

Toyon Research Corporation

# Lab 5: Carrier Recovery

Chilipepper Tutorial Projects

Version 0.7  
5/17/2013

## Table of Contents

Introduction .....	4
Procedure.....	4
Objectives .....	4
Generate HDL Code .....	5
1.1    MATLAB Functions.....	5
1.2    MATLAB Test Bench.....	7
1.3    RX HDL Coder Project .....	9
Create and export Simulink models.....	14
2.1    Create MCU Simulink Design .....	14
2.2    Create ADC driver Simulink Design.....	15
2.3    Create Receiver Core Simulink Design.....	16
2.4    DC Offset Core .....	18
Configure Cores and Export Design .....	20
3.1    Needed IP Cores .....	20
3.2    Configuring the ADC Driver Port.....	20
3.3    Configuring the MCU Port .....	21
3.4    Configuring the dc offset Port .....	21
3.5    Configuring the GPIO Port .....	21
3.6    Configuring the RX Port .....	21
3.7    Configuring the Clock Generator IP Core .....	22
3.8    Pin Assignments.....	23
Create software project .....	25
4.1    Creating a new C Project .....	25
4.2    Adding Supporting files.....	27
4.3    Loading Hardware Platform with iMPACT .....	28
4.4    Debugging with SDK.....	30
Testing and Design Verification.....	31
5.1    Verification with ChipScope Pro .....	31
5.2    Exporting into MATLAB.....	33
5.2    MATLAB Analysis .....	34

---

Creating Wireless Transceivers Using MATLAB to HDL translation and Toyon Chilipepper

Appendix A	MATLAB Frequency Offset .....	36
Appendix B	MATLAB Test Bench .....	38

# Lab 5: Carrier Recovery

---

## Introduction

This lab will show you how to extend the receiver design in the previous lab to allow for Carrier recovery of the transmitted QPSK waveform. The Analog to Digital Conversion (ADC) used to receive the signal will take place on the Chilipepper board. The FMC initialization and microcontroller (MCU) signal control will be handled in software using the Xilinx Software Development Kit (SDK). Finally, the testing of results will be done using ChipScope and MATLAB. This lab assumes prior knowledge of the workings of HDL Coder as well as the Xilinx EDK environment. It is recommended that you complete the previous labs before completing this lab.

This lab is created using:

- MATLAB 2013a
- Xilinx ISE Design Suite 14.4 with EDK and System Generator
- Windows 7, 64-bit

## Procedure

This lab is organized into a series of steps, each including general instructions and supplementary steps, allowing you to take advantage of the lab according to your experience level.

This lab consists of the following basic steps:

- Generate HDL code from a MATLAB algorithm
- Create and export Simulink models using System Generator
- Configure your created PCores and export the design into SDK
- Create software to run your design
- Test and verify your results

## Objectives

After completing this lab, you will be able to:

- Implement Carrier Recovery for a received QPSK Waveform
- Create a Simulink model to implement a basic signal receiver
- Receive a QPSK Waveform using the Chilipepper FMC
- Create a software application to test your design
- Verify your results in ChipScope and analyze them using MATLAB

## Generate HDL Code

## Step 1

This section will show you how to create your MATLAB function and test bench files as well as the process for generating the HDL code used in the Simulink model.

### 1.1 MATLAB Functions

Your MATLAB functions will eventually become a core that will be synthesized into hardware. The algorithm describes the operations in each clock cycle, and processes data on a sample-by-sample basis. This lab requires several MATLAB functions all used in conjunction to receive and process the QPSK waveform. Labs 6 and 7 will build onto the MATLAB code used here, allowing for incremental increases in the processing performed within your FPGA Pcore. The first function used is shown in Figure 1-1.

```
function [r_out, s_f_out, s_c_out, f_est_out] = ...
    qpsk_rx(i_in, q_in, mu_foc_in)

    % scale input data coming from the Chilipepper ADC to be purely fractional
    % to avoid scaling issues
    r_in = complex(i_in, q_in);

    % frequency offset estimation. Note that time constant is input as integer
    [s_f_i, s_f_q, fe] = qpsk_rx_foc(i_in, q_in, mu_foc_in);

    % Square-root raised-cosine band-limited filtering
    [s_c_i, s_c_q] = qpsk_rx_srrc(s_f_i, s_f_q);

    % frequency estimation value
    f_est_out = fe;

    % original signal out (real version)
    r_out = real(r_in);

    % incremental signal outputs after frequency estimation and filtering
    s_f_out = complex(s_f_i, s_f_q);
    s_c_out = complex(s_c_i, s_c_q);
end
```

Figure 1-1: MATLAB function to analyze received signal.

This function is primarily used to call other functions which process the received waveform.

1. Create a directory for the project under C:\QPSK\_Projects\Project\_5.
2. Create a MATLAB directory within the main project directory.
3. Create a new **MATLAB function** with the contents of Figure 1-1.

## Creating Wireless Transceivers Using MATLAB to HDL translation and Toyon Chilipepper

4. **Save** this function as `qpsk_rx.m` inside the project directory.

While this function does not play a huge role in analyzing the modulated signal, it calls other functions which do, the first of which is the function `qpsk_rx_foc.m` which can be seen in **Appendix A**. This function implements a costas loop and its primary purpose is to synchronize the phase (offset of the frequency) so that you can correctly recover the carrier signal.

5. Create a new **MATLAB function** with the contents of Appendix A.
6. Save this function as `qpsk_rx_foc.m` inside the MATLAB project directory

The next function used to analyze the received waveform is called `qpsk_rx_srrc`. The purpose of this function is to apply a square-root-raised-cosine filter (SRRC) to the waveform, and is the same filter used in the previous labs. The MATLAB function is shown in Figure 1-2 below.

```
function [d_i_out, d_q_out] = qpsk_rx_srrc(d_i_in, d_q_in)

persistent buf_i buf_q

OS_RATE = 8;
f = SRRC;

if isempty(buf_i)
    buf_i = zeros(1, OS_RATE*2+1);
    buf_q = zeros(1, OS_RATE*2+1);
end

buf_i = [buf_i(2:end) d_i_in];
buf_q = [buf_q(2:end) d_q_in];

d_i_out = buf_i*f;
d_q_out = buf_q*f;
```

1-2: MATLAB function used to filter the received data.

As you can see from the figure this function uses the same SRRC variable used to filter the transmitted QPSK waveform. This variable is a look up table (LUT) that was created using the MATLAB function `make_srrc_lut.m`. This function, along with the LUT function `make_trig_lut.m`, which is used during frequency offset correction, must also be placed into the project directory. These functions did not change from the previous labs, so you can also copy them from the previous lab directory as well.

7. Create a new **MATLAB function** with the contents of Figure 1-2.
8. Save this function as `qpsk_rx_srrc.m` inside the project directory

## Creating Wireless Transceivers Using MATLAB to HDL translation and Toyon Chilipepper

9. Create or copy the `make_trig_lut.m` and `make_srrc_lut.m` **MATLAB functions** and save them into the project directory.

## 1.2 MATLAB Test Bench

Now that you have created the code needed to analyze the signal, we also need to create a test bench script for the functions. The primary function of this test bench script is to verify the functionality of the MATLAB algorithms. This is done by observing the output graph of the result, which in this case is the signal output of the costas loop and filter. To accomplish this, it is necessary to have a QPSK signal to analyze; therefore this script will also need to simulate a transmitted waveform for the analysis. After the algorithm has been verified using simulated data, and the FPGA design completed, you will again need to verify the algorithms with actual data from ChipScope. This will be done later in Section 6 of this lab. The code used for this script is shown in **Appendix B**.

**Note**

There is a variable called `sim` in the script which allows you to either load your received waveform data from ChipScope or simulate a received QPSK signal in MATLAB and analyze the results. Setting it to 1 simulates the waveform, 0 loads it from a ChipScope prn file.

1. Create a new **MATLAB script** with the contents of Appendix B.
2. Save this function as `qpsk_tb.m` inside the project directory

These are the only files required for the analysis of a received waveform when using exported ChipScope data. However to use simulated data for initial verification of the algorithm, you must also add the files used in the QPSK lab to your project directory. For your reference a list of the required files is given below. Refer to Lab 3 Output QPSK to recreate these files, or download them from the Chilipepper Github Repo<sup>1</sup>.

### Required files for creating Simulated QPSK waveform

- `make_train_lut.m`
- `CreateAppend16BitCRC.m`
- `qpsk_tx.m`
- `qpsk_tx_byte2sym.m`
- `qpsk_srrc.m`
- `mybitget.m`

<sup>1</sup> [https://github.com/Toyon/Chilipepper/tree/master/Labs/Lab\\_5/Matlab](https://github.com/Toyon/Chilipepper/tree/master/Labs/Lab_5/Matlab)

## Creating Wireless Transceivers Using MATLAB to HDL translation and Toyon Chilipepper

Once you have all the required MATLAB files in your project directory, you can run the test bench script to view the waveform analysis. You may have to run the script twice to create the needed LUT files. Your data should look similar to the results shown in Figures 1-3 and 1-4. Be sure you have the `sim` variable in the test bench script set to 1.

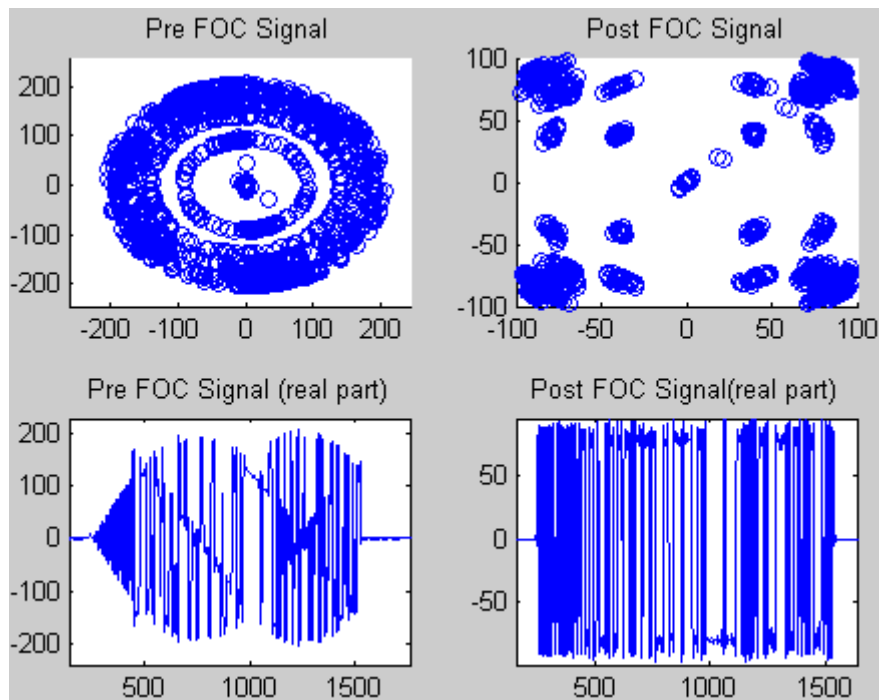


Figure 1-3: Analysis results for simulated QPSK signal

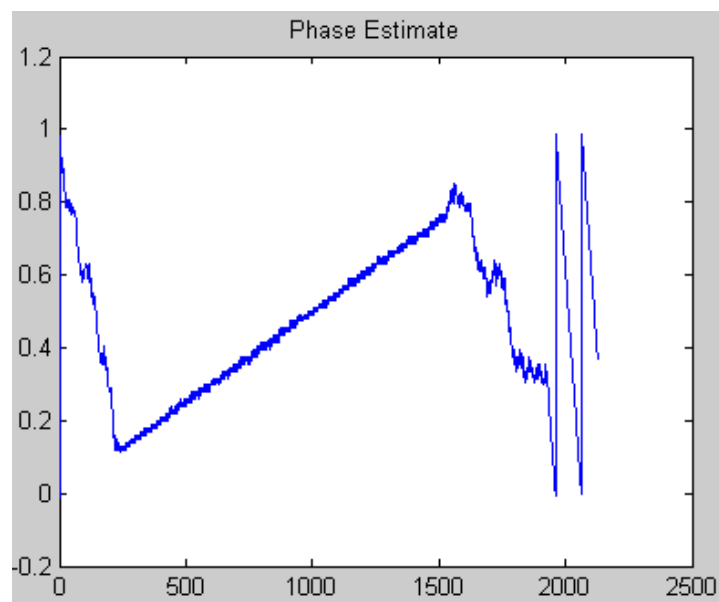


Figure 1-4: Phase estimate for simulated QPSK signal



## Creating Wireless Transceivers Using MATLAB to HDL translation and Toyon Chilipepper

As you can see from the results of the figure, the MATLAB functions allow for capturing the frequency offset, which is used to obtain the correct phase estimate for the QPSK waveform, giving the new signal (post FOC) signal shown in Figure 1-3. This signal is a much closer estimate of the originally transmitted waveform than the one shown before applying the offset correction (Pre FOC) and filtering.

