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 be displayed.
   3. Type hspice.run1 or use default (spice.run1) in Simulation Run Directory
   4. Click OK.
      Initialize Environment form will be displayed.
   5. Choose hspice for Simulator Name
   6. Use default for others.
   7. Click OK.