HSpice simulation tutorial in Linux:

1) Create a netlist using any text editor and save it as [filename].sp.
   You can include the model file by using 'INC [path to model file] ', or you can copy the model file and insert it into the netlist directly (not preferred).
   The model file for this course – par0.18um.net is included in this folder. Please save it and note its path so that you can use it in netlists.

2) Run a Star-Hspice analysis in the terminal by typing:
   hspice [filename].sp >! [filename].lis

   If you use only '>' and [filename].lis already exists, message appears that '[filename].lis: File exists',
   but '>!' does not lead to this problem.

   When the run finishes successfully, the Star-Hspice displays
   >info: ***** hspice job concluded

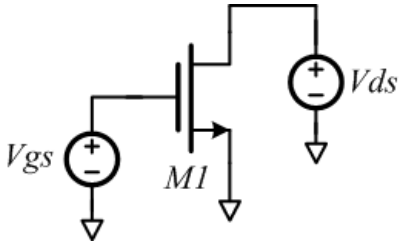
   If not, it will display
   >info: ***** hspice job aborted
   Then you need to examine the syntax of the input netlist file.

3) Open the [filename].lis to view the simulation results.

4) In the terminal, type 'scope' to open CosmosScope to view the waveforms.
   'File' --> 'Open'-->'Plotfiles'-->'Files of type [HSPICE (*.tr*, *.ac, *.sw*, *.ft*)]'

Example 1: NMOS

Testbench for NMOS is shown as below:

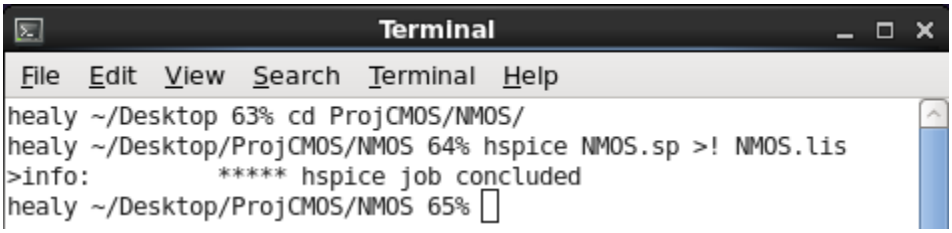


a) Netlist: NMOS.sp

NMOS.sp

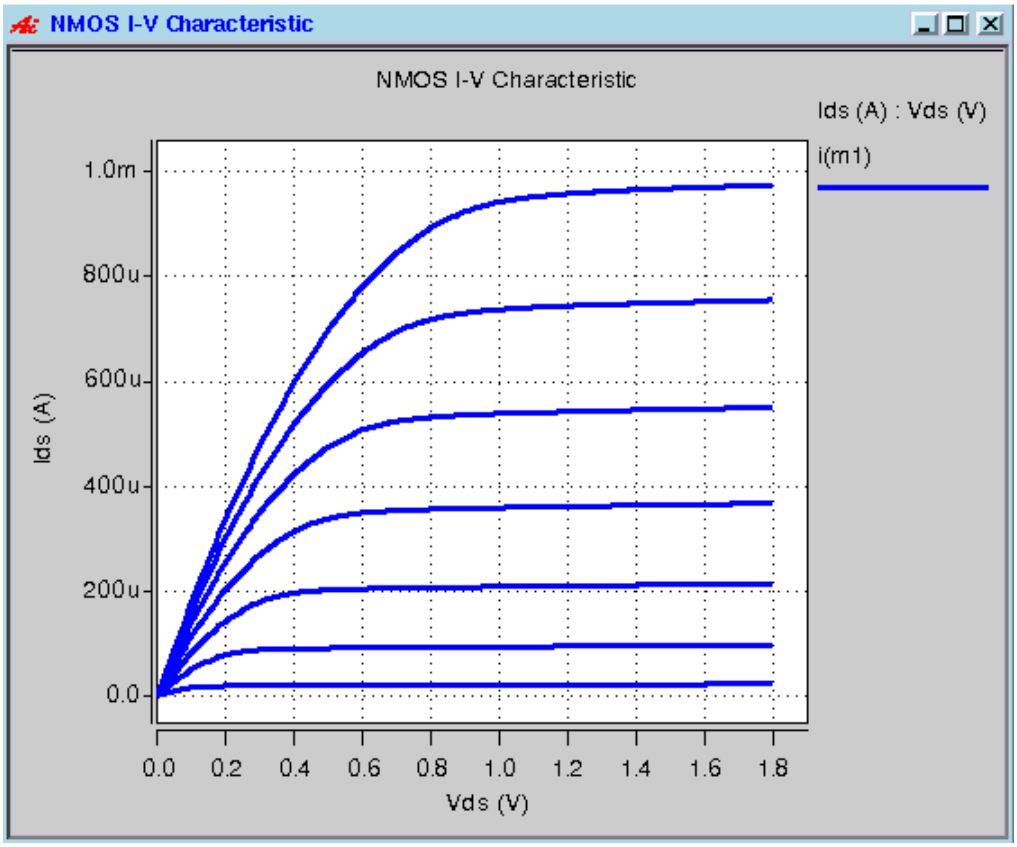
```
.title NMOS
.options post

**** inlude model file ****
.inc  ../par0.18um.net

**** Components ****
M1 Vd Vg 0 0 N_18 W=3u L=1u M=2

**** Sources ****
Vds Vd 0
Vgs Vg 0

**** Controls ****
.dc Vds 0v 1.8v 0.01v Vgs 0.6v 1.8v 0.2v
.op
.print dc i(m1)

.end
```

