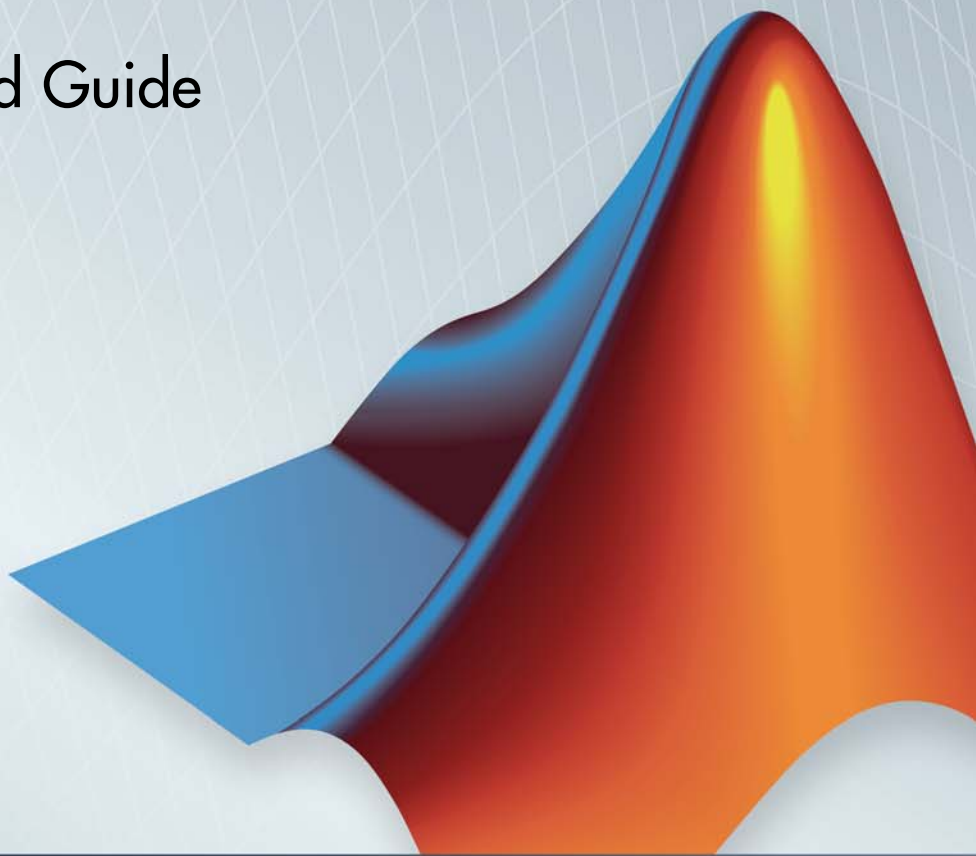


HDL Coder™

Getting Started Guide

R2014a



MATLAB®



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.

HDL Coder™ Getting Started Guide

© COPYRIGHT 2012–2014 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

March 2012	Online only	New for Version 3.0 (Release 2012a)
September 2012	Online only	Revised for Version 3.1 (Release 2012b)
March 2013	Online only	Revised for Version 3.2 (Release 2013a)
September 2013	Online only	Revised for Version 3.3 (Release 2013b)
March 2014	Online only	Revised for Version 3.4 (Release 2014a)

Getting Started with HDL Coder

1

HDL Coder Product Description	1-2
Key Features	1-2
Prepare Simulink Model For HDL Code Generation ..	1-3
Tool Setup	1-4
C/C++ Compiler Setup	1-4
Synthesis Tool Path Setup	1-4
HDL Simulator Setup	1-6
Xilinx System Generator Setup	1-7
Altera DSP Builder Setup	1-8
Xilinx FPGA Floating-Point Library Setup	1-8
Supported Third-Party Tools and Hardware	1-9
Third-Party Synthesis Tools	1-9
FPGA-in-the-Loop Hardware	1-9
Simulink Real-Time FPGA I/O Hardware	1-10
FPGA Turnkey Hardware	1-10
VHDL and Verilog Language Support	1-12

Tutorials

2

HDL Code Generation from a MATLAB Algorithm	2-2
About the Algorithm in This Example	2-2
Copying Files Locally	2-4
Checking Your Synthesis Tool Setup	2-4
Testing the Original MATLAB Algorithm	2-5
Setting Up an HDL Coder Project	2-6

Creating Fixed-Point Versions of the Algorithm and Test Bench	2-8
Generating HDL Code	2-12
HDL Code Generation from a Simulink Model	2-16
Before You Generate Code	2-16
Overview of Exercises	2-17
The <code>sfir_fixed</code> Model	2-17
Generate Code Using the HDL Workflow Advisor	2-20
Generate HDL Code Using the Command Line	2-29
Generate HDL Code Using the Configuration Parameters Dialog Box	2-38
Simulating and Verifying Generated HDL Code	2-49

Getting Started with HDL Coder

- “HDL Coder Product Description” on page 1-2
- “Prepare Simulink Model For HDL Code Generation” on page 1-3
- “Tool Setup” on page 1-4
- “Supported Third-Party Tools and Hardware” on page 1-9
- “VHDL and Verilog Language Support” on page 1-12

HDL Coder Product Description

Generate VHDL® and Verilog® code for FPGA and ASIC designs

HDL Coder™ generates portable, synthesizable VHDL and Verilog code from MATLAB® functions, Simulink® models, and Stateflow® charts. The generated HDL code can be used for FPGA programming or ASIC prototyping and design.

HDL Coder provides a workflow advisor that automates the programming of Xilinx® and Altera® FPGAs. You can control HDL architecture and implementation, highlight critical paths, and generate hardware resource utilization estimates. HDL Coder provides traceability between your Simulink model and the generated Verilog and VHDL code, enabling code verification for high-integrity applications adhering to DO-254 and other standards.

Key Features

- Target-independent, synthesizable VHDL and Verilog code
- Code generation support for MATLAB functions, System objects and Simulink blocks
- Mealy and Moore finite-state machines and control logic implementations using Stateflow
- Workflow advisor for programming Xilinx and Altera application boards
- Resource sharing and retiming for area-speed tradeoffs
- Code-to-model and model-to-code traceability for DO-254
- Legacy code integration

Prepare Simulink Model For HDL Code Generation

To prepare your Simulink model for code generation:

- Use `checkhdl` to check that blocks in your model are supported for HDL code generation.

You can save time by creating your model using blocks from the `hdlsupported` library, which you generate using `hdl1lib`. However, not all available combinations of supported blocks, block implementations, block properties, and HDL optimizations are supported for code generation. `checkhdl` can find many code generation incompatibilities, but you may need to modify your model later, to generate code that meets your hardware requirements.

- If you want to synthesize your generated HDL code, convert your model to fixed point.

To learn how to use Fixed-Point Designer™ to convert your floating-point model to fixed-point, see “Convert Floating-Point Model to Fixed Point”.

You can synthesize some floating-point constructs if your synthesis tool and hardware support it. To learn more, see “FPGA Floating-Point Libraries”.

- Run `hdlsetup` to configure the solver for HDL code generation.

Tool Setup

In this section...

“C/C++ Compiler Setup” on page 1-4

“Synthesis Tool Path Setup” on page 1-4

“HDL Simulator Setup” on page 1-6

“Xilinx System Generator Setup” on page 1-7

“Altera DSP Builder Setup” on page 1-8

“Xilinx FPGA Floating-Point Library Setup” on page 1-8

C/C++ Compiler Setup

HDL Coder requires a C/C++ compiler in order to compile the test bench for faster simulation.

For a list of supported compilers, see at http://www.mathworks.com/support/compilers/current_release/.

Note The LCC compiler is not supported in the current release. Please use one of the other compilers in the list at http://www.mathworks.com/support/compilers/current_release/.

Synthesis Tool Path Setup

- “hdlsetuptoolpath command” on page 1-4
- “Add Synthesis Tool for Current HDL Workflow Advisor Session” on page 1-6
- “Check Your Synthesis Tool Setup” on page 1-6

hdlsetuptoolpath command

To use HDL Coder with one of the supported third-party FPGA synthesis tools, you must add the tool to your system path using the `hdlsetuptoolpath`

command before opening the HDL Workflow Advisor. If you already have the HDL Workflow Advisor open, see “Add Synthesis Tool for Current HDL Workflow Advisor Session” on page 1-6.

The syntax of `hdlsetuptoolpath` is:

```
hdlsetuptoolpath ('ToolName', TOOLNAME, 'ToolPath', TOOLPATH)
```

The input name-value pairs are:

Name	Value	Description
'ToolName'	'Xilinx ISE' or 'Altera Quartus II'	Specifies the synthesis tool name.
'ToolPath'	' <i>full_path_to_executable</i> '	Specifies the full path to the synthesis tool executable.

For example, the following command sets the synthesis tool path to point to an installed Xilinx ISE 13.1 executable.

```
hdlsetuptoolpath('ToolName','Xilinx ISE', ...
'ToolPath', 'C:\Xilinx\13.1\ISE_DS\ISE\bin\nt64\ise.exe');
```

If you have an icon for the synthesis tool on your Windows® desktop, to find the tool path:

1 Right-click the icon and select **Properties**.

2 Click the **Shortcut** tab.

Tip `hdlsetuptoolpath` changes the system path and system environment variables for the current MATLAB session only. To execute `hdlsetuptoolpath` automatically when MATLAB starts, add `hdlsetuptoolpath` to your `startup.m` script.

Add Synthesis Tool for Current HDL Workflow Advisor Session Simulink to HDL Workflow

- 1** At the MATLAB command line, use the “hdlsetuptoolpath command” on page 1-4 to add the synthesis tool.
- 2** In the HDL Workflow Advisor, in the **Set Target > Set Target Device and Synthesis Tool** step, to the right of **Synthesis tool**, click **Refresh**.

The synthesis tool is now available.

MATLAB to HDL Workflow

- 1** At the MATLAB command line, use the “hdlsetuptoolpath command” on page 1-4 to add the synthesis tool.
- 2** In the HDL Workflow Advisor, in the **Select Code Generation Target** step, to the right of **Synthesis tool**, click **Refresh list**.

The synthesis tool is now available.

Check Your Synthesis Tool Setup

To check your Xilinx ISE synthesis tool setup in MATLAB, try launching the tool with the following command:

```
!ise
```

To check your Altera Quartus synthesis tool setup in MATLAB, try launching the tool with the following command:

```
!quartus
```

HDL Simulator Setup

To set up ModelSim®, Questa®, or Incisive® for HDL simulation, or for cosimulation with HDL Verifier™, see “HDL Simulator Startup”.

Add Simulation Tool for Current HDL Workflow Advisor Session MATLAB to HDL Workflow

- 1** Set up your simulation tool.

- 2 In the HDL Workflow Advisor, in the **HDL Verification > Verify with HDL Test Bench** task, click **Refresh list**.

The simulation tool is now available.

Xilinx System Generator Setup

To generate ModelSim simulation scripts for a design containing Xilinx System Generator blocks, you must:

- Have compiled Xilinx simulation libraries.
- Specify the path to your compiled libraries.

Required Libraries

You must have the following compiled Xilinx simulation libraries for your EDA simulator and target language to generate ModelSim simulation scripts:

- unisim
- simprim
- xilinxcorelib

Refer to the Xilinx documentation for `complib` to learn how to compile these libraries.

Specify Path to Required Libraries

Specify the path to your compiled Xilinx simulation libraries by setting the `XilinxSimulatorLibPath` parameter for your model.

For example, you can use `hdlset_param` to set `XilinxSimulatorLibPath`:

```
libpath = '/apps/Xilinx_ISE/XilinxISE-13.4/Linux/ISE_DS/ISE/vhdl/  
mti_se/6.6a/lin64/xilinxcorelib';  
hdlset_param (bdroot, 'XilinxSimulatorLibPath', libpath);
```

Altera DSP Builder Setup

To generate code for a design containing both Altera DSP Builder and Simulink blocks, you must open MATLAB with Altera DSP Builder. Refer to the Altera DSP Builder documentation for details.

Xilinx FPGA Floating-Point Library Setup

To map your design to a Xilinx floating-point library, you must:

- Use Xilinx LogiCORE IP Floating-Point Operator v5.0.
- Have the compiled `xilinxcorelib` simulation library for your EDA simulator and target language.

Refer to the Xilinx `comp1lib` documentation to learn how to compile this library.

- Specify the path to your compiled Xilinx simulation libraries by setting the `XilinxSimulatorLibPath` parameter for your DUT.

For example, you can use `hdlset_param` to set `XilinxSimulatorLibPath`:

```
myDUT = gcb;  
libpath = '/apps/Xilinx_ISE/XilinxISE-13.4/Linux/ISE_DS/ISE/vhdl/  
    mti_se/6.6a/lin64/xilinxcorelib';  
hdlset_param (myDUT, 'XilinxSimulatorLibPath', libpath);
```

Supported Third-Party Tools and Hardware

In this section...

“Third-Party Synthesis Tools” on page 1-9

“FPGA-in-the-Loop Hardware” on page 1-9

“Simulink® Real-Time™ FPGA I/O Hardware” on page 1-10

“FPGA Turnkey Hardware” on page 1-10

Third-Party Synthesis Tools

The HDL Workflow Advisor is tested with the following third-party FPGA synthesis tools:

- Xilinx ISE 14.6
- Altera Quartus II 13.0 SP1
- Xilinx ISE 10.1 is supported only for compatibility with Speedgoat FPGA target devices.

Speedgoat IO301, IO303, and IO311 FPGA IO boards, which use Xilinx Virtex-II FPGAs, are tested with Xilinx ISE version 10.1. Before you select one of these Speedgoat devices in the HDL Workflow Advisor, make sure that you have installed Xilinx ISE 10.1. See “Generate Simulink Real-Time™ Interface for Speedgoat Boards” for more information.

For FPGA-in-the-Loop or Customization for USRP® Device using the HDL Workflow Advisor, a supported synthesis tool must be installed, and the synthesis tool executable must be on the system path.

FPGA-in-the-Loop Hardware

The FPGAs supported for FPGA-in-the-loop simulation with HDL Verifier are listed in the HDL Verifier documentation.

You can also add custom FPGA boards using the FPGA Board Manager. See “FPGA Board Customization” for details.

Simulink Real-Time FPGA I/O Hardware

The FPGA I/O boards supported for use with the Simulink Real-Time FPGA IO workflow are listed in the Simulink Real-Time documentation.

FPGA Turnkey Hardware

The following hardware is supported for the FPGA Turnkey workflow:

- Altera Arria II GX FPGA development kit
- Altera Cyclone III FPGA development kit
- Altera Cyclone IV GX FPGA development kit
- Altera DE2–115 development and education board
- XUP Atlys Spartan-6 development board
- Xilinx Spartan-3A DSP 1800A development board
- Xilinx Spartan-6 SP605 development board
- Xilinx Virtex-4 ML401 development board
- Xilinx Virtex-4 ML402 development board
- Xilinx Virtex-5 ML506 development board
- Xilinx Virtex-6 ML605 development board

For FPGA development boards that have more than one FPGA device, only one such device can be used with FPGA Turnkey.

Supported FPGA Device Families for Board Customization

You can also add custom FPGA boards using the FPGA Board Manager. HDL Coder supports the following FPGA device families for board customization; that is, when you create your own board definition file. See “FPGA Board Customization”.

Device Family	
Xilinx	Kintex7
	Spartan-3A DSP
	Spartan3
	Spartan3A and Spartan3AN
	Spartan3E
	Spartan6
	Virtex4
	Virtex5
	Virtex6
	Virtex7
Altera	Cyclone III
	Cyclone IV
	Arria II
	Stratix IV
	Stratix V

VHDL and Verilog Language Support

The generated HDL code complies with the following standards:

- VHDL-1993 (IEEE® 1076-1993) or later
- Verilog-2001 (IEEE 1364-2001) or later

Tutorials

- “HDL Code Generation from a MATLAB Algorithm” on page 2-2
- “HDL Code Generation from a Simulink Model” on page 2-16

HDL Code Generation from a MATLAB Algorithm

In this section...

