Agilent N5465A InfiniiSim Waveform Transformation Toolset Software

User's Guide

Agilent Technologies

Notices

Manual Part Number

Version 04.60.0000

Edition

September 2013

Available in electronic format only

Agilent Technologies, Inc.
1900 Garden of the Gods Road
Colorado Springs, CO 80907 USA

Warranty

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

Technology Licenses

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.

Restricted Rights Legend

If software is for use in the performance of a U.S. Government prime contract or subcontract, Software is delivered and licensed as "Commercial computer software" as defined in DFAR 252.227-7014 (June 1995), or as a "commercial item" as defined in FAR 2.101(a) or as "Restricted computer software" as defined in FAR 52.227-19 (June 1987) or any equivalent agency regulation or contract clause. Use, duplication or disclosure of Software is subject to Agilent Technologies' standard commercial license terms, and non-DOD Departments and Agencies of the U.S. Government will receive no greater than Restricted Rights as defined in FAR 52.227-19(c)(1-2) (June 1987). U.S. Government users will receive no greater than Limited Rights as defined in FAR 52.227-14 (June 1987) or DFAR 252.227-7015 (b)(2) (November 1995), 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.

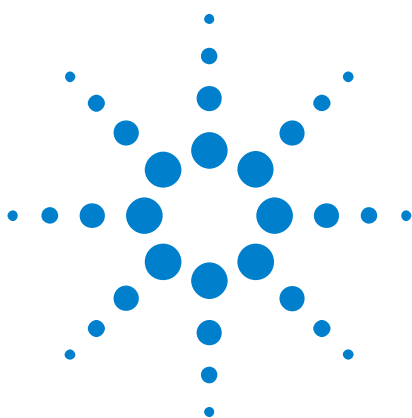**Agilent N5465A InfiniiSim Waveform Transformation Toolset Software User's Guide**   **3**

Contents

1

InfiniiSim Waveform Transformation Toolset

| NOTE | You must have either the InfiniiSim Basic or InfiniiSim Advanced application installed on your oscilloscope to have access to these features. |

The Basic version of InfiniiSim allows you to use only one block in your measurement/simulation circuits (building these circuits is discussed in other InfiniiSim topics) while the Advanced version contains all of the InfiniiSim capabilities.

Application Notes

- *"The ABCs of De-Embedding"*
- *"De-embedding and Embedding S-Parameter Networks"*
- *"De-embedding Techniques in Advanced Design Systems"*

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

InfiniiSim Overview / Theory

The InfiniiSim Waveform Transformation Toolset provides a flexible and accurate way to render waveforms anywhere in a digital serial data link. InfiniiSim transforms waveforms you actually measured into waveforms you wanted to measure, but were unable to for some reason. The highly configurable system modeling in InfiniiSim lets you:

- Remove the deleterious effects of unwanted channel elements.
- Simulate waveforms with channels models inserted.
- View waveforms at physically unprobeable locations.
- Compensate for the loading of probes and other circuit elements.

Next:

For Example: Removing a Cable's or Fixture's Insertion Loss

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.

One thing InfiniiSim lets you do is extract the DUT's signal from the measured signal. For example, look at the following representation of a measurement.
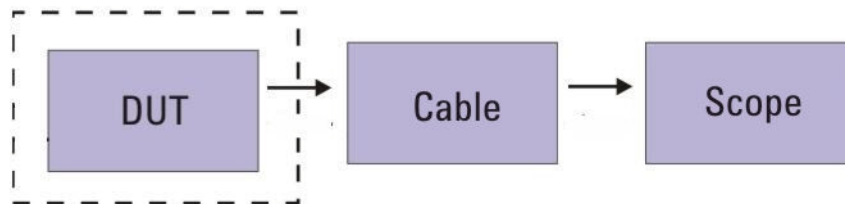
The node that is indicated by the dotted line is the location you want to measure, but the cable distorts this measurement. InfiniiSim can extract the undistorted DUT signal from the distorted measured signal.

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 (30 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.

Next: "InfiniiSim Features" on page 9.

InfiniiSim Features

InfiniiSim can perform the waveform transformation of inserting a cable or fixture (see "For Example: Removing a Cable's or Fixture's Insertion Loss" on page 8), but it can do much more. InfiniiSim is a general transformation tool that can be used to produce more accurate results for a variety of problems. InfiniiSim provides default configurations (called "application presets") for many common use models as well as general purpose configurations for the advanced user. For example, you can:

- **Remove a channel element insertion loss** — Modeled with one block of loss (blocks are described in detail later) such as from a cable or fixture between a digital source and the oscilloscope. The inverse gain of the block (S_{21}^{-1}) is determined, and its time response is convolved with the acquired signal.

- **Insert a channel element insertion loss** — Modeled with one block of loss, such as from a standard cable model, that is inserted before the oscilloscope. The gain of the block (S_{21}) is determined, and its time response is convolved with the acquired signal.

- **Remove scope input reflection** — Modeled using two blocks — one for the transmitter source and the other for the oscilloscope input (so the oscilloscope input reflection can be removed).

- **Remove all effects of a channel element** — By using S-parameter models of source and oscilloscope load, the effects of a channel element are totally removed. This is different from "removing insertion loss" in that the interactions between the elements are taken into account. This provides the most accurate way to remove a channel element.

- **Add all effects of a channel element** — By using S-parameter models of source and oscilloscope load, a complete insertion of a channel element is performed. This is different from "inserting a channel element insertion loss" in that the reflective interactions between the elements are taken into account. This provides the most accurate rendering to insert a channel element.

- **Measurement node relocation** — Measurement 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 "insertion" viewpoint.

- **Remove loading effects of an oscilloscope 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.

- **General purpose configuration** — General-purpose topologies are provided for greater flexibility. General purpose probe, 6 block, and 9 block topologies are available with user defined measurement and simulation points. Additionally, each block 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.
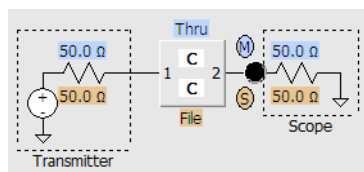
Measurement Circuit / Simulation Circuit Concept

InfiniiSim accomplishes its tasks (see "InfiniiSim Features" on page 9) through the use of correction transfer functions. When using InfiniiSim, you define two electrical circuits.

- One circuit, the **measurement circuit**, models the actual physical electrical circuit that produced the measured waveform.

- The other circuit, the **simulation circuit**, models a hypothetical electrical circuit that exhibits the electrical characteristics that you wished to have measured.

You can build these two circuits 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 as discussed later in other InfiniiSim topics.)

The values highlighted in blue define the measurement circuit while the values in orange define the simulation circuit. In the above 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, etc.). When using more complex preset configurations, you can move the measurement and simulation nodes to a variety of locations within the model.

InfiniiSim analyzes these the measurement and simulation circuits and generates a correction transfer function which, when applied to the measured waveform, produces the desired simulated waveform on the oscilloscope's display.

The convolution process used by InfiniSim requires that the measurement circuit and the simulation circuit be linear and time-invariant (these are small-signal analysis requirements).

Next: "Correction Transfer Functions" on page 11.

Correction Transfer Functions

You now have a basic idea of what InfiniiSim can do, and you know that it generates transfer functions from the two circuits you define in the application (see "Measurement Circuit / Simulation Circuit Concept" on page 10), but how exactly does it do this, and what is a transfer function?

A correction transfer function represents the ratio (in the frequency domain) between voltage waveforms defined by two different conditions — one defined by the *measurement circuit*, and the other defined by the *simulation circuit*. For any given application, the correction transfer function can be calculated from circuit models you create or loaded directly from a transfer function file.

If you create the transfer function through circuit models, you first create the measurement and simulation circuits in the InfiniiSim GUI. The response of the two circuits due to a stimulus from the same voltage source is calculated at their specified observation nodes. The transfer function is then calculated from the ratio (in the frequency domain) of the simulation node's response divided by the measurement node's response. This ratio can then be convolved (in the time domain) with the measured waveform in order to convert it into the desired simulated waveform.

Note that in 4-port mode, there are actually two sources and observation nodes in each circuit. In this case, the ratio of node voltages becomes a matrix division operation generating a set of four frequency domain functions that are convolved with the two measured waveforms to produce the desired simulated waveforms.

It is also 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. For more information, see "S-Parameter Files" on page 85 and "Transfer Function Files" on page 87.

InfiniiSim can be used on differential signals as well as single-ended signals.

• In single-ended applications, the measurement and simulation circuits are constructed from 2-port network model blocks and the analysis is performed on each oscilloscope channel independently.

• In differential applications, you must decide whether to use 2-port analysis or 4-port analysis. 2-port analysis can be used on differential signals or common-mode signals independently, but it does not include cross-coupling components between the differential and common-mode signals.

Note that you would use 2-port analysis on a differential signal if that signal is being converted to a single oscilloscope channel input by a differential probe.

Next: "Circuit Model Block Configurations" on page 12.

Circuit Model Block Configurations

There are a wide variety of configurations possible using the InfiniiSim GUI. This section describes three of the "application preset" configurations to give you a brief overview of how they are used.

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. — defining a circuit model block is discussed in more detail under the "Creating a Transfer Function From a Model" topics).

• **One Block for Simplest Path Compensation**

The configuration shown below is a basic one-block model. Many times you simply want to compensate for the loss of a channel element such as a cable or fixture. This one-block model can perform this task. It can also be extended to a remove / replace operation by changing the simulation parameters.
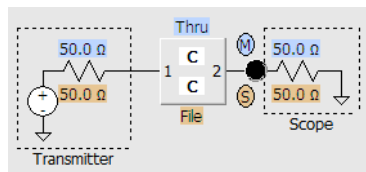
• **Three Block Analysis for True Channel Element Removal and Insertion**

When the most precision for a single channel element removal / insertion is required, you need at least the 3-block model (an example of a 3-block model is shown below). This model uses block descriptions for transmitter (**T**) and receiver (**S**) as well as the channel (**C**) 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 (**M**) to the simulated measurement (**S**).
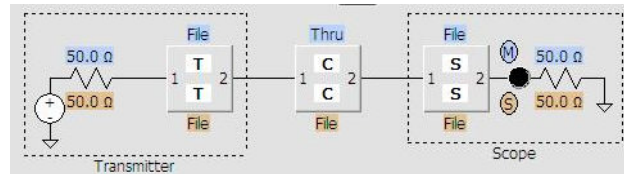
- **General Purpose Configuration**

The InfiniiSim waveform transformation toolset includes three general-purpose topologies to enable detailed tailoring of the description of your circuit:

- For probe modeling, the general-purpose probe topology is used.

- For SMA differential probe usage, the general-purpose 6-block model is used in the majority of cases.

- For those very sophisticated applications (for example, using both high impedance probes and differential SMA probe heads), the general purpose model can be used to describe these complex cases.
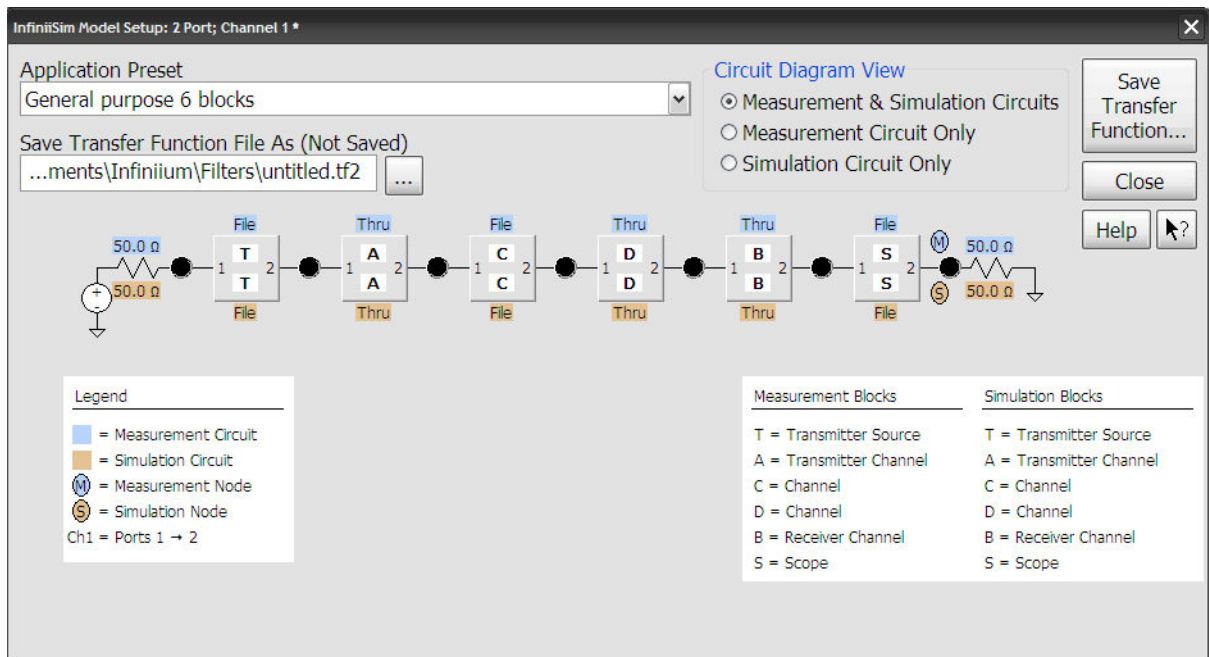
For More Information Now that you have a seen a high-level explanation of the InfiniiSim Waveform Transformation Toolset, refer to the following topics for a more detailed look at the controls as well as additional information regarding the InfiniiSim application.

- "Entering the InfiniiSim Application" on page 15
- "Creating a Transfer Function From a Model (2 Port)" on page 19
- "Creating a Transfer Function From a Model (4 Port)" on page 38
- "Generating / Saving a Transfer Function File" on page 57
- "InfiniiSim Plots" on page 58
- "InfiniiSim Warning / Error Messages" on page 78
- "InfiniiSim Examples" on page 66
- "InfiniiSim Concepts" on page 85
- "InfiniiSim Reference Information" on page 92

Entering the InfiniiSim Application

To enter the InfiniiSim application:

1 Open the Channel Setup dialog box. This dialog box can be accessed by choosing **Setup > Channel 1...** (or **Channel 2**, **Channel 3**, or **Channel 4** depending on the channel you want to use InfiniiSim on). The screen shot below shows the InfiniiSim entry controls in the Channel Setup dialog box.
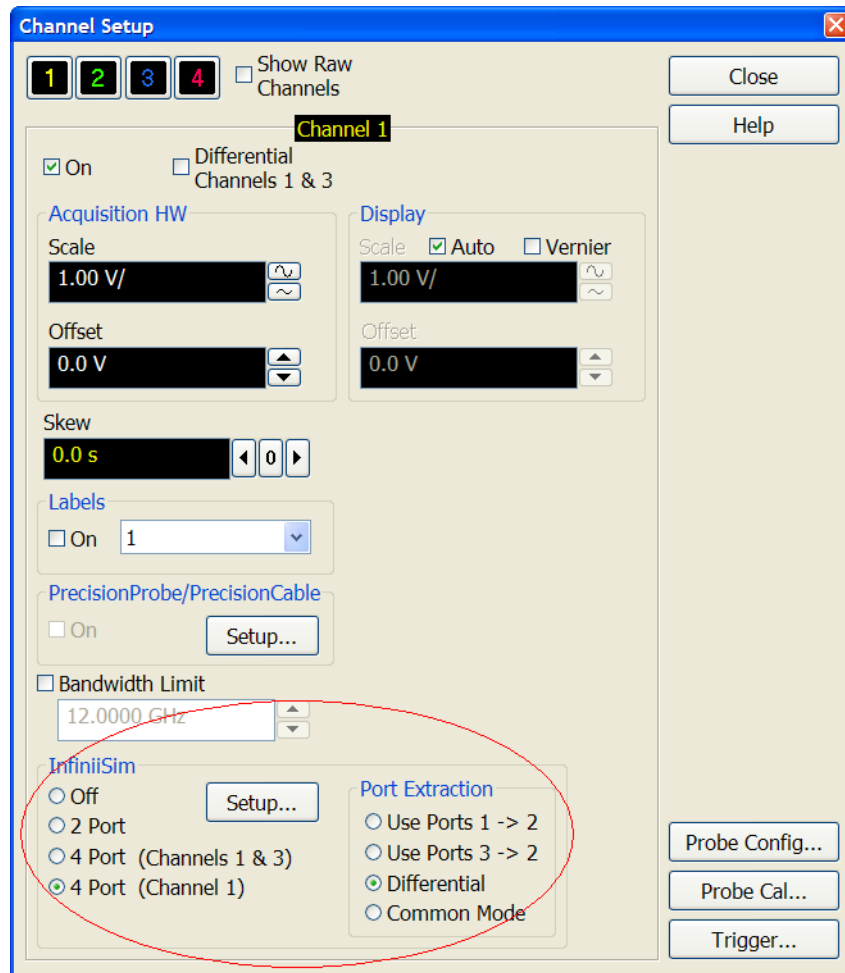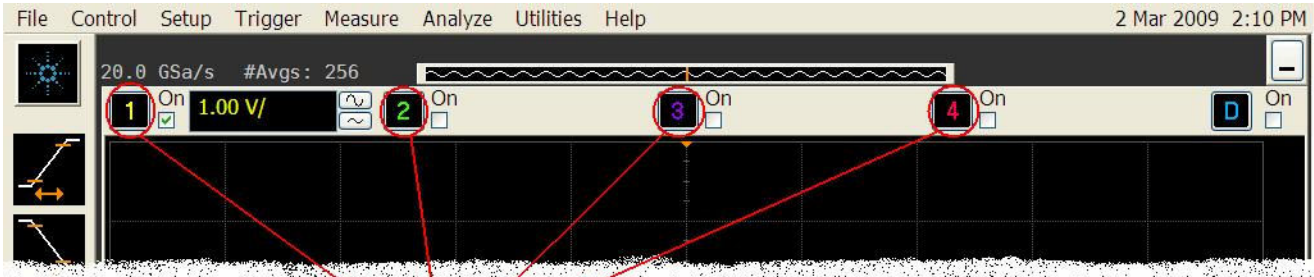

| **NOTE**	You can also use InfiniiSim on a differential or common-mode channel by checking the **Differential** box near the top of the Channel Setup dialog box before proceeding to the next step.

