commSIM 7

Network & Communications Simulation

Getting Started

Electronics
WORKBENCH

CS7-E-1716 Rev. 1

Table of Contents

1. Introduction

2. Commsim Basics

3. Inserting, Setting Up and Wiring Blocks

4. Arranging Blocks

5. Setting Simulation Properties

6. Simulating Block Diagrams

7. Viewing Simulations

Chapter 1
Introduction

The following are described in this chapter.

1.1 About this Chapter

This chapter introduces you to this manual and to Commsim 7 itself. It also gives detailed installation instructions.

1.2 About Commsim 7

Commsim provides the ideal solution for designing and simulating analog and digital end-to-end communication links. The Commsim library supports digital and analog modulation, channel models, demodulation, phase locked loops, error correcting codes, and bit error rate analysis, to mention a few.

Through its support of complex math, Commsim enables the use of complex envelope simulations. By using lowpass equivalent models, you can significantly reduce the computing load required to support most communication analysis problems.

1.3 About this Manual

This manual provides an introduction to the features available in Commsim. Depending on your version of Commsim your software may have some feature limitations. Refer to “1.5 Version Details” on page 1-3 for more information.

1.4 Conventions used in this guide

The following typographical conventions are used in this guide:

Convention	Where it’s used
Shortcut key combinations	Shortcut key combinations are joined with the plus sign (+). For example, the command CTRL+C means to hold down the CTRL key while you press the C key.
Hot keys	Hot keys are the underlined keys in Commsim’s menus, commands, and dialog boxes. To use a hot key, press ALT and then the key for the underlined character. For instance, to execute the File menu’s Save command, hold down the ALT key while you press the F key, then release both keys and press the S key.
SMALL CAPS	To indicate the names of the keys on the keyboard.
ALL CAPS	To indicate directory names, file names, and acronyms.
Italics	To reference a book, chapter, or section. Also used to emphasize certain keywords, and describe block parameter units.
Initial Caps	To indicate menu names, commands names, and dialog box options.

- Block descriptions are arranged alphabetically within each category. Block categories are presented in alphabetical order.
- Unless specifically stated otherwise, use the left mouse button whenever you are choosing a command, selecting a block, or activating a dialog box parameter. For example, when you read *press the OK button...*, you are to position the pointer over the specified object and click the left mouse button.
- To choose a menu command, you can use the mouse or you can press a sequence of keyboard keys. Only the mouse operations are documented.

1.5 Version Details

For details of limitations that you may find in your particular version of Commsim, please refer to the release notes.

1.6 Getting help

To help you get the most out of Commsim, the following online information is available:

- **Online help.** The online help contains step-by-step instructions for using Commsim features.
- **Online release notes.** A file named README.TXT is installed in your main Commsim directory. This file contains last minute information and changes that were discovered after this manual went to print. For your convenience, you should read this file immediately and print a copy of it to keep with this manual.

You may also find it helpful to browse through the sample block diagrams included with Commsim. These diagrams, which are listed in Chapter 13, “Sample Block Diagrams,”of the *Commsim 7 User Guide and Block Reference* demonstrate how Commsim is used to solve a broad spectrum of engineering and scientific problems.

1.6.1 Online help

The Help program provides online instructions for using Commsim.

➢ To open Help, do one of the following:

To	Do this
Access the top level of help	Select Help from the menu bar or press ALT+H.
Access help on the selected block	Click on the Help command button in the dialog box for the block.

➢ To close Help:

1. In the Help window, choose File > Exit, or press ALT+F4.

1.7 Technical support service

When you need assistance you can visit our Web site at www.electronicsworkbench.com and go to the Technical Support area. Alternatively, you may email Technical Support at support@electronicsworkbench.com. Also, the manual, the "readme" file, and the online Help are excellent sources of detailed information.

If you need further assistance contact Technical Support between 9:00 a.m. and 6:00 p.m. Eastern Standard Time, Monday through Friday, excluding holidays at 416-977-5550.

When you call in, please have the following information at hand:

- The version of Windows that you are using
- The type of hardware that you're using
- All screen messages
- What you were doing when the problem happened
- How you tried to solve the problem

Below is a list of support options:

Address/Number	Option
www.electronicsworkbench.com	Web Site
support@electronicsworkbench.com	Technical support
(416) 977-5550	Phone number
(416) 977-1818	Fax number

1.8 License Agreement

Please read this license carefully before installing and using the software contained in this package. By installing and using the software, you are agreeing to be bound by the terms of this license. If you do not agree to the terms of this license, simply return the unused software within thirty days to the place where you obtained it and your money will be refunded.

1. Copyright. The software in this package is copyright Interactive Image Technologies Ltd.(IIT) the parent company and owner of Electronics Workbench. The software is licensed for use only on the terms set forth herein.

Single User Version. This Agreement permits you to install one copy of the software, which is licensed as a single product. The software is installed on a computer when it is loaded into the hard drive or any other storage device of that computer. You must pay for additional copies if you wish to install the software on more than one computer at the same time or on a network. Certain exceptions to this policy exist in specific circumstances. Please contact Electronics Workbench for details.

Floating Network Version. This floating network license allows you to install this software on a network. Each station shall consist of only one computer with only one user at a time. The number of users restricted by this license is determined by the total number of concurrent users running the software from the network at the same time. The total number of concurrent users your particular floating network license is restricted to is _____.

2. Limited Warranty. IIT warrants that under normal use for a period of thirty (30) days from the date of delivery that: (a) the media on which the software is furnished will be free from defects in the material and workmanship; and (b) the software will operate substantially as described in the User's Guide (documentation). In order to make a claim under this warranty, you must call IIT for authorization to return any defective item during the warranty period. If you return merchandise to IIT, you must insure the defective item being returned because IIT does not assume the risk of loss or damage while in transit.

Upon return of a defective item, IIT shall, upon verification of the defect or error, at IIT's option, either repair or replace the defective copy or refund the amount paid for the license. If IIT elects to provide a refund, upon the date you receive notice of such election, this license shall terminate and you must comply with the provisions set out below.

You must assume full responsibility for the selection of this software to achieve your intended purposes, for the proper installation and use of the software and for verifying the results obtained from use of the software. IIT does not warrant in any way that the functions contained in the software will meet your requirements, that the software is fit for any particular purpose or that the operations of the software will be uninterrupted and error-free.

3. Term. The License granted in this Agreement is effective until termination. Your misuse shall automatically terminate this license if you breach any of its terms and conditions.

Upon termination, you shall return all media containing the software and all documentation to IIT and destroy any copies of the software or any portions of it which have not been returned to IIT, including copies resident in computer memory.

4. Copies, Modification or Merger. You shall not copy or modify all or any portion of the software or documentation or merge it into another software program. Copies shall include, without limitation, any complete or partial duplications on any media, adaptations, translations, compilations, partial copies within modifications, mergers with other material from whatever source, and updated works. You shall use your best efforts to prevent any unauthorized copying of the software.

You shall not make any change or modification to any of the executable files, nor shall you reverse engineer, de-compile or disassemble the software or any portion of it, or otherwise attempt to determine the underlying source code of the software or permit any such actions.

5. Disclaimer/Limitation of Liability. IIT expressly disclaims all other warranties, whether oral or written, express or implied, including without limitation warranties of merchantability or fitness for a particular purpose. All warranties shall terminate thirty days from date of delivery of the software to you.

Your exclusive remedy and IIT's entire liability arising from or in connection with the software, the software documentation, and/or this license (including without limitation for breach of warranty) shall be, at IIT's option, the repair or replacement of software diskettes or refund of license fee.

In no event shall IIT's total liability for any damages, direct or indirect, in connection with the software, the software documentation, and/or this license exceed the license fees paid for your right to use this copy of the software, whether such liability arises from any claim based upon contract, warrants, tort or otherwise. In no event shall IIT or its partners be liable for any loss of profit or any other commercial damage, including but not limited to special, incidental, consequential or other damages, resulting from or in any way connected with the use of this software and including but not limited to any damages resulting from the use of the software for any special or high-risk applications such as those relating to or involving nuclear designs, medical devices and other critical or potentially dangerous applications.

IIT specifically disclaims any other warranties, expressed or implied, including but not limited to the implied warranties of merchantability and fitness for a particular purpose.

Some jurisdictions do not allow the exclusion of implied warranties, so the above exclusion may not apply to you. In that event, any implied warranties are limited in duration to thirty (30) days from the date of delivery of the software. This warranty gives you specific legal rights. You may have other rights, which vary from jurisdiction to jurisdiction.

No action for any breach of warranty shall be commenced more than one year following the expiration of such warranty.

6. General. You acknowledge that you have read this agreement, understand it and agree to be bound by its terms and conditions. You further agree that it is the complete and exclusive statement of the agreement between you and IIT and supersedes any proposal or prior agreement or any other communications between IIT and you relating to the use of the software.

If any provision of this Agreement is unenforceable, all others shall remain in effect. This Agreement shall be governed by the internal laws of the Province of Ontario and Canada, including Canadian copyright laws. The exclusive venue in the event of any suit, proceeding or claim brought by you, and at our option, any suit, proceeding or claim brought by IIT, shall be in the Courts located in Toronto, Ontario. If you have any questions regarding this Agreement, you may contact IIT by writing to us at the address set out below:

INTERACTIVE IMAGE TECHNOLOGIES LTD.

111 Peter Street, Suite 801 Toronto, Ontario M5V 2H1

Tel : (416) 977-5550 **Fax:** (416) 977-1818

e-mail: ewb@electronicsworkbench.com

Internet: www.electronicsworkbench.com

1.9 Installing Commsim

To successfully install Commsim, you need up to 50 MB of hard disk space, depending on the options you install. You also need, as a minimum, the following system requirements:

- Windows 98/NT/2000/XP
- 32 MB RAM (64 MB recommended)
- CD-ROM drive
- 800 x 600 minimum screen resolution

Commsim has two types of installation: Single User and Floating Network.

Single User Installation. If you are installing Commsim on a single computer that is not connected to another computer, you are installing for a Single User. For details, see “1.9.1 Single User Installation” on page 1-8.

Floating Network Installation. If you are installing Commsim on a network and the number of concurrent users is being counted, not the number of computers connected to the network, you are installing a Floating Network. For details, see “1.9.2 Installing Commsim Floating Network Version” on page 1-9.

Note **MatLab and Microsoft Visual C/C++:** If you wish to take advantage of Commsim’s full functionality, you will need MatLab and Microsoft Visual C/C++. Please ensure these applications are installed prior to installing Commsim. If you do not have these

applications or do not want to use these interfaces, click No when prompted during installation.

1.9.1 Single User Installation

The CD-ROM you received in your Commsim package is self-starting. Follow the directions below and on the screen during the installation process.

➢ To install Commsim:

1. Copy your serial number in the space provided. You will find the serial number on the front of the CD-ROM package:

   Serial Number _____

2. Exit *all* Windows applications prior to continuing with this installation.
3. Insert the Commsim CD into your CD-ROM drive. When the "Welcome" screen appears, read it. Click **Next** to continue.
4. Read the Commsim License Agreement, which can be found in "1.8 License Agreement" on page 1-5. To accept the terms of the agreement, and to continue to the next screen, click **Yes**. If you do not accept the terms of the agreement, click **No** and the Commsim installation will be terminated.
5. Enter your name, company and the serial number. Click **Next** to continue. A screen will appear verifying the serial number. Click **OK** to continue.
6. Select the location in which you want to install Commsim. Choose the default folder or click **Browse** to select a different location or to enter your own folder name. Click **Next** to continue.
7. Setup will now create the Commsim 7 program folder. Click **Next** to continue.
8. If you wish to install the MatLab Interface, click **Yes**. Enter the full path to MATLAB.EXE in the field provided. Click **OK** to continue.

Note Do not include the MATLAB.EXE file in the path.

9. If you wish to install the Commsim DLL Wizard Interface, click **Yes**. Enter the full path to the BIN\IDE subfolder in MS Visual C++. Click **OK** to continue.

Note Not all editions of Commsim allow you to access the Visual C/C++ interface.

10. The setup routine will now complete the installation. If for any reason you wish to stop the installation, click **Cancel** and the installation of Commsim will be terminated.
11. Restart your computer after completion of the setup routine.

12. There are five icons created:
    - Comm Blocks Help
    - Commsim 7
    - Commsim Help
    - Getting Started
    - User Guide
    - WLAN Help

1.9.1.1 Getting the Release Code for Standalone PCs

You will notice when you run Commsim that the splash screen will prompt you for a release code. The screen indicates that a grace period of 15 days is provided starting on the day of installation. Each day the grace period will decrease by one day. When the grace period reaches the 16th day, Commsim will not run until the release code is provided. Electronics Workbench strongly recommends that you obtain your release code as soon as possible after you have installed Commsim.

To obtain your release code, you must have your serial number and signature code available (located on the splash screen). Contact Electronics Workbench via our web site at www.electronicsworkbench.com and select Product Registration, or call our Customer Service number at 1-800-263-5552.

➢ To enter the release code:

1. At the splash screen, click **Enter Release Code**. This activates the release code text box.
2. Point and click the cursor into the first field of the text box.
3. Enter the 12 digit release code.
4. Press **Accept**. Commsim will start up.

1.9.2 Installing Commsim Floating Network Version

The CD-ROM you received in your Commsim package is self-starting. Follow the directions below and on the computer screen during the installation process.

➢ To install Commsim:

1. Copy your serial number and feature code in the spaces provided. You will find both numbers on the front of the CD-ROM package:

   Serial number _____

   Feature code _____

Note A feature code is not the same as a serial number. The feature code monitors the number of simultaneous users on the network using Commsim.

2. Insert the Commsim CD into your CD-ROM drive. When the “Welcome” screen appears, read it. Click **Next** to continue.
3. Read the License Agreement. To accept the terms of the agreement, and to continue on to the next screen click **Yes**. If you do not accept the terms of the agreement, click **No** and the Commsim installation will be terminated.
4. Enter your name, company and the serial number, click **Next** to continue. A comment box will appear to verify the serial number and version, click **OK** to continue.
5. Enter the feature code, click **Next** to continue. A comment box will appear to verify the number of users on the network.
6. To enter another feature code, click **Yes**. Otherwise, click **No** to continue the setup.
7. Select the network drive/location in which you want to install Commsim. Click the **Browse** button to select a different location or enter your own folder name. Choose **Next** to continue.

Note You MUST install Commsim in a location on the network drive that is accessible to all workstations running Commsim.

8. Setup will now create the Commsim 7 program folder. Click **Next** to continue.
9. If you wish to install the MatLab Interface, click **Yes**. Enter the full path to MATLAB.EXE in the field provided. Click **OK** to continue.

Note Do not include the MATLAB.EXE file in the path.

10. If you wish to install the Commsim DLL Wizard Interface, click **Yes**. Enter the full path to the BIN\IDE subfolder in MS Visual C++. Click **OK** to continue.

Note Not all editions of Commsim allow you to access the Visual C/C++ interface.

11. The setup routine will now complete the installation. If for any reason you wish to stop the installation, click **Cancel** and the installation of Commsim will be terminated.
12. Restart your computer after completion of the setup routine.
13. There are five icons created:
    - Comm Blocks Help
    - Commsim 7
    - Commsim Help
    - Getting Started
    - User Guide
    - WLAN Help

1.9.2.1 Restricting Write Privileges

Access to the Commsim directories should be restricted by placing permissions on various directories. Follow the instructions below to place the appropriate permissions.

Windows NT Client/Server Install

In the main installation directory (N:\Commsim, where N: is the network drive where Commsim is installed, and Commsim is the name of the folder specified at installation), share the directory with share permissions for Everyone set to “Full Control”.

Security on the Commsim directories should be:

Administrator	Full Control
User	(R)ead E(X)ecute

Set up the directory attributes for the subdirectories as follows:

Directory	Attributes - User
Lock	(R)ead (W) rite E(X)ecute

Novell Network Client/Server Install

For the main installation directory (N:\Commsim, where N: is the network drive where Commsim is installed, and Commsim is the name of the folder specified at installation), map a network drive to the directory. Set up security permissions for this directory as follows:

Administrator	(S)upervisor
User	Special directory access (R)ead (F)ile Scan

Set up the directory attributes for the subdirectories as follows:

Directory	Attributes – User
Lock	(R)ead (W)rite (C)reate (F)ile Scan

- To install the Commsim program folder and icons on client computers:
  1. Map a network drive letter to the shared directory on the server.
  2. Navigate to the Commsim folder on the network.
  3. Open file SETUP.BAT and follow the prompts on the screen.

1.9.2.2 Getting the Release Code for a Floating Network Version

You will notice when you run Commsim that the splash screen will prompt you for a release code. The screen indicates that a grace period of 15 days is provided starting on the day of installation. Each day the grace period will decrease by one day. When the grace period reaches the 16th day, Commsim will not run until the release code is provided. Electronics Workbench strongly recommends that you obtain your release code as soon as possible after you have installed Commsim.

To obtain your release code, you must have your serial number and signature code available (located on the splash screen). Contact Electronics Workbench via our web site at www.electronicsworkbench.com and select Product Registration, or call our Customer Service number at 1-800-263-5552.

- To enter the release code:

1. At the splash screen, click **Enter Release Code**. This activates the release code text box.
2. Point and click the cursor into the first field of the text box.
3. Enter the 12 digit release code.
4. Press **Accept**. Commsim will start up.

Note Only one release code is required and the number can be installed from any station.

1.10 Uninstalling Commsim 2001

1.10.1 Single User Uninstall

- To uninstall Commsim from a standalone PC:

1. Ensure you have the serial number written down prior to uninstalling Commsim.
2. Click the Windows **Start** button.
3. Click **Settings** and **Control Panel**.
4. Click **Add/Remove Programs**. The Add/Remove Programs Properties screen appears.
5. From the **Install/Uninstall** list, select **Commsim** and select **Add/Remove**. Commsim will be removed from your computer.

1.10.2 Uninstalling Commsim 2001 Floating Network Version

Only the workstation that was used to install Commsim to the network directory (host computer) can be used to remove it. Performing the steps below on other workstations (client computer) will only remove the Commsim folder.

- To uninstall Commsim from the network directory or the Commsim folder:

1. Ensure you have the serial number and any feature codes written down prior to uninstalling Commsim.
2. Click the Windows **Start** button.
3. Click **Settings** and **Control Panel**.
4. Click **Add/Remove Programs**. The Add/Remove Programs Properties screen appears.
5. From the **Install/Uninstall** list, select **Commsim** and select **Add/Remove**. If the workstation is the host computer, Commsim is removed from the network directory. If the workstation is only a client computer, the Commsim folder is removed.

Chapter 2
Commsim Basics

The following are described in this chapter.

