Getting Started with System Generator for DSP

Lab 7 – Filter Design

Filter Design

Introduction

In this lab you will be using the **Filter Compiler** block to generate optimized filters for the Virtex architectures.

Objectives

After completing this lab, you will be able to:

- Use the MathWorks FDA tool to generate and set the filter coefficients
- Use the Xilinx **FIR Compiler** to generate various filter implementations

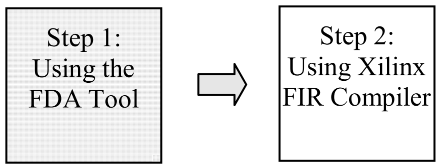
Lab Setup

Please check the System Generator for DSP release notes to insure that the proper versions of ISE Design Suite and MATLAB are installed on your machine. Failure to have the proper tool versions installed may result in unexpected behavior.

Procedure

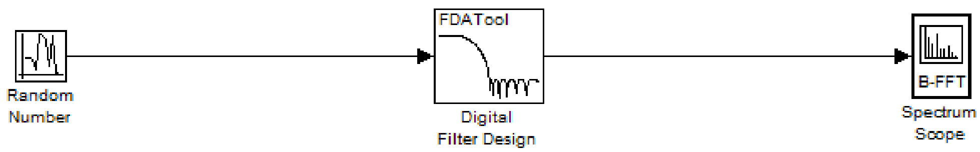
Using the FDA Tool Step 1

General Flow for this Lab:



1. Launch the MATLAB program and change the working directory to:
   …sysgen/examples/getting_started_training/lab7

2. Open the Simulink diagram **lab7.mdl** shown below.



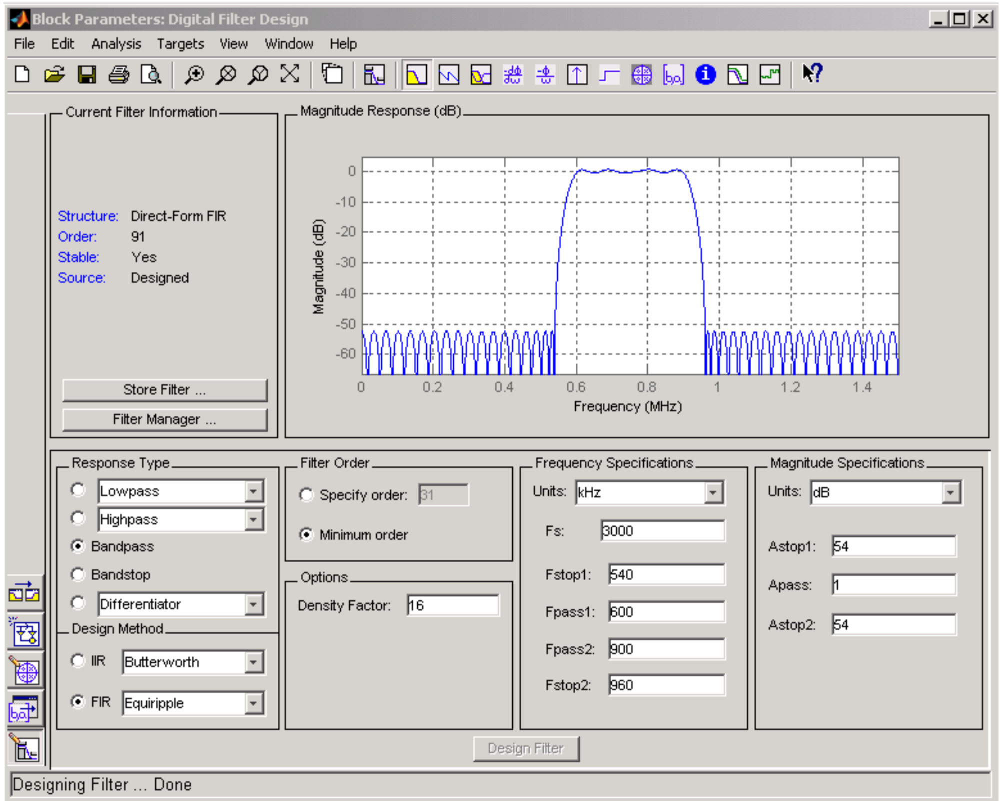
3. Click on **Design Filter** to generate the filter specified by the FDATool parameters.

