TELECOM Paris

IP PARIS

Institut Mines-Télécom

Electronique pour la conception des systèmes embarqués

Electrical simulators

Chadi Jabbour

Année scolaire 2024-2025

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 langage (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 systmes built using circuits of the Shelf (COTS).

Simulator types

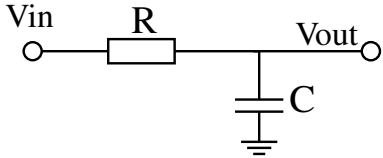
There are many simulators types, there are classified depending

- 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



- Instanciante, 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 instancied using a LTspice directive

Accuracy and reliability of the results

Reliability and accuracy

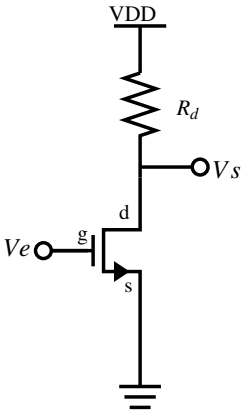
An electrical simulator could give a “good” result for a bad circuit or a “bad“ result for a ”good“ circuit ! It is really important to not trust the simlator blindly

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)
- Unsuitedness 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

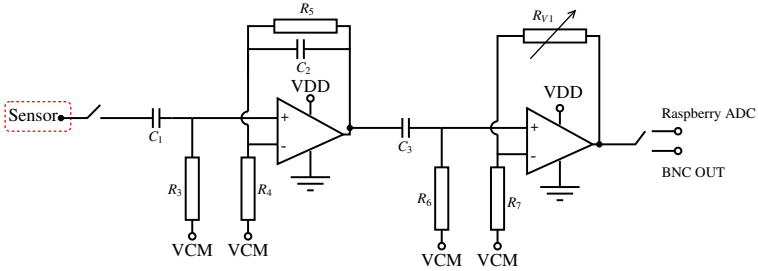


- Instanciante NMOS transistor, resistor (1 kΩ) and VDD=5 V
- Connect them and name nets
- Perform a DC analysis of Ve between 0 and 5 V
- Perform an AC analysis of Ve between 1 kHz and 1 MHz

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 RV_1

Values

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

LTspice help

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 $\underbrace{10k}_{\text{Initial Value}}$ $\underbrace{100k}_{\text{Final Value}}$ $\underbrace{10k}_{\text{step}}$

Merci pour votre attention

Questions ?