



**Agilent N5465A  
InfiniiSim Waveform  
Transformation Toolset  
Software**

**User's Guide**



**Agilent Technologies**

# Notices

© Agilent Technologies, Inc. 2009

No part of this manual may be reproduced in any form or by any means (including electronic storage and retrieval or translation into a foreign language) without prior agreement and written consent from Agilent Technologies, Inc. as governed by United States and international copyright laws.

## Manual Part Number

N5435-97000

## Edition

First edition, June 2009

Printed in USA

Agilent Technologies, Inc.  
1601 California Street  
Palo Alto, CA 94304 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.

# Contents

## 1 License Information

## 2 InfiniiSim Theory / Background

Example: Removing a Cable's or Fixture's Insertion Loss	11
Introduction to InfiniiSim	13
Introduction to the Measurement / Simulation Circuit Concept	14
Introduction to Transfer Functions	15
Overview of Block Configurations	17

## 3 InfiniiSim 2-port Controls

Entering the InfiniiSim Application	21
The InfiniiSim Setup Dialog Box	26
Creating a Transfer Function From a Model - An Overview of the Application Preset and Circuit Diagram View Controls	29
Creating a Transfer Function From a Model - An Overview of the Circuit Component Controls	34
Creating a Transfer Function From a Model - Generating/Saving	49
The InfiniiSim Tab and Plots	50

## 4 InfiniiSim 4-port Controls

Entering the InfiniiSim Application	57
The InfiniiSim Setup Dialog Box	62
Creating a Transfer Function From a Model - An Overview of the Application Preset and Circuit Diagram View Controls	65
Creating a Transfer Function From a Model - An Overview of the Circuit Component Controls	69
Creating a Transfer Function From a Model - Generating/Saving	83
The InfiniiSim Tab and Plots	84

## 5 A Simple Example

Introduction	91
The Problem: Removing the Insertion Loss of a Cable	91
The Solution Using InfiniiSim	92

## **6 Warning / Error Messages and S-parameter File InfiniiSim Keywords**

Warnings/Error Messages 97

S-parameter InfiniiSim Keywords 102













# 1 License Information

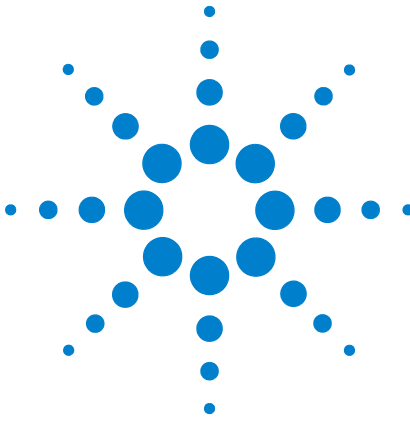
The license information for the InfiniiSim Waveform Transformation Toolset application is:

- DSO90000A-013: InfiniiSim Waveform Transformation Toolset Basic for 90000A Series oscilloscopes (installed on oscilloscope during purchase)
- DSO90000A-014: InfiniiSim Waveform Transformation Toolset Advanced for 90000A Series oscilloscopes (installed on oscilloscope during purchase)
- DSO9000A-013: InfiniiSim Waveform Transformation Toolset Basic for 9000A Series oscilloscopes (installed on oscilloscope during purchase)
- DSO9000A-014: InfiniiSim Waveform Transformation Toolset Advanced for 9000A Series oscilloscopes (installed on oscilloscope during purchase)
- N5465A-001: InfiniiSim Waveform Transformation Toolset Basic for existing Infiniium 9000 or 90000 Series oscilloscopes with software 2.00 or higher
- N5465A-002: InfiniiSim Waveform Transformation Toolset Advanced for existing Infiniium 9000 or 90000 Series oscilloscopes with software 2.00 or higher
- N5435A-026: InfiniiSim Waveform Transformation Toolset Basic server license
- N5435A-027: InfiniiSim Waveform Transformation Toolset Advanced server license

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



## **1 License Information**



## 2 InfiniiSim Theory / Background

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 enables you to remove the deleterious effects of unwanted channel elements, simulate waveforms with channels models inserted, view waveforms in physically unprobable locations, and compensate for the loading of probes and other circuit elements.

Before we detail InfiniiSim's capabilities, it is important to get a feel for the kinds of things it can do. One simple example is the removal of a cable's or fixture's insertion loss from your measurement.

### 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 allows you to do is extract the DUT's signal from the measured signal. For example, look at the following representation of a measurement.

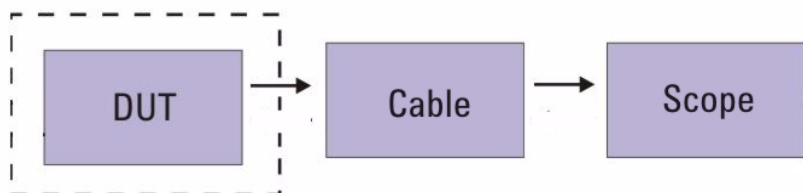


Figure 1 Example representation of a DUT measurement



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 10m (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 10m cable into the measurement.

## Introduction to InfiniiSim

InfiniiSim can perform the previously described waveform transformation of inserting a cable or fixture, but it can also do much more than this. It is a general transformation tool that can be used to produce more accurate results for a variety of problems. InfiniiSim provides default configurations for many common use models as well as general purpose configurations for the advanced user. For example, you can (descriptions of each function are also included):

- **Remove a channel element insertion loss:** Modeled with one block of loss (blocks will be discussed in detail later in this user's guide) 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 acquisition.
- **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  $S_{21}$  of the block is determined and its time response is convolved with the acquisition.
- **Remove all effects of a channel element:** By using S-parameter models of source and scope load, the effects of a channel element are totally removed. This is different from the first bullet above (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 (2nd bullet above) in that the reflective interactions between the elements are taken into account. This provides the most accurate rendering to insert a channel element.
- **Measurement plan relocation:** Measurement plane relocation allows you to view any voltage waveform in a circuit by moving the simulation node to any location you desire. This is an 'in situ' analysis so it is not a 'removal' or 'insertion' viewpoint.

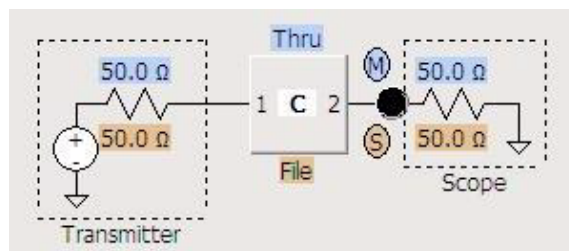
- **Remove loading effects of an oscilloscope probe:** To remove the loading effects of a probe, a topology of circuit blocks is given that allows probe models to be considered in the measurement. An oscilloscope probe, while it might be defined as 'high impedance', really does have a loading effect on the circuit. This effect can be taken into account and removed.
- **General purpose configuration:** General 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.

## Introduction to the Measurement / Simulation Circuit Concept

So, how does InfiniiSim accomplish all of these tasks? It does so through the use of correction transfer functions. When using InfiniiSim, you will 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 your desired electrical characteristics.

You can build these two circuits using the InfiniiSim GUI on your Infiniium oscilloscope as shown in the screen shot below (this is one of the simplest 2-port models you can create. The GUI also allows you to develop much more complex models as discussed later in this manual.).



The values highlighted in blue define the measurement circuit while the values in orange define the simulation circuit. The model block is 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.). You can also move the measurement and simulation nodes to a variety of locations within the model (when you are using more complex preset configurations).

InfiniiSim analyzes these two 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 InfiniiSim requires that the measurement circuit and the simulation circuit be linear and time-invariant (small-signal analysis requirements).

## Introduction to 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, 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 via 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.

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 or common-mode signals, but it does not include cross-coupling components of the differential or common mode signals.



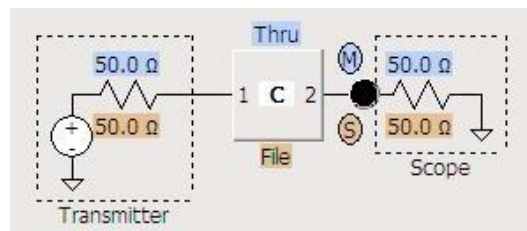
## Overview of Block Configurations

There are a wide variety of configurations possible using the InfiniiSim GUI. This section will cover three of the preset configurations so you get a feel for how these configurations are used. This is just a brief overview. Detailed discussions on the controls needed to set up circuit models will be discussed in subsequent chapters.

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 model block will be discussed in more detail in later chapters).

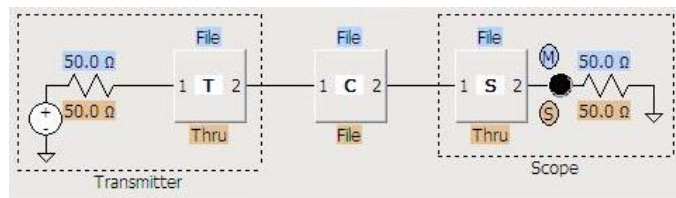
- **One Block for Simplest Path Compensation**

The configuration shown below is a basic one block model. Many times you will 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.



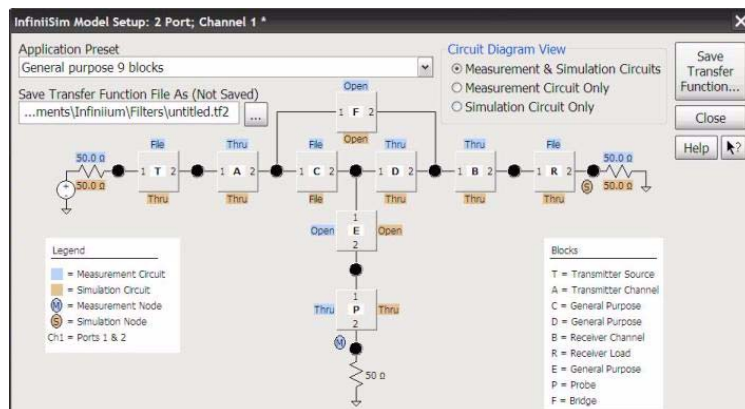
- **Three Block Analysis for True Channel Element Removal and Insertion**

When the most precision for a single channel element removal / insertion is required, you will 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).



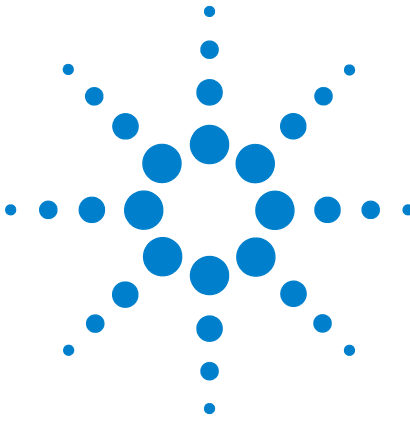
- **General Purpose Configuration**

The InfiniiSim waveform transformation toolset includes three general purpose topologies to enable detail 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 will find use 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.



Now that you have seen a high-level explanation of the InfiniiSim Waveform Transformation Toolset, let us look at the specific controls in a detailed manner.





## 3 InfiniiSim 2-port Controls

*This chapter will discuss the InfiniiSim 2-port controls.*

### Entering the InfiniiSim Application

The entry into the InfiniiSim application is found in the **Channel Setup** dialog box. This dialog box can be accessed via **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 lower right hand corner of the **Channel Setup** dialog box.

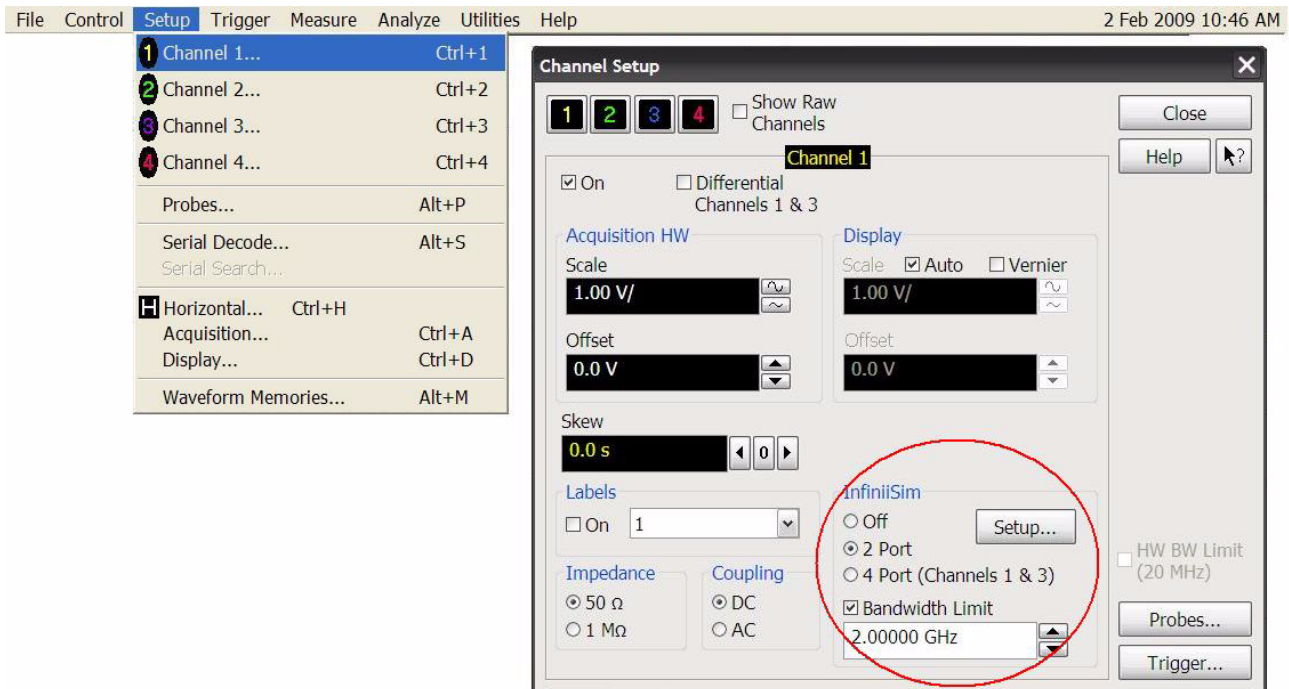


Figure 2 The entry location for the InfiniiSim application



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

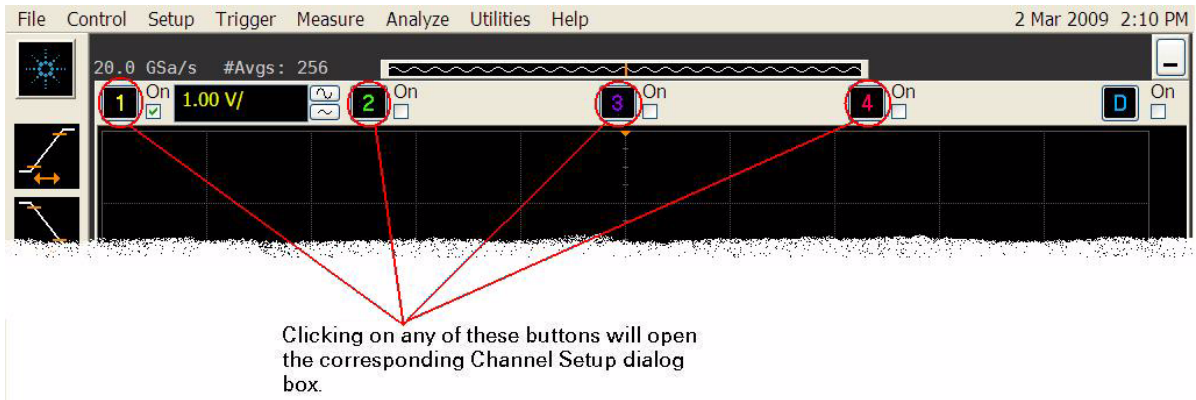


Figure 3 Shortcut buttons for Channel Setup dialog box

**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.

You will now want to set the entry controls for your specific setup. You can see the InfiniiSim entry controls in the screen shot below.

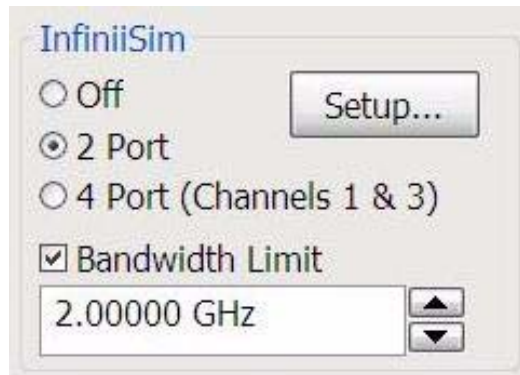


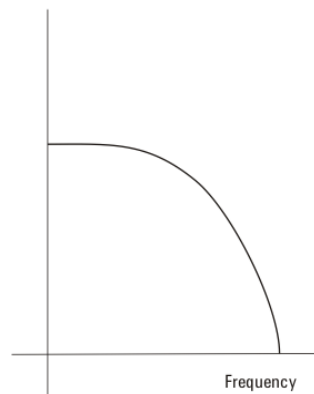
Figure 4 The InfiniiSim section in the Channel Setup dialog box

Start out by simply clicking the radio button next to **2-Port** since this chapter discusses 2-port controls.

Next, you will want to set the **Bandwidth Limit** field to your desired value (if you are going to use it). This field requires some explanation so you know how to properly set it and whether or not it is required for your specific application.

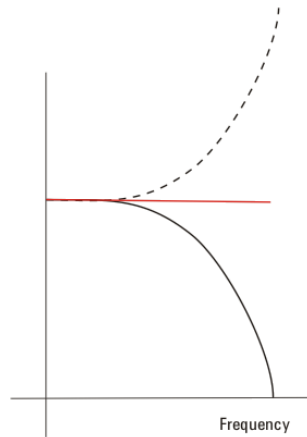
One way to understand the transformations InfiniiSim performs is in terms of the frequency response of the cable, fixture, or probe your signal travels through. Ideally, you would like to transmit a signal through a channel whose frequency response was flat over the full bandwidth of the signal so every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



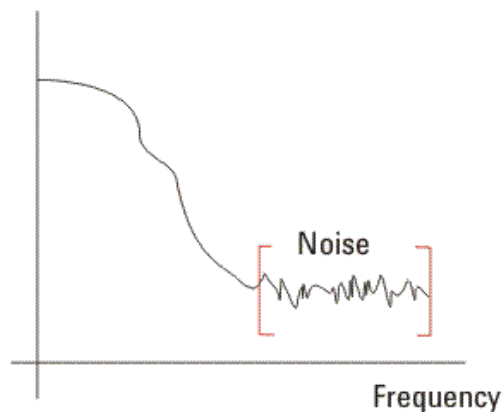
*Figure 5 Simplified plot of signal attenuation at higher frequencies*

Some InfiniiSim applications attempt to remove or correct 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 peaking filter response and the channel response yields a flat frequency spectrum (red line in plot below).



*Figure 6 Applying a peaking filter to the attenuated signal yields a flat response*

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:



*Figure 7 More realistic frequency response plot with mostly noise at high frequencies*



Notice after a certain frequency (or bandwidth) there is very little data and mostly noise. If you apply a correction via InfiniiSim as discussed above, you will end up excessively amplifying the noise. The **Bandwidth Limit** field in this dialog box allows you to minimize this noise gain by attenuating the resulting waveforms above the frequency where there is mostly just noise.

You should examine the Frequency Response Plot to see what bandwidth you should set this field to. This plot along with several others are discussed starting on [page 50](#).

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 should use this field.