This dialog box can also be quickly accessed by clicking the corresponding channel button in the bar immediately above the display (see screen shot below).

Clicking on any of these buttons will open the corresponding Channel Setup dialog box.

2 After opening the Channel Setup dialog box, set the entry controls for your specific setup. You can see the InfiniiSim entry controls in the screen shot below.
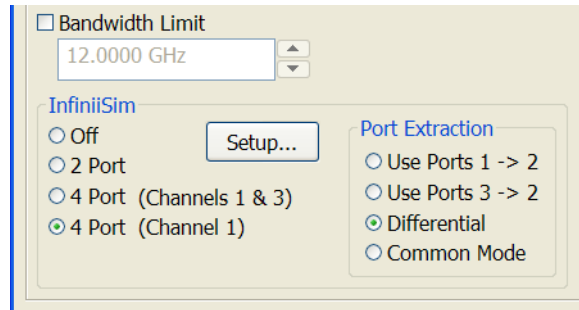
InfiniiSim is enabled by selecting either the **2 Port** or **4 Port** options within the **InfiniiSim** section of the Channel Setup dialog box. If you select 4 Port, you need to specify whether it is two channel or one channel.

Additionally, if you are using 4 Port one channel InfiniiSim, you need to use the **Port Extraction** controls to specify how to properly model your setup. This is a transfer function port extraction.

For channels 1 or 2:

- **Use Ports 1 -> 2** — you are using a single-ended path (port 1 to port 2) for the 4 port one channel analysis.

- **Use Ports 3 -> 2** — you are using a single-ended path (port 3 to port 2) for the 4 port one channel analysis.

- **Differential** — you are using the differential path through the 4 port device.
- **Common Mode** — you are using the common-mode path through the 4 port device.

For channels 3 or 4:

- **Use Ports 3 -> 4** — you are using a single-ended path (port 3 to port 4) for the 4 port one channel analysis.
- **Use Ports 1 -> 4** — you are using a single-ended path (port 1 to port 4) for the 4 port one channel analysis.
- **Differential** — you are using the differential path through the 4 port device.
- **Common Mode** — you are using the common-mode path through the 4 port device.

3 Notice the **Bandwidth Limit** checkbox and value entry field. 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 89.

4 Once you have selected the **2 Port** or **4 Port** options and set the **Bandwidth Limit** field, click **Setup...** to open the InfiniiSim Setup dialog box.
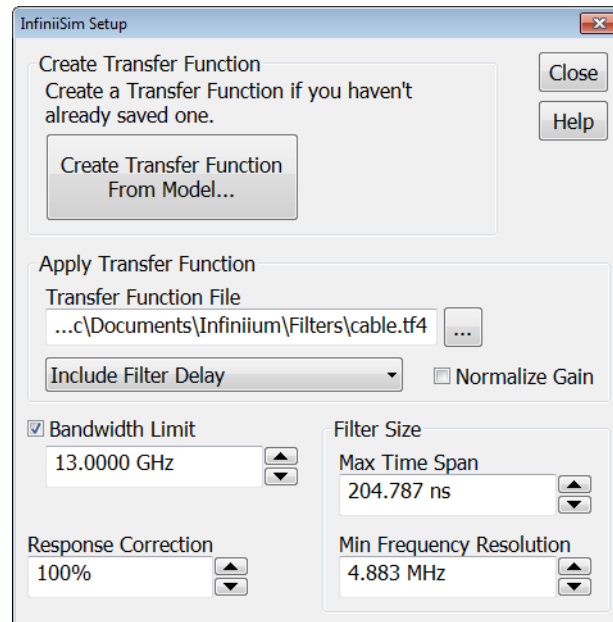
5 If you already have a transfer function created, you can apply it simply by using the **Transfer Function File** field to browse and select the file.

Then, set the other related controls (see "InfiniiSim Setup Dialog Box" on page 92).

If you need to create a transfer function, click **Create Transfer Function From Model...**. For more information, see:

- "Creating a Transfer Function From a Model (2 Port)" on page 19
- "Creating a Transfer Function From a Model (4 Port)" on page 38

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 92).
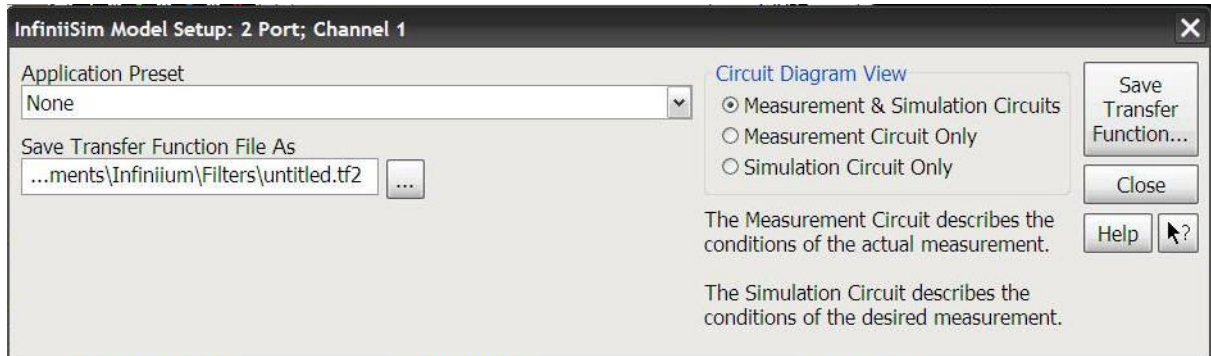
(No models are shown in the InfiniiSim Model Setup dialog box's default view above because the **Application Preset** field is set to **None**.)

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 20.

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 (2 Port)" on page 21.

4 Specify the circuit source and load impedances. See "Circuit Source and Load Impedances (2 Port)" on page 22.

5 Define measurement circuit and simulation circuit blocks. See "Defining a Block (2 Port)" on page 23.

6 Specify measurement and simulation node locations. See "Measurement and Simulation Node Locations (2 Port)" on page 36.

7 Generate and save the transfer function file.

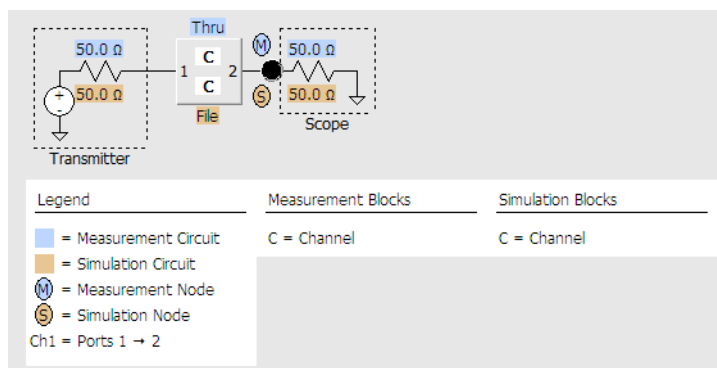| **NOTE**	The transfer function you create in this dialog box is not saved until you click the **Save Transfer Function...** button in the upper right corner. If you close the dialog box without saving, your transfer function is not applied.

See: "Generating / Saving a Transfer Function File" on page 57.

Application Presets (2 Port)

The first field in the InfiniiSim Model Setup dialog box is labeled **Application Preset**. This drop-down field gives you a variety of preset configurations you can work with to build your own model. The available preset configurations 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)
* **Relocate the observation node of a probed measurement** (Advanced)
* **General purpose probe** (Advanced)
* **General purpose 6 blocks** (Advanced)
* **General purpose 9 blocks** (Advanced)

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 there are some values highlighted blue and some highlighted orange. These correspond to the measurement and simulation circuit respectively. Let us first define what we mean by each of these circuits.

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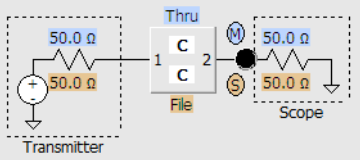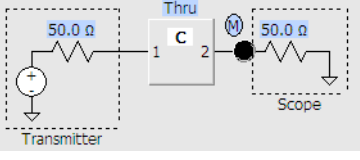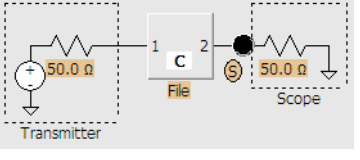
- **Simulation Circuit** – This circuit models a hypothetical electrical circuit that exhibits your desired electrical characteristics.

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.

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 Overview / Theory" on page 8 and "Transfer Function Files" on page 87.

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.
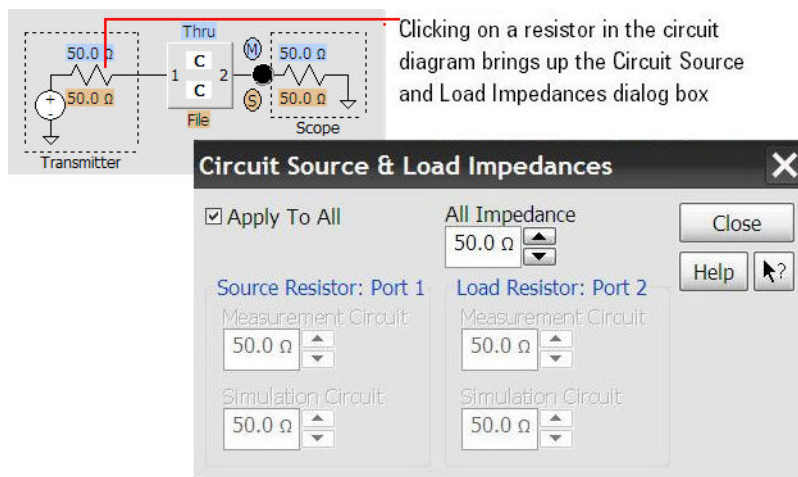
It is important to note that these circuit model representations in the GUI are simplified for ease of viewing. If we were being explicit, the model blocks in the circuit would look like the following for a 2-port device:
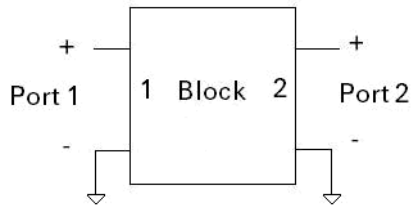
The diagrams in the GUI are completely interactive so you can click various sections or components and make adjustments.

Circuit Source and Load Impedances (2 Port)

The resistor on the left side of the circuit shown below corresponds to the impedance of the transmitter. It is defaulted to 50 Ω, but you can change this if needed. Simply place your cursor over the resistor (it thickens to indicate you are on top of it) and click. You then see the following Circuit Source and Load Impedances dialog box.
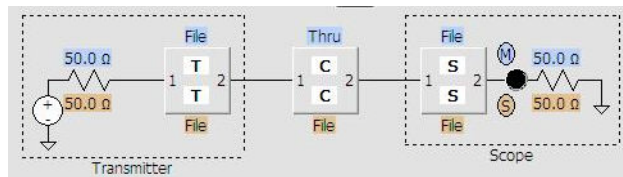

The Circuit Source and Load Impedances dialog box lets you specify all of the source and load impedances within both the measurement and simulation circuits.

Note that if we did not have the **Apply to All** box checked, the **Add Insertion Loss** application preset (used in the screen shot above) would grey-out the load impedance controls because the oscilloscope load impedance is always 50 Ω.

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

Note that by using one of the 3-block models (for example, the **Add/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.
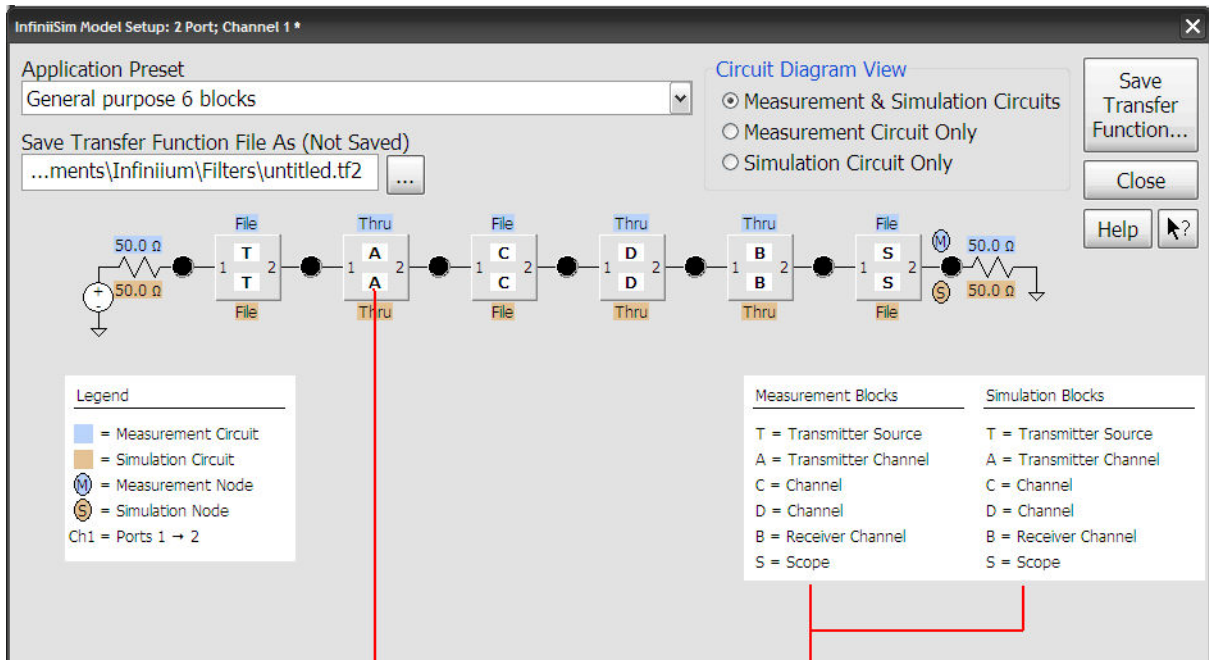

Please note that any termination block located adjacent to a source or load resistor (like the **T** block and **S** block above) can have 1 port S-parameter files assigned to them in 2 port InfiniiSim mode. These are the only locations in a 2 port InfiniiSim circuit where you can place 1 port S-parameter (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 that s1p 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 letters and names to the blocks in the legend for a typical use scenario depending upon 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 Block Setup dialog box (accessed by clicking one of the blocks).

Looking at the Blocks legend to the right tells us that this A stands for Transmitter Channel. However, you are not required to define it as a transmitter channel if you do not want. You can be flexible in your design. You can also change the name of a block by clicking on the block and using the Block Name control.

The Blocks legend shows you what the letters at the center of each block stand for. These are just suggestions, meant to make it easier to understand how the configuration could be used. You do not have to follow these definitions and can be flexible in the design of your circuits.

While blocks represent different parts of the circuit, they have the exact same definition controls to choose from and are simply called different types to help you understand what they might represent in your circuit. The termination blocks, **T** and **S**, 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.
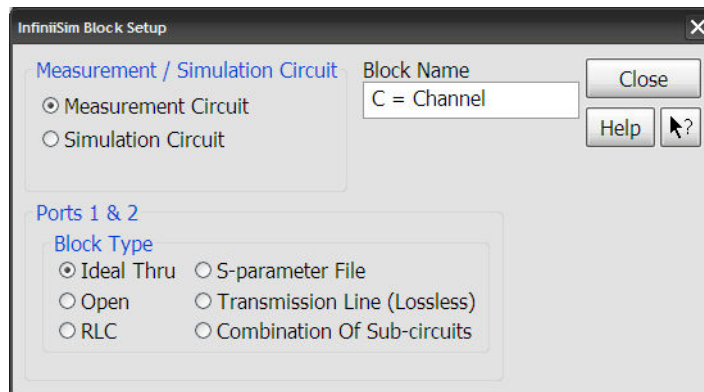
The first section (**Measurement/Simulation Circuit**), lets you specify whether you are currently defining a block in the measurement circuit or the simulation circuit. For instance, if you select **Simulation Circuit** in this section and then set the **Block Type** to **Ideal Thru**, then the corresponding 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. 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 a RLC (resistor, inductor, capacitor) network. See "RLC Block Type (2 Port)" on page 26.

- **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 27.

- **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 29.

- **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 30.

- **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 33.

- **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 35.

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 36), then generate, save, and apply the correction transfer function (see "Generating / Saving a Transfer Function File" on page 57).