“About the Algorithm in This Example” on page 2-2

“Copying Files Locally” on page 2-4

“Checking Your Synthesis Tool Setup” on page 2-4

“Testing the Original MATLAB Algorithm” on page 2-5

“Setting Up an HDL Coder Project” on page 2-6

“Creating Fixed-Point Versions of the Algorithm and Test Bench” on page 2-8

“Generating HDL Code” on page 2-12

About the Algorithm in This Example

For the purpose of this example, you generate and synthesize HDL code for a MATLAB algorithm that implements a simple filter. However, you can use HDL Coder to generate HDL code from MATLAB algorithms for many applications.

This tutorial uses these files:

- `mlhdlc_sfir.m` — Simple filter function from which you generate HDL code.
- `mlhdlc_sfir_tb.m` — Test bench that the HDL Coder project uses to exercise the filter using a representative input range.

`mlhdlc_sfir` Function Code

The following code provides the complete `mlhdlc_sfir` function definition.

```
%#codegen
function [y_out, delayed_xout] = mlhdlc_sfir(x_in, h_in1, h_in2, h_in3, h_in4)
% Symmetric FIR Filter

persistent ud1 ud2 ud3 ud4 ud5 ud6 ud7 ud8;
if isempty(ud1)
```

```

        ud1 = 0; ud2 = 0; ud3 = 0; ud4 = 0; ud5 = 0; ud6 = 0; ud7 = 0; ud8 = 0;
    end

    a1 = ud1 + ud8; a2 = ud2 + ud7;
    a3 = ud3 + ud6; a4 = ud4 + ud5;

    m1 = h_in1 * a1; m2 = h_in2 * a2;
    m3 = h_in3 * a3; m4 = h_in4 * a4;

    a5 = m1 + m2; a6 = m3 + m4;

    % filtered output
    y_out = a5 + a6;
    % delayout input signal
    delayed_xout = ud8;

    % update the delay line
    ud8 = ud7;
    ud7 = ud6;
    ud6 = ud5;
    ud5 = ud4;
    ud4 = ud3;
    ud3 = ud2;
    ud2 = ud1;
    ud1 = x_in;

end

```

mlhdlc_sfir_tb.m Test Bench

The `mlhdlc_sfir_tb` test bench creates an input signal and calls the `mlhdlc_sfir` filter, passing in the input data.

```

clear all;

% input signal with noise
x_in = cos(2.*pi.*(0:0.001:2).*(1+(0:0.001:2).*75)).';

% filter coefficients
h1 = -0.1339; h2 = -0.0838; h3 = 0.2026; h4 = 0.4064;

```

```
len = length(x_in);
y_out = zeros(1,len);
x_out = zeros(1,len);

for ii=1:len
    data = x_in(ii);
    % call to the design 'mlhdlc_sfir' that is targeted for hardware
    [y_out(ii), x_out(ii)] = mlhdlc_sfir(data, h1, h2, h3, h4);
end

figure('Name', [mfilename, '_plot']);
subplot(2,1,1); plot(1:len,x_in);
subplot(2,1,2); plot(1:len,y_out);
```

Copying Files Locally

Before you begin generating code, set up a working folder and copy the tutorial files to this folder.

- 1 Start MATLAB.
- 2 Create a folder named `filter_sfir`, for example:

```
mkdir filter_sfir
```

The folder must not be within the MATLAB directory structure. You must be able to write to this folder.

- 3 Copy the tutorial files, `mlhdlc_sfir.m` and `mlhdlc_sfir_tb.m`, to this folder.

Checking Your Synthesis Tool Setup

Before using HDL Coder to synthesize HDL code, you must set up your synthesis tool path.

To check your Xilinx ISE synthesis tool setup, try launching the tool with the following command:

```
!ise
```

To check your Altera Quartus synthesis tool setup, try launching the tool with the following command:

```
!quartus
```

If the tool does not open, or opens the wrong version, see “Synthesis Tool Path Setup” on page 1-4.

Testing the Original MATLAB Algorithm

Before generating HDL code for this MATLAB algorithm, simulate your MATLAB design to verify that it runs, and to provide a baseline for comparison with the generated HDL code.

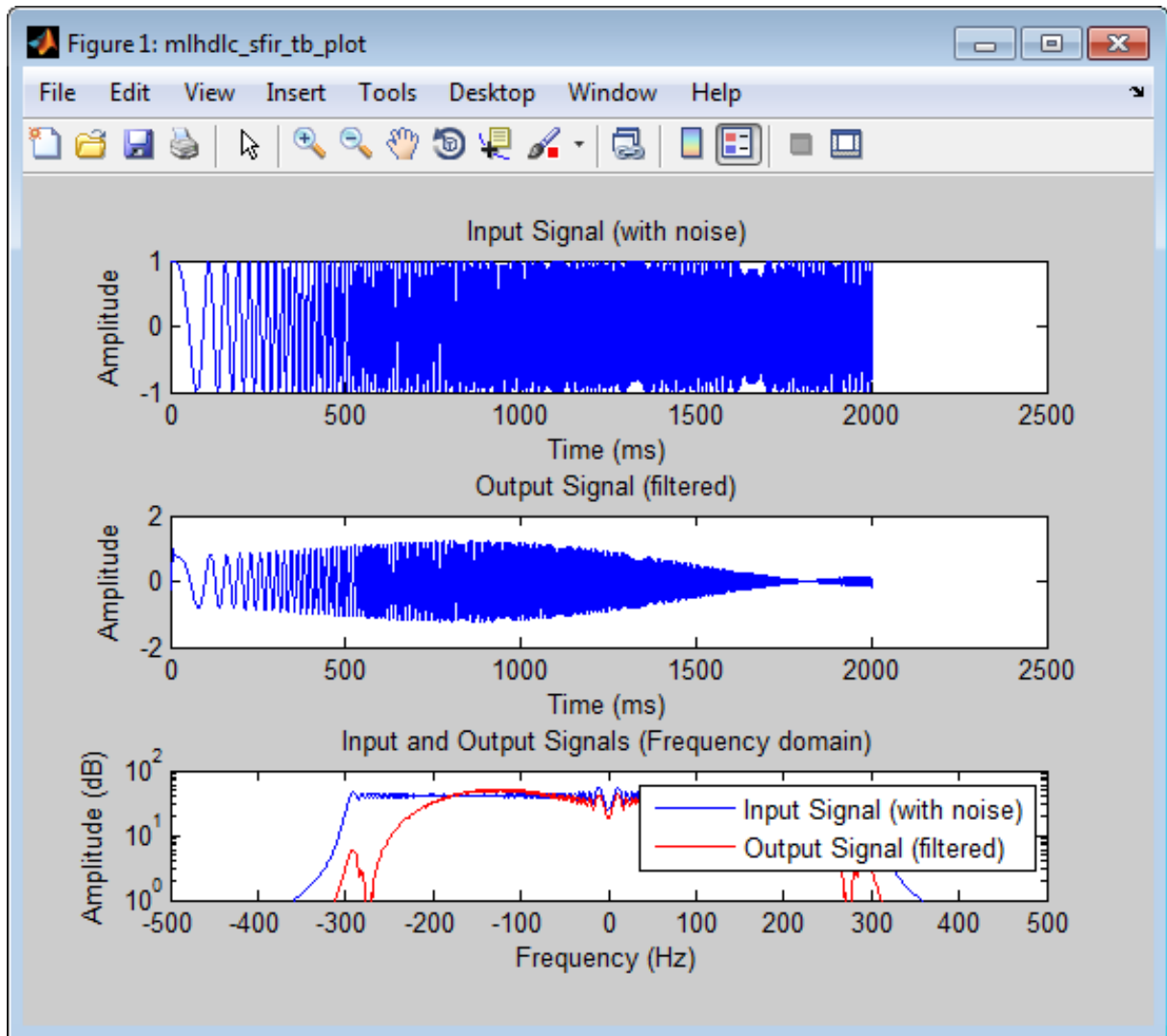
- 1 Make the `filter_sfir` folder your working folder, for example:

```
cd filter_sfir
```


- 2 Run the test bench. At the MATLAB command line, enter:

```
mlhdlc_sfir_tb
```

The test bench runs and plots the input signal and the filtered output.

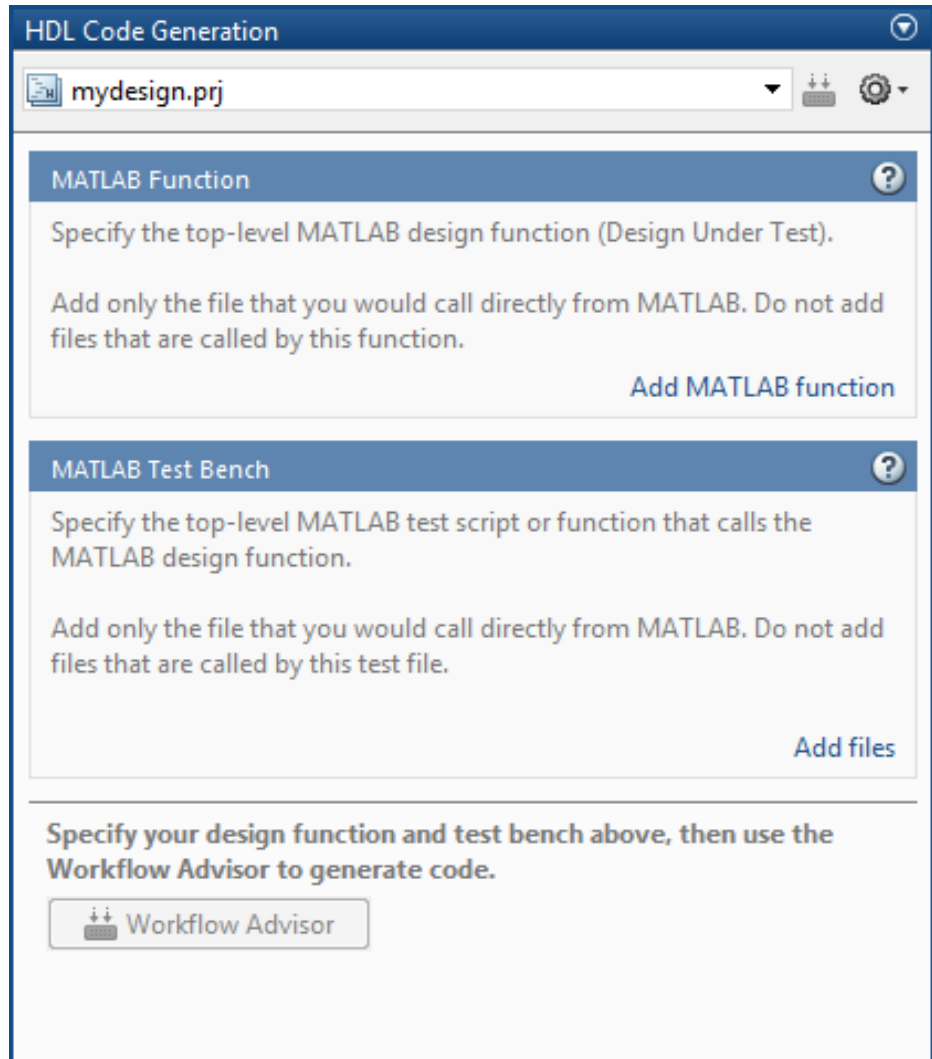


Setting Up an HDL Coder Project

- 1 On the **Apps** tab, on the far right of the **Apps** section, click the arrow .
- 2 Under **Code Generation**, click **HDL Coder**.

- 3 Enter `mydesign` for the project name.

HDL Coder creates the project, `mydesign.prj`, in the local working folder, and opens the project in the right side of the MATLAB workspace.



- 4 Under **MATLAB Function**, click **Add MATLAB function**.

5 In the **Add Files** dialog box, select `mlhdlc_sfir.m` and click **Open**.

HDL Coder adds the file to the project.

6 Under **MATLAB Test Bench**, click **Add MATLAB test bench**.

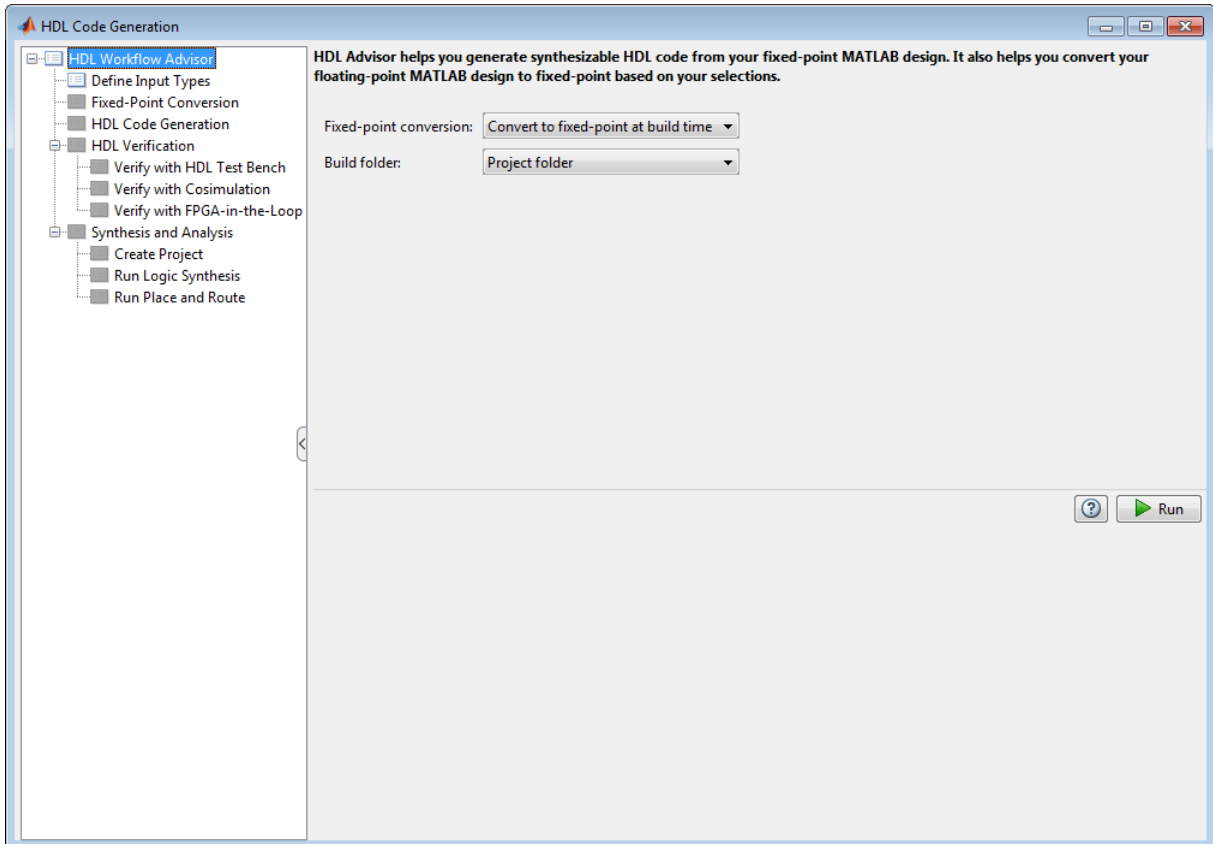
7 In the **Add Files** dialog box, select `mlhdlc_sfir_tb.m` and click **Open**.

HDL Coder adds the test bench file to the project.

You are now ready to convert the code from floating-point to fixed-point.

Creating Fixed-Point Versions of the Algorithm and Test Bench

1 In the project, at the bottom of the pane, click the **Workflow Advisor** button to open the HDL Coder Workflow Advisor.



- 2 Select the **Define Input Types** task and click **Run**.

The coder simulates the algorithm and test bench, and automatically defines input types.

- 3 On the left, select the **Fixed-Point Conversion** task. The Fixed-Point Conversion tool opens in the right pane.

