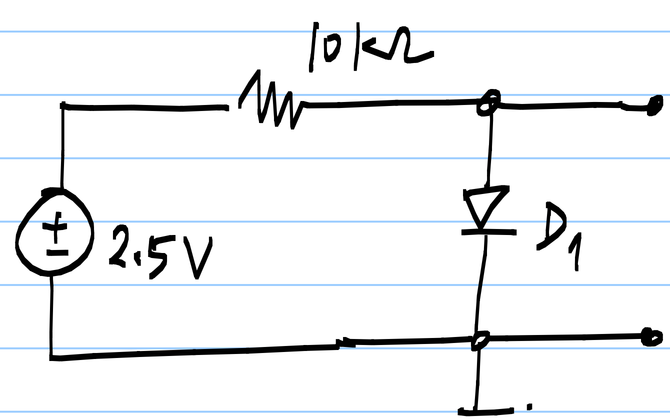
Note Title 3/9/2011

Using spice-like simulators for circuit analysis

Example circuit:

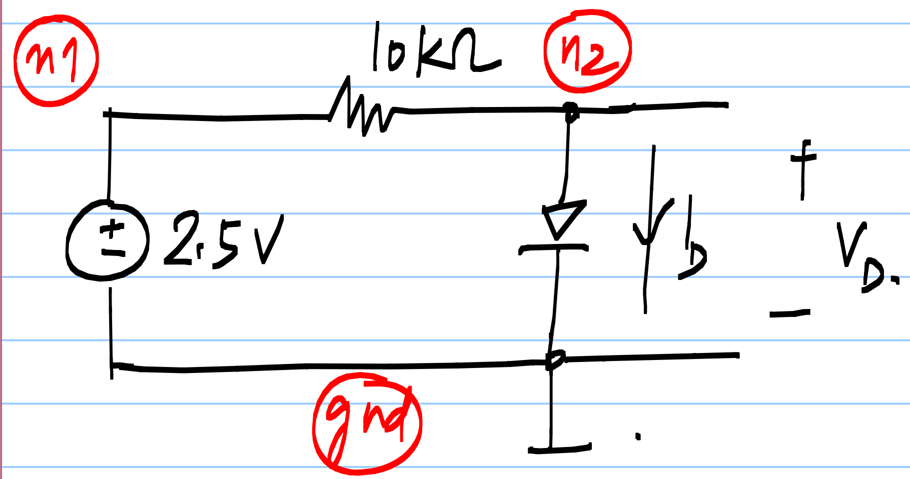


D_1: "ideal_diode"

Analysis that one would like to do:

* Operating point (DC)
* Small signal incremental analysis (AC)
* Analysis with large time varying signals (transient)

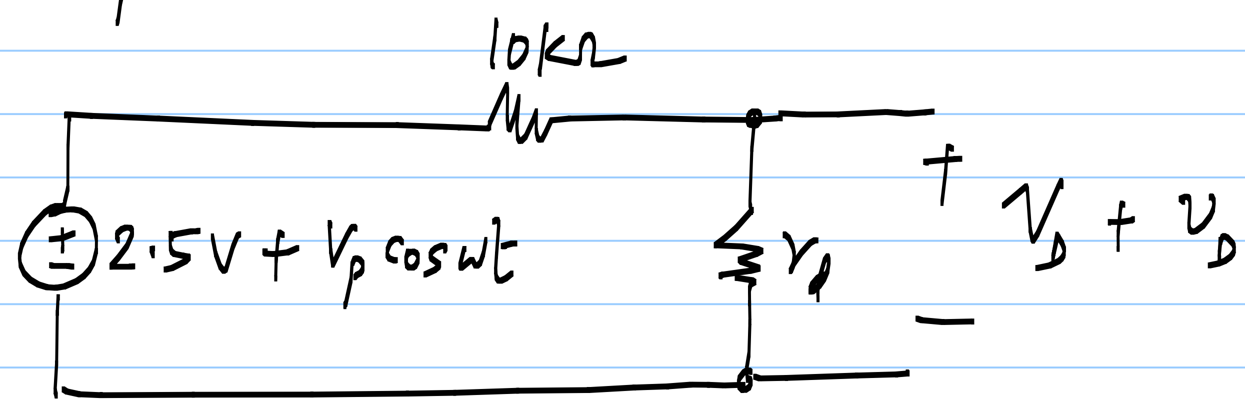
DC operating point analysis:



