Creating Certus-NX-RT Debug with ModelSim

PRODUCT NAME	MANUFACTURER PART NUMBER	DEVICE TYPE
Lattice Certus-NX-RT FPGA	UT24C407	RL01

Table 1: Cross Reference of Applicable Products

1.0 Overview

This document details the process of debugging a **Certus-NX-RT** FPGA project using the **Lattice ModelSim**. For the purposes of this document, create a project named **counter_sim_debug** and configure **ModelSim** to include all the source modules required for a successful build. Using this template, projects are created using (a) the preferred application source directory structures and (b) the directory structure for the **Lattice Radiant**-supplied files. **Figure 1** shows the counter simulation.

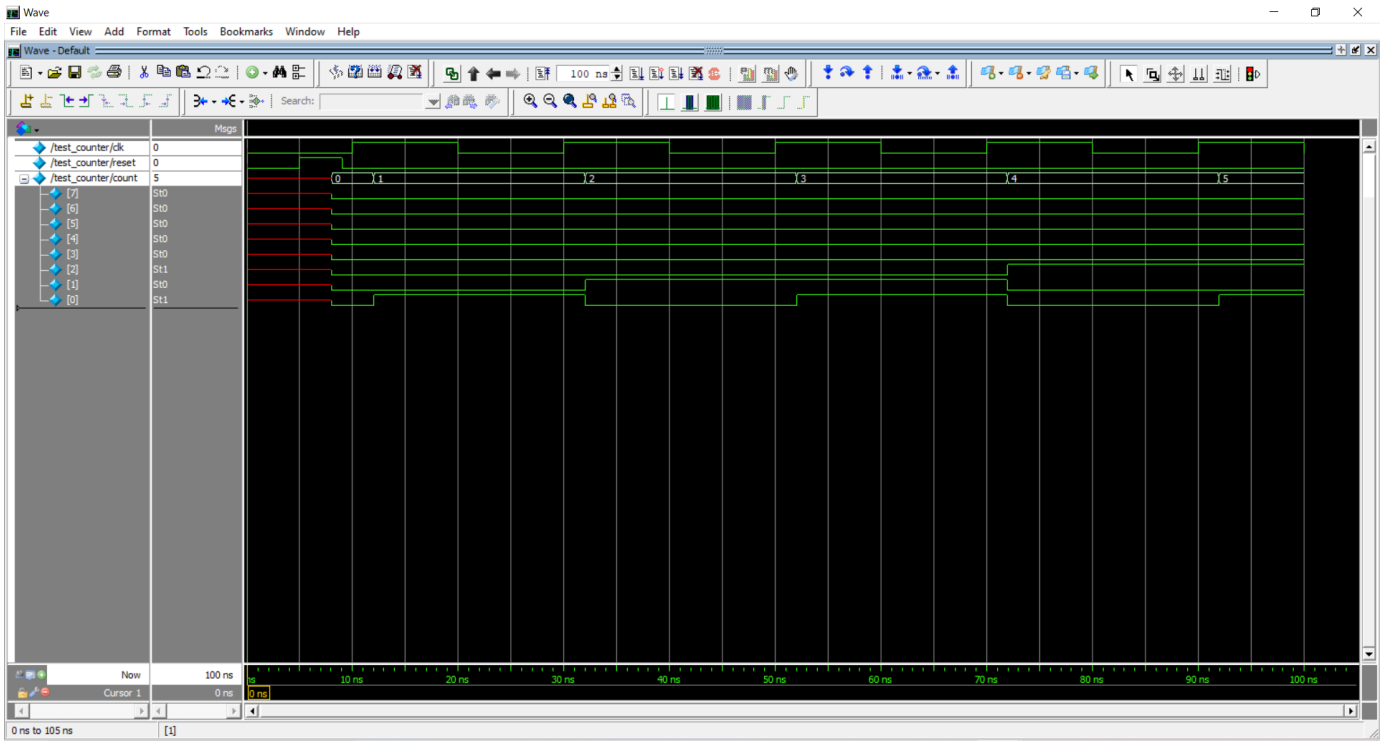


Figure 1: An Example ModelSim Wave

Creating Certus-NX-RT Debug with ModelSim

2.0 Creating a Project

1. Copy **Mentor Graphics** example files from **Lattice Radiant** installation directory:
   <installation dir>\modeltech\examples\tutorials\verilog\basicSimulation

   To:

