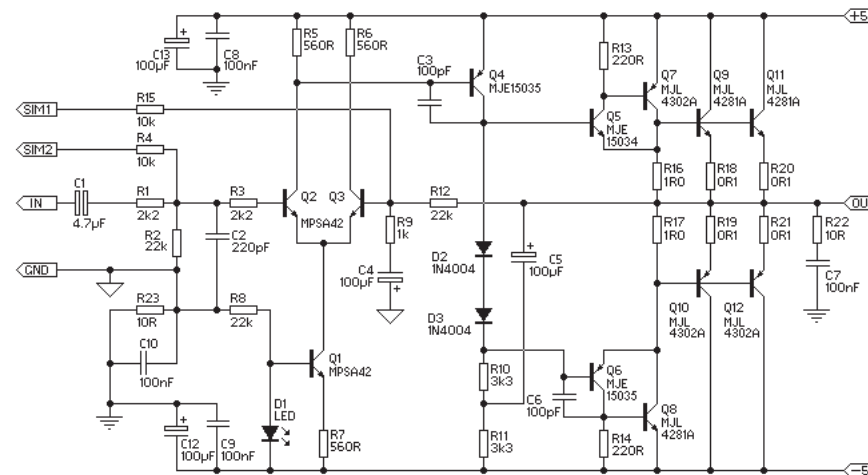
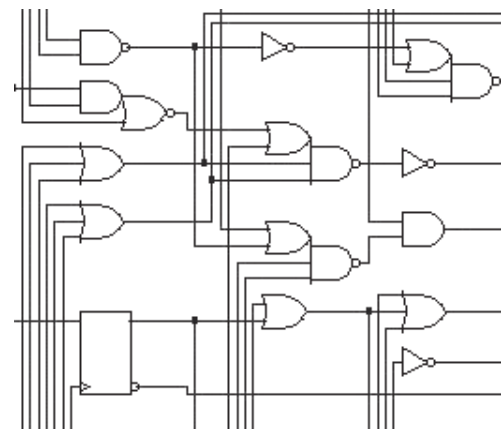


# Analog Simulation

- **Digital simulation**
  - **discrete values**
    - bit, boolean, enumerated, integer
    - exception - floating point
  - **discrete timing**
    - cycle based - uniform time intervals
    - event based - nonuniform time intervals
  
- **Analog simulation**
  - **continuous values**
    - represented as floating point numbers
  - **continuous timing**
    - time or frequency domain





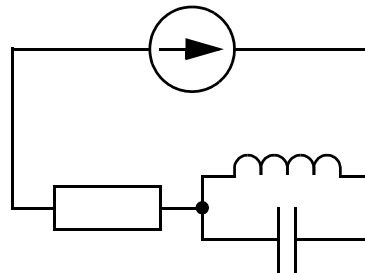
# Analog Simulation

- **Low abstraction level descriptions**
  - detailed delay, power consumption, etc. estimation
  - transistors, polygons, etc.
- **Netlist of components**
  - Differential equations to describe components
  - Small signal modelling
    - frequency domain (Fourier transformation)
  - Large signal modelling
    - time domain (Laplace transformation)
- **SPICE**
  - Simulation Program with Integrated Circuit Emphasis
  - Donald O. Pederson (Prof. Emer. of UC, Berkeley)



## Electric Circuit Model

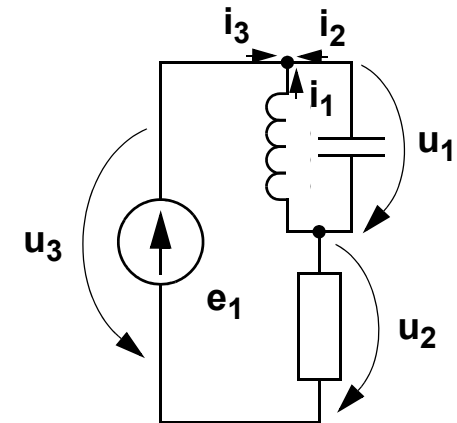
- **Circuit – graphical model**
  - **node** – connection point of three or more components
  - **branch** – path between two nodes
  - **loop** – closed path in a circuit
- **Series connected components – constant current**
- **Parallel connected components – constant voltage**





