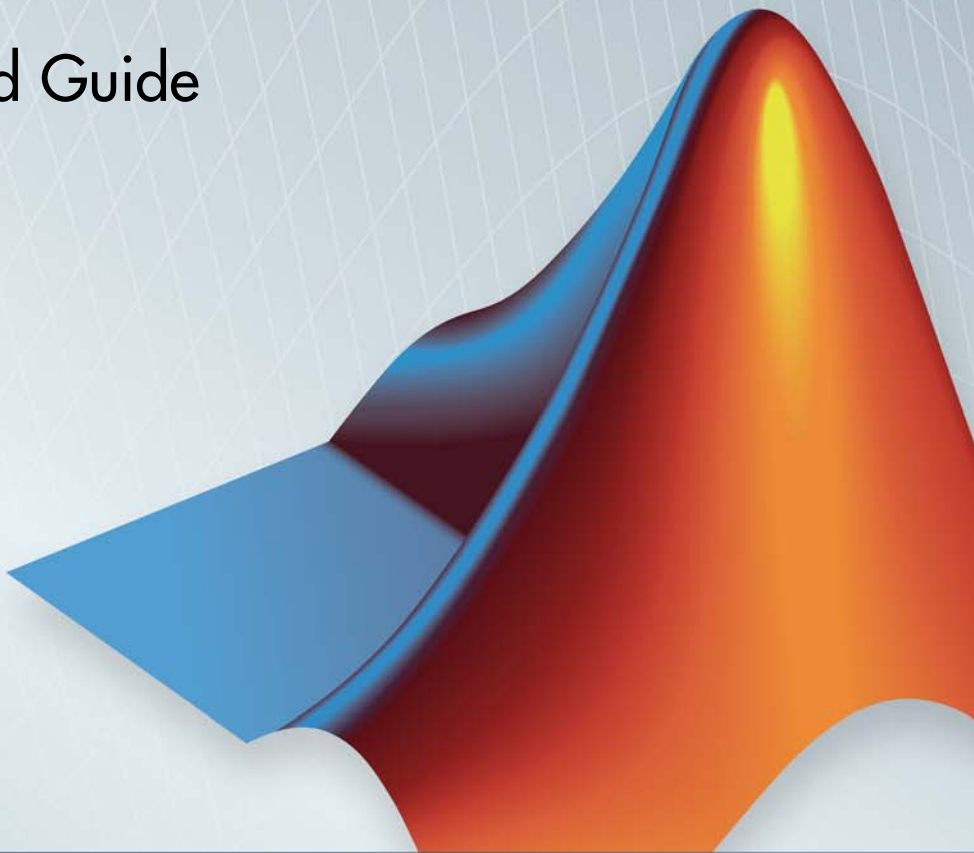


**SimRF™**

# Getting Started Guide

**R2013b**



**MATLAB® & SIMULINK®**



## How to Contact MathWorks



[www.mathworks.com](http://www.mathworks.com) Web  
[comp.soft-sys.matlab](mailto:comp.soft-sys.matlab) Newsgroup  
[www.mathworks.com/contact\\_TS.html](http://www.mathworks.com/contact_TS.html) Technical Support



[suggest@mathworks.com](mailto:suggest@mathworks.com) Product enhancement suggestions  
[bugs@mathworks.com](mailto:bugs@mathworks.com) Bug reports  
[doc@mathworks.com](mailto:doc@mathworks.com) Documentation error reports  
[service@mathworks.com](mailto:service@mathworks.com) Order status, license renewals, passcodes  
[info@mathworks.com](mailto: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 2010–2013 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](http://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](http://www.mathworks.com/patents) for more information.

**Revision History**

September 2010 Online only  
April 2011 Online only  
September 2011 Online only  
March 2012 Online only  
September 2012 Online only  
March 2013 Online only  
September 2013 Online only

New for Version 3.0 (Release 2010b)  
Revised for Version 3.0.2 (Release 2011a)  
Revised for Version 3.1 (Release 2011b)  
Revised for Version 3.2 (Release 2012a)  
Revised for Version 3.3 (Release 2012b)  
Revised for Version 4.0 (Release 2013a)  
Revised for Version 4.1 (Release 2013b)



## Circuit Envelope

### Introduction to Circuit Envelope Simulation

**1**

<b>SimRF Product Description</b> .....	<b>1-2</b>
Key Features .....	1-2
<b>Circuit Envelope Basics</b> .....	<b>1-3</b>
Simulate a Passband Signal in Simulink Software .....	1-4
Compare Passband and Baseband Signals in SimRF Software .....	1-5
Simulate the Envelope of a Baseband Signal .....	1-8
<b>Using SimRF Software for the First Time</b> .....	<b>1-11</b>
Expected Background .....	1-11
Circuit Envelope, Equivalent Baseband, and Idealized Baseband Library Features .....	1-11
Understanding the SimRF Environment .....	1-13

### Frequency Conversion

**2**

<b>Model an RF Mixer</b> .....	<b>2-2</b>
Build the RF System .....	2-2
Define Model Variables .....	2-4
Specify Block Parameters for RF Simulation .....	2-6
Probe Circuit Envelopes Waveforms .....	2-8
Observe Downconverted Envelope Signal at Output Port .....	2-8
<b>Filter Mixing Products</b> .....	<b>2-10</b>
Probe Multiple RF Carriers .....	2-11

Model an RF Filter .....	2-12
Simulate Filtering of RF Signals .....	2-13
Improve Performance by Reducing Total Simulation Frequencies .....	2-14
<b>Measuring Image Rejection Ratio in Receivers .....</b>	<b>2-16</b>

# Equivalent Baseband

## Introduction to Equivalent Baseband Simulation

### 3

<b>SimRF Equivalent Baseband Libraries .....</b>	<b>3-2</b>
Overview of SimRF Equivalent Baseband Libraries .....	3-2
Open SimRF Equivalent Baseband Libraries .....	3-3
Equivalent Baseband Library .....	3-4
Idealized Baseband Library .....	3-6
<b>Equivalent Baseband Workflow .....</b>	<b>3-8</b>
<b>Model an LC Bandpass Filter .....</b>	<b>3-10</b>
Overview of LC Bandpass Filter Example .....	3-10
Select Blocks to Represent System Components .....	3-10
Build the Model .....	3-11
Specify Model Parameters .....	3-12
Validate Filter Components and Run the Simulation .....	3-19
Analyze the Simulation Results .....	3-21

# Circuit Envelope

---





# Introduction to Circuit Envelope Simulation

---

- “SimRF Product Description” on page 1-2
- “Circuit Envelope Basics” on page 1-3
- “Using SimRF Software for the First Time” on page 1-11

## SimRF Product Description

### Design and simulate RF systems

SimRF™ provides a component library and simulation engine for designing RF systems. It includes amplifiers, mixers, S-parameter blocks, and other basic blocks for designing architectures for wireless transceivers used in communication and radar systems. You can connect these blocks arbitrarily to form diverse architectures and to simulate the behavior of the RF front-end at the system level.

SimRF lets you simulate RF amplifiers to estimate gain, noise, even-order, and odd-order intermodulation distortion. The simulation of mixers enables you to predict image rejection, reciprocal mixing, local oscillator phase offsets, and DC conversion. You can also simulate frequency-dependent mismatches between linear and non-linear components in the time and frequency domains.

With SimRF you can model RF systems at different levels of abstraction. The circuit envelope solver enables high-fidelity, multicarrier simulation of networks with arbitrary topologies. The Equivalent Baseband library enables fast, discrete-time simulation of single-carrier cascaded systems.