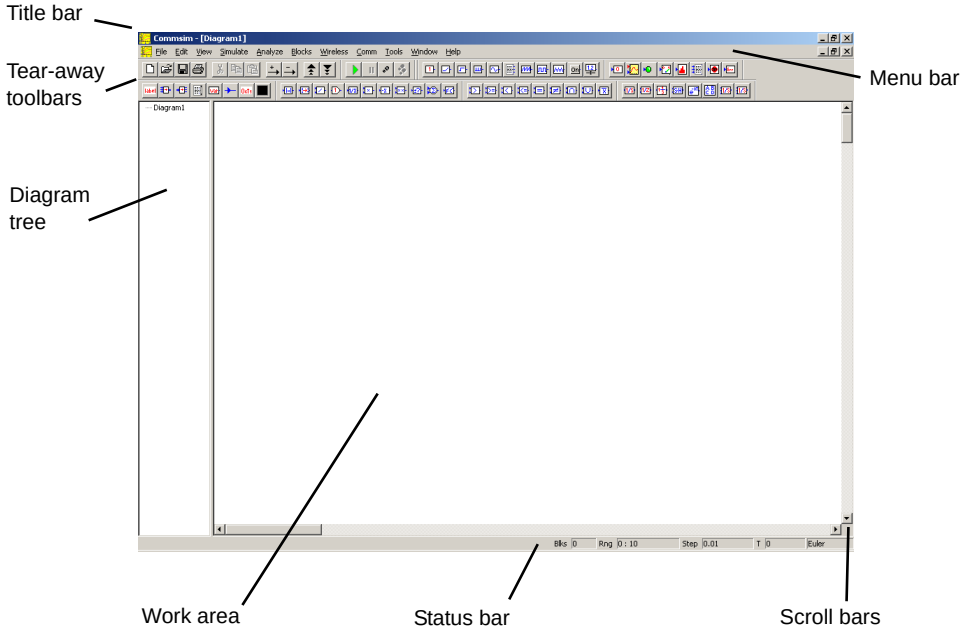
2.1 Starting Commsim

This procedure describes the start-up methods for Commsim. You can customize the start-up procedure by editing the Commsim command line, as described in the *Commsim 7 User Guide and Block Reference*.

1. Click on Start > Programs > Commsim 7.
2. Double-click on the Commsim icon.

2.2 Exploring the Commsim window

When you start Commsim, a new, empty block diagram, like the one shown below, is automatically created for you.



Title bar: Lists the application name and currently opened block diagram. Unnamed diagrams are titled *Diagram1*. The title bar also contains the Minimize, Maximize, and Close buttons. The Minimize button shrinks the Commsim window to an icon; the Maximize button enlarges the Commsim window to fill your entire screen; and the Close button closes the Commsim window.

Menu bar: Lists the seven basic menus available in Commsim: File, Edit, Simulate, Blocks, Tools, View, and Help. Clicking on a menu name displays a list of Commsim commands or blocks.

Tear-away toolbars: The buttons in the toolbars represent commonly used Commsim commands and blocks. To select a toolbar button, click on it.

Each cluster of buttons represents a tear-away toolbar. For example, the Main toolbar consists of the following buttons:

There are eight tear-away toolbars: Main, Sim Control, Annotation Blocks, Arithmetic Blocks, Boolean Blocks, Consumer Blocks, Producer Blocks, and User.

By default, the Main and Sim Control toolbars appear when you start up Commsim. These toolbar buttons are described below.

A **B** **C** **D** **E** **F** **G** **H** **I** **J** **K** **L** **M** **N**

- **A** File > New command
- **B** File > Open command
- **C** File > Save command
- **D** File > Print command
- **E** Edit > Cut command
- **F** Edit > Copy command
- **G** Edit > Paste command
- **H** Edit > Add Connector command
- **I** Edit > Remove Connector command
- **J** Simulate > Go command
- **K** Simulate > Stop command
- **L** Simulate > Single Step command
- **M** Simulate > Continue command
- **N** Help command

As their names imply, the Annotation, Arithmetic, Boolean, Consumer, Producer Blocks, and User toolbars represent blocks in each of the corresponding categories. For descriptions of these blocks, refer to the *Commsim 7 User Guide and Block Reference*.

If a toolbar restricts your view of your work, drag on its background to move it to a new location, or click on its background to display a menu from which to close it. You can also use the View > Toolbar command to close toolbars. For more information, refer to the *Commsim 7 User Guide and Block Reference*

Dimmed toolbar buttons

Sometimes, when a toolbar button is dimmed, it is because the last cursor position was in the left windowpane. Click the mouse anywhere in the right windowpane to activate all available toolbar buttons.

Status bar: Provides simulation information about the current diagram, including the block count, simulation range, integration algorithm, step size, and implicit solver. When you run a simulation, the elapsed simulation time is also displayed.

Displaying menu command and toolbar descriptions in status bar

When you drag the mouse over a menu, menu command, or toolbar button, Commsim displays a brief description of the item in the status bar.

You can show or hide the status bar at any time using the View > Status Bar command.

Scroll bars: Pans the current viewing window. There are three ways to pan with the scroll bars. Click on the scroll arrows to scroll in small increments; click on the scroll bar to scroll in screen increments; or drag the scroll box to a location on the scroll bar that approximates a location in the block diagram.

You can show or hide the scroll bars at any time using the Edit > Preferences command.

Auto-panning

Whenever you drag a block or draw a wire beyond the visible portion of the working area, Commsim will automatically scroll the work area.

Diagram tree: The Commsim window is divided into two panes. The left pane displays a *diagram tree*; that is, an outline of the diagram's compound blocks. At the top of the diagram tree is the Block Diagram icon, which represents the highest level of the currently opened block diagram. Its name appears next to the icon.

Beneath Block Diagram are the names of the compound blocks encapsulated in the block diagram. You can expand and collapse the diagram tree to display more or less detail by

clicking on the plus or minus signs that appear next to the diagram and compound block names.

Whatever you select in the diagram tree is displayed in the right windowpane. For example, if you select Block Diagram, the top level of the block diagram is displayed in the right windowpane. You can jump to a specific compound block without wading through block diagram hierarchy by simply selecting the compound block name in the diagram tree.

If the diagram tree takes up too much space or if you cannot see all the hierarchical information in the tree, you can change its width by dragging its right edge.

2.3 Choosing commands

You can choose menu commands using the mouse or the keyboard. To choose a menu command with the mouse, click on the menu, and then click on the command. To choose a menu command with the keyboard, press ALT to activate the menu bar, then press the key corresponding to the underlined letter in the menu, and finally press the key corresponding to the underlined letter in the command.

For commonly used menu commands, you can either:

- Press shortcut keys, which are listed on the menu to the right of the commands. For example, press CTRL+C to execute Edit > Copy.
- Press a corresponding toolbar button. For example, press to execute Simulate > Go.

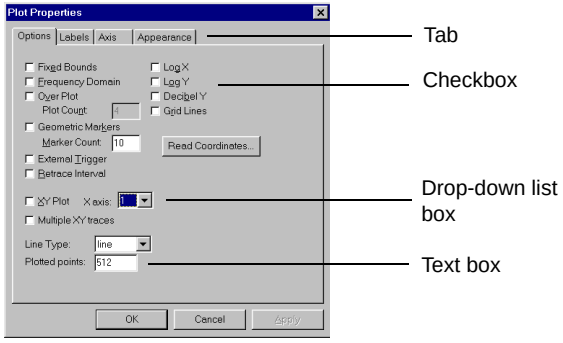
If a menu command is dimmed, it is unavailable for use.

Missing dialog boxes

If you choose a command with an ellipsis (for example, the File > Print command and the dialog box for the command is not displayed, click the mouse anywhere in the right windowpane and re-select the command.

2.4 Using dialog boxes

Commsim uses dialog boxes to gather and display information about a command or block.

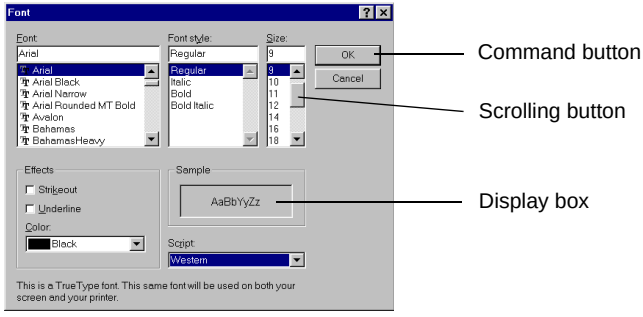


Tab: Allows similar options to be grouped together. When you click on a tab, the corresponding property sheet is brought to the front.

Check box: Sets or clears a particular option. When a checkmark appears in the box, the option is activated.

Drop-down list box: Provides a list of several options. Click on the DOWN ARROW to select from a list of options.

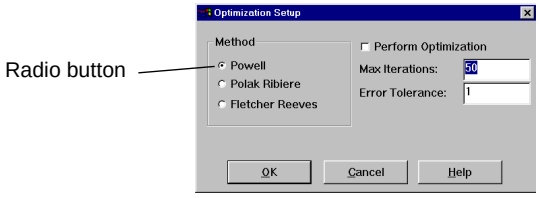
Text box: Allows you to enter text strings. Move the pointer over a text box until it changes into an I-beam; then type in the text.



Command button: Causes an action to happen. Command buttons with ellipsis invoke another dialog box. Command buttons with a darkened rim are the default action. You can press the enter key to execute the default command button.

Scrolling list: Allows you to select from a list. Click on the scroll bar, scroll box, or scroll arrows to scroll through the list.

Display box: Provides a visual representation of your selection.

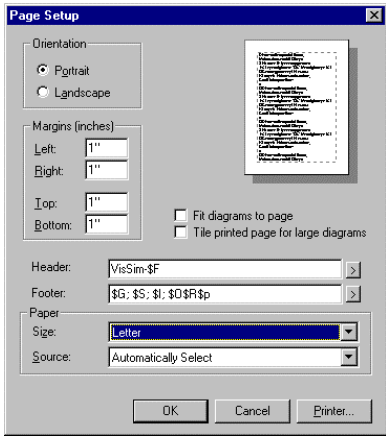


Radio button: Used to present two or more mutually exclusive options. You must pick one of the choices by clicking on it. When a black dot appears in the radio button, it is selected.

2.5 Creating and setting up a new block diagram

To open a new diagram, choose the File > New command, or click on in the toolbar. If you're working on a different diagram and haven't yet saved your changes, Commsim prompts you to save them, then creates a new diagram. Commsim temporarily names the diagram *Diagram1*. The first time you attempt to save it, Commsim asks for a new name.

When you begin working on a new diagram, you usually start by setting up the page with the File > Page Setup command. As you choose options in the Page Setup dialog box, a sample of your selections is displayed in the top right-hand corner of the dialog box.



Orientation: You have the choice of Portrait or Landscape. Activate Portrait for a page that is taller than it is wide. If you want the opposite, activate Landscape.

Margins (Inches): The margins control the distance between diagram elements (blocks and wires) and the edge of the paper. Commsim does not display margins unless you are in print preview mode. In this mode, they appear as blue, nonprinting lines. Headers and footers, if specified, appear inside the margins.

Paper: Click on the DOWN ARROW in the Size box and select a standard paper size from the list; then click on the DOWN ARROW in the Source box to select the paper source (that is, the tray the printer uses to print the diagram).

Fit Diagram to Page: When Fit Diagram to Page is activated, Commsim prints each level of the block diagram on a separate page. When necessary, Commsim reduces diagram text so the level will fit on a single page within the specified margins. Because Commsim prints each level individually with the minimal reduction possible, the levels of a multi-level diagram may be sized differently.

Commsim may not be able to print extremely large block diagrams when Fit Diagram to Page is activated. In these cases, Commsim gives you the opportunity to abort the print operation. If you choose to continue printing, Commsim prints as much of the diagram as will fit on the page.

Tile Printed Page for Large Diagrams: Tile Printed Page for Large Diagrams causes Commsim to print each level using as many pages as necessary to print it without resizing. The margin settings are honored for each page.

Header and Footer: You can create headers and footers by entering text in the Header or Footer box. To view headers and footers, you must be in print preview mode. Headers and footers appear within the established page margins on each printed page of the diagram.

Using fields to enter header and footer information

By using fields, you can automatically insert information into a header or footer. For example, you can use fields to insert the file name of a diagram, the date the diagram was created, and so on.

To enter a field, click on the DOWN ARROW in the Header or Footer box and select one or more fields from the list. When you select a field, it appears as a field code in the Header or Footer box.

Field	Field code
File name	$f
File path	$F
Block path	$H

Field	Field code
Date	$D
Integration method	$I
Optimization	$O
Page number	$p
Range	$G
Step size	$S
Left justify	$L
Center	$C
Right justify	$R

Using File > Print for page setup

You can use the File > Print command to reset the orientation, paper size, paper source, tiling, and fit-to-page options.

2.6 Opening an existing block diagram

You can easily open any of the last 12 block diagrams you worked on. When you click on the File menu, Commsim displays their names at the bottom of the menu.

To open any block diagram, choose the File > Open command. When you assign a title to a block diagram using the Diagram Information command, the title appears in the File Open dialog box when you select the block diagram.

➢ To open a block diagram

1. Do one of the following:
   - From the toolbar, choose  .
   - Choose File > Open.
2. In the File Name box, type or select the name of the block diagram you want to open. If you do not see the block diagram you want, select a new drive or directory.
3. Click on the Open button, or press ENTER.

2.7 Undoing an editing action

You can use the Undo command in the Edit menu to undo one or more previous actions or operations. There is no maximum number of Undo levels. You can also re-instate an “undone” action or operation using the Redo command, also located in the Edit menu. If the Undo command is dimmed, the effect of the command cannot be undone.

The following actions can be undone and then re-instated:

- Block insertion
- Block movement (including dragging, flipping, copying, cutting, deleting, and pasting)
- Wiring
- Creating and dissolving compound blocks

➢ To undo mistakes:

1. Choose Edit > Undo, or press CTRL+Z.
2. If you later decide you didn't want to undo the action, choose Edit > Redo, or press CTRL+A.

2.8 Repainting the screen

Choosing Repaint Screen under the Edit menu redraws blocks and wires, and clears the screen of remnants left over from earlier Commsim manipulations.

2.9 Saving a block diagram

When you open a block diagram, Commsim reads the diagram into your computer’s memory. As you work on the diagram, the changes you make are temporary. To make the changes permanent, you must save them to disk.

Retaining diagram appearance on different graphic resolutions

If you activate Snap To Grid under Preferences in the dialog box for the Edit > Preferences command, Commsim saves block positions in units of ½ the average character size of the currently selected font. This results in a more consistent appearance of the block diagram over different graphic resolutions.

➢ To save an existing block diagram:

1. From the toolbar, choose  .

   Or

   Choose File > Save.

If the diagram contains one or more embedded diagrams, any changes you made to those diagrams are not saved until you use the File > Save Embedded Files command.

2.9.1 The Save All command

The Save All command saves the changes you made to all the opened diagrams. If, however, you made changes to block diagrams embedded in an opened diagram, you must make the opened diagram the active diagram, then use the File >Save Embedded Files command.

2.9.2 The Save As commands

You can use File > Save As to save the block diagram under a new name or to a different directory or device. This command comes in handy when you want to alter the current diagram but keep its original version.

You can use the File > Save As Metafile command to save a block diagram in Windows Metafile Format (WMF). You use this command to insert a picture of your diagram in another application, such as a Word document.

2.10 E-mailing a block diagram

You can route diagrams online by using any 32-bit e-mail program that is compatible with the Messaging Application Programming Interface (MAPI). If your e-mail program is compatible with MAPI, the Send command appears when you point to the File menu.

➢ To send a diagram block:

1. Open the diagram you want to send.
2. Choose the File > Send command. A message appears asking if you want to include the Commsim Viewer install with the current diagram.
3. To include the Viewer in your e-mail, click on Yes, or press the ENTER key. Your default e-mail program starts up. The current e-mail message includes a copy of the opened block diagram.
4. Address the e-mail to the desired recipients and send.

2.11 Previewing before printing

Use the File > Print Preview command to display a block diagram as it will look when printed. Headers and footers, if specified, appear at the top and bottom of the pages according to the specifications established with the File > Page Setup command. Similarly, margins, if specified with File > Page Setup, are displayed in nonprinting, blue ink.

You can zoom in and out of the page using the Zoom buttons in the Print Preview toolbar.

2.12 Printing

The File > Print command lets you choose a printer and select printing options, such as the number of copies, the layers to be printed, and so on.

- To print a block diagram:
  1. From the toolbar, choose  .

     Or

     Choose File > Print.
- To set printing options:
  1. From the toolbar, choose  .

     Or

     Choose File > Print.

2. Do one or more of the following:

To print	Do this
Multiple copies	In the Copies box, enter a number.
The current level of the diagram	Under Print Range, activate Current Level.
The current level and below	Under Print Range, activate Current Level and Below.
All levels of the diagram	Under Print Range, activate All.
Each level of the block diagram on a separate page, and when necessary, reduce diagram text so the level fits on a single page	Activate Fit to Page. Because Commsim prints each level with the minimal reduction possible, the levels of a multi-level diagram may be sized differently. Commsim may not be able to print extremely large block diagrams when Fit to Page is activated. In these cases, Commsim gives you the opportunity to abort the print operation. If you choose to continue printing, Commsim prints as much of the diagram as will fit on the page.
Each level using as many pages as necessary to print it without resizing	Activate Tile Pages. The margin settings are honored for each page.
A version of the block diagram to a file to be printed at a later date or to be used in another program	Activate Print to File and then click on the OK button, or press ENTER. In the ensuing dialog box, specify a name for the block diagram you want to print.

Selecting a printer: The currently selected printer appears in the Printer box when you choose the File > Print command. To select a different printer, choose the Setup command button. If the printer you want to use is not listed, you must install the printer driver software on your system via the system Control Panel. See the *Microsoft Windows User's Guide* for printer installation instructions.

Selecting additional printing options: You can also specify a paper size, orientation, and paper source for the printed diagram by clicking on the Setup command button. The selections you make here override selections you made previously with the File > Page Setup command.

2.13 Setting up the Commsim environment

You can customize Commsim to suit the way you work. You can, for example, use commands in the View menu to change the color of the work area, plotting area, and wires, specify diagram fonts, and switch between presentation modes. Other preferences are set with the Edit > Preferences command. These include coloring compound blocks, alternating between black and white, and color displays, and using training mode.

For more information on setting up the Commsim environment, refer to the *Commsim 7 User Guide and Block Reference*.

2.14 Closing a diagram and exiting Commsim

When you're finished working and decide to exit Commsim, use the File > Exit command, or press ALT+F4 to end your Commsim session. Commsim checks that all your work has been saved. If there are any unsaved changes, Commsim asks if you want them saved before exiting. If you want to work on another diagram in Commsim, close the active diagram with the File > Close command and then open another one.

Chapter 3
Inserting, Setting Up and Wiring Blocks

The following are described in this chapter.

3.1 Block basics

In Commsim, you build system models in the form of block diagrams. Blocks are your basic design component. Each block represents a specific mathematical function. The function can be as simple as a sin function or as complex as a 15^{th} order transfer function.