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

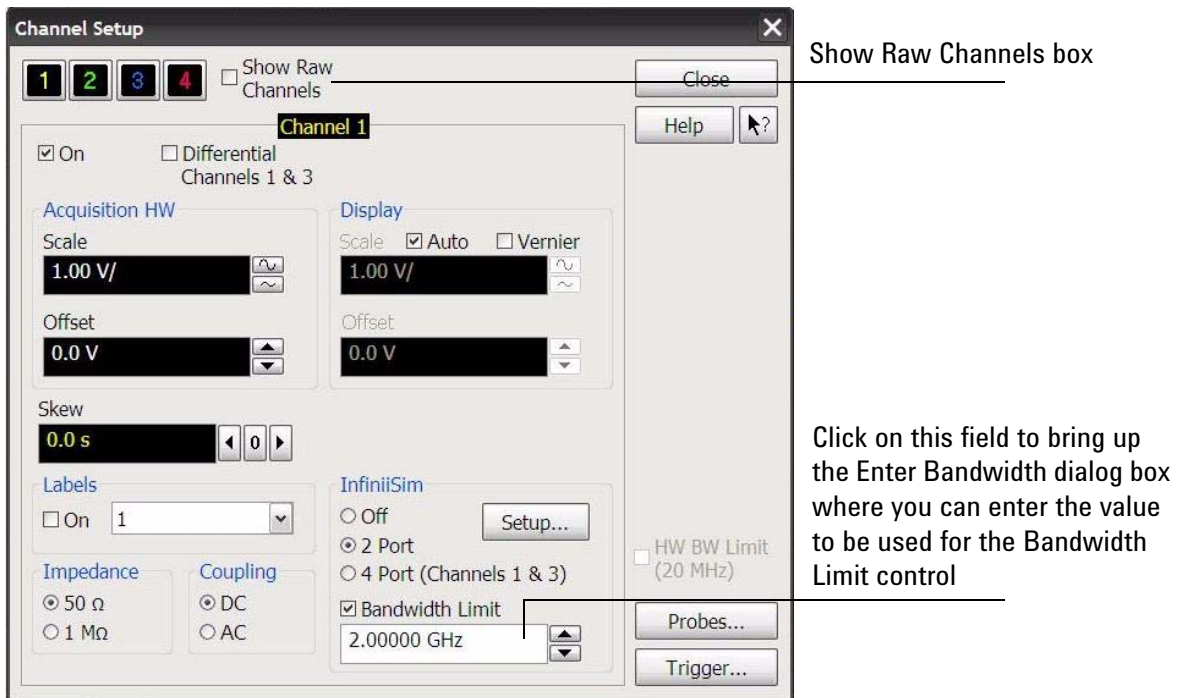


Figure 8 The Signal Bandwidth field

You will also notice that there is a box labeled **Show Raw Channels** in the **Channel Setup** dialog box. When you do things like 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 of 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.

Once you have selected 2-port and have the **Bandwidth Limit** field set, click on the **Setup** button to bring up the **InfiniiSim Setup** dialog box.

## The InfiniiSim Setup Dialog Box

Below is a screen shot showing the **InfiniiSim Setup** dialog box.

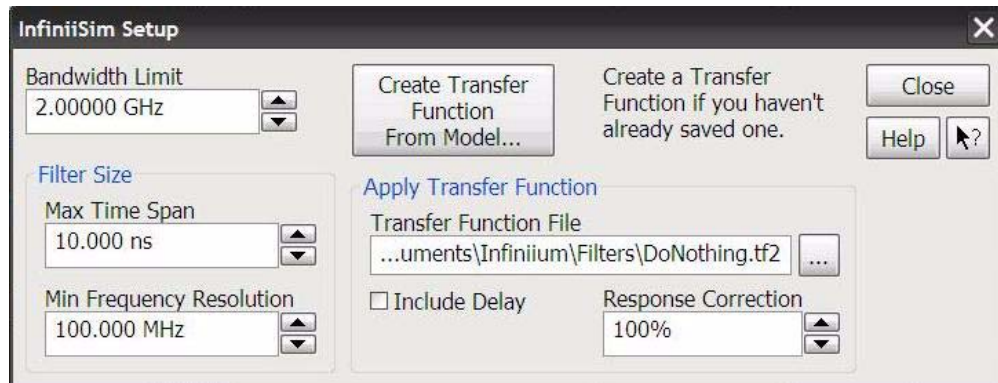


Figure 9 The InfiniiSim Setup dialog box

The first control in this dialog box allows you to adjust the Bandwidth Limit to a different value than the one entered in the **Channel Setup** dialog box (assuming you enabled the feature in the **Channel Setup** dialog box by checking the **Bandwidth Limit** box). This is just for convenience so you do not have to go back to the **Channel Setup** dialog box if you change your mind on what value to set this field to.

In the section of the **InfiniiSim Setup** dialog box entitled *Filter Size*, there are two controls: **Max Time Span** and **Min Frequency Resolution**. These two controls are reciprocals of each other so as you set one to a lower value, the other one raises to a higher value and vice versa. For example:

- Increasing the **Max Time Span** control increases the amount of time it takes for InfiniiSim to run and automatically decreases the **Min Frequency Resolution** control (which corresponds to a finer frequency resolution). Therefore, you would do this if you wanted finer frequency resolution with the trade-off of a longer run time. It basically controls the length of the FIR filter that is generated from the transfer function. You can use either the Step Response or Impulse Response plot to see if the filter is long enough to model your device (see [page 50](#)).
- 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 less frequency resolution. Generally you will want to set the Max Time Span using the Step Response or Impulse Response plot and then this Min Frequency Resolution will be set automatically.

Another control in this dialog box is the **Include Delay** box. 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. Checking this box shifts the simulated waveform in time so the two waveforms are aligned which aids in their comparison.

Finally, the two main controls in this dialog box give you an option of how you want to generate your transfer function. You can see that under the *Apply Transfer Function* section, there is a field labeled **Transfer Function File**. If you have a previously saved transfer function or one you generated elsewhere, you can click on this field and load it. A brief overview of Transfer Function files can be found on [page 15](#).

If you do not have a previously saved transfer function to load, you can create a transfer function from a model (click on the **Create Transfer Function From Model** button).

Please also note the **Response Correction** field. This field linearly applies frequency components above DC from the full potential (100%) to zero gain (0%). This allows you to trade off the amount of correction you want to apply via the transformation function versus the increase in noise at higher frequencies. In other words, you can fine-tune the amount of high-frequency noise versus the crispness of the step response edge. If you are doing rise time or jitter measurements, crisp, fast edges are required and you should, therefore, make the InfiniiSim correction as accurate as it can be by setting this field to 100%. However, if you are working with eye diagrams, you may want to limit the vertical noise apparent in the eye so you would select 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 plot / step response plot ([page 50](#)) while you adjust this field to see its effect on these (for instance, you can see how the frequency response of the transformation filter changes as you change the percentage).

## Creating a Transfer Function From a Model - An Overview of the Application Preset and Circuit Diagram View Controls

This section will discuss how to generate your transfer function from a model (accessed by clicking on the **Create Transfer Function From Model** button in the **InfiniiSim Setup** dialog box).

When you click on this button, you will see the following **InfiniiSim Model Setup** dialog box. Note that no models are shown in this default view because the **Application Preset** field is set to **None**.

The other main control on the left side of the dialog box is the **Save Transfer Function File As** field. The default will be untitled.tf2 for a 2-port transfer function file. Click on this field to bring up a keyboard where you can specify what you want to call the file of the transfer function you are about to create. Be sure to pay attention to where you save the file (note the file path) so you can find this transfer function file later.

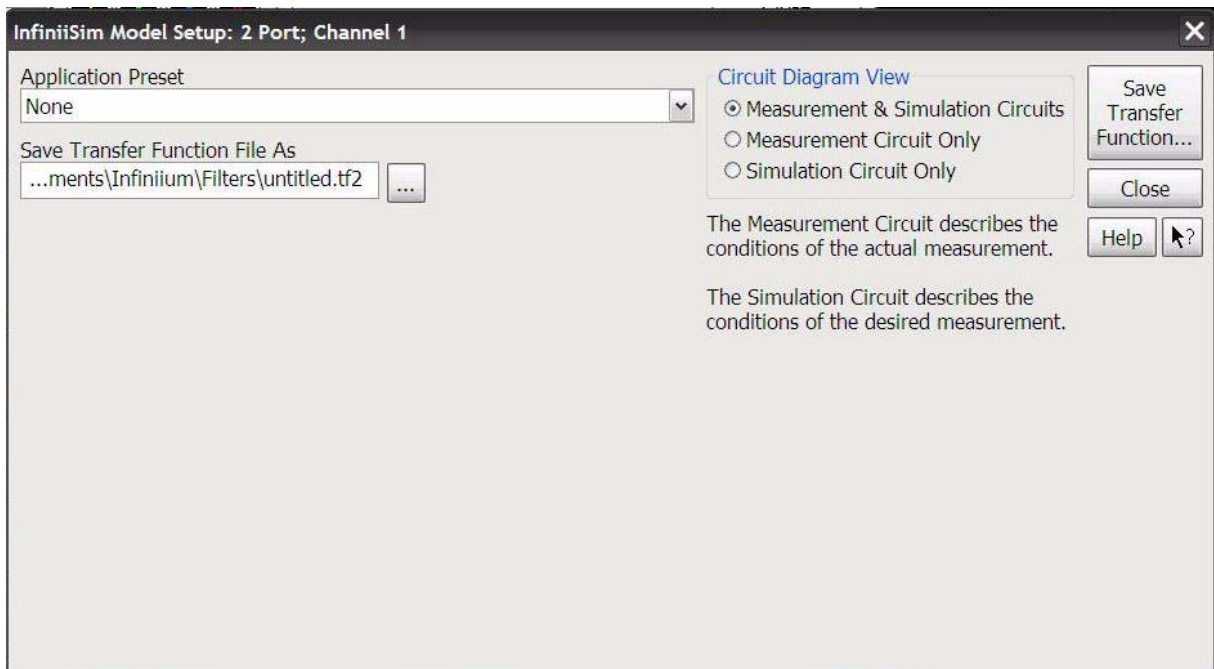


Figure 10 InfiniiSim Model Setup dialog box

*NOTE*

**The transfer function you create in this dialog box is not saved until you press the Save Transfer Function button in the upper right hand corner. If you close the dialog box without saving, your transfer function will not be applied. The application will also warn you that your file is not saved if you try to close this dialog box without saving first.**

---

The first field in this 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 are:

- None
- Add insertion loss of a fixture or cable
- Remove insertion loss of a fixture or cable
- Add all effects of a fixture or cable
- Remove all effects of a fixture or cable
- Replace one channel element with another
- Relocate the observation node of a measurement
- Remove loading effects of a probe
- Relocate the observation node of a probed measurement
- General purpose probe
- General purpose 6 blocks
- General purpose 9 blocks

For the purpose of understanding the basic components that make up these configurations, let us look at a very simple example - the **Add Insertion Loss of a Fixture or Cable** preset application.

The following circuit will appear in the dialog box when you set the **Application Preset** field to **Add Insertion Loss of a Fixture or Cable**.

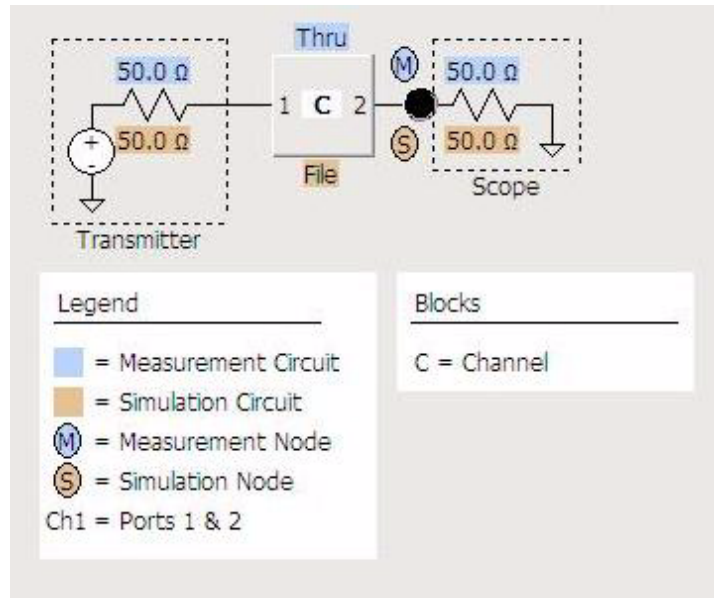


Figure 11 The circuit shown for the Add Insertion Loss of a Fixture or Cable preset

You will 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.

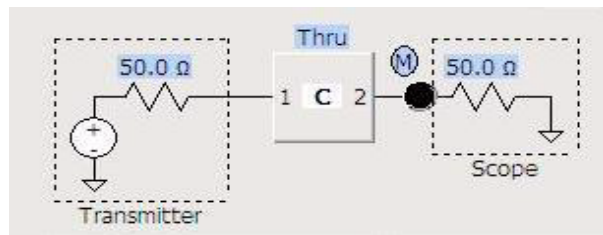
- **Measurement Circuit** - This circuit models the actual physical circuit that produced the measured waveform.
- **Simulation Circuit** - This circuit models a hypothetical electrical circuit that exhibits your desired electrical characteristics.

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.

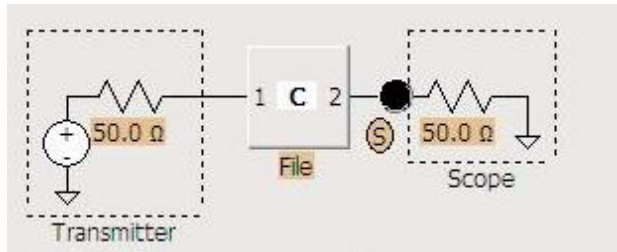
InfiniiSim analyzes these two circuits and generates a correction transfer function which, when convolved with the measured waveform, produces the desired simulated waveform (refer to [page 15](#) for more information on transfer functions and how InfiniiSim generates them).

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

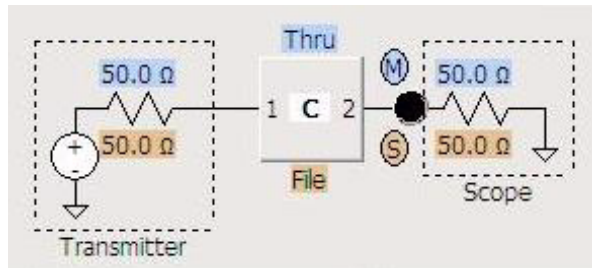
- Measurement Circuit Only



- Simulation Circuit Only



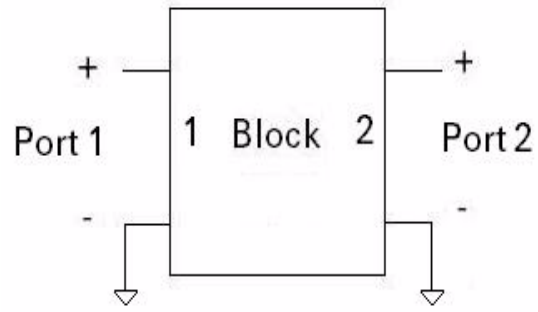
- Both Measurement and Simulation Circuits



These choices allow you to 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 block in the circuit would look like the following for a 2-port device:

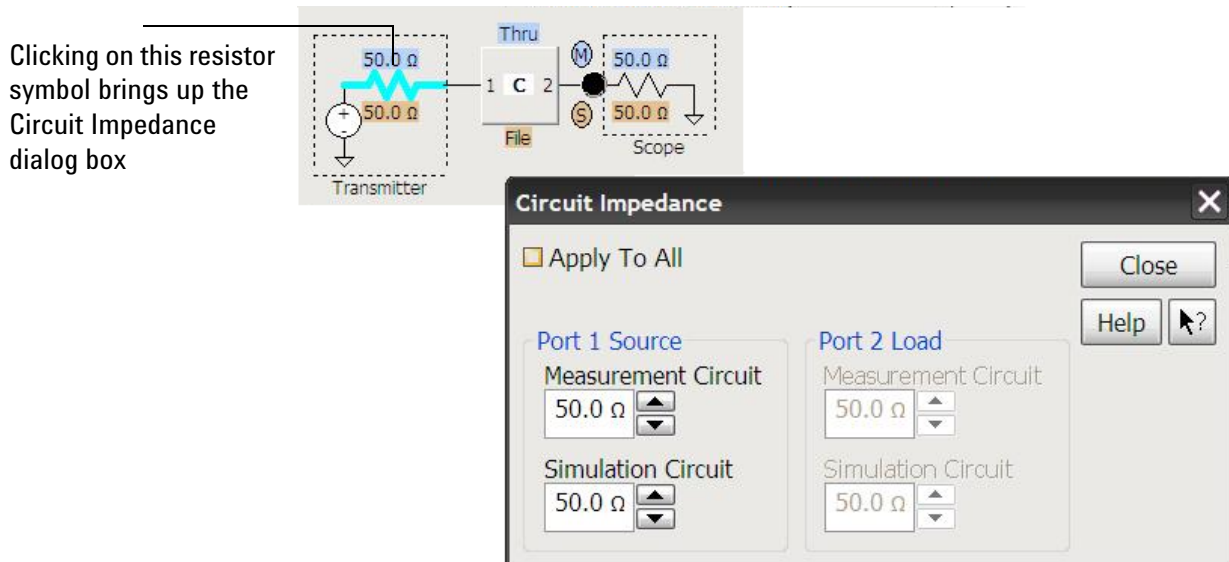


The diagrams in the GUI are completely interactive so you can click on various sections or components and make adjustments. The next section will discuss making changes to these components.

## Creating a Transfer Function From a Model - An Overview of the Circuit Component Controls

### Changing Impedance Values

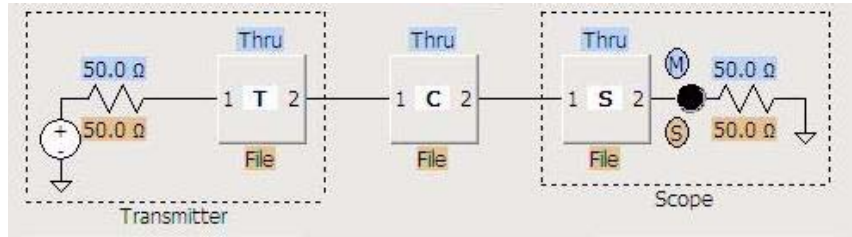
The resistor on the left side of the circuit shown in [Figure 11](#) corresponds to the impedance of the transmitter. It is defaulted to 50 Ohms, but you can change this if you need to. Simply place your cursor over the resistor (it will thicken to indicate you are on top of it) and click your mouse. You will then see the following **Circuit Impedance** dialog box.



*Figure 12 Image showing the thickening of the resistor when you place the cursor over it and the subsequent dialog box that appears when you click on the resistor*

From this dialog box, you can change the Port 1 measurement circuit and simulation circuit transmitter impedances (the resistor in the Transmitter section of the circuits) and the Port 2 scope impedances (the resistor in the Scope section of the circuits). If you want all four impedances to be identical, simply check the **Apply to All** box and set the **All Impedance** field. All of the other impedances will then be 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 scope and/or transmitter impedance because the 3 block models have separate model blocks dedicated to forming the scope impedance and transmitter impedance.



Please note that any block enclosed within dotted lines (like the T block and S block above) can have 1 port s-parameters files assigned to them with 2 port InfiniiSim. These are the only locations in a 2 port InfiniiSim circuit where you can place 1 port s-parameter files.

### Changing Measurement and Simulation Node Locations

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.

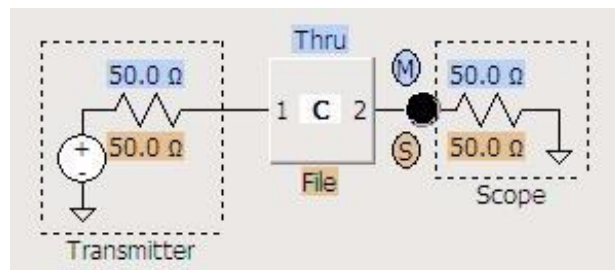
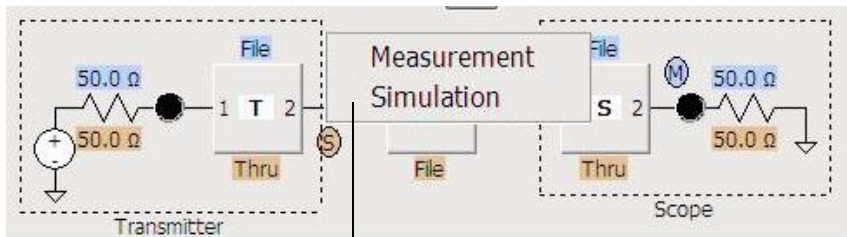


Figure 13 Model showing the measurement and simulation nodes

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). However, the measurement and simulation circuits are not being probed at the same location due to the different definitions of the channel block (the measurement circuit block is defined as Thru and the simulation circuit block is defined as File). Block definitions will be discussed later in this chapter, but this is important to realize now.

You can change the location of the measurement and/or simulation node in more complex configurations such as the 3 block Relocate the Observation Node application preset configuration shown below. You can change the nodes by clicking on one of the node locations (the large black dots in the diagram) and then selecting either measurement or simulation from the popup box (as shown below).