The screenshot shows the HDL Code Generation tool interface. The window title is "HDL Code Generation". On the left is a tree view of the HDL Workflow Advisor with "Fixed-Point Conversion" selected. The main area shows a MATLAB code editor with a comment: "To compute proposed fixed-point types for variables, use Run Simulation, Compute Derived Ranges, or both." Below the code is a table with columns: Variable, Type, Sim Min, Sim Max, Static Min, Static Max, Whole Nu..., and Proposed Type. The table lists variables under three categories: Input (h_in1, h_in2, h_in3, h_in4, x_in), Output (delayed_xout, y_out), and Persistent (ud1, ud2). All variables are of type "double" and have "No" in the "Whole Nu..." column.

Variable	Type	Sim Min	Sim Max	Static Min	Static Max	Whole Nu...	Proposed Type
Input							
h_in1	double					No	
h_in2	double					No	
h_in3	double					No	
h_in4	double					No	
x_in	double					No	
Output							
delayed_xout	double					No	
y_out	double					No	
Persistent							
ud1	double					No	
ud2	double					No	

- At the top left, click **Run Simulation**.

After the simulation, each input, output, and persistent variable has a **Sim Min**, **Sim Max**, and **Proposed Type** in the table.

When proposing fraction lengths for floating-point data types, the coder uses the **Default word length**. In this tutorial, the **Default**

word length is 14. The advisor provides a default **Safety Margin for Simulation Min/Max** of 4%. The advisor adjusts the range of the data by this safety factor. For example, a value of 4 specifies that you want a range of at least 4 percent larger.

In this example, we use only the simulation ranges to infer fixed-point types. **Compute Derived Ranges** gives you the option of using static range analysis. To learn more about options in the fixed-point conversion workflow, see “Automated Fixed-Point Conversion”.

- b** At the top, in the Verification section, click **Validate Types**.

The coder validates the build with the proposed fixed-point types and generates a fixed-point design.

- c** At the top, in the Verification section, click the down-arrow for **Test Numerics** and select **Log inputs and outputs for comparison plots**. Click the top part of the **Test Numerics** button.

The coder simulates the fixed-point design with the original test bench compares the output to the original floating-point design output.

- d** Click the down-arrow to the right of the **Verification Output** tab and select **Type Validation Output**. Click **mlhdlc_sfir_fixpt** to see the fixed-point MATLAB code for the **mlhdlc_sfir** function.

```

%#codegen
function [y_out,delayed_xout] = mlhdlc_sfir_fixpt(x_in,h_in1,h_in2,h_in3,h_in4)

fm = fimath('RoundingMethod', 'Floor', 'OverflowAction', 'Wrap', 'ProductMode',
'FullPrecision', 'MaxProductWordLength', 128, 'SumMode', 'FullPrecision',
'MaxSumWordLength', 128);
% Symmetric FIR Filter
% declare and initialize the delay registers
persistent ud1 ud2 ud3 ud4 ud5 ud6 ud7 ud8
if isempty( ud1 )
    ud1 = fi(0, 1, 14, 12, fm);
    ud2 = fi(0, 1, 14, 12, fm);
    ud3 = fi(0, 1, 14, 12, fm);
    ud4 = fi(0, 1, 14, 12, fm);
    ud5 = fi(0, 1, 14, 12, fm);
    ud6 = fi(0, 1, 14, 12, fm);

```

```

        ud7 = fi(0, 1, 14, 12, fm);
        ud8 = fi(0, 1, 14, 12, fm);
    end
    % access the previous value of states/registers
    a1 = fi(ud1 + ud8, 1, 14, 11, fm);
    a2 = fi(ud2 + ud7, 1, 14, 11, fm);
    a3 = fi(ud3 + ud6, 1, 14, 11, fm);
    a4 = fi(ud4 + ud5, 1, 14, 11, fm);
    % multiplier chain
    m1 = fi(h_in1*a1, 1, 14, 14, fm);
    m2 = fi(h_in2*a2, 1, 14, 15, fm);
    m3 = fi(h_in3*a3, 1, 14, 14, fm);
    m4 = fi(h_in4*a4, 1, 14, 13, fm);
    % adder chain
    a5 = fi(m1 + m2, 1, 14, 14, fm);
    a6 = fi(m3 + m4, 1, 14, 12, fm);
    % filtered output
    y_out = fi(a5 + a6, 1, 14, 12, fm);
    % delayout input signal
    delayed_xout = fi(ud8, 1, 14, 12, fm);
    % update the delay line
    ud8 = fi(ud7, 1, 14, 12, fm);
    ud7 = fi(ud6, 1, 14, 12, fm);
    ud6 = fi(ud5, 1, 14, 12, fm);
    ud5 = fi(ud4, 1, 14, 12, fm);
    ud4 = fi(ud3, 1, 14, 12, fm);
    ud3 = fi(ud2, 1, 14, 12, fm);
    ud2 = fi(ud1, 1, 14, 12, fm);
    ud1 = fi(x_in, 1, 14, 12, fm);
end

```

- e Click the **View report** link.

You can explore the fixed-point code further in the Code Generation Report.

Generating HDL Code

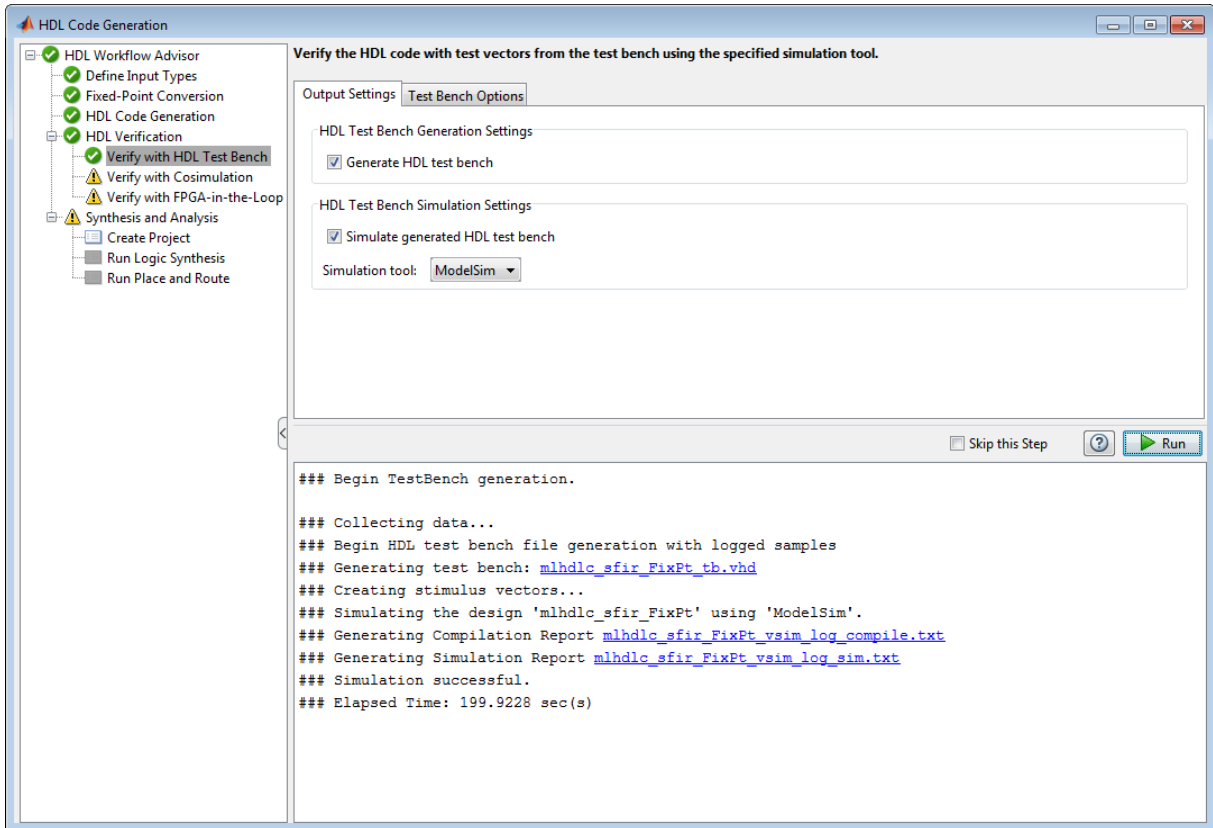
- 1 In the HDL Workflow Advisor left pane, select **HDL Code Generation** and click **Run** to generate HDL code with the default options.

The message window has a links to the generated HDL code and the resource report. Click the links to view the code and resource report.

Tip You can use the **Target**, **Coding Style**, **Clocks and Ports**, **Optimizations**, **Advanced**, and **Script Options** tabs to set code generation options. To learn about the options, click the ? button.

- 2** In the HDL Workflow Advisor left pane, select **HDL Verification > Verify with HDL Test Bench**.
- 3** Enable **Generate HDL test bench** and disable **Skip this step**. Enable **Simulate generated HDL test bench** and select a simulation tool. Click **Run**.

The task generates an HDL test bench, then simulates the fixed-point design using the selected simulation tool, and generates a compilation report and a simulation report.



4 Click **Synthesis and Analysis** and disable **Skip this step**.

- a** Select **Create Project**.
- b** On the right, select a **Synthesis tool** from the list and click **Run**.

This task creates a synthesis project for the HDL code. HDL Coder uses this project in the next task to synthesize the design.

5 Select and run **Run Logic Synthesis**.

This task:

- Launches the synthesis tool in the background.

- Opens the synthesis project created in the previous task, compiles HDL code, synthesizes the design, and emits netlists and related files.
- Generates a synthesis report.

6 Select and run **Place and Route**.

This task:

- Launches the synthesis tool in the background.
- Runs a Place and Route process that takes the circuit description produced by the previous mapping process, and emits a circuit description suitable for programming an FPGA.
- Emits pre- and post-routing timing information for use in critical path analysis and back annotation of your source model.
- Displays results.

HDL Code Generation from a Simulink Model

In this section...

“Before You Generate Code” on page 2-16

“Overview of Exercises” on page 2-17

“The `sfir_fixed` Model” on page 2-17

“Generate Code Using the HDL Workflow Advisor” on page 2-20

“Generate HDL Code Using the Command Line” on page 2-29

“Generate HDL Code Using the Configuration Parameters Dialog Box” on page 2-38

“Simulating and Verifying Generated HDL Code” on page 2-49

Before You Generate Code

Before you generate HDL code from your own models, you should do the following:

- Before generating code, use the `hdlsetup` utility (described in “Initializing Model Parameters with `hdlsetup`” on page 2-30) to set up your model for HDL code generation quickly and consistently.
- Use the `hdl1lib` utility to create a library of blocks that are currently supported for HDL code generation, as described in “Create a Supported Blocks Library”. By constructing models with blocks from this library, your models will be HDL compatible.

The set of supported blocks will change in future releases, so you should rebuild your supported blocks library each time you install a new version of this product.

- Use the **Run Compatibility Checker** option (described in “Selecting and Checking a Subsystem for HDL Compatibility” on page 2-44) to check HDL compatibility of your model or DUT and generate an HDL Code Generation Check Report.

Alternatively, you can invoke the `checkhdl` function (see `checkhdl`) to run the compatibility checker.

Overview of Exercises

The coder supports HDL code generation in your choice of environments:

- The MATLAB Command Window supports code generation using the `makehdl`, `makehdltb`, and other functions.
- The Simulink GUI (the Configuration Parameters dialog box and/or Model Explorer) provides an integrated view of the model simulation parameters and HDL code generation parameters and functions.

The hands-on exercises in this chapter introduce you to the mechanics of generating and simulating HDL code, using the same model to generate code in both environments. In a series of steps, you will

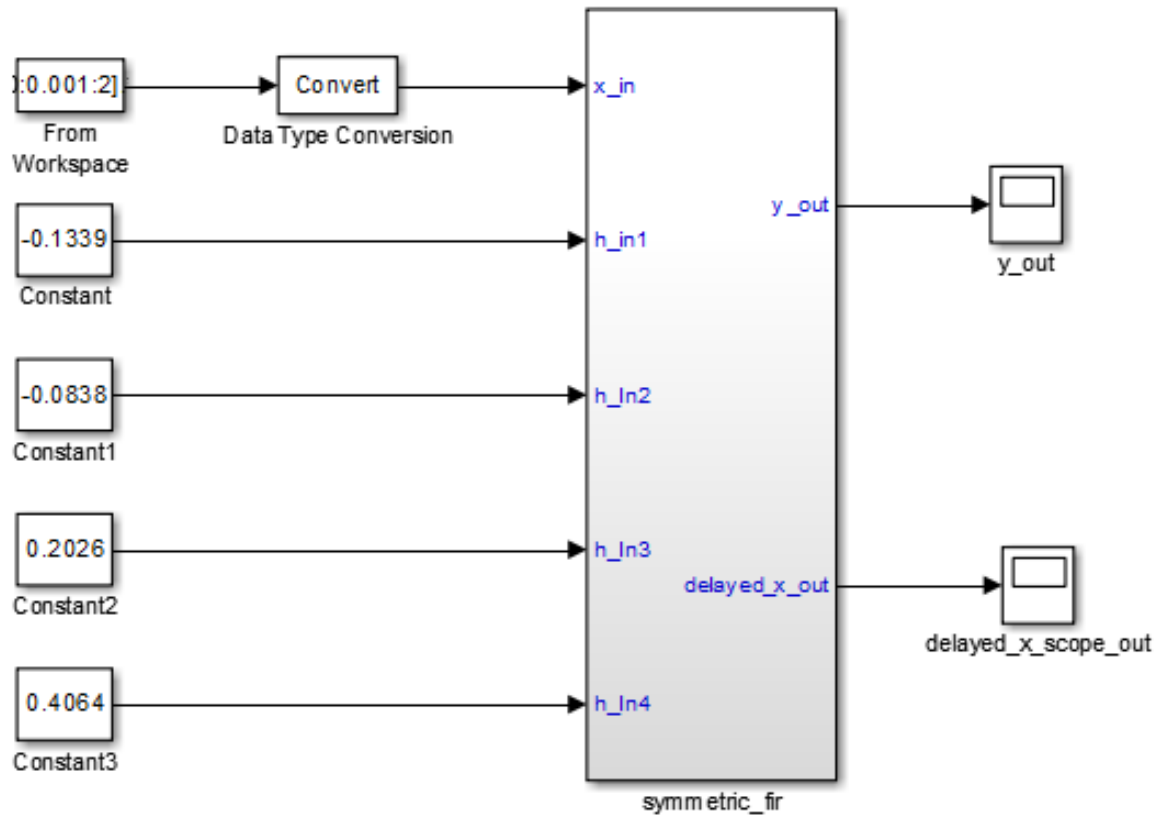
- Configure a simple model for code generation.
- Generate VHDL code from a subsystem of the model.
- Generate a VHDL test bench and scripts for the Mentor Graphics® ModelSim simulator to drive a simulation of the model.
- Compile and execute the model and test bench code in the simulator.
- Generate and simulate Verilog code from the same model.
- Check a model for compatibility with the coder.

The `sfir_fixed` Model

These exercises use the `sfir_fixed` model as a source for HDL code generation. The model simulates a symmetric finite impulse response (FIR) filter algorithm, implemented with fixed-point arithmetic.

The blocks in this example model support HDL code generation, and the model parameters have been configured for HDL code generation. To learn more about preparing your model for code generation, see “Prepare Simulink Model For HDL Code Generation” on page 1-3.

The following figure shows the top level of the model.



