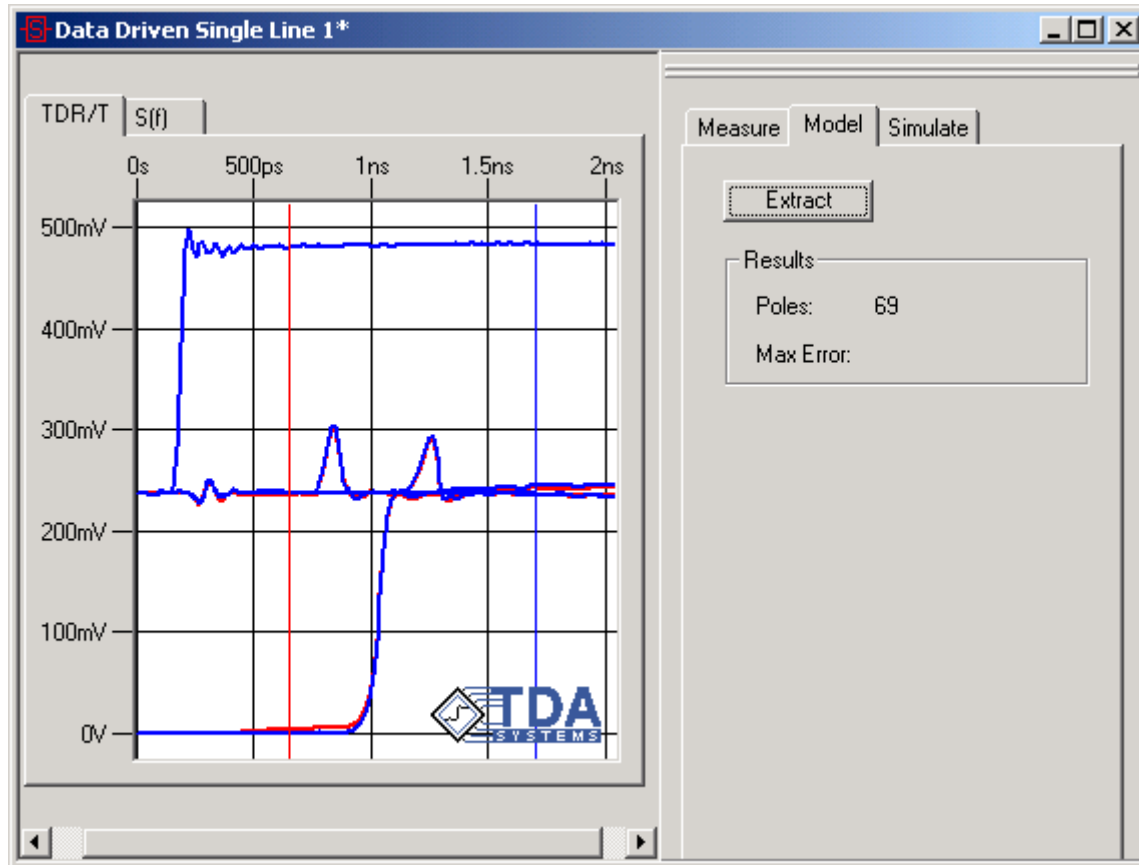


Figure 5-26: The Data Driven Model tab provides single click model extraction capability.

Extract

The simplest part of the data driven models is extraction, thanks to the IConnect MeasureXtractor. Once the data has been entered, you simply press the Extract button and wait for the extraction to complete. You may cancel the extraction at any time. Depending on the complexity of the device you are measuring, an extraction may take anywhere from 30 seconds to 30 minutes.

Poles

After the extraction is complete, IConnect will report how many poles it decided to use to match the behavior of the measured device.

5.11.3 Data Driven Simulate Tabs

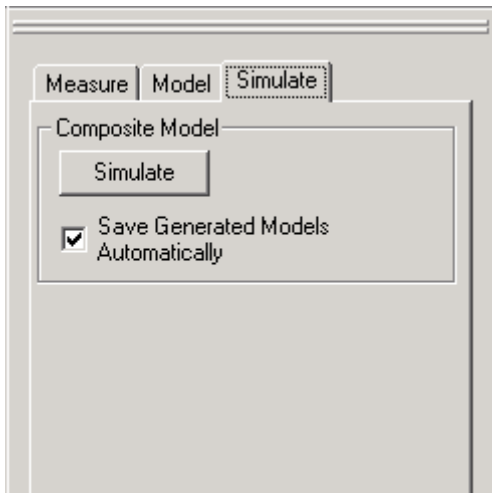


Figure 5-27: Data Driven Simulate tab.

When you press the **Simulate** button, IConnect will generate a composite model for each port of the DUT. Each of these models will be simulated, and the results will be displayed in the **TDR/T** waveform viewing tab.

If the **Save Generated Models Automatically** box is checked, as many models as possible will be saved to default names. If the box is unchecked, you will be asked in succession to save each of the submodel files and the composite model file prior to your initial simulation run.

Note: If files are saved automatically, they will overwrite pre-existing files with the same name. For this reason, users should be careful when working on more than one model in the same directory when automatic saving is being used.

5.12 Data Driven Single Line

The Data Driven Single Line is a two-port data driven model. Most of the description of this model is contained in [“Data Driven Models” on page 5-44](#). The current section will point out any additional information.

Note: [“MeasureXtractor™ Modeling Tutorial” on page 2-97](#) takes you through the measurement and modeling process for the Data Driven Single Line and Data Driven Coupled Lines models.

Data Driven Single Line Measure Tab

The Data Driven Single Line Measure tab guides you through the process of loading the data necessary for the MeasureXtractor to extract a model. For valuable in-depth information on this tab, see [“Data Driven Measure Tabs” on page 5-46](#).

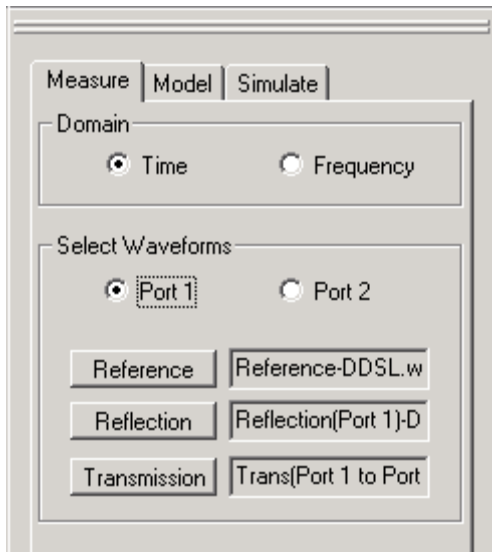


Figure 5-28: Data Driven Single Line Measure tab in time-domain mode.

For port 1, you will need to measure a reference, reflection, and transmission waveform. The reference waveform is simply a measurement of your cable or fixture at port 1 with your DUT removed from the system. Reflection at port 1 is a TDR measurement at port 1 with port 2 terminated in a 50Ω load. This load may be a 50Ω resistance, or more likely it will be a 50Ω cable terminated in channel 2 of your sampling scope. Transmission at port 1 is a TDT measurement from port 1 to port 2.

For port 2, you will need to measure reference and reflection waveforms. The reference waveform is a measurement of your cable or fixture at port 2 with your DUT removed from the system. Reflection at port 2 is a TDR measurement at port 2 with port 1 terminated in a 50Ω load.

Data Driven Single Line Model Tab

[“Data Driven Model Tabs” on page 5-50](#) contains a complete description of the Data Driven Single Line model tab.

Data Driven Single Line Simulate Tab

[“Data Driven Simulate Tabs”](#) on page 5-51 contains a complete description of the Data Driven Single Line Simulate tab.

5.13 Data Driven Coupled Lines

The Data Driven Coupled Lines is a four-port data driven model. Most of the description of this model is contained in [“Data Driven Models” on page 5-44](#). The current section will point out any additional information.

Note: [“MeasureXtractor™ Modeling Tutorial” on page 2-97](#) takes you through the measurement and modeling process for the Data Driven Single Line and Data Driven Coupled Lines models.

Data Driven Coupled Lines Measure Tab

The Data Driven Coupled Lines measure tab guides you through the process of loading the data necessary for the MeasureXtractor to extract a model. For valuable in-depth information on this tab, see [“Data Driven Measure Tabs” on page 5-46](#).

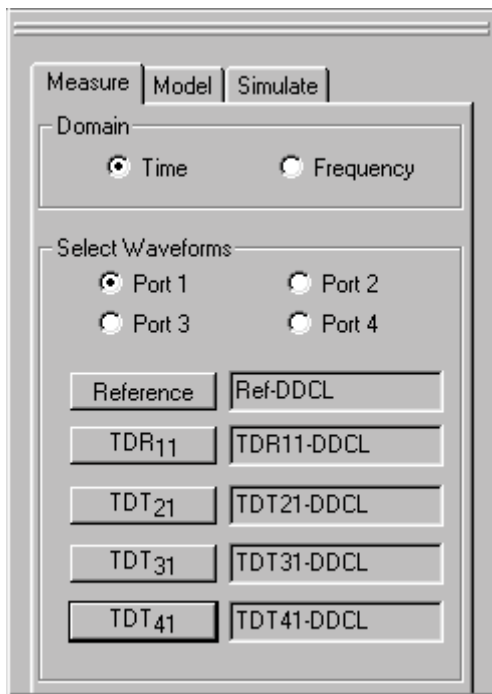


Figure 5-29: Data Driven Coupled Lines Measure tab.

You will need to measure a reference waveform at each of the ports. This is accomplished by removing the DUT from the cable or fixture at that port and measuring an open-circuit waveform. You will also need to measure a reflection waveform at each port. Finally, you will need to measure a varying number of TDT waveforms between the port in question and the other ports. The following table illustrates the excitation and termination scheme used for each measurement:

Waveform	TDR Step	Measurements	Termination
Reference 1	Port 1	Port 1	DUT Removed
Reference 2	Port 2	Port 2	DUT Removed

Waveform	TDR Step	Measurements	Termination
Reference 3	Port 3	Port 3	DUT Removed
Reference 4	Port 4	Port 4	DUT Removed
TDR11	Port 1	Port 1	50Ω
TDT21	Port 1	Port 2	50Ω
TDT31	Port 1	Port 3	50Ω
TDT41	Port 1	Port 4	50Ω
TDR22	Port 2	Port 2	50Ω
TDT32	Port 2	Port 3	50Ω
TDT42	Port 2	Port 4	50Ω
TDR33	Port 3	Port 3	50Ω
TDT43	Port 3	Port 4	50Ω
TDR44	Port 4	Port 4	50Ω

Note: Your 50Ω load may be a 50Ω terminating resistance, or a 50Ω cable terminated in channel 2 of your sampling scope.

Data Driven Coupled Lines Model Tab

“[Data Driven Model Tabs](#)” on page 5-50 contains a complete description of the Data Driven Coupled Lines model tab.

Data Driven Coupled Lines Simulate Tab

“[Data Driven Simulate Tabs](#)” on page 5-51 contains a complete description of the Data Driven Coupled Lines Simulate tab.

5.14 Generic 2-port

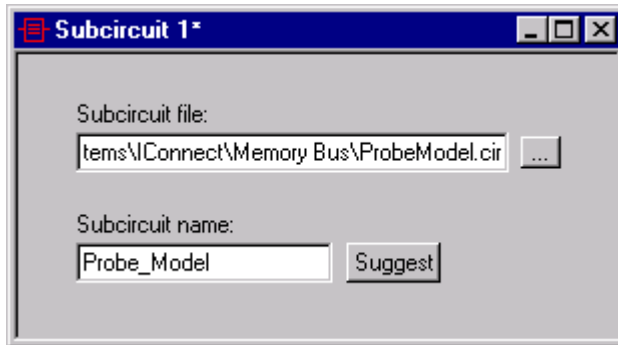
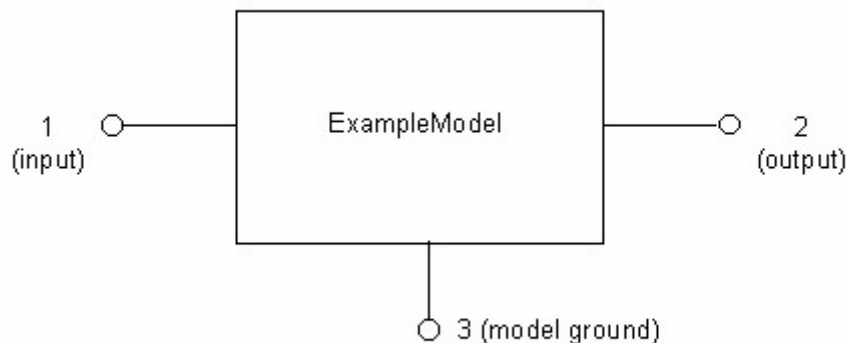


Figure 5-30: Two-port subcircuit model.

The two-port subcircuit model is a powerful way to incorporate the results of other SPICE-based modeling tools into the IConnect measurement-modeling-verification framework. Using the two-port subcircuit, any SPICE model with an input node, an output node, and a ground node may be included as part of a composite model.

Note: Before including an arbitrary two-port netlist using the two-port subcircuit model, ensure that the subcircuit node definitions are according to the requirements below. All two-port subcircuit node definitions must be in the following order: input node, output node, and model ground node. The node ordering convention is shown in the following diagram and corresponding netlist line:



Corresponding SPICE subcircuit line:

```
.subckt ExampleModel 1 2 3
```

IConnect assumes that the external nodes of the subcircuit you are including are defined in the above order. If they are not, your subsequent simulation results will be incorrect.

To incorporate an existing arbitrary two-port model, formatted according to the guidelines given above, into an IConnect simulation, click on the two-port subcircuit button on the model toolbar, or drag a two-port subcircuit block from the model toolbar into an existing composite model and double-click on the empty block. Either action will bring up an empty two-port subcircuit window.

The text box labeled **Subcircuit file** is used to enter the name of the existing arbitrary two-port model file. Click on the ... next to the text box to browse your drives for the desired file. Once you have chosen the subcircuit file, IConnect will attempt to extract the subcircuit name from your netlist. If your netlist contains multiple subcircuits, you may have to enter the name manually.

Note: You may view or edit the arbitrary subcircuit netlist at any time using the **View/Edit Subcircuit Netlist** menu command from the model context menu, accessed by right-clicking on the model window.

After saving the model, you may include it in any composite model where any other two-port model would be appropriate.

5.15 Termination

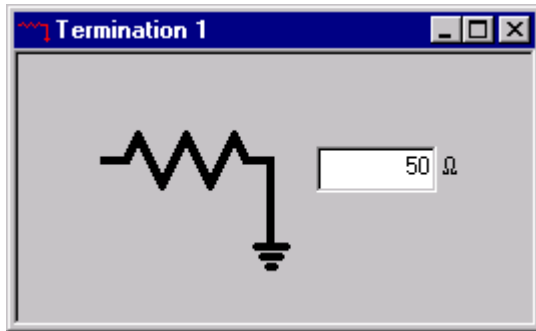


Figure 5-31: Termination model editor.

The termination model is meant to replicate the passive termination conditions of the DUT, and is accessible either from the **Model/Termination** menu command or from the model toolbar. The only model parameter is resistance, since the model consists of a resistor to ground. To emulate an open circuit, a resistance which is large compared to the characteristic impedance should be used, while for a short circuit, a resistance which is small compared to the characteristic impedance of the measurement system should be used. For a 50Ω system, for example, a short circuit might be $1\text{m}\Omega$ whereas an open circuit might be $5\text{k}\Omega$.

5.16 Composite Model

The composite model viewer is the means of verifying your model's behavior against your measured data. It may also be used to cascade several models into a single model.

Note: It is no longer necessary to create a composite model from scratch every time you want to verify an extracted model, as it was in previous versions of IConnect. IConnect's automated Measure/Model/Simulate capability does most composite model creation and population for you.

For validating most single models extracted in IConnect, you can allow IConnect to automatically create and populate the composite model for you, and even launch the simulation. Much of this chapter is devoted to manual creation of composite models. This is typically necessary when you need to cascade several models together to model a communications channel or backplane, for example.

Creating a composite model

A new composite model is manually created by invoking the **File/New...** menu command either from the main menu or from the standard toolbar, and choosing **Composite Model Viewer** from the options presented. This creates an empty composite model.

Composite Model Usage

Once you have manually created a composite model, you will usually begin by populating it with submodel blocks as described in [“Composite Model Schematic Viewer” on page 5-61](#). The next step is to add measured waveforms into the **TDR/T** tab, as described in [“Composite Model Measure Tab” on page 5-60](#), for comparison against the results of subsequent simulations. If you do not add waveforms, you will then need to manually set the simulation timing before launching the simulation, which is covered in [“Composite Model Simulate Tab” on page 5-63](#).

5.16.1 Composite Model Measure Tab

The Composite Model Measure tab is used to load waveforms into the IConnect TDR/T viewer. Since the results of the composite model simulation are also displayed in the TDR/T viewer, it can be used to compare the results of your simulation against the measured waveforms.

The Measure tab also has a waveform legend, which may be used just like the waveform legends in the IConnect standalone waveform viewers to operate on the waveforms in the TDR/T tab. Operations might include waveform selection, deletion, cursor readout, etc. For more information on managing and viewing waveforms, see [“Managing and Viewing Waveforms” on page 7-1](#).

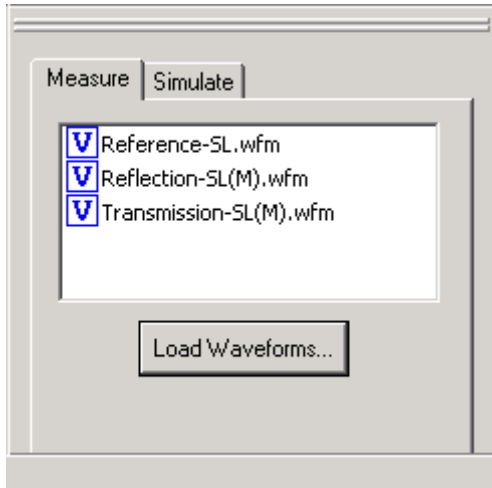


Figure 5-32: The Composite Model Measure tab.

5.16.2 Composite Model Schematic Viewer

The Composite Model Schematic Viewer is accessed by clicking the **Schematic** tab in an open composite model. You may use it to concatenate multiple IConnect models together, and to connect them to source and termination models.

Adding submodels

Submodels are added to the composite model viewer by dragging them from the model toolbar, shown in Figure 5-33. If you click on the toolbar, an empty model is created which is not associated with any composite model. However, if you hold down the mouse button and drag over the composite model window, you will be able to drop a model of that type into one of the slots of the composite model by releasing the mouse button, as shown in Figure 5-34. A model may not be dropped into a position that is not appropriate for that type of model. The model that you drop in is initially empty.

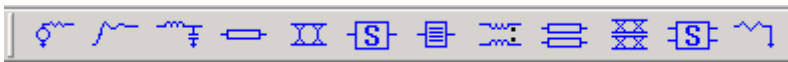


Figure 5-33: Model toolbar.

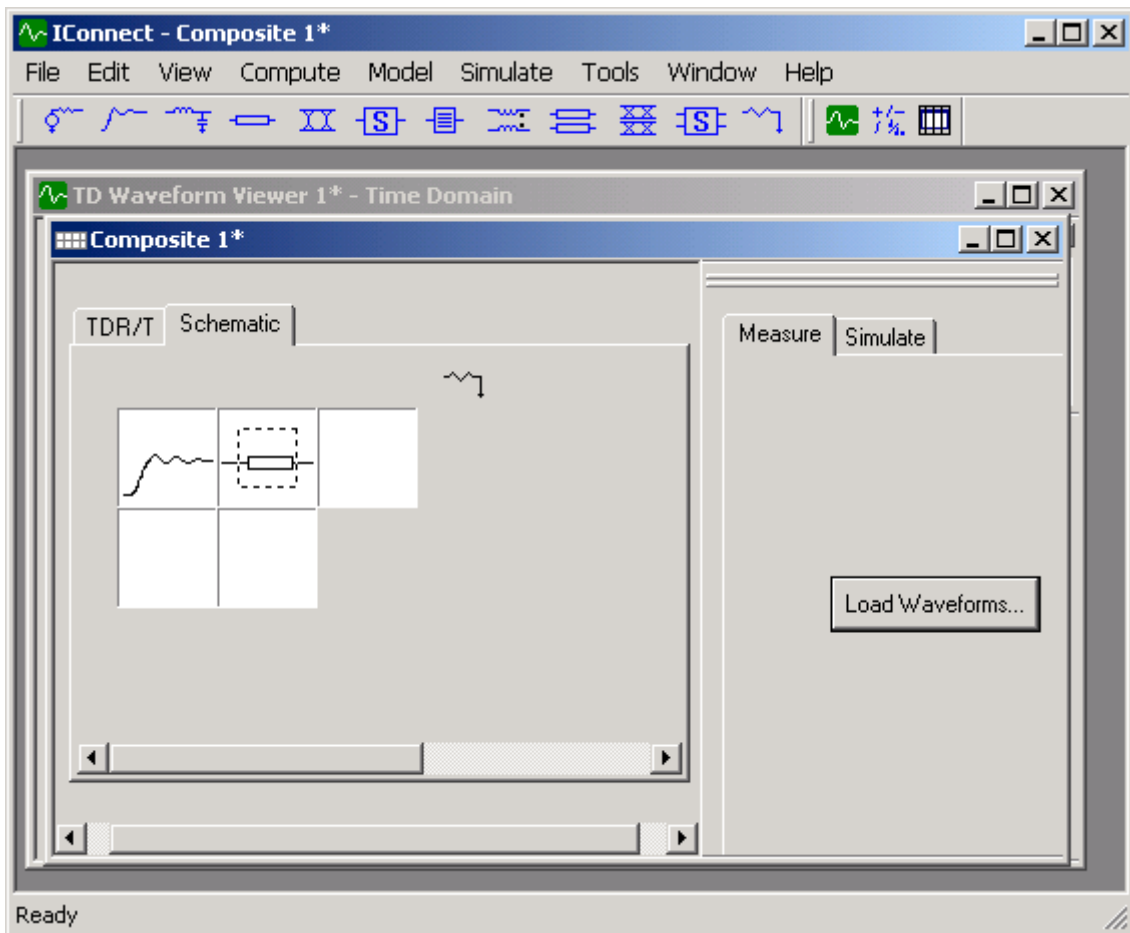


Figure 5-34: Dragging a resistive termination into the composite model viewer.

Models

To load a previously saved model from disk, insert an empty submodel of the type you want to load into the position into which you would like to load it. Right-click on the empty model and use the **Load...** command to choose the model file from disk.

To insert a submodel which is already open within IConnect into a composite model, begin by inserting an empty model of the type you want to load into the position into which you would like to load it. Right-click on the empty model and use the **Attach** menu item which will allow you to choose the model from a list of appropriate open models.

Note: The IConnect framework does a certain amount of design rule checking to ensure that the correct type of model is placed in the correct slot. For this reason, certain types of models will not be allowed in positions that would cause a simulation to subsequently fail due to hanging nodes or other problems.

Selecting submodels

A submodel may be selected by clicking on it. The selection may also be changed from the keyboard by using the arrow keys. If the <Shift> key is held down in conjunction with the right or left arrow key, the selection box moves all the way to either end of the model.

Editing submodels

A submodel may be edited by double-clicking on it, or, if the model is selected, by pressing <Enter>. See the Model Extraction help section for model-specific editing procedures.

Deleting submodels

The selected submodel is deleted by pressing the <Delete> key. It should be emphasized that this does not delete the model from your hard drive, but simply removes it from the composite model.

Viewing the submodel netlist

The submodel netlist may be viewed (and yes, even edited) by right-clicking on the appropriate submodel block and selecting **View Netlist**.

5.16.3 Composite Model Simulate Tab

The Composite Model **Simulate** tab allows you to launch a simulation, and automatically displays the simulation results in the TDR/T viewing window.

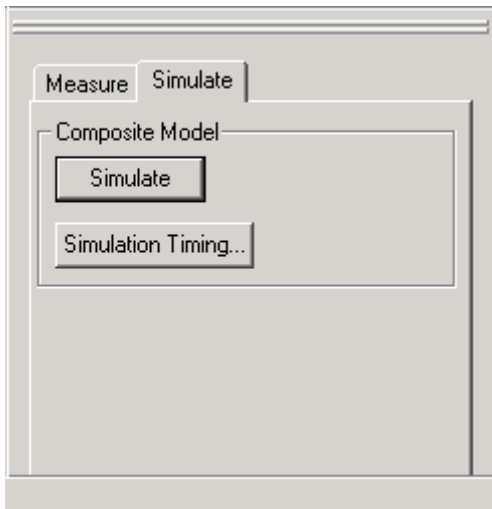


Figure 5-35: Composite Model Simulate tab.

Setting Simulation Timing

To simulate the composite model you have created, you must first ensure that the simulation timing has been set appropriately by pressing the **Simulation Timing** button. This brings up the **Simulation Timing** dialog. The timing setup option allows IConnect to automatically select the timing for comparison with the waveforms in the **TDR/T** viewing tab, or allows you to manually set the timing. If you choose the manual option, you will be required to enter a stop time for the data retrieved from the simulation, as well as the number of data points you want returned from the simulation.

Note: As long as there is a waveform loaded in the TDR/T viewing tab, it is not necessary for you to set the timing manually, since IConnect will do it for you.

Saving the Composite Model

To save the composite model, right click on the composite model window and select **Save** from the resulting drop-down menu, or use the <Ctrl+s> shortcut. You may choose the syntax with which you want to save your model from the drop-down list on the **Save** dialog. IConnect will inform you if various submodels included in your composite models have been saved using different simulator syntaxes. For further information on this subject, see [“Model Verification with Multiple Simulators” on page 6-6](#).

Running the Simulation

After saving the composite model, invoke the simulation by right clicking on the composite model window and choosing **Run** from the resultant drop-down menu, as shown in Figure 5-36. You may also simulate any open composite model by choosing the **Simulate/Run** submenu from the main IConnect menu, which allows you to choose from a list of open composite models. If a model has not been saved, you will be prompted to save it before the simulation will run.

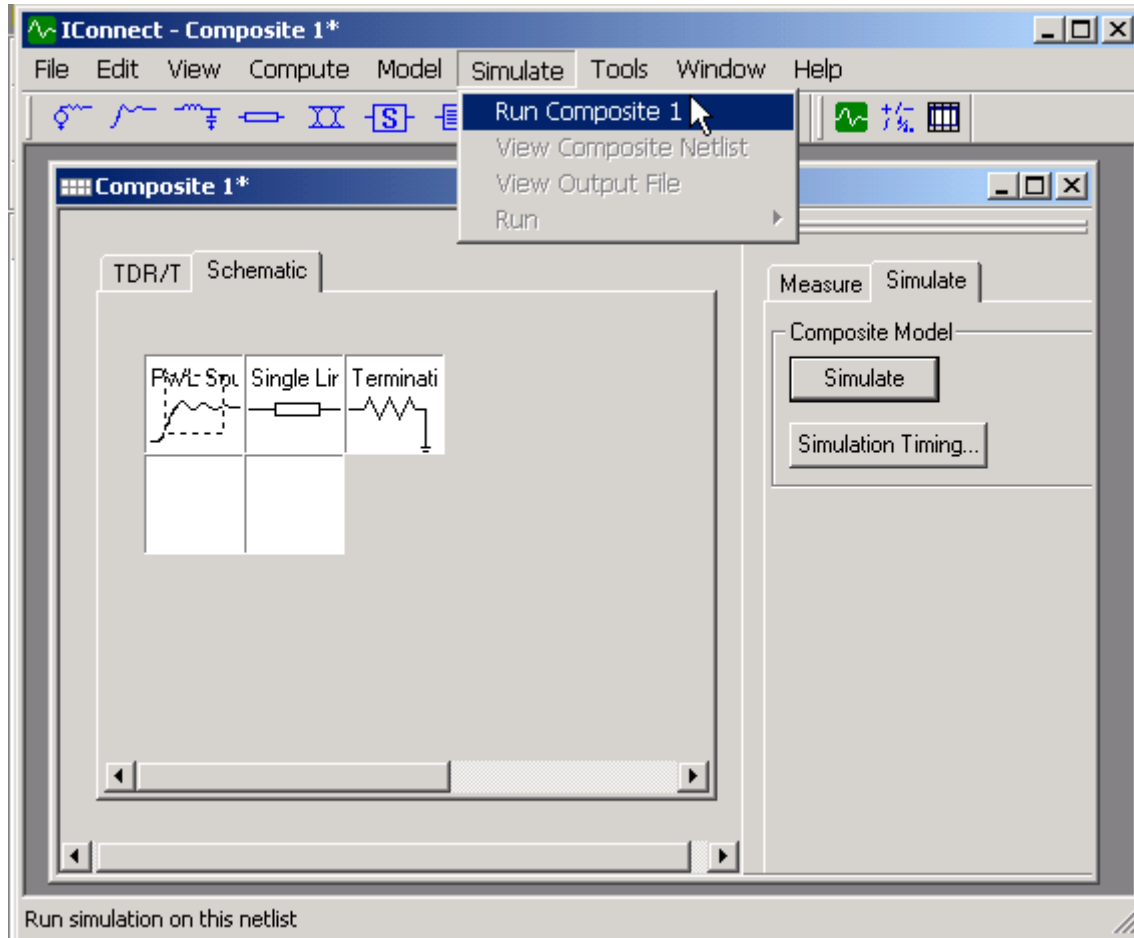


Figure 5-36: Simulating a composite model. Selecting the composite model and using the menu command launches the simulation.

Note: Composite models containing either empty blocks or unsaved submodels cannot be simulated.