## Kirchoff's Laws

- **Kirchoff's first law**
  - applied onto nodes
  - the sum of currents entering a node is zero
  - $i_1 + i_2 + i_3 = 0$
- **Kirchoff's second law**
  - applied onto loops
  - the sum of voltage drops of elements in a loop is equal to the sum of electromotoric powers in the loop
  - $u_1 + u_2 = e_1$
  - $u_3 = e_1$





## Circuit Calculation

- **Set of equations**
- **How to select loops?**
- **$p$  branches and  $q$  nodes -->  $p$  variables**
  - **$p'$  – independent current sources**
    - or the current does not depend on the voltage of the source
  - **$q-1$  equations based on the Kirchoff's first law**
  - **$p-p'-(q-1)$  equations based on the Kirchoff's second law**
- **Inductors and capacitors --> differential equations**
  - **$C = q_C / u_C$        $i_C = dq_C/dt = C(du_C/dt)$**
  - **$L = \Psi_L / i_L$        $u_L = d\Psi_L/dt = L(di_C/dt)$**

# Circuit Calculation

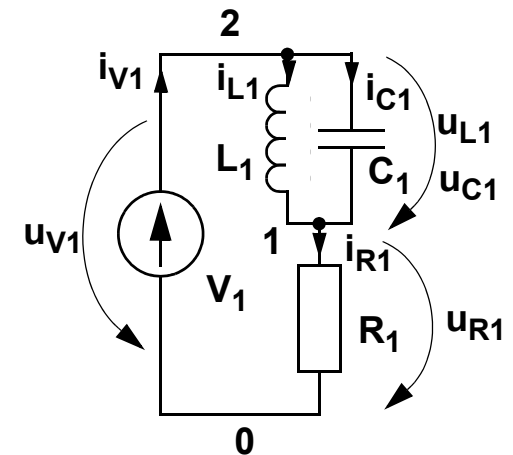
- **Kirchoff's first law**

- (trivial)node 0 -  $i_{R1} = i_{V1}$
- node 1 -  $i_{L1} + i_{C1} = i_{R1}$
- node 2 -  $i_{V1} = i_{L1} + i_{C1}$

- **Kirchoff's second law**

- loop 1 -  $u_{L1} = u_{C1}$
- loop 2 -  $u_{V1} = u_{L1} + u_{R1}$
- loop 3 -  $u_{V1} = u_{C1} + u_{R1}$

- $p = 3, \quad q = 2, \quad p' = 0$

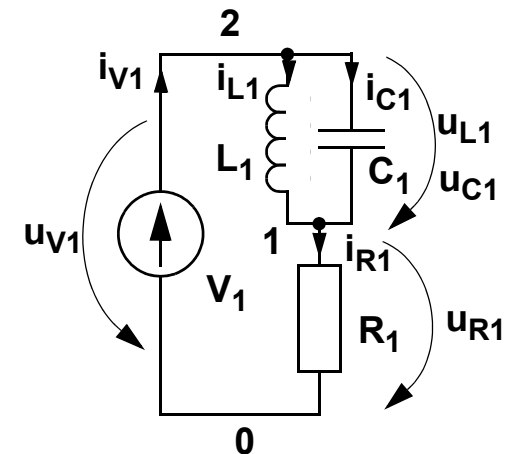


## Circuit Calculation

- $p = 3, \quad q = 2, \quad p' = 0 \quad \rightarrow 1 \text{ K1} \ \& \ 2 \text{ K2}$

- $i_{C1} = C1 \cdot (du_{C1}/dt)$
- $u_{L1} = L1 \cdot (di_{L1}/dt)$
- $u_{R1} = R1 \cdot i_{R1}$
- $e_{V1} = u_{V1}$

- 1:  $i_{L1} + C1 \cdot (du_{C1}/dt) = i_{R1}$
- 2:  $e_{V1} = L1 \cdot (di_{L1}/dt) + R1 \cdot i_{R1}$
- 3:  $L1 \cdot (di_{L1}/dt) = u_{C1}$