This model uses a division of labor that is helpful in HDL design:

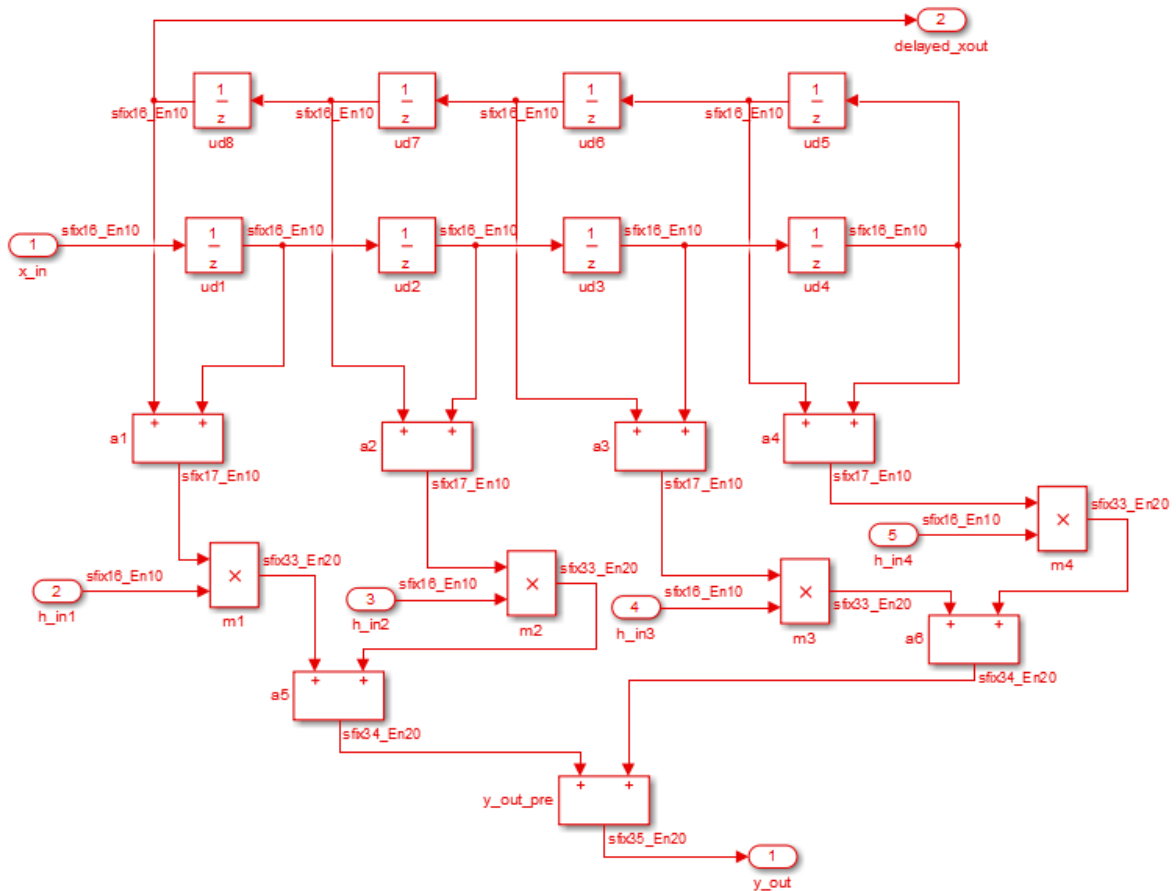
- The `symmetric_fir` subsystem, which implements the filter algorithm, is the device under test (DUT). An HDL entity will be generated, tested, and eventually synthesized from this subsystem.
- The top-level model components that drive the subsystem work as a test bench.

The top-level model generates 16-bit fixed-point input signals for the `symmetric_fir` subsystem. The Signal From Workspace block generates a

test input (stimulus) signal for the filter. The four Constant blocks provide filter coefficients.

The Scope blocks are used in simulation only. They are virtual blocks, and do not generate HDL code.

The following figure shows the `symmetric_fir` subsystem.



The fixed-point data types propagate through the subsystem. Inputs inherit the data types of the signals presented to them. Where required, internal

rules of the blocks determine the output data type, given the input data types and the operation performed (for example, the Product blocks).

The filter outputs a fixed-point result at the `y_out` port, and also replicates its input (after passing it through several delay stages) at the `delayed_x_out` port.

In the exercises that follow, you generate VHDL code that implements the `symmetric_fir` subsystem as an entity. You then generate a test bench from the top-level model. The test bench drives the generated entity, for the required number of clock steps, with stimulus data generated from the Signal From Workspace block.

Generate Code Using the HDL Workflow Advisor

This example shows how to generate HDL code from a Simulink model using the HDL Workflow Advisor.

The model you use in this example, `sfir_fixed`, is already prepared for code generation.

This example uses the Xilinx ISE synthesis tool, and assumes your synthesis tool path is set up. You can also follow this example using Altera Quartus II.

Create Working Folder and Copy Model

1 Start MATLAB.

2 Create a folder named `s1_hdlcoder_work`. For example:

```
mkdir C:\work\s1_hdlcoder_work
```

You will use `s1_hdlcoder_work` to store a local copy of the example model and to store folders and code generated by the coder. The location of the folder does not matter, except that it should not be within the MATLAB folder tree.

3 Make the `s1_hdlcoder_work` folder your working folder. For example:

```
cd C:\work\s1_hdlcoder_work
```

4 Open the `sfir_fixed` model.

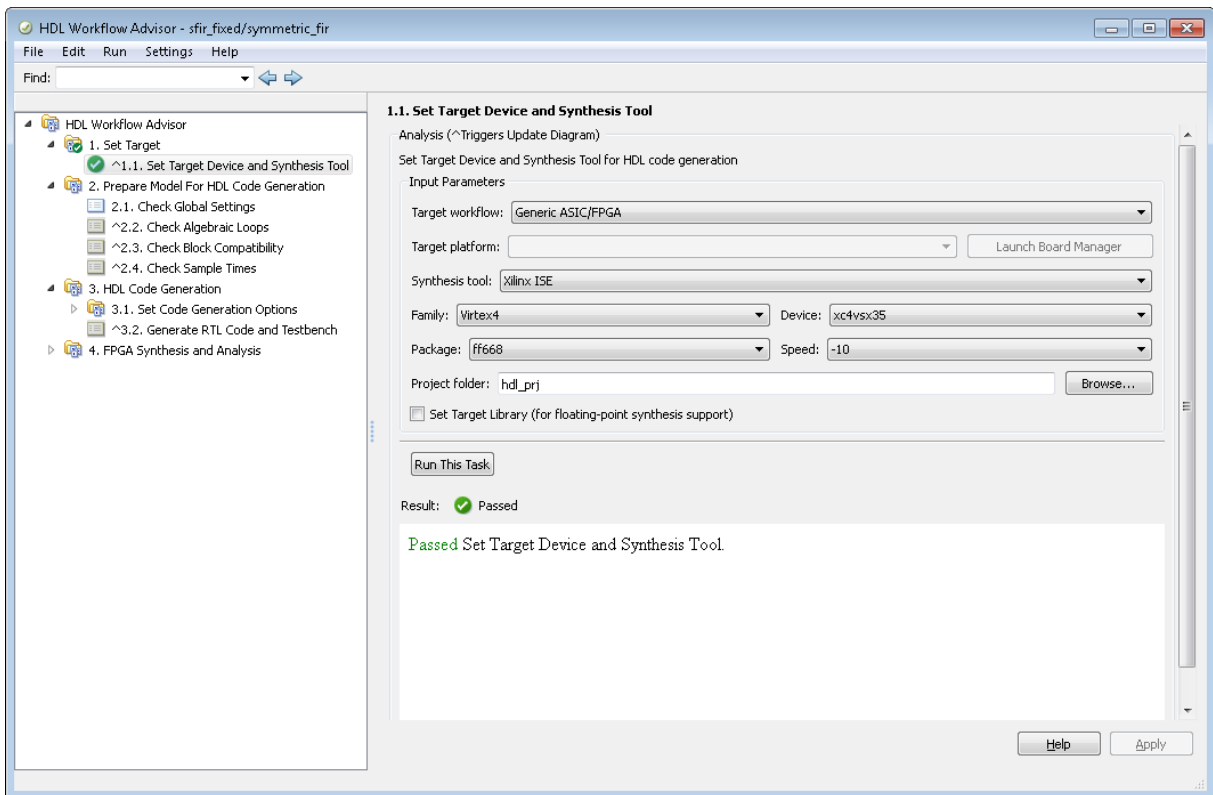
`sfir_fixed`

5 Save a copy of `sfir_fixed` in your `sl_hdlcoder_work` folder.

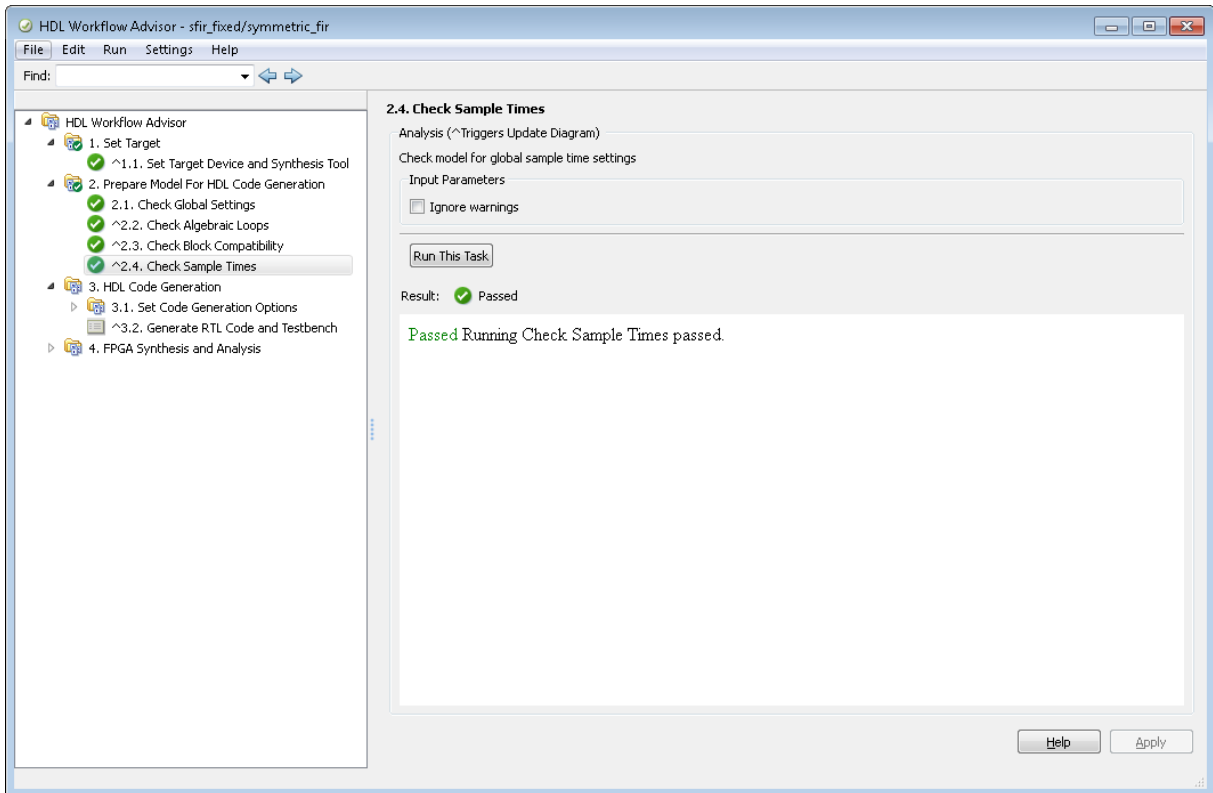
Generate Code Using the HDL Workflow Advisor

1 Right-click the `symmetric_fir` subsystem and select **HDL Code > HDL Workflow Advisor**.

2 In the **Set Target > Set Target Device and Synthesis Tool** step, for **Synthesis tool**, select **Xilinx ISE** and click **Run This Task**.

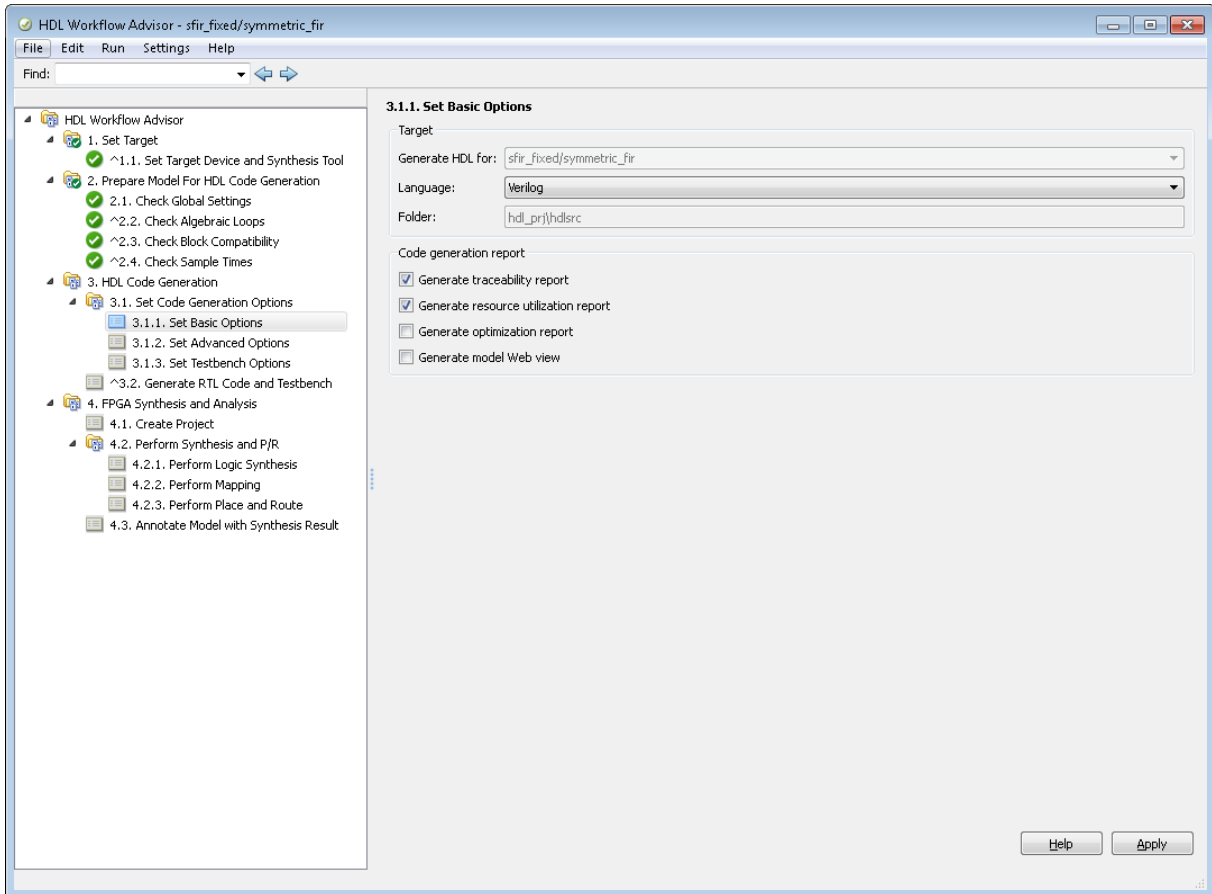


- 3 Right-click **Prepare Model For HDL Code Generation** and select **Run All**. The HDL Workflow Advisor checks the model for code generation compatibility.

