

Communications System Toolbox™

Getting Started Guide



MATLAB® & SIMULINK®

R2015b



How to Contact MathWorks



Latest news: www.mathworks.com
Sales and services: www.mathworks.com/sales_and_services
User community: www.mathworks.com/matlabcentral
Technical support: www.mathworks.com/support/contact_us



Phone: 508-647-7000



The MathWorks, Inc.
3 Apple Hill Drive
Natick, MA 01760-2098

Communications System Toolbox™ Getting Started Guide

© COPYRIGHT 2011–2015 by The MathWorks, Inc.

The software described in this document is furnished under a license agreement. The software may be used or copied only under the terms of the license agreement. No part of this manual may be photocopied or reproduced in any form without prior written consent from The MathWorks, Inc.

FEDERAL ACQUISITION: This provision applies to all acquisitions of the Program and Documentation by, for, or through the federal government of the United States. By accepting delivery of the Program or Documentation, the government hereby agrees that this software or documentation qualifies as commercial computer software or commercial computer software documentation as such terms are used or defined in FAR 12.212, DFARS Part 227.72, and DFARS 252.227-7014. Accordingly, the terms and conditions of this Agreement and only those rights specified in this Agreement, shall pertain to and govern the use, modification, reproduction, release, performance, display, and disclosure of the Program and Documentation by the federal government (or other entity acquiring for or through the federal government) and shall supersede any conflicting contractual terms or conditions. If this License fails to meet the government's needs or is inconsistent in any respect with federal procurement law, the government agrees to return the Program and Documentation, unused, to The MathWorks, Inc.

Trademarks

MATLAB and Simulink are registered trademarks of The MathWorks, Inc. See www.mathworks.com/trademarks for a list of additional trademarks. Other product or brand names may be trademarks or registered trademarks of their respective holders.

Patents

MathWorks products are protected by one or more U.S. patents. Please see www.mathworks.com/patents for more information.

Revision History

April 2011	First printing	New for Version 5.0 (Release 2011a)
September 2011	Online only	Revised for Version 5.1 (Release 2011b)
March 2012	Online only	Revised for Version 5.2 (Release 2012a)
September 2012	Online only	Revised for Version 5.3 (Release 2012b)
March 2013	Online only	Revised for Version 5.4 (Release 2013a)
September 2013	Online only	Revised for Version 5.5 (Release 2013b)
March 2014	Online only	Revised for Version 5.6 (Release 2014a)
October 2014	Online only	Revised for Version 5.7 (Release 2014b)
March 2015	Online only	Revised for Version 6.0 (Release 2015a)
September 2015	Online only	Revised for Version 6.1 (Release 2015b)

1 | **Introduction**

Communications System Toolbox Product Description	1-2
Key Features	1-2
System Setup	1-4
Required Products	1-4
Expected Background	1-4
Configure the Simulink Environment for Communications Models	1-5
About the Communications System Toolbox Simulink Model Template	1-5
Create Model Using the Communications System Toolbox Simulink Model Template	1-5
The Communications System Toolbox Simulink Model Template	1-6
Access the Block Libraries	1-8
Communications System Toolbox Supported Hardware	1-9

2 | **System Simulation**

256-QAM with Simulink Blocks	2-2
Section Overview	2-2
Opening the Model	2-2
Overview of the Model	2-3
Quadrature Amplitude Modulation	2-4
Run a Simulation	2-5

Display the Error Rate	2-6
Set Block Parameters	2-7
Display a Phase Noise Plot	2-9
16-QAM with MATLAB Functions	2-11
Introduction	2-11
Modulate a Random Signal	2-11
Plot Signal Constellations	2-17
Pulse Shaping Using a Raised Cosine Filter	2-21
Error Correction using a Convolutional Code	2-28
Iterative Design Workflow for Communication Systems ..	2-32
Simulate a basic communications system	2-33
Introduce convolutional coding and hard-decision Viterbi decoding	2-37
Improve results using soft-decision decoding	2-41
Use turbo coding to improve BER performance	2-46
Apply a Rayleigh channel model	2-49
Use OFDM-based equalization to correct multipath fading .	2-54
Use multiple antennas to further improve system performance	2-57
Accelerate the simulation using MATLAB Coder	2-61
QPSK and OFDM with MATLAB System Objects	2-63
Accelerating BER Simulations Using the Parallel Computing Toolbox	2-67

Visualization and Measurements

3

Scatter Plot and Eye Diagram with MATLAB Functions ...	3-2
EVM and MER Measurements with Simulink	3-7
ACPR and CCDF Measurements with MATLAB System Objects	3-14
ACPR Measurements	3-14
CCDF Measurements	3-17

What Are System Objects?	4-2
System Objects in Simulink	4-3
System Objects in the MATLAB Function Block	4-3
System Object Methods	4-4
What Are System Object Methods?	4-4
The Step Method	4-4
Common Methods	4-5

Introduction

- “Communications System Toolbox Product Description” on page 1-2
- “System Setup” on page 1-4
- “Configure the Simulink Environment for Communications Models” on page 1-5
- “Access the Block Libraries” on page 1-8
- “Communications System Toolbox Supported Hardware” on page 1-9

Communications System Toolbox Product Description

Design and simulate the physical layer of communications systems

Communications System Toolbox™ provides algorithms and apps for the analysis, design, end-to-end simulation, and verification of communications systems in MATLAB® and Simulink®. Toolbox algorithms, including channel coding, modulation, MIMO, and OFDM, enable you to compose a physical layer model of your system. You can simulate your models to measure performance.

The system toolbox provides constellation and eye diagrams, bit-error-rate, and other analysis tools and scopes for validating your designs. These tools enable you to analyze signals, visualize channel characteristics, and obtain performance metrics such as error vector magnitude (EVM). Channel and RF impairment models and compensation algorithms, including carrier and symbol timing synchronizers, enable you to realistically model your link-level specifications and compensate for the effects of channel degradations.

Using Communications System Toolbox hardware support packages, you can connect your transmitter and receiver models to radio devices and verify your designs with over-the-air testing. The system toolbox supports fixed-point arithmetic and C or HDL code generation.

Algorithms are available as MATLAB functions, System objects, and Simulink blocks.

Key Features

- Algorithms for designing the physical layer of communications systems, including channel coding, modulation, OFDM, MIMO, equalization, and synchronization
- Analysis tools and measurement scopes, including a bit-error-rate app, constellation diagrams, and eye diagrams
- Channel models, including AWGN, multipath Rayleigh fading, Rician fading, MIMO multipath fading, and LTE MIMO multipath fading
- Basic RF impairment models, including nonlinearity, phase noise, thermal noise, and phase and frequency offsets
- Hardware support packages for connecting waveforms to radio devices and verifying designs with over-the-air testing
- GPU-enabled algorithms for computationally intensive algorithms such as Turbo, LDPC, and Viterbi decoders

- Support for fixed-point modeling and C and HDL code generation

System Setup

In this section...
“Required Products” on page 1-4
“Expected Background” on page 1-4

Required Products

The Communications System Toolbox product is part of a family of MathWorks® products. You need to install several products to use this product. For more information about the required products, see the MathWorks website, at <http://www.mathworks.com/products/communications/requirements.html>.

Expected Background

This documentation assumes that you already have background knowledge in the subject of digital communications. If you do not yet have this background, then you can acquire it using a standard communications text or the books listed in the Selected Bibliography subsections that accompany many topics.

The discussion and examples in this section are aimed at new users. Continue reading and try the examples. Then, read the subsequent content that pertains to your specific areas of interest. As you learn which System object™, block, or function you want to use, refer to the online reference pages for more information.

Configure the Simulink Environment for Communications Models


About the Communications System Toolbox Simulink Model Template

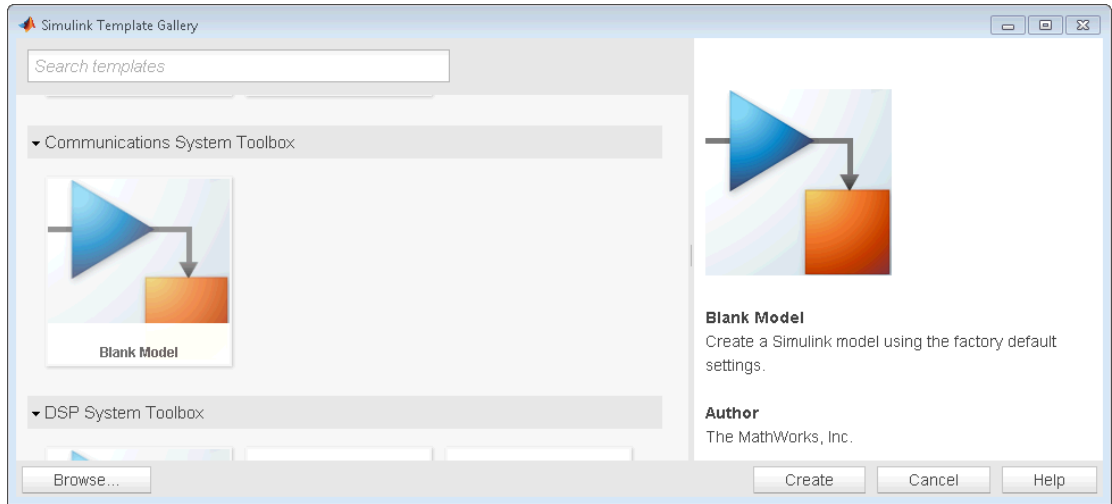
The Communications System Toolbox Simulink model template lets you automatically configure the Simulink environment with the recommended settings for communications modeling. Communications System Toolbox Simulink model templates enable reuse of settings, including configuration parameters. The model you create from the template uses best practices and takes advantage of previous solutions to common problems which helps you get started more quickly.

For more information on Simulink model templates, see “Create a Model Using a Template” in the Simulink documentation.

Create Model Using the Communications System Toolbox Simulink Model Template

To use the Communications System Toolbox Simulink model templates:

- 1 Open the Library Browser by typing `simulink` at the MATLAB command prompt.
- 2 Click the New Model button arrow  and select **From Template**.
- 3 In the Simulink Template Gallery, under Communications System Toolbox, double-click the model template.



The new model using the template settings and contents appears in the Simulink Editor. The model is only in memory until you save it.


The Communications System Toolbox Simulink Model Template

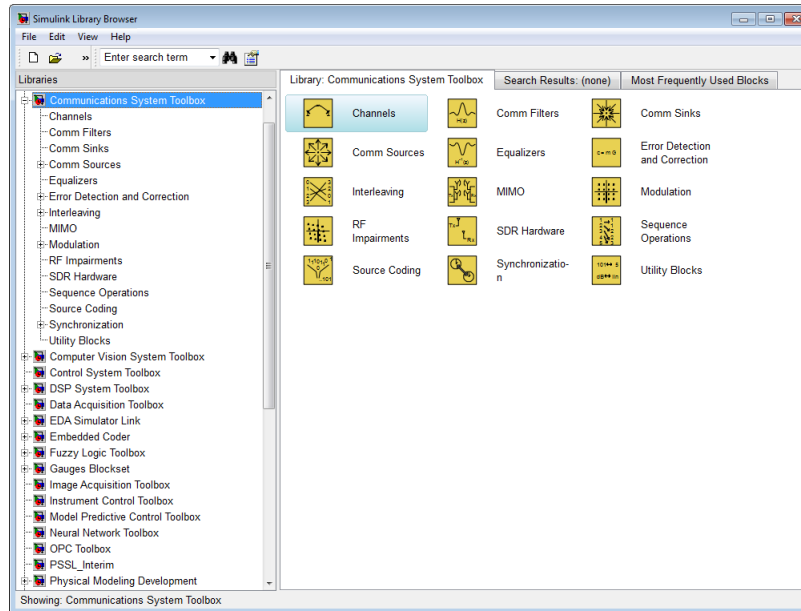
When you create a model by choosing the Communications System Toolbox Simulink model template, the model is configured to use the settings recommended for communications modeling. Some of these settings are:

Configuration Parameter	Setting
'SingleTaskRateTransMsg'	'error'
'Solver'	'VariableStepDiscrete'
'SolverMode'	'SingleTasking'
'MaxStep'	'auto'
'StartTime'	'0.0'
'StopTime'	'inf'
'FixedStep'	'auto'
'SaveTime'	'off'
'SaveOutput'	'off'

Configuration Parameter	Setting
'AlgebraicLoopMsg '	'error '
'RTWInlineParameters '	'on '
'BooleanDataType '	'off '
'UnnecessaryDatatypeConvMsg '	'none '
'LocalBlockOutputs '	'off '

Access the Block Libraries

To view the block libraries for the products you have installed, type `simulink` at the MATLAB prompt (or click the Simulink button  on the MATLAB toolbar). The Simulink Library Browser appears.



Simulink Library Browser

The left pane displays the installed products, each of which has its own library of blocks. To open a library, click the + sign next to the product name in the left pane. This displays the contents of the library in the right pane.

You can find the blocks you need to build communications system models in the Communications System Toolbox, DSP System Toolbox™, and Simulink libraries.

Alternatively, you can access the main Communications System Toolbox block library by entering `commlib` at the MATLAB command line.

Communications System Toolbox Supported Hardware

As of this release, Communications System Toolbox supports the following hardware.

Support Package	Vendor	Earliest Release Available	Last Release Available
RTL-SDR Radio	NooElec™	R2013b	Current
USRP® Radio	National Instruments®	R2011b	Current
Xilinx FPGA-Based Radio	Xilinx®	R2013b	Current
Xilinx Zynq-Based Radio	Xilinx	R2014b	Current

For a complete list of supported hardware, see Hardware Support.

System Simulation

- “256-QAM with Simulink Blocks” on page 2-2
- “16-QAM with MATLAB Functions” on page 2-11
- “Iterative Design Workflow for Communication Systems” on page 2-32
- “QPSK and OFDM with MATLAB System Objects” on page 2-63
- “Accelerating BER Simulations Using the Parallel Computing Toolbox” on page 2-67

256-QAM with Simulink Blocks

In this section...

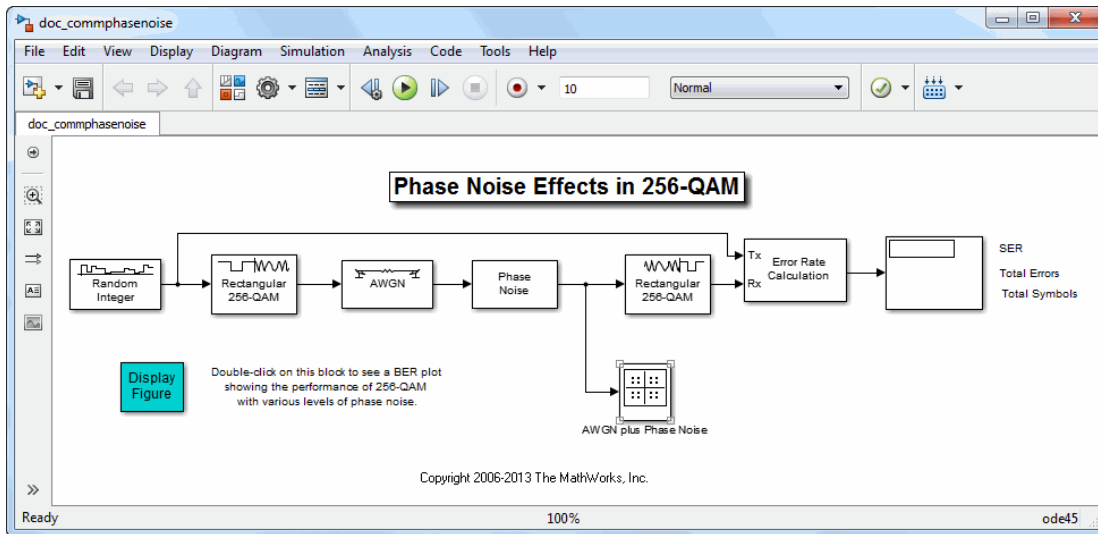
“Section Overview” on page 2-2
“Opening the Model” on page 2-2
“Overview of the Model” on page 2-3
“Quadrature Amplitude Modulation” on page 2-4
“Run a Simulation” on page 2-5
“Display the Error Rate” on page 2-6
“Set Block Parameters” on page 2-7
“Display a Phase Noise Plot” on page 2-9

Section Overview

This section describes an example model of a communications system. The model displays a scatter plot of a signal with added noise. The purpose of this section is to familiarize you with the basics of Simulink models and how they function.

Opening the Model

To open the model, first start MATLAB. In the MATLAB Command Window, enter `doc_comphasenoise` at the prompt. This opens the model in a new window, as shown in the following figure.



Overview of the Model

The model shown in the preceding section, “Opening the Model” on page 2-2, simulates the effect of phase noise on quadrature amplitude modulation (QAM) of a signal. The Simulink model is a graphical representation of a mathematical model of a communication system that generates a random signal, modulates it using QAM, and adds noise to simulate a channel. The model also contains components for displaying the symbol error rate and a scatter plot of the modulated signal.

The blocks and lines in the Simulink model describe mathematical relationships among signals and states:

- The Random Integer Generator block, labeled Random Integer, generates a signal consisting of a sequence of random integers between zero and 255
- The Rectangular QAM Modulator Baseband block, to the right of the Random Integer Generator block, modulates the signal using baseband 256-ary QAM.
- The AWGN Channel block models a noisy channel by adding white Gaussian noise to the modulated signal.
- The Phase Noise block introduces noise in the angle of its complex input signal.
- The Rectangular QAM Demodulator Baseband block, to the right of the Phase Noise block, demodulates the signal.

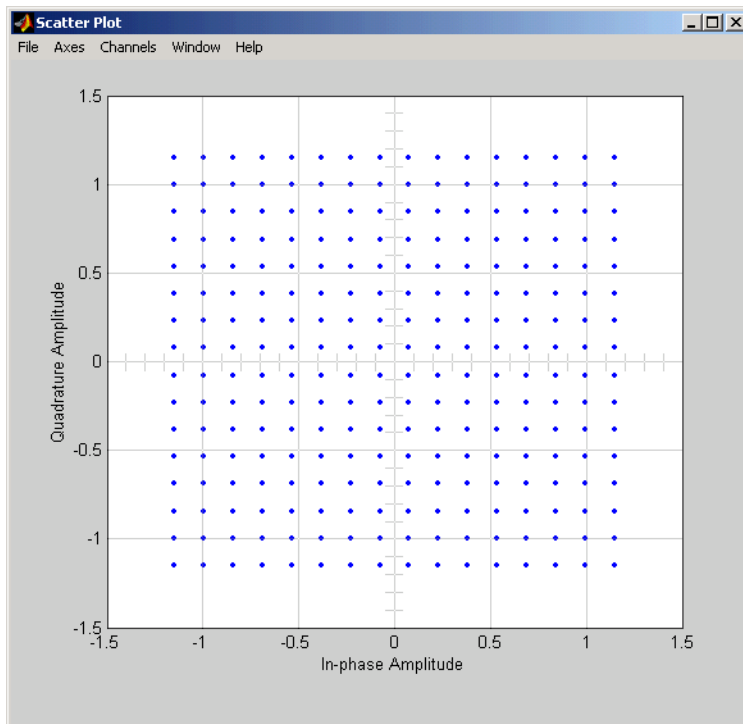
In addition, the following blocks in the model help you interpret the simulation:

- The Constellation Diagram block, labeled AWGN plus Phase Noise, displays a scatter plot of the signal with added noise.
- The Error Rate Calculation block counts symbols that differ between the received signal and the transmitted signal.
- The Display block, at the far right of the model window, displays the symbol error rate (SER), the total number of errors, and the total number of symbols processed during the simulation.

All these blocks are included in Communications System Toolbox. You can find more detailed information about these blocks by right-clicking the block and selecting **Help** from the context menu.

Quadrature Amplitude Modulation

This model simulates quadrature amplitude modulation (QAM), which is a method for converting a digital signal to a complex signal. The model modulates the signal onto a sequence of complex numbers that lie on a lattice of points in the complex plane, called the *constellation* of the signal. The constellation for baseband 256-ary QAM is shown in the following figure.

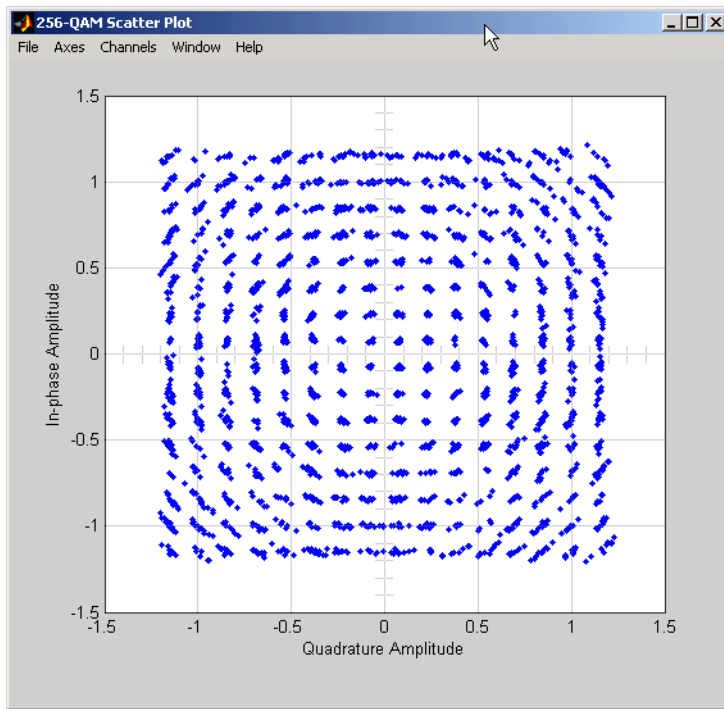


Constellation for 256-ary QAM

Run a Simulation

To run a simulation, click on the **Run** button at the top of the model window. The simulation stops automatically at the **Stop time**, which is specified in the **Configuration Parameters** dialog box. You can stop the simulation at any time by selecting **Stop** from the **Simulation** menu at the top of the model window (or, on Microsoft Windows, by clicking the **Stop** button on the toolbar).

When you run the model, a new window appears, displaying a scatter plot of the modulated signal with added noise, as shown in the following figure.



Scatter Plot of Signal Plus Noise

The points in the scatter plot do not lie exactly on the constellation shown in the figure because of the added noise. The radial pattern of points is due to the addition of phase noise, which alters the angle of the complex modulated signal.

Display the Error Rate

The Display block displays the number of errors introduced by the channel noise. When you run the simulation, three small boxes appear in the block, as shown in the following figure, displaying the vector output from the Error Rate Calculation block.

Note: The image below is a representative example and may not exactly match results you see when running in Simulink.

0.007701	SER
774	Total Errors
1.005e+005	Total Symbols

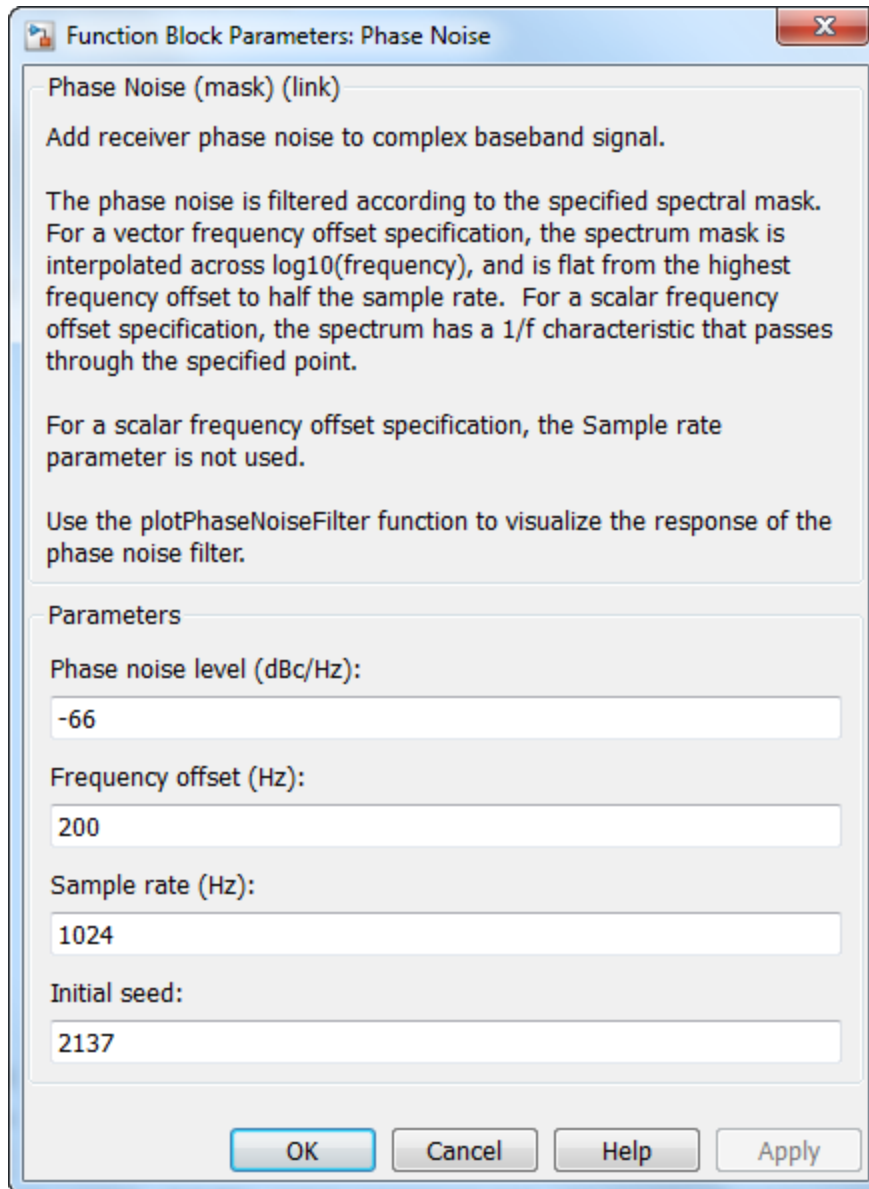
Error Rate Display

The block displays the output as follows:

- The first entry is the symbol error rate (SER).
- The second entry is the total number of errors.
- The third entry is the total number of comparisons made. The notation $1e+004$ is shorthand for 10^4 .