   <project dir>\Lattice\applications

2. Launch **Lattice ModelSim**
3. Click **Jumpstart** to create a project, see **Figure 2**.

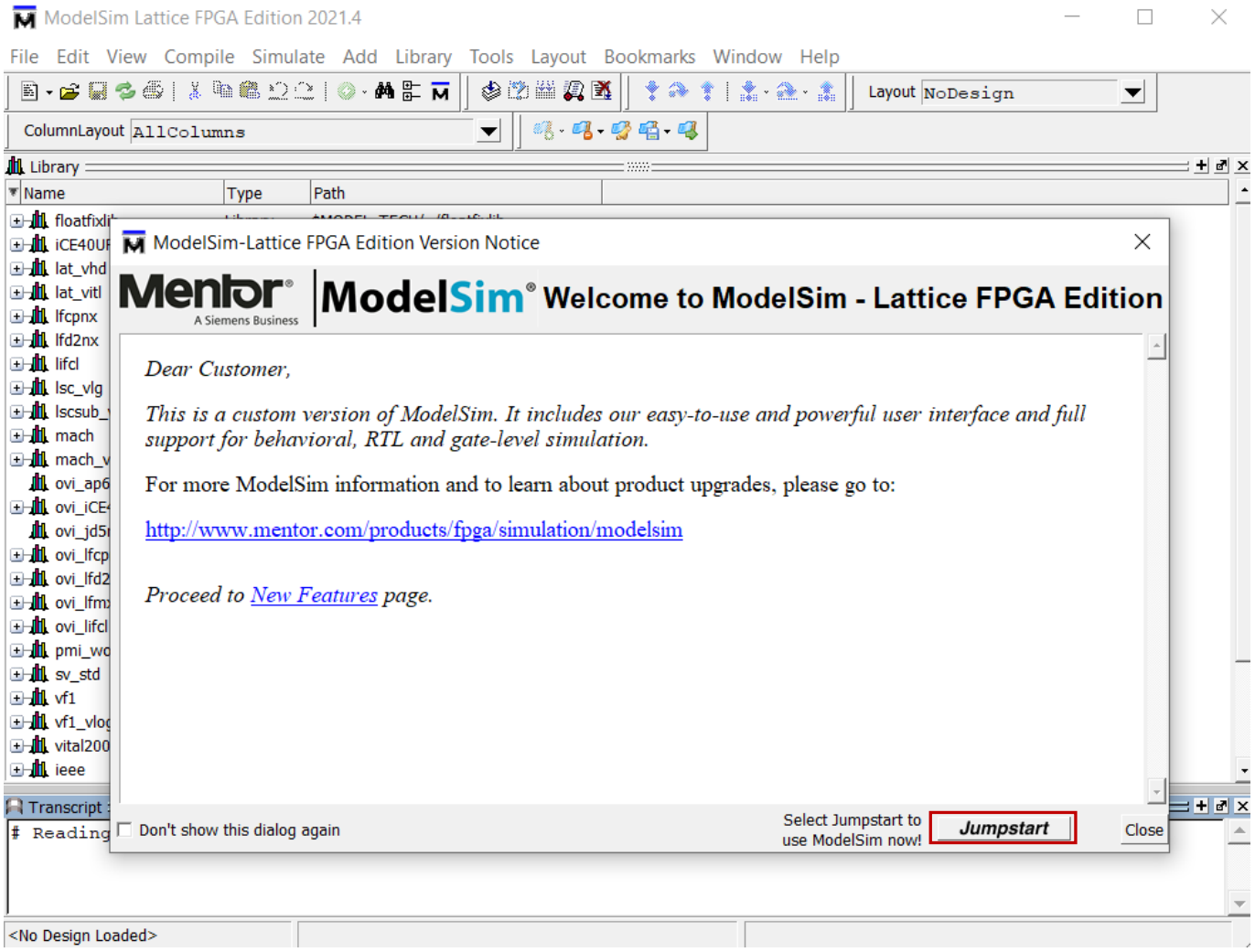


Figure 2: ModelSim-Lattice Project Creation

Creating Certus-NX-RT Debug with ModelSim

4. Specify the project name as **counter_sim_debug**, navigate to the **Project Location** of your choice and click **OK**, see **Figure 3**.