### 1.3 RX HDL Coder Project

Using the same steps outlined in the previous labs, create a new HDL coder project called rx\_qpsk. Add your MATLAB function `qpsk_rx.m` and your test bench script `qpsk_tb.m` to the **MATLAB Function** and **MATLAB Test Bench** categories respectively.

Once you open your Workflow Advisor, you should be greeted with a screen similar to Figure 1-5 which allows you to define input types for your function. You can also allow them to be auto-defined by simply selecting run, and letting MATLAB analyze your design. For the inputs listed for the `qpsk_rx` function, the auto-defined types are fine.

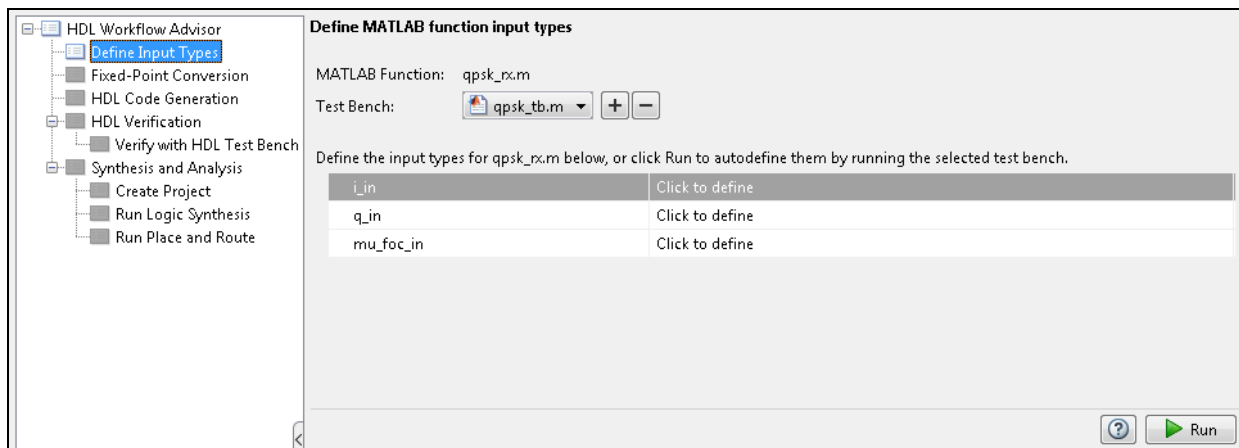


Figure 1-5: HDL Code Generation Workflow Advisor

1. Open Workflow Advisor and select “Run” to define the input types.
2. Click on Fixed-Point Conversion and select run this task. This process may take awhile.

Once the process is completed, you should receive a popup that says “Validation succeeded”. This means that MATLAB has successfully analyzed your design and selected fixed point types to replace the floating point arithmetic required in your algorithm. However, not all of the automatic selections are sufficient for our FPGA design; therefore several of the conversions will need to be modified.

3. Using the function dropdown menu at the top of the HDL Code Generation screen, select each of the functions in the design and make the following modifications.

## Creating Wireless Transceivers Using MATLAB to HDL translation and Toyon Chilipepper

## qpsk\_rx\_foc

Variable	Type	Sim Min	Sim Max	Static Min	Static Max	Whole Number	Proposed Type
<b>Input</b>							
mu_in	double	40	40			Yes	<b>numerictype(0, 12, 0)</b>
y_i	double	-204	205			Yes	<b>numerictype(1, 12, 0)</b>
y_q	double	-205	205			Yes	<b>numerictype(1, 12, 0)</b>
<b>Output</b>							
fe	double	-0.01	0.99			No	<b>numerictype(1, 26, 12)</b>
z_i_out	double	-161.66	160.86			No	<b>numerictype(1, 26, 12)</b>
z_q_out	double	-160.38	159.95			No	<b>numerictype(1, 26, 12)</b>
<b>Persistent</b>							
phi	double	-0.01	1			No	numerictype(1, 14, 12)
<b>Local</b>							
bf	double	-159.06	160.38			No	<b>numerictype(1, 26, 12)</b>
c	double	-0.01	0.01			No	<b>numerictype(1, 12, 12)</b>
e	double	-1	1			Yes	<b>numerictype(1, 2, 0)</b>
f_i	double	-1	1			No	numerictype(1, 14, 12)
f_q	double	-1	1			No	numerictype(1, 14, 12)
lCos	4096 x 1 double	-1	1			No	numerictype(1, 14, 12)
lSin	4096 x 1 double	-1	1			No	numerictype(1, 14, 12)
mu	double	0.01	0.01			No	<b>numerictype(0, 12, 12)</b>
phi12	double	1	4089			Yes	numerictype(0, 13, 0)
phiNew	double	-0.01	0.99			No	<b>numerictype(1, 26, 12)</b>
tf	double	-161.66	157.78			No	<b>numerictype(1, 26, 12)</b>
ti1	double	-177.24	178.86			No	<b>numerictype(1, 26, 12)</b>
ti2	double	-177.03	177.31			No	<b>numerictype(1, 26, 12)</b>
time_diff	double	-84.77	83.27			No	<b>numerictype(1, 26, 12)</b>
tq1	double	-175.07	177.05			No	<b>numerictype(1, 26, 12)</b>
tq2	double	-173.22	175.99			No	<b>numerictype(1, 26, 12)</b>
z_i	double	-161.66	160.86			No	<b>numerictype(1, 26, 12)</b>
z_q	double	-160.38	159.95			No	<b>numerictype(1, 26, 12)</b>

## SRRC

Variable	Type	Sim Min	Sim Max	Static Min	Static Max	Whole Number	Proposed Type
<b>Output</b>							
y	17 x 1 double	-0.05	0.34			No	<b>numerictype(1, 13, 12)</b>

## Creating Wireless Transceivers Using MATLAB to HDL translation and Toyon Chilipepper

**qpsk\_rx**

Variable	Type	Sim Min	Sim Max	Static Min	Static Max	Whole Number	Proposed Type
<b>Input</b>							
i_in	double	-204	205			Yes	<b>numerictype(1, 12, 0)</b>
mu_foc_in	double	40	40			Yes	<b>numerictype(0, 12, 0)</b>
q_in	double	-205	205			Yes	<b>numerictype(1, 12, 0)</b>
<b>Output</b>							
s_c_out	complex double	-97.73	97.11			No	<b>numerictype(1, 26, 12)</b>
s_f_out	complex double	-161.66	160.86			No	<b>numerictype(1, 26, 12)</b>
f_est_out	double	-0.01	0.99			No	<b>numerictype(1, 26, 12)</b>
r_out	double	-204	205			Yes	<b>numerictype(1, 12, 0)</b>
<b>Local</b>							
r_in	complex double	-205	205			Yes	<b>numerictype(1, 12, 0)</b>
fe	double	-0.01	0.99			No	<b>numerictype(1, 26, 12)</b>
s_c_i	double	-97.37	96.43			No	<b>numerictype(1, 26, 12)</b>
s_c_q	double	-97.73	97.11			No	<b>numerictype(1, 26, 12)</b>
s_f_i	double	-161.66	160.86			No	<b>numerictype(1, 26, 12)</b>
s_f_q	double	-160.38	159.95			No	<b>numerictype(1, 26, 12)</b>

**qpsk\_rx\_srrc**

Variable	Type	Sim Min	Sim Max	Static Min	Static Max	Whole Number	Proposed Type
<b>Input</b>							
d_i_in	double	-161.66	160.86			No	<b>numerictype(1, 26, 12)</b>
d_q_in	double	-160.38	159.95			No	<b>numerictype(1, 26, 12)</b>
<b>Output</b>							
d_i_out	double	-97.37	96.43			No	<b>numerictype(1, 26, 12)</b>
d_q_out	double	-97.73	97.11			No	<b>numerictype(1, 26, 12)</b>
<b>Persistent</b>							
buf_i	1 x 17 double	-161.66	160.86			No	<b>numerictype(1, 26, 12)</b>
buf_q	1 x 17 double	-160.38	159.95			No	<b>numerictype(1, 26, 12)</b>
<b>Local</b>							
f	17 x 1 double	-0.05	0.34			No	<b>numerictype(1, 13, 12)</b>
OS_RATE	double	8	8			Yes	<b>numerictype(0, 4, 0)</b>

## Creating Wireless Transceivers Using MATLAB to HDL translation and Toyon Chilipepper

Once all modifications have been made, select “Validate Types” in the top right area of the top toolbar to verify the design for your modified Fixed-Point conversions. Again, once the process is complete, you should get a message saying Validation Succeeded.

4. Select Validate Types to verify the new design
5. Click on HDL Code Generation and modify the settings according to the previous labs. There is no pipelining required for this project. Right Click and select run this task to generate your Xilinx Block Box Design. Your default design should look similar to Figure 1-6 below.