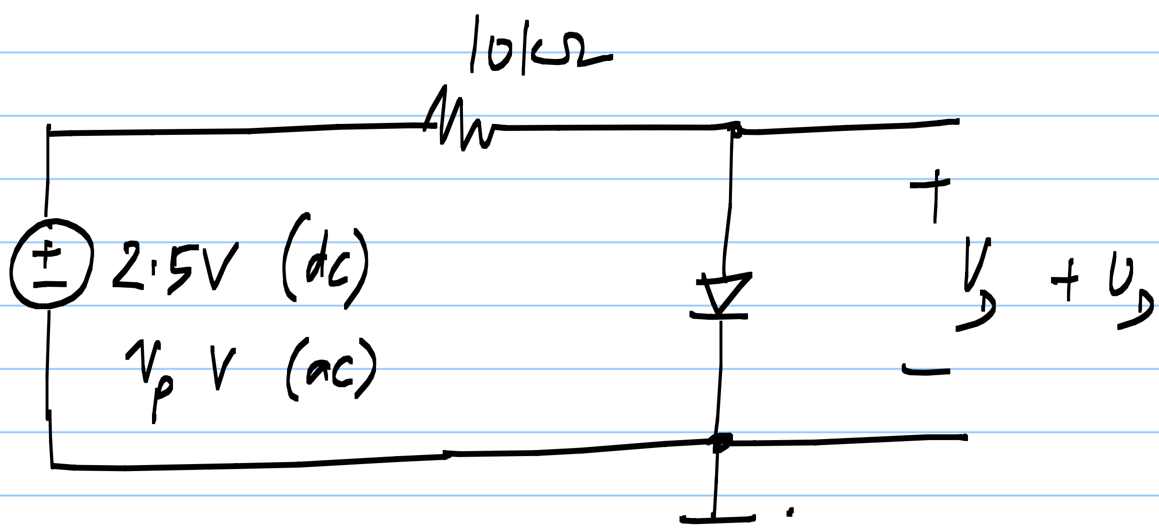
* Do .op analysis & determine V_D, I_D

Small signal incremental analysis:
(Hand calculation)

* Determine the small signal equivalent of the diode.
* Analyze the circuit below.



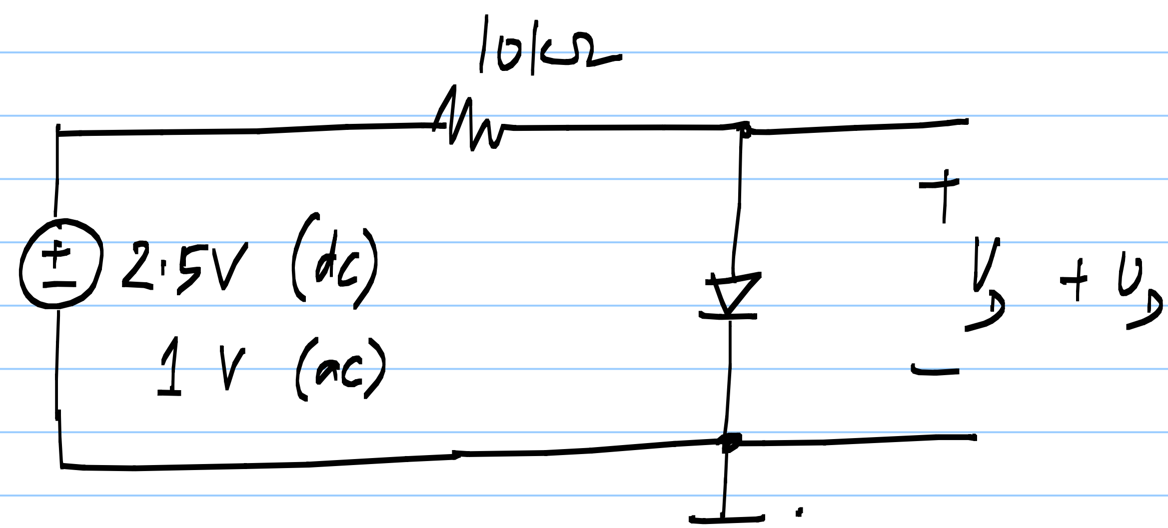
Small signal incremental analysis (simulation)



