Cadence Tutorial 4

Simulating a Schematic with Verilog-XL

EE577b Fall 98

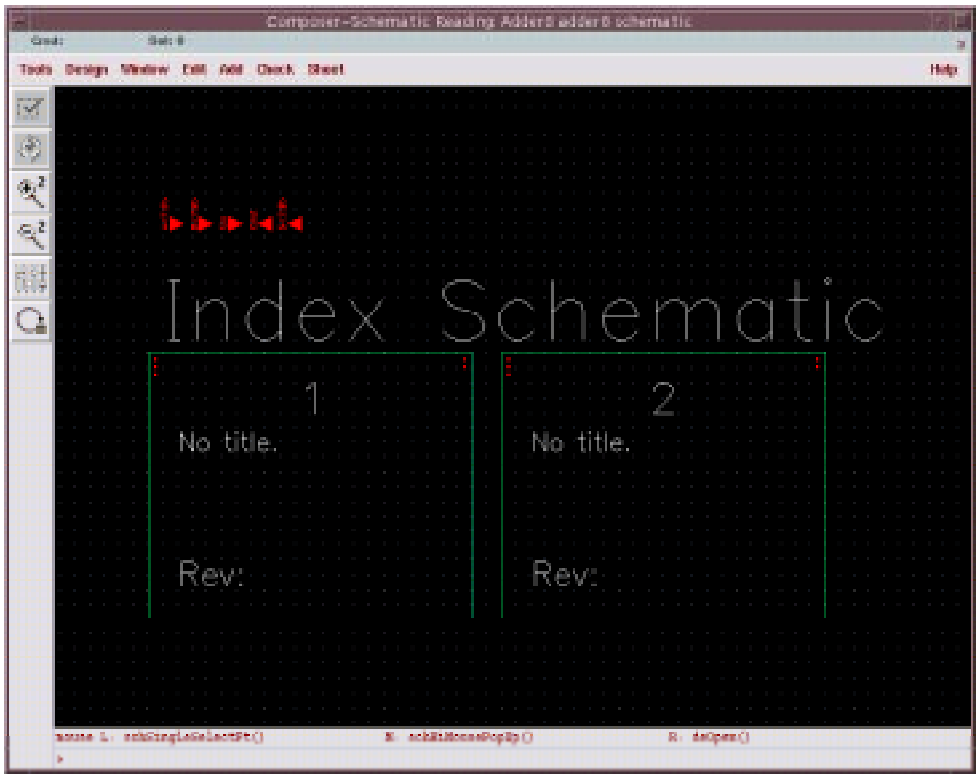
In this tutorial, you will run a Verilog simulation on the schematic cellview of adder8.

1. Tutorial Setup

1. Finish the cadence tutorial 3 before you start this tutorial.
2. Invoke “icds”.

2. Open the 8-bit Adder Schematic Cellview

1. From the Library Manager, read the *adder8 schematic* cellview from the *Adder8* library.
   The *adder8 schematic* cellview is displayed.



3. Initializing Verilog Integration

1. Start Verilog integration by selecting sch : **Tools->Simulation->Verilog-XL** from the *adder8 schematic* cellview.
   The Verilog-XL Setup Environment window is displayed.

jsmoon@ISI.EDU

2. In the Verilog-XL Setup Environment window, enter *adder8.run2* for the run directory. All other default values are correct.
3. Click on OK.

   The Verilog-XL Integration control window is opened. The *adder8.run2* run directory is created and the environment is initialized.