Set Block Parameters

You can control the way a Simulink block functions by setting its parameters. To view or change a block's parameters, double-click the block. This opens a dialog box, sometimes called the block's *mask*. For example, the dialog box for the Phase Noise block is shown in the following figure.



Dialog for the Phase Noise Block

To change the amount of phase noise, click in the **Phase noise level (dBc/Hz)** field and enter a new value. Then click **OK**.

Alternatively, you can enter a variable name, such as `phasenoise`, in the field. You can then set a value for that variable in the MATLAB Command Window, for example by entering `phasenoise = -60`. Setting parameters in the Command Window is convenient if you need to run multiple simulations with different parameter values.

You can also change the amount of noise in the AWGN Channel block. Double-click the block to open its dialog box, and change the value in the **Es/No** parameter field. This changes the signal to noise ratio, in dB. Decreasing the value of **Es/No** increases the noise level.

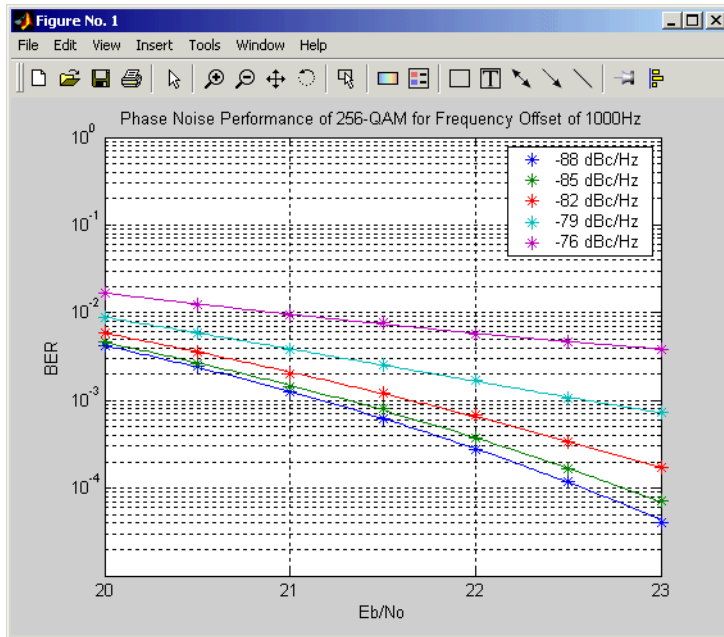
You can experiment with the model by changing these or other parameters and then running a simulation. For example,

- Change **Phase noise level (dBc/Hz)** to -150 in the dialog box for the Phase Noise block.
- Change **Es/No** to 100 in the dialog for the AWGN Channel block.

This removes nearly all noise from the model. When you now run a simulation, the scatter plot appears as in the figure Constellation for 256-ary QAM.

Display a Phase Noise Plot

Double-click the block labeled “Display Figure” at the bottom left of the model window. This displays a plot showing the results of multiple simulations.



BER Plot at Different Noise Levels

Each curve is a plot of bit error rate as a function of signal to noise ratio for a fixed amount of phase noise.

You can create plots like this by running multiple simulations with different values for the **Phase noise level (dBc/Hz)** and **E_s/N_0** parameters.

16-QAM with MATLAB Functions

In this section...

“Introduction” on page 2-11

“Modulate a Random Signal” on page 2-11

“Plot Signal Constellations” on page 2-17

“Pulse Shaping Using a Raised Cosine Filter” on page 2-21

“Error Correction using a Convolutional Code” on page 2-28

Introduction

This section builds an example step-by-step to give you a first look at the Communications System Toolbox software. This section also shows how Communications System Toolbox functionalities build upon the computational and visualization tools in the underlying MATLAB environment.

Modulate a Random Signal

This example shows how to process a binary data stream using a communication system that consists of a baseband modulator, channel, and demodulator. The system's bit error rate (BER) is computed and the transmitted and received signals are displayed in a constellation diagram.

The following table summarizes the basic operations used, along with relevant Communications System Toolbox and MATLAB functions. The example uses baseband 16-QAM (quadrature amplitude modulation) as the modulation scheme and AWGN (additive white Gaussian noise) as the channel model.

Task	Function
Generate a Random Binary Data Stream	<code>randi</code>
Convert the Binary Signal to an Integer-Valued Signal	<code>bi2de</code>
Modulate using 16-QAM	<code>qammod</code>
Add White Gaussian Noise	<code>awgn</code>

Task	Function
Create a Constellation Diagram	scatterplot
Demodulate using 16-QAM	qamdemod
Convert the Integer-Valued Signal to a Binary Signal	de2bi
Compute the System BER	biterr

Generate a Random Binary Data Stream

The conventional format for representing a signal in MATLAB is a vector or matrix. This example uses the `randi` function to create a column vector that contains the values of a binary data stream. The length of the binary data stream (that is, the number of rows in the column vector) is arbitrarily set to 30,000.

The code below also creates a stem plot of a portion of the data stream, showing the binary values. Notice the use of the colon (`:`) operator in MATLAB to select a portion of the vector.

Define parameters.

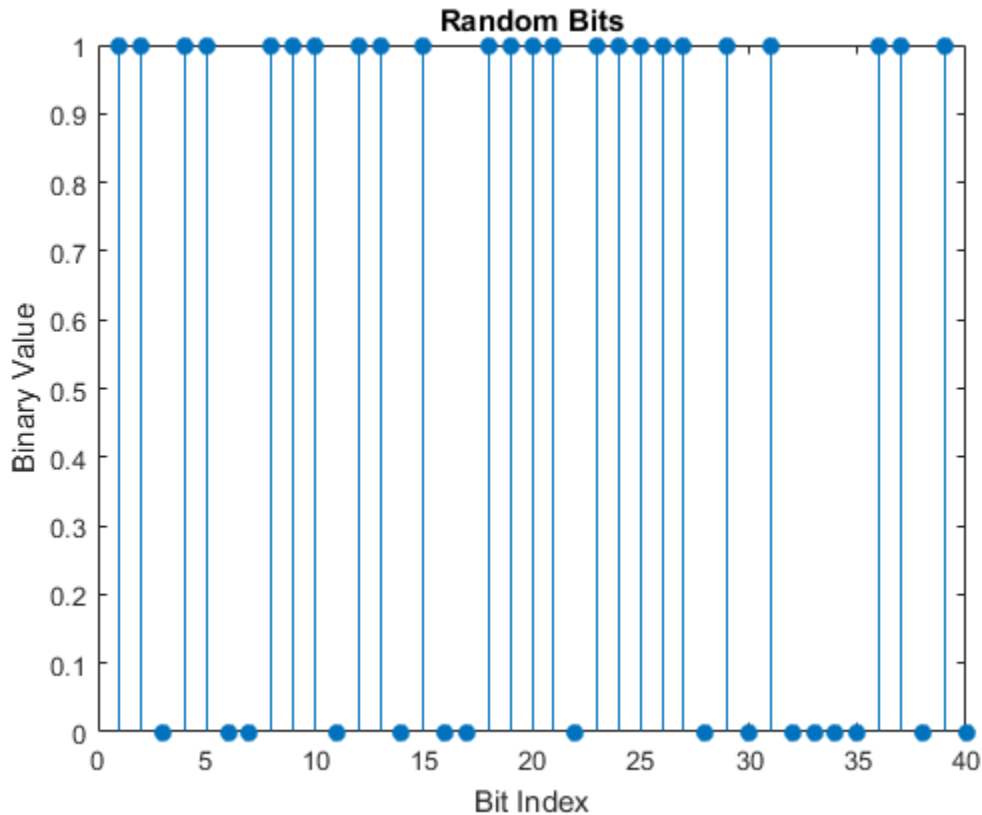
```
M = 16; % Size of signal constellation
k = log2(M); % Number of bits per symbol
n = 30000; % Number of bits to process
numSamplesPerSymbol = 1; % Oversampling factor
```

Create a binary data stream as a column vector.

```
rng default % Use default random number generator
dataIn = randi([0 1],n,1); % Generate vector of binary data
```

Plot the first 40 bits in a stem plot.

```
stem(dataIn(1:40), 'filled');
title('Random Bits');
xlabel('Bit Index');
ylabel('Binary Value');
```



Convert the Binary Signal to an Integer-Valued Signal

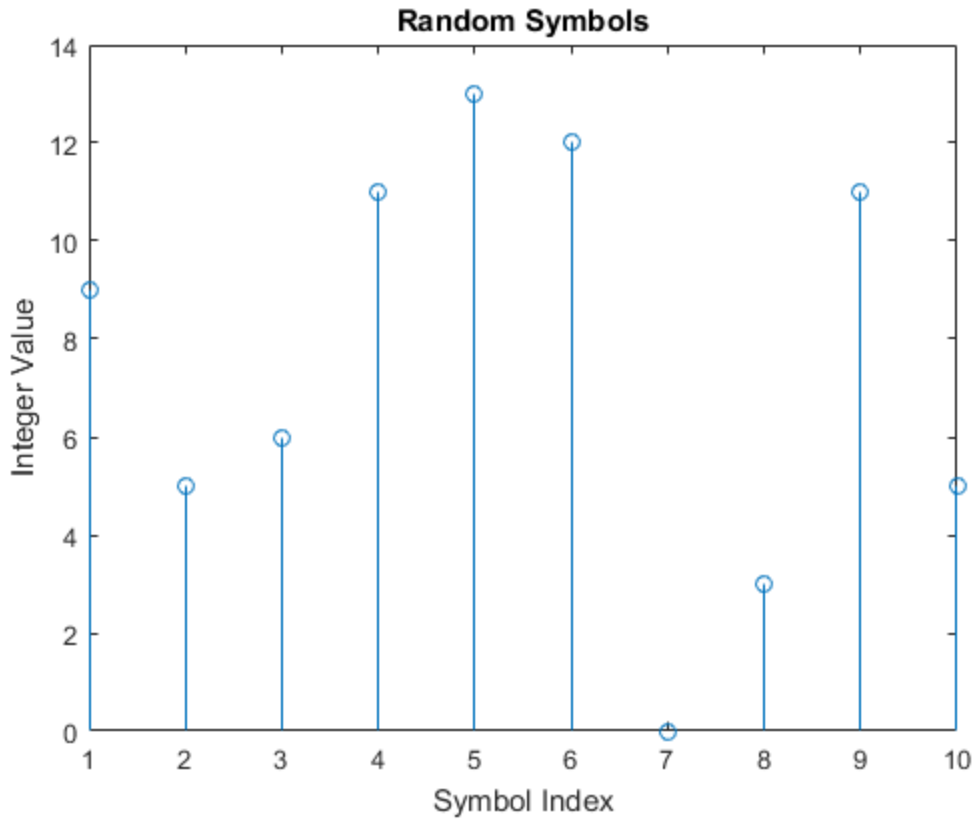
The `qammod` function implements a rectangular, M -ary QAM modulator, M being 16 in this example. The default configuration is such that the object receives integers between 0 and 15 rather than 4-tuples of bits. In this example, we preprocess the binary data stream `dataIn` before using the `qammod` function. In particular, the `bi2de` function is used to convert each 4-tuple to a corresponding integer.

Perform a bit-to-symbol mapping.

```
dataInMatrix = reshape(dataIn,length(dataIn)/4,4); % Reshape data into binary 4-tuples
dataSymbolsIn = bi2de(dataInMatrix); % Convert to integers
```

Plot the first 10 symbols in a stem plot.

```
figure; % Create new figure window.  
stem(dataSymbolsIn(1:10));  
title('Random Symbols');  
xlabel('Symbol Index');  
ylabel('Integer Value');
```



Modulate using 16-QAM

Having generated the `dataSymbolsIn` column vector, use the `qammod` function to apply 16-QAM modulation for both binary and Gray coded bit-to-symbol mappings. Recall that `M` is 16, the alphabet size.

Apply modulation.


```
dataMod = qammod(dataSymbolsIn,M,0);           % Binary coding, phase offset = 0
dataModG = qammod(dataSymbolsIn,M,0,'gray'); % Gray coding, phase offset = 0
```

The results are complex column vectors whose values are elements of the 16-QAM signal constellation. A later step in this example will plot the constellation diagram.

To learn more about modulation functions, see “Digital Modulation”. Also, note that the `qammod` function does not apply pulse shaping. To extend this example to use pulse shaping, see “Pulse Shaping Using a Raised Cosine Filter” on page 2-21. For an example that uses Gray coding with PSK modulation, see Gray Coded 8-PSK.

Add White Gaussian Noise

The ratio of bit energy to noise power spectral density, E_b/N_0 , is arbitrarily set to 10 dB. From that value, the signal-to-noise ratio (SNR) can be determined. Given the SNR, the modulated signal, `dataMod`, is passed through the channel by using the `awgn` function.

Note: The `numSamplesPerSymbol` variable is not significant in this example but will make it easier to extend the example later to use pulse shaping.

Calculate the SNR when the channel has an $E_b/N_0 = 10$ dB.

```
EbNo = 10;
snr = EbNo + 10*log10(k) - 10*log10(numSamplesPerSymbol);
```

Pass the signal through the AWGN channel for both the binary and Gray coded symbol mappings.

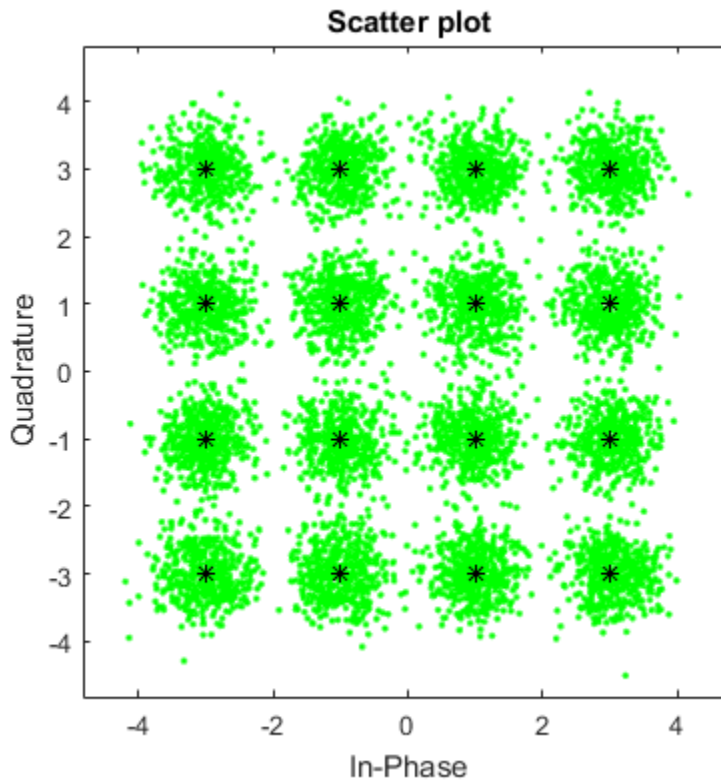
```
receivedSignal = awgn(dataMod,snr,'measured');
receivedSignalG = awgn(dataModG,snr,'measured');
```

Create a Constellation Diagram

The `scatterplot` function is used to display the in-phase and quadrature components of the modulated signal, `dataMod`, and its received, noisy version, `receivedSignal`. By looking at the resultant diagram, the effects of AWGN are readily observable.

Use the `scatterplot` function to show the constellation diagram.

```
sPlotFig = scatterplot(receivedSignal,1,0,'g. ');
hold on
scatterplot(dataMod,1,0,'k*',sPlotFig)
```



Demodulate 16-QAM

The `qamdemod` function is used to demodulate the received data and output integer-valued data symbols.

Demodulate the received signals using the `qamdemod` function.

```
dataSymbolsOut = qamdemod(receivedSignal,M);  
dataSymbolsOutG = qamdemod(receivedSignalG,M,0,'gray');
```

Convert the Integer-Valued Signal to a Binary Signal

The `de2bi` function is used to convert the data symbols from the QAM demodulator, `dataSymbolsOut`, into a binary matrix, `dataOutMatrix` with dimensions of N_{sym} -by-

$N_{\text{bits/sym}}$, where N_{sym} is the total number of QAM symbols and $N_{\text{bits/sym}}$ is the number of bits per symbol, four in this case. The matrix is then converted into a column vector whose length is equal to the number of input bits, 30,000. The process is repeated for the Gray coded data symbols, `dataSymbolsOutG`.

Reverse the bit-to-symbol mapping performed earlier.

```
dataOutMatrix = de2bi(dataSymbolsOut,k);
dataOut = dataOutMatrix(:);           % Return data in column vector
dataOutMatrixG = de2bi(dataSymbolsOutG,k);
dataOutG = dataOutMatrixG(:);        % Return data in column vector
```

Compute the System BER

The function `biterr` is used to calculate the bit error statistics from the original binary data stream, `dataIn`, and the received data streams, `dataOut` and `dataOutG`.

Use the error rate function to compute the error statistics and use `fprintf` to display the results.

```
[numErrors,ber] = biterr(dataIn,dataOut);
fprintf('\nThe binary coding bit error rate = %5.2e, based on %d errors\n', ...
        ber,numErrors)
[numErrorsG,berG] = biterr(dataIn,dataOutG);
fprintf('\nThe Gray coding bit error rate = %5.2e, based on %d errors\n', ...
        berG,numErrorsG)
```

The binary coding bit error rate = 2.40e-03, based on 72 errors

The Gray coding bit error rate = 1.33e-03, based on 40 errors

Observe that Gray coding significantly reduces the bit error rate.

Plot Signal Constellations

The example in the previous section, “Modulate a Random Signal” on page 2-11, created a scatter plot from the modulated signal. Although the plot showed the points in the QAM constellation, the plot did not indicate which integers of the modulator are mapped to a given constellation point. This section illustrates two possible mappings: 1) binary coding, and 2) Gray coding. It was previously demonstrated that Gray coding provides superior bit error rate performance.

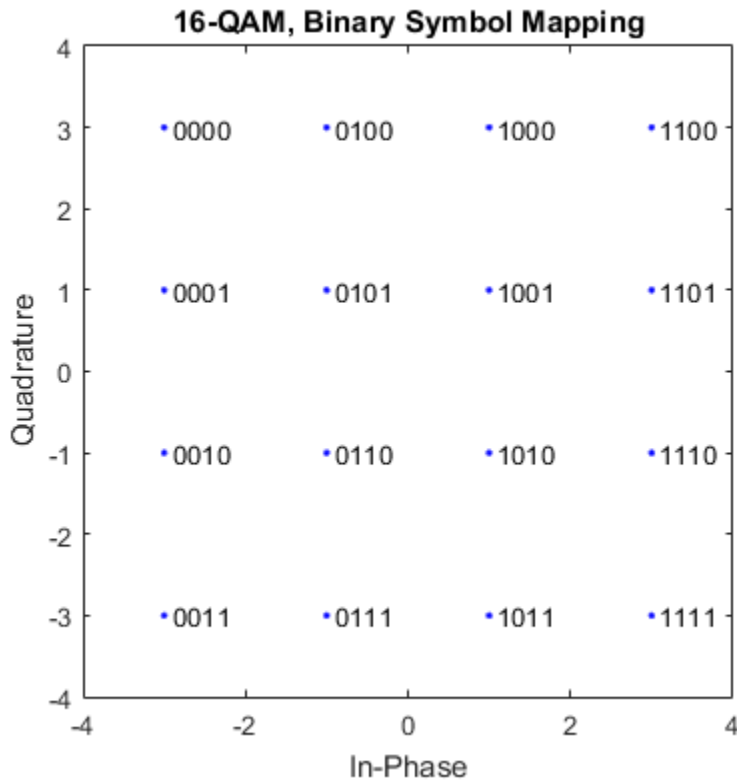
Binary Symbol Mapping for 16-QAM Constellation

Apply 16-QAM modulation to all possible input values using the default symbol mapping, binary.

```
M = 16; % Modulation order
x = (0:15)'; % Integer input
y1 = qammod(x, 16, 0); % 16-QAM output, phase offset = 0
```

Use the `scatterplot` function to plot the constellation diagram and annotate it with binary representations of the constellation points.

```
scatterplot(y1)
text(real(y1)+0.1, imag(y1), dec2bin(x))
title('16-QAM, Binary Symbol Mapping')
axis([-4 4 -4 4])
```



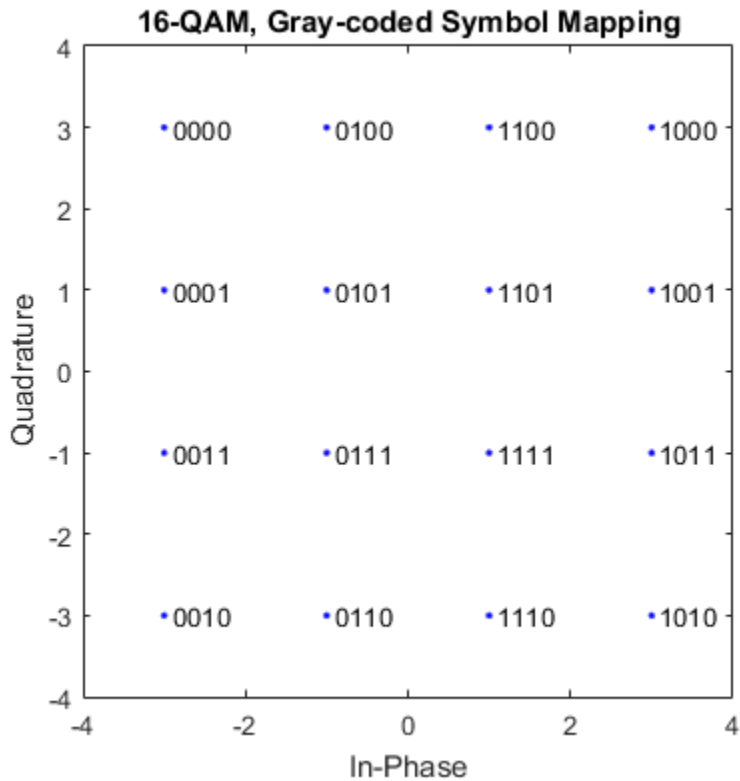
Gray-coded Symbol Mapping for 16-QAM Constellation

Apply 16-QAM modulation to all possible input values using Gray-coded symbol mapping.

```
y2 = qammod(x, 16, 0, 'gray'); % 16-QAM output, phase offset = 0, Gray-coded
```

Use the `scatterplot` function to plot the constellation diagram and annotate it with binary representations of the constellation points.

```
scatterplot(y2)
text(real(y2)+0.1, imag(y2), dec2bin(x))
title('16-QAM, Gray-coded Symbol Mapping')
axis([-4 4 -4 4])
```



Examine the Plots

In the binary mapping plot, notice that symbols 1 (0 0 0 1) and 2 (0 0 1 0) correspond to adjacent constellation points on the left side of the diagram. The binary representations of these integers differ by two bits unlike the Gray-coded signal constellation in which each point differs by only one bit from its direct neighbors, which leads to better BER performance.