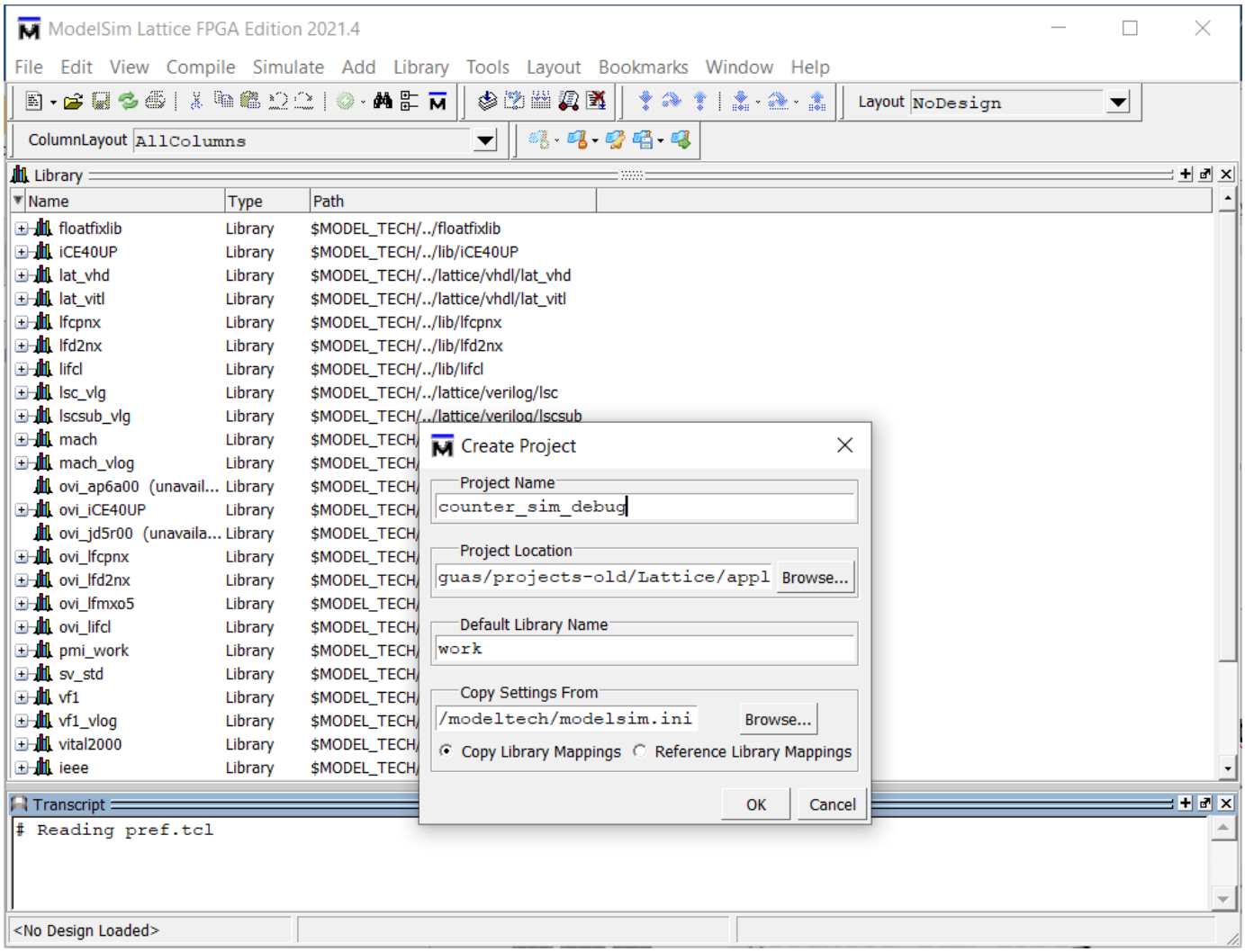


Figure 3: Naming the Project

Creating Certus-NX-RT Debug with ModelSim

5. Click **Add Existing File** and add **counter.v** and **tcounter.v** from <project dir>\Lattice\basicSimulation, see **Figure 4**.

