InfiniiSim Waveform Transformation Toolset Software

KEYSIGHT TECHNOLOGIES

User's Guide

Notices

Revision

Version 11.30.00203

Edition

July 2022

Available in electronic format only

Published by:
Keysight Technologies, Inc.
1900 Garden of the Gods Road
Colorado Springs, CO 80907 USA

Trademarks

Infineon is a registered trademark of Infineon Technologies AG. The Infiniium mark is used with the kind approval of Infineon Technologies, AG.

MIPI® service marks and logo marks are owned by MIPI Alliance, Inc. and any use of such marks by Keysight Technologies is under license. Other service marks and trade names are those of their respective owners.

PCI-SIG®, PCIe®, and the PCI Express® are US registered trademarks and/or service marks of PCI-SIG.

Warranty

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

Technology License

The hardware and/or software described in this document are furnished under a license and may be used or copied only in accordance with the terms of such license.

U.S. Government Rights

The Software is "commercial computer software," as defined by Federal Acquisition Regulation ("FAR") 2.101. Pursuant to FAR 12.212 and 27.405-3 and Department of Defense FAR Supplement ("DFARS") 227.7202, the U.S. government acquires commercial computer software under the same terms by which the software is customarily provided to the public. Accordingly, Keysight provides the Software to U.S. government customers under its standard commercial license, which is embodied in its End User License Agreement (EULA), a copy of which can be found at www.keysight.com/find/sweula. The license set forth in the EULA represents the exclusive authority by which the U.S. government may use, modify, distribute, or disclose the Software. The EULA and the license set forth therein, does not require or permit, among other things, that Keysight: (1) Furnish technical information related to commercial computer software or commercial computer software documentation that is not customarily provided to the public; or (2) Relinquish to, or otherwise provide, the government rights in excess of these rights customarily provided to the public to use, modify, reproduce, release, perform, display, or disclose commercial computer software or commercial computer software documentation. No additional government requirements beyond those set forth in the EULA shall apply, except to the extent that those terms, rights, or licenses are explicitly required from all providers of commercial computer software pursuant to the FAR and the DFARS and are set forth specifically in writing elsewhere in the EULA. Keysight shall be under no obligation to update, revise or otherwise modify the Software. With respect to any technical data as defined by FAR 2.101, pursuant to FAR 12.211 and 27.404.2 and DFARS 227.7102, the U.S. government acquires no greater than Limited Rights as defined in FAR 27.401 or DFAR 227.7103-5 (c), as applicable in any technical data.

Safety Notices

CAUTION

A **CAUTION** notice denotes a hazard. It calls attention to an operating procedure, practice, or the like that, if not correctly performed or adhered to, could result in damage to the product or loss of important data. Do not proceed beyond a **CAUTION** notice until the indicated conditions are fully understood and met.

WARNING

A WARNING notice denotes a hazard. It calls attention to an operating procedure, practice, or the like that, if not correctly performed or adhered to, could result in personal injury or death. Do not proceed beyond a WARNING notice until the indicated conditions are fully understood and met.

Contents

1 InfiniiSim Waveform Transformation Toolset

NOTE

You must have either the D9010DMBA De-embedding Software or the D9020ASIA Advanced Signal Integrity Bundle installed on your oscilloscope to have access to these features.

The De-embedding Software version of InfiniiSim lets you use only one block in your measurement/simulation circuits (building these circuits is discussed in other InfiniiSim topics) while the Advanced Signal Integrity version contains all of the InfiniiSim capabilities.

See Also If you prefer to view a PDF version of this help, see *InfiniiSim User's Guide*.

KEYSIGHT TECHNOLOGIES

What is InfiniiSim?

InfiniiSim is a software tool that, given measurement and simulation circuit models, generates transfer functions (in the frequency domain) that are used to derive finite impulse response (FIR) filters (in the time domain) that, when applied to the measured signal, can:

- Remove the effects of a structure that *is* in the signal path to the oscilloscope (de-embedding).
- Add the effects of a structure that *is not* currently in the path to simulate what the signal would look like if the structure *was* in the path (embedding).
- Simulate observation at a different point than where signal is acquired (virtual probing).
- Simulate loading of a structure (for example, a probe).
- Simulate crosstalk.
- Perform any combination of the above.

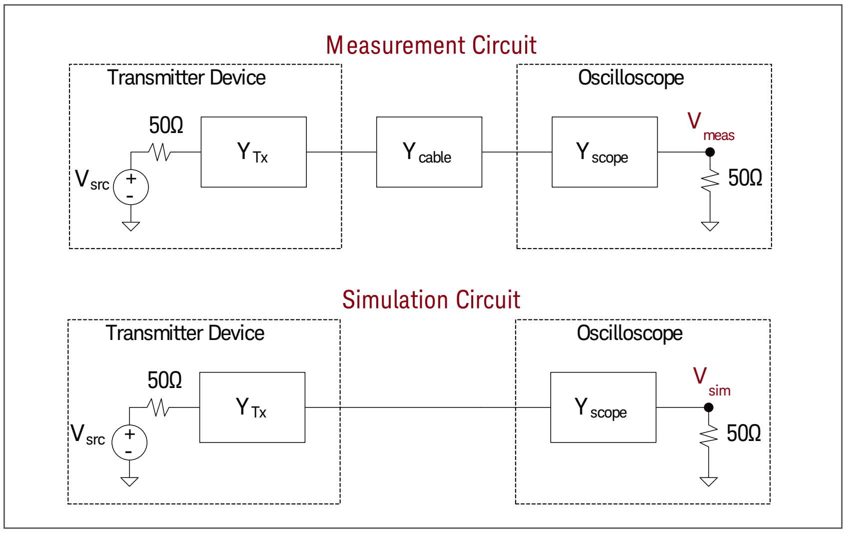
The combination of real-time oscilloscope waveform measurements with circuit simulations is called *co-simulation*: the merging of measurement with simulation.

Next: "Measurement and Simulation Circuit Models" on page 8.

Measurement and Simulation Circuit Models

InfiniiSim generates transfer functions by comparing two circuit models:

- **Measurement Circuit** — This model describes all circuit elements that are truly in place at the time of the measurement and the actual probe point in that circuit, also known as the *measurement observation node*.
- **Simulation Circuit** — This model describes the desired circuit and the desired probe point in that circuit, also known as the *simulation observation node*.



Assuming linearity and time invariance, the difference between the two circuits and their observation nodes is calculated to yield an overall transfer function. On this transfer function, an inverse Fourier transform is performed to yield an impulse response. The impulse response is sampled at the sampling rate of the oscilloscope to yield an FIR filter that is convolved with acquired waveform to render the simulated waveform. See "Transfer Function Derivation" on page 135.

NOTE

In applying the transfer functions, FIR filter coefficients are generated and downloaded to an FPGA in the front end of the Infiniium oscilloscope to enable real-time generation (through convolution) with the measured waveform. This direct connection to hardware control is the reason that the InfiniiSim graphical user interface (GUI) is under the Channel setup dialog box and not an independent analysis application.

Next: "Application Presets for Typical Circuit Topologies" on page 9.

Application Presets for Typical Circuit Topologies

When describing measurement and simulation circuit models, InfiniiSim provides application presets for typical circuit topologies. The application presets vary from a simple one-block model to the most complex 9-block model. All application presets are special cases of the generalized 9-block model with some blocks unused.

In more complex application presets, you are able to locate the measurement and simulation observation nodes.

The application presets for InfiniiSim Basic and InfiniiSim Advanced are:

- **None** (Basic and Advanced)
- **Add insertion loss of a fixture or cable** (Basic and Advanced)

- **Remove insertion loss of a fixture or cable** (Basic and Advanced)
- **Remove scope input reflection** (Advanced)
- **Add all effects of a fixture or cable** (Advanced)
- **Remove all effects of a fixture or cable** (Advanced)
- **Replace one channel element with another** (Advanced)
- **Relocate the observation node of a measurement** (Advanced)
- **Remove loading effects of a probe** (Advanced)
- **Remove loading effects of a DDR interposer and probe** (Advanced)
- **Relocate the observation node of a probed measurement** (Advanced)
- **General purpose 3 port, 6 port** (Advanced, 3 port when using 2-Port InfiniiSim, 6 port when using 4-Port InfiniiSim)
- **General purpose probe** (Advanced)
- **General purpose 6 blocks** (Advanced)
- **General purpose 9 blocks** (Advanced)

For more information on application presets, see:

- "Application Presets (2 Port)" on page 26
- "Application Presets (4 Port)" on page 52

Next: "Defining Circuit Blocks" on page 10.

Defining Circuit Blocks

Each block in a circuit model can be defined in several ways, and blocks can be independently defined in any circuit topology. Circuit elements can be defined using:

- RLC models (resistor, inductor capacitor)
- transmission line models
- shorts (ideal thru)
- opens
- S-parameters

The measurement and simulation circuit definitions are turned into models that are used to generate transfer functions. See "InfiniiSim Circuit Modeling" on page 109.

Next: "Transfer Function Analysis" on page 10.

Transfer Function Analysis

For analysis of generated transfer functions, InfiniiSim provides:

- A *frequency domain response* view to illustrate gain characteristics versus frequency.
- An *impulse response* view of the resulting time domain filter.
- A *step response* view of the resulting time domain filter.

To use InfiniiSim effectively, you must critically assess these portrayals for "reasonableness", and be ready to enter into the "design it, analyze it" loop several times. Depending on the transfer function analysis, you may want to limit the filter bandwidth (see "FIR Filter Bandwidth" on page 149) or adjust the filter length (see "Understanding FIR Filter Length" on page 151).

Return: Chapter 1, “InfiniiSim Waveform Transformation Toolset,” starting on page 7.

InfiniiSim Prerequisites

Circuit models are required

In analyzing a circuit for de-embedding or embedding purposes, you must know the attributes of, and have models (either S-parameter files or idealized components) for the circuits to be analyzed. Fundamental to this is whether the co-simulations are to be of 2-port or 4-port networks, and whether the signals evaluated for waveform measurements are applied directly to the oscilloscope channels or are probed.

- For 2-port InfiniiSim, one channel of the oscilloscope is used, and it can receive the signal directly (front panel input) or from a single-ended (SE) probe.
- For 4-port InfiniiSim, you can have:
  - Two single-ended outputs to be analyzed by direct connection to the oscilloscope (for example, channels 1 and 3).

    In the single-ended case, you have the option of observing the transformed single-ended channels, or transformed differential and common mode versions of these when the differential channels mode is on.
  - One differential probe (for instance, a differential SMA probe head).

Choices in input selection (2-port vs. 4-port, single-ended vs. differential, and probed vs. direct) greatly affect the modeling and the InfiniiSim attributes chosen.

Minimum S-parameter attributes are required

If S-parameter files will be used, they should be evaluated for proper attributes:

- The S-parameter sets must have a maximum frequency higher than the maximum bandwidth of interest.
- The S-parameter resolution (step size) should be linear and small enough to capture the longer time constants present in the circuit impulse response. See "Uniformly Spaced Values and Good Frequency Resolution" on page 114.
- The S-parameter frequency responses must not exhibit quickly moving magnitude and/or phase responses. Responses that are not sampled well enough are nearly impossible to discern after the measurement and will give rise to time domain aliasing.
- Responses must have a wide range of magnitude over bandwidth of interest, for example, >20 dB.

See Also

- "InfiniiSim S-Parameter File Requirements" on page 113
- "S-Parameter and Transfer Function File Viewers" on page 127

Starting InfiniiSim

For a quick overview of the InfiniiSim Waveform Transformation Toolset, see "What is InfiniiSim?" on page 8.

To enter the InfiniiSim application:

1 Open the Channel dialog box. This dialog box can be accessed by:

- Choosing **Setup > Channel N...** where **N** is **1**, **2**, **3**, or **4** depending on the channel you want to use InfiniiSim on.
- Clicking the channel buttons located directly above the waveform window.

NOTE You can also perform InfiniiSim on a differential or common-mode channel by selecting the **Differential** check box near the top of the Channel dialog box before proceeding to the next step. This is called differential channels mode.

2 After opening the Channel dialog box, in the **InfiniiSim** section, select the InfiniiSim mode:

- **2 Port** — Select this mode when measurement and simulation circuits are constructed from 2-port network model blocks, and the simulation analysis will be performed on a single oscilloscope channel.

  Typically, you select this mode when a single-ended signal is being probed by an oscilloscope channel or when a differential signal is being converted to a single oscilloscope channel input by a differential probe.

NOTE

It is possible to use 2-port simulation analysis on differential signals or common-mode signals independently; however, the analysis will not include cross-coupling effects between the differential and common-mode signals.

- **4 Port (two channel)** — Select this mode when measurement and simulation circuits are constructed from 4-port network model blocks, and the simulation analysis will be performed on oscilloscope channel pairs. For example, when the **Differential Partner** is set to **Every Other Channel**, the two channels can be **1 & 3** or **2 & 4**.

  Typically, you select this mode when differential signals are being probed by two single-ended probes or cables on two oscilloscope channels.
- **4 Port (one channel)** — Select this mode when measurement and simulation circuits are constructed from 4-port network model blocks, but the simulation analysis will be performed on a single oscilloscope channel. The single channel can be the current channel, a differential channel (for example **1 - 3** when differential channels are turned on), or a common mode channel (for example **1 + 3** when differential channels are turned on). In this mode, you must identify which ports of the 4-port transfer function to extract and apply to the channel.

  Typically, you select this mode when differential signals are probed by differential or common mode probes, when you want to ignore cross-coupling effects, or when you want to simulate crosstalk.

For more information on when you would select the different InfiniiSim modes, see "InfiniiSim Setup Options and Uses" on page 18.

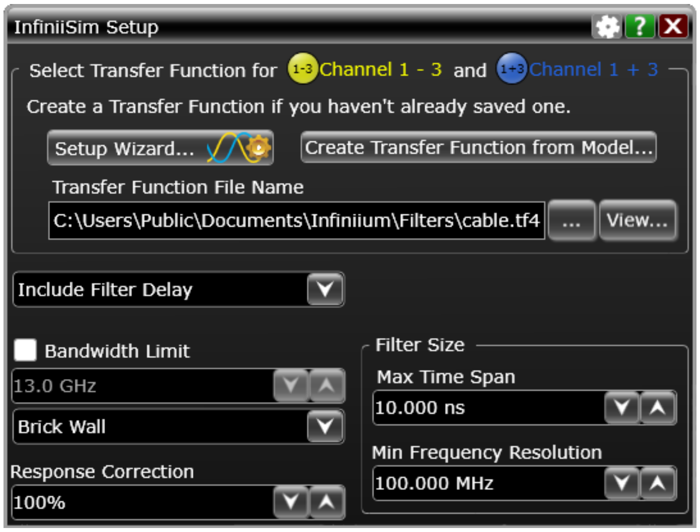
3 If you selected the 4-Port one-channel InfiniiSim mode, you must use the **Port Extraction** control to specify how to properly model your setup. Remember, this is a transfer function port extraction.

- **Use Ports 1 -> 2** — the port 1 to port 2 path of the standard 4-port transfer function.
- **Use Ports 3 -> 2** — the port 3 to port 2 path of the standard 4-port transfer function.
- **Use Ports 3 -> 4** — the port 3 to port 4 path of the standard 4-port transfer function.
- **Use Ports 1 -> 4** — the port 1 to port 4 path of the standard 4-port transfer function.
- **Differential** — the differential path of the mixed-mode 4-port transfer function.
- **Common Mode** — the common-mode path of the mixed-mode 4-port transfer function.

See "Diff Ch Mode=Off, 4 Port (One Channel), Port Extractions" on page 143.

4 If you want to enable or disable the InfiniiSim Plots (see page 79) display, select or clear the **Display InfiniiSim Graphs** check box.

5 Once you have selected the InfiniiSim mode, click **Setup...** to open the InfiniiSim Setup dialog box.



6 If you already have a transfer function created, you can apply it by using the **Transfer Function File Name** field to browse and select the file. The **View...** button opens the InfiniiSimToolbox application for viewing and evaluating transfer function files. See "S-Parameter and Transfer Function File Viewers" on page 127.

If you need to create a transfer function from measurement and simulation circuit models:

- If you are simply adding or removing the insertion loss of a fixture or a cable and would like a wizard to guide you through the process, click **Setup Wizard...**.

  For more information, see "Using the InfiniiSim Wizard" on page 21.

  If you would like to use a more complex application preset than adding or removing insertion loss of a fixture or cable, you can still use the wizard; however, measurement and simulation circuit block setup will have to be performed manually after exiting the wizard.

- If you would like to go through the transfer function file creation process manually without guidance from the wizard, click **Create Transfer Function from Model...**.

  For more information, see:

  - "Creating a Transfer Function From a Model (2 Port)" on page 25
  - "Creating a Transfer Function From a Model (4 Port)" on page 51

Once the transfer function file is created and/or selected, InfiniiSim performs an inverse Fourier transform on it to yield an impulse response. The impulse response is sampled at the sampling rate of the oscilloscope to yield an FIR correction filter that is convolved with acquired waveform to render the simulated waveform.

7 After the transfer function file is created and/or selected, use the remaining controls in the InfiniiSim Setup dialog box to:

- Include or remove the FIR filter delay.
- Apply bandwidth limiting.
- Change the FIR filter's size (length).
- Remove (normalize) any DC gain of the transfer function (when modeling probes).
- Scale the response correction applied to non-DC frequency components of the measured signal.

For more information on these controls, see "InfiniiSim Setup Dialog Box" on page 156.

Show Raw Channels Option

Notice in the Channel dialog box, there is a box labeled **Show Raw Channels**. When you use InfiniiSim, apply filters, or use differential channels, the signal displayed on the oscilloscope may be very different from the signal entering the front end of the oscilloscope. However, the oscilloscope triggers on the signal at the front end, not the signal you see on screen. Also, the signal entering the front end is the one that enters the ADC (analog to digital converter). Therefore, if you are having problems triggering or if you want to ensure you are using the entire vertical range, but not overdriving the ADC, you need to see this front-end signal. Checking this **Show Raw Channels** box turns off all of the transformations or filters applied to the signal and shows you what the signal entering the oscilloscope looks like. You can then make the necessary adjustments and uncheck this box to go back to the signal you were viewing.

See Also

For more information about the theory and operation of InfiniiSim, see:

- "InfiniiSim Circuit Modeling" on page 109
- "S-Parameter Considerations" on page 113
- "InfiniiSim Transfer Functions" on page 134
- "InfiniiSim FIR Filters" on page 149

InfiniiSim Setup Options and Uses

The following table shows:

- InfiniiSim options for channels 1 and 3 and Differential Channels settings
- InfiniiSim math function equivalents
- Inputs and outputs
- The uses for various InfiniiSim options

Table 1 InfiniiSim Setup Options and Uses

Ch	Diff Mode	InfiniiSim Option	Port Extraction Option	InfiniiSim Math Function	Simul. (SnP -> TFn)	Out File	Ch(s) In	Ch Out	Uses	Notes
ch1	off	2 Port		InfiniiSim 2 Port	2-port	.tf2, std	ch1	ch1	se xfer fn with se probe or cable 2-port diff xfer fn with diff probe 2-port cm xfer fn with cm probe	
		4 Port (Channels 1 & 3)		InfiniiSim 4 Port Src1	4-port	.tf4	ch1, ch3	ch1	4-port xfer fn with two se probes or cables	
		4 Port (Channel 1)	Use Ports 1 -> 2	InfiniiSim 4 Port 1 Src	4-port	.tf4	ch1	ch1	4-port xfer fn, ignore cross effects	Does not include 3 -> 2
			Use Ports 3 -> 2	InfiniiSim 4 Port 1 Src	4-port	.tf4	ch1	ch1	4-port xfer fn for xtalk sim	Does not include 1 -> 2
			Use Ports 3 -> 4	InfiniiSim 4 Port 1 Src	4-port	.tf4	ch1	ch1	4-port xfer fn, ignore cross effects	Does not include 1 -> 4
			Use Ports 1 -> 4	InfiniiSim 4 Port 1 Src	4-port	.tf4	ch1	ch1	4-port xfer fn for xtalk sim	Does not include 3 -> 4
			Differential	InfiniiSim 4 Port 1 Src	4-port	.tf4	ch1	ch1	4-port xfer fn with diff probe	
			Common Mode	InfiniiSim 4 Port 1 Src	4-port	.tf4	ch1	ch1	4-port xfer fn with cm probe	

Table 1 InfiniiSim Setup Options and Uses (continued)

Ch	Diff Mode	InfiniiSim Option	Port Extraction Option	InfiniiSim Math Function	Simul. (SnP -> TFn)	Out File	Ch(s) In	Ch Out	Uses	Notes
ch3	off	2 Port		InfiniiSim 2 Port	2-port	.tf2, std	ch3	ch3	se xfer fn with se probe or cable 2-port diff xfer fn with diff probe 2-port cm xfer fn with cm probe	
		4 Port (Channels 1 & 3)		InfiniiSim 4 Port Src2	4-port	.tf4	ch1, ch3	ch3	4-port xfer fn with two se probes or cables	
		4 Port (Channel 3)	Use Ports 1 -> 2	InfiniiSim 4 Port 1 Src	4-port	.tf4	ch3	ch3	4-port xfer fn, ignore cross effects	Does not include 3 -> 2
			Use Ports 3 -> 2	InfiniiSim 4 Port 1 Src	4-port	.tf4	ch3	ch3	4-port xfer fn for xtalk sim	Does not include 1 -> 2
			Use Ports 3 -> 4	InfiniiSim 4 Port 1 Src	4-port	.tf4	ch3	ch3	4-port xfer fn, ignore cross effects	Does not include 1 -> 4
			Use Ports 1 -> 4	InfiniiSim 4 Port 1 Src	4-port	.tf4	ch3	ch3	4-port xfer fn for xtalk sim	Does not include 3 -> 4
			Differential	InfiniiSim 4 Port 1 Src	4-port	.tf4	ch3	ch3	4-port xfer fn with diff probe	
			Common Mode	InfiniiSim 4 Port 1 Src	4-port	.tf4	ch3	ch3	4-port xfer fn with cm probe	

Table 1 InfiniiSim Setup Options and Uses (continued)

Ch	Diff Mode	InfiniiSim Option	Port Extraction Option	InfiniiSim Math Function	Simul. (SnP -> TFn)	Out File	Ch(s) In	Ch Out	Uses	Notes
ch1	on	2 Port		InfiniiSim 2 Port	2-port	.tf2, diff	ch1 - ch3	ch1 - ch3	2-port diff xfer fn	Does not include cm -> diff
		4 Port (Channels 1 & 3)		InfiniiSim 4 Port Diff	4-port	.tf4	ch1 - ch3, (ch1 + ch3)/ 2	ch1 - ch3	4-port xfer fn with two se probes or cables	
		4 Port (Channel 1 - 3)	Differential	InfiniiSim 4 Port 1 Src	4-port	.tf4	ch1 - ch3	ch1 - ch3	4-port xfer fn with two se probes or cables	Does not include cm -> diff
ch3	on	2 Port		InfiniiSim 2 Port	2-port	.tf2, cm	(ch1 + ch3)/ 2	(ch1 + ch3)/ 2	2-port cm xfer fn	Does not include diff -> cm
		4 Port (Channels 1 & 3)		InfiniiSim 4 Port CM	4-port	.tf4	ch1 - ch3, (ch1 + ch3)/ 2	(ch1 + ch3)/ 2	4-port xfer fn with two se probes or cables	
		4 Port (Channel 1 + 3)	Common Mode	InfiniiSim 4 Port 1 Src	4-port	.tf4	(ch1 + ch3)/ 2	(ch1 + ch3)/ 2	4-port xfer fn with two se probes or cables	Does not include diff -> cm

Where:

- SnP -> TFn = S-parameter file to transfer function, where n = 2 for 2-port or n = 4 for 4-port
- se = single-ended
- diff = differential
- cm = common mode
- xfer fn = transfer function
- xtalk sim = crosstalk simulation

Using the InfiniiSim Wizard

The InfiniiSim Wizard helps you quickly set up InfiniiSim by walking you through the necessary steps.

- "General Setup" on page 21
- "Select Model" on page 21
- "Measurement Block Setup" on page 22
- "Simulation Block Setup" on page 23
- "Observation Nodes" on page 24
- "Save File" on page 24

Clicking the **Cancel** button will undo any selections or changes.

General Setup

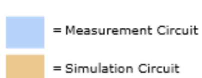
InfiniiSim consists of two circuits:

- The Measurement Circuit is a description of what your system actually is.
- The Simulation Circuit is a description of what you want your system to be like.

InfiniiSim creates a transfer function to convert between the Measurement Circuit (real) and the Simulation Circuit (desired).

For example, you may be using a cable that is having undesired effects on your oscilloscope measurements. In the Measurement Circuit, there would be a block that describes the electrical characteristics of the cable (such as an RLC circuit or S-parameter file). In the Simulation Circuit, it would just be a "Thru" because InfiniiSim creates a transfer function that compensates for the electrical impact of the cable. This means the waveforms shown on the oscilloscope are what you would be seeing if the cable had no electrical impact at all.

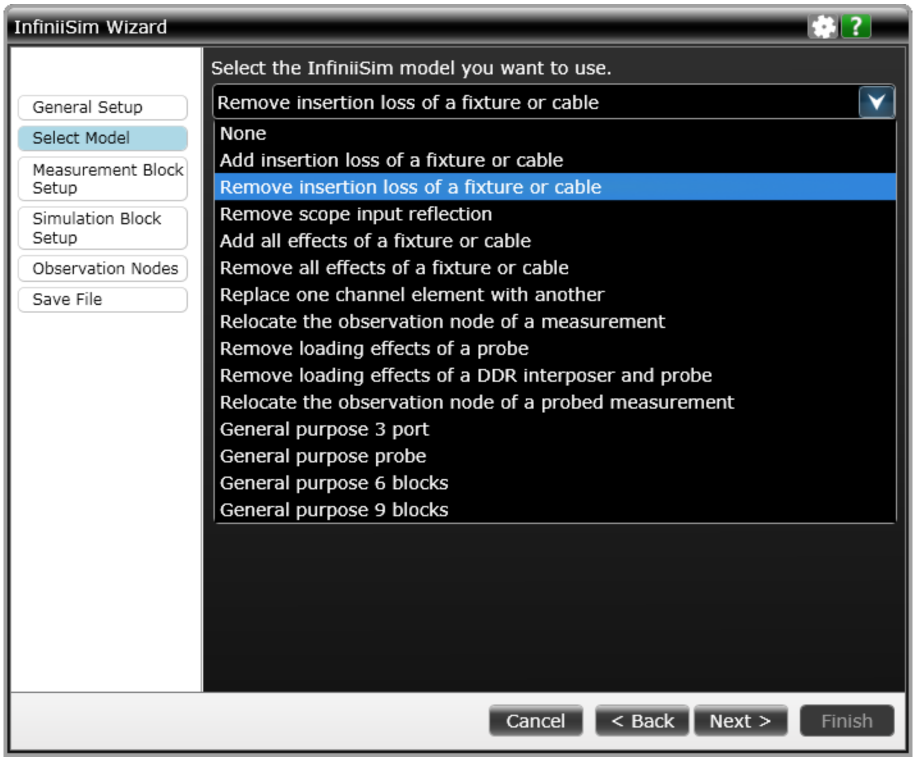
These are the colors for the Measurement (light blue) and Simulation Circuits (light orange).



Next: "Select Model" on page 21.

Select Model

Select the InfiniiSim model (or application preset) you want to use.



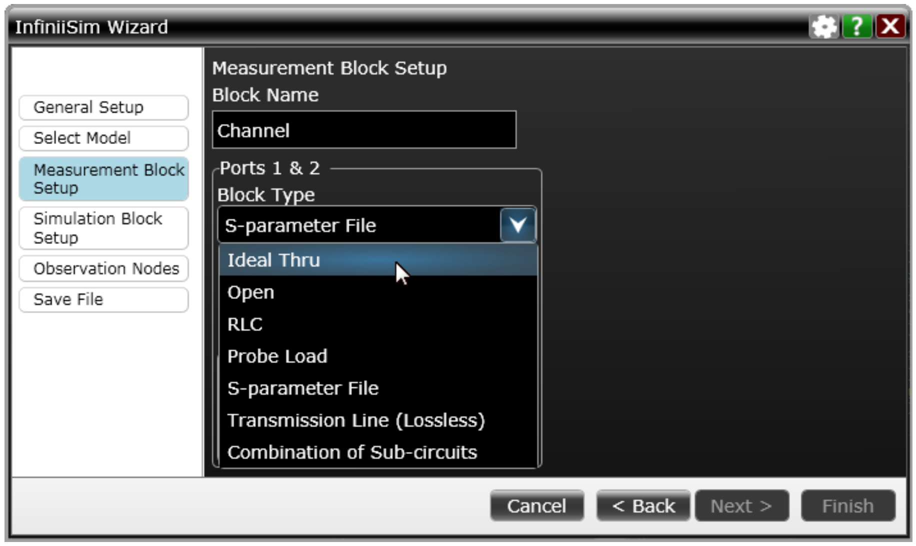
If you select an application preset more complex than adding or removing the insertion loss of a fixture or cable, you will have to set up the measurement and simulation circuit blocks manually after exiting the wizard. In this case, the next wizard step will be "Observation Nodes" on page 24.

For more information on the various InfiniiSim application presets, see "Application Presets for Typical Circuit Topologies" on page 9.

Next: "Measurement Block Setup" on page 22.

Measurement Block Setup

When you have selected either the "Add insertion loss of a fixture or cable" or "Remove insertion loss of a fixture or cable" InfiniiSim model, the next step in the InfiniiSim Wizard lets you set up the measurement block.



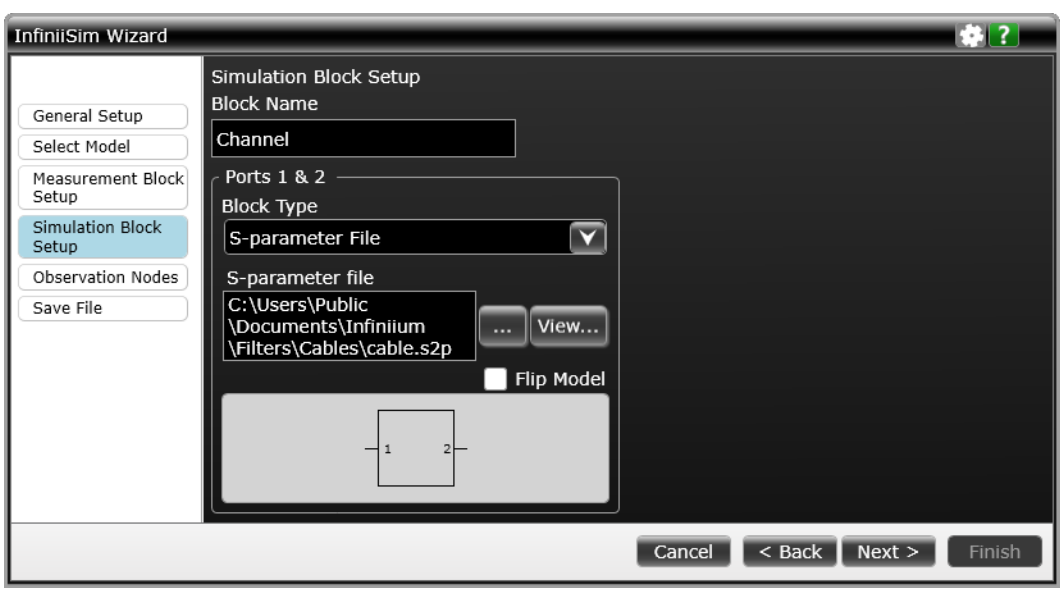
For more information on measurement circuits and setting them up, see:

- "Defining a Block (2 Port)" on page 37
- "Defining a Block (4 Port)" on page 63

Next: "Simulation Block Setup" on page 23.

Simulation Block Setup

When you have selected the measurement block type, the next step in the InfiniiSim Wizard helps you set up the simulation block.



For more information on simulation circuits and setting them up, see:

- "Defining a Block (2 Port)" on page 37
- "Defining a Block (4 Port)" on page 63

Clicking the **View...** button opens the InfiniiSimToolbox application for viewing and evaluating S-parameter files. See "S-Parameter and Transfer Function File Viewers" on page 127.

Next: "Observation Nodes" on page 24.

Observation Nodes

Some models have Observation Nodes, which are indicated by these **M** and **S** icons:

- 

Measurement Observation Node
- 

Simulation Observation Node
- 

Both Measurement and Simulation Nodes

The Measurement Node corresponds to the location in the circuit where you actually measured the original signal.

The Simulation Node corresponds to the location in the circuit where you wished you could have measured the signal.

An Observation Node can be moved by clicking the node where you want it to go. However, there are situations where a node cannot be made into an Observation Node.

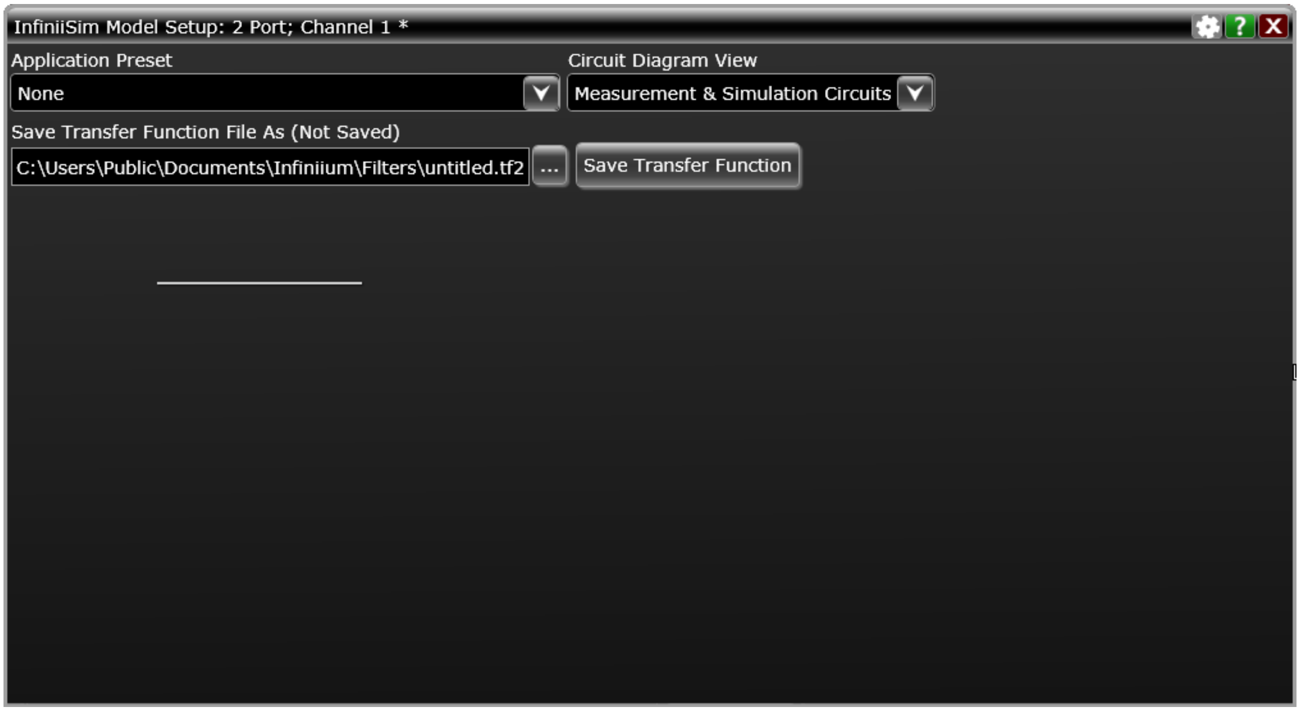
Next: "Save File" on page 24.

Save File

Enter the name of the transfer function file. This file is created by InfiniiSim based on the Measurement and Simulation models you have specified. To use these models again, just use this file instead of running the wizard.

Creating a Transfer Function From a Model (2 Port)

This section describes how to generate a transfer function from a model using the InfiniiSim Model Setup dialog box (accessed by clicking the **Create Transfer Function from Model...** button in the "InfiniiSim Setup Dialog Box" on page 156).



(No models are shown in the InfiniiSim Model Setup dialog box's default view above because the **Application Preset** field is set to **None**.)

Follow these steps to create a transfer function from a model:

1 Select one of the application presets. See "Application Presets (2 Port)" on page 26.

2 In the **Save Transfer Function File As (Not Saved)** field, specify the name of the transfer function file you will create.

3 (Optional) Select the circuit diagram view. See "Circuit Diagram View (2 Port)" on page 34.

4 Specify the circuit source and load impedances. See "Circuit Source and Load Impedances (2 Port)" on page 36.

5 Define measurement circuit and simulation circuit blocks. See "Defining a Block (2 Port)" on page 37.

6 Specify measurement and simulation node locations. See "Measurement and Simulation Node Locations (2 Port)" on page 49.

7 Click **Save Transfer Function** to generate and save the transfer function file. See "Generating/Saving a Transfer Function File" on page 78.

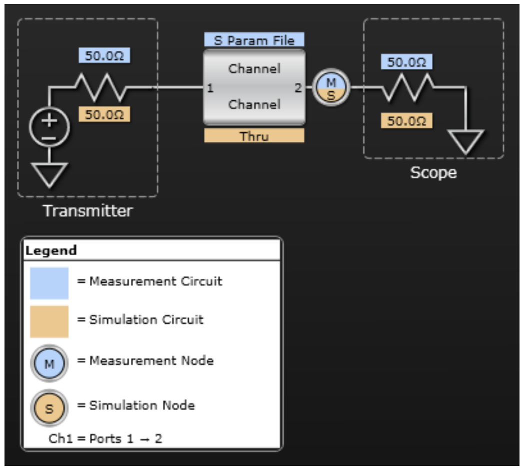
Application Presets (2 Port)

The first field in the InfiniiSim Model Setup dialog box is labeled **Application Preset**. This drop-down field provides a variety of preset configurations you can work with to build your own model. The available application presets are:

- **None**
- "Add insertion loss of a fixture or cable" on page 27
- "Remove insertion loss of a fixture or cable" on page 28
- "Remove scope input reflection (2 Port)" on page 28
- "Add all effects of a fixture or cable" on page 29
- "Remove all effects of a fixture or cable" on page 29
- "Replace one channel element with another" on page 29
- "Relocate the observation node of a measurement" on page 30
- "Remove loading effects of a probe" on page 30
- "Remove loading effects of a DDR interposer and probe (2 Port)" on page 31
- "Relocate the observation node of a probed measurement" on page 30
- "General purpose 3 port (2 Port)" on page 32
- "General purpose probe" on page 31
- "General purpose 6 blocks (2 Port)" on page 33
- "General purpose 9 blocks (2 Port)" on page 33

General-purpose topologies with user-defined measurement and simulation points are provided for greater flexibility.

To understand the basic components that make up these configurations, look at a very simple example. The following circuit appears in the InfiniiSim Model Setup dialog box when you set the **Application Preset** field to **Add insertion loss of a fixture or cable**.



Notice some values are highlighted light blue and some are highlighted light orange. These correspond to the measurement and simulation circuit respectively.

- **Measurement Circuit** — This circuit models the actual physical circuit that produced the measured waveform.
- **Simulation Circuit** — This circuit models a hypothetical electrical circuit that exhibits your desired electrical characteristics.

In other words, the measurement circuit is what you actually measured and the simulation circuit is what you wish you would have been able to measure.

Remember that the model blocks in these configurations are used to represent a certain device and can be defined in a variety of ways (S-parameters, RLC circuits, open, thru, lossless transmission line, etc.).

InfiniiSim analyzes these two circuits and generates a correction transfer function which, when convolved with the measured waveform, produces the desired simulated waveform. For more information on transfer functions and how InfiniiSim generates them, see "InfiniiSim Transfer Functions" on page 134.

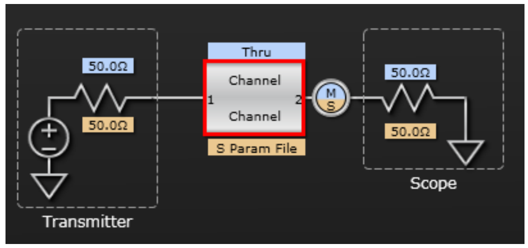
Simple One-Block Presets (2 Port)

The application presets that add or remove insertion loss of a channel element (fixture or cable) are simple one-block models inserted before the oscilloscope.

Many times you simply want to compensate for the loss of a channel element such as a cable or fixture. The one-block models can perform this task. It can also be extended to a remove/replace operation by changing the simulation parameters.

Add insertion loss of a fixture or cable

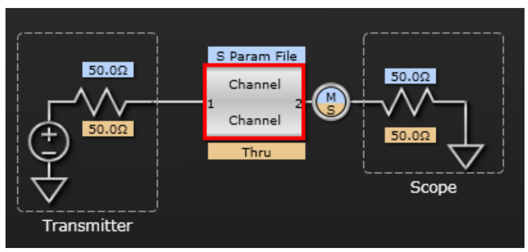
This is an embedding operation where the simulation circuit model has elements that are not in the measurement circuit model.



The gain of the block (S_{21}) is determined, and its time response is convolved with the acquired signal.

Remove insertion loss of a fixture or cable

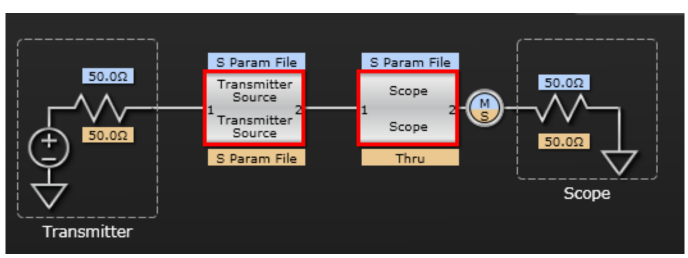
This is a de-embedding operation where the measurement circuit model has elements that are not present in the simulation circuit model.



The inverse gain of the block (S_{21}^{-1}) is determined, and its time response is convolved with the acquired signal.

Remove scope input reflection (2 Port)

Modeled using two blocks — one for the transmitter source and the other for the oscilloscope input (so the oscilloscope input reflection can be removed from the simulation circuit).

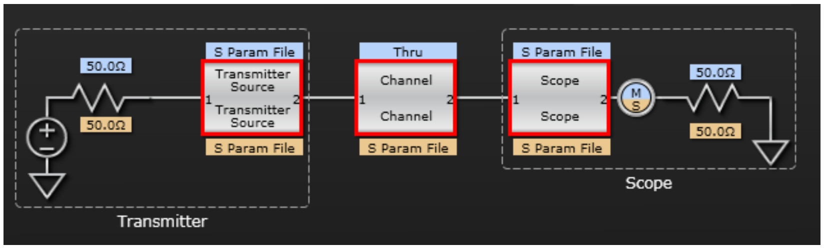


Three-Block Presets (2 Port)

When the most precision for a single-channel element removal/insertion is required, you need at least a 3-block model. These models use block descriptions for transmitter (**Transmitter Source**) and receiver (**Scope**), as well as the channel, to describe the full system. The inclusion of the transmitter and receiver blocks enable the most complete waveform rendering by including the reflective S-parameter elements in the mathematical calculation of the transfer function used to transfer from the measurement node (**M**) to the simulation node (**S**).

