Cadence Tutorial 7

Generating HSPICE Netlist from Schematic

EE577b Spring2000

In this tutorial, I will show how to generate HSPICE netlist from schematic.

1. Tutorial Setup

Tutorial 1,2,4 are necessary to start this tutorial.

2. Open adder8 Schematic

As usual!!

3. Initiate Netlist Generation Tool

- 1. sch:Tools->Other
- 2. sch:Simulation->Initialize Initialize Environment form will b



- 3. Type hspice.run1 or use default (spice.run1) in Simulation Run Directory
- 4. Click OK.

Initialize Environment form will be displayed.

😡 Initialize Environment	
OK Cancel Defaults Apply	Help
Simulation Run Directory	/export/home/jungfrau-00/sangyunk/cds/hspice.r
Simulator Name	hspice =
Design	Browse
Library Name	Adder
Cell Name	adderg
View Name	schematic

- 5. Choose *hspice* for **Simulator Name**
- 6. Use default for others.
- 7. Click OK.