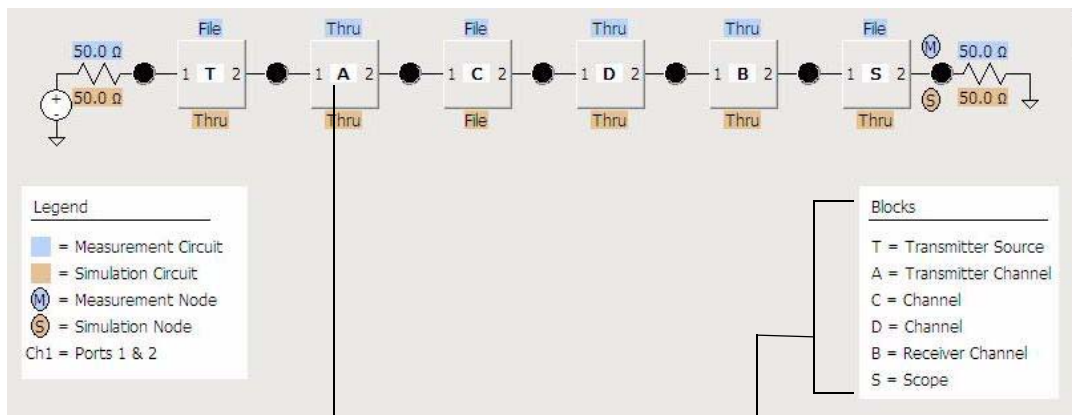


This popup appears when you click on one of the large black dots in the circuit. You can then select either **Measurement** or **Simulation** to move the corresponding observation point to that node

Figure 14 The popup box that appears when you click on one of the node locations

### Defining the Block

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. This is 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.



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

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 these blocks represent different parts of the circuit, there is no difference in terms of defining them. Each block type has 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.

When you click on a Block, you will see the following **InfiniiSim Block Setup** dialog box.

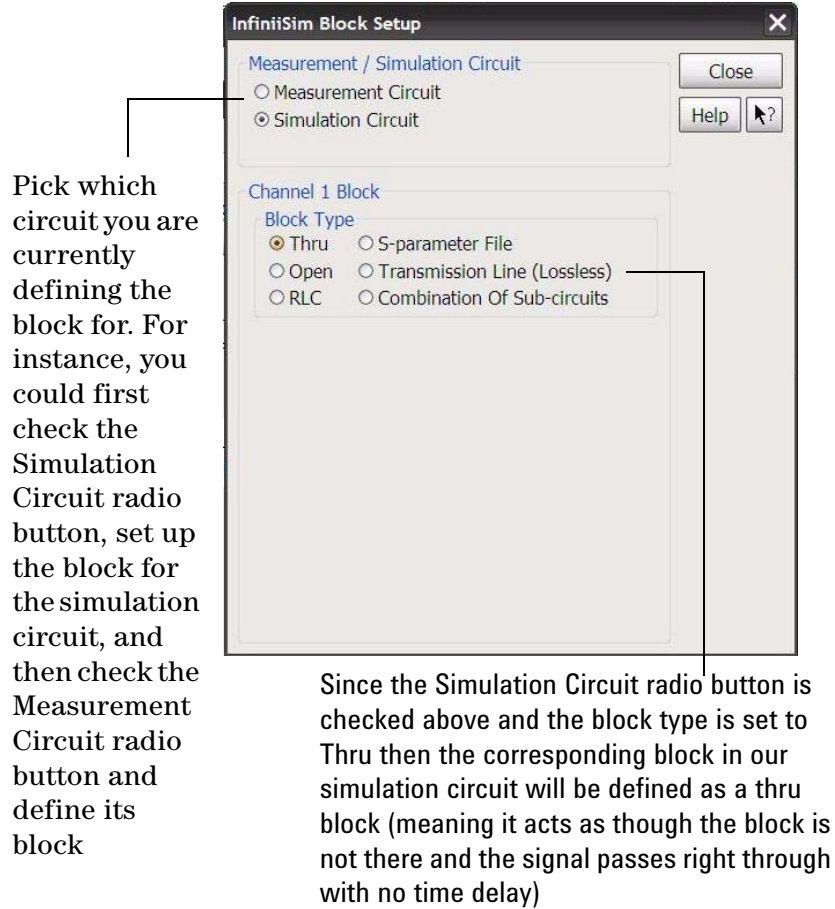


Figure 15 The InfiniiSim Block Setup dialog box

The first section (*Measurement/Simulation Circuit*), allows you to specify whether you are currently defining a block in the measurement circuit or the simulation circuit.

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

- **Thru** - Nothing happens to the signal as it passes through a Thru block. It is as though the block was not there and the signal passed right through it in zero time.
- **Open** - When you set the block to open, no signal passes through it.
- **RLC** - This allows you to set up the block as an RLC (resistor, inductor, capacitor) network.
- **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.
- **Transmission Line (lossless)** - This defines the block as a lossless transmission line. This block is similar to the Thru block except it has a characteristic impedance (whereas the thru matches the impedance to the elements on each side) and it adds some time delay (whereas the Thru block does not).
- **Combination of Subcircuits** - This selection allows you to use up to three blocks in place of the one. These three sub-blocks can be arranged in cascade, parallel, or series.

Starting on the next page, we will look at a couple of these choices in detail.

**RLC**

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

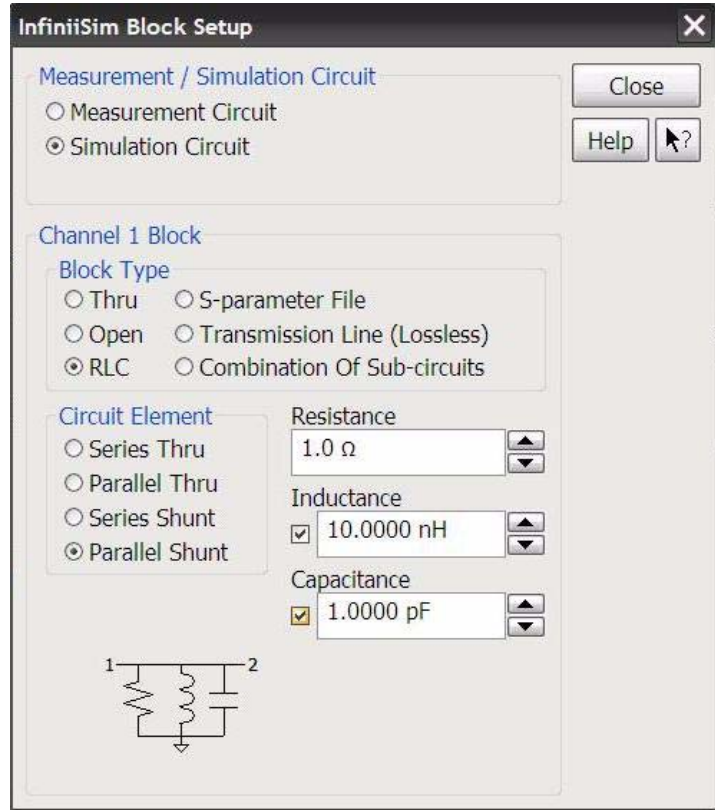


Figure 16 Dialog box when RLC is chosen for the block type

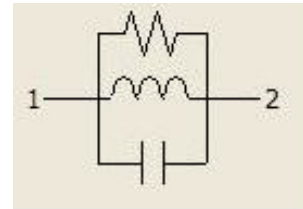


Under the *Circuit Element* section, there are four configuration options.

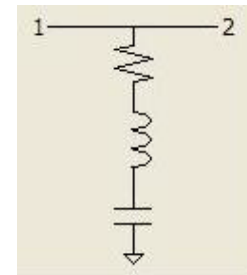
- **Series Thru**



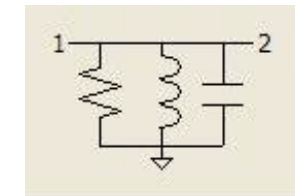
- **Parallel Thru**



- **Series Shunt**



- **Parallel Shunt**



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.

### S-parameter File

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

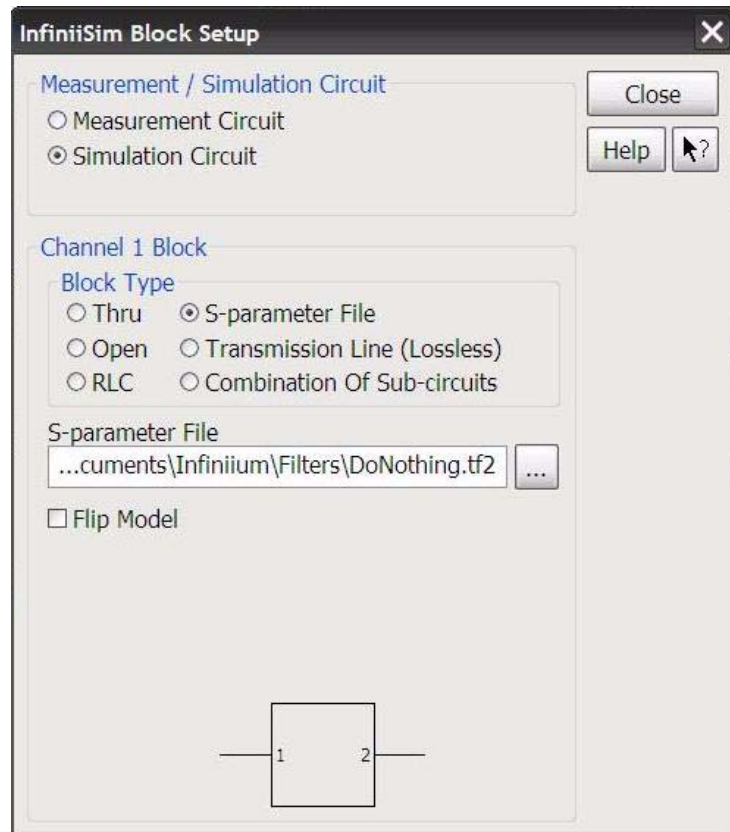


Figure 17 Dialog box when S-parameter File is chosen as the block type

You can click on the **S-parameter File** field to browse to where your saved s-parameter files are located on the oscilloscope (these S-parameter files are typically generated with a VNA, for example, and then saved on the oscilloscope hard drive - see the Application Notes listed on the InfiniiSim Main Page in the 9000 Series online help system for information on generating S-parameter files with VNAs). These can be files you created that model fixtures, cables, devices, or probes in your circuit. InfiniiSim accepts both Touchstone files (s1p, s2p, s4p) and .citi files. It is recommended that you not use pure log sweep 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 align with the way the data is formatted in your s-parameter file. Therefore, you can check the **Flip Model** box to switch ports 1 and 2.

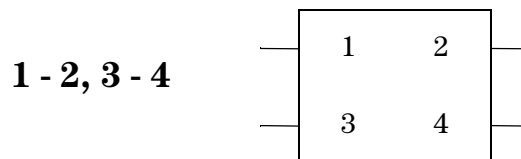
If you select a 4-port s-parameter file (.s4p file) in this step (remember we are dealing with 2-port InfiniiSim in this chapter), the oscilloscope needs to know how to handle the data in the S-parameter file (because there is more than two ports worth of information contained in a 4-port file). In this case, you will see the **Port Extraction** field. This field tells InfiniiSim how to extract the data from the 4-port S-parameter file and use it in your 2-port transformation.

For example, if you selected **1 - 2, 3 - 4** for the **4-port Numbering** field then you will be 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
- **Use Ports 3 & 4** - this means you are only extracting the port 3 and 4 information from the 4-port S-parameter file
- **Differential** - this means you are extracting both channels differentially (each channel with respect to each other)
- **Common Mode** - this means you are extracting both channels in common mode (average of each channel with respect to common ground)

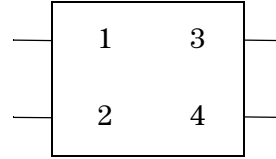
The **4-port Numbering** field enables you to specify how the data is arranged in the 4-port S-parameter file. While there is an industry standard, the InfiniiSim GUI allows you to save the S-parameter file in your desired format and then tell the application how the data is arranged in the file.

The industry standard is the following numbering system.



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

**1 - 3, 2 - 4**

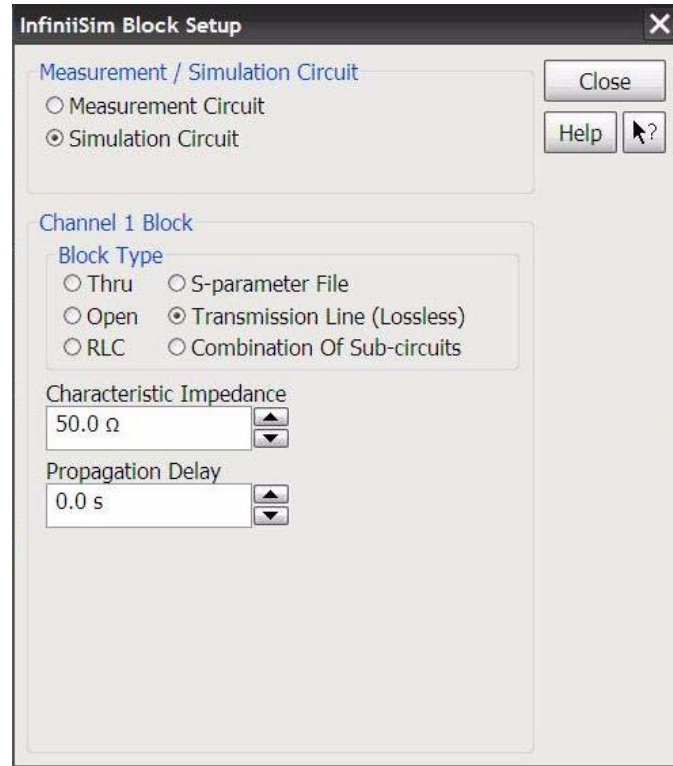


You can also use the **Flip Model** box to flip the numbers between the right hand side and the left hand side.

Flip?	Order	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	

### Transmission Line (lossless)

When you select **Transmission Line (lossless)** for the block type, the following choices appear at the bottom of the dialog box.



*Figure 18 Dialog box when Transmission Line (lossless) is chosen as the block type*

Here you can set the characteristic impedance of the transmission line in your model block as well as its propagation delay.

### Combination of Subcircuits

When you select **Combination of Subcircuits** for the block type, the following choices appear at the bottom of the dialog box.

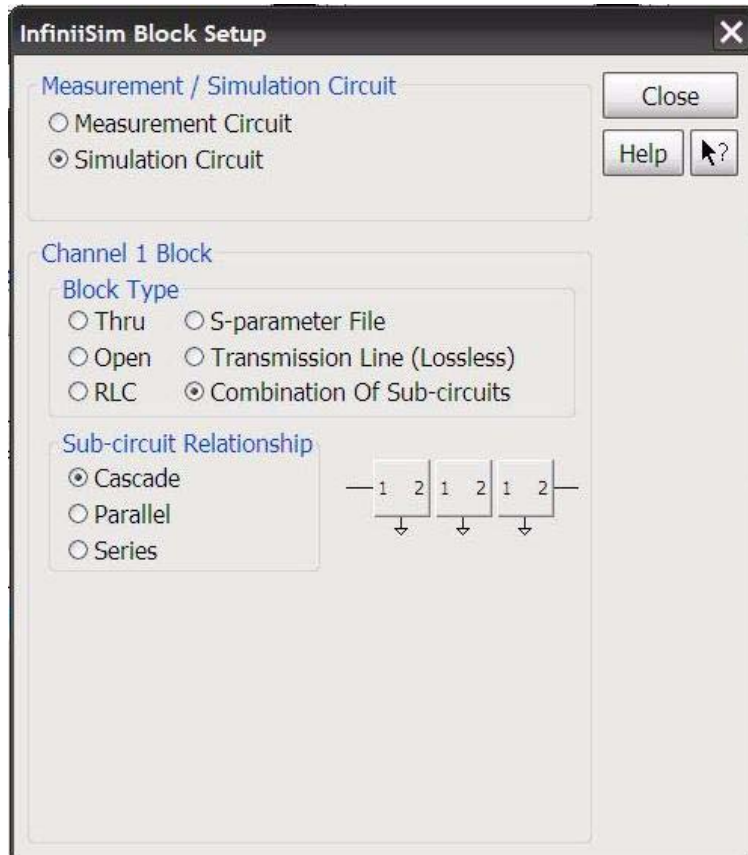
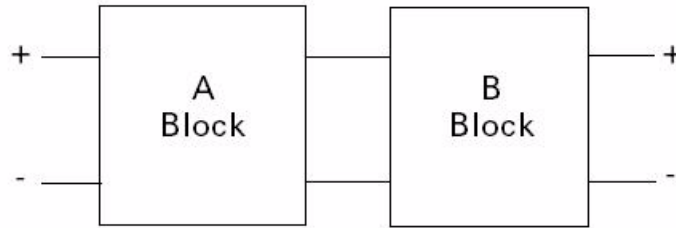


Figure 19 Dialog box when Combination of Subcircuits is chosen as the block type

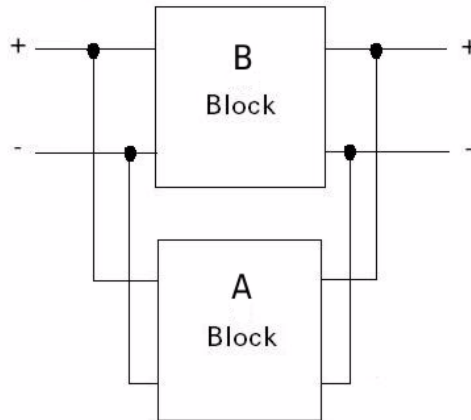
This selection enables you to use a combination of three sub-blocks in place of one block.

You have the following three choices for how to set up the three individual sub-blocks (drawings show connections for two blocks for simplicity).

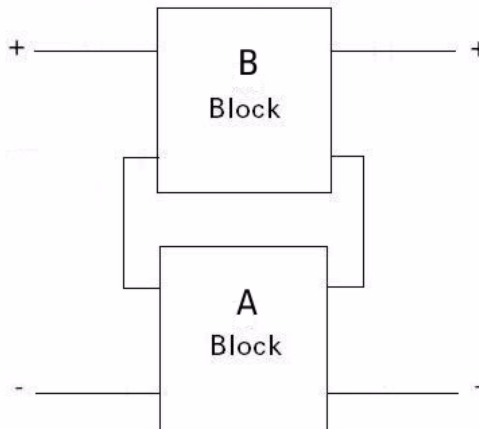
- Cascade



- Parallel



- Series



Once you select one of these three configurations, you can click on each individual block and define what type of block it is (see screen shot below).

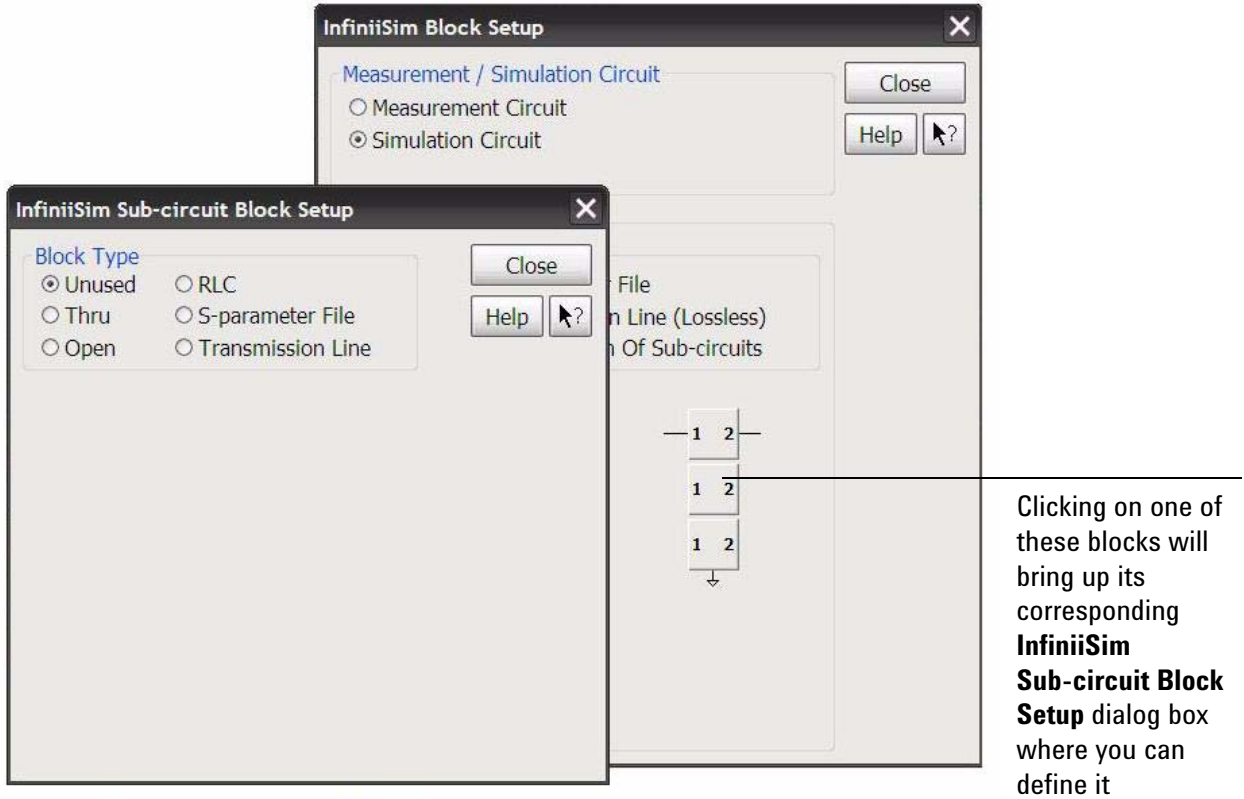


Figure 20 When you click on one of the sub-blocks a new Sub-circuit dialog box appears where you can define its type

Notice that you can choose between Thru, Open, RLC, S-parameter File, and an option called Unused. Unused simply means that the particular sub-block is not used (allowing you to use 2 sub-blocks instead of three, for example).

