I. Basic Usage of HSPICE and Awaves

Step 1. Create an HSPICE file

All HSPICE files should have an .sp extension.

Type in athena prompt:

```
% emacs inv.sp &
```

Simple HSPICE file1:

```
*CMOS Inverter - DC Sweep

* Netlist

* Define Voltage Sources
vdd 1 0 2.5
vin in 0 pvdd

* Define Transistors
m1 out in 1 1 pch l=0.25u w=1.125u as=0.7p ad=0.7p ps=2.375u pd=2.375u
m2 out in 0 0 nch l=0.25u w=0.375u as=0.7p ad=0.7p ps=2.375u pd=2.375u

* Define Output Capacitance
Cout out 0 30f

* Define Parameters
.param pvdd=2.5V

* Models
.lib 'logic025.l' TT

* Analysis

* post option necessary for awaves
.options nomod post
```
DC sweep from 0 to 2.5V at 0.01V increments
 dc vin 0 pvdd 0.01
 .end

Simple HSPICE file2:
-------------------------------
CMOS Inverter - Transient Analysis
-------------------------------

* Netlist

* Define Voltage Sources
 vdd 1 0 2.5
 vin in 0 pulse(0v 2.5v 1n 0.1n 0.1n 4n 8n)

* Define Transistors
 m1 out in 1 1 pch l=0.25u w=1.125u as=0.7p ad=0.7p ps=2.375u pd=2.375u
 m2 out in 0 0 nch l=0.25u w=0.375u as=0.7p ad=0.7p ps=2.375u pd=2.375u

* Define Output Capacitance
 Cout out 0 30f

* Models
 .lib 'logic025.l' TT

* Analysis

* post option necessary for awaves
 .options nomod post

* Transient Analysis for 10ns
 .tran 0.01n 10n
 .end

There are three main sections in the file:

a. the netlist: Netlist is a designation for a computer readable representation of the circuit schematic.
b. the models: A model in spice is a description of the parameters of the equations used by spice to analyze the circuit.
c. the analysis to be performed: here we are requesting a DC sweep from 0 to 2.5 with 0.01V increments and a transient analysis for 10 ns with step 0.01n.
d. the end of the file. This isn’t really a main section, but hspice won’t work without it, and many people forget about it. Always put a .end statement at the end of your file.

Step 2.

If you haven’t done so, add spice (this command also adds awaves):
Run spice in the background, and direct the output to a file for later reference:
% hspice inv.sp > inv.out &

When the simulation is complete, you should have some *.sw* (DC “sweep” data) and/or *.tr* (transient analysis data) files:
% ls
inv.sp  inv.out  inv.sw0  inv.tr0

**Step 3.**

Now, we use awaves to display the data generated by hspice stored in the .sw and .tr files:
% awaves inv &

To view waveforms, select the node number (or node name) that you want to see with the mouse button and drag and drop them in the highlighted panel. Double-clicking on the node numbers also brings up the waveforms. Panels are highlighted when they have a red border. The graph window will display the waveforms which can be measured using a variety of features.

**NOTE:** awaves doesn’t run on Linux.