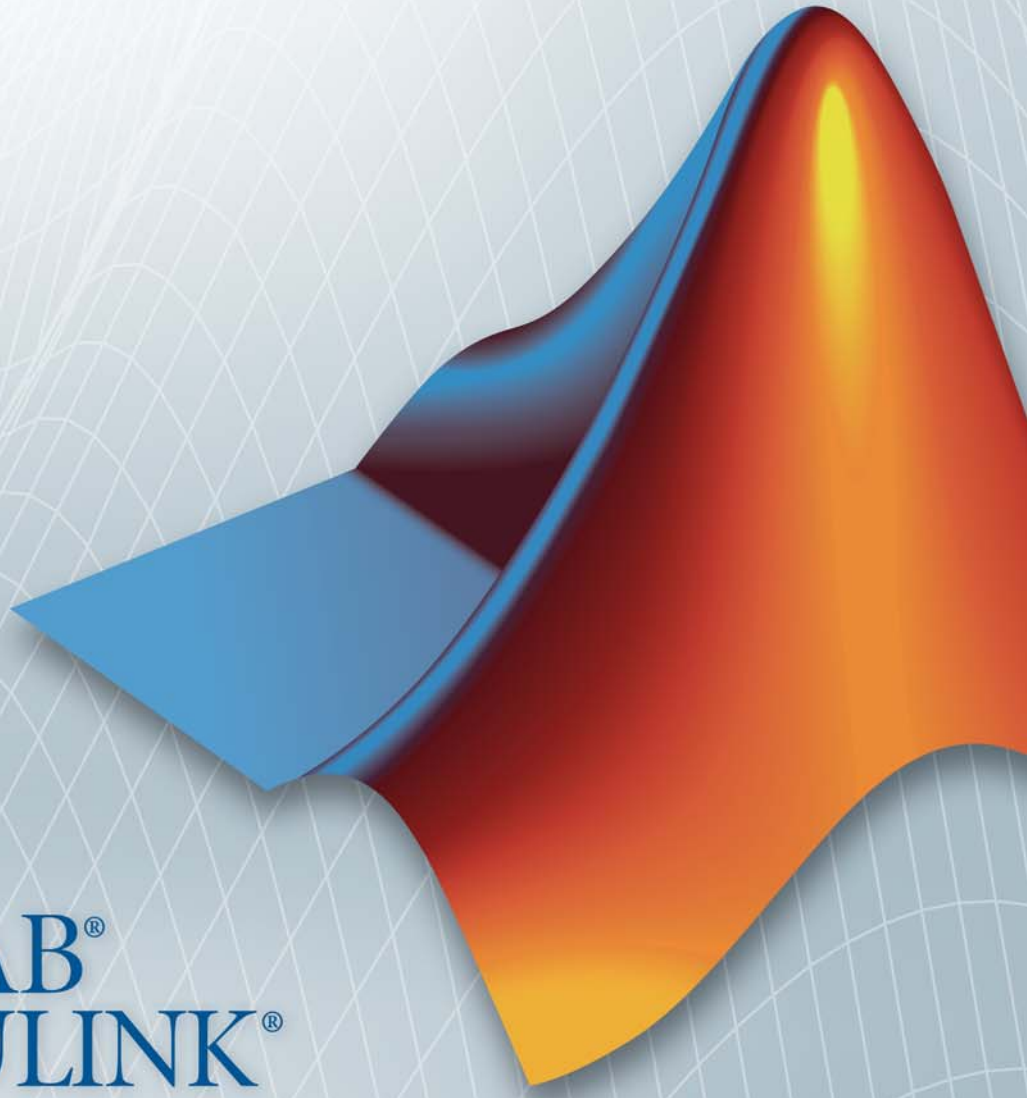
SimRF™

Getting Started Guide

R2011b



MATLAB®
&SIMULINK®

MathWorks®

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 2010–2011 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)

Contents

1 Product Overview

Overview of RF Simulation 1-2

Efficient RF Simulation with SimRF Software 1-3
Simulate a Passband Signal in Simulink Software 1-3
Compare Passband and Baseband Signals in SimRF Software 1-5
Simulate the Envelope of a Baseband Signal 1-9

Working with SimRF Software 1-12
Understanding the SimRF Environment 1-12
Required and Related Products 1-13
Using the SimRF Documentation 1-13