Compare results to hand calculations

* Do .op & .ac analyses & determine V_D & v_D
* Can do .ac over a range of frequencies.
* What happens if v_p (ac magnitude) is changed?

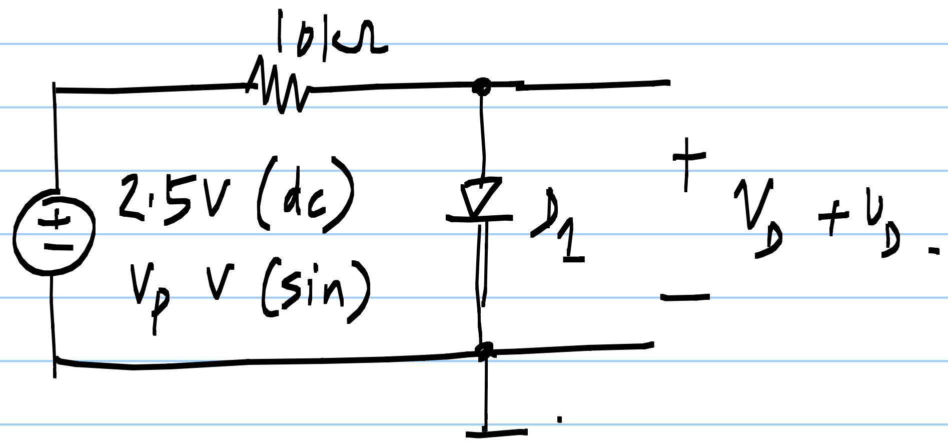
Small signal incremental analysis (simulation)



* Common to use 1V (ac)

* The ac part of the output is the "transfer function" from the input to the output

Large signal analysis with time varying signals



Transient analysis:
can't do by hand – this is why we use a simulator

* Use a small V_p (~ 10mV) and do transient analysis.
* Increase the amplitude and see what happens to the output increment.

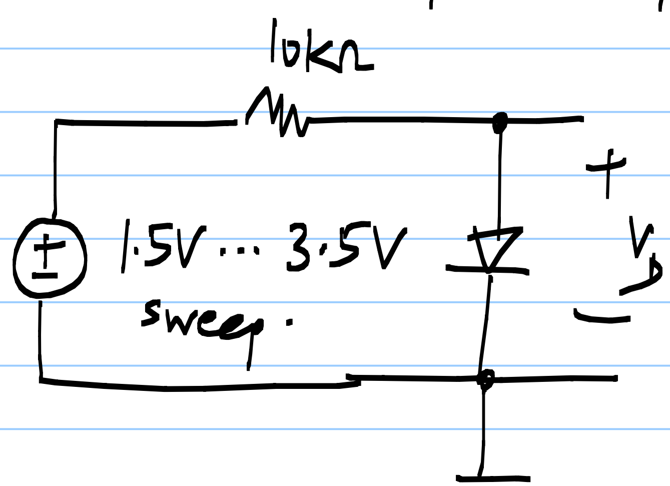
Basic analyses available in SPICE like simulators:

* DC analysis – operating point
* AC analysis – small signal incremental (linear) analysis over the operating point. Also for transfer functions
* TRAN (transient) analysis – Full solution to nonlinear differential equations of the circuit.

Can do all of them in a single run

Other features of a simulator:

* DC sweep: DO dc operating points at multiple input values. Can be used to determine nonlinear input-output (dc) relationships.