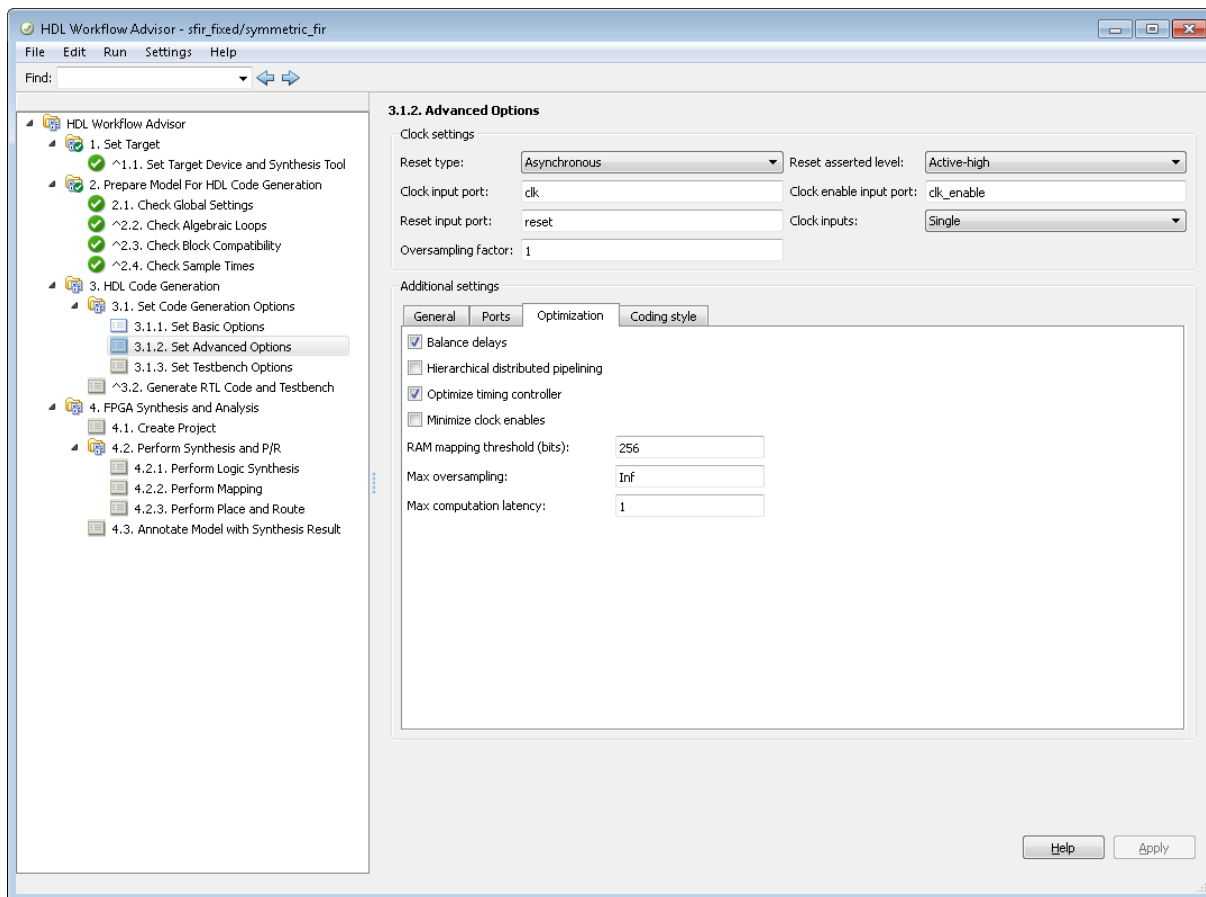


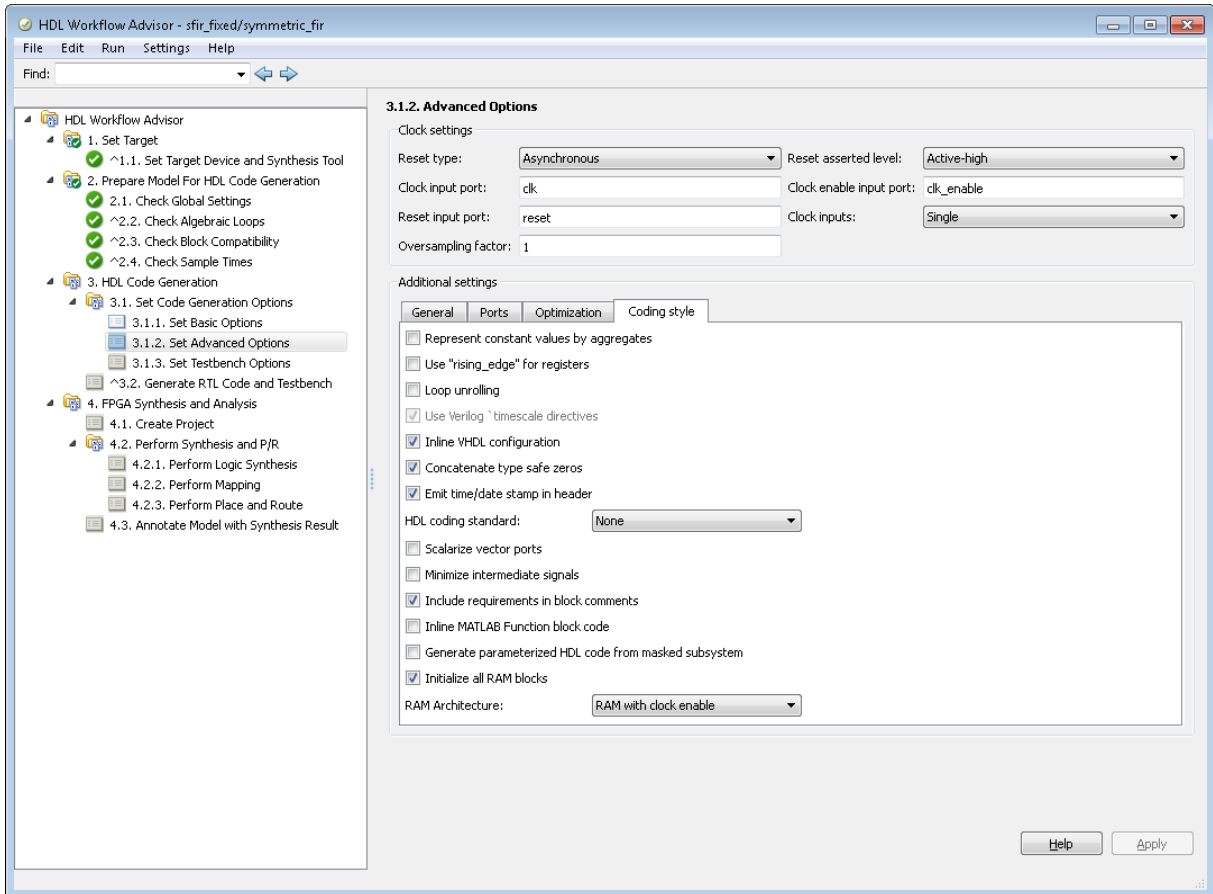
- 4 In the **HDL Code Generation > Set Code Generation Options > Set Basic Options** step, select the following options, then click **Apply**:

- For **Language**, select **Verilog**.
- Enable **Generate traceability report**.
- Enable **Generate resource utilization report**.

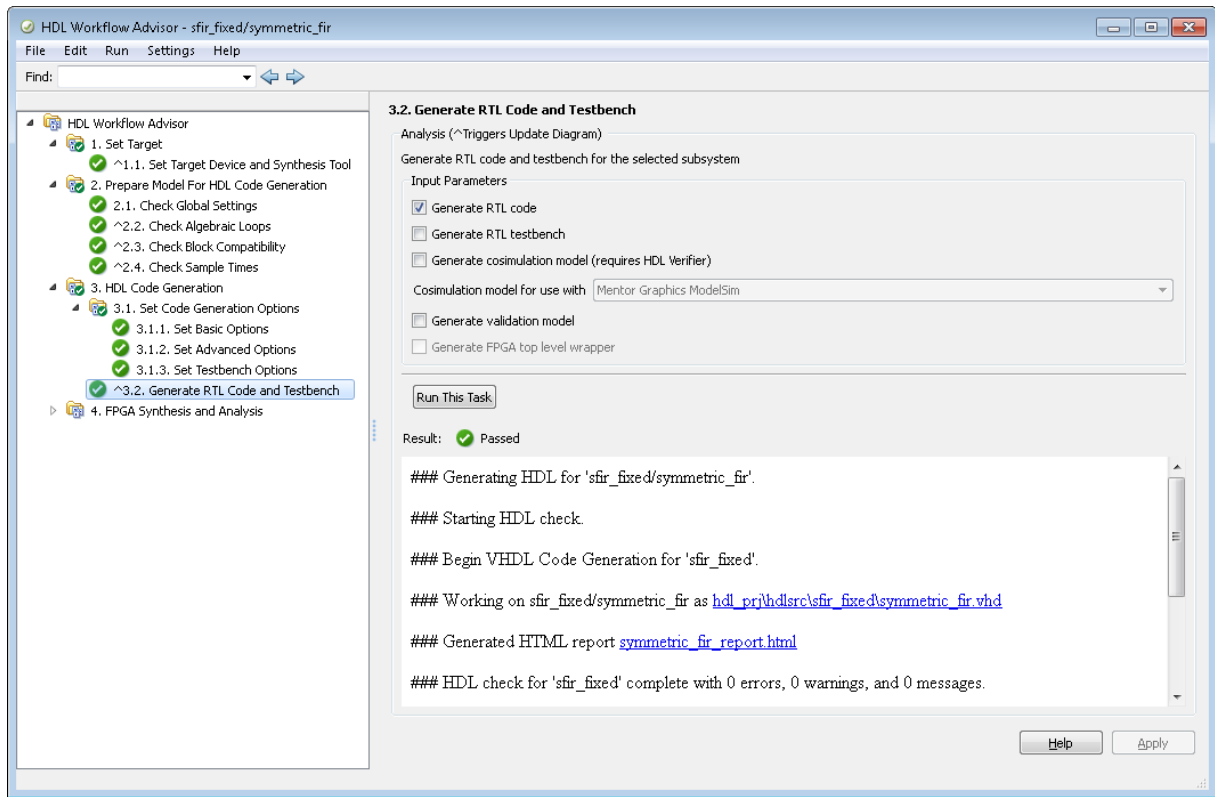


- 5** View the options available in the **Optimization** and **Coding style** tabs. You can use these options to modify the implementation and format of the generated code.

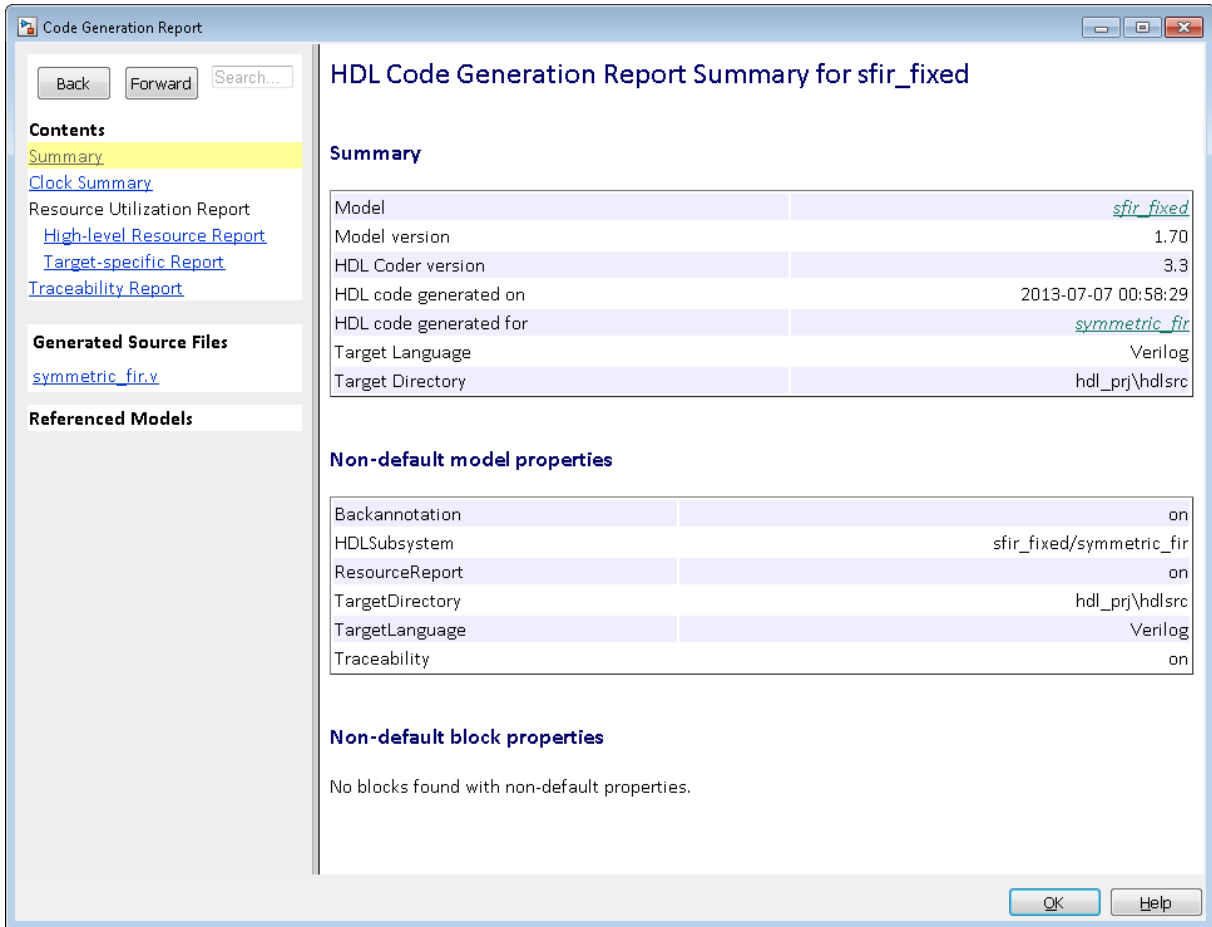




6 Right-click the **HDL Code Generation > Generate RTL Code and Testbench** step, and select **Run to Selected Task**.

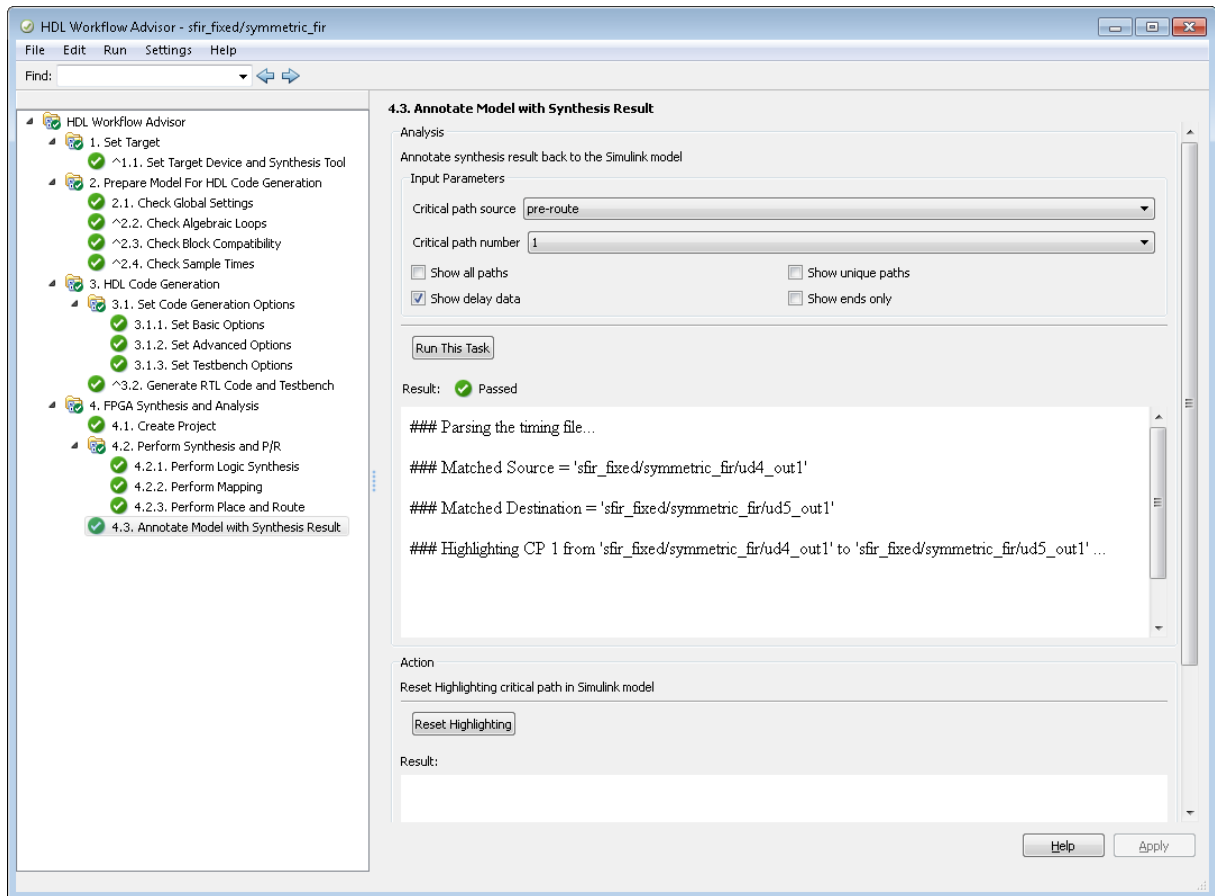


The code generation report, which includes the resource utilization and traceability reports, opens automatically. The resource utilization report shows the hardware resources your design implementation is using. The traceability report enables you to navigate between your model and the generated code.