Pulse Shaping Using a Raised Cosine Filter

The “Modulate a Random Signal” on page 2-11 example was modified to employ a pair of square-root raised cosine (RRC) filters to perform pulse shaping and matched filtering. The filters are created by the `rcosdesign` function. In “Error Correction using a Convolutional Code” on page 2-28, this example is extended by introducing forward error correction (FEC) to improve BER performance.

To create a BER simulation, a modulator, demodulator, communication channel, and error counter functions must be used and certain key parameters must be specified. In this case, 16-QAM modulation is used in an AWGN channel.

Establish Simulation Framework

Set the simulation parameters.

```
M = 16;           % Size of signal constellation
k = log2(M);     % Number of bits per symbol
numBits = 3e5;   % Number of bits to process
numSamplesPerSymbol = 4; % Oversampling factor
```

Create Raised Cosine Filter

Set the square-root, raised cosine filter parameters.

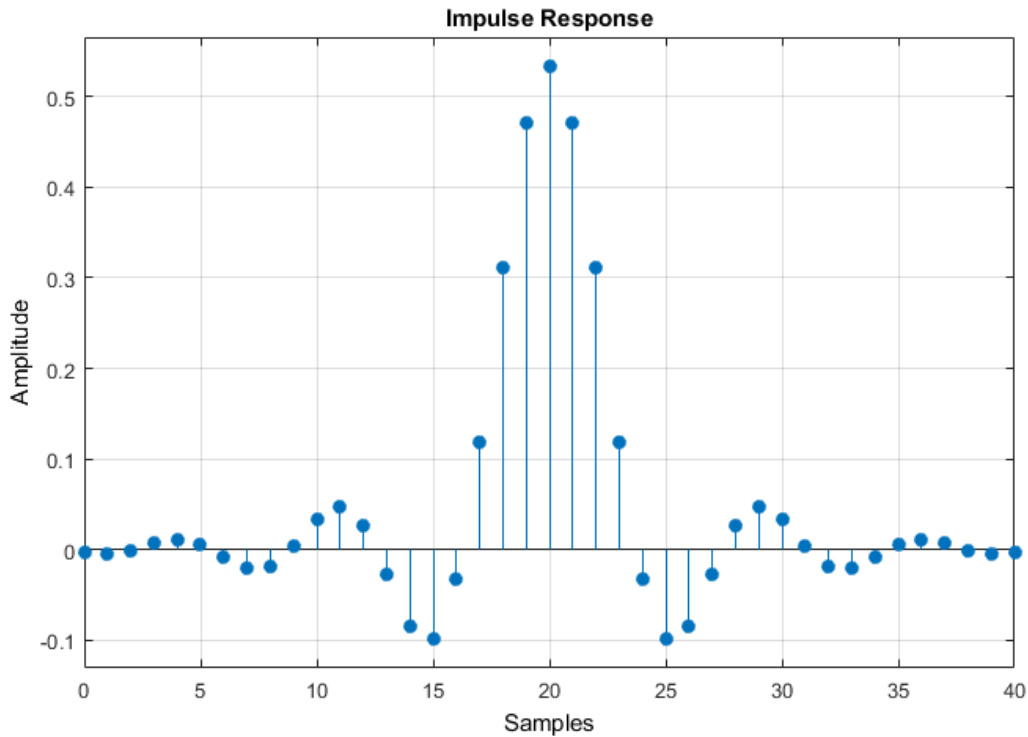
```
span = 10;        % Filter span in symbols
rolloff = 0.25;  % Rolloff factor of filter
```

Create a square-root, raised cosine filter using the `rcosdesign` function.

```
rrcFilter = rcosdesign(rolloff, span, numSamplesPerSymbol);
```

Display the RRC filter impulse response using the `fvtool` function.

```
fvtool(rrcFilter, 'Analysis', 'Impulse')
```



BER Simulation

Use the `randi` function to generate random binary data. The `rng` function should be set to its default state so that the example produces repeatable results.

```
rng default % Use default random number generator
dataIn = randi([0 1], numBits, 1); % Generate vector of binary data
```

Reshape the input vector into a matrix of 4-bit binary data, which is then converted into integer symbols.

```
dataInMatrix = reshape(dataIn, length(dataIn)/k, k); % Reshape data into binary 4-tuple
dataSymbolsIn = bi2de(dataInMatrix); % Convert to integers
```

Apply 16-QAM modulation using `qammod`.

```
dataMod = qammod(dataSymbolsIn, M);
```


Using the `upfirdn` function, upsample and apply the square-root, raised cosine filter.

```
txSignal = upfirdn(dataMod, rrcFilter, numSamplesPerSymbol, 1);
```

The `upfirdn` function upsamples the modulated signal, `dataMod`, by a factor of `numSamplesPerSymbol`, pads the upsampled signal with zeros at the end to flush the filter and then applies the filter.

Set the E_b/N_0 to 10 dB and convert the SNR given the number of bits per symbol, k , and the number of samples per symbol.

```
EbNo = 10;
snr = EbNo + 10*log10(k) - 10*log10(numSamplesPerSymbol);
```

Pass the filtered signal through an AWGN channel.

```
rxSignal = awgn(txSignal, snr, 'measured');
```

Filter the received signal using the square-root, raised cosine filter and remove a portion of the signal to account for the filter delay in order to make a meaningful BER comparison.

```
rxFiltSignal = upfirdn(rxSignal, rrcFilter, 1, numSamplesPerSymbol); % Downsample and filter
rxFiltSignal = rxFiltSignal(span+1:end-span); % Account for delay
```

These functions apply the same square-root raised cosine filter that the transmitter used earlier, and then downsample the result by a factor of `nSamplesPerSymbol`. The last command removes the first `Span` symbols and the last `Span` symbols in the decimated signal because they represent the cumulative delay of the two filtering operations. Now `rxFiltSignal`, which is the input to the demodulator, and `dataSymbolsOut`, which is the output from the modulator, have the same vector size. In the part of the example that computes the bit error rate, it is required to compare vectors that have the same size.

Apply 16-QAM demodulation to the received, filtered signal.

```
dataSymbolsOut = qamdemod(rxFiltSignal, M);
```

Using the `de2bi` function, convert the incoming integer symbols into binary data.

```
dataOutMatrix = de2bi(dataSymbolsOut, k);
dataOut = dataOutMatrix(:); % Return data in column vector
```

Apply the `biterr` function to determine the number of errors and the associated BER.

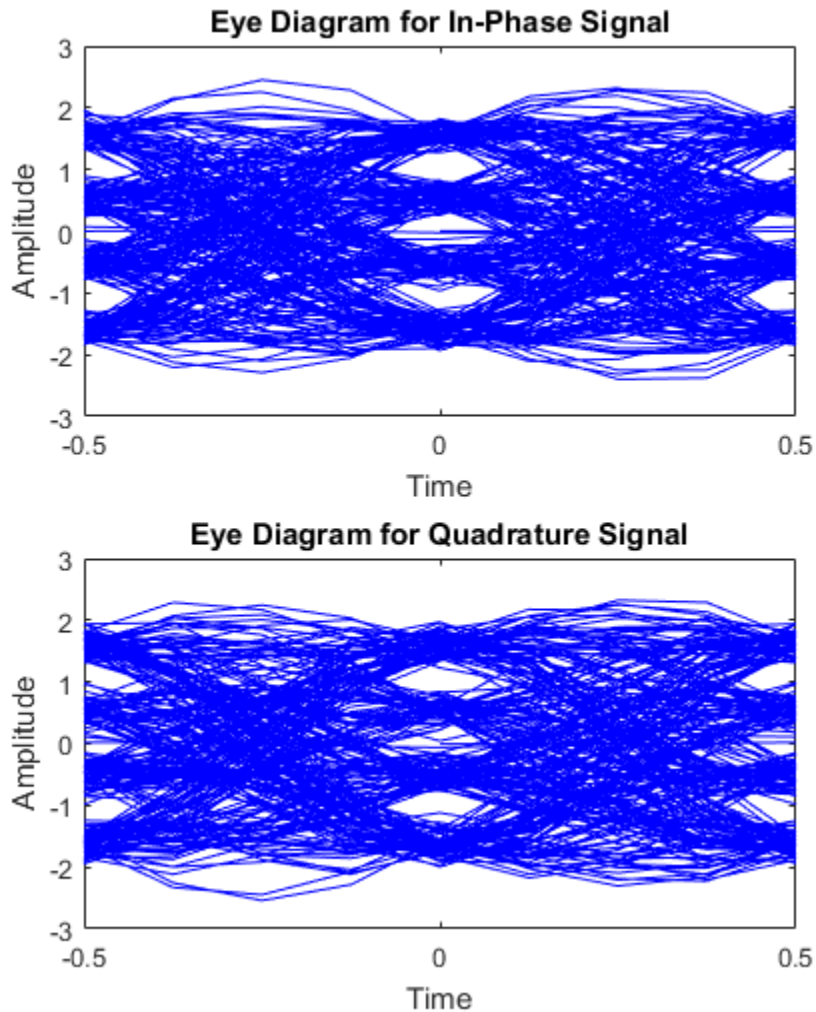
```
[numErrors, ber] = biterr(dataIn, dataOut);  
fprintf('\nThe bit error rate = %5.2e, based on %d errors\n', ...  
        ber, numErrors)
```

The bit error rate = 2.42e-03, based on 727 errors

Visualization of Filter Effects

Create an eye diagram for a portion of the filtered signal.

```
eyediagram(txSignal(1:2000), numSamplesPerSymbol*2);
```

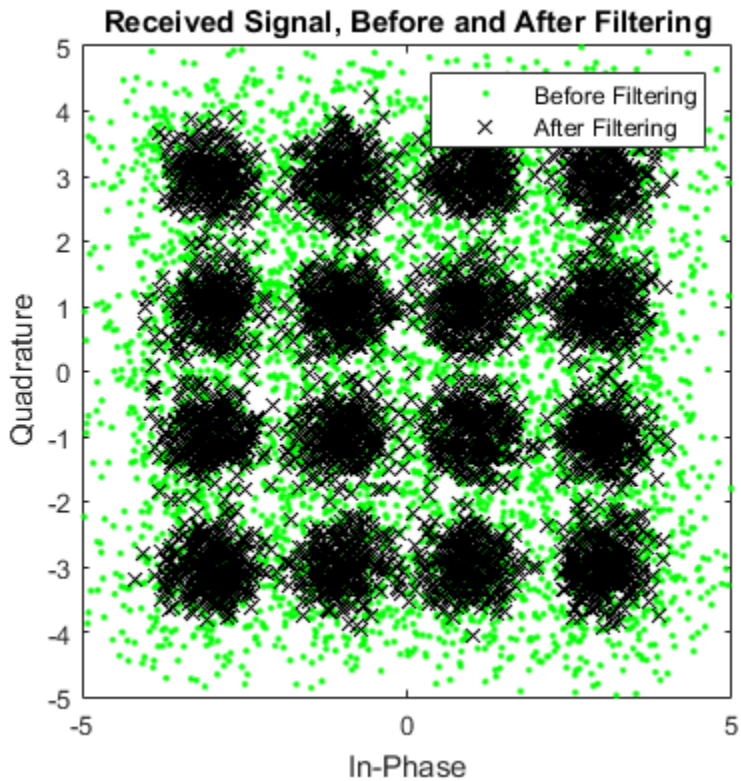


The `eyediagram` function creates an eye diagram for part of the filtered noiseless signal. This diagram illustrates the effect of the pulse shaping. Note that the signal shows significant intersymbol interference (ISI) because the filter is a square-root raised cosine filter, not a full raised cosine filter.

Created a scatter plot of the received signal before and after filtering.

```
h = scatterplot(sqrt(numSamplesPerSymbol)*...  
    rxSignal(1:numSamplesPerSymbol*5e3),...  
    numSamplesPerSymbol,0,'g.');
```

hold on;
scatterplot(rxFiltSignal(1:5e3),1,0,'kx',h);
title('Received Signal, Before and After Filtering');
legend('Before Filtering','After Filtering');
axis([-5 5 -5 5]); % Set axis ranges
hold off;



Notice that the first `scatterplot` command scales `rxSignal` by `sqrt(numSamplesPerSymbol)` when plotting. This is because the filtering operation changes the signal's power.

Error Correction using a Convolutional Code

Building upon the “Pulse Shaping Using a Raised Cosine Filter” on page 2-21 example, this example shows how bit error rate performance improves with the addition of forward error correction, FEC, coding.

Establish Simulation Framework

To create the simulation, a modulator, demodulator, raised cosine filter pair, communication channel, and error counter functions are used and certain key parameters are specified. In this case, a 16-QAM modulation scheme with raised cosine filtering is used in an AWGN channel. With the exception of the number of bits, the specified parameters are identical to those used in the previous example.

Set the simulation variables. The number of bits is increased from the previous example so that the bit error rate may be estimated more accurately.

```
M = 16; % Size of signal constellation
k = log2(M); % Number of bits per symbol
numBits = 100000; % Number of bits to process
numSamplesPerSymbol = 4; % Oversampling factor
```

Generate Random Data

Use the `randi` function to generate random, binary data once the `rng` function has been called. When set to its default value, the `rng` function ensures that the results from this example are repeatable.

```
rng default % Use default random number generator
dataIn = randi([0 1], numBits, 1); % Generate vector of binary data
```

Convolutional Encoding

The performance of the “Pulse Shaping Using a Raised Cosine Filter” on page 2-21 example can be significantly improved upon by employing forward error correction. In this example, convolutional coding is applied to the transmitted bit stream in order to correct errors arising from the noisy channel. Because it is often implemented in real systems, the Viterbi algorithm is used to decode the received data. A hard decision algorithm is used, which means that the decoder interprets each input as either a “0” or a “1”.

Define a convolutional coding trellis for a rate 2/3 code. The `poly2trellis` function defines the trellis that represents the convolutional code that `convenc` uses for encoding

the binary vector, `dataIn`. The two input arguments of the `poly2trellis` function indicate the code's constraint length and generator polynomials, respectively.

```
tPoly = poly2trellis([5 4],[23 35 0; 0 5 13]);
codeRate = 2/3;
```

Encode the input data using the previously created trellis.

```
dataEnc = convenc(dataIn, tPoly);
```

Modulate Data

The encoded binary data is converted into an integer format so that 16-QAM modulation can be applied.

Reshape the input vector into a matrix of 4-bit binary data, which is then converted into integer symbols.

```
dataEncMatrix = reshape(dataEnc, ...
    length(dataEnc)/k, k);           % Reshape data into binary 4-tuples
dataSymbolsIn = bi2de(dataEncMatrix); % Convert to integers
```

Apply 16-QAM modulation.

```
dataMod = qammod(dataSymbolsIn, M);
```

Raised Cosine Filtering

As in the “Pulse Shaping Using a Raised Cosine Filter” on page 2-21 example, RRC filtering is applied to the modulated signal before transmission. The example makes use of the `rcosdesign` function to create the filter and the `upfirdn` function to filter the data.

Specify the filter span and rolloff factor for the square-root, raised cosine filter.

```
span = 10;           % Filter span in symbols
rolloff = 0.25;     % Roloff factor of filter
```

Create the filter using the `rcosdesign` function.

```
rrcFilter = rcosdesign(rolloff, span, numSamplesPerSymbol);
```

Using the `upfirdn` function, upsample and apply the square-root, raised cosine filter.

```
txSignal = upfirdn(dataMod, rrcFilter, numSamplesPerSymbol, 1);
```

AWGN Channel

Calculate the signal-to-noise ratio, SNR, based on the input E_b/N_0 , the number of samples per symbol, and the code rate. Converting from E_b/N_0 to SNR requires one to account for the number of information bits per symbol. In the previous example, each symbol corresponded to k bits. Now, each symbol corresponds to $k \cdot \text{codeRate}$ information bits. More concretely, three symbols correspond to 12 coded bits in 16-QAM, which correspond to 8 uncoded (information) bits.

```
EbNo = 10;  
snr = EbNo + 10*log10(k*codeRate) - 10*log10(numSamplesPerSymbol);
```

Pass the filtered signal through an AWGN channel.

```
rxSignal = awgn(txSignal, snr, 'measured');
```

Receive and Demodulate Signal

Filter the received signal using the RRC filter and remove a portion of the signal to account for the filter delay in order to make a meaningful BER comparison.

```
rxFiltSignal = upfirdn(rxSignal, rrcFilter, 1, numSamplesPerSymbol); % Downsample and filter  
rxFiltSignal = rxFiltSignal(span+1:end-span); % Account for delay
```

Demodulate the received, filtered signal using the `qamdemod` function.

```
dataSymbolsOut = qamdemod(rxFiltSignal, M);
```

Viterbi Decoding

Use the `de2bi` function to convert the incoming integer symbols into bits.

```
dataOutMatrix = de2bi(dataSymbolsOut, k);  
codedDataOut = dataOutMatrix(:); % Return data in column vector
```

Decode the convolutionally encoded data with a Viterbi decoder. The syntax for the `vitdec` function instructs it to use hard decisions. The 'cont' argument instructs it to use a mode designed for maintaining continuity when the function is repeatedly invoked (as in a loop). Although this example does not use a loop, the 'cont' mode is used for the purpose of illustrating how to compensate for the delay in this decoding operation.

```
traceBack = 16; % Traceback length for decoding  
numCodeWords = floor(length(codedDataOut)*2/3); % Number of complete codewords  
dataOut = vitdec(codedDataOut(1:numCodeWords*3/2), ... % Decode data
```



```
tPoly,traceBack,'cont','hard');
```

BER Calculation

Using the `biterr` function, compare `dataIn` and `dataOut` to obtain the number of errors and the bit error rate while taking the decoding delay into account. The continuous operation mode of the Viterbi decoder incurs a delay whose duration in bits equals the traceback length, `traceBack`, times the number of input streams at the encoder. For this rate 2/3 code, the encoder has two input streams, so the delay is $2 \times \text{traceBack}$ bits. As a result, the first $2 \times \text{traceBack}$ bits in the decoded vector, `dataOut`, are zeros. When computing the bit error rate, the first $2 \times \text{traceBack}$ bits in `dataOut` and the last $2 \times \text{traceBack}$ bits in the original vector, `dataIn`, are discarded. Without delay compensation, the BER computation is meaningless.

```
decDelay = 2*traceBack; % Decoder delay, in bits
[numErrors, ber] = ...
    biterr(dataIn(1:end-decDelay),dataOut(decDelay+1:end));

fprintf('\nThe bit error rate = %5.2e, based on %d errors\n', ...
        ber, numErrors)
```

```
The bit error rate = 6.90e-04, based on 69 errors
```

It can be seen that for the same E_b/N_0 of 10 dB, the number of errors when using FEC is reduced as the BER improves from 2.0×10^{-3} to 6.9×10^{-4} .

More About Delays

The decoding operation in this example incurs a delay, which means that the output of the decoder lags the input. Timing information does not appear explicitly in the example, and the duration of the delay depends on the specific operations being performed. Delays occur in various communications-related operations, including convolutional decoding, convolutional interleaving/deinterleaving, equalization, and filtering. To find out the duration of the delay caused by specific functions or operations, refer to the specific documentation for those functions or operations. For example:

- The `vitdec` reference page
- “Delays of Convolutional Interleavers”
- “Fading Channels”

Iterative Design Workflow for Communication Systems

In this section...
“Simulate a basic communications system” on page 2-33
“Introduce convolutional coding and hard-decision Viterbi decoding” on page 2-37
“Improve results using soft-decision decoding” on page 2-41
“Use turbo coding to improve BER performance” on page 2-46
“Apply a Rayleigh channel model” on page 2-49
“Use OFDM-based equalization to correct multipath fading” on page 2-54
“Use multiple antennas to further improve system performance” on page 2-57
“Accelerate the simulation using MATLAB Coder” on page 2-61

This example illustrates a design workflow that represents the iterative steps for creating a wireless communications system with the Communications System Toolbox. Because Communications System Toolbox supports both MATLAB and Simulink, this example showcases design paths using MATLAB code and Simulink blocks. As you progress through the workflow, you may follow the design path for MATLAB, for Simulink, or for both products.

The workflow begins with a simple communications system and performs bit error rate (BER) simulations to gauge system performance. BER simulations are based on simulating a communications system with a given signal-to-noise ratio (E_b/N_0), and then calculating the corresponding bit error rate measurement to determine the number of errors in the transmitted signal. The lower the BER measurement at a given signal-to-noise ratio, the better the system performance.

This workflow starts with a simple communications system, and iteratively adds the algorithmic components necessary to build a more complicated system. These additional components include:

- Convolutional Encoding and Viterbi Decoding
- Turbo Coding
- Multipath Fading Channels
- OFDM-Based Transmission
- Multiple-Antenna Techniques

As you add components to the system, the workflow includes bit error calculations so that you can progressively examine system performance. For some components, theoretical or performance benchmarks are available. In these cases, the workflow shows both the theoretical and measured performance metric.

Simulate a basic communications system

This workflow starts with a simple QPSK modulator system that transmits a signal through an AWGN channel and calculates the bit error rate to evaluate system performance.

In MATLAB

- 1 CD to the following MATLAB folder:

```
matlab\help\toolbox\comm\examples
```

- 2 Type `edit doc_design_iteration_basic_m` at the MATLAB command line.

MATLAB opens a file you will use in this example. Notice that this code employs four System objects from Communications System Toolbox: `comm.PSKModulator`, `comm.AWGN`, `comm.PSKDemodulator`, and `comm.ErrorRate`. For each `EbNo` value, the code runs in a while loop until either the specified number of errors are observed or the maximum number of bits are processed. Notice that the code executes each System object by calling the `step` method. The code outputs BER, defined as the ratio of the observed number of errors per number of bits processed. The subsequent MATLAB functions that this example uses have a similar structure.

- 3 Type `bertool` at the MATLAB command line to open the Bit Error Rate Analysis Tool.
- 4 When the BERTool application appears, click the **Theoretical** tab.

The first plot that you will generate is a theoretical curve.

- 5 Enter `0:9` for the **EbNo range**.

`EbNo` is the ratio of noise power energy per bit. The higher the value, the better the system performance. This simulation will run using different values for the ratio, between 0 and 9.

- 6 Select 4 for **Modulation order**.

The modulation order defines the number of symbols to transmit. Here, each symbol is made up of two bits.

- 7 Click **Plot**.

- The BERTool application generates the theoretical BER curve.
- 8 Click the **Monte Carlo** tab.

Monte Carlo techniques use random sampling to compute data. Therefore, the plot for the second simulation uses random sampling.

- 9 Enter 0:9 for the **EbNo range**.
- 10 Enter ber for the **BER variable name**.
- 11 Enter 200 for the **Number of errors**.

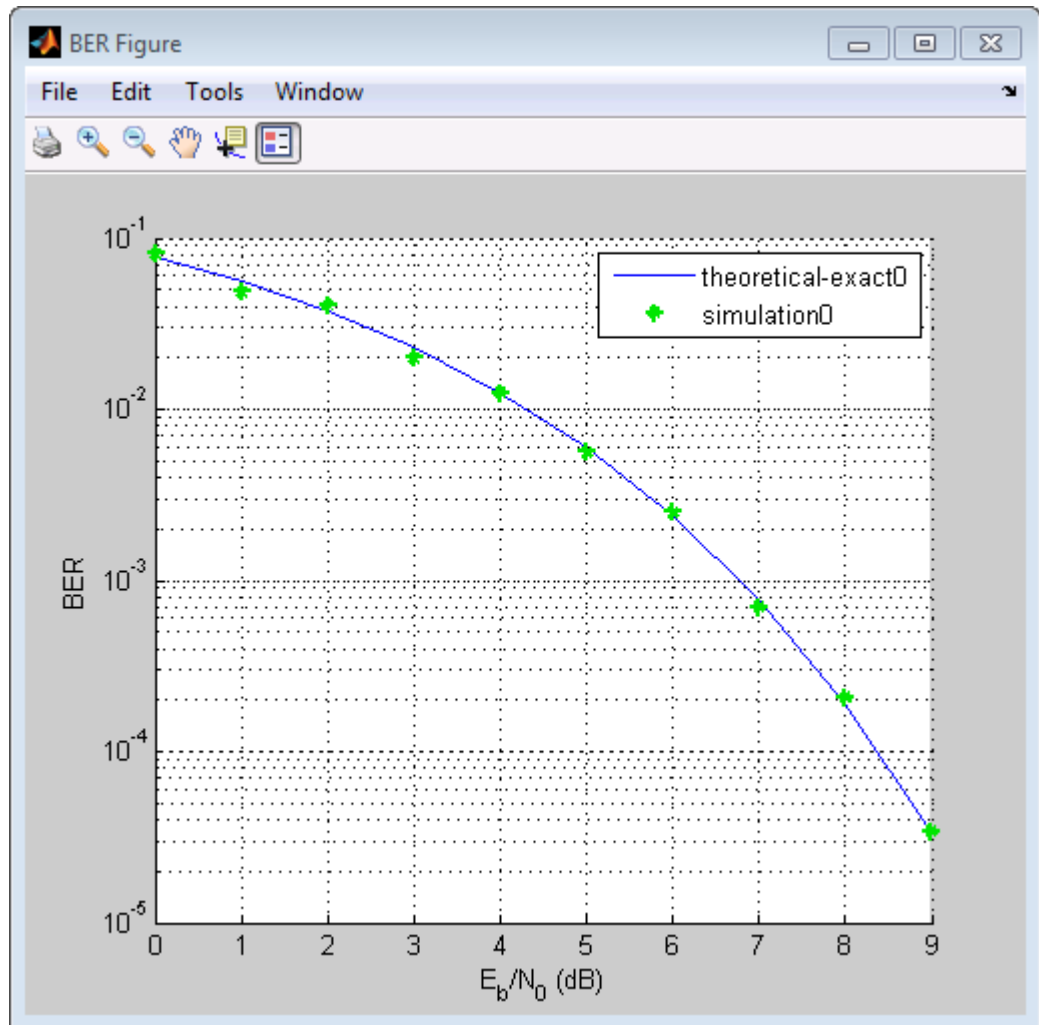
The **Number of errors** is one of the stop criteria for the simulation.