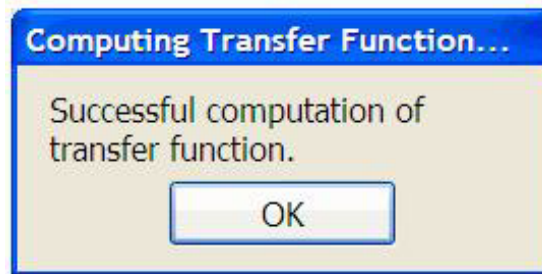
Using this Unused setting has a different impact on this specific block depending on which configuration is being used. For the Cascade structure, for example, Unused acts like a Thru block. For the Parallel structure, Unused acts like an Open block.



## Creating a Transfer Function From a Model - Generating/Saving

Once you have your measurement and simulation circuits built, you are ready to generate and save your transfer function file. Click on the **Save Transfer Function** button in the upper right hand corner of the **InfiniiSim Model Setup** dialog box. This will save it under the name (and to the location) specified in the **Save Transfer Function File As** field.

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



*Figure 21 You will see this message if InfiniiSim successfully calculated your transfer function*

If it does not successfully compute the transfer function and instead gives you an error/warning message, please consult [page 97](#) 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 controls (**Setup** -> **Channel N**) and turning InfiniiSim on or off (by selecting either **Off**, **2-port**, or **4-port**).

## The InfiniiSim Tab and Plots

When you are viewing your simulated signal, you will notice a tab on your oscilloscope's display that is labeled **InfiniiSim**.



Figure 22 The InfiniiSim tab

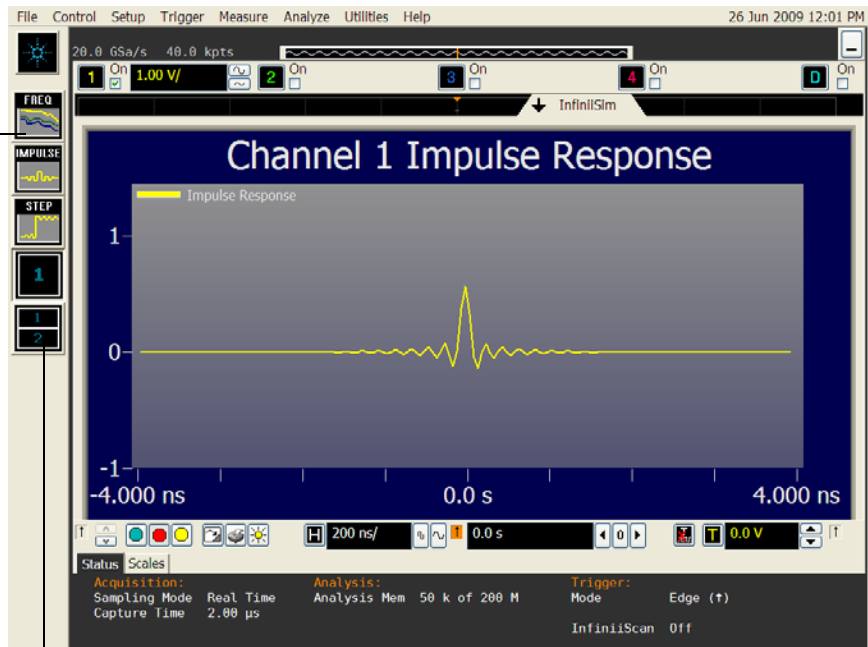
Click on this tab to bring up plots of the frequency response of the original measured signal, the simulated signal, and the filter applied by InfiniiSim. You can also see the impulse response and step response plots.

These plots are very useful when setting some of the controls in the **InfiniiSim Setup** dialog box. Before we discuss this, let us look at some of the basic controls of these plots.

### 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

Figure 23 Screen shot showing the plot icons as well as how to select whether one or two plots are displayed at the same time

Right-click on any of the plots to bring up this menu. This enables you to quickly jump back to the InfiniiSim model builder or the InfiniiSim Setup dialog box to make changes.

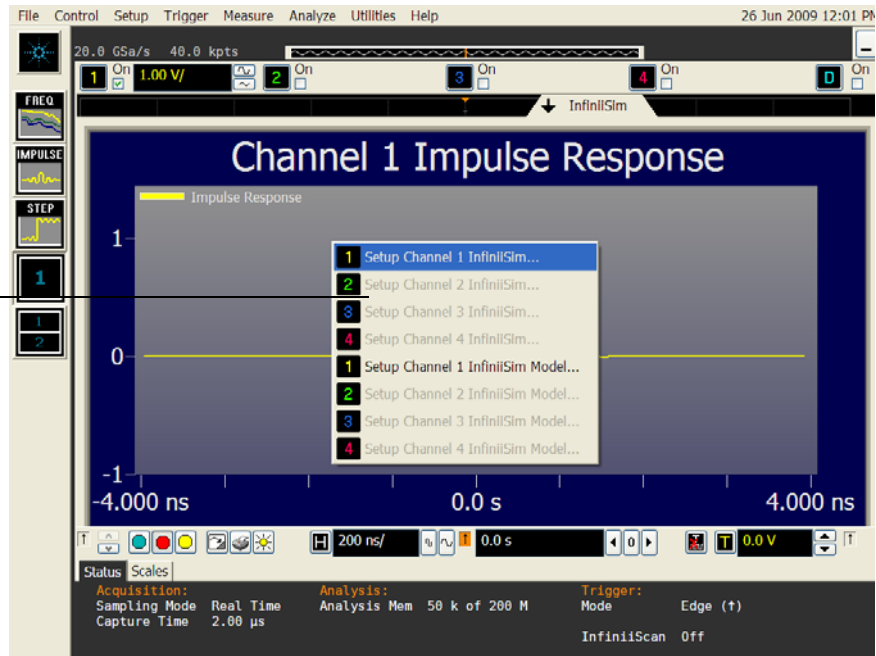


Figure 24 Screen shot showing the menu that appears when you right-click on the plot area

If you click and hold the mouse, drag a rectangle around a certain area, and then release the mouse, the area enclosed by the rectangle will be enlarged (zoom capability)

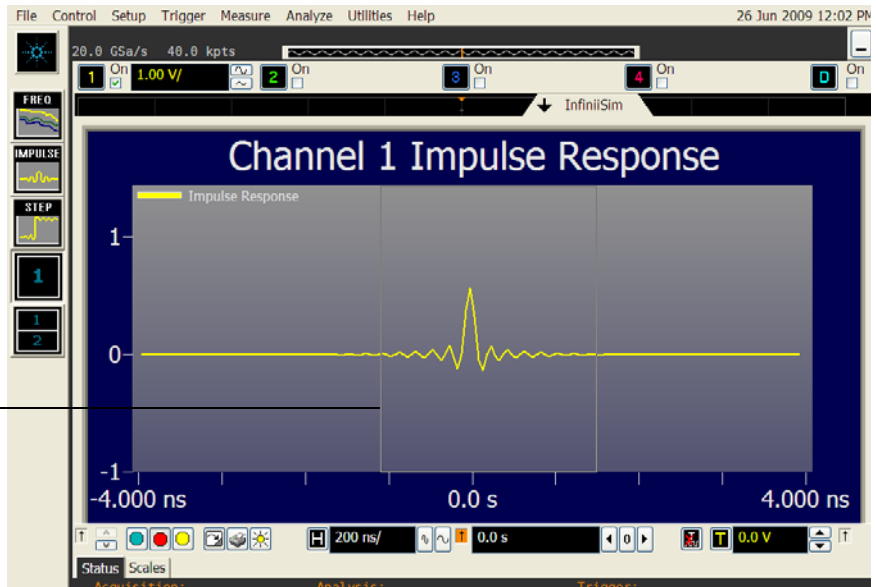
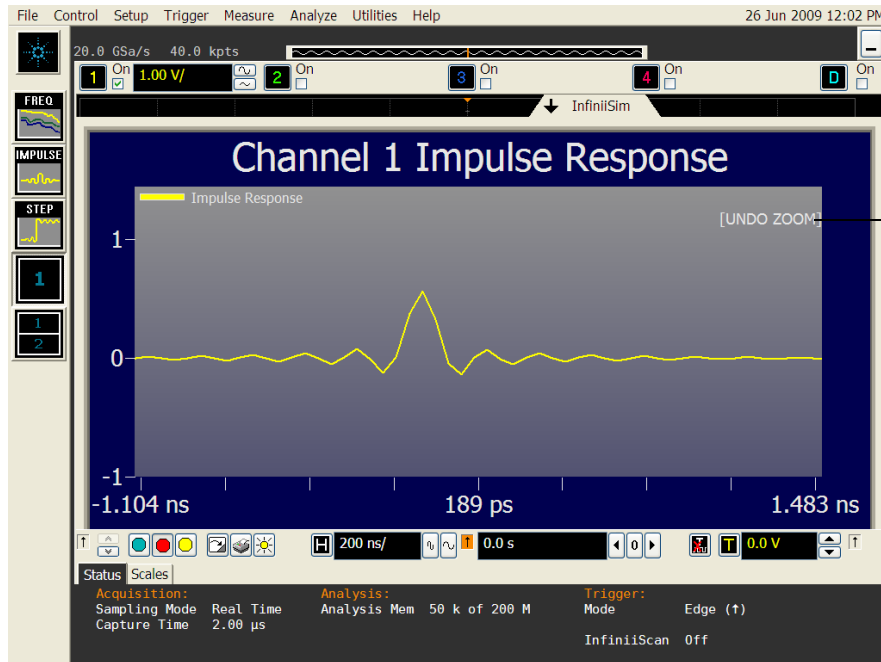


Figure 25 Screen shot showing you how to drag a rectangle around a portion of the plot to zoom in on it



After you zoom in on a plot, this **Undo Zoom** button will appear. To undo the zoom and return to the original view, click on this button.

Figure 26 Screen shot showing the location of the Undo Zoom button

Now that you know the basic controls for these plots, let us look at how they are used.

### Frequency Response Plot

This plot serves two basic functions. First, it enables you to see the measured signal’s frequency response, the simulated signal’s frequency response (the signal after InfiniiSim transforms the measured signal), and the transfer function filter’s frequency response. By investigating these plots, 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.

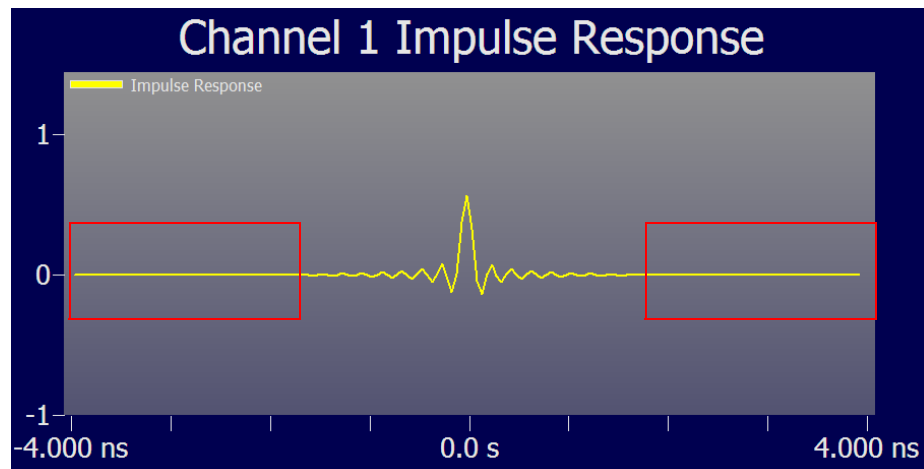
These plots also help you determine what value to use for the **Bandwidth Limit** control. Remember in the previous discussion on the **Bandwidth Limit** control that you need to determine at what frequency your signal’s frequency response is mostly just noise (if you are using this control). By investigating these plots, you can determine this frequency and set the **Bandwidth Limit** control accordingly.

The frequency range shown in the frequency response plot is 1 GHz higher than what you specify in the **Bandwidth Limit** field. This enables you to see the bandwidth start to roll off.

### Impulse Plot

The Impulse 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 the **InfiniiSim Setup** dialog box).

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



*Figure 27 Screen shot showing a simplified Impulse Response plot with the Max Time Span control set to too large of a value*

As you can see, there are large sections of the plot that are just flat. If you see this then you need to reduce the **Max Time Span** control. For example, the screen shot above was captured with the **Max Time Span** set to 8 ns. We need to reduce this until we get to a point where most of these flat regions are not shown. You can bring up the **Infiniium Setup** dialog box by right-clicking on 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 (see next page).

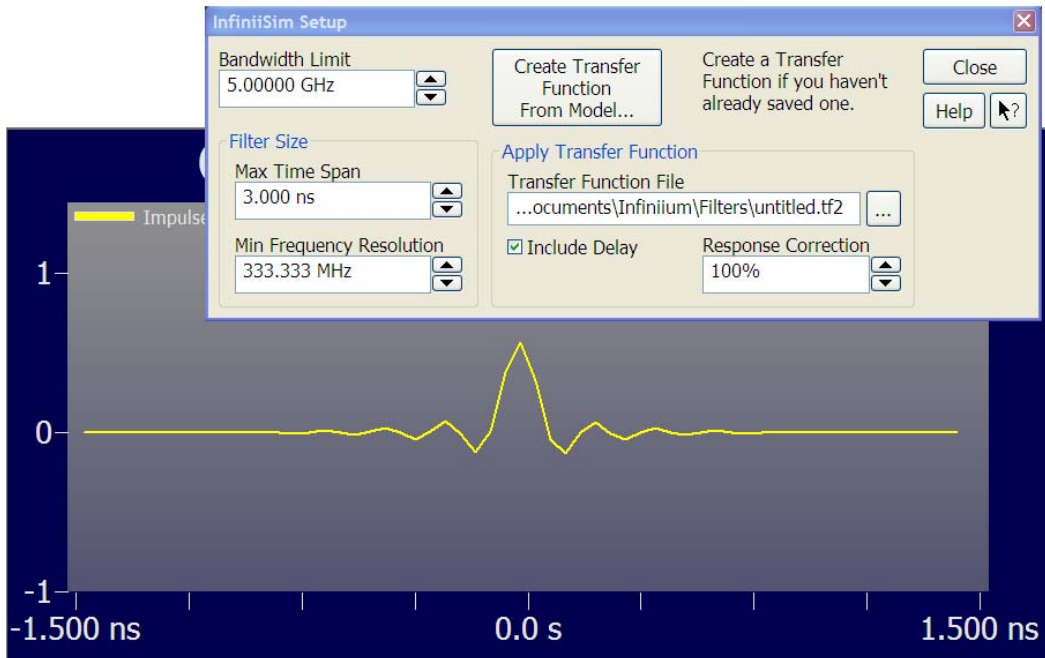


Figure 28 Screen shot showing how to adjust the Max Time Span control to 3 ns and the corresponding change in the Impulse Response plot

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

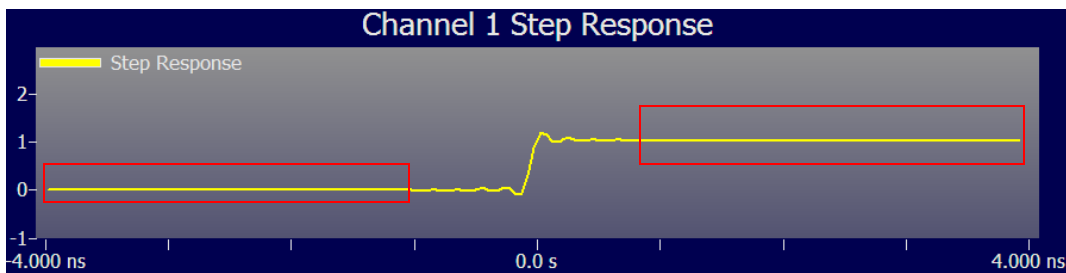
If you display the Impulse Response plot and do not see any flat regions on either side then you should increase the **Max Time Span** control until you start to see some flat regions. Then you can decrease the control and fine-tune it to its correct setting.

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 will automatically adjust since 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.

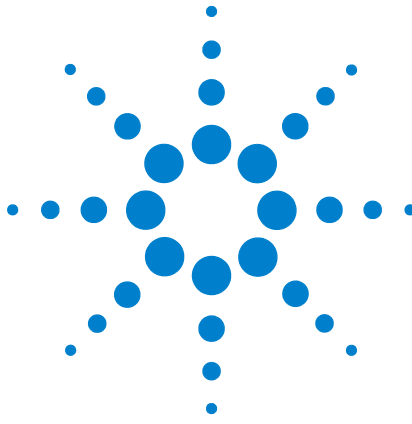
If you look at either side of the step and you see long portions of flatness, you need to shrink the **Max Time Span** control. Similarly, if you plot the Step Response and do not see any flattening off then you should increase this control.



*Figure 29* Screen shot showing a step response with the **Max Time Span** control set to too large of a value

The screen shot above shows the large flat areas enclosed by red rectangles. Reduce the **Max Time Span** control in this instance.





## 4 InfiniiSim 4-port Controls

*This chapter will discuss the InfiniiSim 4-port controls.*

### Entering the InfiniiSim Application

The entry into the InfiniiSim application is found in the **Channel Setup** dialog box. This dialog box can be accessed via **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 lower right hand corner of the **Channel Setup** dialog box.

