

# Laboratory Exercise #9

## Counters, Clock Dividers, and Debounce Circuits

ECEN 248: Introduction to Digital Design  
Department of Electrical and Computer Engineering  
Texas A&M University



## 1 Introduction

The primary objective of this week's lab assignment is to reinforce your knowledge of sequential circuits by introducing an important synchronous sequential circuit, the binary counter. The lab will guide you through the design of a binary up-counter using familiar combinational components as well as sequential components discussed in the previous lab. Furthermore, the lab will demonstrate two important use cases for binary counters, namely clock frequency division and I/O debouncing. Due to the practical nature of this lab assignment, you will be testing all of your designs on the FPGA board.

## 2 Background

The following subsection will review the theory necessary for completion of this lab assignment. The pre-lab will test your knowledge of these concepts.

### 2.1 Counters

With the components you have been introduced to at this point, you can construct a wide variety of useful synchronous circuits. One particularly important circuit we will discuss in this lab is the binary counter. A simple binary up-counter constructed from half-adders and flip-flops is depicted in Figure 1. It is interesting to note that the half-adders are chained together to create a ripple-carry adder that increments the binary value stored in the flip-flops by 1 on every rising-edge of the clock if the enable ( $En$ ) is high. If  $En$  is low, the binary value stored in the counter does not change. Notice the *Reset* signal, which connects to each flip-flop in Figure 1. We have not discussed *Reset* signals yet; however, the idea is quite simple. For certain synchronous circuits, the beginning state is important. *Reset* provides a mechanism to initialize the state of a synchronous circuit. When the *Reset* signal is asserted, the flip-flops will *reset* to a known state. Typically, the *reset* state will be '0'; however, it is not uncommon to *reset* a flip-flop to '1'. Just like the clock signal, the *Reset* signal is a global net distributed to all synchronous components.

Many use cases exist for the binary counter such as input de-bouncing and clock frequency division. The next subsections will expound on those topics, while the experiments in this lab will provide you with experience working with these sorts of circuits.

### 2.2 Clock Division

Clock frequency division refers to the process of generating a slower clock from a faster clock. For example, supposed we have two synchronous circuits driven by separate clocks,  $Clk_1$  and  $Clk_2$ . If the synchronous circuit driven by  $Clk_2$  is slower than the one driven by  $Clk_1$ , then we could use a clock divider circuit to

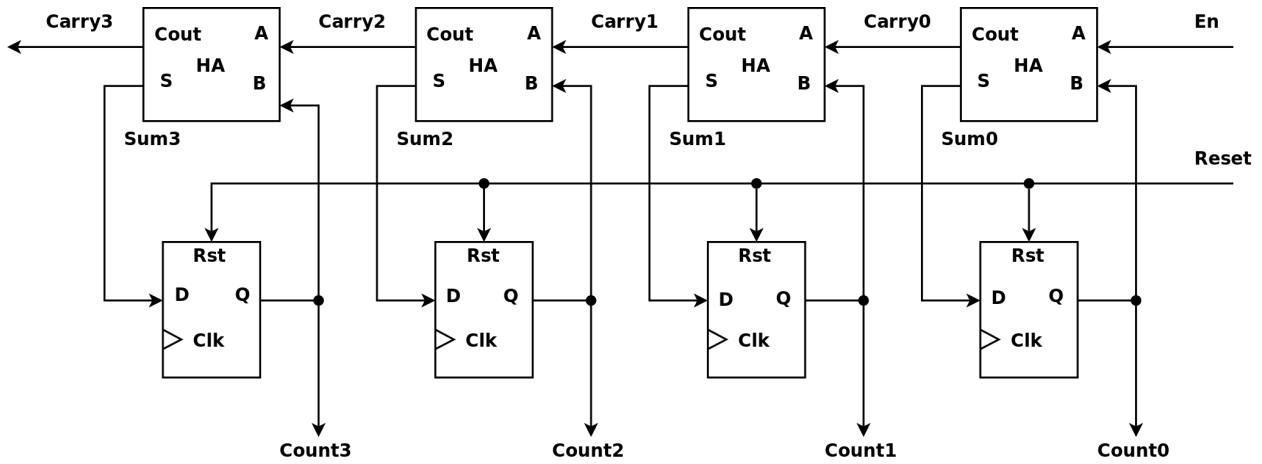


Figure 1: 4-bit Binary Up-Counter

generate the slow clock with the fast clock. Then both circuits can essentially be driven off of the same clock, eliminating the need for two different oscillators. Figure 2 illustrates this concept. If the frequency of the fast clock,  $f_1$ , must be divided by a power-of-2 in order to generate the slow clock,  $f_2$ , then a simple binary counter can be used. If  $f_2 = \frac{f_1}{2^n}$ , then an  $n$ -bit binary counter will be sufficient. We can convince ourselves that this is true by observing the fact that when counting in binary, the least significant bit toggles twice as fast as the next significant bit. The same is true of the next significant bit and so forth.