2 Frequency Conversion

Specify the Circuit Envelope Simulation Environment 2-2
Access Circuit Envelope Libraries 2-2
Ground the RF System 2-3
Configure the SimRF Environment 2-4
Simulate the SimRF Environment 2-5

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 2-13

Filter Mixing Products **2-15**
Probe Multiple RF Carriers **2-16**
Model an RF Filter **2-17**
Simulate Filtering of RF Signals **2-18**

1

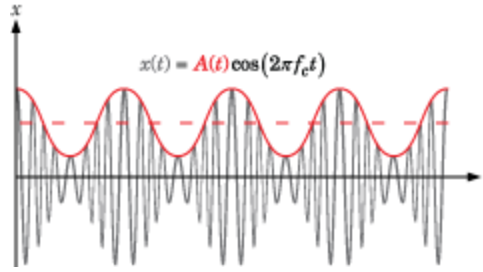
Product Overview

- "Overview of RF Simulation" on page 1-2
- "Efficient RF Simulation with SimRF Software" on page 1-3
- "Working with SimRF Software" on page 1-12

Overview of RF Simulation

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.

Traditional methods of circuit simulation are not ideal for RF systems. High-frequency carriers require very small time steps. This approach results in a long simulation time. Circuit-envelope simulation is a solution for modeling RF signals accurately, while also reducing simulation time.



The preceding figure illustrates a signal $x(t)$ for which circuit-envelope simulation is ideal. The signal consists of a time-varying modulation on a high-frequency carrier. In many RF applications, the frequency of the modulation $A(t)$ is orders of magnitude 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 modulation. The redefined system that the software uses is mathematically equivalent to the original. However, by taking time steps on the scale of the modulation instead of the carrier, the software produces equivalent results in less time.

Efficient RF Simulation with SimRF Software

In this section...

"Simulate a Passband Signal in Simulink Software" on page 1-3

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

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

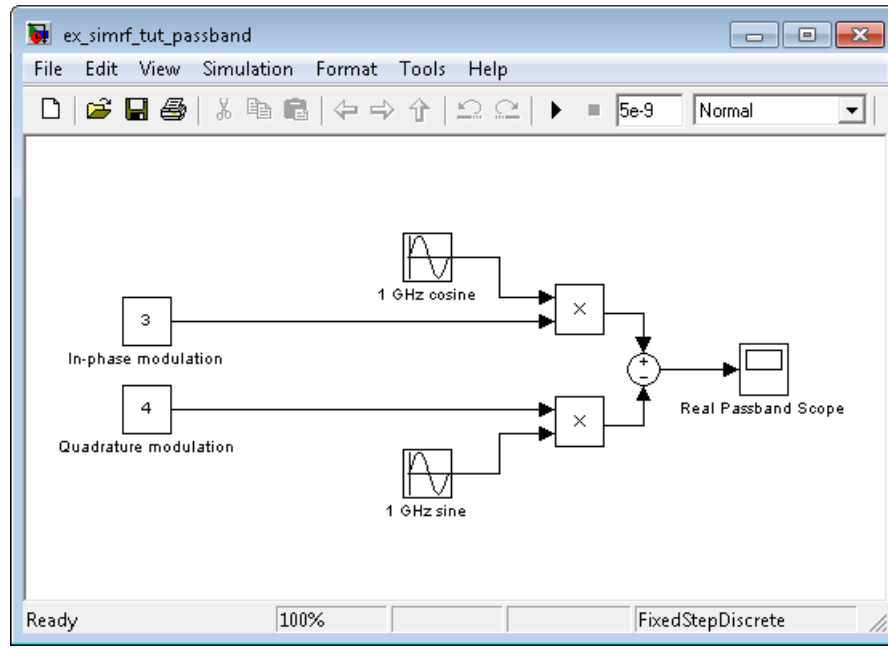
This example uses circuit-envelope simulation to reduce the number of time steps required to simulate a simple signal by a factor of 100.

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 model the modulation of a real passband signal with in-phase and quadrature components.