4. Setting the Netlist and Waveform Options

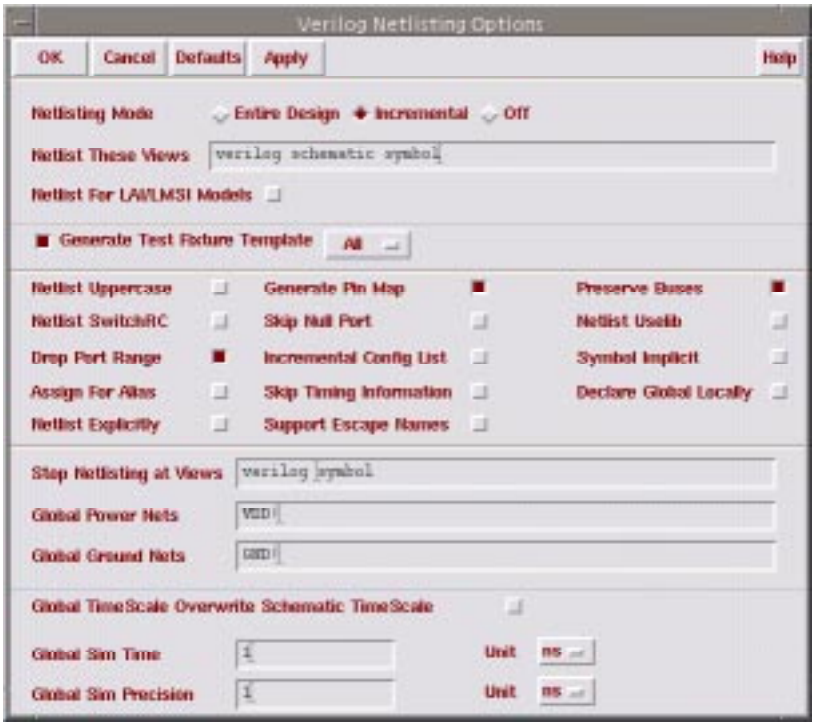
1. In the Verilog-XL window, set the Netlisting Options by selecting **Setup->Netlist**.

   The Netlisting Options form is displayed.
2. In the Netlisting Option window, set **Netlist These Views** list to:

   `verilog schematic symbol`
3. Click on **More>>**

   Additional netlisting options are added to the form.
4. Define **Stop Netlisting at Views** as:

   `verilog symbol`
5. Click on **Generate Pin Map** to turn it on.
6. Click on OK.



5. Creating the Stimulus File

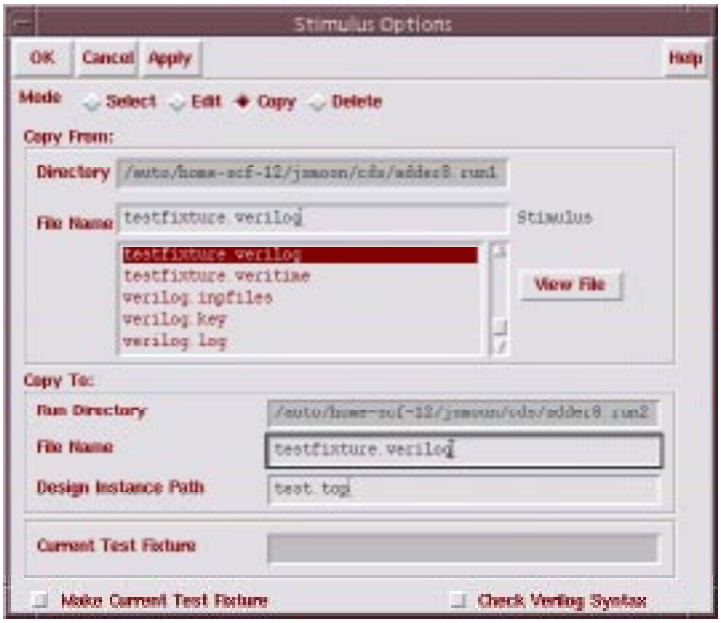
The test fixture file used in the previous tutorial is used for this tutorial as well.

1. Create a Test Fixture (Stimulus) file by selecting

   Stimulus->Verilog
2. Click on No on the dialog box that appears to copy test fixture file on *adder8.run1*.
3. Select *testfixture.verilog* file of *adder8.run1* in **Copy From** and click **copy** on Mode.
4. Type *testfixture.verilog* in **Copy To**.
5. Click OK.

   These steps are for copying *testfixture.verilog* of *adder8.run1* to *adder8.run2*.

jsmoon@ISI.EDU



6. Select *testfixture.verilog* by **Stimulus->Verilog** again.
   Click *testfixture.verilog* and click on **Select** at Mode.

6. Simulation

All simulation steps are same as the previous tutorial. Please look for it to simulate your schematic.
Step 7 to 11 are for simulation and special note for viewing waveform.

7. Netlist Files

Take a look at the netlist files in *ihnl/cds0* directory. You can see how transistor schematic is netlisted in verilog code.

jsmoon@ISI.EDU