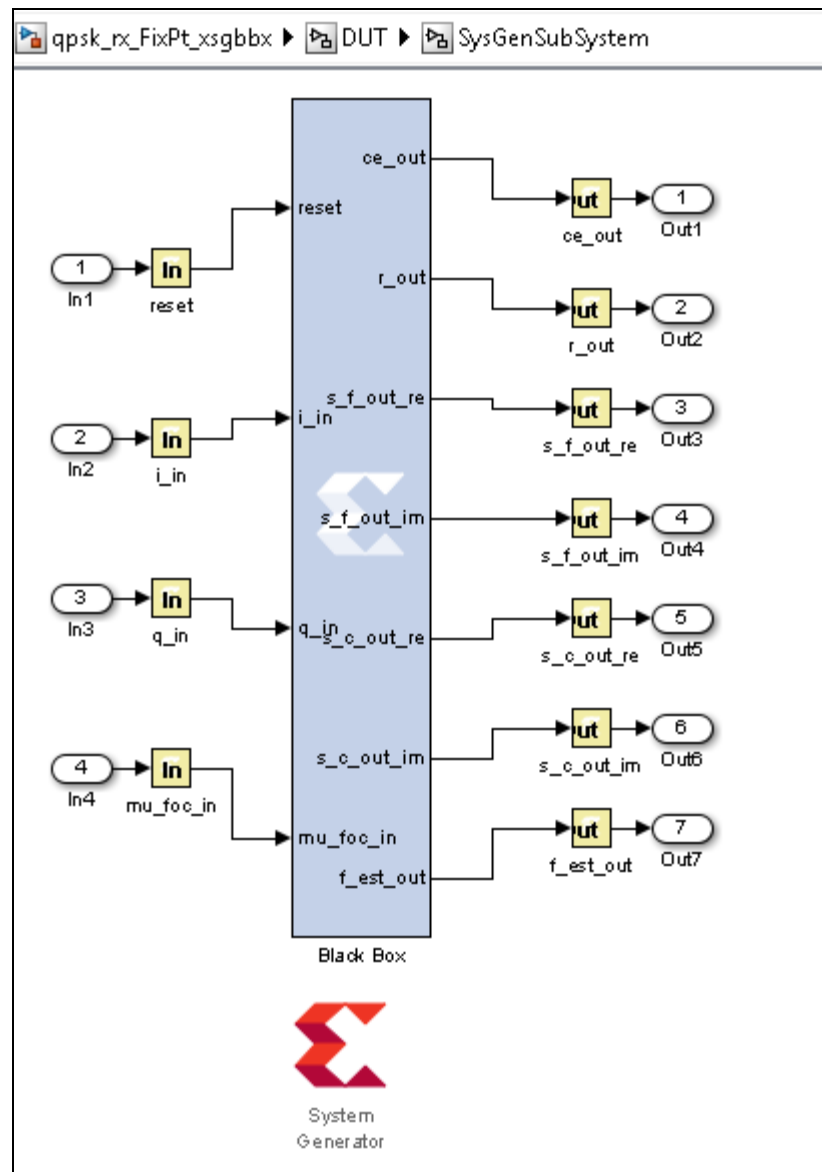


Figure 1-6: Xilinx Black Box generated by HDL Coder

---

Creating Wireless Transceivers Using MATLAB to HDL translation and Toyon Chilipepper

6. Copy the black box and System generator Blocks to a new Model just as in the previous labs.
7. Save the new model as rx.slx into the sysgen directory "Lab\_5\sysgen".
8. Copy the `qpsk_rx_FixPt_xsgbbxcfg.m` file into your Sysgen folder just as in the previous labs.
9. Create the hdl folder inside the Sysgen folder and copy your vhd files into this directory. Make sure you modify the previously copied m file to point to the new location of the vhd files.

## Create and export Simulink models

## Step 2

This section will show you how to create and customize your Simulink models to properly receive a signal and control the Chilipepper MCU and ADC. For additional information on this process, see lab 2.

### 2.1 Create MCU Simulink Design

The **Simulink model**<sup>2</sup> in Figure 2-1 will be used for the control signals to and from the **MCU**.

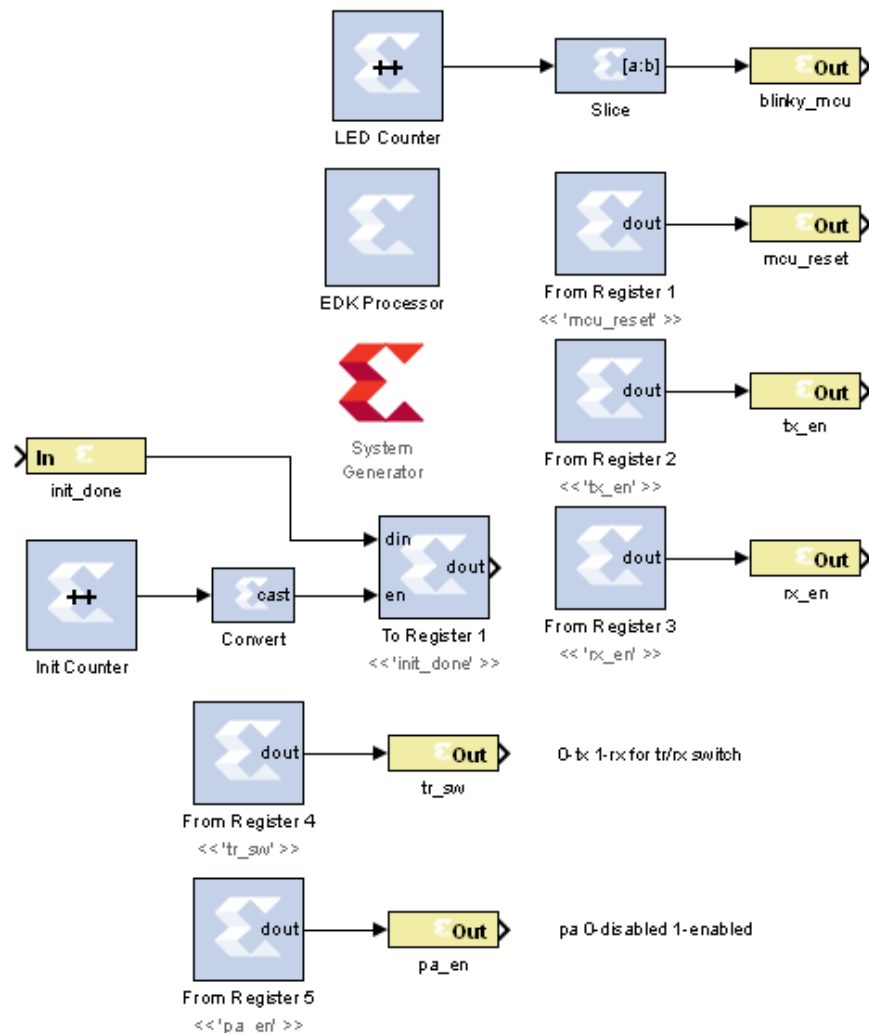


Figure 2-1: Simulink model for MCU control

<sup>2</sup> This model can be downloaded from [https://github.com/Toyon/Chilipepper/tree/master/Labs/Lab\\_5/sysgen](https://github.com/Toyon/Chilipepper/tree/master/Labs/Lab_5/sysgen)

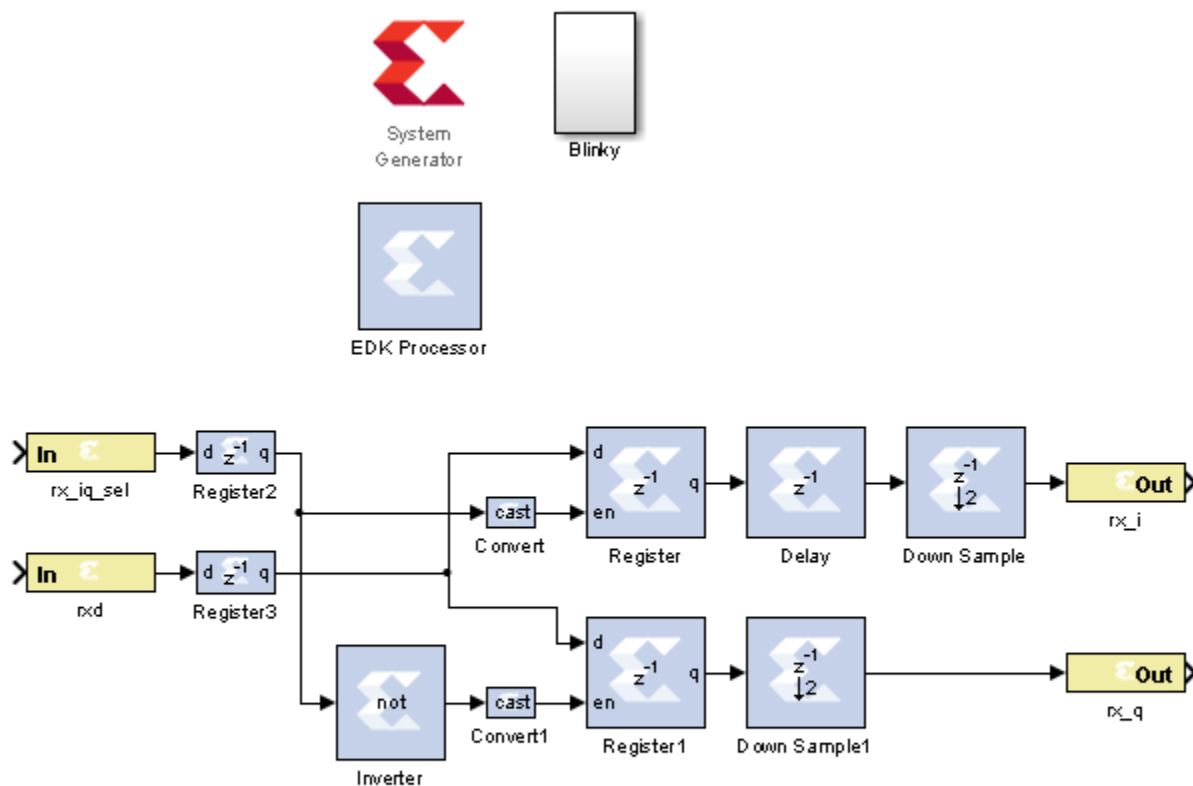
## Creating Wireless Transceivers Using MATLAB to HDL translation and Toyon Chilipepper

1. To create a new Simulink model, open MATLAB and click on the **Simulink Library** button in the Home menu.
2. Select **File → New → Model**
3. **Configure** this model and the system generator the same as in Lab 1, and **save** the design into the Sysgen folder. Name the file **mcu.slx** or something similar.



## 2.2 Create ADC driver Simulink Design

The **Simulink model**<sup>3</sup> In Figure 2-2 will be used for creating the signals which drive the ADC on Chilipepper.



By default inphase is IQ\_sel high and quadrature is IQ\_sel low

This core runs at 40 MHz and demultiplexes the rx\_d data into two 20 MHz streams for I and Q

Figure 2-2: Simulink model for ADC control

<sup>3</sup> This model can be downloaded from [https://github.com/Toyon/Chilipepper/tree/master/Labs/Lab\\_5/sysgen](https://github.com/Toyon/Chilipepper/tree/master/Labs/Lab_5/sysgen)

## Creating Wireless Transceivers Using MATLAB to HDL translation and Toyon Chilipepper

1. **Create** a new Simulink model and add the components from the Simulink blockset.
2. The white box labeled “Blinky” is simply a subsystem of the **Counter Slice** and LED **Gateway Out** blocks. The Blocks used for this subsystem are shown in Figure 2-3. Configure the Blinky subsystem identically to the other LED out systems in the previous labs.