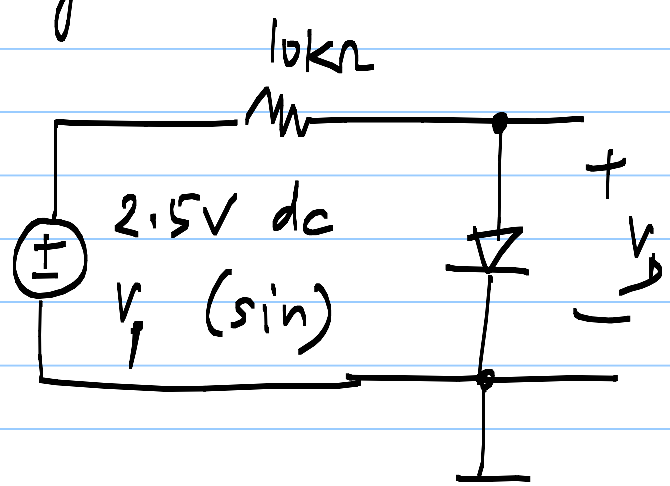
plot V_D vs. V_{in}

plot I_D vs. V_D

Other features of a simulator:

* Parametric sweep: Can do analyses for different component values.

e.g.:

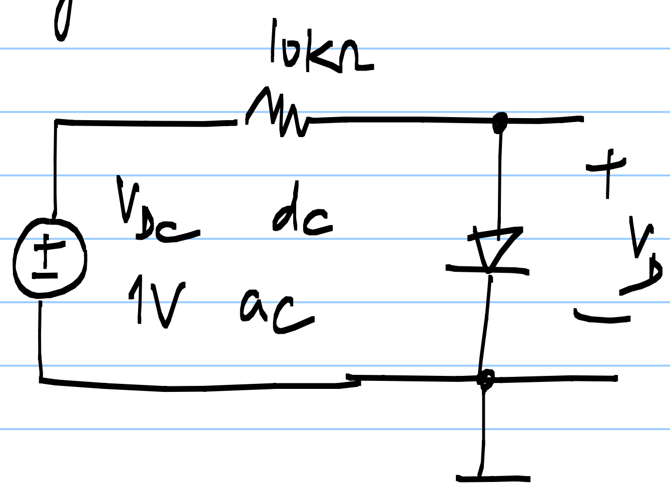


Sweep v_p from 10mV to 1V in decade steps & do transient analysis

Other features of a simulator:

* Parametric sweep: Can do analyses for different component values.

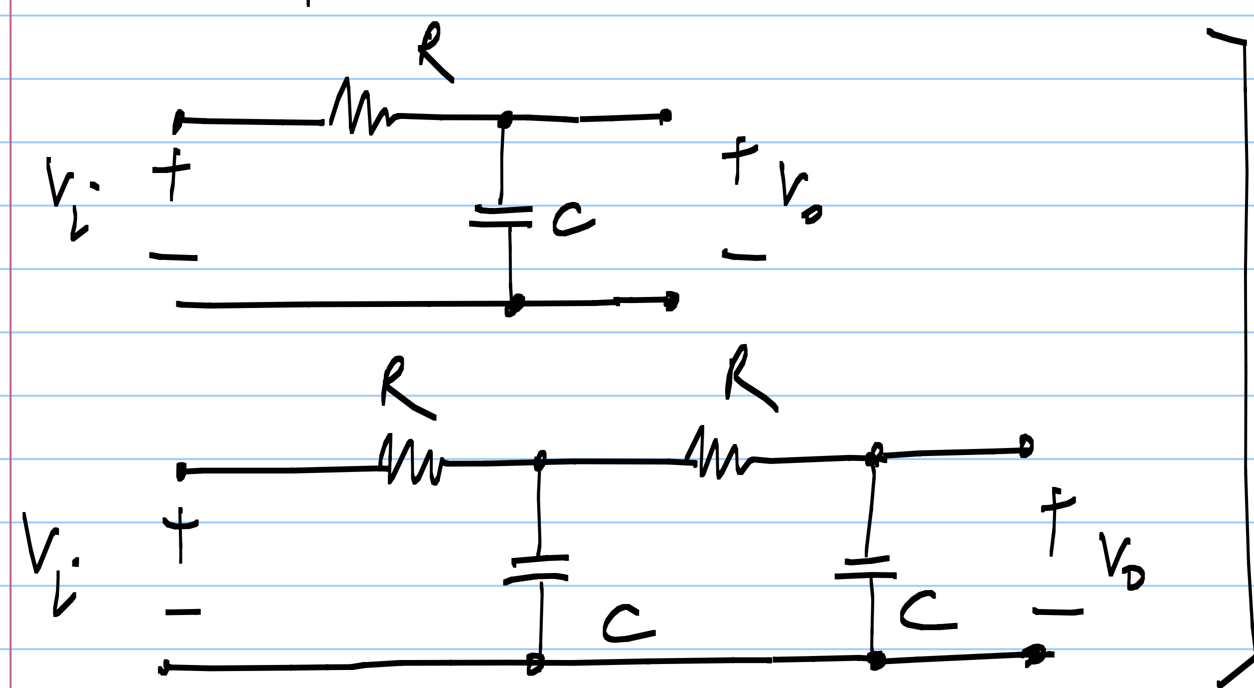
e.g.:



Sweep V_{DC} from 1V to 3V in 1V steps & do ac analysis

Other exercises:

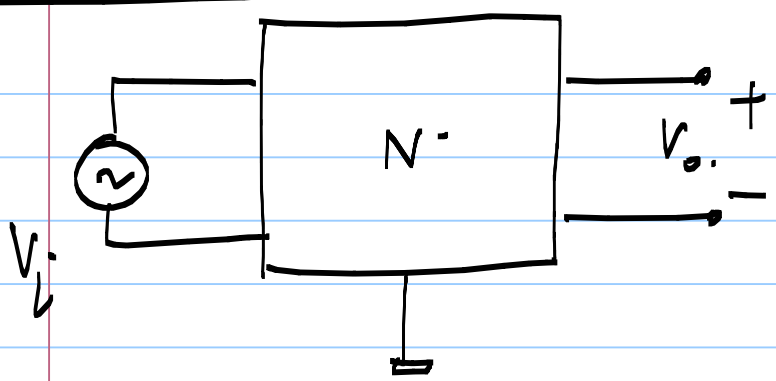
ac analysis for transfer functions:



Plot

$$\left|\frac{v_o}{v_i}\right|, \quad \angle\frac{v_o}{v_i}$$

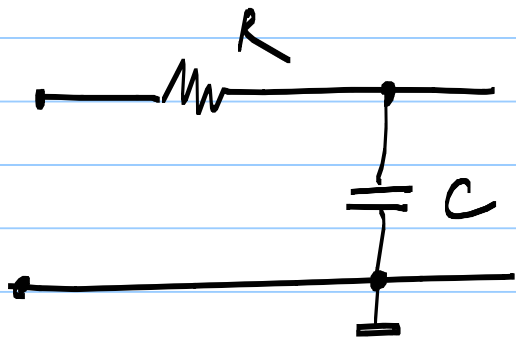
Noise simulation:



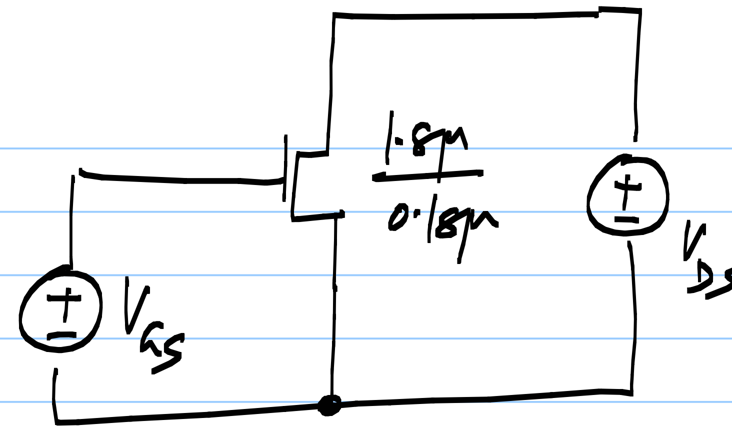
* Can simulate the output noise & input referred noise spectral densities & integrated noise