4. Close FDATool and simulate the filter design in Simulink. Set the Simulation time to **inf**. You should see the **Spectrum Scope** display a waveform that is similar to what was displayed in the FDATool.

5. Double-click on the **Spectrum Scope** and view the properties dialog box. Note that both the **Buffer input** and **Specify FFT length** options are checked. This is necessary to correctly analyze the power spectral density of the streaming signal. This is something to be aware of if you are not familiar with the **Spectrum Scope** block.

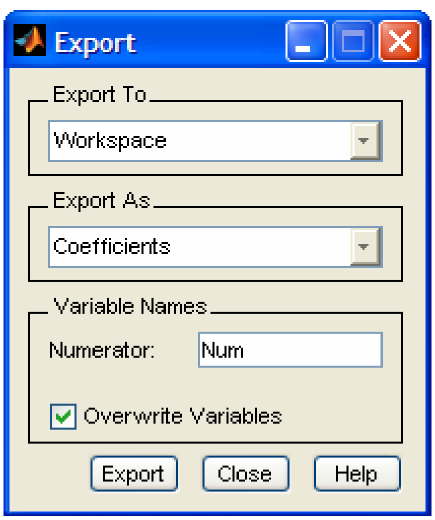
6. Open the **FDATool Properties** dialog box again and reconfigure the filter to be a **Bandpass** filter with the following parameters
   - Type = **Bandpass**
   - Specify Order = **Minimum order**
   - Units = **KHz**
   - Sampling Frequency = **3000 KHz**
   - Fstop 1 = **540 KHz**
   - Fpass1 = **600 KHz**
   - Fpass2 = **900 KHz**
   - Fstop2 = **960 KHz**
   - Attenuation on both sides of the passband = **54 dB**
   - Pass band ripple **= 1**

7. Click on the **Design Filter** button to generate the new filter. The FDATool should now display the following waveform.



The desired filter response has been achieved. The next step is to save the coefficients so they can be used in the hardware implementation

9 Using the FDATool toolbar execute the pulldown menu **File** → **Export**. Export the coefficients to the MATLAB workspace. Use the default variable name **Num**



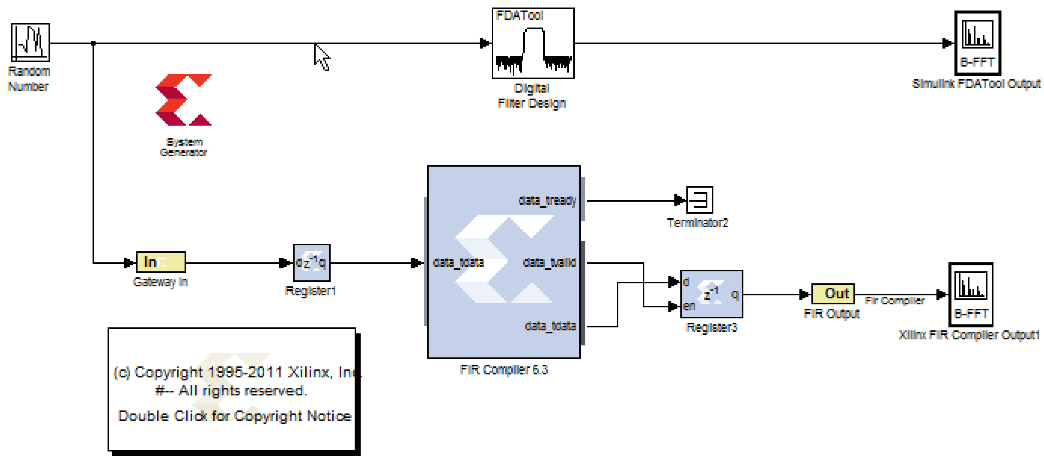
The Coefficients have been saved. In Step 2, you will create the hardware version of this filter.

Using Xilinx FIR Compiler — Step 2

General Flow for this Lab:

Step 1: Using the FDA Tool	⇨	Step 2: Using Xilinx FIR Compiler

1. Create an equivalent FIR filter representation using the FIR Compiler block from the Xilinx DSP blockset as shown below.



2. Double-click on the **FIR Compiler** block and set the parameters as listed below. Use the MATLAB workspace variable name **Num**, which was the variable name used when the coefficients were exported to the MATLAB workspace from the FDATool.