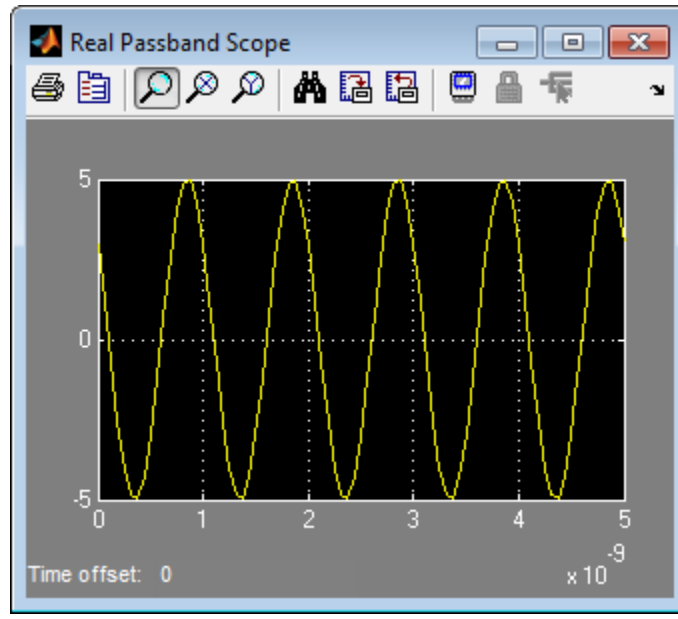
The system specifies a real passband signal $x(t)$ according to the formula

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

where:

- $I(t)$ is the in-phase part of the modulation, equal to 3 in this example, modeled by the Constant block labeled In-phase modulation.
- $Q(t)$ is the quadrature part of the modulation, 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 consistent with the specified in-phase and quadrature amplitudes.

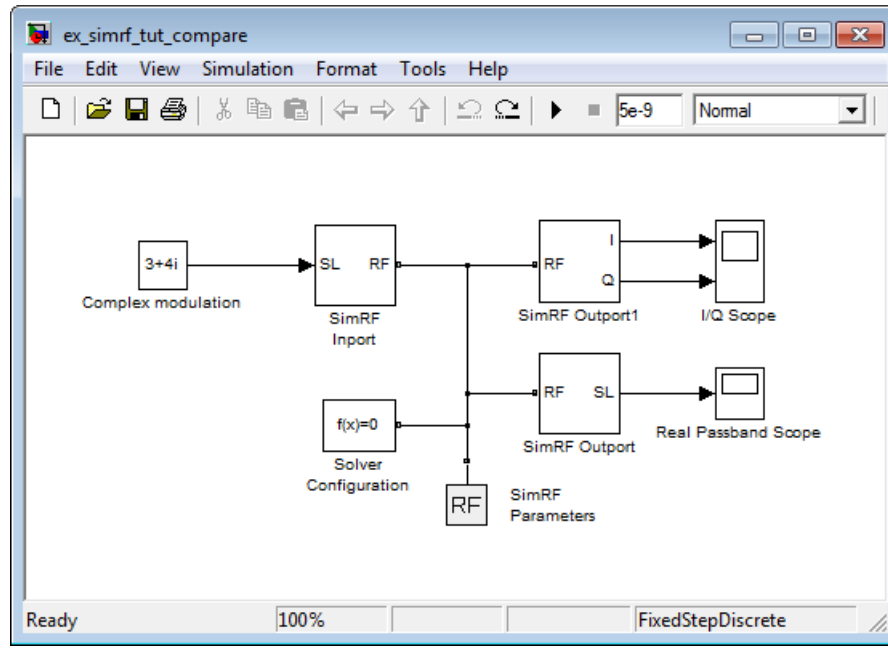
In the Configuration Parameters dialog box, the **Fixed-step size (fundamental sample time)** parameter has been set to `1/20*1e-9`. This value is on the order of the wavelength of the carrier. The simulation takes a total of 101 samples—20 per cycle.

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 1-3.



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

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

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:

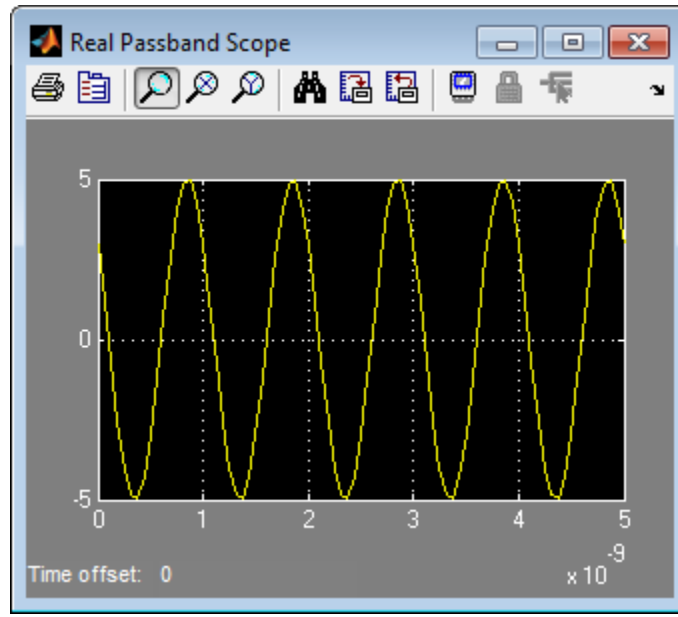
- In the SimRF Inport block, the **Carrier Frequencies** parameter defines the carrier frequencies of the modulations entering from outside the SimRF