* Need to specify
  - output terminals across which the noise voltage is measured (In most simulators, you can't specify a current. Need to convert using a CCVS)
  - input source to which the noise is referred back
  - Range of frequencies

Noise simulation exercises :

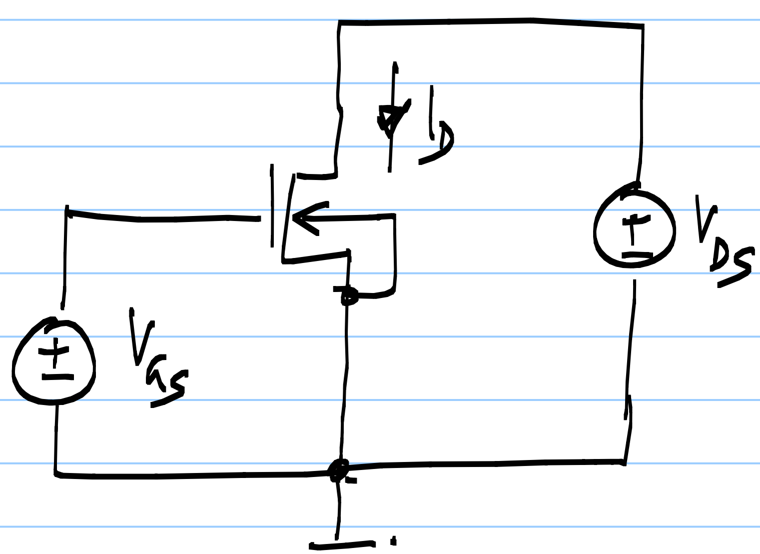


- Verify input referred noise PSD & integrated output noise



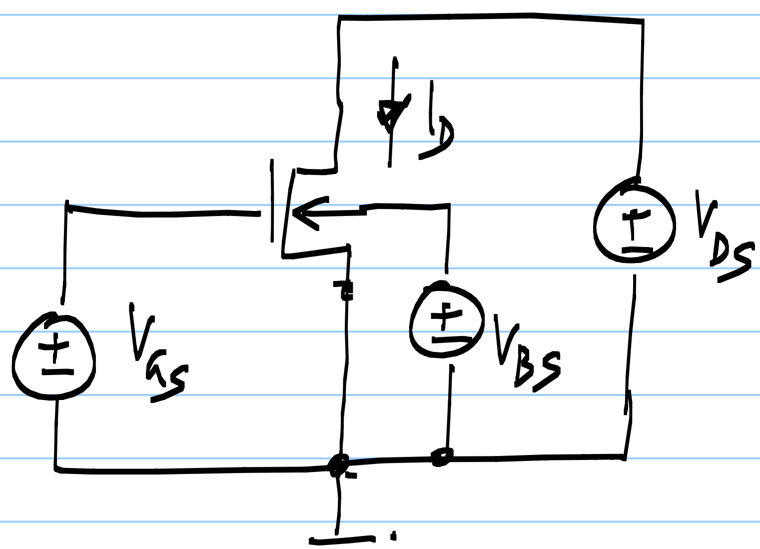
- Bias the MOS transistor at $V_{GS} = 0.7V$, $V_{DS} = 0.7V$ & measure S_{I_D}. Verify thermal noise PSD & flicker noise corner
- Change to $\frac{3.6\mu}{0.36\mu}$ etc. & see

Transistor characteristics:



* Plot I_D vs. V_{DS} for different values of V_{GS}
* Try $\frac{W}{L} = \frac{1.8\mu}{0.18\mu}, \frac{3.6\mu}{0.36\mu}, \frac{5.4\mu}{0.54\mu}, \frac{7.2\mu}{0.72\mu}$
* Estimate μC_{ox}, V_T

Transistor characteristics:



* Plot I_D vs. V_{GS} for different values of V_{BS} (<0) (Fix V_{GS})

* Estimate $\left(\frac{g_{mbs}}{g_m}\right)$