

SimRF™

Getting Started Guide

R2012b

**MATLAB®
& SIMULINK®**

How to Contact MathWorks



www.mathworks.com Web
comp.soft-sys.matlab Newsgroup
www.mathworks.com/contact_TS.html Technical Support



suggest@mathworks.com Product enhancement suggestions
bugs@mathworks.com Bug reports
doc@mathworks.com Documentation error reports
service@mathworks.com Order status, license renewals, passcodes
info@mathworks.com Sales, pricing, and general information



508-647-7000 (Phone)



508-647-7001 (Fax)



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

For contact information about worldwide offices, see the MathWorks Web site.

SimRF™ Getting Started Guide

© COPYRIGHT 2010–2012 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

September 2010	Online only	New for Version 3.0 (Release 2010b)
April 2011	Online only	Revised for Version 3.0.2 (Release 2011a)
September 2011	Online only	Revised for Version 3.1 (Release 2011b)
March 2012	Online only	Revised for Version 3.2 (Release 2012a)
September 2012	Online only	Revised for Version 3.3 (Release 2012b)

Circuit Envelope

Introduction to Circuit Envelope Simulation

1

Product Description	1-2
Key Features	1-2
Minimize Computations for RF Simulations	1-3
Simulate a Passband Signal in Simulink Software	1-4
Compare Passband and Baseband Signals in SimRF Software	1-6
Simulate the Envelope of a Baseband Signal	1-9
Using SimRF Software for the First Time	1-12
Expected Background	1-12
Circuit Envelope, Equivalent Baseband, and Idealized Baseband Library Features	1-12
Understanding the SimRF Environment	1-14

Frequency Conversion

2

Specify the Circuit Envelope Simulation	
Environment	2-2
Access Circuit Envelope Libraries	2-2
Ground the RF System	2-3
Configure the SimRF Environment	2-3
Simulate the SimRF Environment	2-4
Model an RF Mixer	2-6
Build the RF System	2-7
Probe Circuit Envelopes Waveforms	2-7

Define Model Variables	2-8
Specify Block Parameters for RF Simulation	2-11
Simulate Frequency Conversion in an RF Mixer	2-12
Filter Mixing Products	2-14
Probe Multiple RF Carriers	2-15
Model an RF Filter	2-16
Simulate Filtering of RF Signals	2-17

Equivalent Baseband

Introduction to Equivalent Baseband Simulation

3

SimRF Equivalent Baseband Libraries	3-2
Overview of SimRF Equivalent Baseband Libraries	3-2
Open SimRF Equivalent Baseband Libraries	3-3
Physical Library	3-4
Mathematical Library	3-6
Equivalent Baseband Workflow	3-8
Model an LC Bandpass Filter	3-10
Overview of LC Bandpass Filter Example	3-10
Select Blocks to Represent System Components	3-10
Build the Model	3-11
Specify Model Parameters	3-13
Validate Filter Components and Run the Simulation	3-20
Analyze the Simulation Results	3-22

Circuit Envelope

Introduction to Circuit Envelope Simulation

- “Product Description” on page 1-2
- “Minimize Computations for RF Simulations” on page 1-3
- “Using SimRF Software for the First Time” on page 1-12

Product Description

Design and simulate RF systems

SimRF™ provides a component library and simulation engine for designing RF systems. It includes mixers, amplifiers, S-parameter blocks, and other basic blocks for architectural design and modeling of wireless transceivers. You can connect these components arbitrarily to form diverse architectures and to model system-level impairments. SimRF enables simulation of RF gain, noise, and intermodulation distortion commonly attributed to RF amplifiers. It provides circuit envelope technology for simulating RF impairments uniquely associated with mixers, including image rejection, reciprocal mixing, local oscillator phase offsets, DC conversion, and DC offsets. Equivalent baseband technology is also included, enabling fast simulation of single-carrier cascaded systems.

Key Features

- Multiple carrier-frequency models
- General N-port models and S-parameter data files
- Passive components, including resistors, capacitors, inductors, and general impedance blocks
- Weakly nonlinear 3-port mixers and 2-port amplifiers specified by noise figure, IP2, IP3, and data files
- Intra-model signal probing
- Circuit envelop technology for high-fidelity simulation
- Equivalent baseband technology for fast simulation

Minimize Computations for RF Simulations

In this section...

“Simulate a Passband Signal in Simulink Software” on page 1-4

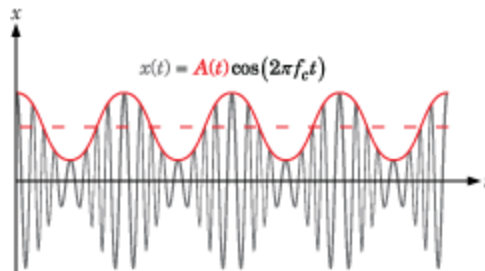
“Compare Passband and Baseband Signals in SimRF Software” on page 1-6

“Simulate the Envelope of a Baseband Signal” on page 1-9

This example shows how to use circuit-envelope simulation to reduce the number of time steps required to simulate a simple signal by a factor of 100.

Traditional methods of circuit simulation are not ideal for RF systems. In Simulink®, simulating signals with high-frequency components requires choosing a proportionately small time step. However, the *envelope* of such signals often varies on a much larger time scale. The SimRF circuit-envelope environment takes advantage of this condition in order to model RF signals accurately, while also reducing simulation time.

The following figure illustrates a signal $x(t)$ for which circuit-envelope simulation is ideal:



The signal consists of a time-varying modulating signal on a high-frequency carrier. In many RF applications, the frequency of the modulating signal $A(t)$ is much less than the frequency of the carrier, f_c .

SimRF software handles the carrier $\cos(2\pi f_c t)$ analytically, so it only needs to simulate the modulating signal. The redefined system that the software uses is mathematically equivalent to the original. However, by taking time

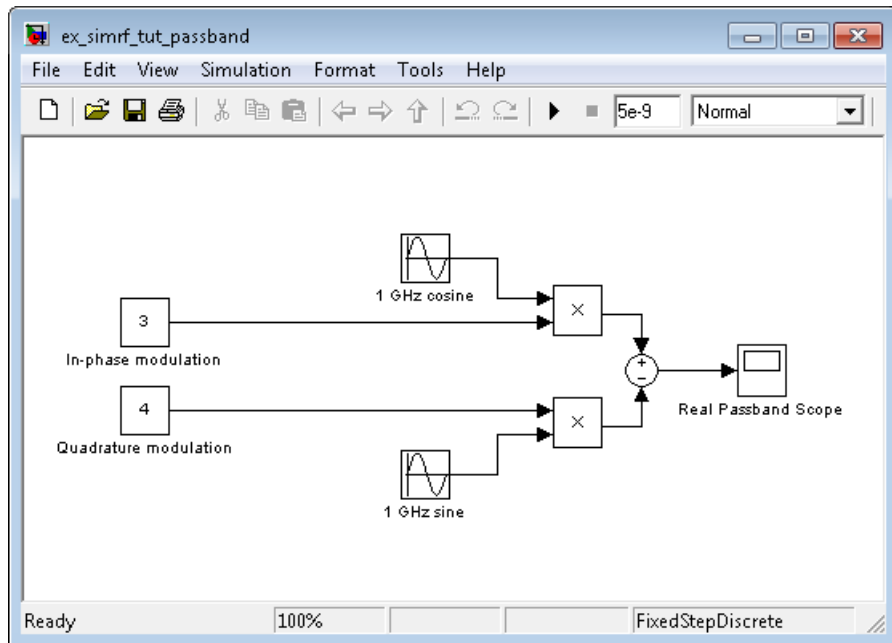
steps on the scale of the modulating signal instead of the carrier, the software produces equivalent results in less time.

Simulate a Passband Signal in Simulink Software

The model

ex_simrf_tut_passband

simulates a 1-GHz sine wave using fundamental Simulink blocks. This section of the example shows you how to modulate a real passband signal with in-phase and quadrature components.



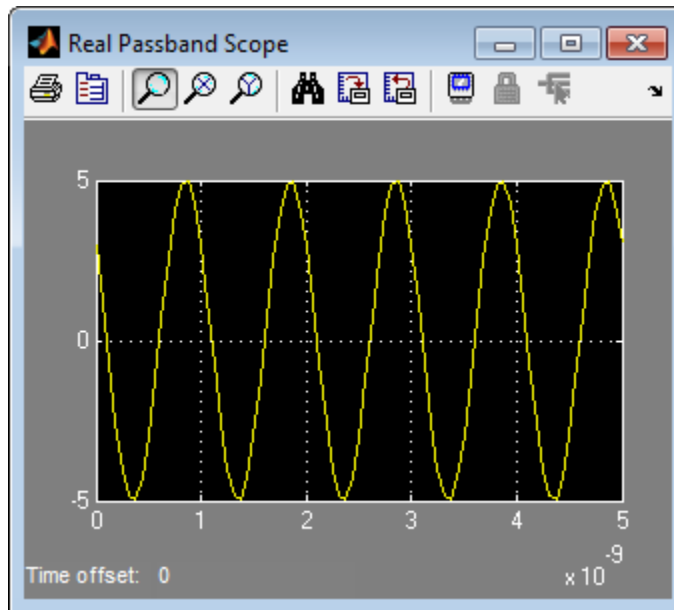
The system specifies a real passband signal $x(t)$ according to the formula

$$x(t) = I(t) \cos(2\pi f_c t) - Q(t) \sin(2\pi f_c t)$$

where:

- $I(t)$ is the in-phase part of the modulating signal, equal to 3 in this example, modeled by the Constant block labeled In-phase modulation.
- $Q(t)$ is the quadrature part of the modulating, equal to 4 in this example, modeled by the Constant block labeled Quadrature modulation.
- f_c is the carrier frequency, equal to 1 GHz in this example.

Running the model produces the following output on the scope.



The output signal at the Real Passband Scope has a magnitude of 5 and a phase shift of $\text{atan2d}(3, -4)$, or about 143° .