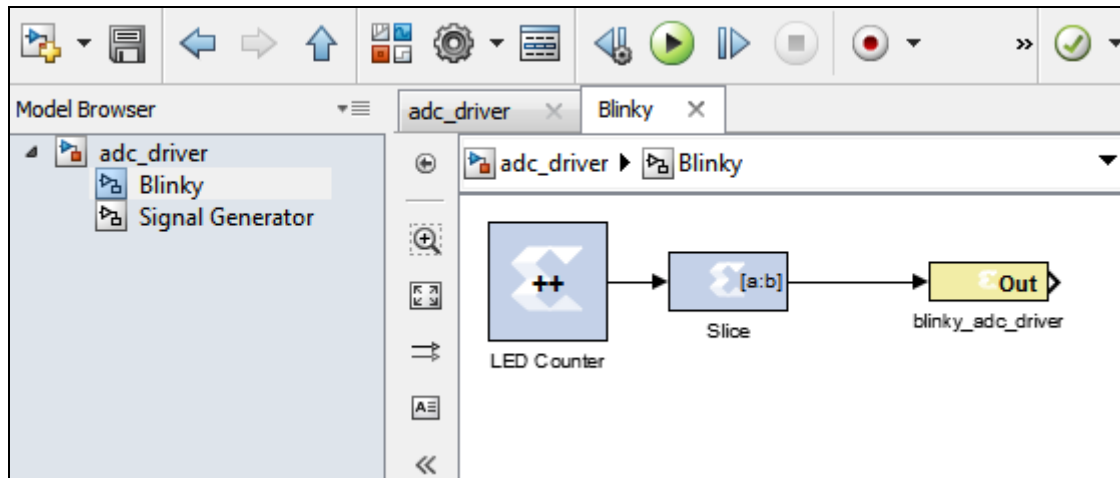


Figure 2-3: Blinky Subsystem

### 2.3 Create Receiver Core Simulink Design

The **Simulink model**<sup>4</sup> exported from your previous HDL Coder project is shown in Figure 2-4. This model will be used for receiving the DC offset correction output and sending the data to ChipScope for extraction.

<sup>4</sup> This model can be downloaded from [https://github.com/Toyon/Chilipepper/tree/master/Labs/Lab\\_5/sysgen](https://github.com/Toyon/Chilipepper/tree/master/Labs/Lab_5/sysgen)



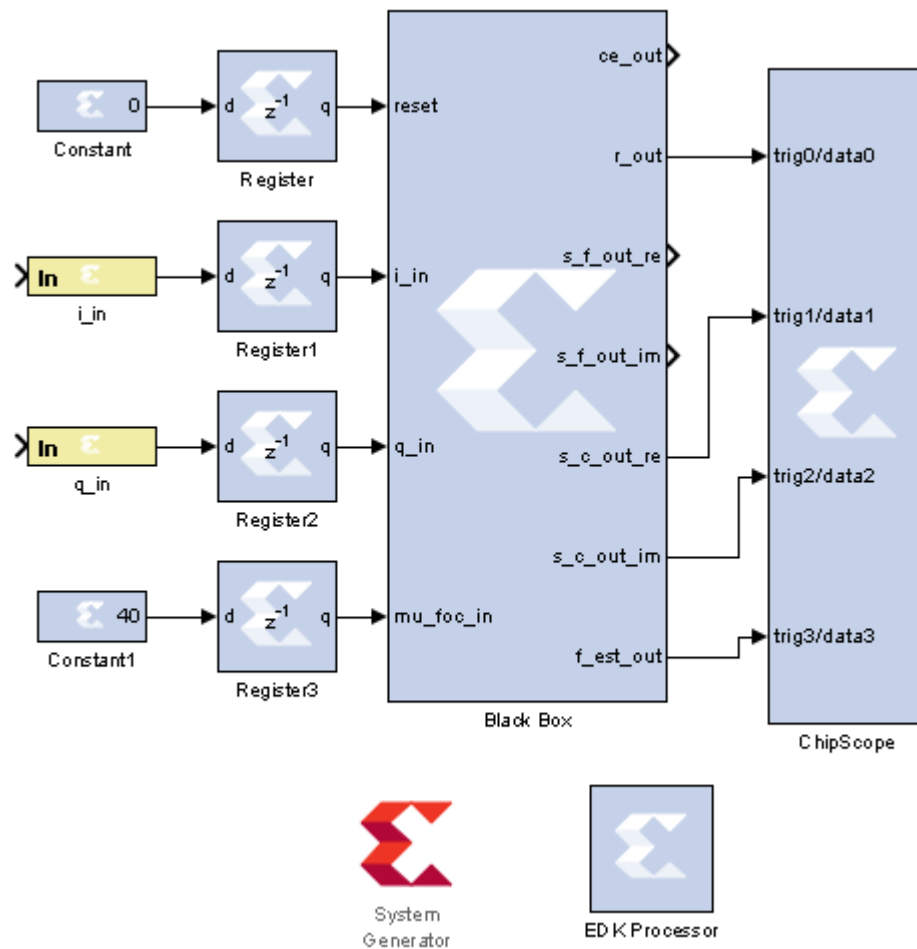


Figure 2-4: Simulink model for receiving DC Offset output

1. **Modify** the Simulink model `rx.slx` created earlier to look similar to Figure 2.4.
2. Both `i_in` and `q_in` should be signed 12 bit (0 decimal bits) inputs. Additionally the constant for `mu_foc_in` should be set to `floor(.01*2^12)`.
3. **Save** the design into the Sysgen folder and copy the vhd file into the Sysgen/hdl directory. If you haven't already done it, **be sure to copy and change the cfg file as well to find the files in your new directory structure.**

**Note**

Additionally, you can add the `s_f_out` real and complex signals to chipscope to analyze the changes to the signal due to band-limited filtering; however this is out of the scope of this Lab.

## 2.4 DC Offset Core

The last **Simulink model**<sup>5</sup> needed for this FPGA design is the DC Offset model created in the previous lab. For your reference this model is shown in Figure 2-5 below.

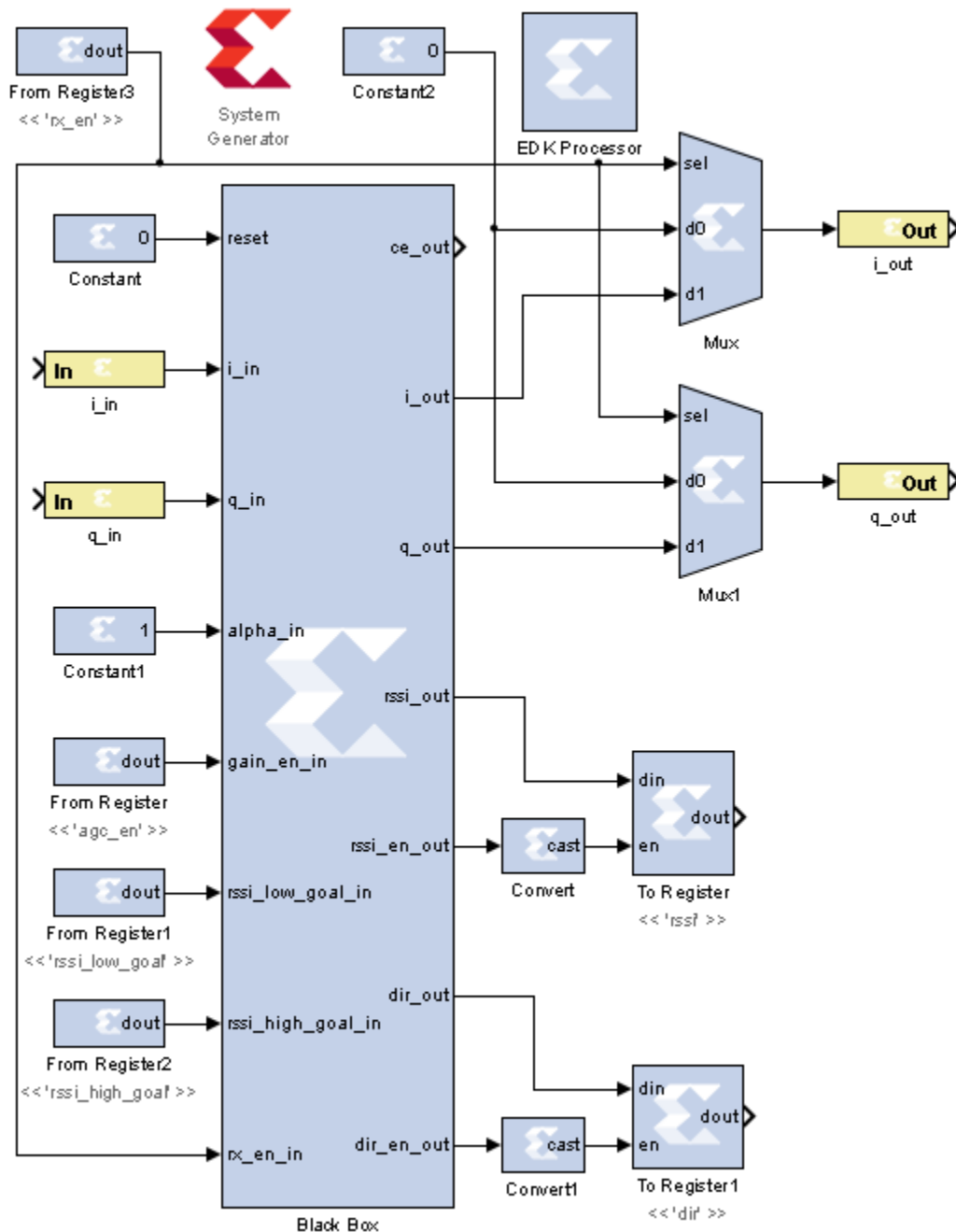


Figure 2-5: Simulink model for receiving ADC output and applying DC Offset Correction

<sup>5</sup> This model can be downloaded from [https://github.com/Toyon/Chilipepper/tree/master/Labs/Lab\\_5/sysgen](https://github.com/Toyon/Chilipepper/tree/master/Labs/Lab_5/sysgen)

---

Creating Wireless Transceivers Using MATLAB to HDL translation and Toyon Chilipepper

---

This Model is slightly different than the one used in Lab 4, as it has output ports rather than ChipScope connections for its output. Additionally, the debug ports have been removed, and the design is assumed to be working correctly. You may use your previous design, and simply remove the ChipScope block and add the output port if you do not want to recreate the HDL coder project, however this lab assumes the project was recreated without the debug ports.

1. Modify the `dc_offset_correction.m` and `testbench` file to remove the debug output. You may use the MATLAB files on the GitHub Repo as a reference.
2. Create a new HDL coder project with the dc offset matlab files, and follow the project setup used in Lab 4.
3. Create the Simulink model shown in Figure 2-5 using your new black box design.

**Note**

Since this Lab has multiple black box designs, be sure that all the appropriate vhd and config.m files are located within the Sysgen directory. Additionally you have the option of combining the black box designs into a single Simulink Model, however this is not described here and is left as an optional exercise.

The last step is to create a new EDK project to export your models to. Refer to Lab 0 Step 3 for more information on how to **Create a New Blank EDK Project**. Be sure to follow the directory structure used. Also verify your **Compilation Settings** are correct as shown in Lab 0 Section 4.1. Once each Simulink model has been exported successfully, you're ready to configure your FPGA design.

## Configure Cores and Export Design

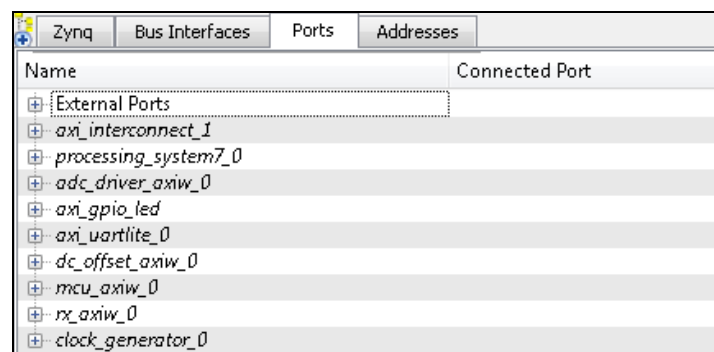
## Step 3

This section will show you how to integrate your PCores into your FPGA design using EDK. There are several components that must be configured for the design of this project. A quick list of the cores needed is given below. Refer to lab 0 sections 4.3 and 5.1 for information on how to add cores to the design.