Add all effects of a fixture or cable

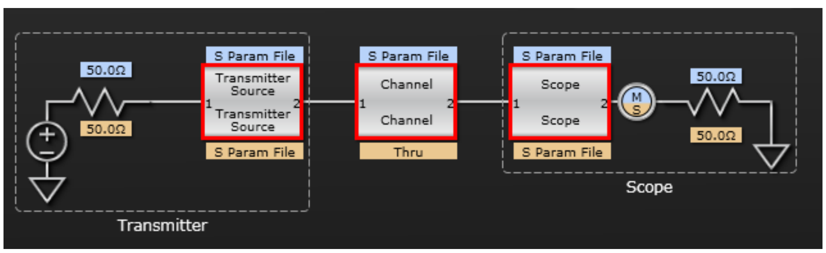
By using S-parameter models of source and oscilloscope load, a complete addition (embedding) of a channel element is performed.



This is different from "Add insertion loss of a fixture or cable" on page 27 in that the reflective interactions between the elements are taken into account. This provides the most accurate rendering to add a channel element.

Remove all effects of a fixture or cable

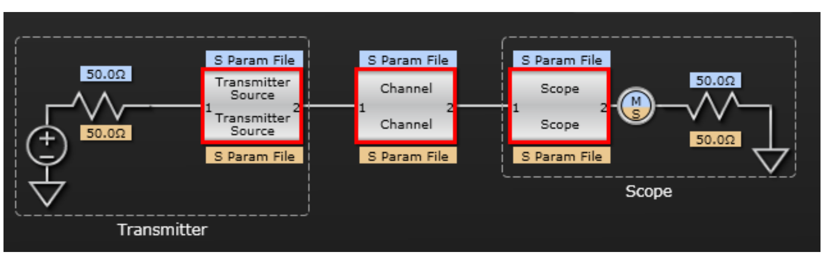
By using S-parameter models of source and oscilloscope load, the effects of a channel element are totally removed (de-embedded).



This is different from "Remove insertion loss of a fixture or cable" on page 28 in that the interactions between the elements are taken into account. This provides the most accurate way to remove a channel element.

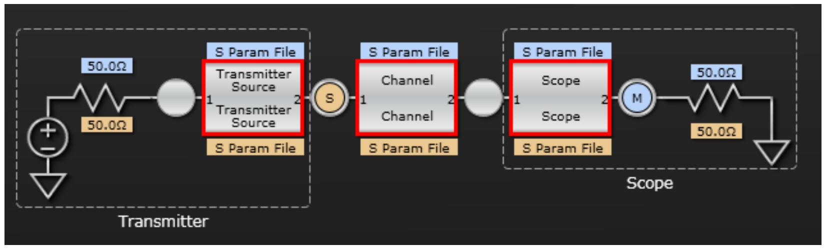
Replace one channel element with another

By using different channel element models for the measurement and simulation circuits, you can replace one channel element with another.



Relocate the observation node of a measurement

Observation node relocation lets you view any voltage waveform in a circuit by moving the observation node to any location you desire. This is an "in situ" analysis so it is not a "removal" or "addition" viewpoint.

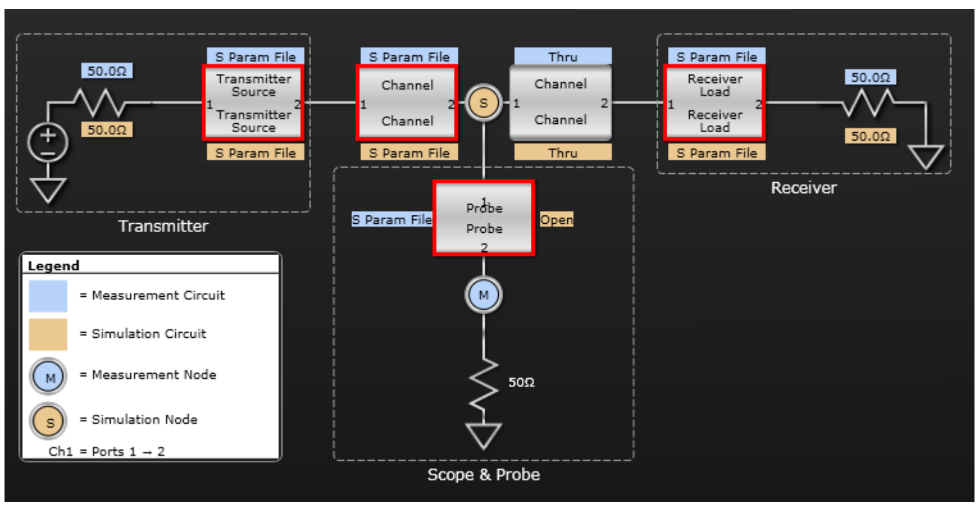


Probe Presets (2 Port)

For probe modeling, use one of the probe application presets.

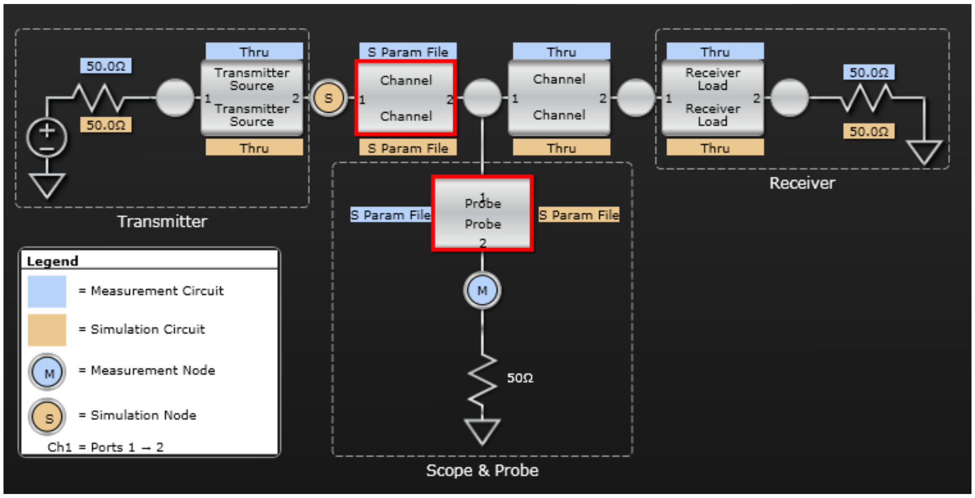
Remove loading effects of a probe

To remove the loading effects of a probe, a topology of circuit blocks is given that lets probe models be considered in the measurement.

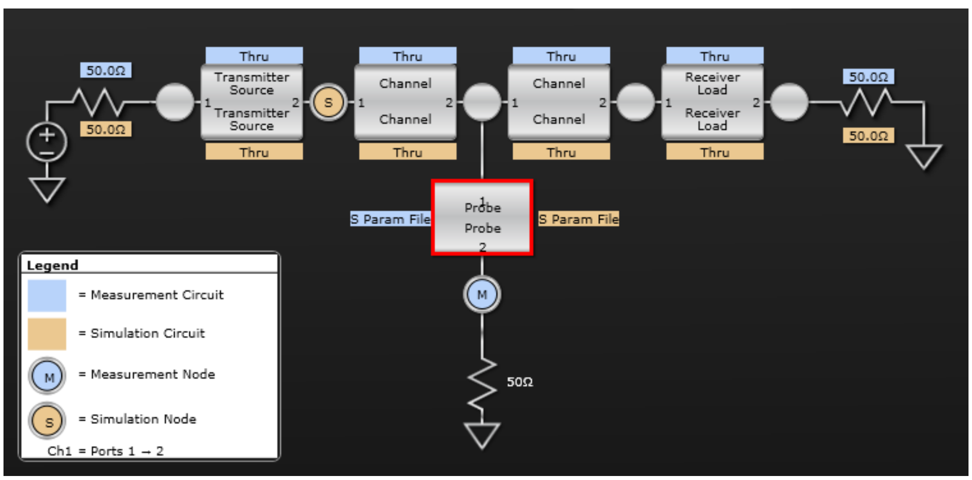


An oscilloscope probe, while it might be defined as "high impedance", really does have a loading effect on the circuit. This effect can be taken into account and removed.

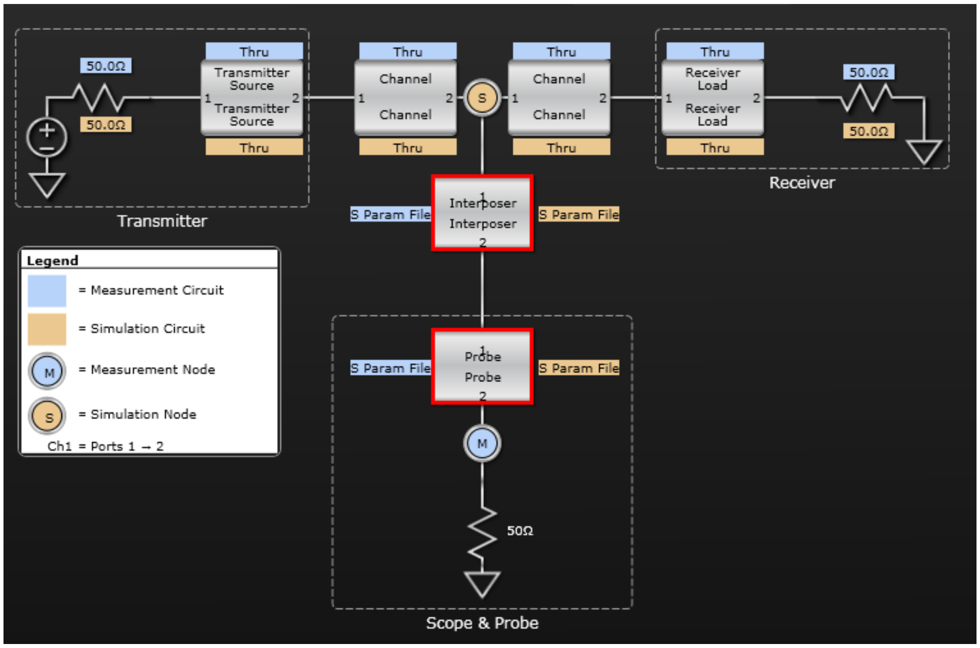
Relocate the observation node of a probed measurement



General purpose probe

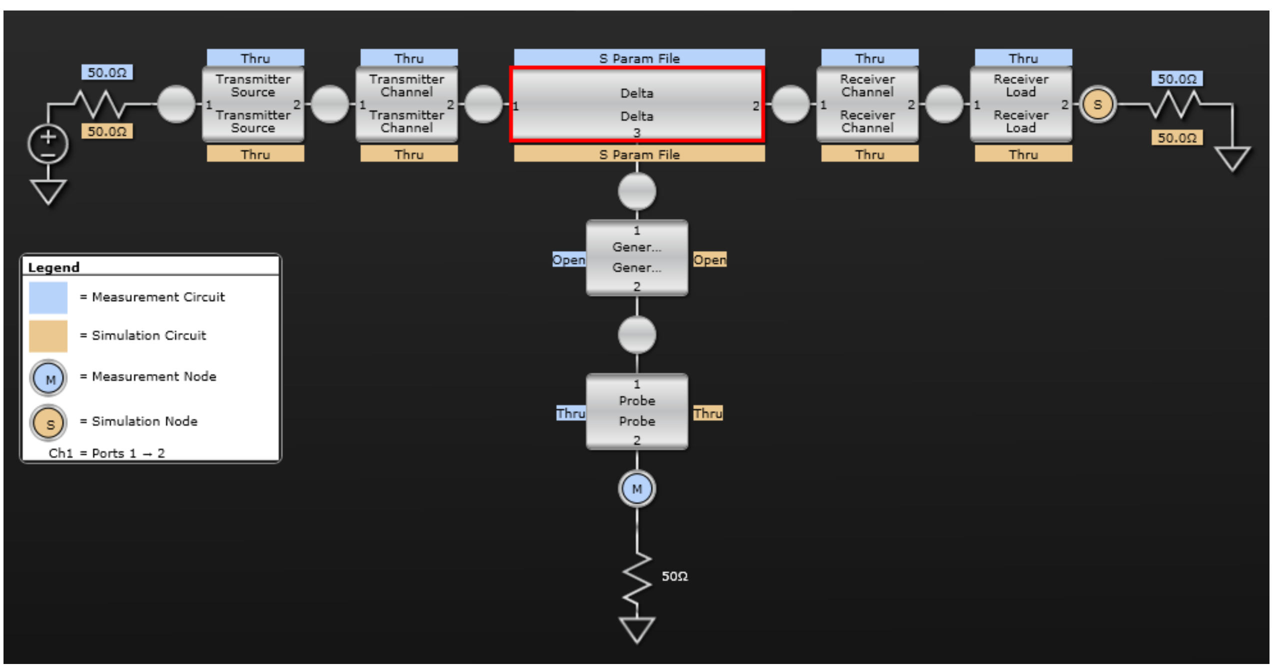


Remove loading effects of a DDR interposer and probe (2 Port)



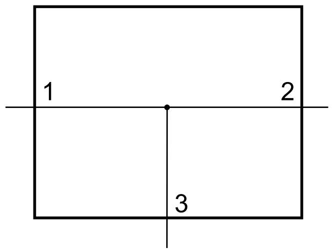
General purpose 3 port (2 Port)

This application preset is similar to **General purpose probe** but different in that it has the **Delta** block into which you can load a 3-port S-parameter (s3p) file.

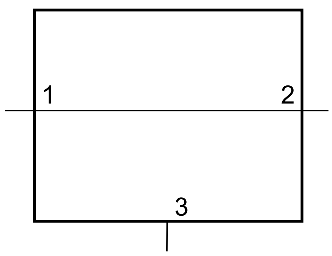


In addition to specifying an S-parameter file for the Delta block, you can also set it to:

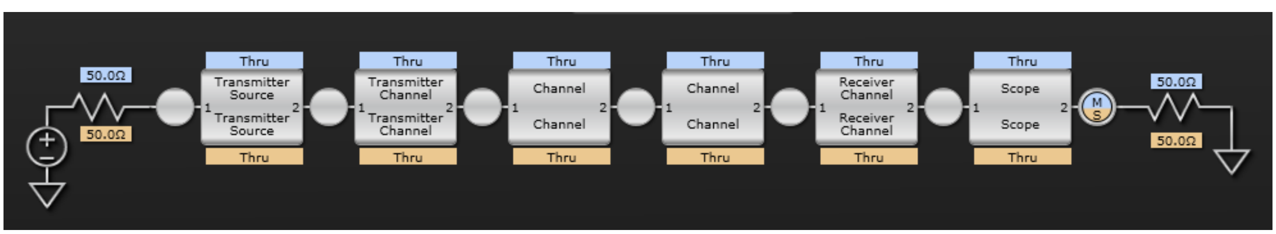
- **Ideal Thru** — In this case, all three ports are shorted together.



- **Open** — In this case, ports 1 and 2 are shorted together and port 3 is open.



General purpose 6 blocks (2 Port)

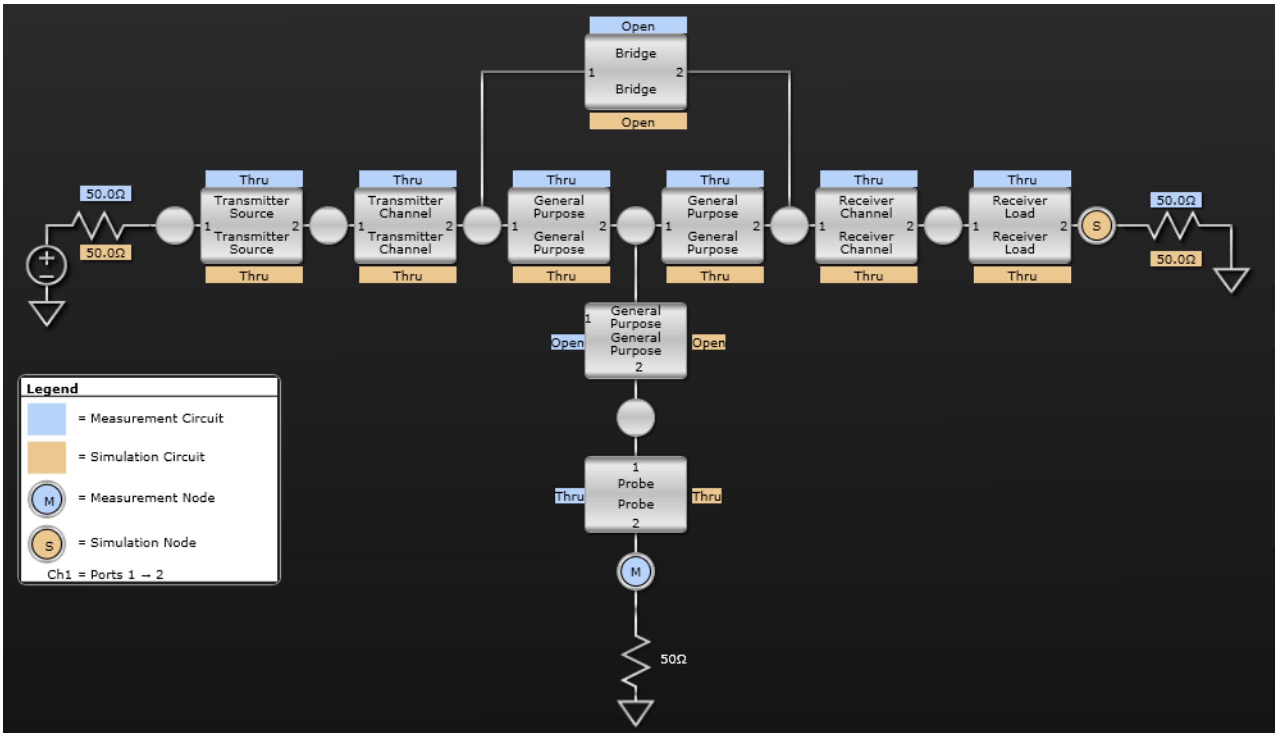


The general-purpose 6-block model is used for SMA differential probe modeling in the majority of cases.

General purpose 9 blocks (2 Port)

For very sophisticated applications (for example, using both high-impedance probes and differential SMA probe heads), the general-purpose 9-block application preset can be used to describe these complex cases.

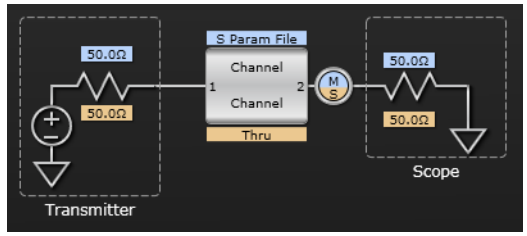
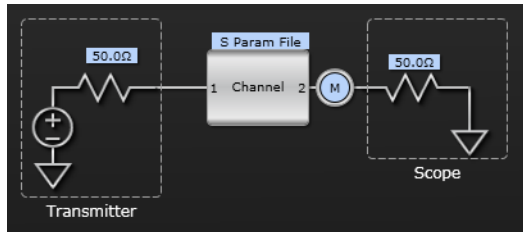
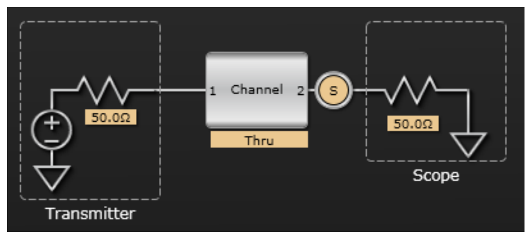
The general-purpose 9-block application preset is the model from which all other application presets can be made.



Each block in the 9-block model can be defined as having a combination of up to three elements in cascade, series, or parallel arrangements, giving you 27 total possible circuit elements to define for the most sophisticated scenarios.

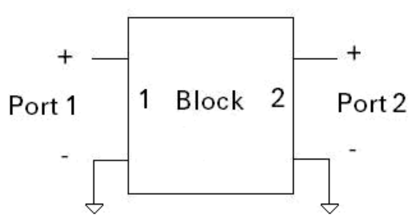
Circuit Diagram View (2 Port)

If you look under the **Circuit Diagram View** section in the InfiniiSim Model Setup dialog box, you see that you have three viewing options.

Measurement & Simulation Circuits	
Measurement Circuit Only	
Simulation Circuit Only	

These choices let you build/view the measurement circuit, the simulation circuit, or both at the same time.

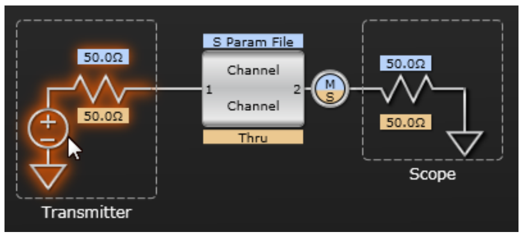
It is important to note that these circuit model representations in the GUI are simplified for ease of viewing. If we were being explicit, the model blocks in the circuit would look like the following for a 2-port device:



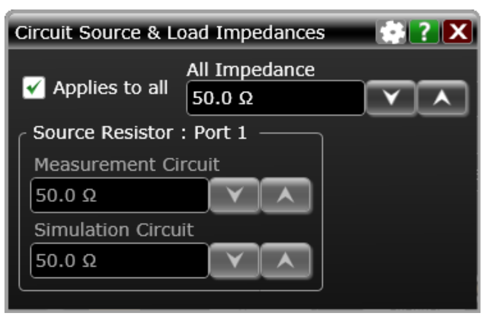
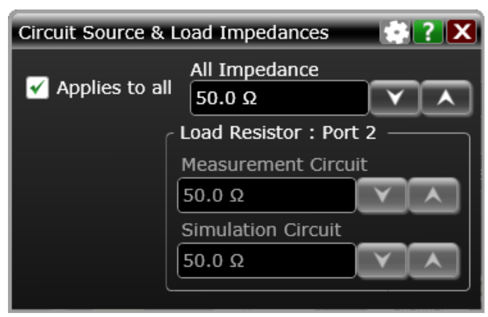
The diagrams in the GUI are completely interactive so you can click various sections or components and make adjustments.

Circuit Source and Load Impedances (2 Port)

The resistor on the left side of the circuit corresponds to the impedance of the transmitter (source). The resistor on the right side of the circuit corresponds to the impedance of the receiver or oscilloscope (load). These impedances are defaulted to 50 **Ω**, but you can change them if needed. Simply place your pointer over the resistor (it thickens to indicate you are on top of it) and click.



You then see one of the following Circuit Source and Load Impedances dialog boxes.

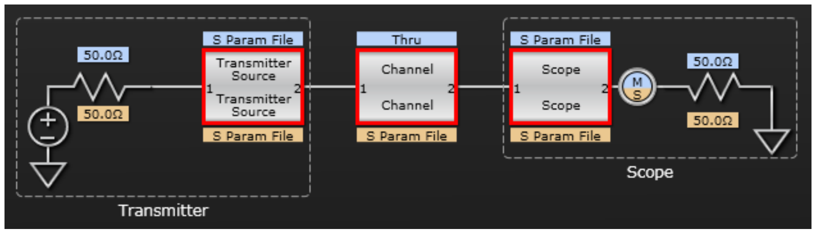
<table>
<tr><th>Source</th><th>Load</th></tr>
<tr><td>
</td><td>
</td></tr>
</table>

The Circuit Source and Load Impedances dialog box lets you specify the source or load impedances within both the measurement and simulation circuits.

If you want all impedances in both the measurement and simulation circuits to be identical, check the **Applies to All** box and set the **All Impedance** value. All other impedances are then changed to match this one.

If the **Applies to All** box was not checked, the **Add Insertion Loss** application preset (used in the preceding screen) would gray-out the load impedance controls because the oscilloscope load impedance is always 50 **Ω**.

Note that by using one of the 3-block models (for example, the **Remove All Effects of a Fixture or Cable** application preset), you can apply an arbitrarily complex transmitter and/or oscilloscope impedance using the additional model blocks dedicated to forming the transmitter source impedance and oscilloscope input impedance.

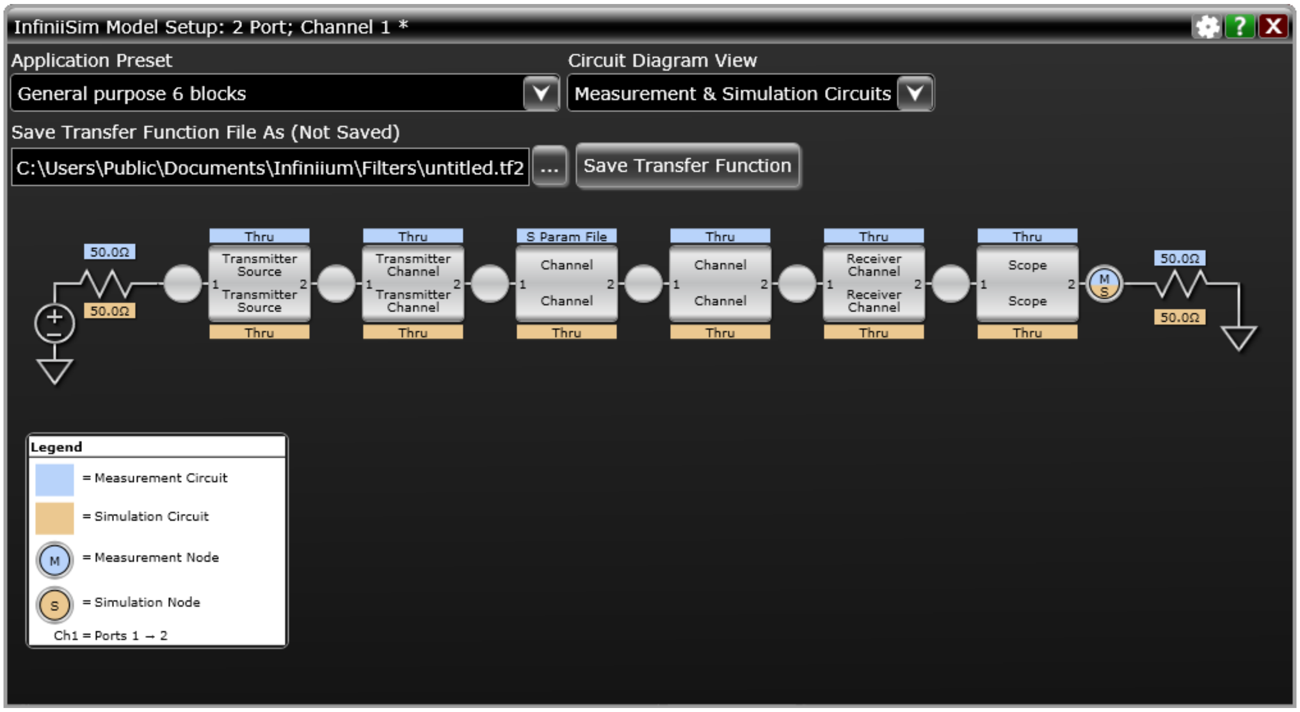


Note that any termination block located adjacent to a source or load resistor (like the **Transmitter Source** block and **Scope** block above) can have 1-port S-parameter files assigned to them in 2-port InfiniiSim mode. These are the only locations in a 2-port InfiniiSim circuit where you can place 1-port S-parameter (s1p) files. Likewise, termination blocks can be defined by 2-port S-parameter files in 4-port mode. You can also use a 1-port S-parameter file for a 4-port application, which will apply those S-parameters to S_{11} and S_{33} and set S_{13} and S_{31} to zero.

Defining a Block (2 Port)

The blocks used in creating the measurement and simulation circuits may be used to model various components (cable loss, probe loading, a fixture, etc.).

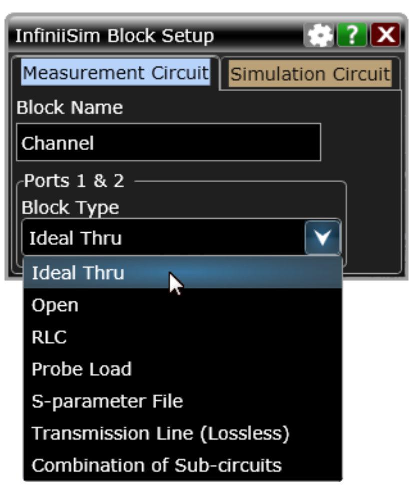
InfiniiSim assigns names to the blocks in the legend for a typical use scenario depending on the application preset chosen. These names are just a convenience for referring to blocks and an example of how the block may be used. However, you can define them any way you like. You can change these names by using the **Block Name** control inside the InfiniiSim Block Setup dialog box (accessed by clicking one of the blocks).



While blocks represent different parts of the circuit, they have the exact same definition controls to choose from and are called different types to help you understand what they might represent in your circuit.

The termination blocks, **Transmitter Source** and **Scope**, are different in one respect—those blocks can be defined by return-loss-only S-parameter files (1-port file in 2-port InfiniiSim analysis mode).

When you click a block, the InfiniiSim Block Setup dialog box opens.



The tabs at the top of the dialog box (**Measurement Circuit** and **Simulation Circuit**) let you specify whether you are currently defining a block in the measurement circuit or in the simulation circuit. For instance, if you select the **Simulation Circuit** tab and then set the **Block Type** to **Ideal Thru**, then the corresponding block in the simulation circuit is set to a Thru block.

The **Block Name** control lets you customize the name of the block.

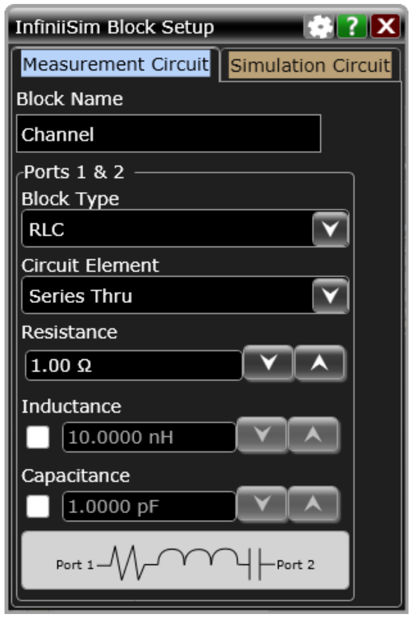
The bottom section (**Block Type**) is where you actually define the block. You can select these block port types:

- **Ideal Thru** — Nothing happens to the signal as it passes through a Thru block. It is as though the block was not there and port 1 is shorted directly to port 2.
- **Open** — When you set the block to open, no signal passes through it. It is as though port 1 and port 2 form an open circuit.
- **RLC** — This lets you set up the block as a RLC (resistor, inductor, capacitor) network. See "RLC Block Type (2 Port)" on page 39.
- **S-parameter File** — If you have an S-parameter file that models a fixture, cable, probe, or device in your circuit, you can load it into InfiniiSim and use it for the block definition. See "S-Parameter File Block Type (2 Port)" on page 40.
- **Transmission Line (Lossless)** — This defines the block as an ideal lossless transmission line. This block is defined by a characteristic impedance value and a time delay value. See "Transmission Line Block Type (2 Port)" on page 43.
- **Combination of Sub-circuits** — This selection lets you use up to three blocks in place of the one. These three sub-blocks can be arranged in cascade, parallel, or series. See "Combination of Sub-Circuits Block Type (2 Port)" on page 43.
- **Probe** — This selection models the effects of the probe that is connected to the current InfiniiSim channel. The model is inserted in series with the signal path and is used when the measured or simulated signal travels through the probe to the input of the oscilloscope. See "Probe Block Type (2 Port)" on page 46.
- **Probe Load** — This selection allows the modeling of a probe load in shunt with the signal path. This model is used when the measured or simulated signal does not travel through the probe, but is only loaded by a probe from a different channel. See "Probe Load Block Type (2 Port)" on page 47.

Once your circuits are built, select the appropriate measurement node location and the desired simulation node location (see "Measurement and Simulation Node Locations (2 Port)" on page 49). Then generate, save, and apply the correction transfer function (see "Generating/Saving a Transfer Function File" on page 78).

RLC Block Type (2 Port)

When you select **RLC** for the block type, additional controls appear at the bottom of the dialog box.



For **Circuit Element**, you can choose:

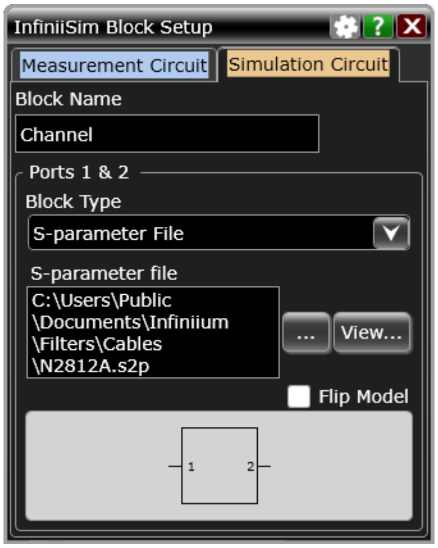
- **Series Thru**
- **Parallel Thru**
- **Series Shunt**
- **Parallel Shunt**

When you select one of these options, the basic circuit configuration is shown in the dialog box (see the preceding screen).

After selecting the configuration of the RLC circuit, you can set the values for the **Resistance**, **Inductance**, and **Capacitance**. If you do not want to use an inductor or capacitor in your RLC circuit, do not check the boxes located next to each entry.

S-Parameter File Block Type (2 Port)

When you select **S-parameter File** for the block type, additional controls appear at the bottom of the dialog box.



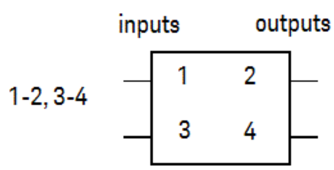
You can click the **S-parameter file** field to browse to your saved S-parameter files. S-parameter files are typically generated using a vector network analyzer or circuit simulator, for example, and then saved on the oscilloscope hard drive. These can be files that model fixtures, cables, devices, or probes in your circuit. InfiniiSim accepts both Touchstone files (.s1p, .s2p, .s4p) and CITIfile (.cti, .cit) files.

Clicking the **View...** button opens the InfiniiSimToolbox application for viewing and evaluating S-parameter files. See "S-Parameter and Transfer Function File Viewers" on page 127.

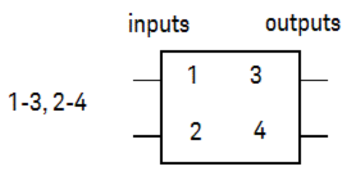
By default, the oscilloscope expects port 1 to be on the left of the block and port 2 to be on the right. However, this may not match the way the data is defined in your S-parameter file. Therefore, you can check the **Flip Model** box to switch ports 1 and 2.

If you select a 4-port S-parameter file (.s4p file) in this step (remember we are performing a 2-port InfiniiSim analysis in this topic), the oscilloscope needs to know how to interpret the data in the S-parameter file (because there are more than two ports worth of information contained in a 4-port file). In this case, you see the additional **4-Port Numbering** and **Port Extraction** controls.

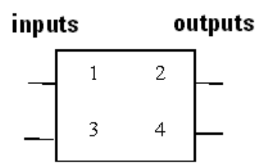
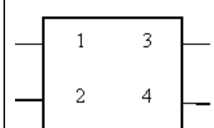
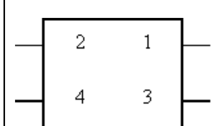
The **4 Port Numbering** control lets you specify how the data is arranged in the 4-port S-parameter file. There is an industry-standard numbering system:



However, you can tell InfiniiSim that your file actually uses the following numbering system.



You can also use the **Flip Model** box to flip the numbers between the right side and the left side.

Flip Model	4 Port Numbering	How ports are connected in a block
No	1→2, 3→4	
No	1→3, 2→4	
Yes	1→2, 3→4	
Yes	1→3, 2→4	

The **Port Extraction** field tells InfiniiSim how to extract a 2-port model from the 4-port S-parameter file and use it in your 2-port transformation.

For example, if you select **1<->2, 3<->4** for the **4-port Numbering** field, you have the following choices:

- **Use Ports 1 -> 2** — this means you are extracting only the port 1 and 2 information from the 4-port S-parameter file (one single-ended path through the differential model)
- **Use Ports 3 -> 4** — this means you are extracting only the port 3 and 4 information from the 4-port S-parameter file (the other single-ended path through the differential model)
- **Differential** — this means you are extracting port D1 and port D2 of a mixed-mode view of the file (differential path through the 4-port file)
- **Common Mode** — this means you are extracting port C1 and port C2 of a mixed-mode view of the file (common-mode path through the 4-port file)

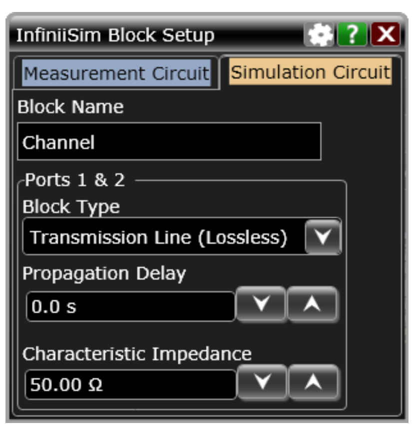
For more information on port extraction from S-parameter files, see:

- "S-Parameters for Single-Ended 4-Port: Flow Graph, Matrix, and Extractions" on page 131
- "Mixed-Mode S-Parameters for Differential 2-Port: Flow Graph, Matrix, and Extractions" on page 132

For more information on S-parameter files in general, see "S-Parameter Considerations" on page 113.

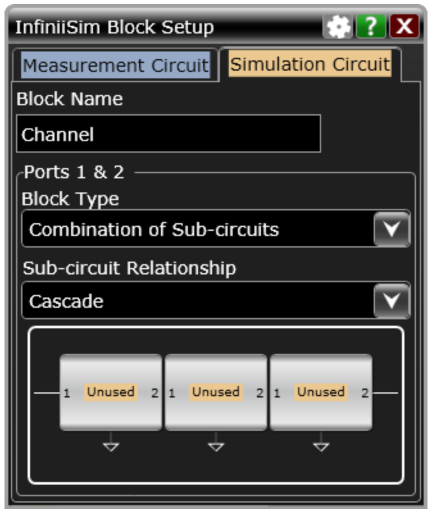
Transmission Line Block Type (2 Port)

When you select **Transmission Line (Lossless)** for the block type, two additional fields appear for setting the **Characteristic Impedance** of the transmission line in your model block as well as its **Propagation Delay**.

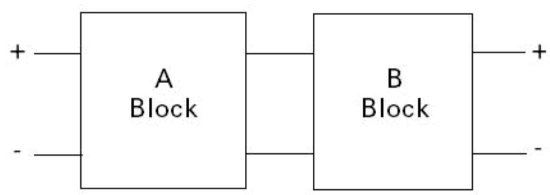


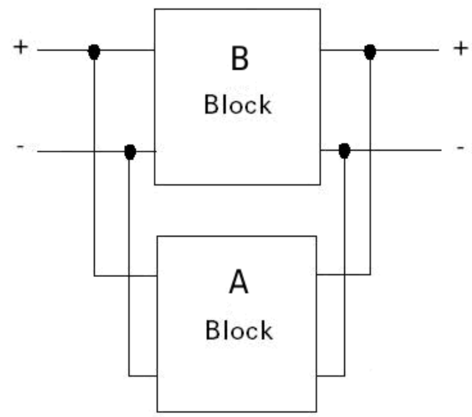
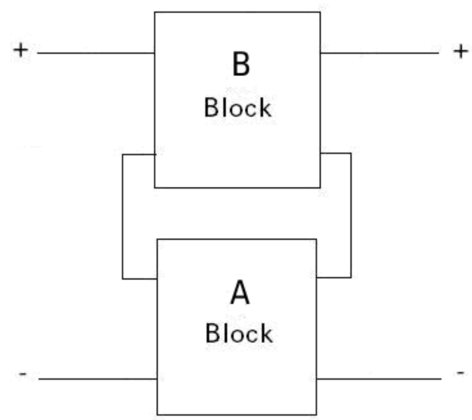
Combination of Sub-Circuits Block Type (2 Port)

The **Combination of Sub-circuits** selection lets you use a combination of three sub-blocks in place of one block. When you make this selection, additional controls appear at the bottom of the dialog box.

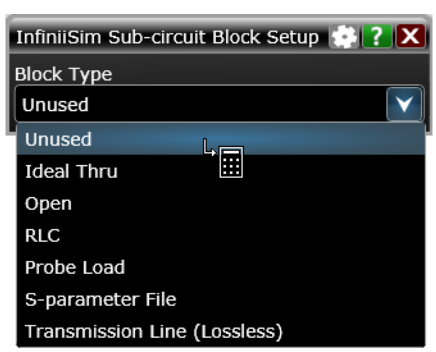


You have the following three choices for how to set up the three individual blocks that now make up the one circuit block (pictures show connections for two of the three blocks for simplicity).

Cascade	

Parallel	
Series	

Once you select one of these three configurations, you can click each individual block and define what type of block it is. When you click a block a new InfiniiSim Sub-circuit Block Setup dialog box appears where you can define its type:



You can choose between **Ideal Thru**, **Open**, **RLC**, **Probe Load**, **S-parameter File**, **Transmission Line (Lossless)**, and an option called **Unused**. Unused means that this particular sub-block is not used (letting you use 2 sub-blocks instead of three, for example).

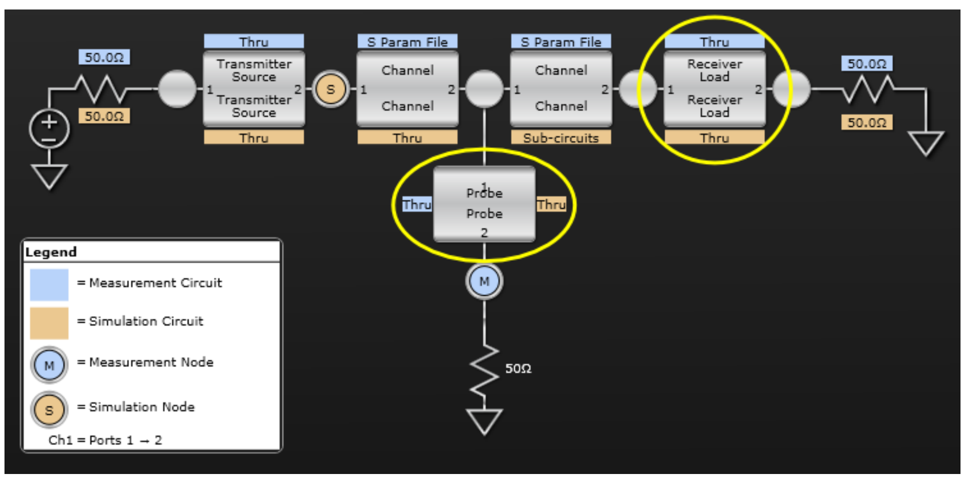
Using the **Unused** setting has a different effect on this specific block depending on which configuration is being used.