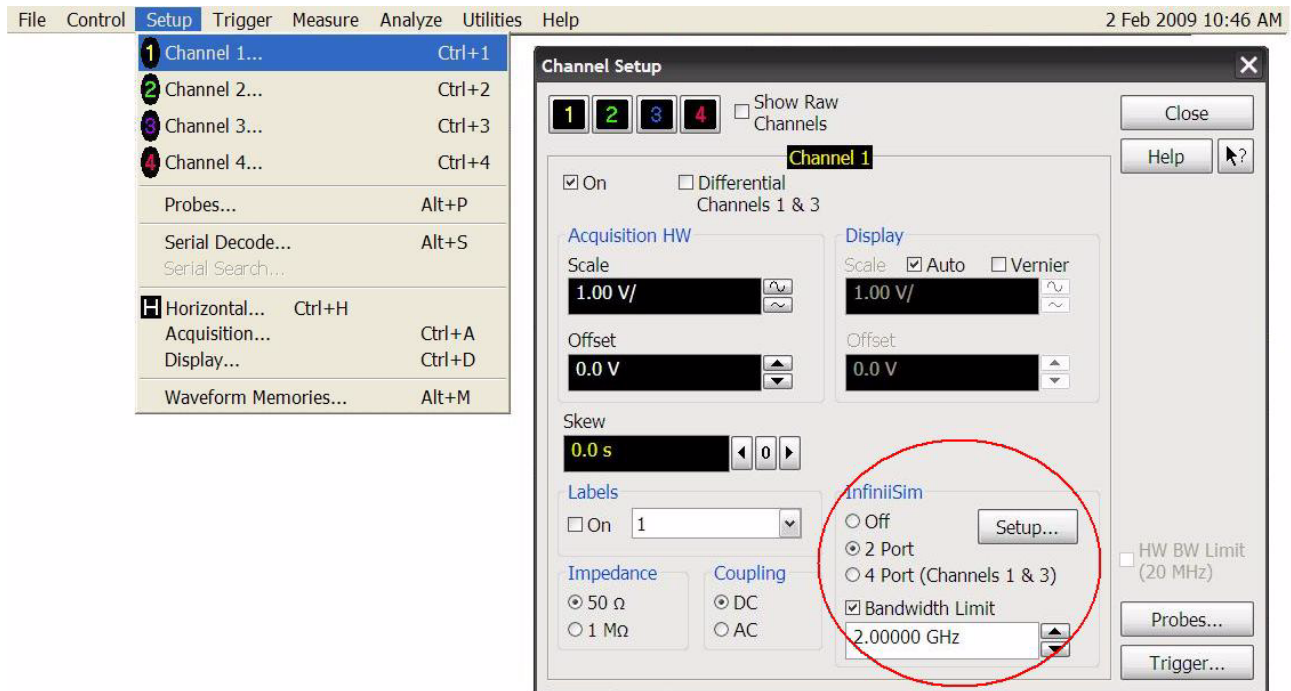


Figure 30 The entry location for the InfiniiSim application



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

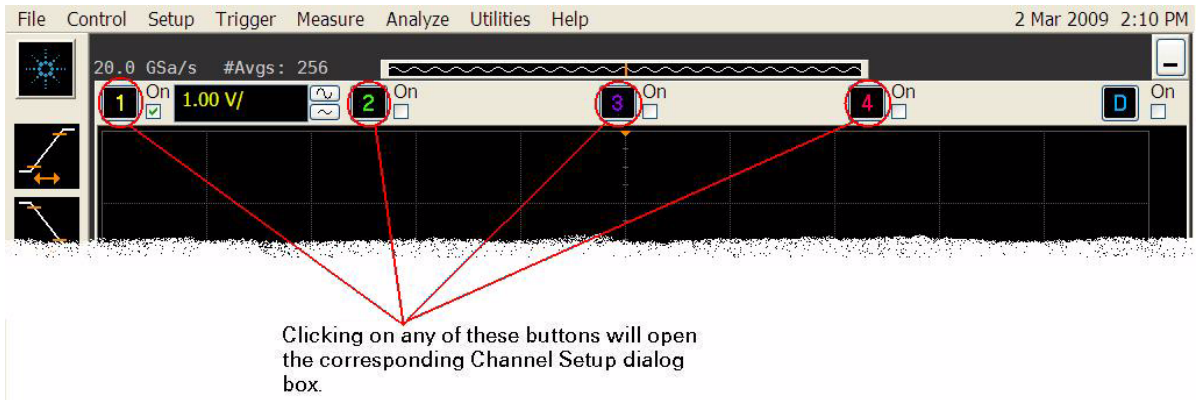


Figure 31 Shortcut buttons for Channel Setup dialog box

**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.

You will now want to set the entry controls for your specific setup. You can see the InfiniiSim entry controls in the screen shot below.



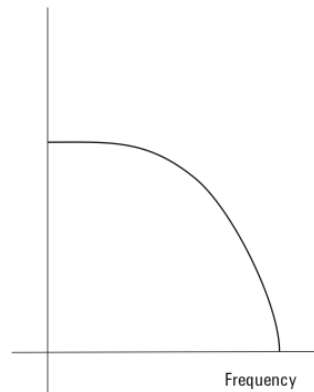
Figure 32 The InfiniiSim section in the Channel Setup dialog box

Start out by simply clicking the radio button next to **4-Port** since this chapter discusses 4-port controls.

Next, you will want to set the **Bandwidth Limit** field to your desired value (if you are going to use it). This field requires some explanation so you know how to properly set it and whether or not it is required for your specific application.

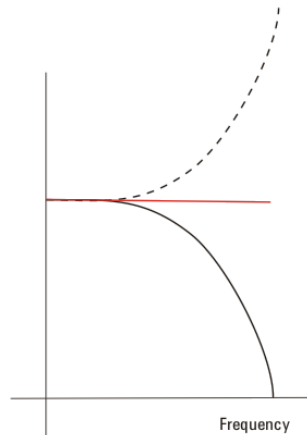
One way to understand the transformations InfiniiSim performs is in terms of the frequency response of the cable, fixture, or probe your signal travels through. Ideally, you would like to transmit a signal through a channel whose frequency response was flat over the full bandwidth of the signal so every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



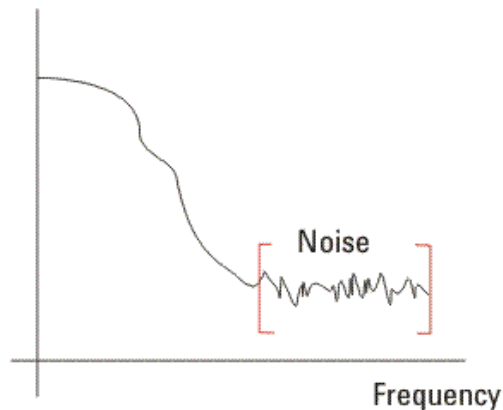
*Figure 33 Simplified plot of signal attenuation at higher frequencies*

Some InfiniiSim applications attempt to remove or correct 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 peaking filter response and the channel response yields a flat frequency spectrum (red line in plot below).



*Figure 34 Applying a peaking filter to the attenuated signal yields a flat response*

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:



*Figure 35 More realistic frequency response plot with mostly noise at high frequencies*

Notice after a certain frequency (or bandwidth) there is very little data and mostly noise. If you apply a correction via InfiniiSim as discussed above, you will end up excessively amplifying the noise. The **Bandwidth Limit** field in this dialog box allows you to minimize this noise gain by attenuating the resulting waveforms above the frequency where there is mostly just noise.

You should examine the Frequency Response Plot to see what bandwidth to set this field to. This plot along with several others are discussed starting on [page 84](#).

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 should use this field.

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

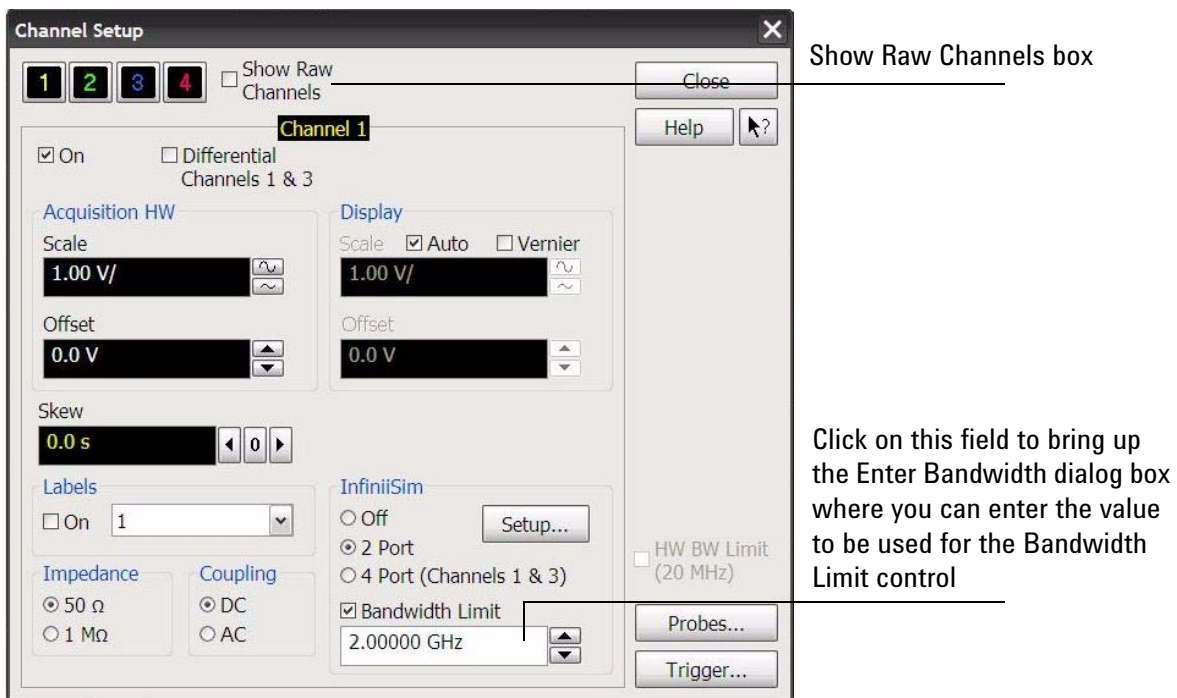


Figure 36 The Signal Bandwidth field

You will also notice that there is a box labeled **Show Raw Channels** in the **Channel Setup** dialog box. When you do things like 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 of 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.

Once you have selected 4-port and have the **Bandwidth Limit** field set, click on the **Setup** button to bring up the **InfiniiSim Setup** dialog box.

## The InfiniiSim Setup Dialog Box

Below is a screen shot showing the **InfiniiSim Setup** dialog box.

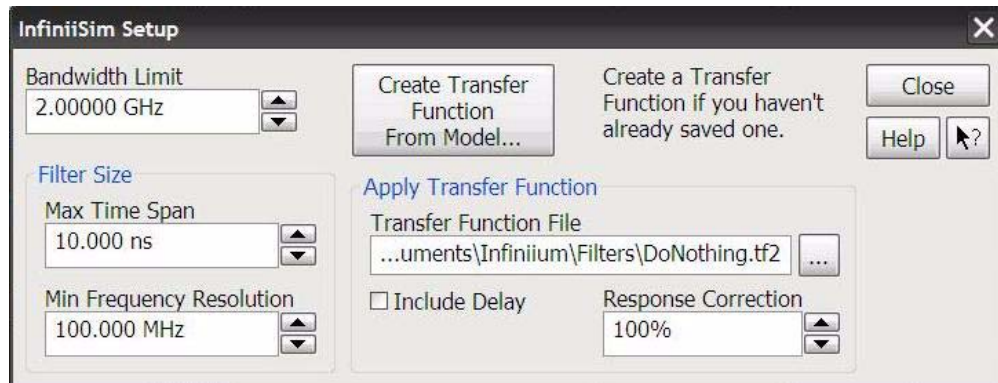


Figure 37 The InfiniiSim Setup dialog box

The first control in this dialog box allows you to adjust the Bandwidth Limit to a different value than the one entered in the **Channel Setup** dialog box (assuming you enabled the feature in the **Channel Setup** dialog box by checking the **Bandwidth Limit** box). This is just for convenience so you do not have to go back to the **Channel Setup** dialog box if you change your mind on what value to set this field to.

In the section of the **InfiniiSim Setup** dialog box entitled *Filter Size*, there are two controls: **Max Time Span** and **Min Frequency Resolution**. These two controls are reciprocals of each other so as you set one to a lower value, the other one raises to a higher value and vice versa. For example:

- Increasing the **Max Time Span** control increases the amount of time it takes for InfiniiSim to run and automatically decreases the **Min Frequency Resolution** control (which corresponds to a finer frequency resolution). Therefore, you would do this if you wanted finer frequency resolution with the trade-off of a longer run time. It basically controls the length of the FIR filter that is generated from the transfer function. You can use either the Step Response or Impulse Response plot to see if the filter is long enough to model your device (see [page 84](#)).
- 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 less frequency resolution. Generally, you will set the Max Time Span using either the Step Response or Impulse Response plot and then this Min Frequency Resolution control will automatically adjust accordingly.

Another control in this dialog box is the **Include Delay** box. 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. Checking this box shifts the simulated waveform in time so the two waveforms are aligned which aids in their comparison.

The two main controls in this dialog box give you an option of how you want to generate your transfer function. You can see that under the *Apply Transfer Function* section, there is a field labeled **Transfer Function File**. If you have a previously saved transfer function or one you generated elsewhere, you can click on this field and load it. A brief overview of Transfer Function files can be found on [page 15](#).

If you do not have a previously saved transfer function to load, you can create a transfer function from a model (click on the **Create Transfer Function From Model** button).

Please also note the **Response Correction** field. This field linearly applies frequency components above DC from the full potential (100%) to zero gain (0%). This allows you to trade off the amount of correction you want to apply via the transformation function versus the increase in noise at higher frequencies. In other words, you can fine-tune the amount of high-frequency noise versus the crispness of the step response edge. If you are doing rise time or jitter measurements, crisp, fast edges are required and you should, therefore, make the InfiniiSim correction as accurate as it can be by setting this field to 100%. However, if you are working with eye diagrams, you may want to limit the vertical noise apparent in the eye so you would select 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 plot / step response plot ([page 84](#)) while you adjust this field to see its effect on these (for instance, you can see how the frequency response of the transformation filter changes as you change the percentage).



## Creating a Transfer Function From a Model - An Overview of the Application Preset and Circuit Diagram View Controls

This section will discuss how to generate your transfer function from a model (accessed by clicking on the **Create Transfer Function From Model** button in the **InfiniiSim Setup** dialog box).

When you click on this button, you will see the following **InfiniiSim Model Setup** dialog box. Note that no models are shown in this default view because the **Application Preset** field is set to **None**.

The other main control on the left side of the dialog box is the **Save Transfer Function File As** field. The default will be untitled.tf4 for a 4-port transfer function file. Click on this field to bring up a keyboard where you can specify what you want to call the file of the transfer function you are about to create. Be sure to pay attention to where you save the file (note the file path) so you can find this transfer function file later.

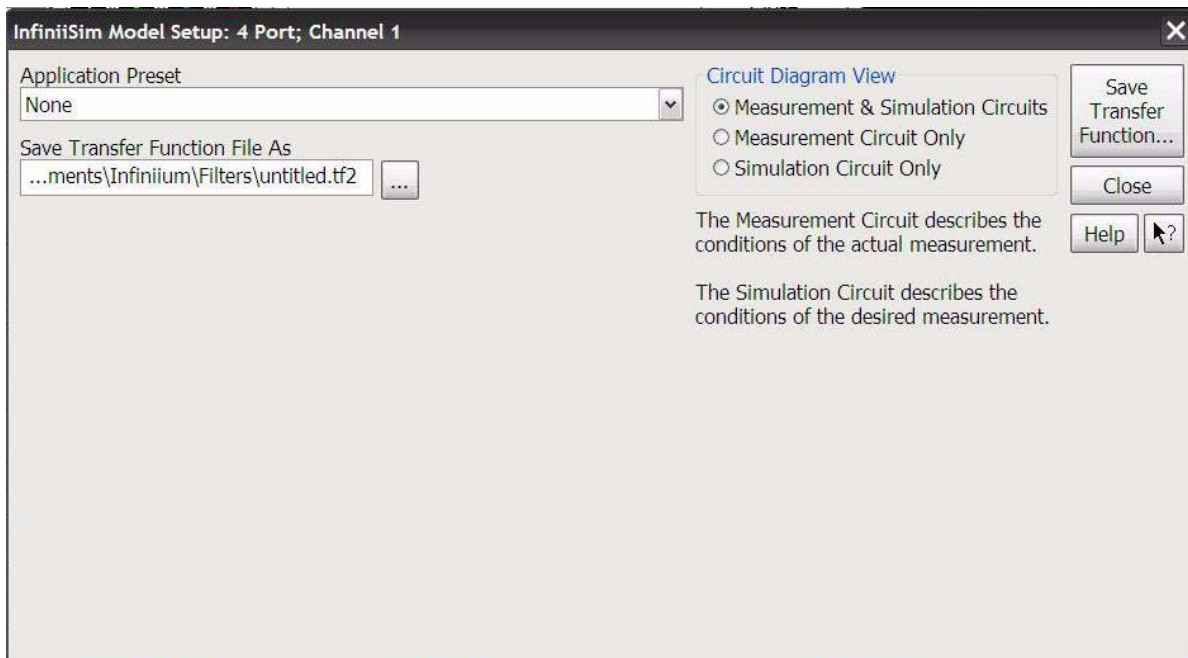


Figure 38 InfiniiSim Model Setup dialog box

*NOTE*

**The transfer function you create in this dialog box is not saved until you press the Save Transfer Function button in the upper right hand corner. If you close the dialog box without saving, your transfer function will not be applied. The application will also warn you that your file is not saved if you try to close this dialog box without saving first.**

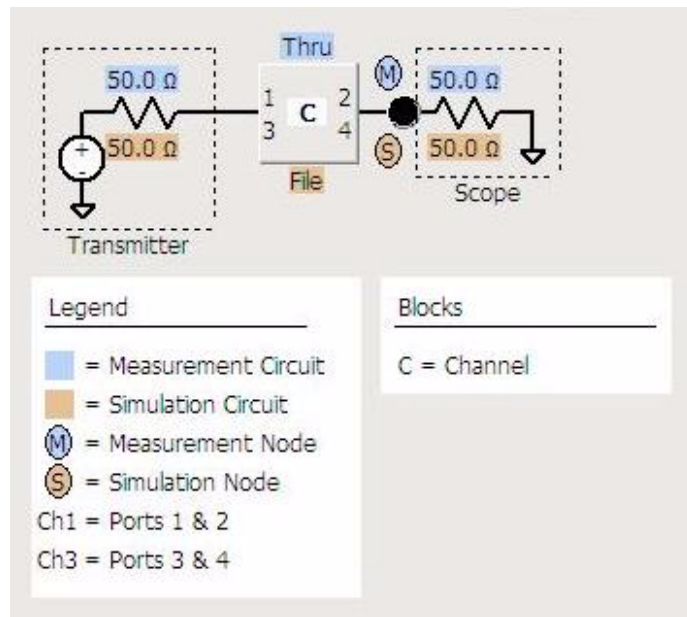
---

The first field in this 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 are:

- None
- Add insertion loss of a fixture or cable
- Remove insertion loss of a fixture or cable
- Remove scope input reflection
- Add all effects of a fixture or cable
- Remove all effects of a fixture or cable
- Replace one channel element with another
- Relocate the observation node of a measurement
- Remove loading effects of a probe
- Relocate the observation node of a probed measurement
- General purpose probe
- General purpose 6 blocks
- General purpose 9 blocks

For the purpose of understanding the basic components that make up these configurations, let us look at a very simple example - the **Add Insertion Loss of a Fixture or Cable** preset application.

The following circuit will appear in the dialog box when you set the **Application Preset** field to **Add Insertion Loss of a Fixture or Cable**.



*Figure 39 The circuit shown for the Add Insertion Loss of a Fixture or Cable preset*

You will 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.

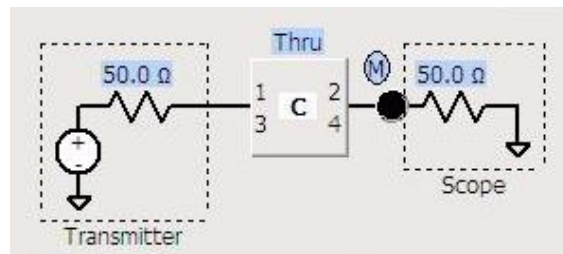
- **Measurement Circuit** - This circuit models the actual physical circuit that produced the measured waveform.
- **Simulation Circuit** - This circuit models a hypothetical electrical circuit that exhibits your desired electrical characteristics.

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.

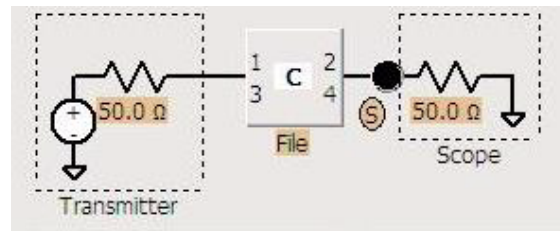
InfiniiSim analyzes these two circuits and generates a correction transfer function which, when convolved with the measured waveform, produces the desired simulated waveform (refer to [page 15](#) for more information on transfer functions and how InfiniiSim generates them).