3.1.1 Types of blocks

Commsim offers over 90 blocks for linear, nonlinear, continuous, discrete-time, time varying, and hybrid system design. Blocks are categorized under the Blocks menu as follows:

- Animation
- Annotation
- Arithmetic
- Boolean
- DDE
- Integration
- Linear Systems
- MatLab Interface
- Matrix Operations
- Nonlinear
- Optimization
- Random Generator
- Signal Consumer
- Signal Producer
- Time Delay
- Transcendental

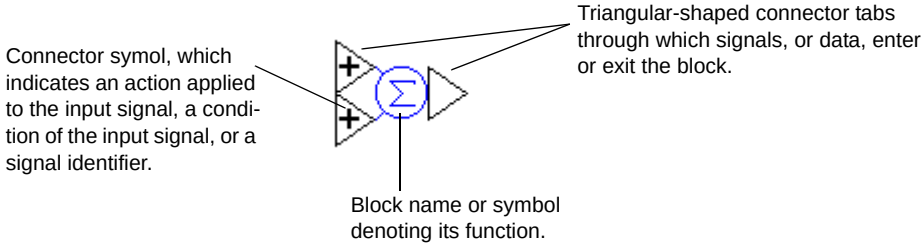
In addition, Commsim supplies four special-purpose blocks: `embed`, `expression`, `OLEobject`, and `userFunction`.

Custom blocks

If your design requirements extend beyond the blocks supplied by Commsim, you can create custom blocks in C, C++, Fortran, or Pascal, as described in the *Commsim 7 User Guide and Block Reference*.

3.1.2 Identifying block parts

When you insert a block into a diagram, various symbols and text appear on it.



You can display additional information on the blocks in your diagram using the View > Block Labels command, as described in the *Commsim 7 User Guide and Block Reference*.

3.2 Inserting blocks

You insert blocks into a diagram by selecting them from the Blocks menu and placing them in the work area. When you click on the Blocks menu, a list of blocks and block categories appears. Block categories are depicted by filled triangles. When you click on a block category, a cascading menu appears listing the additional blocks.

➢ To insert a block from the Blocks menu:

1. Choose Blocks from the menu bar.
2. Point to the block category and click the mouse. For example, point to Nonlinear and click the mouse to display the nonlinear blocks.
3. Point to a block and click the mouse. For example, point to `crossDetect` and click the mouse to choose the `crossDetect` block.

   The Blocks menu closes and a rectangular box appears with the pointer anchored in the upper left-hand corner of the box.
4. Point to the location in the diagram where you want to insert the block and click the mouse.

3.3 Setting up block properties and initial conditions

Most blocks have user-settable properties associated with them that allow you to set simulation invariant parameters of the blocks' functions. You define and change property values for a block through its Properties dialog box. When you change a property value while the simulation is running, Commsim immediately updates the simulation to reflect the change. Initial conditions, which are supplied to the system at the start of a simulation, are also set in the blocks via their Properties dialog boxes.

➢ To set up block properties:

1. Choose Edit > Block Properties.
2. Point to the block whose parameters you want to define or change and click the mouse.
3. In the Properties dialog box, enter or select the new parameter values and options, and then choose the OK button, or press ENTER.

Shortcuts for accessing Properties dialog boxes

A shortcut for accessing Properties dialog boxes for most blocks is to click the right mouse button over the block. For `button`, compound, and `embed`, blocks, hold down the CTRL key while you click the right mouse button to access their Properties dialog boxes.

3.3.1 Entering numeric data

When entering numeric data, Commsim displays values greater than 106 or less than 10-6 in exponential notation. Commsim uses the letter "e" to indicate exponential notation; however, on input, it also recognizes the letter "E." For example, you can enter 6,000,000 in the following ways: 6e6 or 6E6.

3.3.2 Entering arithmetic expressions

Most numeric block parameters can be expressed using the arithmetic operators "+," "-," "*," "/" and the usual rules of precedence. For example:

2 * (5 + 4) = 18

2 * 5 + 4 = 14

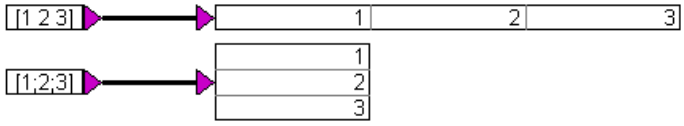
3.3.3 Entering complex numbers

The following blocks support complex numbers: `const`, `mul`, `div`, `add`, `gain`, `merge`, `case`, `unitDelay`, and `sampleHold`. The `const` and `unitDelay` blocks allow you to enter a complex number. You do so in the following format:

(real-part, imaginary-part)

The `const` and `expression` blocks support matrix data.

You enter matrix construction to the Value text box of the `const` block and the Expression text box of the `expression` block. When entering matrix data, enclose it in square brackets ([]) and separate each element with a space (to start a new column) or semi-colon (to start a new row). For example:



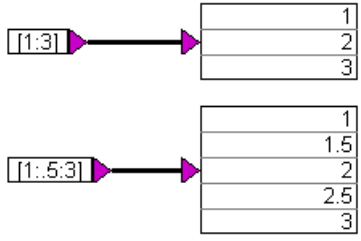
To form an N element vector, use the following notation:

[start:end]

To specify a specific increment, use this notation:

[start:increment:end]

For example:

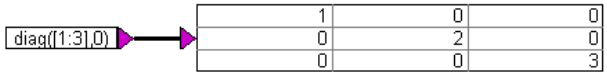


The const and expression blocks also support these matrix constructs:

- diag()
- eye()
- ones()
- zeros()

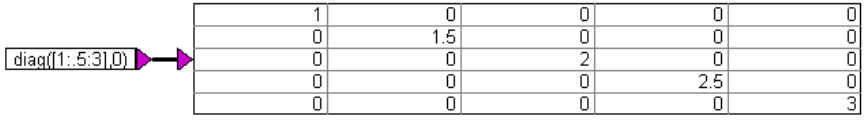
3.3.3.1 diag(V,K)

The diag() construct creates a square matrix with the vector V on the diagonal and zeros elsewhere. K specifies the offset from the main diagonal. If you omit K, it defaults to 0; therefore, diag(V) puts V on the main diagonal.

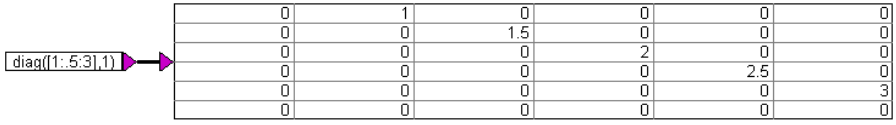


This `const` block creates a 3-by-3 matrix with the elements 1, 2, 3 on the diagonal.

To specify an increment of 0.5, do the following:



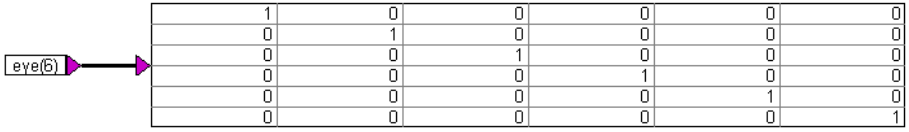
Here, a 5-by-5 matrix is created to include the elements 1, 1.5, 2, 2.5, and 3 on the diagonal. To specify an offset of 1, do the following:



Commsim expands the matrix to 6-by-6 so that the diagonal vector can be placed above the main diagonal.

3.3.3.2 eye(N)

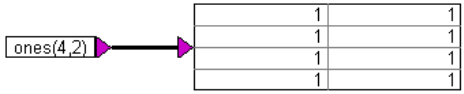
The eye() construct creates an identity matrix N, where N represents the number of rows and the number of columns. Commsim fills in the diagonal with ones and zeros elsewhere. For example:



This `const` block creates a 6-by-6 matrix with ones on the diagonal and zeros elsewhere.

3.3.3.3 ones(R,C)

The ones () construct creates an R-by-C matrix of all ones, where R represents the number of rows and C represents the number of columns. If you omit C, Commsim creates an R-by-R matrix.



This `const` block creates a 4-by-2 matrix filled with ones.

3.3.3.4 zeros(R,C)

The zeros () construct creates an R-by-C matrix of all zeros, where R represents the number of rows and C represents the number of columns. If you omit C, Commsim creates an R-by-R matrix.

This `const` block creates a 1-by-4 matrix filled with zeros.

3.3.4 Entering C expressions

Commsim also recognizes C expressions for numeric data. This means you can build elementary mathematical functions using acos, asin, atan2, cos, cosh, exp, fabs, log, log10, pow, sin, sinh, sqrt, tan, and tanh. For example, if you enter pow (2,3) to the Gain parameter on the `gain` block, Commsim calculates 8. Commsim also interprets the universal constant *pi* as π.

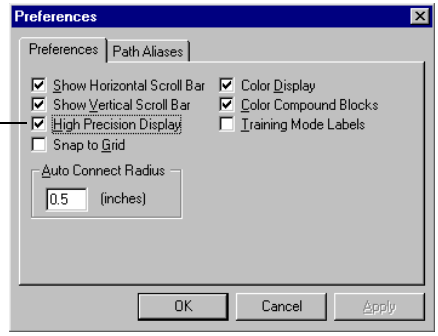
Learning C

If you are unfamiliar with the C language and want to learn how to enter mathematical functions in C format, see *C: A Software Engineering Approach*, (P. Darnell and P. Margolis, Springer-Verlag, 1990).

3.3.5 Controlling the number of displayed significant decimal digits

Numeric block properties are always calculated in up to 15 significant decimal digits; however, you have the choice of displaying them in up to 6 or 15 significant decimal digits. The High Precision Display option under Preferences in the dialog box for the Edit > Preferences command controls the display.

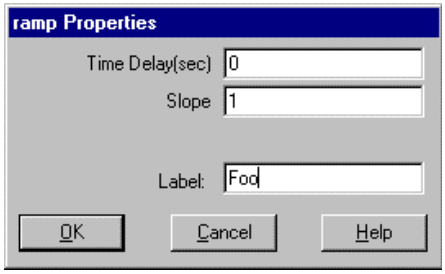
A check activates 15 significant decimal digit display.



3.3.6 Identifying blocks by user-defined labels

For most blocks, a user-defined label can be attached to the block through its Properties dialog box. When View > Block Labels is activated, the labels you specified appear beneath the corresponding blocks. For example, the following ramp block has been assigned the label Foo:

You create a label in the Properties dialog box for a block.



In the above dialog box, the label Foo is assigned to a `ramp` block. When you move the `ramp` block, the label automatically moves with the block.

When searching for a block with the Edit > Find command, you can use the block label as the search item.

➢ To label a block:

1. Choose Edit > Block Properties.
2. Click on the block to be labeled.
3. In the Label box, enter a label. A label cannot be longer than the length of the Label text box. If you need a longer label, use the `label` block under the Annotation category of the Blocks menu.
4. Click on the OK button.
5. Activate the View > Block Labels command. A check mark in front of the command indicates that it is active.

3.4 Wiring basics

By wiring blocks together, Commsim is able to pass signals among blocks during a simulation. Signals are simply data. Input signals (x_n) represent data entering blocks; output signals (y_n) represent data exiting blocks.

Wireless transmission of signals

A `variable` block lets you name and transmit a signal throughout a block diagram without using wires. Typically, you use a `variable` block for system-wide variables or signals that would be laborious or visually messy to represent as wires. For more information, refer to the *Commsim 7 User Guide and Block Reference*.

3.4.1 Types of wires

Commsim offers two types of wires:

- flexWires
- vector wires

A flexWire is a thin wire that allows a single signal to pass through it. A vector wire is a thick wire that contains multiple flexWires. Typically, you use vector wires when performing vector or matrix operations, or to reduce wiring clutter at top-level diagram design.

The table below lists the blocks that accept vector wires:

Block category	Block name
Annotation	index, scalarToVec, variable, vecToScalar, wirePositioner
Arithmetic	1/X, -X, *, /, abs, complexToReIm, convert, gain, phase, power, sign, summingJunction, unitConversion
MatLab Interface	MatLab Expression, MatLab Read Variable, MatLab Write Variable
Matrix Operations	buffer, dotProduct, fft, ifft, inverse, multiply, transpose, vsum
Nonlinear	case, merge
Signal Consumer	display
Signal Producer	const
Time Delay	unitDelay
Other	expression

You can manually bundle and unbundle flexWires using the scalarToVec and vecToScalar blocks.

3.4.2 Wiring rules

You attach flexWires and vector wires to blocks through their connector tabs. Once you have attached a wire to a block, Commsim maintains the connection even as you move the block around the screen.

When you wire blocks, the following rules are in effect:

- Wires can only be drawn between an input and output connector tab pair. The triangular shape of the connector tab lets you easily distinguish inputs from outputs.
- Input connector tabs can only have one wire attached to them; output connector tabs can have any number of wires attached to them. To change the number of connector tabs on a block, follow the procedures in “3.5.1 Adding and removing connector tabs” on page 3-14.
- If you draw multiple wires between two blocks, Commsim automatically skews them.

3.4.3 Wiring blocks together

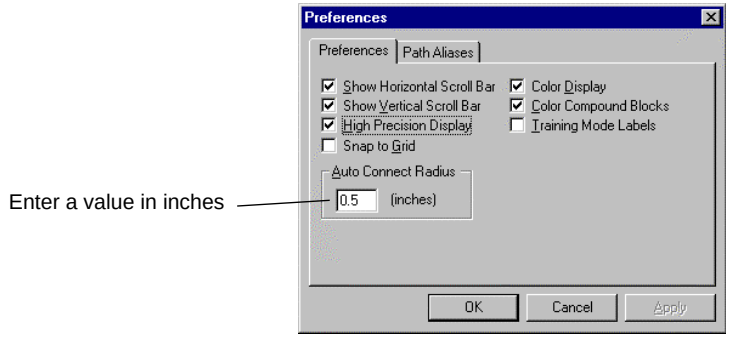
➢ To wire together blocks:

1. Point to a connector tab on one of the blocks to be wired. The pointer becomes an “up-arrow”.
2. Hold down the mouse button and drag the pointer over the connector tab on the destination block.

   As you drag the pointer, Commsim generates a colored line, which represents the wire. Because Commsim draws lines vertically and horizontally, the path of the line may not mimic the path of the cursor.
3. Release the mouse button.

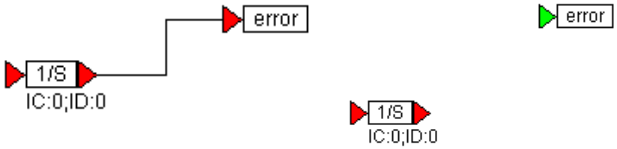
3.4.4 Automatically completing connections

You can control how close the pointer must be to a connector tab to automatically complete a connection with the Auto Connect Radius option under Preferences in the dialog box for the Edit > Preferences command.



3.4.5 Automatically wiring blocks

You can use the Edit menu's Auto Connect and Auto Disconnect commands to automatically wire blocks together. When wiring blocks, they do not have to be vertically parallel; however, if they are too far apart, the command will fail.



The blocks on the left were close enough to be wired, but the blocks on the right were too far apart.

➢ To automatically wire blocks:

1. Select the blocks to be wired.
2. Do one of the following:
   - Choose Edit > Auto Connect.
   - Click the right mouse button over one of the selected blocks to access the pop-up Edit menu and choose Auto Connect.

➢ To automatically disconnect the wires:

1. Select the blocks to be disconnected.
2. Do one of the following:
   - Choose Edit > Auto Disconnect.
   - Click the right mouse button over one of the selected blocks to access the pop-up Edit menu and choose Auto Disconnect.

3.4.6 Positioning wires

Using `wirePositioner` blocks, you can perform a connect-the-dot method of wiring. That is, you insert `wirePositioner` blocks and then manually route the wire through them. Since you control the placement of the `wirePositioner` blocks, it's easy to draw a precise wiring path.

Additionally, because `wirePositioner` blocks do not take any additional computation time, you won't see a decrease in performance during a simulation.

Both flexWires and vector wires can by routed through `wirePositioner` blocks.

3.4.7 Coloring wires

By default, wires are drawn in black. You can, however, change the default color using the View > Colors command, as described in the *Commsim 7 User Guide and Block Reference*.

You can also apply color to specific wires by assigning a connection class to the corresponding connector tabs. This wire coloring method is described in “3.5.3 Setting connection classes” on page 3-15.

3.4.8 Hiding wires

When you activate display mode with the View > Display Mode command, Commsim hides all wiring. Typically, you activate display mode when you want to display a control or instrumentation panel without the underlying connections, or when you want to view an animation.

3.4.9 Deleting wires

You delete a wire by detaching it from an input connector tab. Just point to the tab and hold down the mouse button as you drag the pointer away from the tab. When you release the mouse button, Commsim erases the wire.

3.5 Connector tab basics

All blocks that operate on signals have connector tabs. Commsim distinguishes between input and output connector tabs. Input connector tabs enable signals to enter a block; output connector tabs enable signals to exit a block. The triangular shape of the connector tab lets you easily see the direction in which the signals travel.

Some blocks have symbols on their connector tabs that indicate how the block acts on the data or the type of data the block is expecting. For example, the “-” on the `summingJunction` block means that the input is negated. See the descriptions of the individual blocks in the *Commsim 7 User Guide and Block Reference* for connector tab symbol definitions.

3.5.1 Adding and removing connector tabs

You can add or delete connector tabs on most Commsim blocks. If you delete a connector tab with an attached wire, the wire is also deleted.

Connector tabs on compound blocks

Because additional connector tabs are unconnected in compound blocks, make sure you verify the input and output connections after you complete this procedure.

You can label connector tabs on compound blocks by using the View > Connector Labels command, as described in the *Commsim 7 User Guide and Block Reference*.

➢ To change the number of connector tabs on a block:

1. Do one of the following:
   - From the toolbar, choose  or  .
   - Choose Edit > Add Connector or Edit > Remove Connector.
2. Do one of the following:

To	Do this
Add a connector tab	Point to where you want the tab. The short black line indicates tab placement. Then click the mouse.
Delete a connector tab	Point to the tab to be deleted. The selected tab has a short black line over it. Then click the mouse.

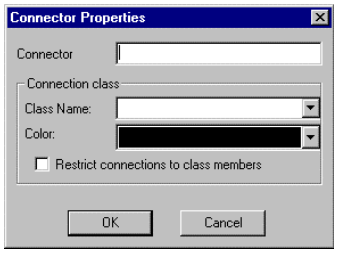
3. Repeat step 2 for as many tabs that you want to add or delete.
4. Click the mouse on empty screen space to exit this command.

3.5.2 Unconnected input connector tabs

Except on * blocks, all unconnected inputs are fed zeros, by default. Unconnected inputs on * blocks are fed ones.

3.5.3 Setting connection classes

Connection classes provide an easy method of organizing your calculations by name and color. You assign connection classes through the Connector Properties dialog box. To access this dialog box, double-click the mouse over a connector tab.



