# 6.374: Analysis and Design of Digital Integrated Circuits Using HSPICE

Fall 2002                                                                 Issued: 9/10/02

---

## I.        Basic Usage of HSPICE and Awaves

**Step 1.**
**Create an HSPICE file**

All HSPICE files should have an .sp extention.

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):

```
% add hspice
```

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.