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

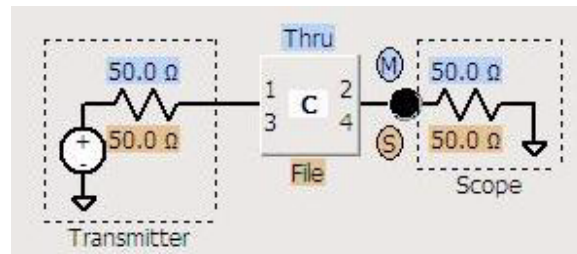
- Measurement Circuit Only



- Simulation Circuit Only



- Both Measurement and Simulation Circuits



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

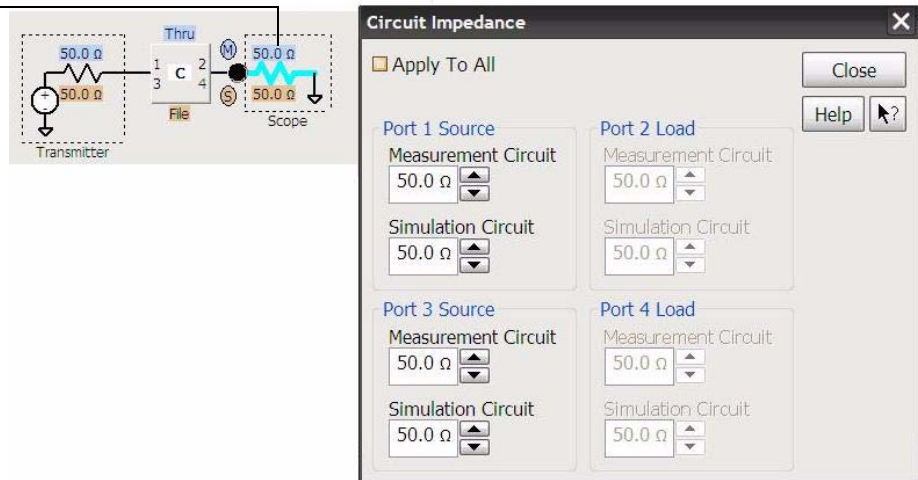
The diagrams in the GUI are completely interactive so you can click on various sections or components and make adjustments. The next section will discuss making changes to these components.

## Creating a Transfer Function From a Model - An Overview of the Circuit Component Controls

### Changing Impedance Values

The resistor on the left side of the circuit shown in [Figure 39](#) corresponds to the impedance of the transmitter. It is defaulted to 50 Ohms, but you can change this if you need to. Simply place your cursor over the resistor (it will thicken to indicate you are on top of it) and click your mouse. You will then see the following **Circuit Impedance** dialog box.

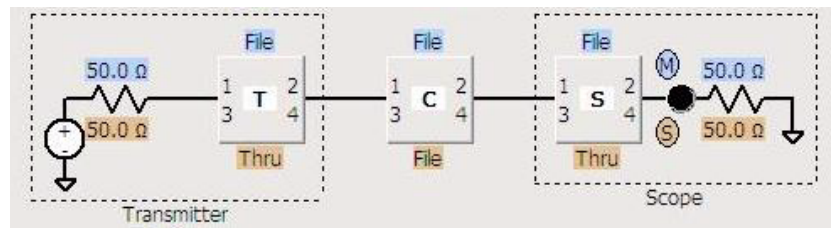
Clicking on this resistor symbol brings up the Circuit Impedance dialog box



*Figure 40 Image showing the thickening of the resistor when you place the cursor over it and the subsequent dialog box that appears when you click on the resistor*

From this dialog box, you can change the Port 1, Port 2, Port 3, and Port 4 impedances for both the measurement and the simulation circuit. If you want all eight impedances to be identical, simply check the **Apply to All** box and set the **All Impedance** field. All of the other impedances will then be 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 scope and/or transmitter impedance because the 3 block models have separate model blocks dedicated to forming the scope impedance and transmitter impedance.



Please also note that any block enclosed within dotted lines (like the T block and S block above) can have 2 port s-parameter files assigned to them. These are the only locations in a 4 port InfiniiSim circuit where you can place 2 port s-parameter files.

### Changing Measurement and Simulation Node Locations

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.

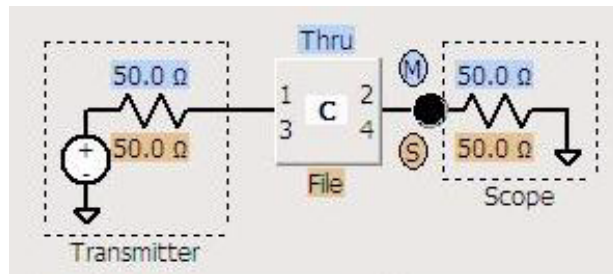
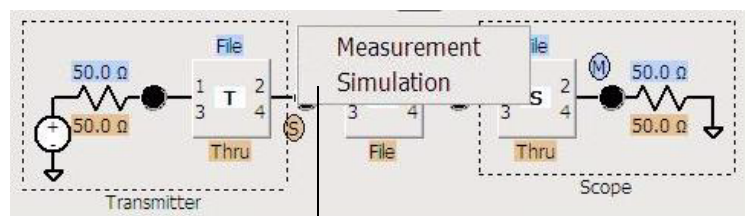


Figure 41 Model showing the measurement and simulation nodes

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). However, the measurement and simulation circuits are not being probed at the same location due to the different definitions of the channel block (the measurement circuit block is defined as Thru and the simulation circuit block is defined as File). Block definitions will be discussed later in this chapter, but this is important to realize now.

You can change the location of the measurement and/or simulation node in more complex configurations such as the 3 block Relocate the Observation Node application preset configuration below. You can change the nodes by clicking on one of the node locations (the large black dots in the diagram) and then selecting either measurement or simulation from the popup box as shown below.

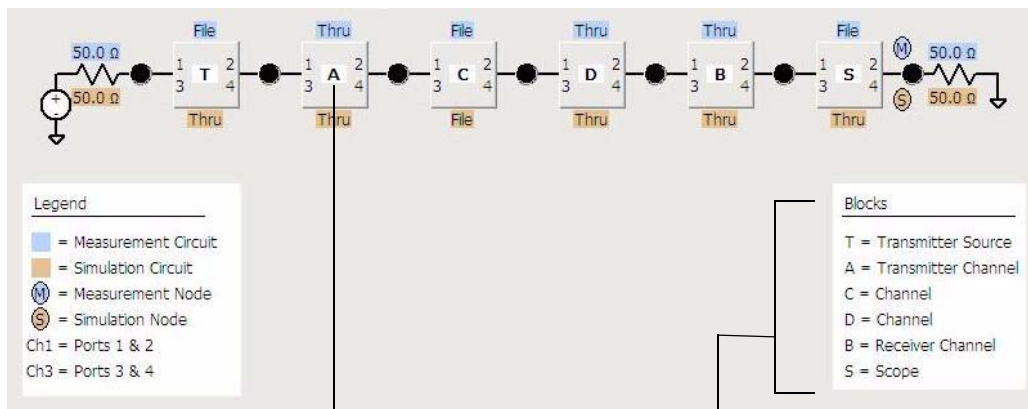


This popup appears when you click on one of the large black dots in the circuit. You can then select either **Measurement** or **Simulation** to move the corresponding observation point to that node

Figure 42 The popup box that appears when you click on one of the node locations

## Defining the Block

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. This is 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.



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

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 these blocks represent different parts of the circuit, there is no difference in terms of defining them. Each block type has 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.



When you click on a Block, you will see the following **InfiniiSim Block Setup** dialog box.

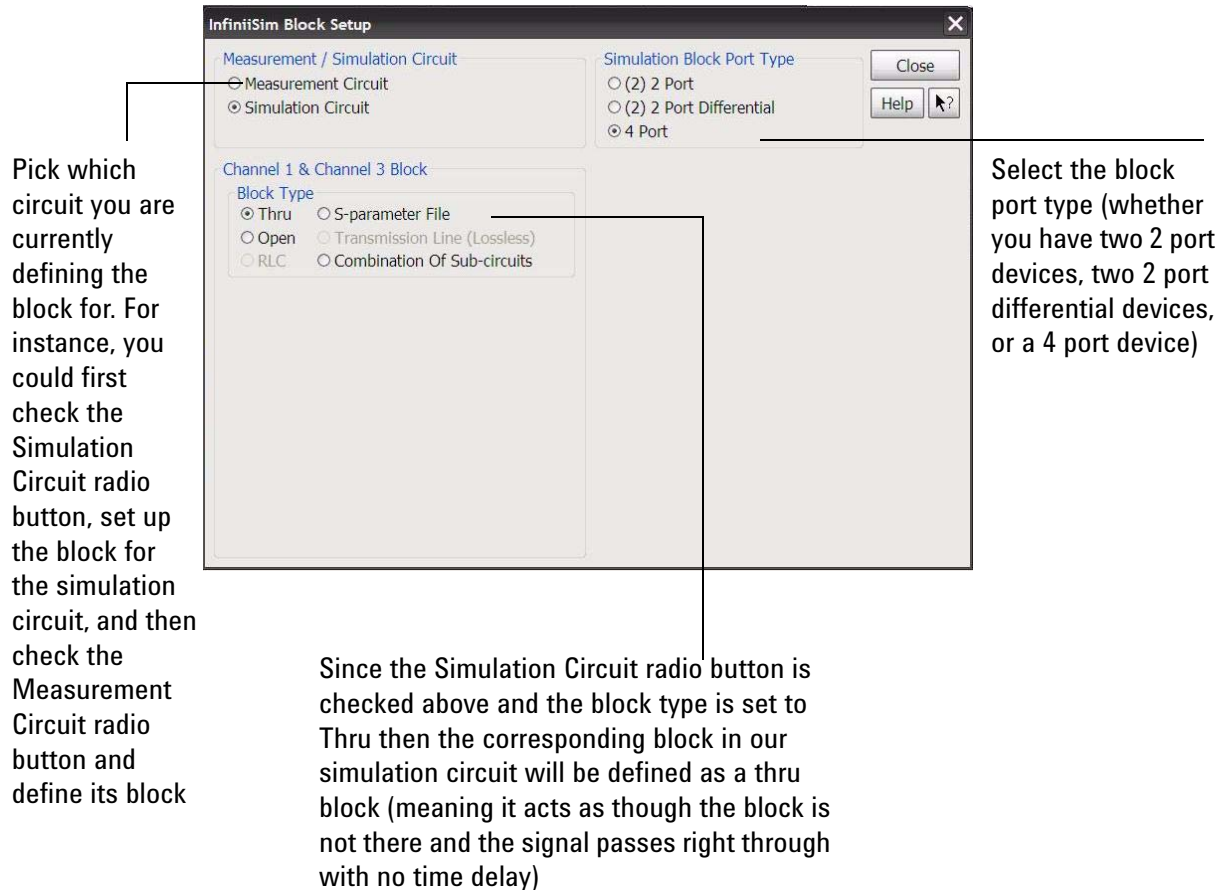


Figure 43 The InfiniiSim Block Setup dialog box

The first section (*Measurement/Simulation Circuit*), allows you to specify whether you are currently defining a block in the measurement circuit or the simulation circuit.

You can also set the Block Port Type in the upper right hand corner. Here you can choose whether there are two 2 port devices, two 2 port differential devices, or a four port device. If you choose either **(2) 2 Port** or **(2) 2 Port Differential** then you will actually be setting up two blocks in this dialog box (the second set of commands will appear when these are selected).

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

- **Thru** - Nothing happens to the signal as it passes through a Thru block. It is as though the block was not there and the signal passed right through it in zero time.
- **Open** - When you set the block to open, no signal passes through it.
- **RLC** - This allows you to set up the block as a RLC (resistor, inductor, capacitor) network.
- **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.
- **Transmission Line (lossless)** - This defines the block as a lossless transmission line. This block is similar to the Thru block except it has a characteristic impedance (whereas the thru matches the impedance to the elements on each side) and it adds some time delay (whereas the Thru block does not).
- **Combination of Subcircuits** - This selection allows you to use up to three blocks in place of the one. These three sub-blocks can be arranged in cascade, parallel, or series.

Starting on the next page, we will look at a couple of these choices in detail.

## RLC

When you select **RLC** for the block type (not available in 4 port Block Port Type mode), the dialog box will look like the following.

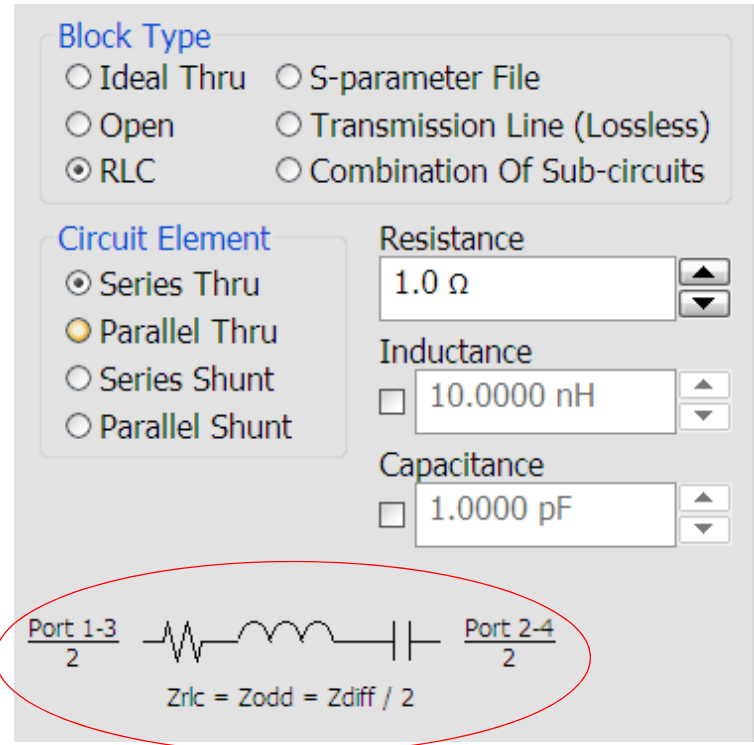


Figure 44 Dialog box when RLC is chosen for the block type

Under the *Circuit Element* section, there are four configuration options. When you select one of these options, the basic circuit configuration will be shown in the dialog box (see circled area 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.

### S-parameter File

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

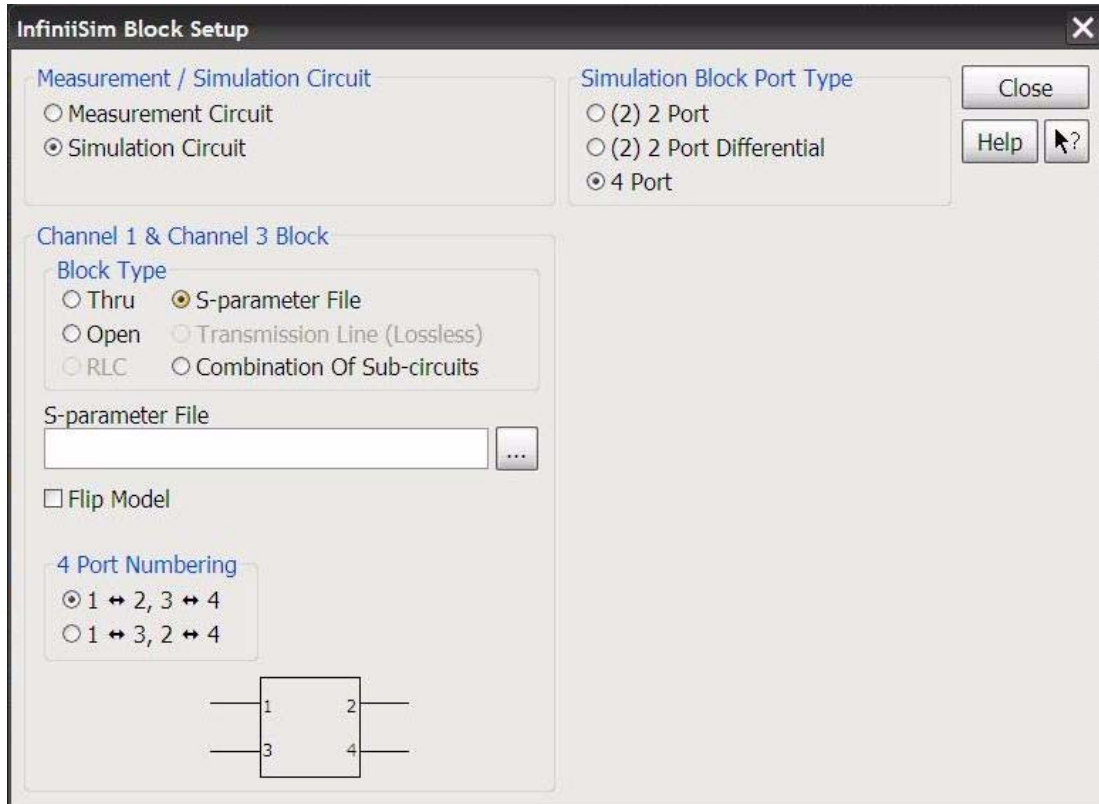


Figure 45 Dialog box when S-parameter File is chosen as the block type

You can click on the **S-parameter File** field to browse to where your saved s-parameter files are located on the oscilloscope (these S-parameter files are typically generated with a VNA, for example, and then saved on the oscilloscope hard drive - see the Application Notes listed on the InfiniiSim Main Page in the oscilloscope help system for information on generating S-parameter files with VNAs). These can be files you created that model fixtures, cables, devices, or probes in your circuit. InfiniiSim accepts both Touchstone files (.s1p, .s2p, .s4p) and .citi files. It is not recommended that you use files that use log sweep.

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

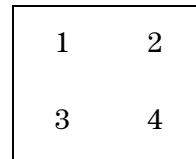
align with the way the data is formatted in your s-parameter file. Therefore, you can check the **Flip Model** box to have the ports switch sides.

Depending upon your settings, you may also see the **4 Port Numbering** field. This field tells InfiniiSim how the data is formatted in the 4-port s-parameter file (it works in conjunction with the **Flip Model** control).

While there is an industry standard, the InfiniiSim GUI allows you to save the s-parameter file in your desired format and then tell the application how the data is arranged in the file.

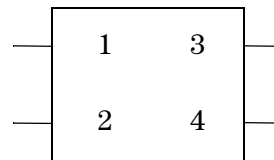
The industry standard is the following numbering system.

**1 - 2, 3 - 4**

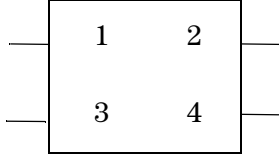
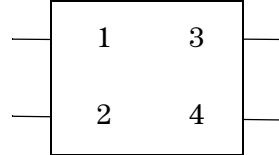
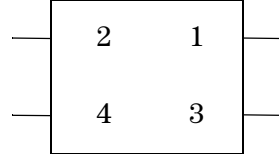
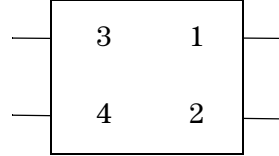


However, you can tell InfiniiSim that your file actually uses the following numbering system.

**1 - 3, 2 - 4**



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

Flip?	Order	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	

### Transmission Line (lossless)

When you select **Transmission Line (lossless)** for the block type, the following choices appear at the bottom of the dialog box.

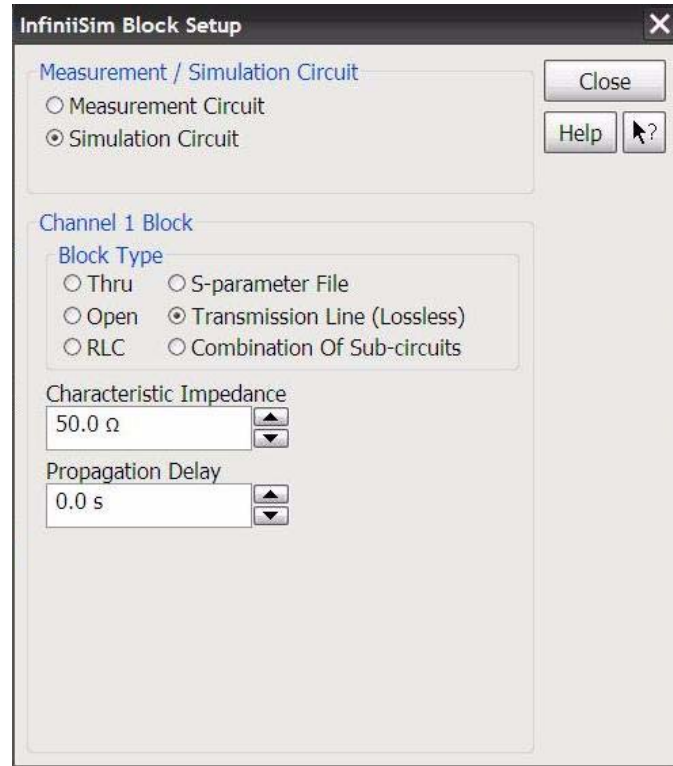


Figure 46 Dialog box when Transmission Line (lossless) is chosen as the block type

Here you can set the characteristic impedance of the transmission line in your model block as well as its propagation delay.

### Combination of Subcircuits

When you select **Combination of Subcircuits** for the block type, the following choices appear at the bottom of the dialog box.

