Simulation Labs

Simulation Labs

Introduction

This lab is a brief introduction to the capabilities of the Foundation Series simulator. It includes both a graphical interactive simulation interface and a script mode simulator. We will go through both examples briefly.

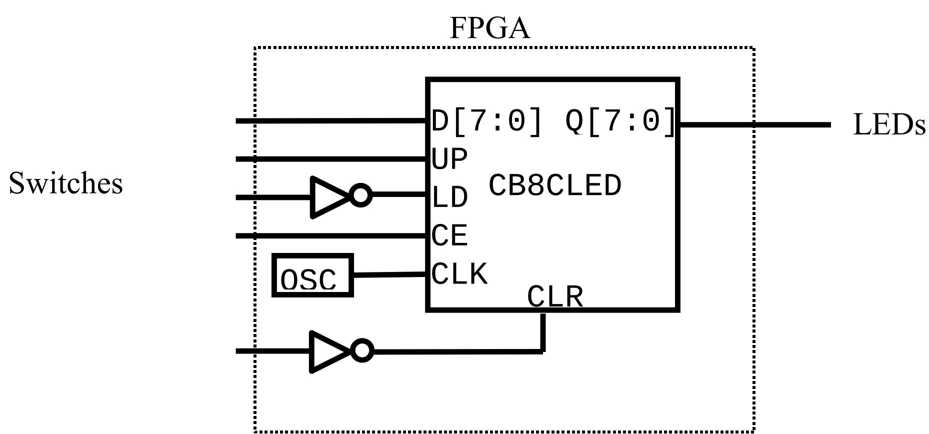
Objective

- Illustrate the use of graphical simulation in conjunction with schematic visualization
- Use the script editor for more advanced simulations.

Procedure

The graphical example is based on the 8-bit counter design, **`COUNT`**, located at **`C:\F15_labs\flow\`**.

This a loadable, bi-directional counter. The load control and data inputs connect to toggle switches on the demo board. The Clock Enable also connects to a toggle switch, while the clock is generated inside the FPGA. After implementation, the outputs would be visible on LEDs on the demo board.



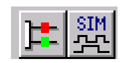
Setting up the Simulator from a Schematic

1) From the Windows Program Manager, <u>click</u> **Start → Programs → Xilinx Foundation Series → Foundation Project Manager**.

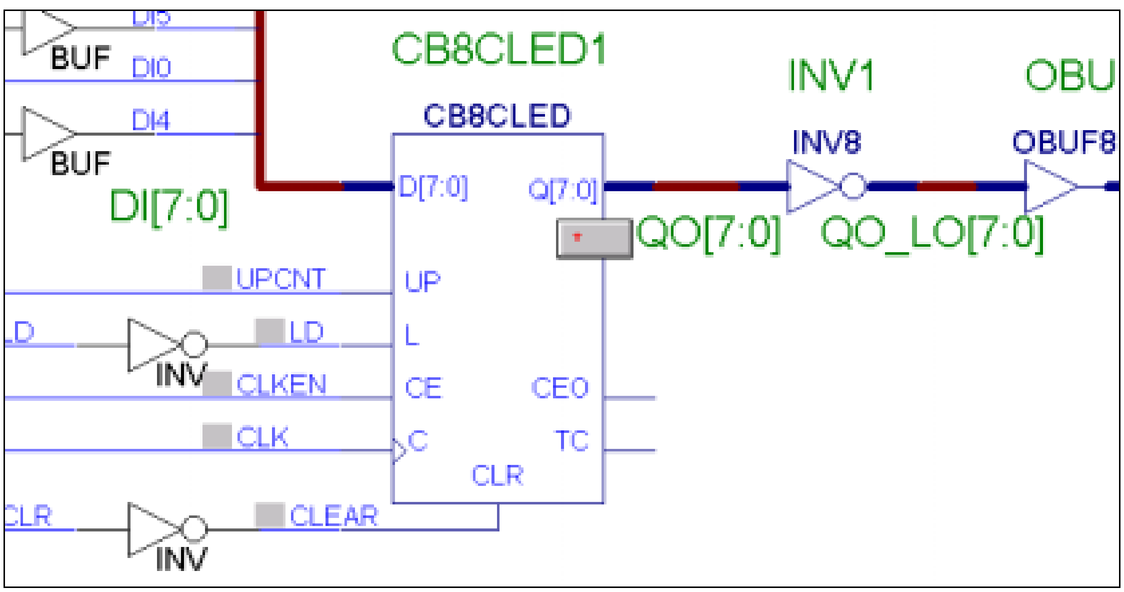
2) To open the **`COUNT`** Project in the **`FLOW`** directory:
   A) **File → Open Project...**
   B) Go to the **`C:\F15_labs\flow`** directory, so that Path says "**`c:\F15_labs\flow`**".
   C) <u>Select</u> the Project **`COUNT`.**
   D) <u>Select</u> **Open**

3) <u>Open</u> the schematic by <u>clicking</u> the **Schematic Editor** icon. Now browse around this design. If needed, take a moment to use the navigation tools and to navigate into and out of schematic blocks.

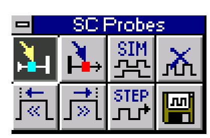
4) <u>Click</u> the **Simulation Toolbox** icon. Your pointer turns into a probe selector. The probe select is the button shown on the left.