- 12 Enter 1e7 for the **Number of bits**.

The **Number of bits** is also a stop criteria for the simulation. The simulation stops when it transmits the number of bits you specify for this parameter. In this example, the simulation either stops when it transmits 10 million bits or when it detects 200 errors.

- 13 Click the **Browse** button.
- 14 Navigate to `matlab/help/toolbox/comm/examples`, and select `doc_design_iteration_basic_m.m`.
- 15 Click **Run**.

BERTool runs the simulation and generates simulation points along the BER curve. Compare the simulation BER curve with the theoretical BER curve.



Every function with two output variables and three input variables can be called using BERTool. This is how you interpret the three input variables:

- The first variable is a scalar number that corresponds to E_b/N_0 .
- The second variable is the stopping criterion based on the maximum number of errors to observe before stopping the simulation.

- The third variable is the stop criterion based on the maximum number of bits to process before observe before stopping the simulation.

In Simulink

- 1 Type `bertool` at the MATLAB command line to open the Bit Error Rate Analysis Tool.
- 2 When the BERTool application appears, click the **Theoretical** tab.
- 3 Enter `0:9` for the **EbNo range**.

EbNo is the ratio of noise power energy per bit. The higher the value, the better the system performance. This simulation will run using different values for the ratio, between 0 and 9.

- 4 Select 4 for **Modulation order**.

The modulation order defines the number of symbols to transmit. Here, each symbol is made up of two bits.

- 5 Click **Plot**.

The BERTool application generates the theoretical BER curve.

- 6 Click the **Monte Carlo** tab.
- 7 Enter `0:9` or the **EbNo range**.
- 8 Enter `ber` for the **BER variable name**.
- 9 Enter 200 for the **Number of errors**.

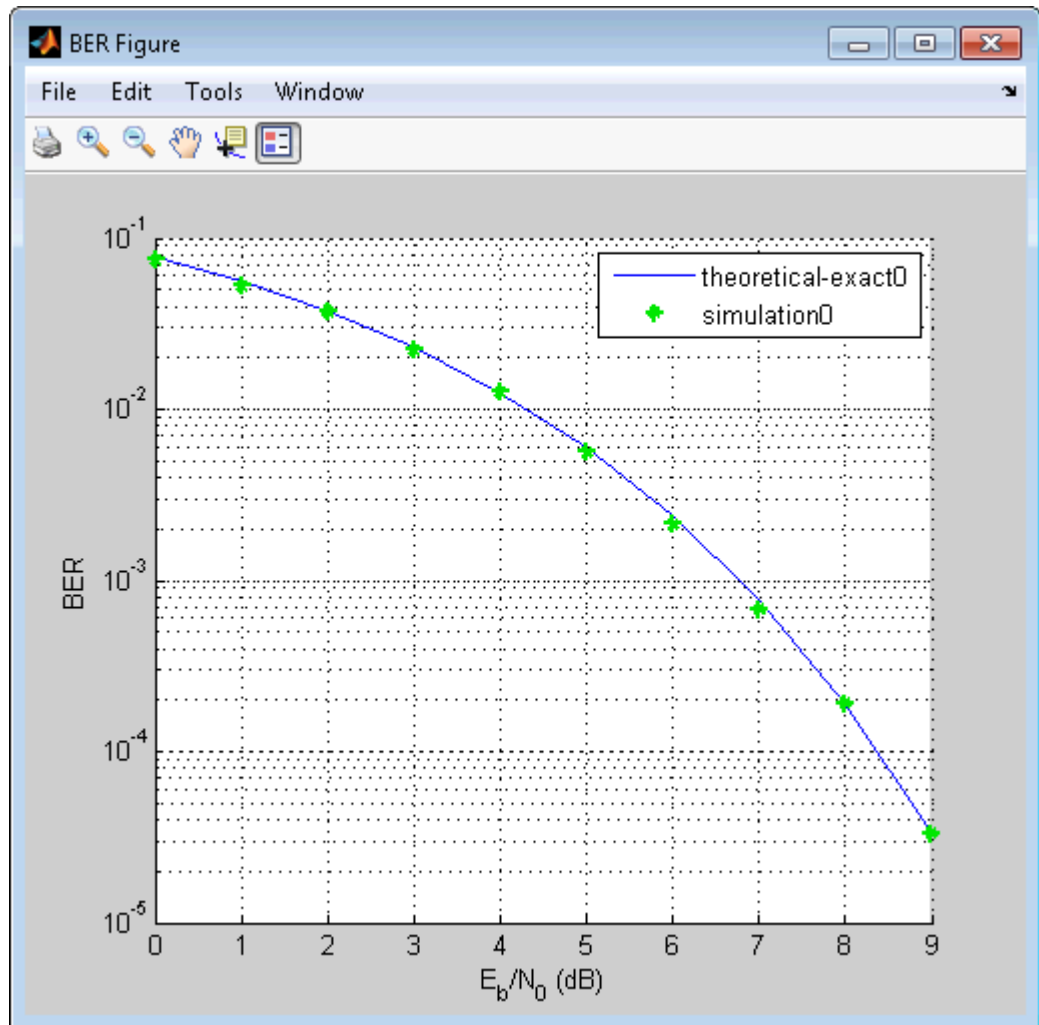
The **Number of errors** is one of the stop criteria for the simulation. The simulation stops when it reaches either the **Number of errors** or the **Number of bits**.

- 10 Enter `1e7` for the **Number of bits**.

The **Number of bits** is also a stop criteria for the simulation. The simulation stops when it transmits the number of bits you specify for this parameter or when it reaches the **Number of errors**. In this example, the simulation either stops when it transmits 10 million bits or when it detects 200 errors.

- 11 Click the **Browse** button, select **All Files** for the **Files of type** field.
- 12 Navigate to `matlab/help/toolbox/comm/examples`, select `doc_design_iteration_basic.slx` and click **Run**.

BERTool runs the simulation and generates simulated points along the BER curve. Compare the simulation BER curve with the theoretical BER curve.



Introduce convolutional coding and hard-decision Viterbi decoding

Modify the basic communications model to include forward error correction. Adding forward error correction to the basic communications model improves system performance. In forward error correction, the transmitter sends redundant bits, along

with the message bits, through a wireless channel. When the receiver accepts the transmitted signal, it uses the redundancy bits to detect and correct errors that the channel may have introduced.

This section of the design workflow adds a convolutional encoder and a Viterbi decoder to the communication system. This communications system uses hard-decision Viterbi decoding. In hard-decision Viterbi decoding, the demodulator maps the received signal to bits, and then passes the bits to the Viterbi decoder for error correction.

In MATLAB

In this iteration of the design workflow, the MATLAB file you use starts from where the one in the previous section ended. This file adds two additional System objects to the communications system, `comm.ConvolutionalEncoder` and `comm.ViterbiDecoder`. The overall structure of the code doesn't change; it simply contains additional functionality.

- 1 Access the BERTool application.
- 2 Clear the **Plot** check boxes for the two plots BERTool generated in the previous step.
- 3 Click **Theoretical**.
- 4 Enter 0:7 for the **EbNo range**.
- 5 Select **Convolutional** for the **Channel Coding**.
- 6 Select **Hard** for the **Decision method**.

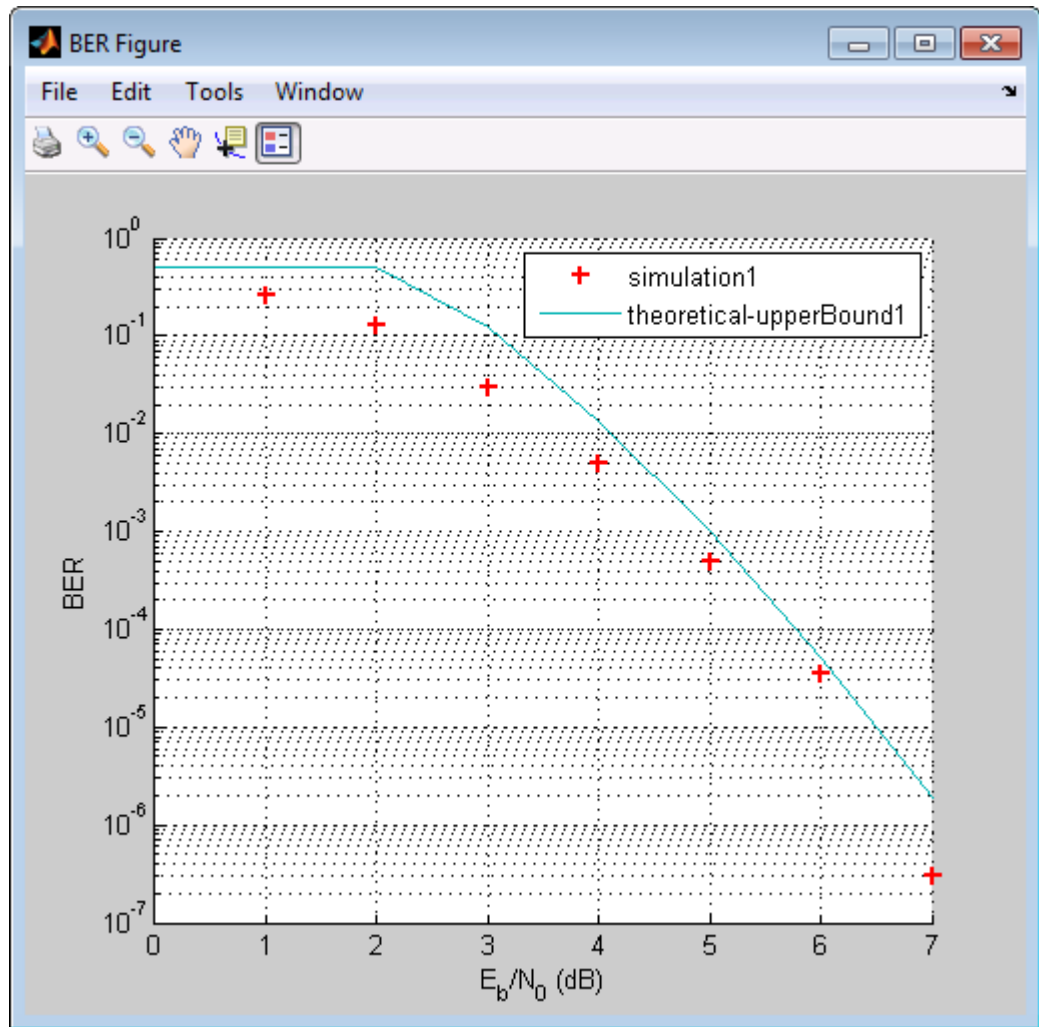
This example uses hard-decision Viterbi decoding. The demodulator maps the received signal to bits, and then passes the bits to the Viterbi decoder for error correction.

- 7 Click **Plot**.

The BERTool application generates the theoretical BER curve.

- 8 Click **Monte Carlo**.
- 9 Enter 0:7 for the **EbNo range**.
- 10 Enter 200 for the **Number of errors**.
- 11 Enter 1e7 for the **Number of bits**.
- 12 Click the **Browse** button.
- 13 Navigate to `matlab/help/toolbox/comm/examples`, select `doc_design_iteration_viterbi_m.m` and click **Open**.
- 14 Click **Run**.

BERTool runs the simulation and generates simulated points along the BER curve. Compare the simulation BER curve with the theoretical BER curve.



In Simulink

- 1 Access the BERTool application.
- 2 Click the **Theoretical** tab.
- 3 Enter 0:7 for the **EbNo range**.
- 4 Select **Convolutional** for the **Channel Coding**.

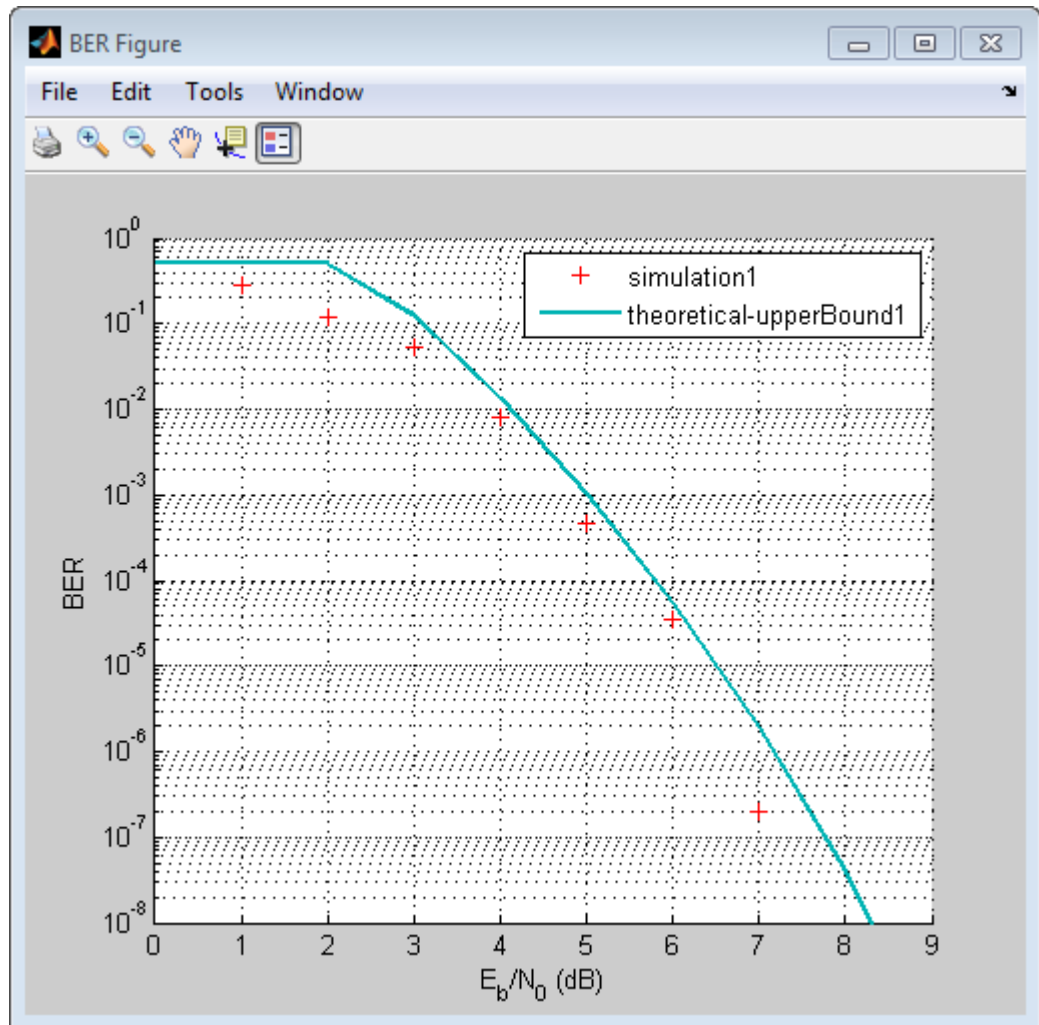
- 5 Select **Hard** for the **Decision method**.

This example uses hard-decision Viterbi decoding. The demodulator maps the received signal to bits, and then passes the bits to the Viterbi decoder for error correction.

- 6 Click **Plot**.

The BERTool application generates the theoretical BER curve.

- 7 Click the **Monte Carlo** tab.
- 8 Enter 0:7 for the **EbNo range**.
- 9 Enter 200 for the **Number of errors**.
- 10 Enter 1e7 for the **Number of bits**.
- 11 Click the **Browse** button, select **All Files** for the **Files of type** field.
- 12 Navigate to `matlab/help/toolbox/comm/examples`, select `doc_design_iteration_viterbi.slx` and click **Open**.
- 13 click **Run**.BERTool runs the simulation and generates simulated points along the BER curve. Compare the simulation BER curve with the theoretical BER curve.



Improve results using soft-decision decoding

Use soft-decision decoding to improve BER performance. The previous section of this workflow uses hard-decision demodulation and hard-decision Viterbi decoding – processes that map symbols to bits. This section of the workflow uses soft-decision

demodulation and soft-decision Viterbi decoding. In soft-decision demodulation, the received symbols are not mapped to bits. Instead, the symbols are mapped to log-likelihood ratios. When the Viterbi decoder processes log-likelihood ratios (LLR), the BER performance of the system improves.

In MATLAB

- 1 Access the BERTool application.
- 2 Clear the **Plot** check boxes for the two plots BERTool generated in the previous step.
- 3 Click **Theoretical**.
- 4 Enter **0:5** for the **EbNo range**.
- 5 Select **Soft** for the **Decision method**.

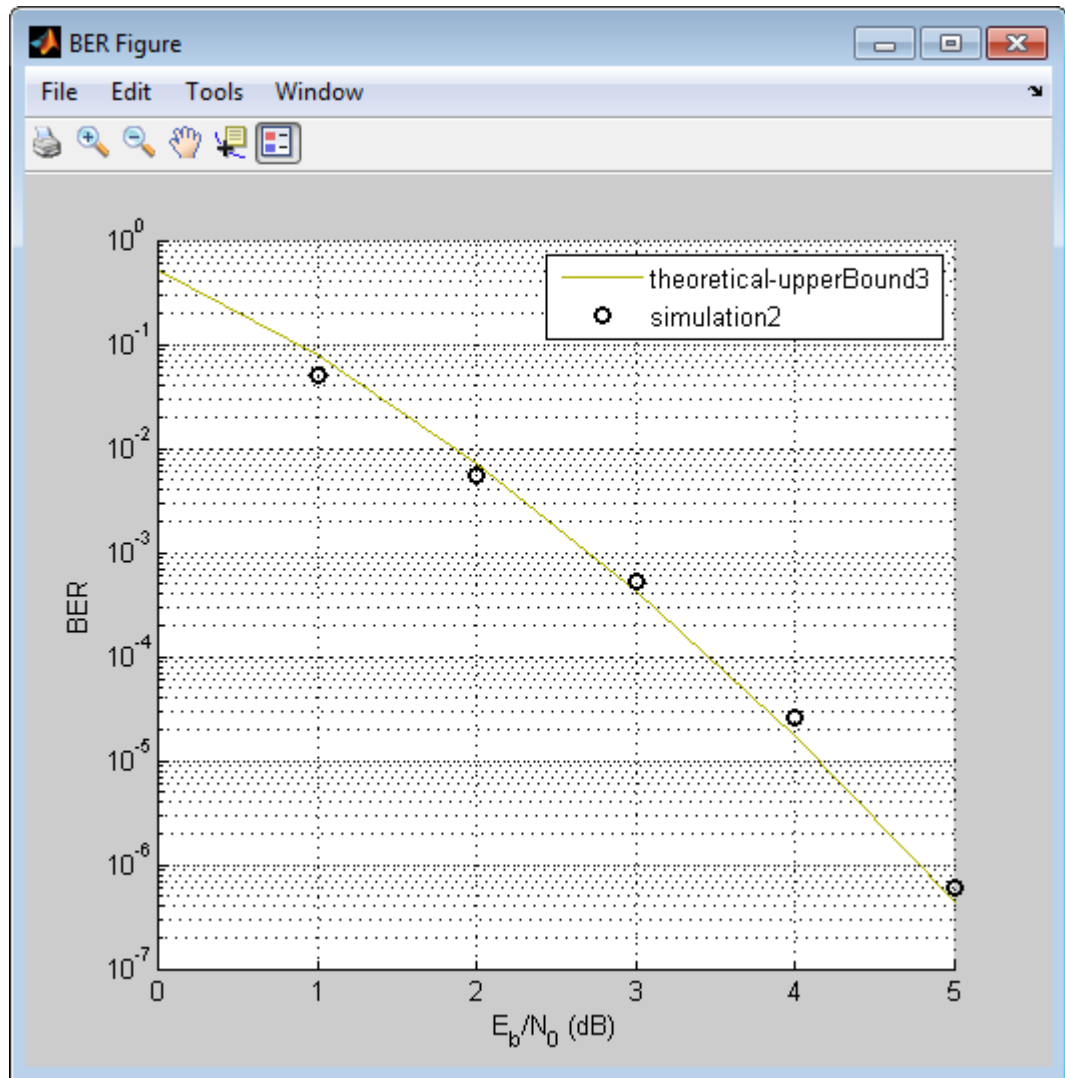
This example uses soft-decision Viterbi decoding. The demodulator maps the received signal to log likelihood ratios, improving BER performance results.

- 6 Click **Plot**.

The BERTool application generates the theoretical BER curve.

- 7 Click **Monte Carlo**.
- 8 Enter **0:5** for the **EbNo range**.
- 9 Enter **200** for the **Number of errors**.
- 10 Enter **1e7** for the **Number of bits**.
- 11 Click the **Browse** button.
- 12 Navigate to `matlab/help/toolbox/comm/examples`, select `doc_design_iteration_viterbi_soft_m.m` and click **Run**.

BERTool runs the simulation and generates the actual simulated points along the BER curve. Compare the simulation BER curve with the theoretical BER curve.



In Simulink

- 1 Access the BERTool application.
- 2 Clear the **Plot** check boxes for the two plots BERTool generated in the previous step.
- 3 Click **Theoretical**.

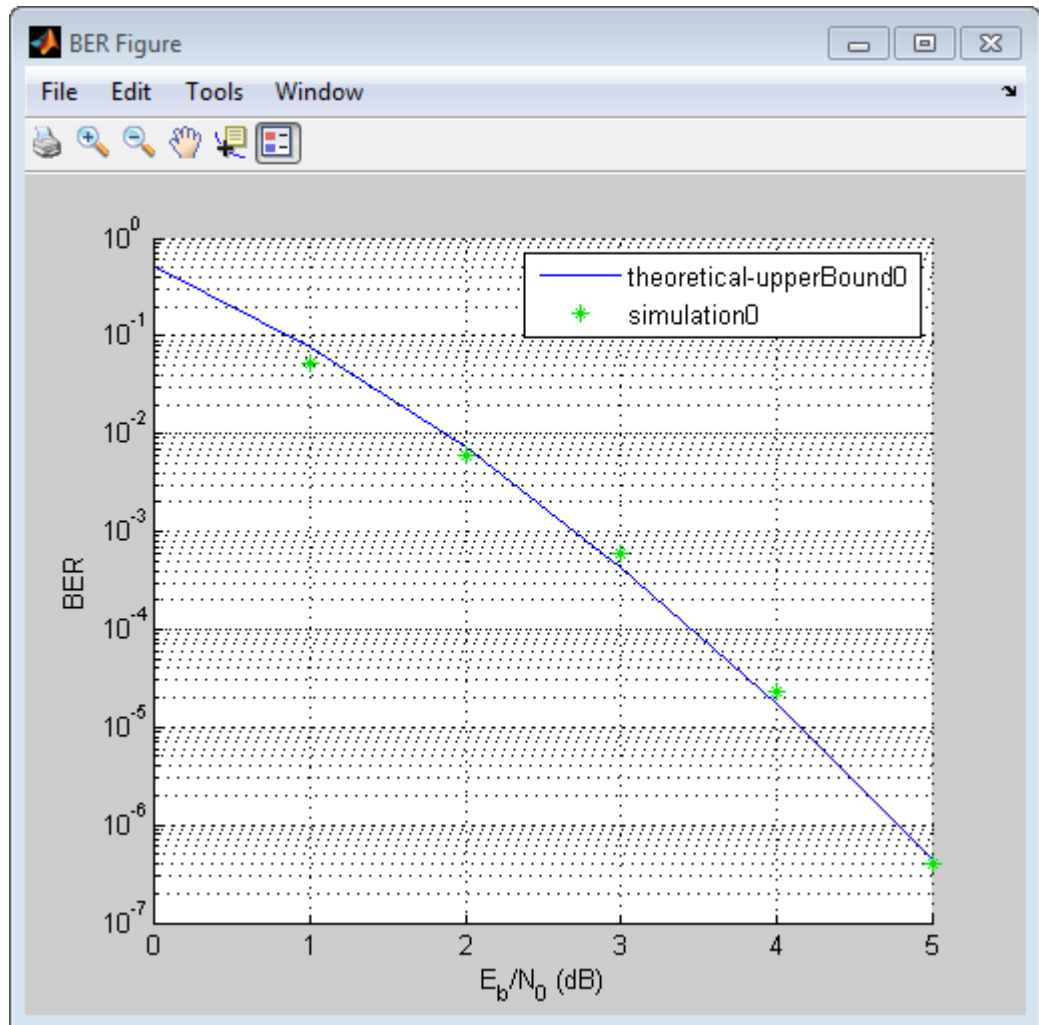
- 4 Enter 0:5 for the **EbNo range**.
- 5 Select **Soft** for the **Decision method**.

