
Version 8.0

VisSim/Model Wizard User's Guide

By Visual Solutions, Inc.

Visual Solutions, Inc.

VisSim/Model Wizard User's Guide Version 8.0

Copyright	© 2010 Visual Solutions, Inc. All rights reserved.	Visual Solutions, Inc. 487 Groton Road Westford, MA 01886
Trademarks	VisSim, VisSim/Analyze, VisSim/CAN, VisSim/C-Code, VisSim/C-Code Support Library source, VisSim/Comm, VisSim/Comm C-Code, VisSim/Comm Red Rapids, VisSim/Comm Turbo Codes, VisSim/Comm Wireless LAN, VisSim/Fixed-Point, VisSim/Knobs & Gauges, VisSim/Model-Wizard, VisSim/Motion, VisSim/Neural-Net, VisSim/OPC, VisSim/OptimizePRO, VisSim/Real-TimePRO, VisSim/State Charts, VisSim/Serial, VisSim/UDP, VisSim Viewer, and flexWires are trademarks of Visual Solutions. All other products mentioned in this manual are trademarks or registered trademarks of their respective manufacturers.	
Copyright and use restrictions	The information in this manual is subject to change without notice and does not represent a commitment by Visual Solutions. Visual Solutions does not assume any responsibility for errors that may appear in this document. No part of this manual may be reprinted or reproduced or utilized in any form or by any electronic, mechanical, or other means without permission in writing from Visual Solutions. The Software may not be copied or reproduced in any form, except as stated in the terms of the Software License Agreement.	
Acknowledgements	The following engineers contributed significantly to preparation of this manual: Mike Borrello, Allan Corbeil, and Richard Kolk.	

Contents

Introduction	1
What is VisSim/Model Wizard.....	1
The VisSim product family	1
Resources for learning VisSim/Model Wizard.....	3
Interactive webinars	3
Sample diagrams	3
Training.....	3
Using the VisSim/Model Wizard	5
Preparing the time history dataset file	5
Starting VisSim/Model Wizard and choosing a dataset file	5
Editing the dataset file.....	7
Specifying the signalset.....	9
Analyzing the signalset.....	9
General information	10
Input signal information	10
Output signal information	11
Plots.....	11
Next step	11
Filtering the output signal.....	11
Automatic filtering.....	12
Manual filtering.....	12
Plots.....	13
Identification iterations.....	13
Step input identification	14
Random input identification.....	14
Tuning parameters for step and random input.....	15
Plots for step input	15
Transfer function construction	15
Returning to VisSim.....	16
Installing VisSim/Model Wizard	17
Installation requirements	17
Installation procedure	17
Index	19

Introduction

This section contains...

What is VisSim/Model Wizard

VisSim/Model Wizard allows you to easily create a linear mathematical model from plant data. It uses optimization techniques to fit a linear single-input-single-output (SISO) transfer function to a measured input or response data set pair.

You can select a transfer function order between 0 and 7 states. In order to properly derive a model, you should select input data that produces interesting variations in the output signal. VisSim/Model Wizard accepts two classes of input signals: step and random.

The resulting model can be represented in VisSim using a single transferFunction block. In VisSim, many such blocks can be composed to represent a larger system, which can then be modeled in time domain, analyzed in frequency domain, or run in real time producing analog responses using the VisSim/Real-Time add-on..

The VisSim product family

The VisSim product family includes several base products and product suites, as well as a comprehensive set of targeted add-on modules that address specific problems in areas such as data communications, data acquisition, linearization and analysis, and digital signal processing.

Base products and product suites

Product	Function
Professional VisSim	Model-based design, simulation, testing, and validation of dynamic systems. A personal version, VisSim PE, is also available. VisSim PE limits diagram size to 100 blocks.
VisSim/Comm Suite	Simulates end-to-end communication systems at the signal level using 200+ communications, signal processing, and RF blocks. Includes Professional VisSim and VisSim/Comm blockset.