If the Composite Model has been saved using a different simulator than the currently selected simulator, IConnect will inform you. For more information, see [“Model Verification with Multiple Simulators”](#) on page 6-6.

After a successful simulation, the resultant waveforms will be placed in the waveform viewer that you selected. If the simulation fails due to corrupted netlists or missing model files, it will exit with a warning. You may choose the **View/Output File** menu command to examine the output listing to determine the source of the error, and you may choose the **Simulate/View Composite Netlist** command to inspect the composite model netlist. You may also want to refer to [“Troubleshooting”](#) on page A-1. TDA Systems will gladly give you any further assistance that you might need.

5.17 Eye Diagram Computation and Mask Testing

Note: “Eye Diagram Tutorial” on page 2-19 contains step-by-step instructions on how to quickly generate an accurate eye diagram.

IConnect has the ability to generate accurate eye diagrams based on TDR measurement data or on model response so that you can predict deterministic jitter and intersymbol interference in your transmission line or communication channel. You may also predict eye closure effects caused by crosstalk from neighboring line pairs. IConnect also assists you in creating and displaying a mask to test the compliance of your communication channel.

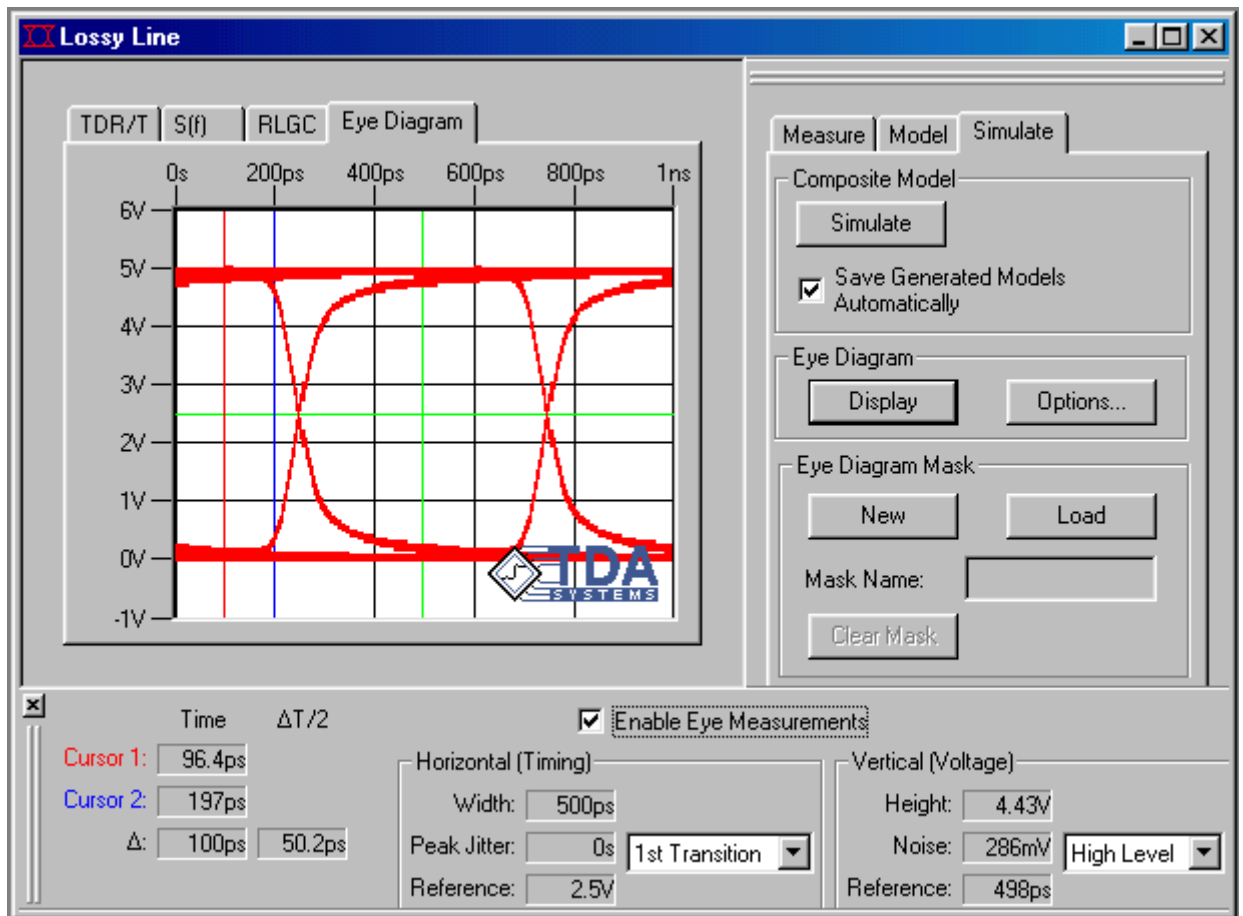


Figure 5-37: Eye diagram computed in the IConnect Lossy Line model.

To compute an accurate eye diagram, you measure your data (or in some cases, provide model parameters), set up the DUT and aggressor options, and press the **Display** button. You may also create a mask for testing your eye against known standards. Each step is outlined in the following sections.

5.17.1 Waveforms for Eye Diagram Computation

In order to compute an eye diagram, you must supply the proper waveforms or model parameters. The waveform data is supplied via the Lossy Line and Symmetric Coupled Lossy Lines **Measure** tabs.

Single Ended Eye Diagram

If you're computing a single-ended eye diagram, you must enter your waveform data in the Lossy Line model **Measure** tab. If you are computing the eye directly from your measured data without extracting a model, you must use the termination type **Matched**, and you must enter at least a reference waveform and a transmission waveform. If you use termination type **Open**, you must create a model from your data prior to creating an eye. For more information on how to enter data into the Lossy Line **Measure** tab, see "[Lossy Line Measure Tab](#)" on page 5-29. Note that the single ended eye diagram capability does not support crosstalk analysis.

Differential Eye Diagram DUT (Victim) Waveforms

If you are computing a differential eye diagram, you must enter your DUT (or victim) waveform data in the Symmetric Coupled Lossy Lines **Measure** tab. If you are computing the eye directly from your measured data without extracting a model, you must use the termination type **Matched**, and you must enter at least a reference waveform and an odd transmission waveform. If you use termination type **Open**, you must create a model from your data prior to creating an eye. For more information on how to enter your DUT data into the Symmetric Coupled Lossy Lines **Measure** tab, see "[Symmetric Coupled Lossy Lines Measure Tab](#)" on page 5-37.

Differential Eye Diagram Aggressor Waveforms

If you are including crosstalk effects in your eye diagram, you will need to enter aggressor waveform information in the Symmetric Coupled Lossy Lines **Measure** tab for the nearby pairs generating the crosstalk. Begin by clicking on the **Use Aggressors** button on the **Measure** tab. A set of eight new buttons will appear, labeled **Aggressor 1** through **Aggressor 8**, allowing you to incorporate crosstalk effects from up to eight nearby line pairs. Clicking on a button opens a window that allows you to enter the appropriate aggressor waveforms.

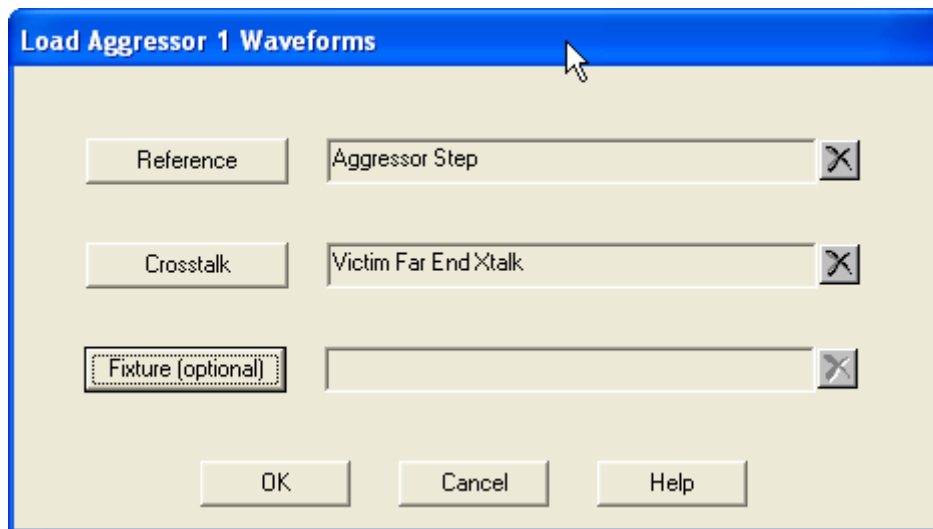


Figure 5-38: Aggressor waveform entry window.

The **Reference** waveform is a TDR waveform measured with the fixture still connected but the aggressor removed from the system. Its purpose is to measure the step waveform used to excite the aggressor.

The **Crosstalk** waveform is typically measured at the same end of the victim at which the transmission waveform was measured. Its purpose is to measure the crosstalk that the aggressor causes on the victim. In specific situations, the aggressor waveform may also be measured at the opposite end of the victim from which the transmission waveform was measured. An example would be if you have reason to believe, due to symmetry arguments, that exciting end A of the aggressor and measuring crosstalk at end B of the victim is the same as exciting end B of the aggressor and measuring crosstalk at end A of the victim. Such symmetry arguments may be useful in applications where you are limited as to which ends of which lines are easily available for measurement.

The optional **Fixture** waveform is a measurement of the crosstalk induced in the victim when the aggressor is replaced by a matched termination. Its purpose is to measure the crosstalk induced in the victim by the fixture alone. It is measured at the same end of the victim as its associated Crosstalk waveform was measured. It is only necessary to supply a fixture waveform if there is significant crosstalk in your fixture compared to the crosstalk between the aggressor and victim.

Additional information on measuring aggressor waveforms can be found in [“Symmetric Coupled Lossy Lines Measure Tab” on page 5-37](#).

5.17.2 Eye Diagram Options

Part of eye diagram computation is setting up the eye diagram options, such as input sequence, data rate and risetime. If you're computing an eye diagram without any crosstalk effects, you only need to supply options for the DUT (victim) line. If you are including crosstalk effects in your eye diagram, you need to supply options for each aggressor line as well.

Where to Set Eye Diagram Options

If you are computing an eye diagram for a single-ended system, you may access the eye diagram options via the Lossy Line **Simulate** tab, by pressing the **Options** button in the **Eye Diagram** group. If you are computing an eye diagram for a differential system, you may access the eye diagram options via the Symmetric Coupled Lossy Line **Simulate** tab.

If your differential eye diagram does *not* include crosstalk effects (that is, you have *not* checked the **Use Aggressors** box on the Symmetric Coupled Lossy Line **Measure** tab), then you may set the DUT eye diagram options by pressing the **Options** button in the **Eye Diagram** group on the **Simulate** tab. If your differential eye diagram *does* include crosstalk effects (that is, you *did* check the **Use Aggressors** box on the Symmetric Coupled Lossy Line **Measure** tab) then the **Options** button has been replaced by a button labeled **DUT** and eight buttons labeled **Aggressor**, as shown in Figure 5-39. The eye diagram options for the DUT are set by clicking the **DUT** button, and the options for each aggressor are set by clicking the appropriate **Aggressor** button.

Note: Some options in the eye diagram options window apply only to the DUT (victim) line, and will be disabled when setting options for the aggressor lines. Some eye diagram options apply only to aggressor lines, and will be disabled when setting options for the DUT (victim) line.

Sequence Settings

Sequence Type is either pseudo-random, K28.5, CJTPAT, or custom. If you choose to use a pseudo-random bit sequence, you must choose a length for it. If you choose a custom sequence, you will be allowed to enter a pattern in the text box labeled **Pattern**. The pattern must consist of ones and zeros. You may enter a repeat count in the **Repeat** box, although regardless of the repeat count, IConnect ensures that the sequence has been repeated enough times to see all bit sequences.

Step Voltages

Step Voltages allows you to enter the minimum and maximum input voltages used to generate the eye diagram.

Compute Response From

The response may be computed from either measured data, or from the model parameters entered in the text boxes, depending on whether **Measurement** or **Model** is selected in the **Compute Response From** group. Note that this option is disabled for aggressor lines, since crosstalk responses may only be computed directly from measured data.

Timing Info

Data Rate is the number of bits which are clocked per second. You may alternately enter this information as **Bit Width** if preferable.

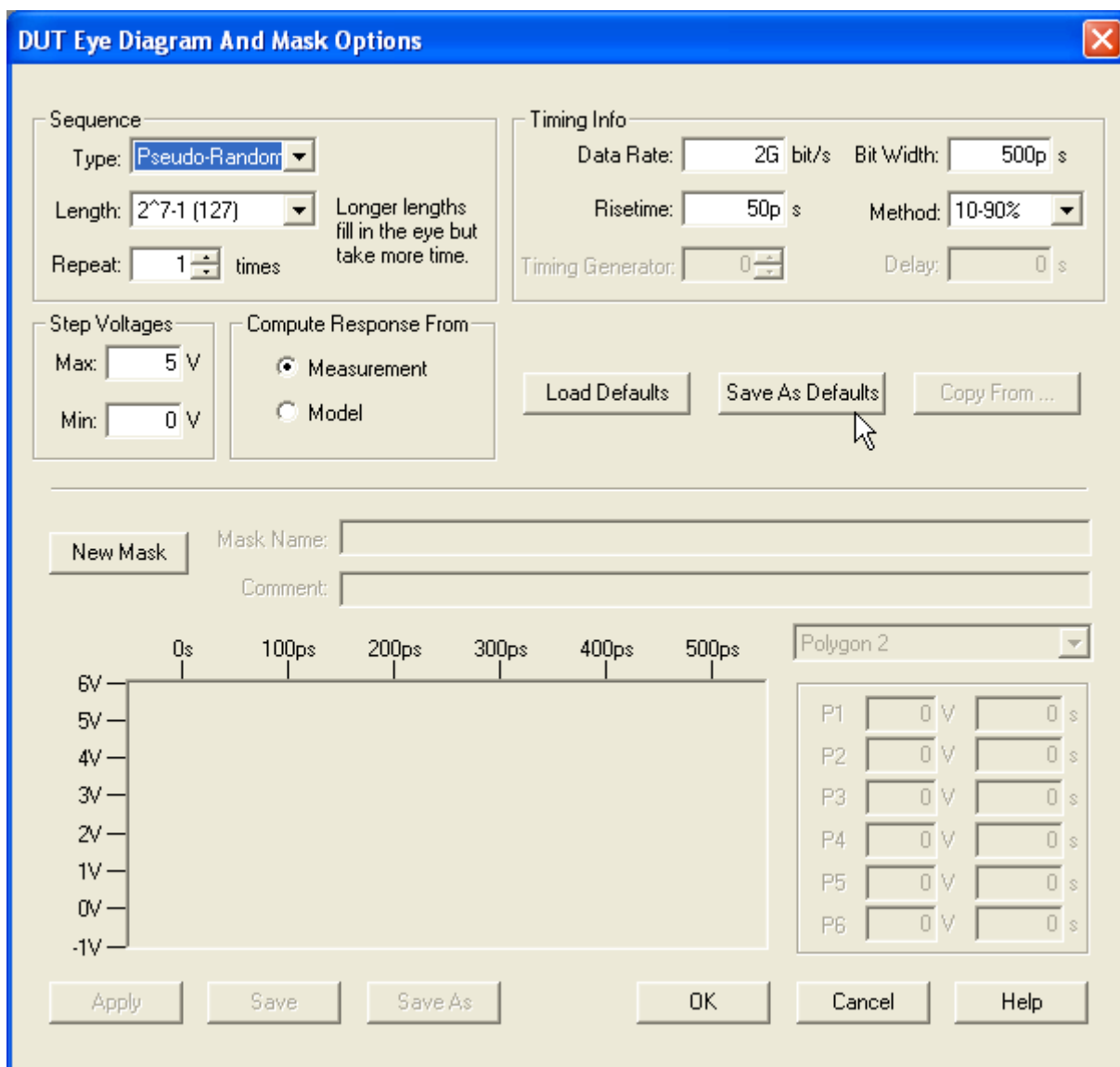


Figure 5-39: Eye diagram DUT options dialog.

Risetime is the risetime of a Gaussian step used to excite the model. In the **Method** dropdown list, you may choose either 10-90% or 20-80% to define your risetime.

Timing Generator allows you to associate multiple signals with a single timing generator. This means that the signals will be assumed to be clocking synchronously with each other for the purposes of computing the eye. If two signals are on different timing generators, even though they are set to the same frequency, they will be treated as asynchronous. Asynchronous signals are assumed to have different frequencies (even if only slightly different), causing them to drift apart over time, resulting in a worst-case eye closure. A set of signals associated with the same timing generator must have data rates which are integral multiples of a base frequency. For important information on the relationship between the data rates of synchronous signals, see [“Adding Crosstalk to an Eye Diagram” on page 5-71](#).

While there are nine timing generators you may choose from for aggressors, the DUT (victim) is required to use timing generator zero.

Note: If you are having trouble assigning compatible data rates to synchronous signals, see [“Adding Crosstalk to an Eye Diagram” on page 5-71](#).

Delay allows you to provide a relative delay between an aggressor signal and the DUT signal. Implicitly, this value can be used to also effect relative delays between aggressors within a synchronous group (that is, a group associated with a single timing generator). Since all signal delays are relative to the DUT, the DUT delay is required to be zero.

Default Options

If you have made changes to the eye diagram options and would like to return to the default settings, click the **Load Defaults** button. To save your current eye diagram options settings as your own default settings, click the **Save as Defaults** button. In the future when you create a new eye diagram, these will be the default settings.

Copy From

If you click **Copy From...** IConnect will allow you to copy the options associated with the signal on one of the other lines to the signal associated with this line, to allow you to rapidly configure a set of similar lines. If you are entering the options describing the signal on a DUT (victim) line, **Copy From...** is disabled.

Mask Creation

The **New Mask** button allows you to create a new mask if one is not already loaded. After clicking it, follow the instructions in [“Mask Testing” on page 5-77](#) to edit the mask.

5.17.3 Adding Crosstalk to an Eye Diagram

In some cases, crosstalk from signals traveling on adjacent line pairs can cause additional eye diagram closure. IConnect allows you to predict the amount of closure that would occur based on measurements of the victim and the aggressor. In order to accurately compute the effects of crosstalk on your eye diagram, you must take crosstalk measurements and set up the proper signals to excite the victim and each aggressor.

Note: The [“Eye Diagram Tutorial” on page 2-19](#) contains an example of how to compute an eye diagram which accurately incorporates the effects of crosstalk.

Crosstalk Measurements

How to take the proper set of crosstalk measurements for eye diagram computation is explained in [“Waveforms for Eye Diagram Computation” on page 5-66](#).

Setting Up Signals on Aggressor Lines

[“Eye Diagram Options” on page 5-68](#) describes how to set up eye diagram options for both the victim (DUT) and aggressors. This includes information on how to provide timing information on the victim (DUT) signal and each of the aggressor signals. However, care must be taken to ensure that the victim and aggressor signals are compatible in terms of data rates and synchronization. In the following subsection, we discuss the how to create a compatible set of signals to drive the victim and aggressors.

Signal Synchronization

Synchronization refers to whether the signals on various lines are switching based the same clock source or different clock sources. Whether or not various signals are synchronized with each other will affect the amount of eye closure observed. IConnect allows you to provide synchronization information in the eye diagram options by identifying each signal with a timing generator. Signals derived from the same timing generator will be synchronized with each other; signals derived from different timing generators will not be.

Note: Even though signals may be assigned identical data rates, if they are derived from different timing generators, they will not be considered synchronous.

Compatible Data Rates for Synchronous Signals

Two signals driven by the same timing generator must have data rates that are simple multiples of some base rate. For example, a single timing generator can drive one signal at 2 Gbit and another at 4 Gbit or perhaps at 2 Gbit and 3 Gbit, but it can not drive signals at 2 Gbit and 2.1 Gbit. IConnect requires that all pairs of signals driven by the same timing generator have data rates that are in the ratio of integers no larger than four, to within four significant digits.

Example 1: The DUT signal runs at 3 Gbit and an aggressor signal at 9 Gbit. These data rates are in the ratio 1:3 which IConnect considers synchronous, so it is acceptable for them to be specified as being driven by the same timing generator.

Example 2: Aggressor signals 1, 2 and 3 run at 1 Gbit, 3 Gbit and 9 Gbit, respectively, and are specified as being driven by the same timing generator. The data rates of aggressor signals 1 and 2 are in the ratio 1:3 and similarly the data rates of aggressor signals 2 and 3 (3 Gbit and 9 Gbit)

are also in the ratio 1:3. However, the data rates of aggressor signals 1 and 3 are in the ratio 1:9. Because this ratio involves an integer larger than 4, IConnect reports an error.

Example 3: One signal runs at 1 Gbit and another at exactly 2/3 that rate. You may specify the rate of the second signal as 0.6666 Gbit. Even though 0.6666 is an approximation to 2/3, IConnect accepts the ratio 1:0.6666 as the ratio 3:2, since it only compares the data rates to four digits.

Synchronization Summary Dialog

Clicking the **Synchronization Summary** button on the **Simulate** tab causes IConnect to display the **Synchronization Summary** dialog. This window summarizes the data rate and timing generator associated with the signal on each line, and groups the lines by timing generator. This information can be very useful in determining the source for any synchronization errors.

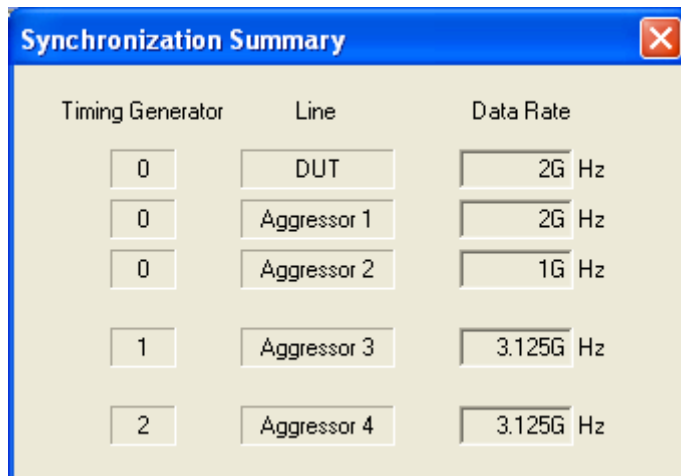


Figure 5-40: Synchronization Summary dialog

Troubleshooting Synchronization Errors

When you either click the **Display** button to calculate and display the eye diagram, or click **Sync Summary** to view the synchronization of the various signals, IConnect validates the various signal data rates that were entered for each timing generator.

When a synchronization error (such as specifying 2 Gbit and 2.1 Gbit for the same timing generator) is detected after clicking the **Display** button, IConnect reports an error, as shown in Figure 5-41. If you click **OK**, IConnect will assign each incompatible signal associated with a timing generator to its own timing generator. Each incompatible signal will essentially become asynchronous with every other signal, leading to worst case eye diagram closure.

You may also choose to cancel the eye diagram computation and find and change the specified timing generators and/or data rates to resolve the compatibility problems. The Sync Summary dialog can help you pinpoint the source for the synchronization error, and the compatibility information (including the three examples) earlier in this section should help you determine the appropriate corrective action. In most cases this will lead to results which are closer to what you originally intended.

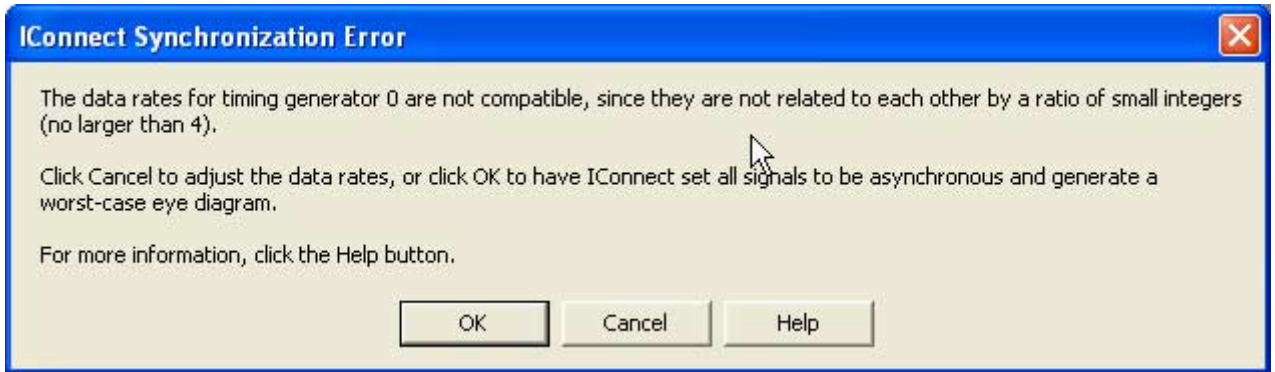


Figure 5-41: Synchronization Error dialog.

5.17.4 Eye Measurements and Analysis

IConnect provides automated eye diagram height, width, jitter and amplitude noise measurements for analyzing your eye diagram. You also have the ability to zoom in and recompute a portion of the eye diagram a higher resolution.

Cursor Measurements

To view the eye diagram measurement readouts, click the **Enable Eye Measurements** check box, as shown in Figure 5-42. Two green reference cursors - one horizontal and one vertical - appear in the viewing area. As you move each reference cursor, the related measurements are updated.

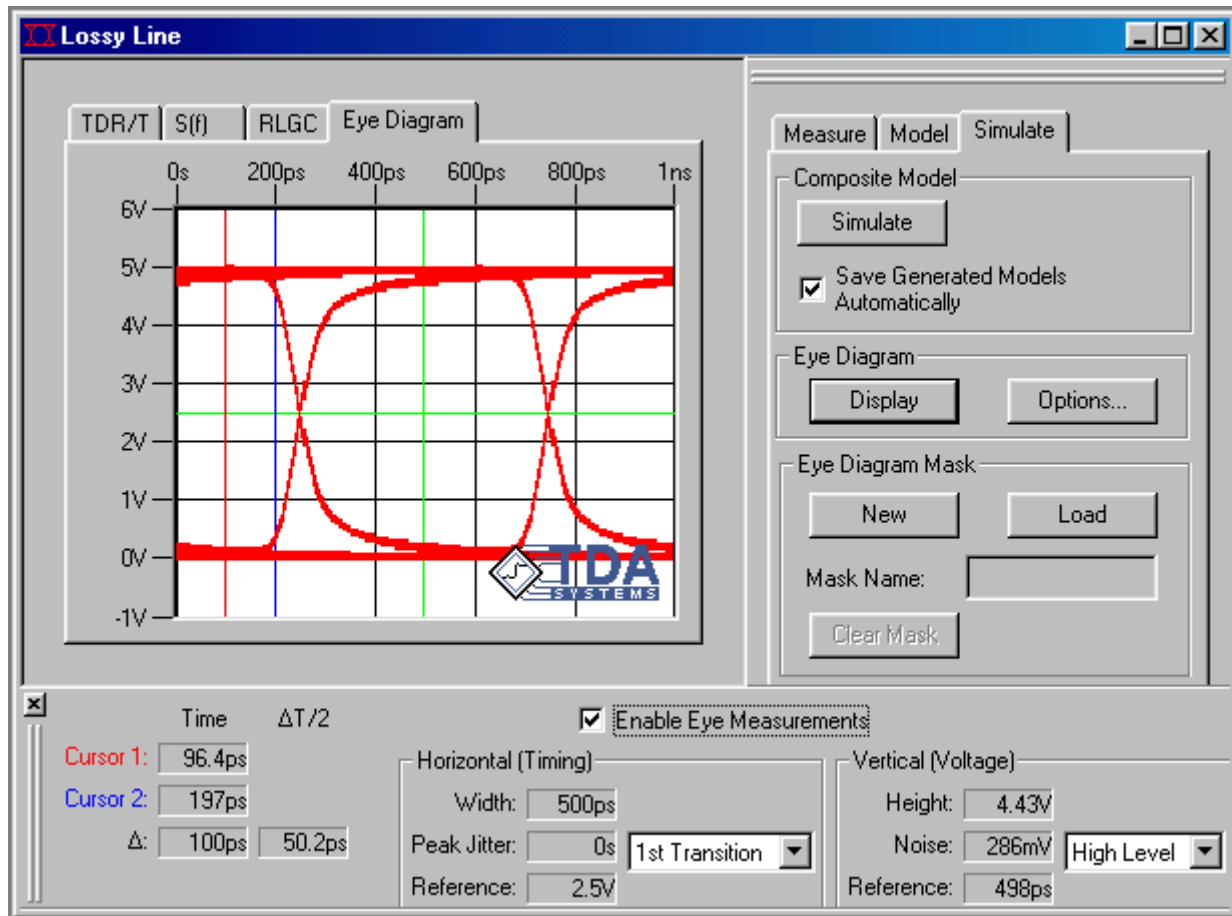


Figure 5-42: Eye measurements

Horizontal (Timing) Measurements

Width measures the width of the eye opening at the reference level determined by the horizontal cursor.

Jitter may be measured at either the first (left-hand) transition or the second (right-hand) transition at the reference level determined by the horizontal cursor. The measurement is from the right-hand side of one eye opening to the left-hand side of the adjacent eye opening, so the most relevant jitter measurement will be taken with the reference near the crossover point.

Vertical (Voltage) Measurements

Height measures the height of the eye opening at the reference time determined by the vertical cursor.

Amplitude Noise may be measured at either the upper rail (high level) or the lower rail (low level) of the eye diagram, at the reference time determined by the vertical cursor. The measurement is from the above (or below) the eye opening to the top (or bottom) of the eye opening.

Zooming in on the Eye Diagram

You can zoom in on areas in the eye diagram by left-clicking in the viewing area and drawing a zoom rectangle around the section you wish to inspect more closely.