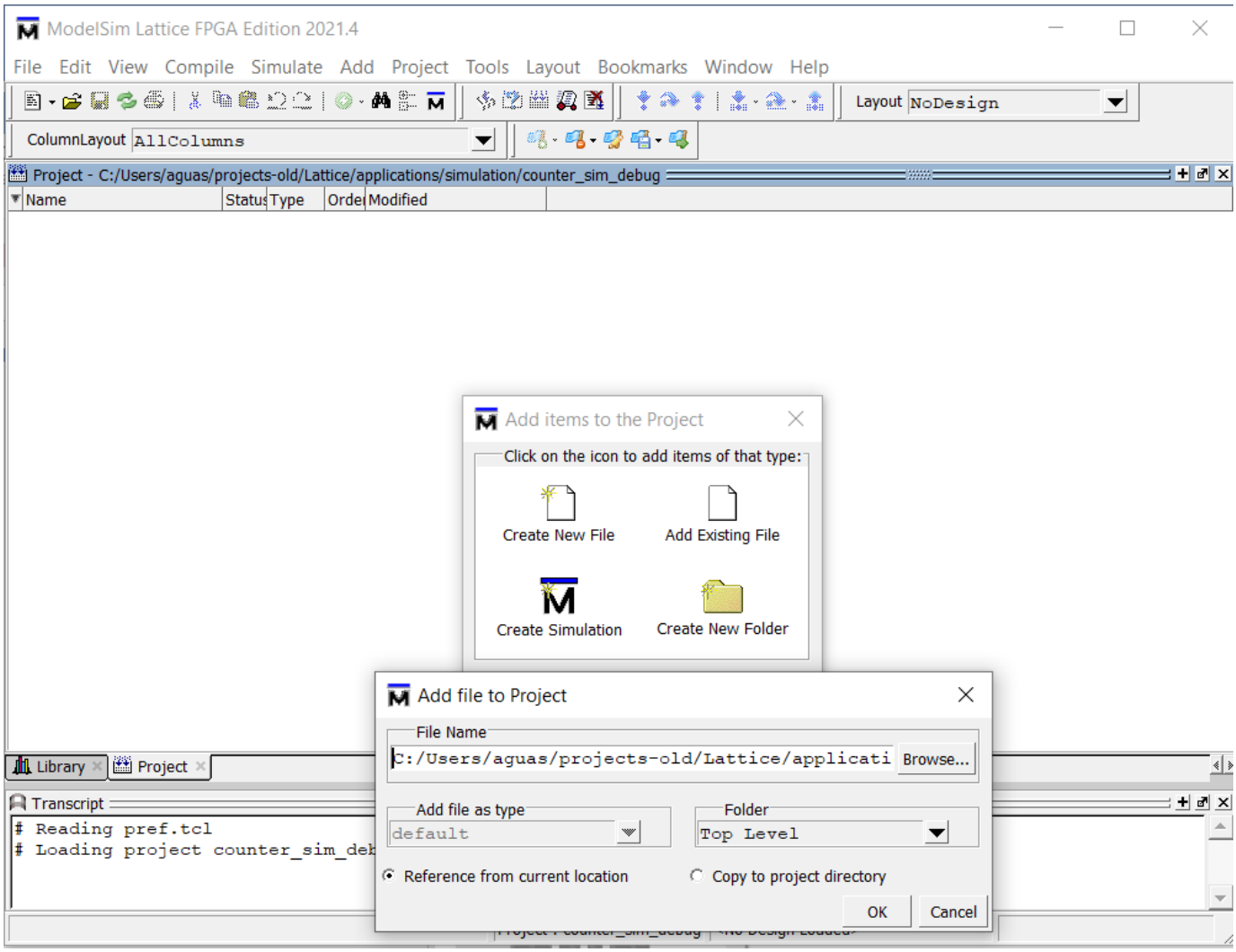


Figure 4: Adding Files to the Project

Creating Certus-NX-RT Debug with ModelSim

6. Form the top menu, select **Compile > Compile All** to compile **tcounter.v** and **counter.v**, see **Figure 5**.

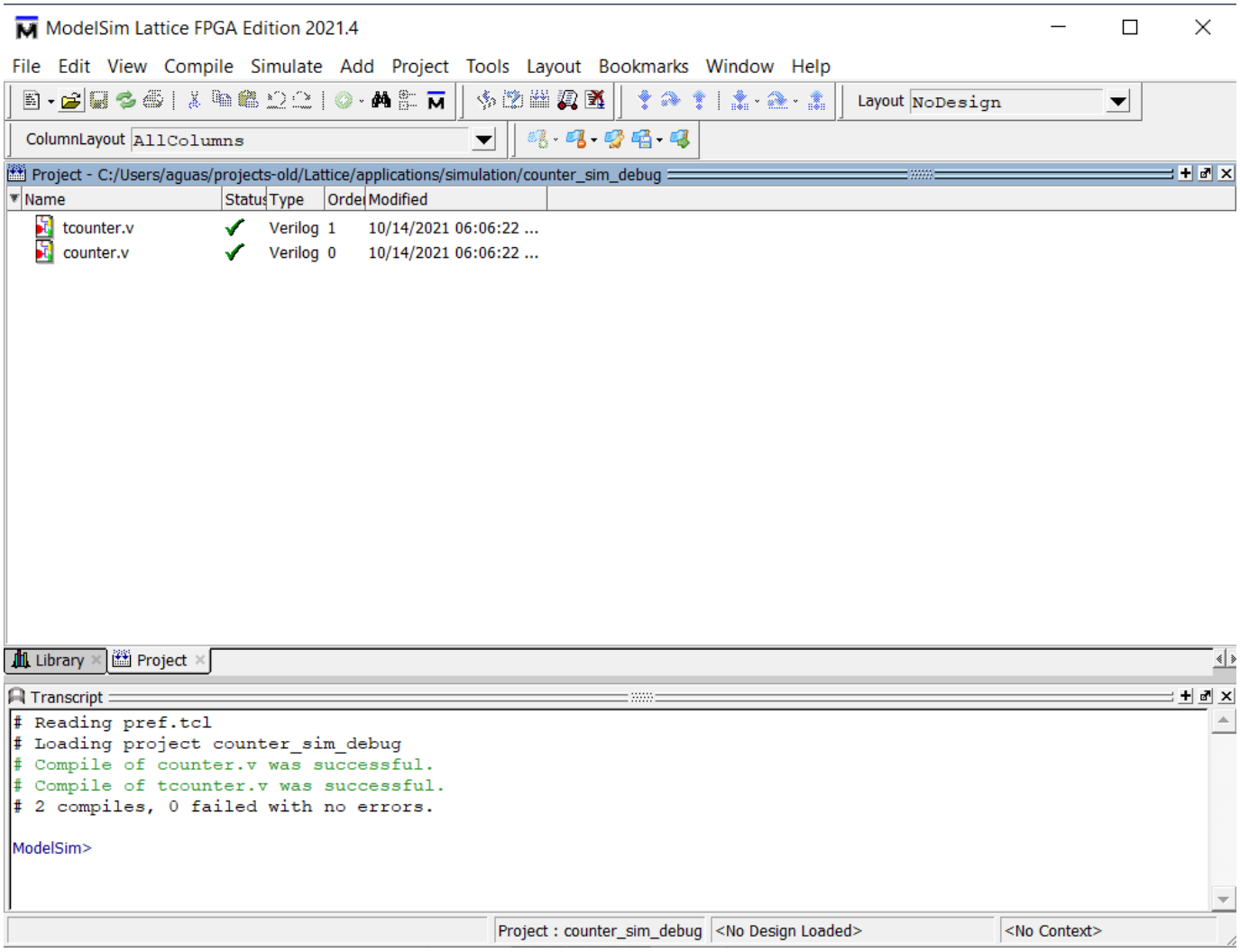


Figure 5: Compiling the Project

Creating Certus-NX-RT Debug with ModelSim

7. Form the top menu, select **Simulate > Start Simulation...** and select **test_counter.v** in the work library and click **OK**, see **Figure 6**.

