# Overview of research activities on the simulation of High frequency devices, circuits and systems using the Scilab/Scicos Environment

R. Sommet*, R. Quéré*, E. Ngoya*, S. Mons*, J.C Nallatamby*, T.Reveyrand**, A Mallet**

*IRCOM - UMR CNRS-Université de Limoges IUT GEII, 7, rue Jules Vallès 19100 BRIVE
**CNES- Avenue E. Belin 31000 TOULOUSE

## 1. INTRODUCTION

Interactions between devices and circuits on one hand and circuits and systems on the other hand are key points of the design of high frequency communications or Radar systems. In order to accurately predict those interactions there is a need for high level tools that are able to cope with different kind of formalisms and equations. Up to now only specialized tools are available for the simulation of devices, circuits and systems individually. A major drawback of these tools is their inability to communicate together. So we begin the integration of a number of tools in the high level Scilab/Scicos in order to develop an integrated and open research tool which will allow to make the various softwares to communicate together.
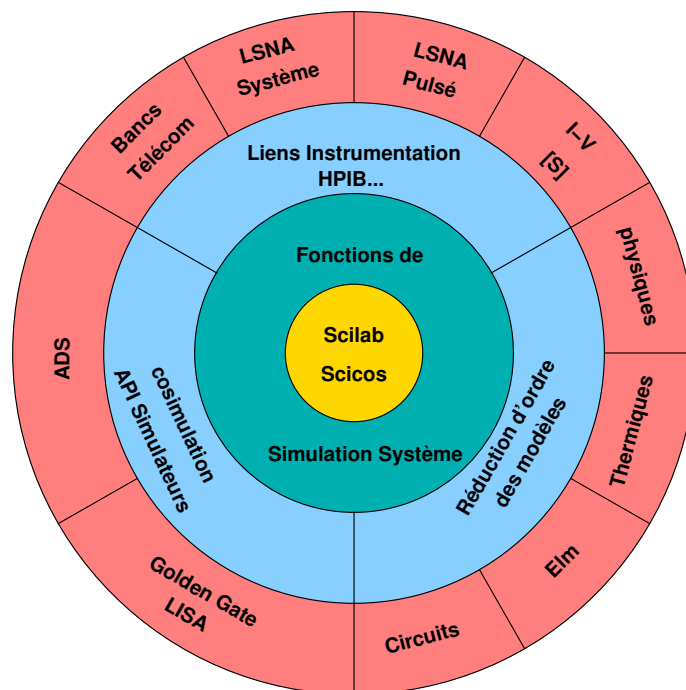


**Figure 1.** General organisation of the research tool for the simulation of devices, circuits and systems

In the general organisation of the previous tool which is represented at Fig-1 it can be seen that Scilab/Scicos constitutes the kernel of this system. In order to make the kernel communicate with device or circuit simulators a set of so-called system simulation functions have to be added to this kernel. Links between those functions and the electomagnetic, thermal or physical models are performed through the use of Model Order Reduction techniques in the linear cases or by direct integration of the simulators in Scilab in the non linear cases as it will be seen in section-2. The integration of commercial high frequency circuit simulation tools will be performed through the call of these tools using API. This will allow the cosimulation of circuits and systems. Finally bridges have to be established between measurement test set-ups and the kernel. This is done by integrating GPIB calls in Scilab functions. In section-2 we present the device circuit interaction where transport equations of semi conductors are embedded in Scilab functions using the `intersci` feature of Scilab. Section-3 will be dedicated to the circuit system interaction by the use of the Scicos environment as a prototyping environment. Emphasis will be put on the ability to simulate mixed (analog-digital) systems such as frequency synthesizers or communication chains.

# 2. DEVICE CIRCUIT INTERACTION

## 2.1. Introduction

Circuit and device interaction determination appears to be one of the major challenge of mobile communication engineering in the next few years [1]. The ability of co-designing simultaneously devices and circuits will be a major feature of CAD tools for the design of MMIC circuits [2]. However, current simulation tools used to simulate the steady state of microwave circuits are often based on the Harmonic Balance (HB) technique and require lumped models for the active devices which are not based on numerical solution of the semiconductor equations and as a consequence are not predictive. They require the active semiconductor components to be characterized in advance and fitted to a lumped equivalent model. The approach presented here deals with the integration of the physical system of semiconductor equations of III-V HBT in a circuit simulator [7] [6]. The main advantages of such simulation tools are

- A direct analysis which take into account of technological parameters on electrical currents voltages, and or any functional values

- A fine description of waveforms as well as a direct access to the internal dynamic of the carriers, ...