This example uses soft-decision Viterbi decoding. The demodulator maps the received signal to log likelihood ratios, improving BER performance results.

- 6 Click **Plot**.

The BERTool application generates the theoretical BER curve.

- 7 Click **Monte Carlo**.
- 8 Enter 0:5 for the **EbNo range**.
- 9 Enter 200 for the **Number of errors**.
- 10 Enter 1e7 for the **Number of bits**.
- 11 Click the **Browse** button, select **All Files** for the **Files of type** field.
- 12 Navigate to `matlab/help/toolbox/comm/examples`, select `doc_design_iteration_viterbi_soft.slx` and click **Run**.



When you plot the soft-decision theoretical curve, you will observe BER curve improvements of about 2 dB relative to the hard-decision decoding. Notice that the simulation results also reflects a similar BER improvement.

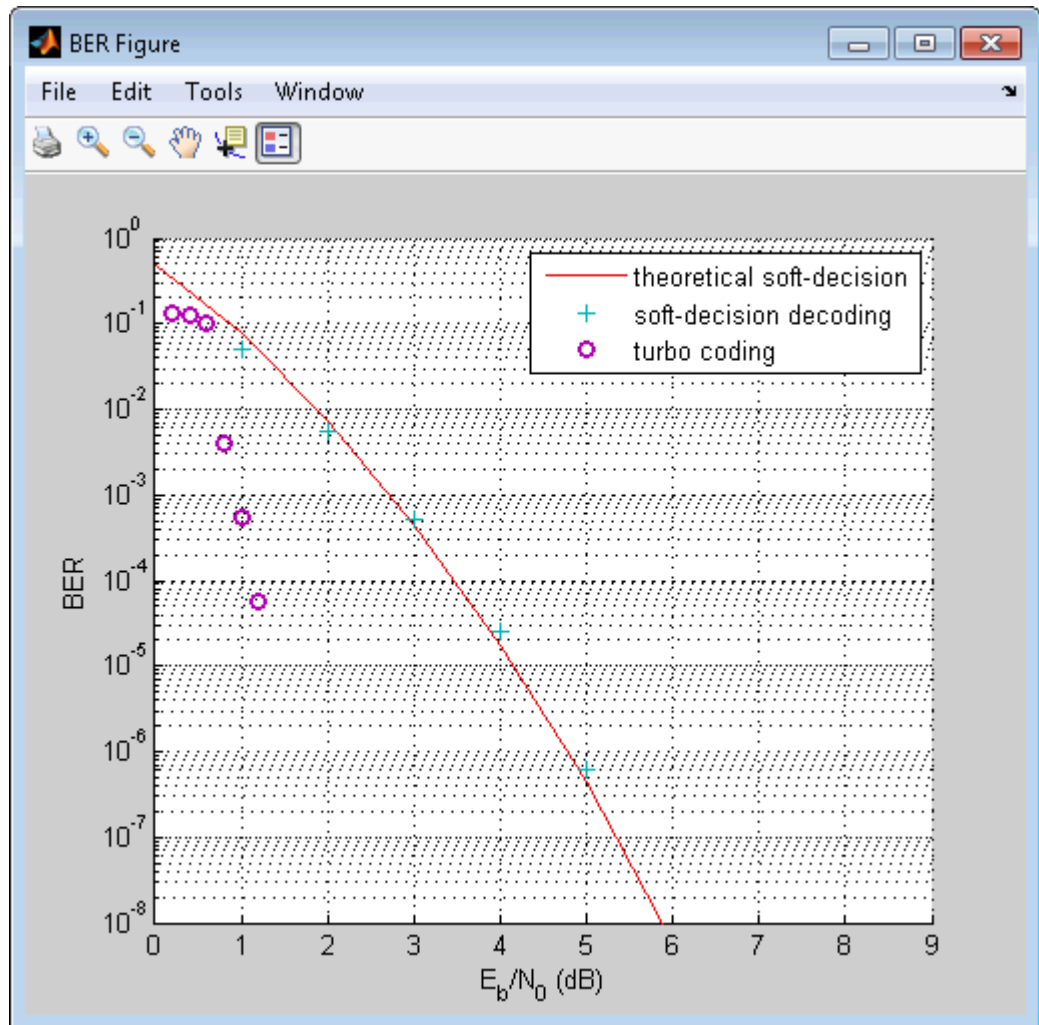
Use turbo coding to improve BER performance

Turbo codes substantially improve BER performance over soft-decision Viterbi decoding. Turbo coding uses two convolutional encoders in parallel at the transmitter and two a posteriori probability (APP) decoders in series at the receiver. This example uses a rate 1/3 turbo coder. For each input bit, the output has 1 systematic bit and 2 parity bits, for a total of three bits. Turbo coders achieve BER performances at much lower SNR values than convolutional encoders. As a result, this iteration uses a lower range of EbNo values than the previous section.

In MATLAB

- 1 Access the BERTool application.
- 2 Click the **Monte Carlo** tab.
- 3 Enter `0:0.2:1.2` for the **EbNo range**.
- 4 Enter 200 for the **Number of errors**.
- 5 Enter `1e7` for the **Number of bits**.
- 6 Click the **Browse** button.
- 7 Navigate to `matlab/help/toolbox/comm/examples`, select `doc_design_iteration_zTurbo_soft_m.m` and click **Run**.

BERTool runs the simulation and generates simulated points along the BER curve.

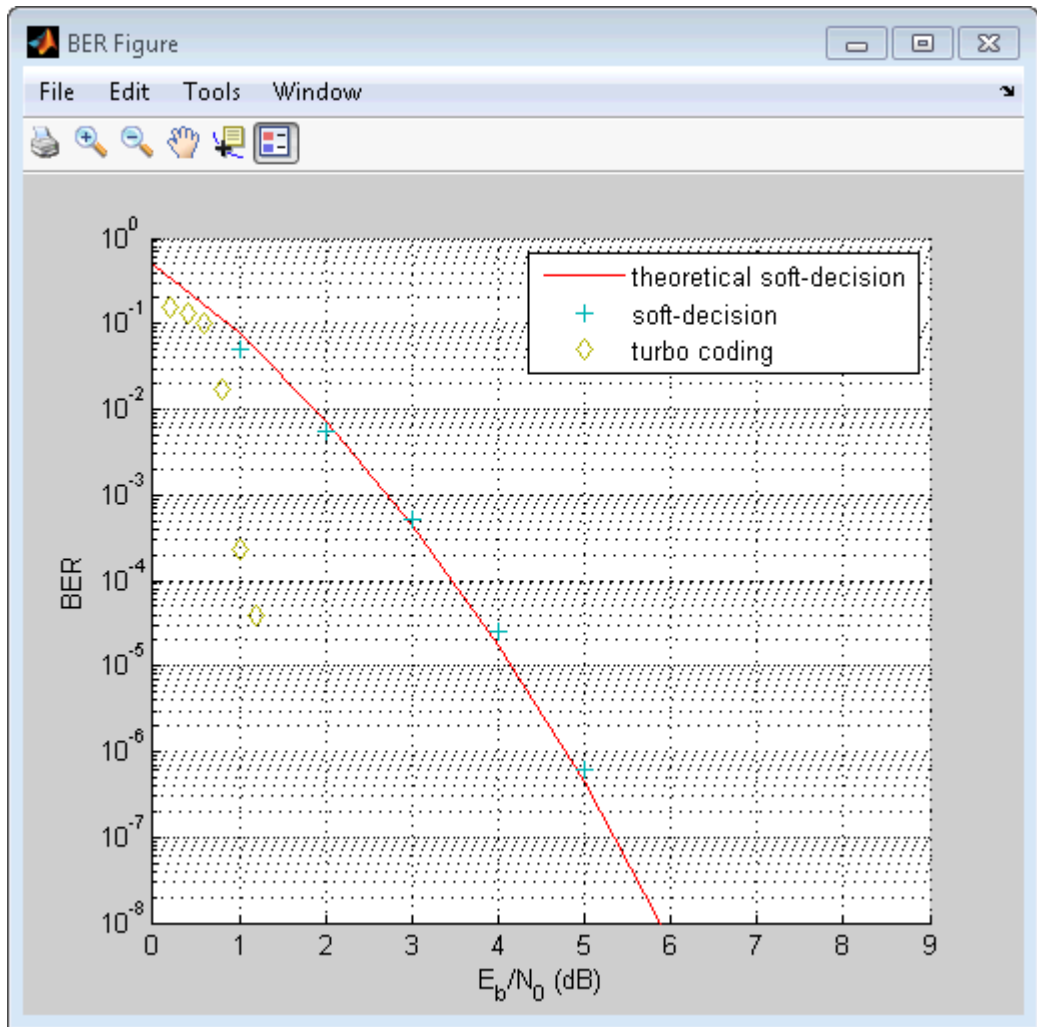


In Simulink

- 1 Access the BERTool application.
- 2 Clear the **Plot** check boxes for the last plot BERTool generated in the previous section.
- 3 Click the **Monte Carlo** tab.

- 4 Enter 0:0.2:1.2 for the **EbNo range**.
- 5 Enter 200 for the **Number of errors**.
- 6 Enter 1e7 for the **Number of bits**.
- 7 Click the **Browse** button, select **All Files** for the **Files of type** field.
- 8 Navigate to `matlab/help/toolbox/comm/examples`, select `doc_design_iteration_turbo.slx` and click **Run**.

BERTool runs the simulation and generates simulated points along the BER curve.



Apply a Rayleigh channel model

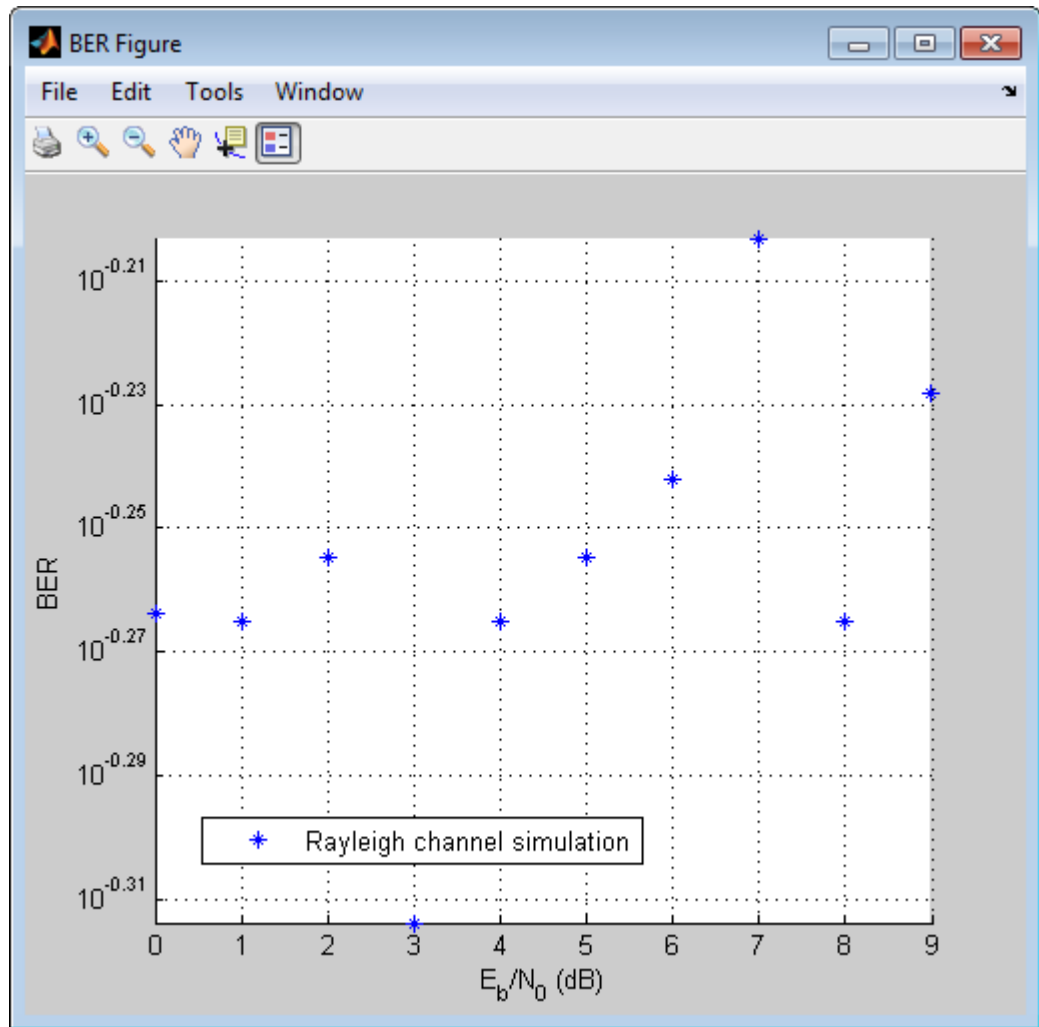
The previous design iterations model narrowband communications systems that can be adequately represented using an AWGN channel. However, high data rate communications systems require a wideband channel. Wideband communications

channels are highly susceptible to the effects of multipath propagation, which introduces intersymbol interference (ISI). Therefore, you must model wideband channels as multipath fading channels. This iteration of the design workflow uses a multipath fading Rayleigh channel, which assumes no direct line-of-sight between the transmitter and receiver.

In MATLAB

- 1 Access the BERTool application.
- 2 Clear the **Plot** check box for the plot BERTool generated in the previous step.
- 3 Click **Monte Carlo**.
- 4 Enter **0:9** for the **EbNo range**.
- 5 Enter **200** for the **Number of errors**.
- 6 Enter **1e7** for the **Number of bits**.
- 7 Click the **Browse** button.
- 8 Navigate to `matlab/help/toolbox/comm/examples`, select `doc_design_iteration_viterbi_rayleigh_m.m` and click **Run**.

BERTool runs the simulation and generates simulated points along the BER curve. Compare the simulation BER curve with the theoretical BER curve.

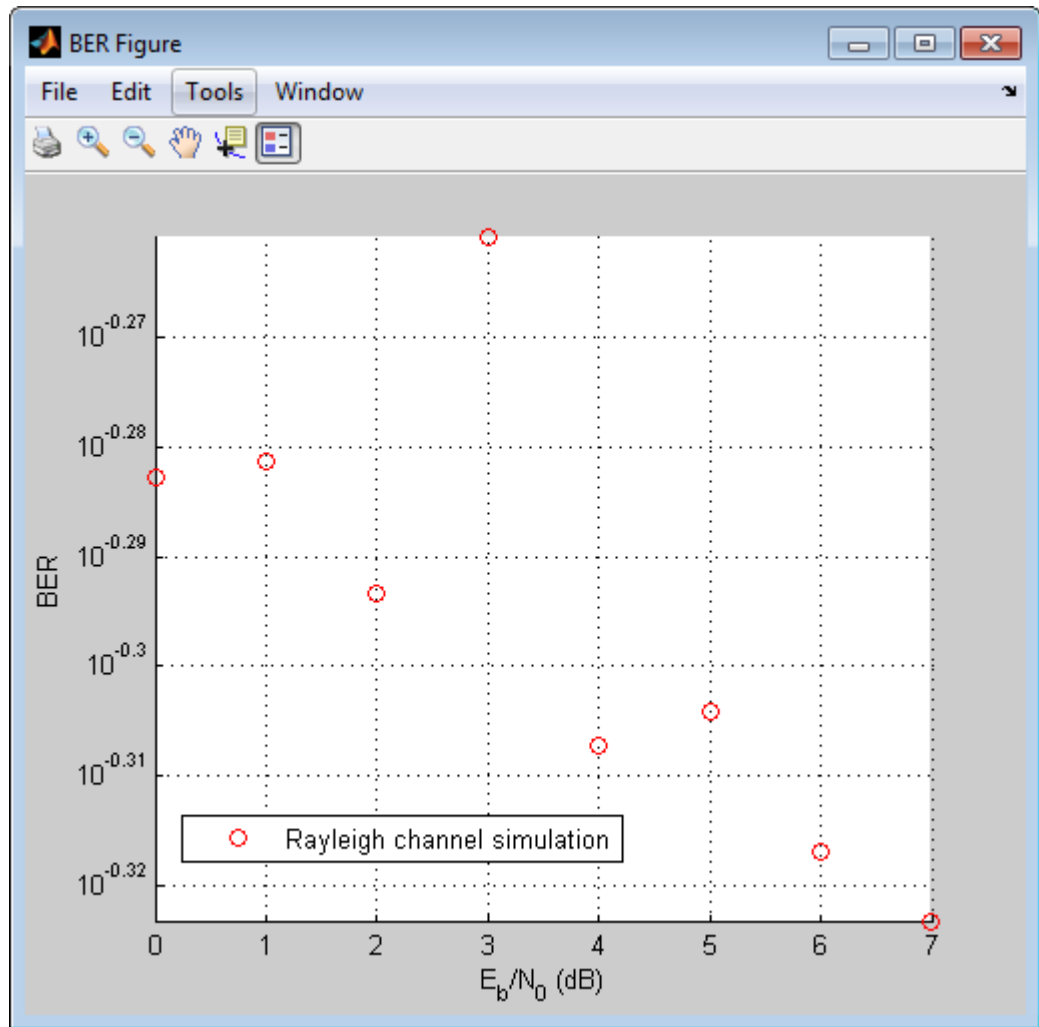


In the presence of multipath fading, the BER performance reduces to that of a binary channel with a consistent value of one-half. To correct the effect of multipath fading, you must add equalization to the communications system.

In Simulink

- 1 Access the BERTool application.
- 2 Clear the **Plot** check box to clear the plot BERTool generated in the previous step.
- 3 Click **Monte Carlo**.
- 4 Enter **0:7** for the **EbNo range**.
- 5 Enter **200** for the **Number of errors**.
- 6 Enter **1e7** for the **Number of bits**.
- 7 Click the **Browse** button, select **All Files** for the **Files of type** field.
- 8 Navigate to `matlab/help/toolbox/comm/examples`, select `doc_design_iteration_viterbi_rayleigh.slx` and click **Run**.

BERTool runs the simulation and generates simulated points along the BER curve. Compare the simulation BER curve with the theoretical BER curve.



In the presence of multipath fading, the BER performance reduces to that of a binary channel with a consistent value of one-half. To correct the effect of multipath fading, you must add equalization to the communications system.

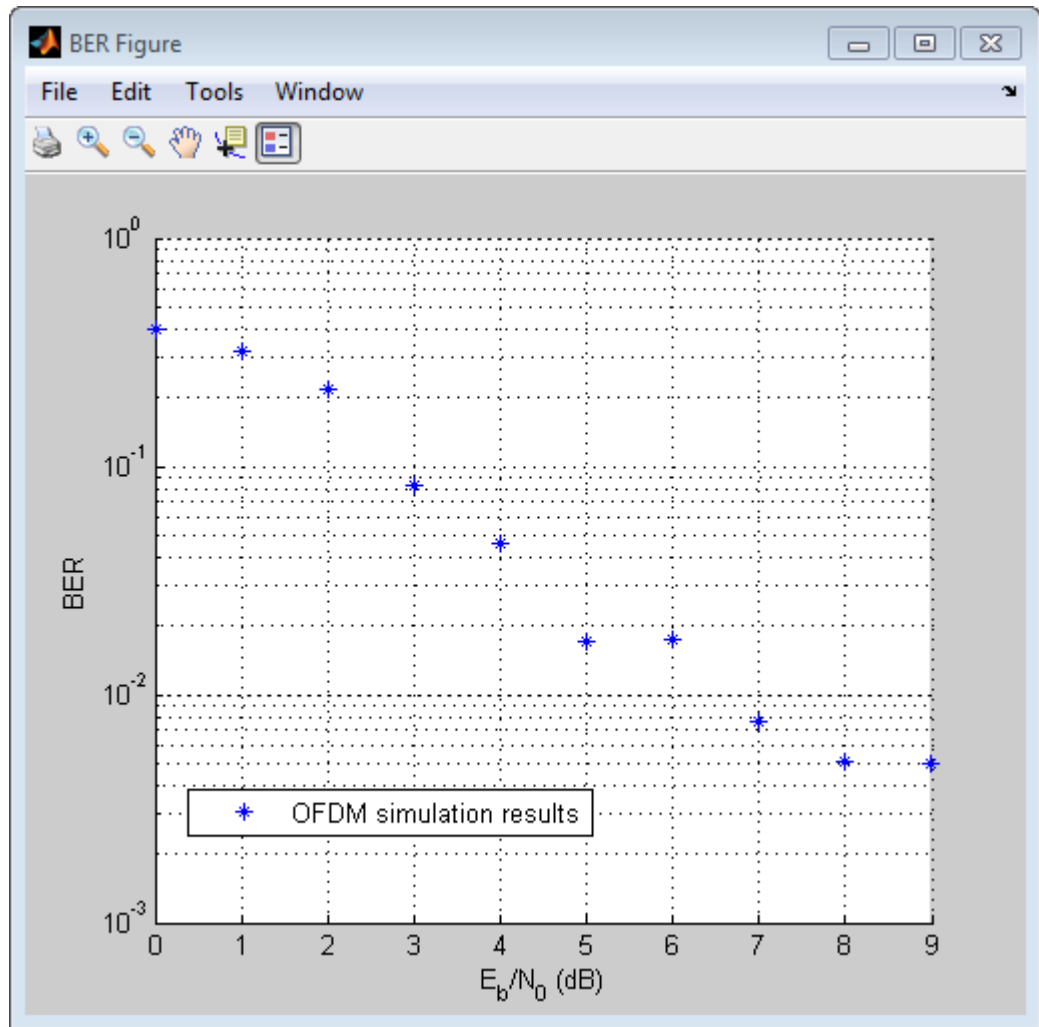
Use OFDM-based equalization to correct multipath fading

Use orthogonal frequency-division multiplexing (OFDM) to compensate for the multipath fading effect introduced by the Rayleigh fading channel. OFDM transmission schemes provides an effective way to perform frequency domain equalization. This design iteration introduces an OFDM transmitter, an OFDM receiver, and a frequency domain equalizer to the communications system.

In MATLAB

- 1 Access the BERTool application.
- 2 Clear the **Plot** check boxes for the simulation plot generated in the previous step.
- 3 Click the **Monte Carlo** tab.
- 4 Enter **0:9** for the **EbNo range**.
- 5 Enter **6000** for the **Number of errors**.
- 6 Enter **1e7** for the **Number of bits**.
- 7 Click the **Browse** button.
- 8 Navigate to `matlab/help/toolbox/comm/examples`, select `doc_design_iteration_viterbi_Rayleigh_OFDM_m.m` and click **Run**.

BERTool runs the simulation and generates simulated points along the BER curve.