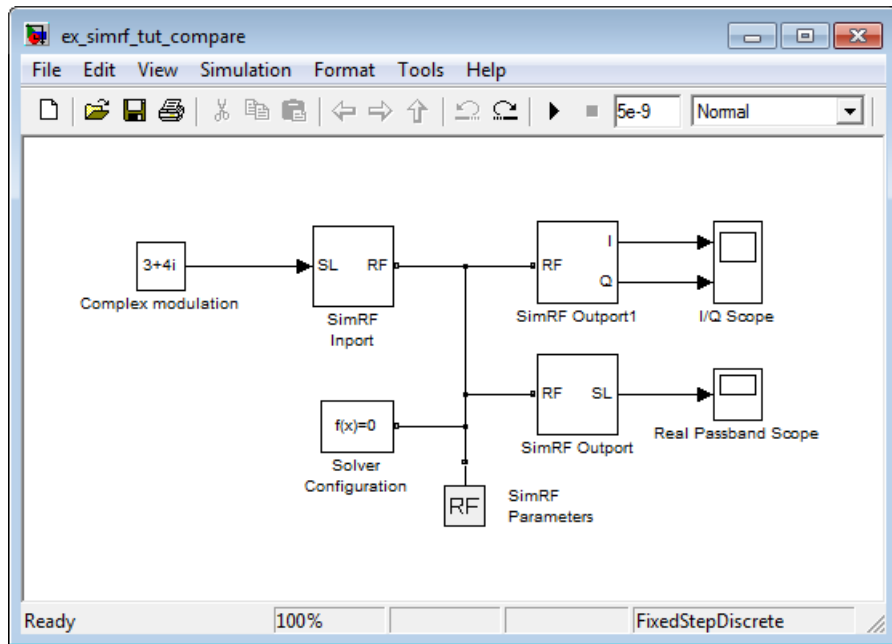
In the Configuration Parameters dialog box, the **Fixed-step size (fundamental sample time)** parameter has been set to $1/20 \times 10^{-9}$. This value is on the order of the wavelength of the carrier. The simulation takes a total of 101 samples—20 per cycle.

Compare Passband and Baseband Signals in SimRF Software

The model

ex_simrf_tut_compare

simulates a 1-GHz sine wave using SimRF blocks. This example builds on the results of the previous section, “Simulate a Passband Signal in Simulink Software” on page 1-4.



The system simulates a real passband signal as the real part of a complex passband signal according to the formula

$$x(t) = \text{Re}[(I(t) + jQ(t))\exp(j \cdot 2\pi f_c t)] = I(t)\cos(2\pi f_c t) - Q(t)\sin(2\pi f_c t)$$

where:

- $I(t) + jQ(t)$ is the complex-valued modulation, equal to $3 + 4j$.

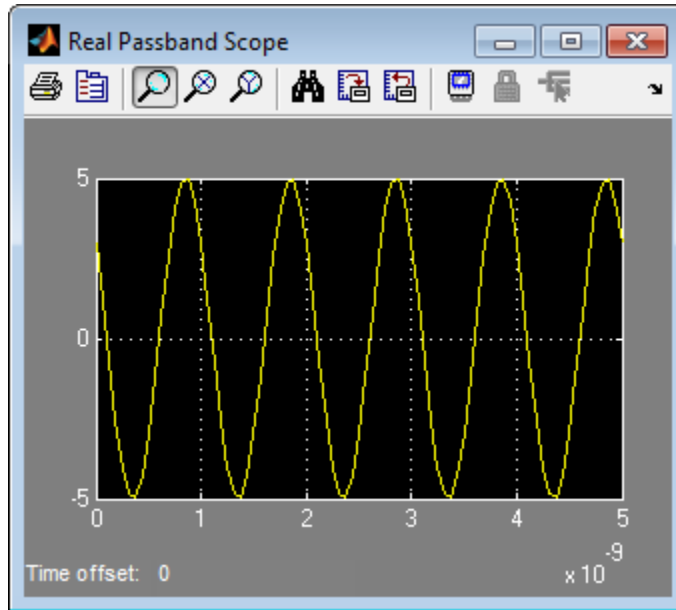
- f_c is the angular frequency, equal to 1 GHz.

Contrary to the Simulink passband implementation in the previous section, the complex baseband signal driving the SimRF system does not include the carrier. Instead, the SimRF environment handles the carrier analytically. The carrier appears in four different blocks in the SimRF environment:

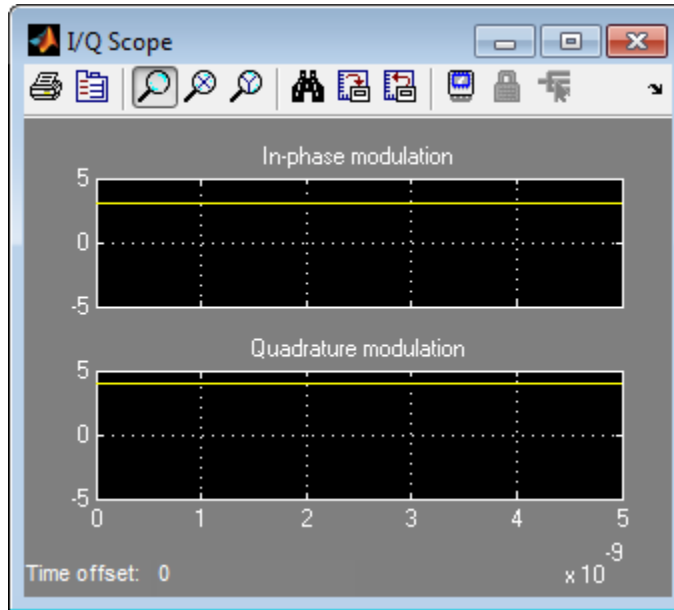
- In the SimRF Inport block, the **Carrier frequencies** parameter defines the carrier frequencies of the modulations entering from outside the SimRF environment. In this example, there is only one input signal, and only one carrier (1 GHz, specified as `1e9 Hz`).
- In the SimRF Outport block, the **Carrier frequencies** parameter specifies the signal on the `1e9 Hz` carrier (1 GHz) as Simulink signals. These signals appear at the I and Q ports. The **Output** parameter is set to **Real Passband**, so this signal represents a real passband signal on the 1-GHz carrier.
- In the block labeled SimRF Outport1 block, also a SimRF Outport block, the **Carrier frequencies** parameter specifies the signal outputted on the `1e9 Hz` carrier (1 GHz) as Simulink signals. These signals appear at the I and Q ports. The **Output** parameter is set to **In-phase and Quadrature Baseband**, so these signals represent the in-phase and quadrature modulations of the signal on the 1-GHz carrier.
- In the SimRF Parameters block, the **Carrier frequencies** parameter specifies all of the carriers to be modeled in the SimRF circuit-envelope simulation environment. In this example, only one carrier is specified.

Note The Solver Configuration block, required in SimRF models, retains its default settings.

Running the model produces the following output on the scopes.



The Real Passband Scope displays the same output as the example in the previous section, “Simulate a Passband Signal in Simulink Software” on page 1-4. The signal has a magnitude of 5 and a phase shift consistent with the specified in-phase and quadrature amplitudes.



The 1-GHz carrier itself does not appear in the output. The results correspond to the real and imaginary parts of the Complex modulation at the input of the system. They also correspond to the In-phase modulation and Quadrature modulation blocks in “Simulate a Passband Signal in Simulink Software” on page 1-4.

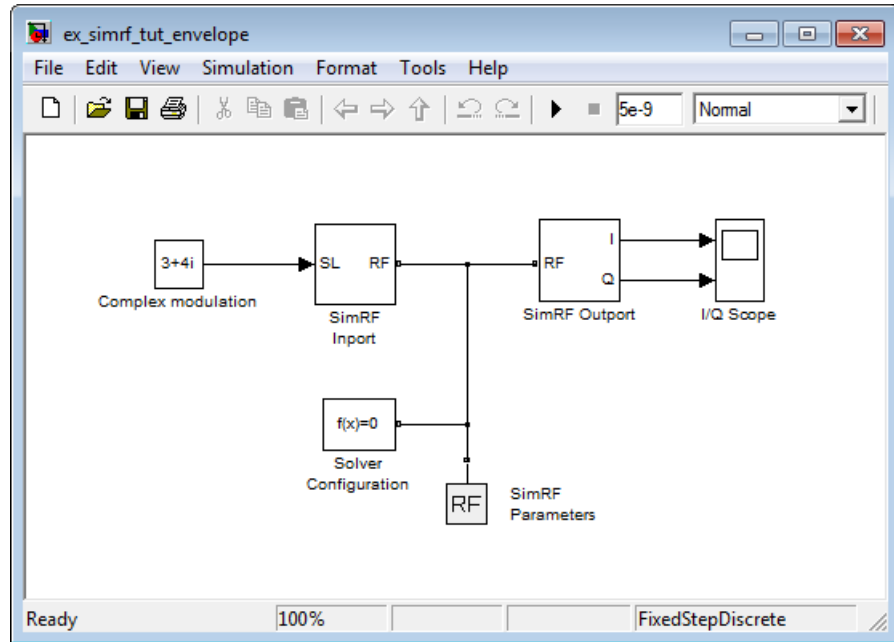
In the Configuration Parameters dialog box, the **Fixed-step size (fundamental sample time)** parameter has been set to $1/20 \times 10^{-9}$. This value is on the order of the wavelength of the carrier. The simulation takes a total of 101 samples—20 per cycle.

Simulate the Envelope of a Baseband Signal

The model

```
ex_simrf_tut_envelope
```

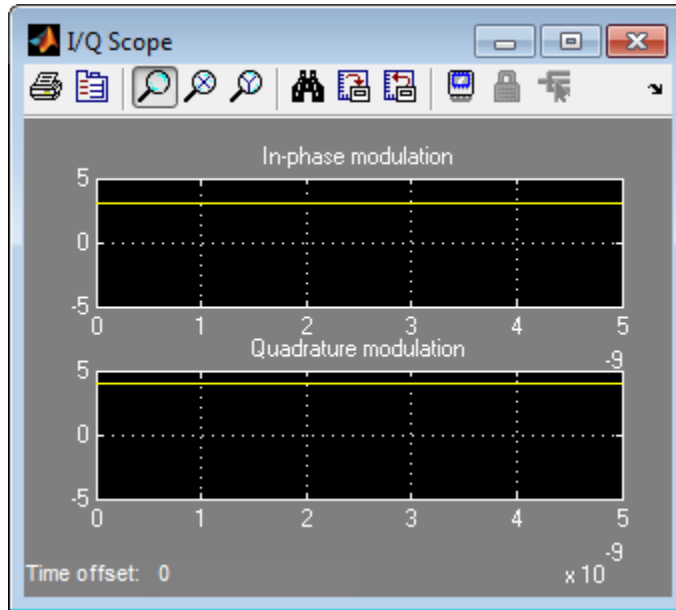
simulates the envelope of a 1-GHz sine wave using SimRF blocks. This example builds on the results of the previous section, “Compare Passband and Baseband Signals in SimRF Software” on page 1-6.



The system is almost identical to the system in the previous section, except:

- The model contains only one SimRF Output block and only one scope. The SimRF environment outputs in-phase and quadrature modulations of the 1-GHz signal. In the SimRF Output block, the **Output** parameter is set to In-phase and quadrature baseband. Since the system is not configured to output a real passband signal, the carrier is not simulated.
- In the Configuration Parameters dialog box, the **Fixed-step size (fundamental sample time)** parameter is greater. Its value is $5e-9$ instead of $1/20 \cdot 1e-9$.