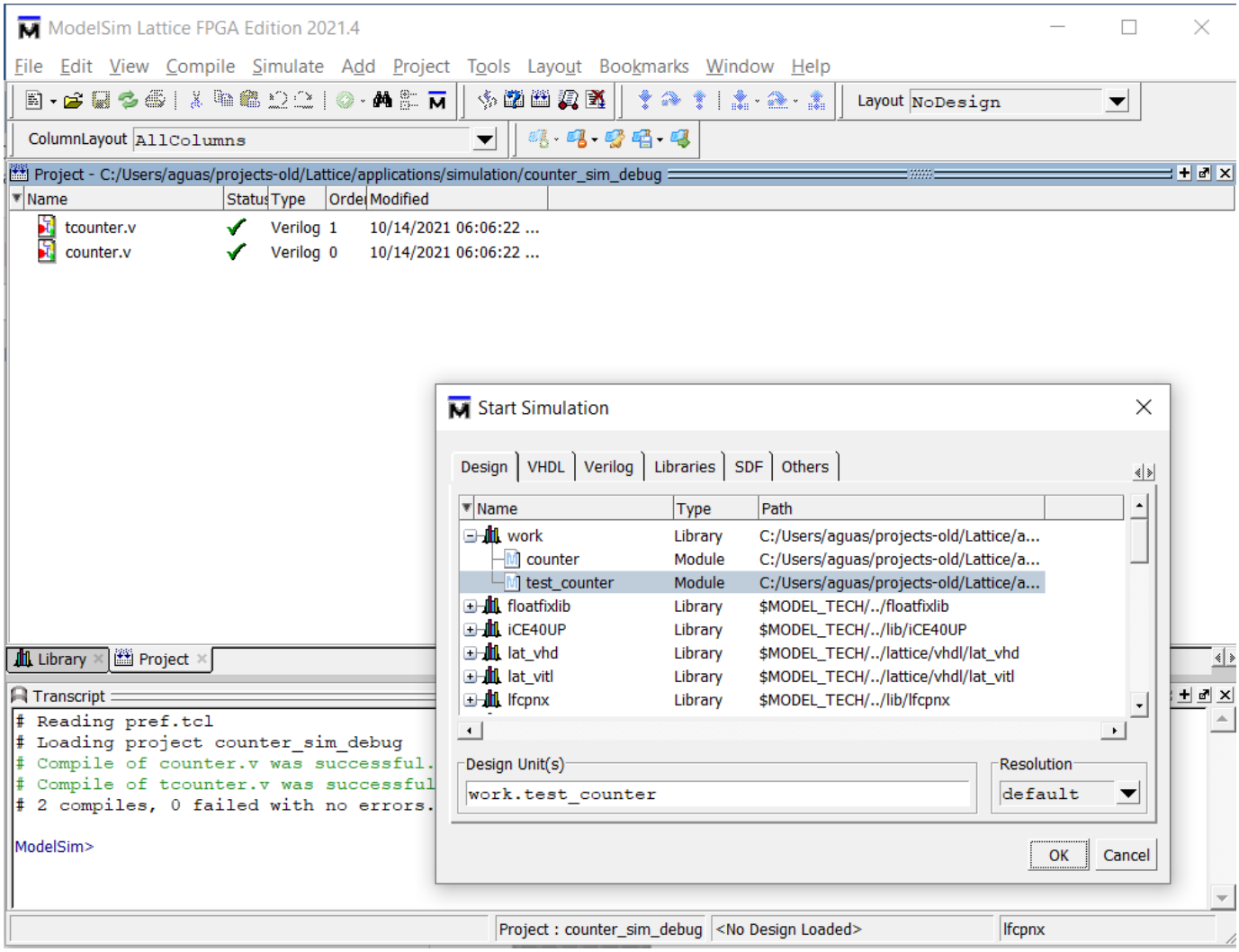


Figure 6: Starting a Simulation

Creating Certus-NX-RT Debug with ModelSim

3.0 Running the Simulation

3.1 Simulating Signals

8. From the **Objects** window, choose the top-level signals and drag them over to the **Wave** window. **Run** for 100 nanoseconds and the **Wave** window will display the timing diagram of the signals. Click **Zoom Full** to see the full picture, see **Figure 7**.