### Key Features

- Circuit envelope solver for multiple carrier-frequency simulation
- Arbitrary network descriptions, enabling N-port block modeling and intra-model signal probing
- S-parameter data files for time-domain and frequency-domain simulations
- Passive components, including resistors, capacitors, inductors, transmission lines, and general impedance blocks
- 3-port mixer and 2-port amplifier models specified by noise figure, IP2, IP3, P1dB, and Psat
- Equivalent Baseband library for discrete-time simulation of single-carrier cascaded systems

## Circuit Envelope Basics

### In this section...

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

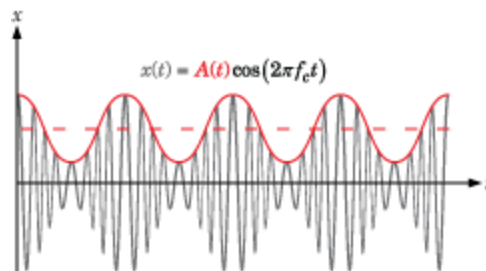
“Compare Passband and Baseband Signals in SimRF Software” on page 1-5

“Simulate the Envelope of a Baseband Signal” on page 1-8

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

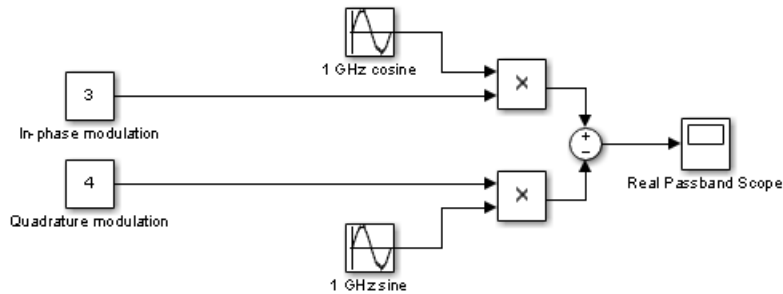
steps on the scale of the modulating signal instead of the carrier, the software produces equivalent results in less time.

## 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.



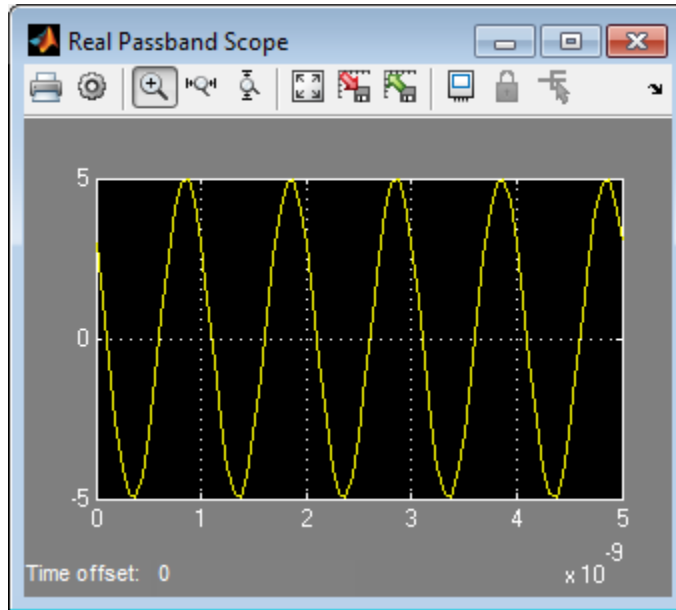
The system specifies a real passband signal  $x(t)$  according to the formula

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

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  $143^\circ$ .

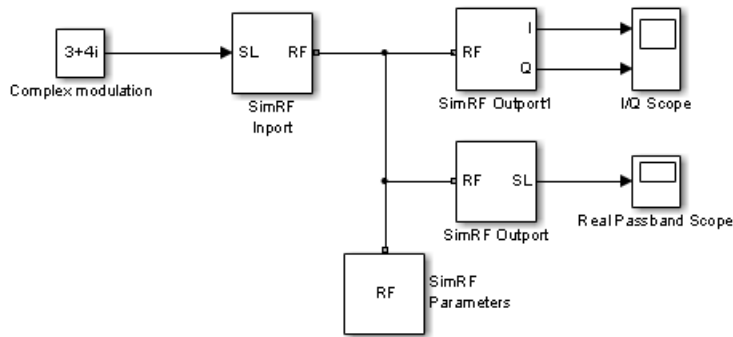
In the Configuration Parameters dialog box, the **Fixed-step size (fundamental sample time)** parameter has been set to  $1/16 \times 10^{-9}$ . This value is on the order of the wavelength of the carrier. The simulation takes a total of 81 samples — 16 per cycle.

## 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.

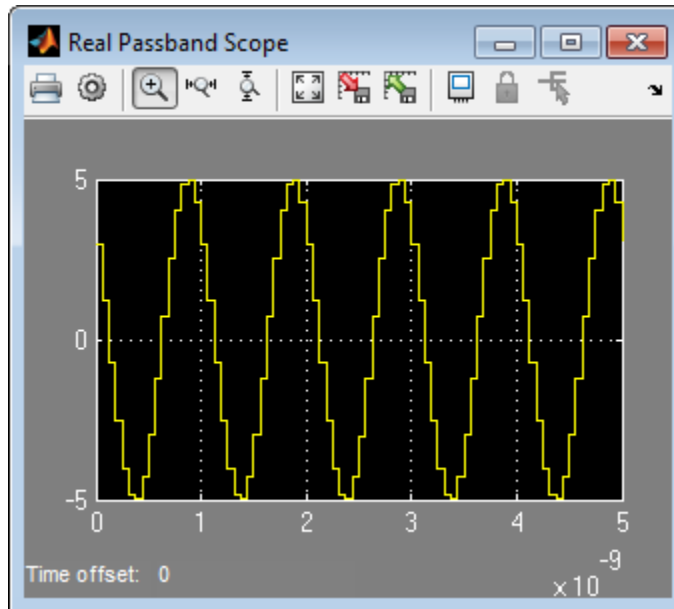
Contrary to the Simulink passband implementation in the previous section, the complex baseband signal driving the SimRF system does not include the carrier. Instead, the SimRF environment handles the carrier analytically. The carrier appears in four different blocks in the SimRF environment:

- In the 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  $1\text{e}9$  Hz).
- In the Outputport block, the **Carrier frequencies** parameter specifies the signal on the  $1\text{e}9$  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 Outputport1 block, also an Outputport block, the **Carrier frequencies** parameter specifies the signal outputted on the

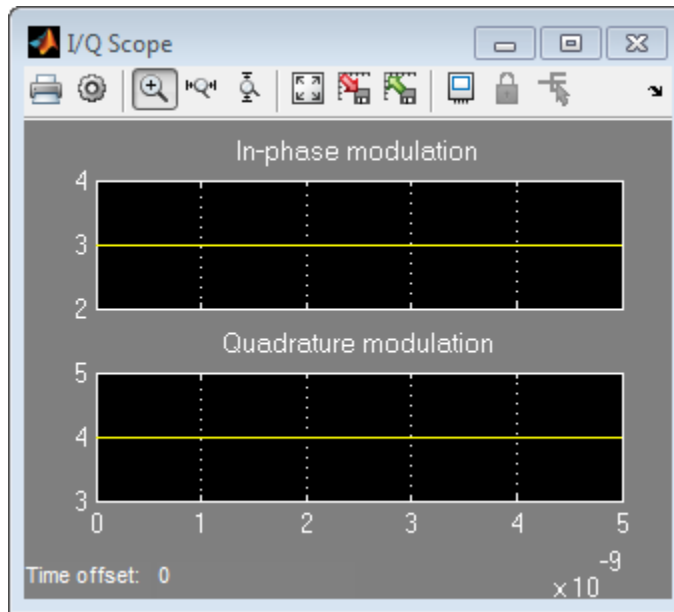
1e9 Hz carrier (1 GHz) as Simulink signals. These signals appear at the I and Q ports. The **Output** parameter is set to In-phase and Quadrature Baseband, so these signals represent the in-phase and quadrature modulations of the signal on the 1-GHz carrier.

- In the Configuration 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. For more options, refer to Configuration block.

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  $1/16 \times 10^{-9}$ . This value is on the order of the wavelength of the carrier. The simulation takes a total of 81 samples — 16 per cycle.

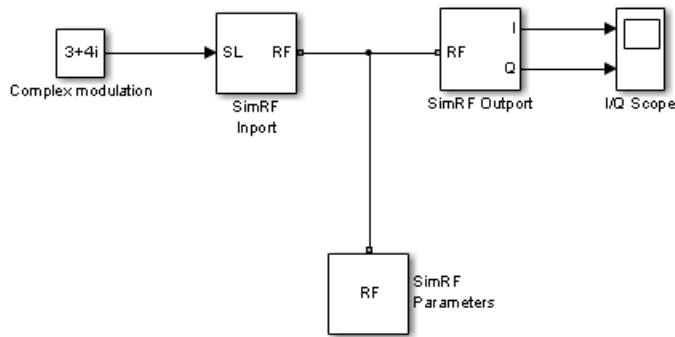
## Simulate the Envelope of a Baseband Signal

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 1-5.