Perform FPGA Synthesis and Analysis

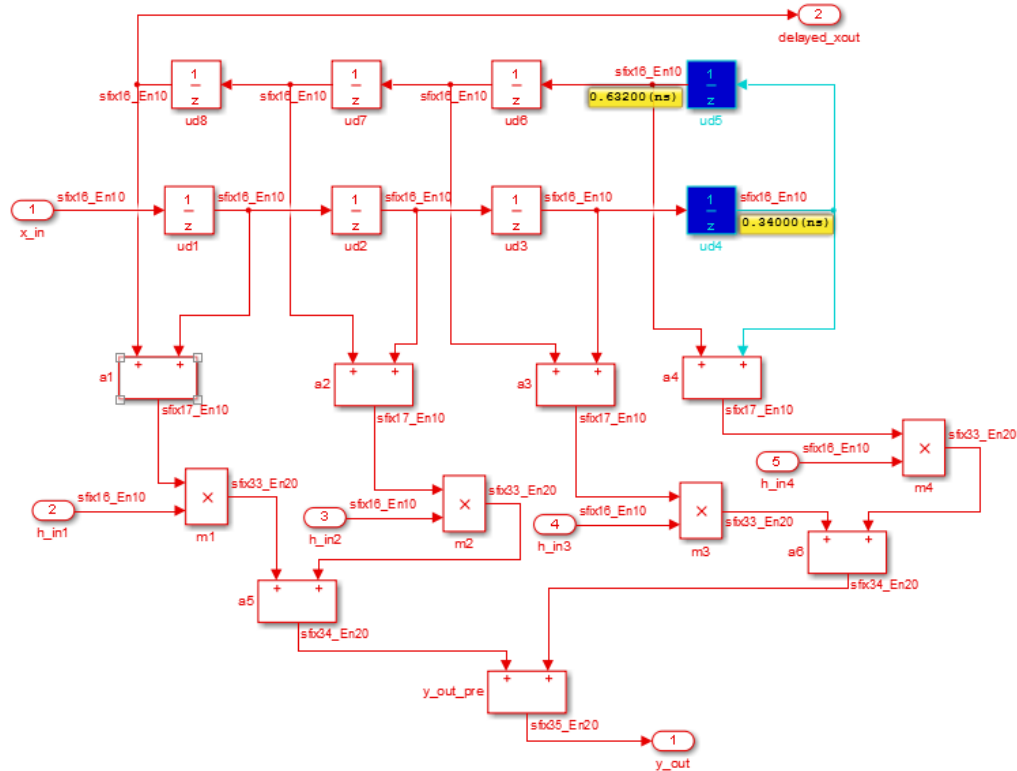
- 1 In the **FPGA Synthesis and Analysis > Perform Synthesis and P/R > Perform Place and Route** task, unselect **Skip this task** and click **Apply**.
- 2 Right-click **Annotate Model with Synthesis Result** and select **Run to Selected Task**.



3 View the annotated critical path in the model.

The critical path is colored cyan.

sfir_fixed symmetric_fir



Generate HDL Code Using the Command Line

- “Overview” on page 2-30
- “Creating a Folder and Local Model File” on page 2-30
- “Initializing Model Parameters with hdlsetup” on page 2-30
- “Generating a VHDL Entity from a Subsystem” on page 2-32
- “Generating VHDL Test Bench Code” on page 2-34
- “Verifying Generated Code” on page 2-36
- “Generating a Verilog Module and Test Bench” on page 2-36

Overview

This exercise provides a step-by-step introduction to code and test bench generation commands, their arguments, and the files created by the code generator. The exercise assumes that you have familiarized yourself with the example model (see “The `sfir_fixed` Model” on page 2-17).

Creating a Folder and Local Model File

Make a local copy of the example model and store it in a working folder, as follows.

- 1 Start the MATLAB software.
- 2 Create a folder named `sl_hdlcoder_work`, for example:

```
mkdir C:\work\sl_hdlcoder_work
```

The `sl_hdlcoder_work` folder will store a local copy of the example model and to store folders and code generated by the coder. The location of the folder does not matter, except that it should not be within the MATLAB tree.

- 3 Make the `sl_hdlcoder_work` folder your working folder, for example:

```
cd C:\work\sl_hdlcoder_work
```

- 4 To open the example model, type the following command at the MATLAB prompt:

```
sfir_fixed
```

- 5 In Simulink, select **File > Save As** and save a local copy of the `sfir_fixed` model to your working folder.
- 6 Leave the `sfir_fixed` model open and proceed to the next section.

Initializing Model Parameters with `hdlsetup`

Before generating code, you must configure the model. You can use the `hdlsetup` command instead of configuring the model manually. The `hdlsetup`

command uses the `set_param` function to set up models for HDL code generation quickly and consistently.

To configure the model for HDL code generation:

- 1 At the MATLAB command prompt, enter:

```
hdlsetup('sfir_fixed')
```

- 2 Select **Save** from the **File** menu, to save the model with its new settings.

Before continuing with code generation, consider the settings that `hdlsetup` applies to the model.

`hdlsetup` configures the **Solver** options that are recommended or required by the coder. These are

- **Type:** Fixed-step. (The coder currently supports variable-step solvers under limited conditions. See `hdlsetup`)
- **Solver:** Discrete (no continuous states). Other fixed-step solvers could be selected, but this option is usually the best one for simulating discrete systems.
- **Tasking mode:** SingleTasking. The coder does not currently support models that execute in multitasking mode.

Do not set **Tasking mode** to Auto.

`hdlsetup` also configures the model start and stop times and fixed-step size as follows:

- **Start Time:** 0.0 s
- **Stop Time:** 10 s
- **Fixed step size (fundamental periodic sample time):** auto

If **Fixed step size** is set to auto the step size is chosen automatically, based on the sample times specified in the model. In the example model, only the Signal From Workspace block specifies an explicit sample time (1 s); the other blocks inherit this sample time.

The model start and stop times determine the total simulation time. This in turn determines the size of data arrays that are generated to provide stimulus and output data for generated test benches. For the example model, computation of 10 seconds of test data does not take a significant amount of time. Computation of sample values for more complex models can be time consuming. In such cases, you may want to decrease the total simulation time.

The remaining parameters set by `hdlsetup` control error severity levels, data logging, and model display options. If you want to view the complete set of model parameters affected by `hdlsetup`, open `hdlsetup.m` in the MATLAB Editor.

The model parameter settings provided by are intended as useful defaults, but they may not be optimal for your application. For example, `hdlsetup` sets a default **Simulation stop time** of 10 s. A total simulation time of 1000 s would be more realistic for a test of the `sfir_fixed` example model. If you would like to change the simulation time, enter the desired value into the **Simulation stop time** field of the Simulink window.

See the “Model Parameters” table in the “Model and Block Parameters” section of the Simulink documentation for a summary of model parameters.

Generating a VHDL Entity from a Subsystem

In this section, you will use the `makehdl` function to generate code for a VHDL entity from the `symmetric_fir` subsystem of the example model. `makehdl` also generates script files for third-party HDL simulation and synthesis tools.

`makehdl` lets you specify numerous properties that control various features of the generated code. In this example, you will use the `makehdl` property defaults.

Before generating code, make sure that you have completed the steps described in “Creating a Folder and Local Model File” on page 2-30 and “Initializing Model Parameters with `hdlsetup`” on page 2-30.

To generate code:

1 Select **Current Folder** from the **Desktop** menu in the MATLAB window. This displays the MATLAB Current Folder browser, which lets you easily access your working folder and the files that will be generated within it.

2 At the MATLAB prompt, type the command

```
makehdl('sfir_fixed/symmetric_fir')
```

This command directs the coder to generate code from the `symmetric_fir` subsystem within the `sfir_fixed` model, using default property values.

3 As code generation proceeds, the coder displays progress messages. The process should complete with the message

```
### HDL Code Generation Complete.
```

Observe that the names of generated files in the progress messages are hyperlinked. After code generation completes, you can click these hyperlinks to view the files in the MATLAB Editor.

`makehdl` compiles the model before generating code. Depending on model display options (such as port data types, etc.), the appearance of the model may change after code generation.

4 By default, `makehdl` generates VHDL code. Code files and scripts are written to a *target folder*. The default target folder is a subfolder of your working folder, named `hdlsrc`.

A folder icon for the `hdlsrc` folder is now visible in the Current Folder browser. To view generated code and script files, double-click the `hdlsrc` folder icon.

5 The files that `makehdl` has generated in the `hdlsrc` folder are

- `symmetric_fir.vhd`: VHDL code. This file contains an entity definition and RTL architecture implementing the `symmetric_fir` filter.
- `symmetric_fir_compile.do`: Mentor Graphics ModelSim compilation script (`vcom` command) to compile the generated VHDL code.
- `symmetric_fir_synplify.tcl`: Synplify® synthesis script

- `symmetric_fir_map.txt`: Mapping file. This report file maps generated entities (or modules) to the subsystems that generated them (see “Trace Code Using the Mapping File”).
- 6** To view the generated VHDL code in the MATLAB Editor, double-click the `symmetric_fir.vhd` file icon in the Current Folder browser.
 - 7** Before proceeding to the next section, close files you have opened in the editor. Then, click the Go Up One Level button in the Current Folder browser, to set the current folder back to your `sl_hdlcoder_work` folder.
 - 8** Leave the `sfir_fixed` model open and proceed to the next section.

Generating VHDL Test Bench Code

In this section, you use the test bench generation function, `makehdltb`, to generate a VHDL test bench. The test bench is designed to drive and verify the operation of the `symmetric_fir` entity that was generated in the previous section. A generated test bench includes

- Stimulus data generated by signal sources connected to the entity under test.
- Output data generated by the entity under test. During a test bench run, this data is compared to the outputs of the VHDL model, for verification purposes.
- Clock, reset, and clock enable inputs to drive the entity under test.
- A component instantiation of the entity under test.
- Code to drive the entity under test and compare its outputs to the expected data.

In addition, `makehdltb` generates Mentor Graphics ModelSim scripts to compile and execute the test bench.

This exercise assumes that your working folder is the same as that used in the previous section. This folder now contains an `hdlsrc` folder containing the previously generated code.

To generate a test bench:

- 1 At the MATLAB prompt, type the command

```
makehdltb('sfir_fixed/symmetric_fir')
```

This command generates a test bench that is designed to interface to and validate code generated from `symmetric_fir` (or from a subsystem with a functionally identical interface). By default, VHDL test bench code, as well as scripts, are generated in the `hdlsrc` target folder.

- 2 As test bench generation proceeds, the coder displays progress messages. The process should complete with the message

```
### HDL TestBench Generation Complete.
```

- 3 To view generated test bench and script files, double-click the `hdlsrc` folder icon in the Current Folder browser. Alternatively, you can click the hyperlinked names of generated files in the code test bench generation progress messages.

The files generated by `makehdltb` are:

- `symmetric_fir_tb.vhd`: VHDL test bench code and generated test and output data.
 - `symmetric_fir_tb_compile.do`: Mentor Graphics ModelSim compilation script (`vcom` commands). This script compiles and loads both the entity to be tested (`symmetric_fir.vhd`) and the test bench code (`symmetric_fir_tb.vhd`).
 - `symmetric_fir_tb_sim.do`: Mentor Graphics ModelSim script to initialize the simulator, set up **wave** window signal displays, and run a simulation.
- 4 If you want to view the generated test bench code in the MATLAB Editor, double-click the `symmetric_fir.vhd` file icon in the Current Folder browser. You may want to study the code while referring to the `makehdltb` reference documentation, which describes the default actions of the test bench generator.
 - 5 Before proceeding to the next section, close files you have opened in the editor. Then, click the Go Up One Level button in the Current Folder browser, to set the current folder back to your `sl_hdlcoder_work` folder.

Verifying Generated Code

You can now take the previously generated code and test bench to an HDL simulator for simulated execution and verification of results. See “Simulating and Verifying Generated HDL Code” on page 2-49 for an example of how to use generated test bench and script files with the Mentor Graphics ModelSim simulator.

Generating a Verilog Module and Test Bench

The procedures for generating Verilog code differ only slightly from those for generating VHDL code. This section provides an overview of the command syntax and the generated files.

Generating a Verilog Module. By default, `makehdl` generates VHDL code. To override the default and generate Verilog code, you must pass in a property/value pair to `makehdl`, setting the `TargetLanguage` property to `'verilog'`, as in this example.

```
makehdl('sfir_fixed/symmetric_fir', 'TargetLanguage', 'verilog')
```

The previous command generates Verilog source code, as well as scripts for the simulation and the synthesis tools, in the default target folder, `hdlsrc`.

The generated files are:

- `symmetric_fir.v`: Verilog code. This file contains a Verilog module implementing the `symmetric_fir` subsystem.
- `symmetric_fir_compile.do`: Mentor Graphics ModelSim compilation script (`vlog` command) to compile the generated Verilog code.
- `symmetric_fir_synplify.tcl`: Synplify synthesis script.
- `symmetric_fir_map.txt`: Mapping file. This report file maps generated entities (or modules) to the subsystems that generated them (see “Trace Code Using the Mapping File”).

Generating and Executing a Verilog Test Bench. The `makehdltb` syntax for overriding the target language is exactly the same as that for `makehdl`. The following example generates Verilog test bench code to drive the Verilog module, `symmetric_fir`, in the default target folder.

```
makehdltb('sfir_fixed/symmetric_fir','TargetLanguage','verilog')
```

The generated files are:

- `symmetric_fir_tb.v`: Verilog test bench code and generated test and output data.
- `symmetric_fir_tb_compile.do`: Mentor Graphics ModelSim compilation script (vlog commands). This script compiles and loads both the entity to be tested (`symmetric_fir.v`) and the test bench code (`symmetric_fir_tb.v`).
- `symmetric_fir_tb_sim.do`: Mentor Graphics ModelSim script to initialize the simulator, set up **wave** window signal displays, and run a simulation.