	<p>A personal version, VisSim/Comm Suite PE, is also available. VisSim/Comm PE limits diagram size to 100 blocks and limits the Communication blockset. See the VisSim/Comm datasheet for details.</p> <p>VisSim/Comm Suite add-on modules are available for real-time data acquisition (Red Rapids digital tuner card); modeling PCCC turbo codes, including UMTS specification; and for support of Bluetooth, 802.11 a/b/g (Wi-Fi), and ultrawideband wireless designs.</p>
VisSim/Embedded Controls Developer Suite	<p>Rapidly prototypes and creates embedded controls for DSPs, DSCs, and MSP430 microcontrollers. You can simulate and generate scaled, fixed-point ANSI C code, as well as code for on-chip peripherals.</p> <p>Includes Professional VisSim, VisSim/C-Code, VisSim/Fixed-Point, and one user-specified target support.</p> <p>A personal version, VisSim/Embedded Controls Developer PE, is also available. VisSim/Embedded Controls Developer PE limits diagram size to 100.</p>
VisSim Viewer (free)	Lets you share VisSim models with colleagues and clients not licensed to use VisSim.

Add-on modules

Add-On Module	Function
VisSim/Analyze	Performs frequency domain analysis of a linearized nonlinear subsystem.
VisSim/CAN	Interfaces with a USB CAN device to read and write CAN messages on the CAN bus.
VisSim/C-Code	Generates highly-optimized, ANSI C code that can be compiled and run on any platform that supports an ANSI C compiler.
VisSim/C-Code Support Library Source	Provides source code for the Support Library.
VisSim/Comm blockset	<p>Simulates end-to-end communication systems at the signal level using 200+ communications, signal processing, and RF blocks.</p> <p>A personal version, VisSim/Comm PE, is also available. VisSim/Comm PE is a subset of the Communication blockset. See the VisSim/Comm datasheet for details</p> <p>You can purchase VisSim/Comm add-on modules for real-time data acquisition (Red Rapids digital tuner cards); for modeling PCCC turbo codes, including UMTS specification; for support of Bluetooth, 802.11 a/b/g (Wi-Fi), and ultrawideband wireless designs.</p>
VisSim/Fixed-Point	Simulates the behavior of fixed-point algorithms prior to code generation and implementation of the algorithm on the fixed-point target.
VisSim/Knobs and Gauges	Provides dynamic gauges, meters, and knobs for process control, and measurement and validation systems.

VisSim/Model-Wizard	Generates transfer function model from historic or real-time data.
VisSim/Motion	Simulates motor control systems with customizable amplifiers, controllers, filters, motors, sensors, sources, tools, and transforms.
VisSim/Neural-Networks	Performs nonlinear system identification, problem diagnosis, decision-making prediction, and other problems where pattern recognition is important.
VisSim/OPC	Connects to any OPC server and log data or run a virtual plant in VisSim for offline tuning.
VisSim/OptimizePRO	Performs generalized reduced gradient method of parameter optimization.
VisSim/Real-TimePRO	Performs real-time data acquisition and signal generation using I/O cards, PLCs, and DCSs.
VisSim/Serial	Performs serial I/O with other computers.
VisSim/State Charts	Creates, edits, and executes event-based systems.
VisSim/UDP	Performs data exchange over the internet using UDP.
VisSim Viewer (free)	Lets you share VisSim models with colleagues and clients not licensed to use VisSim.

Resources for learning VisSim/Model Wizard

For those of you that are new to VisSim, we have provided several free services to make your transition to VisSim fast, smooth, and easy:

Interactive webinars

Interactive webinars offer you the opportunity to meet with Visual Solutions product specialists who will introduce and demonstrate our software products live on your computer and answer any questions you have. Each webinar is approximately 45 minutes long. To learn more about our interactive webinars, go to <http://www.vissim.com/webinars/webinars.html>.

Sample diagrams

VisSim 8.0 includes a directory of fully documented sample diagrams. These diagrams illustrate both simple and complex models spanning a broad range of engineering disciplines, including aerospace, biophysics, chemical engineering, control design, dynamic systems, electromechanical systems, environmental systems, HVAC, motion control, process control, and signal processing.

To access sample diagrams

Click on the **Diagrams** menu in VisSim.

Click on **Examples > Applications**.

Training

Visual Solutions offers training sessions for learning and gaining expertise in VisSim and the VisSim family of add-on products. Training sessions are conducted at Visual

Solutions training facility in Westford, MA, as well as at customer sites and as online webinars.

For information on setting up a training session, contact sales@vissol.com.

Using the VisSim/Model Wizard

This section contains...