## Complex Representation

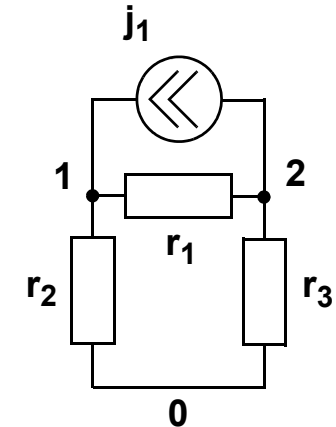
- $u = U_m \cdot \sin(\omega \cdot t + \psi) = J_m(\bar{U}_m \cdot e^{j \cdot \omega \cdot t})$
- $du/dt = J_m((j \cdot \omega \cdot \bar{U}_m) \cdot e^{j \cdot \omega \cdot t})$
- Ohm's law –  $\bar{U} = \bar{I} \cdot Z$  –  $Z = \bar{U} / \bar{I}$
- Resistor –  $i = u / R$  –  $Z = r$
- Capacitor –  $i = C (du/dt)$  –  $Z = 1/(j \cdot \omega \cdot C)$
- Inductor –  $u = L (di/dt)$  –  $Z = j \cdot \omega \cdot L$





## Node Voltage Method

- **$q$  nodes**
  - **$q-1$  equations – base node, Kirchoff's first law**
- **Matrices -  $(u) = (g)^{-1} \cdot (j)$** 
  - **$(u)$  - node voltages, vector  $[q-1]$**
  - **$(g)$  - conductivity between nodes, matrix  $[(q-1) \times (q-1)]$**
  - **$(j)$  - current sources (entering the nodes), vector  $[q-1]$**
- **$(u)^T = [ (u_{10}) (u_{20}) ]$**
- **$(j)^T = [ (j_1) (-j_1) ]$**
- **$(g)$ :**
  - **$g_{11} = (1/r_1) + (1/r_2)$**
  - **$g_{12} = -(1/r_1)$**
  - **$g_{21} = -(1/r_1)$**
  - **$g_{22} = (1/r_1) + (1/r_3)$**





## Node Voltage Method

- $(\mathbf{j}) = (\mathbf{g}) \cdot (\mathbf{u})$        $(\mathbf{u}) = (\mathbf{g})^{-1} \cdot (\mathbf{j})$

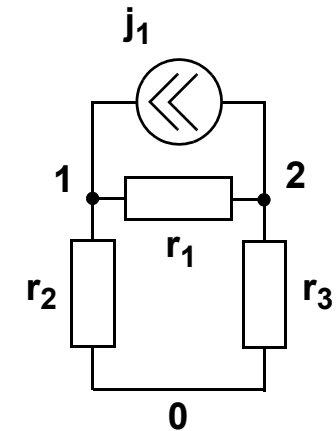
$$\begin{bmatrix} u_{10} \\ u_{20} \end{bmatrix} = \begin{bmatrix} \left(\frac{1}{r_1}\right) + \left(\frac{1}{r_2}\right) & -\left(\frac{1}{r_1}\right) \\ -\left(\frac{1}{r_1}\right) & \left(\frac{1}{r_1}\right) + \left(\frac{1}{r_3}\right) \end{bmatrix}^{-1} \times \begin{bmatrix} j_1 \\ -j_1 \end{bmatrix}$$

- $\mathbf{A}^{-1} = (1/D) \cdot |\mathbf{A}_{ij}|^T$

$$(\mathbf{g})^{-1} = \left(\frac{1}{D}\right) \times \begin{bmatrix} \left(\frac{1}{r_1}\right) + \left(\frac{1}{r_3}\right) & \left(\frac{1}{r_1}\right) \\ \left(\frac{1}{r_1}\right) & \left(\frac{1}{r_1}\right) + \left(\frac{1}{r_2}\right) \end{bmatrix}$$

## Node Voltage Method

- $u_{10} = (1/D) \cdot (j_1 / r_3)$
- $u_{20} = (1/D) \cdot (-j_1 / r_2)$
- $D = 1 / r_1 r_2 + 1 / r_1 r_3 + 1 / r_2 r_3$
- $r_1=2k\Omega, \quad r_2=0.5k\Omega, \quad r_3=1.5k\Omega$
- $j_1=10mA$
- $D=2.6667$
- $u_{10} = 10 / 1.5 \cdot 2.6 = 2.5 \text{ V}$
- $u_{20} = -10 / 0.5 \cdot 2.6 = -7.5 \text{ V}$