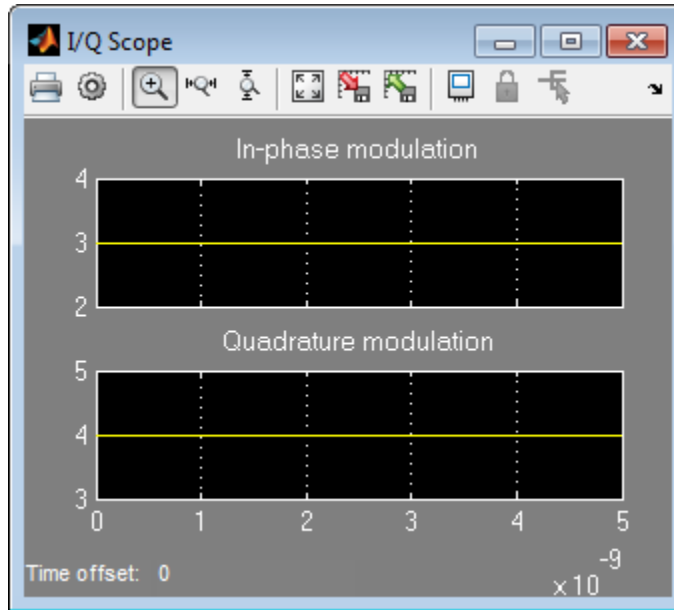




The system is almost identical to the system in the previous section, except:

- The model contains only one SimRF Outport block and only one scope. The SimRF environment outputs in-phase and quadrature modulations of the 1-GHz signal. In the SimRF Outport 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  $1/16 \cdot 1e-9$ .

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  $5\text{e-}9$  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  $1/16 \cdot 1\text{e-}9$ , this model simulates accurately with a time step 80 times larger. This step size results in a reduction of total sample time by a factor of 80, excluding the initial time step at time 0.

## Using SimRF Software for the First Time

### In this section...

“Expected Background” on page 1-11

“Circuit Envelope, Equivalent Baseband, and Idealized Baseband Library Features” on page 1-11

“Understanding the SimRF Environment” on page 1-13

### 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 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)**

<b>Do blocks in this library...</b>	<b>Circuit Envelope Library</b>	<b>Equivalent Baseband Library</b>	<b>Idealized Baseband Library</b>
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 nonlinear elements?	Yes	Yes	Yes
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
Support noise simulation?	Yes	Yes	No

**(Continued)**

<b>Do blocks in this library...</b>	<b>Circuit Envelope Library</b>	<b>Equivalent Baseband Library</b>	<b>Idealized Baseband Library</b>
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 Configuration block**

This blocks manipulates the environment itself. To use this block, connect it to any part of the RF system. Because it is not part of the physical representation of the system, it has the same effect regardless of where you connect it.

To run models containing SimRF blocks, you must connect a Configuration block to the SimRF environment.

# Frequency Conversion

---

- “Model an RF Mixer” on page 2-2
- “Filter Mixing Products” on page 2-10
- “Measuring Image Rejection Ratio in Receivers” on page 2-16

## Model an RF Mixer

### In this section...

“Build the RF System” on page 2-2

“Define Model Variables” on page 2-4

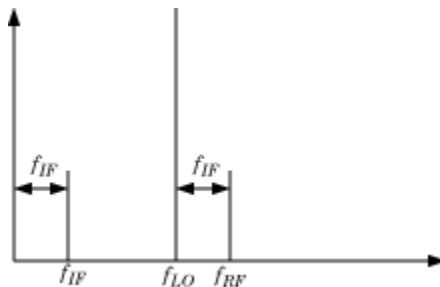
“Specify Block Parameters for RF Simulation” on page 2-6

“Probe Circuit Envelopes Waveforms” on page 2-8

“Observe Downconverted Envelope Signal at Output Port” on page 2-8

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.

### Build the RF System

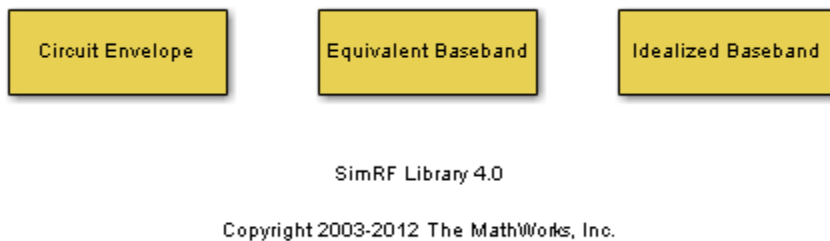
This section shows how to build a SimRF model starting from a blank canvas. To skip this section and start with the completed model, type



```
ex_simrf_tut_mixer
```

at the MATLAB Command Window prompt.

This example model specifies an AM waveform at the input port of the mixer, an LO at the LO port of the mixer, and a  $50\text{-}\Omega$  termination at the output of a mixer. To build this model, open the SimRF library by typing `simrfv2libs` in the MATLAB Command Window.



Double-click the block labeled **Circuit Envelope** to open the Circuit Envelope library.



From the **Elements**, **Sources**, and **Utilities** sublibraries, add the following blocks to your model.

- From the **Elements** library, add a Resistor block.
- From the **Elements** library, add a Mixer block.
- From the **Elements** library, add a Ground block.
- From the **Sources** library, add a Continuous Wave block.

- From the **Sources** library, add a Sinusoid block.
- From the **Utilities** library, add a Configuration block.

Connect the blocks in the same configuration as the `ex_simrf_tut_mixer` model.

### Define Model Variables

For models with blocks that share parameter values, specifying parameter values using variables saves time and effort. To define model variables, use one of the following workflows.

#### Use 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;
```

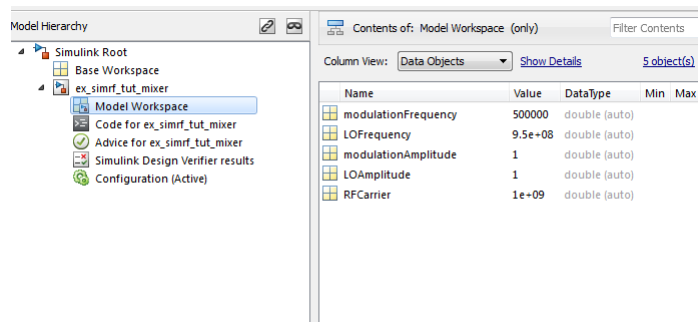
For more information, see Using Callback Functions in the *Simulink User's Guide*.

## Use 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. The `ex_simrf_tut_mixer` model uses this approach.

- 1 Open the Model Explorer window by typing `Ctrl + H`, by selecting **View > Model Explorer > Current System**, 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*.

### Use 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 **Simulation frequencies** parameter in the Configuration 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  $1e-5$ . You can also set the stop time directly on the Simulink canvas.
  - Set **Solver** to `ode23t (mod. stiff/Trapezoidal)`. The SimRF environment does not use the `ode23t` solver. However, since oscillating

signals can be stiff, the solver is a good choice for the Simulink environment when using SimRF blocks.

- 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 Configuration block to open the block dialog box. Set **Step size** to  $1e-7$ .

## 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:

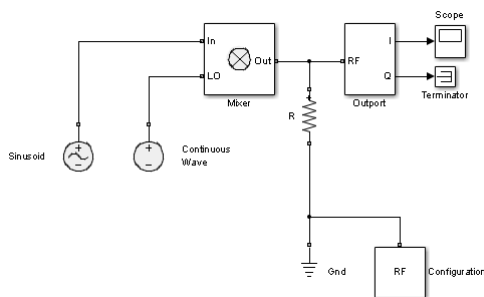
```
hmdl = bdroot;
set_param(hmdl, 'StopTime', '1e-5');
set_param(hmdl, 'Solver', 'ode23t');
set_param([hmdl '/Sinusoid'], ...
  'VA_I','modulationAmplitude', ...
  'Fmod','modulationFrequency', ...
  'CarrierFreq','RFCarrier');
set_param([hmdl '/Continuous Wave'], ...
  'IVoltage','LOAmplitude', ...
  'CarrierFreq','LOFrequency');
set_param([hmdl '/Configuration'], ...
  'StepSize','1e-7');
```

### Probe Circuit Envelopes Waveforms

At the output of the mixer:

- From the SimRF **Utilities** library, drag and drop an **Outport** block onto your model. In the block dialog:
  - Set **Output** to In-phase and quadrature.
  - Set **Carrier frequencies** to  $\text{RFCarrier} - \text{LOFrequency}$ .
- 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.

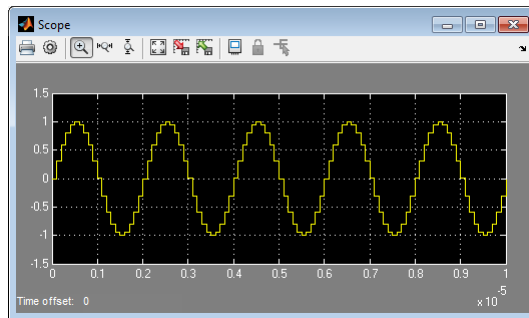


This configuration uses a SimRF **Outport** block as a voltage sensor at the output port of the mixer. The Simulink signal at the output is the envelope of the carrier or carriers specified in the block. A scope attached to the output plots the envelope. The **Output** parameter controls how signals 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 2-6 section of this tutorial.

### Observe Downconverted Envelope Signal at Output Port

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

To view the results of the simulation, double-click the scope, and click the **Autoscale** button.



The Sinusoid specifies a 1-V amplitude for the modulation on  $f_{RF}$ , which the mixer downconverts to  $f_{IF}$ . The SimRF Outport 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 2-11

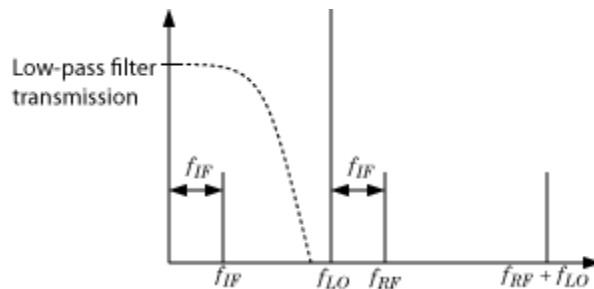
“Model an RF Filter” on page 2-12

“Simulate Filtering of RF Signals” on page 2-13

“Improve Performance by Reducing Total Simulation Frequencies” on page 2-14

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 2-2 section, or type