- The study of various biasing modes (DC, modulation,...)

- The use of the simulator in term of an investigation tool for circuit with new semiconductor devices

The main drawbacks in this type of simulation rely on the computation time which is prohibitive as well as the large memory which is required. The work presented here deals with the development of our platform for global simulation organized around Scilab. This toolbox integrates circuit simulation, physics based simulation of semiconductors equation [4] as well as the 3D thermal simulation [3].

## 2.2. The toolbox

Scilab is an open and free environment for numerical computation. We have chosen this environment for its ability to link dynamically external functions, for its numerous primitives (graphical, linear algebra, eigenvalues...) and for the advantages it gives to link numerous toolbox including the best linear libraries like ATLAS, ARPACK, UMFPACK, TAUCS... We have implemented our physics-based semiconductor simulator be means of interface functions. These calling functions are used to fill in the jacobian matrices of the problem. SCILAB is used to describe the problem and to solve it by powerful solvers. Many different parameters can be achieved with this toolbox : [S] parameters but also currents, voltages, carrier densities inside active device, under large signal conditions. For example, [S] parameters are obtained using the linear perturbation theory corresponding to small signal AC simulation. We consider a small harmonic perturbation equal to $\Delta P^{j\omega t}$ around a bias point [8]. In this case, we can write the transport partial differential equations in a following form (1):

$$F\left(X, P\right) + f\left(X\right) = 0 \quad with \quad f\left(X\right) = D\frac{\partial X}{\partial t} \quad and \quad X = \overline{X} + \Delta X e^{j\omega t} P = \overline{P} + \Delta P e^{j\omega t} \tag{1}$$

$\overline{X} = \left[\overline{\psi}, \overline{n}, \overline{p}\right]$ is the solution vector of static state. $\overline{P}$ : presents all the constant polarizations.
D : is a diagonal matrix where all the elements corresponding to Poisson's equation are zeros and the other elements are equal to one.
With (1), we obtain:

$$F\left(\overline{X} + \Delta X e^{j\omega t}, \overline{P} + \Delta P e^{j\omega t}\right) + j\omega D \Delta X e^{j\omega t} = 0 \tag{2}$$

Applying a Taylor serial development of one order with two variables we find:

$$F\left(\overline{X} + \Delta X e^{j\omega t}, \overline{P} + \Delta P e^{j\omega t}\right) = F\left(\overline{X}, \overline{P}\right) + \left[F_X\left(\overline{X}, \overline{P}\right) \Delta X + F_P\left(\overline{X}, \overline{P}\right) \Delta P\right] e^{j\omega t} \tag{3}$$

with:

$$F_X\left(\overline{X}, \overline{P}\right) = \frac{\partial F\left(X, P\right)}{\partial X} \ et \ F_P\left(\overline{X}, \overline{P}\right) = \frac{\partial F\left(X, P\right)}{\partial P} \tag{4}$$

because $F\left(\overline{X}, \overline{P}\right)$=0 (a static state), so we solve the following system:

$$\begin{pmatrix} F_X & -\omega D \\ \omega D & F_X \end{pmatrix} \begin{pmatrix} \Delta X_r \\ \Delta X_i \end{pmatrix} = \begin{pmatrix} -F_P \Delta P \\ 0 \end{pmatrix} \tag{5}$$

$F_X$ : jacobian matrix of static state build with the physics based semiconductor simulator.
$F_P$ : partial differential report to the different polarizations.

