Tutorial

on

Simulation using ISim

ver. 1.0

Spring 2012
Preparing the Input: Go to the link given above and download following files from the following link:

http://ece.gmu.edu/coursewebpages/ECE/ECE448/S12/labs/448_lab2.htm

VHDL Source Files:

1. jk_ff.vhd
2. four_bit_counter.vhd
3. count_test.vhd

Current Version of Tools: This tutorial has been tested using the following CAD Tools

• Xilinx ISE Versions: 13.1
**Introduction:**

ISim is a simulation tool integrated into Xilinx ISE. Like Modelsim and the ActiveHDL simulation tool, ISim can be used by students to debug and verify their design.

**Objective:**

This tutorial is meant to show students how to perform basic simulation tasks using ISim. This tutorial assumes that students have a working knowledge of the Xilinx ISE design flow and are able to create and manage projects in ISE.
Getting Started

Create a new project in ISE and the source files listed on the first page of this tutorial to the project. Make sure that ISim is set as the simulator in your project settings.

From the simulation view in ISE, highlight the count_test and choose Simulate Behavioral Model.
ISim should now be running, and you should see a window similar to the following:

There are four main sections in the ISim display:

A. **Instances and Processes**: Contains a hierarchical list of all component instantiations in the design you are simulating.

B. **Objects**: Displays all signals, constants, variables in a selected component in your design.

C. **Waveform**: Displays waveforms for selected signals.

D. **Console**: Displays compilation messages and any text output from your testbench.

**Running Simulations**

You can run a simulation using the simulation toolbar:

[Simulation toolbar image]

From right to left, these buttons perform the following functions
1. **Restart**: Restarts a simulation from the beginning without recompiling design files. Use this if you have added more signals and wish to see their waveforms, but you have not changed any of your design files.

2. **Run All**: Runs for an unspecified amount of time.

3. **Run For Specified Time**: Runs for the amount of time specified in the adjacent box.

4. **Step**: A useful tool in debugging code, allows the user to step through their design as it executes.

5. **Break**: Stops the simulation that is currently running.

6. **Re-launch**: Recompiles the design. This is useful to use when you have made changes to the code and wish to re-run your simulation.

### Adding Signals to the Waveform

By default the waveform window begins with the signals from your top-level file, which is the testbench. To add signals from your design to the waveform views, select the component containing the signals from the **Instances and Processes** window and then select the signals of interest. Right click on the signals and then choose “**Add to Wave Window**.” To add all signals from a particular component, right click on the component in the Instances and Process window and choose “**Add to Wave Window**.”

![Waveform Window Screenshot](image)

To view the waveforms for these signals, you must restart the simulation. Click the “**Restart**” button in the simulation toolbar and re-run your simulation.

### Waveform Format

In the screenshot on the previous page, the timescale of the waveform is such that you cannot read the values of some of the signals. To zoom in, use the “**Zoom**” toolbar:
The functions in this toolbar are:

1. **Zoom In**
2. **Zoom Out**
3. **Zoom Fit**: Fits entire waveform into window
4. **Zoom to Cursor**: Zooms in, centered at the location of the cursor.

With more complicated designs, your waveforms will often contain a large number of signals. To keep your waveform organized and readable, you may wish to group related signals together. To do so, select the signals you wish to group, right-click and select “New Group”
The result is a group of signals that can be hidden when not being used and visible when needed.
Often you will want to measure the duration of an event, or the time between two events. One way to do this is by using markers. Markers serve as reference points and can be used to measure time intervals. The commands in the marker toolbar are:

1. **Create Marker**
2. **Move Cursor to Previous Marker**
3. **Move Cursor to Next Marker**

Clicking on a placed marker will display the time relative to that marker. In the example below, you can see that the time between the reset pulses is 220 ns. We measured this by placing a marker at the end of one pulse and the cursor at the next reset pulse:
A shortcut is to click on the waveform at the desired start time, and then drag the mouse cursor to the end point of the interval.

**Saving Waveform Format**

When you close ISim, the simulation data you were using is lost. You will need to rerun the simulation the next time you start ISim in order to recreate the data. It is therefore a good idea to always save the waveform format of your simulations so that you do not have to add all of your signals and create and groups, dividers, or buses every time you wish to run the simulation. To save the waveform format, go to **File ➔ Save As** and save your waveform as a `.wcfg`, the native format for ISim waveform configurations.