SPICE and MATLAB

This document explains how to configure and use SPICE and MATLAB on the EECS instructional servers. You can use a different environment but we cannot support it. Instructions for creating and using instructional accounts are at http://inst.eecs.berkeley.edu.

We will be using Spectre, a version of SPICE from Cadence, for circuit simulation and Matlab for plotting results. Cadence runs under Linux; make sure to connect to an appropriate server (not a Windows computer).

Setting up the Environment

Copy the file `bashrc` from the course website (resources) to your home directory on the instructional machines and rename it .`bashrc` (note the “dot” at the beginning of the filename). If you already have a .`bashrc` file you may want to merge the two files using an editor (e.g. `gedit`).

Then run the following command at the prompt:

```
source ~/.bashrc
```

Verify your Spectre setup. If your setup is correct, typing

```
spectre –h
```

at the command prompt displays the version of the simulator and some documentation.

Running Matlab on a Linux computer requires running X11. Either login locally on one of the instructional machines or follow the instructions at the instructional website for setting up X11 with remote login.

Start Matlab by typing

```
matlab &
```

at the command prompt. Help and documentation are available from the graphical user interface or by typing `help` at the console.

Spectre Documentation

The following command gets you to the directory with the documentation:

```
cd `cds_root spectre`/doc
```

Start with the information in folders `spectreuser` (simulator usage) and `spectreref` (analysis statements) `spectremod` (instances, e.g. `vsource`, `resistor`, …).

Documentation for the Spectre-Matlab Toolbox is available in `MatlabWorkshop.pdf`. Type

```
cd `cds_root spectre`/tools/spectre/examples/SpectreRF_workshop
```

to get to the directory where the file is located.

Simulating a Circuit with Spectre

In this example we simulate a resistively loaded common-source amplifier. Use your preferred editor (e.g. `gedit`) to create `cs.scs`, the file with the simulator input:

```
// NMOS Common-Source Stage

simulator lang=spectre

* device model; download from course website
include "./cmos180.scs" section tt

* save transistor bias (gm, cgs, ...); very helpful for diagnosis
save m1

* circuit model

parameters vdd=1.8
parameters vi=510m
parameters ibias=170u

vin (vi 0) vsource dc=vi mag=1
vob (vdd2 0) vsource dc=vdd/2
ib  (vdd2 vo) isource dc=ibias

m1 (vo vi 0 0) nfet l=180n w=26u
rl (vdd2 vo) resistor r=1.41k
cl (vo 0) capacitor c=2.26p

* analysis

options1 options gmin=0.1p reltol=0.1m vabstol=1u iabstol=100ps
options2 options temp=27 tnom=27
options3 options save=allpub rawfmt=psfbin rawfile="./cs.raw"

dc1 dc param=vi start=0 stop=1.2 lin=101
ac1 ac start=100k stop=10G log=101
noise1 (vo 0) noise start=100k stop=1T log=101
```

Download the technology file (`cmos180.scs` in this example) from the course website and place it in the same directory as `cs.scs`. Then run the simulation from the command prompt:

```
spectre cs.scs
```

Check the output for errors, warnings and notices. Address all issues, then start Matlab:

```
matlab &
```

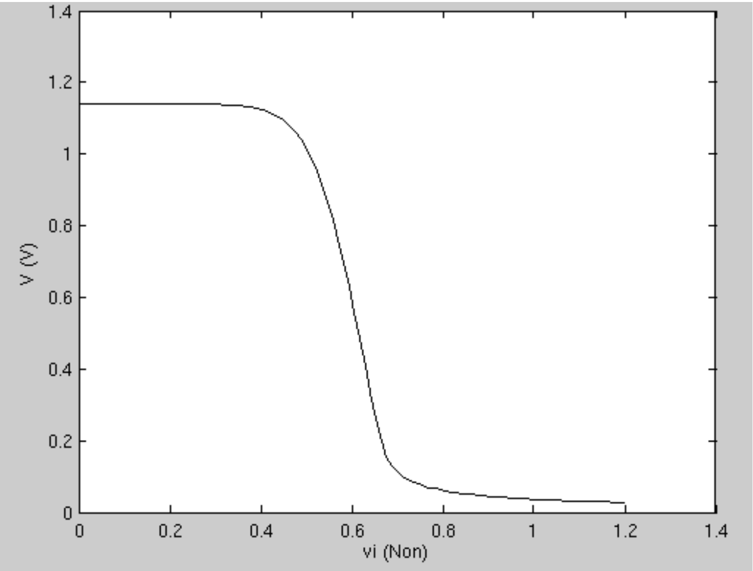
At the matlab command prompt, type

```
    vo = cds_srr('cs.raw', 'dc1-dc', 'vo')
```

to load the simulated dc output voltage vector. The command

```
    cds_plotsig(vo)
```

produces the following figure:

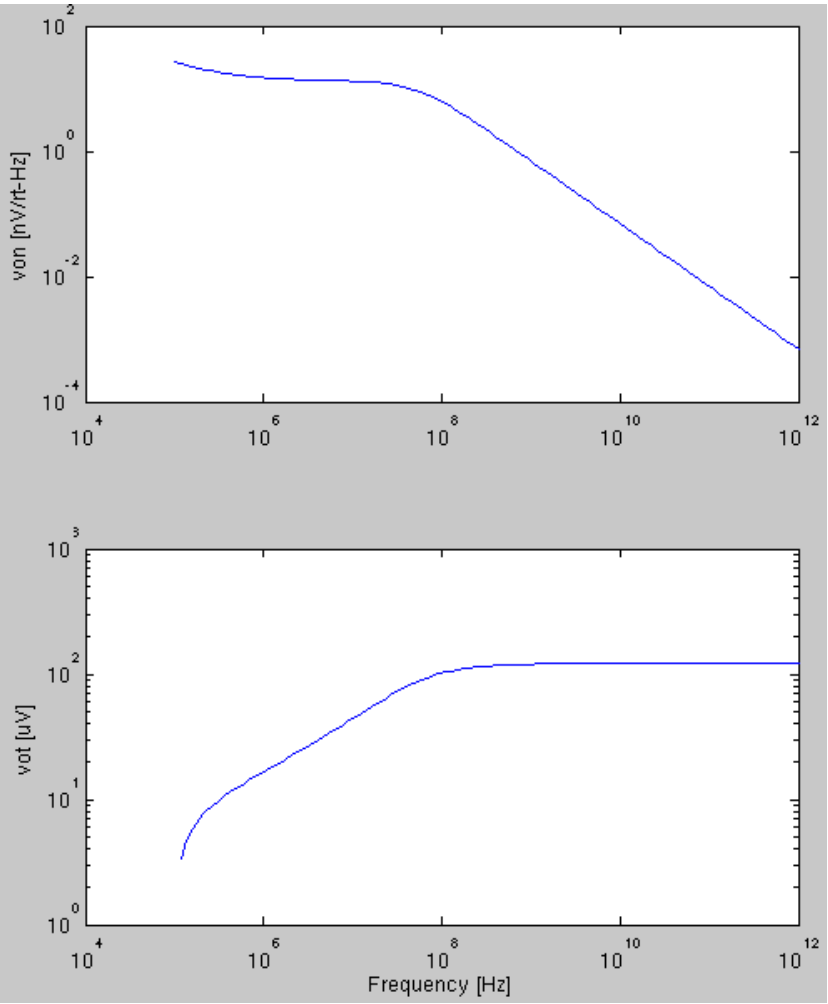


To plot the ac-response type

```
    voac = cds_srr('cs.raw', 'ac1-ac', 'vo')
    cds_plotsig(voac, '', 'freq', 'db20')
    set(gca, 'xscale', 'log')
```

The following commands plot the noise density and total noise at the output of the circuit:

```
    out = cds_srr('cs.raw', 'noise1-noise', 'out');
    von = out.V_sqrt_Hz_;
    f = out.freq;
    vot = sqrt(cumtrapz(f, von.^2));
    subplot(2,1,1);
    loglog(f, von*1e9);
    ylabel('von [nV/rt-Hz]');
    subplot(2,1,2);
    loglog(f, vot*1e6);
    ylabel('vot [uV]');
    xlabel('Frequency [Hz]');
```


von [nV/rt-Hz]
10^{2}
10^{0}
10^{-2}
10^{-4}
10^{4}
10^{6}
10^{8}
10^{10}
10^{12}
vot [µV]
10^{3}
10^{2}
10^{1}
10^{0}
10^{4}
10^{6}
10^{8}
10^{10}
10^{12}
Frequency [Hz]