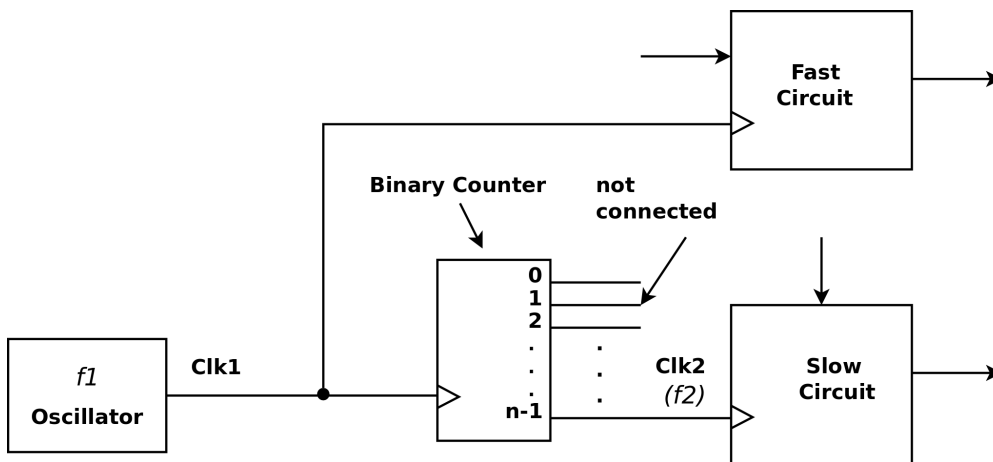


Figure 2: Clock Frequency Division Using A Binary Counter

### 2.3 Signal Debouncing

Up to this point, we have assumed that switches and push-buttons, used for receiving input from the user, are ideal in terms of the output signals that they produce. Unfortunately, this is not the case in the real world because these devices rely on less than ideal electromechanical contacts to generate electrical signals from mechanical events. Electrical chatter as seen in Figure 3 is generated every time a switch is flipped or a button is pressed. This chatter (a.k.a switch bounce) consists of several short duration pulses, which appear as a bouncing signal in the interval immediately after the electromechanical contacts are brought together.

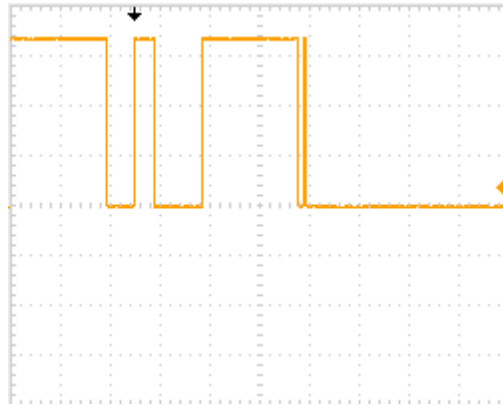


Figure 3: Electrical Chatter

Oftentimes synchronous circuits can be sensitive to this electrical chatter, in which case some additional circuitry must be added to the design to eliminate the bounce in the signal. Many techniques exist for debouncing a signal, but for certain situations, a simple counter circuit will suffice.