- For the **Cascade** structure, for example, **Unused** acts like a **Thru** block.
- For the **Parallel** structure, **Unused** acts like an **Open** block.

Probe Block Type (2 Port)

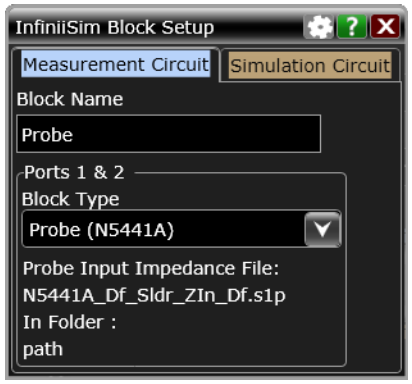
The **Probe** block type selection models the effects of the probe that is connected to the current InfiniiSim channel. The model is inserted in series with the signal path and is used when the measured or simulated signal travels through the probe to the input of the oscilloscope.

You can select the **Probe** block type only in the blocks that are intended to model the oscilloscope input (for example, the ones circled in yellow).



A probe must be connected to the channel for the **Probe** block type selection to be active.

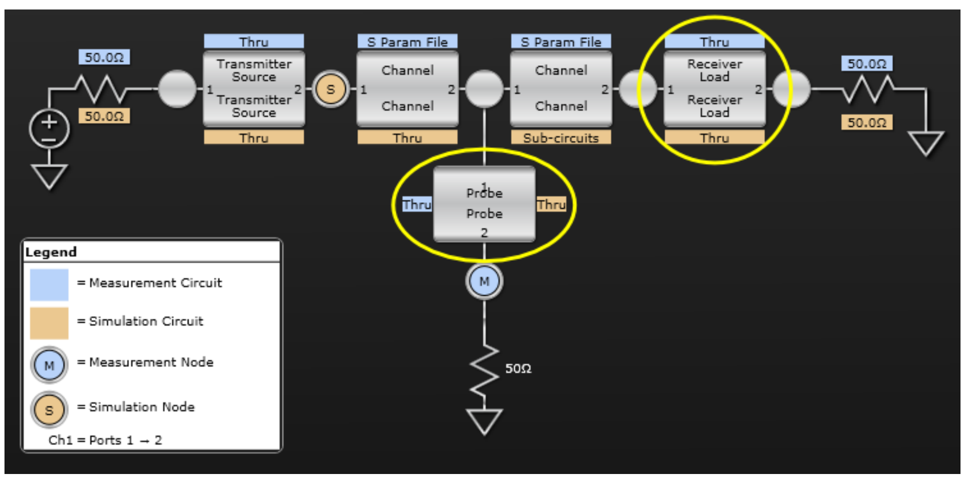
When you make the **Probe** block type selection, the appropriate probe input impedance file is automatically selected, based on your selections in the Probe Configuration dialog box.



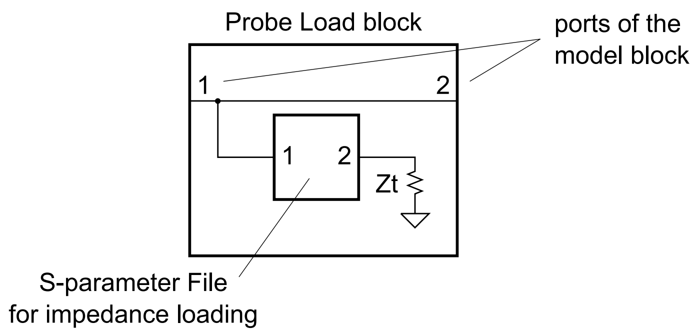
Probe Load Block Type (2 Port)

The **Probe Load** block type selection lets you model a probe load in shunt with the signal path. This model is used when the measured or simulated signal does not travel through the probe, but is only loaded by a probe from a different channel.

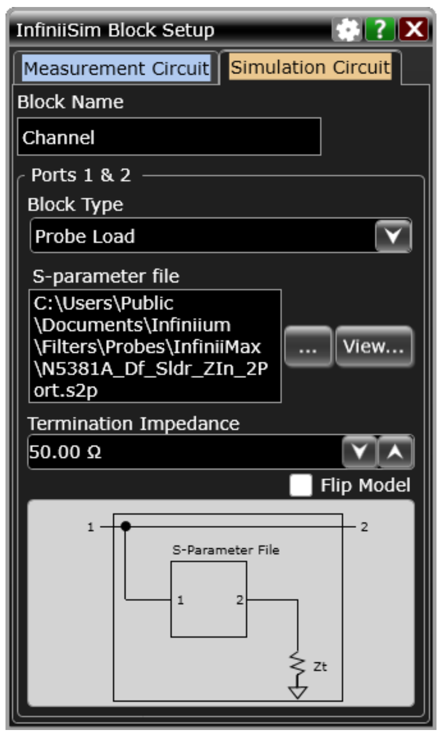
The **Probe Load** block type is not allowed in those blocks that are intended to model the oscilloscope input (for example, the ones not circled in yellow).



When the **Probe Load** block type is selected it does two things: port 1 and 2 of the block are connected together (like an Ideal Thru) and the supplied S-parameter file is used to describe the shunt impedance loading the connected ports.



When you make the **Probe Load** block type selection, additional controls appear at the bottom of the dialog box:



- **S-parameter file** — Used to describe an impedance loading.

  If a 1-port S-parameter (s1p) file is selected, you can ignore port 2 of the load and the Termination Impedance (Zt).

  If a 2-port S-parameter (s2p) file is selected, port 2 of the load is as shown, and the desired Termination Impedance (Zt) should be entered.

  Input impedance files in are located in the "C:\Users\Public\Documents\Infiniium\Filters\Probes\InfiniiMax" directory.

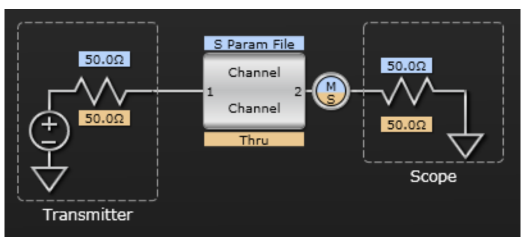
Clicking the **View...** button opens the InfiniiSimToolbox application for viewing and evaluating S-parameter files. See "S-Parameter and Transfer Function File Viewers" on page 127.

- **Termination Impedance** — Again, with a 1-port S-parameter (s1p) file, you can ignore the Termination Impedance (Zt). With a 2-port S-parameter (s2p) file, the desired Termination Impedance (Zt) should be entered.
- **Flip Model** — If your S-parameter file has a different port ordering, you can check this box to switch ports 1 and 2.

Measurement and Simulation Node Locations (2 Port)

You can also change the location of the measurement and simulation nodes. The measurement node corresponds to the location in the circuit where you actually measured the original signal. The simulation node corresponds to the location in the circuit where you wished you could have measured the signal.

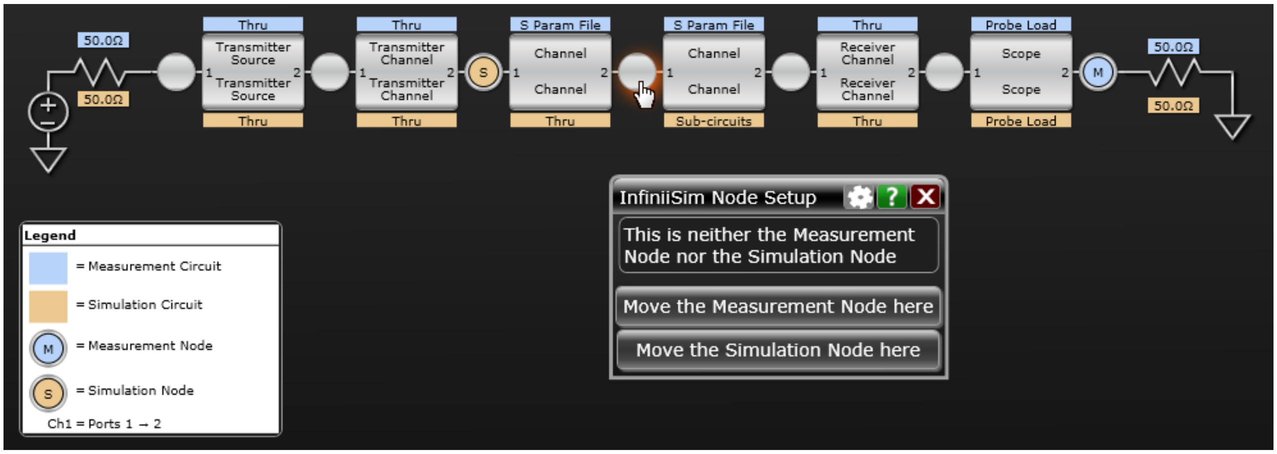
The following picture shows both the measurement and simulation circuit view. The light blue circle labeled **M** is the measurement node and the light orange circle labeled **S** is the simulation node.



In this 1-block model, the measurement and simulation nodes cannot be moved anywhere else in the circuit (there are no empty node locations in the circuit) because this simple preset only lets you add insertion loss of a device.

However, in more complex configurations such as the **General purpose 6 blocks** application preset configuration, you can change the location of the measurement and/or simulation node.

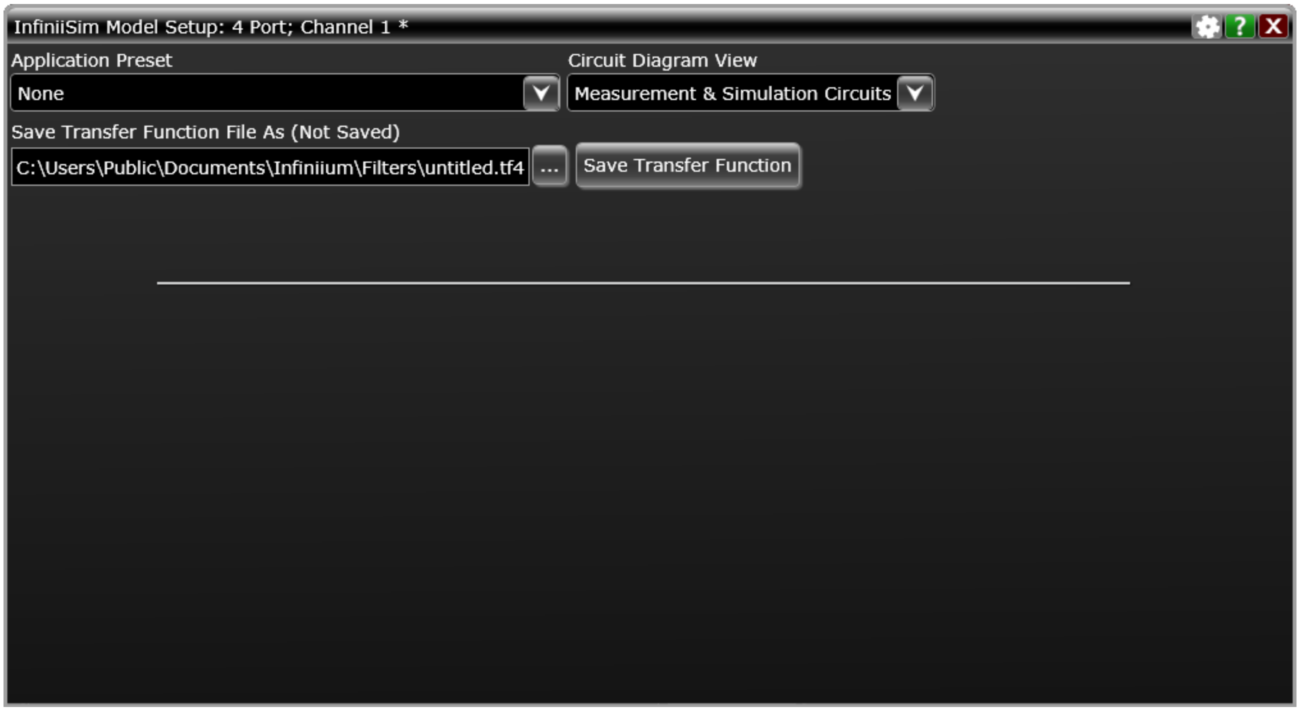
To change the nodes, click one of the empty node locations. Then, select either **Move the Measurement Node here** or **Move the Simulation Node here** from the InfiniiSim Node Setup dialog box.



50.0Ω
50.0Ω
Thru
Transmitter Source
Transmitter Source
Thru
Thru
Transmitter Channel
Transmitter Channel
Thru
S
S Param File
Channel
Channel
Thru
S Param File
Channel
Channel
Sub-circuits
Thru
Receiver Channel
Receiver Channel
Thru
Probe Load
Scope
Scope
Probe Load
M
50.0Ω
50.0Ω
InfiniiSim Node Setup
This is neither the Measurement Node nor the Simulation Node
Move the Measurement Node here
Move the Simulation Node here
Legend
= Measurement Circuit
= Simulation Circuit
= Measurement Node
= Simulation Node
Ch1 = Ports 1 → 2

Creating a Transfer Function From a Model (4 Port)

This section describes how to generate a transfer function from a model using the InfiniiSim Model Setup dialog box (accessed by clicking the **Create Transfer Function from Model...** button in the "InfiniiSim Setup Dialog Box" on page 156).



(No models are shown in the InfiniiSim Model Setup dialog box's default view above because the **Application Preset** field is set to **None**.)

Follow these steps to create a transfer function from a model:

1 Select one of the application presets. See "Application Presets (4 Port)" on page 52.

2 In the **Save Transfer Function File As** field, specify the name of the transfer function file you will create.

3 (Optional) Select the circuit diagram view. See "Circuit_Diagram View (4 Port)" on page 60.

4 Specify the circuit source and load impedances. See "Circuit Source and Load Impedances (4 Port)" on page 62.

5 Define measurement circuit and simulation circuit blocks. See "Defining a Block (4 Port)" on page 63.

6 Specify measurement and simulation node locations. See "Measurement and Simulation Node Locations (4 Port)" on page 76.

7 Click **Save Transfer Function** to generate and save the transfer function file. See: "Generating/Saving a Transfer Function File" on page 78.

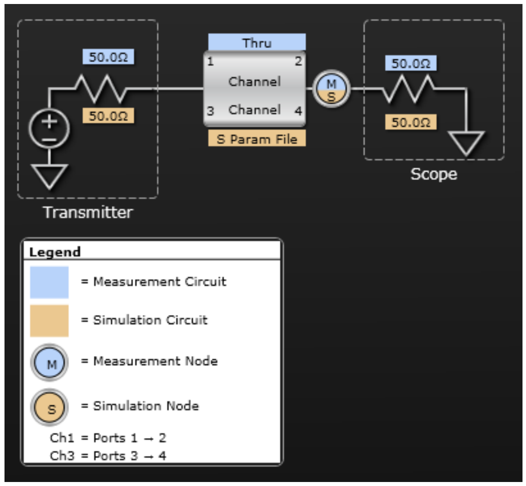
Application Presets (4 Port)

The first field in the InfiniiSim Model Setup dialog box is labeled **Application Preset**. This drop-down field provides a variety of preset configurations you can work with to build your own model. The available application presets are:

- **None**
- "Add insertion loss of a fixture or cable" on page 53
- "Remove insertion loss of a fixture or cable" on page 54
- "Remove scope input reflection (4 Port)" on page 54
- "Add all effects of a fixture or cable" on page 55
- "Remove all effects of a fixture or cable" on page 55
- "Replace one channel element with another" on page 55
- "Relocate the observation node of a measurement" on page 56
- "Remove loading effects of a probe" on page 56
- "Remove loading effects of a DDR interposer and probe (4 Port)" on page 57
- "Relocate the observation node of a probed measurement" on page 56
- "General purpose 6 port (4 Port)" on page 58
- "General purpose probe" on page 57
- "General purpose 6 blocks (4 Port)" on page 59
- "General purpose 9 blocks (4 Port)" on page 60

General-purpose topologies with user-defined measurement and simulation points are provided for greater flexibility.

To understand the basic components that make up these configurations, look at a very simple example. The following circuit appears in the InfiniiSim Model Setup dialog box when you set the **Application Preset** field to **Add insertion loss of a fixture or cable**.



Notice some values are highlighted light blue and some are highlighted light orange. These correspond to the measurement and simulation circuit respectively.

- **Measurement Circuit** — This circuit models the actual physical circuit that produced the measured waveform.
- **Simulation Circuit** — This circuit models a hypothetical electrical circuit that exhibits your desired electrical characteristics.

In other words, the measurement circuit is what you actually measured and the simulation circuit is what you wish you would have been able to measure.

Remember that the model blocks in these configurations are used to represent a certain device and can be defined in a variety of ways (S-parameters, RLC circuits, open, thru, lossless transmission line, etc.).

InfiniiSim analyzes these two circuits and generates a correction transfer function which, when convolved with the measured waveform, produces the desired simulated waveform. For more information on transfer functions and how InfiniiSim generates them, see "InfiniiSim Transfer Functions" on page 134.

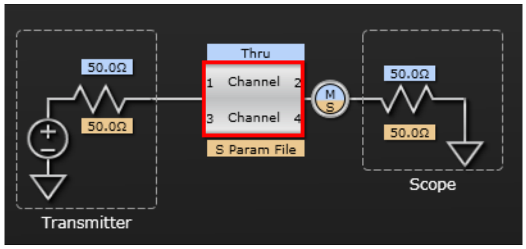
Simple One-Block Presets (4 Port)

The application presets that add or remove insertion loss of a channel element (fixture or cable) are simple one-block models inserted before the oscilloscope.

Many times you simply want to compensate for the loss of a channel element such as a cable or fixture. The one-block models can perform this task. It can also be extended to a remove/replace operation by changing the simulation parameters.

Add insertion loss of a fixture or cable

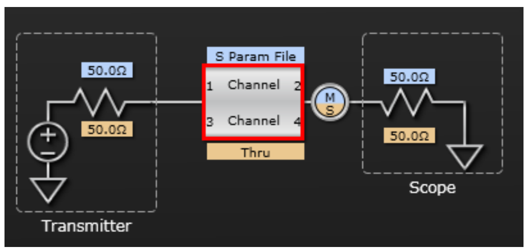
This is an embedding operation where the simulation circuit model has elements that are not in the measurement circuit model.



The gain of the block (S_{21}) is determined, and its time response is convolved with the acquired signal.

Remove insertion loss of a fixture or cable

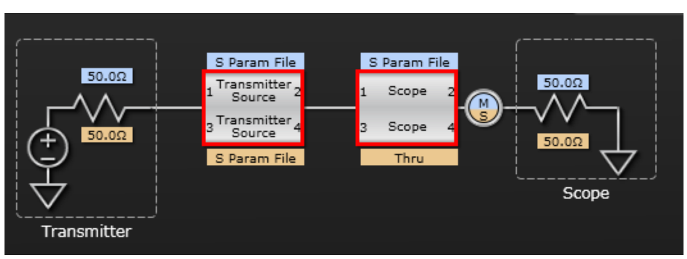
This is a de-embedding operation where the measurement circuit model has elements that are not present in the simulation circuit model.



The inverse gain of the block (S_{21}^{-1}) is determined, and its time response is convolved with the acquired signal.

Remove scope input reflection (4 Port)

Modeled using two blocks — one for the transmitter source and the other for the oscilloscope input (so the oscilloscope input reflection can be removed from the simulation circuit).

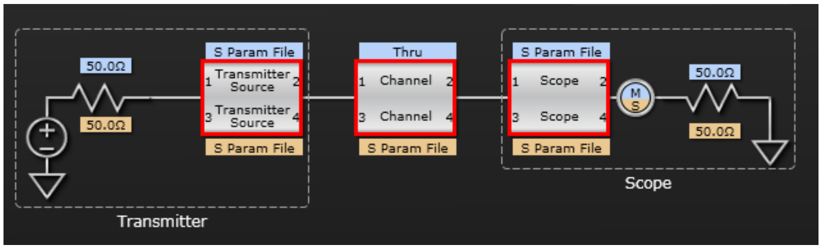


Three-Block Presets (4 Port)

When the most precision for a single-channel element removal/insertion is required, you need at least a 3-block model. These models use block descriptions for transmitter (**Transmitter Source**) and receiver (**Scope**), as well as the channel, to describe the full system. The inclusion of the transmitter and receiver blocks enable the most complete waveform rendering by including the reflective S-parameter elements in the mathematical calculation of the transfer function used to transfer from the measurement node (**M**) to the simulation node (**S**).

Add all effects of a fixture or cable

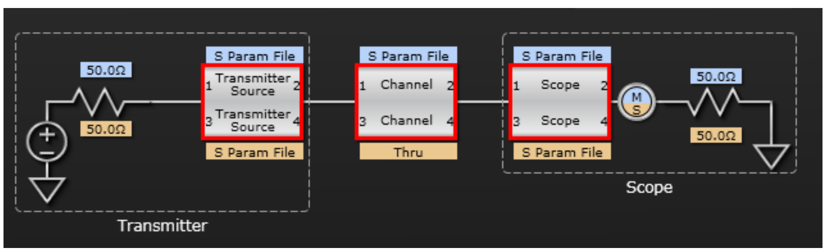
By using S-parameter models of source and oscilloscope load, a complete addition (embedding) of a channel element is performed.



This is different from "Add insertion loss of a fixture or cable" on page 53 in that the reflective interactions between the elements are taken into account. This provides the most accurate rendering to add a channel element.

Remove all effects of a fixture or cable

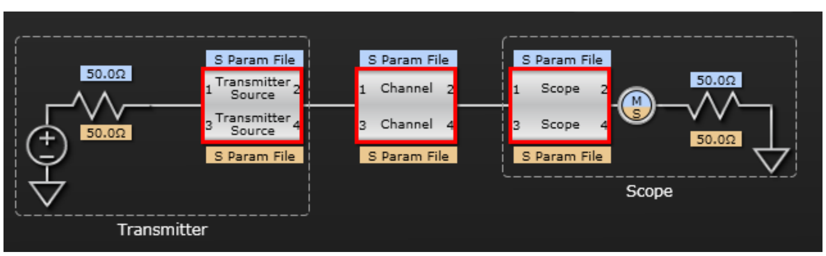
By using S-parameter models of source and oscilloscope load, the effects of a channel element are totally removed (de-embedded).



This is different from "Remove insertion loss of a fixture or cable" on page 54 in that the interactions between the elements are taken into account. This provides the most accurate way to remove a channel element.

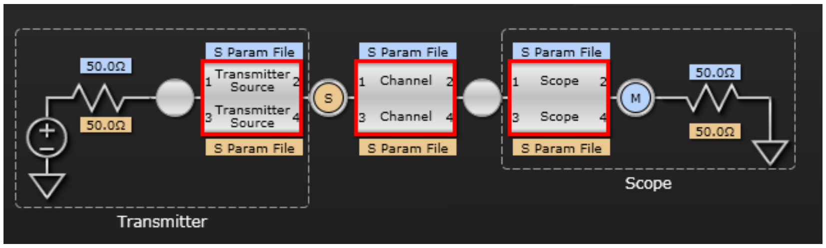
Replace one channel element with another

By using different channel element models for the measurement and simulation circuits, you can replace one channel element with another.



Relocate the observation node of a measurement

Observation node relocation lets you view any voltage waveform in a circuit by moving the observation node to any location you desire. This is an "in situ" analysis so it is not a "removal" or "addition" viewpoint.

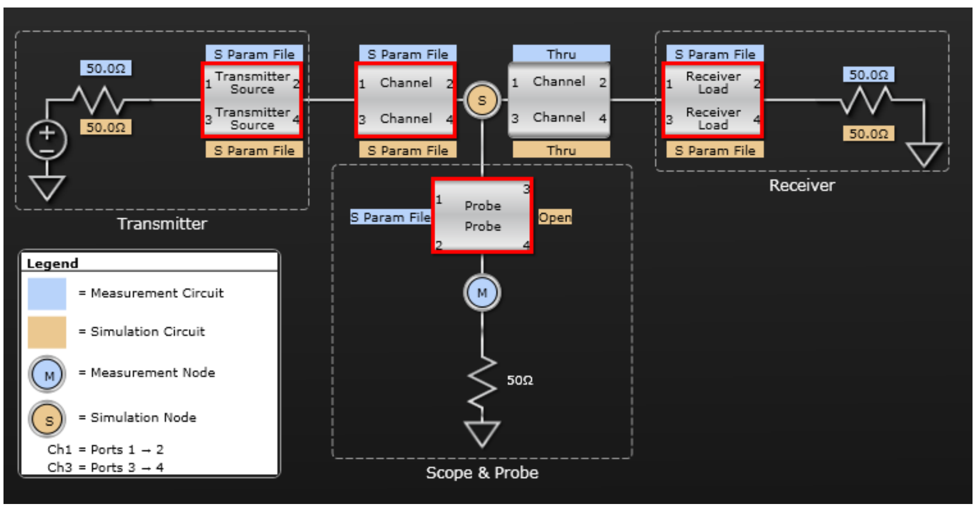


Probe Presets (4 Port)

For probe modeling, use one of the probe application presets.

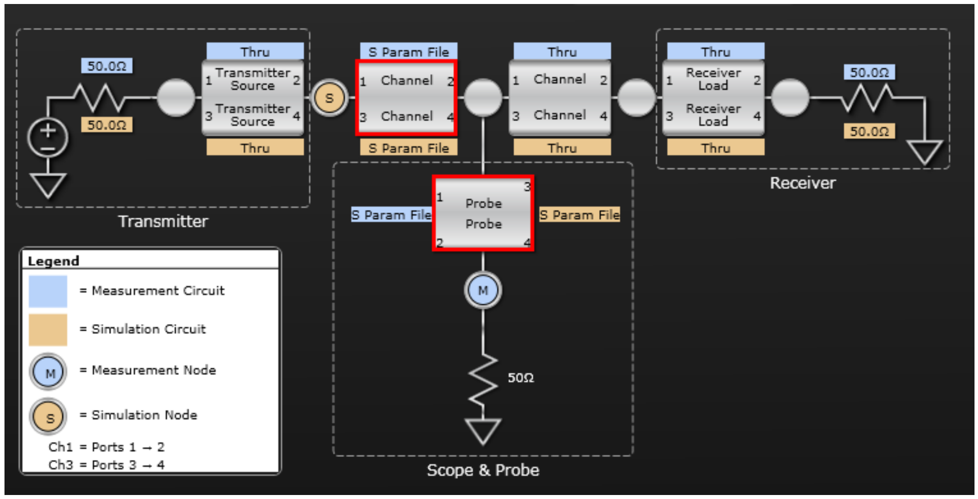
Remove loading effects of a probe

To remove the loading effects of a probe, a topology of circuit blocks is given that lets probe models be considered in the measurement.

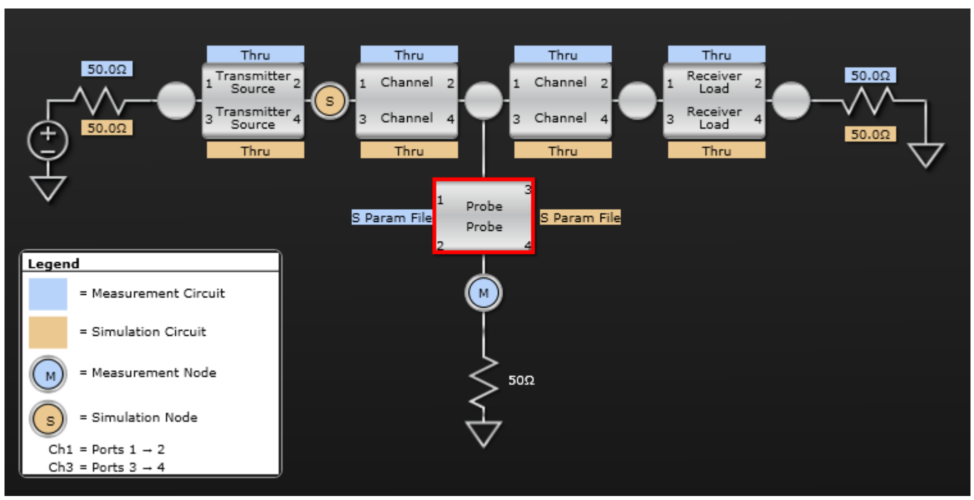


An oscilloscope probe, while it might be defined as "high impedance", really does have a loading effect on the circuit. This effect can be taken into account and removed.

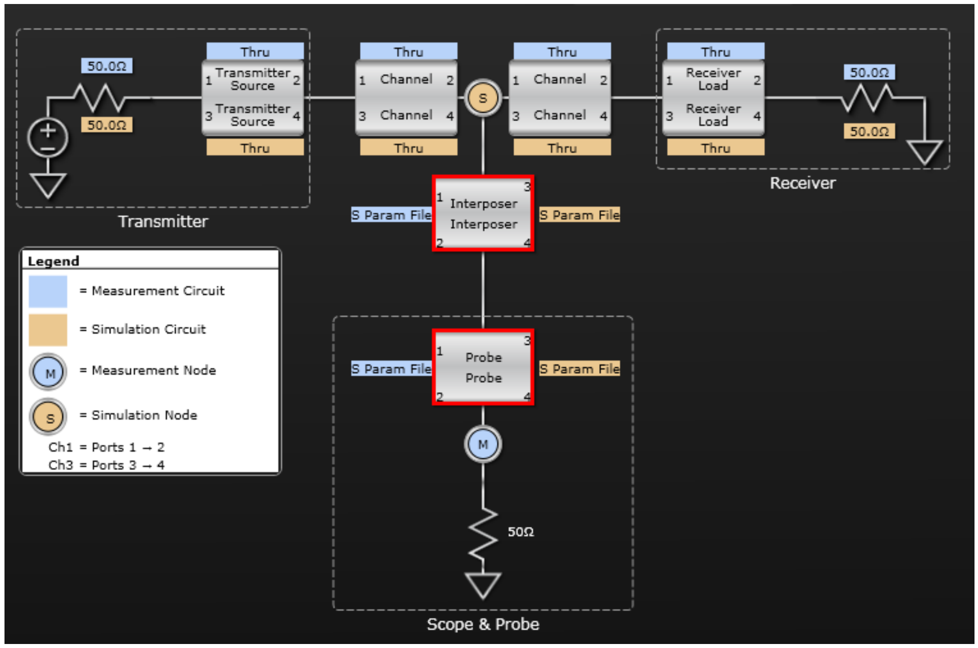
Relocate the observation node of a probed measurement



General purpose probe

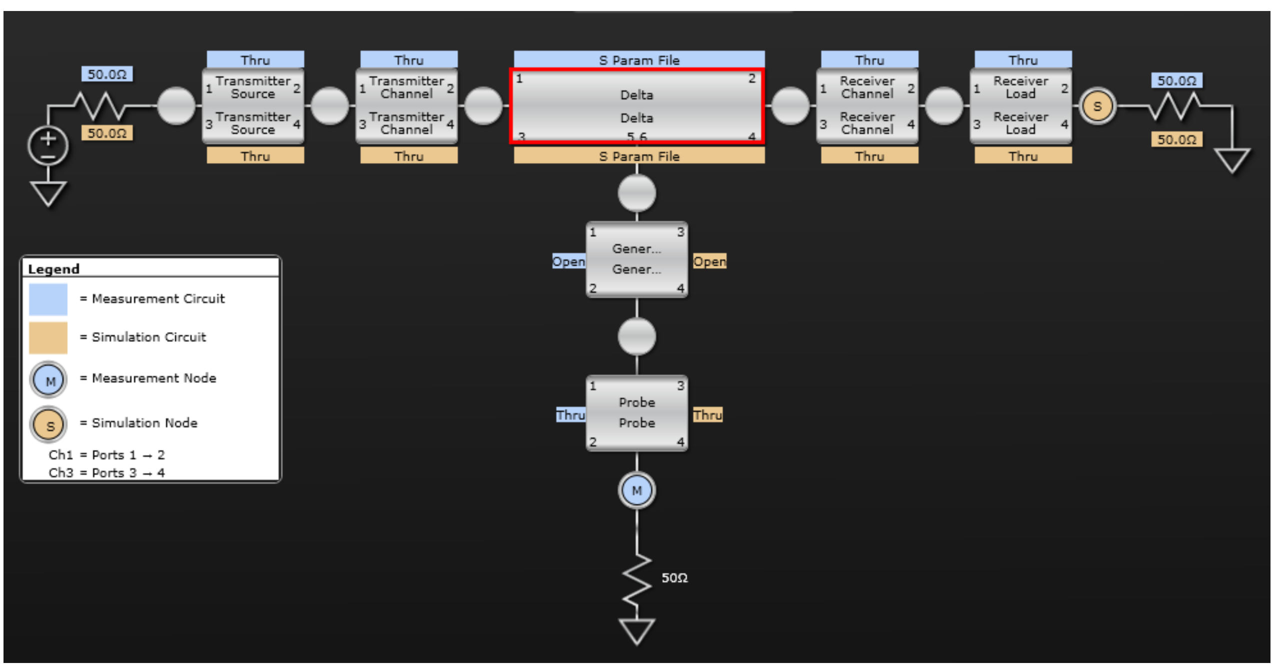


Remove loading effects of a DDR interposer and probe (4 Port)



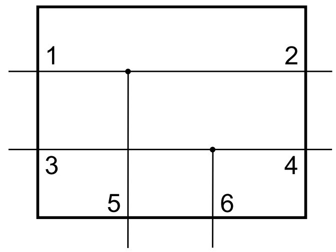
General purpose 6 port (4 Port)

This application preset is similar to **General purpose probe** but different in that it has the **Delta** block into which you can load a 6-port S-parameter (s6p) file.

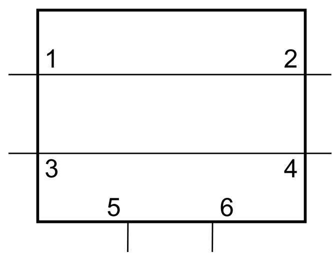


In addition to specifying an S-parameter file for the Delta block, you can also set it to:

- **Ideal Thru** — In this case, ports 1, 2, and 5 are shorted together and it ports 3, 4, and 6 are shorted together.

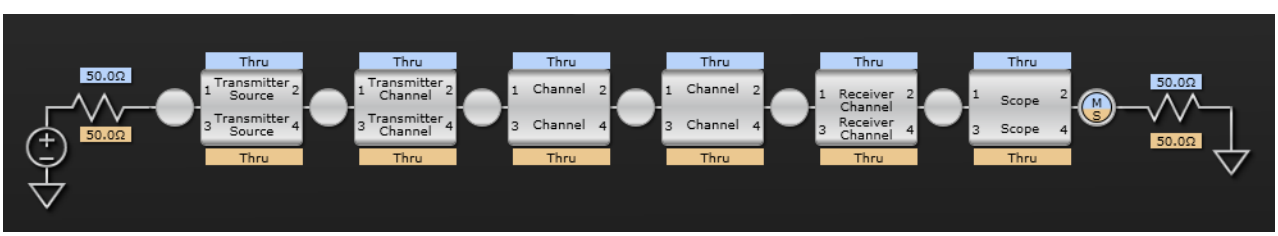


- **Open** — In this case, ports 1 and 2 are shorted together, ports 3 and 4 are shorted together, and ports 5 and 6 are open.



Similar to how you can specify a 4-port block as two 2-port blocks, you can specify a 6-port block as two 3-port blocks. In this case, one of the blocks is for ports 1, 2, and 5, and the other block is for ports 3, 4, and 6. When you do this, it is done for both the measurement circuit and the simulation circuit.

General purpose 6 blocks (4 Port)

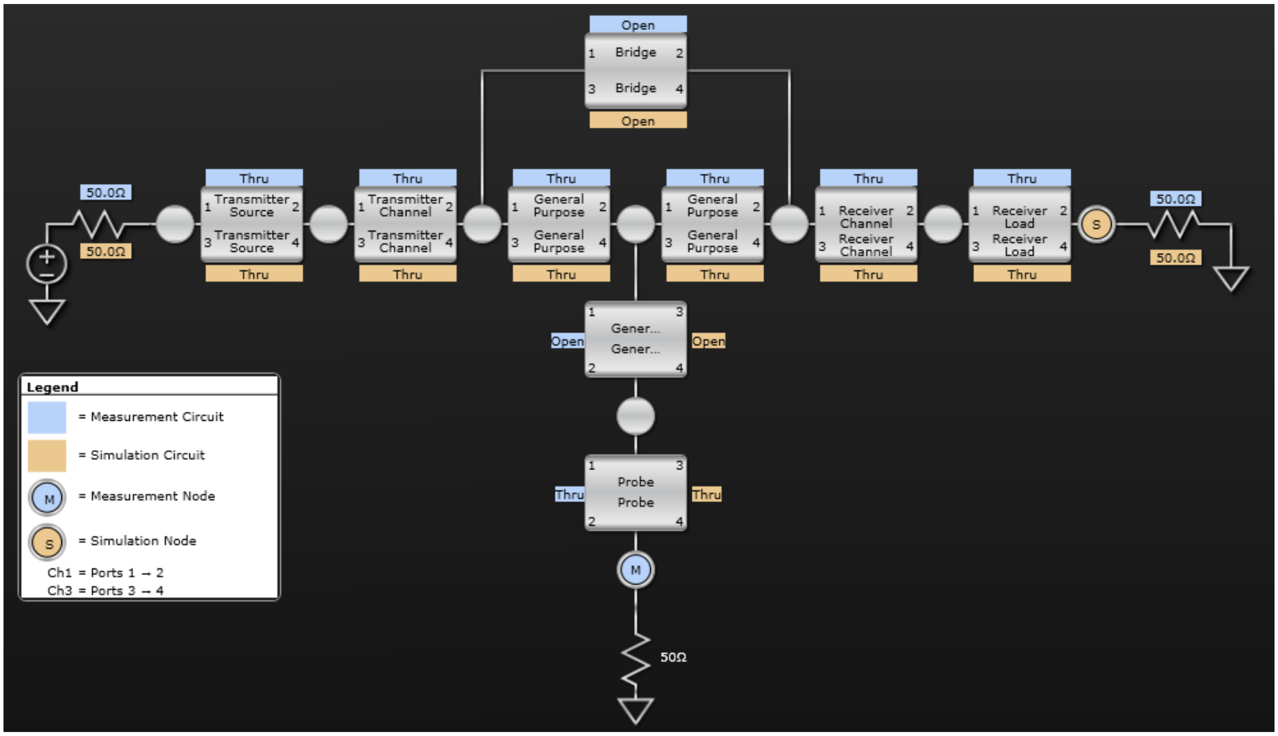


The general-purpose 6-block model is used for SMA differential probe modeling in the majority of cases.

General purpose 9 blocks (4 Port)

For very sophisticated applications (for example, using both high-impedance probes and differential SMA probe heads), the general-purpose 9-block application preset can be used to describe these complex cases.

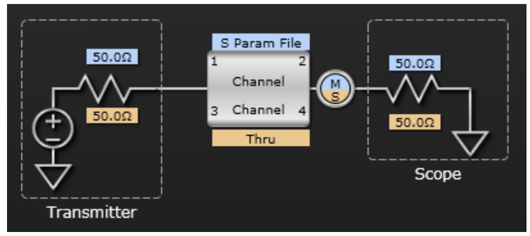
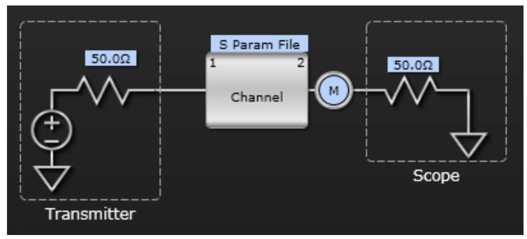
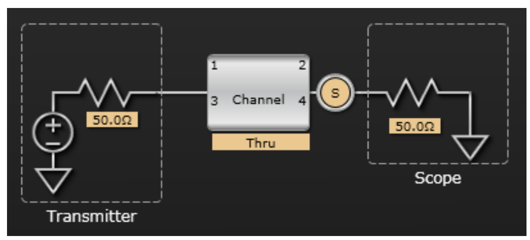
The general-purpose 9-block application preset is the model from which all other application presets can be made.



Each block in the 9-block model can be defined as having a combination of up to three elements in cascade, series, or parallel arrangements, giving you 27 total possible circuit elements to define for the most sophisticated scenarios.

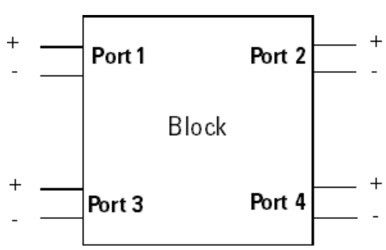
Circuit_Diagram View (4 Port)

If you look under the **Circuit Diagram View** section in the InfiniiSim Model Setup dialog box, you see that you have three viewing options.

Measurement & Simulation Circuits	
Measurement Circuit Only	
Simulation Circuit Only	

These choices let you build/view the measurement circuit, the simulation circuit, or both at the same time.

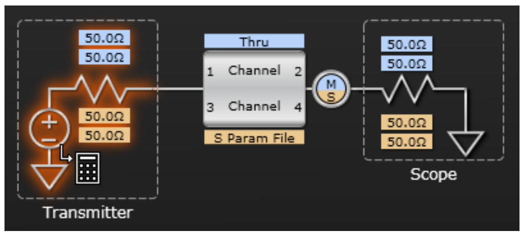
It is important to note that these circuit model representations in the GUI are simplified for ease of viewing. If we were being explicit, the model blocks in the circuit would look like the following for a 4-port device:



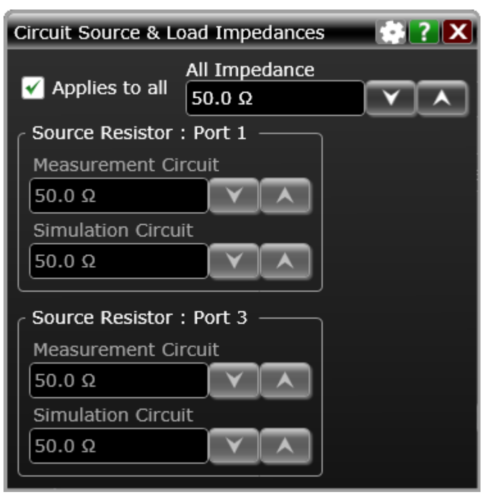
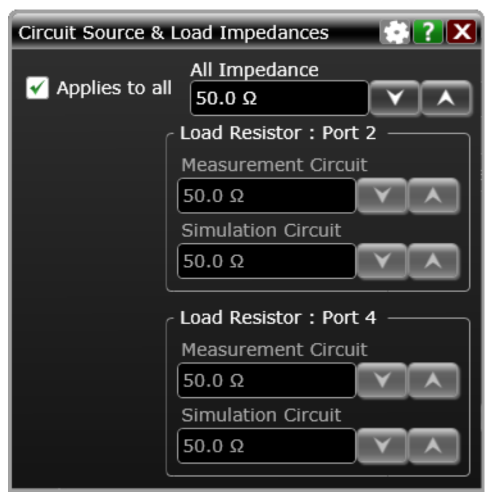
The diagrams in the GUI are completely interactive so you can click various sections or components and make adjustments.