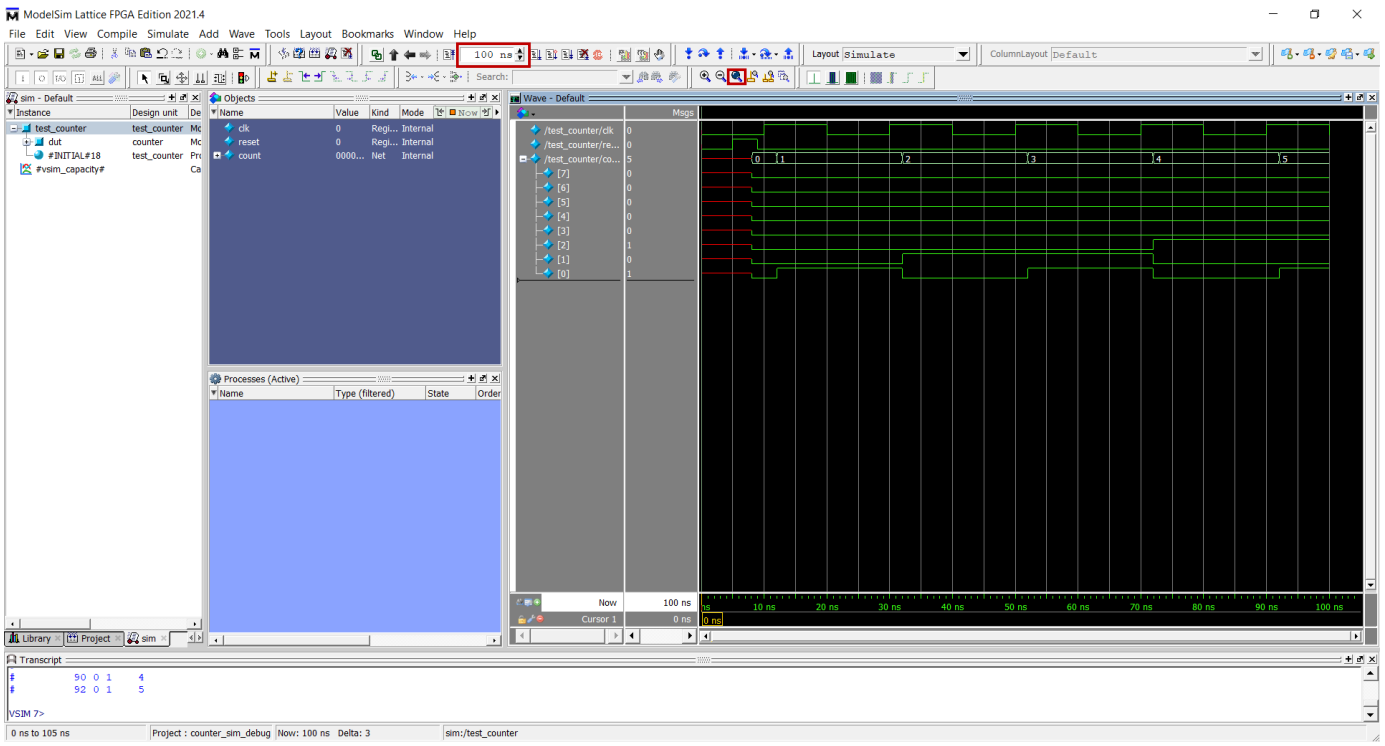


Figure 7: Zooming on the ModelSim Waveform

Creating Certus-NX-RT Debug with ModelSim

4.0 Debugging HDL Code

9. In the Transcript window, enter:
   edit counter.v

10. The **counter.v** file opens next to or over the **Wave** window, see **Figure 8**.

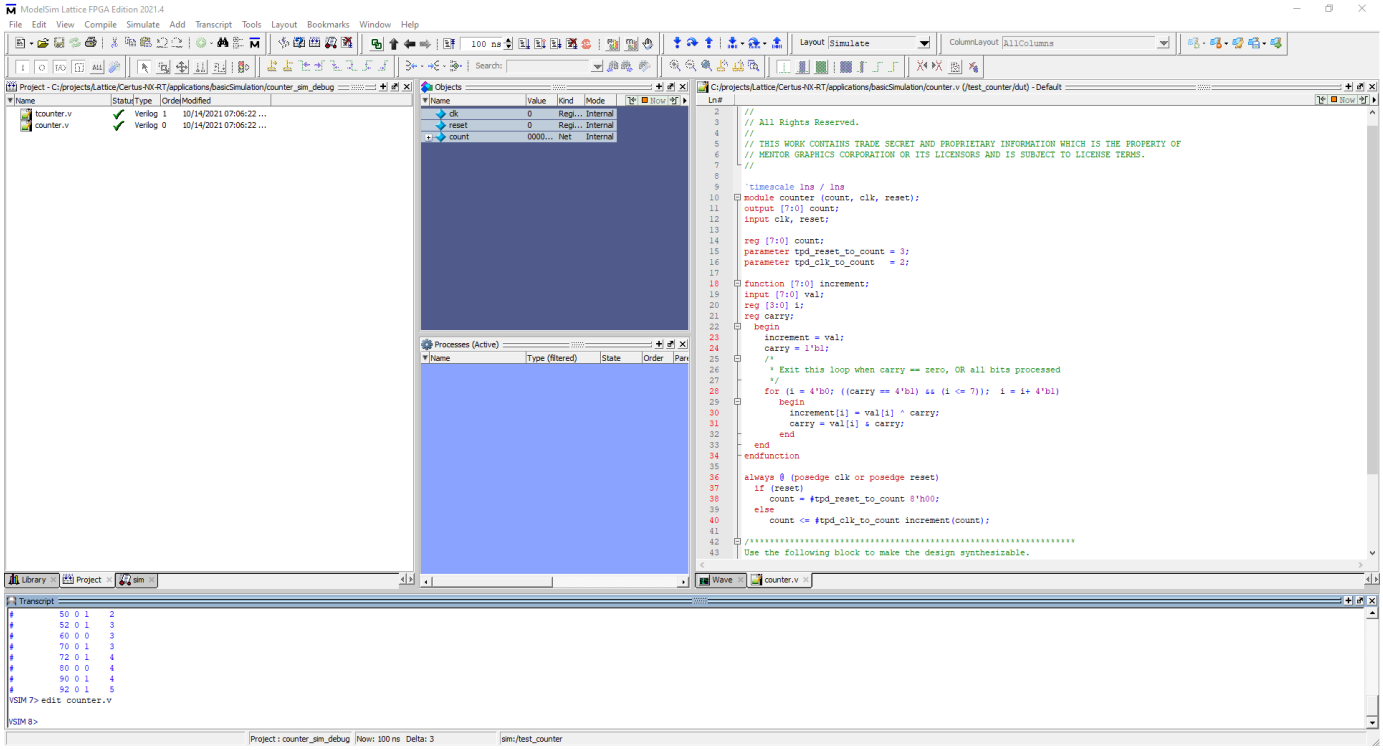


Figure 8: Debugging HDL Code