environment. In this example, there is only one input signal, and only one carrier (1 GHz, specified as `1e9 Hz`).

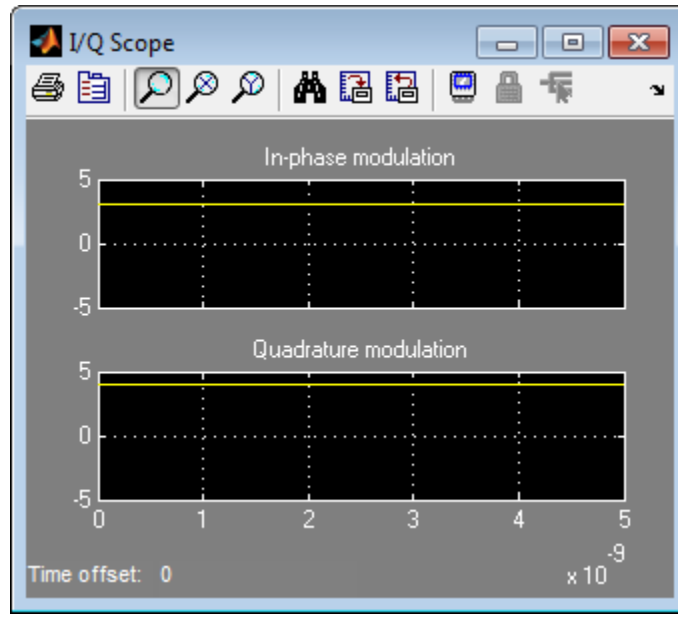
- In the SimRF Outport block, the **Carrier Frequencies** parameter specifies the signal on the `1e9 Hz` carrier (1 GHz) as Simulink signals. These signals appear at the I and Q ports. The **Output** parameter is set to `Real Passband`, so this signal represents a real passband signal on the 1-GHz carrier.
- 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 1-3. 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 1-3.

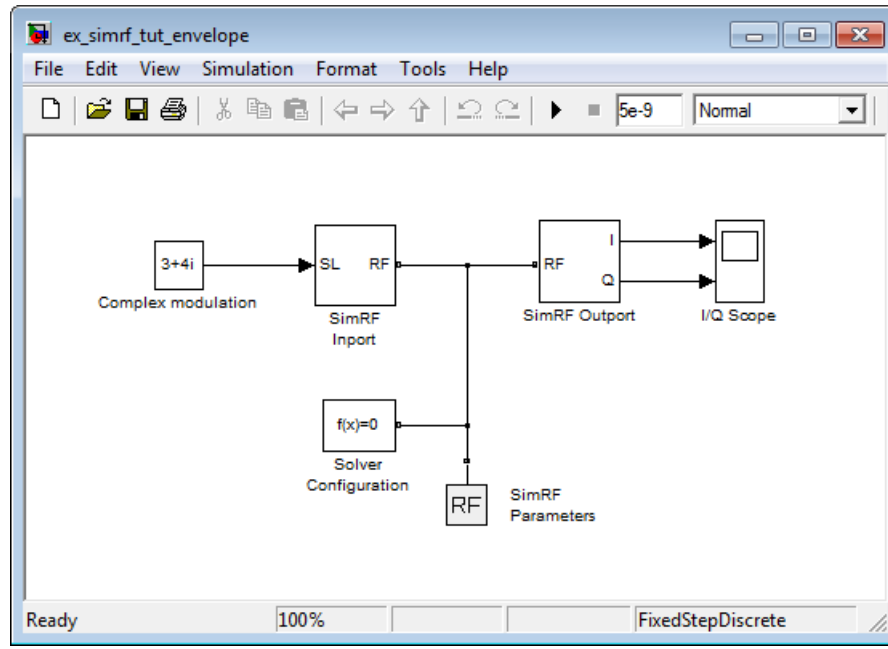
In the Configuration Parameters dialog box, the **Fixed-step size (fundamental sample time)** parameter has been set to `1/20*1e-9`. This value is on the order of the wavelength of the carrier. The simulation takes a total of 101 samples—20 per cycle.