b) Move to the directory where [filename].sp exits and start Hspice to simulate



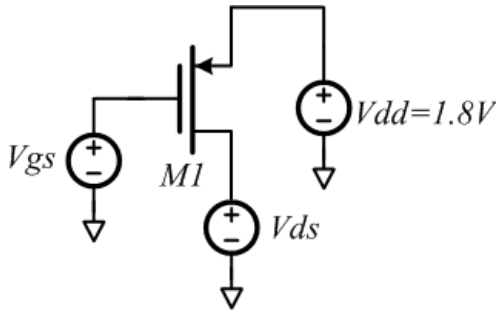
c) View the waveform using CosmosScope

```
healy ~/Desktop/ProjCMOS/NMOS 65% scope
```



1. PMOS

Testbench for PMOS is shown as below:

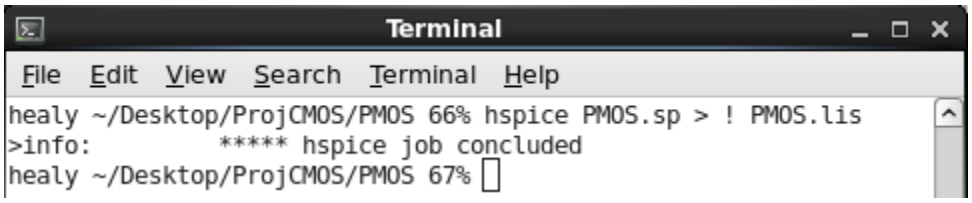


a) Netlist: PMOS.sp

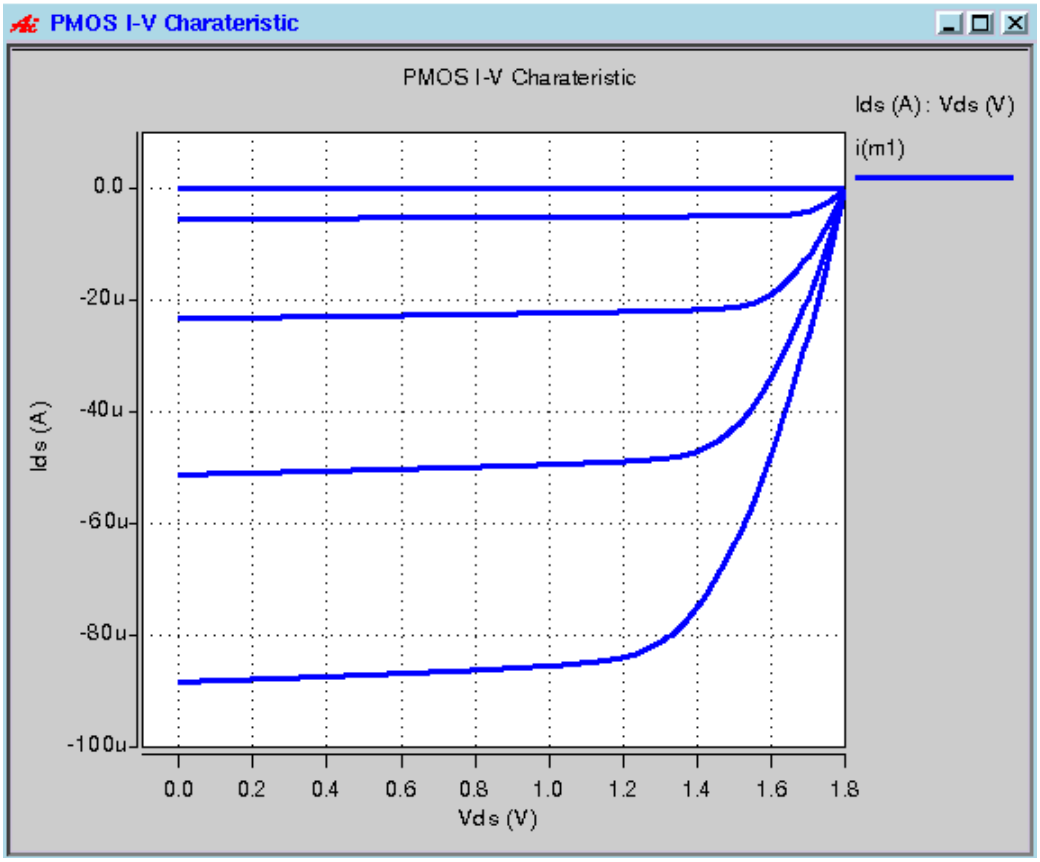
PMOS.sp

```
.title PMOS
.options post

**** inlude model file ****
.inc  ../par0.18um.net

**** Components ****
M1 Vd Vg Vs Vs P_18 W=3u L=1u M=2

**** Sources ****
Vdd Vs 0 DC=1.8v
Vds Vd 0
Vgs Vg 0

**** Controls ****
.dc Vds 0v 1.8v 0.01v Vgs 0.6v 1.8v 0.2v