# SPICE

- **Uses numerical techniques to solve nodal analysis of circuit**
  - Textual input to specify circuit & simulation commands
  - Text or graphical output format for simulation results
  - **Circuit elements:**
    - Resistors
    - Capacitors
    - Inductors
    - Independent sources (V, I)
    - Dependent sources (V, I)
    - Transmission lines
    - Active devices (diodes, BJTs, JFETS, MOSFETS)
  - **Analysis types:**
    - non-linear d.c., non-linear transient
    - linear a.c.
    - noise, temperature



## SPICE

- **A wide variety of active device models**
- **Process parameter variation**
- **Effects of worst / best case & statistical spreads of process**
- **Design optimization**
- **Component Libraries**
- **Behavioral modelling**
- **With some versions of SPICE there can be algorithm related problems (e.g., failure to converge to a solution) which require a repertoire of fixes. With HSPICE many of these problems are avoidable.**
- **For effective application, the designer must have accurate MOSFET models for the process being used, and an understanding of the models and their parameters.**



## Input Format

- **A SPICE file is made up of a series of statements**
- **Each statement is on one line, unless continued onto the next by starting it with '+' as the first character**
- **Each statement is made up of fields**
- **Fields are separated by ',', '=' '(' ')'** or one or more spaces
- **Fields consist of SPICE key words, SPICE symbols, names (alpha-numeric up to 16 characters) numbers (integer or floating point) or scale-factors**



## Input Format

- **The scale-factors recognized by SPICE are:**
  - **T = 1E12**            **G = 1E9**            **MEG = 1E6**
  - **K = 1E3**            **MIL = 25.4E-6**        **M = 1E-3**
  - **U = 1E-6**            **N = 1E-9**            **P = 1E-12**            **F = 1E-15**
- **Letters immediately following a number that are not scale factors are ignored, as are letters immediately following a scale factor, so 1K, 1000volts, 1KV, 1.0E3 are all the same**
- **Comment lines start with '\*' or '\$' as the first character; comments on the same line as a SPICE statement must be at the end and be preceded by '\$' (surrounded by spaces)**
- **A SPICE file must start with a title statement and finish with an end statement**
- **The order of statements between start and end is arbitrary**



# SPICE Syntax

- **Description of the circuit - every element has name and every node has label**
- **The general SPICE syntax for a circuit element is as follows:**
  - **NAME node1 node2 ... nodeN <model ref> value <parameters>**
- **The simplest syntax for each element in the above circuit is:**
  - **Resistor**
    - **Rxxx n1 n2 value**
  - **Capacitor**
    - **Cxxx n1 n2 value**
  - **MOSFET**
    - **Mxxx nd ng ns nb mname L=val W=val**
    - **+ <AD=val> <PD=val> <AS=val> <PS=val>**





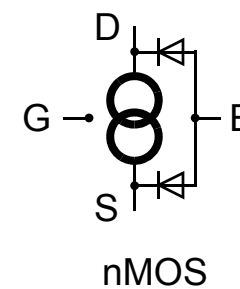
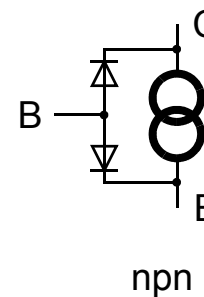
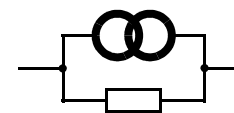
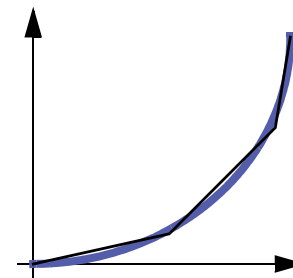
## SPICE Syntax

- **Element identifiers xxx are chosen by the designer and may be numbers or letters**
- **Nodes may be identified by unique numbering; node number 0 is reserved for ground. Alternatively, nodes may be identified by unique names, with gnd reserved for ground.**
- **Note the sequence of MOSFET terminals: drain, gate, source, bulk. L is length, W is width. AD, AS, PD, PS are drain/source areas & perimeters used for estimating capacitances. mname must correspond to a defined model with this identifier.**