Simulate the Envelope of a Baseband Signal

The model

```
ex_simrf_tut_envelope
```

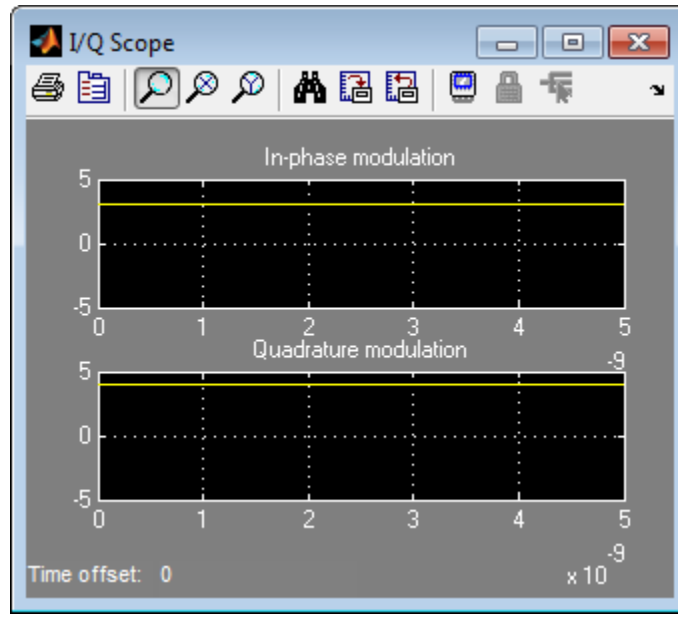
simulates the envelope of a 1-GHz sine wave using SimRF blocks. This example builds on the results of the previous section, "Compare Passband and Baseband Signals in SimRF Software" on page 1-5.



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. However, this system does not output a real passband signal.
- 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*1e-9`.

Running the model produces the following output at the scope.



The I/Q Scope displays the in-phase and quadrature modulations 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. Here, Simulink does not have to simulate this system using a fundamental sample time on the order of the wavelength of the carrier. Because the modulations are constant in this example, only two sample points are needed.

The model uses a value of `5e-9` for the **Fixed-step size (fundamental sample time)** parameter. This value equals the **Stop time** because the modulations are constant. Compared to the preceding examples, which use a sample time of `1/20*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.

Working with SimRF Software

In this section...
"Understanding the SimRF Environment" on page 1-12
"Required and Related Products" on page 1-13
"Using the SimRF Documentation" on page 1-13

Understanding the SimRF Environment

Groups of interconnected SimRF blocks and the algorithms that model the RF system that they represent are collectively referred to in this manual as 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 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 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.

Required and Related Products

Several MathWorks® products are relevant to the tasks that you can perform with the SimRF software and extend the capabilities of MATLAB®.

To install SimRF software, you must have the following products:

- RF Toolbox™ — MATLAB functions for modeling RF systems.
- Simscape — Software tools for modeling and simulating multidomain physical systems.

SimRF software benefits from, but does not require, installation of the following recommended products:

- Communications System Toolbox — Simulink blocks and MATLAB functions for time-domain simulation of modulation and demodulation of a wireless communications signal.
- DSP System Toolbox — Simulink blocks and MATLAB functions for time-domain simulation of filtering of a modulated communications signal.

Using the SimRF Documentation

- “Expected Background” on page 1-14
- “What Chapters Should I Read?” on page 1-14

Expected Background

Topics in this guide assume that you are already familiar with:

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

What Chapters Should I Read?

If you are using SimRF software for the first time, read the "Efficient RF Simulation with SimRF Software" on page 1-3 and Chapter 2, "Frequency Conversion" sections to learn how to build SimRF models and to gain familiarity with the SimRF environment. Once you are familiar with the concepts in this guide, read the following sections of the documentation:

- For information on applying SimRF software to RF receiver design, refer to the *SimRF User's Guide*.
- To learn more about the applications of SimRF software and how SimRF software interacts with other MathWorks products, see the Demos section.
- For a detailed explanation of each SimRF block, see the Blocks — Alphabetical List .
- For information on simulation using the Simscape platform, see Basic Principles of Modeling Physical Networks.