```
ex_simrf_tut_mixer
```

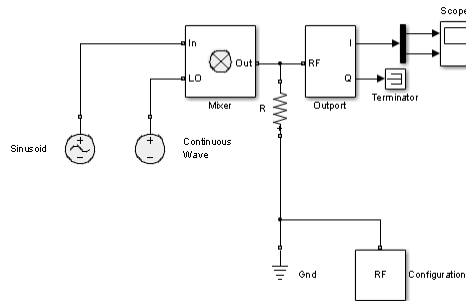
at the Command Window prompt.



## Probe Multiple RF Carriers

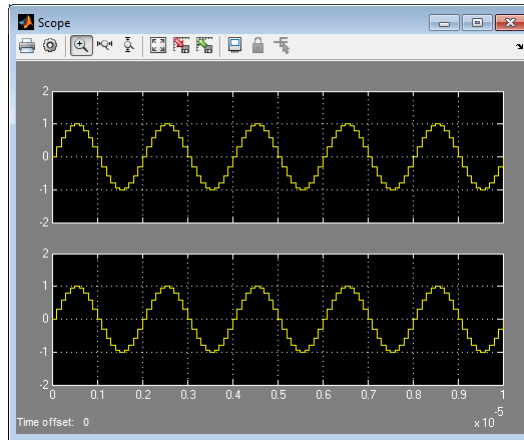
The SimRF environment specifies the carriers  $f_{RF} - f_{LO}$ ,  $f_{RF}$ ,  $f_{LO}$ , and  $f_{RF} + f_{LO}$ , but the SimRF Output block probes only the carrier  $f_{RF} - f_{LO}$ . Examine the carriers  $f_{RF} - f_{LO}$  and  $f_{RF} + f_{LO}$  by changing the model according to the following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

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 output 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.

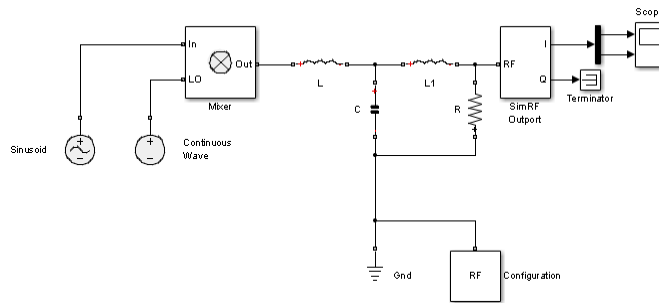
### Model an RF Filter

To begin, open the model that you created in the “Probe Multiple RF Carriers” on page 2-11 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.

You can also use an LC Ladder block to model this filter.

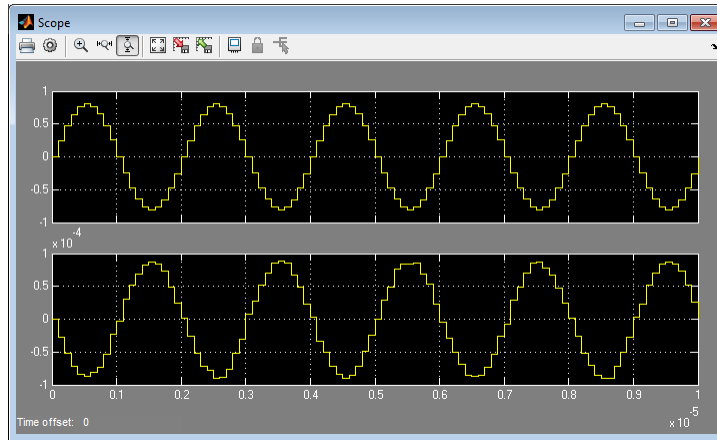
## Simulate Filtering of RF Signals

To begin, open the model that you created in the previous section, “Model an RF Filter” on page 2-12, 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.

such as

### Improve Performance by Reducing Total Simulation Frequencies

The models in the preceding examples use the **Automatically select fundamental tones and harmonic order** setting in the Configuration block. This setting sacrifices performance for compatibility by guaranteeing that every specified **Carrier Frequency** parameter appears in the set of SimRF simulation frequencies. The performance of the `ex_simrf_tut_filter` can be improved by reducing the size of the set of simulation frequencies, as measured by the **Total simulation frequencies** value displayed in the Configuration block dialog.

To enhance performance of the `ex_simrf_tut_filter` model, follow the procedure below.

- 1** Open and simulate the `ex_simrf_tut_filter` model.
- 2** Double-click the Configuration block to open the block dialog box. Note that the block dialog displays a **Total simulation frequencies** value of 121. This value indicates that the environment is running 121 separate simulations.
- 3** Clear the **Automatically select fundamental tones and harmonic order** check box.
- 4** Set **Fundamental tones** to [`RFCarrier`, `LOFrequency`]. This step is not strictly necessary, but setting these values clarifies the meaning of the parameters. Because the mixer has  $f_{RF}$  and  $f_{LO}$  at its input ports, all signals at the output of the mixer have carrier frequencies that are linear combinations of these fundamental tones.
- 5** Set **Harmonic order** to 1. The frequencies  $f_{RF} + f_{LO}$  and  $f_{RF} - f_{LO}$  are the only output carriers of interest. Modeling signals at a higher harmonic order than one is not necessary for this system.
- 6** Click **Apply**. Note that the block dialog displays a **Total simulation frequencies** value of 9.
- 7** Simulate the model.

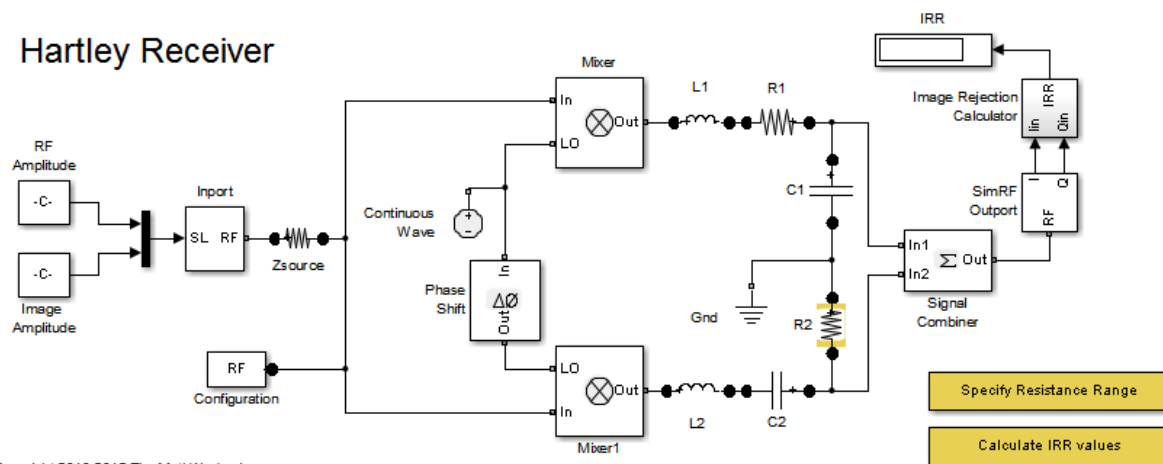
The result of the simulation has not changed because every frequency of importance appears in the new set of simulation frequencies. However, this procedure reduces overall compatibility. If you make modifications to the model, such as adding nonlinear amplification, the resulting signals of interest may not appear in the set of simulation frequencies. You can restore compatibility by restoring the **Automatically select fundamental tones and harmonic order** check box to its default.

## Measuring Image Rejection Ratio in Receivers

This example shows how to use the SimRF™ Circuit Envelope library to calculate the image rejection ratio (IRR) for high-side-injection in Weaver and Hartley receivers. The Weaver receiver shows the effect of phase offset on IRR, and the Hartley receiver shows a similar effect for resistor variation.