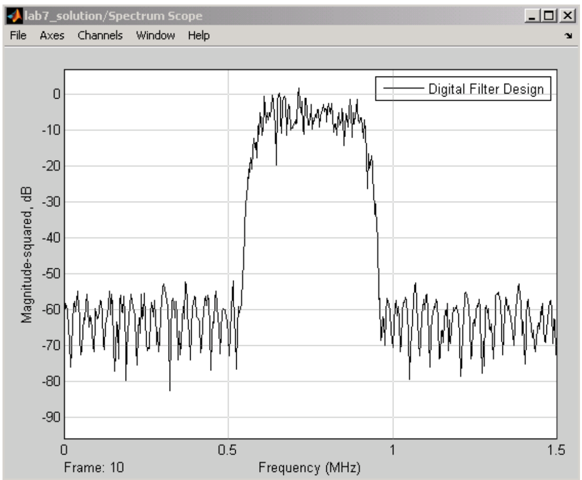
   Filter Specification Tab

   - Coefficient Vector: **Num**
   - Number of coefficient sets: **1**
   - Filter type: **Single_Rate**
   - Number of channels: **1**

   Implementation Tab

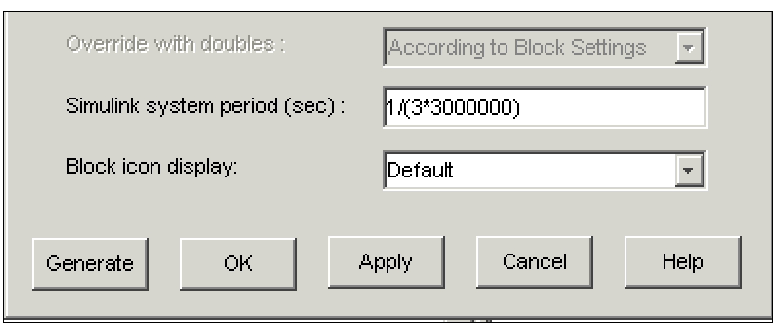
   - Coefficient Type: **Inferred**
   - Quantization: **Maximize_Dynamic_Range**
   - Coefficient Width: **16**
   - Select: **Best Precision Fraction Length**
   - Number of Paths: **1**
   - Output Rounding Mode: **Full_Precision**

3. Simulate the design and compare the results from the two **Spectrum Scopes**. You should see a waveform from the Digital Filter Design but not from the FIR Compiler block. This is caused by incorrect settings of the sample period on the **Gateway In** and **System Generator** blocks

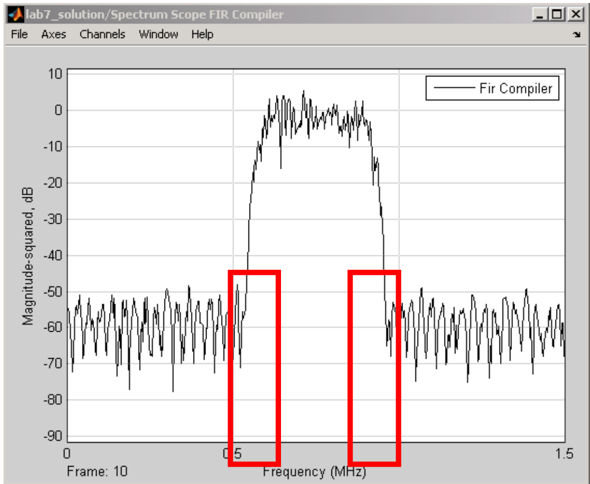


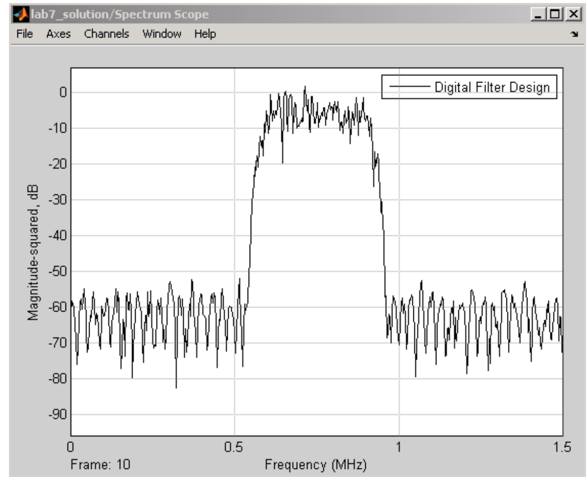
4. Correct the above problem by double-clicking on the **Gateway In** block and set the **Sampling period** to ***1/3000000*** and click **OK**

5. Double-click the **System Generator** block and set the Simulink system period field to ***1/(3*3000000)*** and click **OK**



6. Simulate the design and compare the results from the two **Spectrum Scopes** again. You should now see both waveforms and the results are very similar with some minor fixed-point effects.