Creating Certus-NX-RT Debug with ModelSim

11. **ModelSim** allows to debug the **HDL** code with break points and stepping through the code. Double click on line 40 or right-click on it and select **Set Breakpoint**; the debugger sets a breakpoint at line 40, see **Figure 9**.

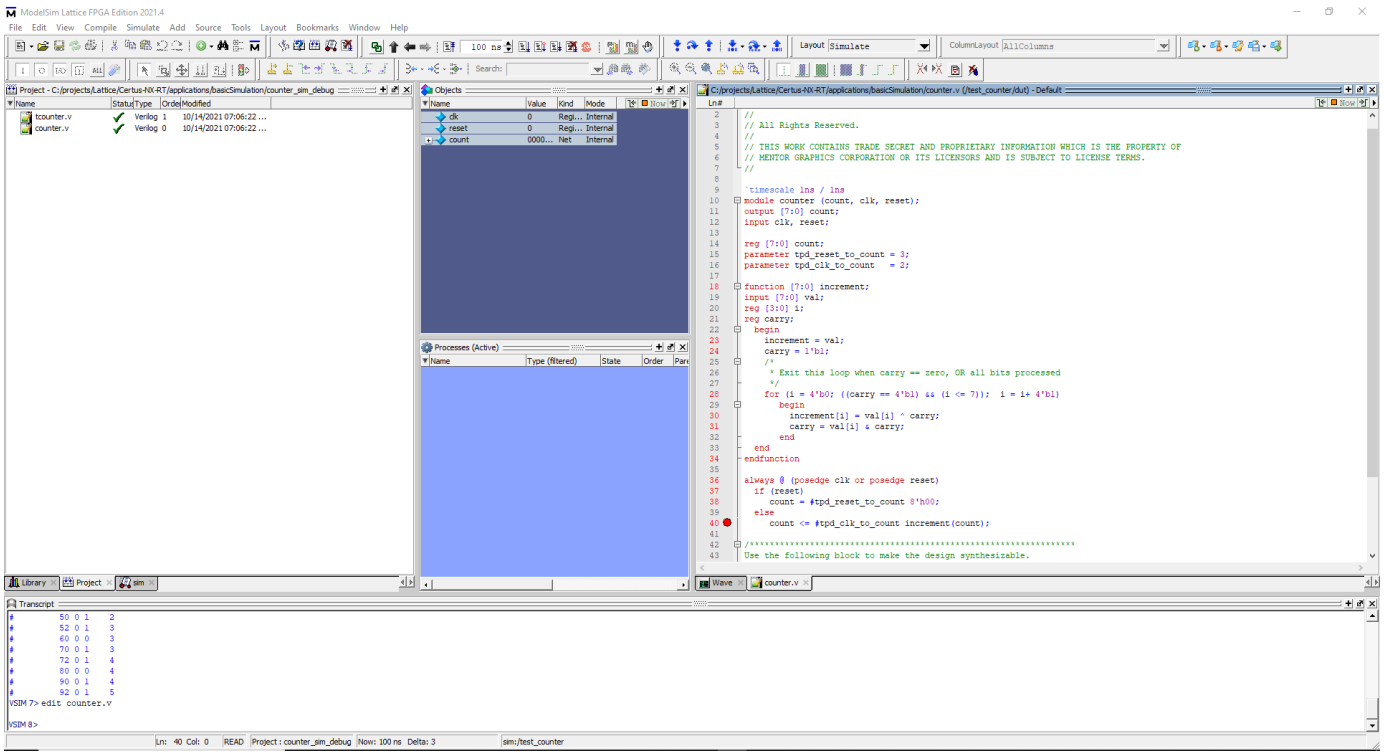
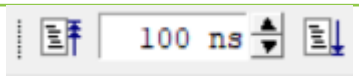