```
model1 = 'simrfV2_hartley';
open_system(model1);
```

### Hartley Receiver



### System Architecture

The RF system consists of:

- An Inport block that assigns multiplexed outputs of the RF and Image, by using Simulink Constant blocks, to the carriers  $f_{c\_RF}$  and  $f_{c\_IM}$  respectively. Real and imaginary values of the Constant blocks are matched to in-phase and quadrature carrier components.
- First stage mixers that mix the input signal with a local oscillator modeled by a Continuous Wave block with frequency  $f_{c\_LO}$ . The LO frequency is the average of the RF and image frequencies, so both signals are mixed down to the same frequency,  $f_{c\_IF}$ . The LO phase is shifted 90 degrees in one mixer relative to the other.

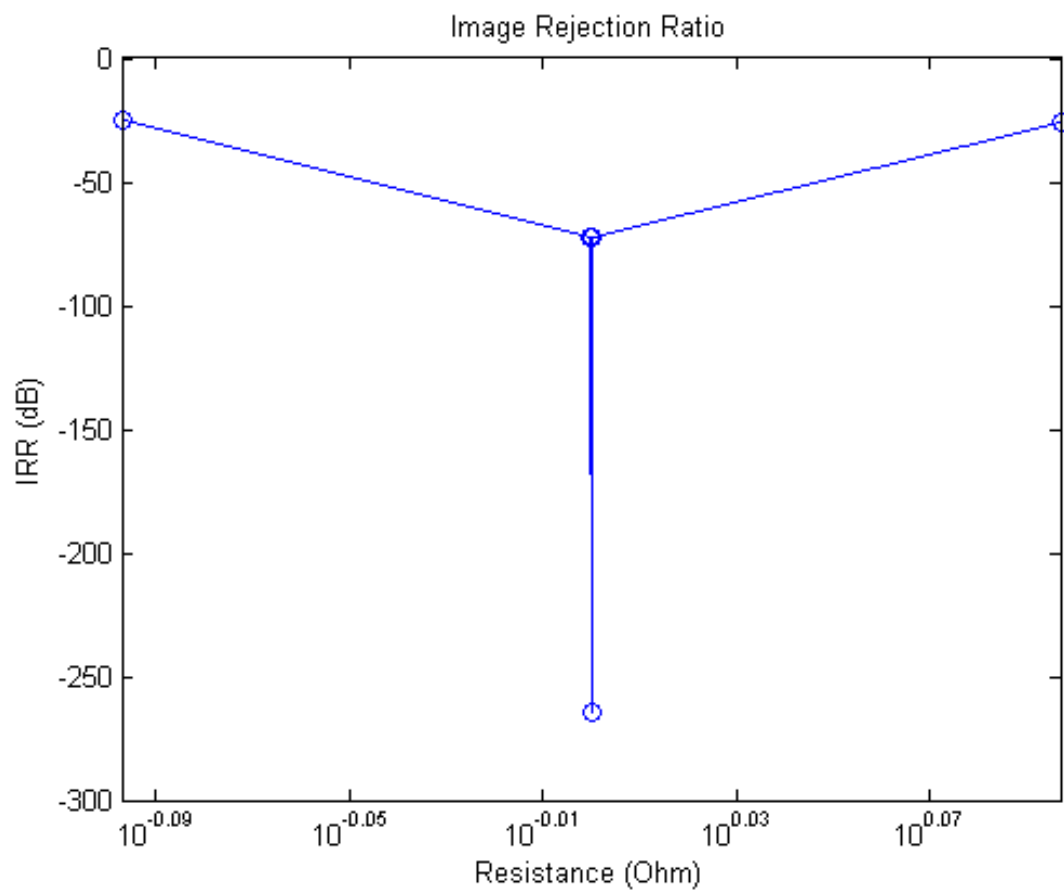
- The second stage of the Hartley uses a frequency independent RC-CR network to produce an additional 90 degree phase shift between the two signal paths, while the Weaver employs two additional mixers for channel selection.
- A Signal Combiner block that sums the voltage signals at its two inputs to yield the RF signal. If the Signal Combiner block is used to perform subtraction, the image can be obtained instead of the RF signal at its output. For low-side-injection, the Signal Combiner block needs to perform subtraction.
- The values of the in-phase and quadrature components of the RF and image signals are chosen to reduce the number of IRR calculations and facilitate reuse of the Image Rejection Calculator.

### Simulating the Hartley Receiver

- 1** Type `open_system('simrfv2_hartley')` at the Command Window prompt.
- 2** Double-click 'Specify Resistance Range' and specify a set of resistance values for the highlighted resistor.
- 3** Double-click 'Calculate IRR values' to execute a script, `simrfv2_hartley_callback`, that simulates the model once for each specified resistance value and generates a plot.

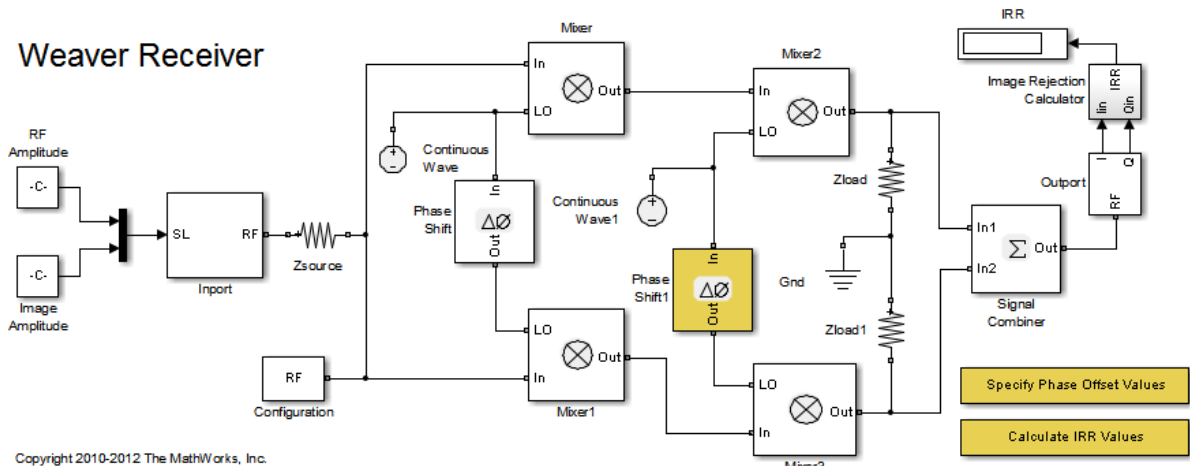
The sensitivity of the architecture to the component variation is shown by simulating the system multiple times, varying the resistance of the highlighted Resistor block at each iteration. When the highlighted resistor has a resistance of 1 Ohm, the images sum to zero in the Signal Combiner block and the IRR is minus infinity.