RLC Block Type (2 Port)

When you select **RLC** for the block type, additional controls appear at the bottom of the dialog box.
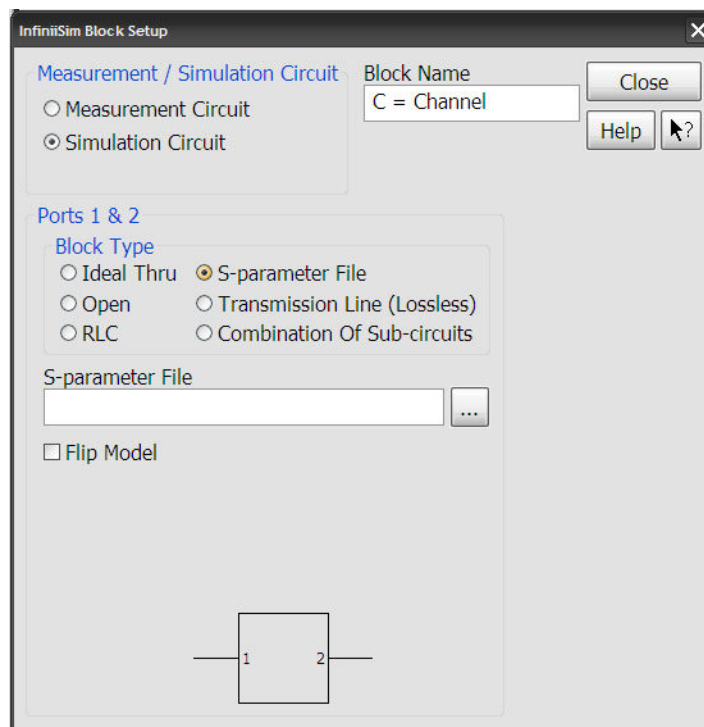
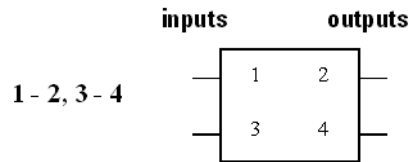In the **Circuit Element** section, there are four configuration options (**Series Thru**, **Parallel Thru**, **Series Shunt**, and **Parallel Shunt**). When you select one of these options, the basic circuit configuration is shown in the dialog box (see screen shot above).

After selecting the configuration of the RLC circuit, you can set the values for the resistor, inductor, and capacitor using the fields to the right. 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) files.

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:

inputs   outputs

1 - 2, 3 - 4   1   2
                    3   4

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

inputs   outputs

1 - 3, 2 - 4   1   3
                   2   4
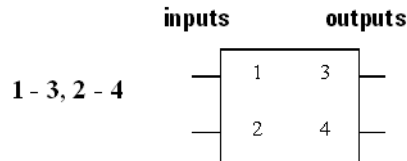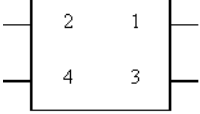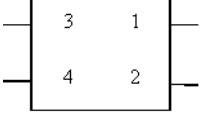
You can also use the **Flip Model** box to flip the numbers between the right side and the left side.

Flip Model	4 Port Numbering	How Ports are Connected in Block
No	1-2, 3-4	inputs outputs 1 2 3 4
No	1-3, 2-4	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 are offered the following choices:

- **Use Ports 1 -> 2** — this means you are only extracting 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 only extracting 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 additional information regarding the reading of S-parameter files, see "S-Parameter Files" on page 85.

### 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**.
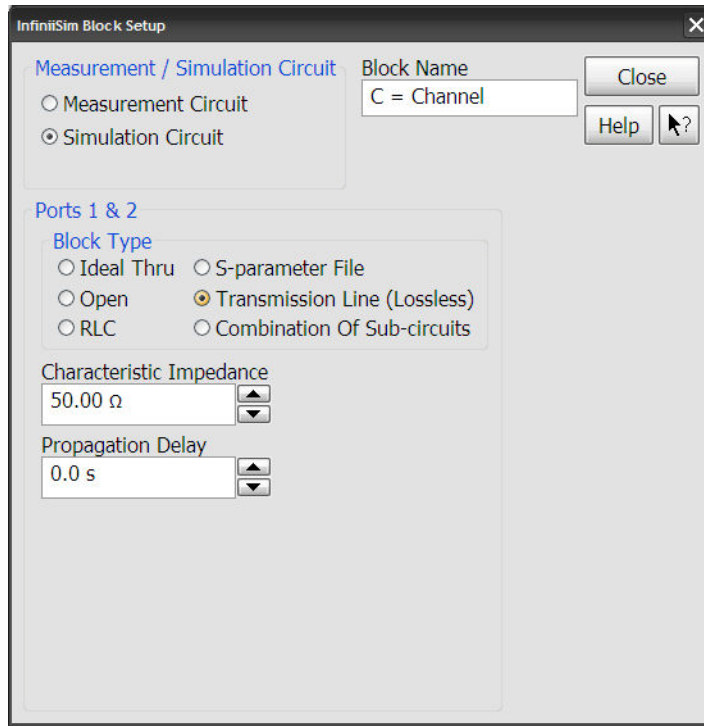
## Combination of Sub-Circuits Block Type (2 Port)

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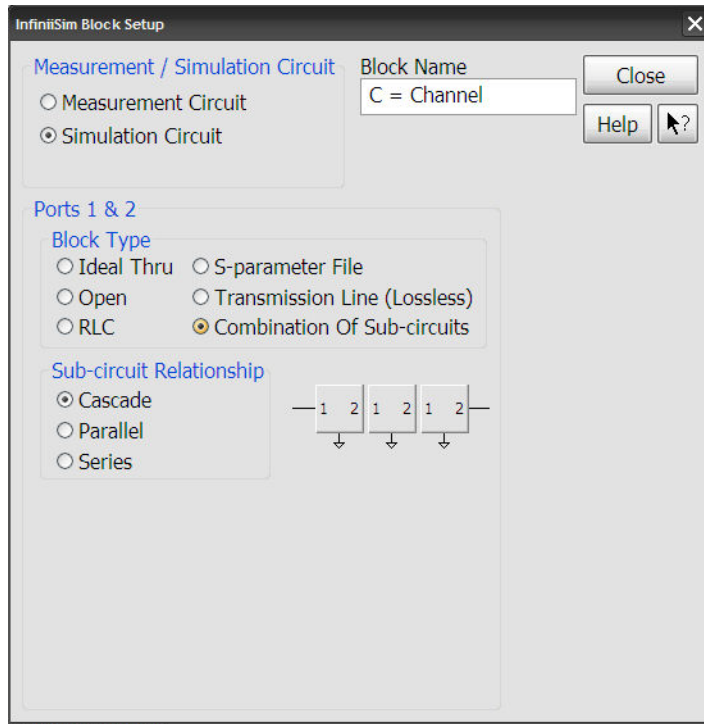
You have the following three choices for how to set up the three individual blocks that now make up the one circuit block (pictures show connections for two of the three blocks for simplicity).
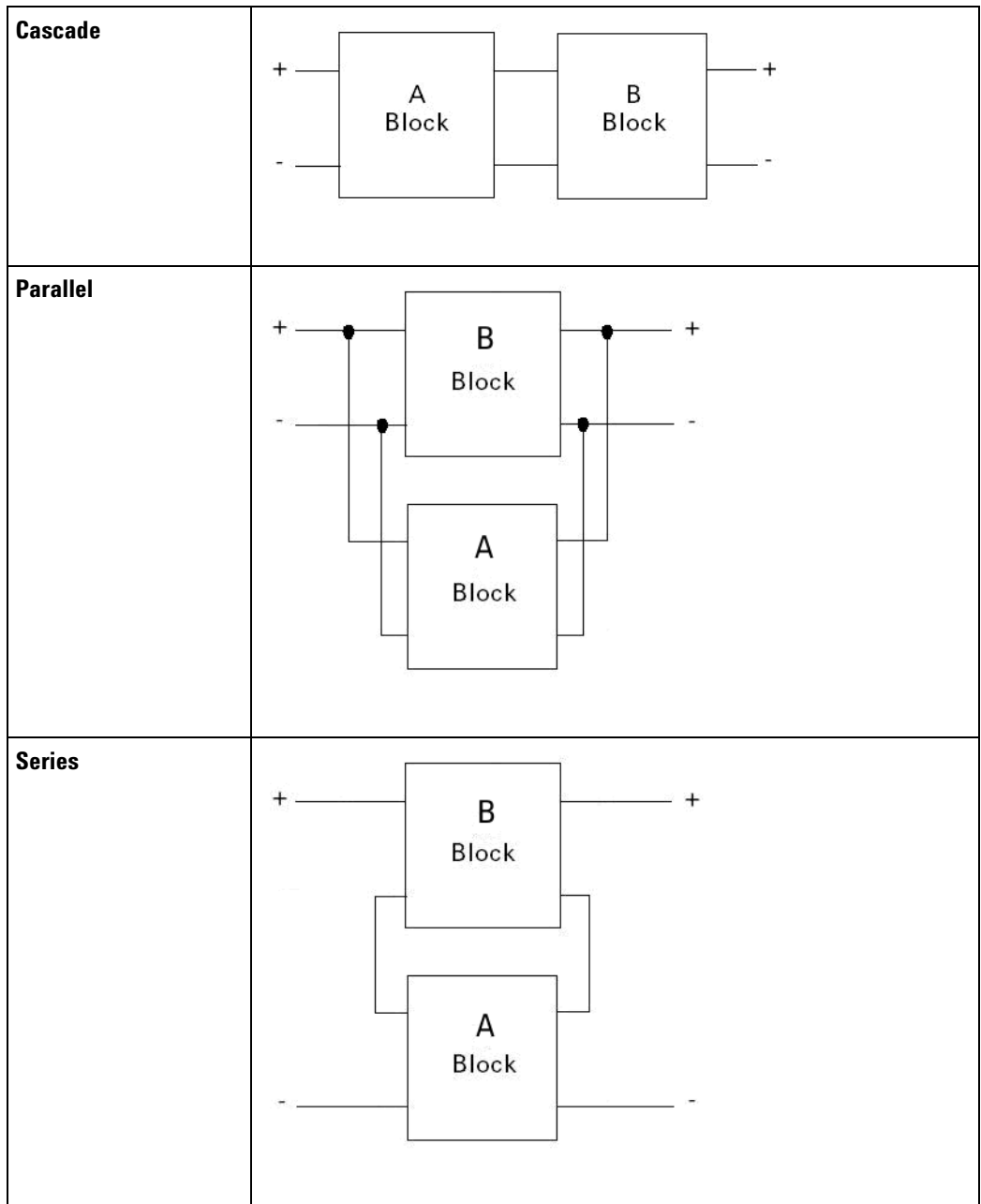
Cascade	
Parallel	
Series	

Once you select one of these three configurations, you can click each individual block and define what type of block it is (see screen shot below).
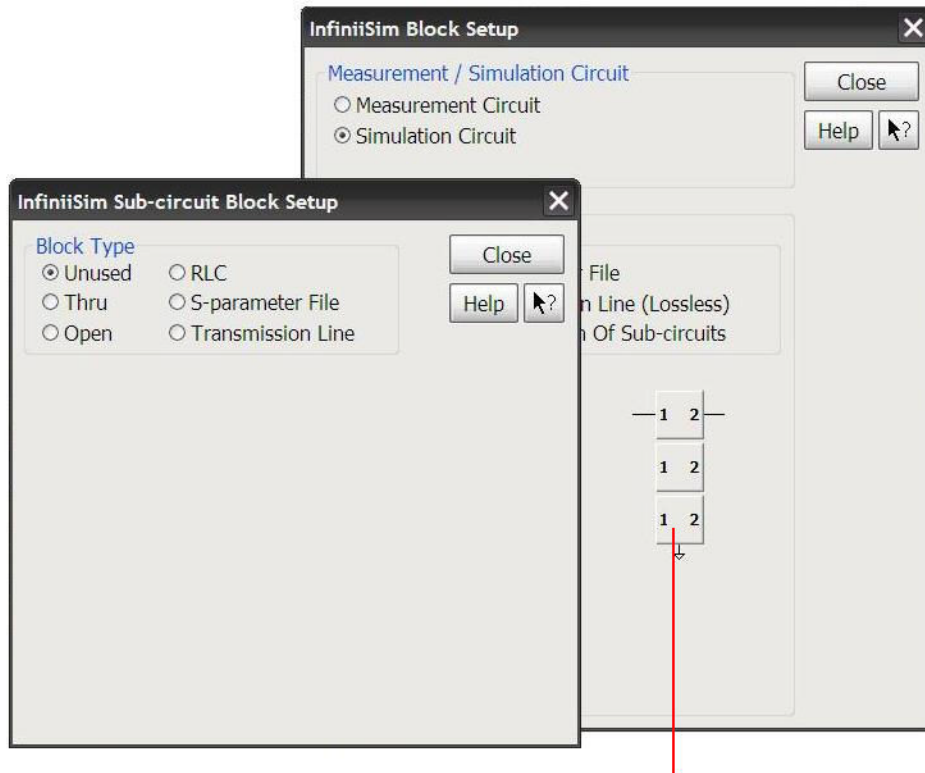
Clicking on one of these blocks will bring up its corresponding InfiniiSim Sub-circuit Block Setup dialog box where you can define it

When you click one of the sub-blocks a new InfiniiSim Sub-circuit Block Setup dialog box appears where you can define its type

Notice that you can choose between **Thru**, **Open**, **RLC**, **S-parameter File**, **Transmission Line**, and an option called **Unused**. Unused simply 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.
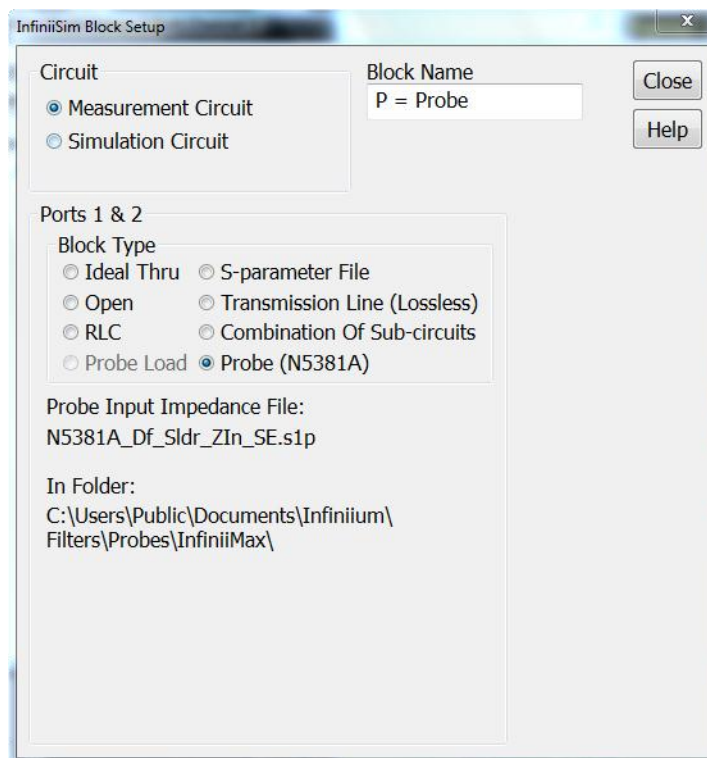
You can select the **Probe** block type only in the blocks that are intended to model the oscilloscope input (for example, the ones circled in red below).

A probe must be connected to the channel in order for the **Probe** block type selection to be active.
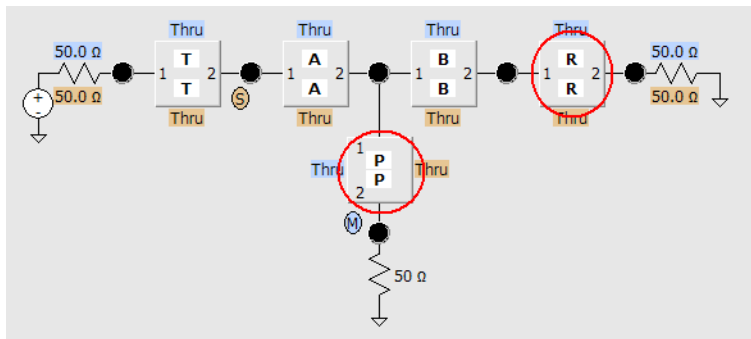
When you make the **Probe** block type selection, the appropriate probe input impedance file is automatically selected, based on your selections in the Probe Configuration dialog box.
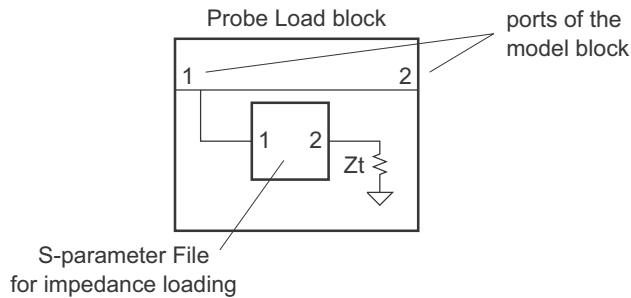
Probe Load Block Type (2 Port)

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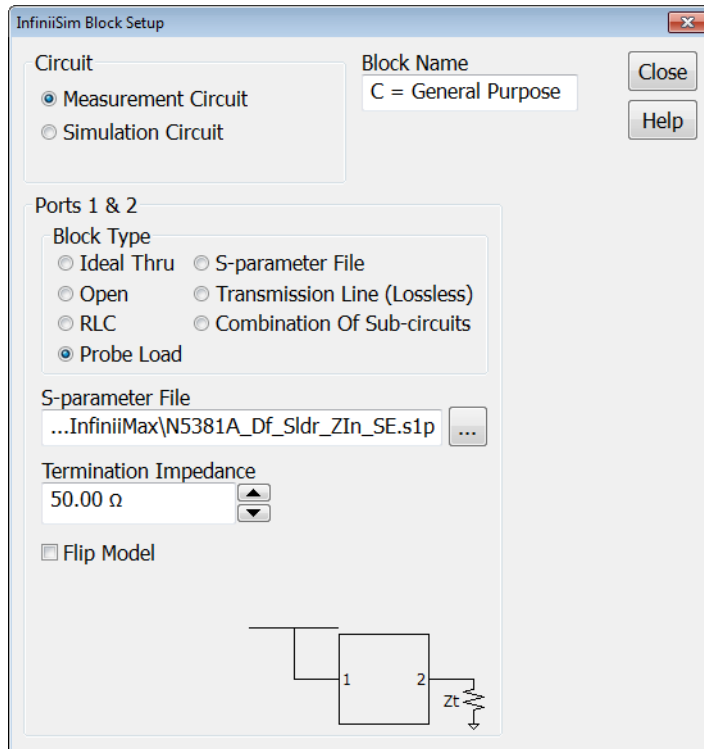
The **Probe Load** block type is not allowed in those blocks that are intended to model the oscilloscope input (for example, the ones not circled in red below).