Preparing the time history dataset file

Before you can use VisSim/Model Wizard, you must prepare a file containing the time history data. This file, referred to as the time history dataset file, is a .TXT or .DAT file. It contains a tab-delimited ASCII text from which the signal set information (time, input signal, output signal) is derived and evaluated.

VisSim/Model Wizard is a data filtering and model identification tool, not a data manipulation tool. Therefore, the dataset file should contain all relevant data, correctly formatted before you begin. Having said that, however, VisSim/Model Wizard does provide rudimentary editing operations that allow you to make temporary patches to the dataset file. For instance, because VisSim/Model Wizard works only with data where time evolves in rows, and many data recorders record time in columns, VisSim/Model Wizard lets you transpose the data. There are several ways to [edit the dataset file](#).

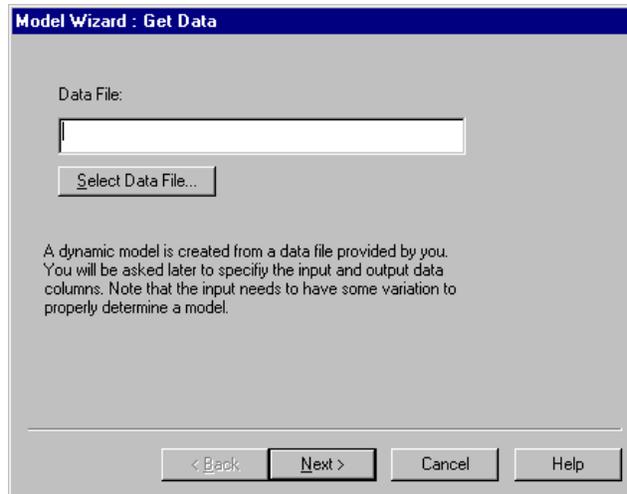
Starting VisSim/Model Wizard and choosing a dataset file

Using the Dynamic Model Wizard command under the Tools menu, you can launch VisSim/Model Wizard and select the time history dataset file.

To start the VisSim/ModelWizard

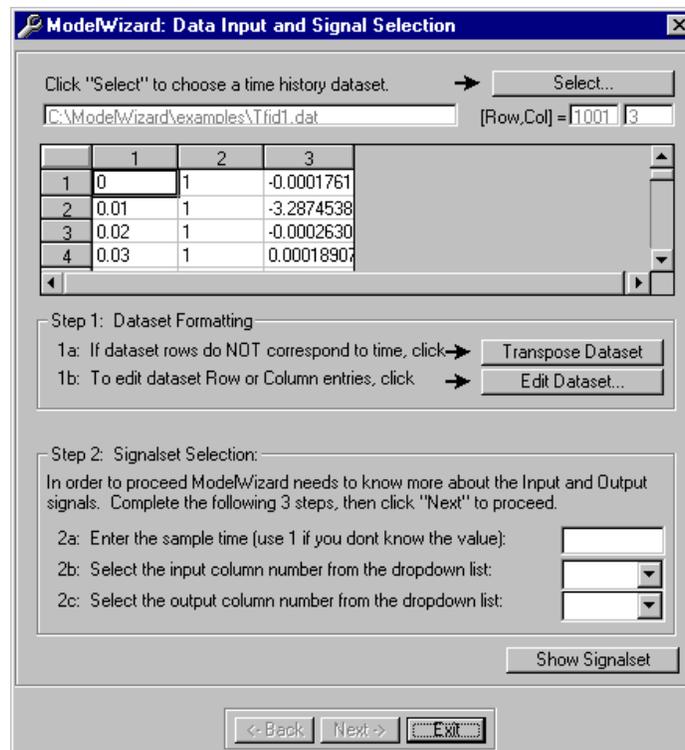
1. Choose **Tools > ModelWizard**.

The following dialog box appears:



2. Click on the **Select Data File** button and select the time history dataset file you want to use.
3. Choose the **OK** button, or press **ENTER**.

The Data Input and Signal Selection dialog box appears, displaying the contents of the specified dataset file in the spreadsheet window.



The name of the currently opened dataset file is displayed in the box at the top of the Data Input and Signal Selection dialog box. If you want to open a different dataset file, click on the Select button and choose from the File Selection dialog box.