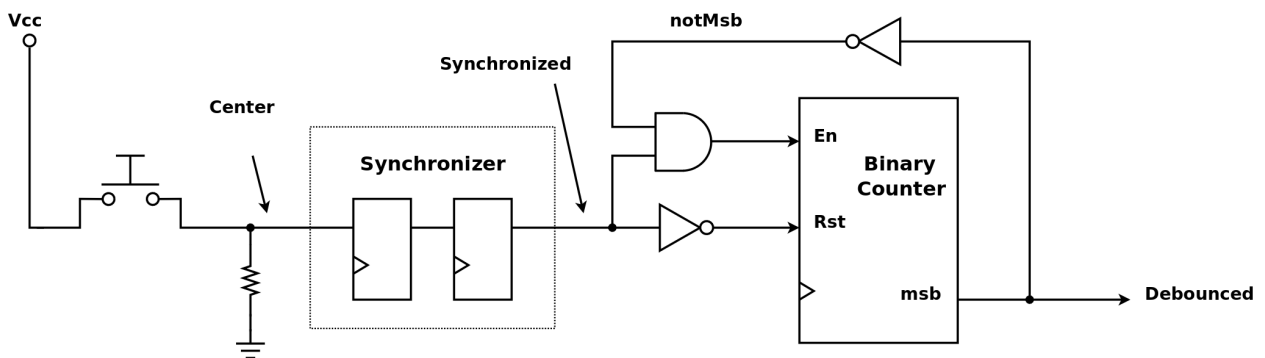


Figure 4: Simple Debounce Circuit



### 3 Pre-lab

The pre-lab questions below will test you on your knowledge of the material presented above. Please answer the following questions completely:

1. Supposed we want to build a 32-bit counter using the circuit in Figure 1. If the gates used to construct the half-adders in the circuit are assumed to have a 2ns delay each and the flip-flop overhead is assumed to be negligible, what is the maximum clock frequency we could use to drive our counter? Given the clock frequency you just calculated, how long would it take this counter to roll over (i.e. return to 0). Please show your calculations.
2. Consider the use of a counter to divide an incoming clock. If our incoming clock signal is 32.768 kHz and we need a 64 Hz signal, how many bits would our counter need to have (i.e. what is  $n$ ) to divide the incoming clock correctly.
3. If the Seconds per Division setting for Figure 3 was set to 1 ms/div, what do you think would be a good bit width for the counter in Figure 4, assuming a 50 MHz clock signal was driving the counter? Explain your answer.

### 4 Lab Procedure

The following lab exercises will guide you through the implementation of the circuits discussed in the background section above. For the first two experiments, we will do things differently by introducing the behavioral description before using structural Verilog. The reasons for this should become clear shortly.

#### 4.1 Experiment 1

The objective of the first experiment is two-fold. First, we would like to give you the opportunity to describe a synchronous sequential circuit in Verilog and actually load it onto the FPGA board. Second, we would like to provide you with hands on experience dealing with counters and clock frequency division. These are important concepts that you will need to understand when you go to design more complex digital circuits.

**Note:** You will need to pair up with someone in the lab for this experiment because we do not have enough logic analyzers for every workstation. Please take a moment to find a friend in the lab who is willing to work with you. If you do not have a friend, ask the TA to find you one.

1. Invoke ISE and create a new project called lab9.
2. The following steps will walk you through the implementation of a simple counter circuit that we will use to divide the rate of our system clock.
  - (a) Type the Verilog code below into a source file and save it as “clock\_divider.v”:

```

1  'timescale 1ns / 1ps
   'default_nettype none
3
   /*This simple module will demonstrate the concept of clock frequency*
5  *division using a simple counter. We will use behavioral Verilog *
   *for our circuit description. */
7
module clock_divider(ClkOut, ClkIn);
9
   /*output port needs to be a reg because we will drive it with *
11  *a behavioral statement */
   output wire [3:0] ClkOut;
13  input wire ClkIn; //wires can drive regs

15  /*-this is a keyword we have not seen yet!*
   *-as the name implies, it is a parameter *
17  * that can be changed at compile time... */
   parameter n = 5;//make count 5-bits for now...
19
   reg [n-1:0] Count; //count bit width is based on n! how cool is that!
21
   /*simple behavioral construct to describe a counter... *
23  *Are you ready for this?? */
   always@(posedge ClkIn) //should look familiar...
25     Count <= Count + 1; //yea that's it

27     /*now we need to wire up our ClkOut which is a 4-bit wire*/
   /*Wire up to most-significant bits*/
29     assign ClkOut[3:0] = Count[n-1:n-4];

31 endmodule //sorry if this was not more exciting