Passband frequency response spectrum of 600 and 900 as set in the FDATool block

Note: verify the frequency response by observing a **Passband** spectrum which should lie between 600 and 900 KHz as previously set in the FDATool.

7. What are the min and max values for the MATLAB vector Num using the "max()" and "min()" MATLAB functions?

   Mim value __________

   Max value __________

8. Notice that the **FIR Compiler** is outputting a large number of bits. No bit truncation is being performed by the **FIR Compiler**. Insert a **Convert Block** to trim the output bits. What is the minimal number of bits that are required to maintain a close match to the golden reference design? ____

   Filter Output Quantization _____________

9. Using the **System Generator** token, generate a bitstream for this design and record the results below:

   \# of DSP48s ______________
   \# of Registers ______________
   \# of slices ______________
   Min clock period _____________

10. Add the Xilinx version of the **FDATool** to the diagram from the Xilinx Blockset/Index library. Setup the parameters to the values below, then click on **Design Filter**.

    Response Type:
    - Type = Bandpass

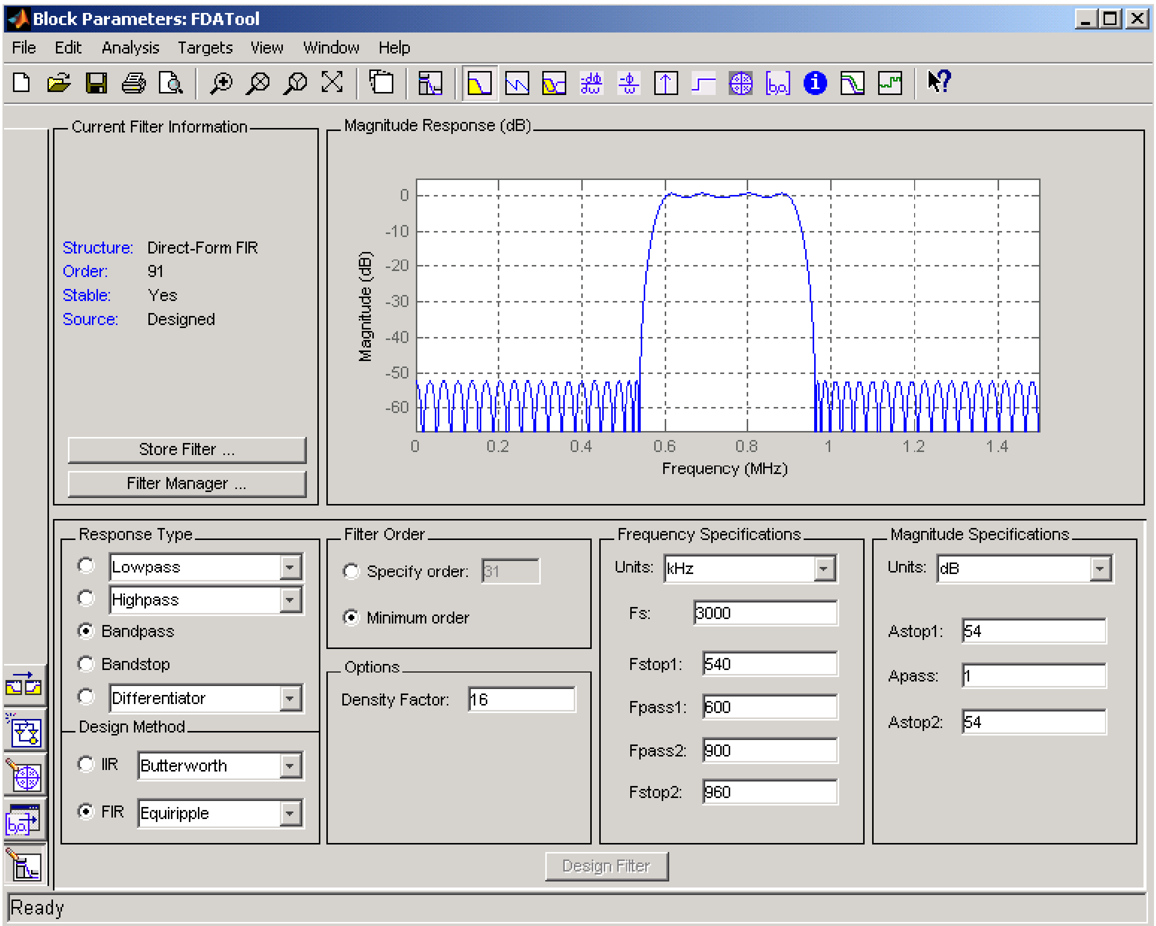
    Filter Order:
    - Select Minimum order