A class connection consists of a class name and corresponding color. The color is applied to the wire attached to the connector. For example, you can assign the class name PRESSURE to all connectors whose input and output signals relate to pressure calculations. All wires entering or exiting those connectors would then be displayed in the same color.

➢ To assign a class:

1. Point to the connector tab to be classified. The pointer turns into an upward pointing arrow.
2. Double-click the mouse.
3. In the Class Name box, enter a name, or click on the DOWN ARROW to select an existing name.
4. In the Color box, click on the DOWN ARROW and select from the drop-down color list.
5. Click on the OK button, or press ENTER.

➢ To change a class color:

1. Point to a connector tab whose class color you want to change. The pointer turns into an upward pointing arrow.
2. Double-click the mouse.
3. In the Color box, click on the DOWN ARROW and select from the drop-down color list.
4. Click on the OK button, or press ENTER.

3.5.3.1 Restricting connections to class members

If you want to prevent wires from being drawn between connector tabs of different classes, activate the Restrict Connections to Class Members box in the Connector Properties dialog box.

3.5.4 Displaying connector tabs in a different view

The View menu's Presentation Mode, Display Mode, and Data Types commands have different affects on how connector tabs are presented. For information on these commands, refer to the *Commsim 7 User Guide and Block Reference*.

Chapter 4
Arranging Blocks

The following are described in this chapter.

4.1 Selecting blocks

Once you have inserted a block into a block diagram, you will probably have to select the block in order to manipulate it. When you select a block, Commsim highlights it in black and outlines it in white.

When you select a compound block, all encapsulated blocks are implicitly selected.

➢ To select a block:

1. Point to the block.
2. Hold down the SHIFT key and click the mouse.

4.1.1 Area Selecting

A quick way to select one or more blocks is to use area select, which lets you draw a bounding box around the blocks you want to select. If any part of a block is contained in the bounding box, it is automatically selected.

➢ To perform an area select:

1. Point to one corner of the area you want to select.
2. To anchor the corner, hold down the mouse button.
3. Drag the pointer until the box encloses all the blocks you want selected.
4. Release the mouse button.

4.1.2 Toggle selecting

This action automatically selects all unselected blocks at the current level, and unselects all selected blocks at the current level.

➢ To toggle select blocks:

1. Point to empty screen space.
2. Hold down the SHIFT key and click the mouse.

4.1.3 Unselecting blocks

You can easily cancel the selection of individual blocks.

➢ To unselect blocks:

1. Point to the selected block.
2. Hold down the SHIFT key and click the mouse.

When blocks are unselected, they are returned to a normal video display.

4.2 Moving and copying blocks

Moving and copying blocks are common operations you'll perform in Commsim. Like many operations, there are several ways to move and copy blocks. For instance, you can move blocks by dragging and dropping them into place or you can cut them to the Windows Clipboard. From there, you can paste them back into your diagram or into another Commsim diagram. You can also paste them into other Windows-based applications.

4.2.1 Rules for moving and copying blocks

The following rules are in effect when you're moving and copying blocks:

- Moved and copied blocks retain the parameter values of the original blocks.
- Moved and copied blocks retain their internal wiring. This means that wires connecting blocks within the group of copied or cut blocks are retained.
- Moved and copied blocks lose their peripheral wiring. This means that wires connecting blocks in the group of blocks being copied or cut to other blocks are not retained.
- When moving or copying a compound block containing a global `variable` block with input, Commsim appends a number to the `variable` block name to keep it unique.

4.2.2 Drag-and-drop editing

An easy way to move or copy blocks within the current level of the diagram is with drag-and-drop editing. If you're moving or copying blocks to another level in the diagram, or to a different block diagram, you have to use the Edit menu's Cut, Copy, and Paste commands.

- To move a single block using: drag-and-drop editing:
  1. Point to the block to be moved and hold down the mouse button.
  2. Drag the block to the new location in the diagram.
  3. Release the mouse button.
- To move a group of blocks using drag-and-drop editing:
  1. Select the blocks to be moved.
  2. Point to one of the selected blocks and hold down the mouse button.

     The selected blocks are replaced with an empty box.
  3. Drag the box to the desired location in the diagram.
  4. Release the mouse button.
- To copy a single block using drag-and-drop editing:
  1. Point to the block to be copied.
  2. Press CTRL+SHIFT while you simultaneously click the mouse.

     As you move the pointer away from the block, a box appears. The box shows where the copy will be placed.
  3. Point to the location where you want the copy inserted and click the mouse.

  If you want to copy a group of blocks, see the description below.

4.2.3 Copying, cutting, and pasting blocks

The Copy, Cut, and Paste commands use the Windows Clipboard to transfer blocks to another block diagram level or to a different block diagram. You can also use the Clipboard to paste blocks into other applications.

The Clipboard can only hold one selection of cut or copied blocks at a time. If you place a new selection in the Clipboard, it overwrites whatever was already there.

➢ To copy or move selected blocks within Commsim:

1. Select the blocks.
2. To copy the blocks, do one of the following:
   - From the toolbar, choose  .
   - Choose Edit > Copy.
   - Press CTRL+C.
   - Click the right mouse button over one of the selected blocks and choose Copy from the pop-up menu.
3. To move the blocks, do one of the following:
   - From the toolbar, choose  .
   - Choose Edit > Cut.
   - Press CTRL+X.
   - Click the right mouse button over one of the selected blocks and choose Cut from the pop-up menu.

   At this point, the blocks are in the Clipboard.
4. Move to where you want the Clipboard contents inserted. If the location is in a different block diagram, use the File > Open command to open the proper block diagram and do one of the following:
   - From the toolbar, choose  .
   - Choose Edit > Paste.
   - Press CTRL+V.

   A rectangular box appears.
5. Position the box where you want the Clipboard contents inserted.
6. Click the mouse.

If blocks and wires overlap as a result of this procedure, you can easily reposition them using drag-and-drop editing.

4.2.4 Copying blocks into other applications

You can use the Copy command to copy pictures of blocks into other Windows-based applications. Common Window elements (like title bars, control-menu boxes, scroll bars, and minimize and maximize boxes) are not copied when you use the Copy command.

Inserting copies of block diagrams in other application files

There are two other methods for inserting a copy of a block diagram in another application. You can use the File > Save As MetaFile command to save a block diagram file as a Windows metafile image. You can then insert the .WMF in an application file that recognizes this file format. For more information, refer to the *Commsim 7 User Guide and Block Reference*.

You can alternatively press PRINT SCRN to copy a picture of the entire Commsim window into the Clipboard. From there, you can paste it into another Windows-based application using the application's paste command.

4.3 Flipping blocks

By allowing you to flip blocks 180^{o}, Commsim can present a more logical representation of right-to-left signal flow. When you flip blocks, Commsim redraws all flexWires attached to the blocks.

➢ To flip a block:

1. Select the blocks to be flipped.
2. Do one of the following:

- Choose Edit > Flip Horizontal.
- Press CTRL+LEFT.
- Click the right mouse button over one of the selected blocks and choose Flip Horizontal from the pop-up menu.

3. Click the mouse on empty screen space to unselect the blocks.

4.4 Aligning blocks vertically and horizontally

There are two ways to align blocks:

- Using the Snap to Grid parameter under Preferences in the dialog box for the Edit > Preferences command.
- Using the Edit menu's Align commands.

4.4.1 Snap to grid

The Snap to Grid parameter under Preferences in the dialog box for the Edit > Preferences command forces blocks to stay on an invisible grid. When you create block diagrams where you want blocks to line up horizontally or vertically, or where you want them to be spaced equally, activate Snap to Grid. When you move a block with Snap to Grid active, the block is forced to the nearest grid point. Blocks that have been inserted into your block diagram before Snap to Grid is active are also affected by this parameter.

4.4.2 The Align commands

Commsim provides commands for the precise alignment of a group of blocks. These commands let you choose whether blocks are lined up by:

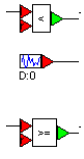
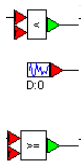
- Right, left, or top edges
- Input wires or output wires

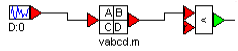
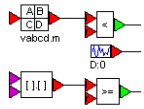
You can also control whether the blocks are stacked vertically.

➢ To align a group of blocks:

1. Select the blocks to be lined up.
2. Do one of the following:
   - Click on the Edit menu
   - Click the right mouse button over one of the selected blocks to access the pop-up Edit menu.

Choose one of the following commands:

Command	What it does	Example
Align Left	Aligns the blocks by their left edge	
Align Right	Aligns blocks by their right edge	

Command	What it does	Example
Align Stack	Aligns blocks vertically and removes empty space between blocks	
Align Top	Aligns blocks horizontally by their top edge	
Align Inputs	Lines up blocks horizontally so that their input wires are straight	
Align Outputs	Lines up blocks horizontally so that their output wires are straight	

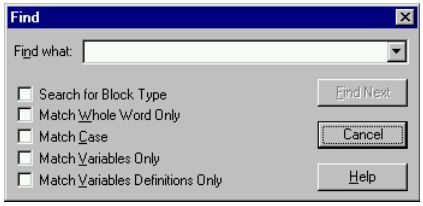
4.5 Finding and replacing blocks

Using the Edit menu's Find and Replace commands, you can search for certain types of blocks (for example, `ramp`, `stripChart`, and `const` blocks) or for certain occurrences of blocks with user-defined names, labels, and text strings.

Once you find the block you're looking for, you can make the appropriate changes to it. Commsim searches the entire block diagram for the search item, regardless of your current location in the diagram.

4.5.1 Finding blocks

When you choose the Find command, Commsim displays a dialog box you can use to specify the block, label, or text string you want to find. If you want to search for `variables`, you can also click on the DOWN ARROW next to the Find What box and select a variable name from the entries. All `variable` blocks in the diagram are listed in the drop-down list.



Once Commsim finds the search item, you can make a change in the diagram and then continue the search by choosing the Find Next button. The dialog box stays open so you can edit the diagram. To move the dialog box out of the way, drag on its title bar.

➢ To find a block:

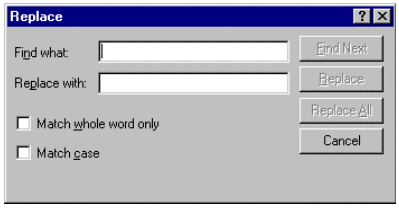
1. Choose Edit > Find.
2. In the Find What box, enter the search item. If you're searching for a `variable`, you can also click on the DOWN ARROW next to the Find What box and select from the variables list.
3. Select any option you want to control the search.

To	Select
Find blocks of a specific type; for example, `ramp` blocks, `animation` blocks, or `gain` blocks.	Search for Block Type box and click on the DOWN ARROW to choose from the list of searchable blocks.
Find whole words and not parts of words.	Match Whole Word Only box.
Find item with same capitalization as the word in the Find What box.	Match Case box.
Find only `variable` block names.	Match Variables Only box.
Find only the defining instance of a `variable`; that is, the `variable` block with an input connection.	Match Variable Definitions Only box.

4. Choose the Find Next button. When Commsim finds a match, it highlights the block in black.
5. To cancel a search or close the dialog box, choose the Cancel button.

4.5.2 Replacing blocks

You use the Replace command to replace the names of the blocks you find. You specify entries in the Replace dialog box in the same way that you do in the Find dialog box.



➢ To replace a block:

1. Choose Edit > Replace.
2. In the Find What box, enter the search item.
3. In the Replace With box, enter the item to replace the search item.
4. Select any option you want to control the search.

To	Select
Find whole words and not parts of words.	Find Whole Word Only box.
Find item with same capitalization as the word in the Find What box.	Match Case box.

5. Choose the Find Next button. When Commsim finds a match, it highlights the block in black.
6. Do one of the following:

To	Select
Replace the block name and find the next occurrence.	The Replace button.
Change all occurrences without confirmation.	The Replace All button.
Leave the block name unchanged and find the next occurrence.	The Find Next button.

7. To cancel a search or close the dialog box, choose the Cancel button.

4.6 Deleting blocks

When your block diagram contains blocks you no longer need, you can delete them using the Edit > Clear command or the DEL key. When you delete blocks, all wires attached to the deleted blocks are also deleted.

➢ To clear selected blocks:

1. Select the blocks to be cleared.
2. Do one of the following:
   - Choose Edit > Clear
   - Press DEL.
3. Click the right mouse button over a selected block and choose Clear from the pop-up menu.

Chapter 5
Setting Simulation Properties

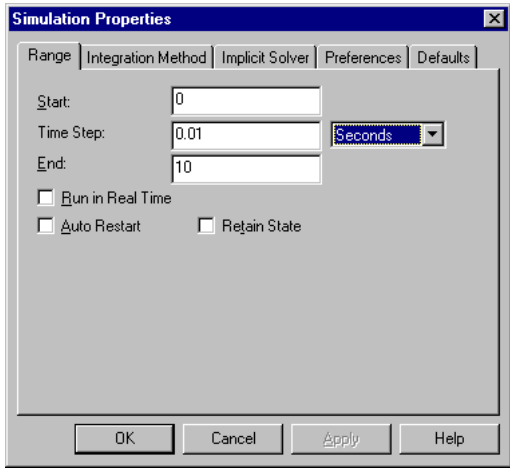
The following are described in this chapter.

5.1 Setting up the simulation range

Setting up the simulation range involves choosing the start and end of the simulation, specifying the step size of the integration algorithm, indicating whether Commsim runs in real-time mode, and indicating whether Commsim automatically restarts the simulation either with or without the last known system states.

- To access the Simulation Range options:
  1. Choose Simulate > Simulation Properties.
  2. Click on the Range tab.

The Range sheet in the Simulation Properties dialog box appears.



3. Choose the options you want, then click on the OK button, or press ENTER. (For more information on the options, see the descriptions below.)

5.1.1 Using the Range property sheet

The Range property sheet options are:

Auto Restart: For real-time control or training neural networks, where multiple data sets must be fed into Commsim repeatedly, you can activate Auto Restart. This parameter restarts and runs the simulation until one of the following conditions is met:

- The signal in the `error` or `stop` block goes to 1.

You manually stop the simulation.

You can keep track of the number of the run by wiring a `$runCount variable` block into your diagram.

To retain the states of blocks each time Commsim automatically restarts a simulation, activate the Retain States parameter, as described below. Blocks that are time based (for example, Signal Producer blocks) are reset to their restart time. For a smooth transition between auto-restart simulation runs, you need to remove the Signal Producer blocks from your diagram. For instance, if your diagram contains a `sinusoid` block, replace it with an `integrator` block with its derivative set to the sinusoidal frequency and feed it to a `sin` block.

Retain States: For a smooth transition between simulation iterations, activate Retain State. When activated, Commsim retains the states of the `integrator`, `stateSpace`, `transferFunction`, and `unitDelay` blocks each time it restarts a simulation.

The Retain State parameter can only be activated when Auto Restart is already activated.

Run in Real Time: With Run in Real Time, Commsim simulates in real-time mode, which has the effect of retarding a simulation so that one simulation second equals one clocked second. This mode comes in handy when a system is exhibiting rapidly varying behavior. In real-time mode, the behavior can be slowed down and more easily analyzed.

Typically, you use real-time mode for hardware-in-the-loop control situations. For this, however, you also need the Commsim/Real-Time software and a PC D/A-A/D card. The Commsim/Real-Time driver lets you configure different analog and digital channels and insert them into a block diagram for reading and writing.

Start/End: Using Start and End, you can set independent variables that indicate when Commsim starts and stops a simulation, as well as when Commsim starts and stops logging data points in the Signal Consumer blocks wired into the block diagram.

You can also set defaults for the start and end, as described in "5.4 Setting simulation defaults" on page 5-9.

Step Size: The step size is the fundamental unit of integration. It indicates the interval at which the integration algorithm computes the integral of the input function and generates a data point in the Signal Consumer blocks wired into the block diagram. You specify the step size in the Step Size box. The default is 0.05.

For adaptive integration methods (adaptive Runge Kutta 5^{th} order and adaptive Bulirsh-Stoer), you can also specify a minimum step size.

You can also set:

- A default step size for the non-adaptive integration methods, as described in "5.4 Setting simulation defaults" on page 5-9
- A local step size for specific subsystems, as described in the *Commsim 7 User Guide and Block Reference*.

You can switch to frequency as the fundamental unit of integration by selecting either Hertz, Kilohertz, or Megahertz from the drop down list box to the right of the Step Size box. When you do so, the Step Size box becomes a Frequency box. Enter the frequency in the box.

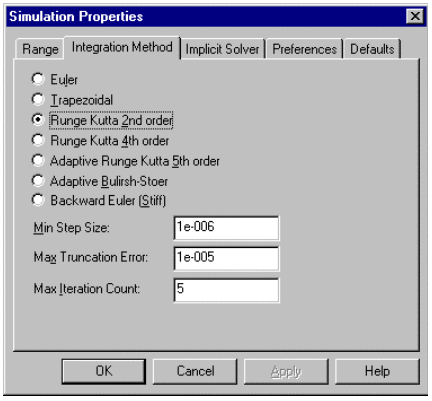
5.2 Setting up an integration method

Setting up the integration algorithm involves choosing the algorithm, and, if you choose an adaptive algorithm, specifying the minimum step size, error tolerance, and iteration count.

➢ To access the integration method options:

1. Choose Simulate > Simulation Properties.
2. Click on the Integration Method tab.

   The Integration Method sheet in the Simulation Properties dialog box appears.



3. Choose the options you want, then click on the OK button, or press ENTER. (For more information on the options, see the descriptions below.)

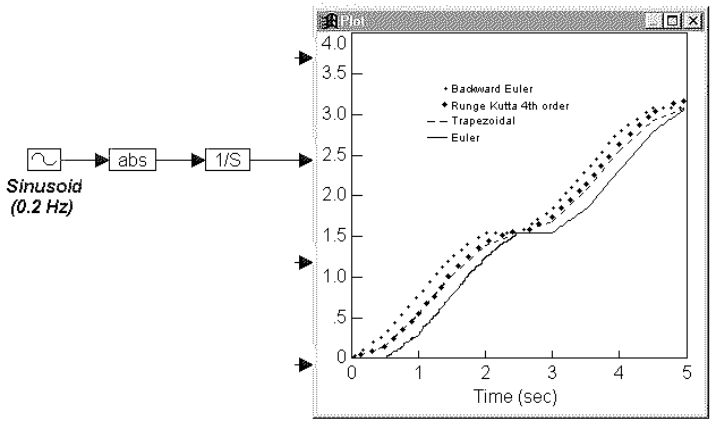
5.2.1 Using the Integration Method property sheet

Commsim provides seven integration algorithms — Euler, trapezoidal, Runge Kutta 2nd order, Runge Kutta 4th order, adaptive Runge Kutta 5th order, adaptive Bulirsh-Stoer, and backward Euler (Stiff) — of varying numerical accuracy for the numerical integration of differential and difference equations.