```

- (b) Now add the above source file and the “clock\_divider.ucf” found in the course directory to your ISE project.
  - (c) Synthesize and Implement the design as outlined in the previous lab.
3. The output of the counter changes far too rapidly for us to view with LEDs so we will need to employ the logic analyzer on our work bench. Unlike the oscilloscope, the logic analyzer on your workbench can probe up to 32 digital signals at once. It also has the capability of grouping signals into buses, which can be viewed in decimal or hexadecimal format. The waveforms that you will see on the logic analyzer today have many similarities to the waveforms you have previously seen in simulation. The difference, however, is that the signals you see on the logic analyzer are being created with actual hardware! The following steps will direct you through the process of connecting the logic analyzer to the Spartan 3E board as well as the initial setup of the logic analyzer.
- (a) Before connecting the logic analyzer to the output of our counter, ensure the FPGA board is turned off. Now, locate the J1 connector towards the bottom-right of the FPGA board. Once you

have found the J1 connector, open the UCF file and compare it to the diagram in Figure 6. You should now be able to locate the *ClkOut* signals of your clock divider.

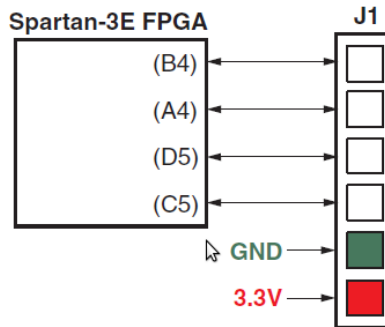


Figure 6: J1 Pinout for the Spartan 3E Board courtesy of Xilinx®

- (b) Two groups of wires (a.k.a channels) are plugged into the back of the logic analyzer. Locate the cable labeled “Port 1.” Now, examine the end of the cable, taking note of the individual channel labels. On the far right of the cable end, you should see a “GND” wire. Use a short jumper wire to connect the probe “GND” to the “GND” on J1. Now connect channels 0 through 3 to ClkOut[0] through ClkOut[3] on J1 in a similar manner. Have the TA inspect your setup before moving on.
  - (c) Turn the logic analyzer on by flipping the power switch towards the bottom of the machine. See Figure 7.
  - (d) Once the Logic Analyzer boots up, press the **Format** menu key to begin setup and use the arrow keys to highlight the “Bus1” label.
  - (e) Press the **Select** key and use the arrow keys to highlight “Modify Label”. Hit **Select** again and use the keypad to change the label name to “COUNT”. Press **Done** to move on.
  - (f) Use the right arrow key to move the cursor all the way to the right of the “COUNT” label in the format screen. Then press **Select** and the appropriate combination of arrow keys to disable bits 15 down to 4 and enable bits 3 down to 0. A ‘●’ indicates the channel is disabled, while an ‘\*’ indicates the channel is enabled.
  - (g) Turn the FPGA board on and program the FPGA with the clock divider circuit. Press **Run** on the logic analyzer once the FPGA has been programmed successfully. If everything is working properly, you should see four separate clock signals on the logic analyzer.
4. To gain a little more experience working with the logic analyzer, we will first perform some simple time measurements, and then we will group our *ClkOut* signals together to view the counter in operation.