Circuit Source and Load Impedances (4 Port)

The resistor on the left side of the circuit corresponds to the impedance of the transmitter (source). The resistor on the right side of the circuit corresponds to the impedance of the receiver or oscilloscope (load). These impedances are defaulted to 50 Ω, but you can change them if needed. Simply place your pointer over the resistor (it thickens to indicate you are on top of it) and click.



You then see one of the following Circuit Source and Load Impedances dialog boxes.

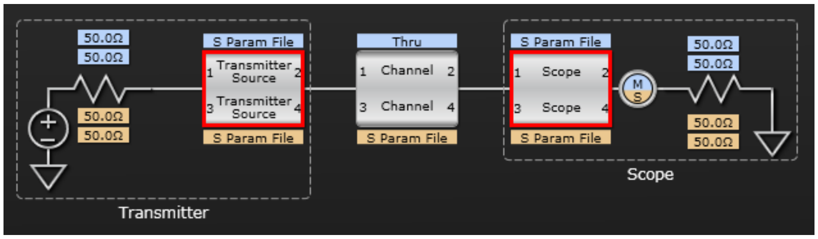
<table>
<tr><th>Source</th><th>Load</th></tr>
<tr><td>
</td><td>
</td></tr>
</table>

The Circuit Source and Load Impedances dialog box lets you specify the source or load impedances within both the measurement and simulation circuits.

If you want all impedances in both the measurement and simulation circuits to be identical, check the **Apply to All** box and set the **All Impedance** value. All other impedances are then changed to match this one.

If the the **Apply to All** box was not checked, the **Add Insertion Loss** application preset (used in the preceding screen) would gray-out the load impedance controls because the oscilloscope load impedance is always 50 Ω.

Note that by using one of the 3-block models (for example, the **Remove All Effects of a Fixture or Cable** application preset), you can apply an arbitrarily complex transmitter and/or oscilloscope impedance using the additional model blocks dedicated to forming the transmitter source impedance and oscilloscope input impedance.

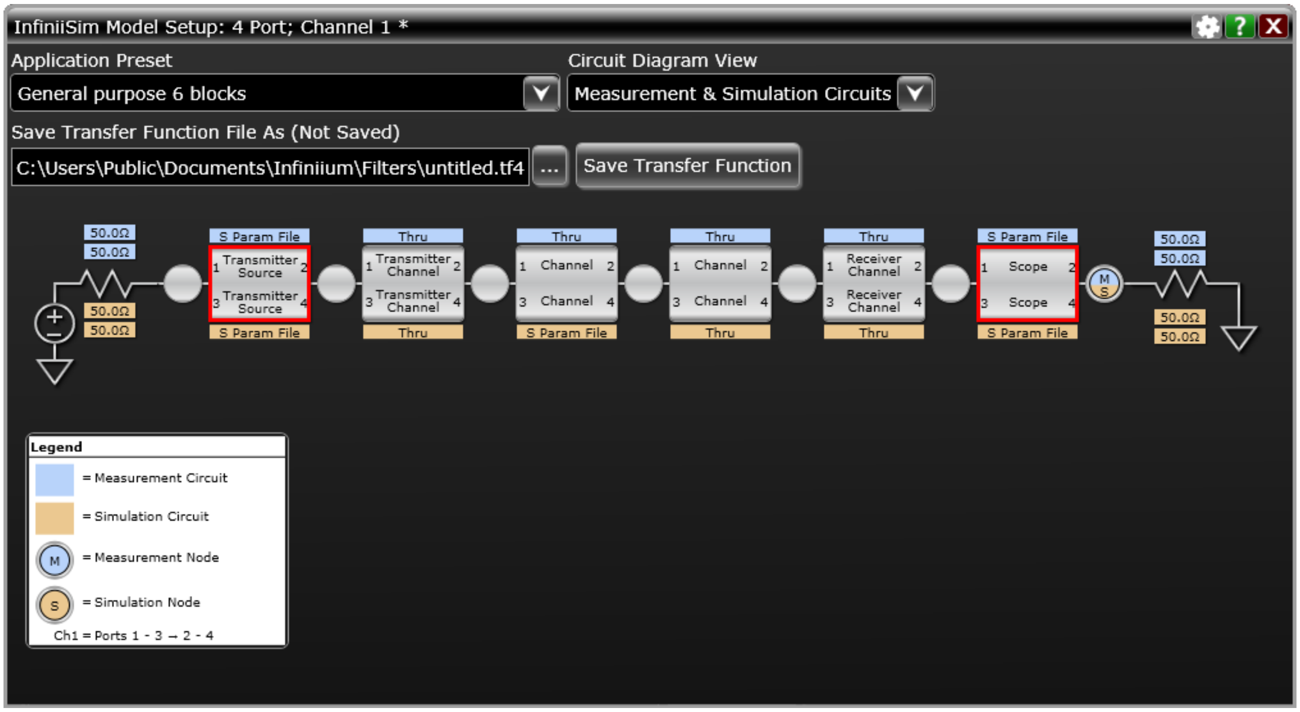


Note that any termination block located adjacent to a source or load resistor (like the **Transmitter Source** block and **Scope** block) can have 1-port S-parameter files assigned to them in 2-port InfiniiSim mode. These are the only locations in a 2-port InfiniiSim circuit where you can place 1-port S-parameter (s1p) files. Likewise, termination blocks can be defined by 2-port S-parameter files in 4-port mode. You can also use a 1-port S-parameter file for a 4-port application, which will apply those S-parameters to S_{11} and S_{33} and set S_{13} and S_{31} to zero.

Defining a Block (4 Port)

The blocks used in creating the measurement and simulation circuits may be used to model various components (cable loss, probe loading, a fixture, etc.).

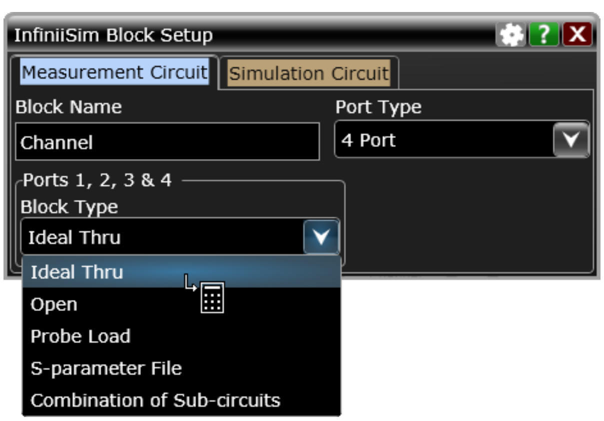
InfiniiSim assigns names to the blocks in the legend for a typical use scenario depending on the application preset chosen. These names are just a convenience for referring to blocks and an example of how the block may be used. However, you can define them any way you like. You can change these names by using the **Block Name** control inside the InfiniiSim Block Setup dialog box (accessed by clicking one of the blocks).



While blocks represent different parts of the circuit, they have the exact same definition controls to choose from and are called different types to help you understand what they might represent in your circuit.

The termination blocks, **Transmitter Source** and **Scope**, are different in one respect—those blocks can be defined by return-loss-only S-parameter files (2-port file in 4-port InfiniiSim analysis mode). You can also use a 1-port S-parameter (s1p) file for a 4-port application, which will apply those S-parameters to S_{11} and S_{33} and set S_{13} and S_{31} to zero.

When you click a block, the InfiniiSim Block Setup dialog box opens.



The tabs at the top of the dialog box (**Measurement Circuit** and **Simulation Circuit**) let you specify whether you are currently defining a block in the measurement circuit or in the simulation circuit. For instance, if you select the **Simulation Circuit** tab and then set the **Block Type** to **Ideal Thru**, then the corresponding block in the simulation circuit is set to a Thru block.

The **Block Name** control lets you customize the name of the block.

Next, you need to select the **Block Port Type**.

Using full 4-port S-parameter models is often necessary because they include cross-coupling components between all four ports in the model. However, it is sometimes more convenient to split the 4-port model into two uncoupled 2-port models. You can, for example, choose to use two 2-port S-parameter files to model a pair of coaxial cables. Another example is when you have an accurate S-parameter file for the differential path through a device, but not for the common-mode path. You can use a 2-port differential S-parameter file to model the differential path through the device and an RLC block to model the common-mode path.

Consequently (in 4-port analysis mode), you can select these block port types:

- **(2) 2 Port** — You can define the 4-port model blocks as a combination of two uncoupled 2-port model blocks—one for the path between ports 1 and 2 and one for the path between ports 3 and 4.

  In this case, you can define each 2-port Block Type independently to be an **Ideal Thru**, **Open**, **RLC**, **Probe Load**, **S-parameter File**, **Transmission Line (Lossless)**, or **Combination of Sub-circuits**.

- **(2) 2 Port Differential** — You can define the 4-port model block as a combination of two different uncoupled 2-port model blocks—one for the differential path through the device and one for the common-mode path through the device.

  In this case, you can define each 2-port Block Type independently to be an **Ideal Thru**, **Open**, **RLC**, **Probe Load**, **S-parameter File**,**Transmission Line (Lossless)**, or **Combination of Sub-circuits**.

- **4 Port** — This uses a standard 4-port model.

  In this case, you can define the Block Type to be an **Ideal Thru**, **Open**, **Probe Load**, 4-port **S-parameter File**, or **Combination of Sub-circuits**. **RLC** and **Transmission Line (Lossless)** block types are not available in 4-port Block Port Type mode.

The bottom section (**Block Type**) is where you actually define the block. There are six choices in this section.

- **Ideal Thru** — Nothing happens to the signal as it passes through a Thru block. It is as though the block was not there and port 1 is shorted directly to port 2.
- **Open** — When you set the block to open, no signal passes through it. It is as though port 1 and port 2 form an open circuit.
- **RLC** — This lets you set up the block as an RLC (resistor, inductor, capacitor) network. See "RLC Block Type (4 Port)" on page 66.

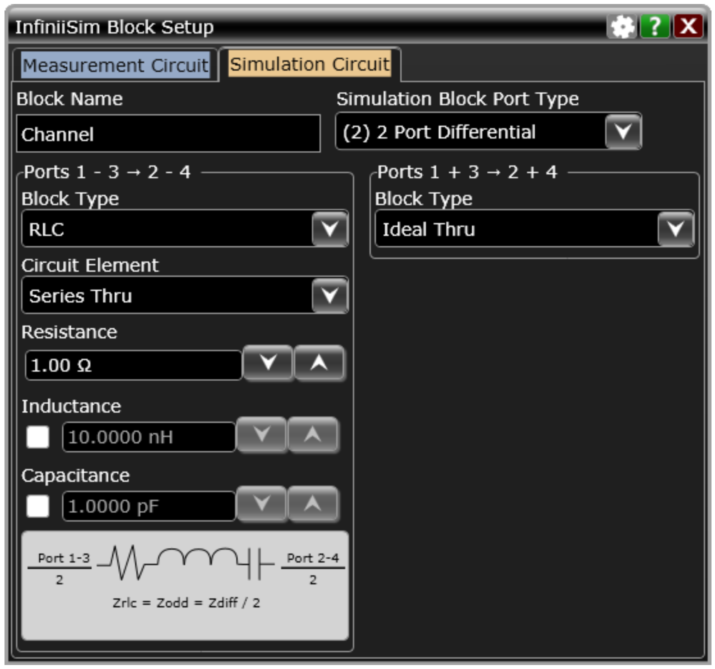
- **Probe Load** — This selection allows the modeling of a probe load in shunt with the signal path. This model is used when the measured or simulated signal does not travel through the probe, but is only loaded by a probe from a different channel. See "Probe Load Block Type (4 Port)" on page 74.
- **S-parameter File** — If you have an S-parameter file that models a fixture, cable, probe, or device in your circuit, you can load it into InfiniiSim and use it for the block definition. See "S-Parameter File Block Type (4 Port)" on page 67.
- **Transmission Line (Lossless)** — This defines the block as an ideal lossless transmission line. This block is defined by a characteristic impedance value and a time delay value. See "Transmission Line Block Type (4 Port)" on page 70.
- **Combination of Sub-circuits** — This selection lets you use up to three blocks in place of the one. These three sub-blocks can be arranged in cascade, parallel, or series. See "Combination of Sub-Circuits Block Type (4 Port)" on page 70.
- **Probe** — This selection models the effects of the probe that is connected to the current InfiniiSim channel. The model is inserted in series with the signal path and is used when the measured or simulated signal travels through the probe to the input of the oscilloscope. See "Probe Block Type (4 Port)" on page 73.

Once your circuits are built, select the appropriate measurement node location and the desired simulation node location (see "Measurement and Simulation Node Locations (4 Port)" on page 76). Then generate, save, and apply the correction transfer function (see "Generating/Saving a Transfer Function File" on page 78).

RLC Block Type (4 Port)

The RLC block type is available only in the **(2) 2 Port** and **(2) 2 Port Differential** port type modes. In the (2) 2 Port mode, either single-ended path of the model block can be defined by a 2-port RLC model. In the (2) 2 Port Differential mode, either the differential or common-mode paths can be defined by a 2-port RLC model.

When you select **RLC** for the **Block Type** (not available in 4-port Block Port Type mode) and the **Block Port Type** is set to **(2) 2 Port Differential**, the dialog box looks like the following.

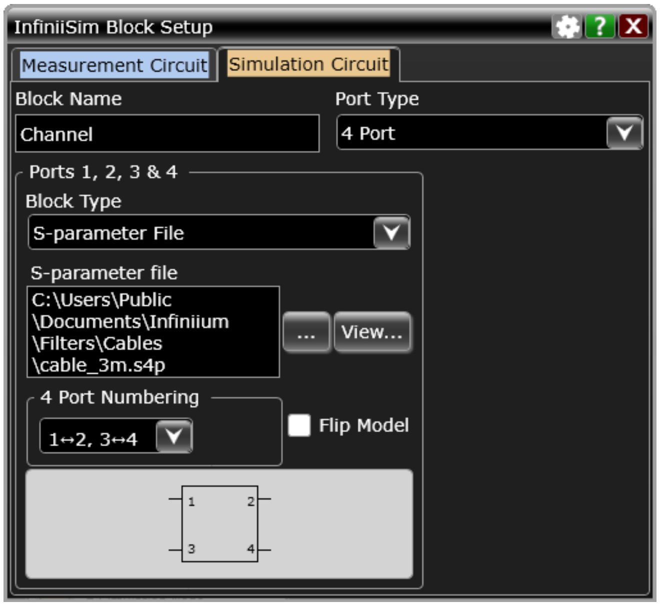


In the **Circuit Element** section, there are four configuration options (**Series Thru**, **Parallel Thru**, **Series Shunt**, and **Parallel Shunt**). When you select one of these options, the basic circuit configuration is shown in the dialog box (see preceding screen).

After selecting the configuration of the RLC circuit, you can set the values for the **Resistance**, **Inductance**, and **Capacitance**. To use an inductor or capacitor in your RLC circuit, check the corresponding box and then enter the value.

S-Parameter File Block Type (4 Port)

When you select **S-parameter File** for the block type, additional controls appear in the dialog box.



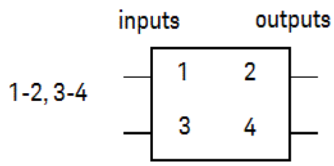
You can click the **S-parameter File** field to browse to your saved S-parameter files. S-parameter files are typically generated using a vector network analyzer or circuit simulator, for example, and then saved on the oscilloscope hard drive. These can be files that model fixtures, cables, devices, or probes in your circuit. InfiniiSim accepts both Touchstone files (.s1p, .s2p, .s4p) and CITIfile (.cti, .cit) files.

Clicking the **View...** button opens the InfiniiSimToolbox application for viewing and evaluating S-parameter files. See "S-Parameter and Transfer Function File Viewers" on page 127.

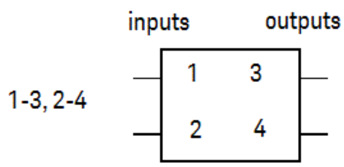
InfiniiSim can read 4-port or 2-port files in **4 Port Block Port Type** mode. It can read 4-port, 2-port, and 1-port files in **(2) 2 Port** or **(2) 2 Port Differential Block Port Type** mode. If you use a 1-port S-parameter (s1p) file for a 4-port application, those S-parameters will apply to S_{11} and S_{33} and set S_{13} and S_{31} to zero. Additional information about reading S-parameter files can be found in "S-Parameter Considerations" on page 113.

By default, the oscilloscope expects ports 1 and 3 to be on the left side of the block (if you have the **Block Port Type** set to **4 Port**) and ports 2 and 4 to be on the right. However, this may not match the way the data is formatted in your S-parameter file. Therefore, you can check the **Flip Model** box to have the ports switch sides.

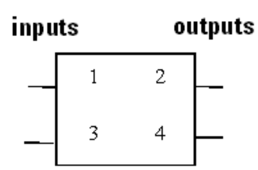
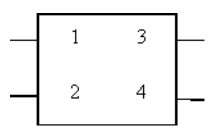
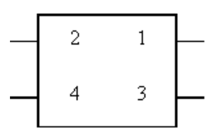
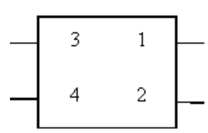
You may also see the **4 Port Numbering** control depending on your settings. The **4 Port Numbering** control lets you specify how the data is arranged in the 4-port S-parameter file. There is an industry-standard numbering system:



However, you can tell InfiniiSim that your file actually uses the following numbering system.



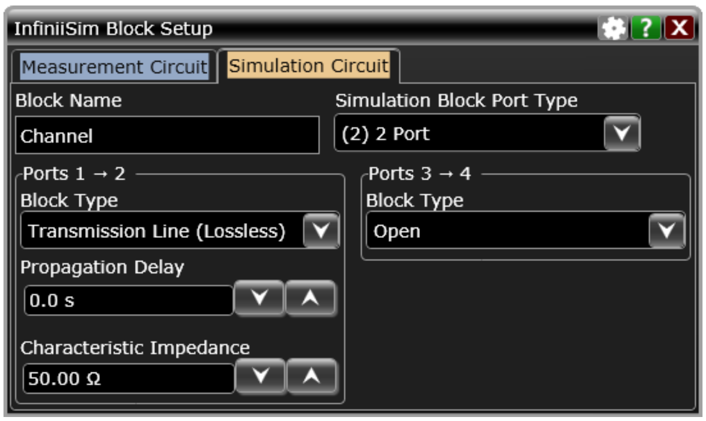
You can also use the **Flip Model** box to flip the numbers between the right side and the left side.

Flip Model	4 Port Numbering	How ports are connected in a block
No	1→2, 3→4	
No	1→3, 2→4	
Yes	1→2, 3→4	
Yes	1→3, 2→4	

For more information on S-parameter files in general, see "S-Parameter Considerations" on page 113.

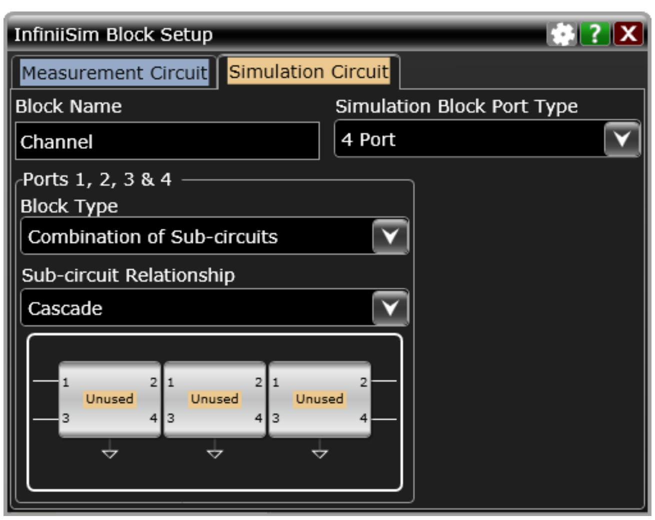
Transmission Line Block Type (4 Port)

When you select **Transmission Line (Lossless)** for the block type, two additional fields appear for setting the **Characteristic Impedance** of the transmission line in your model block as well as its **Propagation Delay**.



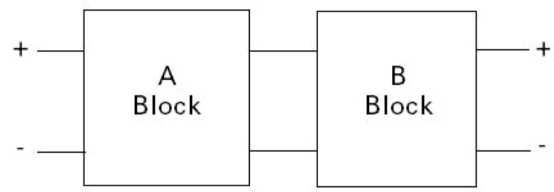
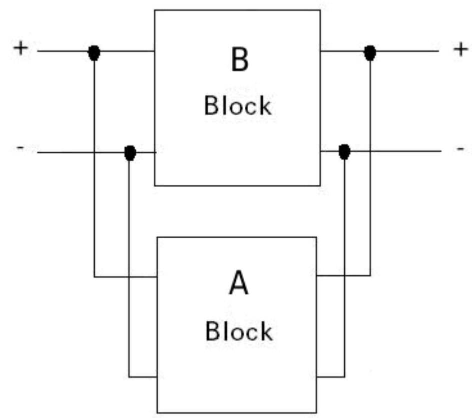
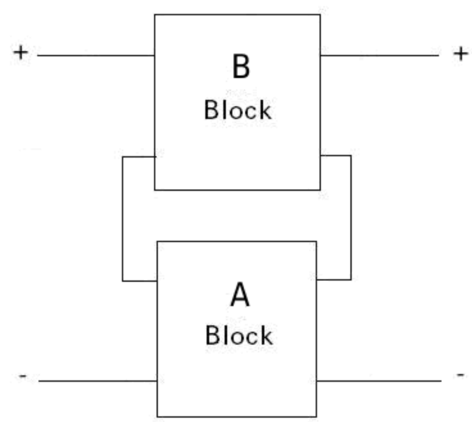
Combination of Sub-Circuits Block Type (4 Port)

The **Combination of Sub-circuits** selection lets you use a combination of three sub-blocks in place of one block. When you make this selection, additional controls appear at the bottom of the dialog box.

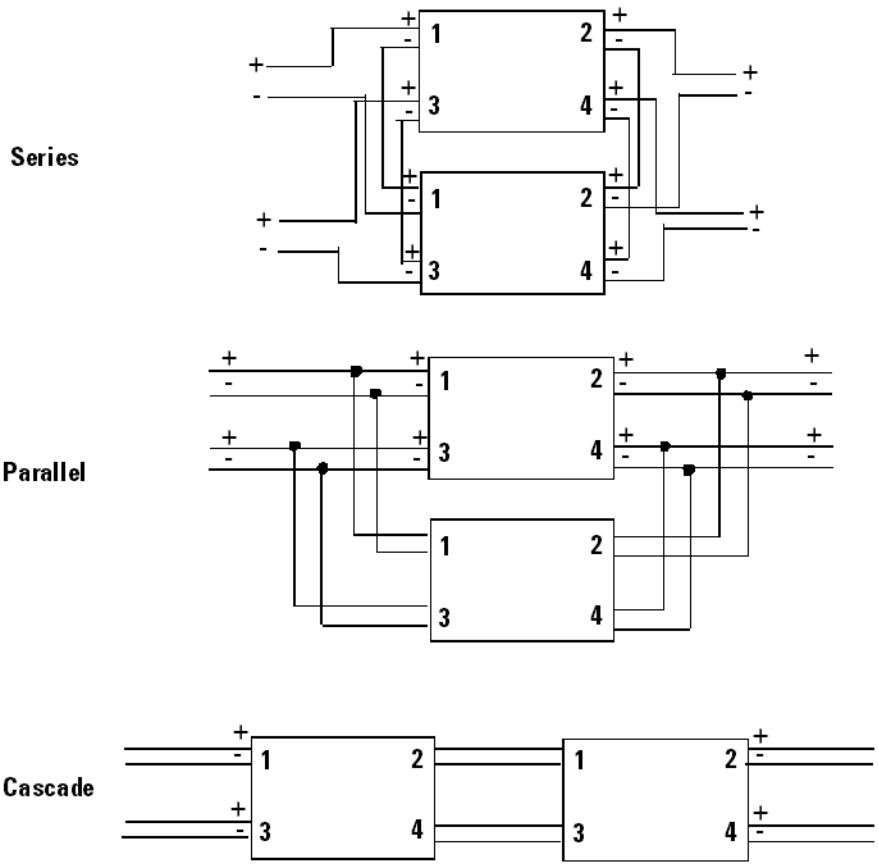


You have the following three choices for how to combine the three individual blocks that now make up the one circuit block: **Cascade**, **Parallel**, and **Series**.

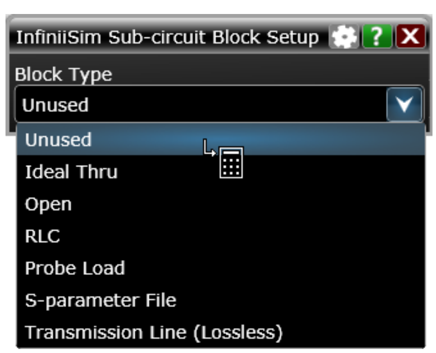
In **(2) 2 Port** and **(2) 2 Port Differential** port type modes, the blocks combine as shown in the following table (pictures show 2-port connections with two blocks for simplicity).

Cascade	
Parallel	
Series	

In **4 Port** port type mode, the 4-port blocks combine as shown in the following picture (pictures show connections for two of the three blocks for simplicity):



Once you select one of these three configurations, you can click each individual block and define what type of block it is. When you click one of the blocks a new InfiniiSim Sub-circuit Block Setup dialog box appears where you can define its type:



Notice that you can choose between **Ideal Thru**, **Open**, **RLC**, **Probe Load**, **S-parameter File**, **Transmission Line (Lossless)**, and an option called **Unused**. Unused means that this particular sub-block is not used (letting you use 2 sub-blocks instead of three, for example).

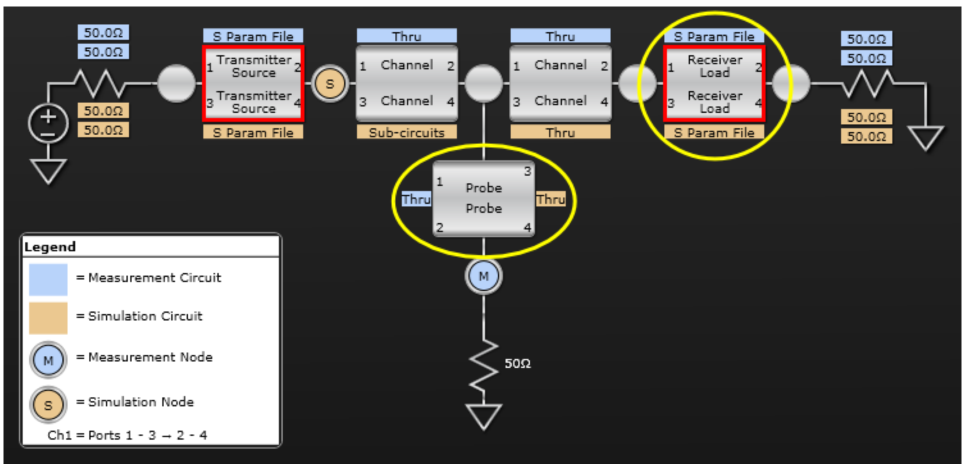
Using the **Unused** setting has a different effect on this specific block depending on which configuration is being used:

- For the **Cascade** structure, for example, **Unused** acts like a **Thru** block.
- For the **Parallel** structure, **Unused** acts like an **Open** block.

Probe Block Type (4 Port)

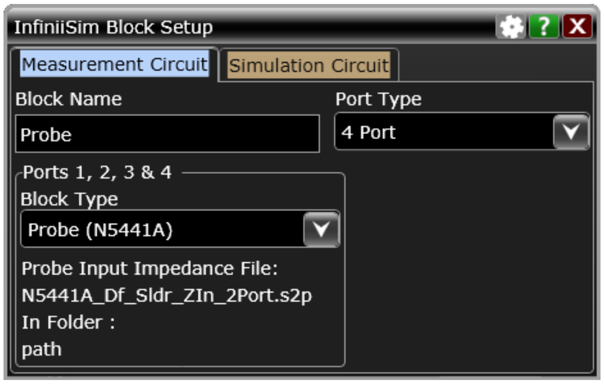
The **Probe** block type selection models the effects of the probe that is connected to the current InfiniiSim channel. The model is inserted in series with the signal path and is used when the measured or simulated signal travels through the probe to the input of the oscilloscope.

You can select the **Probe** block type only in the blocks that are intended to model the oscilloscope input (for example, the ones circled in yellow).



A probe must be connected to the channel for the **Probe** block type selection to be active.

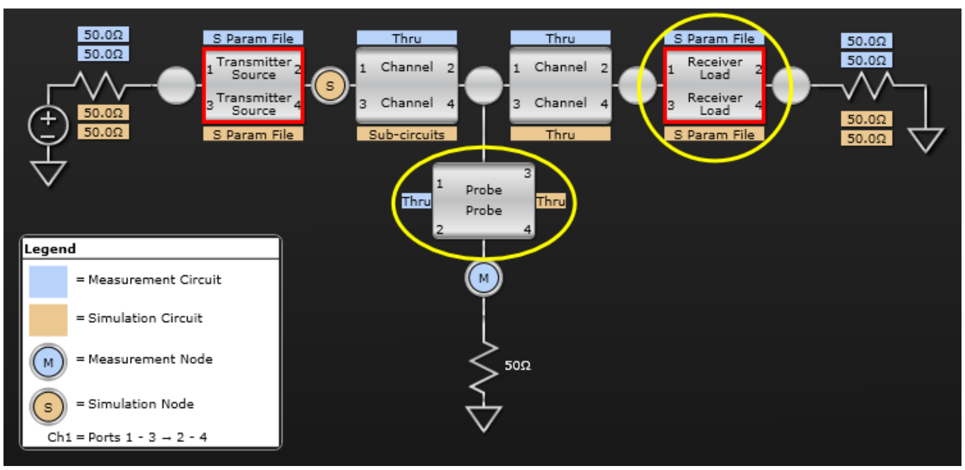
When you make the **Probe** block type selection, the appropriate probe input impedance file is automatically selected, based on your selections in the Probe Configuration dialog box.



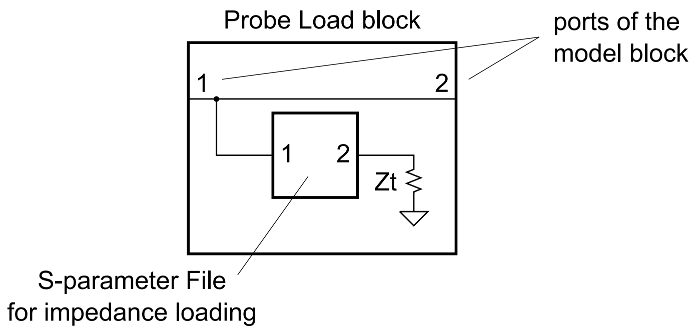
Probe Load Block Type (4 Port)

The **Probe Load** block type selection lets you model a probe load in shunt with the signal path. This model is used when the measured or simulated signal does not travel through the probe, but is only loaded by a probe from a different channel.

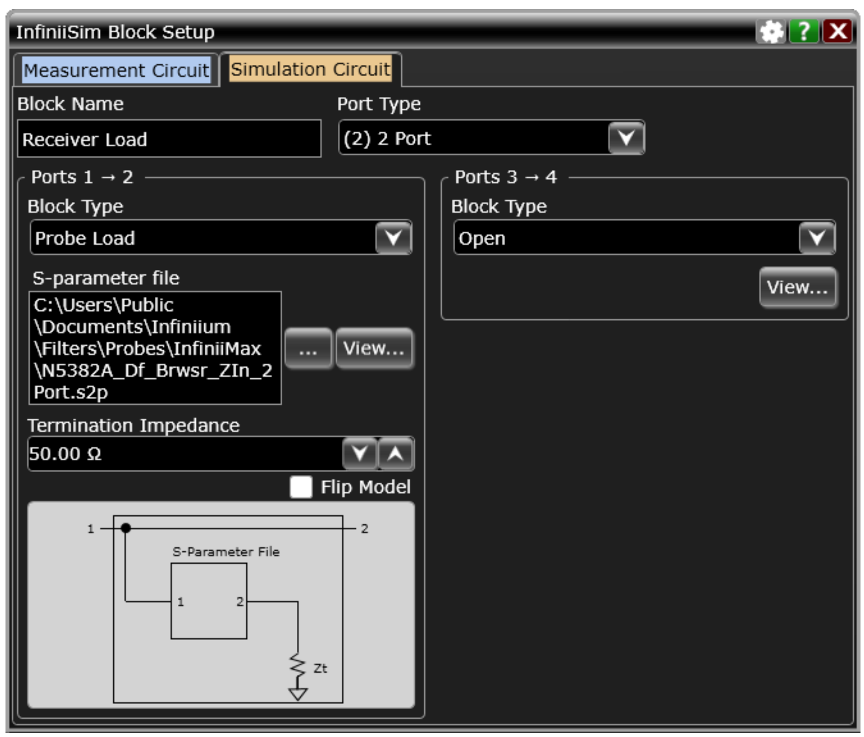
The **Probe Load** block type is not allowed in those blocks that are intended to model the oscilloscope input (for example, the ones not circled in yellow).



When the **Probe Load** block type is selected it does two things: port 1 and 2 of the block are connected together (like an Ideal Thru) and the supplied S-parameter file is used to describe the shunt impedance loading the connected ports.



When you make the **Probe Load** block type selection, additional controls appear at the bottom of the dialog box:



- **S-parameter file** — Used to describe an impedance loading.

  If a 1-port S-parameter (s1p) file is selected, you can ignore port 2 of the load and the Termination Impedance (Zt).

  If a 2-port S-parameter (s2p) file is selected, port 2 of the load is as shown, and the desired Termination Impedance (Zt) should be entered.

  Input impedance files in are located in the "C:\Users\Public\Documents\Infiniium\Filters\Probes\InfiniiMax" directory.

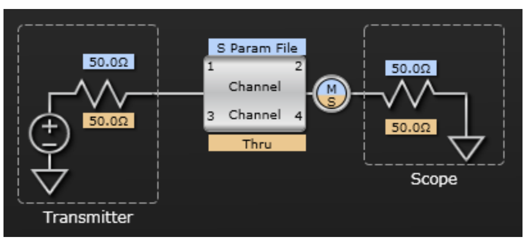
Clicking the **View...** button opens the InfiniiSimToolbox application for viewing and evaluating S-parameter files. See "S-Parameter and Transfer Function File Viewers" on page 127.

- **Termination Impedance** — Again, with a 1-port S-parameter (s1p) file, you can ignore the Termination Impedance (Zt). With a 2-port S-parameter (s2p) file, the desired Termination Impedance (Zt) should be entered.
- **Flip Model** — If your S-parameter file has a different port ordering, you can check this box to switch the selected 4-port numbering.
- **4 Port Numbering** — If your S-parameter file has a different port ordering, you can check this box to switch the selected 4-port numbering.

Measurement and Simulation Node Locations (4 Port)

You can also change the location of the measurement and simulation nodes. The measurement node corresponds to the location in the circuit where you actually measured the original signal. The simulation node corresponds to the location in the circuit where you wished you could have measured the signal.

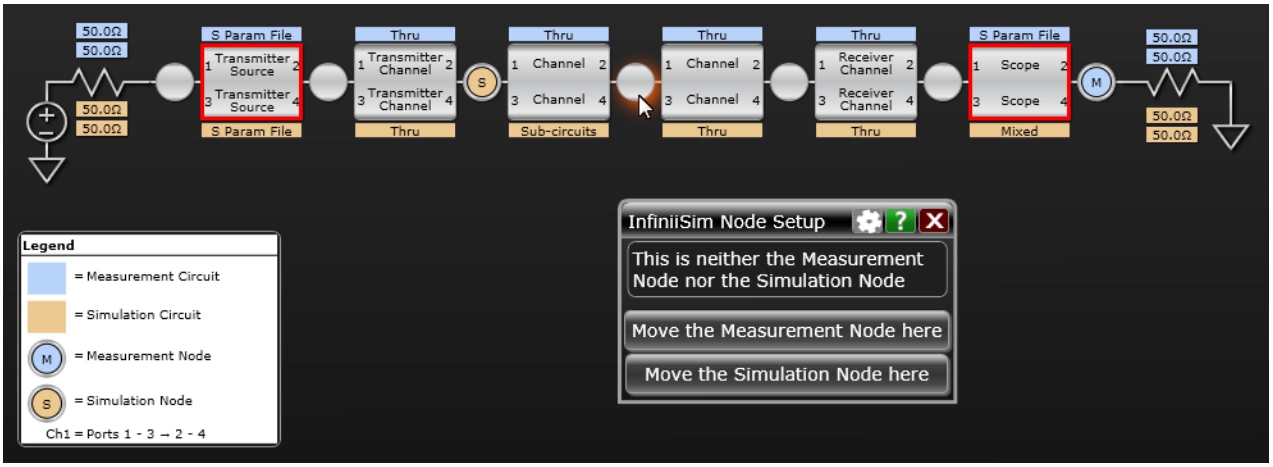
The following model shows both the measurement and simulation circuit view. The light blue circle labeled **M** is the measurement node and the light orange circle labeled **S** is the simulation node.



In this 1-block model, the measurement and simulation nodes cannot be moved anywhere else in the circuit (there are no other large black dots or nodes in the circuit) because this simple preset only lets you add insertion loss of a device.

However, in more complex configurations such as the **General purpose 6 blocks** application preset configuration shown next, you can change the location of the measurement and/or simulation node.

To change the nodes, click one of the empty node locations. Then, select either **Move the Measurement Node here** or **Move the Simulation Node here** from the InfiniiSim Node Setup dialog box (as shown here).



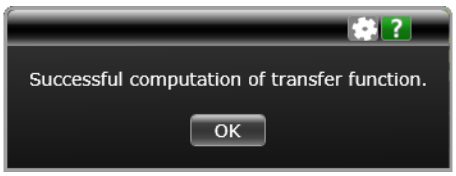
50.0Ω
50.0Ω
50.0Ω
50.0Ω
S Param File
Transmitter Source
Transmitter Source
S Param File
Thru
Transmitter Channel
Transmitter Channel
Thru
S
Thru
Channel
Channel
Sub-circuits
Thru
Channel
Channel
Thru
Thru
Receiver Channel
Receiver Channel
Thru
S Param File
Scope
Scope
Mixed
M
50.0Ω
50.0Ω
50.0Ω
50.0Ω
Legend
= Measurement Circuit
= Simulation Circuit
= Measurement Node
= Simulation Node
Ch1 = Ports 1 - 3 → 2 - 4
InfiniiSim Node Setup
This is neither the Measurement Node nor the Simulation Node
Move the Measurement Node here
Move the Simulation Node here

Generating/Saving a Transfer Function File

Once you have your measurement and simulation circuits built (see "Creating a Transfer Function From a Model (2 Port)" on page 25 or "Creating a Transfer Function From a Model (4 Port)" on page 51), you are ready to generate and save your transfer function file.

In the upper right corner of the InfiniiSim Model Setup dialog box, click the **Save Transfer Function...** button. This saves the transfer function file to the name (and to the location) specified in the **Save Transfer Function File As** field.

After you click the **Save Transfer Function** button, a window appears telling you that InfiniiSim is computing the transfer function. If it successfully computes the transfer function, the following message appears.



If InfiniiSim does not successfully compute the transfer function and instead gives you an error/warning message, see "InfiniiSim Warning/Error Messages" on page 96 for information regarding each of these messages and possible corrective actions.

Close all of the dialog boxes and you will see the simulated waveform on your oscilloscope's display. You can toggle back and forth between the measured waveform and the simulated waveform by entering the Channel dialog box (**Setup > Channel N...**) and turning InfiniiSim on or off (by selecting either **Off**, **2 Port**, or **4 Port**).

Note that when comparing simulated waveforms to measured waveforms (InfiniiSim On/Off), you should use bandwidth limiting to see the transformation difference only. (For more information on the bandwidth-limiting feature, see "Starting InfiniiSim" on page 13.)

InfiniiSim Plots

When InfiniiSim is enabled and the **Display InfiniiSim Graphs** check box in the Channel dialog box is selected, the InfiniiSim window appears on your oscilloscope's display.

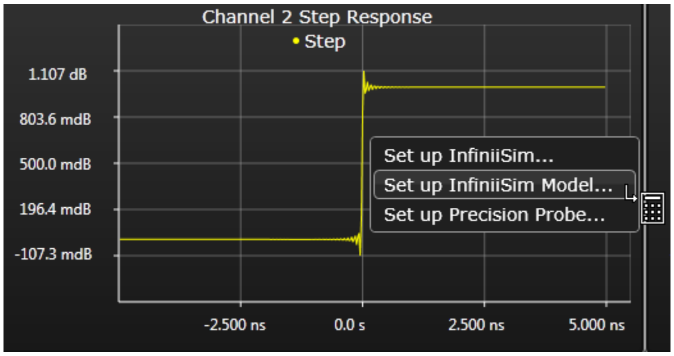
Click the **Type** drop-down menu to see plots of the frequency response of the original measured signal, the simulated signal, and the filter applied by InfiniiSim. You can also see the impulse response and step response plots of the transfer function as well.

- "Frequency Response Plot" on page 80
- "Impulse Response Plot" on page 81
- "Step Response Plot" on page 83

Use the **Graphs** control to specify how many plots you want to see in the InfiniiSim window.

These plots are useful when setting the **Bandwidth Limit** and **Filter Size** controls in the "InfiniiSim Setup Dialog Box" on page 156.

You can right-click in a plot to go directly to the InfiniiSim Setup or InfiniiSim Model Setup dialog boxes.



You can drag a rectangle within a plot to zoom. After you zoom, there will be an **Undo Zoom** selection in the plot to return to the original view.

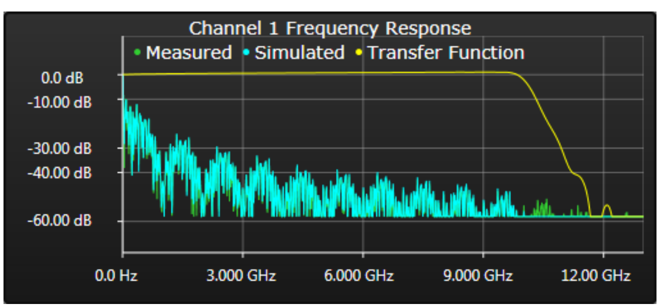
Frequency Response Plot

This plot serves two basic functions. First, it lets you see:

- the measured signal's frequency response
- the simulated signal's frequency response (the signal after InfiniiSim transforms the measured signal)
- the applied transfer function's frequency response

By investigating this plot, you can see whether they make sense for the specific setup you defined. Sometimes errors in the transformation or problems with the transfer function can be found by investigating this plot.