```
evalc('simrfv2_hartley_callback');
```



```
bdclose(model1);  
model2 = 'simrfV2_weaver';  
open_system(model2);
```



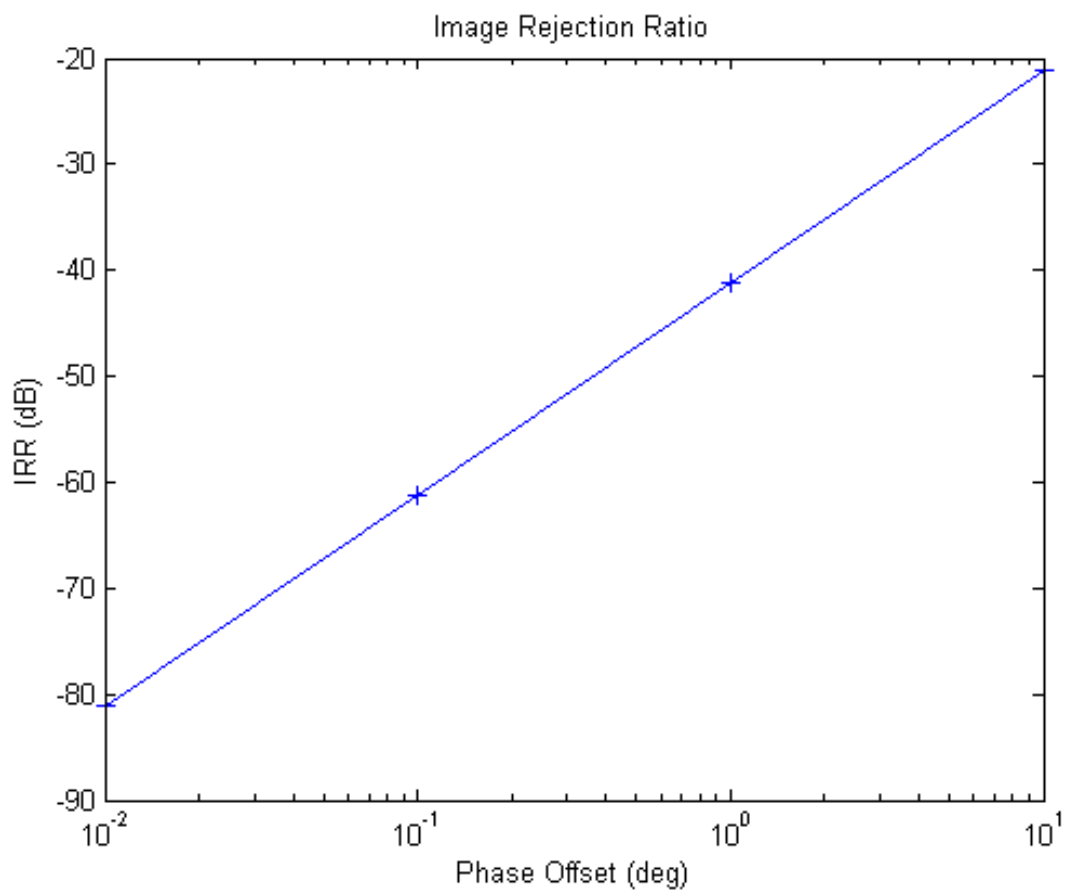


### Simulating the Weaver Receiver

- 1 Type `open_system('simrfv2_weaver')` at the Command Window prompt.
- 2 Double-click 'Specify Phase Offset Values' and specify a set of phase offset values.
- 3 Double-click 'Calculate IRR values' to execute a script, `simrfv2_weaver_callback`, that simulates the model once for each specified offset and generates a plot.

The sensitivity of the architecture to LO phase offset is shown by simulating the system multiple times, varying the phase offset of the highlighted Phase Shift block at each iteration. When the phase offset of the highlighted Phase Shift block is zero, the images sum to zero in the Signal Combiner block and the IRR is minus infinity.