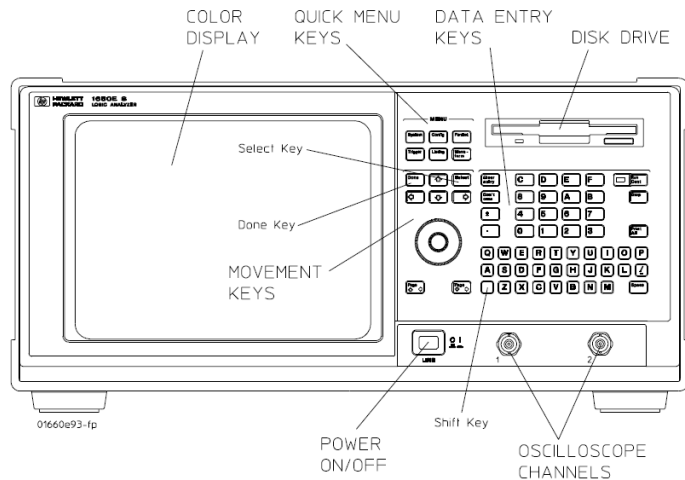


Figure 7: Agilent 1673G Logic Analyzer Front Panel

- With the waveform on the logic analyzer, use the arrow keys to highlight “Markers Off” and then press **Select**. With the arrow keys, highlight “Time” and hit **Select** again.
- A green and a yellow time marker will appear superimposed on the waveform. Use the arrow keys to highlight the box with a green perimeter and rotate the dial to move the green marker to the left or right. Follow the same procedure to move the yellow marker. Displayed in the green box is the measurement of time from the trigger point (red marker) to the green marker. You can calculate the time difference between the green and yellow markers by computing the difference between the two measured times. Measure and record the period of each clock signal using the green and yellow markers. Based on your measurements, what frequency do you think the input clock is running at?
- While viewing the waveform on the logic analyzer, use the arrow keys to highlight the “COUNT” labels on the far left. Rotate the knob to scroll to the bottom of the list of labels and hit **Select** three times. You should see a menu with “Sequential” at the top followed by “Bus”. Highlight “Bus” and hit **Select**.
- A new label should appear at the bottom call “COUNT all”. Use the arrow keys to highlight the “Sec/Div” (seconds per division) and rotate the dial to zoom in or out. Zoom in until you are able to see the hexadecimal values of “COUNT” clearly. You may also highlight “Delay” above the waveform and use the dial to move the waveform to the left or right. Verify that the bus, “COUNT”, is indeed counting upwards.

## 4.2 Experiment 2

For Experiment 2, we will use our Verilog skills to design the up-counter discussed in the background section of this manual. We will drive this up-counter with the clock divider created in the previous experiment and wire the output of the up-counter to the LEDs on the FPGA board. For the *En* signal, we will utilize the North push-button. The steps below will guide you through the experiment.

1. Design the up-counter illustrated in Figure 1.

- (a) Create a source file called “half\_adder.v” and describe the half adder circuit found in Lab 3 using *dataflow* Verilog. You do not need to include delays in your Verilog code. For consistency, use the following module interface:

```
module half_adder(S, Cout, A, B);
```

**Hint:** If you are unsure where to get started, take a look at the source file for the full-adder you designed on Lab 5!

- (b) Create a source file called “up\_counter.v” and construct the circuit found in Figure 1 using Verilog. The code snippet below should help you get started.

```
1 timescale 1ns / 1ps //specify 1ns for each delay unit
  default_nettype none
3
  /*This module describes a simple 4-bit up-counter using *
5  *half-adder modules built in the previous step          */

7 module up_counter(Count, Carry3, En, Clk, Rst);

9     /*Count output needs to be a reg */
     [3:0] Count;
11    output wire Carry3;
     /*inputs are wires*/
13    input wire En, Clk, Rst;

15    /*intermediate nets*/
     wire [3:0] Carry, Sum;
17
     /*instantiate and wire up half-adders here*/
19

21

23
     /*wire up carry3*/
25    assign Carry3 = Carry[3];

27
     /*Describe positive edge triggered flip-flops for count*/
     /*Including “posedge Rst” in the sensitivity list      */
```

```

29     *implies an asynchronous reset!                               */
always@(posedge Clk or posedge Rst)
31     if(Rst)//if Rst == 1'b1
        Count <= 0; //reset count
33     else //otherwise , latch sum
        Count <= Sum;
35
endmodule

```

2. Now simulate the up-counter you just designed and observe the output waveform to ensure your counter is working properly.
  - (a) Copy the “up\_counter\_tb.v” test bench from the course directory and simulate it using ISE.
  - (b) Open up the test bench file and try to understand what is going on. You should see that the test bench produces a *Clk* signal. What is the frequency of that signal?
  - (c) You should also see that the test bench holds the counter in *reset* for a specific interval of time. How long is that interval?
  - (d) After reset is de-asserted, the test bench holds the enable LOW for some amount of time before allowing the counter to run. How long is this time period?
  - (e) From within the waveform window, right-click on “Count” and select *Radix* → *Hexadecimal* to change the display format of the “Count” bus to hexadecimal. See Figure 8.