Running the model produces the following output at the scope.



The I/Q Scope displays the in-phase and quadrature baseband components of the 1-GHz signal. The 1-GHz carrier itself does not appear in the output. The results correspond to the real and imaginary parts of the Complex modulation at the input of the system.

In contrast to the models in the previous two sections, Simulink works differently in this model. Because the modulating signals are constant in this example, only two sample points are needed. To simulate a time-varying modulating signal, Simulink can use a fixed time step on the order of the reciprocal of its bandwidth.

The model uses a value of $5e-9$ for the **Fixed-step size (fundamental sample time)** parameter. This value equals the **Stop time** because, in this case, the modulating signals are constant. Compared to the preceding examples, which use a sample time of $1/20 \cdot 1e-9$, this model simulates accurately with a time step 100 times larger. This step size results in a reduction of total sample time by a factor of 100, excluding the initial time step at time 0.

Using SimRF Software for the First Time

In this section...

“Expected Background” on page 1-12

“Circuit Envelope, Equivalent Baseband, and Idealized Baseband Library Features” on page 1-12

“Understanding the SimRF Environment” on page 1-14

Expected Background

Topics in the SimRF documentation assume that you are already familiar with:

- Using MATLAB® to write and execute scripts and functions.
- Using Simulink to create and simulate block diagrams.

Circuit Envelope, Equivalent Baseband, and Idealized Baseband Library Features

SimRF software offers Circuit Envelope, Equivalent Baseband, and Idealized Baseband libraries for modeling RF networks. Each library represents a distinct simulation paradigm. For certain applications, one library may offer an advantage over another.

- Use the Circuit Envelope library for large, multicarrier simulation of RF networks with arbitrary topologies.
- Use the Equivalent Baseband library for single-carrier simulation of cascaded RF networks.
- Use the Idealized Baseband library for basic, behavioral simulation of highly abstracted, Simulink-domain RF devices.

Except for the cross-domain SimRF Inport, SimRF Outport, Input Port, and Output Port blocks, blocks from the three libraries do not connect to each other. Therefore, to avoid redesigning your model later, choose which library to use based on your application. You can consult the following table for a summary of the features of the three libraries.

(Continued)

Do blocks in this library...	Circuit Envelope Library	Equivalent Baseband Library	Idealized Baseband Library
Connect directly to Simulink blocks?	No, except for cross-domain blocks	No, except for cross-domain blocks	Yes
Support single-carrier simulation?	Yes	Yes	Yes
Support multicarrier simulation?	Yes	No	No
Support third-order nonlinear simulation?	Yes	Yes	No
Support fifth- and seventh-order nonlinear simulation?	No	Yes	No
Support simulation of cascaded networks?	Yes	Yes	Yes
Support simulation of networks with arbitrary topologies?	Yes	No	No
Support signal probing between input and output?	Yes	No	Yes

(Continued)

Do blocks in this library...	Circuit Envelope Library	Equivalent Baseband Library	Idealized Baseband Library
Support noise simulation?	Yes	Yes	No
Support Simscape™ platform features, such as local solvers?	Yes	No	No
Support specification using network parameter data?	Yes	Yes	No

Understanding the SimRF Environment

Groups of interconnected SimRF Circuit Envelope library blocks and the algorithms that model the RF system that they represent comprise the *SimRF environment*. In the SimRF environment, all blocks fall into one of the three following categories.

Blocks that Operate Within the SimRF Environment

These blocks contribute to the physical representation of an RF system. Most SimRF Circuit Envelope library blocks fall into this category, including all blocks in the SimRF Elements and Sources libraries. For example, a Resistor block can model a source impedance or part of a matching network, and an Amplifier block could model a physical RF amplifier. Both of these blocks model physical components.

Blocks within the SimRF environment connect via bidirectional *SimRF signals*. SimRF signals are an implementation of Simscape physical signals whose defined Through and Across variables represent current and voltage, respectively. For an introduction to Simscape software and physical signals, see Basic Principles of Modeling Physical Networks.

Blocks that Convert Between the SimRF and Simulink Environments

These blocks, also called *cross-domain* blocks, provide an interface from an RF system to a larger design. SimRF Inport and SimRF Outport blocks fall into this category. For example, you can construct a signal using blocks from Communications System Toolbox™ or DSP System Toolbox™ libraries, and input that signal into the SimRF environment using a SimRF Inport block.

The SimRF Parameters and Solver Configuration blocks

These blocks manipulate the environment itself. They are not strictly part of the physical system. However, they connect directly with other SimRF blocks using physical connections. To use these blocks, connect them to any part of the RF system. Because they are not part of the physical representation of the system, they have the same effect regardless of where they connect within the environment.

To run models containing SimRF blocks, you must connect a Solver Configuration block to the SimRF environment. Additionally, to simulate carriers besides 0 Hz, or to model noise in the SimRF environment, you must connect a SimRF Parameters block to the SimRF environment.

Frequency Conversion

- “Specify the Circuit Envelope Simulation Environment” on page 2-2
- “Model an RF Mixer” on page 2-6
- “Filter Mixing Products” on page 2-14

Specify the Circuit Envelope Simulation Environment

In this section...

“Access Circuit Envelope Libraries” on page 2-2

“Ground the RF System” on page 2-3

“Configure the SimRF Environment” on page 2-3

“Simulate the SimRF Environment” on page 2-4

This section of the tutorial shows you how to:

- Use a Solver Configuration block to provide local solver settings to the SimRF environment
- Use a SimRF Parameters block to define the carriers and global noise behavior in the SimRF environment.
- Use a Ground block to reference a model to electrical ground.

The Solver Configuration and Ground block are the bare minimum that SimRF software requires in order to perform circuit-envelope simulation. Models that are missing one of these two blocks do not run. In order to simulate the SimRF environment at carrier frequencies other than 0 Hz, a model must also contain a SimRF Parameters block.

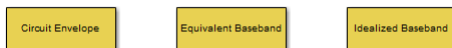
To begin, open a new model by selecting **File > New > Model**.

Access Circuit Envelope Libraries

Open the SimRF library by typing

```
simrflib
```

at the MATLAB Command Window prompt.



SimRF Library 3.3
Copyright 2003-2012 The MathWorks, Inc.

The SimRF library contains three sublibraries:

- The Circuit Envelope library contains blocks for circuit-envelope simulation. This manual often refers to the Circuit Envelope library as the *SimRF library*. The *SimRF Getting Started Guide* and *SimRF User's Guide* pertain exclusively to this library.
- The Equivalent Baseband and Idealized Baseband library contain blocks for equivalent-baseband simulation. Refer to the Equivalent Baseband section of the SimRF documentation for more information on using these blocks.

To continue, double-click the Circuit Envelope library.

Ground the RF System

Place a Ground block in your model:

- 1 In the SimRF library, double-click Elements to open the SimRF Elements library.
- 2 Drag and drop a Ground block onto your model. In the library, the block is abbreviated 'Gnd'.

Connecting SimRF models to ground provides an electrical reference for the system, from which node voltages are defined. A SimRF model is grounded if:

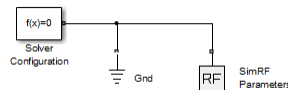
- The model contains a Ground block that connects to the SimRF Environment, or
- The model contains an Amplifier, Continuous Wave, Mixer, Noise, Sinusoid, or S-Parameters block that is internally grounded. To internally ground one of these blocks, select the **Ground and hide negative terminals** check box in the block dialog box.

Configure the SimRF Environment

To continue the tutorial:

- 1 Place a Solver Configuration block in your model. In the SimRF library, double-click Utilities to open the SimRF Utilities library, then drag and drop a Solver Configuration block onto your model.

- 2 Place a SimRF Parameters block in your model. If the SimRF Utilities library is not already open, double-click Utilities to open it, then drag and drop a SimRF Parameters block onto your model.
- 3 Connect the Ground, Solver Configuration, and SimRF Parameters blocks to each other. The following figure shows one possible configuration.



A model can contain multiple Solver Configuration and SimRF Parameters if all of the following conditions are met:

- Each Solver Configuration and SimRF Parameters block connects to a separate SimRF environment.
- No two Solver Configuration or SimRF Parameters blocks connect to the same environment.
- The separate SimRF environments are isolated from each other in separate subsystems.

Warning Simulating more than one SimRF environment without using subsystems is not supported.

Simulate the SimRF Environment

To run the model, select **Simulation > Run**.

The model simulates according to the default settings of the Solver Configuration and SimRF Parameters blocks and produces no output:

- In the SimRF Parameters block dialog, the **Carrier frequencies** parameter specifies a carrier of 0 Hz in the system. In the next section, “Model an RF Mixer” on page 2-6, you specify additional carriers for circuit-envelope simulation.
- In the Solver Configuration block dialog, the **Use local solver** check box is cleared, so the model uses the specified global solver. The examples in the

SimRF User's Guide show you how to configure the SimRF environment to use a local solver.

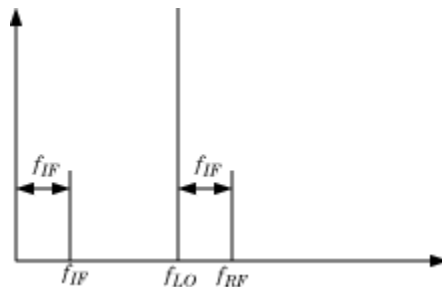
Model an RF Mixer

In this section...

- “Build the RF System” on page 2-7
- “Probe Circuit Envelopes Waveforms” on page 2-7
- “Define Model Variables” on page 2-8
- “Specify Block Parameters for RF Simulation” on page 2-11
- “Simulate Frequency Conversion in an RF Mixer” on page 2-12

This section of the tutorial shows you how to:

- Use a Sinusoid block to model the envelope of an amplitude-modulated (AM) waveform.
- Use a Continuous Wave block to model the constant envelope of an ideal local oscillator (LO).
- Use a Mixer block to downconvert the AM waveform to an intermediate frequency (IF).



The preceding figure illustrates frequency conversion. A signal modulated on the carrier f_{RF} mixes with a local oscillator at f_{LO} . The operation downconverts f_{RF} to $f_{IF} = f_{RF} - f_{LO}$. The upper mixing product $f_{RF} + f_{LO}$ is not modeled.

