



Institut
Mines-Telecom

Electronique des Systèmes Embarqués

Chadi Jabbour

Electrical Simulators



What is an Electrical simulator

Electrical simulator

Is a class of software that allows to simulate the behavior of an Electrical circuits

Difference with a simple programming language (C, python ...)

The main differences are that an electrical simulator

- ▶ is able to emulate simultaneous simulation
- ▶ is able to emulate continuous time simulation
- ▶ allows to obtain for both voltage and current

What are the applications for electrical simulation

Electrical Simulators have two main applications:

- ▶ Simulate circuits in order to build an application-specific integrated circuits (ASIC). This requires to have the design kit of the technology ...
- ▶ Simulate systems built using circuits of the Shelf (COTS).

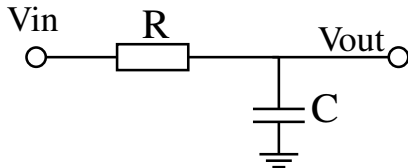
There are many simulators types, there are classified depending

- ▶ Applications: more suited to RF, analog or mixed signals, designing PCB ...
- ▶ Price
- ▶ Operating system
- ▶ ...

Famous simulators

- ▶ Spectre - Cadence
- ▶ Calibre - Siemens
- ▶ LTspice - Analog Devices
- ▶ NGspice (Open Source)

Exercice 1: Transient simulation of an RC filter



- ▶ Instanciate, Resistor, capacitor, input source and ground
- ▶ Connect them and name nets
- ▶ Configure the components
- ▶ Configure the simulation, launch it and analyze the results

Analysis types

- ▶ Transient: calculates a circuit's response over a period of time defined by the user.
- ▶ AC (Alternative current) or small analysis: calculates the frequency behavior of a linearized version of the circuit
- ▶ DC (direct current): calculates the DC operating point of the circuit.
- ▶ Noise is a small signal analysis which calculates the output noise observed in a circuit.

Input types in LTspice

- ▶ DC: Constant voltages used to generate supply and reference voltages
- ▶ Sinewave: Sinewave at given frequency used mainly for the circuit input
- ▶ Pulse: Pulse is square wave whose on and off voltages and duration can be adjusted. It is used mainly for clock and control signals
- ▶ PWL (piece wise linear): PWL source allows to describe a transient voltage waveform by connecting linearly time-voltage pairs. It is used for custom signals
- ▶ Imported source: used to import external signals such as audio. Is instanced using a LTspice directive

Reliability and accuracy

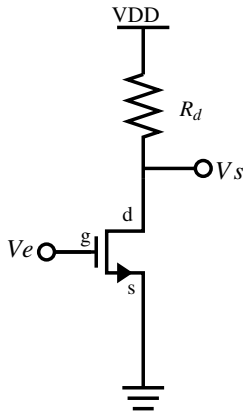
An electrical simulator could give a “good” result for a bad circuit or a “bad” result for a “good” circuit! It is really important to not trust the simulator blindly

The inaccuracy could be

- ▶ Problem of validity of the model
- ▶ High ratio of time constants (ratio between the highest and the lower frequency in the circuit)
- ▶ Unsuiteness of the analyses (for example: simulating non linearity with AC simulation)

It is mandatory to understand the circuit behavior before simulating it!!!!

Exercice 2: DC and AC analysis of an Amplifier

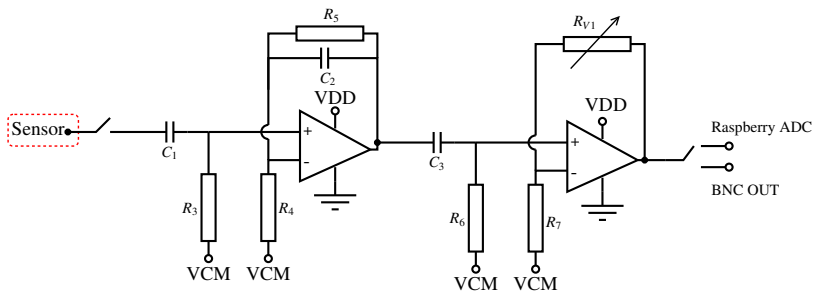


- ▶ Instanciate NMOS transistor, resistor ($1\text{ k}\Omega$) and $V_{DD}=5\text{ V}$
- ▶ Connect them and name nets
- ▶ Perform a DC analysis of V_e between 0 and 5 V
- ▶ Perform an AC analysis of V_e between 1 kHz and 1 MHz

Heart rate sensor Acquisition chain

- ▶ We use a photoplethysmographic sensor
- ▶ Frequency between 0.8 to 3 Hz
- ▶ A variable gain with a factor 10
- ▶ The output should be comprised between 0 to 4 V to be suited for the ADC

Exercice 3: build the amplifier/filter for the heart rate sensor



- ▶ Analyze the circuit
- ▶ Implement it on LTspice
- ▶ Transient simulation
- ▶ AC simulation
- ▶ Parametric simulation with R_{V1}

R_3	R_4	R_5	R_6	R_7
100k	10k	680k	100k	10k
RV_1	C_1	C_2	C_3	VCM
100k	2.2u	68n	2.2u	2 V

Shortcuts:

Copy	Move	Drag	delete	Draw wire	Label net
ctrl+c	F7	F8	del/suppr	F3	F4
Inductor	Capacitor	resistor	Component	Ground	Properties
L	C	R	F2	g	Right click

Units:

10^{-12}	10^{-9}	10^{-6}	10^{-3}	10^3	10^6	10^9
p	n	u	m	k	meg	g

Parametric Simulation :

.step param RV $\underbrace{10k}$ $\underbrace{100k}$ $\underbrace{10k}$
 Initial Value Final Value step

Merci pour votre attention

Questions ?