.op
.print dc i(m1)

.end
```

b) Start Hspice to simulate

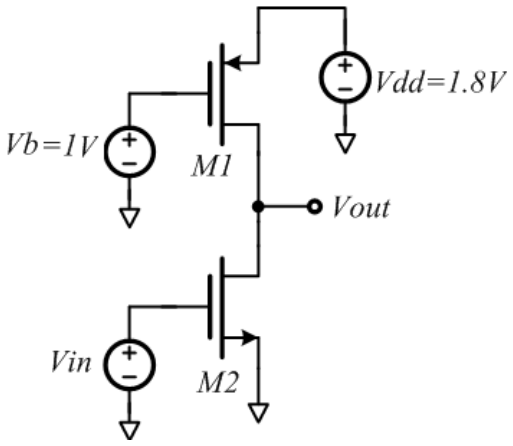


c) View the waveform



2. Common-Source Amplifier with Active Load

Testbench for PMOS is shown as below:



a) Netlist: CommSource.sp

CommSource.sp

```
.title CommSource
.options post

**** inlude model file ****
.inc  ../par0.18um.net

**** Components ****
M1 Vout Vb vd Vd P_18 W=3u L=1u M=2
M2 Vout Vg 0 0 N_18 W=3u L=1u M=2

**** Sources ****
Vdd Vd 0 DC=1.8v
Vbb Vb 0 DC=1v
Vin Vg 0

**** Controls ****
.dc Vin 0v 1.8v 0.01v
.op
.print dc v(Vout)

.end
```

b) View the waveform