To begin, open the model you created in the previous section, “Specify the Circuit Envelope Simulation Environment” on page 2-2, or type

```
ex_simrf_tut_env
```

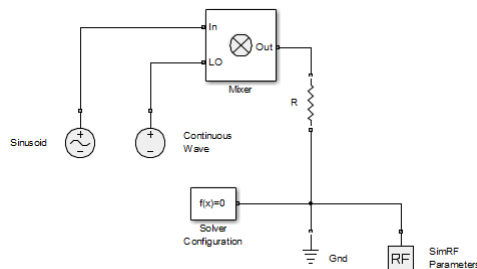
at the MATLAB Command Window prompt.

Build the RF System

Add blocks to the SimRF environment that create a working model for frequency conversion:

- From the SimRF Sources library, drag and drop a Continuous Wave block and a Sinusoid block onto your model.
- From the SimRF Elements library, drag and drop a Mixer block and a Resistor block onto your model.

Connect the blocks as shown in the following figure.



This configuration specifies an AM waveform at the input port of the mixer, an LO at the LO port of the mixer, and a 50- Ω termination at the output of the mixer.

For more information on organizing blocks on the Simulink canvas, see “Change the Appearance of a Block” in the Simulink User’s Guide.

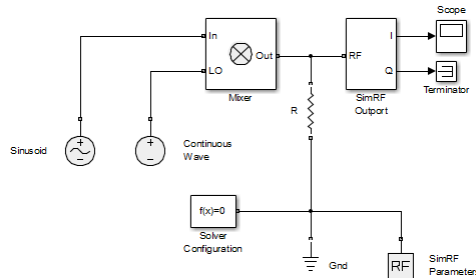
Probe Circuit Envelopes Waveforms

At the output of the mixer:

- From the SimRF Utilities library, drag and drop a SimRF Outport block onto your model.

- From the Simulink Commonly Used Blocks library, drag and drop a Scope block and a Terminator block onto your model.

Connect the blocks as shown in the following figure.



This configuration uses a SimRF Output block as a voltage sensor at the output port of the mixer. The Simulink signal at the output is the envelope of the carrier or carriers specified in the block. A scope attached to the output plots the envelope. The **Output** parameter controls how signals are presented at the output ports. To change the appearance of the block, follow one of the workflows in the “Specify Block Parameters for RF Simulation” on page 2-11 section of this tutorial.

Define Model Variables

For models with blocks that share parameter values, specifying parameters values using variables saves time and effort. For example, a frequency that is specified in the **Carrier frequencies** parameter of a SimRF Inport block must also be specified SimRF Parameters block also appears in the **Carrier frequencies** parameter of a block that uses that carrier. If you specify the parameter with a variable for the model using one of the following procedures.

To define model variables, use one of the following workflows.

Using an Initialization Function to Define Variables

Simulink models run MATLAB code stored in the initialization function (`InitFcn`) each time the model starts. The MATLAB code runs in the base workspace. If the initialization function stores variables in the MATLAB

workspace, the variables are overwritten every time the model executes the initialization function.

Most models in the *SimRF User's Guide* use the `InitFcn` to define model variables.

- 1** Open the Model Properties dialog box by selecting **File > Model Properties**.
- 2** In the **Callbacks** tab, within the **Model callbacks** pane, select the `InitFcn` node.
- 3** In the **Model initialization function** pane, enter the following MATLAB code:

```
modulationAmplitude = 1;  
modulationFrequency = 5e5;  
LOAmplitude = 1;  
LOFrequency = .95e9;  
RFCarrier = 1e9;
```

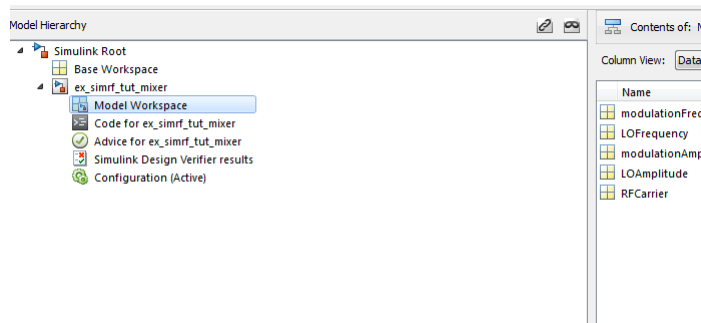
For more information, see Using Callback Functions in the *Simulink User's Guide*.

Using the Model Explorer to Define Variables

From within the Model Explorer, add variables directly to the model workspace. Model workspace variables do not interact with the base MATLAB workspace.

- 1** Open the Model Explorer window by selecting **View > Model Explorer**, or by right-clicking a block in your model and from the context menu, selecting **Explore**.
- 2** On the **Model Hierarchy** pane, select the Model Workspace node for your model.
- 3** On the **Contents** pane, add the following variables to the workspace by selecting **Add > MATLAB Variable** and editing the variable attributes.

Name	Value
modulationAmplitude	1
modulationFrequency	500000
LOAmplitude	1
LOFrequency	.95e+009
RFCarrier	1e+009



For more information, see [The Model Explorer: Overview](#) in the *Simulink User's Guide*.

Using the Command Line to Define Variables

Execute the MATLAB code in this section at the command line to assign variables directly into a model workspace.

In the Command Prompt window, type:

```
hmdl = bdroot;
hws = get_param(hmdl, 'ModelWorkspace');
hws.assignin('modulationAmplitude', 1);
hws.assignin('modulationFrequency', 5e5);
hws.assignin('LOFrequency', .95e9);
hws.assignin('LOAmplitude', 1);
hws.assignin('RFCarrier', 1e9);
```


For more information on manipulating model workspace data, see Using Model Workspaces in the *Simulink User's Guide*.

Specify Block Parameters for RF Simulation

In this section, you:

- Configure the SimRF environment for multi-frequency circuit-envelope simulation by specifying the **Carrier frequencies** parameter in the SimRF Parameters block dialog box.
- Specify the attributes of the waveforms
- Configure global simulation settings.

To specify block parameters, use one of the following workflows.

Using Dialog Boxes to Set Parameters

- 1** Select **Simulation > Model Configuration Parameters** to open the Configuration Parameters dialog box. Specify the following parameters:
 - Set **Stop time** to $1e-5$.
 - Set **Solver** to `ode23t (mod. stiff/Trapezoidal)`.
- 2** Double-click the Sinusoid block to open the Sinusoid Block Parameters dialog box. Specify the following parameters:
 - Set **Sinusoidal amplitude in-phase** to `modulationAmplitude`.
 - Set **Sinusoidal modulation frequency** to `modulationFrequency`.
 - Set **Carrier frequencies** to `RFCarrier`.
- 3** Double-click the Continuous Wave block to open the block dialog box. Specify the following parameters:
 - Set **Constant in-phase value** to `LOAmplitude`.
 - Set **Carrier frequencies** to `LOFrequency`.

- 4 Double-click the SimRF Parameters block to open the block dialog box, and set **Carrier frequencies** to [RFCarrier - LOFrequency, LOFrequency, RFCarrier, RFCarrier + LOFrequency].
- 5 Double-click the SimRF Outport block to open the block dialog box. Specify the following parameters:
 - Set **Output** to In-phase and quadrature.
 - Set **Carrier frequencies** to RFCarrier - LOFrequency.

Using the Command Line to Set Parameters

Execute the MATLAB code in this section at the command line to assign variables directly into a model workspace.

In the Command Prompt window, type:

```
hmdl = bdroot;
set_param(hmdl, 'StopTime', '1e-5');
set_param(hmdl, 'Solver', 'ode23t');
set_param([hmdl '/Sinusoid'], ...
    'VA_I','modulationAmplitude', ...
    'Fmod','modulationFrequency', ...
    'CarrierFreq','RFCarrier');
set_param([hmdl '/Continuous Wave'], ...
    'IVoltage','LOAmplitude', ...
    'CarrierFreq','LOFrequency');
set_param([hmdl '/SimRF Outport'], ...
    'CarrierFreq','RFCarrier - LOFrequency', ...
    'OutputFormat','In-phase and quadrature');
set_param([hmdl '/SimRF Parameters'], ...
    'CarrierFreq',['[RFCarrier - LOFrequency, ' ...
        'LOFrequency, ' ...
        'RFCarrier, '...
        'RFCarrier + LOFrequency]']]);
```

Simulate Frequency Conversion in an RF Mixer

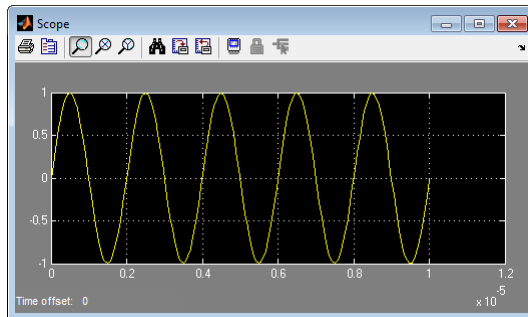
Select **Simulation > Run** to run the model.

Models that contain SimRF Amplifier and Mixer blocks generate files at update time. Before you can successfully update and run models with these blocks, you have to set up a compiler by running `mex -setup`.

By default, SimRF software generates files in the current MATLAB folder. However, you can change the output location for these files by specifying a cache folder in the Simulink Preferences dialog. To specify a cache folder:

- 1** On the **Home** tab, in the **Environment** section, click **Preferences**.
- 2** Click **Simulink**, then click **Launch Simulink Preferences**.
- 3** Specify the **Simulink cache folder** parameter.

To view the results of the simulation, double-click the scope.



The Sinusoid specifies a 1-V amplitude for the modulation on f_{RF} , which the mixer downconverts to f_{IF} . The SimRF Output block probes the intermediate frequency and recovers the 1-V modulation amplitude. This value agrees with a specified conversion gain of 0 dB in the mixer.

Filter Mixing Products

In this section...

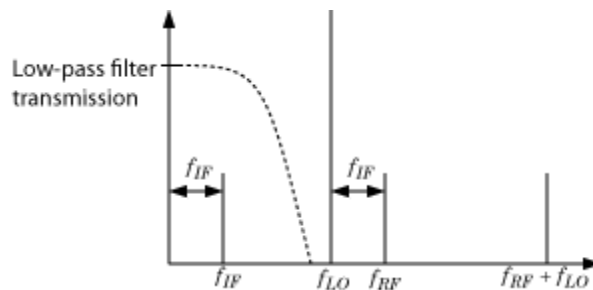
“Probe Multiple RF Carriers” on page 2-15

“Model an RF Filter” on page 2-16

“Simulate Filtering of RF Signals” on page 2-17

This section of the tutorial shows you how to:

- Configure a SimRF Outport block to probe multiple carrier frequencies simultaneously.
- Model an analog filter in the SimRF environment using SimRF Capacitor and Inductor blocks.