This plot also helps you determine what value to use for the bandwidth limit. Remember from the discussion on the bandwidth limit, you need to determine at what frequency your signal's frequency response is mostly just noise (if you are using this control). By investigating these plots, you can determine this and set the bandwidth limit accordingly.



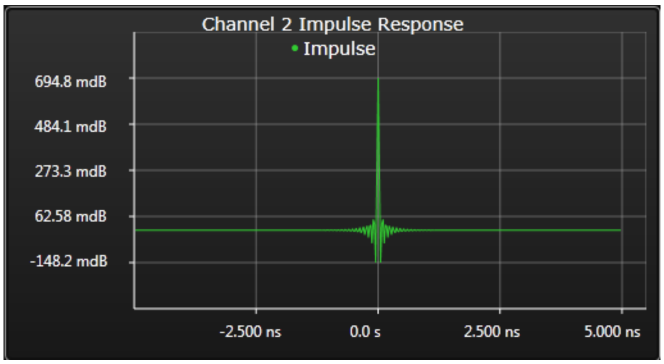
See Also

- "FIR Filter Bandwidth" on page 149

Impulse Response Plot

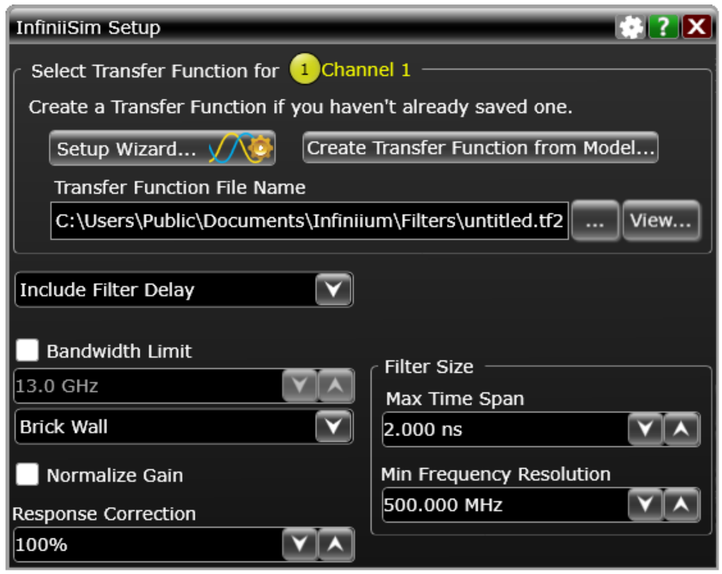
The Impulse Response plot shows a time-domain perspective of the transfer function and is useful in determining how to set the **Max Time Span** control (found in "InfiniiSim Setup Dialog Box" on page 156).

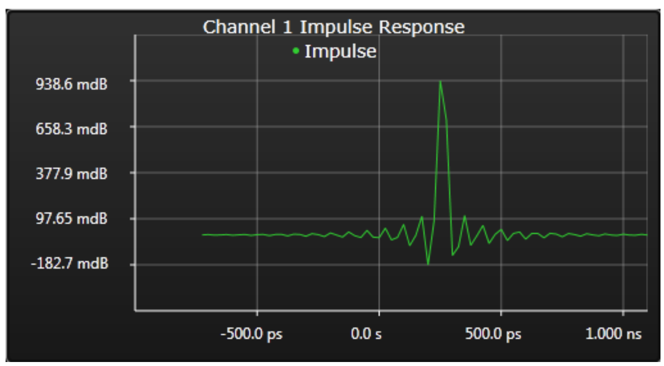
For an example of how the Impulse Response plot lets you decide what to set the **Max Time Span** control to, look at the following simplified plot.



As you can see, large sections of the impulse response are just flat, and therefore contribute insignificantly to the simulated waveform. If you see this, you may be able to increase your measurement update rate by reducing the **Max Time Span** control.

For example, the screen shot above was captured with the **Max Time Span** being set to 8 ns. You can reduce this until you get to a point where most of these flat regions are not shown. You can open the InfiniiSim Setup dialog box by right-clicking the plot area and selecting **Set up InfiniiSim...**. Then, you can move this dialog box to a portion of the display such that you can see both the dialog box and the impulse response plot at the same time. Then, simply reduce the **Max Time Span** control until the flat sections are no longer shown. For example, reducing the control to 3 ns yields the following view of the plot.





As you can see, there are still some flat regions on either side, but they have been dramatically reduced.

If you show the Impulse Response plot and you do not see any flat regions on either side, you may need to increase the **Max Time Span** control until you start to see some flat regions. If increasing the **Max Time Span** control does not change the filter's actual time span, it may be limited by one or more of your S-parameter files.

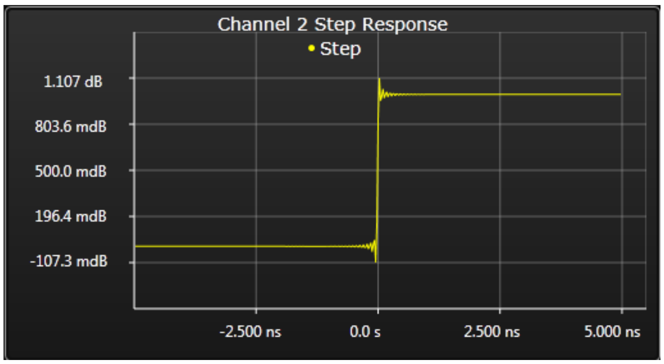
Also, note that you do not have to change both the **Max Time Span** and the **Min Frequency Resolution** controls. When you adjust the **Max Time Span** control, the **Min Frequency Resolution** field automatically adjusts (because they are reciprocals of each other).

See Also

- "Understanding FIR Filter Length" on page 151

Step Response Plot

The Step Response plot is the integration of the Impulse Response plot just discussed. It shows you basically the same information, but in a different way. This plot is also useful in setting the **Max Time Span** control. In fact, it is normally more useful in predicting how the transfer function affects your measured waveform.



The preceding screen shot shows large flat areas. You may need to reduce the **Max Time Span** control in this instance.

See Also

- "Understanding FIR Filter Length" on page 151

InfiniiSim Examples

This section contains the following examples of using InfiniiSim:

- "Removing the Insertion Loss of a Cable" on page 84
- "Probe Loading Example" on page 89
- "Simulating Crosstalk" on page 91

For these additional examples:

- Embedding a cable
- De-embedding a fixture
- Remove the effects of probe loading
- Measuring the eye of signal at the cable end or at the input to the receiver
- De-embedding a DDR interposer

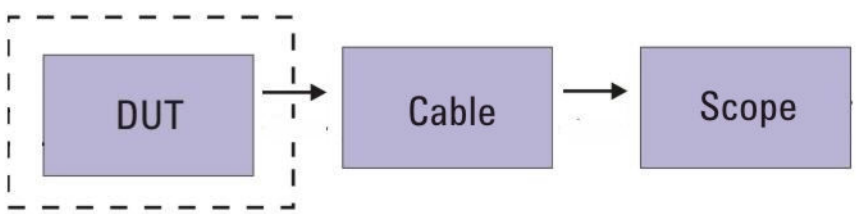
See the Practical Application of the InfiniiSim Waveform Transformation Toolset application note.

Removing the Insertion Loss of a Cable

One of the key features of the Keysight N5465A InfiniiSim Waveform Transformation Toolset is the flexibility it offers in terms of the broad range of models you can build using its GUI.

Many times your device under test (DUT) cannot be immediately accessed by your measurement instrument. Instead, signals from the DUT must pass through devices such as cables or fixtures before being measured. Therefore, you end up measuring a combination of the DUT signal and the effects added from the various other devices used to access the DUT. It would be great if you could isolate the DUT and only measure its performance. InfiniiSim can be used to do just this.

With InfiniiSim, you can extract the DUT's signal from the measured signal. For example, look at the following representation of a measurement.



The node that is indicated by the dotted line is the location you want to measure, but the cable distorts this measurement. InfiniiSim can extract the undistorted DUT signal from the distorted measured signal.

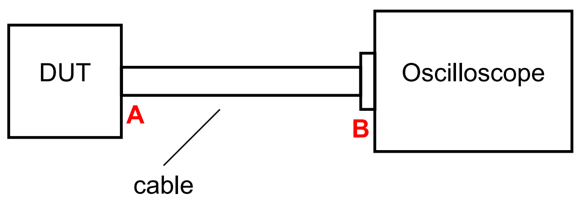
In a similar manner, you could use InfiniiSim to do the exact opposite — add the effects of a cable or fixture to a measured signal. For example, SATA requires that a 10 m (32.8 ft) cable be used in the testing. However, this is a large and cumbersome cable to use. It would be easier to simply take the measurement without the cable and then add the effects of the 10 m cable into the measurement.

The easiest way to learn how to use a flexible application such as this is to first investigate a very simple case and then progress to more advanced models as you get more of a feel for the application and its user interface.

This example walks you through a very simple model you can build using the InfiniiSim application to help you become more familiar with the basic controls.

The Problem: Removing the Insertion Loss of a Cable

For our simple example, let us assume we have the following setup.



The locations marked A and B are the nodes in our measurement. We cannot connect the oscilloscope directly to node A to measure the signal at that location. Instead, we need a cable to connect to the DUT which means we actually end up measuring what the signal looks like at node B after it has passed through the cable. We would like, however, to display the waveform at A if possible.

The Solution Using InfiniiSim

We are now going to walk through the process to accomplish this using InfiniiSim in a step-by-step fashion. Again, understanding this basic model makes it easier to understand how to use the controls when you use it on more complex problems.

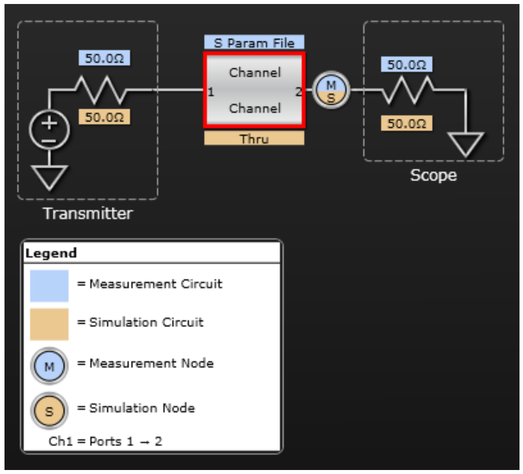
Let us assume the cable is connected to Channel 1 on your oscilloscope. Therefore, click the Channel 1 setup button located directly above the waveform window.

This opens the Channel dialog box. Under the **InfiniiSim** section, select **2 Port** (we assume the signal passing through the cable is single-ended). Then, click the **Bandwidth Limit** button and set the limit of Channel 1 to **4 GHz** because we assume our signal-to-noise ratio past 4 GHz is low (either we already know this or we determined it from the "Frequency Response Plot" on page 80). Click the **Setup...** button from the Channel dialog box to open the InfiniiSim Setup dialog box.

We now want to create our transfer function from a model (we do not have a transfer function previously saved) so click the **Create Transfer Function from Model...** button. This opens the InfiniiSim Model Setup dialog box.

For our simple example, we choose the **Remove Insertion Loss of a Fixture or Cable** as the use model from the **Application Preset** field. This is because we have only one device we are trying to negate and we do not care about multiple reflection effects.

Once you select this use model, you see the following circuit view in the dialog box.

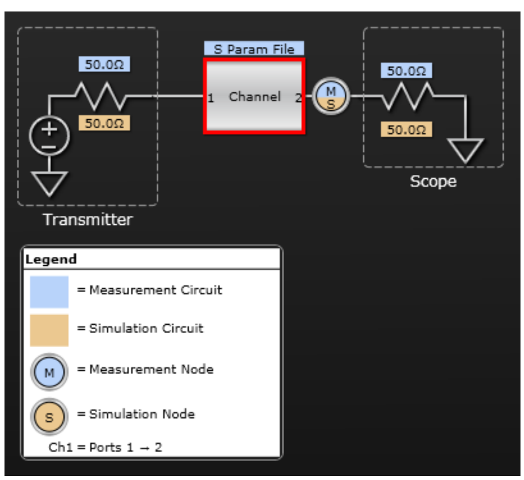


Now, we need to set up the measurement circuit and the simulation circuit.

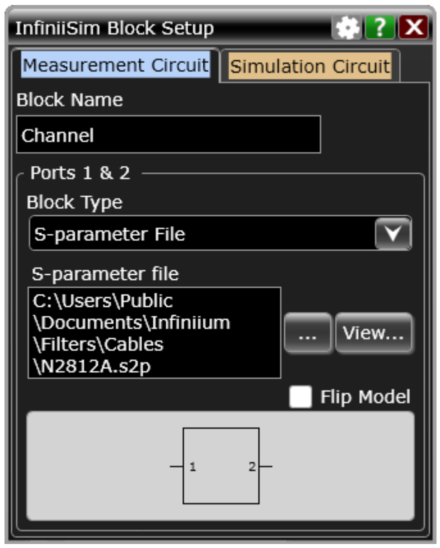
The measurement circuit represents the real physical setup that created the measured waveform. For our example, this would mean there is a cable connected between the DUT and the oscilloscope and we can measure only the signal coming from the DUT after it has passed through the cable.

Let us assume we measured the S-parameters for the cable using a network analyzer, and they have been saved to a file named "cable.s2p". Then, we know we are going to set the measurement view of the model block in the above circuit to this S-parameter file.

This makes sense if you follow along the measurement circuit—first the signal leaves the transmitter (the DUT), travels through the cable (the block has the same characteristics as the cable because we built an S-parameter file for the cable and are using it for the block), and is then measured at the front end of the oscilloscope (the measurement—or **M**—node).



To set it up like this, click the **Channel** block to open the InfiniiSim Block Setup dialog box. Then, click the **Measurement Circuit** tab if it is not already selected. Under Block Type, choose **S-parameter File** and then click the **S-parameter file** field, navigate to where you saved the file on the oscilloscope, and click **Select**. Because the S-parameters represent a cable, $S_{21} = S_{12}$, checking the **Flip Model** option should have no effect.



Because the Simulation circuit block is defaulted to **Ideal Thru** for this application preset, we do not need to change it. If you think about the simulation we want to perform, it makes sense that the block needs to be defined as a Thru (meaning that the block acts as though it is not there). Remember in the measurement circuit that we had the signal exiting the DUT, passing through the cable, and entering the oscilloscope. We could only measure it at the front end of the oscilloscope. We actually wished we could have measured the signal before it entered the cable so we could remove any loss induced by the cable.

Therefore, if we set the simulation circuit to **Ideal Thru**, we have the following situation. The signal will leave the transmitter (the DUT), not pass through any cable (since the block is defined as a Thru), and then immediately enter the oscilloscope's front end. You can now see that we have removed the cable from the circuit and are, in effect, moving the oscilloscope's front end to be connected directly to the DUT.

Now, you have the measurement and simulation circuits built appropriately for your model. In the InfiniiSim Model Setup dialog box, click the **Save Transfer Function File As** field and enter the name you want to give the transfer function file. Then, click the **Save Transfer Function** button and InfiniiSim starts generating and saving the transfer function file. Once complete, you can close all of the dialog boxes to see your simulated waveform on the oscilloscope's display. Remember that this simulated waveform is what the signal would look like with no cable present between the DUT and the oscilloscope.

Probe Loading Example

Oscilloscope probing of circuits is very common and universal. While modern probes are designed to minimize parasitic circuit loading, they generally may have a significant effect on measurement accuracy.

The following figure shows the probe loading effect diagrammatically.

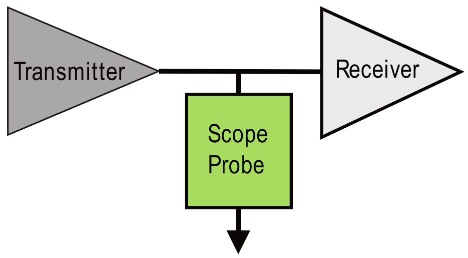


Figure 1 Oscilloscope Probe loading on a digital transmitter and receiver

The following figure shows the probe loading effect schematically, using S-parameter models.

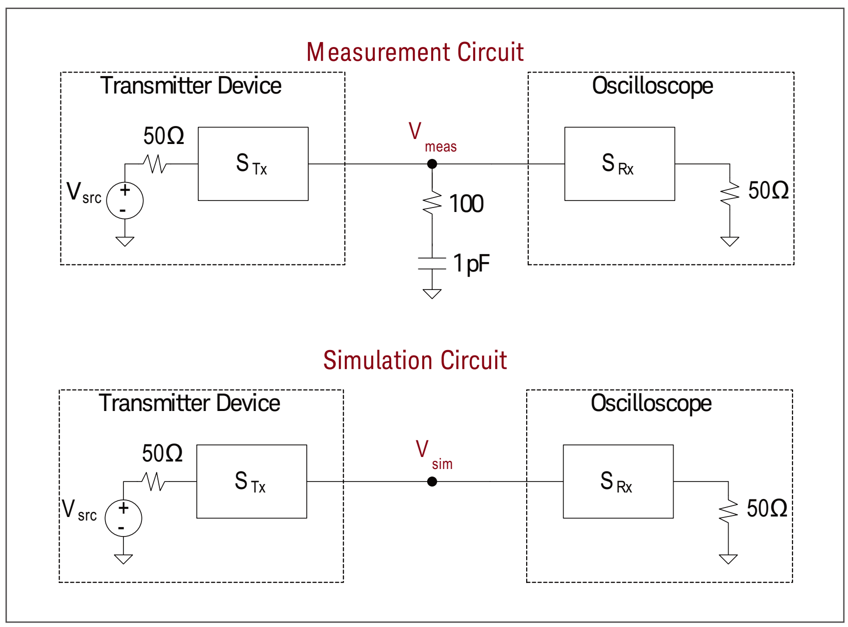


Figure 2 S-parameter modeling of probe loaded circuit and co-simulation of unloaded circuit

The probe loading model in this example is a simple series shunt RC comprised of a lead resistance, R_{Probe} = 100 ohms and capacitance, C_{Probe} = 1 pF. The insertion loss of the transmitter and receiver models will not affect the resultant corrected transfer function provided they are identical for both circuits. However, the transmitter's output return loss and the receiver's input return loss do affect the corrected transfer function. The intermediate transfer functions, $H_m(f)$ and $H_s(f)$ and the resulting corrected transfer function, H(f), are shown in the following figure.

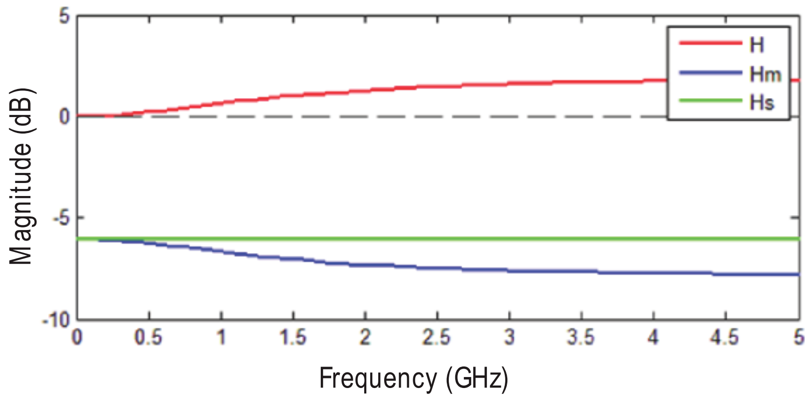


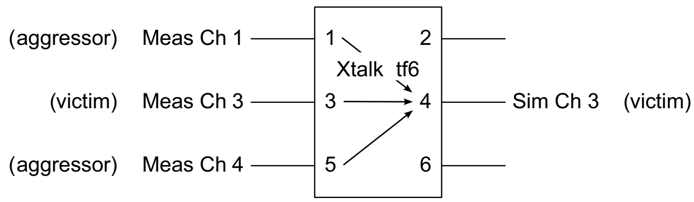
Figure 3 Transfer functions H_m, H_s, and H of probe loading example

The following figure shows the waveforms and the effects of correcting the probe loading in the example. In this case, the effect of the probe as modeled illustrates about a 10% degradation for 0-1-0 and 1-0-1 transitions. Removing the effect of the probe in this case will open the eye commensurately.

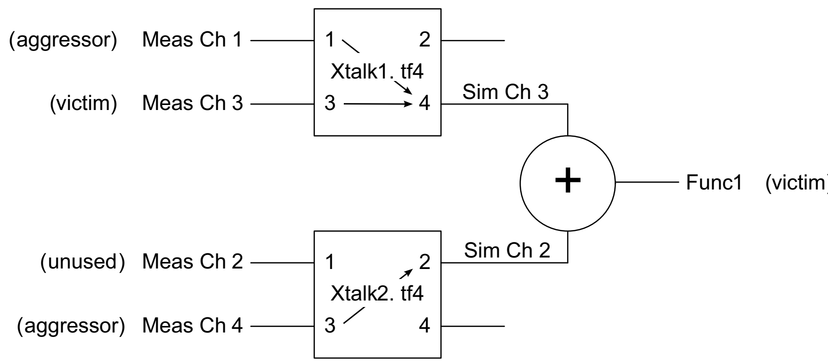
Figure 4 Waveforms V_{meas} and V_{sim} of probe loading example

Simulating Crosstalk

The simplest way to simulate crosstalk is by applying a 6-port transfer function to three measured signals: a victim and 2 aggressors as shown in the following figure.

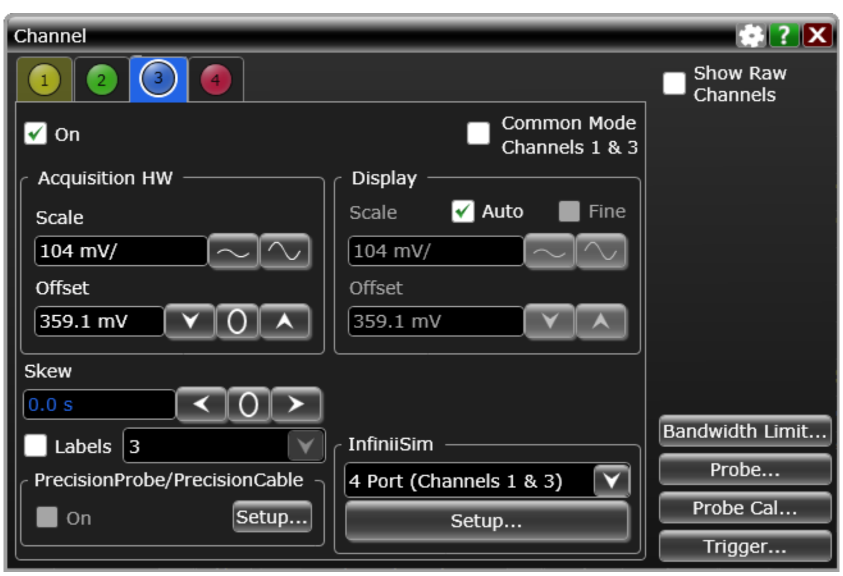


Although InfiniiSim cannot handle 6-port transfer functions, you can simulate crosstalk with two 4-port transfer functions using the setup shown here:

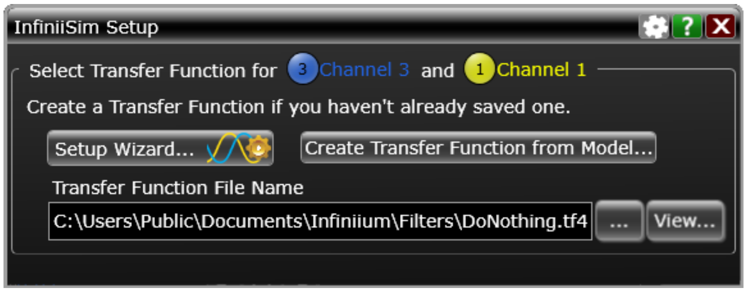


First, create two 4-port TFs (transfer functions) from two 4-port S-parameter models. Then, apply the two 4-port TFs to three measured signals in order to produce two simulated waveforms that can be added together using a waveform math function.

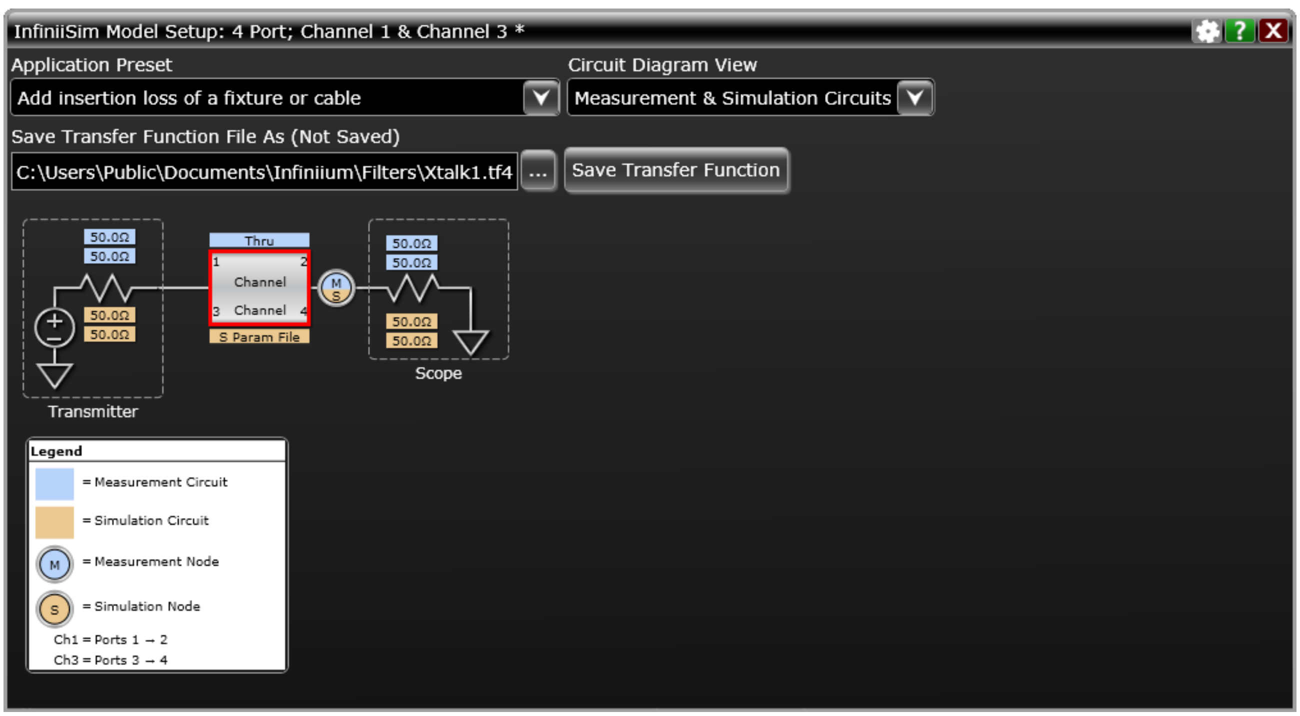
The following figures show how to configure InfiniiSim.



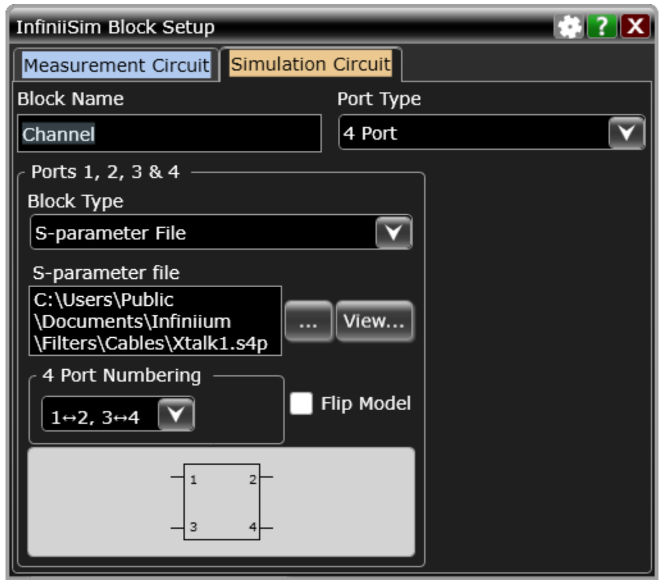
From the Channel 3 Setup dialog box, select InfiniiSim **4 Port (Channels 1 & 3)**. Then click **Setup...** to open the InfiniiSim Setup dialog box:



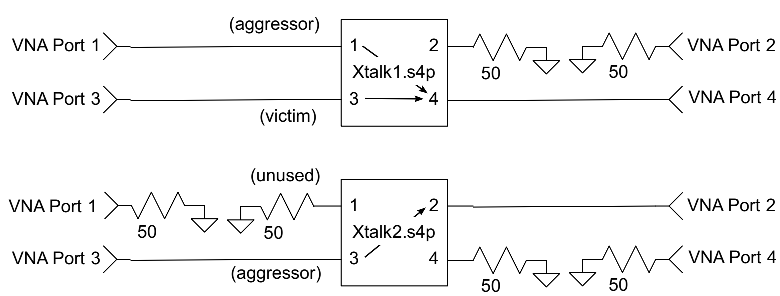
If Xtalk1.tf4 already exists, then select it as the **Transfer Function File Name**. Otherwise, you have to create it from S-parameter models. To create it from models, click **Create Transfer Function from Model...** to open the InfiniiSim Model Setup dialog box. Then choose the **Add insertion loss of a fixture or cable** application preset, and rename the **Save Transfer Function File As** to "Xtalk1.tf4".



Next, click the **Channel** model block to open its InfiniiSim Block Setup dialog box. Configure the Simulation Circuit to use your desired S-parameter file. The Measurement Circuit should remain defined to an **Ideal Thru**.

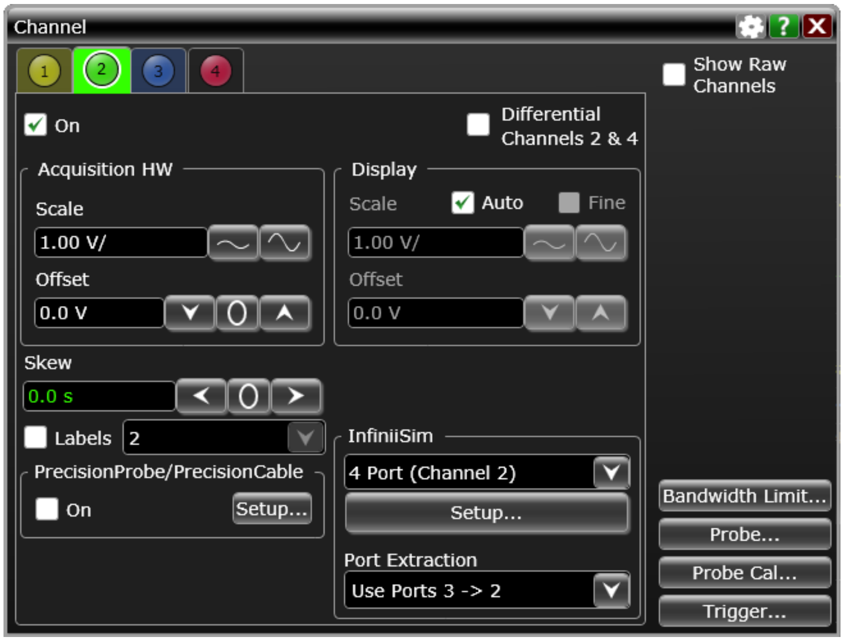


The S-parameter models you use to model crosstalk can be measured directly from your device or exported from your personal simulation tools. In either case, use the following configurations when measuring or simulating the S-parameter crosstalk models, in which the unused ports are terminated into 50 ohms.



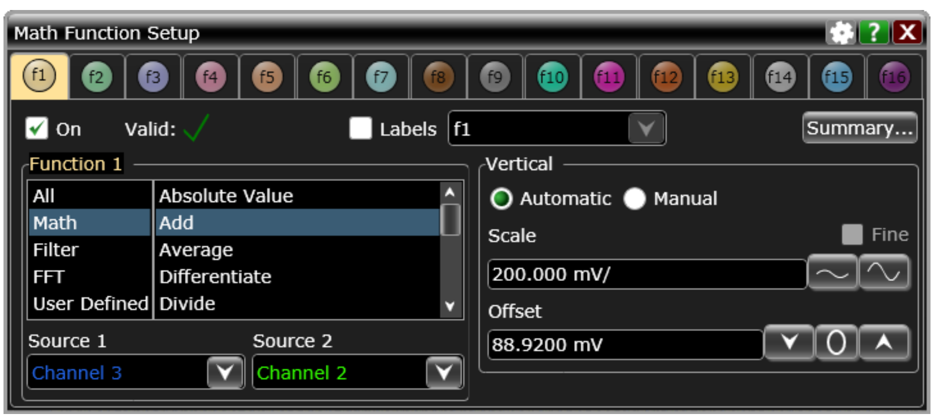
Now, close the InfiniiSim Block Setup dialog box and click **Save Transfer Function**. After it completes calculating and saving the new TF, close the InfiniiSim Model Setup dialog box and the InfiniiSim Setup dialog box.

Next, open the Channel 2 Setup dialog box and configure InfiniiSim as shown below.



Then, configure the InfiniiSim Setup dialog box and generate the Xtalk2.tf4 transfer function using a procedure similar to that explained for Xtalk1.tf4.

Finally, combine the two simulated waveforms into the resultant crosstalk waveform using the "add" math function:



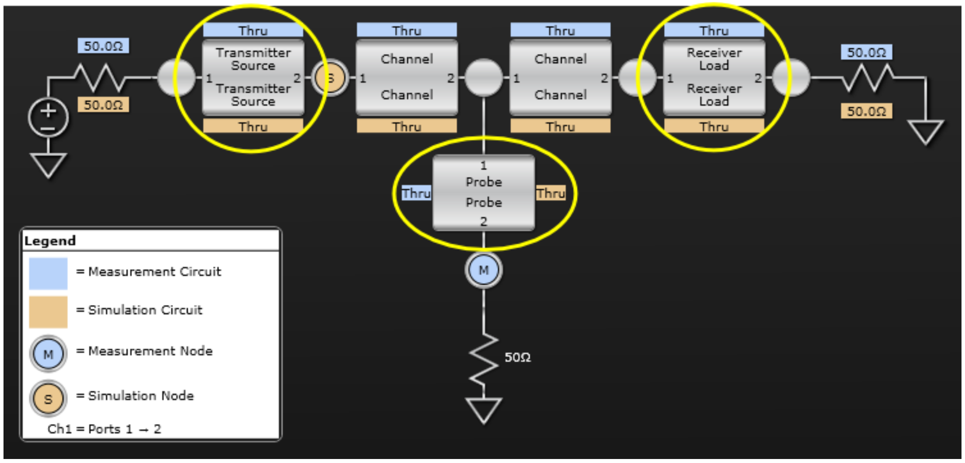
Note that although the example just described only used two aggressors, a third could be added by simply using the port 1 input of Xtalk2.tf4.

InfiniiSim Warning/Error Messages

Sometimes you can get a warning or error message when trying to create a transfer function. Refer to the following alphabetized list for the meaning of each of these messages as well as suggested corrective actions.

1-port files allowed at source, load, or probe blocks only

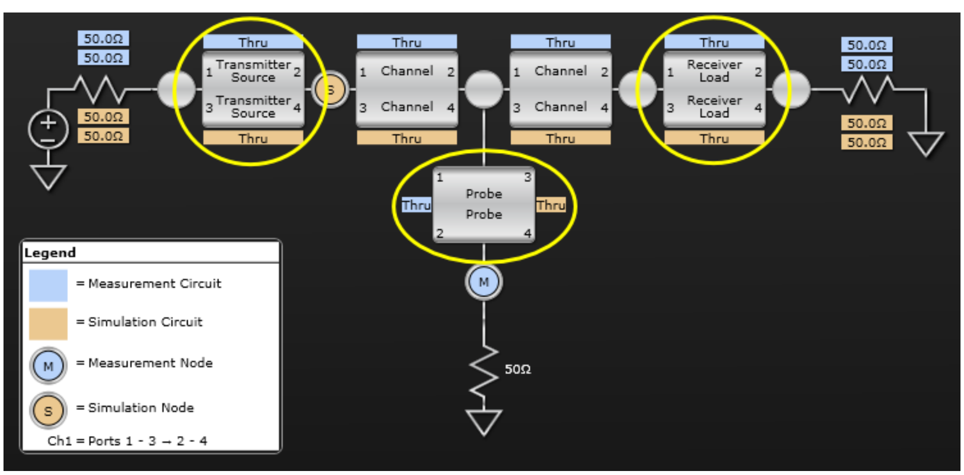
In 2-port InfiniiSim, you are allowed to use only 1-port files in the locations circled below.



In other words, you can use them only for Transmitter, Receiver, and Scope/Probe blocks. If you try to use a 1-port file anywhere else, you get this error.

2-port files allowed at source, load, or probe blocks only

In 4-port InfiniiSim, you are allowed to use only 2-port files in the locations circled below.



In other words, you can use them only for Transmitter, Receiver, and Scope/Probe blocks. If you try to use a 2-port file anywhere else, you get this error.

Calculation Aborted

This error indicates that you chose to abort the transfer function computation before it was written to a file.

DEFAULT_FREQUENCY_RESOLUTION value too small

The DEFAULT_FREQUENCY_RESOLUTION value provided in the S-parameter file is too small and yields more than the allowed 100,000 frequency points (see "S-Parameter File Keywords" on page 116).

Differential transfer function is being applied to non-differential channels

This message means a differential transfer function is being applied in a different way than when it was created. That is, when a 2-port transfer function is computed, the oscilloscope determines whether the channel is in differential/common mode or single-ended mode. The transfer function should then be used only in that mode.

Files may contain only a single data package

Each CITIfile file may model only a single device.

File Not Found

This error indicates that the file is not where you are telling InfiniiSim it is located or it has the wrong name.

Impedance of all ports must be the same

The CITIfile file format lets you specify a unique port impedance for each port. However, they must all be the same for InfiniiSim; otherwise, you get this error.

Invalid data format (DBANGLE, MAGANGLE, RI only)

You are allowed to use only DBANGLE, MAGANGLE, or RI data value formats with CITIfile files. If you use any other data format, you get this error message.

Invalid data format (DB, MA, RI only)

You are allowed to use only DB, MA, or RI data value formats with ATF and Touchstone files. If you use any other data format, you get this error message.

Invalid data value

This error occurs if any data values in an S-parameter or transfer function file are equal to infinity or not-a-number (NaN). Note that magnitude values of negative infinity in dB format are allowed and do not produce an error.

Invalid file format

InfiniiSim could not recognize the indicated S-parameter file format.

Invalid frequency units (Hz, kHz, MHz, GHz only)

You are allowed to use only Hz, kHz, MHz, and GHz frequency units in your S-parameter file. If you use anything else, you get this error.

Invalid MAXIMUM_FREQUENCY value

This error occurs if the MAXIMUM_FREQUENCY keyword value in an S-parameter or transfer function file is less than the second frequency point in the file. InfiniiSim requires at least two valid frequency points to be used from the file.

Invalid output file name

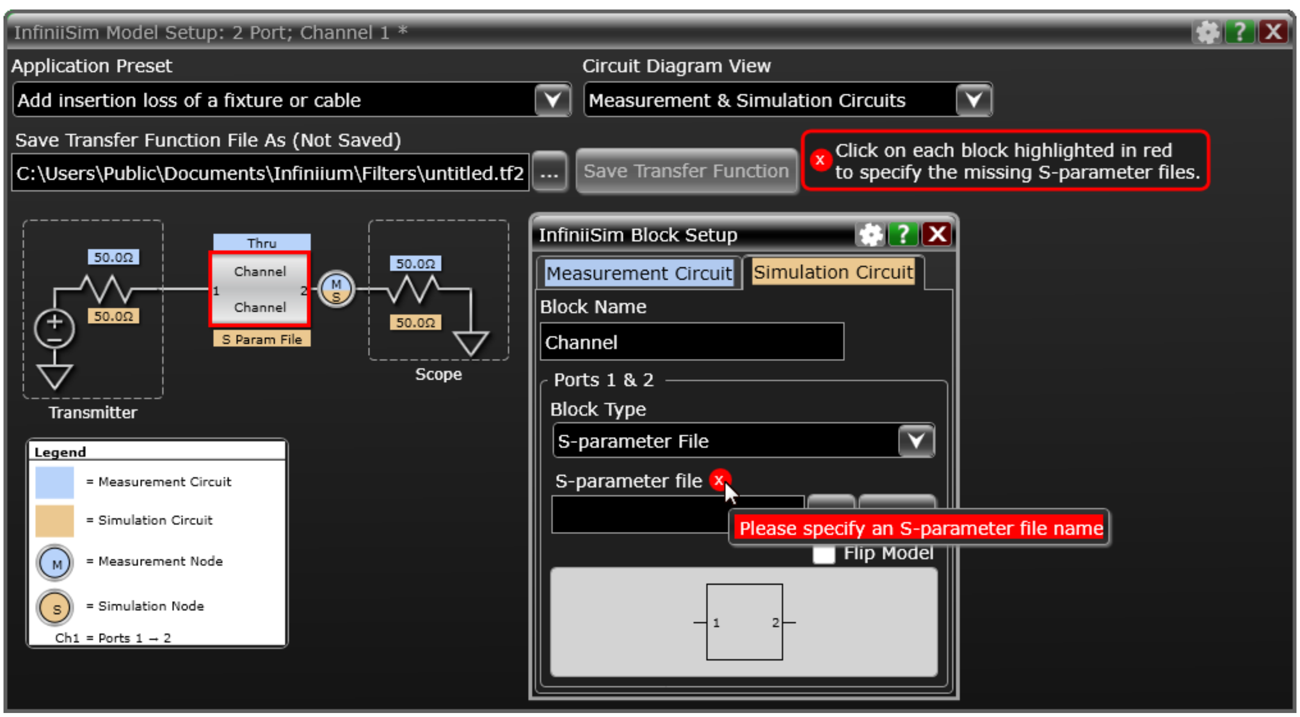
This error means you probably gave the Save As file name an improper file extension.

Invalid RETURN_LOSS_INTERPRETATION value

The RETURN_LOSS_INTERPRETATION value in the S-parameter file must be set to either SCOPE, PROBE, SERIES_RLC, SHUNT_RLC, or DEFAULT (see "S-Parameter File Keywords" on page 116).

Invalid S-Param file name: '<empty string>'

When you are saving a transfer function, this error dialog box appears if no **S-parameter File** name is specified in the InfiniiSim Block Setup dialog box (which appears after clicking the measurement/simulation block in the InfiniiSim Model Setup dialog box).