Each algorithm provides a numerical approximation to continuous integration. The approximation is based on a trade-off between speed of execution and accuracy. Generally speaking, the more complex algorithms yield more stable and numerically correct results; however, they also take longer to run.

For example, the integration of the absolute value of a sinusoid signal with a frequency of 0.2 Hz is plotted below. The output of the `abs` block is a sequence of sinusoid positive half-cycles

with a frequency of 0.4 Hz. Since the simulation range is from 0 to 5 seconds, the output of the `integrator` block is the estimate area under the curve of two positive half-cycles.



While the differences due to the integration algorithms are negligible for this example, more dramatic differences can be observed when comparing simulation methods in diagrams containing differential equations.

A good rule of thumb, then, is to use the least complicated algorithm that provides stable and correct results. To achieve this, start with the most complex integration algorithm and work backwards to simpler algorithms until you see a noticeable change in your results.

Setting a default integration algorithm

If you plan to use a particular integration algorithm a lot, you can set it as the default, as described in “5.4 Setting simulation defaults” on page 5-9.

The Integration Method property sheet options are:

Euler: Evaluates once per simulation time step. This method is least affected by singularities, and is fastest for moderate step sizes.

Trapezoidal: Evaluates twice per simulation time step.

Runge Kutta 2d order: Obtains second order accuracy. This method uses a midpoint step derivative to calculate the final integration value. Specify the length of the step in the Step Size box.

Runge Kutta 4th order: Obtains fourth order accuracy. This method evaluates the derivative four times at each time step: once at the initial point, twice at sample midpoints, and once at a sample endpoint. The final integration value is then derived based on these derivatives.

Adaptive Runge Kutta 5th order: Obtains fifth order accuracy. This algorithm automatically takes small step sizes through discontinuities in the input function and large strides through smooth functions.

Adaptive Bulirsh-Stoer: Uses rational polynomials to extrapolate a series of substeps to a final estimate. This algorithm is highly accurate for smooth functions.

Backward Euler (Stiff): Obtains efficiency for systems with high and low frequencies. The other algorithms would require small step sizes to maintain stability.

Min Step Size: The adaptive Runge Kutta 5^{th} order and adaptive Bulirsh-Stoer integration algorithms exert more control over the accuracy of the solution by letting you specify a minimum step size. The step size is continually adjusted in order to meet the error tolerance and iteration count criteria; however, it is never reduced below the minimum step size, and it will never be greater than the main time step. Thus, inaccurate results may be produced if the minimum step size is too large, the error tolerance is too large, or the iteration count is too small.

The default value for the minimum step size is 1e-006.

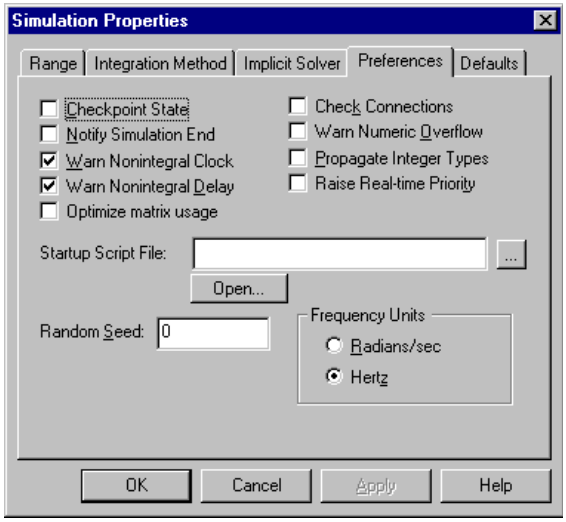
Max Truncation Error: When you choose an adaptive integration algorithm, you can specify the maximum error between the results of two successive adaptive iterations. Commsim uses the truncation error to determine the adaptive step size. The larger the error you're willing to tolerate, the larger the step size. The default value for the maximum truncation error is 1e-005.

Max Iteration Count: When you choose an adaptive integration algorithm, you can also specify the maximum number of times the integration algorithm will vary its time step attempting to meet the maximum truncation error criterion. The default value for the maximum iteration count is 5.

5.3 Setting up simulation preferences

- To access the preferences options:
  1. Choose Simulate > Simulation Properties.
  2. Click on the Preferences tab.

The Preferences property sheet in the Simulation Properties dialog box appears.



3. Choose the options you want, then click on the OK button, or press ENTER. (For more information on the options, see the descriptions below.)

5.3.1 Using the Preferences property sheet

The Preferences property sheet provides options for checking for unconnected blocks, checkpointing a simulation, selecting the frequency units, and more. The property sheet options are:

Check Connections: Warns you at the start of a simulation if the diagram contains unconnected blocks. Unconnected blocks are highlighted in red. A Warning dialog box identifies the unconnected input tab and provides two choices:

- **Abort or Retry** Finishes checking the diagram and then stops the simulation.
- **Ignore** Finishes checking the diagram and then completes the simulation.

Checkpoint State: Saves a temporary copy of your block diagram “as is” at the time you stopped the simulation. To do so, Checkpoint States must be activated before you begin the simulation.

If you activated Checkpoint States, Commsim saves the current values of all system parameters and block outputs, and elapsed simulation time when you stop the simulation. If you close the block diagram and then re-open it, you can continue the simulation from where you left off.

Checkpointing is useful for long simulations because it allows you to stop and save a simulation in the event that you must shut down your computer for a lengthy period of time.

Frequency Units: Sets the frequency units to either radians per second or hertz.

Notify Simulation End: Broadcasts an "End of Simulation" message to your computer when the simulation completes.

Optimize matrix usage: Avoids creating matrices for `scalarToVec` and `vecToScalar` blocks during simulation. Thus when vectors are used for pure connectivity, Commsim will not create matrices internally, which can speed up a simulation.

Propagate Integer Types: Uses C semantics to propagate integer data types. For example, if you add two integers, the result is an integer value.

Raise Real-Time Priority: Gives your process the higher priority to let you achieve reliable real-time sampling without interruptions from other processes running simultaneously. This option is for real-time control applications.

Random Seed: Generates numbers by a random process. The `gaussian` and `uniform` blocks are affected by this option. That is, the numbers exiting these blocks are derived from the value of Random Seed. You can use the $randomSeed `variable` to access the random seed.

Typically, you use Random Seed when input is required to be unpredictable. For example, when modeling the descent path of an airplane, it is impossible to predict the force or direction of the wind. Consequently, you represent it as a function of a random number.

The value of Random Seed ranges from 0 to 65,536. The default is 0.

Altering the sequence of generated random numbers

Commsim generates the sequence of random numbers for each simulation differently depending upon whether the Auto Restart parameter in the dialog box for the Simulate > Simulation Properties command is activated. When Auto Restart is on, Commsim generates a new sequence of random numbers for each simulation. Conversely, when it's turned off, Commsim generates the same sequence of random numbers for each simulation. To change the sequence, you must explicitly enter a new Random Seed value at the start of the simulation.

Warn Nonintegral Clock: Warns you if a pulse is chosen that is not an integral multiple of the simulation step size. This option should always be activated; otherwise, a simulation that inaccurately represents the system you're modeling may go undetected.

Warn Nonintegral Delay: Warns you if a delay is chosen that is not an integral multiple of the simulation step size. This option should always be activated; otherwise, a simulation that inaccurately represents the system you're modeling may go undetected.

Warn Numeric Overflow: Warns you if a `convert` block causes data truncation resulting in the value loosing precision.

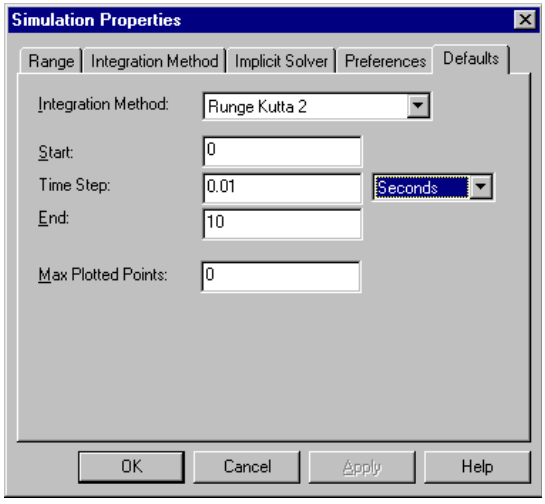
5.4 Setting simulation defaults

For non-adaptive integration methods, you can specify the following default settings that are in effect whenever you create a new block diagram or start a new Commsim session: range, integration algorithm, fixed step size, frequency, and maximum number of plotted points.

➢ To set simulation defaults:

1. Choose Simulate > Simulation Properties.
2. Click on the Defaults tab.

   The Defaults property sheet in the Simulation Properties dialog box appears.



3. Make the selections you want to put into effect.
4. Click on the OK button, or press ENTER.

Chapter 6
Simulating Block Diagrams

The following are described in this chapter.

6.1 Simulation basics

Commsim can simulate linear, nonlinear, continuous, and discrete systems. Commsim can also simulate systems containing both continuous and discrete transfer functions, as well as systems containing multi-rate sampling for discrete transfer functions.

When you initiate a simulation, Commsim first evaluates Signal Producer blocks, like `consts` and `ramps`, then sends the data to intermediate blocks that have both inputs and outputs, like `gains` and `summingJunctions`. Lastly, it sends data to Signal Consumer blocks that have only inputs, such as `plots` and `meters`.

Commsim simulates a system according to:

- Simulation parameters set in the dialog box for the Simulate > Simulation Properties command
- Initial conditions for the system set in the applicable blocks

If the status bar is turned on, Commsim displays current settings for the simulation range, step size, elapsed simulation time, integration algorithm, and implicit solver.

6.1.1 Continuous system simulation

Because integration is a more numerically stable operation than differentiation, you need to transform your ordinary differential equations into ones that use integration operators.

To enter an ordinary differential equation in Commsim, first algebraically solve the equation for the highest derivative. Then, in Commsim, insert the number of `integrator` blocks that equals the order of the highest derivative. Most continuous systems contain one or more differential equations. For example, if you're solving a third order differential equation, insert three `integrator` blocks and supply the equation for the highest order derivative as input to the first integrator block. The output of the first integrator is:

$$\frac{d^{n-1}x}{dt^{n-1}}$$

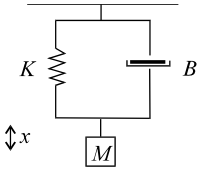
which is the next lower order derivative. The output of the second integrator block is:

$$\frac{d^{n-2}x}{dt^{n-2}}$$

and so on. The outputs of the lower order derivatives can be fed back into the calculation of the highest derivative.

6.1.1.1 Simulating a spring-damper arm

The following example steps you through the process of converting a second order differential equation into Commsim block diagram form. This example involves the ubiquitous damped harmonic oscillator, where a mass *M* is suspended from the ceiling by a spring-damper arm. The mass is attracted back toward the origin by an elastic restoring force proportional to its vertical displacement and is damped by an opposing force that acts in proportion to its velocity.



Where:
K= spring stiffness
B = spring damping
M = mass
x = vertical displacement

Based on Newton's Second Law, the definition of the equation of motion for the damped harmonic oscillator is:

$$M\frac{d^2x}{dt^2} = -Kx - B\frac{dx}{dt}$$

where:

$\frac{d^2x}{dt^2}$ = acceleration

$\frac{dx}{dt}$ = velocity

Because integration is inherently more numerically stable than differentiation, the equation must be expressed in terms of integrals. By definition of the derivative:

$$\frac{dx}{dt} = \int_0^t \frac{d^2x}{dt^2} + v(0)$$

and

$$x = \int_0^t \frac{dx}{dt} + x(0)$$

where:

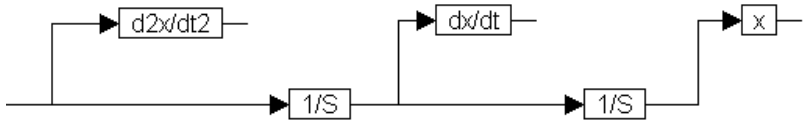
$v(0)$ = initial velocity of the mass

$x(0)$ = initial starting position of the mass

Employing 1/*s* as the operator for integration, and making the initial conditions implicit in 1/*s*, yields the following relationship:

$$x = \frac{1}{s}\left(\frac{dx}{dt}\right) = \frac{1}{s^2}\left(\frac{d^2x}{dt^2}\right)$$

The relationship can be expressed in Commsim block diagram form as:



A more concise representation of this relationship is:

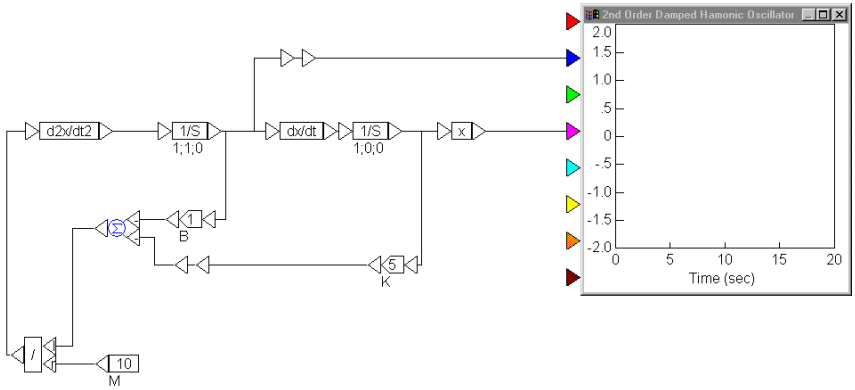
The three `variable` blocks hold the quantities d^2x/dt^2, dx/dt, and x at each instant of time. The `variable` blocks are actually extraneous, because the wires alone can carry the data forward to the next block.

Returning to the original equation of motion and solving for the acceleration yields:

$$\frac{d^2x}{dt^2} = \frac{1}{M}\left(-Kx - B\frac{dx}{dt}\right)$$

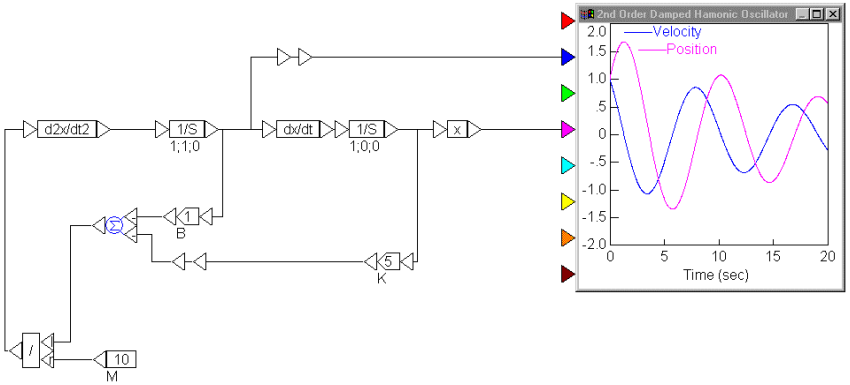
To model this system in Commsim, wire the outputs of the x and dx/dt `variable` blocks through two `gain` blocks (which represent K and B) and into a `summingJunction` block with inputs negated. By dividing the output of the `summingJunction` block by M (which is

represented by a constant block), you produce d^2x/dt^2. Letting $K = 5$, $B = 1$, and $M = 10$, for example, results in the following diagram.



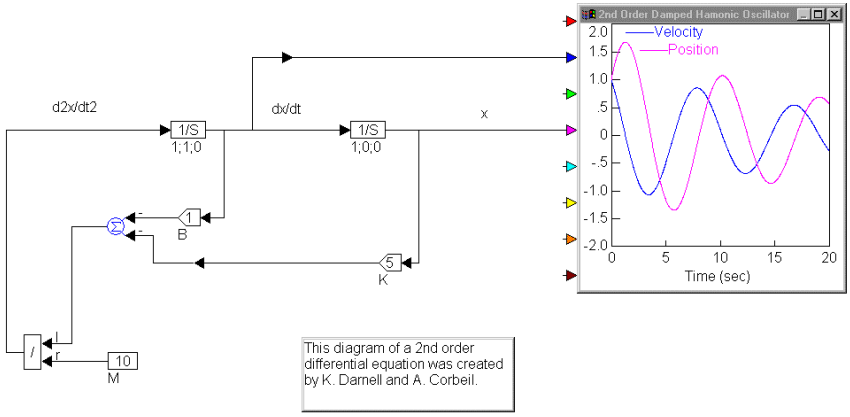
This diagram represents a closed-loop system from which the values for position, velocity, and/or acceleration can be displayed in a `plot` block, as was done here. The initial conditions of starting position $x(0)$ and velocity $v(0)$ of M are specified within the `integrator` blocks preceding the respective `variable` blocks.

Letting $x(0) = 0$ and $v(0) = 1$, and setting the simulation range from 0 to 20 and the step size to 0.05 yields the following results:



Note that the characteristic decay that is observed depends on the parameters M, K, and B. Different values for these quantities and initial conditions can be entered into the appropriate blocks to simulate any system.

You can simplify the diagram by replacing the `variable` blocks that denote *x*, *dx*/*dt*, and d^2x/dt^2 with optional `label` blocks.



Other physical effects can now be added, such as static and sliding friction, or external driving forces.

A coupled system can also be modeled by interconnecting two separate block diagrams. This permits extremely complex systems to be modeled without the need for a closed mathematical solution.

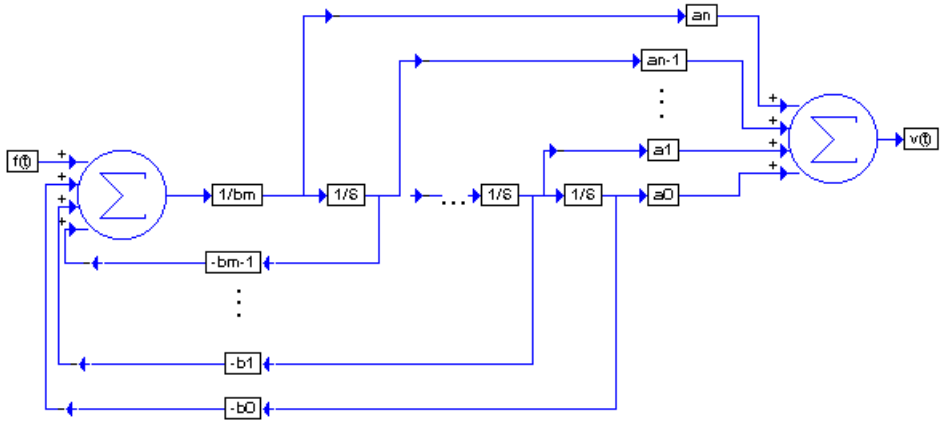
6.1.1.2 Entering continuous time transfer functions

A transfer function is a ratio of polynomials in the Laplacian *s* operator that models the ratio of the output signals divided by the input signals. There are two ways to enter a transfer function in Commsim. The more common method is via the `transferFunction` block, which you use when entering coefficients as numeric constants.