Note: If your eye diagram contains crosstalk effects, you may only zoom a total of 4x in the x and y directions.

When zooming in on an eye generated with long data sequences, you will be asked to choose the zoom method as shown in Figure 5-43. If you choose **Continue** you will not have to wait for the eye to be recomputed, but you will be viewing the zoom area at less than full resolution. If you choose **Recompute**, the visible portion of the eye will be recomputed at higher resolution for you to view. However, you should note that the portion of the eye outside the zoom area is no longer available for viewing unless it is recomputed. Choosing **Resize** will return you to the eye diagram you originally viewed.

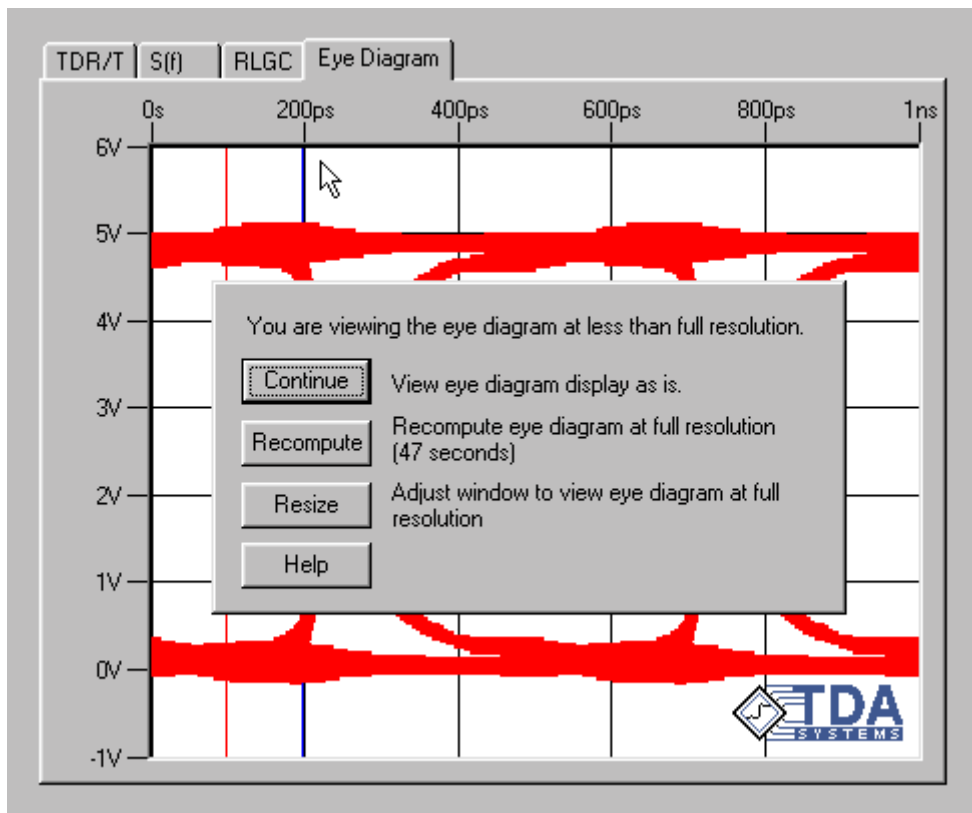


Figure 5-43: Choosing the appropriate zoom method.

Models

An estimated length of time for the recompute is displayed next to the **Recompute** button to help you determine which method is best. When recomputing the eye diagram, if the computation takes too long, you can use the **Cancel** button to cancel it.

If you choose to recompute the eye at higher resolution when you zoom in, the prior eye diagram data will be discarded, and only the visible portion will be computed at high resolution. For this reason, if you attempt to scroll an eye which is displayed at full resolution, or if you zoom to 100%, you will be given the same set of options for recomputing or resizing the eye in order to be able to view the portions which formerly were not visible.

5.17.5 Mask Testing

You may create a mask in IConnect to test the compliance of your communication channel with the relevant standards. Mask creation consists of manipulating the mask polygons, after which the mask is superposed over the eye diagram for compliance testing. The mask polygons may be graphically manipulated in the **Eye Diagram and Mask Options** dialog, and the mask may also be saved for later use with a different eye diagram.

Creating a New Mask

To create a new eye diagram mask, click the **New** button in the **Eye Diagram Mask** group box on the **Simulate** tab, as shown in Figure 5-44. The **Eye Diagram and Mask Options** dialog appears

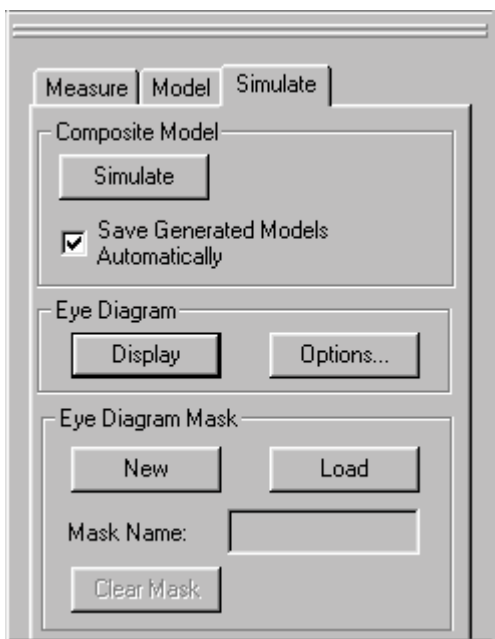


Figure 5-44: Lossy Line/ Symmetric Coupled Lossy Lines Simulate tab.

with a default mask in the viewing area, as shown in Figure 5-45. You can enter a mask name and descriptive comment in the text boxes provided.

Loading an Existing Eye Diagram Mask

To load an existing mask file, click the **Load** button in the **Eye Diagram Mask** group box on the **Simulate** tab, as shown in Figure 5-44. Choose a file in the browser and click **Open**. The mask will be displayed in the Eye Diagram viewing tab.

Modifying an Eye Diagram Mask

After creating a new mask or loading an existing one, you can manipulate it in the mask editor. The active polygon is displayed with red squares on its vertices, which are identified by names (P1, P2, etc.). To activate a different polygon, select it in the polygon drop-down list to the right of the mask editor. Below the polygon drop-down, the coordinates of the active polygon are displayed.

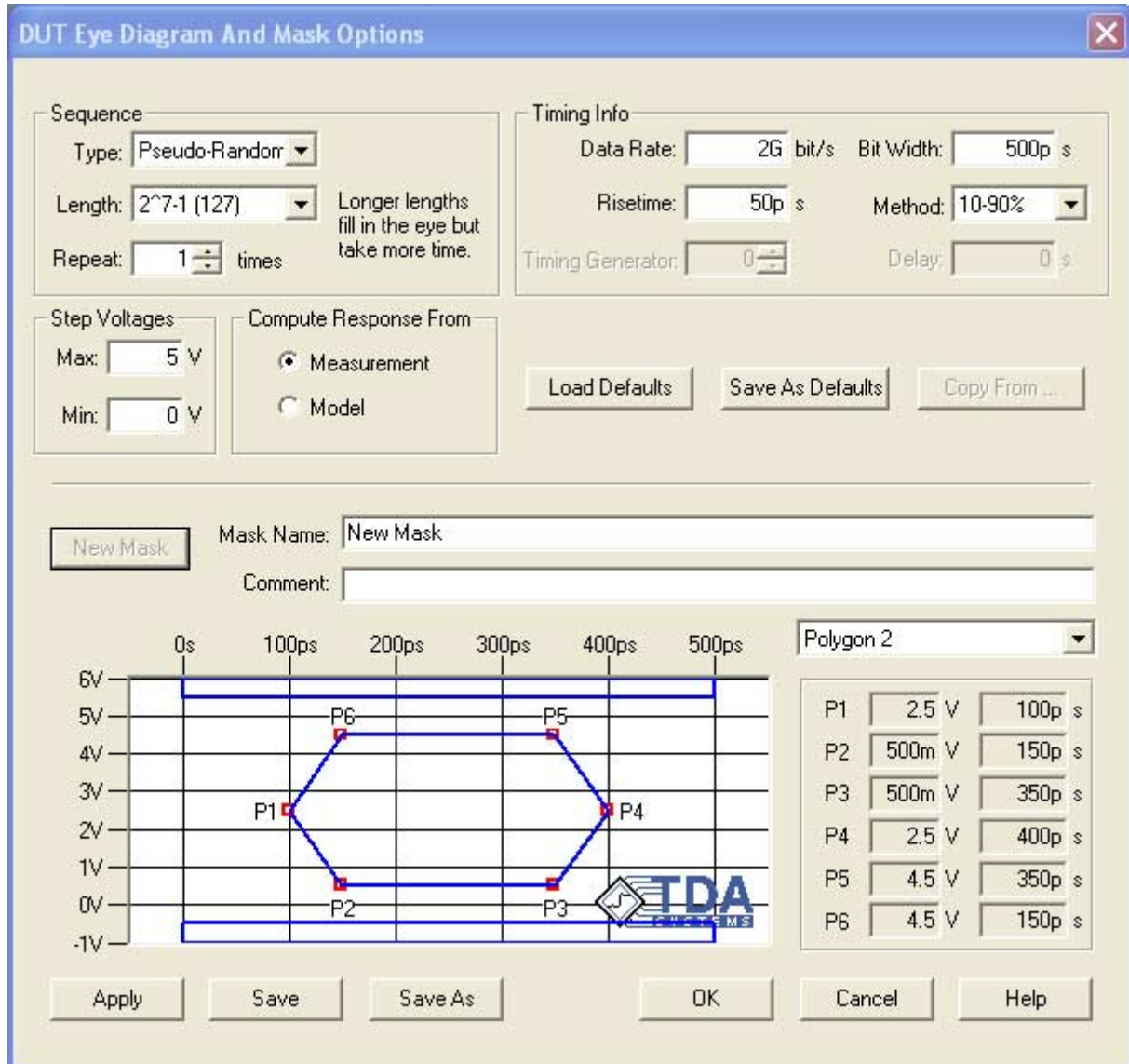


Figure 5-45: Eye Diagram Mask dialog.

You can change the shape of the mask by dragging the red squares on the vertices of the active polygon. As the vertices are moved, their coordinates are updated in the group box. In order to maintain mask symmetry, other vertices may be moved automatically in response.

Applying the Eye Diagram Mask

When you have finished defining the mask, press **OK**, and you will be returned to the Eye Diagram viewing tab. The eye diagram will be updated (if you have made changes to the eye diagram options) and the mask will be displayed. If you would like to view intermediate results of your mask editing while still keeping the **Eye Diagram and Mask Options** dialog open, press **Apply**.

Saving the Mask

To save the mask separately from the model so that it can be applied to other models, press the **Save** or **Save As** button. You will then be able to load it into other Lossy Line models at a later time.

Note: When you save a model with an eye diagram mask in it, the mask will be saved with the model, even if it has not been saved separately using the **Save** or **Save As** button. The only reason to save a mask separately is to be able to import it into other models.

Clearing the Mask

To clear a mask from the Eye Diagram viewing tab, click the **Clear Mask** button in the **Eye Diagram Mask** group box on the **Simulate** tab.

Note: Clearing the mask removes it from the display, but does not delete a mask file that has been saved using the **Save** or **Save As** buttons on the **Eye Diagram and Mask Options** dialog.

5.18 Partitioning Waveforms in IConnect

IConnect's PWL source models, single line models, and symmetric coupled models (collectively known as the "partition-based models", require that a waveform or waveforms to be divided into partitions, and possibly that the characteristics of partitions be defined by the user. This is accomplished using the **Model** tab in each model's window. Although each model's Model tab will vary from the others, each of them has several characteristics in common. This section outlines generally how to efficiently use the mouse and keyboard interfaces to quickly partition waveforms and create models, without delving into the specifics of partitioning each model. Such specific information is contained in each model's section earlier in this chapter.

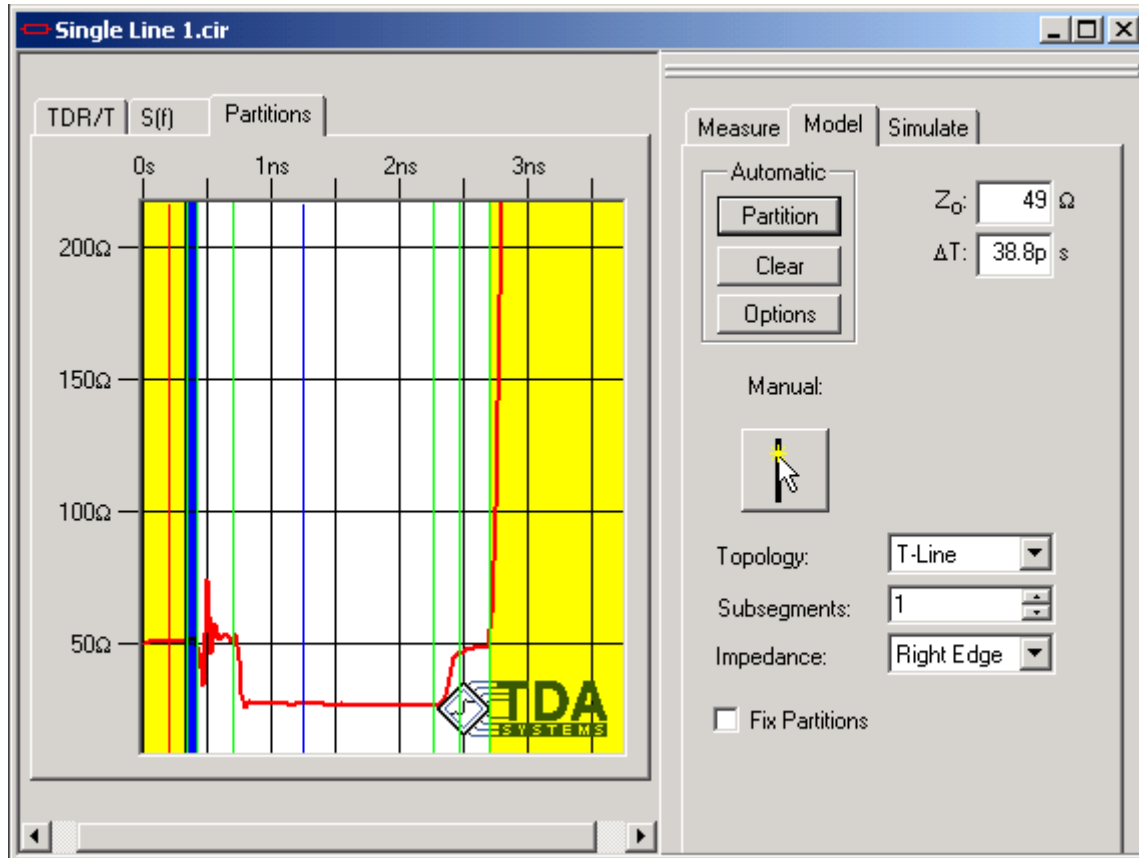


Figure 5-46: Single Line model tab used to partition the Single Line impedance profile waveform.

Each of the partition-based models has a degree of automatic partitioning capability, which will suffice as a starting point for most partitioning tasks. However, this capability is not meant to substitute for modeling expertise, but to speed the modeling process. As a result, it may be necessary to adjust the results of the automatic partitioning by adding, deleting, and modifying some partitions.

Creating partitions

A new partition is created by clicking on the partition button (with the new partition symbol, shown above), and then clicking on the point in the waveform viewer where a new partition is desired.



Figure 5-47: Manual partition creation button.

Note: To create multiple partitions rapidly, hold down the <Shift> key while you click in the waveform viewer, and you can continue to create new partitions until the <Shift> key is released.

Selecting partitions

One partition at a time may be selected, and clicking the mouse inside of the partition toggles the selection state of a partition. The selection may be changed to an adjacent partition using the right and left arrow keys. If a partition is selected, the readouts in the partition window will refer to the characteristics of that partition, although which characteristics are listed will depend on the model being extracted and the type of segment being employed.

Note: If you want to zoom in on a waveform without selecting a partition, hold down the <Ctrl> and <Shift> keys simultaneously while zooming.

Sizing partitions

A partition is sized by grabbing the boundary on either edge and moving it to the desired position.

Deleting partitions

A partition is deleted by selecting it and pressing the <Delete> key.

Automatic partitioning

As a starting point for each of the partition-based models, you may request IConnect to automatically partition the waveform using the **Partition** button. The autopartitioning algorithms are not meant to substitute for modeling expertise, and are tuned to work best for typical applications and reasonably clean data. For best results, you may need to set the autopartitioning options using the **Options** button as discussed in each model's specific manual section in this chapter.



Chapter 6 Simulation

One of the most crucial steps in modeling is verification. This chapter contains the information you will need to interface IConnect to your simulator or simulators, and describes the steps necessary to validate an interconnect model in IConnect by simulating it and comparing it with your original measured data.

Contents

6.1	Verification Overview	6-2
6.2	Simulator Installation	6-3
6.3	Configuring IConnect to Communicate with your Simulator	6-4
6.4	Model Verification with Multiple Simulators.....	6-6
6.5	Model Verification Tips.....	6-8

6.1 Verification Overview

Whether extracted using a measurement-based approach or a basic-principles approach, no model can truly be considered valid until it has been compared against measurement at some level. IConnect streamlines this process by linking directly to simulation tools to allow models to be evaluated, and subsequently loads the results into a waveform viewer for comparison with measured data. As a modeling tool, IConnect relies on third-party simulators to verify extracted models. The benefits of this relationship are that:

1. Users are able to choose from a number of quality, tried and tested simulators written by industry-leading companies.
2. Modelers need not migrate to an unfamiliar simulator to be able to do measurement-based interconnect modeling.

The verification process consists of setting up a simulation to recreate the measurement conditions, and seeing if the model behavior under such conditions is acceptably close to the behavior of the actual DUT. If it is, then the modeling session is complete. If it is not, the model must be altered in some way to improve its accuracy.

IConnect has a composite model viewer for easily recreating the measurement conditions, submitting the composite model to the simulator, and returning the results to the waveform viewer for comparison. This automated sequence represents significant time savings to the modeler.

6.2 Simulator Installation

IConnect supports direct model verification by automatically submitting your simulation netlist to the simulator and reading the results into IConnect when the simulation has completed. IConnect supports the following simulators for model creation and verification:

1. PSpice from Cadence
2. HSpice from Synopsys (formerly from Avant!)
3. Berkeley Spice 3F from the University of California

A precompiled version of Berkeley Spice 3F is included on the IConnect distribution CD. This simulator is provided as is, and TDA Systems makes no guarantee as to its fitness for any particular purpose.

Installation Tips

The full version of Berkeley Spice 3F comes pre-installed with IConnect, and is sufficient to perform model verification. However, users may find that transmission line simulation in Berkeley Spice is slow compared to commercial simulators. In addition, since Berkeley Spice is freeware, the University of California offers no support for users.

6.3 Configuring IConnect to Communicate with your Simulator

Before IConnect can utilize your installed simulator to perform model validation, it will probably be necessary to configure IConnect to locate your simulator executable. To inform IConnect of the location of your simulator, select the **Tools/Options...** menu command and click the **Simulation** tab.

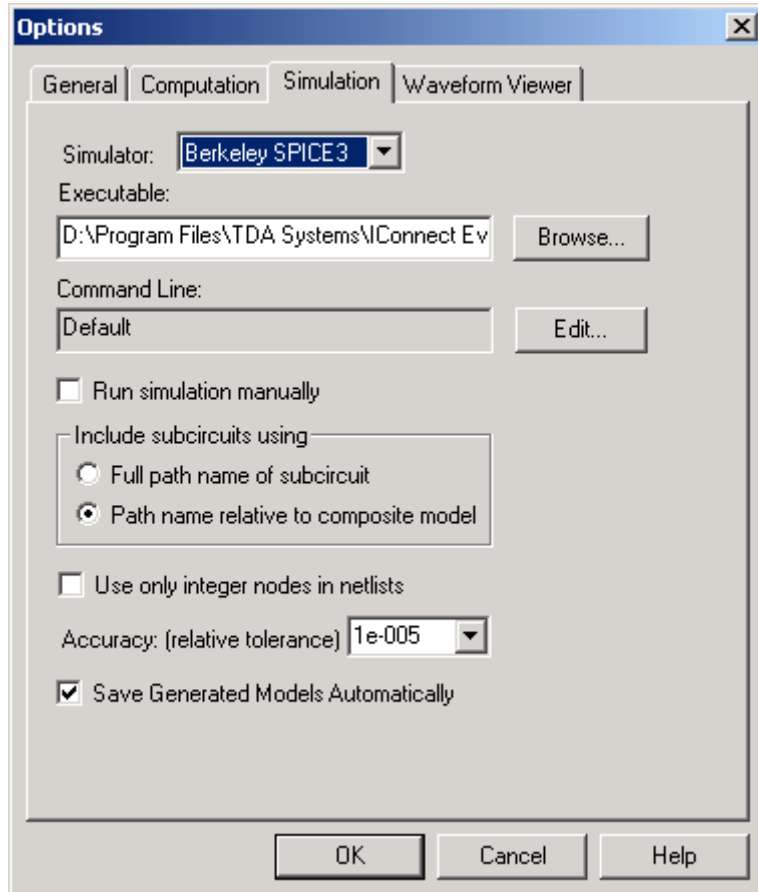


Figure 6-1: Setting simulation options.

In the drop-down list entitled **Simulator:**, choose the simulator you will be using to verify your models. In the text box labeled **Executable:**, type in the name of your simulator's executable file, or alternately the name of a script you wish to use to call your simulator. You may use the **Browse** button if necessary to help choose the file.

If you are using Berkeley Spice as installed during the IConnect installation, the executable will be located at **%IConnect%\Program Files\bspice.exe**, where **%IConnect%** is the directory in which you installed IConnect, e.g. **C:\Program Files\TDA Systems\IConnect**.

In the text box marked **Command Line:** is the command that will be used by IConnect to launch your simulation. In most cases, the default command will correctly launch your simulation and read its results.

Note: The IConnect PSpice command line is configured to work with PSpice 9.2 (Lite) so the current command line is ("%exe" -e -r -o "%output" "%input"). If you would like to run PSpice 8.0 then you will manually need to change the command line to read "%exe" -wONLY "%input". Choose 'PSpice' in the **Simulator** drop-down and click the **Edit** button to edit the command line.

Finally, if you would like to run your simulation manually and inform IConnect to read in the results when you have completed the simulation, you may check the **Run simulation manually** box. This option may be useful for running simulations remotely, although this process can be automated by writing an appropriate script file, as outlined below.

Custom Simulation Commands

In most instances, the default simulation command supplied by IConnect will launch your simulation correctly. However, if you would like to change any aspect of the command, you may do so by pressing the **Edit** button and entering the desired command line in the dialog box. The command submitted to the operating system is identical to that displayed in the **Command Line:** text box, except that the following substitutions are made:

%exe – Simulator executable fully qualified filename, as entered in the **Executable:** text box

%input – Input netlist unqualified filename (no path prepended), which is the name of the composite model submitted for simulation

%output – Output listing unqualified filename, from which the waveforms will be read into IConnect following the simulation.

It is recommended that users enclose all three of the above keywords in double quotes, so that the operating system will correctly interpret filenames containing special characters such as spaces. However, HSpice tends to misinterpret quotes, and as a result they should not be used in a command line passed directly into HSpice.

Note that the simulation command may in fact be any executable (e.g., script) file which takes as inputs the simulation input and output filenames, and which causes the simulation output to be placed in a file with the correct output filename and in the correct format for IConnect to read. IConnect launches the simulation in the directory in which the composite model resides.

Users running custom simulations on platforms other than Microsoft Windows, or on simulators other than the above list of supported simulators, may find it necessary to perform some file format translations before IConnect can read their simulation output files. TDA Systems can provide you with formatting information if you are creating such a translator. Note, however, that IConnect can read simulation results output by both the Microsoft Windows version and the various Unix versions of HSpice with no special file format translation necessary on the part of the user.

6.4 Model Verification with Multiple Simulators

IConnect supports direct model verification interfaces to PSpice, HSpice, and Berkeley Spice 3, by automatically submitting your simulation netlist to the simulator and reading the results into IConnect when the simulation has completed. With some care on the part of the user, IConnect easily handles the different netlist syntaxes and output formats of the various SPICE flavors. If you use a single simulator to perform model verification, the information contained in this section may not be necessary.

Current Simulator

The current simulator is used to automatically simulate your models when you use one of the **Simulate/Run** menu commands. The current simulator is chosen and configured using the **Simulation** tab of the program options property sheet which is accessed via the **Tools/Options** menu command, as discussed in [“Configuring IConnect to Communicate with your Simulator” on page 6-4](#). This simulator also dictates the syntax with which new models are created in IConnect.

Model Syntax

Each model and submodel has its own syntax. When a new model is created within IConnect, its syntax is initially that of the current simulator. However, if a model is read from disk, its syntax is that with which it was last saved to disk. When saving a model, you may override its syntax by choosing an alternate syntax from the choices in the **Syntax**: drop-down list at the bottom of the **Save As** dialog box. If you choose to override the current model syntax, you will be asked to confirm.

When you save a composite model, IConnect checks all submodels to ensure that the syntax of each submodel is the same as that of the composite model. If any discrepancies are encountered, you will be given the option to overwrite the submodel with the syntax of the composite model, leave the syntax of the submodel as is, or cancel the save operation. You are also given the option of resolving remaining syntax discrepancies in other submodels by overwriting them with the syntax of the composite model. It is suggested that you overwrite the submodels with the syntax of the composite model (write it to a new filename if you wish to preserve the existing submodel) to avoid potential simulation errors caused by syntax discrepancies. While some models are generic enough to use identical syntax in all simulators, many contain simulator-specific syntax.

Simulation

When you ask to run a simulation, IConnect performs a non-exhaustive set of consistency checks to attempt to catch possible errors before the netlist is submitted to the simulator. One of the checks is to ensure that the composite model syntax is the same as that of the currently selected simulator which will be used to launch the simulation. If this condition is not met, a simulation error will almost certainly occur. If a discrepancy arises, you will be given the option to save the model using the current simulator syntax, continue with the simulation (not advisable in most cases), or cancel the simulation.

Note: If you choose to resave the composite model using the current simulator syntax, it will likely trigger the need to resave all submodels as well, since they were probably saved using the same syntax as the composite model.

Simulator Configuration in IConnect

For detailed information on how to configure IConnect to interface transparently with your simulator, please see the section [“Configuring IConnect to Communicate with your Simulator”](#) on [page 6-4](#).

6.5 Model Verification Tips

The following suggestions will help verify whether or not your model behaves the same way that your DUT would behave in a circuit.

Be sure of the model netlists you are working with. If you mix submodel netlists between different composites, it is possible to overwrite an existing submodel, causing a composite model created in an earlier modeling session to become invalid. Instead of failing to simulate, it will give incorrect results. It is recommended to keep each modeling session stored in a different directory.

Know your measurements. Inaccuracies in your measurement should not be confused with inaccuracies in your model. If your measurement reads 5 mV when it should read 1 mV, ensure that your model reads 1 mV. If parasitics in your measurement system cause glitches that are not part of the DUT response, don't expect your model to replicate them. If it did, it would not be accurate.

Know your operating edge rates/frequencies. It is only necessary that your model be valid in the frequency range or under the rise times to which it will be subjected in use. If it doesn't match your measurement at faster risetimes than it will see in practice, this is not a problem. Be aware that later, more demanding applications may require you to tweak the model. You may also use the waveform filtering capability of IConnect to approximate your DUT and model responses at slower risetimes.



Chapter 7 Managing and Viewing Waveforms

Efficient use of the IConnect waveform viewers is the key to easily developing models in IConnect. This chapter covers the use of the IConnect waveform viewers for managing and viewing waveforms in both the time and frequency domains.

Contents

7.1	Using the Waveform Viewers.....	7-2
7.2	Waveform Selection.....	7-4
7.3	Waveform File Operations	7-6
7.4	Waveform Viewing Operations	7-8
7.5	Using Labels in the Waveform Viewers	7-11
7.6	Time Domain Waveform Viewer	7-14
7.7	Frequency Domain Waveform Viewer.....	7-16
7.8	Importing and Exporting Waveform Data	7-17
7.8.1	Importing and Exporting Tabular Data	7-18
7.8.2	Importing and Exporting S-Parameter Data in Touchstone Format	7-21

7.1 Using the Waveform Viewers

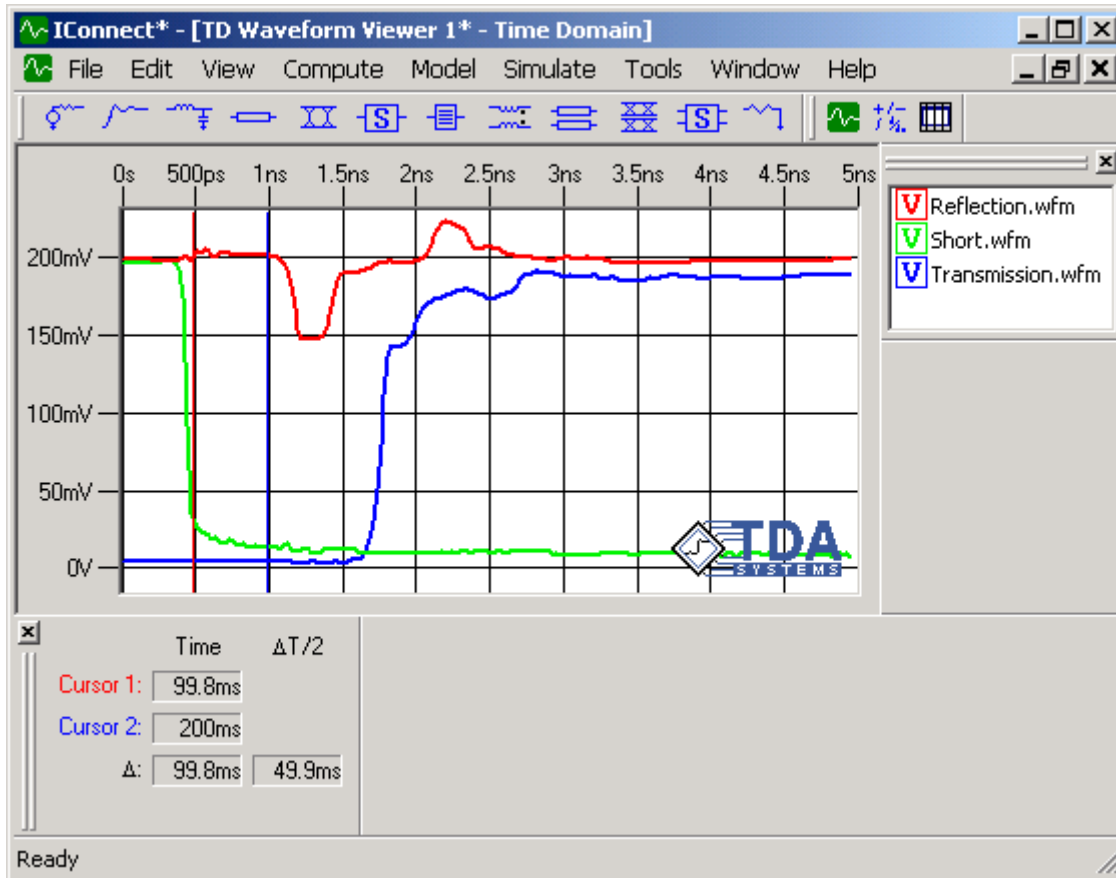


Figure 7-1: The IConnect time-domain waveform viewer.