The size of the dataset file is displayed in the Row,Col box. This is a read-only field.

The contents of the dataset file is displayed in the spreadsheet window in the Data Input and Signal Selection dialog box. To examine different cell values, use the scroll bars on the bottom and right-hand side of the spreadsheet.

If you do not need to make changes to the dataset file, your next step is to create a [three-column data signalset](#).

Editing the dataset file

In some cases, the dataset file may require minimal editing before you can proceed. Using the Edit Dataset button in the Data Input and Signal Selection dialog box, you can make the following temporary edits to the data:

- Transpose rows and columns
- Cut, copy, and paste data
- Replace text strings

Transposing data

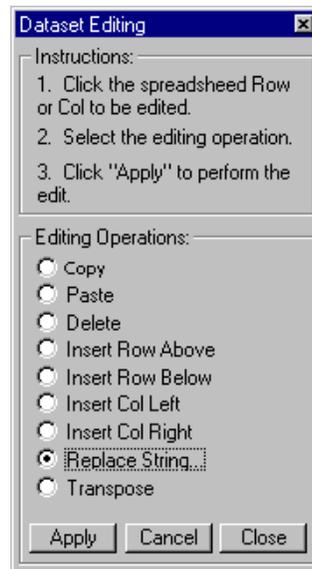
Signal time histories must evolve in rows, not columns. However, depending on the format in which the data was recorded, time may be in columns.

To transpose data

- Do one of the following:
 - Click on the Transpose Dataset button.
 - Click on the Edit Dataset button; select Transpose; and click on the Apply button and then the Close button.

Editing data

The Edit Dataset button brings up the Dataset Editing dialog box, which provides a set of rudimentary signal editing functions.



To copy or move data to a different row or column

1. In the spreadsheet, click on the row or column to be copied or moved.
2. Click on the **Edit Dataset** button.
3. In the **Dataset Editing** dialog box under **Editing Operations**, do one of the following:

- Select **Copy** to copy the selected data.
 - Select **Delete** to move the selected data.
 - Click on the **Apply** button.
4. In the spreadsheet, click on the row or column into which the data is to be copied or moved.
 5. Click on the **Edit Dataset** button.
 6. In the **Dataset Editing** dialog box, select **Paste**; then click on the **Apply** button and the **Close** button.

To delete a row or column of data

1. In the spreadsheet, click on the row or column to be deleted.
2. Click on the **Edit Dataset** button.
3. In the **Dataset Editing** dialog box under **Editing Operations**, click on the **Delete** button; then the **Apply** button and the **Close** button.

To insert a row or column

Before you can insert a new row or column, you must select the row or column in the spreadsheet next to which the new row or column will be placed. For example, if you want to insert a new row before row 4, you either select row 3 or row 4, then use the appropriate Insert Row editing command to make the insertion.

1. In the spreadsheet, click on the row or column next to which the new row or column will be inserted.
2. Click on the Edit Dataset button.
3. In the Dataset Editing dialog box under Editing Operations, do one of the following:

To insert this	Do this
A row above the selected row	Select Insert Row Above.
A row below the selected row	Select Insert Row Below.
A column to the left of the selected column	Select Insert Col Left.
A column to the right of the selected column	Select Insert Col Right.

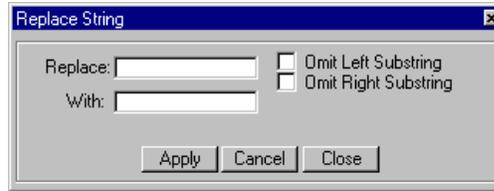
4. Click on the Apply button and then the Close button.

To replace text string with numeric data

Some recorders record using ON/OFF or YES/NO rather than discrete data. The Replace String command lets you replace all occurrences of the specified text with zeros or ones.

1. Click on the **Edit Dataset** button.
2. In the **Dataset Editing** dialog box under **Editing Operations**, select **Replace Text** and click on the **Apply** button.

The Replace Text dialog box appears.



3. In the Replace box, enter the text to be replaced.
4. In the With box, enter the new text.

The Omit Left Substring and Omit Right Substring options let you delete unwanted text to the left and to the right of the replacement text. For example, to replace XYZ ON with 1, in the Replace box, enter ON; in the With box, enter 1; and activate the Omit Left Substring box.

