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 cirucits

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 cicruits of the Shelf (COTS).

Simulator types

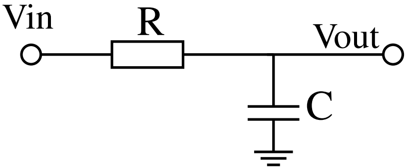
There are many simulators types, there are classified depending

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

Famous simulators

- Cadence
- Mentor
- Orcade
- Ltspice

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

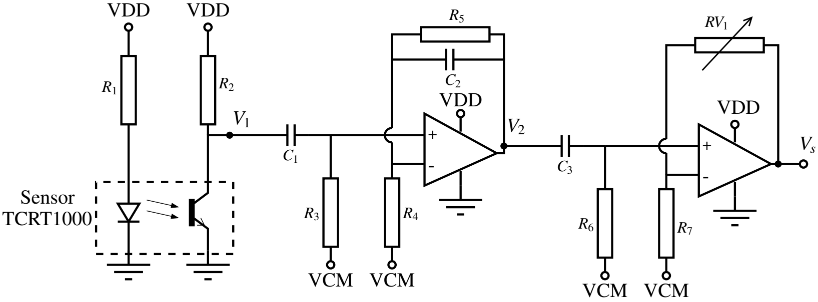
- Problem of validity of the model
- High ratio of time constants (ratio between the highest and the lower frequency in the circuit)
- Unsuitedness of the analyses (for example: simulating non linearity with AC simulation)

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

Heart rate sensor Acquisition chain

- We use a photoplethysmographic sensor TCRT1000
- Frequency between 0.8 to 3 Hz
- A gain between 100 to 1000 is needed
- Th output should be comprised between 0 to 4 V to be suited for the ADC

Exercice 2: 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_1	R_2	R_3	R_4	R_5	R_6
150	1k	100k	10k	1meg	100k
R_7	RV_1	C_1	C_2	C_3	VCM
10k	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 $\underbrace{\text{10k}}_{\text{Initial Value}}$ $\underbrace{\text{100k}}_{\text{Final Value}}$ $\underbrace{\text{10k}}_{\text{step}}$

Merci pour votre attention

Questions ?