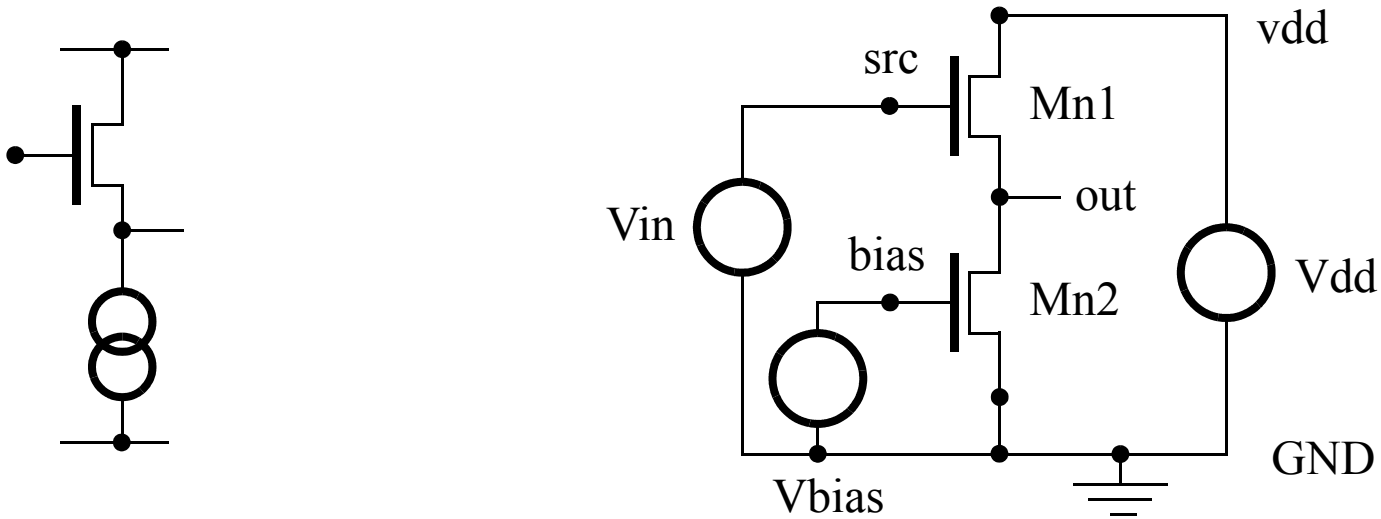
## Non-linear components

- **Non-linear models are replaced with a set of linear models**
  - Newton Raphson algorithm
  - working point determines which of the linear models is used
- **Diodes**
  - resistor + voltage controlled current source
- **Transistors**
  - different models for small and large signals
  - bipolar – diodes + current controlled current source
    - diodes – pn-junctions
  - MOS – diodes + voltage controlled current source
    - diodes – substrate



## Example – SAQ Circuit

- The current sink has been replaced by a transistor and the circuit has been annotated as follows:





## Example – SAQ Circuit

\*single transistor source-follower - dc sweep of voltage I/O - no body effect

\*file:fol-2.hsp replace ideal current sink by single transistor current sink

\*parameters

```
.PARAM ref=2.0
```

\*circuit net list

```
Vdd vdd GND DC 5.0
```

```
Vin src GND DC 0.0
```

```
Vbias bias GND DC ref
```

\*assume twin-tub or p-well process so  $V_{bs}=0$  can be designed

```
Mn1 vdd src out out nmos11 W=8 L=8
```

```
Mn2 out bias GND GND nmos11 W=2 L=16
```

\*options & analysis

```
.OPTIONS POST=2 SCALE=1U
```

```
.DC Vin 0 5 0.1
```

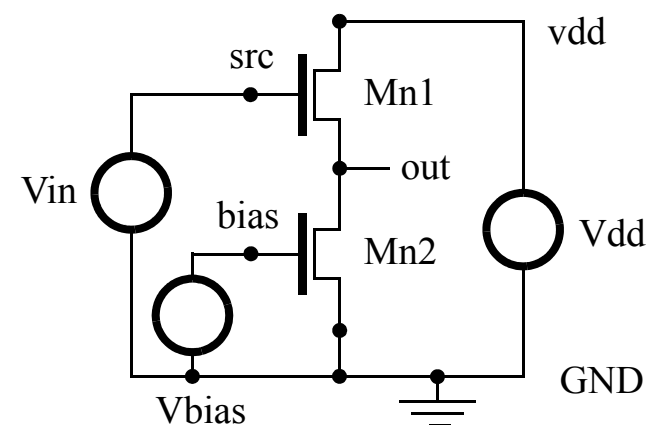
\*output

```
.plot DC v(src) v(out) $cp. Lect1-4.vwg slide No 9
```