5. Click on the **Apply** button and then the **Close** button.

Specifying the signalset

At this point, you are ready to derive a standard three-column data signalset, consisting of time, input signal, and output signal, from the time history dataset file.

To specify the signalset

- In the Data Input and Signal Selection dialog box, under Signalset Selection, do the following:
 - Enter the sample time in the 2a option box. You must enter the sample time manually because many data records do not include time as a recorded signal. The sample time must be a positive number greater than 0. If you do not know the sample time, enter 1 as the sample time.
 - Enter the input column number in the 2b option box and the output column number in the 2c option box.
 - The input and output column numbers are specified based on the current dataset. The Show Signalset button toggles the Signalset Selection view from current view to data view. This allows you to see the three-column data signalset and editing changes you may have made so far, before continuing.

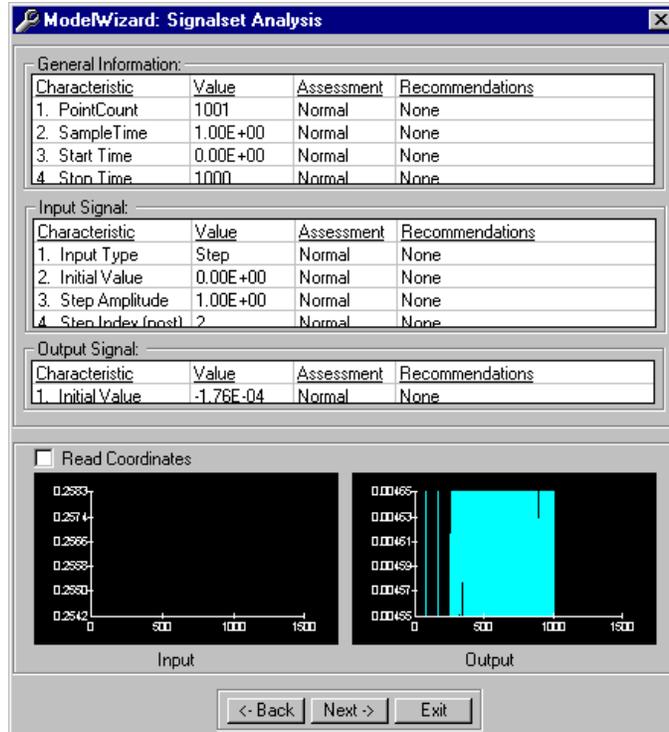
After all the information has been correctly entered into the Data Input and Signal Selection dialog box, click on the Next button to begin signalset analysis.

If the Next button is dimmed, you have probably omitted signalset information and will need [to re-enter it](#).

Analyzing the signalset

The Signalset Analysis dialog box summarizes and evaluates the three-column signalset. After verifying the information, you need only to click on the Next button to proceed, or the Back button to make changes to the signalset data.

The Signalset Analysis dialog box categorizes information as general information, input signal information, and output signal information. The information value is analyzed and assessed as either Normal, N/A = not available, or Abnormal. When possible, a corrective action is presented as a recommendation.



General information

The following information is included in this box:

- **PointCount:** Displays the number of rows in the sampleset. The ModelWizard checks for too few and too many rows. A minimum of 10 rows is required and a maximum of 3,000 is suggested. When you have more than 3,000 rows, the identification process is significantly slowed down.
- **SampleTime:** Displays the sample time specified in the Data Input and Signal Selection dialog box.
- **Start Time** Displays the start time of the signalset. This number must be greater than or equal to 0.
- **Stop Time:** Displays the stop time of the signalset. This number must be greater than 0.

Input signal information

The following information is included in this box:

- **Input Type:** Displays the input type. Either step or random is permitted.
- **Initial Value:** Displays the initial value of the input signal. This is the first value in the signalset, which is also assumed to be the pre-step amplitude if the input is a step input.
- **Step Amplitude:** Displays the amplitude of the step discontinuity. This is only displayed if the input is a step. For random inputs, the value is set to 0 and the assessment is N/A. If a step input is applied and the step amplitude is 0, a warning is issued.

- **Step Index:** Displays the index of the step discontinuity immediately after it has occurred. This is only displayed if the input is a step. For random inputs, the value is set to 0 and the assessment is N/A.