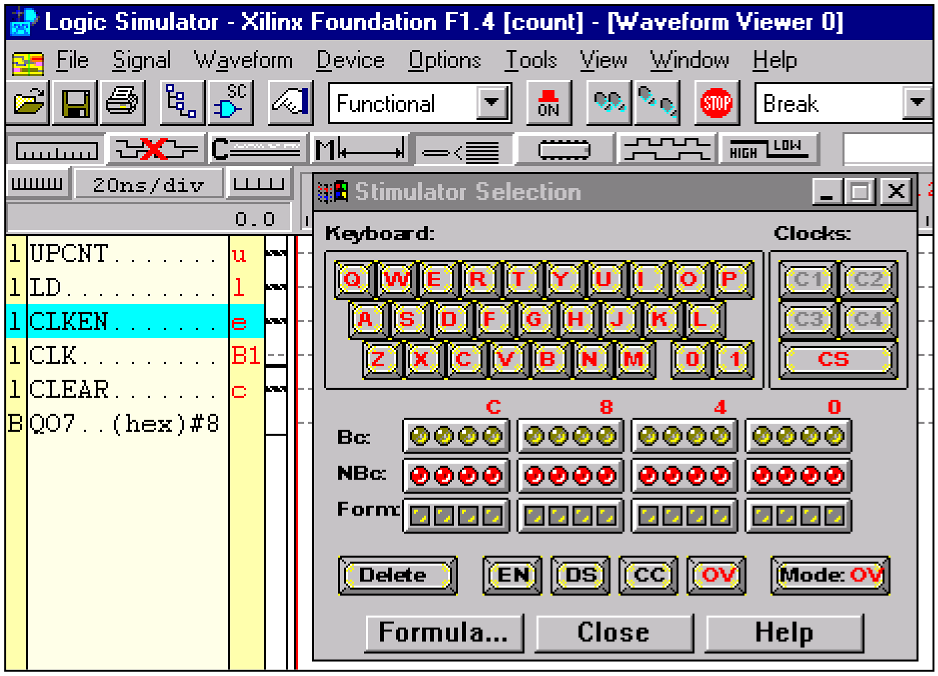
5) <u>Click</u> on these ***net names***, not the wires themselves**: `UPCNT, LD, CLKEN, CLK, CLEAR, QO[7:0]`**. You should see a light grey box near each selected net.



6) <u>Click</u> on the **SIM**ulator icon (top row, 3rd from the left) in the **SC Probes** window. This brings up the simulator tool.



7) Select **Signal → Add Stimulators**. As you select to highlight each input signal, each will turn blue in color. Then click a letter on the **selection keyboard**. For example, **CLKEN** is highlighted blue. Click on the letter "**e**". Now the letter "**E**" on your physical keyboard can toggle this input signal. Similarly, select **Binary Clock, Bc1** (Clock divided by 2^1) for **CLK**, and the other signals as shown to the right.



8) Click **Close** when done, and then you can step through your design. Press the letters "**U**", "**L**", "**E**", and "**C**" to toggle each signal from unknown to a known state. By doing so, and stepping, you can watch the waveform change. The optimum position to show the counter counting at each step is with **UPCNT** and **CLKEN** high and **LD** and **CLEAR** low. You may also need to expand the time per division to see the values of the output bus.

9) Once this is working, go back to your **Schematic Editor**. Click the **STEP** button within the small **SC Probes** box. Now you can see the values of the nets change from within your schematic each time you step!

10) There are numerous options and variations to this. After implementation to a Xilinx device, you can come back to this and see real timing driven, rather than the basic **functional simulation**.

Using the Simulation Script Editor

As a pre-requisite for this second section, you must have completed the Multiplier Core – Schematic lab. Otherwise, instead of referring to the design at **`C:\F15_labs\Cores`**, you will need to refer to the completed shortcut at **`C:\F15_labs\shortcut\cores`** .

11) Return to the **Project Manager**. You can use **<Alt>** + **<Tab>** to change windows, or **File → Exit** to close the Schematic Editor.

12) To open the **SCHVER** Project in the **CORES** directory:

A) **File → Open Project...**
B) Go to the **`C:\F15_labs\cores`** directory.
C) Select the Project **`SCHVER`.**
D) Select **Open.**

13) Open the schematic by clicking the **Schematic Editor** icon. Now browse around this design. If needed, take a moment to use the navigation tools and to navigate into and out of schematic blocks**.**

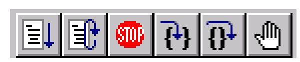
14) Go to the Simulator by selecting **File → Go To Simulator**.

15) Select **Tools → Script Editor**. Choose **Open** : **Existing Script File**. <**OK**>.

16) Browse to find **`C:\F15_labs\cores\schver\schver.cmd`** .<**Open**>.

17) Now you can look over the script command sequence for a moment. When ready, position your windows so that the "**Logic Simulator – Waveform Viewer**" is behind the "**Script Editor**" window.

18) Using the various script stepping icons, you can **Go** through the entire 1000ns simulation, **Restart** the simulation, **Stop** the simulation, **Trace**, **Step**, or **Toggle a Breakpoint** in the command script. <**F7**> is a shortcut key to step through the simulation. As you go through the stepping, watch the waveform window in the background.



.