The preceding figure illustrates low-pass filtering of a low-side injection system. Mixing f_{RF} and f_{LO} produces signals on the carriers $f_{IF} = f_{RF} - f_{LO}$ and $f_{RF} + f_{LO}$. Adding a low-pass filter to the model reduces the power present in the high-frequency signal.

To begin, open the model that you created in the “Model an RF Mixer” on page 2-6 section, or type

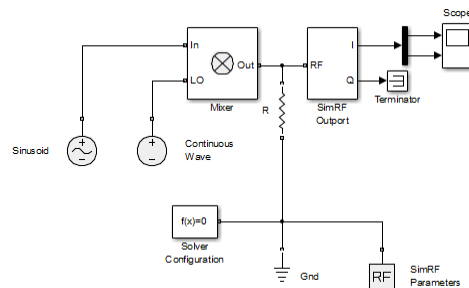
```
ex_simrf_tut_mixer
```

at the Command Window prompt.

Probe Multiple RF Carriers

The SimRF environment specifies the carriers $f_{RF} - f_{LO}$, f_{RF} , f_{LO} , and $f_{RF} + f_{LO}$, but the SimRF Outport block probes only the carrier $f_{RF} - f_{LO}$. Examine the carriers $f_{RF} - f_{LO}$ and $f_{RF} + f_{LO}$ by changing the model according to the following workflow:

- 1 From the Simulink Commonly Used Blocks library, add a Demux block to your model.
- 2 In the Scope block dialog box, click the **Parameters** button, then set **Number of Axes** to 2.
- 3 In the SimRF Outport block dialog box, set **Carrier frequencies** to [RFCarrier - LOFrequency, RFCarrier + LOFrequency]
- 4 Connect the blocks as shown in the following figure.



- 5 Select **Simulation > Run** to run the model.

Models that contain SimRF Amplifier and Mixer blocks generate files at update time. Before you can successfully update and run models with these blocks, you have to set up a compiler by running `mex -setup`.

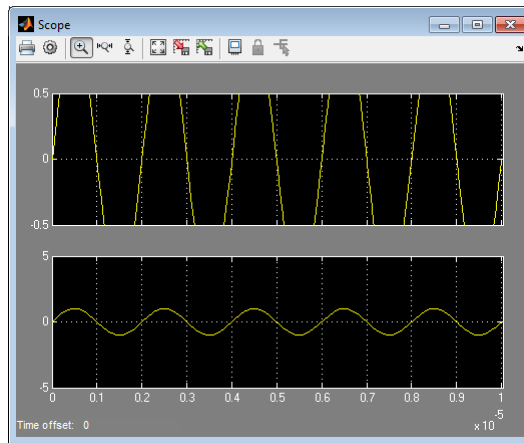
By default, SimRF software generates files in the current MATLAB folder. However, you can change the output location for these files by specifying a cache folder in the Simulink Preferences dialog. To specify a cache folder:

- 1 On the **Home** tab, in the **Environment** section, click **Preferences**.

2 Click **Simulink**, then click **Launch Simulink Preferences**.

3 Specify the **Simulink cache folder** parameter.

To view the results of the simulation, double-click the scope.



The first set of axes displays the modulation of specified carrier $f_{IF} = f_{RF} - f_{LO}$. This carrier appears on the first set of axes because the **Carrier frequencies** parameter of the outport specifies it first. The second set of axes displays the modulation at $f_{RF} + f_{LO}$. The modulation of the upper mixing product has the same amplitude as the modulation of the downconverted signal.

Model an RF Filter

To begin, open the model that you created in the “Probe Multiple RF Carriers” on page 2-15 section, or type

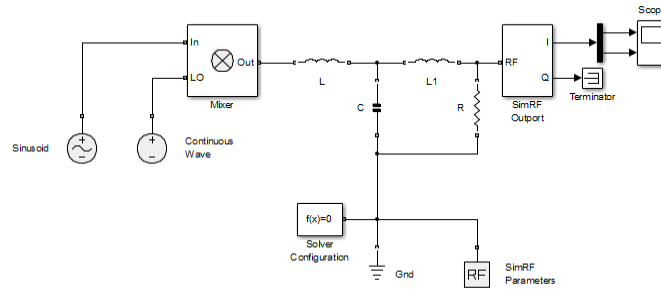
```
ex_simrf_tut_probe
```

at the MATLAB Command Window prompt.

1 From the Elements library, drag and drop one Capacitor and two Inductor blocks onto your model

2 Set the **Capacitance** parameter of the capacitor to $40e-12$ F.

- 3 Set the **Inductance** parameter of both inductors to $50e-9$ H.
- 4 Connect the blocks as shown in the following figure.



This configuration models a third-order low-pass Butterworth filter with a cutoff frequency of 1 rad/ns, or about 0.159 GHz. The high-frequency mixing product is in the stopband of the filter. The low-frequency product is in the passband.

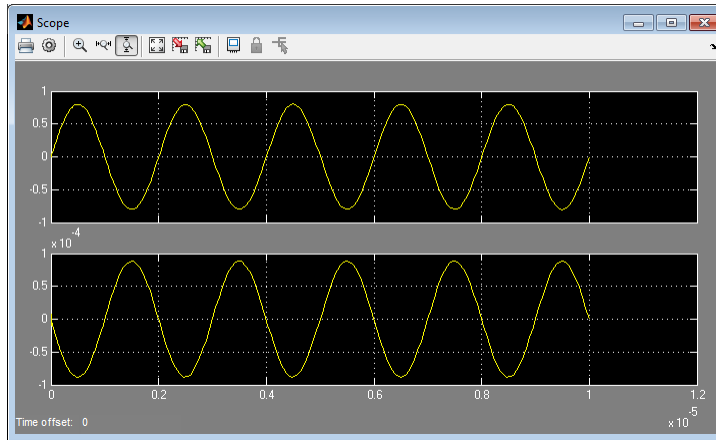
Simulate Filtering of RF Signals

To begin, open the model that you created in the previous section, “Model an RF Filter” on page 2-16, or type

```
ex_simrf_tut_filter
```

at the MATLAB Command Window prompt. Select **Simulation > Run** to run the model.

To view the results of the simulation, double-click the scope.



The filter attenuates the high-frequency signal at $f_{RF} + f_{LO}$ and transmits the signal at $f_{RF} - f_{LO}$ with minimal loss.

The SimRF documentation contains additional RF filter analysis and simulation examples:

- The example Model an RF Filter Using S-Parameter Data uses an S-Parameters block to model the filter shown in this tutorial.
- The example AC Analysis of an RF System calculates the voltage transfer function of an RF filter using harmonic balance analysis.

Equivalent Baseband

Introduction to Equivalent Baseband Simulation

- “SimRF Equivalent Baseband Libraries” on page 3-2
- “Equivalent Baseband Workflow” on page 3-8
- “Model an LC Bandpass Filter” on page 3-10

SimRF Equivalent Baseband Libraries

In this section...
“Overview of SimRF Equivalent Baseband Libraries” on page 3-2
“Open SimRF Equivalent Baseband Libraries” on page 3-3
“Physical Library” on page 3-4
“Mathematical Library” on page 3-6

Overview of SimRF Equivalent Baseband Libraries

The SimRF Equivalent Baseband libraries consist of the Equivalent Baseband (physical) and Idealized Baseband (mathematical) libraries of components for modeling RF systems within the Simulink environment. An RF model can contain blocks from both the physical and mathematical libraries. It can also include Simulink blocks and blocks from other blocksets.

SimRF Equivalent Baseband software extends your Simulink modeling environment with a library of blocks for modeling RF systems that include RF filters, transmission lines, amplifiers, and mixers.

You use SimRF Equivalent Baseband library blocks to represent the components of your RF system in a Simulink model. The blockset provides several types of component representations using network parameters (S, Y, Z, ABCD, H, and T format), mathematical descriptions, and physical properties.

In the Simulink model, you cascade the components to represent your RF architecture and run the simulation. During the simulation, the model computes a time-domain, complex-baseband representation. This method results in fast simulation of the quadrature modeling schemes used in modern communication systems and enables compatibility with other Simulink blocks.

The blocks let you visualize their specified network parameters using plots and Smith[®] Charts.

A validated Simulink model of an RF system can provide an executable specification for RF circuit design for wireless communication systems.

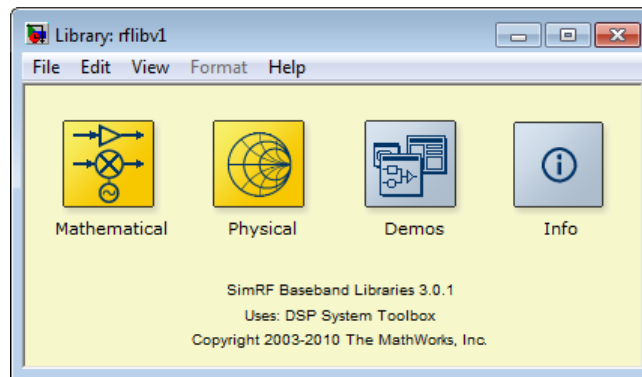
You can also use the blockset with Simulink Coder™ software to generate embeddable C code for real-time execution.

Open SimRF Equivalent Baseband Libraries

To open the main library window, type the following at the MATLAB prompt:

```
rflib
```

Each yellow icon in the window represents a library. Double-click an icon to open the corresponding library.



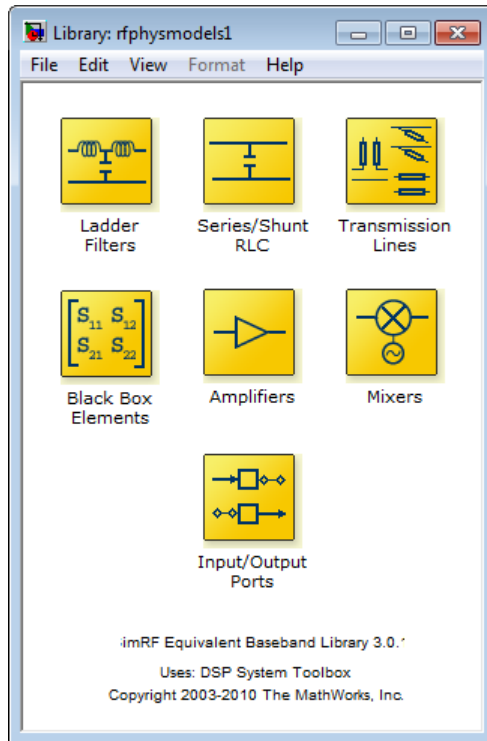
For a discussion of the Physical and Mathematical libraries, see the following sections.

Note The blue icons take you to the MATLAB Help browser.