When the coefficients are polynomial constants, begin by defining the transfer function in operator notation. The transfer function should be proper; that is, the highest degree of the denominator polynomial *m* must be greater or equal to that of the numerator *n*. The general transfer function representation is:

$$\frac{N(s)}{D(s)} = \frac{a_n s^n + a_{n-1} s^{n-1} \ldots + a_1 s + a_0}{b_m s^m + b_{m-1} s^{m-1} \ldots + b_1 s + b_0}$$

You represent this in Commsim as follows:



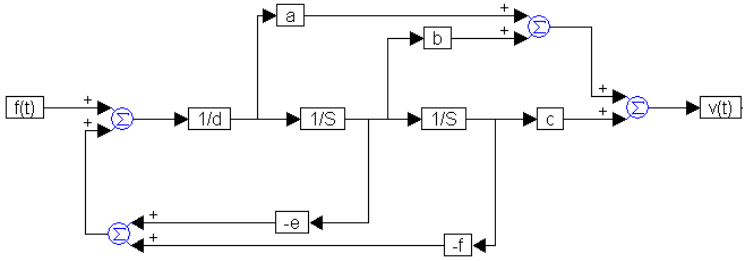
This diagram represents the condition in which the numerator and denominator degrees are equal ($m = n$). When the numerator is of a degree less than the denominator, the output paths are removed from the left. For example, if n is two less than m, the a_n and a_{n-1} output paths would be removed.

Note also that for each k^{th} polynomial term, you add an `integrator` and a corresponding a_k, b_k set of gains flow to the upper and lower `summingJunction` blocks in the diagram.

The second degree transfer function is:

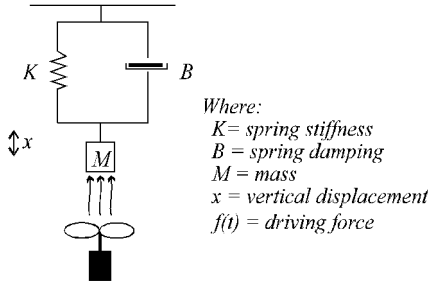
$$\frac{N(s)}{D(s)} = \frac{as^2 + bs + c}{ds^2 + es + f}$$

The diagram for this transfer function is:



6.1.1.3 Applying an external driving force to a spring-damper arm

By modifying the damped harmonic oscillator, created earlier, to include an external driving force *f*(*t*), you can create a system that contains a transfer function. An illustration of the modified system is shown below:



In this system, a vertical draft is produced by a strong fan positioned below the mass. This driving force sustains the motion of the damped oscillator and represents an input to the system. The output is the instantaneous velocity *v* of the mass. To derive the transfer function for this simple system, you will use the Laplace transform. The modified equation of motion for this system is:

$$M\frac{d^2x}{dt^2} = -Kx - B\frac{dx}{dt} + f(t)$$

Accounting for non-zero initial conditions, the Laplace transform becomes:

$$Ms^2x(s) - Msx(0) - M\frac{dx}{dt}(0) = -Kx(s) - Bsx(s) + Bx(0) + F(s)$$

Regrouping the equation yields:

$$x(s)\left(Ms^2 + Bs + K\right) - x(0)(Ms - B) - \frac{dx}{dt}(0)M = F(s)$$

Transfer function representation requires all initial conditions be equal to zero, specifically:

$$x(0) = \frac{dx}{dt}(0) = 0$$

The equation reduces to:

$$x(s)\left(Ms^2 + Bs + K\right) = F(s)$$

whose transfer function is:

$$\frac{x(s)}{F(s)} = \frac{1}{Ms^2 + Bs + K}$$

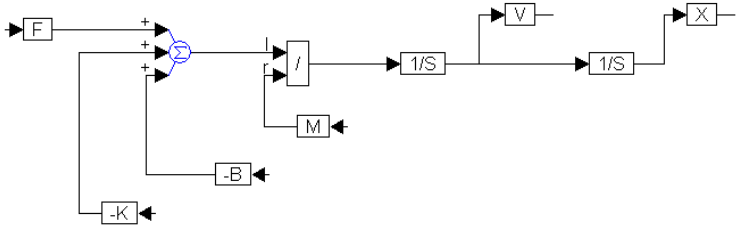
Since velocity rather than displacement is the desired output, the substitution:

$$V(s) = sx(s)$$

is made to produce the transfer function:

$$\frac{V(s)}{F(s)} = \frac{s}{Ms^2 + Bs + K}$$

The denominator remains unchanged; however, the numerator coefficients are different. The block diagram becomes:



The non-zero initial conditions can be easily included by specifying their values on each of the two `integrator` blocks. For example, suppose that the spring was initially stretched one inch. An initial condition of one would be placed on the rightmost `integrator`. Assuming an initial velocity of zero, the initial condition on the leftmost `integrator` would still be zero.

6.1.2 Discrete time system simulation

You can simulate models of discrete time systems using `unitDelay`, `transferFunction`, and `stateSpace` blocks. These discrete blocks have built-in samplers on their inputs and zero-order holds on their outputs.

You set the sample time of `transferFunction` and `stateSpace` blocks in the dT parameters of their Properties dialog boxes. The dT parameter sets the sample time at which

the blocks' states are updated. The `unitDelay` block has a Boolean clock at its input to set the sample time.

Simulating multi-rate systems: Discrete time systems in Commsim can be formulated as multi-rate systems. This means that a single model can contain blocks with different sampling rates. This capability is particularly useful in the simulation of discrete Multiple-Input-Multiple-Output (MIMO) systems. For a system with significant differences in its time constants in some natural modes or control loops, you can achieve improved performance by sampling different subsystems at different rates.

To specify multi-rate subsystems, use different sample times in the corresponding discrete `transferFunction` or discrete `stateSpace` blocks. The simulation time step must be set to a value equal to or less than the smallest value of all the sample times used in the discrete blocks.

6.1.2.1 Entering difference equations

A difference equation (DE) is similar to an ordinary differential equation, but instead of continuous functions, functions in a DE take on values only at discrete instances of time. Just as the operator in an ordinary differential equation is the integrator, the operator in the difference equation is the unit delay.

To understand how to represent a DE in block diagram form, consider the following example of the trapezoidal integration algorithm in difference equation form:

$$Y_k = Y_{k-1} + \frac{dt\left(R_k + R_{k-1}\right)}{2}$$

where:

R = input

Y = output

Here, *dt* is the fixed discrete update time and the subscript k and k-1 denote time in integer multiples of *dt*. Thus:

$$R_k = R\left(kdt\right)$$

and

$$R_{k-1} = R\left(\left(k-1\right)dt\right)$$

A DE is converted to a transfer function in terms of the *Z* operator by replacing occurrences of F_{k-n} with $F(z^{-n})$. Thus:

$$Y_k \rightarrow Y\left(z^0\right) \rightarrow Y$$
$$Y_{k-1} \rightarrow Y\left(z^{-1}\right)$$
$$R_k \rightarrow R\left(z^0\right) \rightarrow R$$
$$R_{k-1} \rightarrow R\left(z^{-1}\right)$$

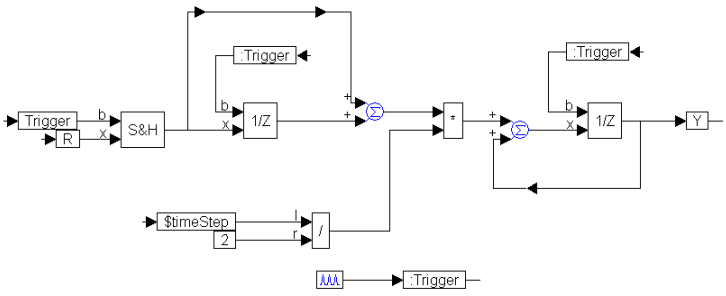
Performing the replacement and solving for $\frac{Y}{R}$ yields:

$$\frac{Y}{R} = \frac{dt}{2}\frac{1+z^{-1}}{1-z^{-1}}$$

Since transfer functions are conventionally expressed in positive powers of *z*, you must multiply the right-hand side of the equation by *z*/*z* to produce:

$$\frac{Y}{R} = \frac{dt}{2}\frac{z+1}{z-1}$$

To create a Commsim block diagram, the procedure is similar to that used for continuous time transfer functions. However, the `unitDelay` block replaces the `integrator` block. The resulting block diagram becomes:



The continuous input signal, R, is made a discrete function by passing it through a `sampleHold` block to effectively sample and hold its value every time the trigger is activated. The trigger is activated every *dt* seconds using the `pulseTrain` block, and must be fed into every `unitDelay` block to synchronize the Commsim data flow.

6.1.3 Hybrid system simulation

In Commsim, discrete and continuous time blocks can be used together in a model. Such systems are called hybrid systems. In hybrid systems, the outputs of the discrete blocks are held constant between successive sample times, and updated at times that correspond to the specified discrete sample time. The outputs of continuous blocks are updated at every time step. Similarly, the inputs to the discrete blocks are updated at times that correspond to the discrete time interval while the inputs to continuous blocks are updated at every time step.

Hybrid systems can also be multi-rate. To specify multi-rate subsystems, use different sample times in the corresponding discrete `transferFunction` or discrete `stateSpace` blocks. For hybrid system simulation, the simulation time-step must be set to a value equal to or less than the smallest value of all the sample times used in the discrete blocks.

6.2 Controlling a simulation

There are two ways to control a simulation:

- Using the Simulate menu Go, Stop, Continue, and Reset commands or corresponding toolbar buttons
- Using the simulation Control Panel

Both ways provide the same level of interactive control over the simulation.

6.2.1 The Control Panel

The Control Panel provides fast and easy interactive control over a simulation.

Go, Stop, and Cont pushbuttons: These buttons allow you to start, stop, and continue a simulation. They are equivalent to the Go, Stop, and Continue commands in the Simulate menu, and the  ,  , and  buttons in the toolbar.

Step pushbutton: This button allows you to single-step through a simulation. Each time you press the Step pushbutton, the simulation advances one time step. The Step pushbutton is equivalent to  button in the toolbar.

Reset pushbutton: When you're single-stepping or proceeding normally through a simulation, the Go pushbutton is replaced with the Reset pushbutton. If you click on Reset, Commsim resets the system to its initial conditions.

➢ To activate the Control Panel:

1. Choose View > Control Panel.

6.2.2 Starting a simulation

➢ To start a simulation:

1. Do one of the following:
   - From the toolbar, choose  .
   - From the Control Panel, press the Go pushbutton.
   - Choose Simulate > Go command.

6.2.3 Stopping a simulation

➢ To stop a simulation:

1. Do one of the following:
   - From the toolbar, choose  .
   - From the Control Panel, press the Stop pushbutton.
   - Choose Simulate > Stop .

6.2.4 Continuing a simulation

➢ To continue a simulation:

1. Do one of the following:
   - From the toolbar, choose  .
   - From the Control Panel, press the Cont pushbutton.
   - Choose Simulate > Continue .

6.2.5 Single-stepping a simulation

➢ To single step

1. Do one of the following:
   - From the toolbar, choose  .
   - From the Control Panel, press the Step pushbutton.

6.2.6 Resetting a simulation

The Reset Sim command resets states of `unitDelay` blocks, `integrator` blocks, `resetIntegrator` blocks, `limitedIntegrator` blocks, `stateSpace` blocks, and `transferFunction` blocks to their initial conditions. In addition, simulation time is turned back to 0.

➢ To reset, do one of the following:

- From the Control Panel, press the Reset pushbutton.
- Choose Simulate > Reset Sim.

6.3 More on controlling a simulation

6.3.1 Dynamically modifying signal values

You can dynamically modify a signal value during a simulation using the slider block. This block lets you set upper and lower bounds in one and 10 percent increments.

➢ To modify a signal value:

1. Insert and wire a `slider` block into your diagram.
2. Using the scroll bar, adjust the value to be applied to the signal.
3. As the simulation proceeds, re-adjust the value of the `slider` block as necessary.

6.3.2 Probing signal values

There are two ways to probe signal values at each time step of a simulation:

To	Do this
Monitor signals entering or exiting a specific block	Hold down the right mouse button over a connector tab on the block.
Monitor signal values emitted from multiple blocks simultaneously	Wire `display` blocks to the output connector tabs of the blocks.

6.3.3 Trimming a system

Commsim's `unknown` and `constraint` blocks can be used to trim a simulation to begin at a desired non-zero point. This technique is especially useful for slow-running simulations in which the interesting region lies later in the trajectory. By trimming the conditions at the interesting region, you save time.

The initial condition of the integrator can be set externally using a `summingJunction` block. (The actual initial condition on the integrator is set to 0.) The goal is to drive the derivative signal to zero on the first pass of the simulation by adjusting the value of the unknown blocks, which is the integrator initial condition.

6.3.4 Resetting error conditions

If a simulation fails as a result of a math fault — for example, a negative argument to a log function — Commsim displays a dialog box stating the nature of the error and highlights the offending block in red. To reset the error condition, point to the offending block and click the right mouse button. If the offending block is encapsulated within one or more compound blocks, each compound block is also highlighted in red. Note that you'll have to drill into highlighted compound blocks to find the offending block.

If multiple blocks contain errors, use the Edit > Clear Errors command to clear all the errors.

6.3.5 Snapping and resetting system states

You can snap and reset the states of `unitDelay` blocks, `integrator` blocks, `resetIntegrator`, `limitedIntegrator`, `stateSpace`, and `transferFunction` blocks.

Snapping system states: When you snap states, Commsim overwrites the initial conditions of the blocks with their current output states. Snapping states is useful when you want to run a simulation to a stable operating point and, from there, experiment with the system.

State values are saved in memory; to save them to disk, use the File > Save command.

Resetting system states: When you reset system states, Commsim sets the states of the blocks to zero.

- To snap state values to memory:
  1. Run the simulation to a specific point of interest.
  2. Choose Simulate > Snap States.
- To reset initial conditions:
  1. Choose Simulate > Reset States.

6.4 Troubleshooting

What should I do when the input function to an integrator block contains discontinuities?

If the input function to an `integrator` block contains discontinuities, use the adaptive Runge Kutta 5th order or adaptive Bulirsh-Stoer integration algorithm.

How do I stabilize rapidly oscillating behavior?

A simulation that exhibits an oscillating behavior that increases rapidly in amplitude points to unstable integration settings. When this occurs, decrease the integration step size or switch to an integration algorithm that yields more accurate results and produces less accumulated errors over the course of the simulation, such as the adaptive Runge Kutta 5th order or adaptive Bulirsh-Stoer integration algorithm.

For highly nonlinear systems or stiff systems, you should use backward Euler.

How can I speed up my simulations?

When speed is a factor, there are several things you can do:

1. Disconnect all Signal Consumer blocks at the currently displayed level.
2. Encapsulate the portions of the computation that are used infrequently (like simulation start-up and shutdown) in an enabled compound block that is executed only under specific conditions. For more information on triggering the execution of compound blocks, refer to the *Commsim 7 User Guide and Block Reference*.

Is there an easy way to check for complete wiring?

A faulty simulation can be the result of incomplete wiring. Commsim automatically assigns zeros to all unsatisfied connector tabs (except on `variable` and `*` blocks) before it begins a simulation. To ensure that all blocks are fully connected, activate Check Connections under the Preferences property sheet in the dialog box for the Simulate > Simulation Properties command. When this parameter is activated, Commsim warns you at the start of the simulation if the diagram contains unconnected blocks.

Why is my feedback loop causing errors?

If you create a feedback loop that does not contain integration or delay blocks, it is referred to as an *algebraic loop*. Commsim is not equipped to solve algebraic loops. Hence, during simulation, Commsim flags the loop-head block in red and issues a notification message. To fix the error, rework the loop to introduce a delay.

If you are trying to solve an implicit equation, refer to the *Commsim 7 User Guide and Block Reference* for information on using `unknown` and `constraint` blocks.

Chapter 7
Viewing Simulations

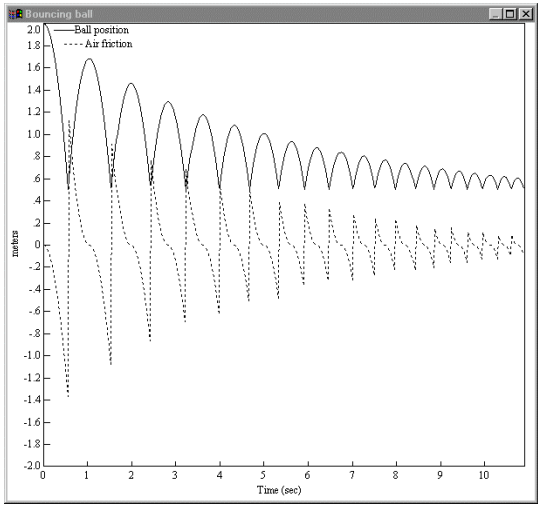
The following are described in this chapter.

7.1 Plots

The `plot` block displays data in a two-dimensional time domain plot. You can customize the plot and control how data is presented through the Plot Properties dialog box for the `plot` block.

7.1.1 Basic time domain plots

When you wire a `plot` block into your diagram and run a simulation, the simulation data is initially presented in time domain. All the signals are plotted on the y-axis; the x-axis represents time. As data points arrive to be plotted, Commsim dynamically rescales plot bounds and connects the data points with line segments.



In the above plot, ball position and air friction are displayed as functions of time. The peak ball position follows an exponential decay, governed by air viscosity. The signals are distinguished by line patterns, a feature the `plot` block automatically performs when displaying or printing on monochrome devices. To make the plot more meaningful, signal labels and a title were also added.

7.1.2 Sizing a plot block

You might want to change the size or shape of a `plot` block for better viewing. You can expand it to full screen size with the Maximize button in the upper right-hand corner of the block, or you can drag the plot's borders or corners to adjust its size.

7.1.3 Zooming

You can zoom in on data points to view them at a magnified size and zoom back out to display them at their normal size. You can zoom in several times in a row for greater magnification.

If the area you're zooming in on does not contain at least one data point, the magnified area will be blank.

- To zoom in:
  1. Point to one corner of the area you want to select.
  2. To anchor the corner, hold down the mouse button and CTRL key simultaneously.
  3. Drag the pointer until the box encloses the area you want to magnify. A status box in the lower left-hand corner of the plot displays the pointer position.
  4. Release the mouse button and CTRL key.
- To zoom out:
  1. Hold down the CTRL key and click the right mouse button over the plot.

7.1.4 Printing a plot block

To print just a plot, click on the control-menu box in the upper left-hand corner of the `plot` and select the Print command.

7.1.5 Memory usage of plot block

The plot block uses eight bytes per data point of RAM. If you are running a long simulation at a small step size, it is possible to exceed your RAM limit. For example, a simulation with a step size of 0.005 and duration of 32236 would require 6.4 million points of data per plot trace. At eight bytes per point, each plot trace uses 51 MB of RAM. If you used all eight traces on a plot block, you would exceed 412 MB RAM.