\*mosfet models - level 1 included from separate file

```
.INC 'l1typ.inc' $Note this is lower case L 1 ...
```

```
.END
```



## Example #2 – NAND gate with load

### Hands-on exercise

- **NAND gate loaded with an inverter**

- 180 nm technology
- wire length - 10  $\mu\text{m}$

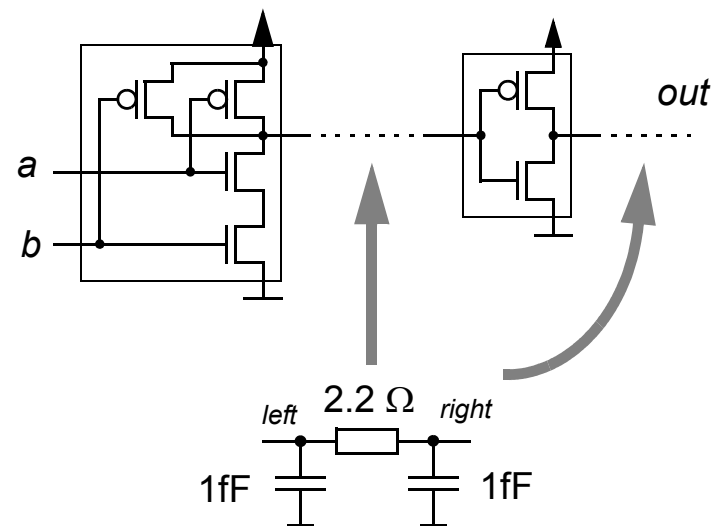
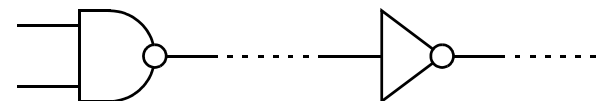
- **Modeling the system**

- **gate models**

- NAND gate - 2 nMOS & 2 pMOS transistors
- inverter - 1 nMOS & 1 pMOS transistors
- $L = 180 \text{ nm}$  &  $W \geq 360 \text{ nm}$

- **wire model**

- $\Pi$ -model - 1 resistor & 2 capacitors
- $10.0 \cdot 0.36 \mu\text{m} \rightarrow 2.2 \Omega$  &  $2 \text{ fF}$



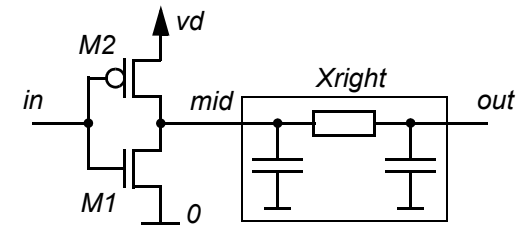
## Example #2 – NAND gate with load

- **Wire model**

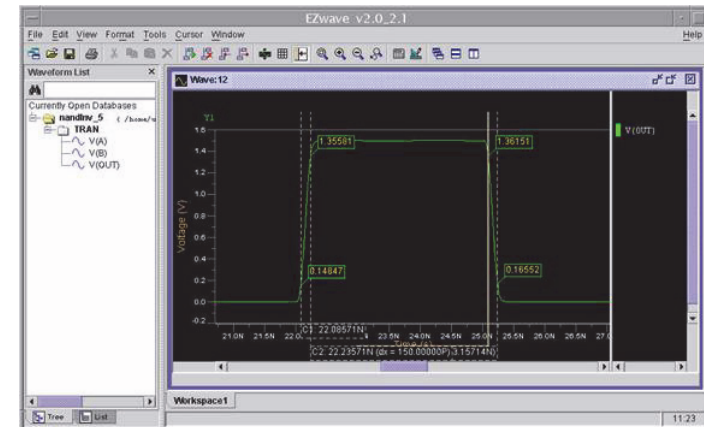
```
.SUBCKT wire left right
R1 left right 2.2
C1 left 0 1f
C2 right 0 1f
.ENDS
.end
```