- Double-click the Demos icon to open the SimRF Equivalent Baseband examples.
 - Double-click the Info icon to open the SimRF Equivalent Baseband documentation.
-

Physical Library

Use blocks from the Physical library to model physical and electrical components by specifying physical properties or by importing measured data. The Physical library includes several sublibraries, as shown in the following figure.



The following table describes the sublibraries and how to use them.

Sublibrary	Description
Amplifiers	RF amplifiers, specified using network parameters, noise data, and nonlinearity data, or a data file containing these parameters.
Ladder Filters	RF filters, specified using LC parameters. The software calculates the network parameters and noise data of the blocks from the topology of the filter and the LC values.
Series/Shunt RLC	Series and shunt RLC components for designing lumped element cascades, specified using RLC parameters. The software calculates the network parameters and noise data of the blocks from the topology of the components and the RLC values.
Mixers	RF mixers that contain local oscillators, specified using network parameters, noise data, and nonlinearity data, or a data file containing these parameters.
Transmission Lines	RF filters, specified using physical dimensions and electrical characteristics. The software calculates the network parameters and noise data of the blocks from the specified data.

Sublibrary	Description
Black Box	Passive RF components, specified using network parameters, or a data file containing these parameters. The software calculates the network parameters and noise data of the blocks from the specified data.
Input/Output Ports	Blocks for specifying simulation information that pertains to all blocks in a physical <i>subsystem</i> , such as center frequency and sample time. <hr/> <p>Note A physical subsystem is a collection of one or more physical blocks bracketed by an Input Port block and an Output Port block that bridge the physical and mathematical parts of the model.</p>

For more information on defining components, see “Specify or Import Component Data”.

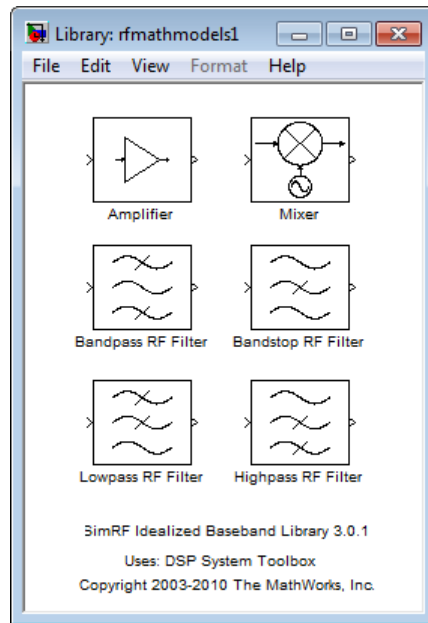
Mathematical Library

The Mathematical library contains mathematical representations of the amplifier, mixer, and filter blocks. Use a block from the Mathematical library to model an RF component in terms of mathematical equations that describe how the block operates on an input signal.

Mathematical blocks assume perfect impedance matching and a nominal impedance of 1 ohm. This means there is no loading and the power flow is unidirectional. As such, they are similar to standard Simulink blocks. In contrast, the physical blocks do not assume perfect matching—these blocks model the reflections that occur between blocks. Physical blocks model bidirectional power flow, and include loading effects. For these blocks, you

can specify the source and load impedances using the Input Port and Output Port blocks.

The mathematical library is shown in the following figure.



Equivalent Baseband Workflow

When you analyze an RF system using SimRF Equivalent Baseband software, your workflow might include the following tasks:

- 1 Create a Simulink model that includes RF components.

For more information, see “Model RF Components”.

- 2 Define component data by

- Specifying network parameters, mathematical relationships, or physical properties
- Importing data from an industry-standard Touchstone file, a MathWorks™ AMP file, an Agilent® P2D or S2D file, or the MATLAB workspace

The product lets you access component data in Touchstone SnP, YnP, ZnP, HnP, and GnP formats. You can also import amplifier network parameters and power data from a MathWorks AMP file.

For more information, see “Specify or Import Component Data”.

- 3 Where applicable, add the following information to the component definition:

- Operating condition values (see “Specify Operating Conditions”).
- Nonlinearity data (see “Model Nonlinearity”).
- Noise data (see “Model Noise”).

- 4 Validate the behavior of individual blocks by plotting component data.

Note You can plot data for individual blocks from the Physical library that model physical components either before or after you run a simulation.

For more information, see “Create Plots”.

- 5 Run the simulation.

For more information on how the product performs time-domain simulation of an RF system, see “Simulate an RF Model”.

6 Generate plots to gain insight into system behavior.

For more information, see “Create and Modify Subsystem Plots”.

The following plots and charts are available:

- Rectangular plots
- Polar plots
- Smith Charts
- Composite plots
- Budget plots

Model an LC Bandpass Filter

In this section...
“Overview of LC Bandpass Filter Example” on page 3-10
“Select Blocks to Represent System Components” on page 3-10
“Build the Model” on page 3-11
“Specify Model Parameters” on page 3-13
“Validate Filter Components and Run the Simulation” on page 3-20
“Analyze the Simulation Results” on page 3-22

Overview of LC Bandpass Filter Example

In this example, you model the signal attenuation caused by an RF filter by comparing the signals at the input and output of the filter.

The RF filter you use in this example is an LC bandpass filter with a bandwidth of 200 MHz, centered at 700 MHz. You use a three-tone input signal to stimulate a range of in-band and out-of-band frequencies of the filter. The input signal has the following tones:

- 700 MHz — Center of the filter
- 600 MHz — Lower edge of the filter passband
- 900 MHz — Outside the filter passband

You simulate the effects of the filter over a bandwidth of 500 MHz.

Select Blocks to Represent System Components

In this part of the example, you select the blocks to represent the input signal, the RF filter, and the signal displays.

You model the RF filter using a physical *subsystem*, which is a collection of one or more physical blocks bracketed by an Input Port block and an Output Port block. The RF filter subsystem consists of an LC Bandpass Pi block, and the Input Port and Output Port blocks. The function of the Input Port and

Output Port blocks is to bridge the physical part of the model, which uses bidirectional RF signals, and the rest of the model, which uses unidirectional Simulink signals.

The following table lists the blocks that represent the system components and a description of the role of each block.

Block	Description
Sine Wave	Generates a three-channel signal.
Matrix Sum	Combines the three channel signal into a single three-tone source signal.
Input Port	Establishes parameters that are common to all blocks in the RF filter subsystem, including the source impedance of the subsystem that is used to convert Simulink signals to the SimRF Equivalent Baseband physical modeling environment.
LC Bandpass Pi	Models the signal attenuation caused by the RF filter which, in this example, is the LC Bandpass Pi filter.
Output Port	Establishes parameters that are common to all blocks in the RF filter subsystem. These parameters include the load impedance of the subsystem, which is used to convert RF signals to Simulink signals.
Spectrum Scope	Displays signals at the input to and output of the filter.

Build the Model

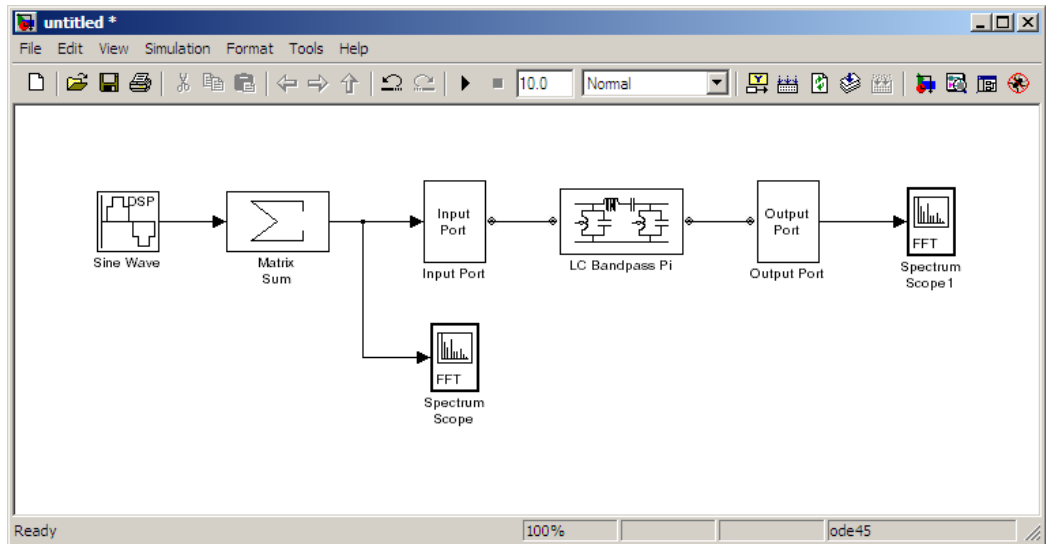
In this part of the example, you create a Simulink model, add blocks to the model, and connect the blocks.

- 1 Create a model with the blocks shown in the following table. The Library column of the table specifies the hierarchical path to each block.

Block	Library Path	Quantity
Sine Wave	DSP System Toolbox > Sources	1
Matrix Sum	DSP System Toolbox > Math Functions > Matrices and Linear Algebra > Matrix Operations	1
Spectrum Scope	DSP System Toolbox > Sinks	2
Input Port	SimRF > Equivalent Baseband > Equivalent Baseband > Input/Output Ports	1
LC Bandpass Pi	SimRF > Equivalent Baseband > Ladder Filters	1
Output Port	SimRF > Equivalent Baseband > Input/Output Ports	1

2 Connect the blocks as shown in the following figure.

For more information on connecting physical and mathematical blocks, see “Connect Model Blocks”.



Now you are ready to specify block parameters.

Specify Model Parameters

In this part of the example, you specify the following parameters to represent the behavior of the system components:

- “Input Signal Parameters” on page 3-13
- “Filter Subsystem Parameters” on page 3-15
- “Signal Display Parameters” on page 3-19

Input Signal Parameters

You generate the three-tone source signal using two blocks. You use the Sine Wave block to generate a complex three-channel signal, where each channel corresponds to a different frequency. Then, you use the Matrix Sum block to combine the channels into a single three-tone source signal. Without this block, the signal in all subsequent blocks would have three independent channels.

The SimRF Equivalent Baseband simulation algorithm requires you to shift the frequencies of the input signal. The software simulates the filter subsystem using a complex-baseband modeling technique, which automatically shifts the filter response and centers it at zero. You must shift the frequencies of the signals outside the physical subsystem by the same amount.

For more information on complex-baseband modeling, see “Create a Complex Baseband-Equivalent Model”.