7.1.6 Changing plot properties

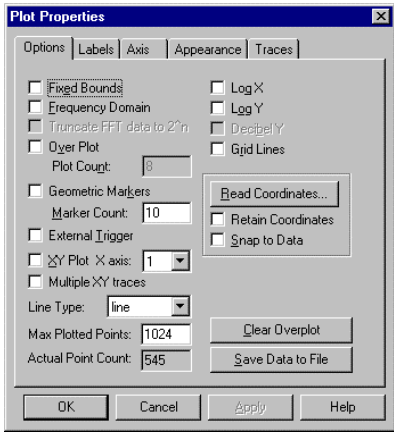
The Plot Properties dialog box controls how simulation data is presented. You can control how the data is presented by selecting parameters from the five property sheets that make up the dialog box.

➢ To access the dialog box:

1. Choose Edit > Block Properties.
2. Click the mouse over the `plot` block.
3. Click on the tab that corresponds to the property sheet containing the plotting parameters you want to change.
4. Select the plotting parameters. (See the descriptions below for information about each parameter.)
5. Click on the OK button, or press ENTER.

7.1.6.1 Options property sheet

The Option property sheet lets you choose between XY and frequency domain plots; select logarithmic scaling, fixed axis bounds, or a time axis scale; display signal traces as individual data points, line segments, or stepped line segments; and more.



Fixed Bounds: Specifies the region of the plot you want to view by letting you select the plotting bounds. When Fixed Bounds is activated, Commsim uses the values for the X Upper Bound, X Lower Bound, Y Upper Bound, and Y Lower Bound parameters in the Axis property sheet.

Frequency Domain: Obtains the frequency power spectrum through the use of the Fast Fourier Transform (FFT) algorithm.

Do not obtain frequency power spectrum data until after you have run a simulation. If you halt the simulation prematurely, the fidelity of the FFT is diminished.

Unexpected peaks

If your frequency domain plot produces unexpected peaks, check the simulation step size to verify that your sampling rate is adequate for obtaining accurate results. Then, based on the simulation step size and range, check the Plotted Points parameter to verify that you are indeed plotting each time step.

Truncate FFT Data To 2^n: When activated, this option truncates data down to the nearest power of 2. If you do not activate this option, the data buffer is padded with zeros to round up to the nearest power of 2.

This option can be turned on only when the Frequency Domain option is activated.

Over Plot, Plot Count, and Clear Overplot: When activated, Over Plot displays the results of multiple simulation runs in a single plot. This allows for better insight into how small changes can affect overall system performance.

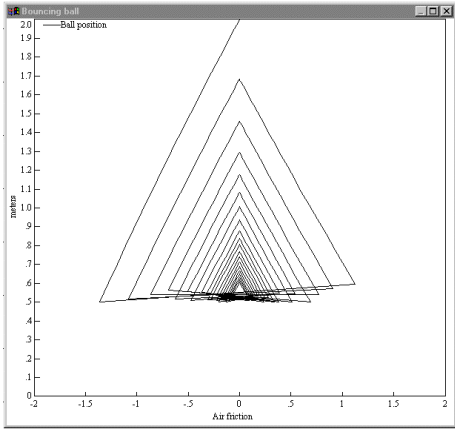
You can select the number of overlapping plots by entering a number into the Plot Count box. To clear all signal traces from a plot, click on the Clear Overplot button.

Geometric Markers and Marker Count: When activated, Geometric Markers overlays signal traces with geometric markers. To change the shape of a marker, click on the Traces tab in the Plot Properties dialog box. For more information, see “7.1.6.5 Traces property sheet” on page 7-11.

By default, Commsim overlays each signal trace with 10 markers; however, if this is not satisfactory, you can enter a new number in the Marker Count box.

External Trigger: Determines whether Commsim displays simulation data in the plot based on the value of an external trigger. When activated, External Trigger causes Commsim to place a round input connector on the `plot` block. When signal values entering the external trigger are 1, simulation data is plotted; when signal values entering the external trigger are 0, simulation data is not plotted.

XY Plot and X Axis: Together, XY Plot and X Axis let you use one input signal to represent X coordinate generation. As time advances, the remaining input signals are plotted relative to the X-axis signal.



In the XY plot above, ball position is plotted against air friction. At time 0, the ball position is at 2 and air friction is at 0. Over the course of the simulation, the ball position moves counter-clockwise, following a three-sided decaying cycle.

- To specify an XY plot:

  1. Activate the XY Plot parameter.
  2. Under the X Axis parameter, choose the input signal to be used for X coordinate generation: 1 represents the input signal attached to the top input connector tab on the `plot` block, 2 represents the input signal attached to the second to the top input connector tab on the `plot` block, and so on.
  3. Click on the OK button, or press ENTER.

- To label the x-axis on an XY plot:

  In an XY plot, Commsim automatically labels the x-axis with the label for the input signal used for x-coordinate generation. For example, if you activate XY Plot and choose 2 under X Axis, Commsim uses the label assigned to input signal 2.

  1. Click on the Labels tab.
  2. Enter a label for the input signal you chose to be used for X coordinate generation. The Trace 1 box corresponds to 1 in the X Axis parameter, the Trace 2 box corresponds to 2 in the X Axis parameter, and so on.
  3. Click on the OK button, or press ENTER.

  Multiple XY Traces: Creates two independent XY plots, which allows two signals to be superimposed. The XY Plot option must be activated in order to use this option.

Line Type: Click on the DOWN ARROW and choose Point, Line, or Discrete.

- Point displays signal values as individual data points. The primary advantage of point plots is that you can see the separation of data as time advances.
- Line connects data points with solid line segments. On color displays, line segments are keyed to the color to their corresponding input connector tab. On monochrome displays and printers, Commsim automatically uses line patterns (solid, dot, dash, and dot-dash) to distinguish multiple signals. You may have to lower the point count in the Plotted Points parameter to allow enough room between data points for the pattern to be displayed. If this is not satisfactory, you can overlay signal traces with geometric markers.
- Discrete displays signal values as stepped line segments. In a discrete plot, Commsim holds the Y value constant from point to point. A discrete plot is helpful when data points are irregularly spaced and you don't know where the curve is accurate.

Max Plotted Points: Determines the smoothness and accuracy of a plot. The more data points you plot, the smoother and more accurate the plot. However, increasing the number of plotted data points also increases the time it takes to print and display the plot.

The maximum number of data points that can be plotted is 250 million.

Setting a maximum default number of plotted points

If you know the maximum number of data points you want plotted in all your plots, you can set it as the default, as described in "5.4 Setting simulation defaults" on page 5-9.

Actual Point Count: Displays the number of data points plotted.

Log X and Log Y: Allow data to be plotted in logarithmic and semi-logarithmic coordinate systems. When you specify a logarithmic or semi-logarithmic plot, you cannot plot negative values on the log axis. Any negative value will be clipped to the low end of the scale. When neither parameter is activated, the plot defaults to linear.

To	Do this
Create a semi-logarithmic plot where the x-axis is $\log_{10}$ and the y-axis is linear	Activate the Log X parameter.
Create a semi-logarithmic plot where the y-axis is $\log_{10}$ and the x-axis is linear	Activate the Log Y parameter.
Create a plot using $\log_{10}$- $\log_{10}$ scales	Activate the Log X and Log Y parameters.

Decibel Y: Rescales the y-axis to display the values in decibels.

Grid Lines: Extends grid lines from the vertical and horizontal axis coordinates. Grid frequency — that is, the vertical and horizontal spacing of grid lines — is controlled by the spacing of the axis coordinates. Commsim automatically establishes reasonable axis coordinate spacing and hence controls the grid frequency.

Read Coordinates: Overlays the plot with a crosshair and displays its (*x*,*y*) coordinate position at the bottom of the plot.

When you click the left or right mouse button, Commsim freezes the crosshair. Click the left mouse button again to erase the crosshair.

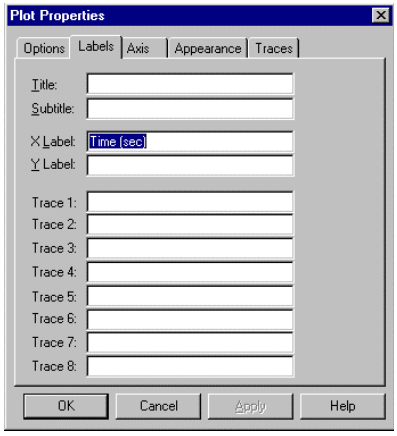
You have two crosshair options:

- **Snap to Data:** Causes the data point closest to the pointer to have a magnetic-like pull on the crosshair. As you move the pointer, the crosshair continues to jump to the closest data point.
- **Retain Coordinates:** Displays two crosshairs. One crosshair is frozen at the last known (*x*,*y*) position. The other is controlled by the position of the pointer.

Save Data To File: Opens the Select File dialog box to specify a file to which the plot data is to be saved. Click on the DOWN ARROW in the Files of Type box to choose a file format.

7.1.6.2 Labels property sheet

The Labels property sheet lets you name your plots, label the x- and y-axes, and apply names to signal traces.



Title and Subtitle: The Title and Subtitle parameters let you provide names for your plots. Titles and subtitles can be up to 80 alphanumeric characters. The title appears in the plot title bar; the subtitle is displayed in the top area of the plot. By default, plots are titled *Plot* and have no subtitles.

X Label and Y Label: The X Label parameter specifies a label for the x-axis. The Y Label parameter specifies a label for the y-axis.

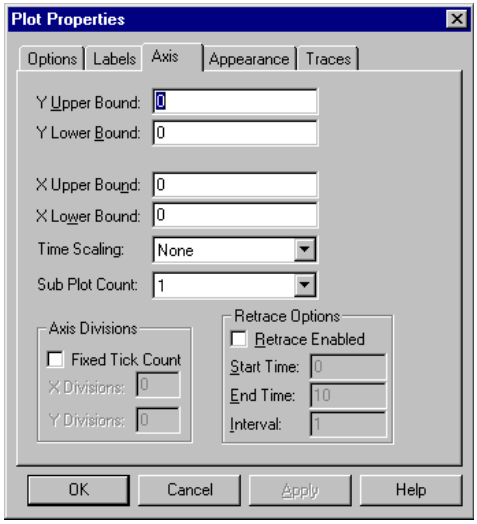
Axis labels can contain up to 80 alphanumeric characters.

Trace 1, Trace 2, Trace 3, Trace 4, Trace 5, Trace 6, Trace 7, and Trace 8: Let you specify labels for up to eight input signals. The Trace 1 box corresponds to the top input connector tab, the Trace 2 box corresponds to the next lower tab, and so on.

Signal labels can contain up to 80 alphanumeric characters.

7.1.6.3 Axis property sheet

The Axis property sheet lets set upper and lower bounds for the x- and y-axes, choose a time scale, specify axis divisions, and more.



Y Upper Bound, Y Lower Bound, X Upper Bound, and X Lower Bound: Specify the upper and lower bounds for the x- and y-axes. These bounds are in effect when you activate the Fixed Bounds parameter in the Options property sheet.

Time Scaling: Specifies x-axis scaling in microseconds, milliseconds, seconds, minutes, hours, and days. When you select a different time axis scale, Commsim re-calculates the values in the X Upper Bound and X Lower Bound boxes. When you close the dialog box, the x-axis is scaled to the time you chose.

Sub Plot Count: Lets you specify between 1 and 8 subplot windows, allowing you to plot each signal trace in a separate window. This feature is useful when signal traces overlap in range and obscure each other.

Axis Divisions: You can override the plot's grid tick division by activating Fixed Tick Count and entering values into the X Divisions and Y Divisions boxes. The numbers you enter indicate the number of grid ticks on each axis.

Retrace Options: Configures a plot as an *eye* diagram. Activate Retrace Enabled and specify the desired interval in the Interval box. In the Start Time and End Time boxes, enter the start and end times for the eye diagram. Eye diagrams are particularly useful for analyzing digital data waveforms.

7.1.6.4 Appearance property sheet

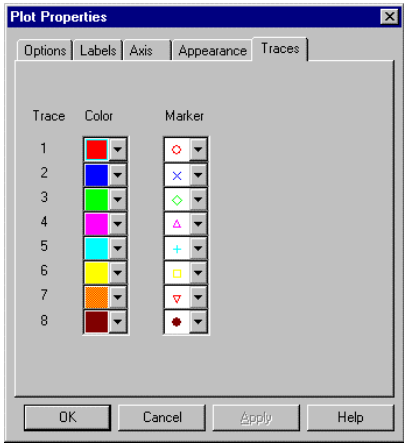
With the Appearance property sheet, you can add color and background patterns to your plots.

Color: Click on Foreground to color the axis labels and scaling text; click on Background to color the plotting area. Activate the Override Default Colors to override the color specified in the View > Colors command.

Bitmap: You can specify a bitmap image background for the plotting area. Type the file name directly into the Bitmap box or select one using Image command button. The specified bitmap image file overrides any background color selection.

7.1.6.5 Traces property sheet

Up to eight signal traces can be displayed on a single plot. You can select the color of the trace as well as the type of marker that appears on the trace.



➢ To choose a color and marker shape:

1. Invoke the Plot Properties dialog box.
2. Click on the Traces tab.
3. Select the color and marker shape from the drop-down list boxes that correspond with the signal trace.
4. Click on the OK button, or press ENTER.

7.2 Strip charts

The `stripChart` block displays up to eight signals in a scrolling window. You define the display width and scrollable width of the window. To scroll back and forth through the window, use the horizontal scroll bar at the bottom of the `stripChart` block.

You can customize the strip chart and control how data is presented in the following ways:

- Choose frequency domain strip charts.
- Select y-axis scaling, fixed bounds, or a time axis scale.
- Display signal traces as individual data points, line segments, or stepped line segments.
- Overlay signal traces with geometric markers.

You can also save simulation data to file in .DAT, .M, .MAT, and .WAV formats.

7.2.1 Basic time domain strip chart

Like the `plot` block, a `stripChart` block initially displays data in the time domain. All signals are plotted on the y-axis; x-axis represents time. As data points arrive to be plotted, Commsim dynamically rescales the plot bounds and connects the data points with line segments.

7.2.2 Sizing a stripChart block

To change the size or shape of the `stripChart` block for better viewing, drag the `stripChart's` borders or corners to adjust its size.

7.2.3 Printing a stripChart block

To print just a strip chart, click on the control-menu box in the upper left-hand corner of the `stripChart` and select the Print command. Commsim prints the strip chart in horizontal bands, with a maximum of four bands per page. Commsim uses as many pages as necessary to print all the data. Commsim also honors the margin settings specified by the File > Page Setup command.

7.2.4 Changing stripChart properties

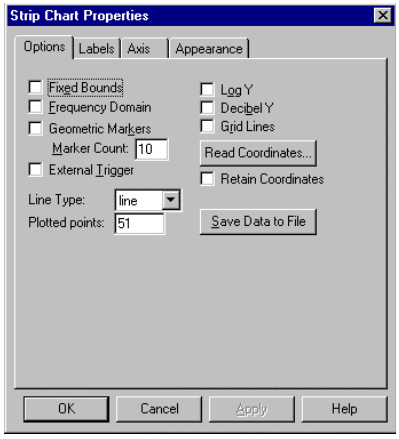
The Strip Chart Properties dialog box controls how simulation data is presented.

➢ To access the dialog box:

1. Choose Edit > Block Properties.
2. Click the mouse over the `stripChart` block.
3. Select the strip chart parameters. (See the descriptions below for information about each parameter.)
4. Click on the OK button, or press ENTER.

7.2.4.1 Options property sheet

The Option property sheet lets you select fixed axis bounds, logarithmic y scaling, or decibel y scaling; activate frequency domain plotting; enable an external trigger; display signal traces as individual data points, line segments, or stepped line segments; and more.



Fixed Bounds: Specifies the region of the strip chart you want to view by letting you select the plotting bounds. When Fixed Bounds is activated, Commsim uses the values for the X Upper Bound, X Lower Bound, Y Upper Bound, and Y Lower Bound parameters in the Axis property sheet. Out-of-range signal values are clipped to the existing strip chart bounds.

Frequency Domain: Obtains the frequency power spectrum through the use of the Fast Fourier Transform (FFT) algorithm.

Do not obtain frequency power spectrum data until after you have run a simulation. If you halt the simulation prematurely, the fidelity of the FFT is diminished.

Unexpected peaks

If your frequency domain plot produces unexpected peaks, check the simulation step size to verify that your sampling rate is adequate for obtaining accurate results. Then, based on the simulation step size and range, check the Plotted Points parameter to verify that you are indeed plotting each time step.

Geometric Markers and Marker Count: The Geometric Markers parameter overlays signal traces with geometric markers (squares, diamonds, circles, and triangles). These markers are particularly useful for monochrome displays and printers.

By default, Commsim overlays each signal trace with 10 markers; however, if this is not satisfactory, you can change the number of markers with the Marker Count parameter.

External Trigger: Determines whether Commsim displays simulation data in the strip chart based on the value of an external trigger. When activated, External Trigger causes Commsim to place a round input connector on the `stripChart` block. When signal values entering the external trigger are 1, simulation data is displayed in the strip chart; when signal values entering the external trigger are 0, simulation data is not displayed.

Line Type: Click on the DOWN ARROW and choose Point, Line, or Discrete.

- Point displays signal values as individual data points. The primary advantage of point plots is that you can see the separation of data as time advances.
- Line connects data points with solid line segments. On color displays, line segments are keyed to the color to their corresponding input connector tab. On monochrome displays and printers, Commsim automatically uses line patterns (solid, dot, dash, and dot-dash) to distinguish multiple signals. You may have to lower the point count in the Plotted Points parameter to allow enough room between data points for the pattern to be displayed. If this is not satisfactory, you can overlay signal traces with geometric markers.
- Discrete displays signal values as stepped line segments. In a discrete plot, Commsim holds the Y value constant from point to point. A discrete plot is helpful when data points are irregularly spaced and you don't know where the curve is accurate.

Plotted Points: Determines the smoothness and accuracy of a plot. The more data points you plot, the smoother and more accurate the plot. However, increasing the number of plotted data points also increases the time it takes to print and display the plot.

The maximum number of data points that can be plotted is 250 million.

Log Y: Enables a logarithmic y-axis. Note that you cannot plot negative values on a log axis. Any negative value is clipped to the low end of the scale.

Decibel Y: Rescales the y-axis to display the values in decibels.

Grid Lines: Extends grid lines from the vertical and horizontal axis coordinates. Grid frequency — that is, the vertical and horizontal spacing of grid lines — is controlled by the spacing of the axis coordinates. Commsim automatically establishes reasonable axis coordinate spacing and hence controls the grid frequency.

Read Coordinates and Retain Coordinates: The Read Coordinates command button overlays the strip chart with crosshair and displays its (*x*,*y*) coordinate position at the bottom of the chart.