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.
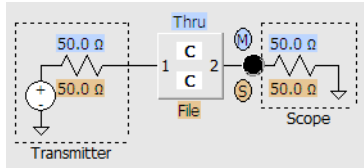
  On Infiniium oscilloscopes with the Windows 7 operating system, input impedance files in are located in the "C:\Users\Public\Documents\ Infiniium\Filters\Probes\InfiniiMax" directory.

- **Termination Impedance** — Again, with a 1-port S-parameter (1sp) 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.
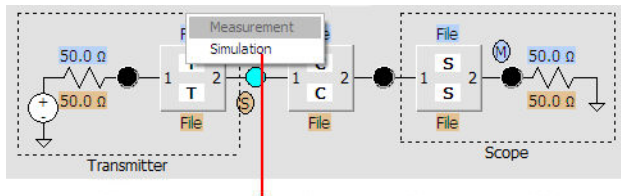
Look at the model below that shows both the measurement and simulation circuit view. The blue circle labeled **M** is the measurement node and the orange circle labeled **S** is the simulation node.

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 3 block **Relocate the Observation Node** application preset configuration shown below, you can change the location of the measurement and/or simulation node.

To change the nodes, click one of the node locations (the large black dots in the diagram); then, select either **Measurement** or **Simulation** from the popup box (as shown below).

This popup appears when you click on one of the large black dots in the circuit. You can then select either Measurement or Simulation to move the corresponding observation point to that node.

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 92).
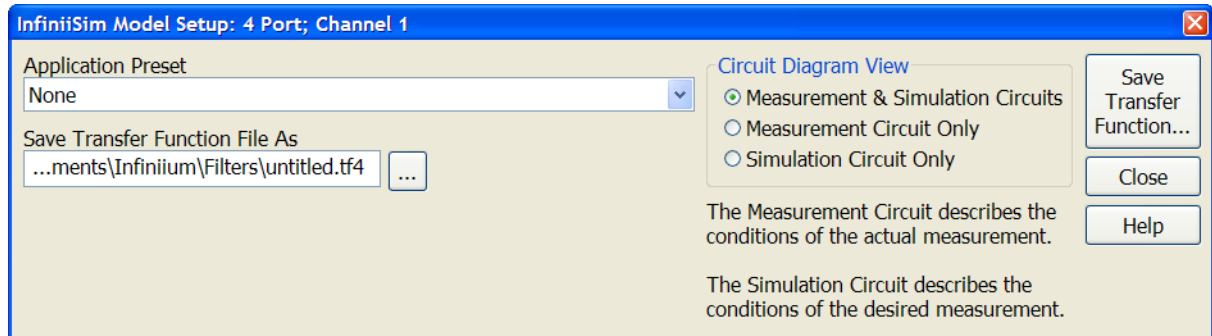
(No models are shown in the InfiniiSim Model Setup dialog box's default view above because the **Application Preset** field is set to **None**.)

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 39.

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 40.

4 Specify the circuit source and load impedances. See "Circuit Source and Load Impedances (4 Port)" on page 41.

5 Define measurement circuit and simulation circuit blocks. See "Defining a Block (4 Port)" on page 42.

6 Specify measurement and simulation node locations. See "Measurement and Simulation Node Locations (4 Port)" on page 56.

7 Generate and save the transfer function file.

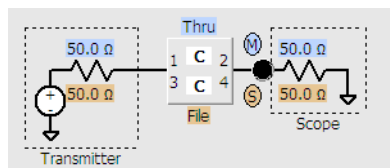| **NOTE**	The transfer function you create in this dialog box is not saved until you click the **Save Transfer Function...** button in the upper right corner. If you close the dialog box without saving, your transfer function is not applied.

See: "Generating / Saving a Transfer Function File" on page 57.

Application Presets (4 Port)

The first field in the InfiniiSim Model Setup dialog box is labeled **Application Preset**. This drop-down field gives you a variety of preset configurations you can work with to build your own model. The available preset configurations 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)
- **Relocate the observation node of a probed measurement** (Advanced)
- **General purpose probe** (Advanced)
- **General purpose 6 blocks** (Advanced)
- **General purpose 9 blocks** (Advanced)

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 there are some values highlighted blue and some highlighted orange. These correspond to the measurement and simulation circuit respectively. Let us first define what we mean by each of these circuits.
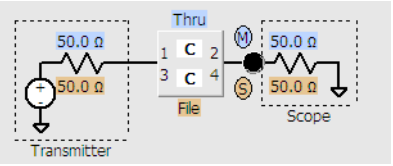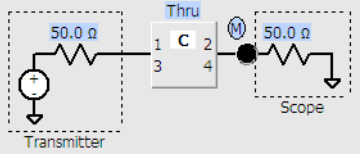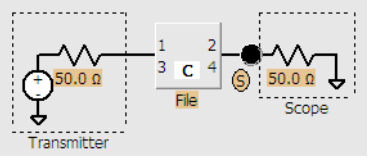
- **Measurement Circuit** — This circuit models the actual physical circuit that produced the measured waveform.
- **Simulation Circuit** — This circuit models a hypothetical electrical circuit that exhibits your desired electrical characteristics.

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.

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 Overview / Theory" on page 8 and "Transfer Function Files" on page 87.

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.
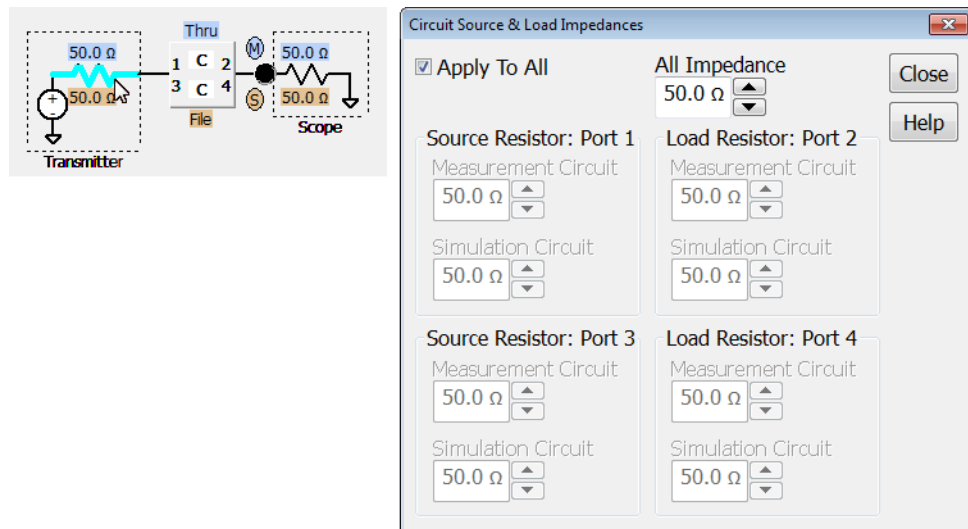
It is important to note that these circuit model representations in the GUI are simplified for ease of viewing. If we were being explicit, the model blocks in the circuit would look like the following for a 4-port device:
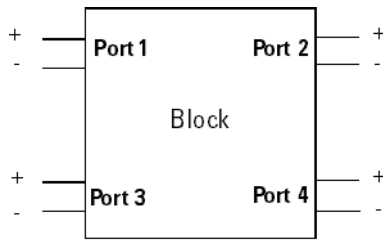
The diagrams in the GUI are completely interactive so you can click various sections or components and make adjustments.

Circuit Source and Load Impedances (4 Port)

The resistor on the left side of the circuit shown below corresponds to the impedance of the transmitter. It is defaulted to 50 Ω, but you can change this if needed. Simply place your cursor over the resistor (it thickens to indicate you are on top of it) and click. You then see the following Circuit Source and Load Impedances dialog box.
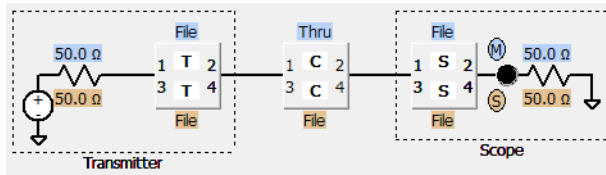

The Circuit Source and Load Impedances dialog box lets you specify all of the source and load impedances within both the measurement and simulation circuits.

Note that if we did not have the **Apply to All** box checked, the **Add Insertion Loss** application preset (used in the screen shot above) would grey- out the load impedance controls because the oscilloscope load impedance is always 50 Ω.

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

Note that by using one of the 3-block models (for example, the **Add/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.

Please note that any termination block located adjacent to a source or load resistor (like the **T** block and **S** 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 that s1p 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 letters and names to the blocks in the legend for a typical use scenario depending upon 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 Block Setup dialog box (accessed by clicking one of the blocks).

Looking at the Blocks legend to the right tells us that this A stands for Transmitter Channel. However, you are not required to define it as a transmitter channel if you do not want. You can be flexible in your design.

You can also change the name of a block by clicking on it and using the Block Name control in the subsequent dialog box.

The Blocks legend shows you what the letters at the center of each block stand for. These are just suggestions, meant to make it easier to understand how the configuration could be used. However, you do not have to follow these definitions and can be flexible in the design of your circuits

While blocks represent different parts of the circuit, they have the exact same definition controls to choose from and are simply called different types to help you understand what they might represent in your circuit. The termination blocks, **T** and **S**, 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 that s1p 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.
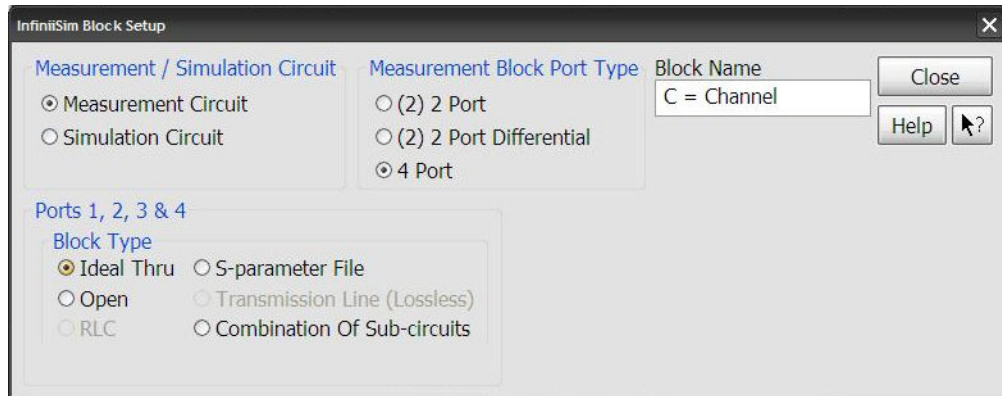
The first section (**Measurement/Simulation Circuit**), lets you specify whether you are currently defining a block in the measurement circuit or the simulation circuit. For instance, if you select **Simulation Circuit** in this section and then set the **Block Type** to **Ideal Thru**, then the corresponding 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**, **Transmission Line (Lossless)**, **S-parameter File**, 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**, **Transmission Line (Lossless)**, **S-parameter File**, 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**, 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 a RLC (resistor, inductor, capacitor) network. See "RLC Block Type (4 Port)" on page 46.

- **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 46.

- **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 49.

- **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 49.

- **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 52.

- **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 54.

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 56), then generate, save, and

apply the correction transfer function (see "Generating / Saving a Transfer Function File" on page 57).

RLC Block Type (4 Port)

The RLC block type is only available 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.
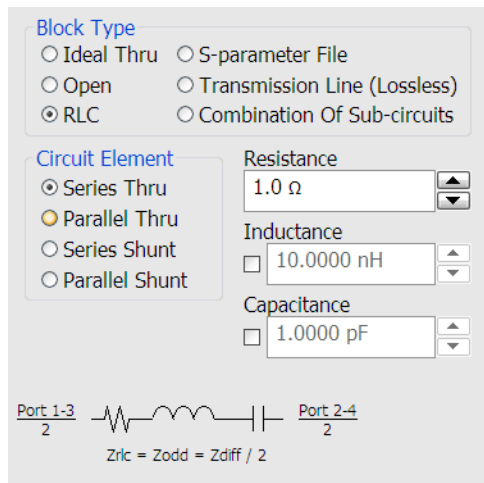
In the **Circuit Element** section, there are four configuration options (**Series Thru**, **Parallel Thru**, **Series Shunt**, and **Parallel Shunt**). When you select one of these options, the basic circuit configuration is shown in the dialog box (see screen shot above).

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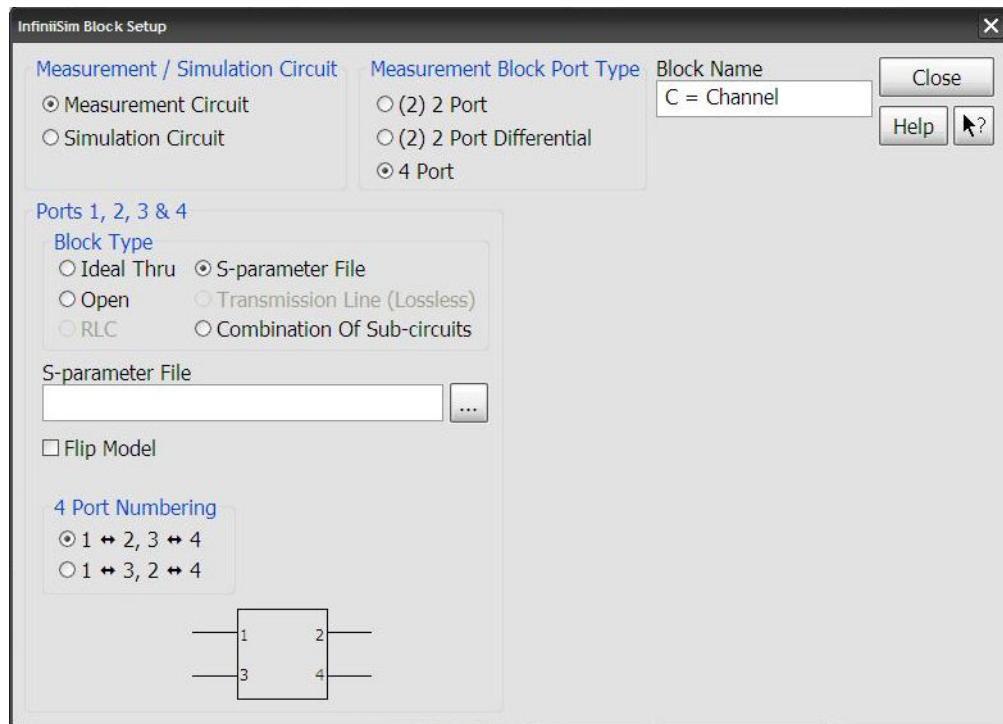
You can click the **S-parameter File** field to browse to your saved S-parameter files. S-parameter files are typically generated using a vector network analyzer or circuit simulator, for example, and then saved on the oscilloscope hard drive. These can be files that model fixtures, cables, devices,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

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, that s1p 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 Files" on page 85.

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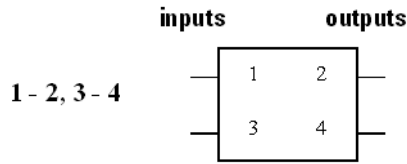
However, you can tell InfiniiSim that your file actually uses the following numbering system.

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 Block
No	1-2, 3-4	
No	1-3, 2-4	
Yes	1-2, 3-4	
Yes	1-3, 2-4	

For additional information regarding the reading of S-parameter files, see "S-Parameter Files" on page 85.

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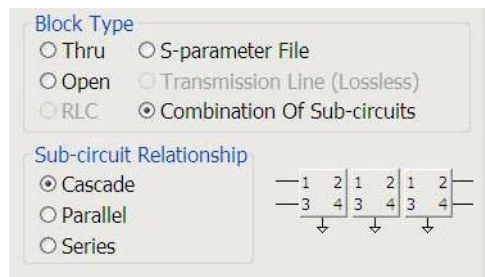
You have the following three choices for how to combine the three individual blocks that now make up the one circuit block: **Cascade**, **Parallel**, and **Series**.