The primary method for visualizing time-domain and frequency-domain data in IConnect is waveform viewers. The waveform viewers are highly configurable to allow aesthetic changes for production of presentation-quality plots. In addition to data visualization, the viewers support waveform file operations and comparison of waveforms. The time-domain viewer also supports waveform math, which is discussed in [“Waveform Math” on page 8-20](#).

Frequency-domain display, shown in Figure 7-2, is a valuable way to view a device’s response to signals of varying rates of change. Making an analogy to time-domain data, high-frequency behavior tends to represent a device’s response to an input signal’s corners and sharp edges, whereas low frequency behavior corresponds to a device’s response later in the time record after a rapid change. Discontinuities in a device tend to create resonances which are centered around a frequency or a periodic set of frequencies.

An empty waveform viewer is opened from the **File/New...** command by selecting **TD Waveform Viewer** (Time Domain viewer) or **FD Waveform Viewer** (Frequency Domain viewer) from the list shown in Figure 7-3. A more typical way to open a new waveform viewer is simply by acquiring a waveform with the **New Waveform Viewer** option in the acquisition window.

Most single waveform operations are performed either by using the left mouse button or by using the waveform context menu, which is accessed by right clicking the mouse over the waveform in question. Waveforms themselves behave much as file icons in an Explorer-style window.

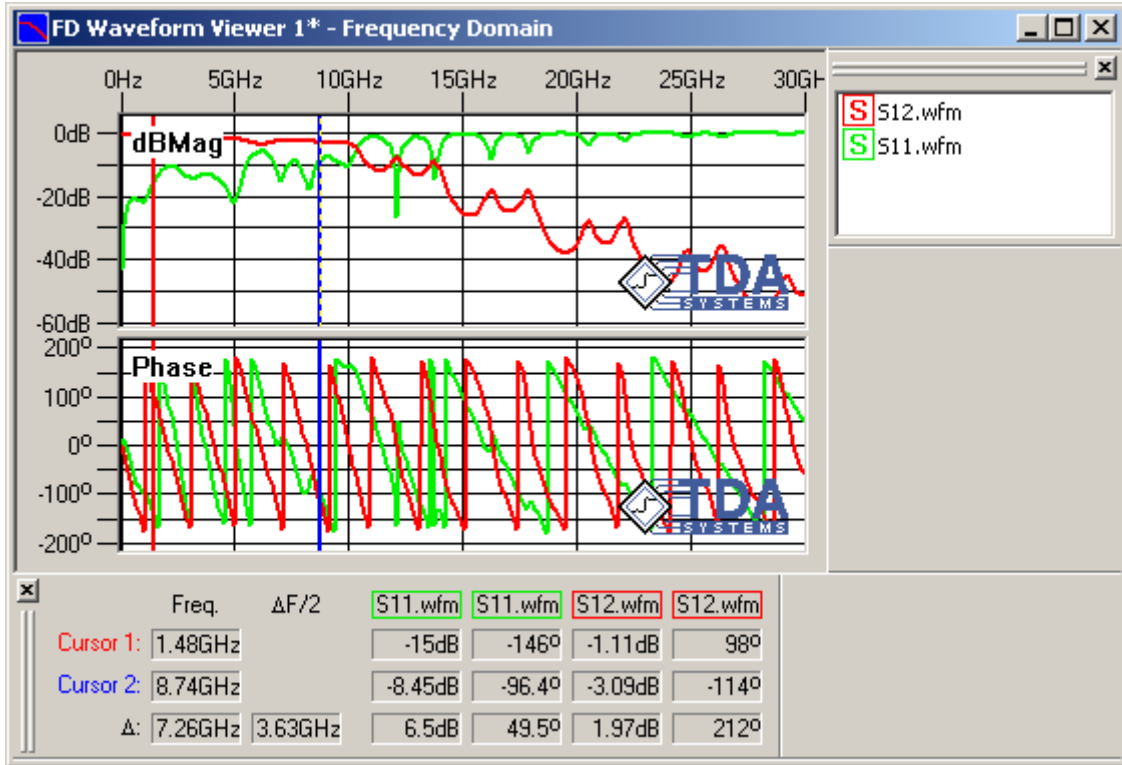


Figure 7-2: The IConnect frequency-domain waveform viewer.

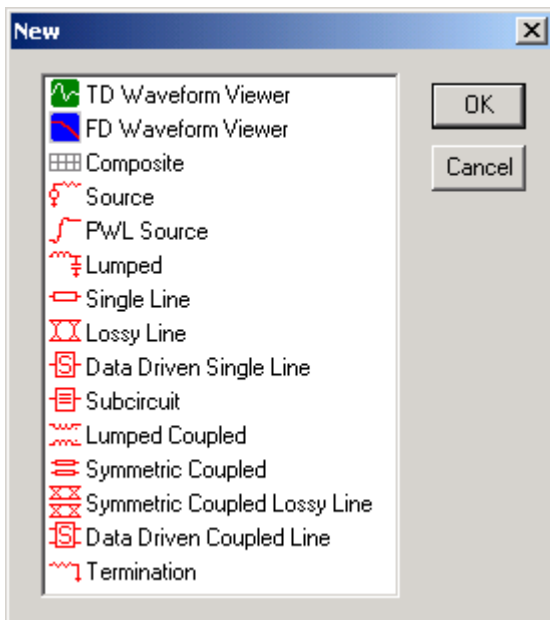


Figure 7-3: Creating an empty waveform viewer.

7.2 Waveform Selection

The key to efficiently operating on waveforms within the waveform viewers is waveform selection. The viewer and the waveform legend window both support selection of multiple waveforms simultaneously. This allows all appropriate waveform operations to be performed on multiple waveforms. Selection is performed similarly to an Explorer-style window.

Selection within the waveform legend window

A single click on a waveform title cancels the current selection and selects that waveform. If the <Ctrl> key is held down while clicking on a waveform, the current selection is maintained and the selection state of the clicked-on waveform is toggled. This allows a simple mechanism for adding or subtracting waveforms from the current selection. If the left mouse button is clicked in any part of the window other than a waveform title, all waveforms are de-selected, unless the <Shift> or <Ctrl> keys are held down, in which case nothing happens. If the <Shift> key is held down and the left button is clicked on a waveform title, the clicked-on waveform and all waveforms between it and the last non-shifted clicked-on waveform are selected. The <Ctrl-A> keyboard shortcut may be used to select all open waveforms.

Note: The waveform legend window is resizable to allow you to view a large number of waveforms or waveforms with long names. You may also undock it from the waveform viewer window to maximize your viewer space.

Selection within the waveform viewer window

Waveforms may be selected within the waveform viewer windows by clicking on the waveform itself. Selection operations within the waveform viewer window are identical to those within the legend window, except that in the viewer window, holding down the <Shift> key is equivalent to holding down the <Ctrl> key. The <Ctrl-A> keyboard shortcut may be used to select all open waveforms.

Waveform context menu

The waveform context menu, activated with the right mouse button, is the key to efficient management of waveforms. If the right button is clicked on a selected waveform, the selection is maintained, and all appropriate menu commands refer to the entire set of selected waveforms. If the right mouse button is clicked on an unselected waveform, the current selection is canceled and the waveform is selected, in which case the commands in the context menu refer to that waveform alone.

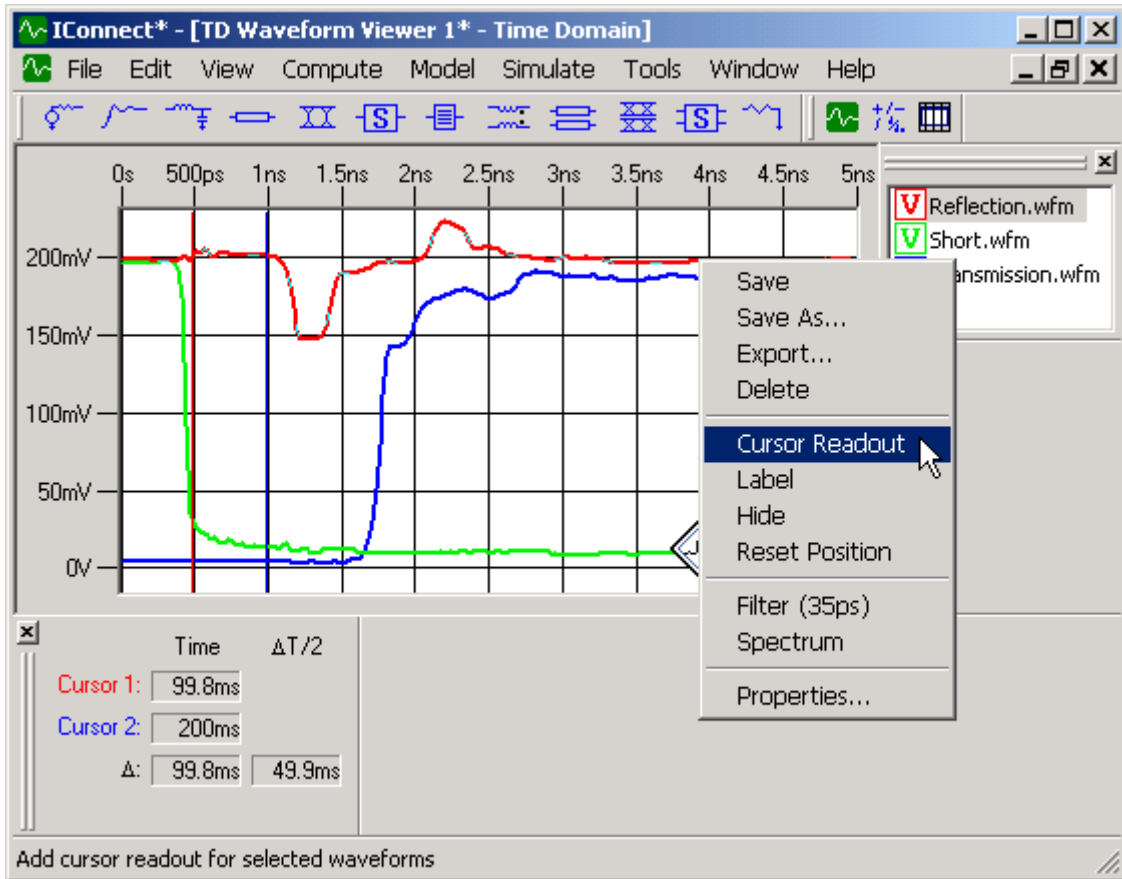


Figure 7-4: The waveform context menu for Reflection.wfm is accessed by right-clicking on the waveform or on its name in the legend. The waveform context menu is the key to efficient waveform management.

7.3 Waveform File Operations



Figure 7-5: The waveform acquisition window.

Loading Waveforms

The most common way to load waveforms from disk is via the acquisition window using the virtual scope. The waveform is loaded into the viewer listed in the **Waveform Viewer:** dropdown list. Waveforms may also be loaded using the **File/Open...** menu command or by dragging them from the Windows Explorer and dropping them on the IConnect window, although these approaches lack the flexibility of the acquisition window in that the IConnect framework decides which waveform viewer will receive the data. The waveform(s) will be loaded into the viewer that was most recently used, if it is still open. If the viewer is no longer open, a new one is created to receive the waveform(s). IConnect is capable of reading Tektronix IPA510 waveform files in addition to its own waveform files.

Saving Waveforms

Waveforms are saved via the waveform context menu accessed by clicking the right mouse button on the desired waveform, as shown in Figure 7-6. **Save** and **Save As...** commands are both available from the menu. **Save As...** prompts the user for a filename under which to save the waveform. If the file already exists, the user is offered the opportunity to cancel the operation. **Save** will only prompt the user for a filename if the waveform has never been saved to disk.

Note: The **File/Save** and **File/Save As...** main menu commands are used to save waveform viewer information to disk, and do not necessarily save individual waveforms.

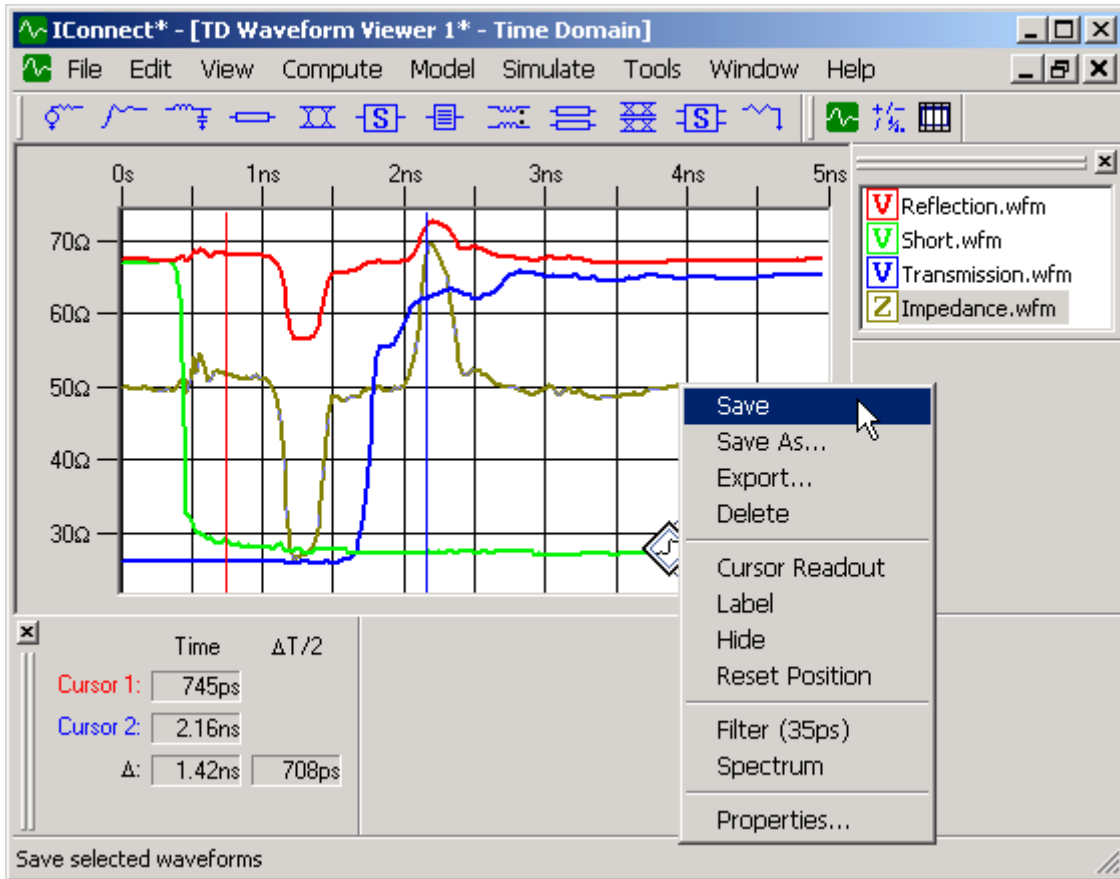


Figure 7-6: Waveforms are saved via the waveform context menu. The File/Save command in the main menu saves waveform viewer information, but does not save individual waveforms.

Waveform Import/Export

Waveforms may be imported from and exported to a variety of software tools using tabular data separated by comma (CSV), space, or tab characters, as described in [“Importing and Exporting Tabular Data”](#) on page 7-18. However, it should be noted that IConnect does not automatically detect CSV or other tabular formats when it reads a file. As a result, tabular files may not be read from the **File/Open** menu item or from the virtual scope, but must instead be read in using the **Import** utility accessible from the **File/Import** menu item, or from the **Import** item of the waveform viewer context menu.

7.4 Waveform Viewing Operations

Within the time-domain and frequency-domain waveform viewers, the selected waveform(s) may be moved and hidden, as well as deleted. Cursor readouts are available as well.

Moving waveforms

At times it is convenient to be able to move waveforms, particularly for comparison purposes. Simply clicking on one with the left mouse button and dragging the selected waveforms to the desired position moves the selected waveforms. If the <Shift> key is held down while the waveforms are being dragged, their motion is constrained to be either vertical or horizontal. This is a convenient method of introducing either delay or DC offset into a waveform without introducing the other.

Alternately, the selected waveforms may be moved using the arrow keys. If the <Shift> key is held down while the arrow keys are used, the waveforms move by larger increments. Note that if you have just been operating in the Waveform Legend window, you will need to press the <TAB> key in order for the arrow keys to work in the Waveform Viewer.

Cursor readouts corresponding to a waveform are disabled when it is moved because their meaning would be ambiguous regarding a translated waveform, and are re-enabled when it returns to its original position. The best way to return the selected waveforms to their original positions is by using the **Reset Position** command from the waveform context menu.

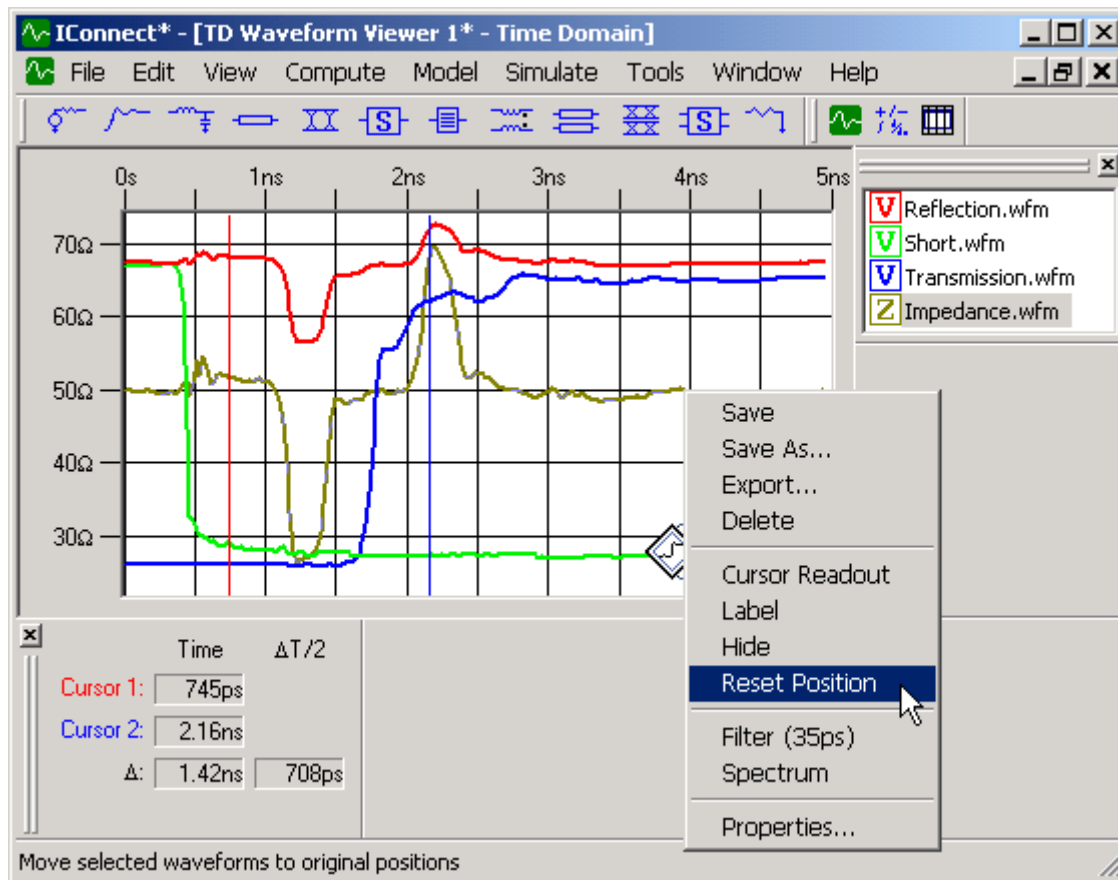


Figure 7-7: Resetting the position of a waveform that has been moved. The waveform context menu can perform a variety of operations on waveforms.

Zooming

To inspect an area of a waveform more closely, left-click on an empty area of the display and drag a rectangle over the area to which you would like to zoom. Make sure that the rectangle you draw is large enough to cause the zoom to occur. When you are finished zooming, choose **View/Zoom to 100%** from the main menu or from the waveform viewer context menu to return to the original unzoomed view. Additionally, the **Zoom In** and **Zoom Out** commands, available from the **View** menu, zoom the plot in and out respectively, by a factor of two.

Autoscaling

The viewer may be autoscaled to set the scales to achieve maximal use of the viewing area while showing every waveform. This is useful if the scales are such that a large amount of empty space is present in the viewer. It is also a quick way to zoom out.

Manual Scaling

In some instances it will be convenient to set the axis limits manually. This may be accomplished by selecting the **Set Scale...** item from the **View** menu. A window will open which will allow you to manually enter the axis limits for the waveform viewer. These limits define the size of the unzoomed waveform viewer until either another manual scaling or an autoscaling operation is performed.

Hiding and showing waveforms

In order to hide certain waveforms from view to reduce screen clutter or to isolate certain waveforms for inspection, the selected waveforms may be hidden using the **Hide** command from the waveform context menu. They may be viewed again by choosing the **Unhide** command from the context menu. If a heterogeneous group of waveforms is selected (i.e., some visible and some hidden), the entire group is first hidden, at which point it becomes a homogeneous selection with respect to its visibility.

Deleting waveforms

The selected waveforms may be deleted from the viewer by means of the **Delete** command from the waveform context menu, or by pressing the <Delete> key. Deleted waveforms may always be retrieved by the **Undo** command, available from the **File** menu or from the standard toolbar.

Note: Waveforms are **not** deleted from the disk by this method, but only from the waveform viewer. To delete the waveform from disk, use Windows Explorer or a similar application to remove the waveform files.

Cursor readouts

Cursor readouts may be viewed for any set of the waveforms loaded into the viewer. The cursor readout status of the selection may be toggled via the **Cursor Readout** command from the waveform context menu. If the selection is heterogeneous with respect to cursor readout (i.e., some selected waveforms have readouts, some don't), cursor readout is turned on for all selected waveforms, at which point the selection becomes homogeneous.

Waveform properties

The **Properties...** command from the waveform context menu or the <Alt-Enter> keyboard shortcut may be used to edit multiple waveform characteristics at once for the currently selected set of waveforms. Properties that may be edited include color, position, visibility, cursor readout status, and waveform comments.

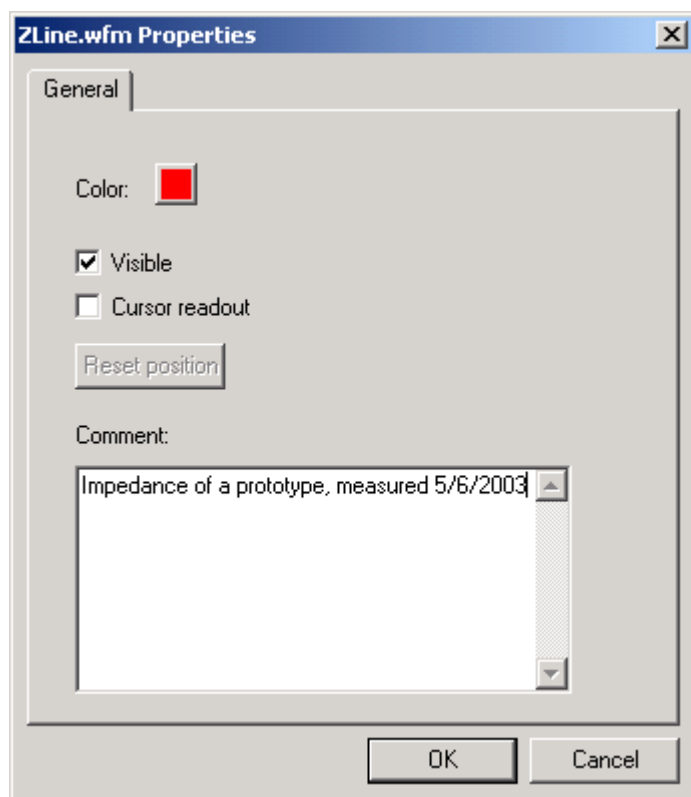


Figure 7-8: Changing waveform properties. This property sheet is accessed via the waveform context menu.

7.5 Using Labels in the Waveform Viewers

Labels are a convenient way to textually enhance and document your IConnect waveform displays. Labels are customizable in terms of their color, size, borders, and backgrounds. They can be associated with a particular waveform on the display, or they can be independent, such as a title or note. When a waveform viewer is saved, the size, position, and other properties of each label are saved as part of the file, which may then be recalled at a later date.

Creation

Labels may be created from either the Waveform Viewer context menu, accessed by right clicking on an empty portion of the waveform viewer, or from the waveform context menu, which is accessed by right clicking on a waveform. Labels created from the waveform context menu are by default attached to the selected waveform, and the text of such labels defaults to the name of the waveform. Labels created from the Waveform Viewer context menu are by default not connected to any waveform, and initially have no text.

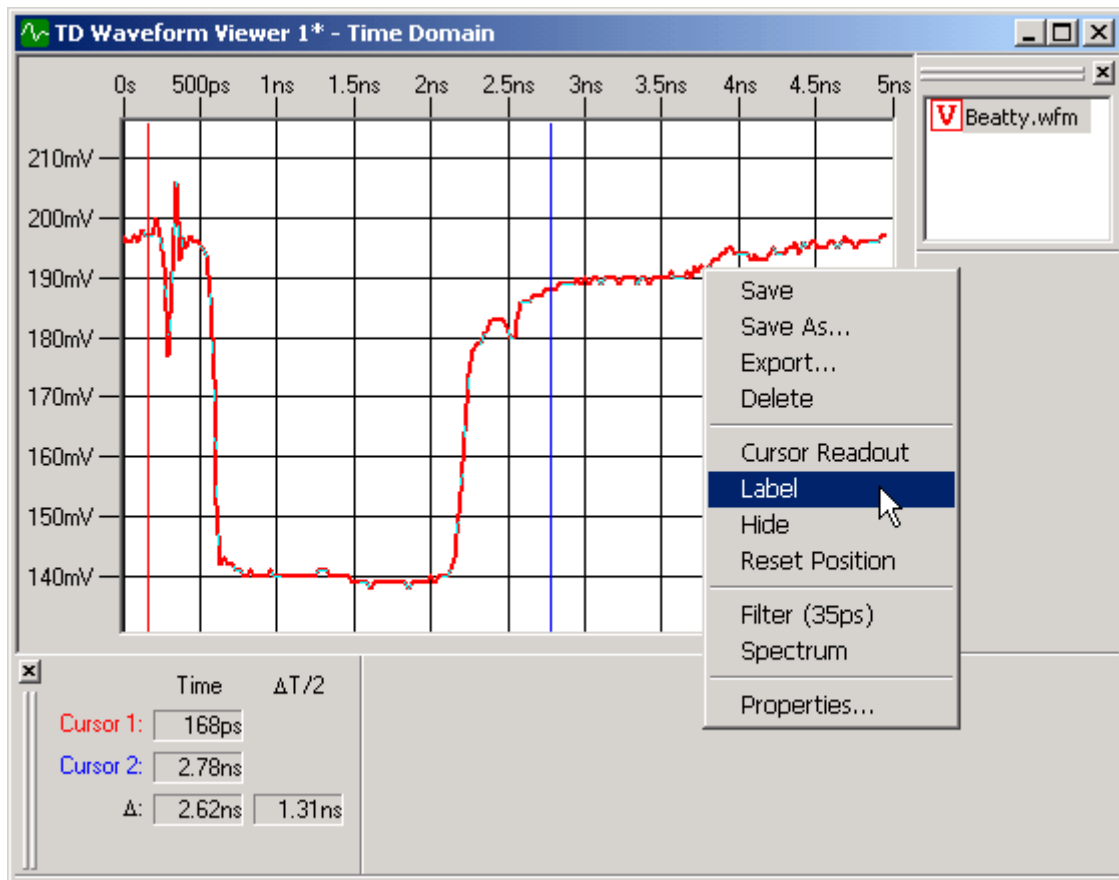


Figure 7-9: Creating a label.