```
evalc('simrfv2_weaver_callback');
```



`bdclose(model2)`

# 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
“Equivalent Baseband Library” on page 3-4
“Idealized Baseband 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<sup>®</sup> Charts.

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

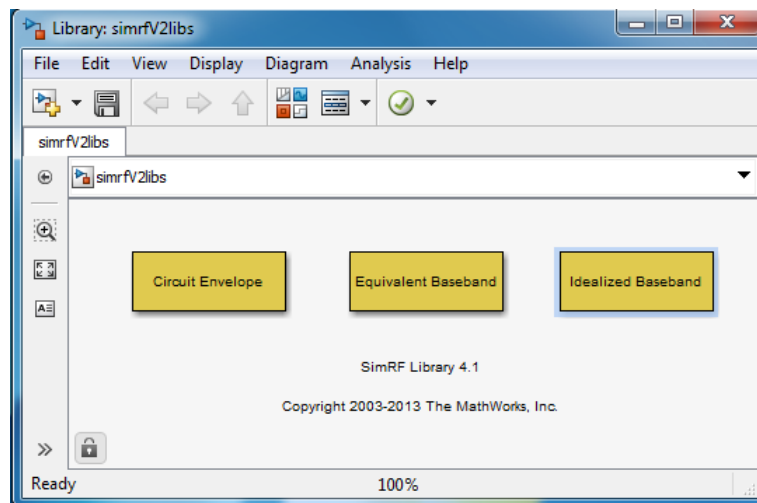
You can also use the blockset with Simulink Coder™ software to generate embeddable C code for real-time execution.

## 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.



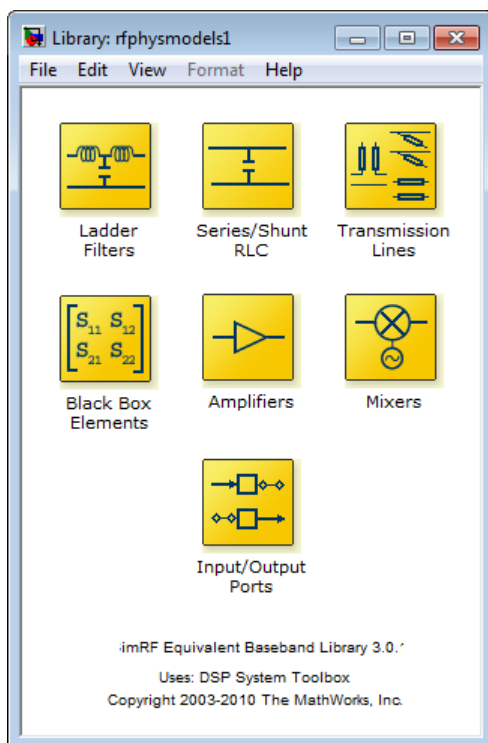
For a discussion of the Equivalent and Idealized Baseband libraries, see the following sections.

**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.
- 

## Equivalent Baseband Library

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





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

<b>Sublibrary</b>	<b>Description</b>
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><b>Note</b> 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”.

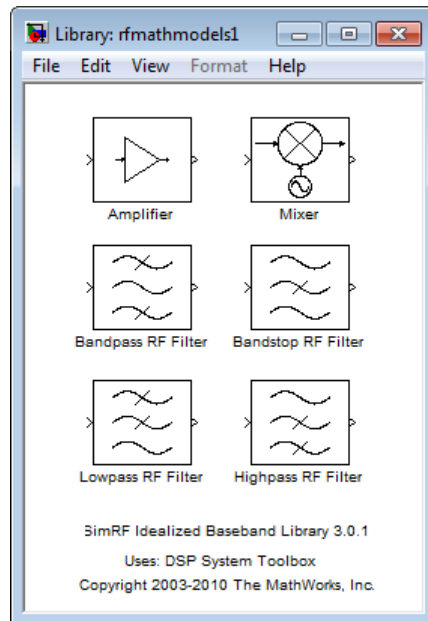
### **Idealized Baseband Library**

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

Idealized Baseband 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 Equivalent Baseband 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 Equivalent Baseband 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 3-12
“Validate Filter Components and Run the Simulation” on page 3-19
“Analyze the Simulation Results” on page 3-21

### 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.

<b>Block</b>	<b>Description</b>
<b>Sine Wave</b>	Generates a three-channel signal.
<b>Matrix Sum</b>	Combines the three channel signal into a single three-tone source signal.
<b>Input Port</b>	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.
<b>LC Bandpass Pi</b>	Models the signal attenuation caused by the RF filter which, in this example, is the LC Bandpass Pi filter.
<b>Output Port</b>	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.
<b>Spectrum Analyzer</b>	Displays signals at the input to and output of the filter.

## Build the Model

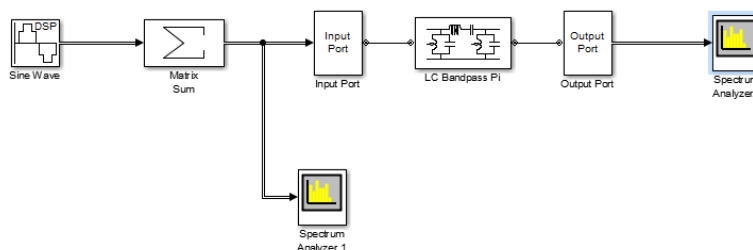
In this part of the example, you create a Simulink model, add blocks to the model, and connect the blocks.

- 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 Analyzer	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 3-18

### 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”.

---

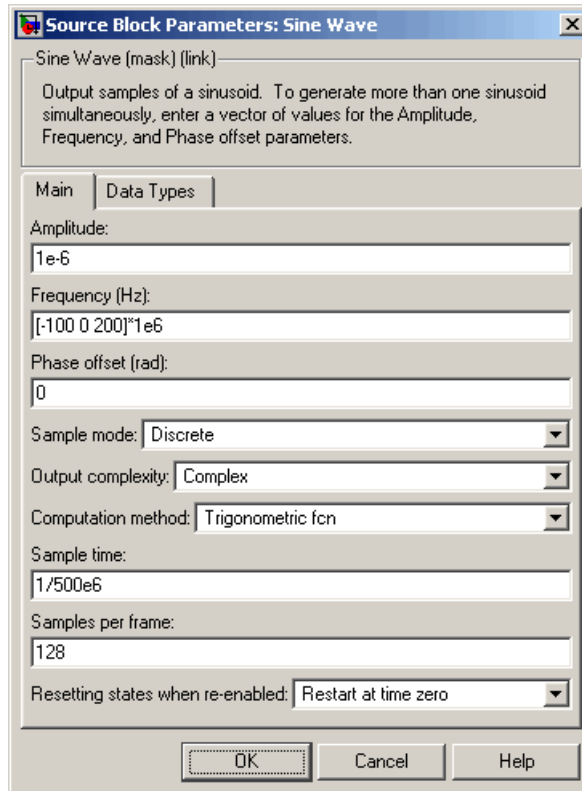
**Note** All signals in the RF model must be complex to match the signals in the physical subsystem, so you create a complex input signal.

---

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.

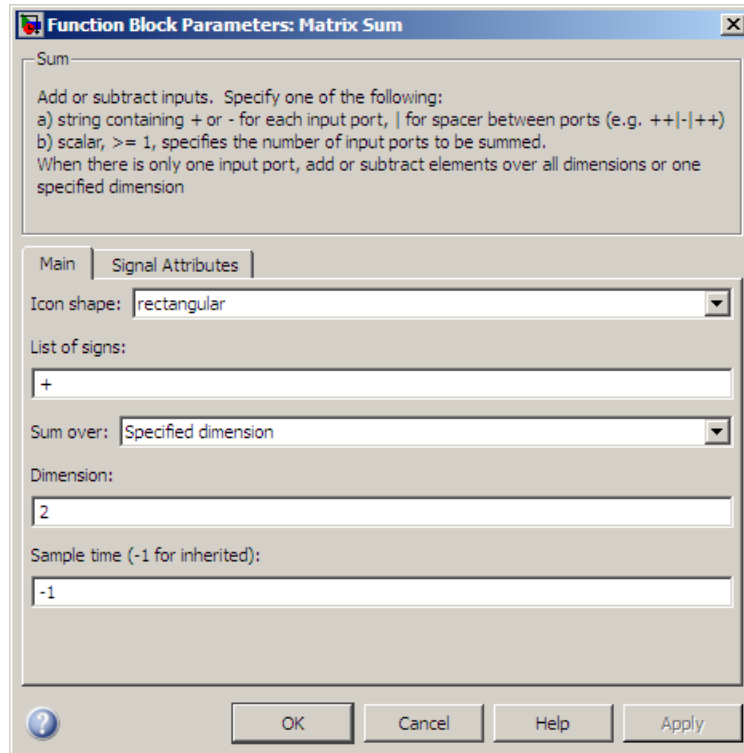
**1** In Sine Wave block dialog box:

- Set the **Amplitude** parameter to  $1e-6$ .