### 3.1 Needed IP Cores

- ADC Driver PCore created in Simulink
- MCU PCore created in Simulink
- rx PCore created in Simulink
- DC Offset PCore created in Simulink
- Clock Generator IP Core
- Processing System IP Core
- AXI Interconnect IP Core
- GPIO Cores for LEDs
- AXI\_UART (Lite) Core

In addition, several of these cores will require external ports. Be sure that you have access to modifying the external port settings. Refer to Figure 3-1 Below.



Name	Connected Port
External Ports	
axi_interconnect_1	
processing_system7_0	
adc_driver_axiw_0	
axi_gpio_led	
axi_uartlite_0	
dc_offset_axiw_0	
mcu_axiw_0	
rx_axiw_0	
clock_generator_0	

Figure 3-1: EDK project ports list

### 3.2 Configuring the ADC Driver Port

Expand the **ADC Driver** port. There are 6 individual I/O pins which need to be routed on this port.

---

Creating Wireless Transceivers Using MATLAB to HDL translation and Toyon Chilipepper

---

1. The first three are the `rx_iq_sel`, the `rx_d` and the `bliky_adc_driver` pins. Each of these pins can be assigned as **External ports**.
2. Next are the `rx_i` and the `rx_q` output pins. Connect these pins to the `i_in` and `q_in` pins of the `dc_offset` PCore created in Simulink.
3. The `sysgen_clk` pin can be skipped for now and will be connected later in **section 3.4**.

### 3.3 Configuring the MCU Port

Expand the **MCU** port. There are 8 individual I/O pins which need to be routed on this port.

1. Configuring this port is very simple as all of the pins with the exception of the `sysgen_clk` are assigned as **External ports**.

### 3.4 Configuring the dc offset Port

Expand the **dc\_offset** port. There are 5 individual I/O pins which need to be routed on this port.

1. Ignore the `Sysgen_clk` pin for now. Verify that the `i_in` and `q_in` pins are connected to the `adc_driver` as a result of step 3.2.
2. Next are the `i_out` and the `q_out` output pins. Connect these pins to the `i_in` and `q_in` pins of the `rx` PCore created in Simulink.

### 3.5 Configuring the GPIO Port

This port will be used later in SDK to verify the functionality of the receiver.

1. Add an AXI General Purpose IO to the design. Check the box to Enable Channel 2 and give each channel a width of 1 bit. Name the port `axi_gpio_led` or something similar.
2. Expand the `IO_IF` section of the GPIO, and assign the `GPIO_IO` and `GPIO2_IO` pins to external ports. The other pins can be left blank.

### 3.6 Configuring the RX Port

Expand the **RX** port. There are 3 individual I/O pins which need to be routed on this port.

1. Ignore the `Sysgen_clk` pin for now. Verify that the `i_in` and `q_in` pins are connected to the `dc_offset` core as a result of step 3.4.

### 3.7 Configuring the Clock Generator IP Core

The Clock Generator is used in this project to distribute the appropriate clock signals to each of the PCores, as well as any external hardware which may require a clock signal. For this project, the Clock Generator is sourced from the 40 MHz `pll_clk_out` on the Chilipepper radio board (as described in the **Chilipepper user's guide**).

1. **Double click** the Clock Generator PCore and **configure** the settings as follows
  - Input Clock Frequency of **40Mhz**
  - CLKFBIN Required Frequency of **40Mhz** with **no Clock Deskew**
  - CLKFBOUT Required Frequency of **40Mhz**, Required Group **PLLE0**, and **Buffered True**
  - CLKOUT0 Required Frequency of **20MHz**, 0Phase, **PLLE0** group and **Buffered true**
  - CLKOUT1 Required Frequency of **40MHz**, 0Phase, **PLLE0** group and **Buffered true**
  - CLKOUT2 Required frequency of **20Mhz**, 0Phase, **PLLE0** group and **Buffered true**
  - CLKOUT3 Required Frequency of **40MHz**, **90 Phase**, **PLLE0** group and **Buffered true**
  - CLKOUT4 Required Frequency of **40MHz**, 0Phase, **PLLE0** group and **Buffered true**

Now that the settings are configured you should have several clocks in your clock generator list.

2. **Connect** the pins according to the following.
  - CLKIN       $\longrightarrow$       External Ports
  - CLKOUT0  $\longrightarrow$       `mcu::sysgen_clk`
  - CLKOUT1  $\longrightarrow$       `adc_driver::sysgen_clk`
  - CLKOUT2  $\longrightarrow$       `rx:: sysgen_clk && dc_offset::sysgen_clk`
  - CLKOUT3  $\longrightarrow$       External Ports
  - CLKOUT4  $\longrightarrow$       External Ports
  - CLKFBIN  $\longrightarrow$       CLKFBOUT
  - RST         $\longrightarrow$       `net_gnd`
  - LOCKED     $\longrightarrow$       External Port

**Note**

The CLKOUT3 pin has a 90 degree phase shift as mentioned in the **Chilipepper user's guide**. This line will be used as the RX\_CLK signal from the FPGA to the radio board.

Your Clock Generator port should look similar to Figure 3-2.

clock_generator_0		
CLKIN	External Ports::clock_generator_0_pll_pin	I
CLKOUT0	mcu_axiwr_0::sysgen_clk	O
CLKOUT1	adc_driver_axiwr_0::sysgen_clk	O
CLKOUT2	rx_axiwr_0::sysgen_clk	O
CLKOUT3	dc_offset_axiwr_0::sysgen_clk	O
CLKOUT4	External Ports::clock_generator_0_rx_clk_pin	O
CLKFBIN	External Ports::clock_generator_0_tx_clk_pin	O
CLKFBOUT	clock_generator_0::CLKFBOUT	I
CLKFBOUT	clock_generator_0::CLKFBIN	O
RST	net_gnd	I
LOCKED	External Ports::clock_generator_0_LOCKED_pin	O

Figure 3-2: Clock Generator port configuration

### 3.8 Pin Assignments

Once the clock generator is configured correctly, the sysgen clock for the other cores should be set as well. The last step is to setup the **pin assignments** for the external ports.

1. Rename the pins of the external ports so they are easily identifiable. Figure 3-3 shows the names used in this Lab, however you don't have to use the same naming convention.
2. Open the **Project** tab.
3. Double-click on the **UCF File: data\system.ucf** from this panel, to open the constraints file.
4. Fill in the pin out information for your design using Figure 3-3 below as a reference.



Be sure that the **orientation** of the RXD pins is set correctly. If you follow the pin list in the figure above, you must **reverse** the RXD pins in the external ports assignment section. This is done using the same method used in Lab 0 Section 5.2 for the LEDs.

5. Prior to EDK version 14.4, Xilinx had a [documented issue](http://www.xilinx.com/support/answers/51739.htm)<sup>6</sup> with AXI-bus generation for Simulink PCores targeting the Zynq FPGA. Refer to this issue for more information. As in

<sup>6</sup> Issue can be found at <http://www.xilinx.com/support/answers/51739.htm>

## Creating Wireless Transceivers Using MATLAB to HDL translation and Toyon Chilipepper

Lab 0 section 5.2, this bug must be corrected if your **EDK version is 14.3 or lower**. The steps to perform are identical to those in the previous labs; however they must be performed for **all** of the PCores used in this lab.

6. Select the **Export Design** button from the navigator window on the left. Click the **Export and Launch SDK** button. This process may take awhile.

```
##### PL clocks and reset #####
NET clock_generator_0_pll_pin          LOC = D18 | IOSTANDARD = LVCMOS25;
NET clock_generator_0_pll_pin          TNM_NET = clock_generator_0_pll;
TIMESPEC TS_clock_generator_0_pll = PERIOD clock_generator_0_pll 40.000 MHz;
##### Rx – FMC interface at 2.5V #####
NET clock_generator_0_rx_clk_pin        LOC = J18          | IOSTANDARD = LVCMOS25 | DRIVE = 4 | SLEW = FAST;
NET adc_driver_axiw_0_rx_iq_sel_pin     LOC = N19          | IOSTANDARD = LVCMOS25;
NET adc_driver_axiw_0_rxd_pin[0]        LOC = M21          | IOSTANDARD = LVCMOS25;
NET adc_driver_axiw_0_rxd_pin[1]        LOC = J21          | IOSTANDARD = LVCMOS25;
NET adc_driver_axiw_0_rxd_pin[2]        LOC = M22          | IOSTANDARD = LVCMOS25;
NET adc_driver_axiw_0_rxd_pin[3]        LOC = J22          | IOSTANDARD = LVCMOS25;
NET adc_driver_axiw_0_rxd_pin[4]        LOC = T16          | IOSTANDARD = LVCMOS25;
NET adc_driver_axiw_0_rxd_pin[5]        LOC = P20          | IOSTANDARD = LVCMOS25;
NET adc_driver_axiw_0_rxd_pin[6]        LOC = T17          | IOSTANDARD = LVCMOS25;
NET adc_driver_axiw_0_rxd_pin[7]        LOC = N17          | IOSTANDARD = LVCMOS25;
NET adc_driver_axiw_0_rxd_pin[8]        LOC = J20          | IOSTANDARD = LVCMOS25;
NET adc_driver_axiw_0_rxd_pin[9]        LOC = P21          | IOSTANDARD = LVCMOS25;
NET adc_driver_axiw_0_rxd_pin[10]       LOC = N18          | IOSTANDARD = LVCMOS25;
NET adc_driver_axiw_0_rxd_pin[11]       LOC = J16          | IOSTANDARD = LVCMOS25;
##### MCU Interface #####
NET clock_generator_0_tx_clk_pin        LOC = C17          | IOSTANDARD = LVCMOS25 | DRIVE = 4 | SLEW = FAST;
##### MCU Interface #####
NET axi_uartlite_0_RX_pin              LOC = R19          | IOSTANDARD = LVCMOS25 | DRIVE = 4 | SLEW = FAST;
NET axi_uartlite_0_TX_pin              LOC = L21          | IOSTANDARD = LVCMOS25 | DRIVE = 4 | SLEW = FAST;
NET mcu_axiw_0_mcu_reset_pin           LOC = K20          | IOSTANDARD = LVCMOS25 | DRIVE = 4 | SLEW = FAST;
NET mcu_axiw_0_tx_en_pin                LOC = D22          | IOSTANDARD = LVCMOS25 | DRIVE = 4 | SLEW = FAST;
NET mcu_axiw_0_tr_sw_pin                LOC = D20          | IOSTANDARD = LVCMOS25 | DRIVE = 4 | SLEW = FAST;
NET mcu_axiw_0_rx_en_pin                LOC = C22          | IOSTANDARD = LVCMOS25 | DRIVE = 4 | SLEW = FAST;
NET mcu_axiw_0_pa_en_pin                LOC = E21          | IOSTANDARD = LVCMOS25 | DRIVE = 4 | SLEW = FAST;
NET mcu_axiw_0_init_done_pin            LOC = K19          | IOSTANDARD = LVCMOS25;
##### LEDs #####
NET axi_gpio_led_GPIO_IO_pin           LOC = T22 | IOSTANDARD=LVCMOS33; # "LD0"
NET axi_gpio_led_GPIO2_IO_pin          LOC = T21 | IOSTANDARD=LVCMOS33; # "LD1"
NET mcu_axiw_0_blinky_mcu_pin          LOC = U22 | IOSTANDARD=LVCMOS33; # "LD2"
NET adc_driver_axiw_0_blinky_adc_driver_pin LOC = U21 | IOSTANDARD=LVCMOS33; # "LD3"
NET clock_generator_0_LOCKED_pin        LOC = V22 | IOSTANDARD=LVCMOS33; # "LD4"
```

Figure 3-3: EDK project pin assignments



## Create software project

## Step 4

---

Once the design is compiled and exported, you'll be greeted with a screen asking you where you would like to store your software project. It is very helpful to create the workspace folder in the same directory as your Sysgen and EDK folders. Doing this will keep all relevant files in the same location.