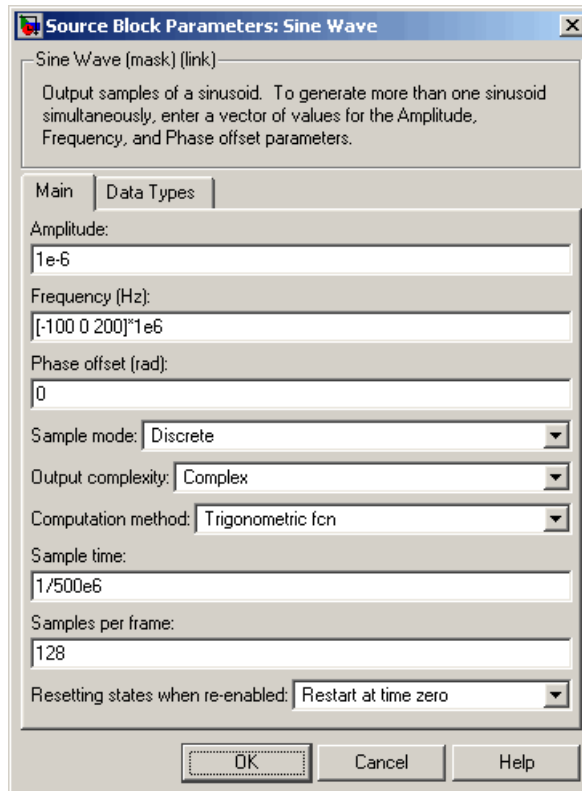
Note All signals in the RF model must be complex to match the signals in the physical subsystem, so you create a complex input signal.

The center frequency of the LC bandpass filter is 700 MHz, so you use a three-tone source signal with tones that are 700 MHz below the actual tones, at -100 MHz, 0 MHz, and 200 MHz, respectively.

1 In Sine Wave block dialog box:

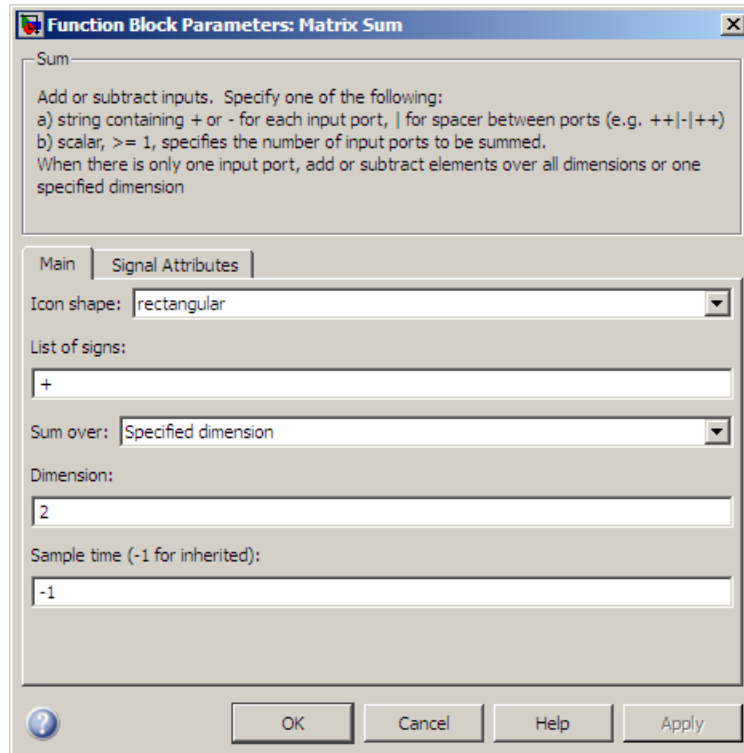
- Set the **Amplitude** parameter to $1e-6$.

- Set the **Frequency (Hz)** parameter to $[-100 \ 0 \ 200] \cdot 1e6$.
- Set the **Output complexity** parameter to Complex.
- Set the **Sample time** parameter to $1/500e6$.
- Set the **Samples per frame** parameter to 128 .



2 In the Matrix Sum block dialog box:

- Set the **Sum over** parameter to Specified dimension.
- Set the **Dimension** parameter to 2.



Filter Subsystem Parameters

In this part of the example, you configure the blocks that model the RF filter subsystem—the Input Port, LC Bandpass Pi, and Output Port blocks.

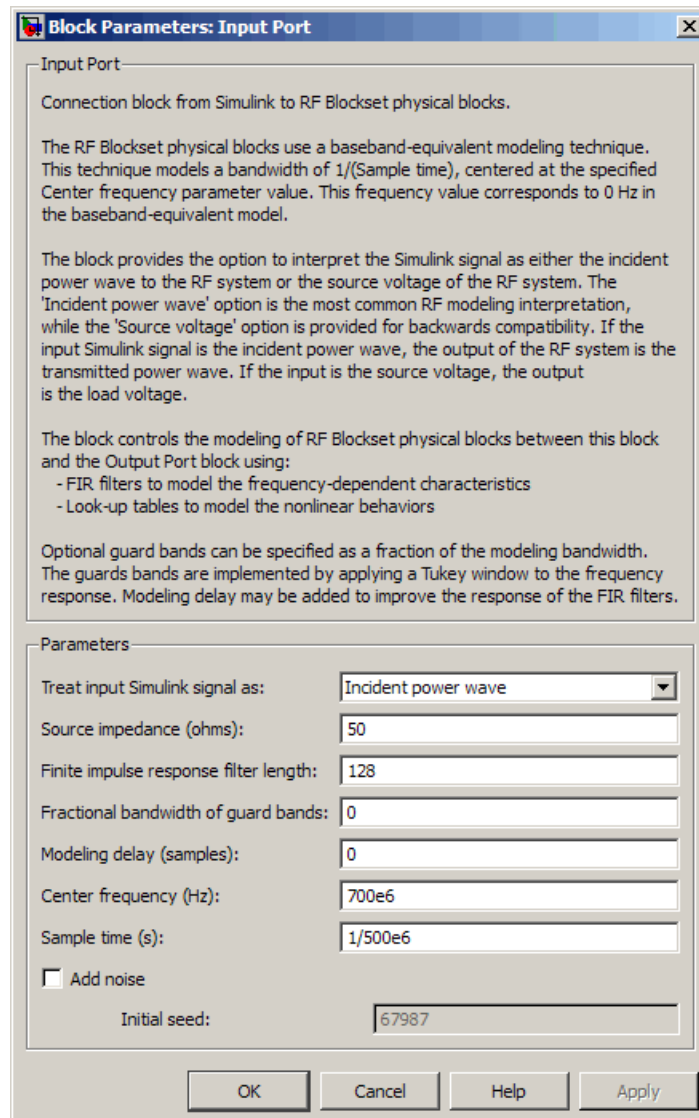
1 Set the Input Port block parameters as follows:

- **Treat input Simulink signal as = Incident power wave**

This option tells the blockset to interpret the input signal as the incident power wave to the RF subsystem, and not the source voltage of the RF subsystem.

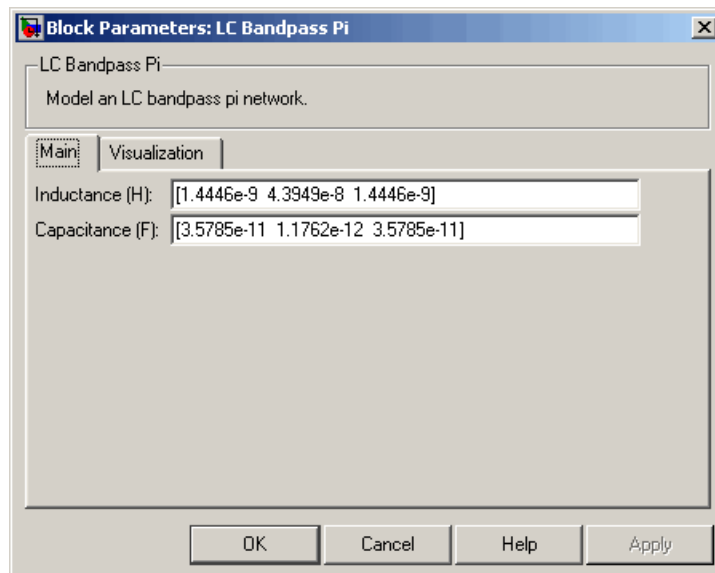
Note If you use the default value for this parameter, the software interprets the input Simulink signal as the source voltage. As a result, the source and the load that model the Input Port and Output Port blocks, respectively, introduce 6 dB of loss into the physical system at all frequencies. For more information on why this loss occurs, see the note in “Convert to and from Simulink Signals”.

- **Center frequency** = $700\text{e}6$
- **Sample time (s)** = $1/500\text{e}6$
- Clear the **Add noise** check box so the software does not include noise in the simulation. To learn how to model noise, see “Model Noise”.

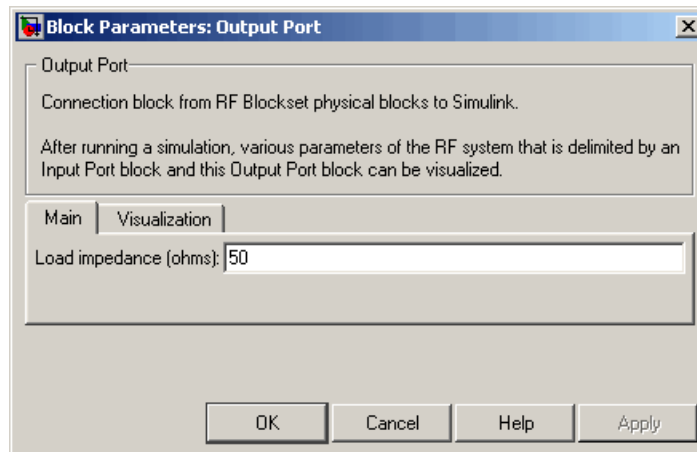


Note You must enter the **Sample time (s)** because the Input Port block does not inherit a sample time from the input signal. The specified sample time must match the sample time of the input signal. The **Sample time (s)** of $1/500e6$ second used in this example is equivalent to a bandwidth of 500 MHz.

- 2 Accept default parameters for inductance and capacitance in the LC Bandpass Pi block. These parameters create a filter with the desired bandwidth of 200 MHz, centered at 700 MHz.



- 3 Accept the default parameters for the Output Port block to use a load impedance of 50 ohms.



Signal Display Parameters

In this part of the example, you specify:

- The parameters of the Spectrum Scope block to display the source signal
- The parameters of the Spectrum Scope1 block to display the filtered signal

For each scope, you set the range of the x - and y -axes to make sure that the entire signal is visible.

By default, the scope displays appear stacked on top of each other on the screen when you run the simulation, so you can only see one of them. To ensure that both scopes are visible during the simulation, you specify a different position for each scope on the screen.