- Set the **Frequency (Hz)** parameter to  $[-100\ 0\ 200]*1e6$ .
- 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**

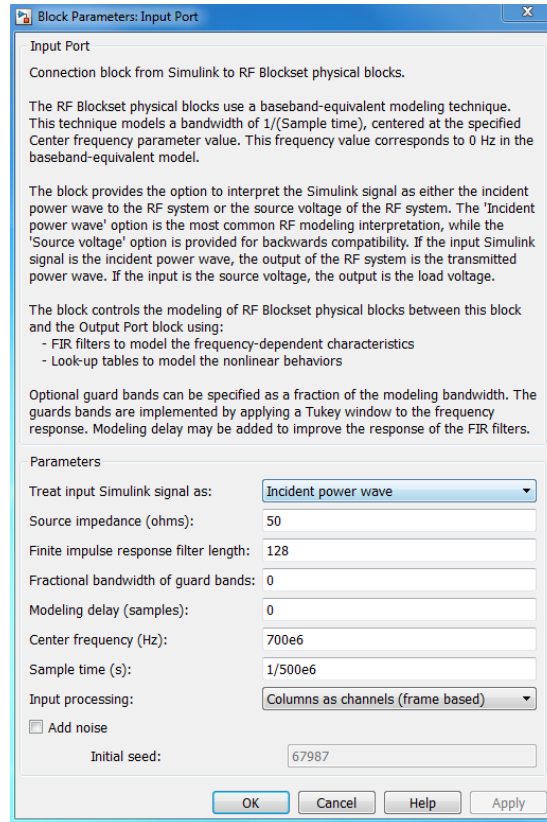
This option tells the blockset to interpret the input signal as the incident power wave to the RF subsystem, and not the source voltage of the RF subsystem.

---

**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** = 700e6
- **Sample time (s)** = 1/500e6
- **Input Processing** = Columns as channels (frame based)
- 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”.

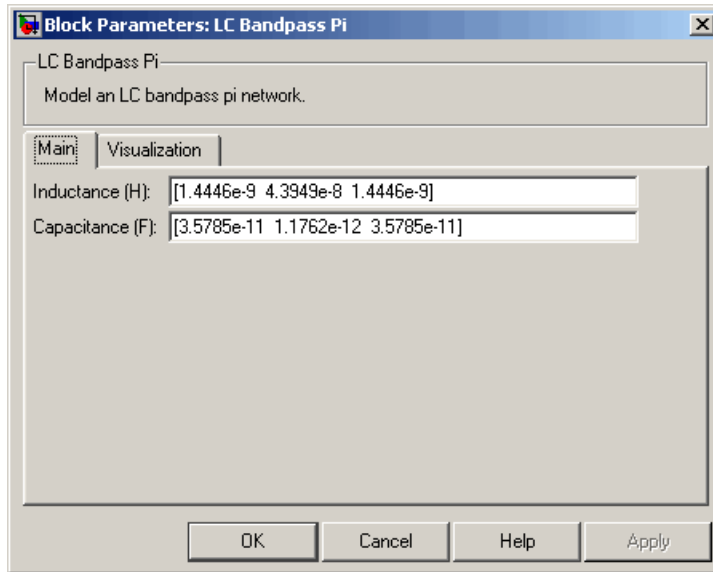


---

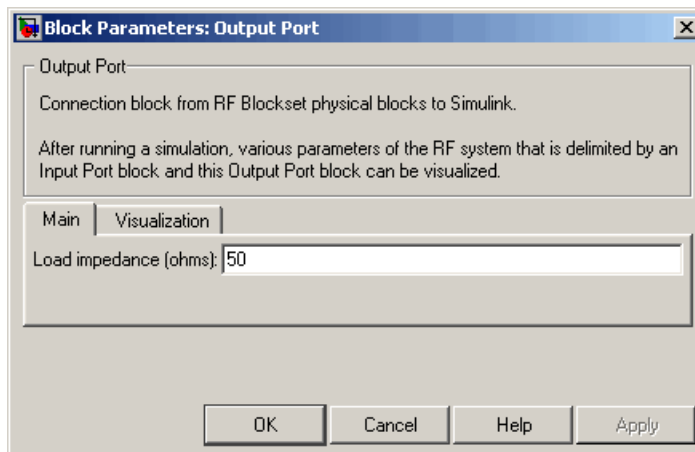
**Note** You must enter the **Sample time (s)** because the Input Port block does not inherit a sample time from the input signal. The specified sample time must match the sample time of the input signal. The **Sample time (s)** of  $1/500e6$  second used in this example is equivalent to a bandwidth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

- 1 Set the Spectrum Analyzer 1 parameters as follows:
  - In the **View** tab check **Spectrum Settings**. In Trace options set the **Units** parameter to dBm.
  - In the **View** tab open **Configuration Properties**. Set the **Minimum Y-limit** parameter to -291 and the **Maximum Y-limit** parameter to -67. Also, set the **Y-axis label** parameter to dBm.
- 2 Set the Spectrum Analyzer 1 parameters as follows:
  - In the **View** tab check **Spectrum Settings**. In Trace options set the **Units** parameter to dBm.
  - In the **View** tab open **Configuration Properties**. Set the **Minimum Y-limit** parameter to -291 and the **Maximum Y-limit** parameter to -67. Also, set the **Y-axis label** parameter to dBm.

## Validate Filter Components and Run the Simulation

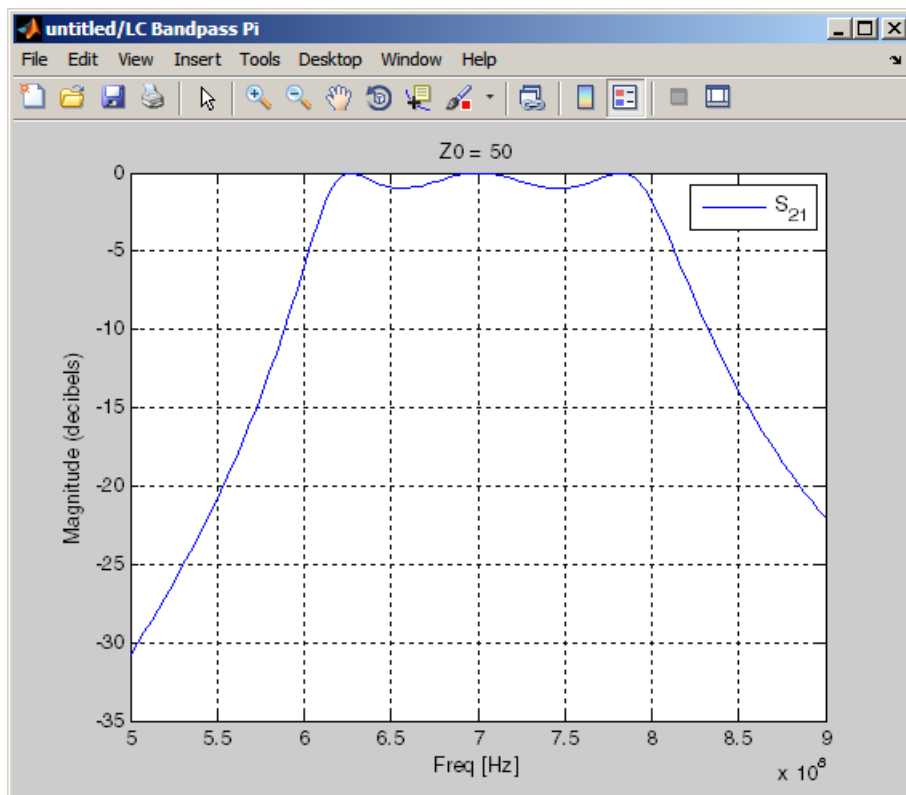
In this part of the example, you validate the behavior of the LC Bandpass Pi filter block by plotting its frequency response and then run the simulation.

---

**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 3-21
- “Plot Model Parameters of the Filter Subsystem” on page 3-23

### Compare the Input and Output Signals of the RF Filter

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

The Spectrum Analyzer 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 Analyzer 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 Analyzer block dialog boxes. For more information on complex-baseband modeling, see “Create a Complex Baseband-Equivalent Model”.

The Spectrum Analyzer 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 Analyzer 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).

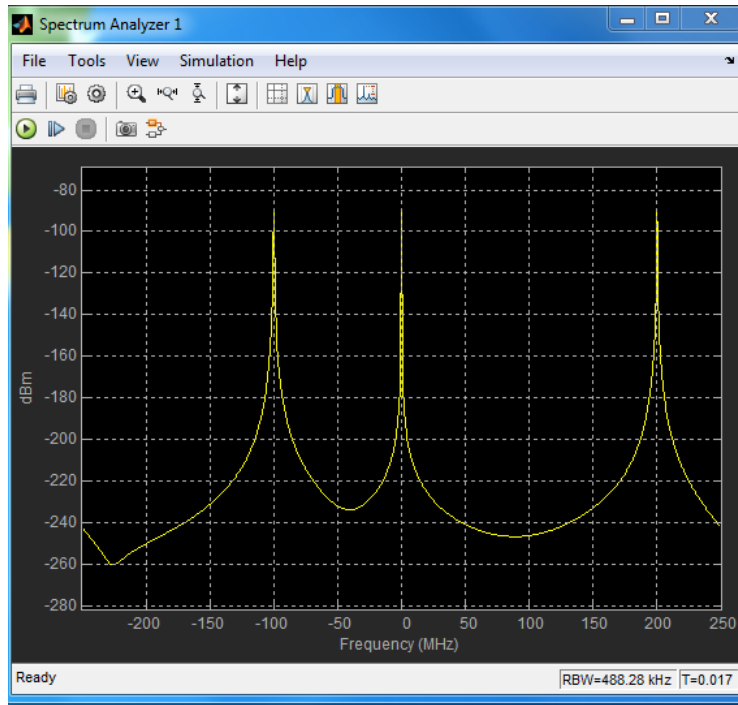
---

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

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

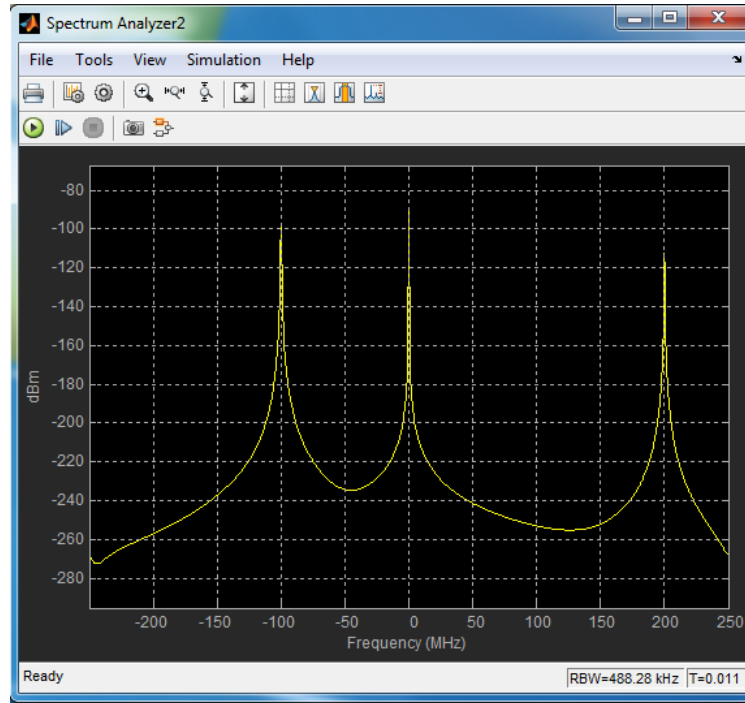

---

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.

---