Frequency Specification:

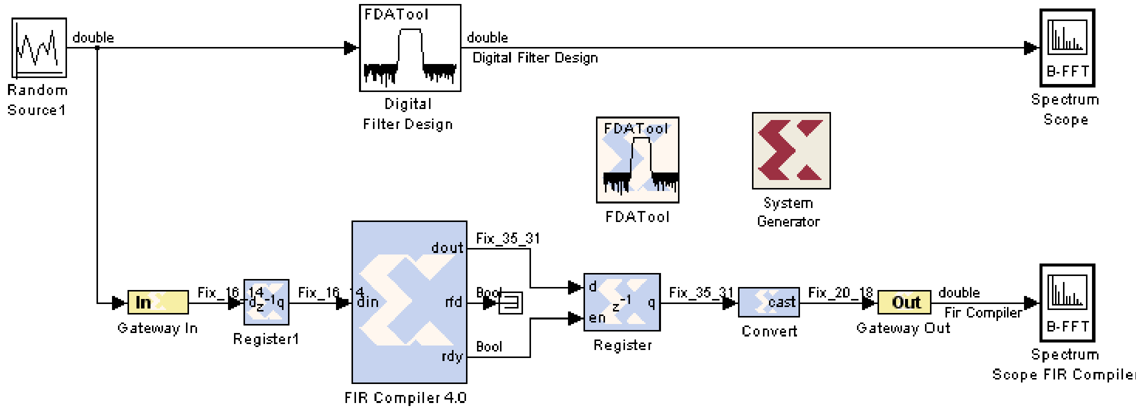
- Units = KHz
- Sampling Frequency = 3000 Khz
- Fstop 1 = 540 Khz
- Fpass1 = 600 Khz
- Fpass2 = 900 Khz
- Fstop2 = 960 Khz

Magnitude Specifications:

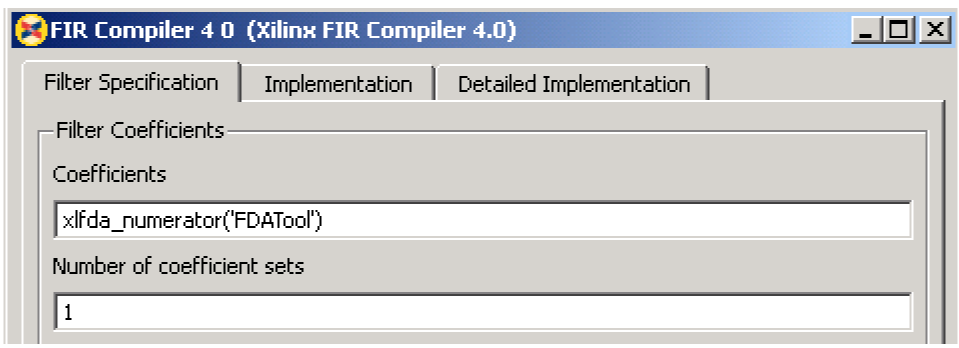
- Attenuation on both sides of the passband = 54 db
- Pass band ripple = 1



The problem with using The MathWorks FDATool is that each time the Simulink model is opened, the filter coefficients must be re-exported to the MATLAB workspace. The Xilinx version of the FDATool can be polled by the **FIR compiler** for the filter coefficients automatically eliminating the need to re-export to the workspace each time. At this point, your model should look similar to the one below.



10 As shown below, double-click on the **FIR Compiler** block and change the **Coefficients** field to **xlfda_numerator('FDATool').** This is a System Generator function that will call the Xilinx FDATool and return the filter coefficients of the designed filter.



11 Close the diagram, save the model and rerun the simulation again. The filter should display the correct waveforms as shown in **Step.2-6** above.

Solution

The complete solution to this lab is in the following location:

...<*sysgen_tree*>**/examples/getting_started_training/lab7/solution**