Less than 2 data points in file

You must have at two least data points in your S-parameter and transfer function files.

Non-differential transfer function is being applied to differential channels

This message means a non-differential transfer function is being applied in a different way than when it was created. That is, when a 2-port transfer function is computed, the oscilloscope determines whether the channel is in differential/common mode or single-ended mode. The transfer function should then be used only in that mode.

Signal amplitude too small at measurement node

This message usually means you made a mistake when setting up your measurement circuit (for example, placing an open circuit block somewhere between the source and measurement nodes). This may also occur if an S-parameter file in the measurement circuit has a resonant 'suck-out' that the transfer function is trying to compensate for.

What this warning means is that the voltage seen at the measurement node is very small. It could be that your specific setup does indeed cause this to occur and there was no error made, but more often than not, you should go back and check your measurement circuit setup.

Source signals insufficiently unique at measurement node

4-port transfer functions must determine how each of the two independent input ports affect the measurement and simulation nodes. This error occurs if the two input ports do not affect measurement and simulation nodes independently, such as if they are shorted together. In this case, InfiniiSim would not return a valid transfer function.

Transfer function data must contain a 0 Hz point

This is not true for S-parameter files, but is true for transfer function files.

Unspecified error

An internal error was detected by InfiniiSim. Please contact Keysight Technical Support.

Unspecified file read error

This indicates that the file has an invalid format.

Unspecified file write error

An internal error was detected in InfiniiSim. Please contact Keysight Technical Support.

Unstable or ill-conditioned circuit definition

This message indicates one of your circuits is invalid. An infinite voltage is present in the circuit (analogous to an oscillation); in other words, the gain becomes infinite.

Warning: File data truncated to 250 kpts

This warning means there are too many points in your S-parameter file. You need to reduce the number of points.

Warning: Frequency resolution may be too large for slow time constants

This is similar to the next warning except that instead of a model's long time delay causing the problem, it is the model's slow time constants. These kinds of models require finer frequency resolution. Therefore, you need to have a finer sampling of frequencies in your S-parameter file.

Warning: Frequency resolution may be too large for time delay

S-parameter models that have long time delays require finer frequency resolution because the model's impulse response needs to be longer. This warning indicates that your S-parameter file may not have enough resolution to accurately represent the modeled device. Therefore, you need to have a finer sampling of frequencies in your S-parameter file.

If you see this warning and your S-parameter file does not contain a DC frequency, InfiniiSim may not extrapolate the DC values correctly. InfiniiSim looks at the lowest frequency points in the file to determine if each DC value should be a positive or negative. If the frequency resolution in the file is so large compared to the time delay of the S-parameter model that it causes the phase linear component (delay component) of the phase variation to change more than 90 degrees per frequency step, then the DC value will have the wrong polarity. When the DC value of the insertion loss is incorrectly determined to be negative, but it should actually be positive, you will likely also see the "Transfer function appears to invert signal" warning message.

Warning: Insufficient uniqueness at measurement node prevented correction above <value> Hz

4-port transfer functions must determine how each of the two independent input ports affect the measurement and simulation nodes. An error occurs if the two input ports do not affect measurement and simulation nodes independently, such as if they are shorted together. If the error occurs only above a certain frequency, the InfiniiSim returns a band-limited transfer function and displays this warning. This warning condition can occur when the short is caused by a capacitance, for example.

Warning: Interpolated S-parameters limited to 100 kpts

This warning indicates that the interpolation performed by InfiniiSim on your file data results in too many points. InfiniiSim sets the resolution by looking at the separation between the last two points in the file. If these values are too close together then it can cause the resolution to be too fine.

There is also a command you can place within the S-parameter file that forces InfiniiSim to have a different resolution than that specified by the frequency spacing in the file (see "S-Parameter Considerations" on page 113). If you used this command and set it to too fine of a resolution, this could result in this warning message.

Warning: Non-uniformly sampled data

If you give InfiniiSim S-parameter files with uniformly spaced points, it uses the data as is (if there is only one file). If your points are not uniformly spaced (for example, log sweep), InfiniiSim will try to interpret the points, but you may get a bad result (especially if you are using a pure log sweep—not recommended). If you have nonuniform spacing between the points in your S-parameter file, InfiniiSim sets the resolution by looking at the separation between the last two points in the file. See "S-Parameter Considerations" on page 113.

Warning: Port order appears to be incorrect

InfiniiSim supports the most common 4-port S-parameter order configurations:

- 1→2, 3→4
- 1→3, 2→4
- 2→1, 4→3
- 3→1, 4→2

If your configuration is non-standard, such as 1→3, 4→2, this warning appears.

Warning: Resampled S-param file data truncated to max 25 kpts

InfiniiSim will read S-parameter files up to 250 kpts, but it needs to truncate and/or re-sample them down to a size less than or equal to 25 kpts. You can include the DEFAULT_FREQUENCY_RESOLUTION keyword in the S-parameter file header to tell InfiniiSim how to truncate and/or re-sample the data. See "DEFAULT_FREQUENCY_RESOLUTION" on page 116.

Warning: Resampled transfer function file data truncated to max 100 kpts

InfiniiSim will read transfer function files up to 250 kpts, but it needs to truncate and/or re-sample them down to a size less than or equal to 100 kpts. You can include the DEFAULT_FREQUENCY_RESOLUTION keyword in the transfer function file header to tell InfiniiSim how to truncate and/or re-sample the data. See "DEFAULT_FREQUENCY_RESOLUTION" on page 148.

Warning: Return loss interpretation defaulted. See Help.

When InfiniiSim is provided a 1-port file to represent the return loss-only portion of a 2-port terminating model block, it needs to convert it to a full 2-port model. The formula used to perform this conversion depends on what type of terminating circuit it is. The RETURN_LOSS_INTERPRETATION keyword can be used to tell InfiniiSim the type of terminating circuit. When no keyword is present, the termination type is defaulted. The termination type is also defaulted if the keyword is unrecognized. This warning also applies to 2-port files used to model the return loss of 4 ports.

Warning: Small amplitude at measurement node prevented correction above.

Much loss occurs at higher frequencies, and the S-parameter shows the loss.

Warning: Transfer function appears to invert signal

This warning typically happens with 4-port InfiniiSim. There is one particular non-standard port ordering that can cause the 4-port model to invert the signal polarity. Unfortunately, this uncommon port ordering cannot be corrected with any combination of the **Flip Model** and **4 Port Numbering** controls.

Again, it could be that your specific setup does indeed cause signal polarity inversion to occur and there was no error made (for example, if you are modeling an inverting amplifier), but more often than not there is a problem with the S-parameter file port ordering.

If you see the "Frequency resolution may be too large for time delay" message and the S-parameter file does not contain a DC frequency, InfiniiSim may not extrapolate the DC values correctly. InfiniiSim looks at the lowest frequency points in the file to determine if each DC value should be a positive or negative. If the frequency resolution in the file is so large compared to the time delay of the S-parameter model that it causes the phase linear component (delay component) of the phase variation to change more than 90 degrees per frequency step, then the DC value will have the wrong polarity. When the DC value of the insertion loss is incorrectly determined to be negative, but it should actually be positive, you will likely also see this "Transfer function appears to invert signal" warning message.

Warning: Transfer function gain is unusually large

This warning indicates that the voltage seen at the simulation node is huge when compared with the voltage seen at the measurement node (when driven by the same source). This usually means you made a mistake when setting up your measurement circuit (for example, placing an open circuit block somewhere that is preventing the source from driving the measurement node) so you should first check your measurement circuit to ensure there are no errors. This may also occur if an S-parameter file in the measurement circuit has a resonant 'suck-out' that the transfer function is trying to compensate.

It could be that your specific setup does indeed cause a large gain in the transfer function. Examination of the frequency response plot can help verify if it is correct (see "Frequency Response Plot" on page 80).

Warning: Unrecognized transfer function version

This warning indicates that InfiniiSim does not support the ATF file you are trying to use. InfiniiSim supports only A.01.00 ATF files.

Warning: Unrecognized CITIfile version (A.01.00 and A.01.01 only)

This warning indicates that InfiniiSim does not support the CITIfile you are trying to use. InfiniiSim supports only A.01.00 and A.01.01 CITIfiles.

InfiniiSim Math Functions

The InfiniiSim math functions let you apply transfer functions to waveform sources similarly to the way the InfiniiSim Waveform Transformation Toolset application applies transfer functions to analog input channels.

You can create transfer functions using the InfiniiSim Waveform Transformation Toolset application.

To create InfiniiSim math functions:

1. Choose **Setup > Math Functions...** or **Math > Math Functions...**.
2. In the Function dialog box, select the tab of function you want to define, **F1-F16**.
3. In the left side of the Function box, select **All**.
4. In the right side of the Function box, select the desired operator:
   - "InfiniiSim 2 Port" on page 104
   - "InfiniiSim 4 Port 1 Src" on page 105
   - "InfiniiSim 4 Port CM" on page 106
   - "InfiniiSim 4 Port Diff" on page 106
   - "InfiniiSim 4 Port Src1" on page 107
   - "InfiniiSim 4 Port Src2" on page 107
5. Then, select the waveform source(s), transfer function file, and any other function parameters.

   These parameters are similar to the selections you can make when setting up InfiniiSim in the Channel dialog box.
6. Check **On** to enable the math function waveform.

See Also

- "InfiniiSim Setup Options and Uses" on page 18

InfiniiSim 2 Port

The InfiniiSim 2 Port math function applies a 2-port transfer function to the selected source waveform.

This is for transfer functions used when selecting **2 Port** InfiniiSim in the Channel dialog box.

When this math function is selected, you can also set these parameters:

- **Source 1**:

If the transfer function was generated for the InfiniiSim option:	**Then select this type of waveform:**
2 Port	single-ended
2 Port, with the **Differential Channels N & N** check box selected	differential
2 Port, with the **Common Mode Channels N & N** check box selected	common mode

- **Transfer Function File** — Select the 2-port transfer function file (*.tf2).
- **Max Time Span** — Lets you increase or reduce the time span of the correction transfer function's impulse response (filter size). See the discussion of **Max Time Span** in "InfiniiSim Setup Dialog Box" on page 156.
- **Signal Bandwidth** — Lets you apply a bandwidth limit to the filter.
- **Remove Delay** — Lets you either apply or remove the delay imposed by the model to the waveform. Because different frequencies can have different delays, this is an approximation of the group delay. Select **Off** to accurately shift the waveform in time according to the model.

InfiniiSim 4 Port 1 Src

The InfiniiSim 4 Port 1 Src math function applies a 4-port transfer function to the selected source waveform.

This is for transfer functions used when selecting **4 Port (Channel N)** InfiniiSim in the Channel dialog box.

When this math function is selected, you can also set these parameters:

- **Source 1**:

If the transfer function was generated for the InfiniiSim option:	**Then select this type of waveform:**
4 Port (Channel N)	single-ended
4 Port (Channel 1 - 3), with the **Differential Channels N & N** check box selected	differential
4 Port (Channel 1 + 3), with the **Common Mode Channels N & N** check box selected	common mode

- **Transfer Function File** — Select the 4-port transfer function file (*.tf4).
- **Signal Bandwidth** — Lets you apply a bandwidth limit to the filter.
- **Port Extraction** — Specifies how, in the transfer function, a 2-port model is extracted from the 4-port model:
  - **Ports 1 -> 2** — the port 1 to port 2 single-ended path is used.

- **Ports 3 -> 2** — the port 3 to port 2 single-ended path is used.
- **Ports 3 -> 4** — the port 3 to port 4 single-ended path is used.
- **Ports 1 -> 4** — the port 1 to port 4 single-ended path is used.
- **Differential** — the differential path through the 4-port device is used.
- **Common Mode** — the common-mode path through the 4-port device is used.

- **Max Time Span** — Lets you increase or reduce the time span of the correction transfer function's impulse response (filter size). See the discussion of **Max Time Span** in "InfiniiSim Setup Dialog Box" on page 156.
- **Remove Delay** — Lets you either apply or remove the delay imposed by the model to the waveform. Because different frequencies can have different delays, this is an approximation of the group delay. Select **Off** to accurately shift the waveform in time according to the model.

InfiniiSim 4 Port CM

The InfiniiSim 4 Port 1 CM (common-mode) math function applies a 4-port transfer function to the selected source waveforms.

This is for transfer functions used when selecting the **Common Mode Channels N & N** check box in the Channel dialog box and selecting **4 Port (Channel N & N)** InfiniiSim.

When this math function is selected, you can also set these parameters:

- **Source 1** — This is the differential waveform.
- **Source 2** — This is the common mode waveform.
- **Transfer Function File** — Select the 4-port transfer function file (*.tf4).
- **Signal Bandwidth** — Lets you apply a bandwidth limit to the filter.
- **Max Time Span** — Lets you increase or reduce the time span of the correction transfer function's impulse response (filter size). See the discussion of **Max Time Span** in "InfiniiSim Setup Dialog Box" on page 156.
- **Remove Delay** — Lets you either apply or remove the delay imposed by the model to the waveform. Because different frequencies can have different delays, this is an approximation of the group delay. Select **Off** to accurately shift the waveform in time according to the model.

InfiniiSim 4 Port Diff

The InfiniiSim 4 Port 1 Diff (differential) math function applies a 4-port transfer function to the selected source waveforms.

This is for transfer functions used when selecting the **Differential Channels N & N** check box in the Channel dialog box and selecting **4 Port (Channel N & N)** InfiniiSim.

When this math function is selected, you can also set these parameters:

- **Source 1** — This is the differential waveform.

- **Source 2** — This is the common mode waveform.
- **Transfer Function File** — Select the 4-port transfer function file (*.tf4).
- **Signal Bandwidth** — Lets you apply a bandwidth limit to the filter.
- **Max Time Span** — Lets you increase or reduce the time span of the correction transfer function's impulse response (filter size). See the discussion of **Max Time Span** in "InfiniiSim Setup Dialog Box" on page 156.
- **Remove Delay** — Lets you either apply or remove the delay imposed by the model to the waveform. Because different frequencies can have different delays, this is an approximation of the group delay. Select **Off** to accurately shift the waveform in time according to the model.

InfiniiSim 4 Port Src1

The InfiniiSim 4 Port 1 Src1 math function applies a 4-port transfer function to the selected source waveforms.

This is for transfer functions used on the first channel when selecting **4 Port (Channel N & N)** InfiniiSim in the Channel dialog box.

When this math function is selected, you can also set these parameters:

- **Source 1** — This is the first single-ended waveform.
- **Source 2** — This is the second single-ended waveform.
- **Transfer Function File** — Select the 4-port transfer function file (*.tf4).
- **Signal Bandwidth** — Lets you apply a bandwidth limit to the filter.
- **Max Time Span** — Lets you increase or reduce the time span of the correction transfer function's impulse response (filter size). See the discussion of **Max Time Span** in "InfiniiSim Setup Dialog Box" on page 156.
- **Remove Delay** — Lets you either apply or remove the delay imposed by the model to the waveform. Because different frequencies can have different delays, this is an approximation of the group delay. Select **Off** to accurately shift the waveform in time according to the model.

InfiniiSim 4 Port Src2

The InfiniiSim 4 Port 1 Src2 math function applies a 4-port transfer function to the selected source waveforms.

This is for transfer functions used on the second channel when selecting **4 Port (Channel N & N)** InfiniiSim in the Channel dialog box.

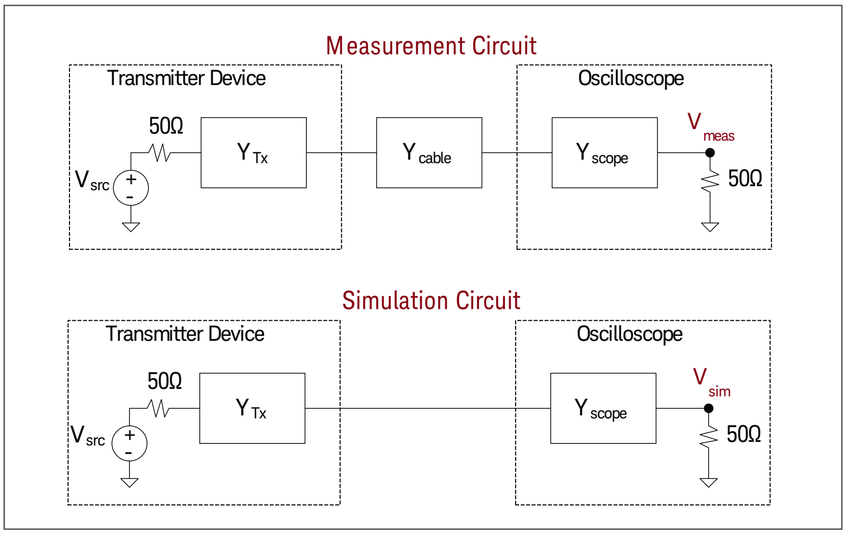
When this math function is selected, you can also set these parameters:

- **Source 1** — This is the first single-ended waveform.
- **Source 2** — This is the second single-ended waveform.
- **Transfer Function File** — Select the 4-port transfer function file (*.tf4).

- **Signal Bandwidth** — Lets you apply a bandwidth limit to the filter.
- **Max Time Span** — Lets you increase or reduce the time span of the correction transfer function's impulse response (filter size). See the discussion of **Max Time Span** in "InfiniiSim Setup Dialog Box" on page 156.
- **Remove Delay** — Lets you either apply or remove the delay imposed by the model to the waveform. Because different frequencies can have different delays, this is an approximation of the group delay. Select **Off** to accurately shift the waveform in time according to the model.

InfiniiSim Circuit Modeling

The calculation of transfer functions requires complete and accurate descriptions of both the measurement circuit and the simulation circuit. The following circuits are good examples.



Note that these circuits are single-ended 2-port models, and although they are normally drawn with only two connections, they all have an implied connection to ground.

S-parameters are commonly used to describe measurement and simulation circuit models because S-parameters are easy to measure directly.

Observation Nodes

Calculation of transfer functions also requires properly locating the measurement and simulation observation voltage nodes of the two circuits. The previous example used the same observation node for both circuits, and removed the cable. Alternatively, the cable could have been left in place while the simulation observation node was relocated.

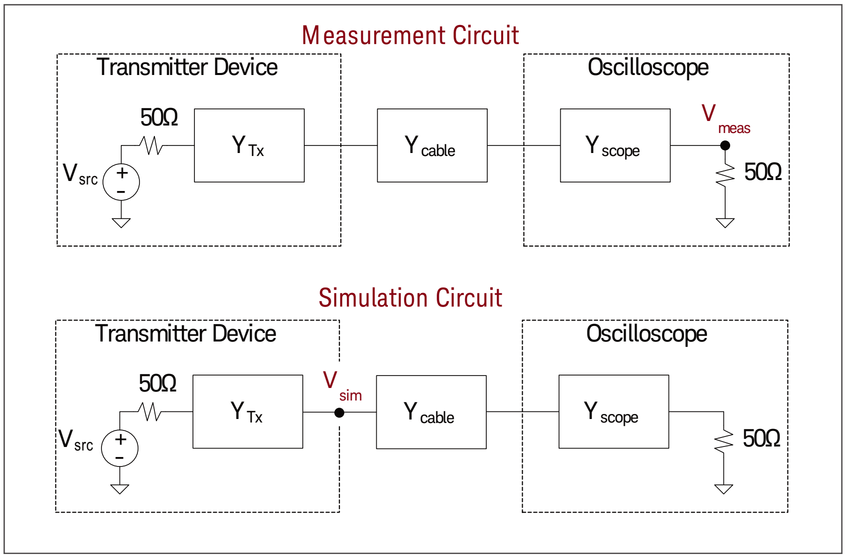


Figure 5 Relocating the Simulation observation node

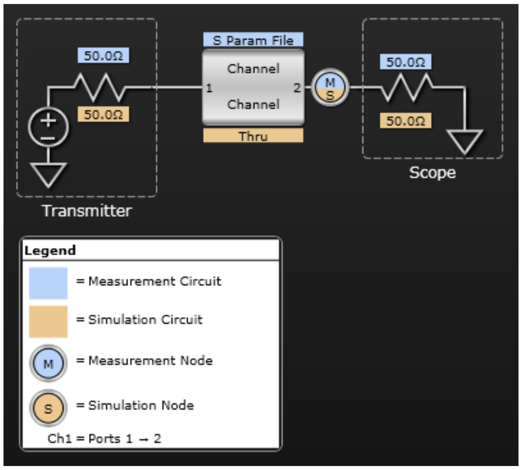
NOTE

These examples locate the oscilloscope's measurement observation node behind the oscilloscope's model block instead of in front of it. This is because high bandwidth oscilloscopes with 50Ω inputs are calibrated to measure the incident voltage and not the node voltage at the input. If converted to an S-parameter model, the calibrated oscilloscope input model, V_{scope} would have a non-ideal return loss and an ideal insertion loss. This not true, however, for a calibrated probe. Probes are calibrated to measure the node voltage at the probe tip. Therefore, it does not matter whether the measurement observation node is located before or after the probe input model block.

- "Measurement and Simulation Circuit Models in the InfiniiSim GUI" on page 110
- "Circuit Models Converted to Y-Parameters Internally" on page 111

Measurement and Simulation Circuit Models in the InfiniiSim GUI

InfiniiSim generates transfer functions by comparing measurement and simulation circuit models. You can build measurement and simulation circuit models using the InfiniiSim GUI shown below.



This is one of the simplest 2-port models you can create. The GUI also lets you develop much more complex models using other application presets.

The values highlighted in light blue define the measurement circuit while the values in light orange define the simulation circuit. In this example, the single-block model is used to represent the connection between the DUT and the oscilloscope, and can be defined in a variety of ways (S-parameters, RLC circuits, open, thru, lossless transmission line, or a combination of sub-circuits). When using more complex application presets, you can move the measurement and simulation nodes to a variety of locations within the model.

InfiniiSim analyzes the measurement and simulation circuits and generates a correction transfer function. Once a transfer function is generated, InfiniiSim performs an inverse Fourier transform on it to yield an impulse response. The impulse response is sampled at the sampling rate of the oscilloscope to yield an FIR correction filter that is convolved with acquired waveform to render the simulated waveform.

The InfiniiSim convolution process requires that the measurement and simulation circuits be linear and time-invariant (these are small-signal analysis requirements).

See Also

- "Application Presets (2 Port)" on page 26
- "Application Presets (4 Port)" on page 52

Circuit Models Converted to Y-Parameters Internally

To aid in deriving a transfer function, each circuit element is internally converted to a set of Y-parameters using a common frequency set to be used in the overall analysis. The frequency set is usually driven by the S-parameter frequencies used in the files that define the circuits.

Using Y-parameters enables the simplicity of nodal analysis of each circuit (measurement and simulation) at each frequency.

A Y-parameter model is one form of N-port network model that describes port currents as a function of port voltages. Y-parameter models are convenient for performing nodal circuit analysis. However, any N-port network model type could be used to describe these circuits because all N-port model types (Z, Y, S, H, and so on) can be converted into any other N-port model type.

When nodal analysis is performed on each circuit (measurement and simulation) and related to one another, the result is the frequency domain transfer function.

S-Parameter Considerations

S-parameter files describe electrical n-port network models of physical devices as a tabular list of complex S-parameter values at specific frequencies.

S-parameters can be measured, directly, with vector Network Analyzers (VNAs) and Time Domain Reflectometers (TDRs). They can also be created from simulation tools such as the Keysight Advanced Design System (ADS).

- "InfiniiSim S-Parameter File Requirements" on page 113
- "Measuring S-Parameters" on page 117
- "Common Issues in Measuring S-Parameters" on page 119
- "Other Issues for Consideration" on page 124
- "S-Parameter and Transfer Function File Viewers" on page 127
- "Port Ordering in S-Parameter Models" on page 128
- "S-Parameters for Single-Ended 2-Port: Flow Graph and Matrix" on page 130
- "S-Parameters for Single-Ended 3-Port: Flow Graph and Matrix" on page 130
- "S-Parameters for Single-Ended 4-Port: Flow Graph, Matrix, and Extractions" on page 131
- "Mixed-Mode S-Parameters for Differential 2-Port: Flow Graph, Matrix, and Extractions" on page 132

For more information on measuring or creating S-parameters, see these application notes:

- S-Parameter Measurements Basics for High Speed Digital Engineers (using a VNA)
- Using the Time-Domain Reflectometer (to measure S-parameters)
- Using De-embedding Tools for Virtual Probing (using simulation tools)

InfiniiSim S-Parameter File Requirements

When measuring S-parameters or creating them from simulation tools, keep in mind the requirements InfiniiSim has for S-parameter information.

- "S-Parameter File Formats" on page 114
- "Model Information at DC (0 Hz)" on page 114
- "Uniformly Spaced Values and Good Frequency Resolution" on page 114
- "Values with Increasing Frequencies, Limited Number of Frequency Points" on page 115
- "S-Parameters with Return Loss Only" on page 115
- "S-Parameter File Keywords" on page 116

NOTE An S-parameter file must not include any DC blocking capacitors.

See Also
- "Measuring S-Parameters" on page 117
- "Common Issues in Measuring S-Parameters" on page 119
- "Other Issues for Consideration" on page 124

S-Parameter File Formats

InfiniiSim supports two standard file formats used to store S-parameters:

- Touchstone (version 1.1)

  Touchstone files are identified by the file extensions .s1p, .s2p, .s3p, .s4p, or .s6p (where the number indicates the number of ports in the network model).
- CITIfile (version A.01.00 and A.01.01)

  CITIfile files are identified by either .cti or .cit.

  In addition, InfiniiSim supports CITIfiles that contain only a single data package (each file may model only a single device).

NOTE InfiniiSim assumes all 4-port S-parameter files are stored in standard (non-mixed-mode) form.

Formatted as ASCII text, S-parameter files are easily viewed or edited using any text-based file editor.

See Also
- "S-Parameter File Keywords" on page 116

Model Information at DC (0 Hz)

InfiniiSim requires model information at DC (0 Hz). However, S-parameter files often do not contain DC values, especially if they originate from network analyzer measurements. When files do not contain DC values, InfiniiSim will, by default, copy the file's lowest frequency values to 0 Hz and set their phase components to zero. This technique may not always yield appropriate DC S-parameter values. Ideally, you should measure or simulate the true DC values and manually add them to the S-parameter file with a text editor.

Uniformly Spaced Values and Good Frequency Resolution

InfiniiSim also requires S-parameter values that are uniformly spaced in frequency with a spacing or resolution that is small enough to accurately represent the slowest time constants and time delays exhibited by the modeled device.

InfiniiSim checks the data to ensure that it is uniformly sampled in frequency (except for the interval between DC and the next lowest frequency point) and generates a warning if the data is not uniformly sampled.

If an S-parameter file does not contain uniformly spaced frequency data, you can insert the special InfiniiSim keyword DEFAULT_FREQUENCY_RESOLUTION into the S-parameter file to cause the data to be resampled. See "DEFAULT_FREQUENCY_RESOLUTION" on page 116.

Values with Increasing Frequencies, Limited Number of Frequency Points

InfiniiSim expects S-parameter files to list their frequency point values in monotonically increasing frequency order and ignores those values at which frequencies are less than or equal to the frequencies of previously listed values.

The size of S-parameter files depends directly on the frequency resolution and maximum frequency of the data. Although the supported S-parameter file formats do not limit the size of files, hardware I/O can significantly slow the transfer of large file function generation in de-embedding applications, especially for 4-port models.

InfiniiSim reads up to the first 250 k frequency points of an S-parameter file, but it needs to truncate and/or re-sample data down to 25 kpts or less for the simulation engine. InfiniiSim can simply use the first 25 kpts of the S-parameter file, or, if you add the DEFAULT_FREQUENCY_RESOLUTION keyword to the S-parameter file header, InfiniiSim will read the first 250 kpts and down-sample the data before it truncates it. See "DEFAULT_FREQUENCY_RESOLUTION" on page 116.

InfiniiSim does not re-sample unless the frequency points in the file are not uniformly spaced or unless a different frequency resolution is specified using the DEFAULT_FREQUENCY_RESOLUTION keyword.

It is always best to use S-parameter files whose frequency points are uniformly sampled, contain a DC point, and have less than or equal to 25 kpts. In this case, InfiniiSim uses the file data exactly as provided and does not have to truncate or re-sample.

S-Parameters with Return Loss Only

InfiniiSim calculates transfer functions using only full 2-port (in 2-port analysis mode) or 4-port (in 4-port analysis mode) S-parameter models that contain both return loss and insertion loss information.

By default, InfiniiSim assumes that the return loss data models a simple passive shunt RLC circuit, and it calculates the insertion loss of the same circuit that would produce the file data.

However, blocks that model source impedance or load impedance (transmitter source, oscilloscope/probe, or oscilloscope input) can be defined by S-parameter files that contain only return loss information. In this case, you can insert the special InfiniiSim keyword RETURN_LOSS_INTERPRETATION into the S-parameter file to change how InfiniiSim interprets the return loss data. See "RETURN_LOSS_INTERPRETATION" on page 117.

Selecting a 1-port S-parameter (s1p) file for a 4-port application applies those S-parameters to S_{11} and S_{33}, and sets S_{13} and S_{31} to zero (see "S-Parameters for Single-Ended 4-Port: Flow Graph, Matrix, and Extractions" on page 131).

S-Parameter File Keywords

Some de-embedding applications let you insert special keywords into the comment sections of S-parameter files. These keywords are read by the application and let you control how the application interprets the file data. This is especially useful if the S-parameter measurements of one device will be used for multiple purposes. Also, this keyword technique is valuable for specifying other resolutions or maximum frequencies for different applications, or to limit file size used for de-embedding.

InfiniiSim scans S-parameter files for special/optional keywords that communicate to InfiniiSim how the data within the file was generated or how it should be interpreted. The following keywords are the currently supported:

- "DEFAULT_FREQUENCY_RESOLUTION" on page 116
- "RETURN_LOSS_INTERPRETATION" on page 117

The format of keyword lines differs slightly between Touchstone and CITIfile files. In Touchstone files, keywords must be preceded by an exclamation point (!).

InfiniiSim keywords are case-insensitive and permit additional comments on the same line.

DEFAULT_FREQUENCY_RESOLUTION

```
#DSO DEFAULT_FREQUENCY_RESOLUTION <value>
```

where <value> may be a numeric value in Hz or the text strings AUTO or AUTOMATIC.

If the DEFAULT_FREQUENCY_RESOLUTION keyword is used and its <value> is numeric, the data is resampled using linear interpolation and the frequency resolution specified by <value>.

If the keyword is used and its <value> is either of the text strings "AUTO" or "AUTOMATIC", the data is resampled using linear interpolation and a frequency resolution equal to the frequency interval between the last two frequency points in the file. No warning is generated even if the file data is non-uniformly sampled.

In the absence of the keyword or if <value> is an unrecognized text string, InfiniiSim checks the data to ensure it is uniformly sampled in frequency (except the interval between DC and then next lowest frequency point). If the data is not uniformly sampled, InfiniiSim resamples the data using a frequency resolution equal to the frequency interval between the last two frequency points in the file and generates a warning.

RETURN_LOSS_INTERPRETATION

```
#DSO RETURN_LOSS_INTERPRETATION <string>
```

where <string> is one of the following legal strings: SCOPE, PROBE, SERIES_RLC, SHUNT_RLC, or DEFAULT.

In the absence of this keyword, or when <string> equals SHUNT_RLC, InfiniiSim assumes that the return loss data models a simple passive shunt RLC circuit and calculates the insertion loss of the same circuit that would produce the file data ($S_{21} = S_{12} = 1 + S_{11}$, $S_{22} = S_{11}$).

When <string> equals SERIES_RLC, InfiniiSim assumes that the return loss data models a simple passive series RLC circuit ($S_{21} = S_{12} = 1 - S_{11}$, $S_{22} = S_{11}$).

When <string> equals SCOPE, InfiniiSim assumes that the return loss data models an oscilloscope input ($S_{21} = 1$, $S_{12} = S_{22} = 0$).

When <string> equals PROBE, InfiniiSim assumes that the return loss data models a probe input ($S_{21} = 1 + S_{11}$, $S_{12} = 0$, $S_{22} = -1$).

When <string> equals DEFAULT, InfiniiSim uses its default interpretation without creating the default return loss warning.

Measuring S-Parameters

The detailed use of a vector network analyzer (VNA) to measure the S-parameter file for a circuit element is not described here. However, there are certain general procedures and factors that should be observed to help ensure good VNA measurements.

Frequency Sweep Because de-embedding applications transform between the time and frequency domains using Fourier transforms, they must operate on uniformly sampled data that extends across the entire frequency range; f_{DC} to f_x. S-parameter data can be generated/modeled using either a linear or logarithmic frequency sweep. If a logarithmic sweep is used, then that data necessarily must be linearly re-sampled or interpolated by the de-embedding application. This resampling will be valid only if the amplitude and phase characteristics are adequately sampled. Logarithmic swept data can sometimes be used, but linearly swept data is always preferred.

IF Bandwidth It is recommended that 1 kHz or lower be used as the IF (intermediate frequency) bandwidth. 1 kHz IF bandwidth provides superior signal-to-noise (S/N) at the price of only slightly increased measurement time.

Minimum Frequency

Exercise caution in choosing the minimum measurement frequency. The accuracy of most microwave VNAs degrades considerably at low frequencies. It is often necessary to break VNA measurements into two separate ranges using a separate low-frequency VNA for the low frequency range and then combining the S-parameter data into a single file.

Frequency Resolution

Whether synthesizing S-parameters with a circuit simulator or measuring them directly with a vector network analyzer, it is important to select an appropriate frequency resolution for the S-parameter data. Obviously, using too fine of resolution will produce very large data files and increase measurement and processing times. Using too coarse a resolution reduces file size but can make the data too granular to analyze accurately.

Note that there are several important limitations to maximizing the frequency resolution. The first is interpolation. The S-parameter data must be "interpolatable" in the frequency domain. This will be achieved if the resolution used ensures the data contains all of the significant information needed for the application. Having too fine a resolution does not add any information to the frequency domain sampling and will merely cost more time in the measurement process.

Settling Time

Settling time presents a limitation as well. Resolution in the frequency domain corresponds to range in the time domain. The frequency resolution must be set be fine enough in the S-parameters to model the lowest time-constant elements of the circuits. The best method for determining how much frequency resolution the models require is by observing how much time range is required for the step response to settle in the time domain. Then, the resolution can be determined by:

FreqRes = 1 / TimeRange

Also, the derived S-parameters need to have enough frequency resolution to accurately represent the full group delay of the circuits that they are modeling. The recommended maximum resolution to model coax cable for example is:

FreqRes = (1/4) / PropDelay

This may appear to be twice as fine as needed, but it is often required to extrapolate measured data down to DC or zero Hz. Extrapolating the phase down to DC with only half of this resolution is prone to error, and therefore, the higher frequency resolution is warranted.

Maximum Frequency

As with frequency resolution, the maximum frequency of the S-parameters also needs to be chosen with care. The primary effect of choosing an insufficiently large maximum frequency is unwanted preshoot and ringing as described in the following section about causality (see "Causality" on page 125). The models themselves may have preshoot and ringing as long as they are stimulated by the spectral content of the measured signal. In most cases, some preshoot and ringing is acceptable in the simulated signal as long as it is insignificant relative to the random noise in the signal. Average-mode measurements (on the oscilloscope) have a much higher signal-to-noise ratio and, therefore, afford the use of models with higher maximum frequency.

The signal-to-noise ratio of the measured signal has a major effect on the maximum frequency to which lossy test fixtures and cables can be de-embedded. It is usually recommended to limit the bandwidth of de-embedding applications when applying the transfer function and allow the models themselves to have a higher bandwidth. This is so that you can, if desired, adjust the de-embedding bandwidth by observing the simulated signal and re-computing (see "FIR Filter Bandwidth" on page 149). New transfer functions take much longer to compute than using previously generated transfer functions with different bandwidths.

Network Analyzer Calibration

Finally, network analyzer calibration is critical. If the network analyzer is not calibrated, all the data acquired can be corrupt. Newer models of VNAs are easier and less complicated to calibrate than older models. However, close attention must be paid to the calibration process of all models because even the slightest calibration error voids all subsequent data acquisitions. It is highly recommended that the VNA's calibration is verified by measuring the calibration standards after each calibration.

Common Issues in Measuring S-Parameters

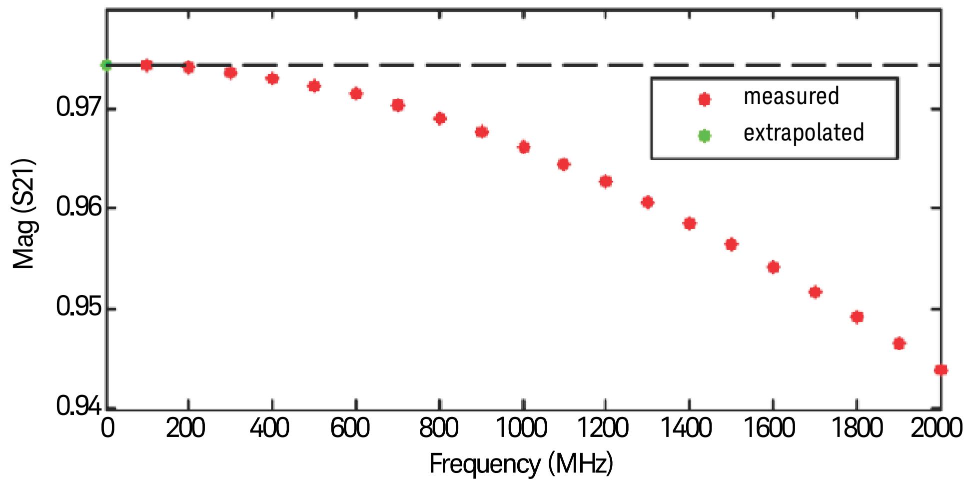
If the case arises that the S-parameter file set must be evaluated and processed further in order to deliver optimal accuracy in the process of de-embedding or embedding, the following provides insights to the issues and guidance in addressing them.

- "Extrapolation To DC" on page 119
- "Modeling Noisy Measurement Data" on page 121
- "Averaging S-Parameters" on page 122
- "Frequency Response Evaluation of Insertion Loss" on page 123

Extrapolation To DC

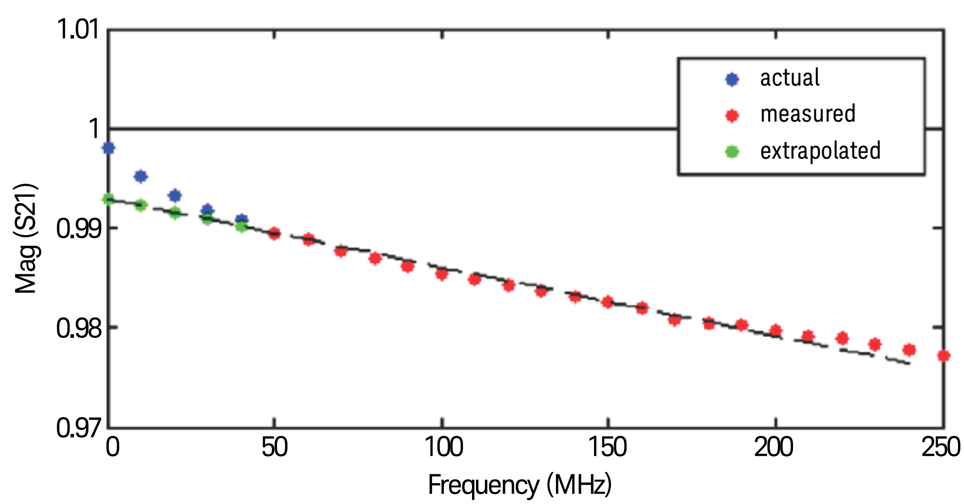
S-parameters for de-embedding require values that extend all the way down to and including 0 Hz. If the S-parameters do not extend all the way down to DC and cannot be measured with a VNA, the de-embedding applications will be forced to extrapolate them to DC. Two common extrapolation techniques are:

- Simply copying the first measured point to DC.

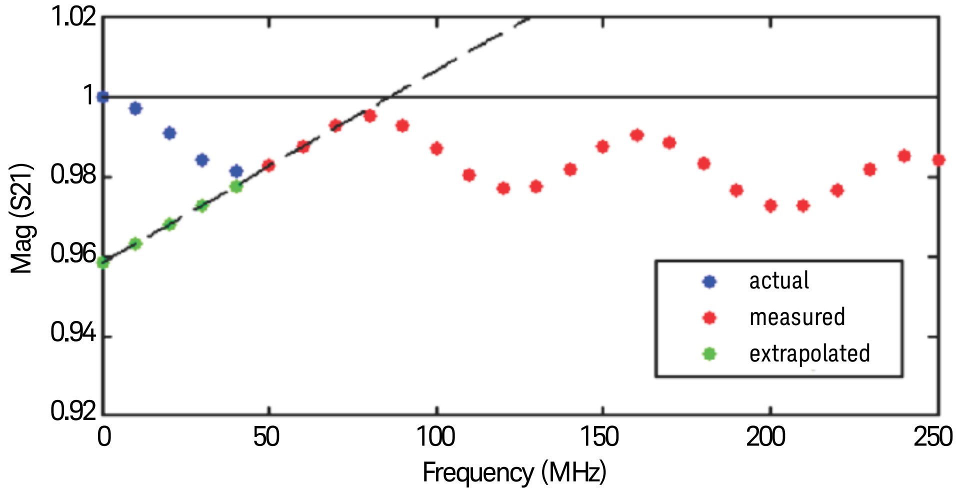


When files do not contain DC values, InfiniiSim will use this method and copy the file's lowest frequency values to 0 Hz and set their phase components to zero. This technique may not always yield appropriate DC S-parameter values. Ideally, you should measure or simulate the true DC values and manually add them to the S-parameter file with a text editor.

- Linearly fitting the first two measured points. Some de-embedding tools (other than InfiniiSim) use this method.



The linear fitting extrapolation method can sometimes provide a better match to the actual value, but is particularly susceptible to exaggerating the DC error either from measurement noise or from ripple in the model's response as in the following figure.



In either event, while extrapolating S-parameters data to DC will generally provide valid data, in some cases it can be problematic and you should take care to ensure the data is accurate.

Both the copy method and the linear fitting method are valid extrapolation methods provided that the lowest measured frequency points are sufficiently close to DC. However, high-frequency VNAs, the most common tool for measuring S-parameters, often cannot sweep low enough, or their accuracy degrades so much that their lowest measured values cannot be accurately extrapolated. In such cases, the best procedure is to measure the S-parameters in two separate frequency bands using two different VNAs and then merge them together. In any event, it is prudent to always verify that the S-parameters contain valid DC values or that they can be extrapolated to DC without introducing significant error.