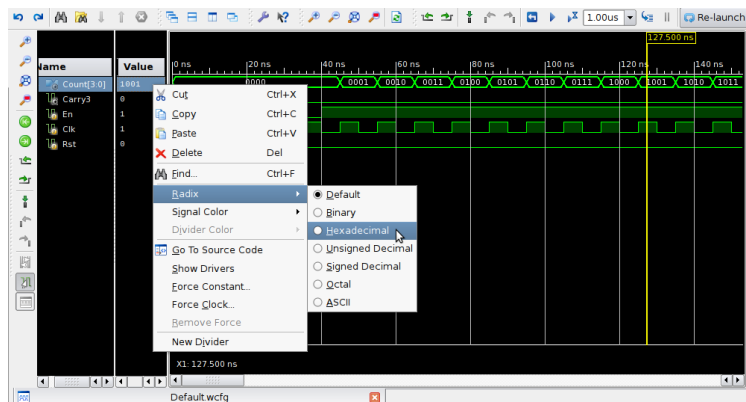


Figure 8: Change Radix to Hexadecimal

- (f) Finally, you should notice that the counter will roll over (i.e. return to 0) after reaching a maximum value. What is this maximum count value and what signal in the waveform could we use to know exactly when the counter is going to roll over?

3. Create a top-level Verilog module that can be used to drive our counter on the FPGA and then load the final design onto the Spartan 3E board.
- Open the “clock\_divider.v” source file and modify the module such that the *Count* signal is 26 bits wide. This will allow us to divide the clock by  $2^{26} = 67,108,864$ . If we use a 50MHz clock to drive our frequency divider, what rate will the most significant bit of the divider oscillate at?
  - Create a source file called “top\_level.v” and use the code snippet below to describe the circuit in Figure 9.

```

1  timescale 1ns / 1ps //specify 1ns for each delay unit
2  default_nettype none

4  /*This is the top-level module which wires all
   *of our synchronous components together. This module *
6  *does NOT include synchronizers for the inputs (we *
   *will discuss them shortly) so just don't use this in*
8  *real application! */

10 module top_level(LEDs, SWs, North, South, FastClk);

12     /* all ports will be wires because we will use
        structural Verilog to wire everything up*/
14
16     /*intermediate nets*/
17     wire [3:0] Clocks;
18     reg SlowClk; //will use an always block for mux

20     /*behavioral description of a mux which
        selects between the four available clock signals*/
21     always@(*) //combinatorial logic
22         case(SWs) //SWs is a 2-bit bus
23             2'b00: SlowClk = Clocks[0]; //use blocking statement for
24                                     //combinatorial logic
25
26             //finish this up
27
28         endcase

30     /*instantiate your up_counter here*/
31     /*Hint: if you want to wire a port to just the first 4 *
32     *bits of a bus, you can do something like this: LEDs[3:0]*/

34
35     /*instantiate the clock divider. I will do that for you...*/
36     clock_divider clk_div0(
37         .ClkOut(Clocks),
38         .ClkIn(FastClk)
39     );