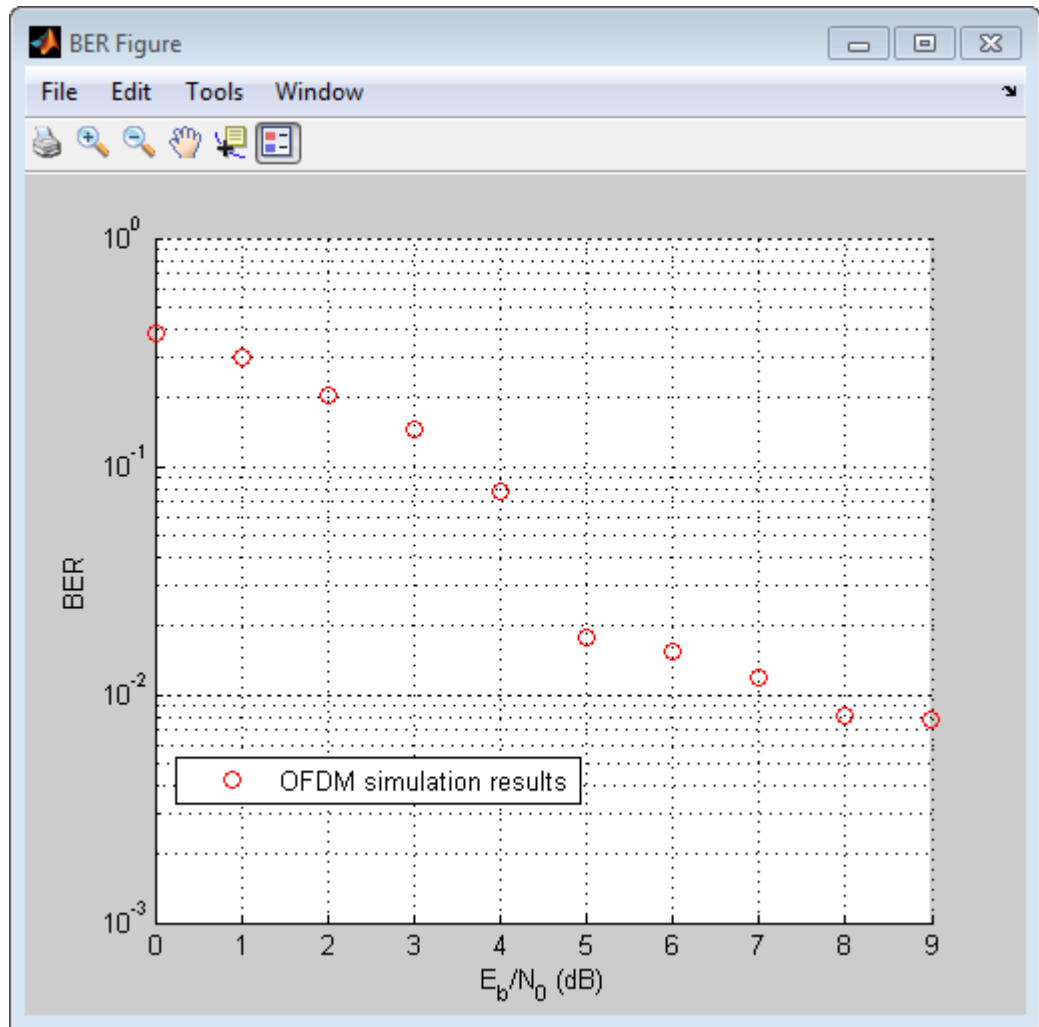


In Simulink

- 1 Access the BERTool application.
- 2 Clear the **Plot** check boxes for the plots BERTool generated in the previous step.
- 3 Click the **Monte Carlo** tab.
- 4 Enter 0:9 for the **EbNo range**.

- 5 Enter 6000 for the **Number of errors**.
- 6 Enter $5e7$ for the **Number of bits**.
- 7 Click the **Browse** button, select **All Files** for the **Files of type** field.
- 8 Navigate to `matlab/help/toolbox/comm/examples`, select `doc_design_iteration_viterbi_rayleigh_OFDM.slx` and click **Run**.

BERTool runs the simulation and generates simulated points. Compare the simulation BER curve with the theoretical BER curve.



Use multiple antennas to further improve system performance

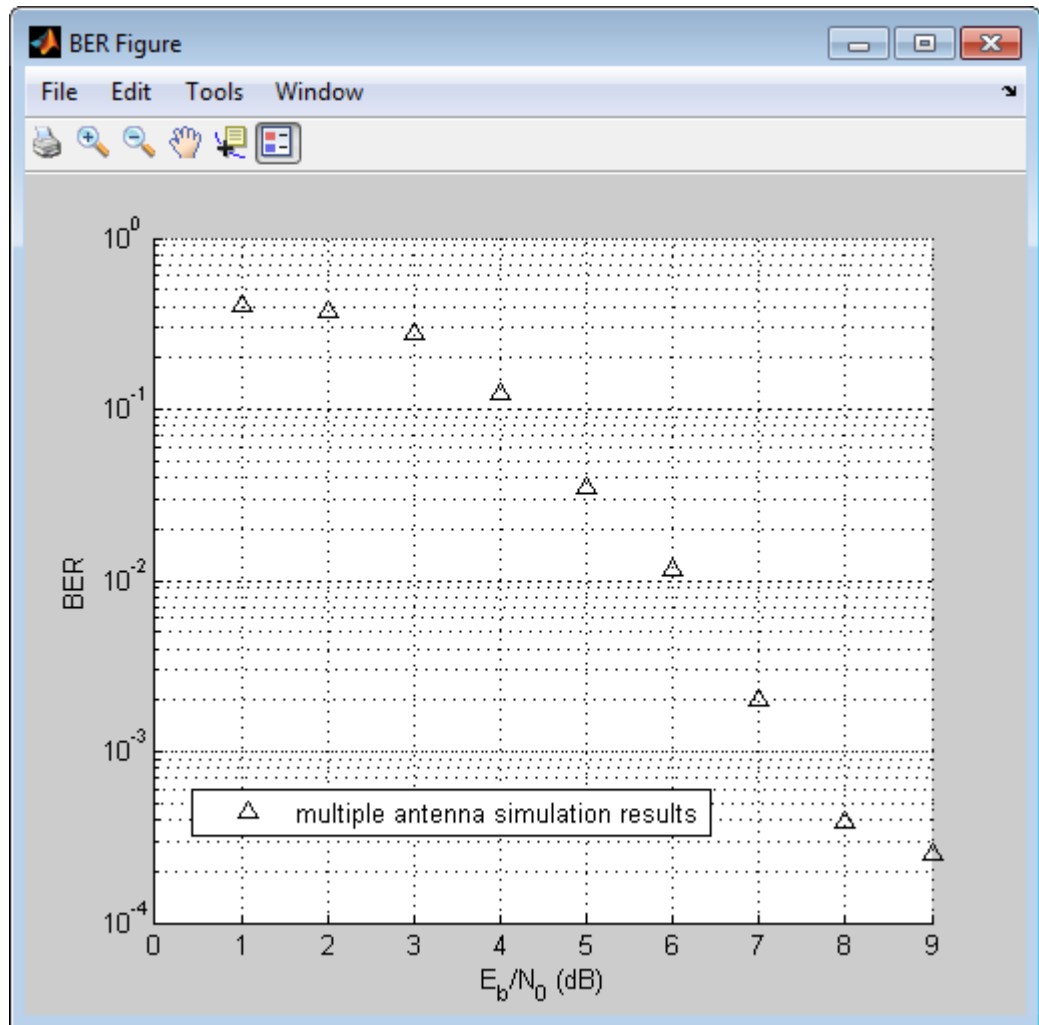
Simultaneously transmitting copies of a signal using multiple antennas can significantly increase the likelihood that the receiver correctly recovers the transmitted signal. This

phenomenon is known as transmit diversity. However, this performance improvement comes at the expense of introducing additional computational complexity in the receiver.

In MATLAB

- 1 Access the BERTool application.
- 2 Clear the **Plot** check box to clear the simulation plot generated in the previous step.
- 3 Click the **Monte Carlo** tab.
- 4 Enter **0:9** for the **EbNo range**.
- 5 Enter **1000** for the **Number of errors**.
- 6 Enter **1e7** for the **Number of bits**.
- 7 Click the **Browse** button.
- 8 Navigate to `matlab/help/toolbox/comm/examples`, select `doc_design_iteration_viterbi_rayleigh_OFDM_MIMO_m.m` and click **Run**.

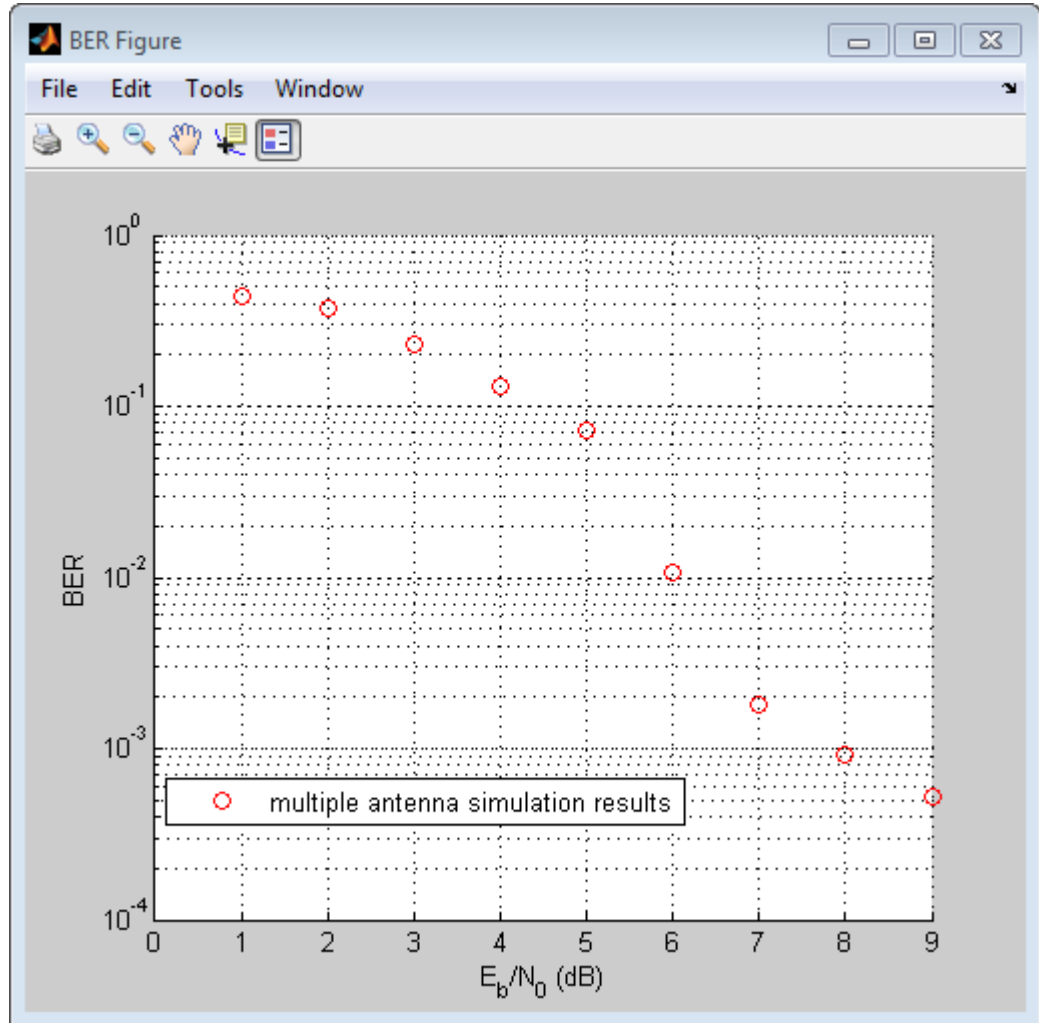
BERTool runs the simulation and generates simulated points along the BER curve. Compare the simulation BER curve with the theoretical BER curve.



In Simulink

- 1 Access the BERTool application.
- 2 Click the **Monte Carlo** tab.
- 3 Enter 0:9 for the **EbNo range**.
- 4 Enter 700 for the **Number of errors**.

- 5 Enter 1e7 for the **Number of bits**.
- 6 Click the **Browse** button, select **All Files** for the **Files of type** field.
- 7 Navigate to matlab/help/toolbox/comm/examples, select doc_design_iteration_viterbi_rayleigh_OFDM_MIMO.slx and click **Run**.



Accelerate the simulation using MATLAB Coder

All of the functions and System objects that this design iteration workflow uses support C code generation. If you have a MATLAB Coder™ license, you can accelerate simulation speed by generating a .mex file using the `codegen` command.

In MATLAB

- 1 Copy the `doc_design_iteration_viterbi_rayleigh_OFDM_MIMO_m.m` file to a folder that is not on the MATLAB path. For example, `C:\Temp`.
- 2 Change your working directory to the folder you just created.
- 3 Execute the following commands to set a numerical value for each of the input arguments in the `doc_design_iteration_viterbi_rayleigh_OFDM_MIMO_m` function. For example:

```
EbNo=1;
MaxNumErrs=200;
MaxNumBits=1e7;
```

- 4 Execute the `codegen` command to generate the executable MATLAB file.

```
codegen -args {EbNo,MaxNumErrs,MaxNumBits}
doc_design_iteration_viterbi_rayleigh_OFDM_MIMO_m
```

- 5 The file extension of the MATLAB executable file that gets generated depends upon your operating system. For example, on 64-bit Windows® the file extension will be `.mexw64`, and the full file name will be `doc_design_iteration_viterbi_rayleigh_OFDM_MIMO_m_mex.mexw64`.

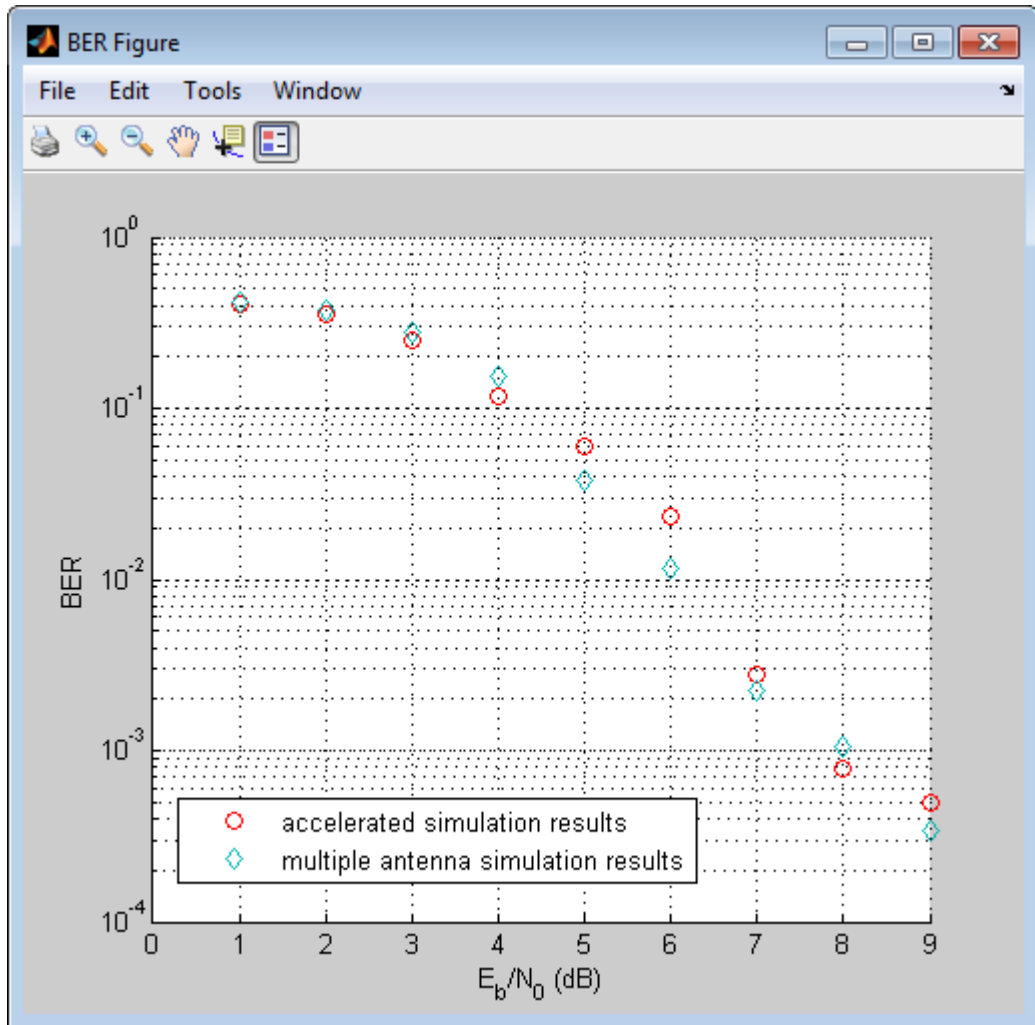
If you run the mex file you just generated in BERTool, you will obtain the simulation results more quickly.

- 6 Access the BERTool application.
- 7 Click the **Monte Carlo** tab.
- 8 Enter `0:9` for the **EbNo range**.
- 9 Enter `700` for the **Number of errors**.
- 10 Enter `1e7` for the **Number of bits**.
- 11 Click the **Browse** button, and select **All Files**.

Navigate to folder you created in step 1 and click **Run**.

BERTool runs the simulation and generates simulated points along the BER curve. Compare the simulation BER curve with the previous curve. Any variation in the BER curve of the mex file and the MATLAB file from which it was generated is

related to the seed of the random number generator and is statistically insignificant. In this example, BERTool generates the curve much more quickly when you use MATLAB Coder to generate C code. Notice that BERTool generates similar BER results in about 1/4 of the time that it took for the original simulation to complete.



QPSK and OFDM with MATLAB System Objects

This example shows how to simulate a basic communication system in which the signal is first QPSK modulated and then subjected to Orthogonal Frequency Division Multiplexing. The signal is then passed through an additive white Gaussian noise channel prior to being demultiplexed and demodulated. Lastly, the number of bit errors are calculated. The example showcases the use of MATLAB® System objects™.

Set the simulation parameters.

```
M = 4;           % Modulation alphabet
k = log2(M);    % Bits/symbol
numSC = 128;    % Number of OFDM subcarriers
cpLen = 32;     % OFDM cyclic prefix length
maxBitErrors = 100; % Maximum number of bit errors
maxNumBits = 1e7; % Maximum number of bits transmitted
```

Construct System objects needed for the simulation: QPSK modulator, QPSK demodulator, OFDM modulator, OFDM demodulator, AWGN channel, and an error rate calculator. Use name-value pairs to set the object properties.

Set the QPSK modulator and demodulator so that they accept binary inputs.

```
hQPSKMod = comm.QPSKModulator('BitInput',true);
hQPSKDemod = comm.QPSKDemodulator('BitOutput',true);
```

Set the OFDM modulator and demodulator pair according to the simulation parameters.

```
hOFDMmod = comm.OFDMModulator('FFTLength',numSC,'CyclicPrefixLength',cpLen);
hOFDMdemod = comm.OFDMDemodulator('FFTLength',numSC,'CyclicPrefixLength',cpLen);
```

Set the `NoiseMethod` property of the AWGN channel object to `Variance` and define the `VarianceSource` property so that the noise power can be set from an input port.

```
hChan = comm.AWGNChannel('NoiseMethod','Variance', ...
    'VarianceSource','Input port');
```

Set the `ResetInputPort` property to `true` to enable the error rate calculator to be reset during the simulation.

```
hError = comm.ErrorRate('ResetInputPort',true);
```

Use the `info` function of the `hOFDMmod` object to determine the input and output dimensions of the OFDM modulator.

```
ofdmInfo = info(hOFDMmod)
```

```
ofdmInfo =
```

```
    DataInputSize: [117 1]  
    OutputSize: [160 1]
```

Determine the number of data subcarriers from the `ofdmInfo` structure variable.

```
numDC = ofdmInfo.DataInputSize(1)
```

```
numDC =
```

```
    117
```

Determine the OFDM frame size (in bits) from the number of data subcarriers and the number of bits per symbol.

```
frameSize = [k*numDC 1];
```

Set the SNR vector based on the desired E_b/N_0 range, the number of bits per symbol, and the ratio of the number of data subcarriers to the total number of subcarriers.

```
EbNoVec = (0:10)';  
snrVec = EbNoVec + 10*log10(k) + 10*log10(numDC/numSC);
```

Initialize the BER and error statistics arrays.

```
berVec = zeros(length(EbNoVec),3);  
errorStats = zeros(1,3);
```

Simulate the communication link over the range of E_b/N_0 values. For each E_b/N_0 value, the simulation runs until either `maxBitErrors` are recorded or the total number of transmitted bits exceeds `maxNumBits`.

```
for m = 1:length(EbNoVec)  
    snr = snrVec(m);  
  
    while errorStats(2) <= maxBitErrors && errorStats(3) <= maxNumBits  
        dataIn = randi([0,1],frameSize); % Generate binary data
```

```

    qpskTx = step(hQPSKMod,dataIn);           % Apply QPSK modulation
    txSig = step(hOFDMmod,qpskTx);          % Apply OFDM modulation
    powerDB = 10*log10(var(txSig));         % Calculate Tx signal power
    noiseVar = 10.^(0.1*(powerDB-snr));     % Calculate the noise variance
    rxSig = step(hChan,txSig,noiseVar);     % Pass the signal through a noisy channel
    qpskRx = step(hOFDMdemod,rxSig);       % Apply OFDM demodulation
    dataOut = step(hQPSKDemod,qpskRx);     % Apply QPSK demodulation
    errorStats = step(hError,dataIn,dataOut,0); % Collect error statistics
end

    berVec(m,:) = errorStats;               % Save BER data
    errorStats = step(hError,dataIn,dataOut,1); % Reset the error rate calculator
end

```

Use the `berawgn` function to determine the theoretical BER for a QPSK system.

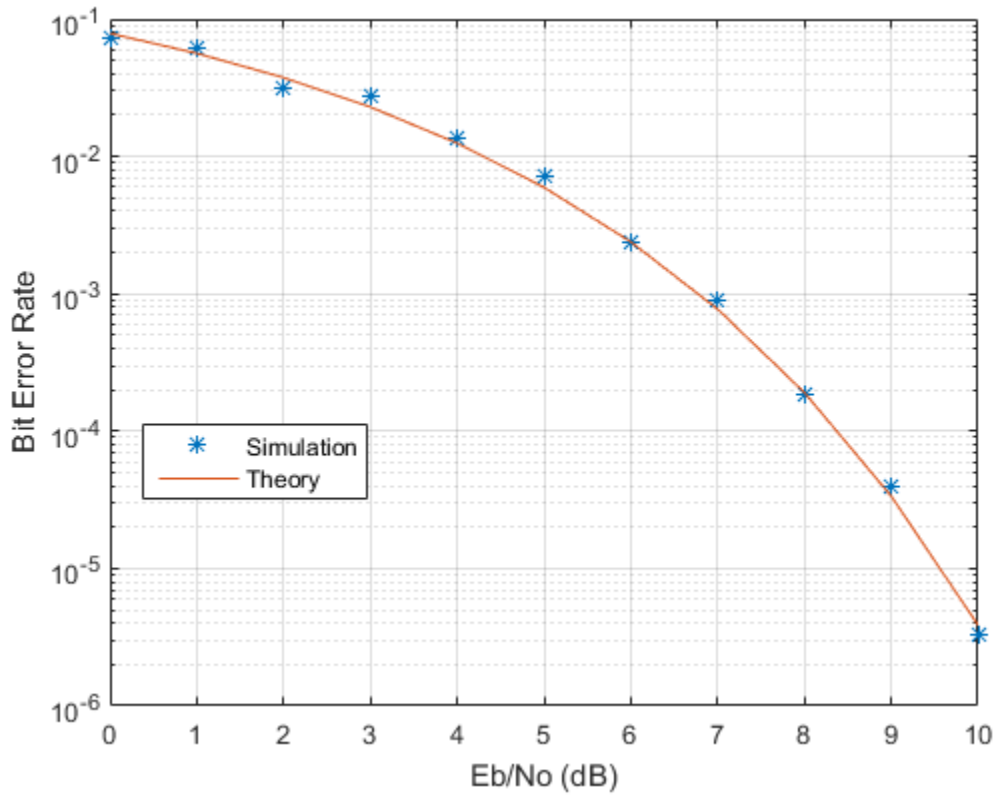
```
berTheory = berawgn(EbNoVec, 'psk',M, 'nondiff');
```

Plot the theoretical and simulated data on the same graph to compare results.

```

figure
semilogy(EbNoVec,berVec(:,1),'*')
hold on
semilogy(EbNoVec,berTheory)
legend('Simulation','Theory','Location','Best')
xlabel('Eb/No (dB)')
ylabel('Bit Error Rate')
grid on
hold off

```



Observe that there is good agreement between the simulated and theoretical data.

Accelerating BER Simulations Using the Parallel Computing Toolbox

This example shows how to use the Parallel Computing Toolbox™ to accelerate a simple, QPSK bit error rate (BER) simulation. The system consists of a QPSK modulator, a QPSK demodulator, an AWGN channel, and a bit error rate counter. In this example, four parallel processors are used.