Modeling Noisy Measurement Data

Simulated S-parameters are generally superior to measured S-parameters for de-embedding applications for a variety of reasons. The obvious ones are that they are noiseless and they inherently include a 0 Hz value. They also do not suffer from reduced accuracy at low frequencies. Measured S-parameters can include all characteristics of the measured device, but they also offer the option of correcting only selected characteristics. For example, multiple impedance discontinuities on a long transmission line can produce ripple in the insertion loss of the cable. As is often the case, the correction should be applied to the overall loss of the cable, but not the ripple of that particular cable. This is because the ripple is highly sensitive to the exact delay between the discontinuities of that particular cable, but the loss is not and can be applied to a variety of similar cables. Simulating S-parameters using a lossy transmission line model that matches the loss of the measurement allows modeling the loss only.

Another area where measured data can give improved results by optimization with a simulator is where VNA noise or bias error yields a non-passive result on a known passive device. In this case, the simulator tool performs an optimization of given circuit structures to the measured data.

Averaging S-Parameters

There are several situations where averaging S-parameters is appropriate. One instance would be for creating a nominal model set of test fixtures or reducing the noise in a set of measured S-parameters. While it appears to be a straight-forward proposition, it turns out that averaging complex values is more problematic than initially assumed.

The reason for this is that there are two ways to average complex numbers—averaging the real and imaginary values or averaging the magnitude and phase values. And, the method chosen will depend on the specific application. The following discussion details the approaches to averaging.

TIP **Be careful when de-embedding using nominal models. Correction transfer functions can be very sensitive to the absolute time delay of some circuit elements. De-embedding with a nominal delay model may actually make the measurement results worse.**

Random Noise Measurement

This measurement most generally appears as a two-dimensional Gaussian error vector added to the actual device model data vector at each frequency in the data set. In other words, each real and imaginary component of noise is comprised of independent Gaussian distributions. So for reducing measurement noise, the approach is to average its real and imaginary components. Visualize, for example, averaging out the noise of an S_{11} measurement of a perfect 50 **Ω** termination. The real and imaginary components would average down to zero, but the magnitude and phase would not.

Assume that the objective is to determine the nominal model for a set of coax cables. Also assume that the insertion loss variation and time delay variation of the cables is much larger than the measurement noise. In this case, where the goal is to average the S_{21} data for a set of cables that all had the same loss, but different time delays, it is better to average magnitude and phase. The result is that the magnitude and phase would average to correct values, but real and imaginary components would not.

It is usually better to average the real and imaginary components of insertion loss values, whether reducing measurement noise or generating nominal models. That is because the process variation of S_{11} for many devices is large relative to its nominal value, producing the effect described for measurement noise reduction. Also, note that averaging magnitude, in dB, is not possible. The approach is to convert it to a simple gain value before averaging. Phase values typically need to be unwrapped prior to averaging.

Frequency Response Evaluation of Insertion Loss

It is imperative to examine the frequency response of the insertion loss of the circuit element being measured.

One issue is interpolate-ability. This implies that the frequency resolution is fine enough to capture all the "bumps and wiggles" of the device. This is directly analogous to sampling in time domain. The directive is to always sample quickly enough to capture all trends of the response.

A second issue is quickly changing insertion loss responses. Because such device responses may change significantly over time and temperature, the magnitude and phase is affected, proportionally. Unless you are aware of this and compensates accordingly, it can make a de-embed correction worse than the non-de-embedded result. For accurately de-embedding and embedding, correct phase is just as important as correct magnitude.

Finally, when using this approach, consider limiting the bandwidth of the measurement when a device exceeds ~15 dB of gain range. In the case of de-embedding a lossy channel, for example, the frequency bands with greatest loss will have the greatest de-embedded gain. If there is a section with 20 dB of loss, that band may have only a 6 dB of S/N ratio (function of signal spectrum) and amplifying it by 20 dB (for de-embedding) would result in a very noisy waveform. In the example in the following figure, the reduction of signal-to-noise at around 4.5 GHz because of the channel, gives rise to oscillatory behavior in the step function.

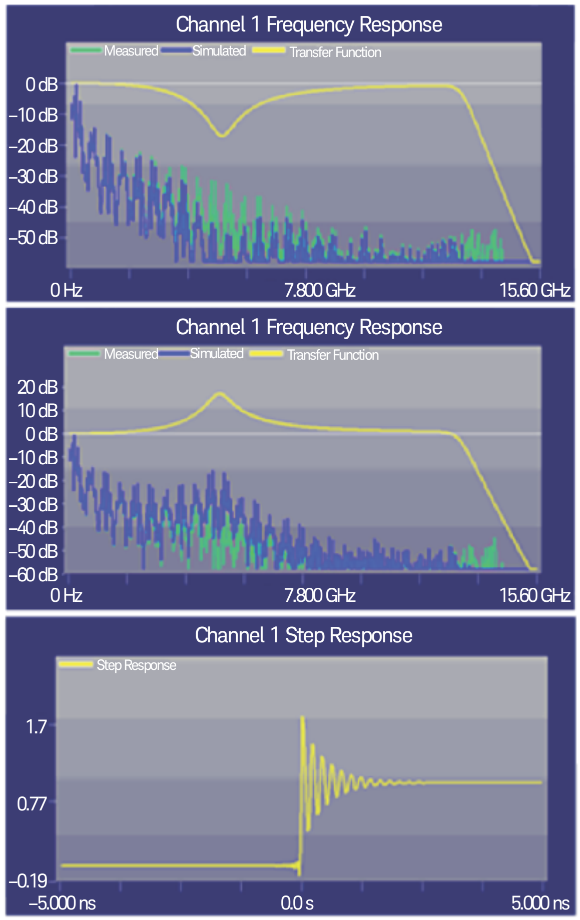


Figure 6 Result of de-embedding a section of lossy channel

Other Issues for Consideration

The preceding discussion has addressed the best practice approach for addressing the issues common to S-parameter comprehension, manipulation, and pitfalls. We hope you now have a reasonable understanding of the global approach to obtaining reliable S-parameter readings. There can be a couple of peripheral effects that can skew otherwise seemingly valid measurements:

- "Passivity" on page 125
- "Causality" on page 125

Passivity

Concern over the passivity of N-port network models arose from the possibility that some circuit simulators become unstable when ideally passive model elements possess a slight gain. This can happen, for example, when measurement noise causes the measured insertion loss of a passive connector to have a magnitude greater than one at some frequency.

While this is not a problem for de-embedding applications, it is a condition of the process and you should be aware of it. It is mentioned here for completion, and you should be aware that forcing measured S-parameter data to be passive (say by truncation) actually creates a less tolerable error than it removes, and is not recommended. When possible, measured S-parameters should be replaced by simulated models that are fit to the measured data, as described below. This not only ensures passivity, it removes all noise from the S-parameters.

Causality

As with passivity, the concern over the causality of S-parameter models causes many engineers to corrupt their otherwise accurate circuit models in order to make them causal. You know that the device is causal and therefore assume that the respective model must be causal as well. However, that is not always the case.

One source of the concern about causality originates from some over-simplified commercially available lossy transmission line models. These models do not model phase at all and are highly non-causal. They are inappropriate for any time-domain analysis and should not be used for oscilloscope de-embedding applications.

Another source of concern is the Gibbs phenomenon that occurs when converting a band-limited frequency domain model to the time-domain using Fourier transforms. Consider the RC high-pass filter model whose frequency response is shown in the following figure. Notice that it is only modeled up to 2 GHz.

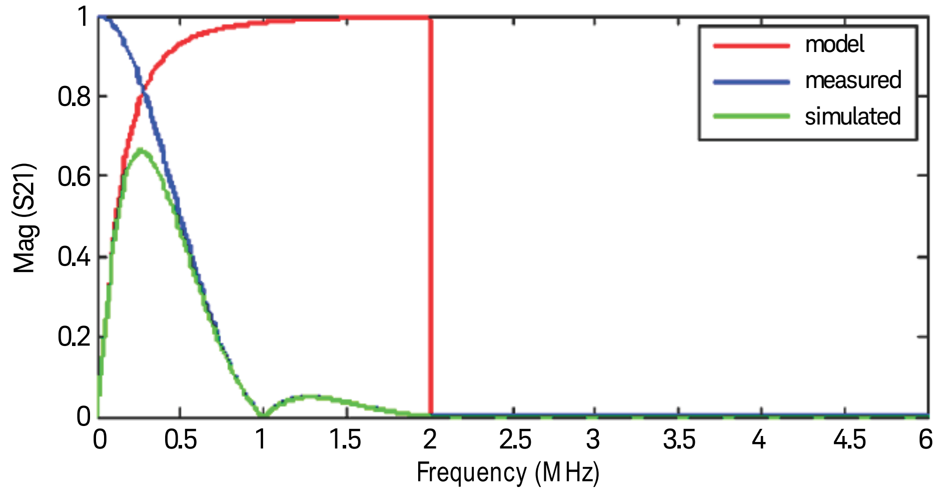


Figure 7 RC high-pass filter model frequency response

Viewed in the time-domain, the model's impulse response appears to be non-causal as shown in the following figure. But, the preshoot seen on the impulse response of the model does not appear on the simulated signal as long as the measured signal's significant spectral content does not extend above 2 GHz.

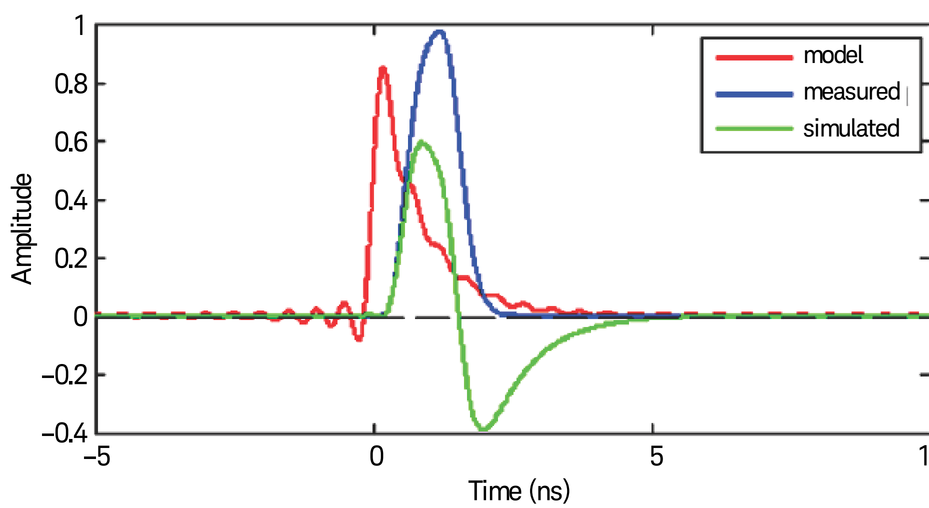


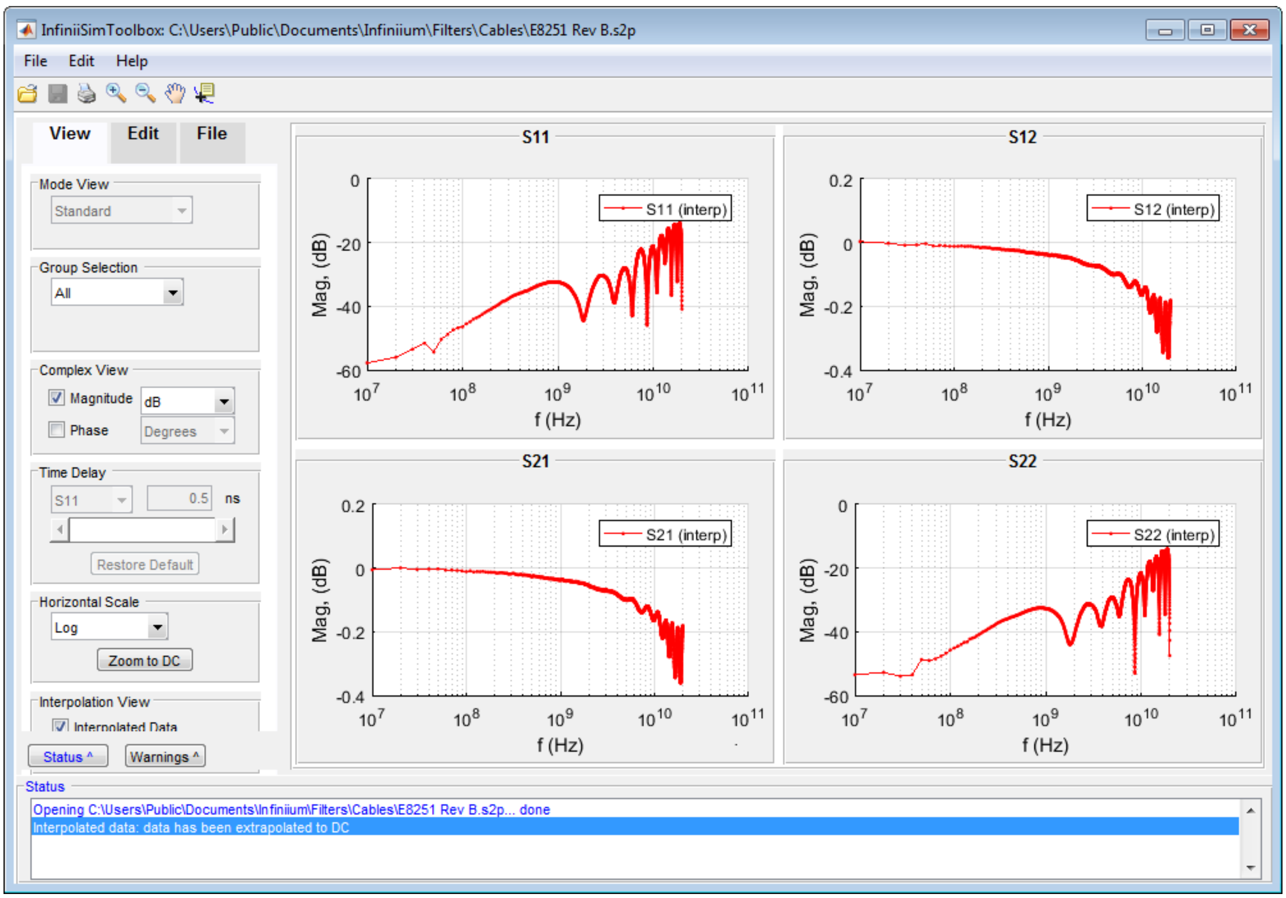
Figure 8 RC high-pass filter model impulse response

It is worth noting that, although most S-parameters used for oscilloscope de-embedding applications do not contain meaningful non-causal information, S-parameter data does support negative time information. The requirement that S-parameter data be "interpolate-able" in the frequency domain also ensures the data supports negative time.

S-Parameter and Transfer Function File Viewers

You can evaluate S-parameter and transfer function files using a viewer.

In the Infiniium user interface, wherever you can select an S-parameter or transfer function file, there is a **View...** button you can click to open the InfiniiSimToolbox, a MATLAB application for viewing these files. Once in the InfiniiSimToolbox application, you can use the **File** menu to open and evaluate files.



The InfiniiSimToolbox application has options for:

- Viewing **All**, **Individual**, **Reflection**, or **Transmission** S-parameters.
- Viewing **Magnitude** as well as **Phase** responses (and specifying the Y axis units).
- Changing **Time Delays** in the phase responses.

- Choosing a Log or Linear **Horizontal Scale**.
- Viewing **Interpolated Data**, **File Data**, or both.
- Displaying **Status** or **Warning** messages.
- Editing the port connections used in the response plots.
- Viewing the S-parameter file as a text file.

For more detail on how the InfiniiSimToolbox application works, see its online help.

Examples of some other third-party S-parameter viewers are:

- S-Parameter Explorer (http://www.eecircle.com/downloads/spex.html)
- RF Toolbox in MATLAB (http://www.mathworks.com/products/rftoolbox/)
- scikit-rf package for RF/Microwave engineering (http://scikit-rf-web.readthedocs.org/)

Port Ordering in S-Parameter Models

Network elements are represented as n-port, where n is usually two or four. The value of "n" could be as high as six, eight, or even 16, where crosstalk between lanes is desired to be analyzed. Realistically, however, such values are rarely seen for oscilloscope applications. Single-ended systems usually consist of 2-port elements. Differential systems, as many of the high-speed interfaces today are, require 4-port models.

In such network representations, the major issues revolve around frequency. Frequent anomalies arise in the inconsistent ordering of the ports between measurement and transfer function generation.

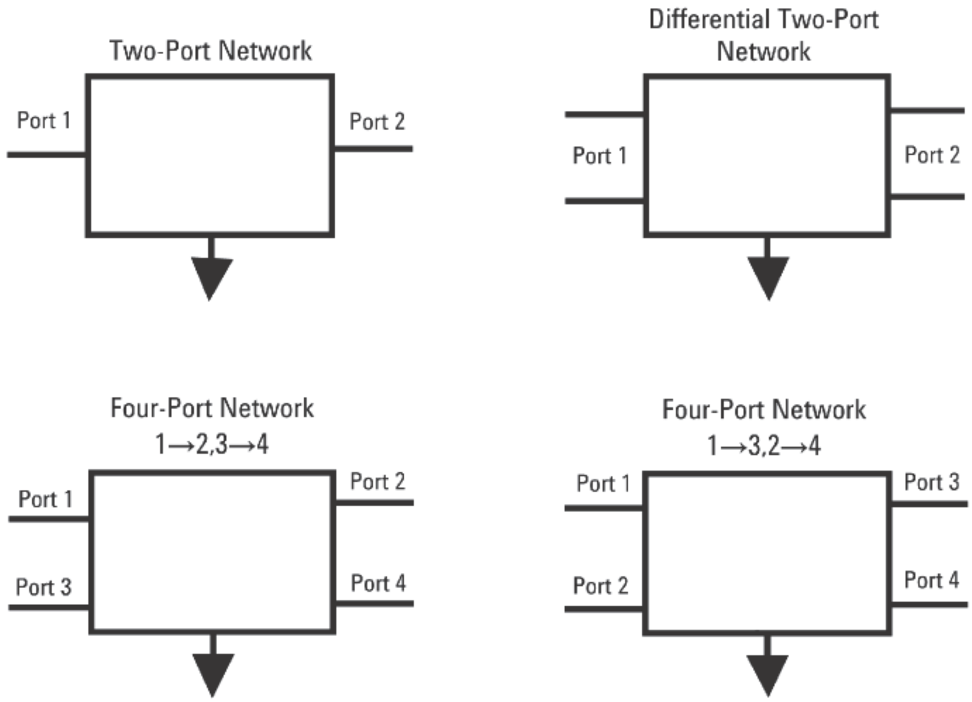


Figure 9 Generalized two- and four-Port models of circuit elements

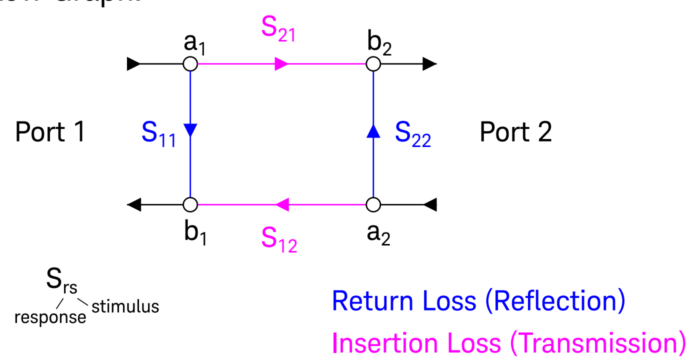
For industry standard two-ports, port 1 is the input and port 2 is the output. (1→2).

For four-ports, standard usage allows ports 1 and 3 or ports 1 and 2 as the inputs. (1→2, 3→4 or 1→3, 2→4 respectively).

Port assignments such as 1→4, 3→2, while possible to re-order to standard configuration, are undesirable for many tools and so should be avoided.

S-Parameters for Single-Ended 2-Port: Flow Graph and Matrix

Flow Graph:

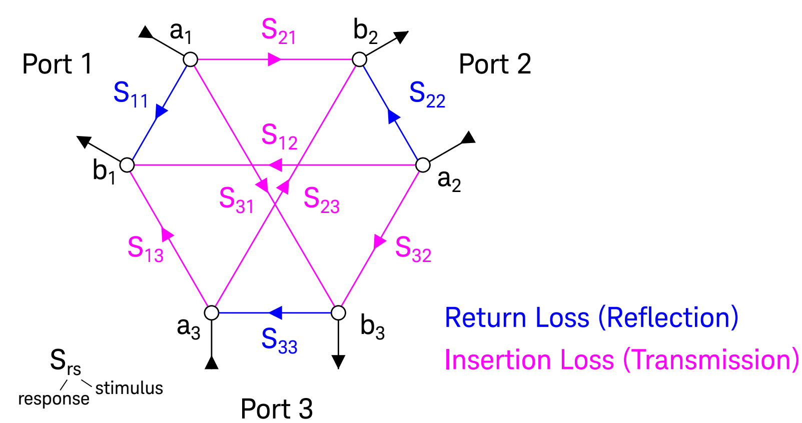


Matrix:

$$\begin{Bmatrix} b_1 \\ b_2 \end{Bmatrix} = \begin{bmatrix} S_{11} & S_{12} \\ S_{21} & S_{22} \end{bmatrix} \begin{Bmatrix} a_1 \\ a_2 \end{Bmatrix}$$

S-Parameters for Single-Ended 3-Port: Flow Graph and Matrix

Flow Graph:

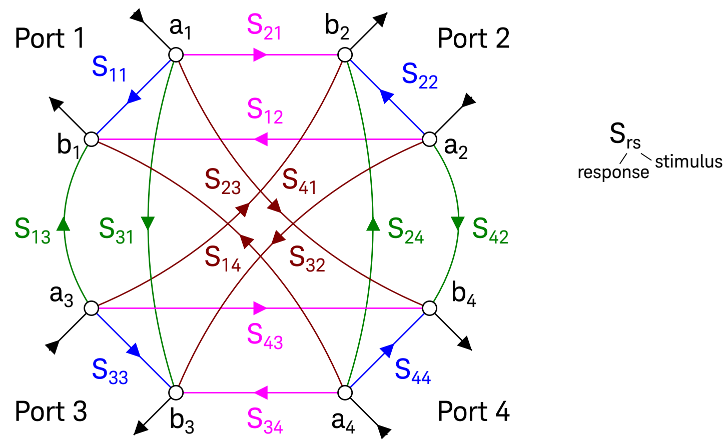


Matrix:

$$\begin{Bmatrix} b_1 \\ b_2 \\ b_3 \end{Bmatrix} = \begin{bmatrix} S_{11} & S_{12} & S_{13} \\ S_{21} & S_{22} & S_{23} \\ S_{31} & S_{32} & S_{33} \end{bmatrix} \begin{Bmatrix} a_1 \\ a_2 \\ a_3 \end{Bmatrix}$$

S-Parameters for Single-Ended 4-Port: Flow Graph, Matrix, and Extractions

Flow Graph:



Matrix:

$$\begin{Bmatrix} b_1 \\ b_2 \\ b_3 \\ b_4 \end{Bmatrix} = \begin{bmatrix} S_{11} & S_{12} & S_{13} & S_{14} \\ S_{21} & S_{22} & S_{23} & S_{24} \\ S_{31} & S_{32} & S_{33} & S_{34} \\ S_{41} & S_{42} & S_{43} & S_{44} \end{bmatrix} \begin{Bmatrix} a_1 \\ a_2 \\ a_3 \\ a_4 \end{Bmatrix}$$

Return Loss (Reflection)
Insertion Loss (Transmission)
Near-End Crosstalk (NEXT)
Far-End Crosstalk (FEXT)

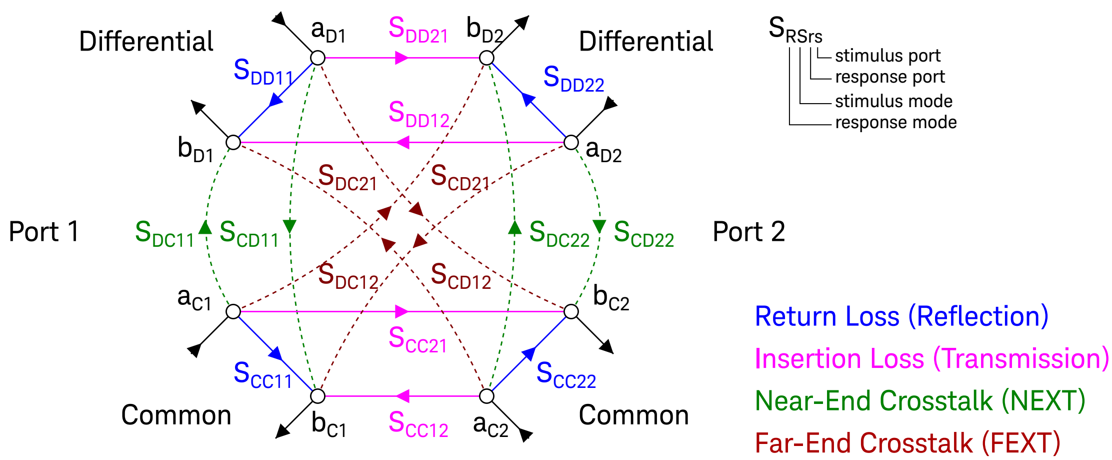
Extractions:

Ports 1 -> 2 = $\begin{bmatrix} S_{11} & S_{12} \\ S_{21} & S_{22} \end{bmatrix}$ Ports 3 -> 2 = $\begin{bmatrix} S_{33} & S_{23} \\ S_{32} & S_{22} \end{bmatrix}$

Ports 3 -> 4 = $\begin{bmatrix} S_{33} & S_{34} \\ S_{43} & S_{44} \end{bmatrix}$ Ports 1 -> 4 = $\begin{bmatrix} S_{11} & S_{14} \\ S_{41} & S_{44} \end{bmatrix}$

Mixed-Mode S-Parameters for Differential 2-Port: Flow Graph, Matrix, and Extractions

Flow Graph:



Matrix:

		Diff Stimulus Port 1	Diff Stimulus Port 2	Common Mode Stimulus Port 1	Common Mode Stimulus Port 2
Differential Response	Port 1	S_{DD11}	S_{DD12}	S_{DC11}	S_{DC12}
	Port 2	S_{DD21}	S_{DD22}	S_{DC21}	S_{DC22}
Common Md Response	Port 1	S_{CD11}	S_{CD12}	S_{CC11}	S_{CC12}
	Port 2	S_{CD21}	S_{CD22}	S_{CC21}	S_{CC22}

$$\begin{Bmatrix} b_{D1} \\ b_{D2} \\ b_{C1} \\ b_{C2} \end{Bmatrix} = \begin{bmatrix} \begin{bmatrix} S_{DD11} & S_{DD12} \\ S_{DD21} & S_{DD22} \end{bmatrix} & \begin{bmatrix} S_{DC11} & S_{DC12} \\ S_{DC21} & S_{DC22} \end{bmatrix} \\ \begin{bmatrix} S_{CD11} & S_{CD12} \\ S_{CD21} & S_{CD22} \end{bmatrix} & \begin{bmatrix} S_{CC11} & S_{CC12} \\ S_{CC21} & S_{CC22} \end{bmatrix} \end{bmatrix} \begin{Bmatrix} a_{D1} \\ a_{D2} \\ a_{C1} \\ a_{C2} \end{Bmatrix}$$

Extractions:

$$\text{Differential} = \begin{bmatrix} S_{DD11} & S_{DD12} \\ S_{DD21} & S_{DD22} \end{bmatrix}$$

$$\text{Common Mode} = \begin{bmatrix} S_{CC11} & S_{CC12} \\ S_{CC21} & S_{CC22} \end{bmatrix}$$

Equal to zero in ideal, balanced device:

Differential Resonse to Common Mode Stimulus $= \begin{bmatrix} S_{DC11} & S_{DC12} \\ S_{DC21} & S_{DC22} \end{bmatrix}$

Common Mode Response to Differential Stimulus $= \begin{bmatrix} S_{CD11} & S_{CD12} \\ S_{CD21} & S_{CD22} \end{bmatrix}$

The signals input to the network are defined as having both differential and common components. To support this more intuitive treatment, the standard 4-port S-parameter set can be transformed to comprise four matrices. The first matrix (top left) relates differential responses to differential inputs. The second matrix (bottom right) relates common mode responses to common mode inputs.

The remaining two matrices relate conversions between differential and common mode inputs. In equation form, the differential and common mode outputs are represented as:

$O_{DM} = I_{Diff} * G_{DM} + I_{CM} * G_{Common\_to\_Diff}$

$O_{CM} = I_{CM} * G_{CM} + I_{DM} * G_{Diff\_to\_Common}$

The mixed-mode description is more intuitive because you can view the S-parameter data and comprehend where the problems are in both the frequency domain and the observe mode conversions.

A simplification on the mixed-mode is a Differential 2-port. In this case, only a portion of the second equation above is used and takes into account only differential input and output:

$O_{DM} = I_{Diff} * G_{DM}$

This is effectively a 2-port analysis as the inputs are a pure differential signal. This assumes common mode input is small, that cross conversion terms and common mode gain terms are insignificant. With this simplification, you needs to validate that the assumptions are true. In more complex circuits, where a number of differential structures exist, conversion terms will not be zero and there is always some common mode component in an input signal.

InfiniiSim Transfer Functions

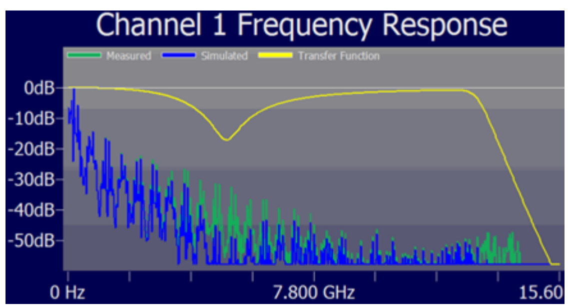
InfiniiSim generates transfer function files by comparing measurement and simulation circuit models. Transfer function files describe, in the frequency domain, the complex magnitude and phase relationship between two nodes (measurement and simulation) as a tabular list of values at specific frequencies.

- "The Idea of the Transfer Function" on page 134
- "Transfer Function Derivation" on page 135
- "Two-Port vs. Four-Port Transfer Functions" on page 139
- "Transfer Function Inputs and Outputs" on page 140
- "Transfer Function Files" on page 146

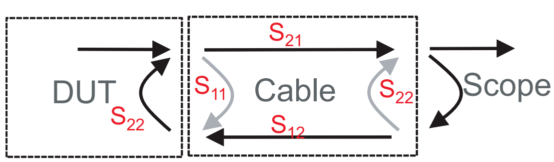
Once a transfer function is generated, InfiniiSim performs an inverse Fourier transform on it to yield an impulse response. The impulse response is sampled at the sampling rate of the oscilloscope to yield an FIR correction filter that is convolved with acquired waveform to render the simulated waveform.

The Idea of the Transfer Function

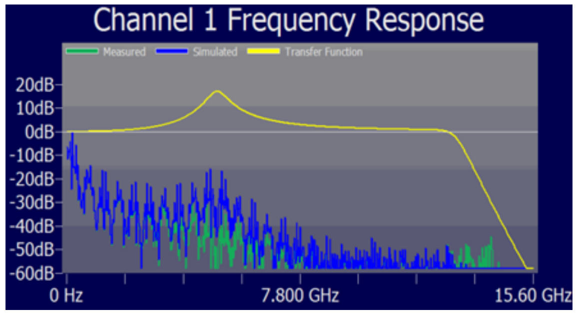
For example, if you have a cable with the following frequency response, measured as S-parameters using a vector network analyzer (VNA):



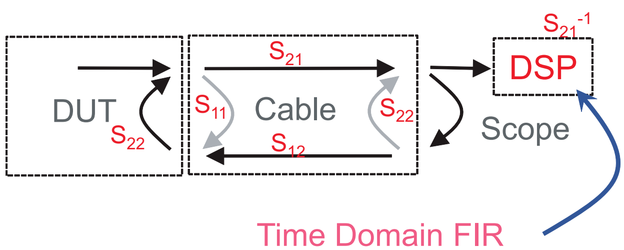
And considering *only* the transmission characteristic for a "de-embed" (that is, in S-parameter terms, consider S_{21}):



InfiniiSim can generate a transfer function to invert the transmission characteristic (S_{21}^{-1}):



InfiniiSim then performs an inverse Fourier transform on the transfer function to yield an impulse response. The impulse response is sampled at the sampling rate of the oscilloscope to yield an FIR filter that is convolved with acquired waveform to render the simulated waveform.



InfiniiSim can create FIR filters for embedding, de-embedding, and virtual probing, not only for this simple case, but in much more varied cases.

Transfer Function Derivation

Deriving the filter transfer function requires the definition of two circuits. The first circuit accurately reflects the existing measurement conditions on the bench, including the oscilloscope observation point (either oscilloscope's front-panel input connector or a probe). The second circuit reflects the desired measurement conditions.

As an example, assume a cable exists between the source device under test (DUT) and the oscilloscope. The objective is to remove (de-embed) the effects of the cable. In this case, the measurement circuit would manifest a single cable between the source and an oscilloscope channel input.

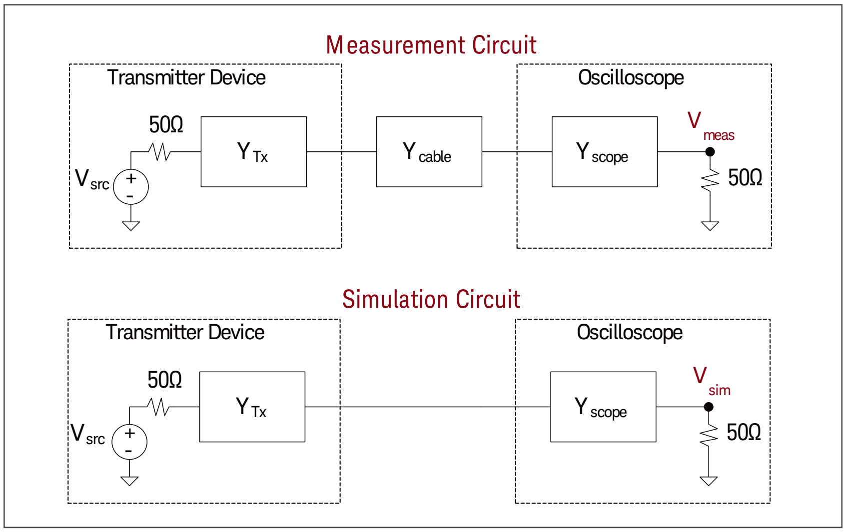


Figure 10 Measurement and simulation circuit models for observing waveform without cable

As shown in the previous figure, the simulation circuit would be identical to the measurement circuit except that its cable has been removed (in other words, a simulation of the oscilloscope directly connected to the source). By assuming that these two circuits are linear and time-invariant, the simulation can be performed by basic nodal analysis. The analysis of these circuits results in two transfer functions (in the frequency domain), from a common source to each circuit's observation node.

- Measurement circuit transfer function (H_m)
- Simulation circuit transfer function (H_s)

Given:

$$H_m(f) = \frac{(V_{meas}(f))}{(V_{src}(f))}$$

$$H_s(f) = \frac{(V_{sim}(f))}{(V_{src}(f))}$$

Taking the ratio of these transfer functions produces the desired correction transfer function, H(f), which is be used to convert, or transform the measured waveform to the simulated waveform.

$$H(f) = \frac{H_s(f)}{H_m(f)} = \frac{V_{sim}(f) \,/\, V_{src}(f)}{V_{meas}(f) \,/\, V_{src}(f)} = \frac{V_{sim}(f)}{V_{meas}(f)}$$

$$V_{meas}(f) * H(f) = V_{meas}(f) * \frac{V_{sim}(f)}{V_{meas}(f)} = V_{sim}(f)$$

Which yields the filtered, or simulated, result desired.

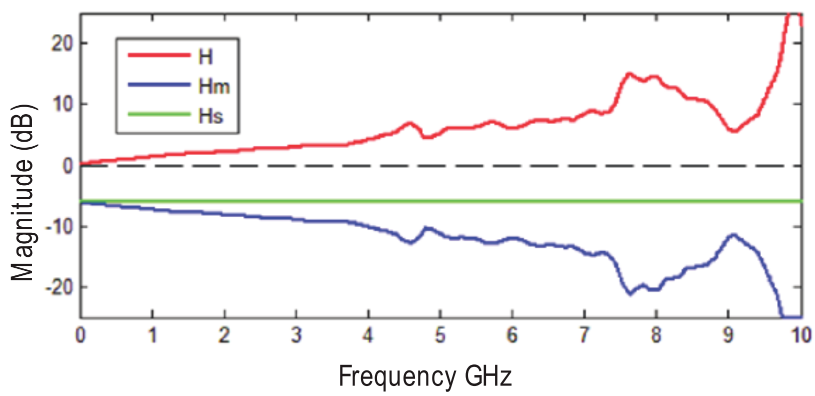


Figure 11 Transfer functions Hm, Hs, and H of the cable de-embedding example

Because the oscilloscope inherently operates in the time domain, it applies the transfer function to the measured waveform, $V_{meas}(t)$ by convolution with the impulse response, h(t). This becomes the inverse Fourier transform of the frequency domain transfer function, H(f), which is:

$$V_{sim}(t) = V_{meas}(t) * h(t)$$

The oscilloscope performs this convolution using a finite impulse response (FIR) digital filter that is implemented using either digital signal processing (DSP) hardware (as in Keysight Infiniium oscilloscopes) or using software (as in the InfiniiSim math functions). While both are accurate and effective, the obvious benefit of a hardware implementation is speed—the processing and updating the display is much faster. The following figure shows the impulse response, h(t) and a filter that could be used to apply it to the measured signal, $V_{meas}(t)$.

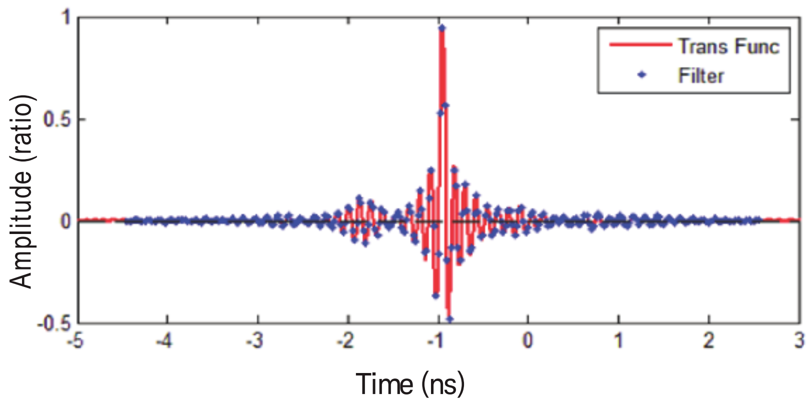


Figure 12 Correction filter (blue) and impulse response of de-embedding transfer function (red) from lossy cable example

The following figure compares the correction filter's frequency response to that of the de-embedding transfer function to be applied to the measured waveform. In this example, the targeted transfer function is a band-limited version of the one calculated previously.

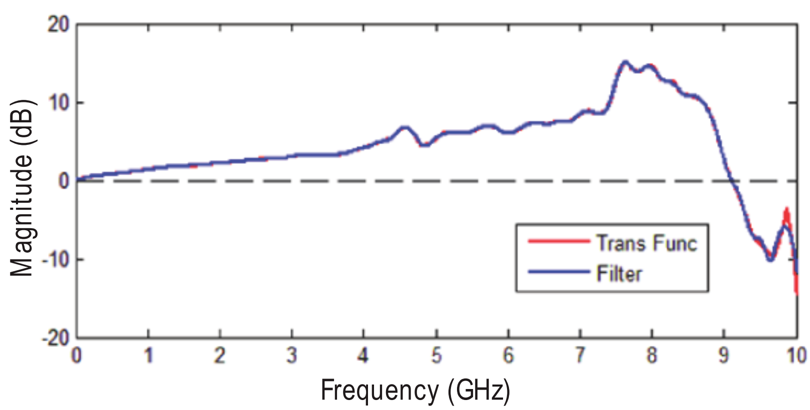


Figure 13 Comparison of correction filter's frequency response (blue) and the targeted de-embedding transfer function's frequency response (red) from the lossy cable example

The waveforms in the following figure represent the effects of applying the correction transfer function shown in the previous impulse response figure, to a measured waveform.

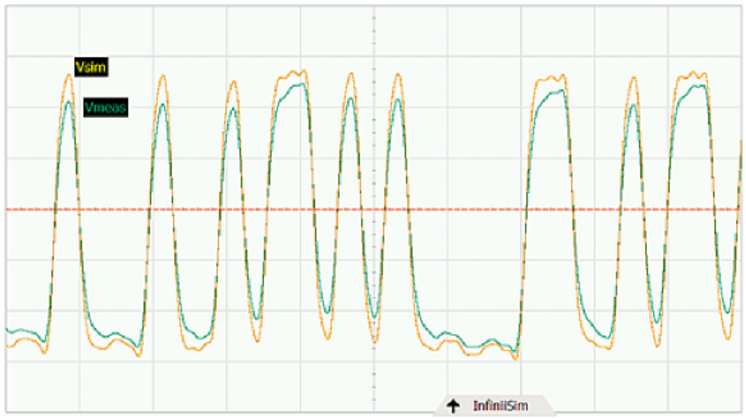


Figure 14 Waveforms V_{meas} and V_{sim} of lossy cable example

Two-Port vs. Four-Port Transfer Functions

The "Transfer Function Derivation" on page 135 topic described de-embedding using single-ended 2-port networks. However, de-embedding can be applied to any N-port network.

Many (if not most) de-embedding applications involve differential signals, and differential de-embedding applications use either differential 2-port or 4-port network models.

Differential 2-Port Networks

Differential 2-port networks use simplified models of the actual physical circuits in which only the odd mode of the signals are modeled and the even mode is ignored. Odd mode is the differential response to differential stimulus and common mode response to common mode stimulus. Even mode is the differential response to common mode stimulus and common mode response to differential stimulus.

Using differential 2-port networks in de-embedding applications can be a very effective way to simplify and accelerate your measurements. However, they can be used only to model highly balanced differential circuits in which there is little to no coupling between the even and odd modes of the circuit.