Output signal information

The Output Signal information box displays the initial value of the output signal regardless of the input type.

Plots

There are two plots at the bottom of the Signalset Analysis dialog box. The left plot presents the input signal versus time (as defined by the sample time and the pointcount of the signal set). The right plot presents the output signal versus the same time.

To zoom in

1. You can zoom in and out of the plots when the Read Coordinates box is not activated.
2. Point to the corner of the area to be magnified.
3. Hold down the left mouse button.
4. Drag the pointer to encompass the area to magnify. As you drag the mouse, a box outlines the area of magnification.
5. Release the mouse button.

To zoom out

- Click the right mouse button.

Reading coordinates

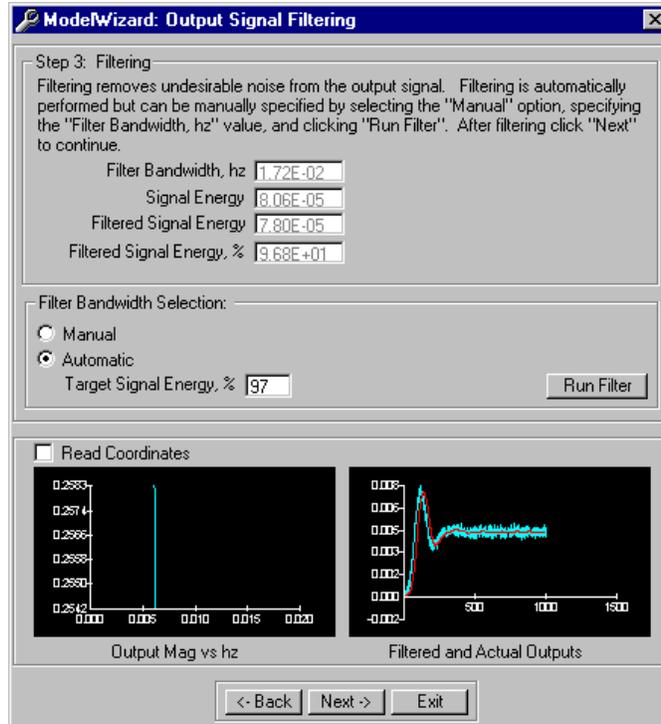
When you activate the Read Coordinates checkbox, you can use the pointer to display plot coordinates. As you move the pointer, plot coordinates are displayed next to the Read Coordinates checkbox.

Next step

If all assessments are normal or N/A, click the Next button to proceed to initiate output signal filtering. If you need to make changes in the previous dialog box, click on the Back button.

Filtering the output signal

The Output Signal Filtering dialog box lets you automatically or manually perform low-pass filtering on the output signal in the signalset. Low-pass filtering removes high frequency noise from the output signal. The assumption is that the signal is primarily low frequency and the noise is primarily high frequency.



Automatic filtering

The ModelWizard automatically initiates low-pass filtering as soon as the Output Signal Filtering dialog box is displayed. Plots at the bottom of the dialog box display the filtering results. In addition, the output signal is automatically filtered to have 97% of its original energy.

You can, however, specify other energy percents in the Target Signal Energy box and rerun the automatic filtering process. Note that the smaller the target signal energy, the more filtering is applied to the output signal. The maximum value for the Target Signal Energy is 100%. In this case no filtering is applied and the original signal is used.

To rerun automatic filtering with a different target signal energy

1. In the **Filter Bandwidth Selection** box, select **Automatic** and enter a new value in the **Target Signal Energy** box.
2. Click on the **Run Filter** button.

Manual filtering

You can alternatively run the filter manually; however you cannot specify the target signal energy percentage.

To manually filter

1. In the **Filter Bandwidth Selection** box, select **Manual**.
2. In the **Filter Bandwidth** box, enter the desired value in hertz.
3. Click on the **Run Filter** button.

Plots

Two plots are presented at the bottom of the Output Signal Filtering dialog box

The left plot presents the effect of filtering in frequency. The filtered output signal magnitude (as computed from a Fast Fourier Transform, FFT) versus frequency is shown. Peaks in this plot reveal the frequencies at which signal power (system information) is concentrated. Most systems have significant DC (zero-frequency) power and possibly one or more peaks at non-zero frequencies, called modes.