Deletion

A Label may be deleted by right clicking on it and selecting **Delete**, or by selecting it and pressing the <Delete> key.

Repositioning

A label may be moved by clicking on it and dragging it to the new location.

Editing

Label text may be edited by single clicking in the label when it is already selected, and then using standard editing keystrokes to delete or enter text. Editing is completed by clicking elsewhere in the viewer window, or in another window. Note that if you wish to cancel any changes, you may press the <Escape> key prior to completing the edit. Standard **Copy**, **Cut** and **Paste** commands also work during label editing.

Special Text

If a label has a waveform associated with it, the text **%name** in the label text is automatically translated into the name of the waveform.

If a label has a waveform associated with it, the text **%comment** in the label text is automatically translated into the waveform comment as specified in the waveform properties.

Sizing

A label may be resized by selecting and dragging its border during an edit operation.

Autosizing

A label may be autosized by right clicking on it and selecting **Autosize**. Its width and height are set to be flush against the label text on all sides.

Properties

Individual or group label properties are set in the label properties window, shown in Figure 7-10, which is accessed by right clicking on a label or a set of selected labels, and selecting **Properties**. The following properties are settable:

Foreground – Text and border color

Background – Color behind text, inside of border

Background Type:

Opaque – Use requested background color

Transparent – No background. Allows viewer display to show through label. Also turns off shadow

Viewer Color – Use whatever color the Waveform Viewer has as its background

Border Type – Single Line or no border

Shadow – Draws a shadow behind the label

Waveform – Associates the label with a waveform

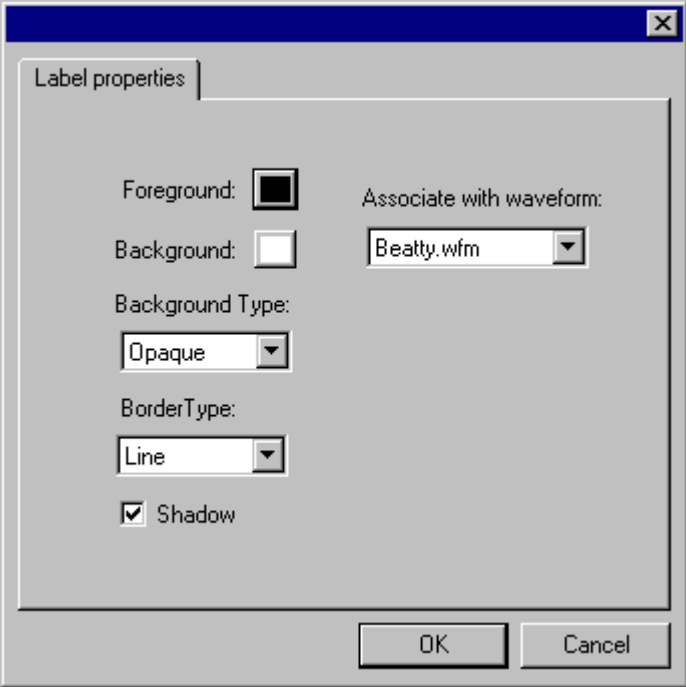


Figure 7-10: Changing label properties.

7.6 Time Domain Waveform Viewer

In the time domain, data may either be plotted against time or against distance. To switch between different plotting modes, right click in the waveform viewer, and choose the Plot submenu from the resultant context menu. Then choose the desired plotting mode, as shown in Figure 7-11. Inappropriate plotting modes will be grayed out.

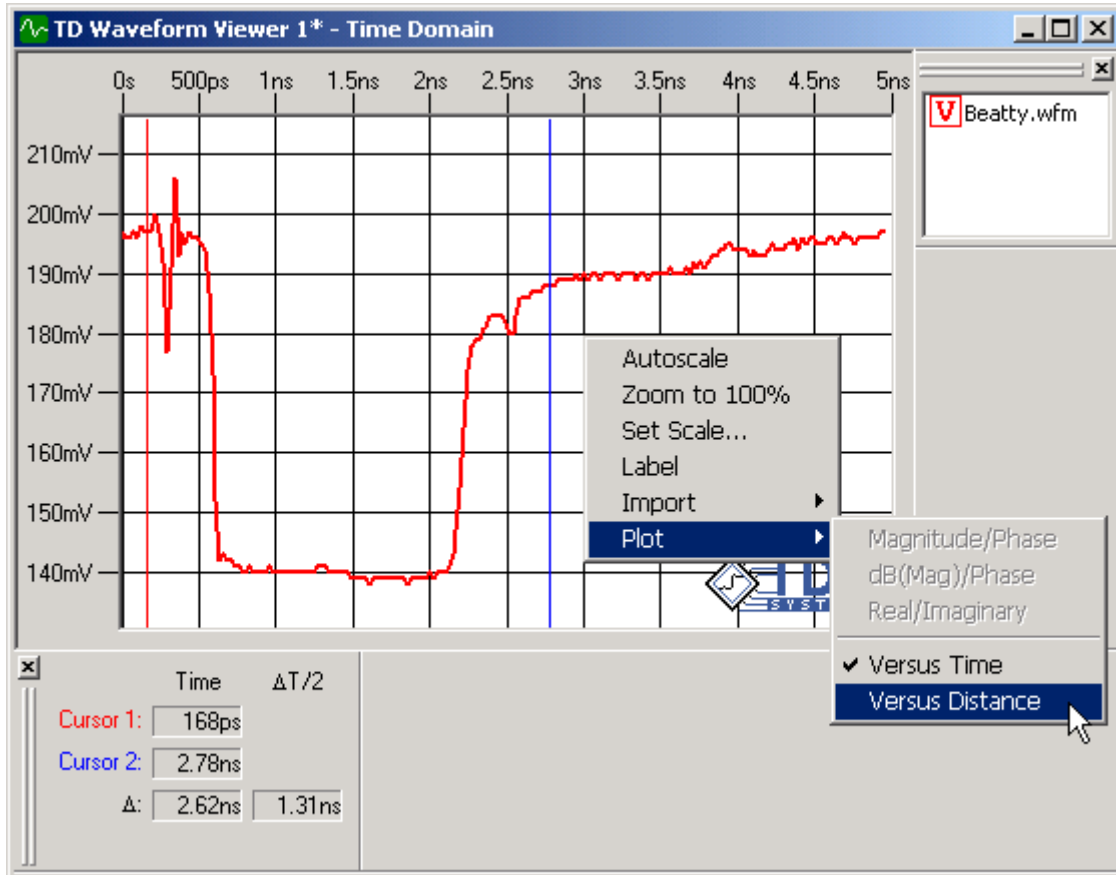


Figure 7-11: Plotting waveforms versus distance. Time and distance are related by the velocity of propagation in the material.

Plotting vs. Time

This is the default plotting mode for IConnect in the time domain. The data is plotted exactly as it was transferred from a measurement instrument or from a file, or computed via waveform computations.

Plotting vs. Distance

In this mode, time values are linearly scaled to become distance values, according to the equation

$$\Delta x = \frac{c\Delta t}{\sqrt{\epsilon_r}}$$

where c is the speed of light in a vacuum, and ϵ_r is the effective relative dielectric constant of the dielectric material from which the structure is fabricated. The dielectric constant is displayed in the lower left corner of the waveform viewer, and is set by choosing the **Tools/Options...** command

from the main IConnect menu, and selecting the **Waveform Viewer** tab. The distance units used for display on the x-axis may be set on the same options page.

It should be noted that the effective dielectric constant of the DUT and that of the measurement interface cabling will most likely be different, so that absolute distance measurements have little meaning. Furthermore, in many cases the effective dielectric constant will vary significantly within the DUT itself, due to geometric and materials variations. Any distance measurements should be conducted between waveform features corresponding to points on the DUT which are separated by a signal path over which the dielectric constant does not vary longitudinally.

7.7 Frequency Domain Waveform Viewer

Note: This section describes how to manage waveforms **within** the frequency-domain viewer. To understand how to convert time-domain data **into** frequency-domain data, refer to “[Step Spectrum Computation](#)” on page 8-9 and “[S-Parameter Computation](#)” on page 8-11.

In the frequency domain, data may either be represented as Magnitude/Phase, dB(Mag)/Phase, or Real/Imaginary. To switch between different plotting modes, right click in the waveform viewer, and choose the Plot submenu from the resultant context menu. Then choose the desired plotting mode, as shown in Figure 7-12. Inappropriate plotting modes will be grayed out

There is a settable maximum frequency above which frequency-domain data will not be displayed. This is to allow users to window out noise which is often present in high-frequency waveform data. To set the maximum plot frequency, go to **Tools/Options...** in the main IConnect menu, select the **Waveform Viewer** tab, and set the value in the **Max. Frequency:** text box.

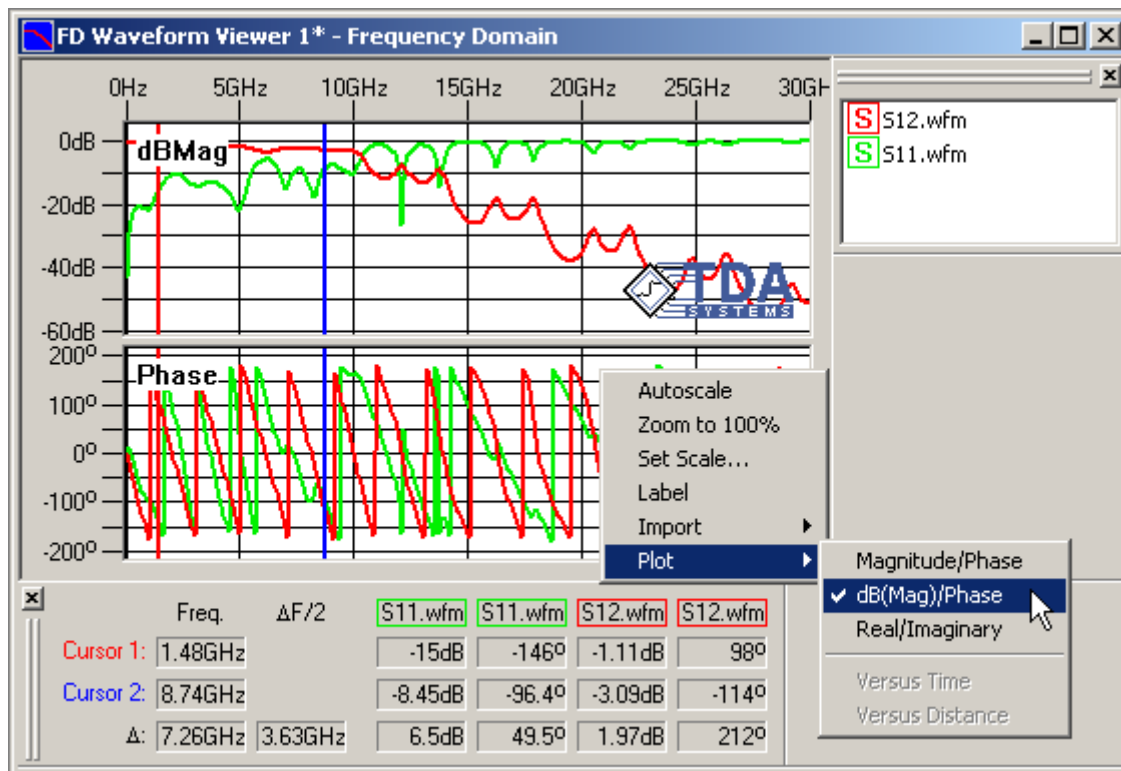


Figure 7-12: Plotting frequency-domain data.

7.8 Importing and Exporting Waveform Data

IConnect comes with the ability to easily share data with other applications. IConnect is capable of writing and reading time- and frequency-domain data to and from tabular data formats separated by space, comma (CSV), or tab characters. These formats are written and read by many different applications, including Microsoft Excel and other spreadsheet tools. IConnect also reads and writes frequency domain data in touchstone format, which is supported by many simulation tools and measurement instruments. Finally, IConnect reads file formats created by the Tektronix IPA 510 product, but does not write these formats.

Note: If you only want to use your waveforms within IConnect, they should be saved using the **File/Save** command, or the **Save** command which is available when you right-click on a waveform or on its name in the legend window. Import and export capabilities are only for sharing data with other applications.

Waveform data files from other applications may be imported and exported from IConnect using either the IConnect main menu or from various context menus accessed via the right mouse button.

7.8.1 Importing and Exporting Tabular Data

IConnect is also capable of writing and reading time- and frequency-domain data to and from tabular data formats separated by space, comma (CSV), or tab characters. These formats are written and read by many different applications, including Microsoft Excel and other spreadsheet tools.

Note: If you only want to use your waveforms within IConnect, they should be saved using the **File/Save** command, or the **Save** command which is available when you right-click on a waveform or on its name in the legend window. Import and export capabilities are only for sharing data with other applications.

Exporting tabular data

In order to export a waveform or multiple waveforms, select the waveforms in the waveform viewer or select their names in the waveform legend window. Right-click on one of the waveforms (or its name in the legend window) and select **Export/CSV..** to bring up the waveform export window. From the dropdown box labeled Save As Type, you can choose to use either commas, spaces, or tabs to separate the columns of data that will be saved. Which format you use will depend primarily on how you intend to utilize the exported data. If you will be importing the data into Microsoft Excel, comma-separated values, using a CSV file extension, is often the best choice. If you are using Matlab, space-separated columns (with no column headers included) is generally the best choice.

If you would like to include each waveform's name as text in the first row of the file, you may do so by checking the box titled **Include column headers**. This will make your files more readable to others, although it may cause errors in third-party programs attempting to read pure numeric values.

Note: Including waveform names which contain the separation character (e.g., a space-separated file with waveform names containing spaces of their own) will generate errors in many programs, including IConnect, when they attempt to read in the resulting data file.

If you are using commas to separate your data values, you will also have the option to omit writing the first column heading to the file, although still including the subsequent comma. This is accomplished by checking the box titled **Suppress first column header**, and it allows data to be imported directly into Excel and used for plotting. The specifics of this format variation are discussed in the subsection below entitled **CSV format specifics**.

When you click OK, the waveforms will be saved to a single file in which the first column is the x values of each point, and each of the subsequent columns contains the y values of each waveform. However, if you have checked the box entitled **Save waveforms to different files**, each waveform will be saved in a separate file.

Importing tabular data

Importing tabular data into IConnect is seamless, provided that the data is in the proper format. Begin by right-clicking in a waveform viewer and selecting **Import/Waveforms...** from the resulting menu. Choose one of the selections from the **Files of type** dropdown box based on whether your columns of data are separated by commas, spaces, or tabs. If there are no textual column headers in your file, you can choose the **ASCII Data** selection, which allows IConnect to automatically sense which delimiter character the file uses, and read it accordingly. If there are column headers in the file, and you would like the waveforms to be named based on the column headers when they

are read in, it is recommended that you explicitly select the file type based on the delimiter character.

You must ensure that the correct domain is chosen in the **x-axis** dropdown, and that the correct waveform type is chosen in the **y-axis** dropdown. If you are reading frequency domain data, you must also select the representation from the **Complex data** dropdown that corresponds to the format in which the data is stored in the file: Mag/Phase, Real/Imaginary, or dBMag/Phase. If the file you are importing contains header lines before the tabular columns begin, you can force IConnect to skip a fixed number of lines before it looks for the tabular data.

IConnect automatically computes the time or frequency step information by subtracting the first value of the first column from the second value. If a value obtained in such a way is not accurate, you may insert your own time or frequency step value using the group of windows labeled **Step size**.

CSV format specifics

IConnect seamlessly imports and exports files stored in the "CSV" file format. CSV format refers to comma-separated values, in which rows and columns are stored in a text file with columns separated by commas and rows separated by carriage returns. There are many variations of the loosely defined CSV format, and the IConnect implementation is geared to be most compatible with Microsoft Excel. An example of IConnect CSV format would be

```
0,0.4,1.2
2e-11,0.5,1.1
4e-11,0.6,1.2
6e-11,0.5,1.5
8e-11,0.4,1.4
1e-10,0.6,1.5
...
```

in which the number of lines (or rows) of data is arbitrary. The number of columns is also arbitrary, and each column except the first is read in as a single waveform. In the example above, the first column is assumed by IConnect to represent the time values associated with the rest of the data points in the row. It should be noted that when importing CSV files, IConnect assumes a constant sample interval, or time between data points, by reading only the first two data points in the time column and taking the sample interval to be the difference between these two values. This means that if your data is not evenly sampled, it is necessary to interpolate it prior to importation into IConnect. The remaining two columns in the above example are interpreted as two waveforms.

A variation on the basic CSV format may be utilized, if desired, to associate a name with a waveform, and also to identify the first column as x-axis data to graphing programs such as Microsoft Excel. This may be done by including a single row of comma-separated text values at the beginning of the file, which serve as names for the waveforms. If one waveform is named, it is necessary for every waveform to have a name, although as described next, the x-axis values need not have a name. To identify the first column as x-axis data, the first text value may be omitted. *However, the comma following the value must not be omitted*, or the data will not be read properly. An example of two waveforms in this format follows:

```
,Waveform 1,Waveform 2
0,0.4,1.2
2e-11,0.5,1.1
4e-11,0.6,1.2
6e-11,0.5,1.5
```

8e-11,0.4,1.4

1e-10,0.6,1.5

...

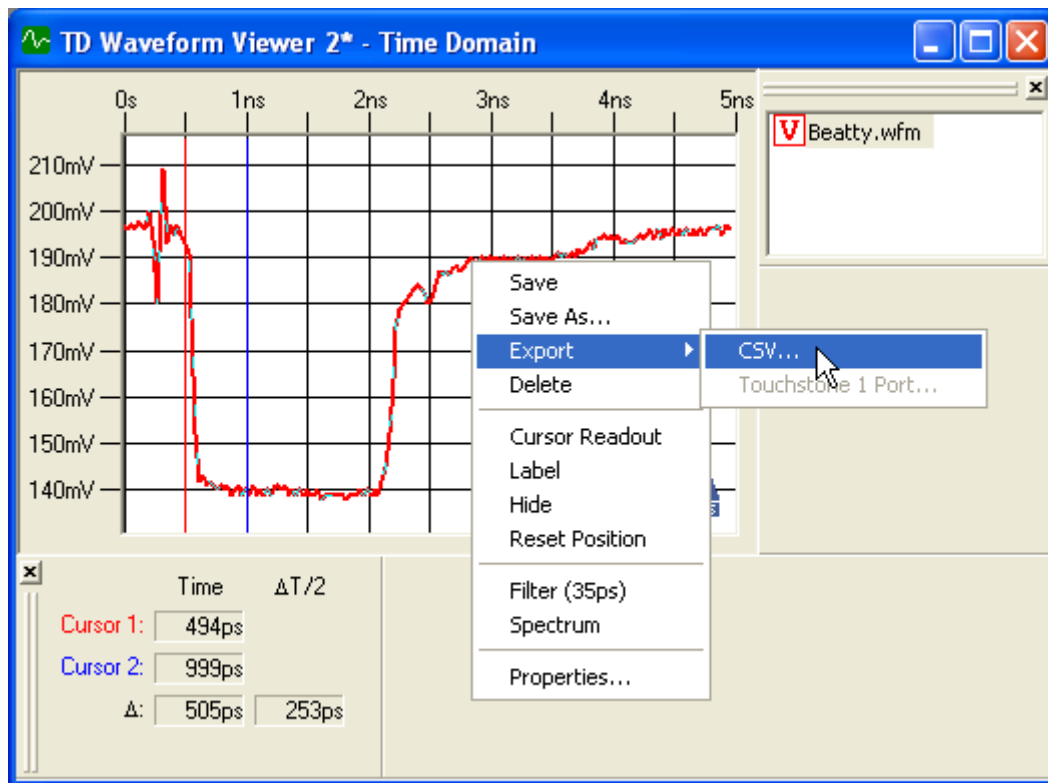


Figure 7-13: Exporting waveform data as tabular ASCII text.

Troubleshooting tabular waveform data import problems

The most common problem encountered when importing tabular data into IConnect is faulty column headers. If your data includes column headers, make sure that they do not contain spaces, particularly in space-separated files. Similarly, ensure that column headers do not contain commas in comma-separated files. If either of these situations presents itself, either edit or remove the header line, or force IConnect to skip the line using the **Skip first n lines** text box. Furthermore, if a column header contains any spaces, tabs, or commas, IConnect may not be able to autosense which delimiter is being used, in which case you must explicitly inform it what delimiter character is being used in the file using the **Files of type** dropdown box.

A common error message is that IConnect expected to read a certain number of columns, and did not find that many columns to read. This is because IConnect reads the first line of the file (after skipping any lines which you have requested) and uses that information to determine how many columns are in the file. If subsequent rows do not have the same number of columns, an error message is generated. Often a column header with spaces in it can be read as multiple columns, causing an error to be generated when the subsequent rows do not appear to contain the same number of columns in them.

7.8.2 Importing and Exporting S-Parameter Data in Touchstone Format

IConnect supports import and export of frequency-domain data in touchstone format. Any S-Parameter waveform in IConnect can be exported as touchstone from either the IConnect main menu or from various context menus accessed via the right mouse button. You may also export a group of waveforms as a two- or four-port network in touchstone format. Two-port and four-port touchstone data import is handled in the data driven single and coupled line models.

Note: If you only want to use your waveforms within IConnect, they should be saved using the **File/Save** command, or the **Save** command which is available when you right-click on a waveform or on its name in the legend window. Import and export capabilities are only for sharing data with other applications.

Exporting single waveforms in touchstone format

The simplest way to export a single S-Parameter waveform to a touchstone file is to right-click on the waveform or on its name in the waveform legend. When the context menu appears, choose **Export** and then **Touchstone 1 Port**. You will be asked to choose a filename, and the touchstone file will be written to disk. If multiple waveforms were selected, you will be asked to supply a filename for each waveform. If you prefer, you may use the IConnect main menu to save the data, by selecting the desired waveforms and choosing **File/Export/Touchstone/1 Port**.

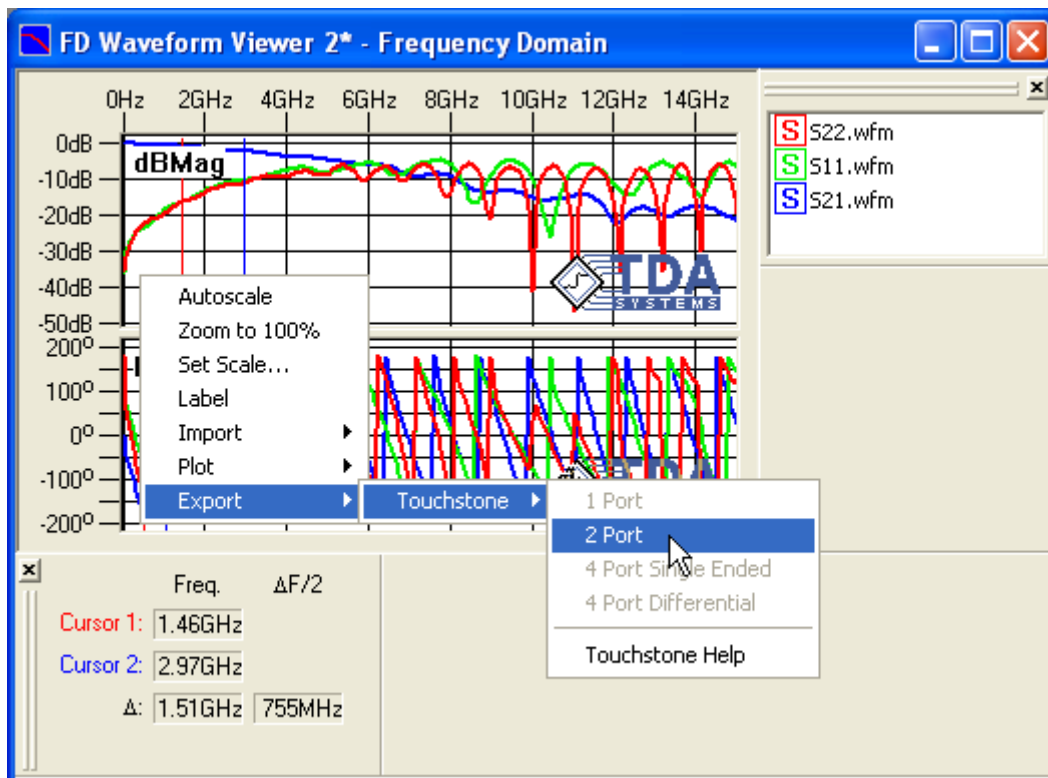


Figure 7-14: Exporting frequency domain waveform data to a touchstone file.

Exporting two-port and four-port networks in touchstone format

The simplest way to export a set of S-Parameter waveforms as a network to a single touchstone file is to right-click in an empty area of the waveform viewer that contains the desired waveforms. When the context menu appears, choose **Export** and then choose the size of the network you would like to export, as shown in Figure 7-14.

Note: If you want to export a two-port network, you must have at least three S-Parameter waveforms present in the waveform viewer. If you want to export a four-port network, you must have at least ten S-Parameter waveforms present in the waveform viewer.

If you are exporting a four-port network, IConnect gives you the choice of providing either single-ended or differential S-parameters for export.

Note: IConnect does not perform transformations on the waveforms you provide between single-ended and differential representations. The waveforms are written to the touchstone file as you have provided them.

IConnect will display a window that allows you to choose the waveforms you would like to include in the network. Enter each waveform in the appropriate dropdown list box. If you leave the **Reciprocal** box checked, all waveforms entered in the lower half of the matrix will be copied to the upper half of the matrix when the touchstone file is written. If you want to enter waveforms explicitly in the upper half of the matrix, uncheck the **Reciprocal** box. Once you have selected the waveforms you want, press **Continue**. You will be asked to provide a filename, and the touchstone file will be written to it.

Importing two-port and four-port networks in touchstone format

IConnect allows you to import two-port and four-port networks in touchstone format via the measure tabs of the Data Drive Single Line and Data Driven Coupled Line models, respectively. For information on how to import two- and four-port data, [“Data Driven Single Line” on page 5-52](#) and [“Data Driven Coupled Lines” on page 5-54](#).



Chapter 8 Waveform Computations

IConnect's waveform computation capabilities are the key to accurately interpreting your TDR measurements and easily developing accurate models in IConnect. This chapter covers the use of the waveform computation window for extracting model parameters and doing waveform math and basic signal processing.

Contents

8.1	Waveform Computations Overview	8-2
8.2	Waveform Filtering	8-4
8.3	Impedance Profile Computation	8-6
8.4	Step Spectrum Computation	8-9
8.5	S-Parameter Computation.....	8-11
8.6	Capacitance and Inductance Computation	8-14
8.6.1	Self Inductance Computation.....	8-15
8.6.2	Self Capacitance Computation	8-16
8.6.3	Mutual Inductance Computation	8-17
8.6.4	Mutual Capacitance Computation.....	8-18
8.7	Flip Waveform	8-19
8.8	Waveform Math	8-20

8.1 Waveform Computations Overview

IConnect is capable of performing a number of waveform computations, including impedance profile computation, which is the precursor to many extraction techniques. General waveform math operations such as addition, subtraction, multiplication and division of two waveforms are available, as well as integration, differentiation, and risetime filtering. IConnect can also compute self and mutual inductances as well as self and mutual capacitances according to JEDEC standard techniques.

IConnect is capable of computing scattering parameters from TDR measurements, allowing you to use your oscilloscope like a network analyzer, and also computes the spectrum of a step-like function such as a TDR input step using FFT-based methods.

Note: No computations are currently supported for frequency-domain data, which is why frequency-domain waveforms may not be selected as computation operands, and frequency-domain viewers may not be selected in the **Waveform Viewer** dropdown.



Figure 8-1: Waveform computation window.

Computation Window

Computations are performed from the computation window, which is a dockable window associated with the main IConnect window. You may toggle its visibility using the **View/Computation Window** menu command. Or, use the **View/Toolbars/Auxiliary Toolbars** menu command to

enable visibility of the auxiliary toolbar, after which you can toggle the visibility of the computation window using the toolbar button.

Computation Selection

The desired computation is selected using the drop-down list at the top of the computation window. The other choices available from the computation window will change in response to your selected computation.

Waveform Viewer Selection

It is necessary to choose which waveform viewer to use as a source for computation waveforms. This selection is made using the drop-down list box marked **TD Source Waveform Viewer:** near the top of the computation window. The list box contains the names of all currently open time domain waveform viewers. If no viewers are available, the viewer selection list box and the remainder of the computation window are disabled.

For computations which transform time-domain data into frequency-domain data, an additional drop-down list box titled **FD Target Waveform Viewer:** is displayed. The list box contains the names of all currently open frequency domain waveform viewers. The selected waveform viewer is used to display the resultant waveforms. If no frequency domain viewers are available, a new frequency-domain viewer is created to display the results of the computation.

Waveform Selection

Once the desired computation and waveform viewer(s) are selected, the waveforms available as operands are listed in the group of one to three drop-down list boxes in a group titled **Waveforms**. Each list box contains a list of waveforms from the selected waveform viewer which are pertinent to the desired computation.

Note: An exception is when **Waveform Math** is selected, in which case one of the boxes is used to select a math operator.

The number of list boxes and the type of waveform available in each will depend on which computation has been selected. In each list box, choose the waveforms upon which the desired operation is to occur.

Additional Parameters

Depending on the selected computation, one or more additional parameters may be available at the bottom of the computation window for selection.

Computation

Once the appropriate parameters have been selected, the computation is performed on the selected waveforms by pressing the **Compute** button at the bottom of the computation window. The computed waveform is inserted into the chosen time domain waveform viewer, unless the computation transforms time-domain data into frequency-domain data. In this case, the computed waveform is inserted into the chosen target frequency domain waveform viewer.