### 4.1 Creating a new C Project

This section will show you how to create a C program to test your receive tone project.

1. Select **File → New → Application Project**.
2. Name the project `qpsk_rx` or something similar and leave the other settings at their defaults. Be sure to select **Hello World** from the **Select Project Template** section.
3. Click **Finish**. You should now see your project folder, as well as a **board support package** (bsp) folder.
4. If you navigate into the project folder, and into the src folder, you should see a `helloworld.c` file. This is the file we will be using to create our software design. Feel free to give the file a more descriptive name such as `main.c` or something similar.
5. **Double click** the file to open it and **replace** all of its contents with the code in Figure 4-1.

---

#### Note

It would be helpful if you have completed the Embedded System Design tutorial in the *ZedBoard AP SoC Concepts Tools and Techniques Guide*. Refer to Lab 1 for more information on the MCU signal control using C code within SDK.

---

## Creating Wireless Transceivers Using MATLAB to HDL translation and Toyon Chilipepper

```

#include <stdio.h>
#include "platform.h"
#include "xbasic_types.h"
#include "xgpio.h"
#include "xparameters.h"
#include "xstatus.h"
#include "chilipepper.h"
#include "xuartps.h"
#include "xil_printf.h"
#include "xscugic.h"
#include "xil_exception.h"

XGpio gpio_blinky;

int SetupPeripherals( void );

int main()
{
    int aliveLed = 0;
    static int BlinkCount = 0;
    init_platform();
    if(SetupPeripherals() != XST_SUCCESS)
        return -1;

    if ( Chilipepper_Initialize() != 0 )
        return -1;

    Chilipepper_SetPA( 1 );
    Chilipepper_SetTxRxSw( 1 ); // 0- transmit, 1-receive
    while (1)
    {
        Chilipepper_ControlAgc(); //update the Chilipepper AGC
        BlinkCount += 1;
        if (BlinkCount > 500000)
        {
            if (aliveLed == 0)
                aliveLed = 1;
            else
                aliveLed = 0;
            BlinkCount = 1;
            XGpio_DiscreteWrite(&gpio_blinky, 2, aliveLed); //blink LEDs
            XGpio_DiscreteWrite(&gpio_blinky, 1, ~aliveLed);
        }
    }
    cleanup_platform();
    return 0;
}

int SetupPeripherals( void )
{
    XGpio_Initialize(&gpio_blinky, XPAR_AXI_GPIO_LED_DEVICE_ID);
    XGpio_SetDataDirection(&gpio_blinky, 2, 0);
    XGpio_SetDataDirection(&gpio_blinky, 1, 0);
    XGpio_DiscreteWrite(&gpio_blinky, 1, 0);
    XGpio_DiscreteWrite(&gpio_blinky, 2, 0);
    return XST_SUCCESS;
}

```

Figure 4-1: Main C function for rx\_qpsk project

## 4.2 Adding Supporting files

In addition to the main c file, you need the library files for the Chilipepper board. The 2 required files for this Lab are Chilipepper.c and Chilipepper.h and can be found on the github repo. Place these files in the src directory of your project workspace.

1. [Chilipepper.c](#) – This file is the primary library file for the Chilipepper board. It contains functions for modifying the MCU registers as well as basic helper functions for tasks such as initialization, transmitting, and receiving.
2. [Chilipepper.h](#) – This file holds the function prototypes for the Chilipepper.c functions.

---

**Note**

In addition to the Library files, you also need to include a Math library which contains the `pow` function that is used when creating the CRC. See Lab 3 section 4.1 for more information on how to add the Math Library to your project.

---

The Chilipepper.c library file is configured for both TX and RX cores as well as a UART to talk to the on board MCU and configure its settings. To use the library file properly, you must specify which of these features you will use. To do this, modify lines 8-12 of the Chilipepper.c file to define a variable for the cores you will be using. Your code should resemble the following, as we will be using all cores except the TX\_DRIVER for this Lab.

```
#define MCU_UART
#define MCU_DRIVER
#define DC_OFFSET
//#define TX_DRIVER
#define RX_DRIVER
```



If you are still not able to compile your C design due to include errors, you may need to tell SDK where your PCore drivers are stored. If you click on Xilinx Tools → Repositories, you can specify (in Global Repositories) where the EDK directory of your project is. (This will need to be changed for each new project)

---

### 4.3 Loading Hardware Platform with iMPACT

Once your program is written and compiled you are ready to test the design! This is done by programming the FPGA with your hardware descriptions defined in the bit file generated in EDK, and running your software on top of this design. For this lab, you can verify your design by connecting two Chilipepper boards together using an attenuator. On one board, you should run the latest version of Lab 3 which allows you to either send a single packet via button press, or multiple packets using the switch on the FPGA.

1. Connect the Chilipepper to the FPGA board and verify all cables are connected properly and the jumper settings are correct. Verify this by using the *Chilipepper user guide* and the *ZED Board Hardware users guide* as a reference. Also See Lab 0 for details on Jumper Configuration.
2. Once the FPGA and radio board are connected correctly, turn on the board.
3. Open iMPACT in the ISE Design tools.
4. Select no if Impact asks you to load the last saved project.
5. Select yes to allow iMPACT to automatically create a new project for you. If you receive any connection errors, verify your USB or JTAG programmer cables are connected properly.
6. Select the Automatic option for the JTAG boundary scan setting and click ok.
7. Hit yes to assign configuration files. Bypass the first file selection, but for the second selection, browse to the location of your system.bit file. It should be inside the "Implementation" folder of your EDK project folder.
8. Select ok on the next screen verifying that the board displayed is your Zynq xc7z020 board. It should look similar to Figure 4-2 below.
9. Right click on the xc7z020 board icon (should be on the right), select program and hit ok.

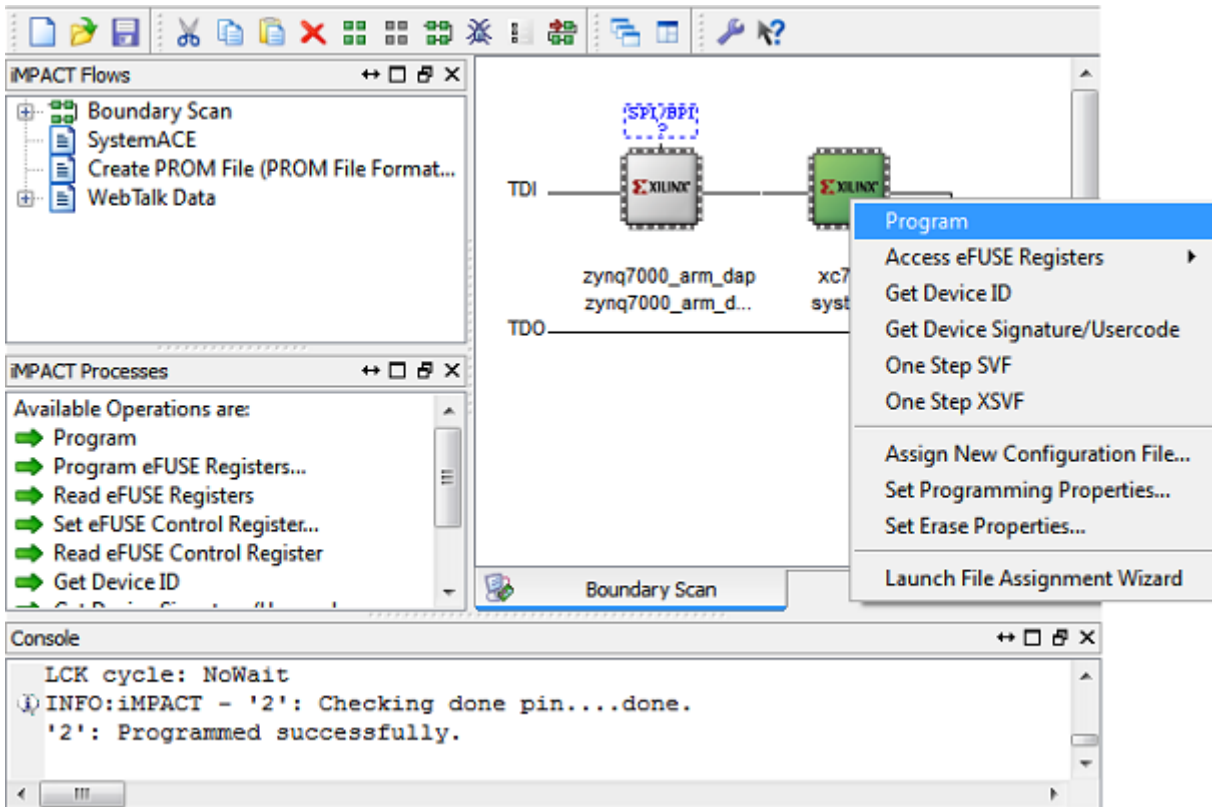
---

**Note**

If you are running lab 3 from a second PC, you will need to repeat this process for the second board using the Lab 3 system.bit file. Alternatively, you can run Lab 3 directly from the SD card by loading a standard SD card with the Boot.bin file for lab 3, which can be found on the github repo.

---

## Creating Wireless Transceivers Using MATLAB to HDL translation and Toyon Chilipepper



4-2: iMPACT configuration screen

To load Lab 3 via SD card:

1. Place the file on the SD card, and place the card inside the SD slot of the FPGA.
2. Configure the jumpers on the FPGA as shown in Figure 4-3.
3. Turn on the board, and the program should load after about 30 seconds. Check for the blue light, indicated the load was successful.

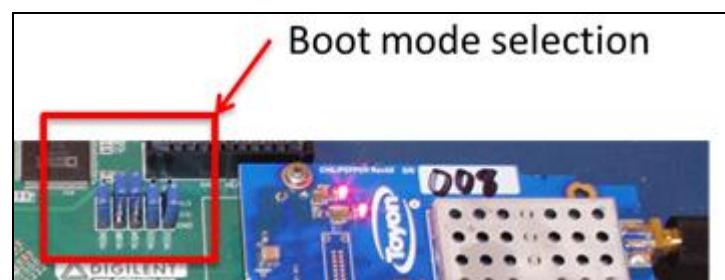



Figure 4-3: Jumper configuration needed to load a project via SD card

#### 4.4 Debugging with SDK

If the hardware design is correct, you should see the LEDs start blinking on the board, as well as a blue light indicating the program was successful. You should also see the LED blinking on the second FPGA indicating the Lab 3 project is working properly. You can now return to the SDK project screen to test your software.

1. Test it by **right clicking** the project name folder and selecting **Debug As → Launch on Hardware**.
2. You should now be taken to a screen which shows the first pointer initialization as highlighted. You can now start the software program by clicking the  (play) button in the top menu.


If the software initialization worked, you should see a green light on the Chilipepper as well as LED0 and LED1 blinking alternatively.