Figure 9: Setting a Breakpoint

Creating Certus-NX-RT Debug with ModelSim

12. Run for 100ns 

and the debugger will stop at line 40; signal values can be seen in the **Objects** window. As with any other **IDE** running **C** code, you can **Step Into**  , **Step Over**  , and **Step Out**  ,see **Figure 10**.

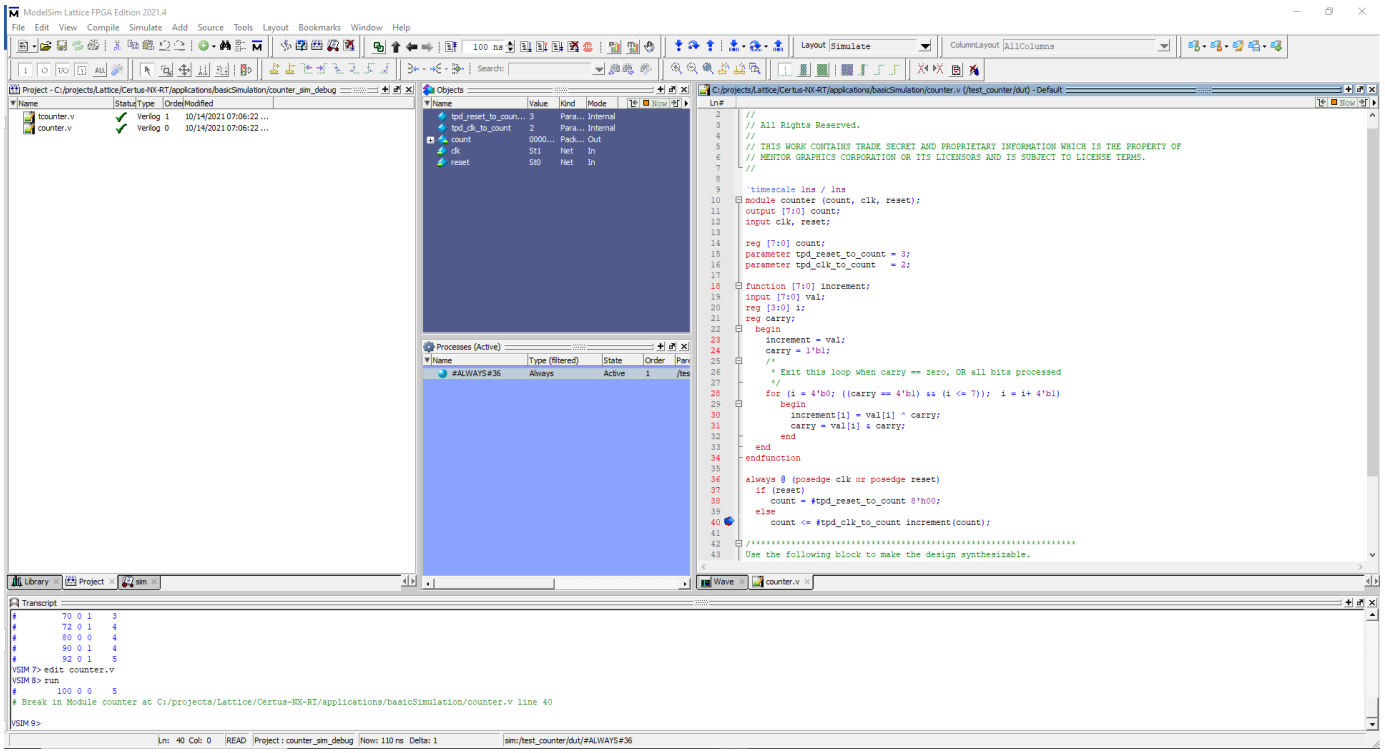


Figure 10: Stepping Around a Breakpoint

Creating Certus-NX-RT Debug with ModelSim

5.0 REVISION HISTORY

Date	Rev. #	Author	Change Description
3/18/2022	1.0.0	JA	Initial Release.
4/21/23	1.0.1	JB	Document reformatted to match Frontgrade template.

The following United States (U.S.) Department of Commerce statement shall be applicable if these commodities, technology, or software are exported from the U.S.: These commodities, technology, or software were exported from the United States in accordance with the Export Administration Regulations. Diversion contrary to U.S. law is prohibited.