8.2 Waveform Filtering

A lowpass filter may be applied to a waveform to remove excessive noise that is present, or to simulate how a measured TDR waveform or simulated waveform would appear in response to a slower-risetime input signal. For technical discussion on the effective use of filtering, see “References” on page A-5.

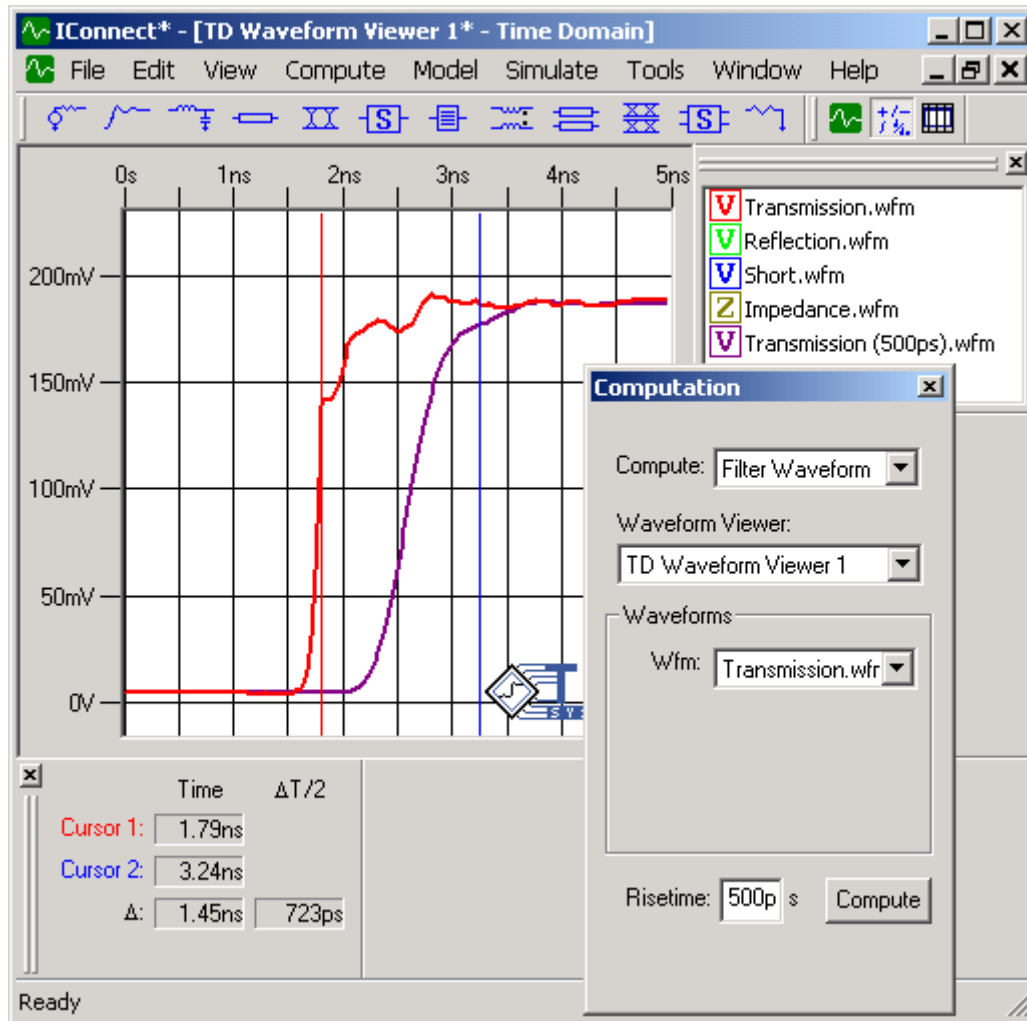


Figure 8-2: Transmission waveform that has been filtered to emulate a 500ps risetime.

Waveform filtering is executed from the computation window by choosing **Filter** from the **Compute** list box. It may be necessary to choose an operand waveform from the drop-down list box in the group titled **Waveforms**. It may also be necessary to enter a desired filter risetime in the parameter entry box at the bottom of the computation window. Press the **Compute** button, and the resultant waveform will be placed in the selected waveform viewer.

Note: Waveform filtering may be easily applied to a group of selected waveforms or to a single waveform from the waveform context menu, which is accessed by right-clicking on a waveform or on its name in the waveform legend window.

The applied filter is a gaussian pulse which, if applied to an ideal step, would result in a 10%-90% risetime equal to the number entered in the **Risetime** parameter entry box at the bottom of the computation window. The pulse is delayed by three times its standard deviation to avoid undesirable edge effects.

8.3 Impedance Profile Computation

The impedance profile of a device is its characteristic impedance as a function of distance along its length. Use of a post processing algorithm such as that implemented in IConnect is almost always necessary to get accurate impedance profile information from measured waveforms, since in raw TDR waveforms the effects of any multiple discontinuities are superposed on each other. Computation of the impedance profile gives the user access to information on the DUT which would otherwise be hidden.

The existence of an impedance profile assumes that the DUT can be modeled as a nearly lossless transmission line of non-uniform characteristic impedance. The impedance profile can be thought of as answering the question, "If this DUT can be modeled as a series of cascaded transmission line segments, what would be the impedance of each segment?" The nearly lossless assumption is a valid approximation in a large number of circuit board and packaging applications currently in use and development. For further technical discussion on impedance profile, the user is referred to "[References](#)" on page A-5.

In IConnect, the impedance profile of a DUT is computed from its response to a TDR step input, based on how a non-uniform transmission line would respond to such a step. For this reason, computation of the impedance profile requires both a TDR input step and a TDR response as parameters.

The impedance profile is computed via the computation window by choosing **Z-Line** from the **Compute** list box, or by simply pressing the red **Z** on the algorithms toolbar. It may be necessary to choose two operand waveforms from the drop-down list box in the group titled **Waveforms**. In the list box labeled **DUT:**, identify the waveform that is the reflection from the DUT. In the list box labeled **Step:**, identify the waveform which was the open-circuit or short-circuit response at the interface to the DUT. It may be necessary to change the value for the characteristic impedance of the measurement system in the parameter entry box labeled **Z₀**: at the bottom of the computation window, and it may also be necessary to adjust the threshold value in the parameter entry box labeled **Threshold**: at the bottom of the computation window. Press the **Compute** button, and the resultant waveform will be placed in the selected waveform viewer.

Calibration

You may also want to supply an optional 50Ω calibration waveform to improve the accuracy of the computations by calibrating out some launch discontinuities. This waveform is measured by attaching a 50Ω termination in place of the DUT and measure a TDR reflection waveform, and is subtracted from the DUT and reference waveforms by IConnect prior to computing the impedance profile. However, the Z-Line algorithm calibrates out many of the discontinuities even without a 50Ω calibration waveform, so in most cases supplying the extra waveform will not improve accuracy significantly.

Notes on Impedance Profile

The impedance profile is computed via a deconvolution technique, and as such requires accurate input data. If you experience difficulty, first check to ensure that you have used the correct waveforms as inputs to the algorithm. It is also important to be aware that because of the deconvolution nature of the algorithm, noise originally present in measured waveforms may be amplified, even to the point of algorithm breakdown. If your resultant waveform appears to be at all unstable, adjust the threshold value to a higher level. In many cases, this will be enough to achieve an accurate result. For further reading, see "[References](#)" on page A-5.

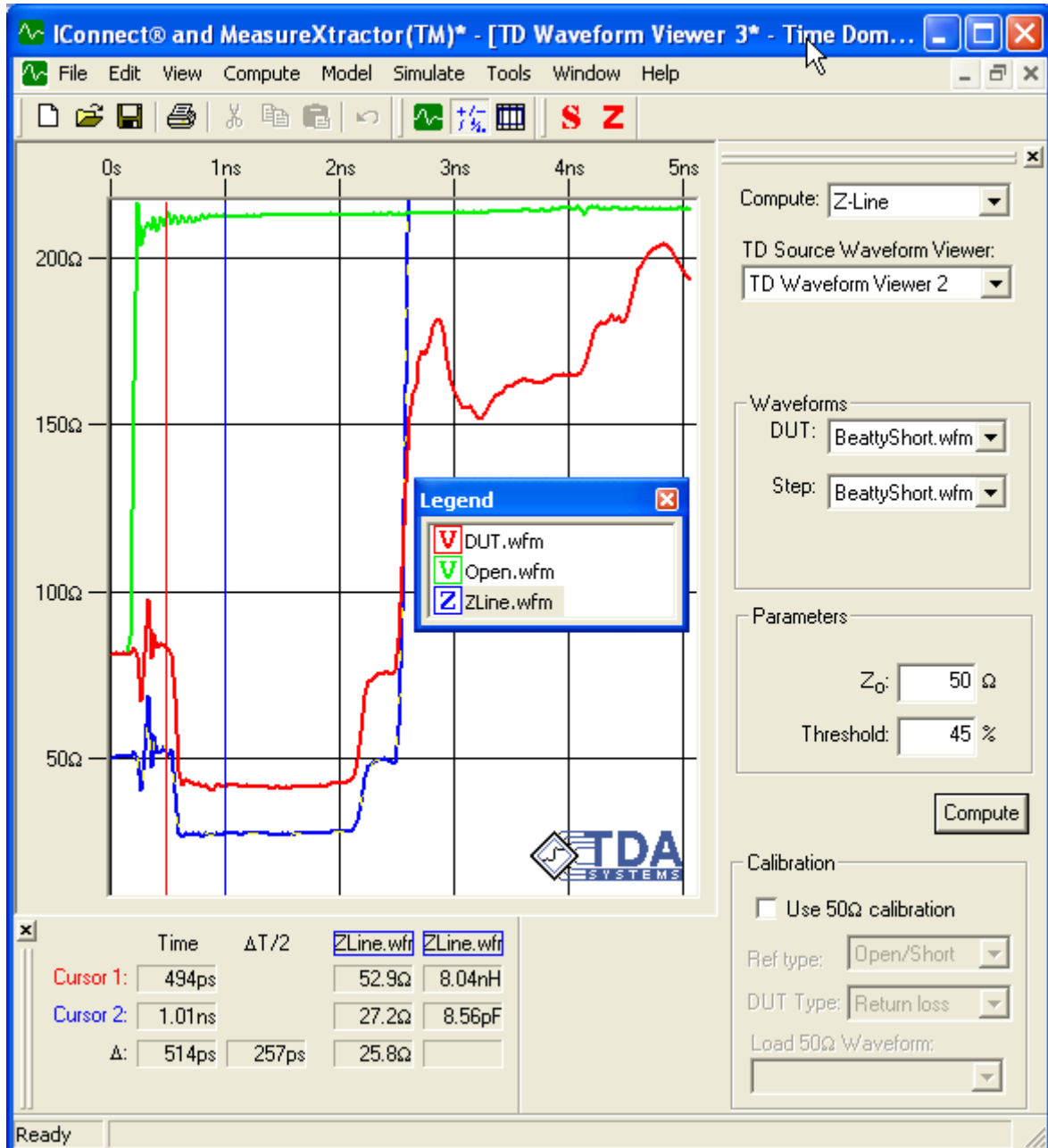


Figure 8-3: Calculating the impedance profile.

Threshold Parameter

The settable threshold parameter is a method for reducing the sensitivity of the computed impedance profile to measurement noise and error. It reduces the effects of time offsets and vertical noise present in the measured data so they do not have an artificially large effect on the computation. It is commonly set between 25% and 75%, depending on the noise level present in the measured data. Values lower than 25% may not have an appreciable effect, and values higher than 75% may damp out the signal along with the noise.

Typically a higher threshold setting is necessary when the risetime of the signal is fast compared to the sampling interval of the data, since this condition results in increased digitization error in the

Waveform Computations

signal transition, which can in turn affect the impedance profile computation. A higher setting may also be necessary if your signal launch has significant discontinuities.

8.4 Step Spectrum Computation

It is often useful to have information on the frequency content present in a signal used to excite a device under test. This information gives us an upper bound on the validity of any model extracted from the DUT response.

Mathematically speaking, the excitation and the corresponding response of the device are related in the frequency domain by $y(j\omega) = H(j\omega)x(j\omega)$, where $x(j\omega)$ is the excitation, $H(j\omega)$ is the system transfer function (such as scattering parameters) and $y(j\omega)$ is the response. From the equation, it is apparent that if there are frequencies at which $x(j\omega)$ is negligibly small, the measured response will contain little or no information about the DUT at those frequencies, and consequently we have no way of validating our model at those frequencies.

Step-like FFT

The fast fourier transform (FFT) can be used to compute samples of the spectrum of a time-limited signal. However, since a TDR step is not time limited, but instead continues indefinitely at steady state, special techniques must be utilized to get an accurate estimate of its frequency content. In effect, the waveform is differentiated such that it becomes time limited, and the FFT is computed. The resultant frequency-domain data, $h[k]$, is divided by the sequence

$$1 - e^{\frac{j2\pi k}{N}}$$

to regain the spectrum of the original signal. However, inspection of the equations reveals that the resultant value for $h[0]$ will be 0/0, and indeed the frequency spectrum of an ideal step waveform is $1/j\omega$, which is infinite at DC. For this reason, IConnect does not report a DC value for step spectra, but instead artificially sets the DC value to zero in an attempt to remind the user that this data point is invalid.

Readers interested in further details of FFTs of step-like signals are referred to [“References” on page A-5](#).

Performing the Computation

The step spectrum is computed from the computation window by choosing **Step Spectrum** from the **Compute** list box.

From the **Wfm**: drop-down list box in the group box titled **Waveforms**, choose an open or short circuit step to approximate the TDR waveform used to excite the DUT.

Once you select your source waveform, IConnect will attempt to choose appropriate default values for Δf , the sample interval of the computed frequency data, and F_{\max} , the maximum value of the computed frequency data. If you feel that you need to override the default values, you may do so by checking the **Set Manually** check box and entering the desired values for Δf and F_{\max} . Note that choosing a large value of F_{\max} and a small value of Δf can result in long computation times and high memory usage. The maximum allowed value of F_{\max} is also limited by the time step in

your source waveform by the equation $F_{\max} \leq \frac{1}{2\Delta T}$. If you choose an F_{\max} value which is too

high to meet this criteria, IConnect will inform you of the maximum value you may choose based on your source waveform.

Click the **Compute** button, and the step spectrum will be displayed in the frequency-domain waveform viewer designated in the **FD Target Waveform Viewer**: drop-down list box. If no frequency domain waveform viewer is currently open at the time the computation is performed, a new fre-

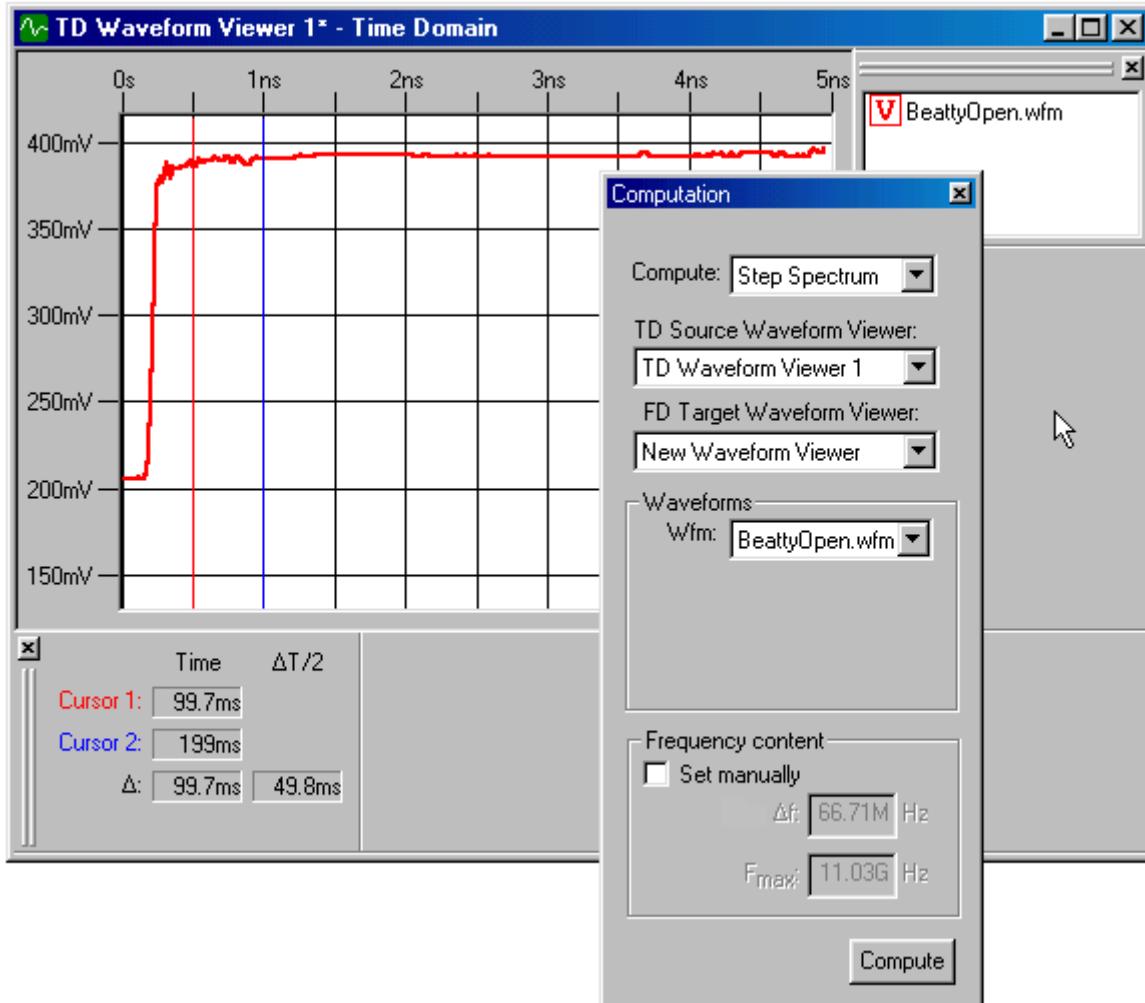


Figure 8-4: Computing the frequency-domain spectrum of an input step.

quency domain viewer will be opened with the step spectrum displayed in it. Note that this operation may be performed more readily by right clicking on the step waveform and choosing **Spectrum** from the resulting context menu.

8.5 S-Parameter Computation

For more information on how to compute S-Parameters, refer to [“S-parameter Computation Tutorial” on page 2-37](#).

Frequency-domain scattering parameters are a useful way to visualize the electrical properties of a system, and they are more recently being used to test compliance with certain interconnect standards. Although they have historically been measured in the frequency domain using a network analyzer, much of the same information is contained in TDR and TDT measurements which can be made for a fraction of the cost of network analyzer measurements. However, in order to obtain a frequency-domain representation of the data from TDR/T measurements, several signal processing steps are required.

Mathematically, the input and the output of the system are related in the frequency domain by

$$b(j\omega) = S(j\omega)a(j\omega)$$

where $a(j\omega)$ is the incident voltage step from the TDR, and $b(j\omega)$ is either the reflected voltage step or the transmitted voltage step. Transforming our time-domain measurements of a and b into the frequency domain and inverting the equation allows us to compute $S(j\omega)$, which is the scattering parameter element relating $a(j\omega)$ and $b(j\omega)$.

Note: Scattering-parameter computation is just as valid for TDT (through) measurements as it is for TDR (reflection) measurements. TDT results in insertion loss (S_{21}), TDR in return loss (S_{11}).

The observant reader will note that at frequencies at which $a(j\omega)$, the input signal, is zero, the equation is not invertible. In practice a measured value of a will rarely be identically zero; however, above a certain frequency it will typically be very small, and above this frequency the computed value for $S(j\omega)$ will be meaningless. (The frequency at which the input signal spectrum is too small to return accurate data is dependent on the risetime and shape of the input signal, although the effect can easily be observed qualitatively using the Step Spectrum computation capability of IConnect.) IConnect allows the user to select a cutoff frequency above which frequency domain samples are not computed, in order to keep noisy high-frequency data from obscuring the accurate low-frequency data.

Performing the Computation

The scattering parameter is computed from the computation window by choosing **S-Parameter** from the **Compute** list box.

From the drop-down list box titled **Ref:**, choose an open circuit step to approximate the TDR waveform used to excite the DUT. From the list box titled **DUT:**, choose the measured response of the DUT to the input step.

Once you select your source waveform, IConnect will attempt to choose appropriate default values for Δf , the sample interval of the computed frequency data, and F_{\max} , the maximum value of the computed frequency data. If you feel that you need to override the default values, you may do so by checking the **Set Manually** check box and entering the desired values for Δf and F_{\max} . Note that choosing a large value of F_{\max} and a small value of Δf can result in long computation times and high memory usage. The maximum allowed value of F_{\max} is also limited by the time step in

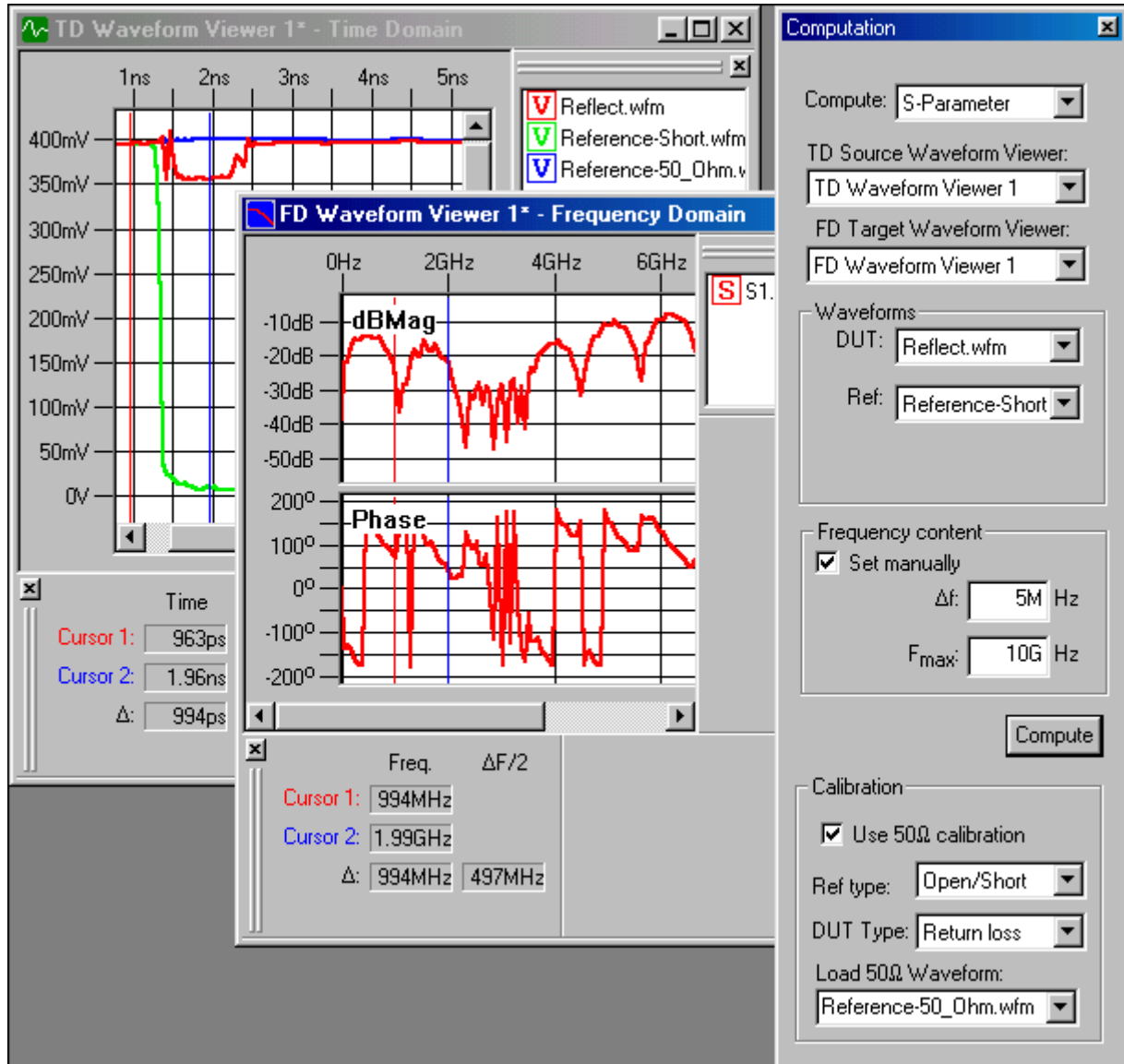


Figure 8-5: Computing the scattering response of a device based on the input and output TDR waveforms.

your source waveform by the equation $F_{max} \leq \frac{1}{2\Delta T}$. If you choose an F_{max} value which is too high to meet this criteria, IConnect will inform you of the maximum value you may choose based on your source waveform.

Click the **Compute** button, and the scattering response will be displayed in the frequency-domain waveform viewer designated in the **FD Target Waveform Viewer**: drop-down list box. If no frequency domain waveform viewer is currently open at the time the computation is performed, a new frequency domain viewer will be opened with the scattering response displayed in it.

Calibration

For additional information on calibration, see [“Calibration” on page 2-47](#).

IConnect allows you to perform a partial calibration at your discretion when computing S-parameters by supplying a matched 50Ω waveform. The waveform is measured by removing your DUT from your fixture and replacing it with a 50Ω load. By subtracting this waveform from your measured TDR waveforms prior to computation, IConnect removes certain fixturing and measurement equipment effects. Because it is not a complete calibration, users should be aware that the frequency to which their scattering parameter measurements will be accurate is limited by the frequency at which their fixturing becomes nonideal. However, quality cables and connectors are often accurate well past 12 GHz. It is important to be aware that if your fixturing setup is clean up to your frequency of interest, it may not be worth the effort to supply a calibration waveform, since you will not gain much by supplying the calibration waveform to IConnect.

Use 50Ω calibration

Check this box if you wish to supply a 50Ω matched waveform for calibration. Whether you need a calibration waveform will depend on how clean your fixture setup is at the highest frequency of interest.

Ref type:

IConnect will perform the calibration differently depending on which type of waveform you supplied as a reference. If you supplied an open-circuit or short-circuit TDR waveform as your reference waveform, select **Open/Short**. If you supplied a matched TDT waveform as your reference waveform, select **Thru**.

DUT type:

IConnect will perform calibration differently depending on which type of waveform you supplied as DUT. If you supplied a TDR waveform to measure DUT return loss, select **Open/Short**. If you supplied a TDT waveform to measure DUT insertion loss or crosstalk, select **Thru**.

Load 50Ω waveform:

Select your calibration waveform from this drop down box. Your calibration waveform should be a TDR waveform measured by replacing the DUT with a 50Ω termination.

8.6 Capacitance and Inductance Computation

IConnect is capable of computing lumped self- and mutual- capacitance and inductance terms by examining the behavior of TDR waveforms from $t=0$ to steady state. Some of the relevant mathematical derivation and measurement issues are discussed in ["References" on page A-5](#). This manual is limited to the procedural details of each operation.

The asymptotic capacitance and inductance computations are accessed via the computation window. The computation to be performed is chosen from the computation drop-down, which causes the drop-down list boxes and their labels in the **Waveforms** group to adjust for data entry. It may be necessary to choose a waveform viewer from which to choose the input waveforms for the computation. This viewer will also receive the resulting waveform once it is computed.

General tips for lumped inductance/capacitance computations

Wait for steady state. Each of these methods is based on the DC relationships between charge and voltage (capacitance) and flux and current (inductance). For this reason, it is necessary that all waveforms have reached a steady state value in order to get accurate results.

Use the correct reference waveform. Parasitic measurement requires information on what the ideal response would be – open circuit for self capacitance, short circuit for self inductance. Mutual measurements only require input information to determine the initial and final voltage and values.

Pay attention to quiescent nearby leads or lines, and the far ends of the lines under test. For capacitance measurements, the lead(s) in question should be open-circuited at the far end, and adjacent leads short-circuited, and vice versa for inductance measurements.