4-Port Networks

When the ultimate accuracy is required or your differential circuits are not well balanced, you must use 4-port networks for your de-embedding applications. 4-port de-embedding circuits necessarily become more complicated than 2-port circuits because they must model twice as many signals (see the following figure). As a result, the transfer functions derived from these circuits become matrix functions, similar to the 4-port network models themselves. Given:

$$H(f) = \begin{bmatrix} H_{21}(f) & H_{23}(f) \\ H_{41}(f) & H_{43}(f) \end{bmatrix}$$

$$V_{sim-}(f) = H_{21}(f) \cdot V_{meas-}(f) + H_{23}(f) \cdot V_{meas+}(f)$$

$$V_{sim+}(f) = H_{41}(f) \cdot V_{meas-}(f) + H_{43}(f) \cdot V_{meas+}(f)$$

Or in matrix form:

$$V_{sim}(f) = H(f) \cdot V_{meas}(f)$$

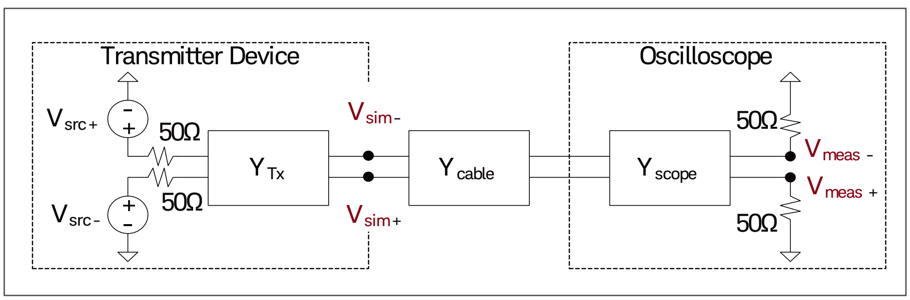


Figure 15 Circuit model of differential application using 4-port Y-parameters

When measurement and simulation circuit models are compared using 2-port network models, 2-port transfer function files are generated.

When measurement and simulation circuit models are compared using 4-port network models, 4-port transfer function files are generated.

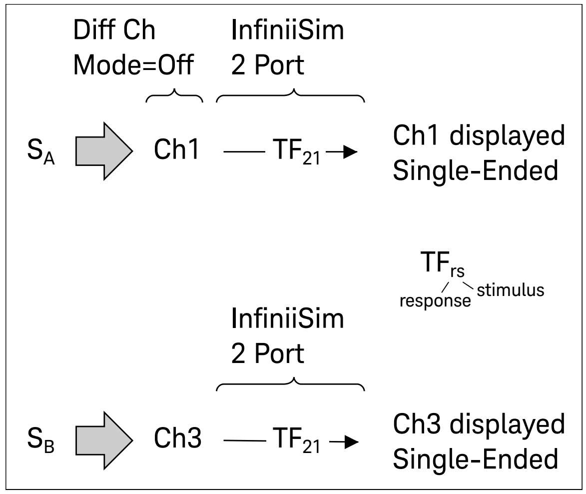
Transfer Function Inputs and Outputs

The following table shows how generated transfer functions are applied to channel 1 and 3 inputs when the various differential channel and InfiniiSim modes are selected.

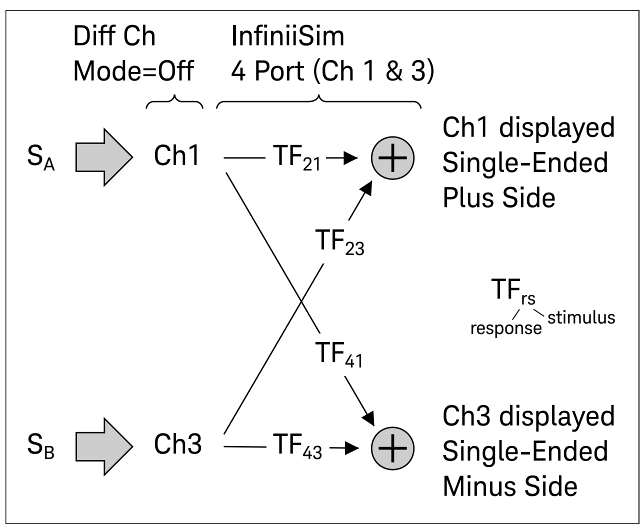
Physical Signal	Scope Ch	Diff Ch Mode	InfiniiSim Mode	TFn Input(s)	Channel Output Displayed
S_A	ch1	off	2 Port	S_A	Single-ended (see page 142)
			4 Port (Channels 1 & 3)	S_A, S_B	Single-ended sum (see page 142)
			4 Port (Channel 1)	S_A	(Extraction dependent) (see page 143)
S_B	ch3		2 Port	S_B	Single-ended (see page 142)
			4 Port (Channels 1 & 3)	S_A, S_B	Single-ended sum (see page 142)
			4 Port (Channel 3)	S_B	(Extraction dependent) (see page 143)
S_A	ch1	on	2 Port	S_A - S_B	Differential (see page 144)
			4 Port (Channels 1 & 3)	S_A - S_B, $(S_A + S_B)/2$	Differential sum (see page 144)
			4 Port (Channel 1 - 3)	S_A - S_B	Differential (with Differential port extraction) (see page 145)
S_B	ch3		2 Port	$(S_A + S_B)/2$	Common Mode (see page 144)
			4 Port (Channels 1 & 3)	S_A - S_B, $(S_A + S_B)/2$	Common Mode sum (see page 144)
			4 Port (Channel 1 + 3)	$(S_A + S_B)/2$	Common Mode (with Common Mode port extraction) (see page 145)

The table for channels 2 and 4 would be similar.

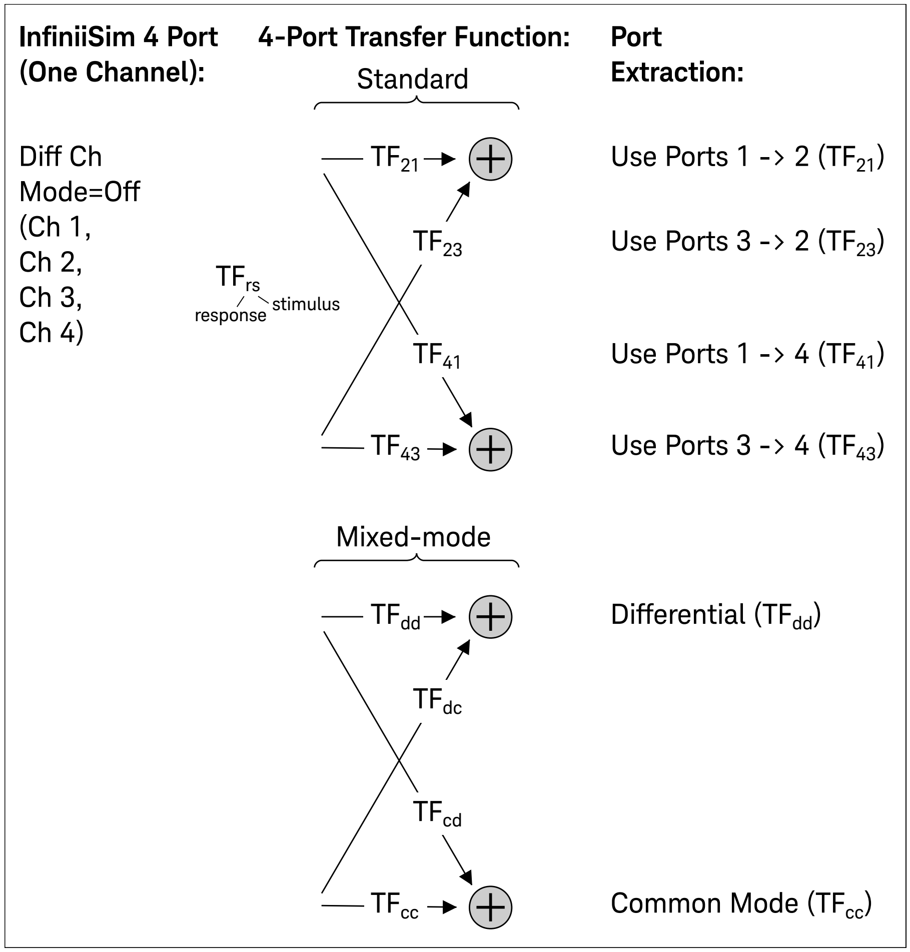
Diff Ch Mode=Off, 2 Port



Diff Ch Mode=Off, 4 Port (Channels 1 & 3)



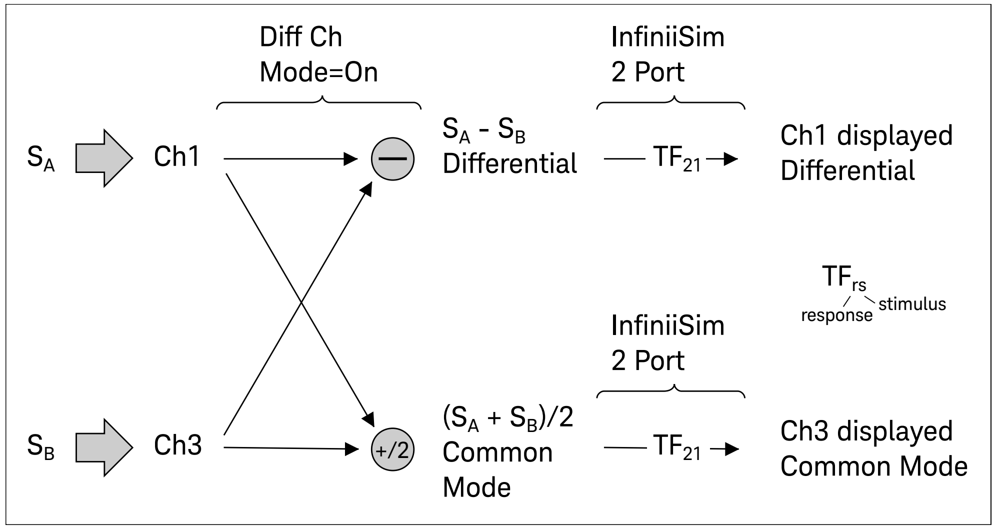
Diff Ch Mode=Off, 4 Port (One Channel), Port Extractions



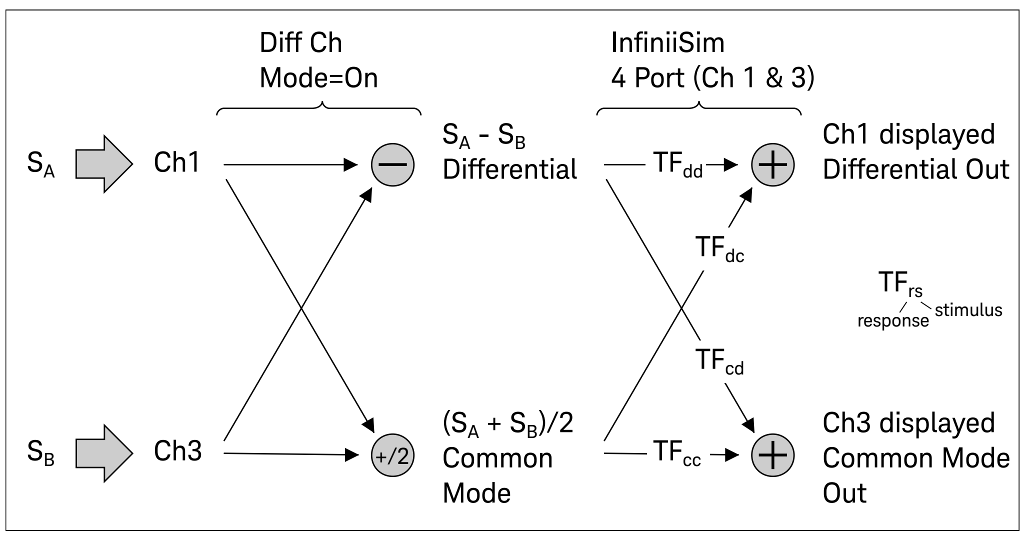
NOTE

InfiniiSim always saves transfer function files in the standard form (TF_{21}, TF_{23}, TF_{41}, TF_{43}) and converts them internally to mixed-mode form (TF_{dd}, TF_{dc}, TF_{cd}, TF_{cc}) when differential channels mode is turned on or when a differential or common mode extraction is selected for the 4-port, one channel InfiniiSim mode.

Diff Ch Mode=On, 2 Port



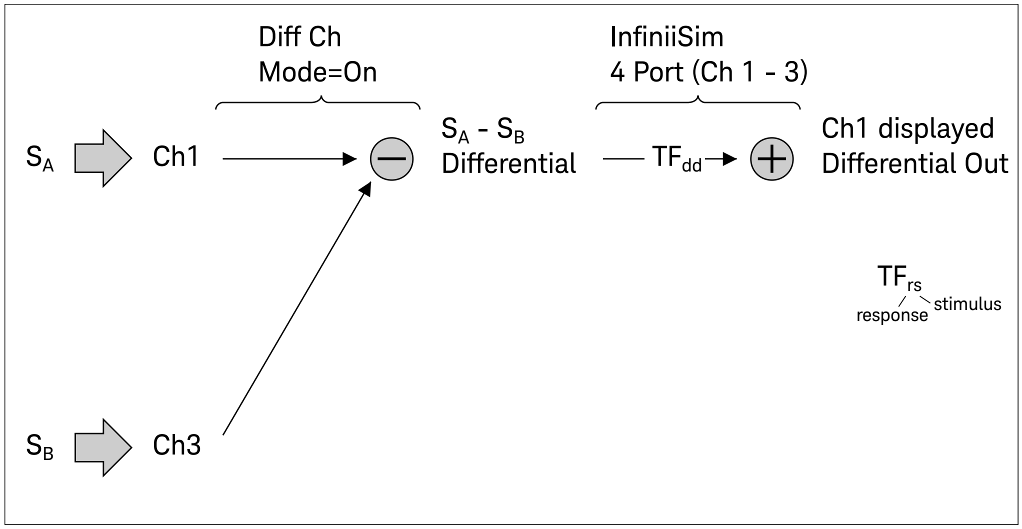
Diff Ch Mode=On, 4 Port (Channels 1 & 3)



NOTE

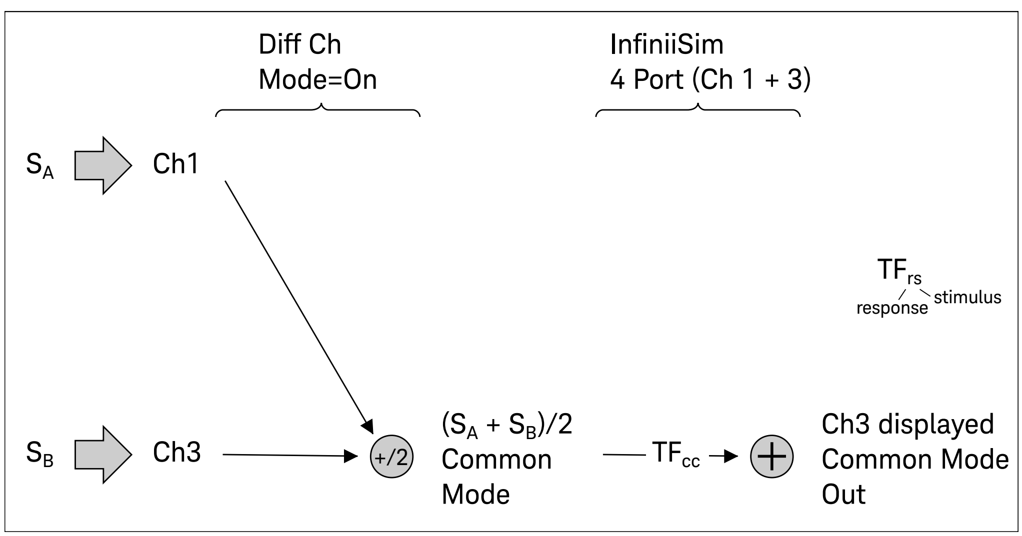
InfiniiSim always saves transfer function files in the standard form (TF_{21}, TF_{23}, TF_{41}, TF_{43}) and converts them internally to mixed-mode form (TF_{dd}, TF_{dc}, TF_{cd}, TF_{cc}) when differential channels mode is turned on or when a differential or common mode extraction is selected for the 4-port, one channel InfiniiSim mode.

Diff Ch Mode=On, Port (Channels 1 - 3), Differential Port Extraction



NOTE InfiniiSim always saves transfer function files in the standard form (TF_{21}, TF_{23}, TF_{41}, TF_{43}) and converts them internally to mixed-mode form (TF_{dd}, TF_{dc}, TF_{cd}, TF_{cc}) when differential channels mode is turned on or when a differential or common mode extraction is selected for the 4-port, one channel InfiniiSim mode.

Diff Ch Mode=On, 4 Port (Channels 1 + 3), Common Mode Port Extraction



NOTE

InfiniiSim always saves transfer function files in the standard form (TF_{21}, TF_{23}, TF_{41}, TF_{43}) and converts them internally to mixed-mode form (TF_{dd}, TF_{dc}, TF_{cd}, TF_{cc}) when differential channels mode is turned on or when a differential or common mode extraction is selected for the 4-port, one channel InfiniiSim mode.

Transfer Function Files

Transfer function files describe the complex magnitude and phase relationship between two nodes as a tabular list of values at specific frequencies.

- "Transfer Function File Format" on page 146
- "Transfer Function Values at DC (0 Hz)" on page 146
- "Uniformly Spaced Values and Good Frequency Resolution" on page 147
- "Values with Increasing Frequencies, Limited Number of Frequency Points" on page 147
- "Transfer Function File Keywords" on page 147

It is important to realize that transfer function files and S-parameter files are not the same thing. S-parameter files represent complete electrical models of physical elements and can be used to define specific model blocks within the circuit models. Transfer function files represent the relationship between measured and simulated voltage waveforms.

Transfer Function File Format

InfiniiSim supports a single proprietary transfer function file format:

- ATF (version A.01.00) derived from the Touchstone 1.1 format.

  ATF files are identified by the file extensions .tf2 or .tf4 (for 2-port and 4-port network models).

Formatted as ASCII text, transfer function files are easily viewed or edited using any text-based file editor.

NOTE

InfiniiSim always saves transfer function files in the standard form (TF_{21}, TF_{23}, TF_{41}, TF_{43}) and converts them internally to mixed-mode form (TF_{dd}, TF_{dc}, TF_{cd}, TF_{cc}) when differential channels mode is turned on or when a differential or common mode extraction is selected for the 4-port, one channel InfiniiSim mode.

See Also

- "Transfer Function File Keywords" on page 147

Transfer Function Values at DC (0 Hz)

All transfer function files must include values at DC (0 Hz).

Uniformly Spaced Values and Good Frequency Resolution

InfiniiSim requires transfer function values that are uniformly spaced in frequency with a spacing or resolution that is small enough to accurately represent the slowest time constants and time delays exhibited by the modeled device.

InfiniiSim checks the data to ensure it is uniformly sampled in frequency (except for the interval between DC and the next lowest frequency point) and generates a warning if the data is not uniformly sampled.

If a transfer function file does not contain uniformly spaced frequency data, you can insert the special InfiniiSim keyword DEFAULT_FREQUENCY_RESOLUTION into the transfer function file to cause the data to be resampled. See "DEFAULT_FREQUENCY_RESOLUTION" on page 148.

Values with Increasing Frequencies, Limited Number of Frequency Points

InfiniiSim expects transfer function files to list their values in monotonically increasing frequency order and ignores those values at which frequencies are less than or equal to the frequencies of previously listed values.

InfiniiSim reads up to the first 250 k frequency points of a transfer function file, but it needs to truncate and/or re-sample data down to 100 kpts or less for the simulation engine. InfiniiSim can simply use the first 100 kpts of the transfer function file, or, if you add the DEFAULT_FREQUENCY_RESOLUTION keyword to the transfer function file header, InfiniiSim will read the first 250 kpts and down-sample the data before it truncates it. See "DEFAULT_FREQUENCY_RESOLUTION" on page 148.

InfiniiSim does not re-sample unless the frequency points in the file are not uniformly spaced or unless a different frequency resolution is specified using the DEFAULT_FREQUENCY_RESOLUTION keyword.

It is always best to use transfer function files whose frequency points are uniformly sampled, contain a DC point, and have less than or equal to 100 kpts. In this case, InfiniiSim uses the file data exactly as provided and does not have to truncate or re-sample.

Transfer Function File Keywords

The following keywords are the currently supported optional keywords you can place in your transfer function files. InfiniiSim scans transfer function files for special keywords that communicate to InfiniiSim how the data within the file was generated or how it should be interpreted. InfiniiSim keywords are case insensitive and permit additional comments on the same line.

- "ATF_FILE_VERSION" on page 148
- "DEFAULT_FREQUENCY_RESOLUTION" on page 148
- "TRANSFER_FUNCTION_DEFINITION_STRING" on page 148

ATF_FILE_VERSION `! #DSO ATF_FILE_VERSION <string>`

The value of <string> represents the format version of the ATF file.

DEFAULT_FREQUENCY_RESOLUTION

```
! #DSO DEFAULT_FREQUENCY_RESOLUTION <value>
```

where <value> may be a numeric value in Hz or the text strings AUTO or AUTOMATIC.

If the DEFAULT_FREQUENCY_RESOLUTION keyword is used and its <value> is numeric, the data is resampled using linear interpolation and the frequency resolution specified by <value>.

If the keyword is used and its <value> is either of the text strings "AUTO" or "AUTOMATIC", the data is resampled using linear interpolation and a frequency resolution equal to the frequency interval between the last two frequency points in the file. No warning is generated even if the file data is non-uniformly sampled.

In the absence of the keyword or if <value> is an unrecognized text string, InfiniiSim checks the data to ensure it is uniformly sampled in frequency (except the interval between DC and then next lowest frequency point). If the data is not uniformly sampled, InfiniiSim resamples the data using a frequency resolution equal to the frequency interval between the last two frequency points in the file and generates a warning.

TRANSFER_FUNCTION_DEFINITION_STRING

```
! #DSO TRANSFER_FUNCTION_DEFINITION_STRING <string>
```

InfiniiSim uses this keyword to include a complete description of the circuit models from which it calculates the transfer function file. When loading transfer function file data, InfiniiSim checks for this keyword and if present, uses this information to re-populate the circuit model descriptions.

InfiniiSim FIR Filters

Once a transfer function is generated, InfiniiSim performs an inverse Fourier transform on it to yield an impulse response. The impulse response is sampled at the sampling rate of the oscilloscope to yield an FIR correction filter that is convolved with acquired waveform to render the simulated waveform.

Because the FIR correction filter is created from the transfer function, it could apply to the full bandwidth and frequency resolution represented by the transfer function. However, the frequency content of the signal used may not require the full bandwidth, and you can therefore limit the filter bandwidth to reduce noise.

The resolution of the transfer function determines the length of the FIR correction filter (that is, the number of coefficients, or points, or taps). Filter length in turn affects the waveform processing time. You can shorten the filter length to shorten processing time, but you must understand how this affects the frequency response of the system.

- "FIR Filter Bandwidth" on page 149
- "Understanding FIR Filter Length" on page 151

FIR Filter Bandwidth

The time domain FIR correction filter, h(t), is derived from the frequency domain transfer function H(f). This filter is convolved with the measured waveform to yield the co-simulated result desired. Because the correction filter is created from the transfer function, it could apply to the full bandwidth and frequency resolution represented by the transfer function, but that may not be best choice for a particular application. Therefore, it becomes prudent to analyze the characteristics of this filter and optimize it to meet specific needs.

Consider the graphical representation of the transfer function in the following figure. It was calculated to de-embed cable loss up to 9 GHz. Note that the frequency content of the measured signal (green trace) extends only to about 5 GHz. Above that frequency is just noise. This is an example of why analyzing the characteristics of this filter becomes prudent. If the full 9 GHz of correction of this measurement is applied, it will simply increase, or boost the noise above 5 GHz without any appreciable benefit to the measured signal.

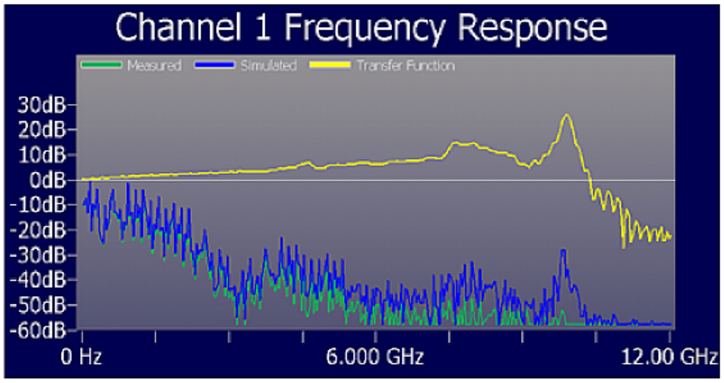


Figure 16 Correction transfer function for de-embedding a lossy cable (yellow), frequency spectrum of measured signal (green), and frequency spectrum of simulated signal (blue)

Filter Bandwidth Effects on the Eye Diagram

Notice how limiting the bandwidth of the correction filter in the previous figure improves the eye diagram measurements (see the figure that follows). The top eye used no de-embedding. The middle eye portrays de-embedded the cable loss, reducing its inter-symbol interference (ISI), but also significantly increasing the noise. The bottom eye portrays de-embedded the cable loss only up to 5.5 GHz, also reducing its ISI, but without increasing its noise.

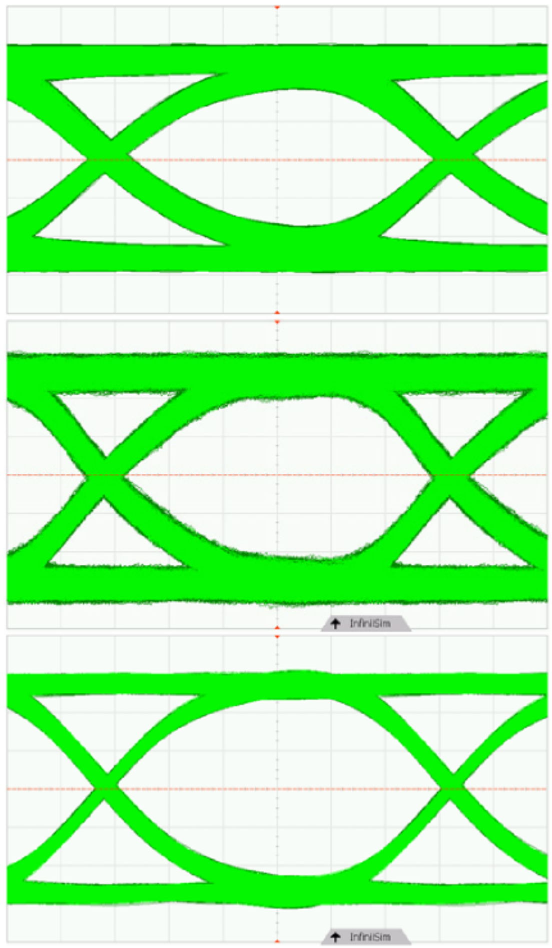


Figure 17 Eye diagram measurements using; no de-embedding, de-embedding to 9 GHz, and de-embedding

See Also

- "Using the Bandwidth Limit Control" on page 158
- "Frequency Response Plot" on page 80

Understanding FIR Filter Length

The length (number of coefficients, or points, or taps) of the FIR correction filter determines its ability to apply long-time constant or slowly varying characteristics. In other words, longer filters correspond to transfer functions with finer frequency resolution. However, the filter's length also affects waveform processing time. Longer filters slow down oscilloscope display and measurement update rate. So, there is a tradeoff in choosing the filter length.

Refer back to the filter shown in "Transfer Function Derivation" on page 135. It was chosen to have a length of ~7 ns (280 points). Now observe the filter shown in the following figure. This new filter was designed to apply the same transfer function, but it is only ~2.5 ns long (100 points). Notice that this shorter version is not long enough to represent all of the bumps and wiggles in the target transfer function (see the frequency response figure that follows).

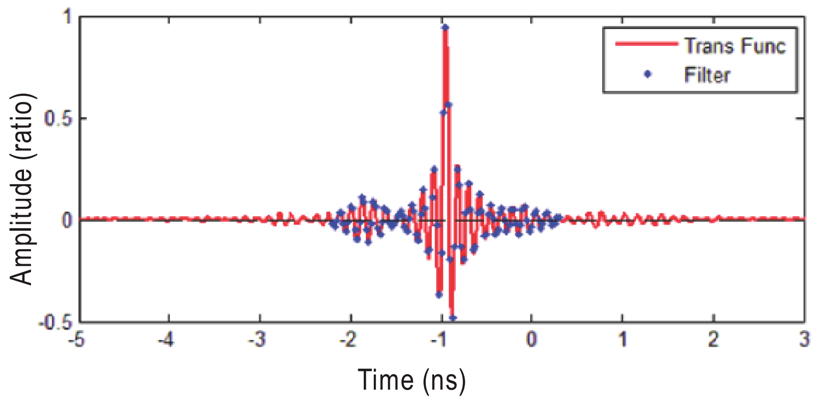


Figure 18 Impulse responses of filter (blue) and de-embedding transfer function (red) from lossy cable example

Although the filter's length can often be optimized solely by examining the filter itself, it is usually better to view the filter's step response. That is because a step response visually emphasizes the lower frequency components of the filter more than an impulse response does (see the following figure).

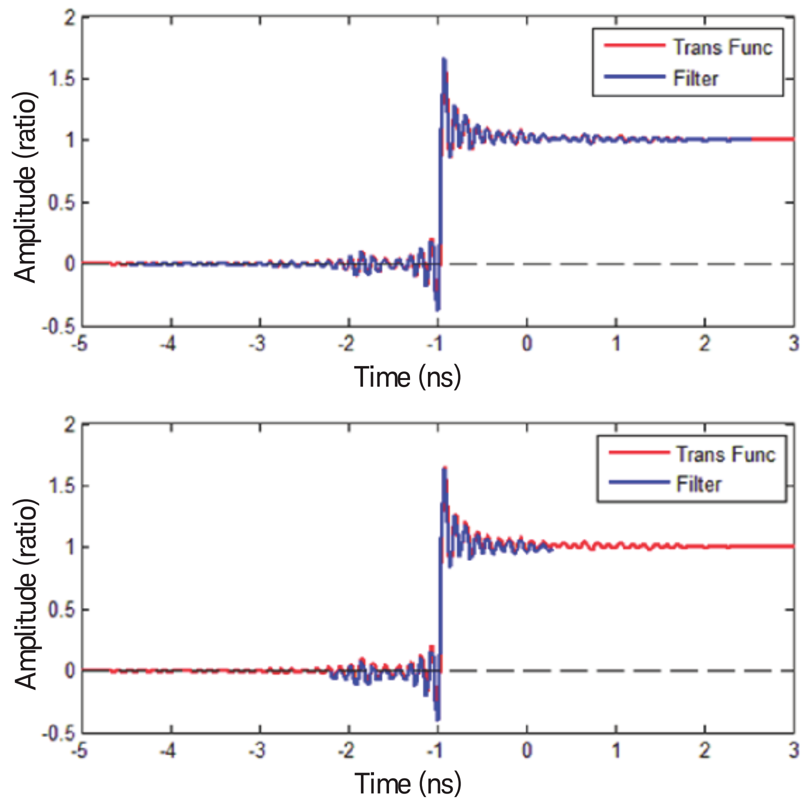


Figure 19 Step responses of filters (blue) and de-embedding transfer functions (red) from the lossy cable example using two different length filters

The step responses for both filters shown in the previous figure are nearly identical over the reduced filter length. Shortening the transfer function, however, does alter the frequency response of the transfer function and serves to smooth the filter's effective frequency response (see the following figure). Shortening the filter to produce a smoothed filter frequency response can be appropriate, but make sure that it is appropriate for your application. In particular, the transfer function should be evaluated for quickly changing phase or magnitude responses.

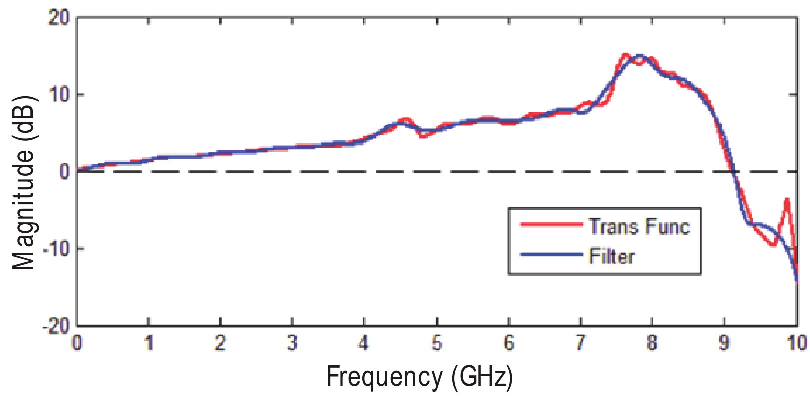


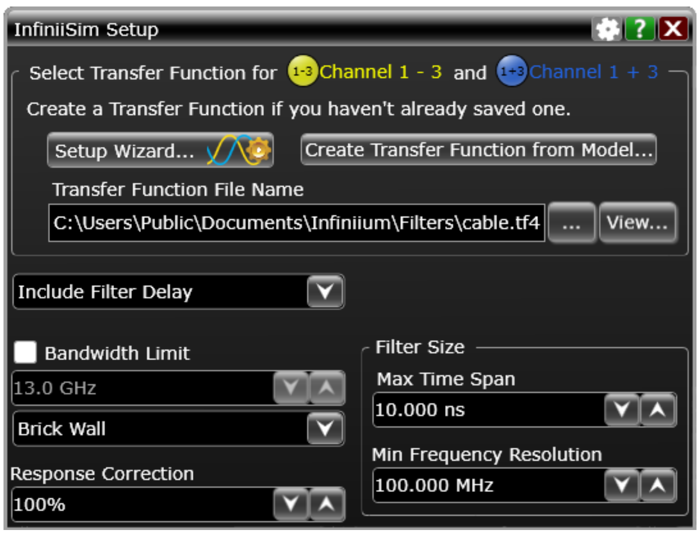
Figure 20 Frequency response of filter from the lossy cable example with reduced length

InfiniiSim Application Notes

- Transforming Oscilloscope Acquisitions for De-Embedding, Embedding and Simulating Channel Effects
- S-Parameter Requirements for Oscilloscope De-Embedding Applications
- S-Parameter Measurements Basics for High Speed Digital Engineers (using a VNA)
- Using the Time-Domain Reflectometer (to measure S-parameters)
- Using De-embedding Tools for Virtual Probing (using simulation tools)
- Practical Application of the InfiniiSim Waveform Transformation Toolset
- The ABCs of De-Embedding
- De-embedding and Embedding S-Parameter Networks
- De-embedding Techniques in Advanced Design Systems
- S-Parameter Design

InfiniiSim Dialog Boxes

InfiniiSim Setup Dialog Box



- **Create a Transfer Function if you haven't already saved one.** — If you do not have a previously saved transfer function to load, you can create a transfer function from a model (click **Create Transfer Function from Model...**). For information on creating a transfer function from a model, see "InfiniiSim Model Setup Dialog Box" on page 160.
- **Transfer Function File Name** — Lets you select a transfer function file to apply. If you have a previously saved transfer function or one you generated elsewhere, you can click this field and load it. For an overview of transfer function files, see "Transfer Function Files" on page 146. The **View...** button opens the InfiniiSimToolbox application for viewing and evaluating transfer function files. See "S-Parameter and Transfer Function File Viewers" on page 127.

  Also in the **Transfer Function File Name** area is a control for including or removing delay. Cables and fixtures delay the signal in time in addition to changing the waveform's shape. This can make it difficult to compare the simulated and measured waveforms because as you turn InfiniiSim on and off, the waveforms are not aligned in time. This may make one or the other shift entirely off screen, for example.

- **Include Filter Delay** — Apply the delay imposed by the model to the waveform. This selection will accurately shift the waveform in time according to the model.
- **Remove Filter Delay** — Remove the delay imposed by the model on the waveform. Because different frequencies can have different delays, this is an approximation of the group delay. Use this mode to compare the response correction of the signal with InfiniiSim turned on and off.
- **Include Trigger Corrected Delay** — This selection includes filter delay. If the InfiniiSim channel is the trigger source, this selection also applies the trigger's jitter error correction after the de-embedding filter has been applied.

The **Normalize Gain** option removes any DC gain of the InfiniiSim transfer function and can be used when modeling probes.

You may have to apply a different transfer function before the remaining controls become visible.

- **Bandwidth Limit** — This is the same control located in the Bandwidth Limit dialog box (accessed by clicking **Bandwidth Limit...** in the Channel dialog box). It is duplicated here for convenience.

  Bandwidth limiting is independent of InfiniiSim waveform transformations and can be applied to measured or simulated waveforms. It is implemented using a FIR (finite impulse response) filter that has a size (time span) determined by the filter size controls located inside the InfiniiSim Setup dialog box. For more information, see: "Using the Bandwidth Limit Control" on page 158.

- **Response Correction** — This field linearly scales the amount of correction applied to the non-DC frequency components of the measured signal. This lets you trade off the amount of correction to apply through the transformation function versus the increase in noise it may create at higher frequencies. In other words, you can fine-tune the amount of high-frequency noise versus the sharpness of the step response edge.

  If you are making averaged mode measurements or applying a transfer function that does not magnify the noise, use the full correction by setting this field to 100%. However, if you are working with eye diagrams or making jitter measurements and the transfer function is magnifying the noise, you may want to limit the correction by selecting a lower percentage.

  Once you have your InfiniiSim-transformed signal displayed on screen, you can adjust this value in the dialog box to see its effect on the signal.

  It is also useful to display the frequency response plots and step response plot while you adjust this field to see its effect on these plots. For instance, you can see how the frequency response of the transformation filter changes as you change the percentage. See "Frequency Response Plot" on page 80 and "Step Response Plot" on page 83.

- **Filter Size** — This area has two controls: **Max Time Span** and **Min Frequency Resolution**. These two controls are reciprocals of each other and tied together, so as you set one to a lower value, the other one raises to a higher value and vice versa. InfiniiSim applies the correction transfer function by means of a FIR digital filter. The size of this filter determines the time span of the transformation it can apply. However, there is a trade-off. Using a longer filter corrects longer time constants, but it can take longer to calculate. By default, InfiniiSim tries to choose a filter size that preserves the coarsest frequency resolution used by all of the model blocks. However, you can limit the filter size by reducing the **Max Time Span** value.

  A quick summary of the two controls is:

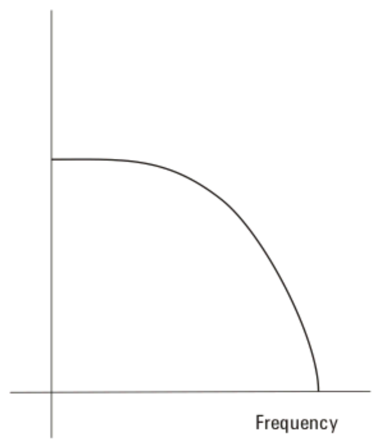
  - Increasing the **Max Time Span** control increases the maximum time span of the correction transfer function's impulse response. Increasing this control enables InfiniiSim to apply transformations that have slower time constants. You can use either the Step Response or Impulse Response plot (see page 79) to see if the time span is long enough to model your device adequately.
  - Increasing the **Min Frequency Resolution** control makes the frequency resolution less fine and automatically decreases the **Max Time Span** control (which corresponds to a higher update rate). Therefore, you would do this if you wanted a higher update rate with the trade-off of a shorter transfer function impulse response. Generally, you want to set the **Max Time Span** using the Step Response or Impulse Response plot (see page 79) and then this **Min Frequency Resolution** is set automatically.

Using the Bandwidth Limit Control

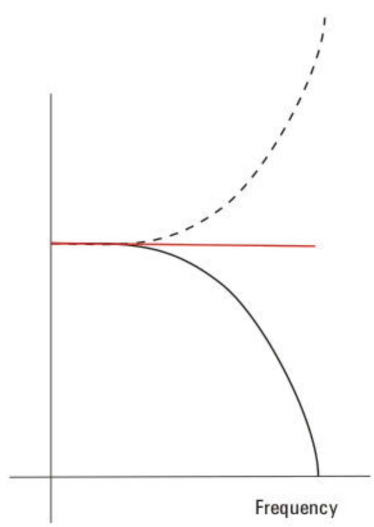
The **Bandwidth Limit** control requires some explanation so you know how to properly set the value and whether or not it is required for your specific application.

One way to understand the transformations InfiniiSim performs is in terms of the frequency response of the cable, fixture, or probe your signal travels through. Ideally, you would like to transmit a signal through a channel whose frequency response was flat over the full bandwidth of the signal so every frequency component of the signal was equally represented.

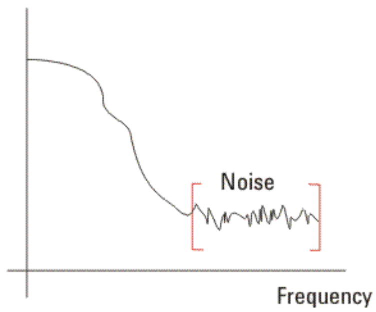
In practice, your frequency response plot may look something like the plot below where the signal attenuates at higher frequencies (this is a very simplified example).



Some InfiniiSim applications try to remove or correct this insertion loss in the channel. In those cases, the correction transfer function becomes the inverse of the channel's insertion loss (dashed line in following plot) such that the product of the "peaked" correction transfer function response and the channel response yields a flat frequency spectrum (red line in plot).



In the previous example, the response plot before InfiniiSim was applied was very simplistic. A more realistic frequency response plot might look more like this:



Notice that above a certain frequency (or bandwidth) there is very little signal and mostly noise. If you apply a correction through InfiniiSim as discussed above, you may end up excessively amplifying the noise. Setting the bandwidth in the Bandwidth Limit dialog box lets you minimize this noise gain by attenuating the resulting waveforms above the frequency where there is mostly just noise.

Examining the Frequency Response Plot helps you see what bandwidth you should set this field to (see "Frequency Response Plot" on page 80).

Normally you do not need to set a bandwidth limit if you are adding loss (such as applying the loss of a cable). However, if you are applying gain (such as removing the loss of a cable), you may need to use the Bandwidth Limit.

See Also

- "FIR Filter Bandwidth" on page 149

InfiniiSim Model Setup Dialog Box

If you are using 2-port models, see:

- "Creating a Transfer Function From a Model (2 Port)" on page 25

If you are using 4-port models, see:

- "Creating a Transfer Function From a Model (4 Port)" on page 51

Circuit Source and Load Impedances Dialog Box

If you are using 2-port models, see:

- "Circuit Source and Load Impedances (2 Port)" on page 36

If you are using 4-port models, see:

- "Circuit Source and Load Impedances (4 Port)" on page 62

InfiniiSim Block Setup Dialog Box

If you are using 2-port models, see:

- "Defining a Block (2 Port)" on page 37

If you are using 4-port models, see:

- "Defining a Block (4 Port)" on page 63

InfiniiSim Sub-circuit Block Setup Dialog Box

If you are using 2-port models, see:

- "Combination of Sub-Circuits Block Type (2 Port)" on page 43

If you are using 4-port models, see:

- "Combination of Sub-Circuits Block Type (4 Port)" on page 70

InfiniiSim Node Setup Dialog Box

If you are using 2-port models, see:

- "Measurement and Simulation Node Locations (2 Port)" on page 49

If you are using 4-port models, see:

- "Measurement and Simulation Node Locations (4 Port)" on page 76

Index

G

I

K

L

M

N

O

P

R

S

T

U

V

W

Y