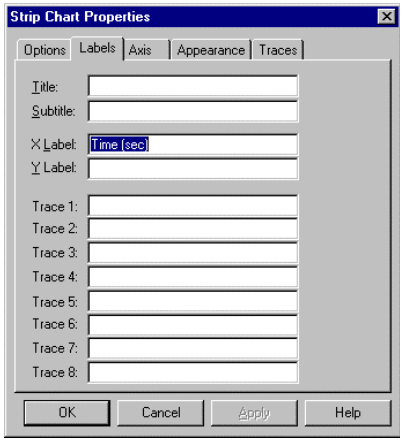
When you click the left or right mouse button, Commsim freezes the crosshair. Click the left mouse button again to erase the crosshair.

You can optionally activate the Retain Coordinates parameter. When you do so, the strip chart appears with two crosshairs. One crosshair is frozen at the last known (*x*,*y*) position. The other is controlled by the position of the pointer.

Save Data To File: Opens the Select File dialog box to specify a file to which the strip chart data is to be saved. Click on the DOWN ARROW in the Files of Type box to choose a file format.

7.2.4.2 Labels property sheet

The Labels property sheet lets you name your strip charts, label the x- and y-axes, and apply names to signal traces.



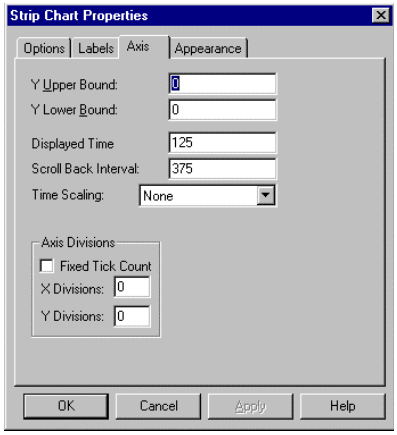
Title and Subtitle: The Title and Subtitle parameters let you provide names for your strip charts. Titles and subtitles can be up to 80 alphanumeric characters. The title appears in the plot title bar; the subtitle is displayed in the top area of the plot. By default, plots are titled *Strip Chart* and have no subtitles.

X Label and Y Label: The X Label and Y Label parameters specify labels for the x- and y-axes. Axis labels can contain up to 80 alphanumeric characters.

Trace 1, Trace 2, Trace 3, Trace 4, Trace 5, Trace 6, Trace 7, Trace 8: Let you specify labels for up to eight input signals. The Trace 1 box corresponds to the top input connector tab, the Trace 2 box corresponds to the next lower tab, and so on. Signal labels can contain up to 80 alphanumeric characters.

7.2.4.3 Axis property sheet

The Axis property sheet lets you define the display width and scrollable width of the window; set upper and lower bounds for the y-axis; choose a time scale; and specify axis divisions.



Y Upper Bound and Y Lower Bound: Specify the upper and lower bounds for the y-axis. These bounds are in effect when you activate the Fixed Bounds parameter in the Options property sheet.

Time Scaling: Specifies x-axis scaling in microseconds, milliseconds, seconds, minutes, hours, and days. When you select a different time axis scale, Commsim re-calculates the values in the X Upper Bound and X Lower Bound boxes. When you close the dialog box, the x-axis is scaled to the time you chose.

Axis Divisions: You can override the strip chart's grid tick division by activating Fixed Tick Count and entering values into the X Divisions and Y Divisions boxes. The numbers you enter indicate the number of grid ticks on each axis.

Displayed Time: Indicates the amount of units to be displayed in the strip chart window at any given time. The default value is ¼ of the total simulation time.

Scroll Back Interval: Indicates how much data (in x units) is saved for scrolling through. To conserve memory, keep this value low. To retain more data points, but use more memory, raise this value. The default value is the total simulation time.

7.2.4.4 Appearance property sheet

With the Appearance property sheet, you can add color and background patterns to your strip charts.

Color: Click on Foreground to color the axis labels and scaling text; click on Background to color the plotting area. Activate the Override Default Colors to override the color specified in the View > Colors command.

Bitmap: You can specify a bitmap image background for the plotting area. Type the file name directly into the Bitmap box or select one using Image command button. The specified bitmap image file overrides any background color selection.

7.3 Histograms

The `histogram` block shows how data are distributed over the course of a simulation. At each time step, a data point is placed in a bin that corresponds to a specific range. You can select the number of bins and the maximum and minimum bin value for the histogram. You can also select the maximum displayed bin height or have the `histogram` block dynamically rescale the bins as the data points arrive. The bins are spaced equally between the minimum and maximum bin values.

7.3.1 Sizing a histogram block

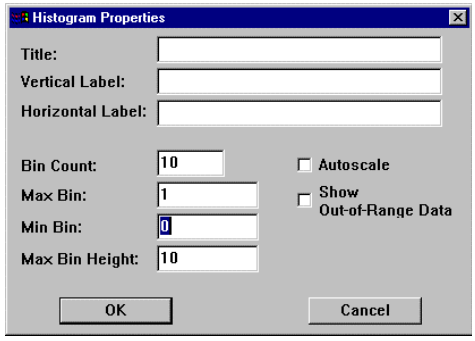
You might want to change the size or shape of the `histogram` block for better viewing. You can expand it to full screen size with the maximize button in the upper right-hand corner of the histogram or you can drag the histogram's borders or corners to adjust its size.

7.3.2 Changing histogram properties

The Histogram Properties dialog box controls how simulation data is presented.

➢ To access the dialog box:

1. Choose Edit > Block Properties.
2. Click the mouse over the `histogram` block.
3. Select the plotting parameters. (See the descriptions below for information about each parameter.)
4. Click on the OK button, or press ENTER.



Title: Specifies a title for the histogram. The default title is Histogram.

Vertical Label: Specifies a vertical axis label.

Horizontal Label: Specifies a horizontal axis label.

Bin Count: Indicates the number of bins. If you change the bin count, the bin values are reset. The default is 10.

Max Bin: Indicates the maximum value of the data. The default is 1.

Min Bin: Indicates the minimum value of the data. The default is 0.

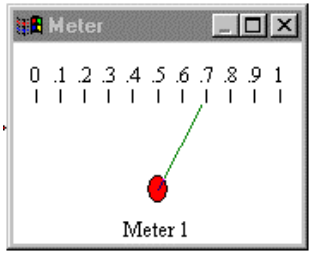
Max Bin Height: Indicates the maximum height of the bin. The default is 10.

Autoscale: Rescales plot when the maximum bin height is exceeded.

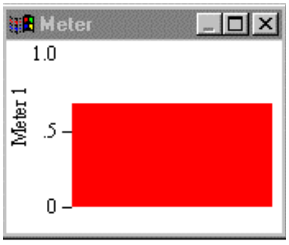
Show Out-Of-Range Data: Displays bins before and beyond the minimum and maximum bins to hold out-of-range data points.

7.4 Bar and needle graphs

The `meter` block displays signals in either a gauge- or a bar-style display. Initially, the `meter` block appears as a gauge-style display with one input connector tab.

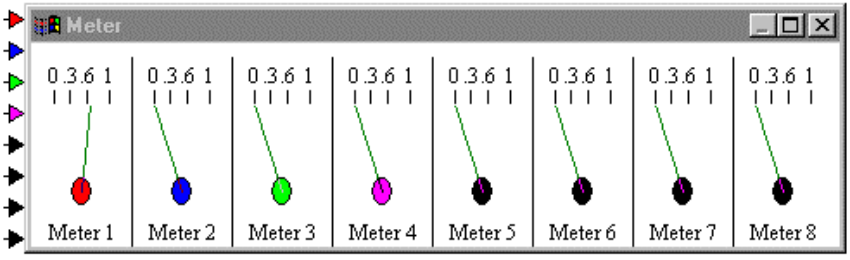


Gauge display



Bar display

You can display up to eight signals in a `meter` block.



Commsim displays each signal in a separate meter window. The color of the input connector tab corresponds to the bar (in a bar display) or bulb (in a gauge display) of the same color. You have the option of changing these colors.

The default number of input connector tabs is one. To change the number of input connector tabs, use the + and − toolbar buttons, or the corresponding Edit > Add Connector and Edit > Remove Connector commands.

7.4.1 Sizing a meter block

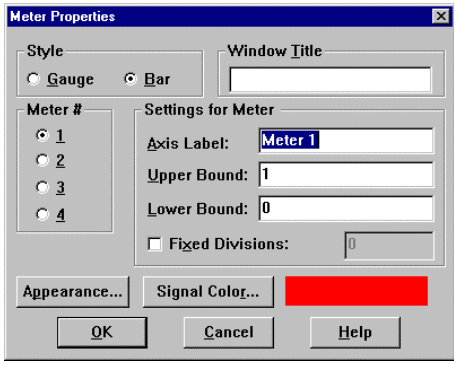
You might want to change the size or shape of the `meter` block for better viewing. You can expand it to full screen size with the Maximize button in the upper right-hand corner of the `meter` or you can drag the `meter's` borders or corners to adjust its size.

7.4.2 Changing meter properties

The Meter Properties dialog box controls how simulation data is presented.

➢ To access the dialog box:

1. Choose Edit > Block Properties.
2. Click the mouse over the `meter` block.
3. Select the plotting parameters. (See the descriptions below for information about each parameter.)
4. Click on the OK button, or press ENTER.



Style: Switches between a bar and gauge display.

Meter #: 1, 2, 3, and 4 indicate the signal whose settings are to be examined or changed. The text in the Axis Label, Upper Bound, and Lower Bound boxes correspond to the selected signal.

Window Title: Indicates a title for the `meter` block. The title appears in the title bar that runs across the top of the `meter` block. The default title is *Meter.*

Axis Label: Indicates a name for the axis on which the signal is displayed. In a gauge display, the axis label is displayed horizontally across the top of the display; in a bar display, the axis label is displayed vertically along the left-hand side of the display.

Upper Bound and Lower Bound: Control the upper and lower bound of the meter display. The defaults are 1 and 0, respectively.

Fixed Division: Indicates the number of grid ticks.

Appearance: Opens the Appearance dialog box. Click on Foreground to color the axis label and scale text; click on Background to color the plotting area. The color you specify overrides the color specified in the View > Colors command.

You can alternatively specify a bitmap image background for the plotting area. Type the file name directly into the Bitmap box or select one using the Select Bitmap command button. The specified bitmap image file overrides any background color selection.

Signal Color: Opens the Color dialog box in which to specify a color for the input connector tab, the bulb and needle (in a gauge-style display), and the bar (in a bar-style display).

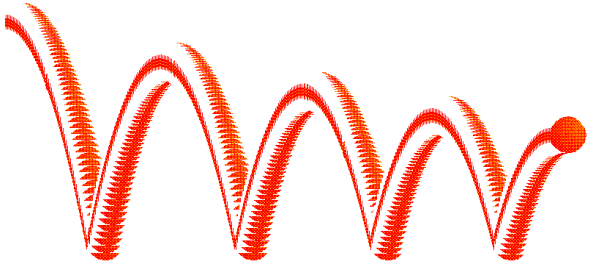
7.5 Creating animation

Animation is a series of images that, during a simulation, creates the illusion of movement. Commsim provides two blocks to create animations: the `animate` block, for animating an image, and the `lineDraw` block, for animating a line.

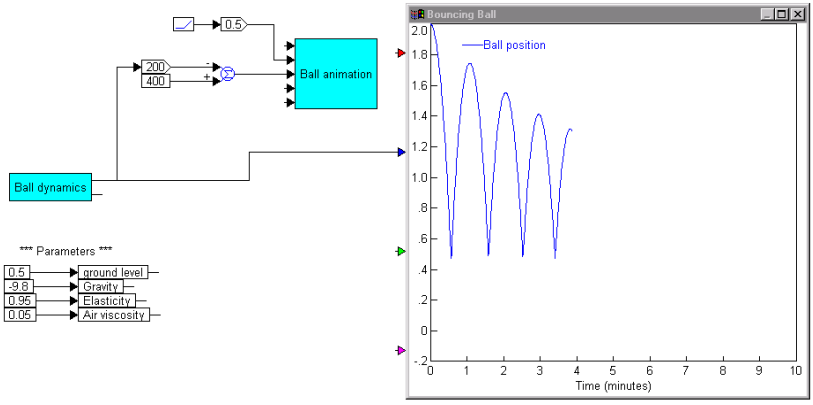
7.5.1 Animation basics

Animation occurs only when you initiate a simulation with display mode active. In this mode, all wires are hidden, all blocks are frozen in place, and with the exception of the interactive elements on `button` and `slider` blocks, block parameter values cannot be changed. To activate display mode, click on the View > Display Mode command. A check mark in front of the command indicates that it is active.

Animation works by feeding signals into an animation block. The signals drive the coordinates of the animation block, which result in motion. For example, consider the bouncing ball animation, shown below:



The ball is represented as a single `animate` block. Movement is introduced by changing the ball's *x,y* screen position coordinates. As the simulation progresses, the signals entering the `animate` block continually update its position. The diagram below drives the simulation of the bouncing ball.



To create the illusion of depth, you can vary the ball's *w,h* size coordinates.

7.5.2 Using the animate block

The `animate` block lets you animate an image during simulation. Animation occurs through movement and changes in the size or appearance of the image.

The Animate Properties dialog box controls how animation data is presented.

➢ To access the dialog box:

1. Choose Edit > Block Properties.
2. Click the mouse over the `animate` block.
3. Select the animation parameters. (See the descriptions below for information about each parameter.)

4. Click on the OK button, or press ENTER.

7.5.2.1 Applying pictures

The pictures you apply to an `animate` block must be in bitmap file format (.BMP). Commsim supports Windows-formatted bitmaps with up to 256 colors.

➢ To apply pictures to an animate block:

1. In the Number of Images box, enter then number of pictures to be applied to the block. You can have up to 16 different pictures
2. In the States box, select state 0.
3. Do one of the following:
   - Click on the Associate Bitmap command button. Commsim displays the File Open dialog box in which you can select a .BMP file to be associated with state 0. The .BMP file name appears in the File Name box.
   - In the File Name box, enter the complete pathname of the .BMP file to be associated with state 0.
4. To apply a second bitmap image to the `animate` block, select state 1 from the States box and repeat step 3. Continue to repeat steps 3 and 4, incrementing the state number, for each bitmap image you want to apply to the `animate` block.
5. Click on the OK button, or press ENTER.

7.5.2.2 Creating animation

Signals fed into the `animate` block drive the animation during simulation. The `animate` block accepts five input signals.

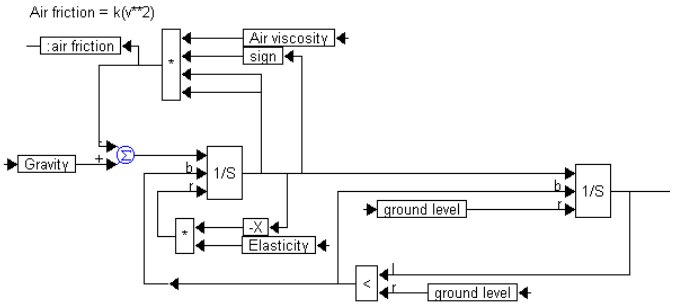
Signals fed into the top input: The top input connector tab determines which image is applied to the `animate` block. An input signal value of 0 causes the bitmap image file corresponding to state 0 to be displayed; an input signal value of 1 causes the bitmap image file corresponding to state 1 to be displayed; and so on. Signal values entering the top input adhere to these rules:

- When a signal value is a non-integer, the `animate` block truncates it to integer form.
- When a signal value is greater than the number of set states, the `animate` block uses the highest set state.
- When the signal value is negative, the `animate` block uses state 0.

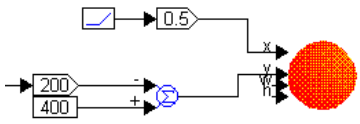
Signals fed into the x, y, w, and h inputs: The "x" and "y" connectors provide the *x*,*y* screen position coordinates for the image. The input connector tabs labeled "w" and "h" provide the width and height of the image. By varying these values, you can create movement and depth.

The values fed into the x, y, w, and h inputs represent display pixels. The *x, y* position (0,0) is the upper left corner of the Commsim window. Positive values extend to the right and down. For your image to appear on most video screens, keep its position within the bounds of a VGA screen (640x480).

You must perform all coordinate conversion manually. For example, the equations that determine the position of a bouncing ball are shown below:



However, before the output of the `resetIntegrator` block can be fed into the `animate` block, the position of the ball must be mapped to screen coordinates, as shown below:



The objective is to animate the bouncing of the ball as a function of time. This means that time is the independent variable, or the x-axis. A `ramp` block is used to generate time, which ranges between 0 and 1000. A gain of 0.5 is used to scale time so that the amount of space used on the screen in the horizontal direction is limited in the range 0 to 500 pixels.

The ball position, *y*, varies from 0.5 to 2 in the simulation. This is scaled 0 to 300 pixels by using the scaling shown. Note that the pixel location (0,0) corresponds to the top left corner of your screen, and the largest pixel location is the bottom right corner of your screen.

Creating a trail: To leave a trail of the picture as the simulation progresses, activate the Leave Trail on Motion parameter.

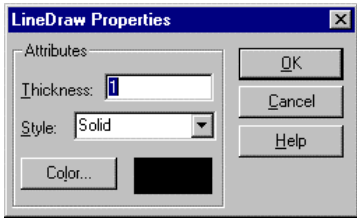
7.5.3 Using the lineDraw block

The `lineDraw` block lets you animate a line during simulation. You define the line by specifying two sets of *x*,*y* screen coordinate endpoints. You can also set the color, thickness, and style of the line.

The LineDraw Properties dialog box controls how a line is animated.

➢ To access the dialog box:

1. Choose Edit > Block Properties.
2. Click the mouse over the `lineDraw` block. The lineDraw Properties dialog box appears.



3. Do the following:

To specify	Do this
Color	Click on the Color command button and choose a color from the color palette.
Line style	Click on the DOWN ARROW for the Style box and select a style from the drop down list.
Line thickness	In the Thickness box, enter a value. Values are specified in points.

4. Click on the OK button, or press ENTER.

7.5.3.1 Creating line motion

Like the `animate` block, the `lineDraw` block uses the signal fed into its inputs to create motion. The top two inputs provide one set of *x*,*y* screen coordinate endpoints for the line; the bottom two inputs provide the other set. By varying these values, you can create motion.

The signal values fed into the inputs represent display pixels. Position (0,0) is the upper left corner of the Commsim window. Positive values extend to the right and down. For your line to appear on most video screens, keep its position within the bounds of a VGA screen (640x480).

7.5.4 Other ways to create animation

Animation can also be applied to a simulation using the `light`, `button`, and `bezel` blocks.

Use this block	To	For more information
light	Alternate among three images	Refer to the *Commsim 7 User Guide and Block Reference*
button	Alternate among 16 images	Refer to the *Commsim 7 User Guide and Block Reference*
bezel	Create operator control panels	Refer to the *Commsim 7 User Guide and Block Reference*

Index

C

R

S

T

U

V

W

X