```

40

endmodule

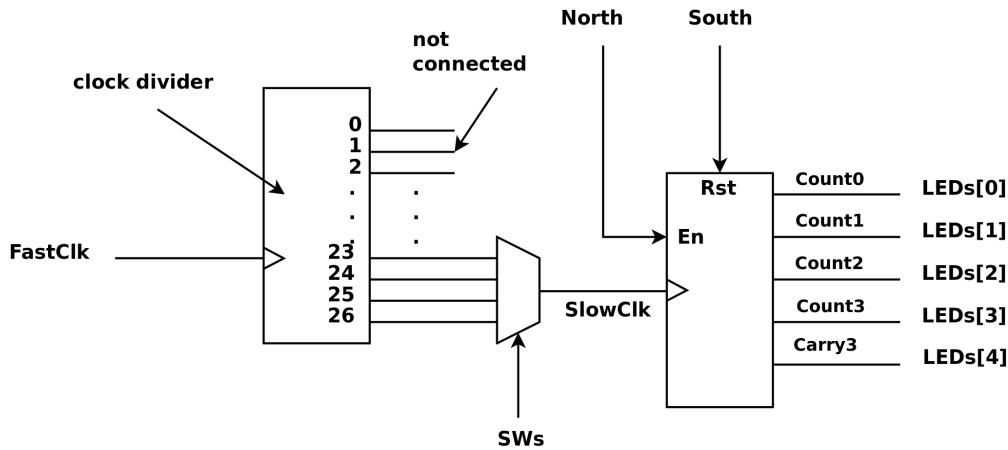


Figure 9: Top-Level Diagram

- (c) Create a file called “top\_level.ucf” and save it in your lab9 directory. Type the UCF starter code shown below. Unfortunately, portions of the UCF are missing. Using the UCFs from the previous labs and Figure 9 as a guide, fix this UCF. You may also find the Spartan 3E User Guide from Xilinx helpful.

```

1 #Switches
  NET "SWs[0]" LOC = "L13" | IOSTANDARD = LVTTTL; #SW0
3 # fill in for SWs[1], connect to SW1

5 #Push-buttons
  NET "North" LOC = "V4" | IOSTANDARD = LVTTTL | PULLDOWN; #North
7 #fill in for South

9 # LEDs
  NET "LEDs[0]" LOC = "F12" | IOSTANDARD = LVTTTL; #LD0
11
  #Something is missing here...
13
  NET "LEDs[4]" LOC = "C11" | IOSTANDARD = LVTTTL; #LD4
15
  NET "FastClk" LOC = "C9" | IOSTANDARD = LVCMOS33 ;
17 # Define clock period for 50 MHz oscillator
  NET "FastClk" PERIOD = 20.0ns HIGH 40%;

```

- (d) Add the UCF you just created to your ISE project and then Synthesize and Implement the design. Correct any errors and then load the design onto the FPGA board.

**Hint:** Errors generated during the synthesis process will be due to issues introduced in your Verilog code, while errors reported during the Implementation phase will be related to the UCF.

- (e) Before continuing, it is important that we take note of the warnings that have been generated during the Implementation of this design. For demonstration purposes, we are violating “good design practice” for FPGA development by create a clock signal from combinational logic. However, the clock signal that we are creating is *very* slow relative to the speed at which the FPGA fabric is able to operate. Thus, our circuit will function as expected. For faster clocks, we should utilize the built-in clock-dividers available on the FPGA.
- (f) Experiment with the various combinations of SW1 and SW0, while pressing the “North” push-button. Observe the rate at which the LEDs toggle. Make note in your lab write-up how the switches affect the LED outputs.
- (g) Additionally, press the “South” push-button and note the result in your lab write-up. Once the design has been found to work, demonstrate your progress to the TA.

### 4.3 Experiment 3

The purpose of the last experiment is to demonstrate switch bounce and its effects on an example synchronous circuit. Additionally, we will learn how to eliminate switch bounce with the debounce circuit discussed in the background section of this manual. We will begin by employing the oscilloscopes in lab to view switch bounce produced by one of the push buttons. We will then connect that push-button to the input of a simple synchronous circuit (as you may have guess, it will be yet another counter!) without proper debouncing. Finally, we will add signal debouncing and compare the results. The following steps will guide you through the experiment.

1. The first step will be to connect the oscilloscope to the “Center” push-button (the one built into the rotational knob) and observe switch bounce.
  - (a) Add the source file, “switch\_bounce.v”, and the UCF, “switch\_bounce.ucf”, to your ISE project.
  - (b) Set the “switch\_bounce.v” module as the top-level module. Ensure the appropriate UCF is listed under your top-level module and start the Implementation process.
  - (c) While the design is building, open up both “switch\_bounce” files and make a note in your lab write-up what it is these file describe.
  - (d) Connect the Channel 1 scope probe to the J1 connector. Use the UCF and Figure 6 to determine the exact wiring necessary. Have the TA inspect your setup before continuing.
  - (e) Turn the FPGA board on and load the current design onto the FPGA.
  - (f) Turn the oscilloscope on. Once it boots up, set the Volts/Division for channel 1 to 1 volt/div. Also, set the Seconds per Division to 1 ms/div.

- (g) Now, press the “Trigger Menu” button and ensure the following settings are correct:
- Type: Edge
  - Source: Ch1
  - Slope: Rising Edge
  - Mode: Normal
  - Coupling: DC
- (h) Adjust the “Trigger Level” knob until the yellow arrow on the right of the screen is set to about 1.5 volts.
- (i) Press the “Run/Stop” button and the scope. Ensure the scope is in run mode (the screen will display “Read” in yellow at the top of the screen).
- (j) Finally, press the rotational knob and observe the output on the scope. Repeat this step until you observe switch bounce (it may not happen every time). Copy the waveform on the scope into your lab write-up. Demonstrate your progress to the TA.
2. Now that we have viewed switch bounce or electrical chatter on the oscilloscope, it would be beneficial to observe how it might affect an example synchronous circuit. The following steps will help you do just that.
- (a) Figure 10 provides a simple synchronous circuit that will be sensitive to switch bounce. This circuit contains a binary counter with an enable controlled by the output of an edge-detector. The edge-detection circuitry serves two purposes in this circuit. First of all, it synchronizes the asynchronous input from the push-button, *Center*. Second of all, it detects when that signal transitions from LOW to HIGH. Thus, it detects a positive-edge. The output of the edge-detector is a pulse that immediately follows the rising-edge of *Center* and is one clock cycle wide. The designer intended to create a circuit that will increment a counter by one every time the “Center” button on the Spartan 3E board is pressed. Let us see if that is what actually happens.