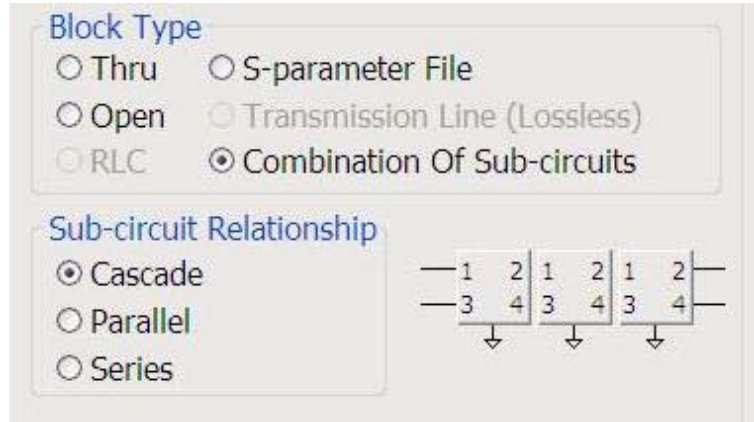


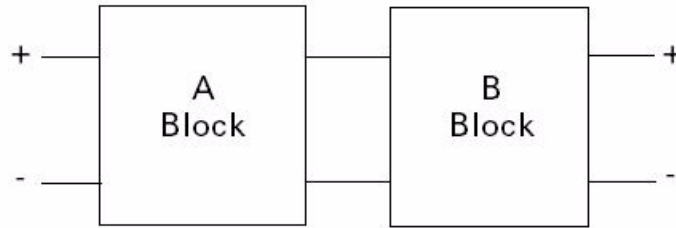
Figure 47 Dialog box when Combination of Subcircuits is chosen as the block type

This selection enables you to use a combination of three sub-blocks in place of one block.

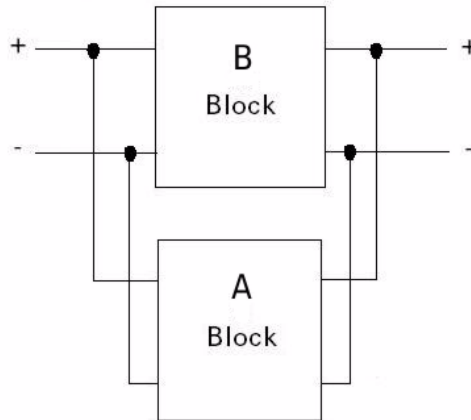


You have the following three choices for how to configure the three individual sub-blocks (pictures show 2 port connections with two blocks for simplicity).

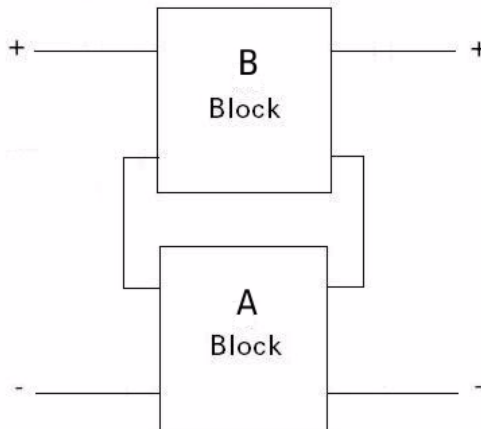
- **Cascade**



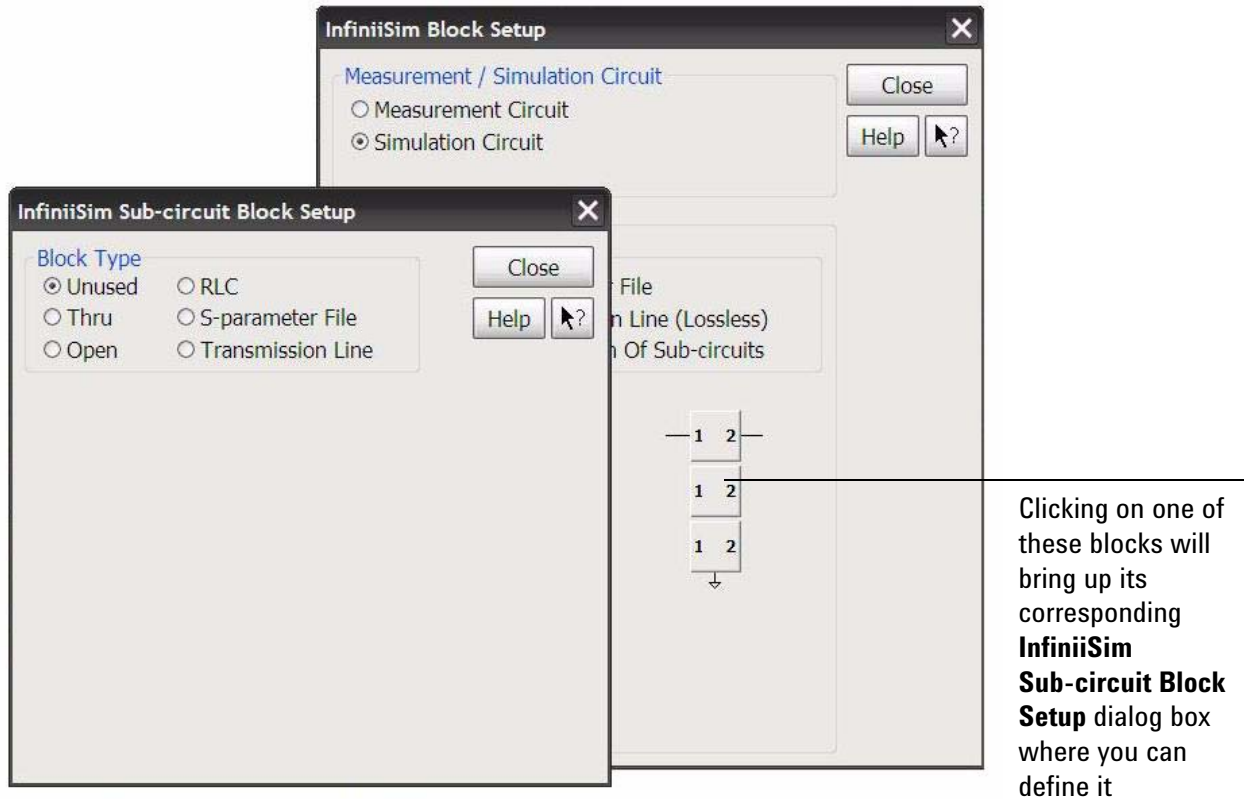
- **Parallel**



- **Series**



Once you select one of these three configurations, you can click on 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

*Figure 48 When you click on one of the sub-blocks a new Sub-circuit dialog box appears where you can define its type*

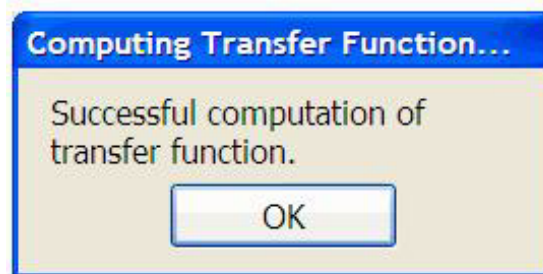
Notice that you can choose between Thru, Open, RLC, S-parameter File, and an option called Unused. Unused simply means that the particular sub-block is not used (allowing you to use 2 sub-blocks instead of three, for example).

Using this Unused setting has a different impact 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.

## Creating a Transfer Function From a Model - Generating/Saving

Once you have your measurement and simulation circuits built, you are ready to generate and save your transfer function file. Click on the **Save Transfer Function** button in the upper right hand corner of the **InfiniiSim Model Setup** dialog box. This will save it under the name (and to the location) specified in the **Save Transfer Function File As** field.

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



*Figure 49 You will see this message if InfiniiSim successfully calculated your transfer function*

If it does not successfully compute the transfer function and instead gives you an error/warning message, please consult [page 97](#) 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 controls (**Setup -> Channel N**) and turning InfiniiSim on or off (by selecting either **Off**, **2-port**, or **4-port**).

## The InfiniiSim Tab and Plots

When you are viewing your simulated signal, you will notice a tab on your oscilloscope's display that is labeled **InfiniiSim**.



Figure 50 The InfiniiSim tab

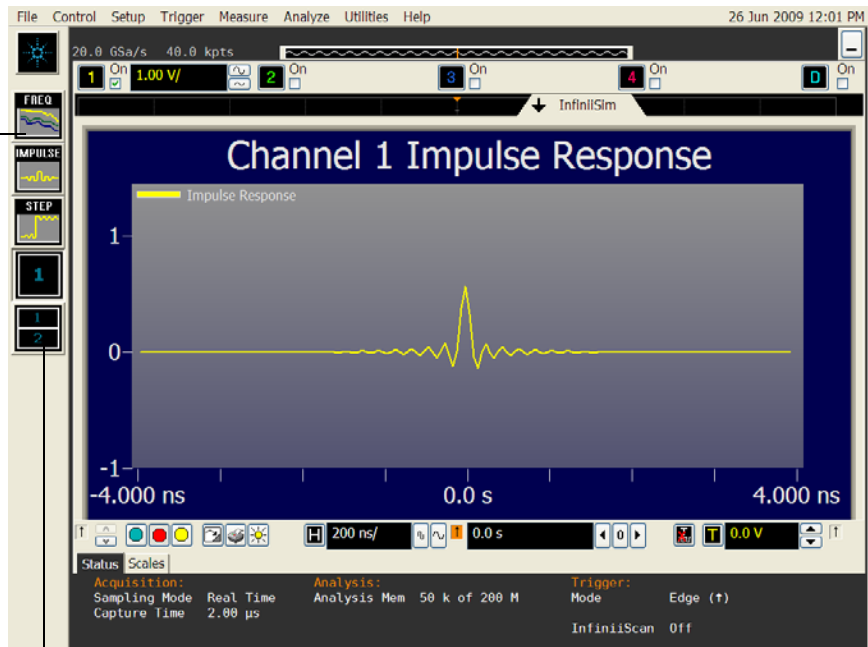
Click on this tab to bring up plots of the frequency response of the original measured signal, the simulated signal, and the filter applied by InfiniiSim. You can also see the impulse response and step response plots.

These plots are very useful when setting some of the controls in the **InfiniiSim Setup** dialog box. Before we discuss this, let us look at some of the basic controls of these plots.

### 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

*Figure 51* Screen shot showing the plot icons as well as how to select whether one or two plots are displayed at the same time

Right-click on any of the plots to bring up this menu. This enables you to quickly jump back to the InfiniiSim model builder or the InfiniiSim Setup dialog box to make changes.

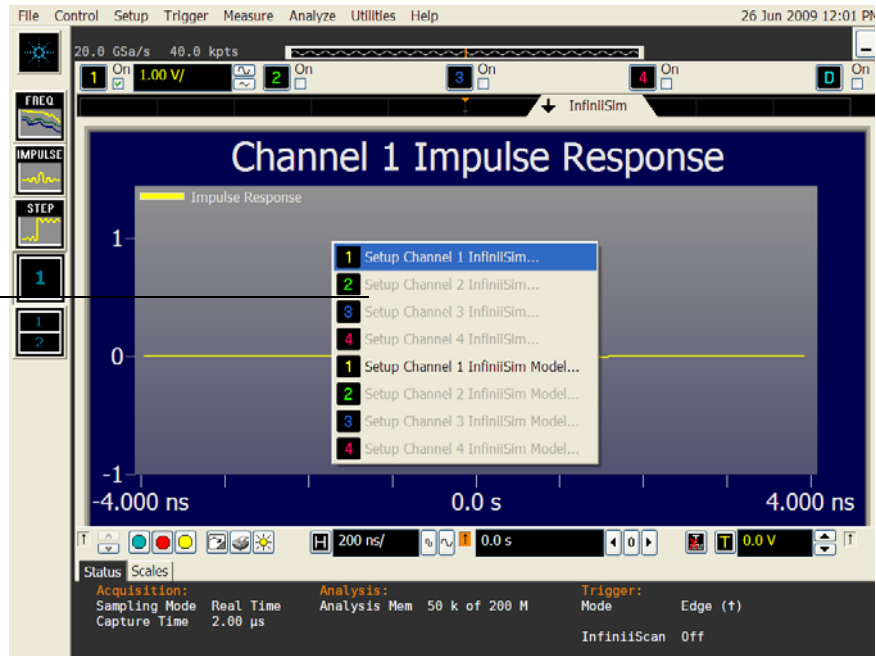


Figure 52 Screen shot showing the menu that appears when you right-click on the plot area

If you click the mouse, drag a rectangle around a certain area, and then release the mouse, the area enclosed by the rectangle will be enlarged (zoom capability)

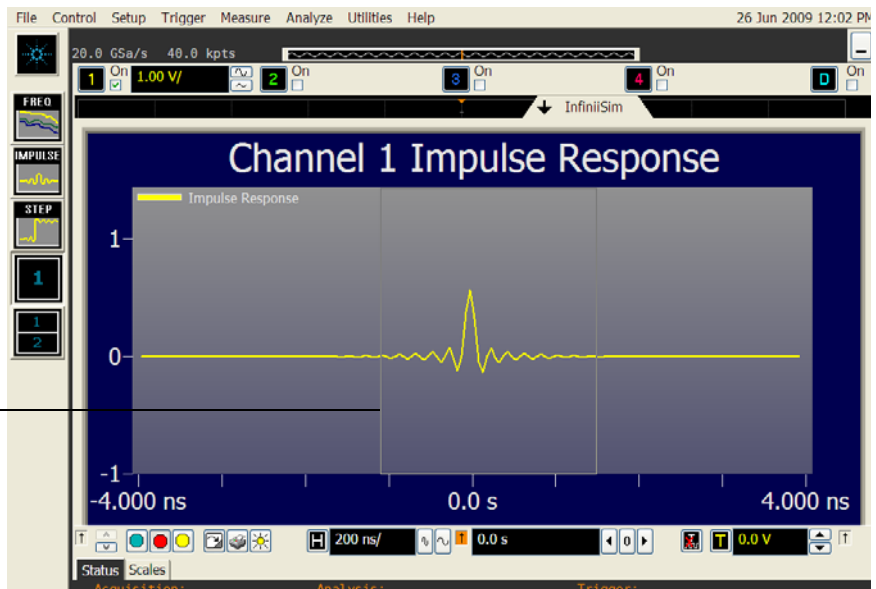
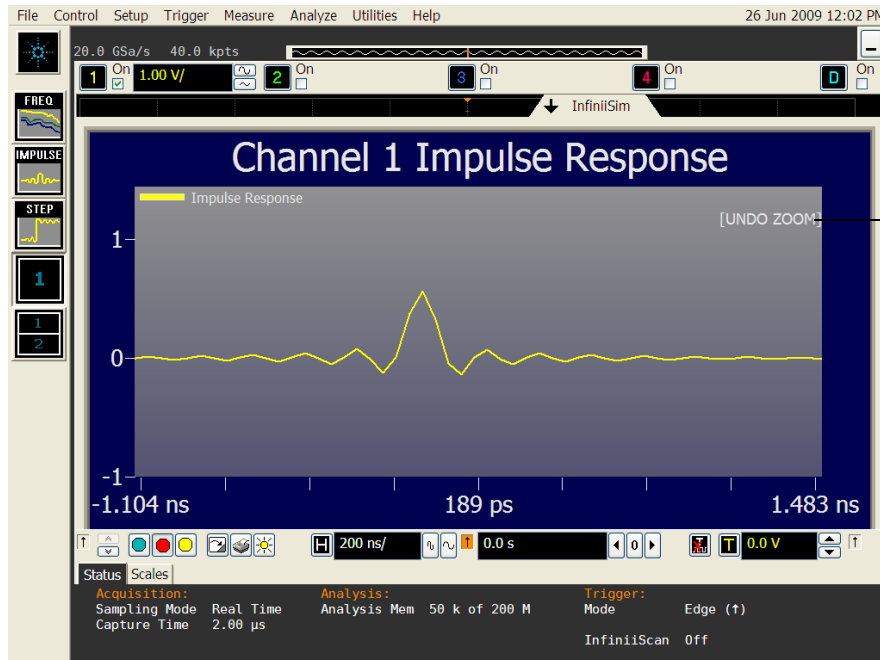


Figure 53 Screen shot showing you how to drag a rectangle around a portion of the plot to zoom in on it



After you zoom in on a plot, this **Undo Zoom** button will appear. To undo the zoom and return to the original view, click on this button.

Figure 54 Screen shot showing the location of the Undo Zoom button

Now that you know the basic controls for these plots, let us look at how they are used.

### Frequency Response Plot

This plot serves two basic functions. First, it enables you to see the measured signal's frequency response, the simulated signal's frequency response (the signal after InfiniiSim transforms the measured signal), and the transfer function filter's frequency response. By investigating these plots, 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.

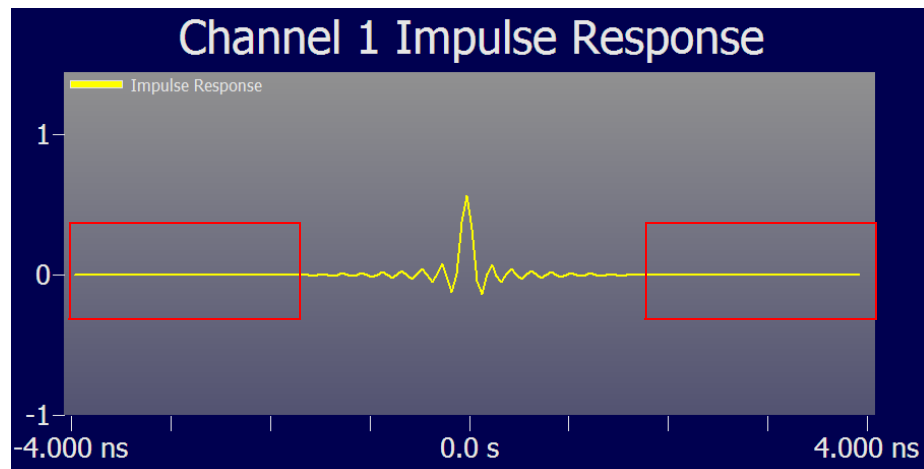
These plots also help you determine what value to use for the **Bandwidth Limit** control. Remember in the previous discussion on the **Bandwidth Limit** control that you need to determine at what frequency your signal's frequency response is mostly just noise (if you are using this control). By investigating these plots, you can determine this frequency and set the **Bandwidth Limit** control accordingly.

The frequency range shown in the frequency response plot is 1 GHz higher than what you specify in the **Bandwidth Limit** field. This enables you to see the bandwidth start to roll off.

### Impulse Plot

The Impulse 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 the **InfiniiSim Setup** dialog box).

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



*Figure 55* Screen shot showing a simplified Impulse Response plot with the **Max Time Span** control set to too large of a value

As you can see, there are large sections of the plot that are just flat. If you see this then you need to reduce the **Max Time Span** control. For example, the screen shot above was captured with the **Max Time Span** set to 8 ns. We need to reduce this until we get to a point where most of these flat regions are not shown. You can bring up the **InfiniiSim Setup** dialog box by right-clicking on the plot area and selecting **Setup Channel N InfiniiSim**. Next 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 (see next page).