The following listing shows the commands and responses from a test bench session using the generated scripts:

```
ModelSim> do symmetric_fir_tb_compile.do
# Model Technology ModelSim SE vlog 6.0 Compiler 2004.08 Aug 19 2004
# -- Compiling module symmetric_fir
#
# Top level modules:
# symmetric_fir
# Model Technology ModelSim SE vlog 6.0 Compiler 2004.08 Aug 19 2004
# -- Compiling module symmetric_fir_tb
#
# Top level modules:
# symmetric_fir_tb
ModelSim>do symmetric_fir_tb_sim.do
# vsim work.symmetric_fir_tb
# Loading work.symmetric_fir_tb
# Loading work.symmetric_fir
# **** Test Complete. ****
# Break at
C:/work/sl_hdlcoder_work/vlog_code/symmetric_fir_tb.v line 142
# Simulation Breakpoint:Break at
C:/work/sl_hdlcoder_work/vlog_code/symmetric_fir_tb.v line 142
# MACRO ./symmetric_fir_tb_sim.do PAUSED at line 14
```

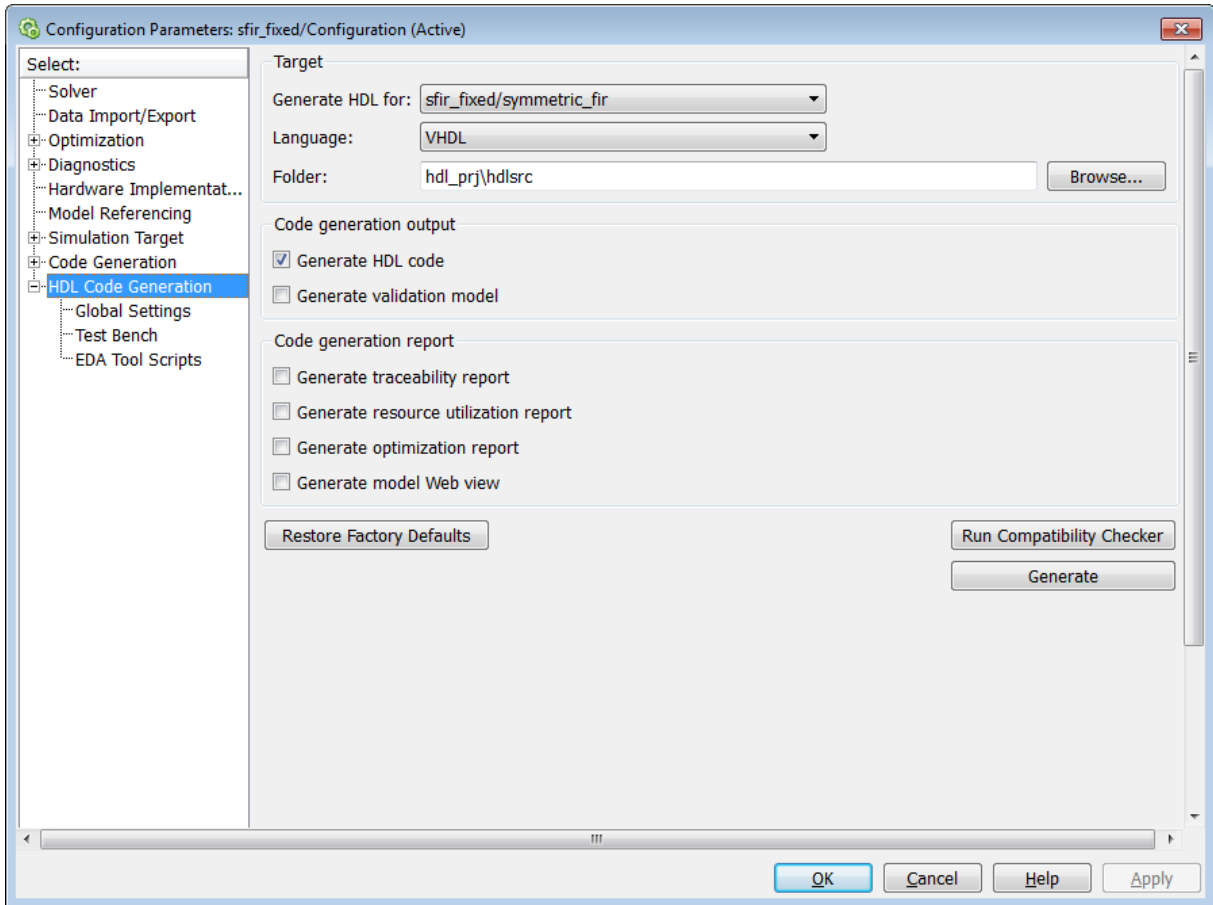
Generate HDL Code Using the Configuration Parameters Dialog Box

- “HDL Coder GUI Overview” on page 2-38
- “Creating a Folder and Local Model File” on page 2-40
- “Viewing Coder Options in the Configuration Parameters Dialog Box” on page 2-41
- “Initializing Model Parameters with hdlsetup” on page 2-42
- “Selecting and Checking a Subsystem for HDL Compatibility” on page 2-44
- “Generating VHDL Code” on page 2-45
- “Generating VHDL Test Bench Code” on page 2-47
- “Verifying Generated Code” on page 2-49
- “Generating Verilog Model and Test Bench Code” on page 2-49

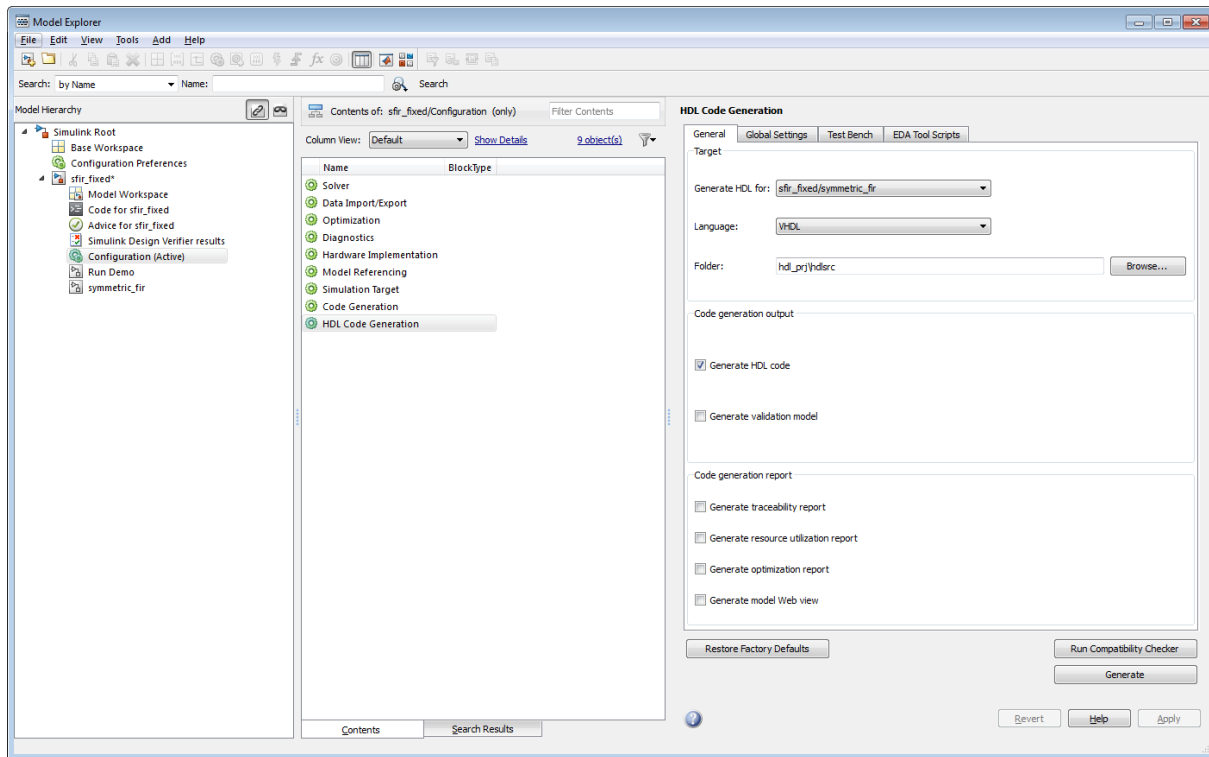
HDL Coder GUI Overview

You can view and edit options and parameters that affect HDL code generation in the Configuration Parameters dialog box, or in the Model Explorer.

The following figure shows the top-level **HDL Code Generation** pane in the Configuration Parameters dialog box.



The following figure shows the top-level **HDL Code Generation** options pane in the Model Explorer.



In the code generation exercises that follow, you use the Configuration Parameters dialog box to view and set the coder options and controls. The exercises use the `sfir_fixed` model (see “The `sfir_fixed` Model” on page 2-17) in basic code generation and verification steps.

Creating a Folder and Local Model File

In this section you will setup the folder and a local copy of the example model.

Creating a Folder. Start by setting up a working folder:

- 1 Start MATLAB.
- 2 Create a folder named `s1_hdlcoder_work`, for example:

```
mkdir C:\work\s1_hdlcoder_work
```


You will use `sl_hdlcoder_work` to store a local copy of the example model and to store folders and code generated by the coder. The location of the folder does not matter, except that it should not be within the MATLAB folder tree.

- 3 Make the `sl_hdlcoder_work` folder your working folder, for example:

```
cd C:\work\sl_hdlcoder_work
```

Making a Local Copy of the Model File. Next, make a copy of the `sfir_fixed` model:

- 1 To open the model, type the following command at the MATLAB prompt:

```
sfir_fixed
```

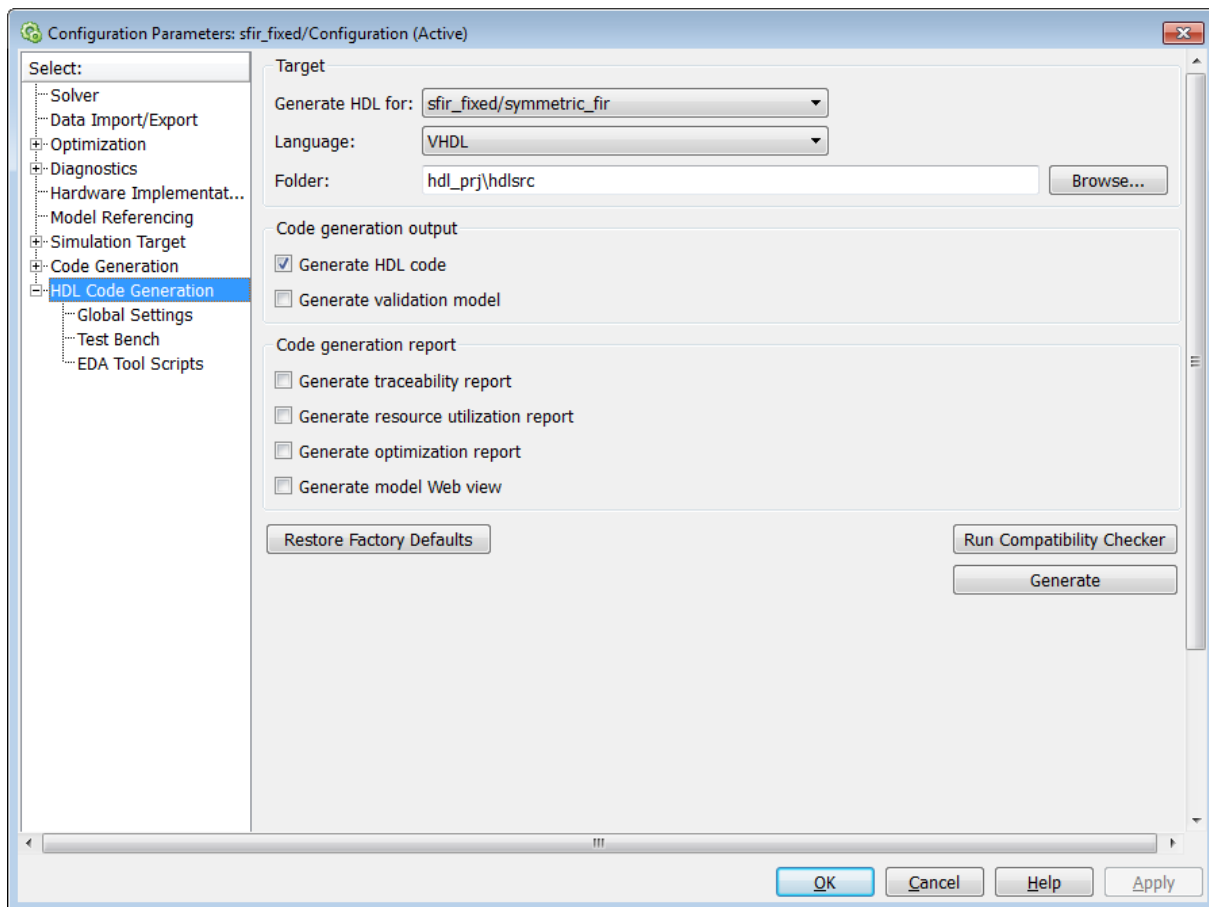
- 2 Save a local copy of the `sfir_fixed` model to your working folder.
- 3 Leave the `sfir_fixed` model open and proceed to the next section.

Viewing Coder Options in the Configuration Parameters Dialog Box

The coder option settings are displayed as a category of the model's active configuration set. You can view and edit these options in the Configuration Parameters dialog box, or in the Model Explorer. This discussion uses the Configuration Parameters dialog box.

To access the coder settings:

- 1 Open the Configuration Parameters dialog box.
- 2 Select the **HDL Code Generation** pane.



The **HDL Code Generation** pane contains top-level options and buttons that control the HDL code generation process. Several other categories of options are available under the **HDL Code** entry. This exercise uses a small subset of these options, leaving the others at their default settings.

Initializing Model Parameters with `hdlsetup`

Before generating code, you must set some parameters of the model. Rather than doing this manually, use the `hdlsetup` command. The `hdlsetup` command uses the `set_param` function to set up models for HDL code generation quickly and consistently.

To set the model parameters:

- 1 At the MATLAB command prompt, type:

```
hdlsetup('sfir_fixed')
```

- 2 Save the model with its new settings.

Before continuing with code generation, consider the settings that `hdlsetup` applies to the model.

`hdlsetup` configures **Solver** options that are recommended or required by the coder. These options are:

- **Type:** Fixed-step. (The coder currently supports variable-step solvers under limited conditions. See `hdlsetup`.)
- **Solver:** Discrete (no continuous states). Other fixed-step solvers could be selected, but this option is usually the best one for simulating discrete systems.
- **Tasking mode:** SingleTasking. The coder does not currently support models that execute in multitasking mode.

Do not set **Tasking mode** to Auto.

`hdlsetup` also configures the model start and stop times and fixed-step size as follows:

- **Start Time:** 0.0 s
- **Stop Time:** 10 s
- **Fixed step size (fundamental periodic sample time):** auto

If **Fixed step size** is set to auto the step size is chosen automatically, based on the sample times specified in the model. In the example model, only the Signal From Workspace block specifies an explicit sample time (1 s); the other blocks inherit this sample time.

The model start and stop times determine the total simulation time. This in turn determines the size of data arrays that are generated to provide stimulus and output data for generated test benches. For the example model,

computation of 10 seconds of test data does not take a significant amount of time. Computation of sample values for more complex models can be time consuming. In such cases, you may want to decrease the total simulation time.

The remaining parameters set by `hdlsetup` control error severity levels, data logging, and model display options. If you want to view the complete set of model parameters affected by `hdlsetup`, open `hdlsetup.m` in the MATLAB Editor.

The model parameter settings provided by `hdlsetup` are intended as useful defaults, but they may not be optimal for your application. For example, `hdlsetup` sets a default **Simulation stop time** of 10 s. A total simulation time of 1000 s would be more realistic for a test of the `sfir_fixed` example model. If you would like to change the simulation time, enter the desired value into the **Simulation stop time** field of the Simulink Editor.

See the “Model Parameters” table in the “Model and Block Parameters” section of the Simulink documentation for a summary of model parameters.

Selecting and Checking a Subsystem for HDL Compatibility

The coder generates code from either the current model or from a subsystem at the root level of the current model. You use the **Generate HDL for** menu to select the model or subsystem from which code is to be generated. Each entry in the menu shows the full path to the model or one of its subcomponents.

The `sfir_fixed` model is configured with the `sfir_fixed/symmetric_fir` subsystem selected for code generation. If this is not the case, make sure that the `symmetric_fir` subsystem is selected for code generation, as follows:

- 1 Select `sfir_fixed/symmetric_fir` from the **Generate HDL for** menu.
- 2 Click **Apply**.