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

Cascade	
Parallel	
Series	
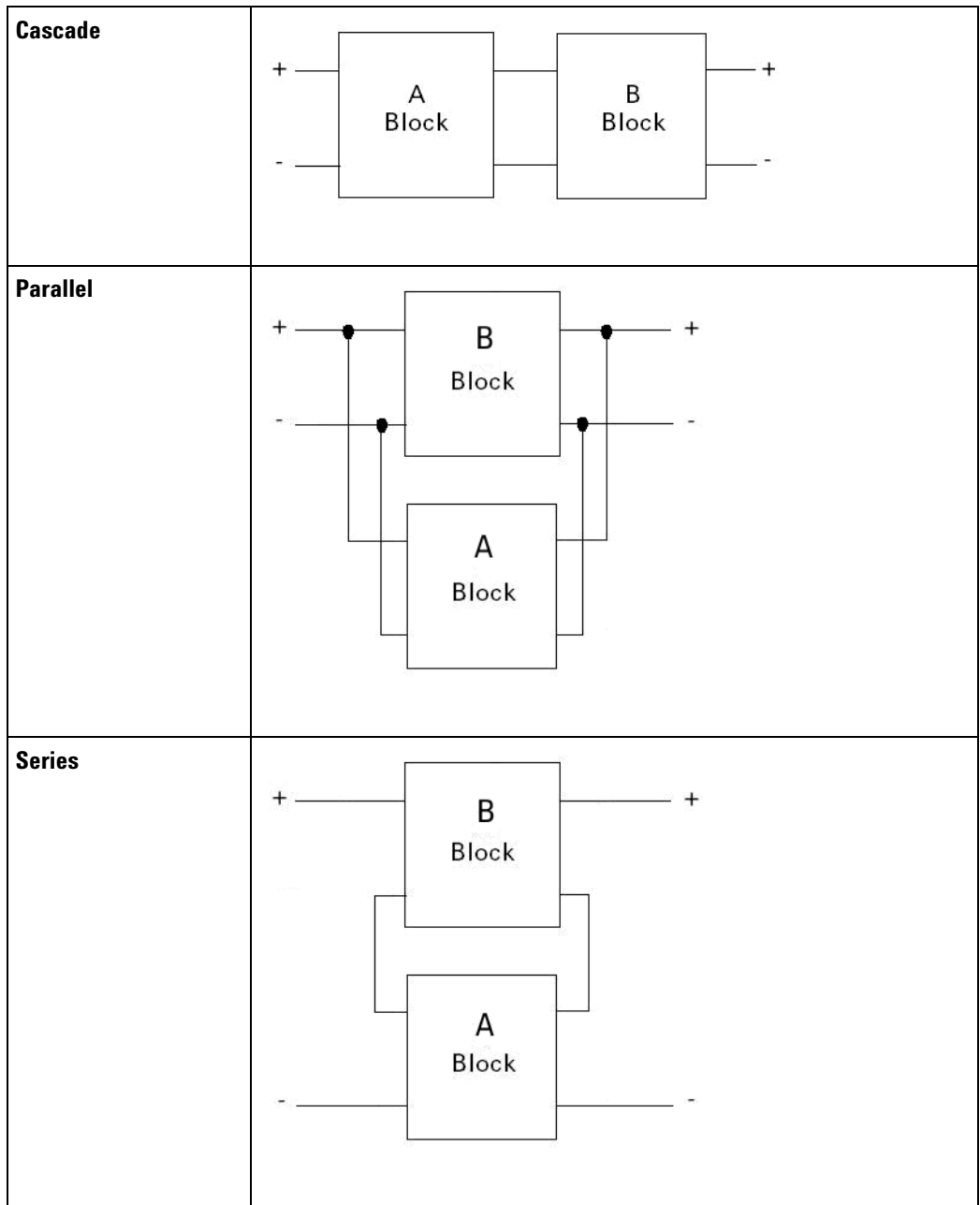
In **4 Port** port type mode, the 4 port blocks combine as shown below (pictures show connections for two of the three blocks for simplicity):
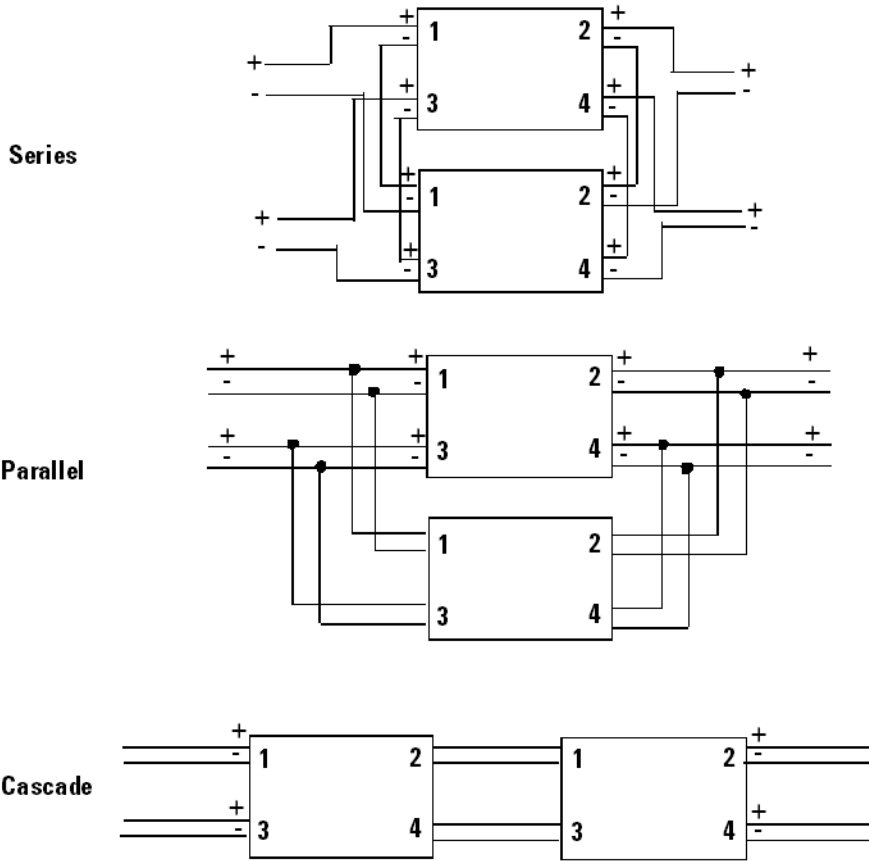
Series

Parallel

Cascade

Once you select one of these three configurations, you can click each individual block and define what type of block it is (see screen shot below).
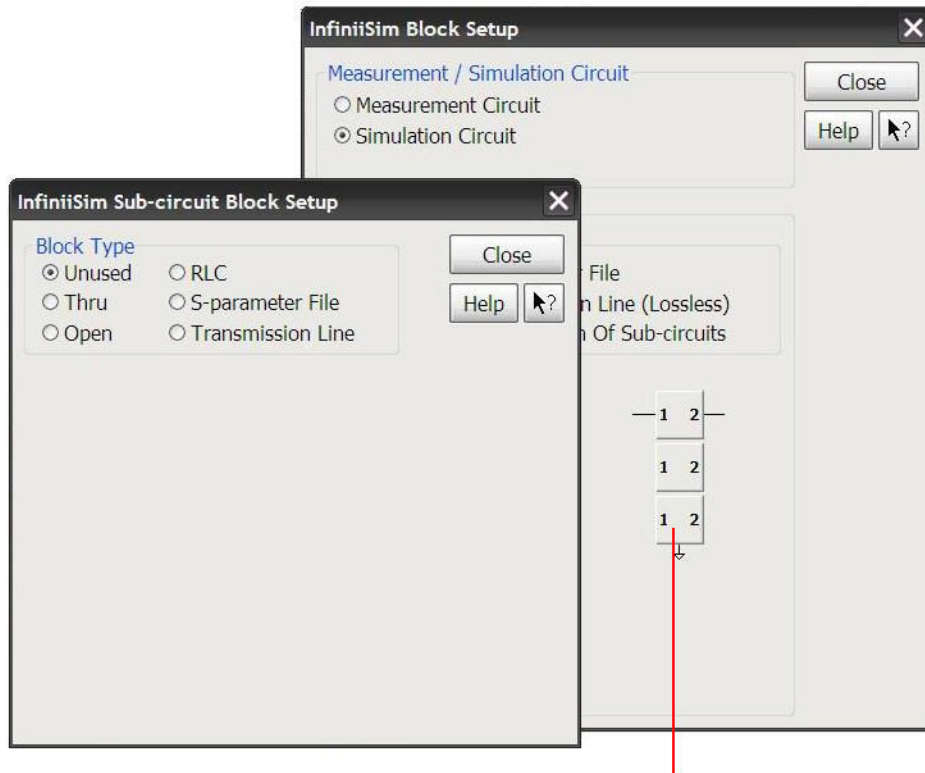
Clicking on one of these blocks will bring up its corresponding InfiniiSim Sub-circuit Block Setup dialog box where you can define it

When you click one of the sub-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**, **S-parameter File**, **Transmission Line**, and an option called **Unused**. Unused simply 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 red below).

A probe must be connected to the channel in order 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.
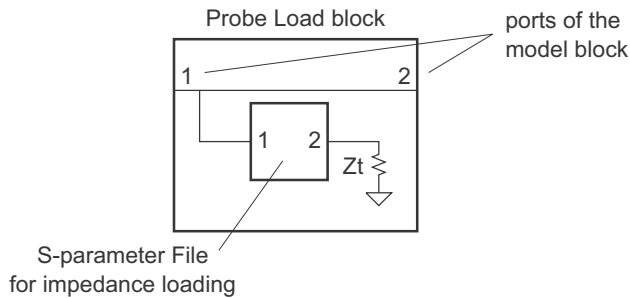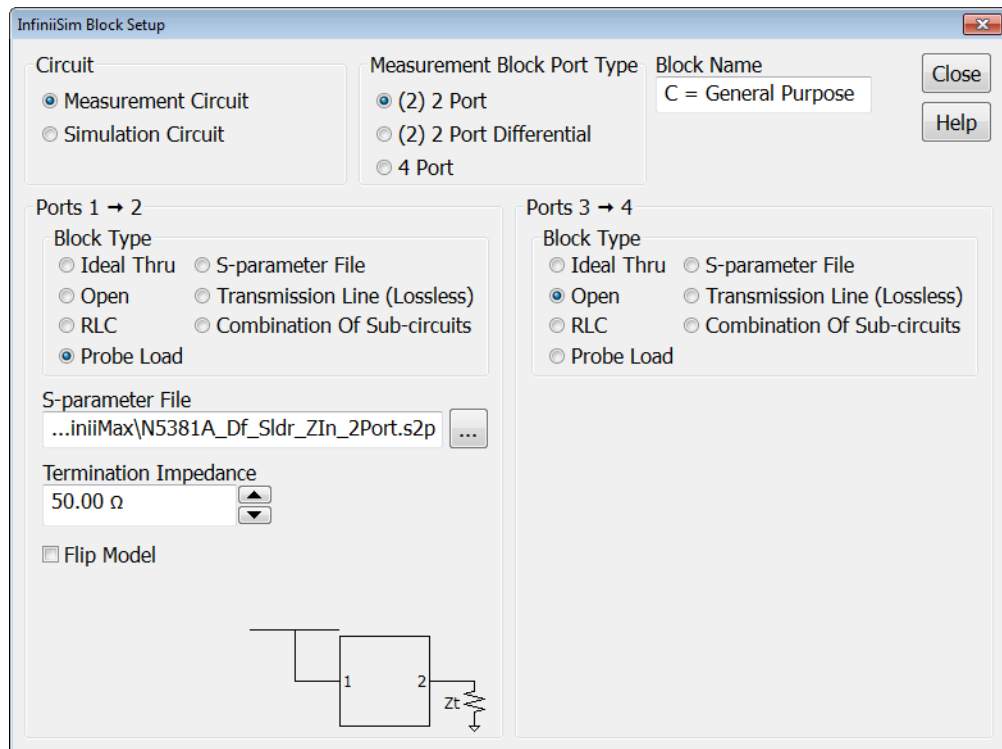
Probe Load Block Type (4 Port)

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 red below).

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.

  On Infiniium oscilloscopes with the Windows 7 operating system, input impedance files in are located in the "C:\Users\Public\Documents\Infiniium\Filters\Probes\InfiniiMax" directory.

- **Termination Impedance** — Again, with a 1-port S-parameter (1sp) 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.

Look at the model below that shows both the measurement and simulation circuit view. The blue circle labeled **M** is the measurement node and the 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 3 block **Relocate the Observation Node** application preset configuration shown below, you can change the location of the measurement and/or simulation node.

To change the nodes, click one of the node locations (the large black dots in the diagram); then, select either **Measurement** or **Simulation** from the popup box (as shown below).

This popup appears when you click on one of the large black dots in the circuit. You can the select either Measurement or Simulation to move the corresponding observation point to that node

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 19 or "Creating a Transfer Function From a Model (4 Port)" on page 38), 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.
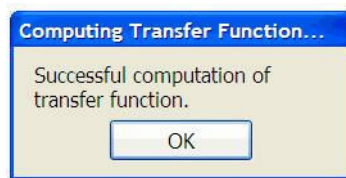
If InfiniiSim does not successfully compute the transfer function and instead gives you an error/warning message, see "InfiniiSim Warning / Error Messages" on page 78 for information regarding each of these messages and possible corrective actions.

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 Setup 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 (see the "Entering the InfiniiSim Application" on page 15 topic for more information on the bandwidth limiting feature).

InfiniiSim Plots

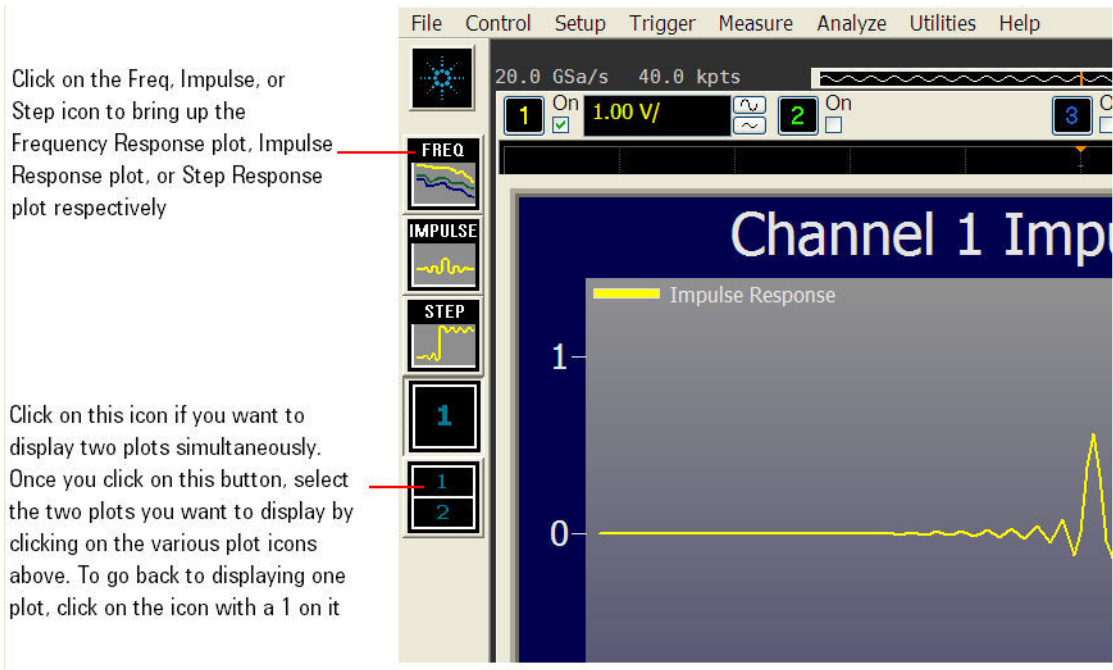When InfiniiSim is enabled, a tab labeled **InfiniiSim** appears on your oscilloscope's display.

Click this tab 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.

These plots are useful when setting some of the controls in the "InfiniiSim Setup Dialog Box" on page 92.

Basic Plot Controls

Below is an example of one of the plots you can display (the Impulse Response plot). The screen shots below highlight the basic plot controls.