Set the simulation parameters.

```
EbNoVec = 5:8;      % Eb/No values in dB
totalErrors = 200; % Number of bit errors needed for each Eb/No value
totalBits = 1e7;   % Total number of bits transmitted for each Eb/No value
```

Allocate memory to the arrays used to store the data generated by the function, `doc_fcn_qpsk_sim_with_awgn`.

```
[numErrors, numBits] = deal(zeros(length(EbNoVec),1));
```

Run the simulation and determine the execution time. Only one processor will be used to determine baseline performance. Accordingly, observe that the normal for-loop is employed.

```
tic
```

```
for idx = 1:length(EbNoVec)
    errorStats = doc_fcn_qpsk_sim_with_awgn(EbNoVec, idx, ...
        totalErrors, totalBits);
    numErrors(idx) = errorStats(idx,2);
    numBits(idx) = errorStats(idx,3);
end
```

```
simBaselineTime = toc;
```

Calculate the BER.

```
ber1 = numErrors ./ numBits;
```

Rerun the simulation for the case in which the Parallel Computing Toolbox is available. Create a pool of workers.

```
pool = gcp;
```

```
Starting parallel pool (parpool) using the 'local' profile ... connected to 4 workers.
```

Determine the number of available workers from the `NumWorkers` property of `pool`. The simulation runs the range of E_b/N_0 values over each worker rather than assigning a single E_b/N_0 point to each worker as the former method provides the biggest performance improvement.

```
numWorkers = pool.NumWorkers;
```

Determine the length of `EbNoVec` for use in the nested `parfor` loop. For proper variable classification, the range of a for-loop nested in a `parfor` must be defined by constant numbers or variables.

```
lenEbNoVec = length(EbNoVec);
```

Allocate memory to the arrays used to store the data generated by the function, `doc_fcn_qpsk_sim_with_awgn`.

```
[numErrors, numBits] = deal(zeros(length(EbNoVec),numWorkers));
```

Run the simulation and determine the execution time.

```
tic
```

```
parfor n = 1:numWorkers
```

```
    for idx = 1:lenEbNoVec
        errorStats = doc_fcn_qpsk_sim_with_awgn(EbNoVec, idx, ...
            totalErrors/numWorkers, totalBits/numWorkers);
        numErrors(idx,n) = errorStats(idx,2);
        numBits(idx,n) = errorStats(idx,3);
    end
```

```
end
```

```
simParallelTime = toc;
```

Calculate the BER. In this case, the results from multiple processors must be combined to generate the aggregate BER.

```
ber2 = sum(numErrors,2) ./ sum(numBits,2);
```

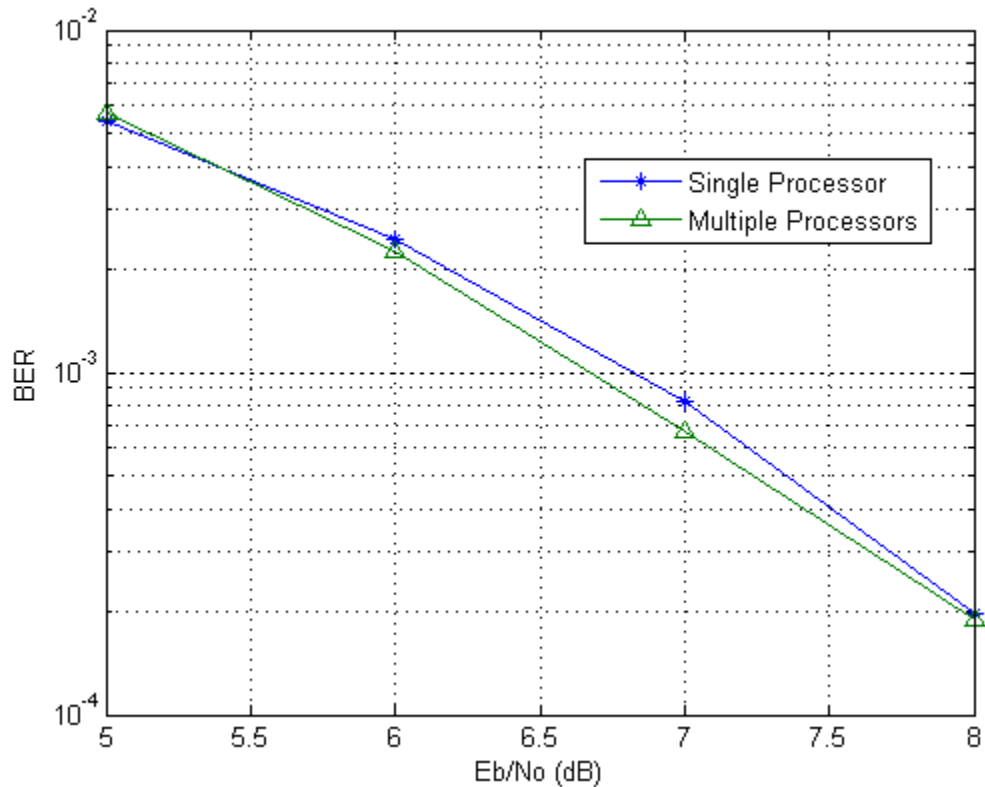
Compare the BER values to verify that the same results are obtained independent of the number of workers.

```
semilogy(EbNoVec',ber1,'-*',EbNoVec',ber2,'-^')
legend('Single Processor','Multiple Processors','location','best')
```

```

xlabel('Eb/No (dB)')
ylabel('BER')
grid

```



You can see that the BER curves are essentially the same with any variance being due to differing random number seeds.

Compare the execution times for each method.

```

fprintf(['\nSimulation time = %4.1f sec for one worker\n', ...
        'Simulation time = %4.1f sec for multiple workers\n'], ...
        simBaselineTime, simParallelTime)

```

```
Simulation time = 170.1 sec for one worker
```

Simulation time = 52.7 sec for multiple workers

In this case where four processor cores were used, the speed improvement factor was approximately four.

Visualization and Measurements

- “Scatter Plot and Eye Diagram with MATLAB Functions” on page 3-2
- “EVM and MER Measurements with Simulink” on page 3-7
- “ACPR and CCDF Measurements with MATLAB System Objects” on page 3-14

Scatter Plot and Eye Diagram with MATLAB Functions

This example shows how to use the Communications System Toolbox to visualize signal behavior through the use of eye diagrams and scatter plots. The example uses a QPSK signal which is passed through a square-root raised cosine (RRC) filter.

Scatter Plot

Set the RRC filter, modulation scheme, and plotting parameters.

```
span = 10;           % Filter span
rolloff = 0.2;      % Rolloff factor
sps = 8;           % Samples per symbol
M = 4;            % Modulation alphabet size
k = log2(M);       % Bits/symbol
phOffset = pi/4;   % Phase offset (radians)
n = 1;           % Plot every nth value of the signal
offset = 0;       % Plot every nth value of the signal, starting from offset+1
```

Create the filter coefficients using the `rcosdesign` function.

```
filtCoeff = rcosdesign(rolloff,span,sps);
```

Generate random symbols for an alphabet size of M .

```
rng default
data = randi([0 M-1],5000,1);
```

Apply QPSK modulation.

```
dataMod = pskmod(data,M,phOffset);
```

Filter the modulated data.

```
txSig = upfirdn(dataMod,filtCoeff,sps);
```

Calculate the SNR for an oversampled QPSK signal.

```
EbNo = 20;
snr = EbNo + 10*log10(k) - 10*log10(sps);
```

Add AWGN to the transmitted signal.

```
rxSig = awgn(txSig,snr, 'measured');
```

Apply the RRC receive filter.

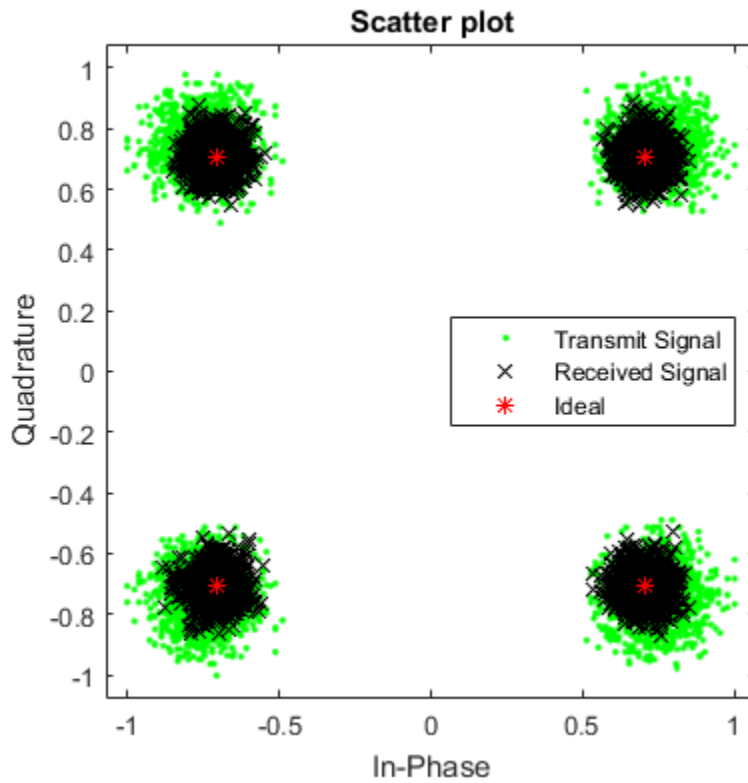
```
rxSigFilt = upfirdn(rxSig, filtCoeff,1,sps);
```

Demodulate the filtered signal.

```
dataOut = pskdemod(rxSigFilt,M,phOffset, 'gray');
```

Use the `scatterplot` function to show scatter plots of the signal before and after filtering. You can see that the receive filter improves performance as the constellation more closely matches the ideal values. The first `span` symbols and the last `span` symbols represent the cumulative delay of the two filtering operations and are removed from the two filtered signals before generating the scatter plots.

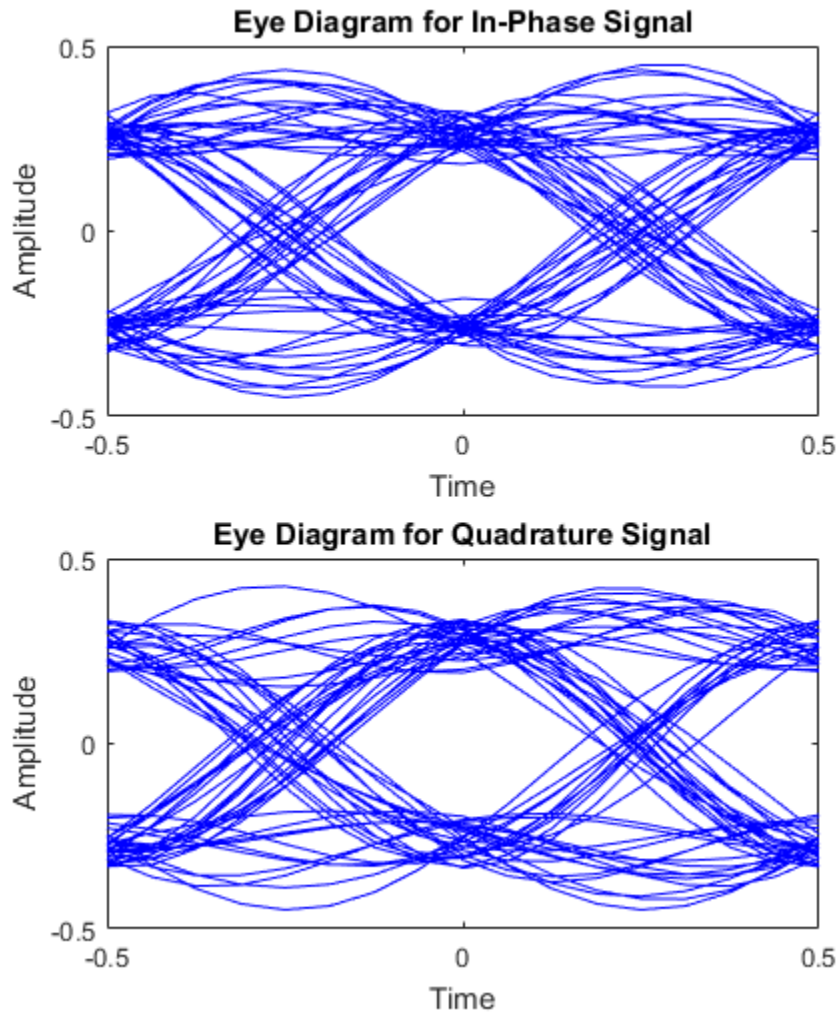
```
h = scatterplot(sqrt(sps)*txSig(sps*span+1:end-sps*span),sps,offset, 'g. ');  
hold on  
scatterplot(rxSigFilt(span+1:end-span),n,offset, 'kx',h)  
scatterplot(dataMod,n,offset, 'r*',h)  
legend('Transmit Signal','Received Signal','Ideal','location','best')
```



Eye Diagram

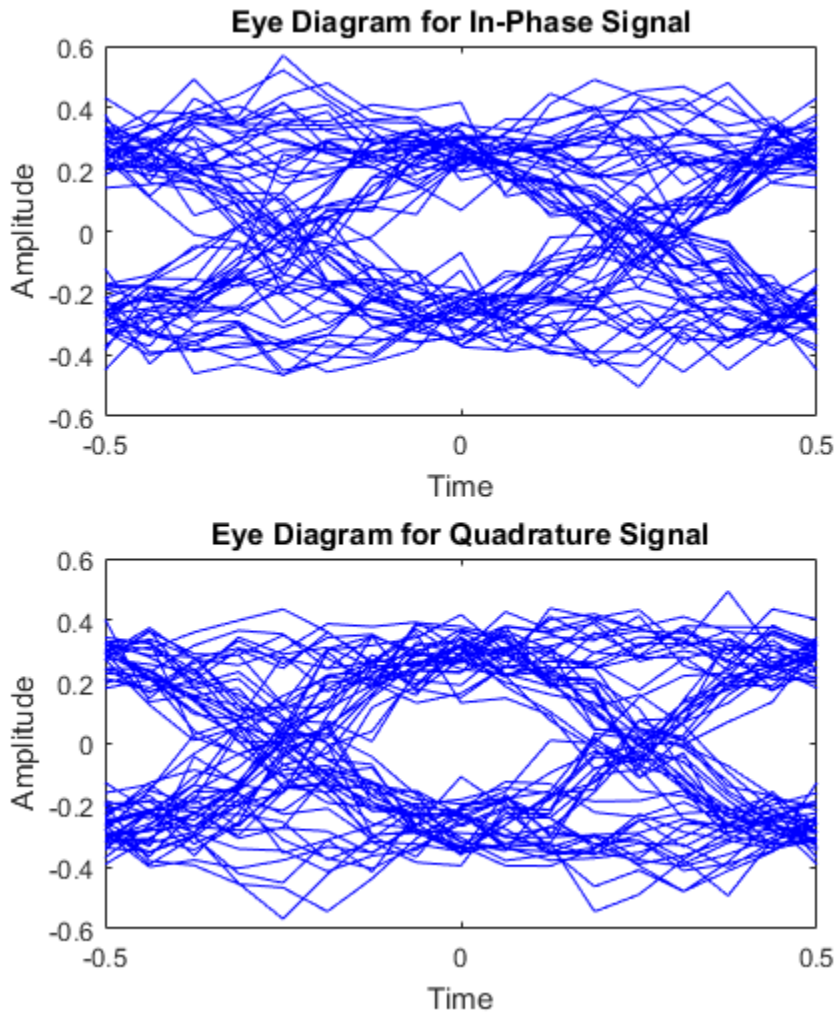
Display 1000 points of the transmitted signal eye diagram over two symbol periods.

```
eyediagram(txSig(sps*span+1:sps*span+1000),2*sps)
```



Display 1000 points of the received signal eye diagram.

```
eyediagram(rxSig(sps*span+1:sps*span+1000),2*sps)
```



Observe that the received eye diagram begins to close due to the presence of AWGN. Moreover, the filter has finite length which also contributes to the non-ideal behavior.

EVM and MER Measurements with Simulink

This model shows how error vector magnitude (EVM) and modulation error rate (MER) measurements are made using Simulink blocks.

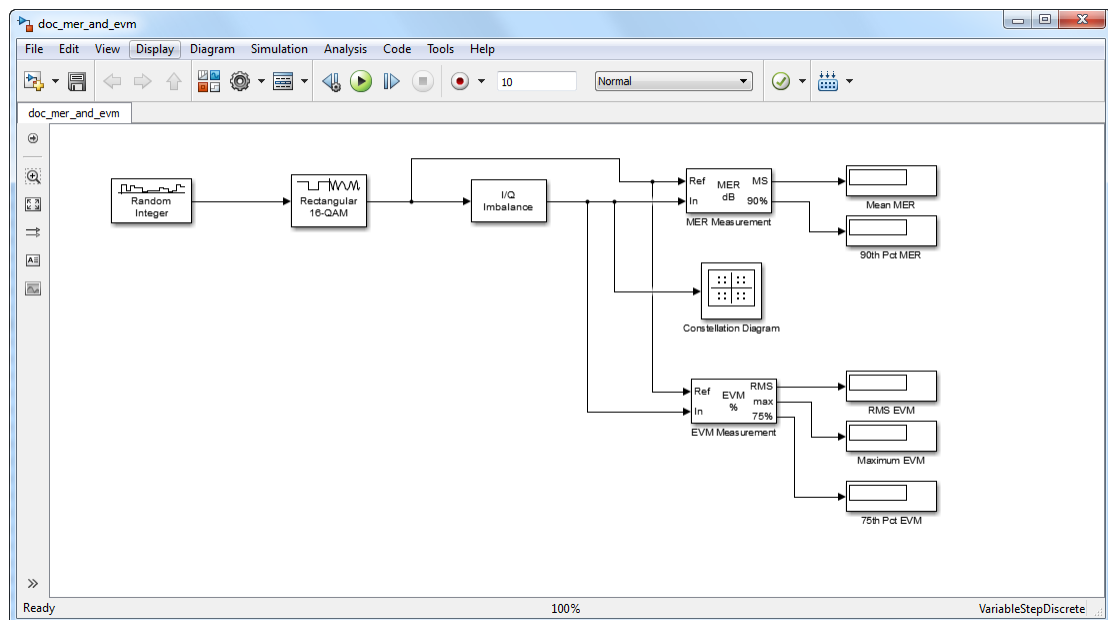
Load the model `doc_mer_and_evm` from the MATLAB command prompt.

```
doc_mer_and_evm
```

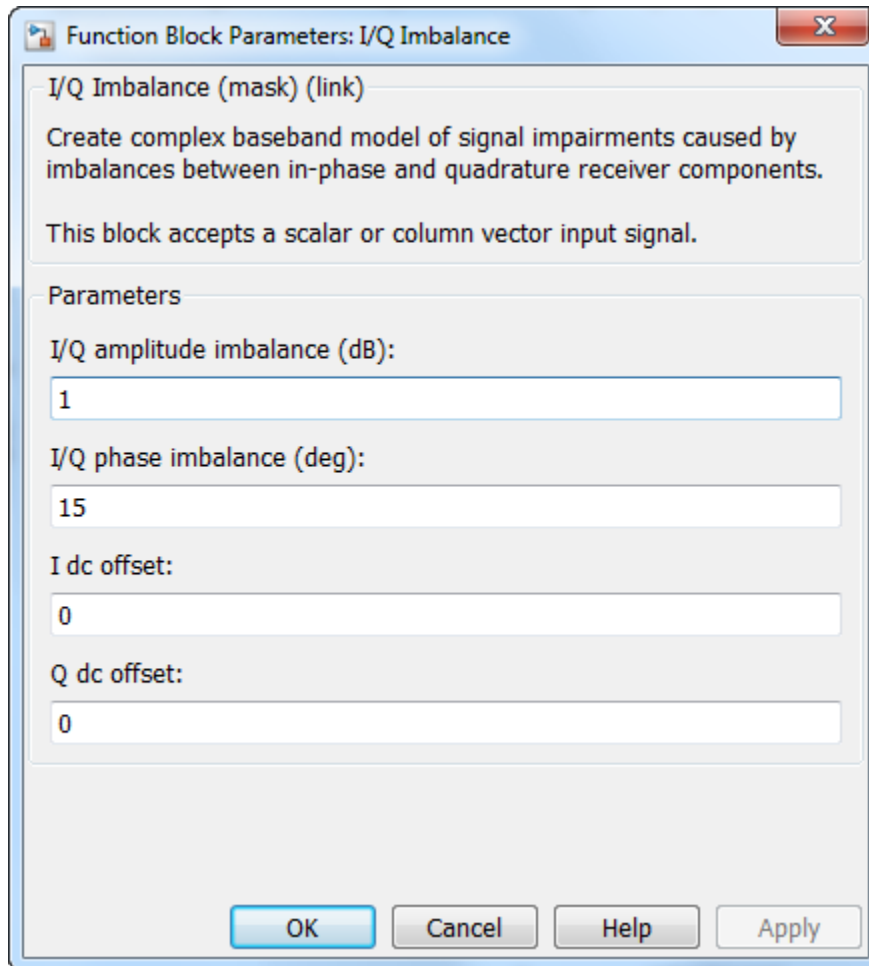
This example includes:

- A 16-QAM modulated signal
- An I/Q imbalance
- A constellation diagram block
- EVM Measurement and MER Measurement blocks

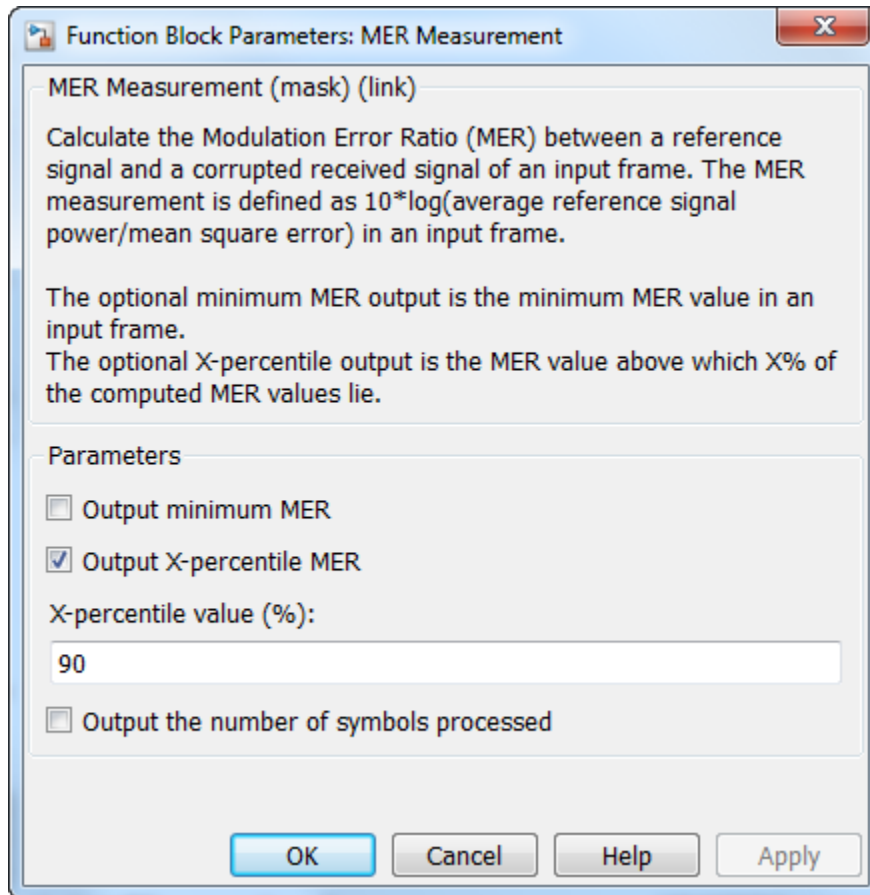
The model applies an I/Q imbalance to a QAM-modulated signal at which point MER and EVM measurements are made. The constellation diagram provides a visual representation of the effects the imbalance has on the modulation performance indicators.