The right plot shows the filtered and raw output signals plotted against time. This plot shows the effect of filtering on the output signal.

Zooming

You can zoom in and out of the plots when the Read Coordinates box is not activated.

To zoom in

1. Point to the corner of the area to be magnified.
2. Hold down the left mouse button.
3. Drag the pointer to encompass the area to magnify. As you drag the mouse, a box outlines the area of magnification.
4. Release the mouse button.

To zoom out

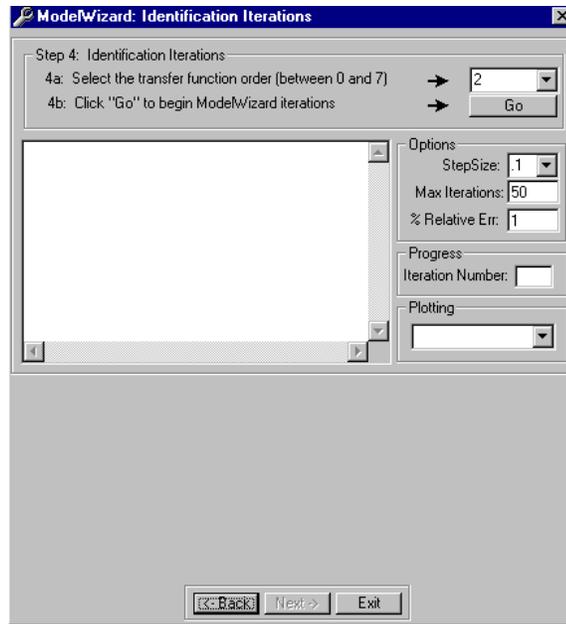
- Click the right mouse button.

Reading coordinates

When you activate the Read Coordinates checkbox, you can use the pointer to display plot coordinates. As you move the pointer, plot coordinates are displayed next to the Read Coordinates checkbox.

Identification iterations

The Identification Iterations dialog box lets you set up and perform the identification iterations. The plotting options vary depending on whether you are using step or random input. The ModelWizard automatically determines whether to use the step input identification mode or the random input identification mode by examining the input dataset.



To set up and perform identification iterations

- Under Identification Iterations, do the following
 - In the 4a text box, specify the transfer function order.
 - Click on the **Go** button.
 - A series of results scroll through the large window in the dialog box.

Step input identification

Step input identification uses signal deflation to fit successively higher frequency modes beginning at DC to the output signal. Initially the DC (or steady state) mode, referred to as mode 1, is fit. After fitting, the mode model is displayed in the textbox and the mode model and true mode (remnant) or actual output signal are plotted in the plotting window at the bottom of the dialog box.

The mode model may be either a constant, an exponential, or a damped sinusoid.

Mode model	Equation	Where
Constant	$y(t) = k$	k is determined from the signalset
Exponential	$y(t) = ke^{at}$	k and a are determined from the signalset
Damped sinusoid	$y(t) = e^{at}[k_1 \cos(\omega t) + k_2 \sin(\omega t)]$	a , k_1 , k_2 , and w are determined from the signalset

Random input identification

Random input identification uses an AutoRegressive Moving Average (ARMA) model to fit the entire transfer function at once. This mode of identification is useful when there is no step change in the input dataset.

After fitting, the model is displayed in the textbox along with a model plot and a plot of the true output signal at the bottom of the screen. No modes are used in the ARMA fitting process.

The ARMA process used in the ModelWizard is iterative. Beginning with an initial best guess estimate of the transfer function using least squares, this estimate is refined iteratively using gradient adjustment.

Tuning parameters for step and random input

There are three tuning parameters that affect the speed and stability of the ModelWizard. These parameters are available under Options in the Identification Iterations dialog box.

- **StepSize:** This is a normalized value which controls how aggressively the iterations are executed. For well-behaved linear problems (which rarely occur) the Stepsize may be set to a large value (not to exceed 1). For most normal problems with mild to medium nonlinear behavior the Stepsize is defaulted to 0.1. For more nonlinear problems a value of 0.01 or even 0.001 may be used. Use of a stepsize that is too large for a given problem will result in iteration divergence and premature termination of the identification procedure.
- **Max Iterations:** This is one of two termination criteria for the signal deflation iterations. If you wish to terminate sooner than the default 50 iterations just click and reenter a smaller value. A default value of 50 iterations is reasonable, for many problems, termination is usually reached due to the second criteria, % Relative Error, before the Max Iterations are reached.
- **% Relative Error:** This is the second of the two termination criteria for the signal deflation iterations. If the relative error between two consecutive Root Sum Squared (RSS) objectives falls below this value the deflation iteration will terminate. Normally a value of 1% is adequate for most problems.

