

## ELE447 Lab 1: Introduction to HSPICE

In this lab, you will learn how to use HSPICE for simulating the electronic circuits.

To be able to simulate a circuit using HSPICE, we need to write a text file that describes all nodes, power supplies and elements (such as transistors, resistor and capacitors, ...) in the circuit. The text file including the circuit information is called the **netlist of the circuit**. First the Commands necessary to define circuit's parameters and elements are explained and a simple RC circuit is simulated and finally you should write the netlist for a simple inverter and simulate it using HSPICE.

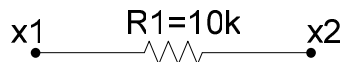
### Defining Circuit Parameters

**1. Elements:** each circuit consists of some elements like transistor, resistor and capacitor. In this section we learn how to introduce different elements.

#### Resistor:

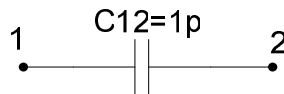
Rxxx n+ n- value

R1 x1 x2 10k



#### Capacitor:

Cxxx n+ n- value



C12 1 2 1p

#### Inductor:

Lxxx n+ n- value

#### Diode:

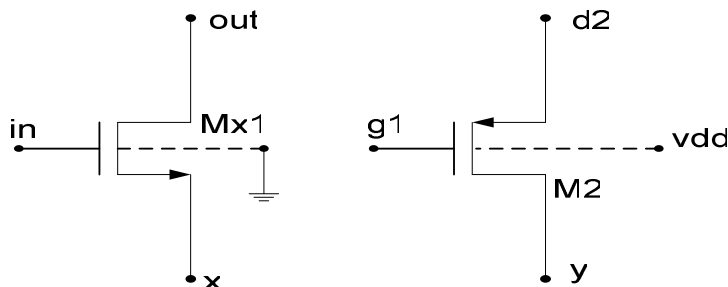
Dxxx n+ n- modelname

#### JFET:

Jxxx nd ng ns modelname

#### MOSFET:

Mxxx nd ng ns nb modelname W=value L=value Ad As Pd Ps



```

Mx1 out in x Gnd nfet W=10u L=1u
M2 y g1 d2 Vdd pfet W=8u L=0.6u

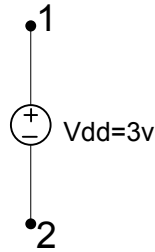
```

**Voltages sources:**

A) Independent voltage sources:

Vxxx n+ n- DC value AC value

Vdd 1 2 DC 3v



B) Voltage Controlled Voltage source (VCVS):

Exxx n+ n- in+ in- gain

C) Current Controlled Voltage source (CCVS):

Hx2 20 10 VCUR MAX=+10 MIN=-10 1000

The example above selects a linear current controlled voltage source. The controlling current flows through the dependent voltage source called VCUR. The defining equation of the CCVS is:

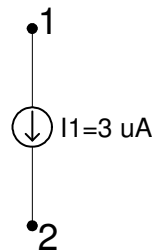
$$HX2 = 1000 \cdot VCUR$$

**Current sources:**

A) Independent current source:

Ixxx n+ n- DC value AC value

I1 1 2 DC 3u



B) Current Controlled Current source (CCCS):

Fxxx n+ n- vn1 gain

vn1: names of voltage sources through which the controlling current flows.

( to use a current controlled current source, a dummy independent voltage source is often placed into the path of the controlling current).

F1 13 5 VSENS MAX=+3 MIN=-3 6

C) Voltage Controlled Current source (VCCS):

Gxxx n+ n- in+ in- transconductance

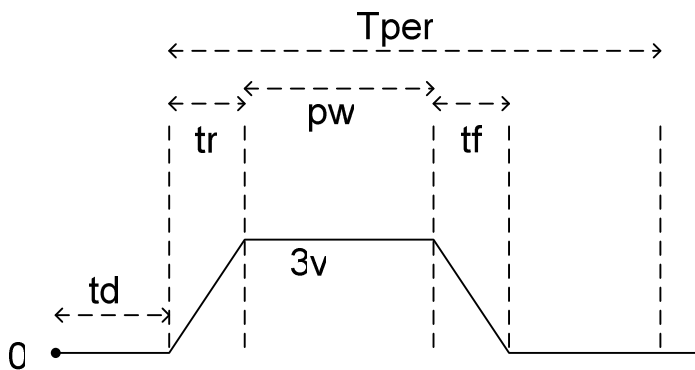
A voltage-controlled resistor represents a basic switch characteristic. The resistance between nodes 2 and 0 varies linearly from 10 meg to 1 m ohms when voltage across nodes 1 and 0 varies between 0 and 1 volt. Beyond the voltage limits, the resistance remains at 10 meg and 1 m ohms, respectively.

Gsw 2 0 VCR PWL(1) 1 0 0v,10meg 1v,1m

## 2. Different types of voltage and current sources:

Pulse:

Vxxx n+ n- pulse v1 v2 td tr tf pw T(period)



Vpulse 1 2 pulse 0 3 0 1n 1n 98n 200n

Sinusoid:

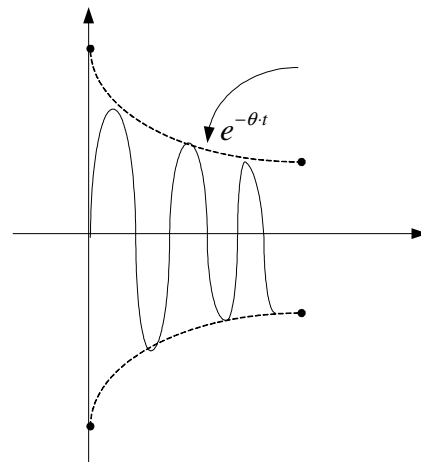
Vxxx n+ n- SIN Vdc A freq. td  $\theta$   $\phi$

Vdc: The DC value of sinusoid.

td: time delay before starting the sinusoid.