To check HDL compatibility for the subsystem:

- 1 Click the **Run Compatibility Checker** button.
- 2 The HDL compatibility checker examines the system selected in the **Generate HDL for** menu for compatibility problems. In this case, the

selected subsystem is fully HDL-compatible, and the compatibility checker displays the following message:

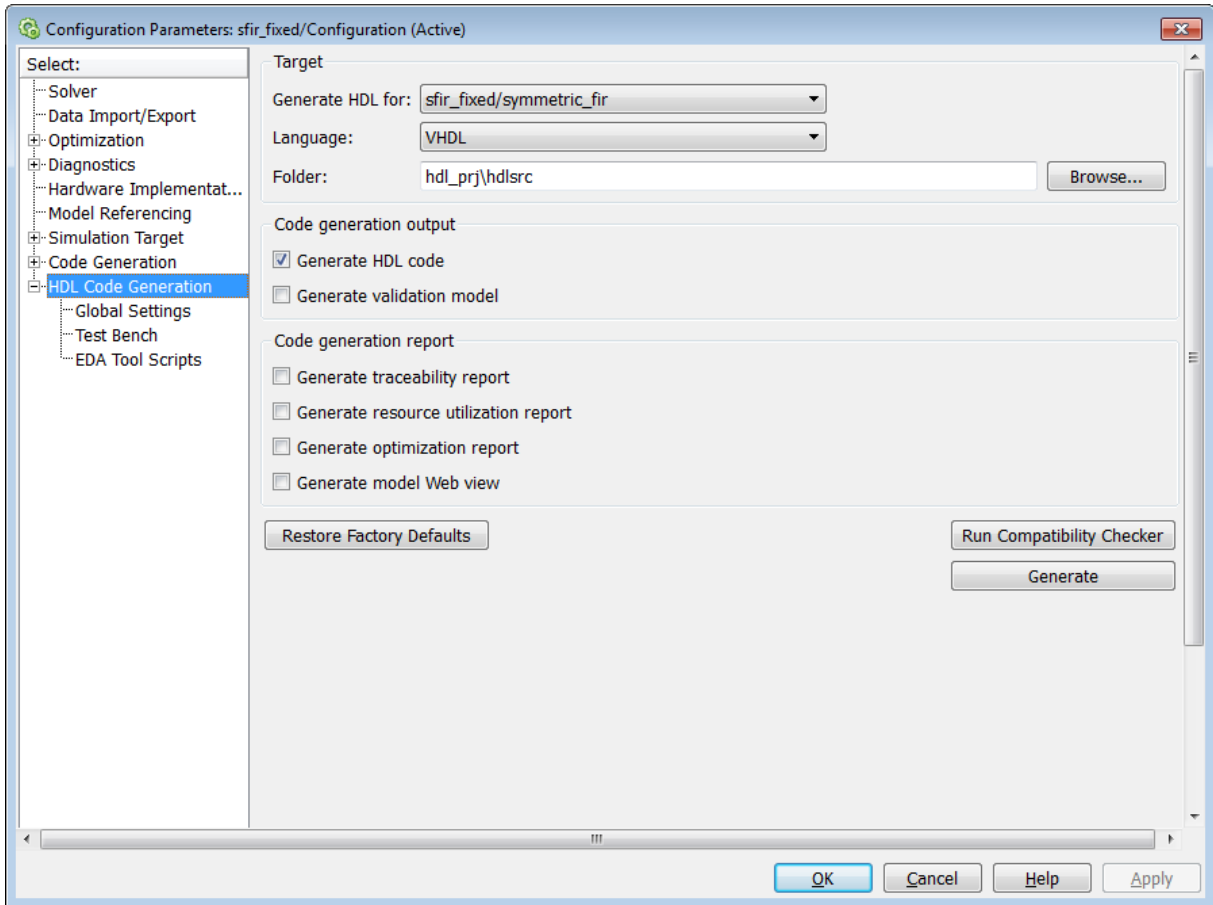
```
### Starting HDL Check.  
### HDL Check Complete with 0 errors, warnings and messages.
```

The compatibility checker also displays a report in a new window.

Generating VHDL Code

The top-level **HDL Code Generation** options are now set as follows:

- The **Generate HDL for** field specifies the `sfir_fixed/symmetric_fir` subsystem for code generation.
- The **Language** field specifies (by default) generation of VHDL code.
- The **Folder** field specifies a *target folder* that stores generated code files and scripts. The default target folder is a subfolder of your working folder, named `hdlsrc`.



Before generating code, select **Current Folder** from the **Desktop** menu in the MATLAB window. This displays the Current Folder browser, which lets you access your working folder and the files that will be generated within it.

To generate code:

- 1 Click the **Generate** button.
- 2 As code generation proceeds, the coder displays progress messages. The process should complete with the message

```
### HDL Code Generation Complete.
```

Observe that the names of generated files in the progress messages are hyperlinked. After code generation completes, you can click these hyperlinks to view the files in the MATLAB Editor.

The coder compiles the model before generating code. Depending on model display options (such as port data types, etc.), the appearance of the model may change after code generation.

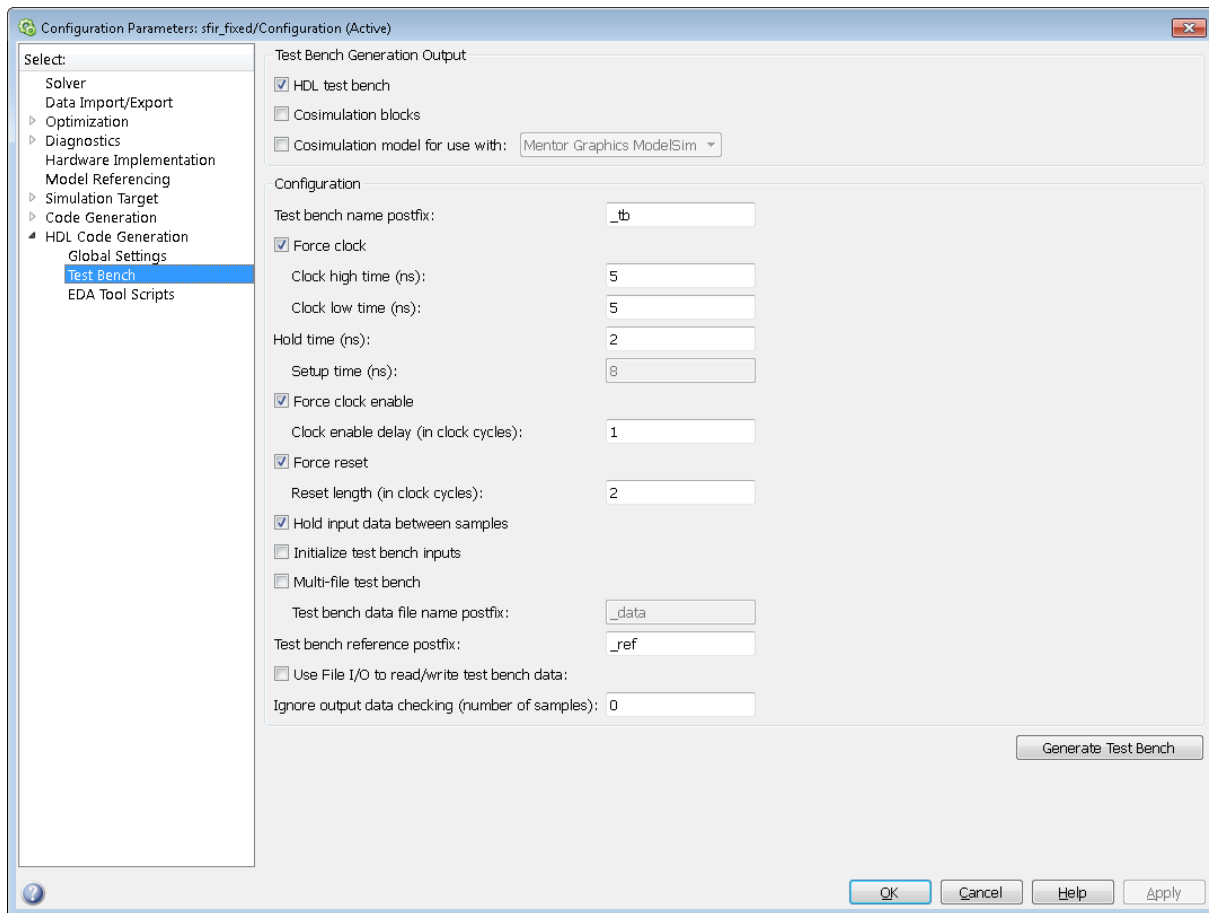
- 3** A folder icon for the `hdlsrc` folder is now visible in the Current Folder browser. To view generated code and script files, double-click the `hdlsrc` folder icon.
- 4** The files that were generated in the `hdlsrc` folder are:
 - `symmetric_fir.vhd`: VHDL code. This file contains an entity definition and RTL architecture implementing the `symmetric_fir` filter.
 - `symmetric_fir_compile.do`: Mentor Graphics ModelSim compilation script (vcom command) to compile the generated VHDL code.
 - `symmetric_fir_synplify.tcl`: Synplify synthesis script.
 - `symmetric_fir_map.txt`: Mapping file. This report file maps generated entities (or modules) to the subsystems that generated them (see “Trace Code Using the Mapping File”).
- 5** To view the generated VHDL code in the MATLAB Editor, double-click the `symmetric_fir.vhd` file icon in the Current Folder browser.
- 6** Before proceeding to the next section, close files you have opened in the editor. Then, click the Go Up One Level button in the Current Folder browser, to set the current folder back to your `sl_hdlcoder_work` folder.

Generating VHDL Test Bench Code

At this point, the **Generate HDL for**, **Language**, and **Folder** fields are set as they were in the previous section. Accordingly, you can now generate VHDL test bench code to drive the VHDL code generated previously for the `sfir_fixed/symmetric_fir` subsystem. The code will be written to the same target folder as before.

To generate a VHDL test bench:

1 Select the **HDL Code Generation > Test Bench** pane.



2 Select **HDL test bench**.

3 Click the **Generate Test Bench** button.

4 As test bench generation proceeds, the coder displays progress messages. The process should complete with the message


```
### HDL TestBench Generation Complete.
```

5 The generated files in the `hdlsrc` folder are:

- `symmetric_fir_tb.vhd`: VHDL test bench code, with generated test and output data.
- `symmetric_fir_tb_compile.do`: Mentor Graphics ModelSim compilation script (`vcom` commands). This script compiles and loads the entity to be tested (`symmetric_fir.vhd`) and the test bench code (`symmetric_fir_tb.vhd`).
- `symmetric_fir_tb_sim.do`: Mentor Graphics ModelSim script to initialize the simulator, set up **wave** window signal displays, and run a simulation.

Verifying Generated Code

You can now take the generated code and test bench to an HDL simulator for simulated execution and verification of results. See “Simulating and Verifying Generated HDL Code” on page 2-49 for an example of how to use generated test bench and script files with the Mentor Graphics ModelSim simulator.

Generating Verilog Model and Test Bench Code

The procedure for generating Verilog code is the same as for generating VHDL code (see “Generating a VHDL Entity from a Subsystem” on page 2-32 and “Generating VHDL Test Bench Code” on page 2-34), except that you select Verilog from the **Language** field of the **HDL Code Generation** options.

Simulating and Verifying Generated HDL Code

Note This section requires the use of the Mentor Graphics ModelSim simulator.

This section assumes that you have generated code from the `sfir_fixed` model as described in either of the following exercises:

- “Generate HDL Code Using the Command Line” on page 2-29

- “Generate HDL Code Using the Configuration Parameters Dialog Box” on page 2-38

In this section you compile and run a simulation of the previous generated model and test bench code. The scripts generated by the coder let you do this with just a few simple commands. The procedure is the same, whether you generated code in the command line environment or in the GUI.

To run the simulation:

- 1** Start the Mentor Graphics ModelSim software.
- 2** Set the working folder to the folder in which you previously generated code.

```
ModelSim>cd C:/work/sl_hdlcoder_work/hdlsrc
```

- 3** Use the generated compilation script to compile and load the generated model and text bench code. The following listing shows the command and responses.

```
ModelSim>do symmetric_fir_tb_compile.do
# Model Technology ModelSim SE vcom 6.0 Compiler 2004.08 Aug 19 2004
# -- Loading package standard
# -- Loading package std_logic_1164
# -- Loading package numeric_std
# -- Compiling entity symmetric_fir
# -- Compiling architecture rtl of symmetric_fir
# Model Technology ModelSim SE vcom 6.0 Compiler 2004.08 Aug 19 2004
# -- Loading package standard
# -- Loading package std_logic_1164
# -- Loading package numeric_std
# -- Compiling package symmetric_fir_tb_pkg
# -- Compiling package body symmetric_fir_tb_pkg
# -- Loading package symmetric_fir_tb_pkg
# -- Loading package symmetric_fir_tb_pkg
# -- Compiling entity symmetric_fir_tb
# -- Compiling architecture rtl of symmetric_fir_tb
# -- Loading entity symmetric_fir
```

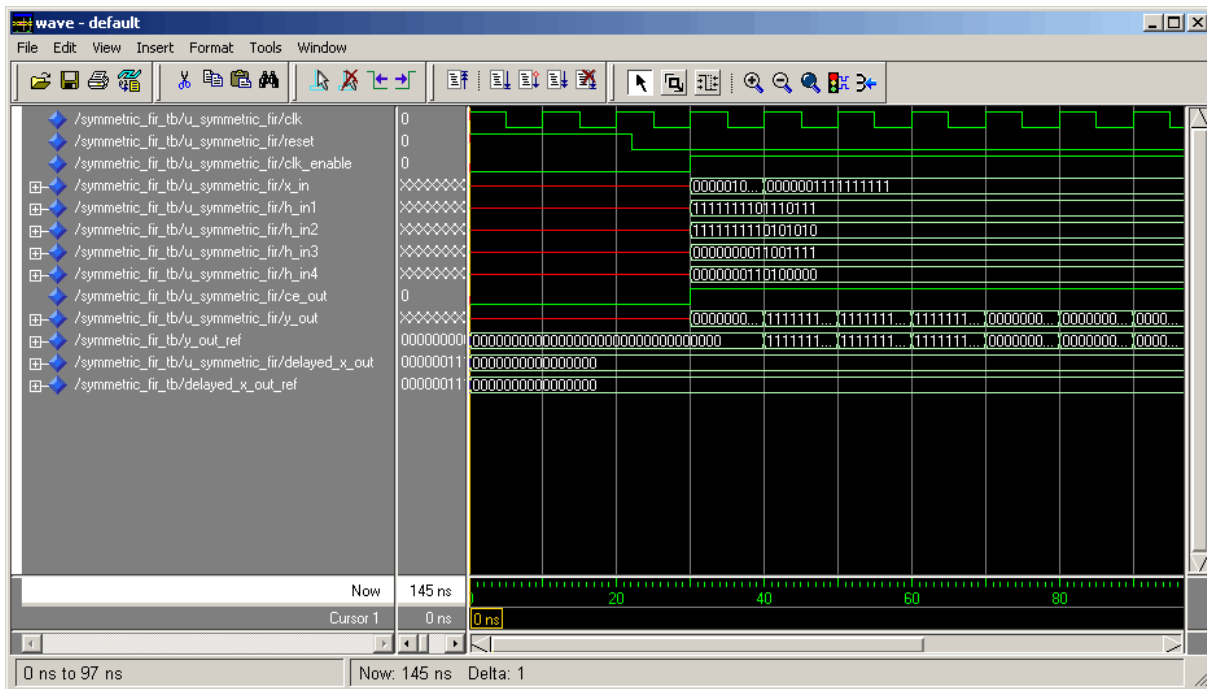
- 4** Use the generated simulation script to execute the simulation. The following listing shows the command and responses. The warning messages are benign.

```
ModelSim>do symmetric_fir_tb_sim.do
# vsim work.symmetric_fir_tb
# Loading C:\Applications\ModelTech_6_0\win32\..\std.standard
# Loading C:\Applications\ModelTech_6_0\win32\..\ieee.std_logic_1164(body)
# Loading C:\Applications\ModelTech_6_0\win32\..\ieee.numeric_std(body)
# Loading work.symmetric_fir_tb_pkg(body)
# Loading work.symmetric_fir_tb(rtl)
# Loading work.symmetric_fir(rtl)
# ** Warning: NUMERIC_STD."<": metavalue detected, returning FALSE
#   Time: 0 ns   Iteration: 0   Instance: /symmetric_fir_tb
.
.
.
# ** Warning: NUMERIC_STD.TO_INTEGER: metavalue detected, returning 0
#   Time: 0 ns   Iteration: 1   Instance: /symmetric_fir_tb
# ** Note: *****TEST COMPLETED *****
#   Time: 140 ns  Iteration: 1   Instance: /symmetric_fir_tb
```

The test bench termination message indicates that the simulation has run to completion without comparison errors.

```
# ** Note: *****TEST COMPLETED *****
```

- 5** The simulation script displays inputs and outputs in the model (including the reference signals `y_out_ref` and `delayed_x_out_ref`) in the Mentor Graphics ModelSim **wave** window. The following figure shows the signals displayed in the **wave** window.



- 6 Exit the Mentor Graphics ModelSim simulator when you finish viewing signals.
- 7 Close files you have opened in the MATLAB Editor. Then, click the **Go Up One Level** button in the Current Folder browser, to set the current folder back to your work folder.