2

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 2-15

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 2-4

"Simulate the SimRF Environment" on page 2-5

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.

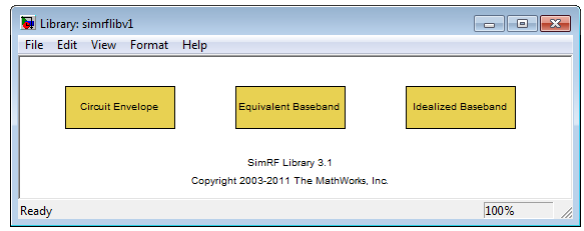
To begin, open a new model by selecting **File > New > Model**.

Access Circuit Envelope Libraries

Open the SimRF library by clicking the **Start** button and selecting **Simulink > SimRF > Block Library (Circuit Envelope)**, or type

```
simrflib
```

at the MATLAB Command Window prompt.



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.

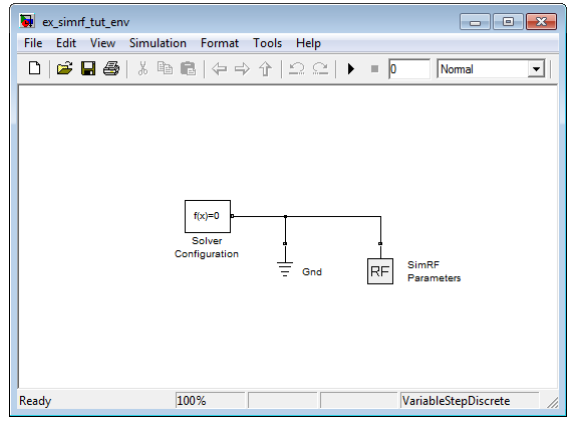
Configure the SimRF Environment

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 > Start**.

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.