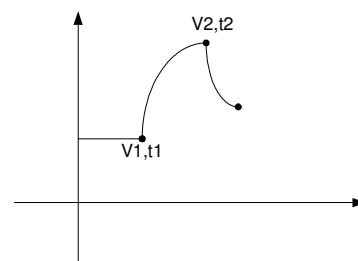
A: amplitude of sinusoid

Vdc=0,  $\phi = 0$



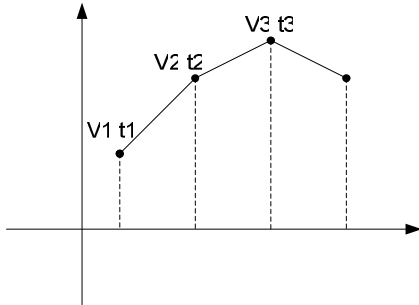
Exponential source:

Vxxx n+ n- exp v1 v2 td1  $\tau 1$  td2  $\tau 2$



Piecewise linear source:

**Vxxx n+ n- PWL t1 v1 t2 v2 t3 v3 .....**



### 3. Different types of simulation:

-DC simulation: to find the DC operating points of nodes.

**.DC var start stop incr**

-AC simulation: to find the frequency response of a circuit.

**.AC type np fstart fstop sweep var type np start stop**

-Transient simulation: to find the time domain response of circuit to a signal that changes over time( like pulse or a sinusoidal source)

**.Tran step stop start**

### 4. Exporting the information of a variable:

After we simulate a circuit spice builds different files based on type of simulation performed. One of these files is .lis file (HSPICE builds .lis file regardless of simulation type). Command .print, writes the information to the .lis file.

**.print V(1) V(1,2) Vdb(1) Vp(1) I(M1)**

### 5. Commands used at the beginning and the end of each netlist:

At the beginning:

**.include 'library path (model path)'**

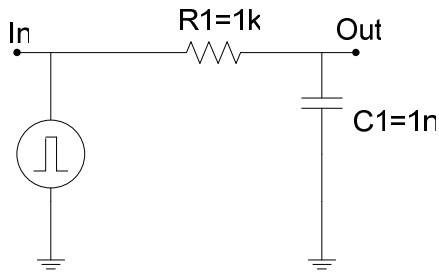
At the end:

**.options node list post**

**.op**

**.end**

Now we are ready to write the netlist to describe a simple RC circuit in Figure 1. The corresponding netlist is listed below:

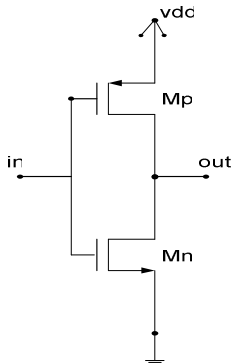


**Figure 1.**

```
R1 in out 1k
C1 out 0 1n
Vin in 0 pulse 0 3 0 1n 1n 20n 40n
.tran 1n 200n

.print V(in) V(out) V(in,out) I(Vin)
* .IC V(out)=1v
* .pz V(out) V(in) or .pz V(out) I(in)
.op
.options node list post
.end
```

After simulating a simple RC circuit, create the netlist of the CMOS inverter in Figure 2. Apply a positive pulse from 0 to 3 volts to the input of the inverter (and then a negative pulse from 3 volts to 0) and plot the output using the scope.



**Figure 2.**

The transistor geometry is:

(a)  $(W/L)_p = (W/L)_n = 4/2$

(b)  $(W/L)_p = 2.5 (W/L)_n$

In both cases, find the rise time, fall time, propagation delay, trip point and noise margin of inverter.

Compare the results of (a) and (b) in a table and explain their differences with analytical expressions.

## Appendix

### Running HSPICE, CSCOPE, MAGIC, IRSIM

To avoid confusion in finding the files, create a folder ELE447 in your home directory and make 4 subfolders HSPICE, CSCOPE, MAGIC, IRSIM for their corresponding files.

Home folder: /u/grads/your home folder/ELE447 -----> make four different folders in ELE447 named HSPICE, CSCOPE, MAGIC and IRSIM to save their corresponding files.

#### **HSPICE:**

- 1- Create a new text file.
- 2- Write the netlist necessary for simulating the circuit and save the file with .sp extension for example RCsimple.sp .
- 3- Open a terminal and type HSPICE Complete Path/RCsimple.sp and click enter.

#### **CSCOPE:**

After you runned the netlist. HSPICE creates a file based on type of simulation that you have done. For example name.tr for transoent simulation and name.ac file for AC simulation which can be used to plot the data of the circuit simulated.

- 1- Open a terminal and type cscope.
- 2- Go to File/open/plotfiles and look up for .tr or .ac files depending on type of performed simulation and open it.
- 3- Upon opening the file, CSCOPE opens a window showing the voltage and current of different nodes on the circuit.
- 4- Click on any current/voltage of any node to see the corresponding waveform.

#### **MAGIC and IRSIM:**

**Run the commands below before using magic/irsim.**

- 1- open a terminal.
- 2- emacs .cshrc
- 3- alias magic50 'magic -d ogl -T SCN3ME\_SUBM.30'
- 4- alias irsim50 'irsim -s scmos30.prm'
- 5- save your configuration and exit.

#### **Library:**

Professor Fischer has provided a library of layout of basic cells like and, nor, adder and a lot more. You can access those files in path below.(you can copy them to your home directory and use them.)

Path: /net/common/uricells1/ele447