- **Invertor with wire**

```
.INCLUDE wire.cir
.MODEL nM NMOS LEVEL=2
.MODEL pM PMOS LEVEL=2
Vdd vd 0 1.5
M1 mid in 0 0 nM L=180n W=360n
M2 mid in vd vd pM L=180n W=936n
Xright mid out wire
Vin in 0 pulse(0 1.5 0 10p 10p 5n 10n)
.TRAN 1ps 50ns
.plot TRAN v(in) v(out)
.end
```



Add the NAND gate and get slopes right...



# How to optimize?

## Complexity of the task...

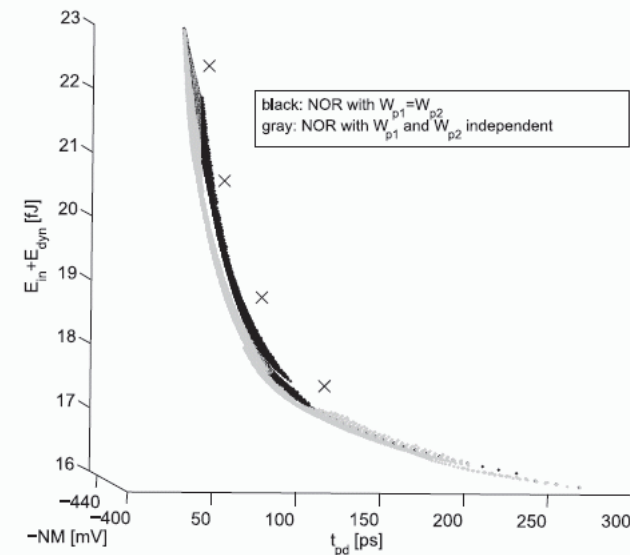
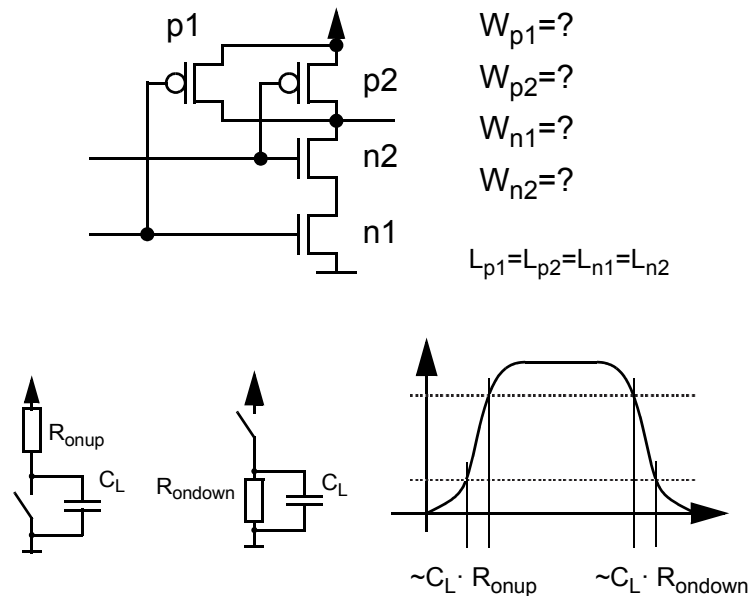


Fig. 6. Pareto fronts of the 65nm CMOS NOR gate considering the optimization variables  $W_n$ ,  $W_{p1}$  and  $W_{p2}$

- Matthias Blesken, Ulrich Rückert, Dominik Steenzen, Katrin Witting, Michael Dellnitz, "Multiobjective Optimization for Transistor Sizing of CMOS Logic Standard Cells Using Set-Oriented Numerical Techniques." The 27th Norchip Conference, Trondheim, Norway, Nov. 2009.**

# How to optimize?

## Approximation... a.k.a. Why NAND?

- Mobility –**

$$\mu_n = 1250 \text{ cm}^2 / \text{V sec} \quad \& \quad \mu_p = 480 \text{ cm}^2 / \text{V sec}$$

- depends on the actual technology

- $R \sim \mu^{-1}$     &     $R \sim L w^{-1}$  (L-constant)