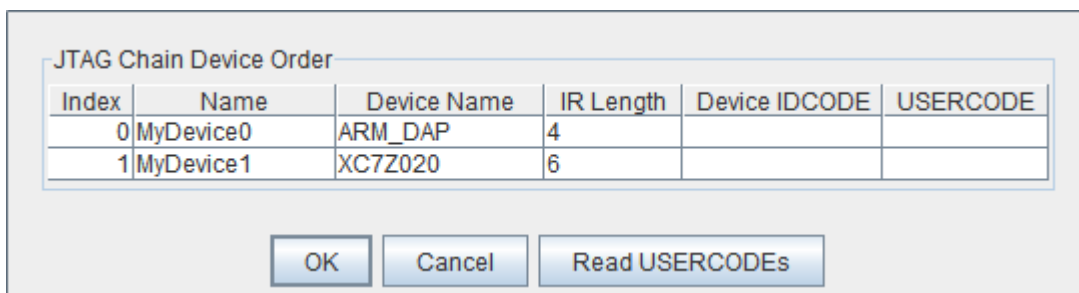
## Testing and Design Verification

## Step 5

### 5.1 Verification with ChipScope Pro

There are several methods available for verifying the received QPSK transmission. This lab focuses on verification using ChipScope Pro, as well as exporting to MATLAB for further analysis.

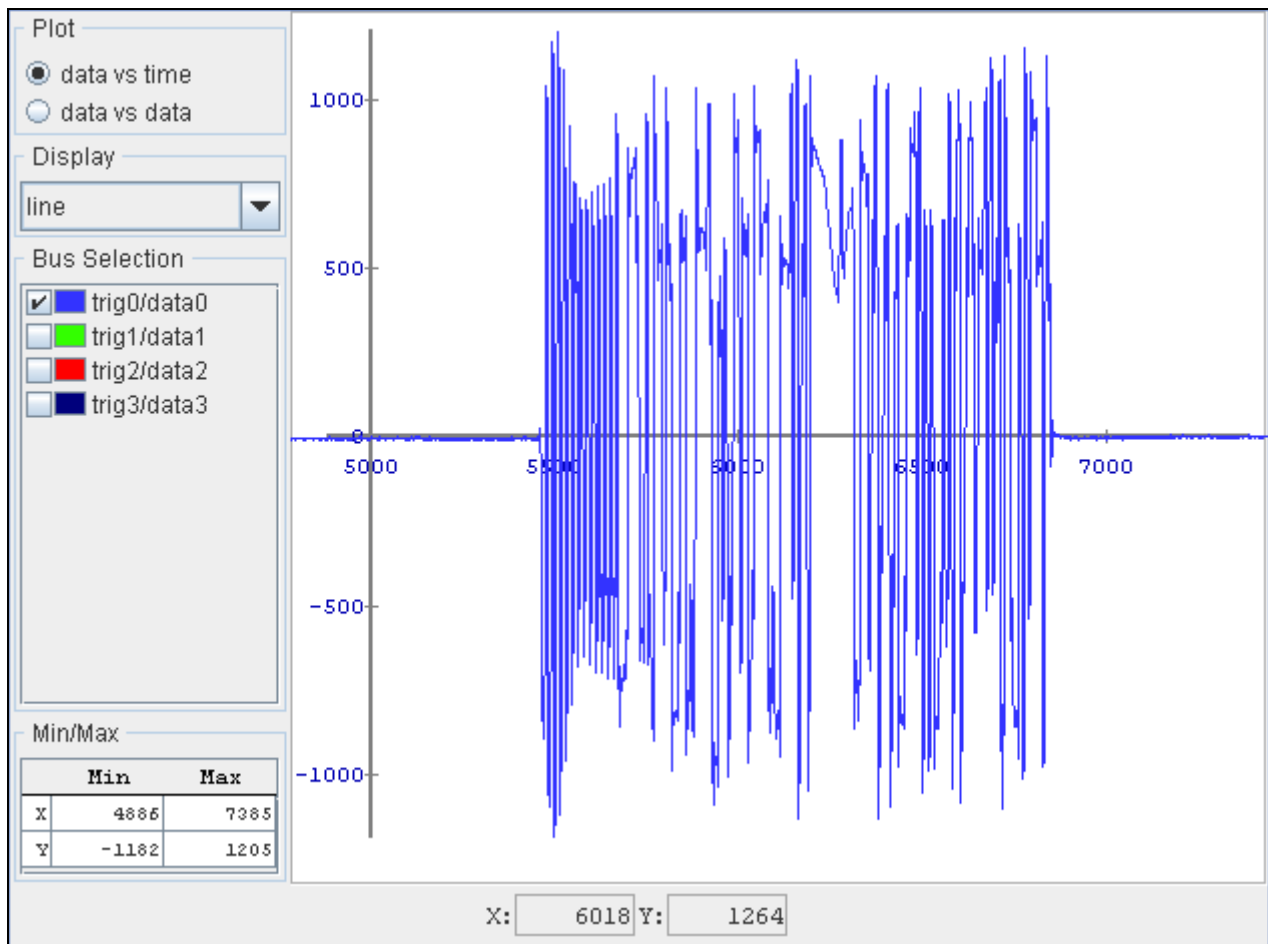
1. To verify the received signal, you will need to open **ChipScope Pro Analyzer**. Be sure that the JTAG cable is connected to the FPGA board properly (or the 6 pin output of the Chilipepper).
2. Once the program opens, click the  (open cable) button to open your JTAG connection to the board. If your jumpers are configured correctly, you should see the following devices on the cable.



3. Select ok to get to the Analyzer main screen. Open the file menu and select **Import**.
4. Click **Select New File**, and browse to the location of your ChipScope **CDC file**, which is located in the Sysgen/netlist folder of your project directory. This file was created for you when you generated your PCores from your Simulink Model design. It tells the ChipScope program how to interpret the data it is receiving from the JTAG port.
5. Next double click on the **bus plot** option in the New Project menu in the top left hand side of the screen. This will open a window which allows you to view a signal **value vs. time** plot of your waveforms.
6. Under Data Port in the Signals Dev menu on the left side of the screen, right click on each of the ports, and change their **bus radix** to **signed decimal**. Click OK to accept the default decimal values.
7. On the Bus Plot screen, you can change the color of each of the signals to get a better view of each individual signal. Click the **check box** next to any of the signals you wish to see on the plot.

## Creating Wireless Transceivers Using MATLAB to HDL translation and Toyon Chilipepper

8. To correctly view the received signal in ChipScope, you must catch the signal in the window which is currently being viewed in ChipScope. This is much easier to do if you flip the switch on the Lab 3 demo to allow for continuous packet transmission.
9. Click the **play button** in the top menu bar until you get a full display of the signal. Additionally you can set up triggering options for periodic or continuous playback of the received signal. Your received signal should look similar to Figure 5-1 below.



5-1: real part of the received QPSK waveform in ChipScope Pro

This is only the real part of the received signal, and is before any of the frequency offset correction and filtering. To see the signal post processing, change the bus plot view to see the data1 and data2 values. Be sure to set the correct bus radix for your signal. Your i and q signals post processing should resemble Figure 5-2 below.



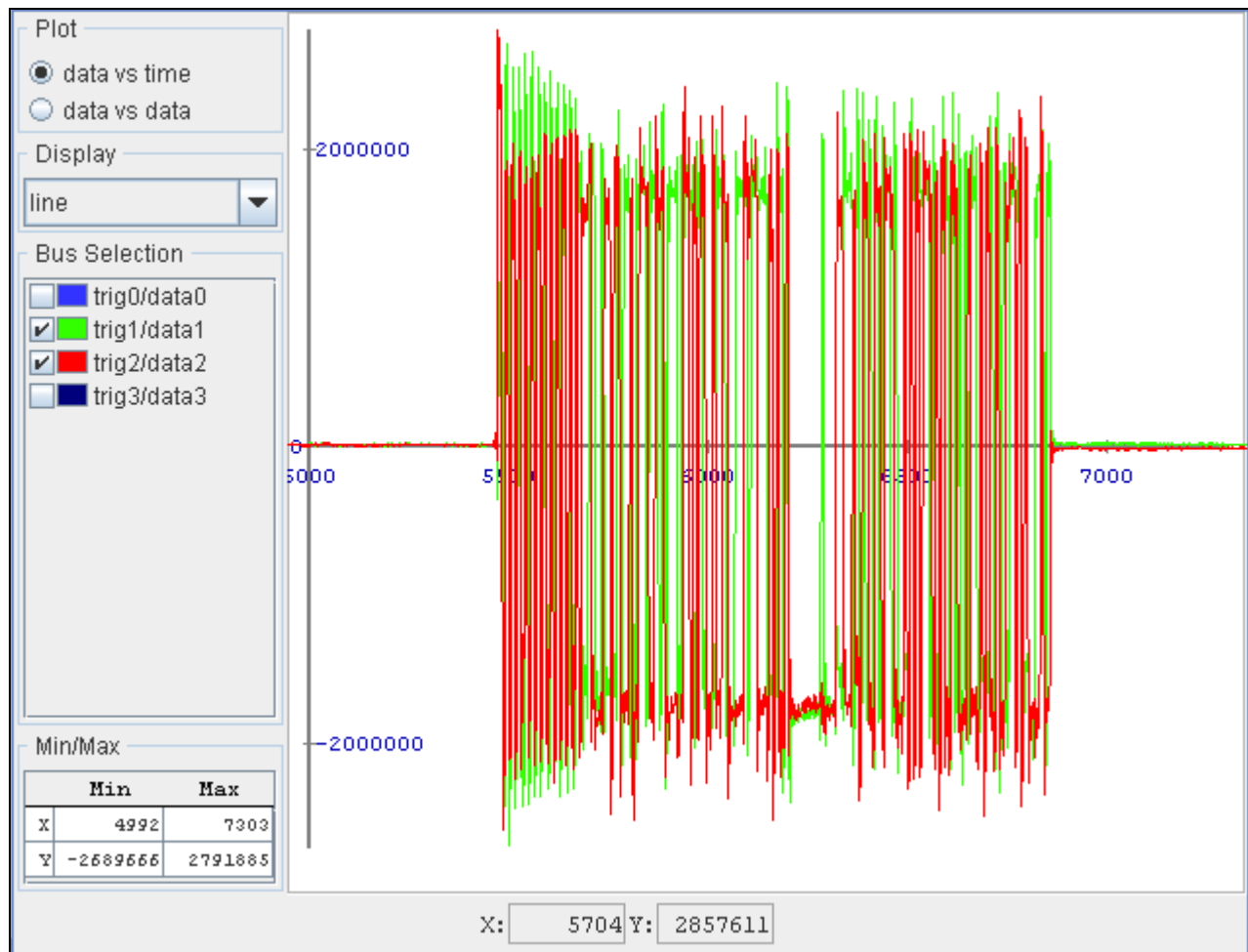


Figure 5-2: Plot of the i and q components of the received QPSK signal post F.O.C. and filtering

## 5.2 Exporting into MATLAB

Now that you have verified the received signal, you can get a pretty good idea of what your QPSK waveform looks like in the time domain. However, ChipScope allows you to export the data received directly into MATLAB for further analysis.

1. It will be helpful later in your MATLAB code if you rename your **Data Port variables**. Right click on the Ports, and **change the names** to something more descriptive, such as `real_out`, `rx_i` and `rx_q` respectively. If needed, you can use the Simulink model to find which signal each port has.

## Creating Wireless Transceivers Using MATLAB to HDL translation and Toyon Chilipepper

2. Open the file menu and select **Export**.
3. Click the **ASCII** radio box, select **Bus Plot Buses** under Signals to export, and then click **export**.
4. It is recommended that you save this file into the project directory with your MATLAB files. Call it something descriptive such as **rx.prn**.