1 In the Spectrum Scope block dialog box:

- In the **Scope Properties** tab, set the **Spectrum units** parameter to dBm.
- In the **Display Properties** tab, set the **Scope position** parameter to `get(0, 'defaultfigureposition').* [.15 1 1 1]`
- In the **Axis Properties** tab, set the **Minimum Y-limit** parameter to -220.

- In the **Axis Properties** tab, set the **Maximum Y-limit** parameter to -80
 - In the **Axis Properties** tab, set the **Y-axis label** parameter to dBm.
- 2** Set the Spectrum Scope1 block parameters as follows:
- In the **Scope Properties** tab, set the **Spectrum units** parameter to dBm.
 - In the **Display Properties** tab, set the **Scope position** parameter to `get(0,'defaultfigureposition').*[1.85 1 1 1]`
 - In the **Axis Properties** tab, set the **Minimum Y-limit** parameter to -220.
 - In the **Axis Properties** tab, set the **Maximum Y-limit** parameter to -80
 - In the **Axis Properties** tab, set the **Y-axis label** parameter to dBm.

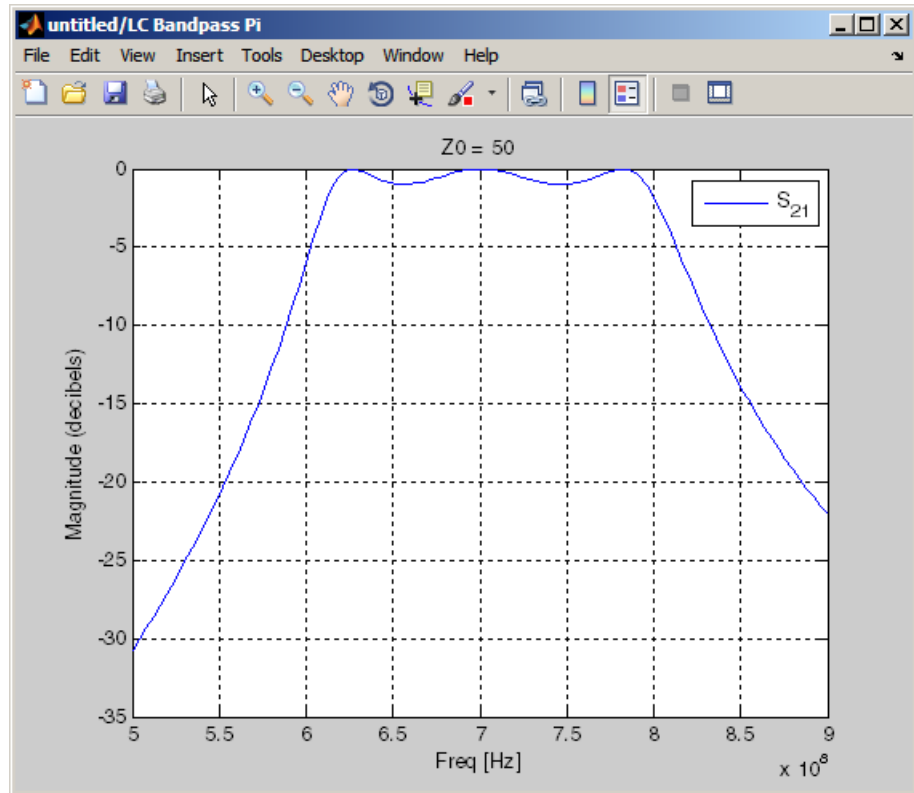
Note If you do not specify the position of the scopes using the **Display Properties** tab, you can click and drag the displays to arrange them on the screen after the simulation starts.

Validate Filter Components and Run the Simulation

In this part of the example, you validate the behavior of the LC Bandpass Pi filter block by plotting its frequency response and then run the simulation.

Note When you plot information about a physical block, the plot displays the actual frequency response of the block at the selected passband (i.e., the response at the unshifted frequencies), not the response at the shifted frequencies. For more information on this shift, see “Input Signal Parameters” on page 3-13.

- 1** Double-click the LC Bandpass Pi block to open the block dialog box.
- 2** Select the **Visualization** tab and click **Plot** to plot the frequency response of the filter. This plots the magnitude of S_{21} as a function of frequency, which represents the gain of the filter.



Filter Gain

Note The physical blocks only model a band of frequencies around the center frequency of the physical subsystem. You must choose the sample time and center frequency such that all important frequency characteristics of your physical subsystem fall in this band of frequencies. The plot shows the frequency response of the filter for the portion of the RF spectrum that the physical blocks model. In this example, the physical blocks model a 500-MHz band centered at 700 MHz, as defined by the Input Port block.

- 3 In the model window, select **Simulation > Start** to run the simulation.

Analyze the Simulation Results

In this part of the example, you analyze the results of the simulation. This section contains the following topics:

- “Compare the Input and Output Signals of the RF Filter” on page 3-22
- “Plot Model Parameters of the Filter Subsystem” on page 3-24

Compare the Input and Output Signals of the RF Filter

You can view the source signal and the filtered signal in the Spectrum Scope windows while the model is running. These windows appear automatically when you start the simulation.

The Spectrum Scope blocks display the signals at the shifted (baseband-equivalent) frequencies, not at the selected passband frequencies. You can relabel the *x*-axes of the Spectrum Scope windows to display the passband signal by entering the **Center frequency** parameter value of 700e6 (from the Input Port block) for the **Frequency display offset (Hz)** parameter in the **Axis Properties** tab of the Spectrum Scope block dialog boxes. For more information on complex-baseband modeling, see “Create a Complex Baseband-Equivalent Model”.

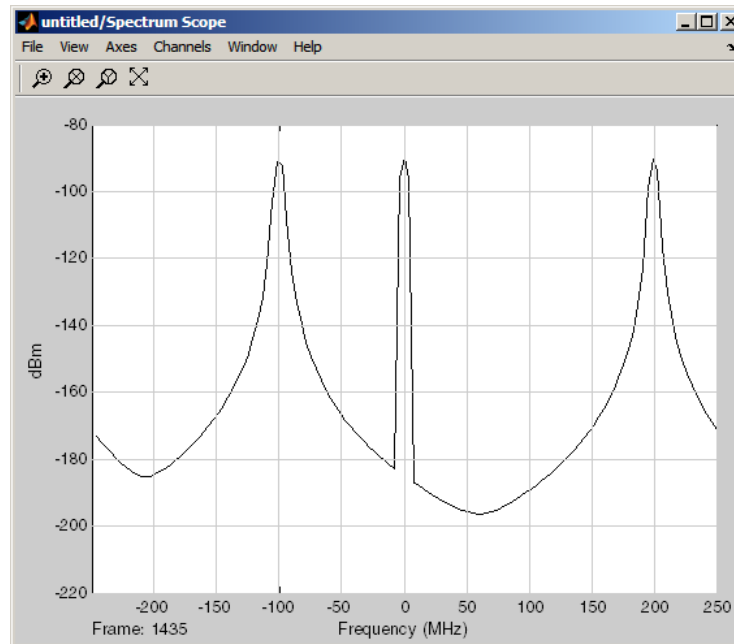
The Spectrum Scope blocks display power spectral density normalized to unit sampling frequency. To display power per channel, insert a Gain block with the **Gain** parameter set to 1/sqrt(N) before each Spectrum Scope block. N is the number of channels. The Gain block is in the **Simulink > Commonly Used Blocks** library.

In this example, N is 128 (the value of the **Samples per frame** parameter of the Sine Wave block, 128).

Note SimRF Equivalent Baseband signals represent amplitudes, not voltages. This means that in the product, power is defined as:

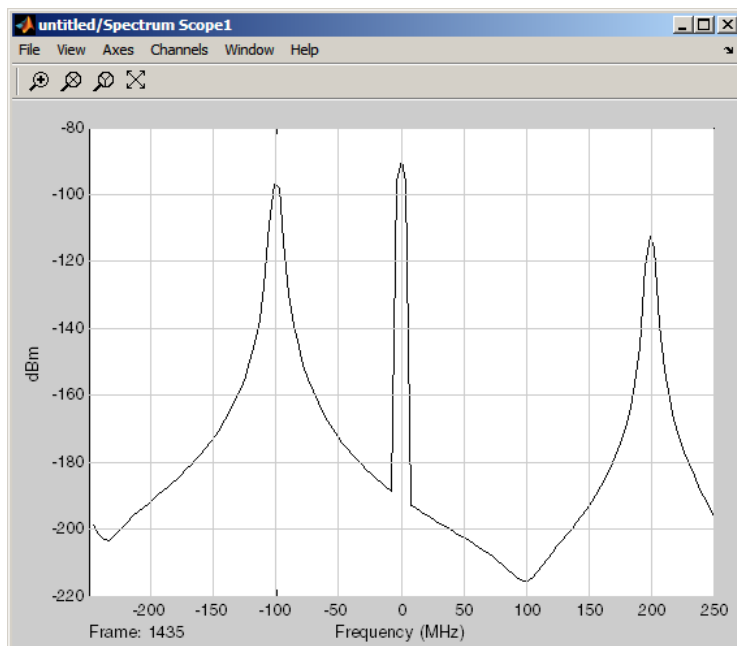
$$Power(in\ watts)=[Amplitude(in\ volts / \sqrt{ohm})]^2$$

The following plot shows the RF filter input signal you specified in the Sine Wave block.



Input to RF Filter

The next plot shows the filtered signal. Notice the RF filter does not attenuate the signal at the center frequency.



Attenuated Output of RF Filter

Plot Model Parameters of the Filter Subsystem

After you simulate an RF model, you can evaluate the behavior of the physical subsystem by plotting the network parameters of the Output Port block.

Note When you plot information about a physical subsystem, the plot displays the actual frequency response of the subsystem at the selected passband (i.e. the response at the unshifted frequencies), not the response at the shifted frequencies.

To understand the frequency response of the filter, examine the S-parameters as a function of frequency for the RF filter subsystem on a composite plot.

- 1 Open the dialog box of the Output Port block by double-clicking the block.
- 2 Select the **Visualization** tab, and click **Plot**.

The composite plot, shown in the following figure, contains four separate plots in one figure. For the Output Port block, the composite plot shows the following as a function of frequency (counterclockwise from the upper-left plot):

- An X-Y plane plot of the magnitude of the filter gain, S_{21} , in decibels.
- An X-Y plane plot of the phase of the filter gain, S_{21} , in degrees.
- A Z Smith chart showing the real and imaginary parts of the filter reflection coefficient, S_{11} .
- A Polar plane plot showing the magnitude and phase of the filter reflection coefficient, S_{11} .

Note In this example, the response of the filter subsystem is the same as the response of the filter block because the subsystem contains only a filter block.