Plots for step input

You can plot various modes and their individual mode models, the sum total fit model and the signalset output signal, and finally the total remnant or that part of the output that is unfit by the model.

Note that plotting is disabled for random input identification. Because there are no modes there is nothing additional to plot in this situation.

Transfer function construction

Step input: After the designated number of modes has been fit (the designated modes is related to the selected transfer function order), the mode collection is assembled and the individual mode parameters are polished using a least square objective to best match the output signal. Once the polishing operation is complete the transfer function (in s -domain) is constructed from the individual time modes and displayed under the Results section in the scrolling window. This transfer function is in the s -domain and has a monic denominator (leading coefficient = 1). All gain is absorbed into the numerator polynomial.

Random input: After the iterated ARMA model with the order you specified (as the transfer function order), the transfer function's coefficients are polished using a least square objective to best match the output signal. Once the polishing operation is complete the transfer function is converted from discrete to continuous domain and the transfer function information is displayed under the Results section in the scrolling window. This transfer function is in the s -domain and has a monic denominator (leading coefficient = 1). All gain is absorbed into the numerator polynomial.

Returning to VisSim

Upon successful completion of the model identification phase, ModelWizard automatically creates an output file, named MODWIZOUT.M, containing the coefficients of the numerator and denominator polynomials of the identified model transfer function.

When you click on the Exit button to close the ModelWizard and return to the VisSim design space, VisSim automatically places a new `transferFunction` block in the workspace. The `transferFunction` block is automatically embedded with the gain and the polynomial coefficients from MODWIZOUT.M..

Installing VisSim/Model Wizard

The Install program that comes on your VisSim/Model Wizard disk installs the VisSim/Model Wizard program and other utility files on your hard disk.

Installation requirements

VisSim/Model Wizard runs on personal computers using the Intel 80286 or higher processor, including the IBM Personal System/2 Series, the IBM PC AT, and 100% compatibles. To use VisSim/Model Wizard, your computer must have the following components:

- Visual Solutions VisSim 8.0+
- 3MB of free hard disk space

Installation procedure

You use the Install program to install VisSim/Model Wizard on your computer for the first time or to upgrade your existing copy of VisSim/Model Wizard to a more recent version of the software.

To install VisSim/Model Wizard

This procedure assumes that you are installing from drive A to your hard disk. If you are installing from a different drive, substitute the correct drive designation in the installation procedure.

1. Insert the disk labeled VisSim/Model Wizard into drive A.
2. Do one of the following:
 - Click on Start and choose Run.
 - Select File from the Program Manager menu bar and choose the Run command.
 - In the Command Line box, type A:INSTALL and click on the OK button, or press ENTER.
3. Follow the on-screen instructions.

Index

A

Analyzing the signalset 9
Automatic filtering 12

E

Editing data 7
Editing the dataset file 7

F

Filtering the output signal 11

G

General information 10

I

identification iterations
 maximum iterations 15
 relative error 15
 step size 15
Identification iterations 13
Input signal information 10
Installation procedure 17
Installation requirements 17
Installing VisSim/Model Wizard 17
Interactive webinars 3
Introduction 1

M

Manual filtering 12

N

Next step 11

O

Output signal information 11

P

Plots 11, 13
Plots for step input 15
Preparing the time history dataset file 5

R

Random input identification 14
Reading coordinates 11, 13
Resources for learning VisSim/Model Wizard 3
Returning to VisSim 16

S

Sample diagrams 3
Specifying the signalset 9
Starting VisSim/Model Wizard and choosing a dataset file 5
Step input identification 14

T

The VisSim product family 1
Training 3
Transfer function construction 15
Transposing data 7
Tuning parameters for step and random input 15

U

Using the VisSim/Model Wizard 5

W

What is VisSim/Model Wizard 1

Z

Zooming 13