8.6.1 Self Inductance Computation

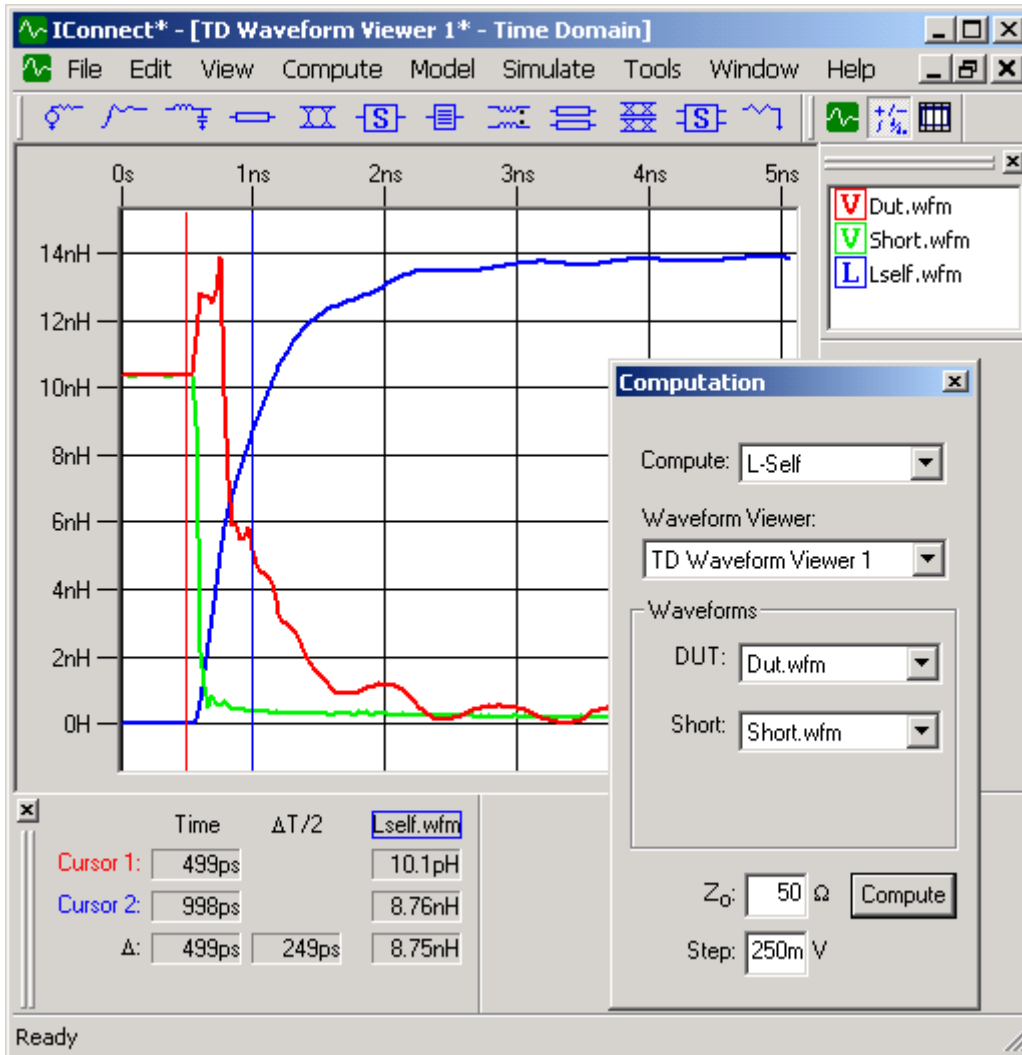


Figure 8-6: Self inductance computation.

In the computation window, choose **L-Self** as your computation. In the **DUT**: waveform choice, select the measurement of your DUT short-circuited at the far end. For the **Short**: waveform, select the reference waveform that corresponds to your measurement system shorted where the DUT would normally be placed. It may be necessary to enter the characteristic impedance and step amplitude for your measurement system prior to performing the computation.

Once these parameters have been chosen, pressing the **Compute** button performs the computation. The resultant waveform will be placed in the chosen waveform viewer and the inductance is measured as the final asymptotic level that the inductance waveform reaches. If the cursors are used to make the measurement, they read directly in units of Henries.

8.6.2 Self Capacitance Computation

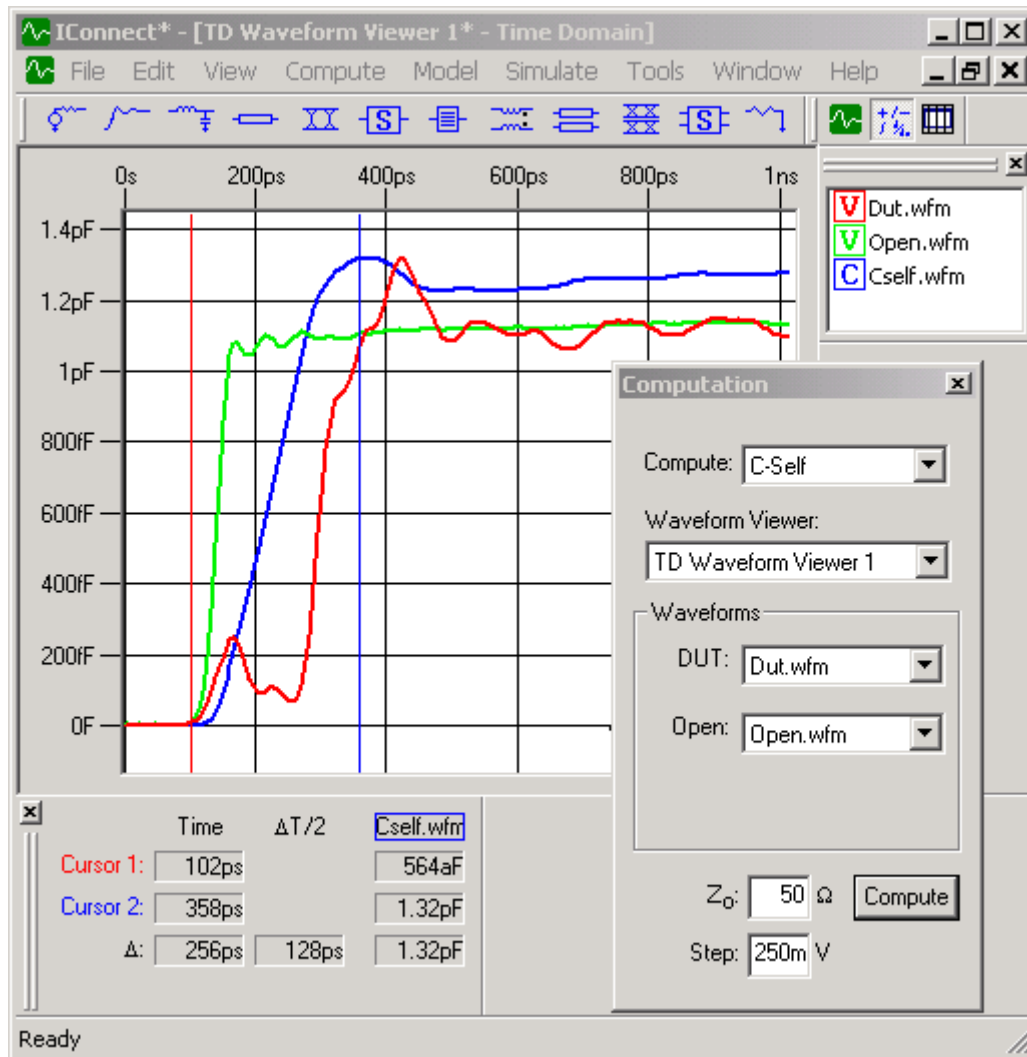


Figure 8-7: Self capacitance computation.

In the computation window, choose **C-Self** as your computation. In the **DUT:** waveform choice, select the measurement of the DUT open-circuited at the far end. In the **Open:** waveform option, select the reference waveform, which corresponds to the measurement system open-circuited where the DUT would normally be placed. It may be necessary to enter the characteristic impedance and step amplitude for your measurement system prior to performing the computation.

Once these parameters have been chosen, pressing the **Compute** button performs the computation. The resultant waveform will be placed in the chosen waveform viewer, and the capacitance is typically measured as the final asymptotic level that the capacitance waveform reaches. If the cursors are used to make the measurement, they read directly in units of Farads.

8.6.3 Mutual Inductance Computation

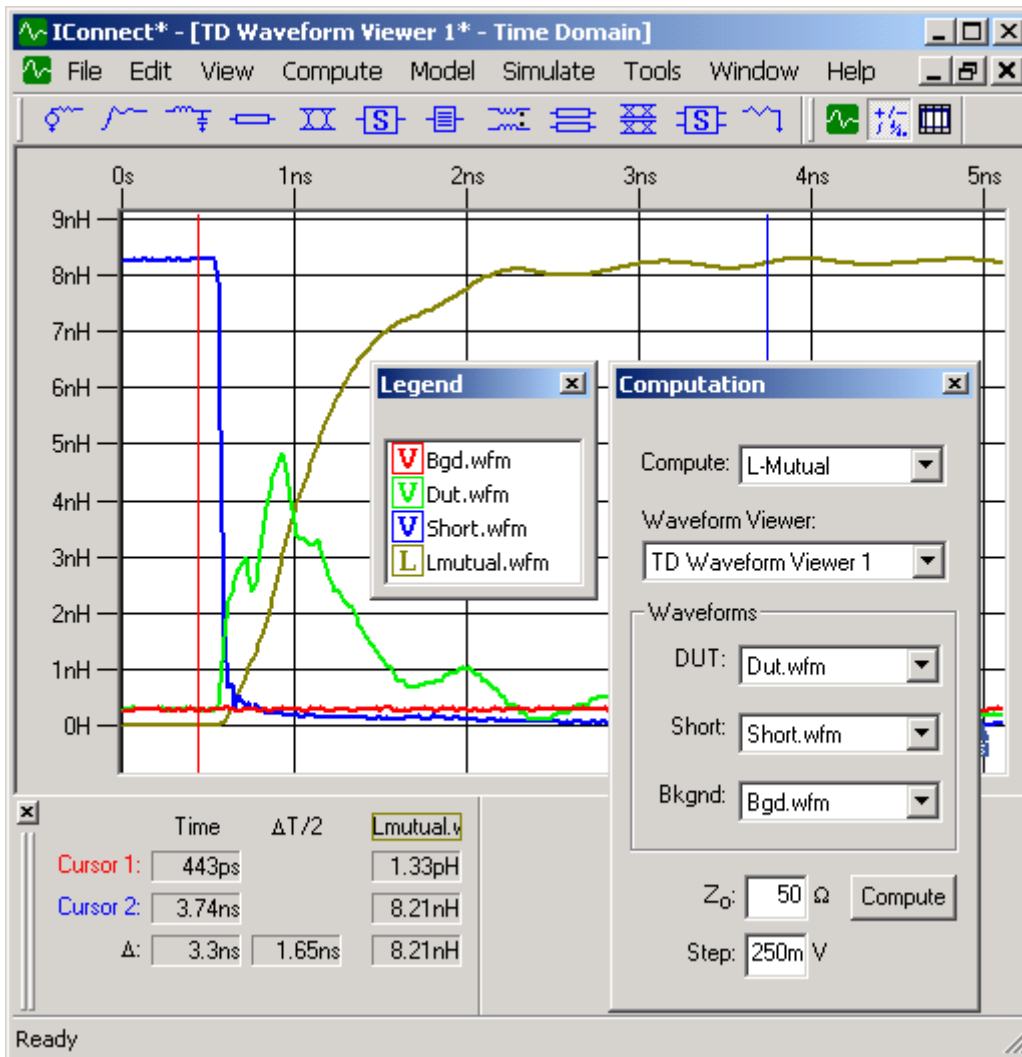


Figure 8-8: Mutual inductance computation.

In the computation window, choose **L-Mutual** as your computation. In the **DUT:** waveform option, select the measurement of the near-side crosstalk on the quiescent line with adjacent leads open at the near end, short-circuited at the far end. In the **Short:** waveform choice, select the waveform that corresponds to your measurement system shorted where the DUT would normally be placed. In the **Bkgnd:** waveform option, select the background waveform, which is acquired on the channel used to measure the DUT waveform when the DUT is disconnected. It may be necessary to enter the characteristic impedance and step amplitude for your measurement system prior to performing the computation.

Once the above parameters have been chosen, pressing the **Compute** button performs the computation. The resultant waveform will be placed in the chosen waveform viewer, and the inductance is typically measured as the final asymptotic level that the inductance waveform reaches. If the cursors are used to make the measurement, they read directly in units of Henries.

8.6.4 Mutual Capacitance Computation

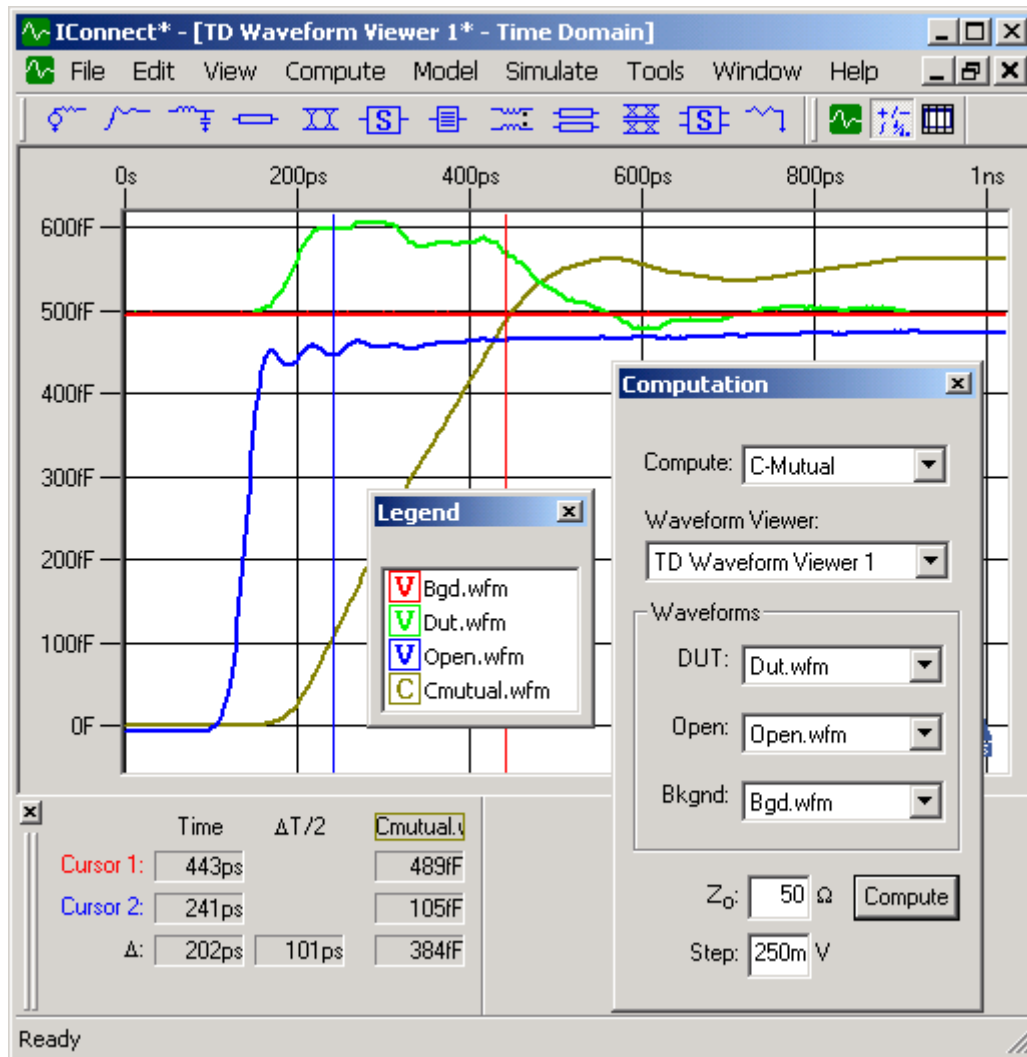


Figure 8-9: Mutual capacitance computation.

In the computation window, choose **C-Mutual** as your computation. In the **DUT:** waveform option, select the measurement of the near-side crosstalk on the quiescent line with adjacent leads shorted at the near end, open-circuited at the far end. In the **Open:** waveform choice, select the waveform which corresponds to your measurement system open-circuited, where the DUT would normally be placed. In the **Bkgnd:** waveform option, select the background waveform, which is acquired on the channel used to measure the DUT waveform when the DUT is disconnected. It may be necessary to enter the characteristic impedance and step amplitude for your measurement system prior to performing the computation.

Once the above parameters have been chosen, pressing the **Compute** button performs the computation. The resultant waveform will be placed in the chosen waveform viewer, and the capacitance is typically measured as the final asymptotic level that the waveform reaches. If the cursors are used to make the measurement, they read directly in units of Farads.

8.7 Flip Waveform

In certain applications it is useful to flip a waveform end for end. A typical application is one in which the impedance profile of a long line has been computed. If some of the detail at the far end of the line has been masked, it may be useful to make a reflection measurement from the far end of the line, compute the impedance profile, and flip the resulting impedance waveform. The flipped waveform may then be used to build a model of the far end of the line.

8.8 Waveform Math

IConnect is capable of performing addition, subtraction, multiplication, and division of two waveforms. It will also perform integration, differentiation, and filtering of a waveform. Since a constant may be substituted for one of the waveforms, waveform math also allows convenient scaling and offsetting of waveforms.

Waveform math is executed from the computation window by choosing **Waveform Math** from the **Compute** list box. It may then be necessary to choose the desired operator from the **Operator** drop-down list located in the **Waveforms** group. Depending on the nature of the operator, it may be necessary to choose either one or two operand waveforms in the remaining drop-down list boxes in the group. Press the **Compute** button, and the resultant waveform will be placed in the selected waveform viewer. To use a constant as one of your operands, choose **Constant** from one of the waveform selection boxes, and enter the value of the constant in the text box at the bottom of the Computation Window.

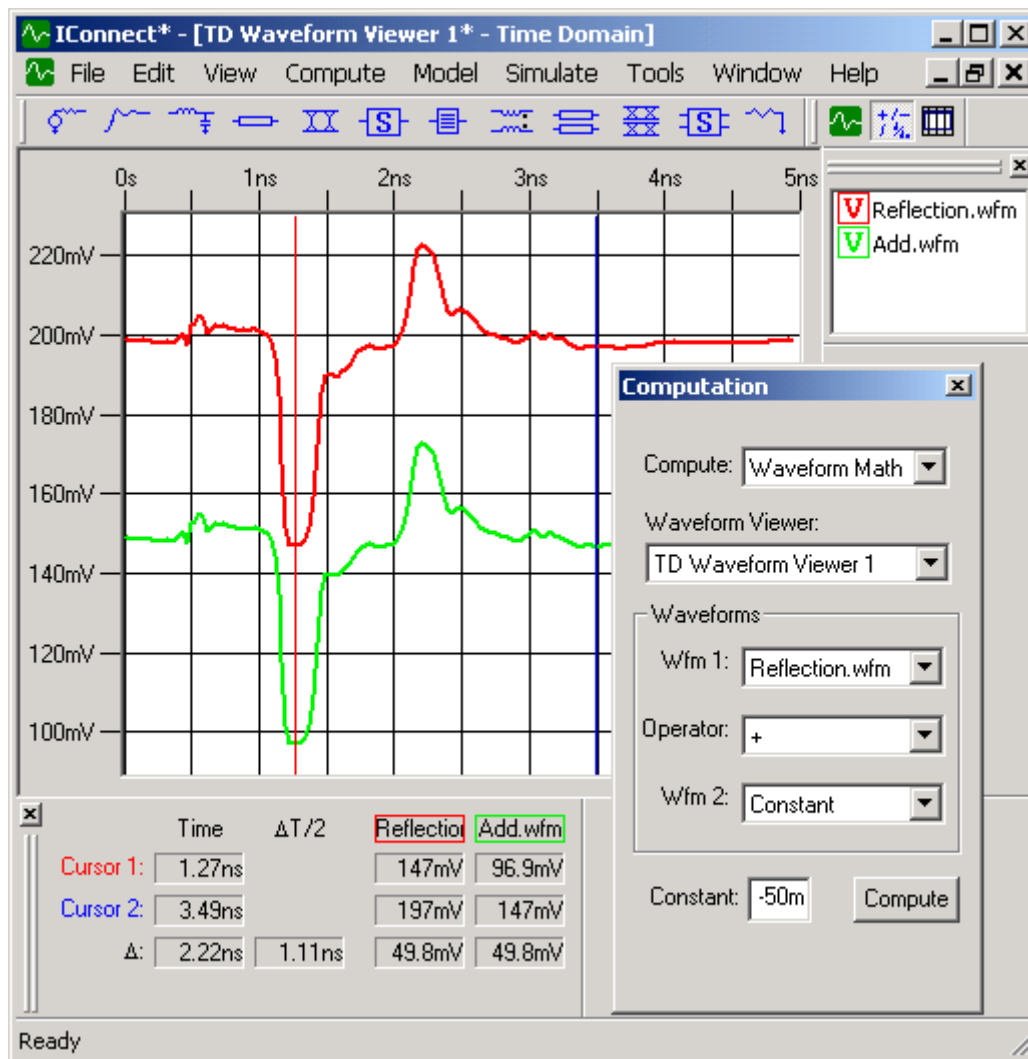


Figure 8-10: Reflection waveform that has been adjusted by -50mV.

Notes on integration and differentiation

Integration is performed as the running sum of the waveform data points, multiplied by the time step between data points. For this reason, the magnitude of the integrated waveform will typically be several orders of magnitude smaller than that of the original waveform.

Differentiation is performed as the difference between consecutive waveform data points, divided by the time step between data points. For this reason, the magnitude of the differentiated waveform will typically be several orders of magnitude larger than that of the original waveform.

Note: Differentiation attenuates low-frequency components and amplifies high-frequency components. Therefore, it is not uncommon for high-frequency noise to be visibly present in a differentiated waveform.



Chapter 9 Utilities

IConnect[®] and MeasureXtractor[™] TDR and VNA software comes with the ability to capture screen shots of windows to a file, the clipboard, or a printer. It also comes with the ability to easily send waveforms by E-mail, provided that you have Microsoft's Messaging Application Programming Interface (MAPI) installed.

Contents

9.1	Screen Capture	9-2
9.2	E-mail Utilities	9-4

9.1 Screen Capture

IConnect comes with its own screen capture capability for straightforward visual documentation of measurement and modeling results. Captured bitmaps may be easily transferred into any software tool that is capable of reading bitmaps either from the clipboard or a file. The captured bitmap may also be sent directly to a printer for output.

The screen capture window

The screen capture is accessed via the **Tools/Screen Capture...** command or from the screen capture icon on the auxiliary toolbar. You may need to select one or more parameters prior to performing your capture. The options are described below.

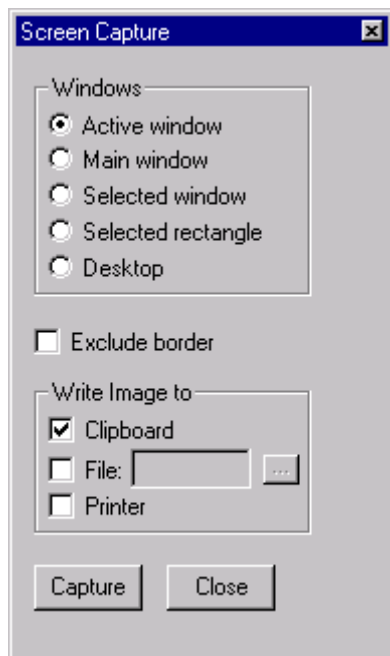


Figure 9-1: IConnect screen capture window.

Window:

Active window - the currently active model or waveform viewer window. If none is present, the main IConnect window is captured.

Main window - the main IConnect window and everything on top of it.

Selected window - any window on your computer screen. You will subsequently select the window with your mouse.

Selected rectangle - any area of your computer screen. You will subsequently select the area with your mouse.

Desktop - the entire area of your computer screen, including the taskbar if it is visible.

Exclude border - determines whether or not the border of the window will be included in the captured bitmap.

Write Image to:

Clipboard - image is written to the system clipboard.

File - image is written to the selected file. You may use the ... button to choose a file, or you will be prompted if no file is selected.

Printer - image is sent to the printer. You will be prompted for printer options after the capture takes place.

Capturing the window

Capture is initiated by pressing the **Capture** button at the bottom of the screen capture window. Depending on what type of capture you are performing, different steps must be taken.

If you are capturing the **Active Window**, **Main Window**, or **Desktop**, you simply press and release the **Capture** button to complete the capture.

If you are capturing a **Selected Window**, you must hold the mouse button down after pressing the **Capture** button, and release it over the window you want to capture. As you point across the screen with the button depressed, each window to be captured will be outlined. When you are above the desired window, release the mouse button, and the window will be captured.

If you are capturing a **Selected Rectangle**, you must hold down the mouse button after pressing the **Capture** button. When you get to a corner of the rectangle you want to capture, press the <Ctrl> key, continuing to hold down the mouse button. While the mouse button is still depressed, drag the pointer to the opposite corner of the rectangle and release the mouse button. Once the mouse button is released, the outlined rectangle will be captured.

9.2 E-mail Utilities

IConnect comes with built-in E-mail utilities for easily sharing data and models. These utilities are available via the File/Mail menu and they enable the user to send waveforms and SPICE models as E-mail attachments. To be able to send E-mail directly from IConnect, it is necessary to have Windows Messaging installed as part of your operating system. More information on Windows Messaging can be found in [“The MAPI interface” on page 9-4](#).

Sending Waveforms

To send one or more waveforms via electronic mail, select the waveforms within the waveform viewer and choose the **File/Mail/Selected Waveforms...** menu item. An E-mail window for your default mail program will pop up, with the selected waveforms included as attachments to the E-mail. You are free to edit the recipient list as well as the subject line and the body of the message before sending the message. The mail will be addressed by default to the E-mail address you have selected in the General program options, accessible via the **Tools/Options...** menu item.

To send a waveform viewer with its associated waveforms:

- Select all waveforms and send them.
- Send waveform viewer using File/Mail/Current Document menu command.

To receive a waveform viewer with its associated waveforms:

- Place all attachments in one directory.
- Open the waveform viewer file (*.wfv) in IConnect.

Sending a Model

To send a model via electronic mail, select the model's window so that it is active. Then go to the **File/Mail** menu and the document should appear as one of the menu choices. Choose the document name and a mail window for your default mail program will pop up with the model included as an attachment. You are free to edit the recipient list as well as the subject line and the body of the message before sending the message.

Note: Models that include waveforms as part of the model will not be sent in their entirety. The waveform must be sent separately. If the waveform is not sent, the model may be evaluated by a simulator, but may not be edited. Likewise, waveforms must be sent separately from the waveform viewer.

The MAPI interface

The E-mail connection is implemented via the Microsoft Messaging Application Programming Interface (MAPI), which allows end-user programs to manage electronic mail using a well-defined interface. Although it is not necessary that you use Microsoft products to send and receive E-mail, it is necessary to have the Microsoft Messaging option installed as part of your operating system. To determine whether Windows Messaging is installed on your system, check to see if the Inbox icon is present on your desktop. If it is absent, Windows Messaging has not been installed and the mail utilities of IConnect will be grayed out and inaccessible.

Windows Messaging is installed using the Add/Remove programs icon in the control panel. For further information, see the documentation shipped with your operating system, or contact TDA Systems.

Note: For further help on installation of Windows Messaging, use the keywords “e-mail” or “windows messaging” to search the operating system Help.

Using non-Microsoft E-mail Clients

If your preferred E-mail client is not Microsoft Exchange, you may still be able to use the IConnect E-mail utilities, provided that your mail implements the MAPI interface. The configuration of your mailer as the default MAPI client will vary from product to product. As an example, the following steps configure Netscape 6:

- Choose the **Edit/Preferences...** menu item.
- Choose the **Mail & Newsgroups** item in the Preferences window.
- Select **Use Netscape Mail as the default mail application**.
- Click **OK**.

Configuration approaches for other E-mail clients will vary, and are not always available. However, it is always possible to save files to disk and send them as E-mail attachments outside of the IConnect framework.



Chapter 10 User Options

IConnect[®] and MeasureXtractor[™] TDR and VNA software has many settable user options. They are accessible by choosing the **Tools/Options...** menu command, and clicking on one of the tabs: General, Computations, Simulation, or Waveform Viewer. The available options are described in the following sections.

Contents

10.1 General Options	10-2
10.2 Computation Options	10-3
10.3 Simulation Options	10-5
10.4 Waveform Viewer Options	10-7

10.1 General Options

Several general program options are set within IConnect. They are accessible by choosing the **Tools/Options...** menu command, and clicking on the **General** tab. The available options are described below.

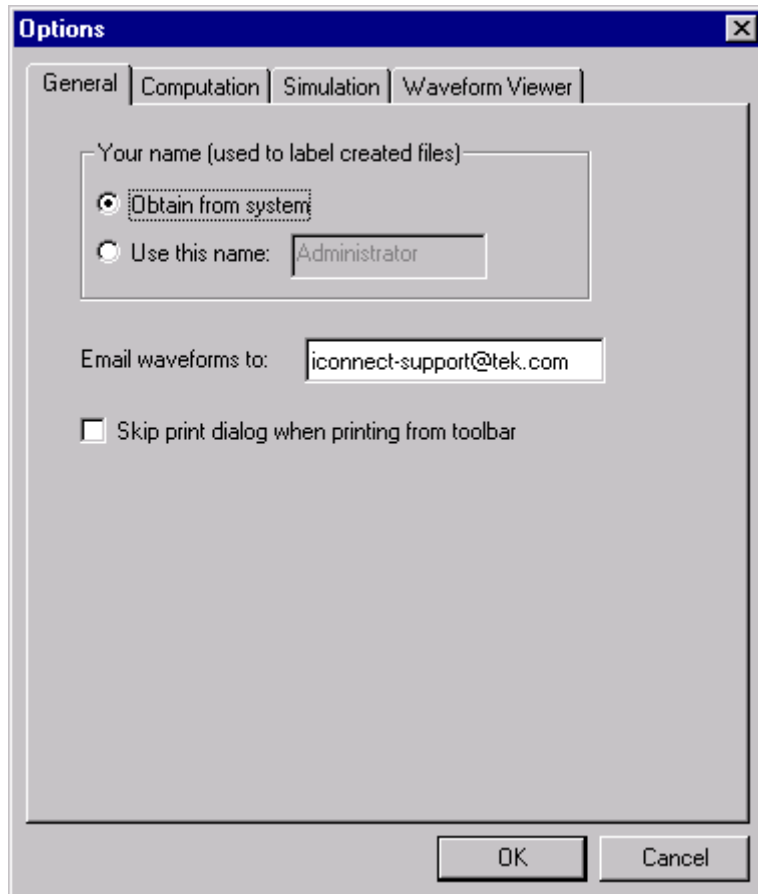


Figure 10-1: Setting general options via the Tools/Options menu command.

Your Name

IConnect uses your name to label files that you have created, so that different users may keep track of their work. You may choose to have IConnect obtain your name from the system, or you may enter a name yourself.

Email Waveforms To

You may enter an e-mail address to which IConnect will automatically address waveforms to be sent by email. Default is iconnect-support@tek.com.

Skip Print Dialog when Printing from Toolbar

Some people prefer that the print toolbar bypass the print setup dialog and print with the set of values used for the previous printing. If you prefer this behavior, check the box.