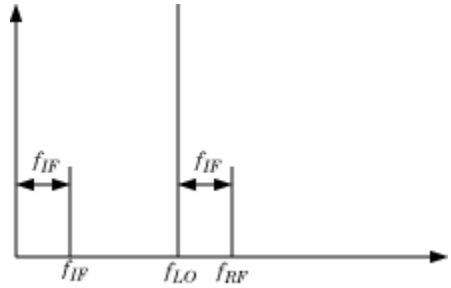
Model an RF Mixer

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 2-13

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
```

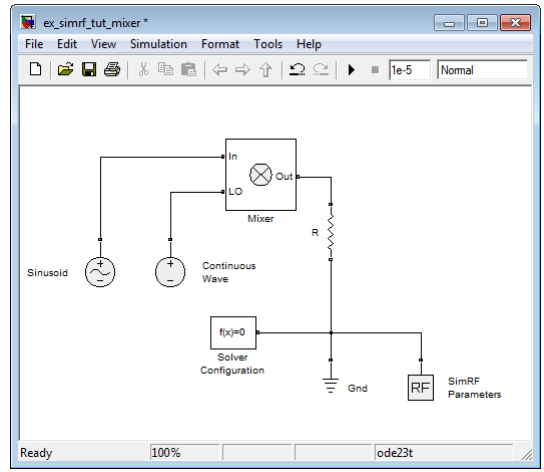
at the MATLAB Command Window prompt.

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.

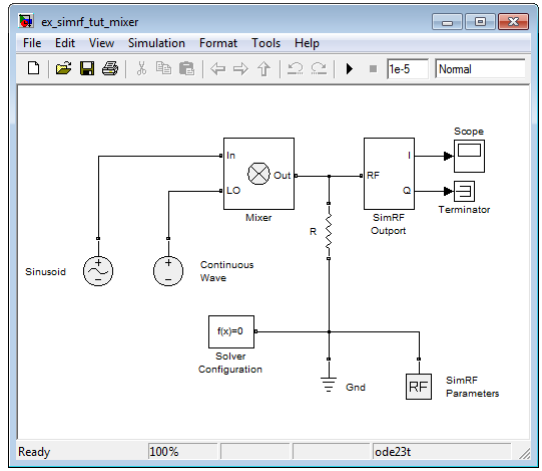
For more information on organizing blocks on the Simulink canvas, see "Changing a Block's Appearance" in the Simulink User's Guide.

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 Outport block as a voltage sensor at the output port of the mixer. The Simulink signal at the outport is the envelope of the carrier or carriers specified in the block. A scope attached to the outport 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*.

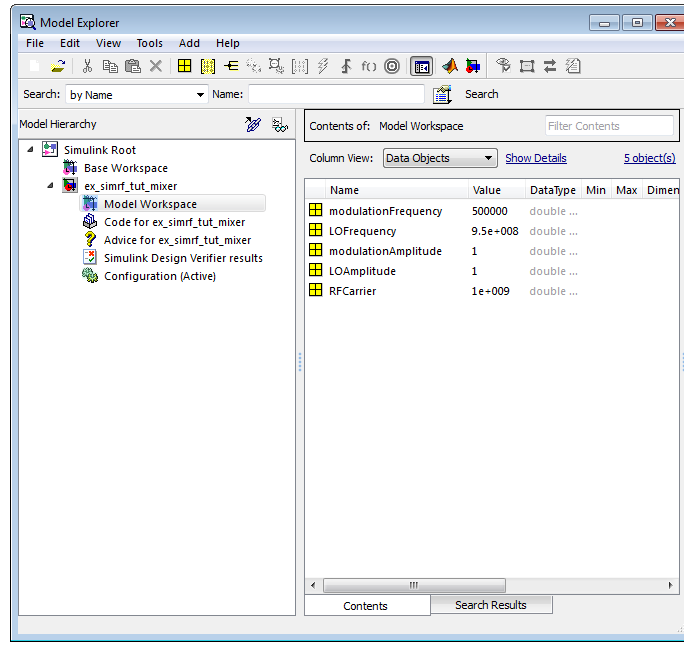
Using the Model Explorer to Define Variables

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 > 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 > Start** to run the model.

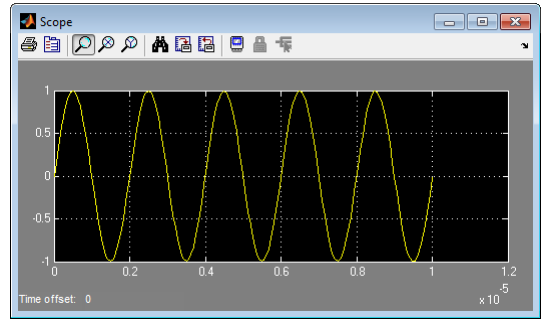
Models that contain SimRF Amplifier and Mixer blocks generate files at update time. Before you can successfully update and run models with these blocks, you have to set up a compiler by running `mex -setup`.

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 box. To specify a cache folder:

1. Open the Simulink Preferences dialog box (**File > Preferences**).
2. Specify a location for the **Simulink cache folder** parameter.

For more information about the Simulink interface, see Simulink Preferences Window.

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



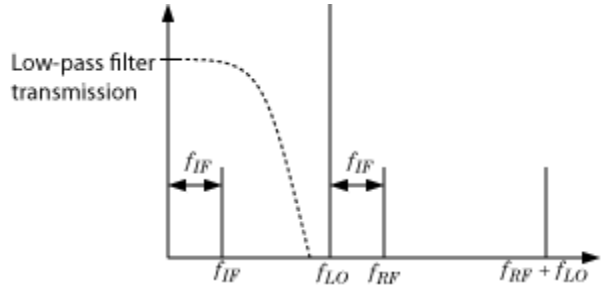
The Sinusoid specifies a 1-V amplitude for the modulation on f_{RF}, which the mixer downconverts to f_{IF}. The SimRF Outport block probes the intermediate frequency and recovers the 1-V modulation amplitude. This value agrees with a specified conversion gain of 0 dB in the mixer.

Filter Mixing Products

In this section...
"Probe Multiple RF Carriers" on page 2-16
"Model an RF Filter" on page 2-17
"Simulate Filtering of RF Signals" on page 2-18

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

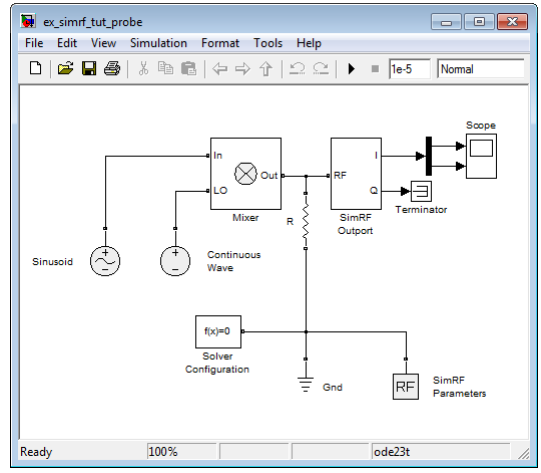
The SimRF environment specifies the carriers $f_{RF} - f_{LO}$, f_{RF}, f_{LO}, and $f_{RF} + f_{LO}$, but the SimRF Outport block probes only the carrier $f_{RF} - f_{LO}$. Examine the carriers $f_{RF} - f_{LO}$ and $f_{RF} + f_{LO}$ by changing the model according to the following workflow:

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 > Start** to run the model.

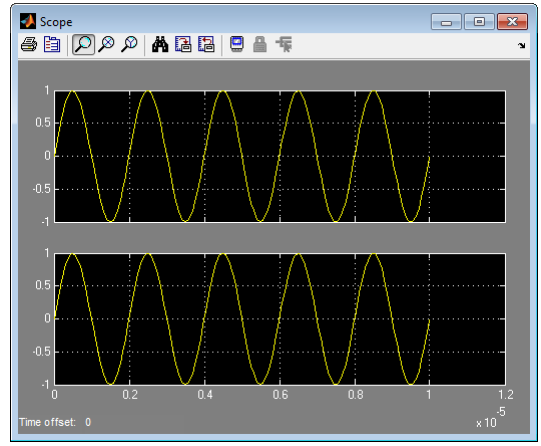
Models that contain SimRF Amplifier and Mixer blocks generate files at update time. Before you can successfully update and run models with these blocks, you have to set up a compiler by running `mex -setup`.

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 box. To specify a cache folder:

1. Open the Simulink Preferences dialog box (**File > Preferences**).
2. Specify a location for the **Simulink cache folder** parameter.

For more information about the Simulink interface, see Simulink Preferences Window.

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 2-16 section, or type