Click on the Freq, Impulse, or Step icon to bring up the Frequency Response plot, Impulse Response plot, or Step Response plot respectively

Click on this icon if you want to display two plots simultaneously. Once you click on this button, select the two plots you want to display by clicking on the various plot icons above. To go back to displaying one plot, click on the icon with a 1 on it

Right-click any of the plots to open the popup menu shown in the screen shot below. This lets you quickly jump back to the InfiniiSim model builder or the InfiniiSim Setup dialog box to make changes.
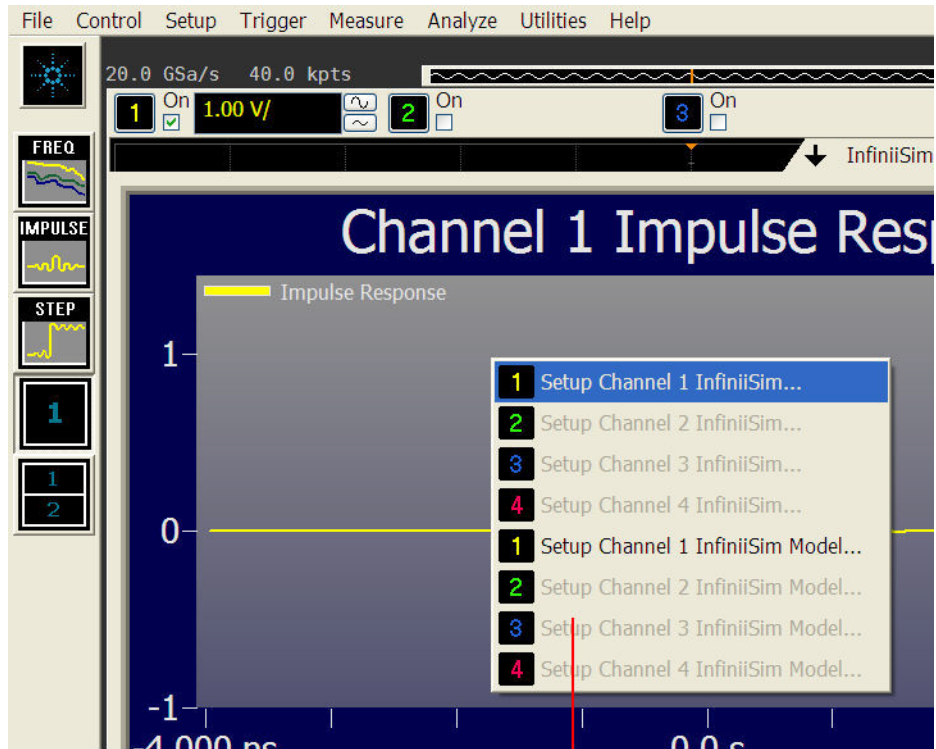
Right click on the plot display to make this popup window appear
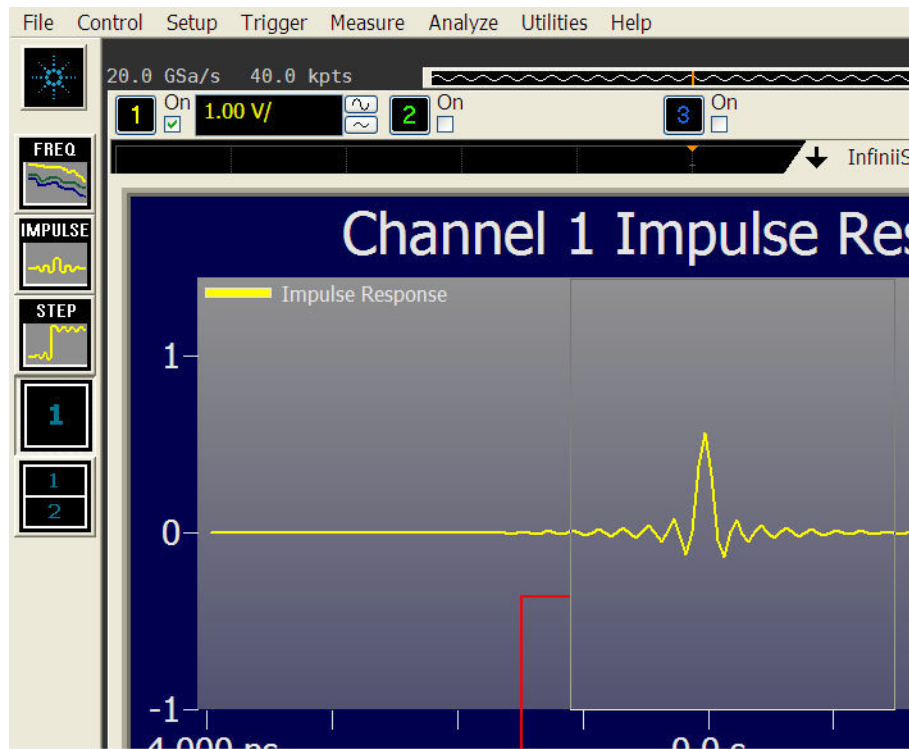
If you click and hold on a portion of the plot, drag a rectangle around a certain area, and then release the mouse, the area enclosed by the rectangle will be enlarged horizontally (zoom capability)
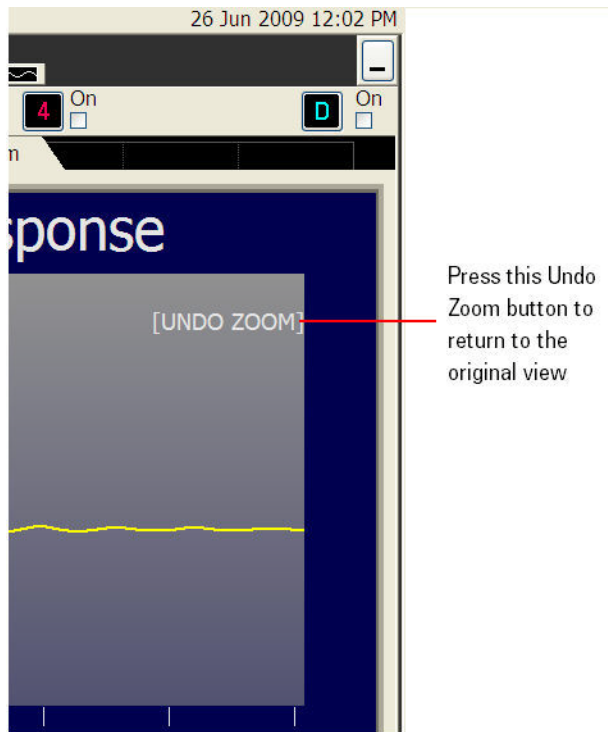
If you have InfiniiSim acting on multiple channels, buttons appear in your plots that let you switch between the different channels.

Also, it is important to note that the plots shown in this topic correspond to 2-port analysis. If you were performing a 4-port analysis, there would be two plots shown in each graph (a plot for the differential mode and a plot for the common-mode).

See Also • "Frequency Response Plot" on page 63

• "Impulse Response Plot" on page 63

• "Step Response Plot" on page 65

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), and 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** control. Remember in the discussion on the **Bandwidth Limit** control (found in "InfiniiSim Setup Dialog Box" on page 92), 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** control accordingly.

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 92).

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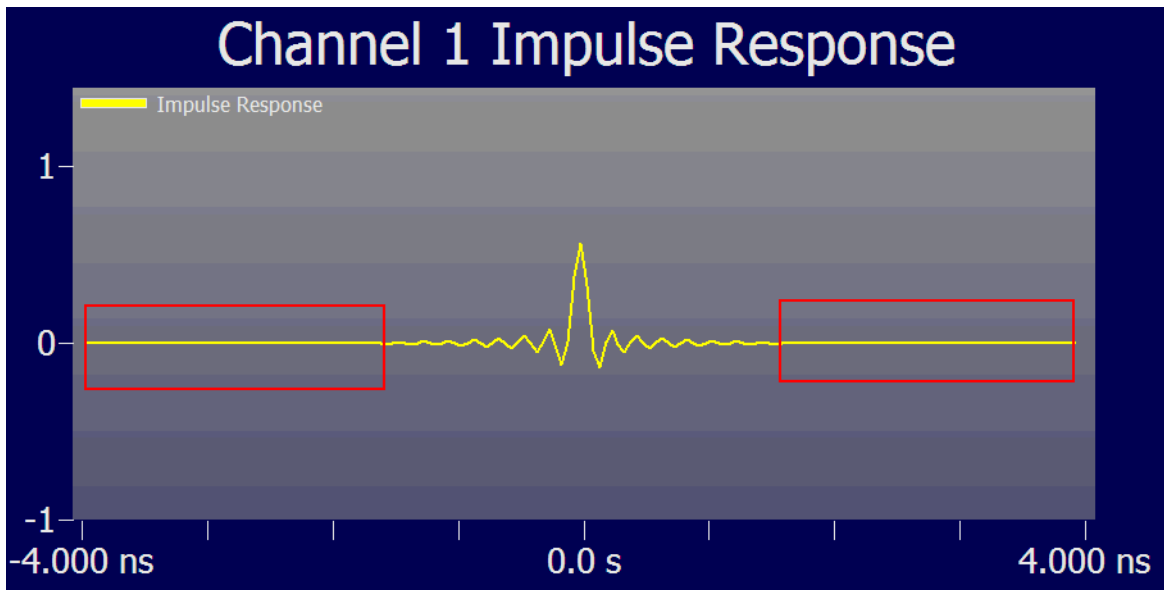
As you can see, there are large sections of the impulse response that are just flat and, therefore, contribute insignificantly to the simulated waveform. If you see this, you may be able to increase your measurement update rate by reducing the **Max Time Span** control.

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 **Setup Channel N 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, please 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).

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.
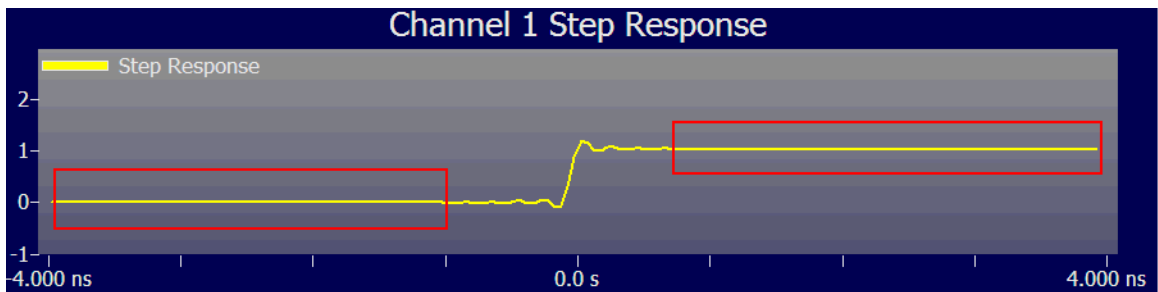
The screen shot above shows the large flat areas enclosed by red rectangles. You may need to reduce the **Max Time Span** control in this instance.

InfiniiSim Examples

Removing the Insertion Loss of a Cable

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

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.
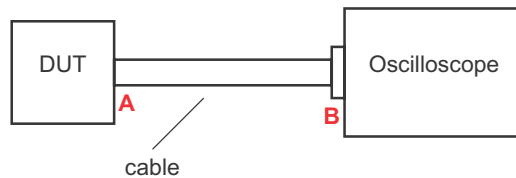
The locations marked A and B are the nodes in our measurement. We cannot connect the oscilloscope directly to node A to measure the signal at that location. Instead, we need a cable to connect to the DUT which means we actually end up measuring what the signal looks like at node B after it has passed 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 above your oscilloscope's display.
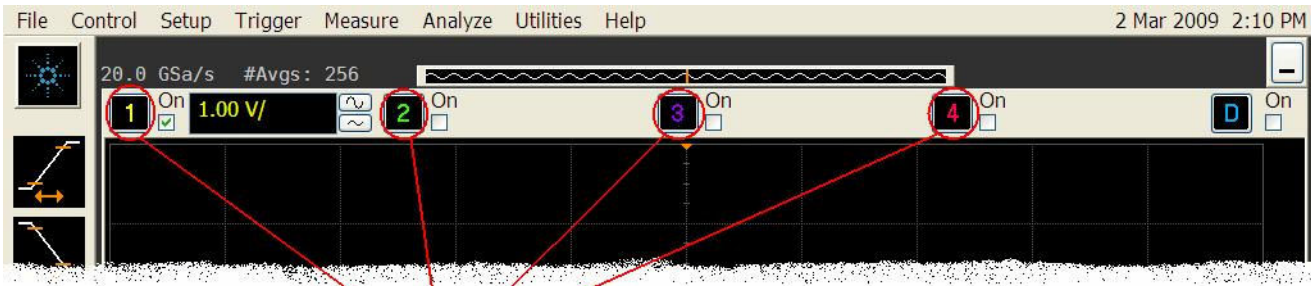
Clicking on any of these buttons will open the corresponding Channel Setup dialog box.

This opens the Channel Setup dialog box. Under the **InfiniiSim** section, select **2 Port** (we assume the signal passing through the cable is single-ended). Then, enable the **Bandwidth Limit** field and set it to 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). Click the **Setup…** button to open the InfiniiSim Setup dialog box.
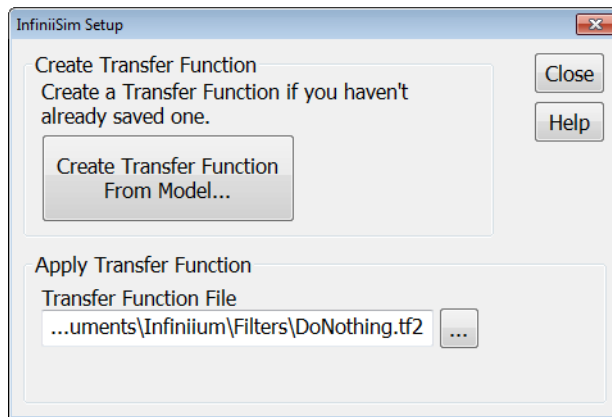
We now want to create our transfer function from a model (we do not have a transfer function previously saved) so click the **Create Transfer Function From Model…** button. This opens the InfiniiSim Model Setup dialog box.
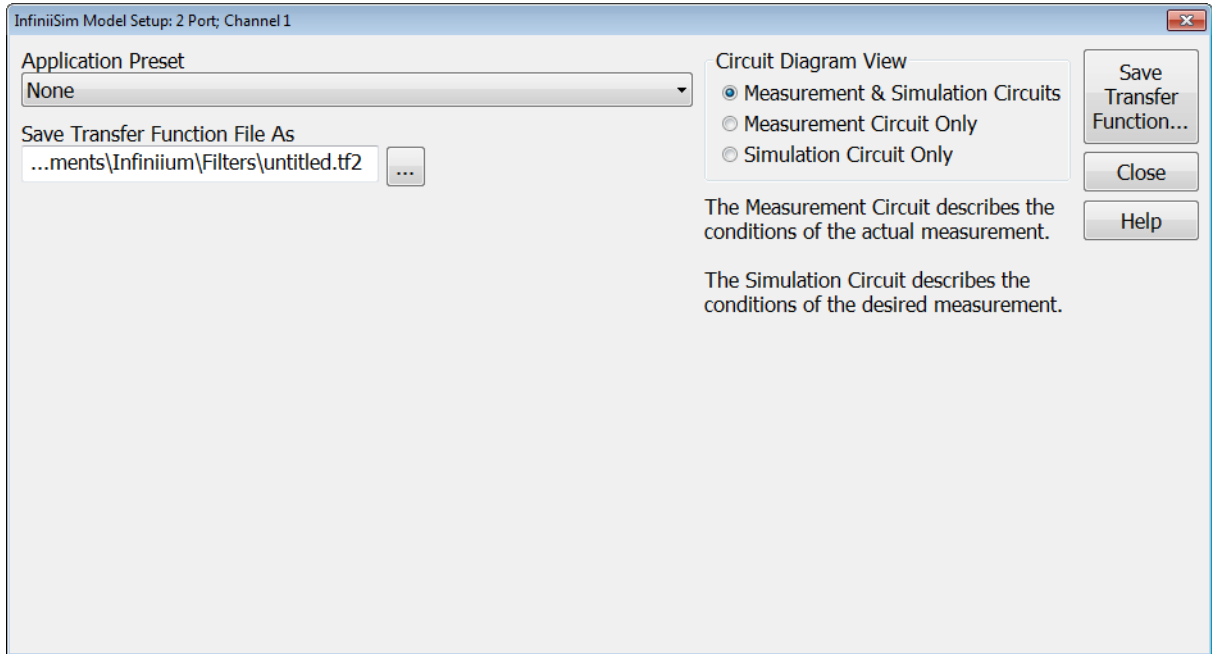
For our simple example, we choose the **Remove Insertion Loss of a Fixture or Cable** as the use model from the **Application Preset** field. This is because we only have one device we are trying to negate and we do not care about multiple reflection effects.

Once you select this use model, you see the following circuit view in the dialog box.
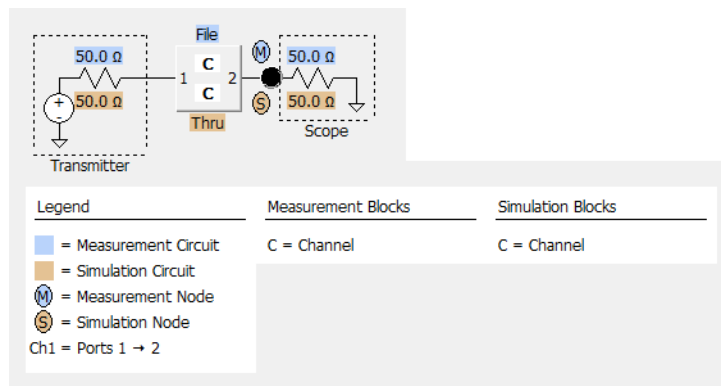
Now, we need to set up the two circuits: the measurement circuit and the simulation circuit.