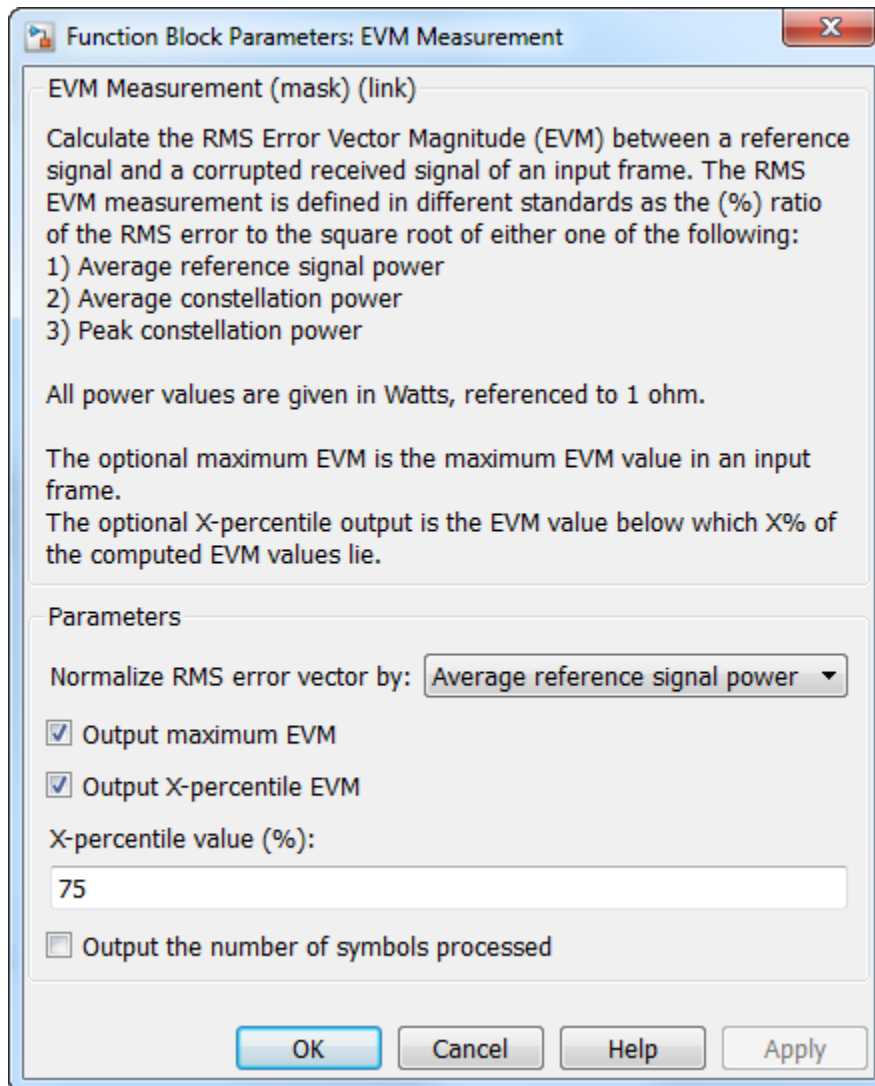
The I/Q Imbalance block is set to that the **I/Q amplitude imbalance (dB)** is set to 1 and the **I/Q phase imbalance (deg)** is set to 15.



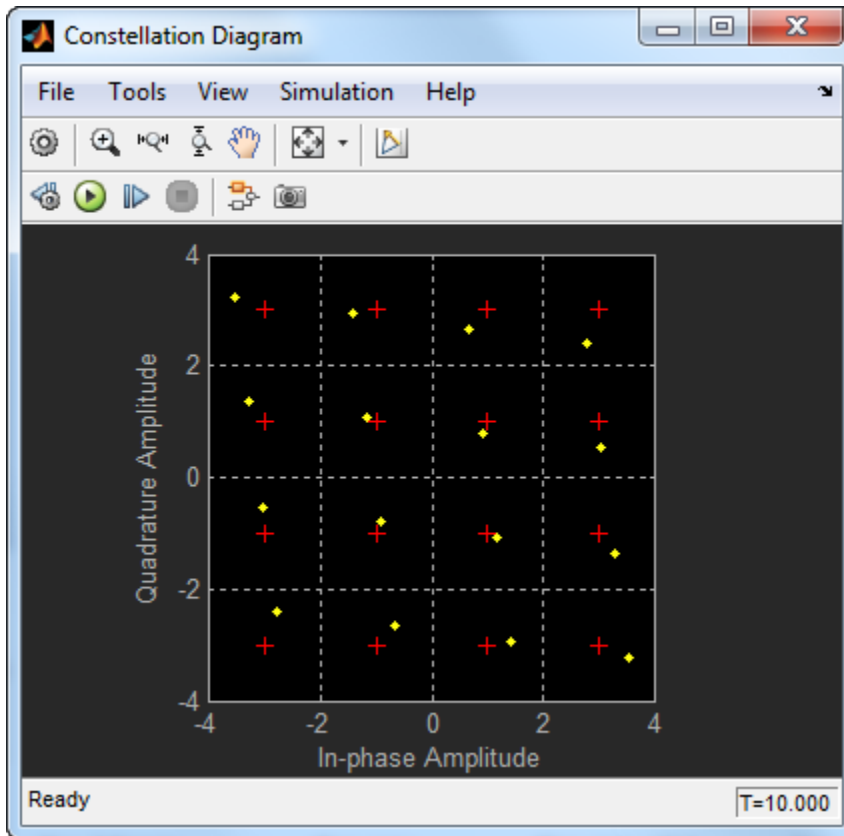
The MER Measurement block is set so that it outputs the X-percentile MER value which is set to 90%.



The EVM Measurement block is set to output the maximum and 75th percentile EVM values.

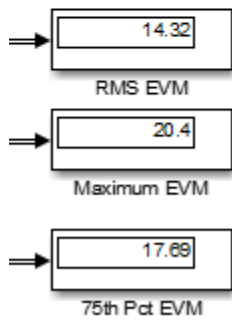
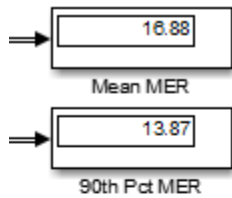


Run the model.

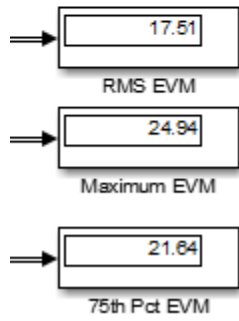
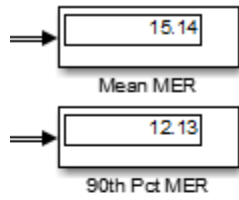


You can see that I/Q amplitude and phase imbalance has shifted the constellation diagram so that each symbol is not exactly equal to its reference symbol (shown in red). Change the I/Q Imbalance block to see the effects of differing imbalances on the constellation diagram.

Observe the EVM and MER values. For the default configuration of the model, the mean MER is approximately 16.9 dB and the 90th percentile MER is 13.9 dB. The RMS EVM is, approximately, 14.3%, the maximum EVM is 20.4%, and the 75th percentile EVM is 17.7%.



Change the **I/Q amplitude imbalance (dB)** value in the I/Q Imbalance block to 2 dB. You can see that the all the MER and EVM metrics degrade.



ACPR and CCDF Measurements with MATLAB System Objects

In this section...
“ACPR Measurements” on page 3-14
“CCDF Measurements” on page 3-17

ACPR Measurements

This example shows how to measure the adjacent channel power ratio (ACPR) from a baseband, 50 kbps QPSK signal. ACPR is the ratio of signal power measured in an adjacent frequency band to the power from the same signal measured in its main band. The number of samples per symbol is set to four.

Set the random number generator so that results are repeatable.

```
prevState = rng;  
rng(577)
```

Set the samples per symbol (sps) and channel bandwidth (bw) parameters.

```
sps = 4;  
bw = 50e3;
```

Generate 10,000 4-ary symbols for QSPK modulation.

```
data = randi([0, 3], 10000, 1);
```

Construct a QPSK modulator and then modulate the input data.

```
hMod = comm.QPSKModulator;  
x = step(hMod, data);
```

Apply rectangular pulse shaping to the modulated signal. This type of pulse shaping is typically not done in practical system but is used here for illustrative purposes.

```
y = rectpulse(x, sps);
```

Construct an ACPR System object. The sample rate is the bandwidth multiplied by the number of samples per symbol. The main channel is assumed to be at 0 while the adjacent channel offset is set to 50 kHz (identical to the bandwidth of the main channel).

Likewise, the measurement bandwidth of the adjacent channel is set to be the same as the main channel. Lately, enable the main and adjacent channel power output ports.

```
hACPR = comm.ACPR('SampleRate',bw*sps,...  
    'MainChannelFrequency',0,...  
    'MainMeasurementBandwidth',bw,...  
    'AdjacentChannelOffset',50e3,...  
    'AdjacentMeasurementBandwidth',bw,...  
    'MainChannelPowerOutputPort', true,...  
    'AdjacentChannelPowerOutputPort',true);
```

Use the step method of the `comm.ACPR` System object to output the ACPR, the main channel power, and the adjacent channel power for the signal, `y`.

```
[ACPR, mainPower, adjPower] = step(hACPR, y)
```

```
ACPR =  
-9.5103
```

```
mainPower =  
28.9634
```

```
adjPower =  
19.4531
```

Change the frequency offset to 75 kHz and determine the ACPR. Since the `AdjacentChannelOffset` property is nontunable, you must first release `hACPR`. Observe that the ACPR improves when the channel offset is increased.

```
release(hACPR)  
hACPR.AdjacentChannelOffset = 75e3;  
ACPR = step(hACPR, y)
```

```
ACPR =  
-13.2317
```

Reset hACPR and specify a 50 kHz adjacent channel offset.

```
release(hACPR)
hACPR.AdjacentChannelOffset = 50e3;
```

Create a raised cosine filter and filter the modulated signal.

```
hFilt = comm.RaisedCosineTransmitFilter('OutputSamplesPerSymbol', sps);
z = step(hFilt, x);
```

Measure the ACPR for the filtered signal, z . You can see that the ACPR improves from -9.5 dB to -17.7 dB when raised cosine pulses are used.

```
ACPR = step(hACPR, z)
```

```
ACPR =
    -17.7338
```

Plot the adjacent channel power ratios for a range of adjacent channel offsets. Set the channel offsets to range from 30 kHz to 70 kHz in 10 kHz steps. Recall that you must first release hACPR to change the offset.

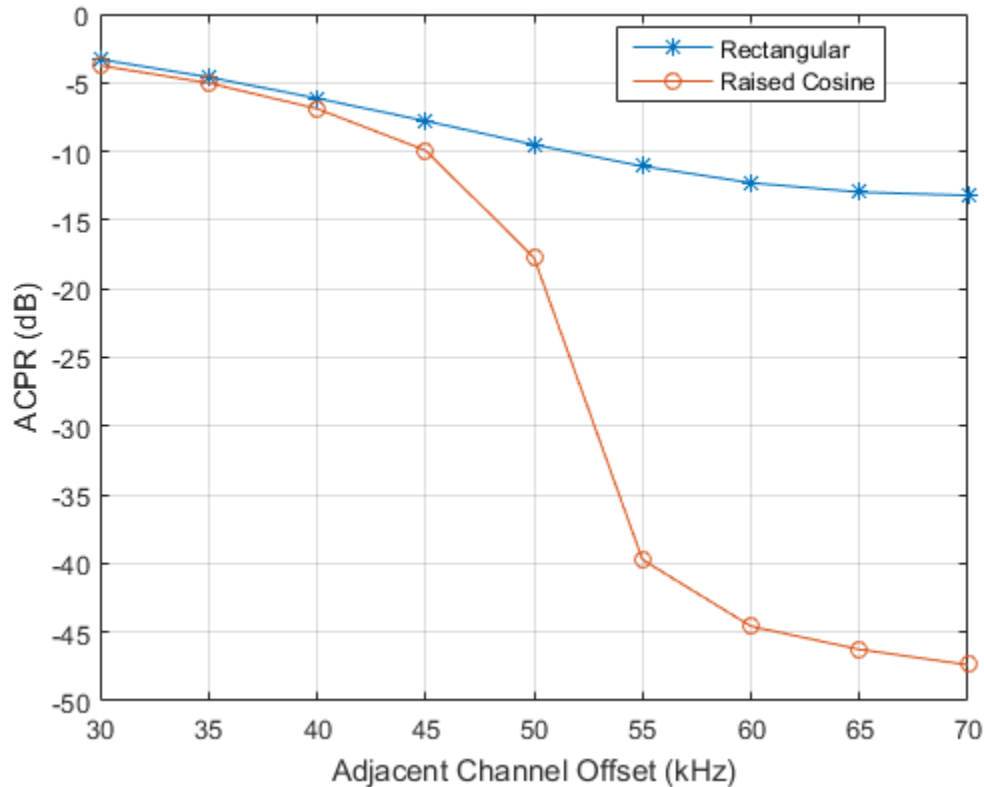
```
freqOffset = 1e3*(30:5:70);
release(hACPR)
hACPR.AdjacentChannelOffset = freqOffset;
```

Determine the ACPR values for the signals with rectangular and raised cosine pulse shapes.

```
ACPR1 = step(hACPR, y);
ACPR2 = step(hACPR, z);
```

Plot the adjacent channel power ratios.

```
plot(freqOffset/1000, ACPR1, '*-', freqOffset/1000, ACPR2, 'o-')
xlabel('Adjacent Channel Offset (kHz)')
ylabel('ACPR (dB)')
legend('Rectangular', 'Raised Cosine', 'location', 'best')
grid
```

Return the random number generator to its initial state.

```
rng(prevState)
```

CCDF Measurements

This example shows how to use the Complementary Cumulative Distribution Function (CCDF) System object to measure the probability of a signal's instantaneous power being greater than a specified level over its average power. Construct the `comm.CCDF` object, enable the PAPR output port, and set the maximum signal power limit to 50 dBm.

```
hCCDF = comm.CCDF('PAPROutputPort',true, ...  
    'MaximumPowerLimit', 50);
```

Create a 64-QAM modulator and an OFDM modulator. The QAM modulated signal will be evaluated by itself and evaluated again after OFDM modulation is applied.

```
hMod = comm.RectangularQAMModulator('ModulationOrder',64);  
hOFDM = comm.OFDMModulator('FFTLength', 256, 'CyclicPrefixLength', 32);
```

Determine the input and output sizes of the OFDM modulator object using the `info` method of the `comm.OFDMModulator` object.

```
info(hOFDM)  
ofdmInputSize = hOFDM.info.DataInputSize;  
ofdmOutputSize = hOFDM.info.OutputSize;
```

```
ans =
```

```
    DataInputSize: [245 1]  
    OutputSize: [288 1]
```

Set the number of OFDM frames.

```
numFrames = 20;
```

Allocate memory for the signal arrays.

```
qamSig = repmat(zeros(ofdmInputSize), numFrames, 1);  
ofdmSig = repmat(zeros(ofdmOutputSize), numFrames, 1);
```

Use the default random number generator to ensure repeatability.

```
rng default
```

Generate the 64-QAM and OFDM signals for evaluation.

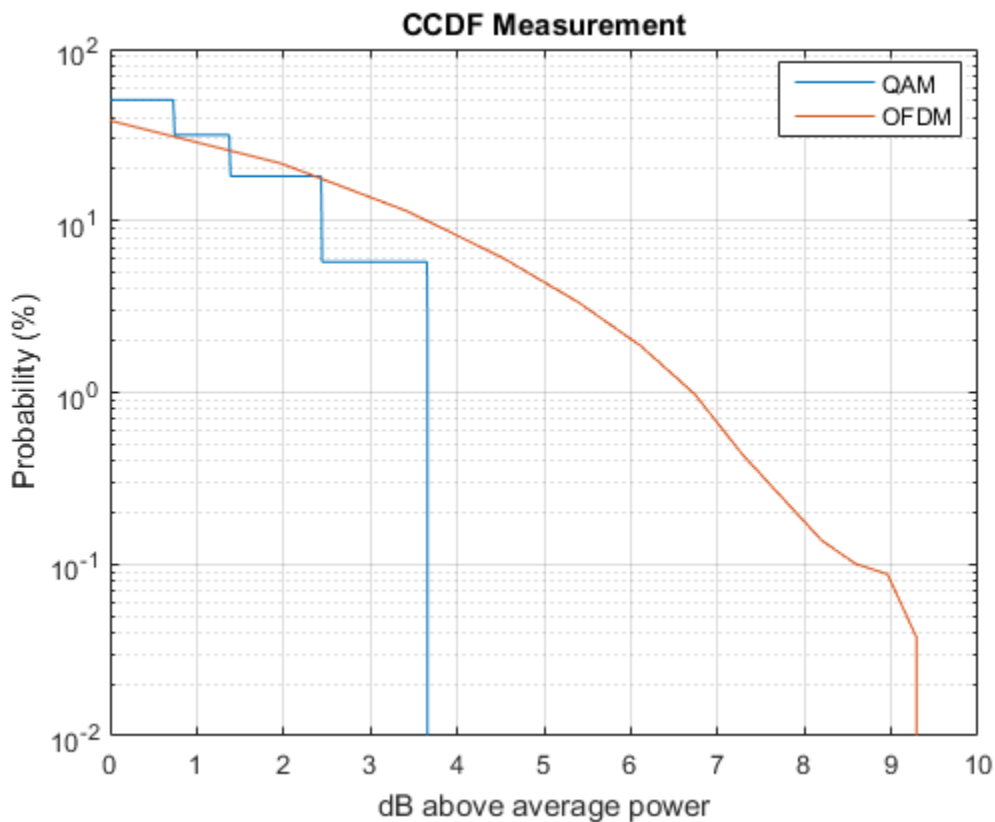
```
for k = 1:numFrames  
    % Generate random data symbols  
    data = randi([0, 63], ofdmInputSize);  
    % Apply 64-QAM modulation  
    tmpQAM = step(hMod, data);  
    % Apply OFDM modulation to the QAM-modulated signal  
    tmpOFDM = step(hOFDM, tmpQAM);  
    % Save the signal data  
    qamSig((1:ofdmInputSize)+(k-1)*ofdmInputSize(1)) = tmpQAM;  
    ofdmSig((1:ofdmOutputSize)+(k-1)*ofdmOutputSize(1)) = tmpOFDM;  
end
```

Determine the average signal power, the peak signal power, and the PAPR ratios for the two signals. The two signals being evaluated must be the same length so the first 4000 symbols are evaluated.

```
[Fy, Fx, PAPR] = step(hCCDF, [qamSig(1:4000), ...  
    ofdmSig(1:4000)]);
```

Plot the CCDF data. Observe that the likelihood of the power of the OFDM modulated signal being more than 3 dB above its average power level is much higher than for the QAM modulated signal.

```
plot(hCCDF)  
legend('QAM', 'OFDM', 'location', 'best')
```



Compare the PAPR values for the QAM modulated and OFDM modulated signals.

```
fprintf('\nPAPR for 64-QAM = %5.2f dB\nPAPR for OFDM = %5.2f dB\n', ...  
        PAPR(1), PAPR(2))
```

```
PAPR for 64-QAM = 3.65 dB  
PAPR for OFDM = 9.44 dB
```

You can see that by applying OFDM modulation to a 64-QAM modulated signal, the PAPR increases by 5.8 dB. This means that if 30 dBm transmit power is needed to close a 64-QAM link, the power amplifier needs to have a maximum power of 33.7 dBm to ensure linear operation. If the same signal were then OFDM modulated, a 39.5 dBm power amplifier is required.

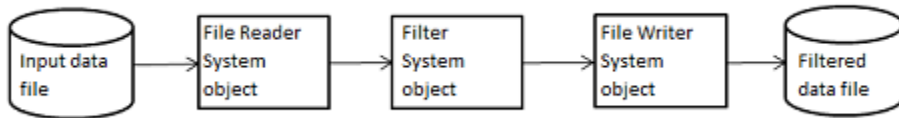
System Objects

- “What Are System Objects?” on page 4-2
- “System Objects in Simulink” on page 4-3
- “System Object Methods” on page 4-4

What Are System Objects?

A System object is a specialized kind of MATLAB object. System Toolboxes include System objects and most System Toolboxes also have MATLAB functions and Simulink blocks. System objects are designed specifically for implementing and simulating dynamic systems with inputs that change over time. Many signal processing, communications, and controls systems are dynamic. In a dynamic system, the values of the output signals depend on both the instantaneous values of the input signals and on the past behavior of the system. System objects use internal states to store that past behavior, which is used in the next computational step. As a result, System objects are optimized for iterative computations that process large streams of data, such as video and audio processing systems.

For example, you could use System objects in a system that reads data from a file, filters that data and then writes the filtered output to another file. Typically, a specified amount of data is passed to the filter in each loop iteration. The file reader object uses a state to keep track of where in the file to begin the next data read. Likewise, the file writer object keeps tracks of where it last wrote data to the output file so that data is not overwritten. The filter object maintains its own internal states to assure that the filtering is performed correctly. This diagram represents a single loop of the system.



Many System objects support:

- Fixed-point arithmetic (requires a Fixed-Point Designer™ license)
- C code generation (requires a MATLAB Coder or Simulink Coder license)
- HDL code generation (requires an HDL Coder™ license)
- Executable files or shared libraries generation (requires a MATLAB Compiler™ license)

Note: Check your product documentation to confirm fixed-point, code generation, and MATLAB Compiler support for the specific System objects you want to use.

System Objects in Simulink

System Objects in the MATLAB Function Block

You can include System object code in Simulink models using the `MATLAB Function` block. Your function can include one or more System objects. Portions of your system may be easier to implement in the MATLAB environment than directly in Simulink. Many System objects have Simulink block counterparts with equivalent functionality. Before writing MATLAB code to include in a Simulink model, check for existing blocks that perform the desired operation.

System Object Methods

In this section...
“What Are System Object Methods?” on page 4-4
“The Step Method” on page 4-4
“Common Methods” on page 4-5

What Are System Object Methods?

After you create a System object, you use various object methods to process data or obtain information from or about the object. All methods that are applicable to an object are described in the reference pages for that object. System object method names begin with a lowercase letter and class and property names begin with an uppercase letter. The syntax for using methods is `<method>(<handle>)`, such as `step(H)`, plus possible extra input arguments.

System objects use a minimum of two commands to process data—a constructor to create the object and the step method to run data through the object. This separation of declaration from execution lets you create multiple, persistent, reusable objects, each with different settings. Using this approach avoids repeated input validation and verification, allows for easy use within a programming loop, and improves overall performance. In contrast, MATLAB functions must validate parameters every time you call the function.

These advantages make System objects particularly well suited for processing streaming data, where segments of a continuous data stream are processed iteratively. This ability to process streaming data provides the advantage of not having to hold large amounts of data in memory. Use of streaming data also allows you to use simplified programs that use loops efficiently.

The Step Method

The `step` method is the key System object method. You use `step` to process data using the algorithm defined by that object. The `step` method performs other important tasks related to data processing, such as initialization and handling object states. Every System object has its own customized `step` method, which is described in detail on the `step` reference page for that object. For more information about the step method and other available methods, see the descriptions in “Common Methods” on page 4-5.

Common Methods

All System objects support the following methods, each of which is described in a method reference page associated with the particular object. In cases where a method is not applicable to a particular object, calling that method has no effect on the object.

Method	Description
<code>step</code>	Processes data using the algorithm defined by the object. As part of this processing, it initializes needed resources, returns outputs, and updates the object states. After you call the <code>step</code> method, you cannot change any input specifications (i.e., dimensions, data type, complexity). During execution, you can change only tunable properties. The <code>step</code> method returns regular MATLAB variables. Example: <code>Y = step(H,X)</code>
<code>release</code>	Releases any special resources allocated by the object, such as file handles and device drivers, and unlocks the object. For System objects, use the <code>release</code> method instead of a destructor.
<code>reset</code>	Resets the internal states of a locked object to the initial values for that object and leaves the object locked
<code>getNumInputs</code>	Returns the number of inputs (excluding the object itself) expected by the <code>step</code> method. This number varies for an object depending on whether any properties enable additional inputs.
<code>getNumOutputs</code>	Returns the number of outputs expected from the <code>step</code> method. This number varies for an object depending on whether any properties enable additional outputs.
<code>getDiscreteState</code>	Returns the discrete states of the object in a structure. If the object is unlocked (when the object is first created and before you have run the <code>step</code> method on it or after you have released the object), the states are empty. If the object has no discrete states, <code>getDiscreteState</code> returns an empty structure.
<code>clone</code>	Creates another object of the same type with the same property values
<code>isLocked</code>	Returns a logical value indicating whether the object is locked.

Method	Description
isDone	Applies to source objects only. Returns a logical value indicating whether the step method has reached the end of the data file. If a particular object does not have end-of-data capability, this method value returns false .
info	Returns a structure containing characteristic information about the object. The fields of this structure vary depending on the object. If a particular object does not have characteristic information, the structure is empty.