10.2 Computation Options

Several computation options are set within IConnect, to allow users to customize IConnect for their particular measurement setups and usage habits. The computation options are accessible by choosing the **Tools/Options...** menu command, and clicking on the **Computation** tab. The available options are described below.

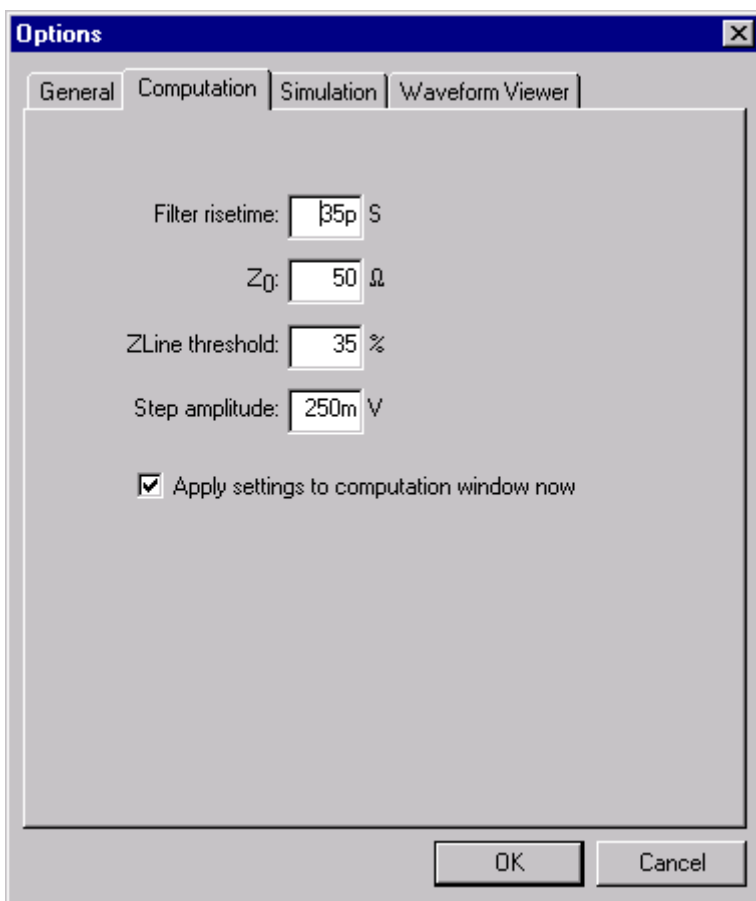


Figure 10-2: Setting computation options via the Tools/Options menu command.

Filter risetime

The filter risetime may be entered into the text box entitled **Filter risetime:**. This value will be the first to come up in the computation window when IConnect is started.

Z₀

The characteristic impedance of your measurement system may be entered in the text box labeled **Z₀:**. This allows IConnect to properly interpret voltage values measured from your scope for computing inductance, capacitance, and impedance profile.

Z-Line threshold

The default threshold value for the Z-Line algorithm may be entered in the text box labeled **Z-Line threshold:**. The threshold value is used to improve the stability of the algorithm by filtering out high-frequency noise.

Step amplitude

The step amplitude of your TDR oscilloscope may be entered in the text box labeled **Step amplitude**. The value entered is the amplitude of the step measured at DC at the end of an open-circuited cable of impedance Z_0 , which is half the value of the entire voltage swing output by the step generator itself. Nominal values are 200mV for Agilent scopes, 250mV for Tektronix scopes.

Apply settings to computation window now

Check this box if you wish for the settings to take effect immediately. Otherwise they will be preserved as defaults to be used the next time you start IConnect.

10.3 Simulation Options

Several simulation options are set within IConnect, both to configure the software to interface with the simulation engine, and to allow increased flexibility for the modeler. The simulation options are accessible by choosing the **Tools/Options...** menu command, and clicking on the **Simulation** tab. The available options are described below.

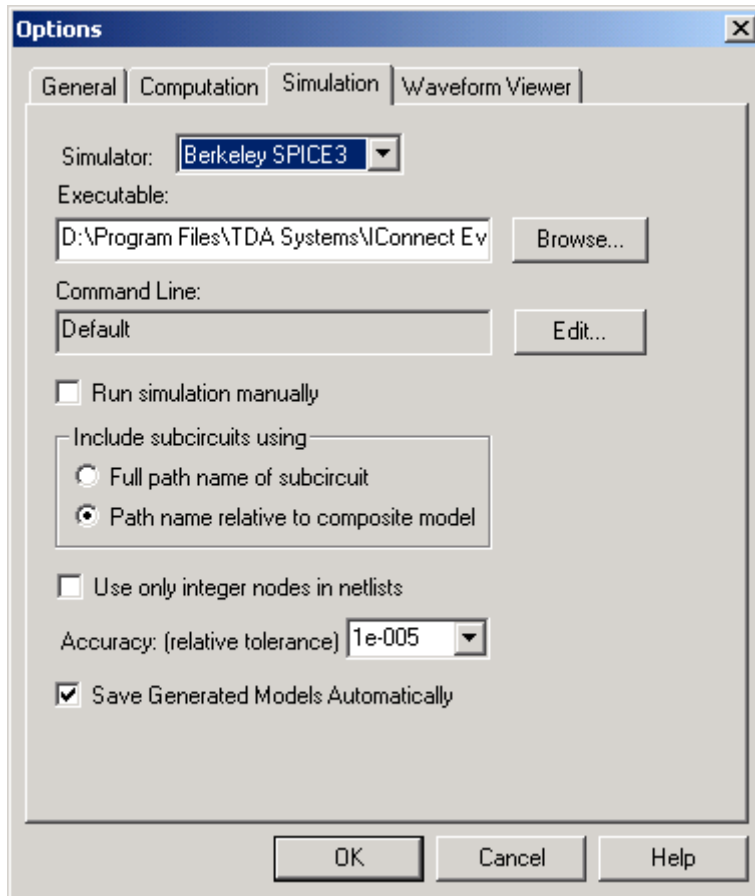


Figure 10-3: Setting simulation options via the Tools/Options menu command.

Simulator

The simulator may be chosen from the drop-down list box entitled **Simulator**. Currently supported simulators are PSpice from Cadence, HSpice from Synopsys, and Berkeley Spice 3F from the University of California. The chosen simulator determines what syntax is used to write model netlists.

Simulator executable

The path to the simulator executable may be typed in the box titled **Executable**, or selected using the **Browse** button. This setting allows IConnect to find the simulator to perform verification. The file must be accessible via your path or the simulation will not launch.

Command line

This is the command line that IConnect will submit to the operating system when you request it to run a simulation. For either of the three directly supported simulators running on your local machine, the default calling syntax should be correct. However, if you are running your simulation

on a remote computer or even running an unsupported simulator, you may have to edit this field by pressing the **Edit** button. Note that this field may call a script or other executable file if necessary. For more information on how the simulator is called, see [“Configuring IConnect to Communicate with your Simulator” on page 6-4.](#)

Run simulation manually

Instead of submitting your simulation job to the simulator, IConnect pauses and allows you to submit the simulation manually. After you inform IConnect that you have completed the simulation, it reads the simulation from the output file. This option is useful if you are running your simulation from a computer or network on which it cannot be launched directly or by a script from within IConnect.

Subcircuit include mode

You can have IConnect include your subcircuits into a composite model either by a fully qualified path name, including a drive letter, or by a path name relative to the directory containing the composite model. In most cases, it is advisable to use relative path names, since they allow a composite model and its included subcircuits to be relocated to another directory or computer without directly editing the composite netlist.

Use only integer nodes in netlist

Checking this box causes submodels to be saved using only integer nodes. This option should be set if your simulator treats numeric node names differently than it treats alphabetic node names. An example of such a simulator is HSpice, which considers node 1A to be identical to node 1. Using only integer nodes can negatively affect the readability of the model netlist, but ensures that the above ambiguity will never be encountered.

Accuracy (relative tolerance)

You may select or type the desired relative accuracy you would like your simulator to achieve for each of the circuit variables such as voltage, current, and charge. This number corresponds directly to the RELTOL option found in most SPICEs. Note that this setting applies to all subsequently saved composite models, so previously-saved composite netlists must be resaved after changing this option in order for the setting to apply to those simulations.

Save Generated Models Automatically

Before automatically validating your model, IConnect will save your files automatically (in the directory of your choice) as long as this box is checked. If you prefer the default to be that you manually save all your files before simulating, uncheck this box. For more information on automatic validation of models, see [“Configuring IConnect to Communicate with your Simulator” on page 6-4.](#)

10.4 Waveform Viewer Options

The waveform viewer may be customized by setting a variety of options that are accessed by invoking the **Tools/Options...** menu command and pressing the **Waveform Viewer** tab. The options are outlined below.

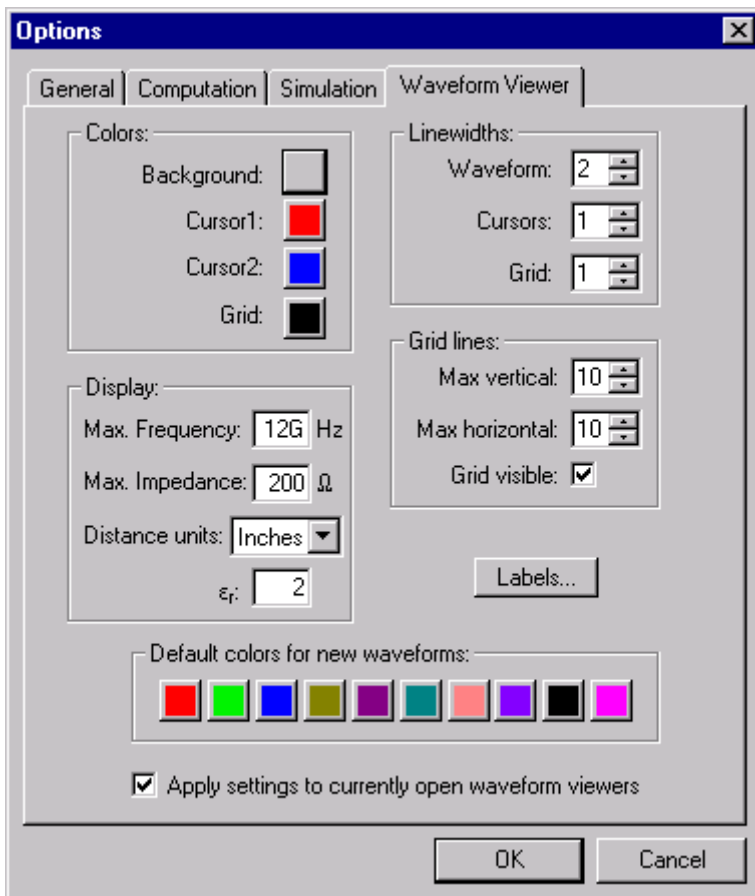


Figure 10-4: Setting waveform viewer options via the Tools/Options menu command.

Colors:	
Background	sets the background color for the graph
Cursor 1	sets the color for cursor 1
Cursor 2	sets the color for cursor 2
Grid	sets the color for the waveform grid

Linewidths: (given in pixels)	
Waveform	width of waveform line
Cursor	width of cursor lines
Grid	width of grid lines

User Options

Display:	
Max. Frequency	highest frequency for frequency-domain display
Max. Impedance	maximum impedance value plotted
Distance Units	units for distance display in x-axis
ϵ_r	relative dielectric constant for converting time to distance

Grid Lines:	
Vertical Lines	maximum number of vertical lines shown in grid
Horizontal Lines	maximum number of horizontal lines shown in grid
Grid Visible	toggles whether the grid is visible in the graph window

Labels	Default label properties. See "Using Labels in the Waveform Viewers" on page 7-11
--------	---

Default Colors for New Waveforms:	These are the ten colors assigned in that order to waveforms added to a waveform viewer. After each set of ten waveforms, IConnect cycles through the colors again.
-----------------------------------	---

Apply settings to currently open waveform viewers:	If box is unchecked, settings apply only to newly opened waveform viewers. If this box is checked, settings also apply to currently open waveform viewers.
--	--

Appendix 1 Troubleshooting

Overview

If you encounter difficulties while using IConnect® and MeasureXtractor™, the topics below may help you quickly find a solution. If you are unable to find the help you need, please do not hesitate to [contact TDA Systems](#). You may also visit the [TDA Systems web site](#) and read our FAQ list.

If there are additional topics or issues you would like to see addressed in this section, please let us know.

Installation problems

Installation doesn't start automatically

If the setup program does not automatically start when you place the IConnect installation CD in the CD reader, you may have to go to explorer, go into the CD, and double click on **Setup.exe**.

Sentinel hardware key not recognized or will not install

If you encounter problems with the installation of the Sentinel hardware key or its drivers, contact TDA Systems for further assistance. Occasional incompatibilities have been experienced with certain hardware configurations.

Problems Viewing On-line Documentation

Unable to view HTML Help

If you are running Windows 95 or NT4.0 with Internet Explorer 3.0 or earlier, you should check to see if the file IConnect.chm exists in the Help directory.

Note: If you chose the default installation options, the directory will likely be C:\Program Files\TDA Systems.

If you cannot find the file IConnect.chm, then you should manually install the HTML Help Viewer as follows:

1. Using Windows Explorer or a similar application, go to the directory in which you installed IConnect. If you chose the default installation options, the directory will likely be C:\Program Files\TDA Systems.
2. From there, choose the directory IConnect\Help.
3. Double-click on the Microsoft applet file hhupd.exe, which stands for "HTML Help Update"
4. Answer yes to the first dialog, and HTML Help will be updated.

Note: Microsoft states that this installation step is not necessary if you are running Windows 98, ME, 2000, or XP.

Unable to View IConnect PDF Manual

If you are unable to view the IConnect PDF Manual, then you need to install Adobe Acrobat Reader 4.0 or later. We recommend installing version 6.0, which can be found on the IConnect installation CD in the PDF Manual folder.

GPIB Card not recognized

Verify that your card is from National Instruments. IConnect only supports the use of National Instruments GPIB cards.

Ensure that your GPIB card is seated and that the drivers have been correctly installed. It may be helpful to download the most recent version of your drivers.

In some situations it may be necessary to resolve hardware conflicts over interrupt lines or I/O mapping. The GPIB drivers are typically configured via an applet in the control panel.

TDA Systems is happy to help with issues that arise during GPIB installation. However, it may be necessary on occasion to refer you to National Instruments customer support.

Oscilloscope connection will not work

If you are experiencing difficulty connecting your oscilloscope, the following steps may be helpful.

Ensure that your GPIB interface is correctly installed, and that it passes the manufacturer's diagnostic test.

It may be necessary to reset the scope GPIB address, or reconfigure IConnect to communicate using the current scope address.

If your scope is in local lockout mode, IConnect will not be able to communicate with it.

Simulation

Simulator not seen or simulation will not launch

If your simulation fails to launch, it may be necessary to reconfigure IConnect to inform it of the location of your simulator executable. This may be done via the **Tools/Options...** menu item.

If your simulation launches and either your simulator reports an error or IConnect is unable to read in the results of your simulation, refer to [“Simulation errors encountered”](#) below.

Simulation errors encountered

IConnect implements numerous checks to ensure the veracity of your models and simulation setup prior to invocation of the simulator. However, due to differences between simulators and other external factors, it may be necessary at times to troubleshoot a failed simulation.

View the output listing. The first step in troubleshooting a simulation is to choose the **Simulate/View Output File** menu item, which brings up the SPICE output listing in a text editor. (Or you may conveniently right click on the composite model in question and choose **View Output File**.) Scan this output listing for errors. If the error messages are difficult to understand, it might be helpful to consult your simulator documentation. Although TDA Systems cannot

guarantee that we can resolve problems occurring with simulations, in most cases we can, and are happy to assist you if necessary.

Common simulation errors. It should be noted that the most common error in a simulation involves the simulator not being able to locate a model file which was to be included in the simulation. Often this occurs because either the included file or the including file has been moved since the last time the simulation was run. It may be necessary to locate the file and copy it back to the proper directory before running the simulation. For this reason it is not advised to move submodel files or composite model files with respect to each other.

It is also possible to introduce simulation errors by directly editing the SPICE netlist. Although users are not discouraged from editing the netlists by hand, it should be done with caution by someone familiar with SPICE simulation, and only after saving a copy of the file as a backup.

Another common error is trying to submit a composite model for HSpice simulation that has spaces in its name. The HSpice command line parser does not handle filenames with spaces.

Please let us know!

If any other simulation errors occur, please report them to TDA Systems. It is our intent that IConnect trap the majority of such errors before submitting the simulation job to the simulator.

Printing

Print Preview looks poor

Print preview will not operate properly unless you have a printer driver installed on your computer.

Appendix 2 References

The following reference documents can be found at the Tektronix web site by going to

http://www.tek.com/products/oscilloscopes/sampling/interconnect_analysis/index.html

and clicking on **Technical Information** to the right under **Produce Resources**.

Application Notes

Modeling of Infiniband Cable Assembly from Measurements. Appnote analyzing an effective approach of InfiniBand cable assembly modeling from measurements.

Effects of Measurement Fixtures of S-Parameters Compliance Testing of Infiniband Cable Assemblies. TDNA is the most cost-effective and fast way to de-embed fixture effects and perform compliance testing for standards such as PCI-Xpress, Serial ATA, Infiniband, and Gigabit Ethernet.

Modeling of Gigabit Backplanes from Measurements. A detailed procedural document on Signal Integrity modeling and analysis of gigabit backplanes.

Electronic Package Fault Isolation Using TDR. A comprehensive appnote on package FA and fault isolation. *To be published in 5th Edition of EDFAS Microelectronic Failure Analysis Desk Reference.*

TDR and VNA Measurement Primer. A quick overview of the basics of TDR and VNA measurements and a comparison between the two.

A Hybrid Measurement and Field Solver Approach for the Design of High-Performance SMA Transitions. A verifiable design methodology that enables leading edge performance from off-the-shelf components.

Cost-efficient Cable Assembly Compliance Testing and Modeling. Save money by using a single instrument for all your cable assembly compliance testing and modeling needs.

Using Measure-Based Modeling to Analyze Backplane Deterministic Jitter. Understand and separate the contribution of different deterministic jitter components.

Complete Methodology for Signal Integrity Analysis of Gigabit Interconnects. A short overview of measurement-based signal integrity analysis methodology. *A re-print of our article in the December 8 issue of Electronic Engineering Times.*

Simple Approach to Modeling of Power Delivery Networks and Components. The TDR/T approach for PDN analysis which produces models which will accurately represent the PDN behavior in both time and frequency domains.

Lossy Line Simulation and Analysis. A comprehensive discussion about the different methods for evaluation of lossy transmission line signal integrity effects in simulation.

Signal Integrity Modeling of Gigabit Backplanes, Cables and Connectors Using TDR. A complete TDR-based methodology for gigabit interconnect signal integrity analysis.

Measurement of Input and Output Die Capacitance for M-LVDS and Other Signal Standards. Use this technique to measure your Cin and Cout for M-LVDS, PCI Express, Infiniband, Rapid IO, HyperTransport, Gigabit Ethernet and other standards.

References

TDR Test Primer. A detailed introduction to TDR measurements and analysis. *Published in Printed Circuit Design Magazine, April 2002.*

Choosing Signal Integrity Measurement Tools: Time or Frequency Domain? *Presented at DesignCon2002, Paper HP-7.*

Signal Trace and Power Plane Shorts Fault Isolation Using TDR. *Presented at International Symposium for Testing and Failure Analysis 2001.*

Practical Characterization of Lossy Transmission Lines Using TDR. *Based on presentation at DesignCon 2001.*

SCSI Connector and Cable Modeling from TDR Measurements. *SCSI Signal Modeling Study Group presentation.*

TDR Techniques for Characterization and Modeling of Electronic Packaging. Comprehensive application note of package modeling techniques. *Published at HDI Magazine in March and April 2001 in two parts.*

Use TDR and Frequency Domain Analysis to Ensure Signal Integrity in Microprocessor Motherboards and Memory Modules. *Reprint of our article in March 2001 issue of Printed Circuit Design Magazine.*

Improved Method for Characterizing and Modeling Gigabit Flex-Circuit Based Interconnects. *Presented at DesignCon 2001.*

Disk Drive Flexible Interconnect Characterization Using TDR. *Reprint of our article in June 2000 issue of Electronic Design News Magazine.*

Characterization of Differential Interconnect from TDR Measurements. *Reprint of our article in March 2000 issue of Microwave Journal.*

Electronic Package Measurements and Failure Analysis Using TDR. *Presented at International Symposium for Testing and Failure Analysis 2000*

PCB Interconnect Characterization from TDR Measurements. *Reprint of our article from the May 1999 issue of Printed Circuit Design Magazine.*

Technical Briefs

S-parameters, Insertion, and Return Loss Measurements Using TDR Oscilloscope. Discussion about what single ended and differential S-parameters are and how to properly measure them. Also, download our S-parameter Measurement Quick Tour.

Eye Diagram Measurements Using TDR Oscilloscope Transmission Data. The easiest way to obtain information about eye diagram degradation due to interconnect performance at 10Gbit and beyond.

Computing Self and Mutual Capacitance and Inductance Using Even and Odd Mode TDR Measurements. A novel technique for computing self and mutual inductance and capacitance based on differential and common mode TDR measurements

Remotely Launching Unix Spice from within IConnect. This technical brief discusses interfacing from IConnect to HSpice via network for model validation

Presentations

You will find that some material in these presentations overlap; we recommend that you download one or two that best suit your application.

[Time Domain Network Analysis: Getting S-parameters from TDR/T Measurements.](#) *TDA Systems' Presentation at the Infiniband PlugFest, August 2004*

[Cost-Efficient Cable Assembly Compliance Testing and Modeling for Serial ATA and PCI Xpress.](#) *TDA Systems' presentation at Intel Technology Symposium / IDF.*

[Electrical Characterization of Serial ATA Interconnection at Gigabit Speeds](#) by Michael Cheong, Molex, and Dima Smolyansky, TDA. Both Serial ATA specification testing and Molex cable assembly modeling are discussed.

[Lossy Line Characterization and Modeling for SPICE and IBIS.](#) *IBIS Summit Presentation, January 28, 2002.*

[High-Speed Interconnect Measurements and Modeling Using TDR.](#) *Presented at DesignCon2002, TecForum HP-TF5.*

[FibreChannel Interconnect Signal Integrity Measurement and Modeling Methodologies.](#) *Presented at FibreChannel T11 Modeling Committee meeting, June 6, 2001.*

[IC Package Measurements, Modeling and Failure Analysis Using TDR.](#) *Presented at Sematech Assembly Analytical Forum in Chandler, AZ, January 20, 2000.*

[High-Speed Digital Interconnect Modeling Using TDR.](#) *DesignCon2000 presentation, Santa Clara, CA, February 2, 2000.*

[SCSI Connector and Cable Modeling from TDR Measurements.](#) *SCSI Signal Modeling Study Group presentation.*

[Connector model Extraction Based on Differential TDR.](#) *Presentation on connector and differential line modeling at EIA IBIS Summit Meeting, Marlborough, MA, October 14, 1999. It can also be downloaded from <http://www.eda.org/pub/ibis/summits/oct99/corey.zip>.*

[Direct Rambus Signal Integrity Measurements.](#) *Presented at Hewlett Packard Users Group Meeting, San Diego, CA, June 1999.*

Quick Guides

[TDR Interconnect Modeling Quick Guide.](#) A basic guide to interconnect modeling from TDR measurements. Understand the TDR measurements and select the right circuit topology for the device under test.

[Interconnect Probing Quick Guide.](#) If you are new to interconnect measurements, review this guide to find out about the basics of interconnect probe and probing system selection. Determine whether probing or fixturing (or both) is the right approach to your interconnect device characterization.

References

Also See

[TDR Measurement and Modeling FAQ page](#). Questions and answers about TDR, VNA and IConnect® measurement and modeling methodology. If you can't find an answer to your question, submit it to iconnect-support@tektronix.com.

Index

Numerics

8510 4-5

A

Accuracy 6-2

Acquisition 4-9

Acquisition window 4-9

Agilent 8510 4-5

C

Calibration

 S-parameters 2-46, 2-47, 8-12

 Z-line 8-6

Capacitance 8-14

 mutual 8-18

 self 8-16

Color

 waveform viewer, setting 10-7

Composite model 5-59

 creating 5-59

Computation options 10-3

Computation window 8-2

Configuring IConnect for measurement .. 4-6

Configuring your E-mail client to work with

 IConnect 9-4

Crosstalk

 discussion 5-11

 in eye diagram 5-71

CSV

 import/export 7-7, 7-18

 supported formats 7-19

Cursors 7-9

D

Data-driven models 2-97, 5-44

Dielectric constant, setting 10-8

differential lines 5-19

Distance

 plotting against 7-14

 setting display units 10-8

E

E-mail

 options 10-2

 utilities 9-4

Epsilon, setting 10-8

Even mode 5-19

Excel

 import/export 7-18

Exporting data 7-17

Eye diagram 2-19, 5-65

 analysis 5-74

 computation 5-66

 crosstalk 5-71

 mask testing 5-77

 measurements 5-74

 options 2-28, 5-68

 zooming 2-31, 5-75

F

Failure analysis 2-10

Filtering waveforms 8-4

Flip Waveform 8-19

Frequency

 setting maximum in display ... 7-16, 10-8

Frequency content 2-46

G

General options 10-2

GPIB

 installation 4-3

 troubleshooting A-2

GPIB Commands

 sending to instrument 4-11

Grid

 controlling visibility 10-8

 setting colors and linewidths 10-7

H

Hardware key

 troubleshooting A-1

HP8510 4-5

HSpice 2-89, 2-90, 6-3, 6-6, 10-6

 custom simulation commands 6-5

 simulation error A-3

I

IBIS 2-54

Impedance

 setting maximum in display 10-8

Impedance profile 2-8, 8-6

Importing data 7-17

 troubleshooting 7-20

Inductance	8-14
mutual	8-17
self	8-15
Input step	5-6
Insertion loss	2-37
Installation	
troubleshooting	A-1
Installation notes	1-6
Instrument selection	4-9
Instrument settings	
restoring	4-11
saving	4-11

J

JEDEC	2-54
Jitter	2-35
measurement	2-27, 2-33

K

Keyboard shortcuts	1-8
--------------------------	-----

L

Labels	7-11, 7-12
default properties	10-8
Lossy Line model	2-80
Lumped Coupled model	5-11

M

Mask testing	2-32
Measurement	4-1
eye diagram	2-33
overview	4-2
tips	4-13
MeasureXtractor	2-97, 5-44, 5-52, 5-54
Model	
extraction	5-3
Model extraction tips	5-4

O

Odd mode	5-19
Oscilloscope connection	
troubleshooting	A-2
Oscilloscope setup	4-5

P

Partition window	5-80
------------------------	------

Partitions	
creating	5-80
deleting	5-81
selecting	5-81
Plotting	
vs. distance	7-14
vs. time	7-14
Print preview	
troubleshooting	A-3
PSpice	6-3
custom simulation commands	6-5
PWL Source model	5-6

R

References	A-5
Restoring instrument settings	4-11
Return loss	2-37
Risetime	
coupled lines model discussion	2-71
lossy line model discussion ..	2-20, 2-25, 2-34
setting filter default	10-3
setting for an eye diagram	2-28
setting for filter	2-79, 8-4
single line model discussion	2-71
touchstone import	5-46

S

Saving instrument settings	4-11
Scale	
setting manually	7-9
Screen capture	9-2
Sending data by E-mail	9-4
Sending instrument commands	4-11
Shortcuts, keyboard and mouse	1-8
Simulation	5-63
controlling error	10-6
running manually	10-6
timing	5-63
troubleshooting	A-2
viewing output listing	5-64
Simulation options	6-4, 10-5

- Simulator
 - configuring IConnect 6-4
 - installation 6-3
 - specifying command-line syntax 10-5
 - specifying executable 10-5
 - specifying subcircuit handling 10-6
 - specifying syntax 10-5
- Single Line model 2-63
- S-Parameters
 - computation 2-37, 2-45, 8-11
- Spectrum
 - step-like computation 8-9
- SPICE
 - controlling error 10-6
 - model creation 2-63
 - using only integer nodes 10-6
- Step amplitude
 - setting the default 10-4
- Symmetric Coupled Lines model . 2-63, 5-19

- T**
- TDA Systems
 - contact info 1-9
- TDS/CSA8000 and TDS/CSA8200
 - oscilloscope setup 4-5
 - running locally 4-4
- Termination model 5-58
- Time
 - plotting against 7-14
- Touchstone
 - import/export 7-21
 - importing 2-99, 2-110
- Troubleshooting A-1

- V**
- Verification 6-2
 - tips 6-8
 - with multiple simulators 6-6
- Virtual scope
 - configuring 4-7
- Voltage ramp 5-5
- Voltage step 5-5

- W**
- Waveform computations 8-2
- Waveform file filter example 4-7
- Waveform viewer
 - using 7-2
- Waveform viewer options 10-7
- Waveforms
 - context menu 7-4
 - controlling visibility 7-9
 - deleting 7-9
 - filtering 8-4
 - flipping 8-19
 - integration and differentiation 8-21
 - loading and saving 7-6
 - math operations 8-20
 - moving 7-8
 - offsetting 8-20
 - saving with a comment 7-9
 - scaling 8-20
 - selecting 7-4
 - setting color 7-9
 - setting properties 7-9
 - zooming 7-9

- Z**
- Z0
 - setting the default 10-3
- Z-Line
 - computation 8-6
 - setting the default threshold 10-3
- Zooming 7-9
 - in eye diagram 2-31, 5-75
 - in partition-based model window 5-81