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 only measure 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 are 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 a 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).
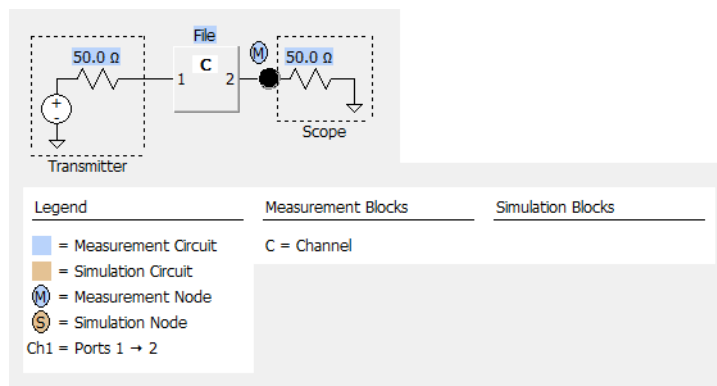
To get it set up like this, click the **C** block to open the InfiniiSim Block dialog box. Then, select the **Measurement Circuit** option in the Circuit section. Under Block Type, choose **S-parameter File** and then click the **S-parameter File** field, navigate to where you saved the file on the oscilloscope, and click **Load**. Because the S-parameters represent a cable, S_{21} = S_{12}, checking the **Flip Model** option should have no effect.
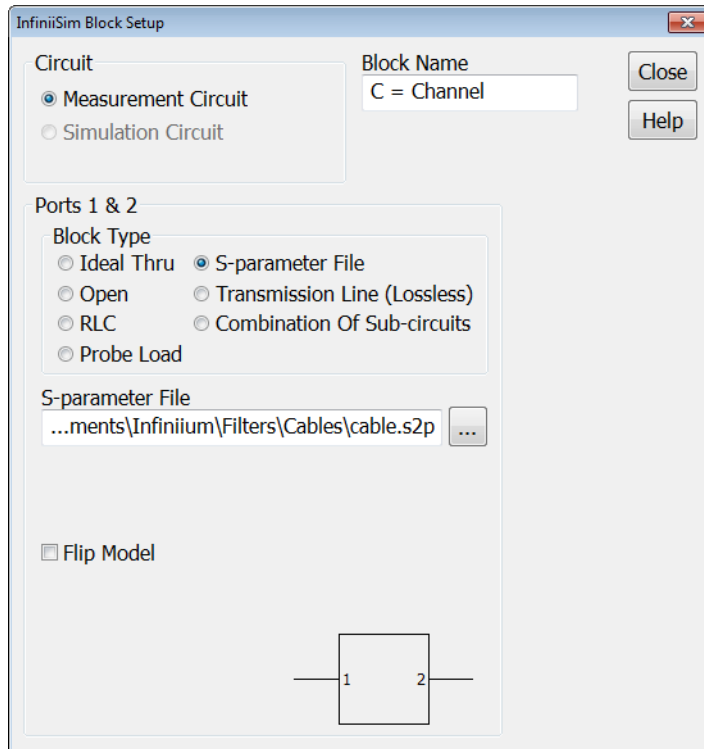
Because the Simulation circuit block is defaulted to **Ideal Thru** for this application preset, we do not need to change it. If you think about the simulation we want to perform, it makes sense that the block needs to be defined as a Thru (meaning that the block acts as though it is not there). Remember 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 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 it is complete, you can then 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 looked like with no cable present between the DUT and the oscilloscope.

You can also click the **InfiniiSim** tab to see the various plots associated with your signal, the simulated signal, and the transfer function.
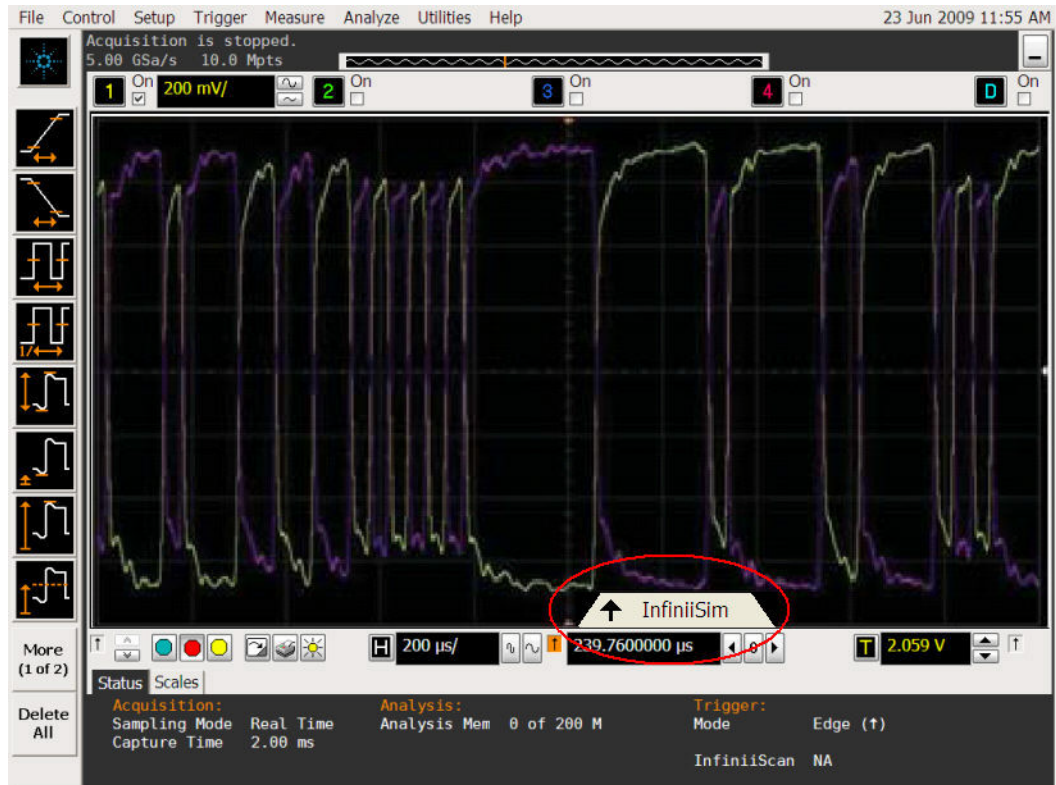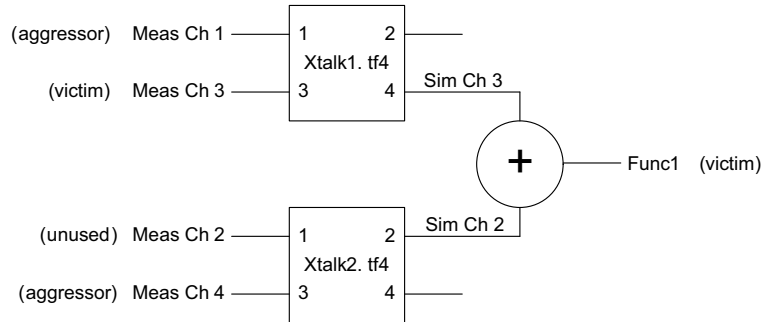

Simulating Crosstalk

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.
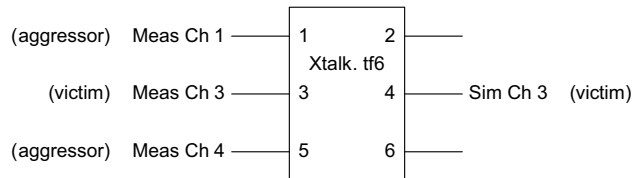
However, InfiniiSim cannot handle 6-port transfer functions. But, you can simulate crosstalk with two 4-port transfer functions using the setup shown in the following figure.

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 shown below.
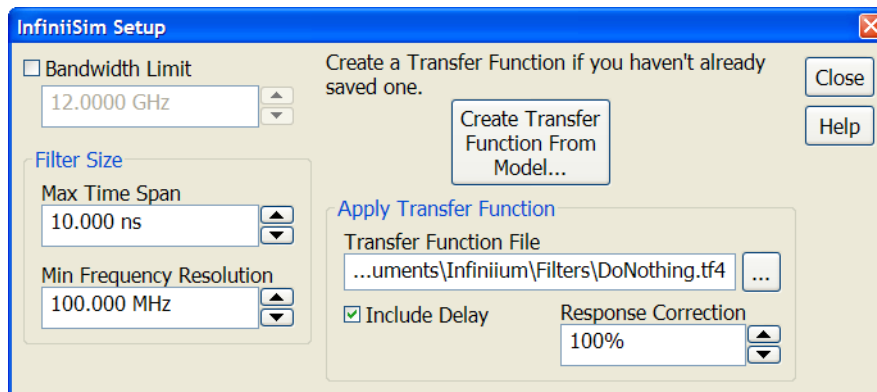
If Xtalk1.tf4 already exists, then select it as the **Transfer Function File**. Otherwise, you have to create it from S-parameter models. To create it from models, click **Create Transfer Function From Model…** to open the InfiniiSim Model Setup dialog box; then, choose the **Add insertion loss of a fixture of cable** application preset, and rename the **Save Transfer Function File As** to "Xtalk1.tf4".
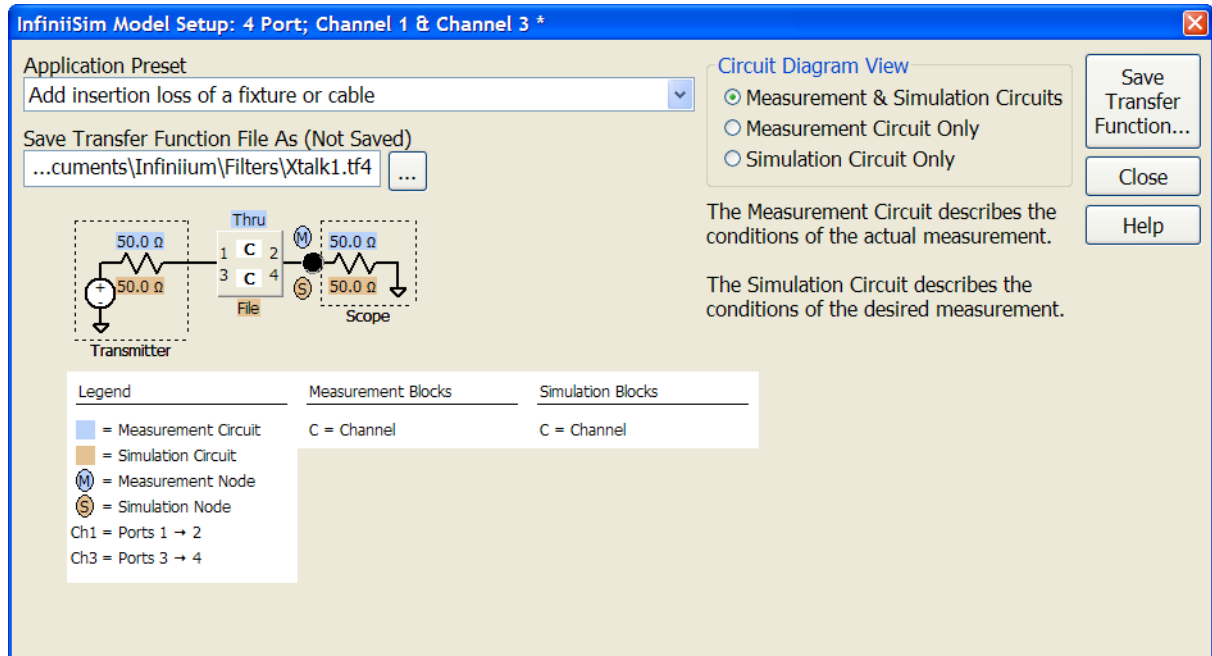
Then, click the **C** model block to open its Block Setup dialog box. Configure the Simulation Circuit to use your desired S-parameter file. The Measurement Circuit should remain defined to an **Ideal Thru**.
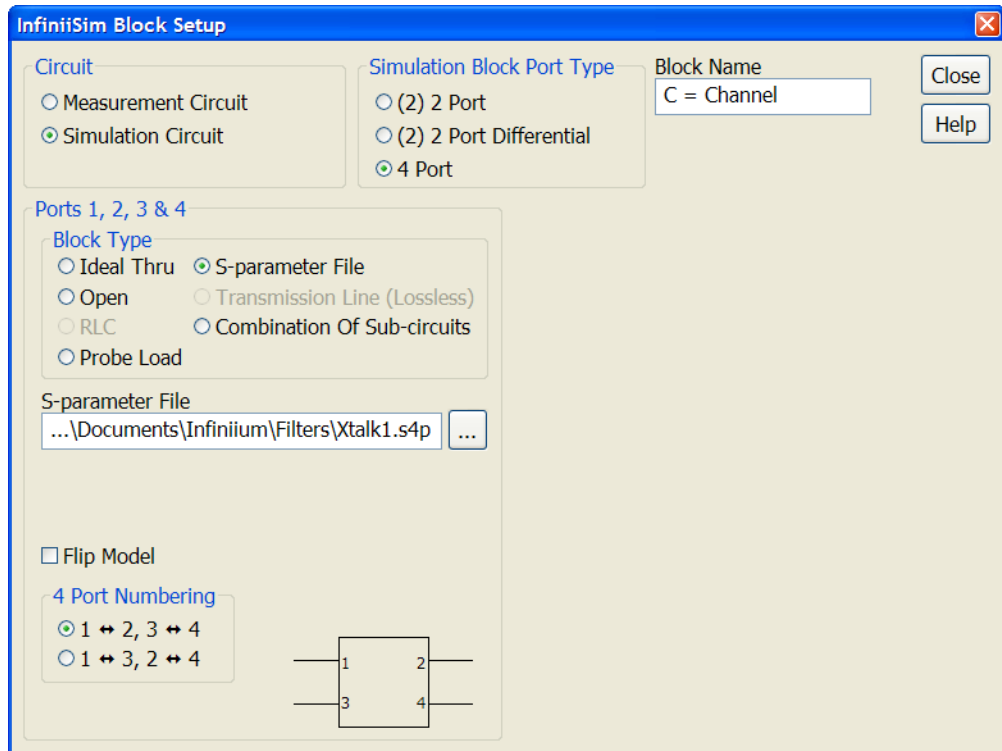
The S-parameter models you use to model crosstalk can be measured
directly from your device or exported from your personal simulation tools.
In either case, use the following configurations when measuring or
simulating the S-parameter crosstalk models, in which the unused ports
are terminated into 50 ohms.
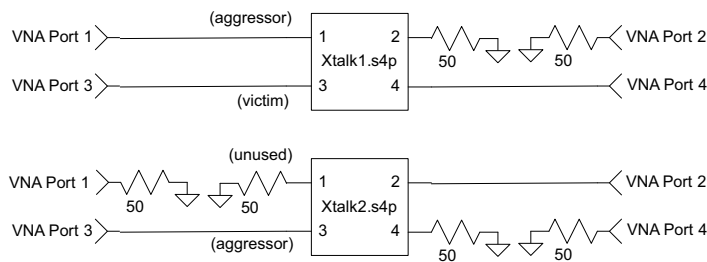
Now, close the Block Setup dialog box and click **Save Transfer Function...**.
After it completes calculating and saving the new TF, close the Model
Setup dialog box and the InfiniiSim Setup dialog box.

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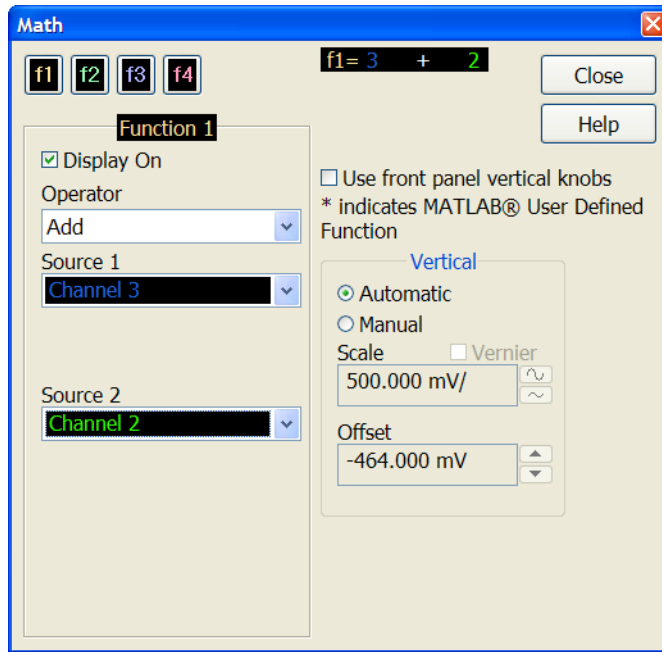
Note that although the example just described only used two aggressors, a third could be added by simply using the port 1 input of Xtalk2.tf4.

InfiniiSim Warning / Error Messages

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

1 port Files Allowed at Source and Load Blocks Only

In 2 port InfiniiSim, you are only allowed to use 1 port files in the location circled below.
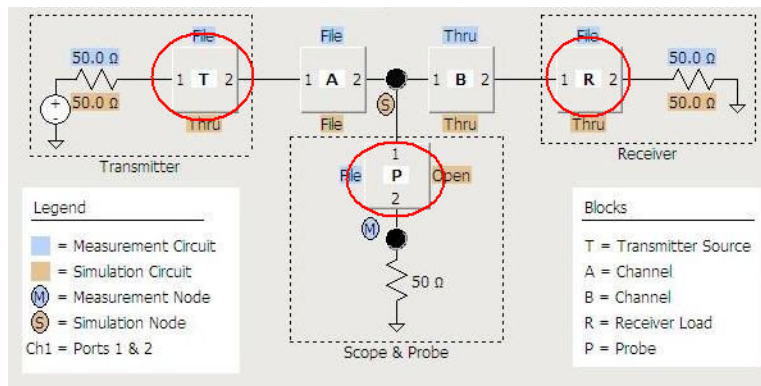
In other words, you can only use them for Transmitter, Receiver, and Scope / Probe blocks. If you try to use a 1-port file anywhere else, you get this error.