```
ex_simrf_tut_probe
```

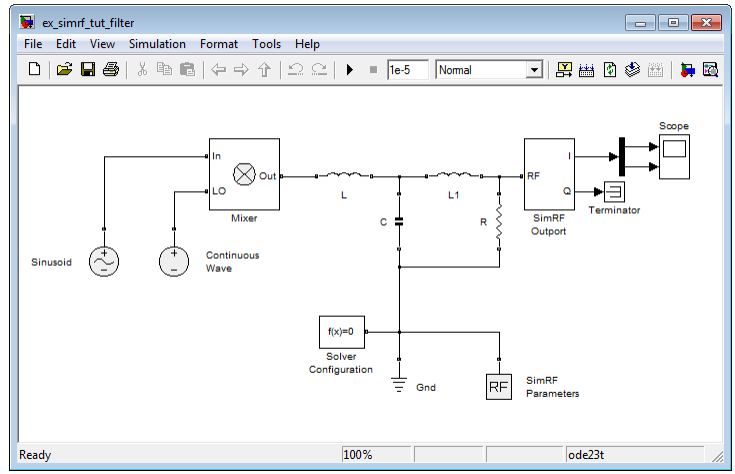
at the MATLAB Command Window prompt.

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.

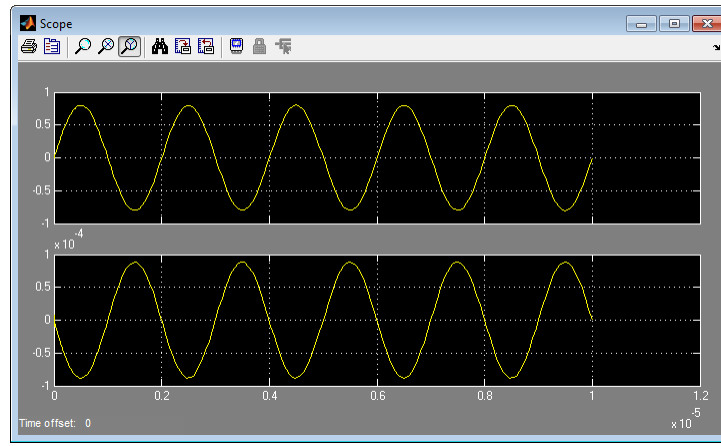
Simulate Filtering of RF Signals

To begin, open the model that you created in the previous section, “Model an RF Filter” on page 2-17, or type

```
ex_simrf_tut_filter
```

at the MATLAB Command Window prompt. Select **Simulation > Start** 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 demo AC Analysis of an RF System calculates the voltage transfer function of an RF filter using harmonic balance analysis.