### 5.3 MATLAB Analysis

The last step is to verify the correctness of the rx pcore by plotting the resultant data received from the core. The rx module designed earlier in MATLAB returns the i and q channels of a QPSK waveform which has gone through Frequency offset estimation via a costas loop. Therefore, we would expect that a scatter plot of the result should yield a constellation plot that has a majority of its signal data at the correct pre-specified symbol locations.

1. Verify this plot by running the following MATLAB script. You may have to change the load values depending on which variables you exported from ChipScope.

```
fid = fopen('rx.prn');
M = textscan(fid, '%d %d %d %d %d %d', 'Headerlines', 1);
fclose(fid);
i_in = double(cell2mat(M(3)));
i_out = double(M{4});
q_out = double(M{5});
offset = double(M{6});

figure(1)
scatter(i_out, q_out)
title('Scatter Plot of rx with FOC');

figure(2)
subplot(1,2,1)
plot(i_in);
title('OTA Receive Signal (real part)');
subplot(1,2,2)
plot(i_out);
title('Signal Post FOC (real part)');

figure(3)
plot(offset);
title('Phase Estimate');
```

Your plot should look similar to Figures 5-3 below.

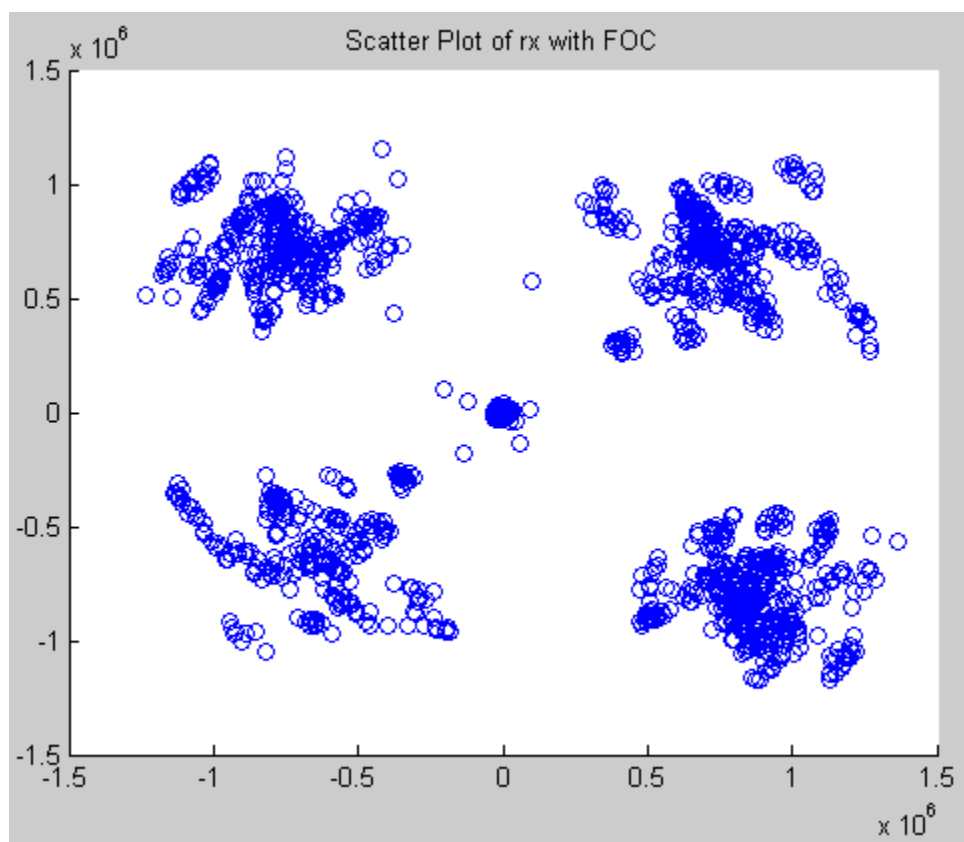


Figure 5-2: Scatter Plot of the transmitted and received QPSK waveform post Frequency Offset Correction.

## Appendix A MATLAB Frequency Offset

MATLAB script qpsk\_rx\_foc.m

```

%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%
% QPSK demonstration packet-based transceiver for Chilipepper
% Toyon Research Corp.
% http://www.toyon.com/chilipepper.php
% Created 10/17/2012
% embedded@toyon.com
%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%
% Demonstration of a Costas Loop. Refer to:
% Telecommunications Breakdown: Concepts of Communication Transmitted via
% Software-Defined Radio C. Richard Johnson
% We employ a hard-decision feedback in order to get rid of the loop
% filters.
%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%
%#codegen
function [z_i_out, z_q_out, fe] = qpsk_rx_foc(y_i, y_q, mu_in)

    persistent phi

    lSin = SIN;
    lCos = COS;

    if isempty(phi)
        phi = 0;
    end

    mu = mu_in/2^12;

    % create the VCO signal
    if phi >= 1
        phi = phi - 1;
    end
    if phi < 0
        phi = phi + 1;
    end

    phi12 = round(phi*2^12)+1;
    if phi12 >= 2^12
        phi12 = 1;
    end
    if phi12 < 0
        phi12 = 0;
    end
end

```

## Creating Wireless Transceivers Using MATLAB to HDL translation and Toyon Chilipepper

```
f_i = lCos(phi12+1);
f_q = lSin(phi12+1);
ti1 = y_i*f_i;
ti2 = y_q*f_q;
tq1 = y_q*f_i;
tq2 = -y_i*f_q;
z_i = ti1 + ti2;
z_q = tq1 + tq2;

% generate the error term to drive VCO generateion
if z_q < 0
    tf = -z_i;
else
    tf = z_i;
end
if z_i < 0
    bf = -z_q;
else
    bf = z_q;
end
% using sign of error in order to make it gain invariant
time_diff = tf-bf;
if time_diff < 0
    e = -1;
else
    e = 1;
end

c = mu*e;
phiNew = phi - c;
phi = phiNew;

fe = phiNew;

z_i_out = z_i;
z_q_out = z_q;
end
```

## Appendix B MATLAB Test Bench

MATLAB function qpsk\_tb.m

```

%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%
% Model/simulation parameters
%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%
sim=1;
OS_RATE = 8;
SNR = 1;
fc = 10e3/20e6; % sample rate is 20 MHz, top is 10 kHz offset
muFOC = floor(.01*2^12)/2^12;
%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%

%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%
% Initialize LUTs
%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%
make_src_lut;
make_train_lut;
make_trig_lut;
%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%
if (sim)
    %%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%
    % Emulate microprocessor packet creation
    %%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%
    % data payload creation
    messageASCII = 'hello world!';
    message = double(unicode2native(messageASCII));
    % add on length of message to the front with four bytes
    msgLength = length(message);
    messageWithNumBytes = [ ...
        mod(msgLength,2^8) ...
        mod(floor(msgLength/2^8),2^8) ...
        mod(floor(msgLength/2^16),2^8) ...
        1 ... % message ID
        message];
    % add two bytes at the end, which is a CRC
    messageWithCRC = CreateAppend16BitCRC(messageWithNumBytes);
    m1 = length(messageWithCRC);
    %%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%

    %%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%
    % FPGA radio transmit core
    %%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%
    data_in = 0;
    empty_in = 1;
    tx_en_in = 0;
    numBytesFromFifo = 0;
    num_samp = m1*8*2*2*3;
    x = zeros(1,num_samp);
    CORE_LATENCY = 4;
    data_buf = zeros(1,CORE_LATENCY);

```

## Creating Wireless Transceivers Using MATLAB to HDL translation and Toyon Chilipepper

```

empty_buf = ones(1,CORE_LATENCY);
clear_buf = zeros(1,CORE_LATENCY);
tx_en_buf = zeros(1,CORE_LATENCY);
for i1 = 1:num_samp
    % first thing the processor does is clear the internal tx fifo
    if i1 == 1
        clear_fifo_in = 1;
    else
        clear_fifo_in = 0;
    end

    if i1 == 5 % wait a little bit then begin to load the fifo
        empty_in = 0;
        numBytesFromFifo = 0;
    end

    data_buf = [data_buf(2:end) data_in];
    empty_buf = [empty_buf(2:end) empty_in];
    clear_buf = [clear_buf(2:end) clear_fifo_in];
    tx_en_buf = [tx_en_buf(2:end) tx_en_in];
    [i_out, q_out, re_byte_out, tx_done_out, d1, d2, d3] = ...
        qpsk_tx(data_buf(1),empty_buf(1),clear_buf(1),tx_en_buf(1));
    x_out = complex(i_out,q_out)/2^11;
    x(i1) = x_out;

    %% Emulate read FIFO AXI interface
    if re_byte_out == 1 && numBytesFromFifo < length(messageWithCRC)
        data_in = messageWithCRC(numBytesFromFifo+1);
        numBytesFromFifo = numBytesFromFifo + 1;
    end
    % processor loaded all bytes into FIFO so begin transmitting
    if numBytesFromFifo == length(messageWithCRC)
        empty_in = 1;
        tx_en_in = 1;
    end
end

index = find(abs(x) > sum(SRRC))+24*8; % constant is pad bits
offset = index(1)+6+length(TB_i)*OS_RATE;
idx = offset:8:(offset+8*ml*4-1);
y = x(idx); % four symbos per byte of data
sc = zeros(1,18*8);
sc(1:2:end) = real(y);
sc(2:2:end) = imag(y);
sh = sign(sc);
sb = (sh+1)/2;
d = zeros(1,ml);
for i1 = 1:ml
    si = sb(1+(i1-1)*8:i1*8);
    d(i1) = bi2de(si);
end
%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%

```

## Creating Wireless Transceivers Using MATLAB to HDL translation and Toyon Chilipepper

```

%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%
% Emulate channel
%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%
% pad on either side with zeros
p = complex(zeros(1,100),zeros(1,100));
xp = [p x p]; % pad

% Apply frequency offset and receive/over-the-air AWGN
y = xp.*exp(1i*2*pi*fc*(0:length(xp)-1));
rC = y/max(abs(y)).*1*2^11; % this controls receive gain
r = awgn(rC,SNR,0,1);
r1 = rC;
%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%
end
%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%
% Load Chipscope samples
%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%
if (~sim)
    fid = fopen('rx.prn');
    M = textscan(fid, '%d %d %d %d', 'Headerlines', 1);
    fclose(fid);
    is = double(M{3});
    qs = double(M{4});
    r = complex(is,qs);
end
%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%
% Main receiver core
%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%
r_out = zeros(1,length(r));
s_f = zeros(1,length(r));
s_c = zeros(1,length(r));
f_est = zeros(1,length(r));
for i1 = 1:length(r)+200
    if i1 > length(r)
        r_in = 0;
    else
        r_in = r(i1);
    end
    i_in = round(real(r_in));
    q_in = round(imag(r_in));
    [r_out(i1), s_f(i1), s_c(i1), f_est(i1)] = ...
        qpsk_rx(i_in, q_in, floor(muFOC*2^12));
end
%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%

```



## Creating Wireless Transceivers Using MATLAB to HDL translation and Toyon Chilipepper

```
figure(2)
subplot(2,2,1)
scatter(real(r),imag(r))
title('Pre FOC Signal');
subplot(2,2,3)
plot(real(r_out));
title('Pre FOC Signal (real part)');
subplot(2,2,2)
scatter(real(s_c1),imag(s_c1))
title('Post FOC Signal');
subplot(2,2,4)
plot(real(s_c1));
title('Post FOC Signal(real part)');

figure(3)
plot(f_est);
title('Phase Estimate');
```