2 port Files Allowed at Source and Load Blocks Only

In 4 port InfiniiSim, you are only allowed to use 2-port files in the location circled below.
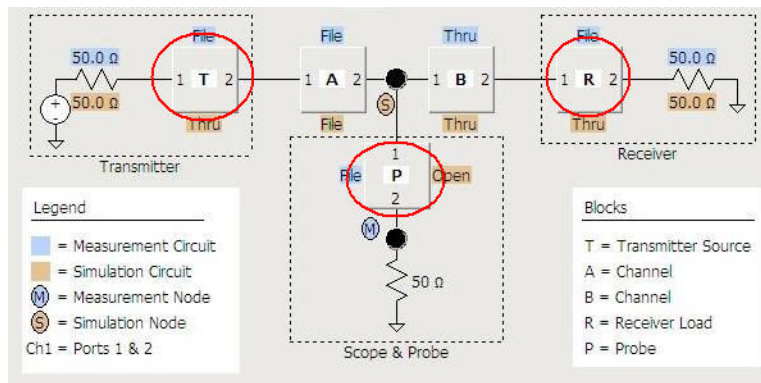
In other words, you can only use them for Transmitter, Receiver, and Scope / Probe blocks. If you try to use a 2-port file anywhere else, you get this error.

Calculation Aborted

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

Cannot Read 4 Port Data From a 2 Port File

This error means that you have attempted to load 2-port file data into a 4-port model block. If you want to define a 4-port block with 2-port files, you need to use the **(2) 2 Port** or **(2) 2 Port Differential** Block Port Type settings.

DEFAULT_FREQUENCY_RESOLUTION Value is 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 86).

Files May Only Contain a Single Data Package

Each CITIfile file may only model a single device.

File Not Found

This 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 .cti 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 only allowed to use 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 only allowed to use BE, BD, MA, or RI data value formats with ATF and Touchstone files. If you use any other data format, you get this error message.

Invalid File Format

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

Invalid File Name

This indicates that the file extension is wrong or invalid for the type of file needed.

Invalid Frequency Units (Hz, kHz, MHz, GHz only)

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

Invalid Output Name

This 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 86).

Less Than 2 Data Points in the File

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

Signal Amplitude at Measurement Node is Too Small

This message usually means you made a mistake when setting up your measurement circuit (example, placed 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.

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 Agilent 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 Agilent Technical Support.

Unstable or Ill-conditioned Circuit Definition

When you get this message it means one of your circuits is invalid. This warning indicates that an infinite voltage is present in the circuit (analogous to an oscillation) or in other words, the gain becomes infinite.

Warning: File Data Truncated to 100 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 warning immediately below except that instead of a model's long time delay causing the problem, it is the model's slow time constants. These kind 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 the 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 in order 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 the signal" warning message.

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 Files" on page 85). 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 a 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 Files" on page 85.

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 the time delay" 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 in order 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 the 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 (example, placed 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 63).

Warning: Unrecognized ATF File Version (A.01.00 Only)

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

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).
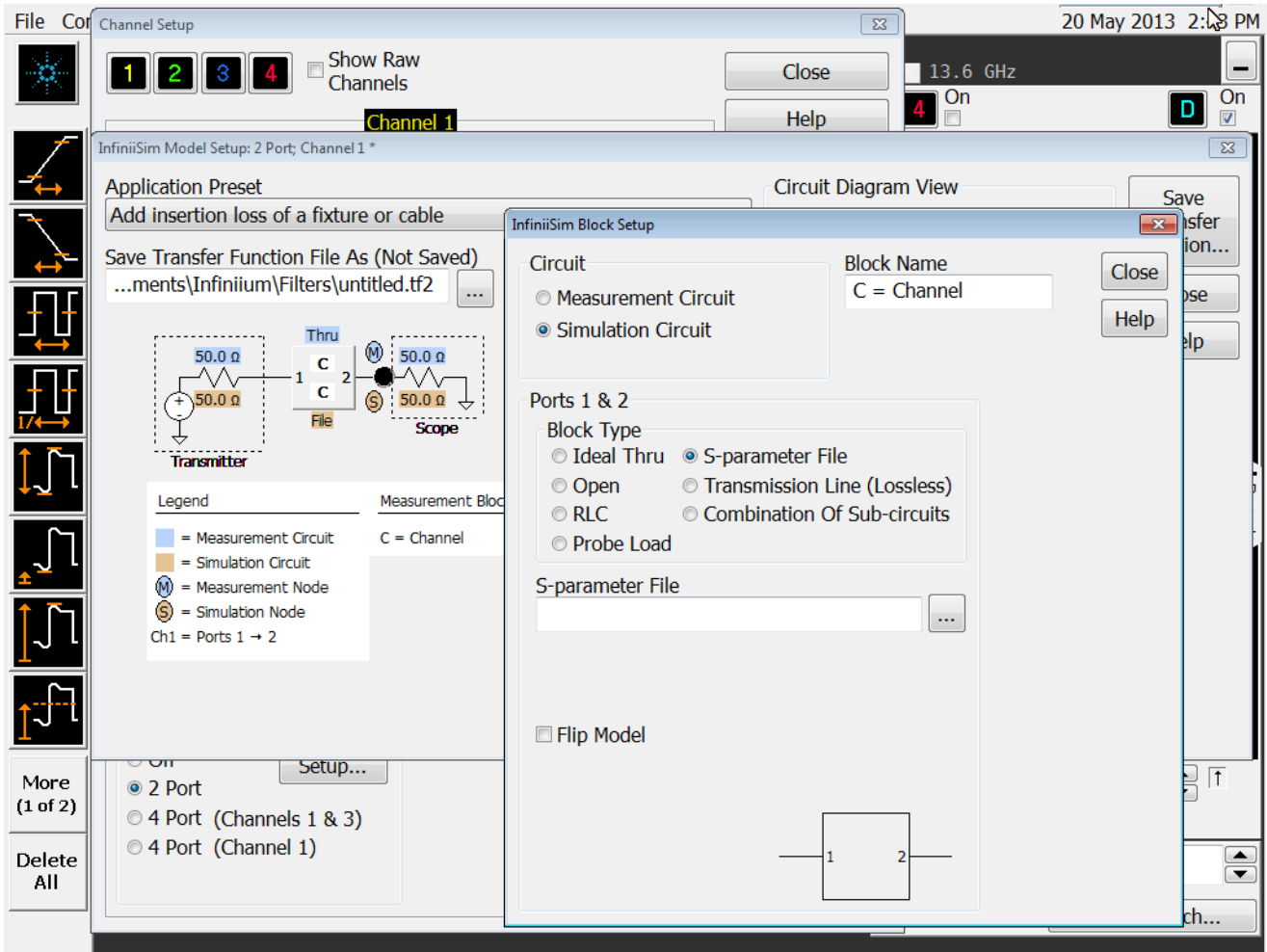
InfiniiSim Concepts

S-Parameter Files

S-parameter files describe electrical n-port network models of physical devices as a tabular list of complex S-parameter values at specific frequencies. Formatted as ASCII text, S-parameter files are easily viewed or edited using any text-based file editor. InfiniiSim supports two standard file formats:

- Touchstone (version 1.1).

   Touchstone files are identified by the file extensions .s1p, .s2p, or .s4p.

- CITIfile (version A.01.00 and A.01.01).

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

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

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.

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. However, this does not mean that all S-parameter files must contain uniformly spaced frequency data. Uniformly spaced file data is usually preferred, but not required if you are knowledgeable about how InfiniiSim interprets file data.

In the absence of the special InfiniiSim keyword DEFAULT_FREQUENCY_RESOLUTION (see "S-Parameter File Keywords" on page 86), InfiniiSim checks the data to ensure that it is uniformly sampled in frequency (with exception to the interval between DC and the next lowest frequency point) and generates a warning if the data is not uniformly sampled. If the DEFAULT_FREQUENCY_RESOLUTION keyword is accompanied by a numeric value, the data is resampled using linear interpolation and the

frequency resolution specified by that value. Otherwise, InfiniiSim resamples the file data using a frequency resolution equal to the frequency interval between the last two frequency points in the file.

InfiniiSim expects S-parameter 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 also truncates all file data to a maximum of 100,000 frequency points.

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. However, those blocks that model source impedance or load impedance (transmitter source, oscilloscope/probe, or oscilloscope input) can be defined by S-parameter files which contain only return loss information. In the absence of the special InfiniiSim keyword RETURN_LOSS_INTERPRETATION, 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.

Selecting a 1-port S-parameter (s1p) file for a 4 port application applies that s1p to s_{11} and s_{33}, and sets s_{13} and s_{31} to zero.

S-Parameter File Keywords

The following are the currently supported optional keywords you can place in your S-parameter files. InfiniiSim scans S-parameter files for special keywords which communicate to InfiniiSim how the data within the file was generated or how it should be interpreted. The format of these lines differs slightly between Touchstone and CITIfile files in that they must be proceeded by an exclamation point (!) within Touchstone files. InfiniiSim keywords are case insensitive and permit additional comments on the same line.

#DSO DEFAULT_FREQUENCY_RESOLUTION <value>

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

In the absence of this keyword, InfiniiSim checks the data to ensure that it is uniformly sampled in frequency (with exception to the interval between DC and the next lowest frequency point) and generates a warning if the data is not uniformly sampled. If <value> is a numeric value, the data is resampled using linear interpolation and the frequency resolution specified by <value>. Otherwise, InfiniiSim resamples the file data using a frequency resolution equal to the frequency interval between the last two frequency points in the file. If <value> is equal to either of the text strings (AUTO or AUTOMATIC), InfiniiSim does not report a warning about non-uniformly sampled data.

#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, 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} = $_{12}$ = 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.

Transfer Function Files

Transfer function files describe the relationship between two sets of time varying node voltages as a tabular list of complex transfer function values at specific frequencies. Formatted as ASCII text, transfer function files are easily viewed or edited using any text-based file editor. 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.

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

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. However, this does not mean that all transfer function files must contain uniformly spaced frequency data. Uniformly spaced file data is usually preferred, but not required if you are knowledgeable about how InfiniiSim interprets file data.

In the absence of the special InfiniiSim keyword DEFAULT_FREQUENCY_RESOLUTION (see "Transfer Function File Keywords" on page 88), InfiniiSim checks the data to ensure that it is uniformly sampled in frequency (with exception to the interval between DC and the next lowest frequency point) and generates a warning if the data is not uniformly sampled. If the DEFAULT_FREQUENCY_RESOLUTION keyword is accompanied by a

numeric value, the data is resampled using linear interpolation and the frequency resolution specified by that value. Otherwise, InfiniiSim resamples the file data using a frequency resolution equal to the frequency interval between the last two frequency points in the file.

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 also truncates all file data to a maximum of 100,000 frequency points.

Transfer Function File Keywords

The following are the currently supported optional keywords you can place in your transfer function files. InfiniiSim scans transfer function files for special keywords which 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.

! #DSO ATF_FILE_VERSION <string>

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

! #DSO DEFAULT_FREQUENCY_RESOLUTION <value>

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

In the absence of this keyword, InfiniiSim checks the data to ensure that it is uniformly sampled in frequency (with exception to the interval between DC and the next lowest frequency point) and generates a warning if the data is not uniformly sampled.

If <value> is a numeric value, the data is resampled using linear interpolation and the frequency resolution specified by <value>. Otherwise, InfiniiSim resamples the file data using a frequency resolution equal to the frequency interval between the last two frequency points in the file.

If <value> is equal to either of the text strings (AUTO or AUTOMATIC), InfiniiSim does not report a warning about non-uniformly sampled data.

! #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.

Using the Bandwidth Limit Control

The **Bandwidth Limit** field requires some explanation so you know how to properly set it 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).
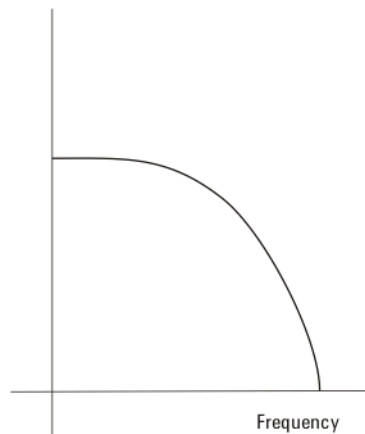
Some InfiniiSim applications attempt to remove or correct this insertion loss in the channel. In those cases, the correction transfer function becomes the inverse of the channel's insertion loss (dashed line in plot below) such that the product of the "peaked" correction transfer function response and the channel response yields a flat frequency spectrum (red line in plot below).
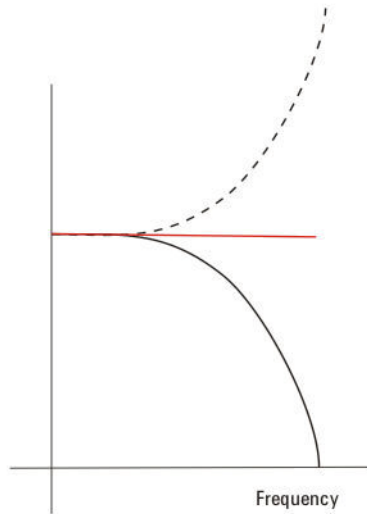
Now, in the above example, the response plot before InfiniiSim was applied was very simplistic. A more realistic frequency response plot might look more like the following:
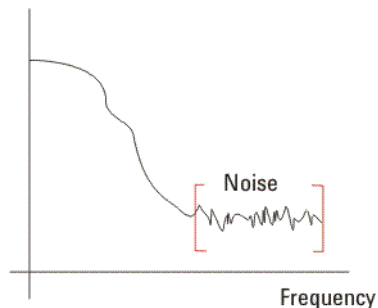
Notice that above a certain frequency (or bandwidth) there is very little signal and mostly noise. If you apply a correction through InfiniiSim as discussed above, you may end up excessively amplifying the noise. The **Bandwidth Limit** field in this dialog box lets you minimize this noise gain by attenuating 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 63).

Also, note that normally you do not need to apply Bandwidth Limit if you are adding loss (applying the loss of a cable, etc.). However, if you are applying gain (removing the loss of a cable, etc.), you may need to use this field.

Click the **Bandwidth Limit** field to open the Enter Bandwidth dialog box. You can then enter the value for this field.

Notice in the Channel Setup 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.

InfiniiSim Reference Information

InfiniiSim Setup Dialog Box

Below is a screen shot showing the InfiniiSim Setup dialog box.
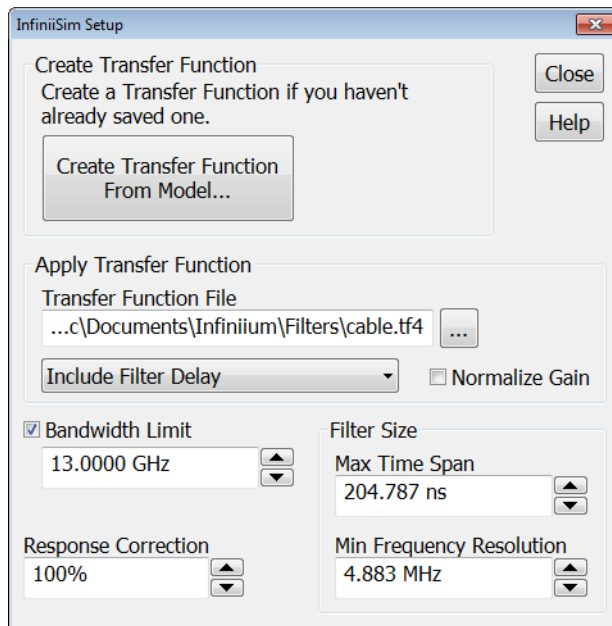
- **Create Transfer Function** — If you do not have a previously saved transfer function to load, you can create a transfer function from a model (click **Create Transfer Function From Model...**). For information on creating a transfer function from a model, see "InfiniiSim Model Setup Dialog Box" on page 94.

- **Apply Transfer Function** — 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 87.

  Also in the **Apply Transfer Function** 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** — This selection shifts the simulated waveform in time so the two waveforms are approximately aligned which aids in their comparison.

- **Remove Filter Delay** — No waveform shift is applied.

- **Include Trigger Corrected Delay** — This selection 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 Channel Setup dialog box. It is duplicated here for convenience. See "Using the Bandwidth Limit Control" on page 89.

- **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 63 and "Step Response Plot" on page 65.

- **Filter Size** — This area has two controls: **Max Time Span** and **Min Frequency Resolution**. These two controls are reciprocals of each other and slaved 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 which have slower time constants. You can use either the Step Response or Impulse Response plot (see) 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) and then this **Min Frequency Resolution** is set automatically.

InfiniiSim Model Setup Dialog Box

If you are using 2-port models, see:

- "Creating a Transfer Function From a Model (2 Port)" on page 19

If you are using 4-port models, see:

- "Creating a Transfer Function From a Model (4 Port)" on page 38

Circuit Source and Load Impedances Dialog Box

If you are using 2-port models, see:

- "Circuit Source and Load Impedances (2 Port)" on page 22

If you are using 4-port models, see:

- "Circuit Source and Load Impedances (4 Port)" on page 41

InfiniiSim Block Setup Dialog Box

If you are using 2-port models, see:

- "Defining a Block (2 Port)" on page 23

If you are using 4-port models, see:

- "Defining a Block (4 Port)" on page 42

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 30

If you are using 4-port models, see:

- "Combination of Sub-Circuits Block Type (4 Port)" on page 49

Index