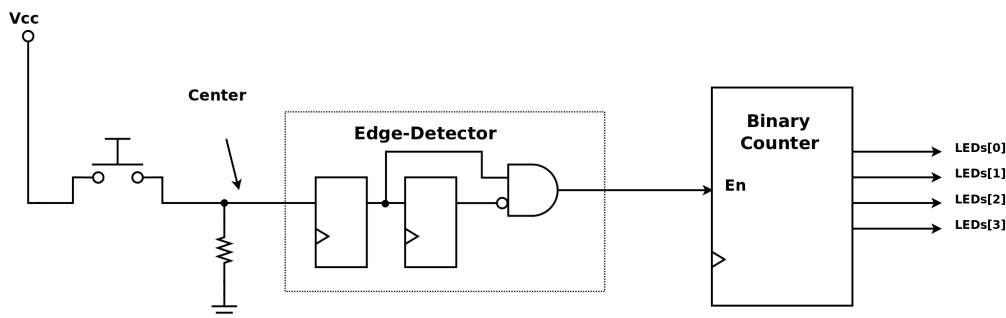


Figure 10: Example Synchronous Circuit without Debounce Circuitry

- (b) The circuit in Figure 10 has been implemented for you in “noDebounce.v”. Copy this file along with the associated UCF, “noDebounce.ucf”, from the course directory to your lab9 directory.
  - (c) Add both files to your ISE project and start the implementation process.
  - (d) While the design is building, open up both files and verify that the Verilog code and the UCF pin assignments match the provided diagram.
  - (e) Once the design has completed the Implementation process load it onto the FPGA.
  - (f) Press the rotational knob (i.e. the “Center” button) and observe the output of the LEDs. Does the design work as intended? Why or why not?
3. Now that we have seen first hand how switch bounce can affect a synchronous circuit, we should look into how to correct for it.
- (a) Add “withDebounce.v” and ”withDebounce.ucf” to your ISE project and set “withDebounce” as the top module. Start the Implementation process.
  - (b) Study the circuit in Figure 4. Now take a moment to examine the uncommented Verilog code in “withDebounce.v”. You should find that this source file contains the circuit in Figure 10 and the circuit in Figure 4 combined into one. Please add comments to the source code and include it in your lab write-up.
  - (c) Explain in your lab write-up the operation of the circuit in described in “withDebounce.v”. A block diagram might help with the explanation but is not required.
  - (d) Once the program file has been created, load it onto the FPGA. Test the circuit with the debounce logic and note the results in your lab write-up.

## 5 Post-lab Deliverables

Please include the following items in your post-lab write-up in addition to the deliverables mentioned in the *Policies and Procedures* document.

1. Include the source code with comments for **all** modules in lab. You do **not** have to include test bench code. Code without comments will not be accepted!
2. Include any UCFs that you wrote or modified.
3. Include screenshots of all waveforms captured during simulation in addition to the test bench console output for each test bench simulation.
4. Answer all questions throughout the lab manual.
5. (**Honors**) For the debounce circuit in the last experiment, what might be the effect of increasing the **parameter**,  $n$ , in the Verilog code? Similarly, what would be the effect of lowering that **parameter**?



## 6 Important Student Feedback

The last part of lab requests your feedback. We are continually trying to improve the laboratory exercises to enhance your learning experience, and we are unable to do so without your feedback. Please include the following post-lab deliverables in your lab write-up.

**Note:** If you have any other comments regarding the lab that you wish to bring to your instructor's attention, please feel free to include them as well.

1. What did you like most about the lab assignment and why? What did you like least about it and why?
2. Were there any section of the lab manual that were unclear? If so, what was unclear? Do you have any suggestions for improving the clarity?
3. What suggestions do you have to improve the overall lab assignment?