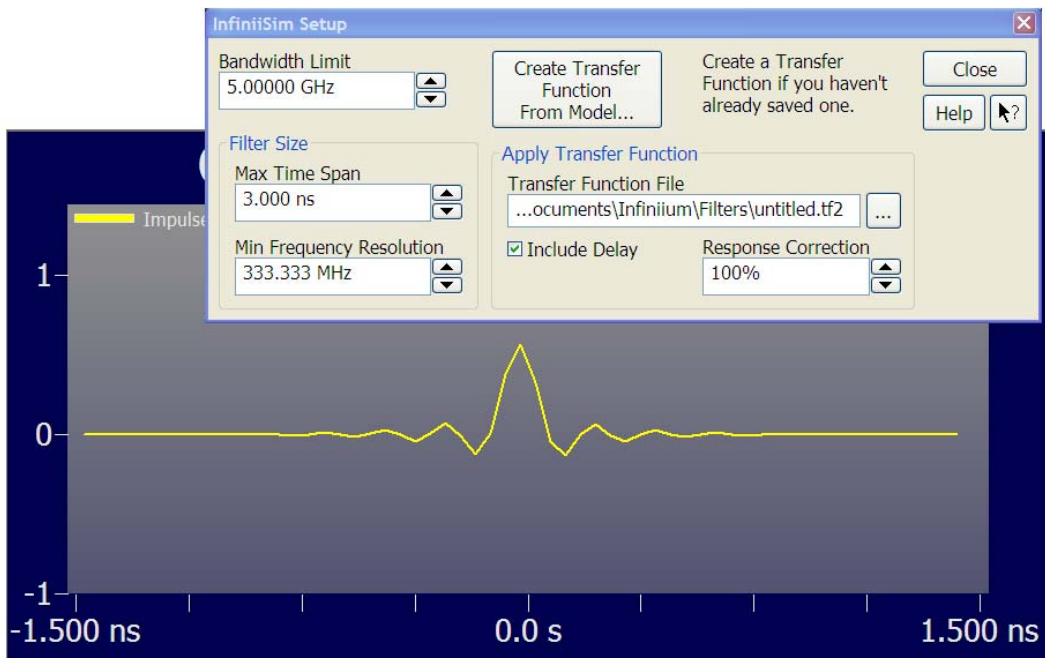


Figure 56 Screen shot showing how to adjust the Max Time Span control to 3 ns and the corresponding change in the Impulse Response plot

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

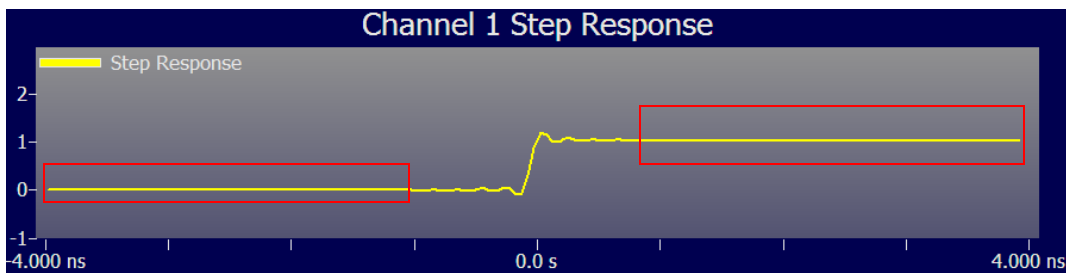
If you look at the Impulse Response plot and do not see any flat regions on either side then you should increase the **Max Time Span** control until you start to see some flat regions. Then you can decrease the control and fine-tune it to its correct setting.

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 will automatically adjust since they are reciprocals of each other.

### Step Response Plot

The Step Response plot is the integration of the Impulse Response plot. It shows you basically the same information, but in a different way. This plot is also useful in setting the **Max Time Span** control.

If you look at either side of the step and you see long portions of flatness, you need to shrink the **Max Time Span** control. Similarly, if you plot the Step Response and do not see any flattening off then you should increase this control.



*Figure 57* Screen shot showing a step response with the Max Time Span control set to too large of a value

The screen shot above shows the large flat areas enclosed by red rectangles. Reduce the **Max Time Span** control in this instance.



## 5 A Simple Example

### Introduction

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 chapter will walk you through a very simple model you can build using the InfiniiSim application so 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.

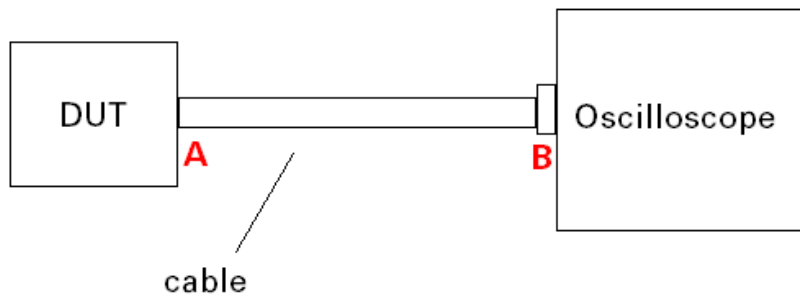


Figure 58 The setup for this simple 2-port example

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 what the waveform looks like at A if possible.

## The Solution Using InfiniiSim

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

Let us assume the cable is connected to Channel 1 on your oscilloscope. Therefore, click on the Channel 1 setup button above your oscilloscope's display.

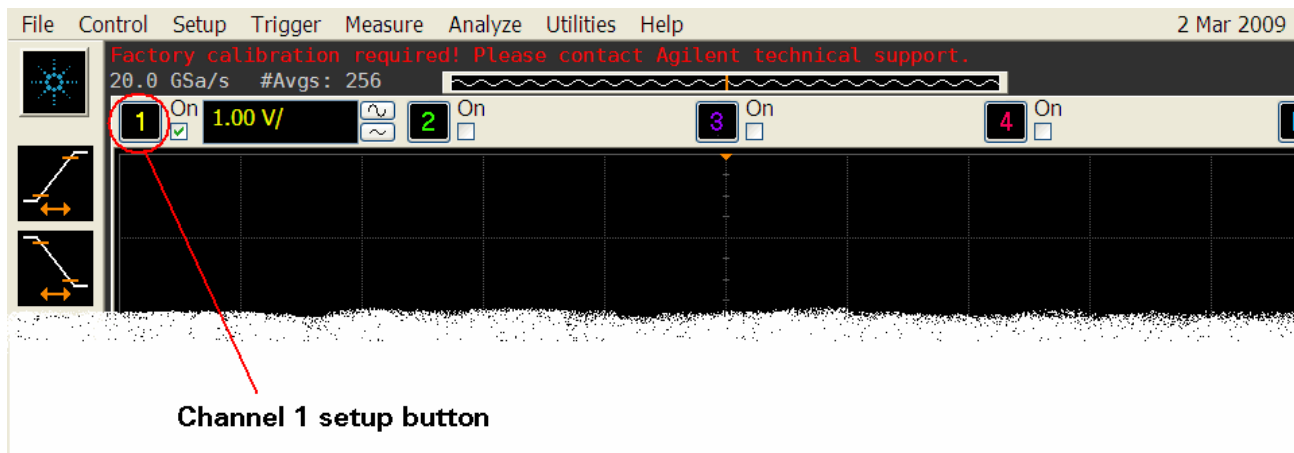


Figure 59 Location of Channel 1 setup button

This will bring up the **Channel Setup** dialog box. Under the *InfiniiSim* section, click on the **2-port** radio button (we will assume the signal passing through the cable is single-ended). Then set the **Signal Bandwidth** field to 4 GHz because we'll assume our signal to noise ratio past 4 GHz is low (either we already know this or we determined it from the Frequency Response plot - discussed on [page 50](#)). Click on the **Setup** button to bring up the **InfiniiSim Setup** dialog box.

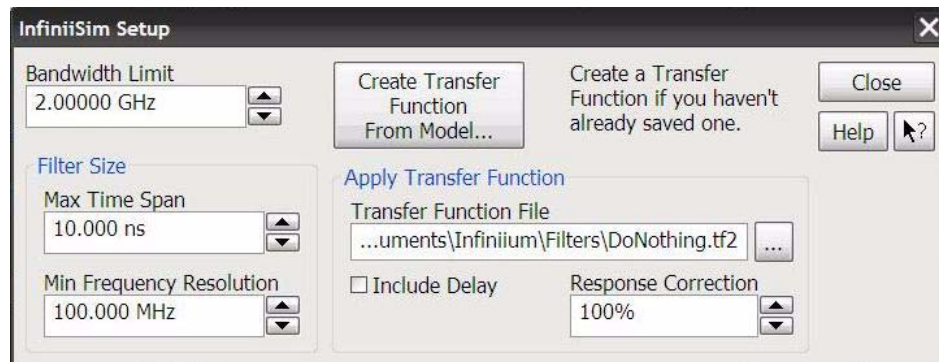


Figure 60 InfiniiSim Setup dialog box

For this simple example, leave the fields under *Apply Transfer Function Accuracy* at their default values. For now we will leave the Filter Size controls at their default values as well. We will adjust these after generating the transfer function.

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

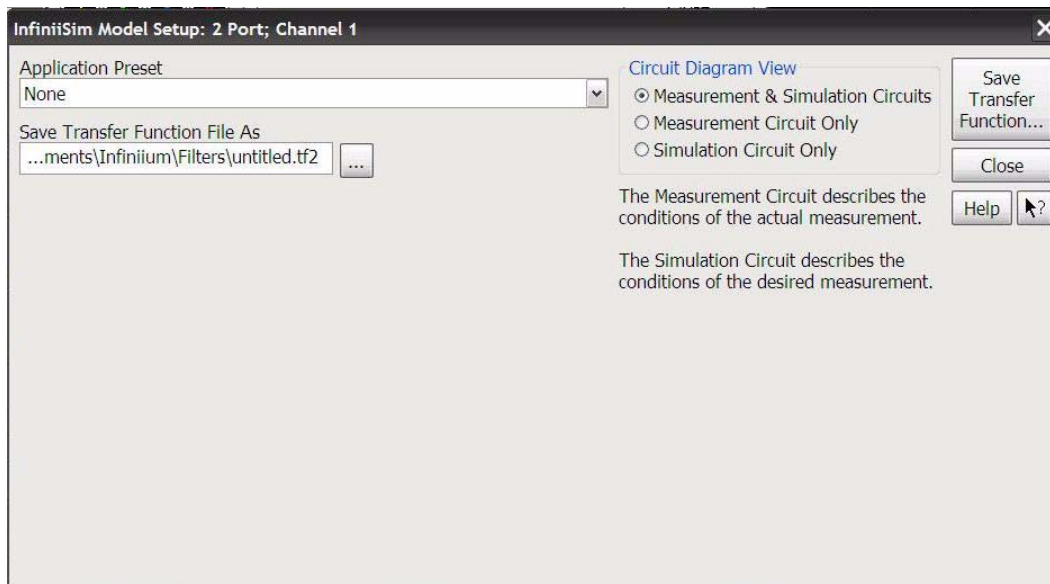


Figure 61 InfiniiSim Model Setup dialog box

For our simple example, we will choose the **Remove Insertion Loss of a Fixture or Cable** as the use model from the **Application Preset** field. This is because we have one device's effects we are trying to negate.

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

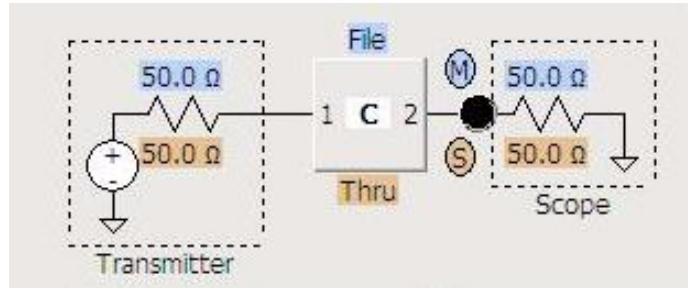


Figure 62 Default measurement and simulation circuit views for Remove Insertion Loss of a Fixture or Cable preset

Now, we need to setup 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 VNA (they are saved to a file named cable.s2p). Therefore, we can define the C block in the measurement circuit using this s-parameter file and the block will act like the cable in terms of its impact on the signal passing through it.

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

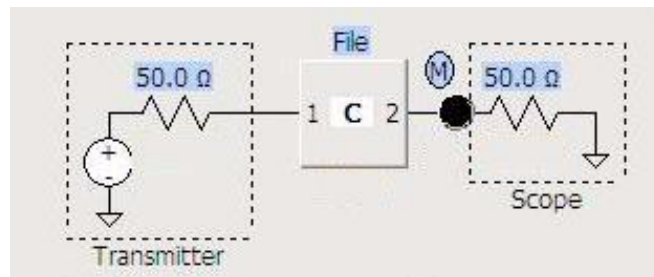


Figure 63 The measurement circuit for our simple example

To set up the measurement circuit like this, click on the block to open the **InfiniiSim Block** dialog box. Then click on the **Measurement Circuit** radio button in the *Measurement / Simulation Circuit* section. Under *Block Type*, choose **S-parameter File** and then click on the **S-parameter File** field, navigate to where you saved the file on the oscilloscope, and select **Load**. Since the s-parameters represent a cable,  $S_{21} = S_{12}$ , so you do not need to check the **Flip Model** box.

Since the Simulation circuit block is defaulted to **Thru** for this application preset, we do not need to adjust 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 - no time delay or characteristic impedance). Remember in the measurement circuit that we had the signal exiting the DUT, passing through the cable, and entering the oscilloscope. Also, we could only measure the signal at the front end of the oscilloscope.

If we set the simulation circuit to Thru, we will 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 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.

## 5 A Simple Example

Now that you have the measurement and simulation circuits built appropriately for your model, click on the **Save Transfer Function As** field and enter the name you want to give it. Then click on the **Save Transfer Function** button and InfiniiSim will start 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 on the InfiniiSim tab to see the various plots associated with your signal, the simulated signal, and the transfer function.

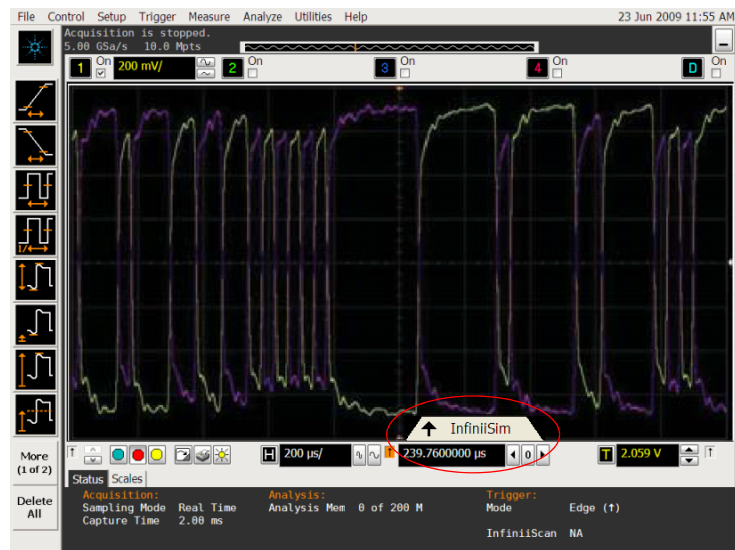


Figure 64 The InfiniiSim tab

Specifically, we should look at either the Step Response plot or the Impulse Response plot to make sure that the default **Max Time Span** and **Min Frequency Resolution** controls were set correctly. See [page 50](#) for information on how to do this.





## Warning / Error Messages and S-parameter File InfiniiSim Keywords

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

### Warnings/Error Messages

#### **Warning: Transfer Function Gain is Unusually Large**

This message usually means you made a mistake when setting up your measurement circuit (example, placed an Unused block somewhere that made it act like an open). What this warning means is 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). 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.

#### **Warning: Transfer Function Appears to Invert Signal**

This warning typically happens with 4 port InfiniiSim. If the ports in your s-parameter file are arranged in such a way that InfiniiSim cannot read them, it may cause the signal to actually become inverted. You should create your s-parameter file with such a format that the port ordering is valid in InfiniiSim. This is not a problem that can be corrected by changing the **Flip Model** or **4 Port Numbering** controls. You must change the format in your s-parameter file. Again, it could be that your specific setup does indeed cause this to occur and there was no error made, but more often than not there is a problem with the s-parameter file port ordering.



**Warning: Frequency Resolution May Be Too Large for Time Delay**

Signals that have long time delays require finer frequency resolution. Therefore, you need to have a finer sampling of frequencies in your s-parameter file. 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 [page 102](#) for details on this command). If you used this command then you need to set it to a finer value.

**Warning: Interpolated S-parameters Limited to 100 kpts**

This warning indicates that the interpolation performed by InfiniiSim results in too many points. This could be because you used the resolution command discussed in the previous warning and set it to too fine of a resolution. Also, 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. If these values are too close together then it can cause the resolution to be too fine.

**Warning: Frequency Resolution May Be Too Large for Slow Time Constants**

This is similar to the warning two above this except that instead of a signal's long time delay causing the problem, it is the signal's slow time constant. These kind of signals require finer frequency resolution. Therefore, you need to have a finer sampling of frequencies in your s-parameter file. 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 [page 102](#) for details on this command). If you used this command then you need to set it to a finer value.

**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. If these values are too far apart then it can cause you to lose the fine resolution at lower frequencies. It is recommended that you use uniform spacing if possible.

### Warning: File Data Truncated to 100 kpts

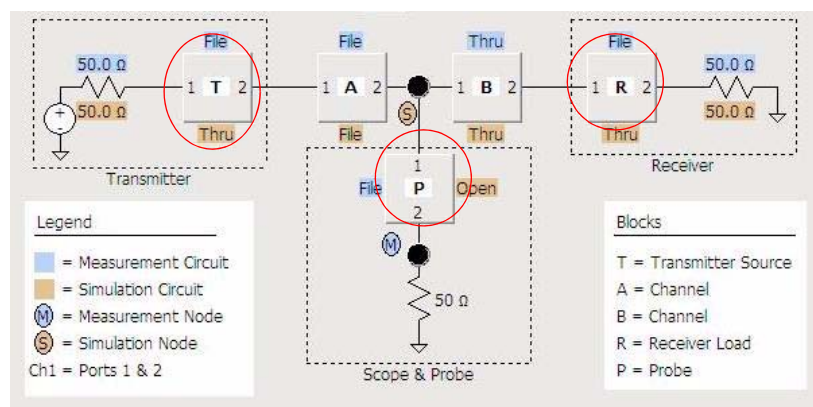
This warning means there are too many points in your s-parameter file. You need to reduce the number of points or use the command you can insert into the s-parameter file to specify a coarser frequency resolution than just the spacing between the points in the file (see [page 102](#) for details on this command).

### 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 Unused block somewhere that makes it act like an open). 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.

### 1 port Files Allowed at Source and Load Blocks Only

For 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 on the blocks enclosed within a Transmitter section, Receiver section, or Scope / Probe section. If you try to use a 1 port file anywhere else, you will get this error.

#### **Cannot Read 4 Port Data From a 2 Port File**

This is the same as the error above, but with 4 port InfiniiSim, you can only use 2 port s-parameter files on the end sections circled above. If you try to use one anywhere else in a circuit, you will get this message.

#### **Unstable or Ill-conditioned Circuit Definition**

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

#### **Less Than 2 Data Points in the File**

You must have at least two data points in your s-parameter file.

#### **Delta Frequency Value is Too Small**

As mentioned above, when s-parameter files with non-uniformly spaced points are used, InfiniiSim figures out the resolution by seeing how closely spaced the last two points in the file are. It then uses this sampling space for the resolution for the rest of the file. If the spacing between these last two points is too small, you will get this error message.

#### **Unspecified File Read Error**

This indicates that the file has an invalid format.

#### **Invalid File Name**

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

**File Not Found**

This indicates that the file is not where you are telling InfiniiSim it is located.

**Invalid Frequency Units**

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

**Invalid Data Format**

You are only allowed to use dB, magnitude/angle, or real/imaginary (complex). If you use any other data format, you will get this error message.

**Transfer Function Must Contain a 0 Hz Point**

This is not true for s-parameter functions, but is true for transfer functions.

**Impedance of All Ports Must Be the Same**

The .citi format allows you to specify port impedances. They must all be the same or you will get this error.

**Invalid Output Name**

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

## S-parameter InfiniiSim Keywords

The following are the currently supported keywords you can place in your s-parameter file. InfiniiSim scans the data file for these special keywords which communicate to InfiniiSim how data was generated or how to interpret the data. The format of these lines is slightly different between Touchstone and CITI files in that they must be preceded by a ! inside Touchstone files. InfiniiSim keywords are case insensitive. Additional comments are permitted on the same line as those containing the keywords.

### **#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 special comment line, the data is checked to ensure that it is uniformly sampled in frequency. The interval between DC and the lowest non-zero frequency point is not checked. A warning is generated if the data is not uniformly sampled. When <value> is actually a numeric value, the data is resampled using the specified frequency resolution value.

### **#DSO TRANSFER\_FUNCTION\_DEFINITION\_STRING <string>**

where <string> is an optionally formatted string.

For every occurrence of this comment line in the file, the entire contents of that line are appended to the return string frequency resolution value.

**#DSO RETURN\_LOSS\_INTERPRETATION <string>**

where <string> is one of the following legal strings: SCOPE, PROBE, SERIES\_RLC, SHUNT\_RLC, or DEFAULT.

Nothing unique happens if <string> equals DEFAULT. Each of the other <string> values cause the 1-port (2-port) return loss file data to be converted into 2-port (4-port) data. A value of PROBE causes the insertion loss portion of the data to model that of a probe. A value of SCOPE models a scope, etc..







[www.agilent.com](http://www.agilent.com)

© Agilent Technologies, Inc. 2009

First edition, June 2009



N5435-97000