This linear perturbation theory is not only dedicated to [S] parameters, where the perturbations are only applied at the terminals of the device. In fact, the perturbation can be applied anywhere in the device. So we can

evaluate the effect of a physical noise source connected at each node of the mesh on the macroscopic parameters of circuits (currents and voltages in different semiconductor device terminals) [5]. The implementation of noise sources within the drift diffusion equations [5, 9, 10] can be performed by adding fluctuations in the continuity equations. It is the Langevin approach. The nonlinear approach is more complicated. We write the whole set of equations for each sample of the large applied signals to the device. In the mathematical formulation we obtain at each step of a Newton Raphson algorithm a sparse system that is solved with UMFPACK through the SCISPT interface. Typically A is a matrix of 750x750, M is a very sparse matrix which represents the memory effects of the devices. For N equal 32, the size of the global matrix is about $24000x24000$, with 168192 non zero elements.

$$
\begin{bmatrix}
A_0^n & 0 & 0 & \cdots & 0 & M_{N-1}^n \\
M_1^n & A_1^n & 0 & \ddots & \ddots & 0 \\
0 & M_2^n & \ddots & \ddots & \ddots & \vdots \\
\vdots & \ddots & \ddots & \ddots & \ddots & \vdots \\
\vdots & \ddots & \ddots & \ddots & \ddots & 0 \\
0 & \cdots & \cdots & 0 & M_{N-1}^n & A_{N-1}^n
\end{bmatrix}
\begin{bmatrix}
\Delta X_1^n \\
\vdots \\
\vdots \\
\vdots \\
\vdots \\
\Delta X_{N-1}^n
\end{bmatrix}
=
\begin{bmatrix}
B_1^n \\
\vdots \\
\vdots \\
\vdots \\
\vdots \\
B_{N-1}^n
\end{bmatrix}
$$

This matrix represents the problem for one transistor in a simple circuit. The size of the matrix increases rapidly if we consider severals transistor devices. To solve such a problem we have developed a collaboration with INRIA LIP laboratory of Lyon. It relies on the scheme (client/agent/server) shown Fig-2. The use of parallelism, metacomputing through an ASP (Application Service Provider) approach. More details will be given at the conference. Some results concerning circuit and device co simulation will be also presented.
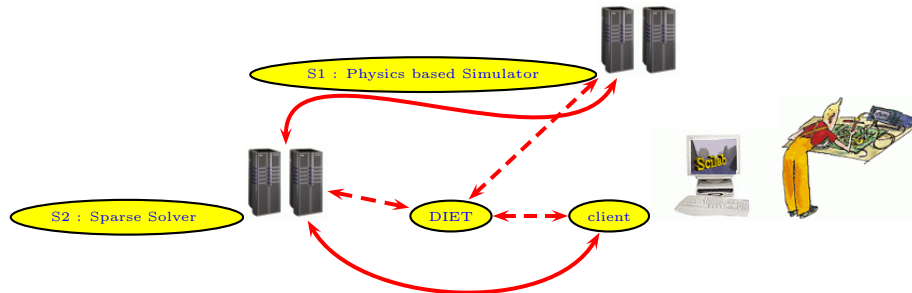


**Figure 2.** The ASP approach

# 3. CIRCUIT SYSTEM INTERACTION

## 3.1. Communication chains

A typical, yet simplified, communication chain is given at Fig-3. This communication chain is constituted of pure analog circuits such as Power Amplifiers (PA), Low Noise Amplifiers (LNA), filters and antenna as well as pure digital circuits such as source and channel coders and decoders. At the interface between those two worlds we find some mixed circuits such as Analog to Digital Converters (ADC) or modulators and Local Oscillators. The Scicos environment allows to describe all those components in a simple way. The option chosen in our lab has been to integrate models already developed in our laboratory as "C" routines linked with Scicos. Beyond the integration of known models an effort has been done to develop specific scopes used in telecommunications designs [11] such as eye-diagram scope, constellation scope, spectrum analyser, probability densities scope...

The development of such a toolbox which will be available soon, has permitted to simulate spatial communications channels in collaboration with the CNES. However calculation of low Bit Error Rates (BER) still remains a real computing challenge due to the huge number of bits errors to calculate ($> 10^8$) so that often pure Monte Carlo approaches are not tractable at all. In order to circumvent this problem a parallelization scheme has been developes using the Parallel Virtual Machine capabilities of Scilab [11]. In this case Scicos is used as a rapid prototyping tool. Indeed the simulations are performed in the Scilab environment in parallel, each simulation running for a specified block of data. This provides a considerable improvement for the total simulation time.

Communications chains are generally strongly affected by the nonlinearity of the Power Amplifier (PA). This circuit exhibits very complex behavior because of the nonlinear memory effects that take place in the amplifier. Models, based on dynamic Volterra series, are able to cope with such effects. So a dynamic nonlinear model of power amplifier has been introduced in the Scicos environment [14] which gives accurate results for the complete simulation.
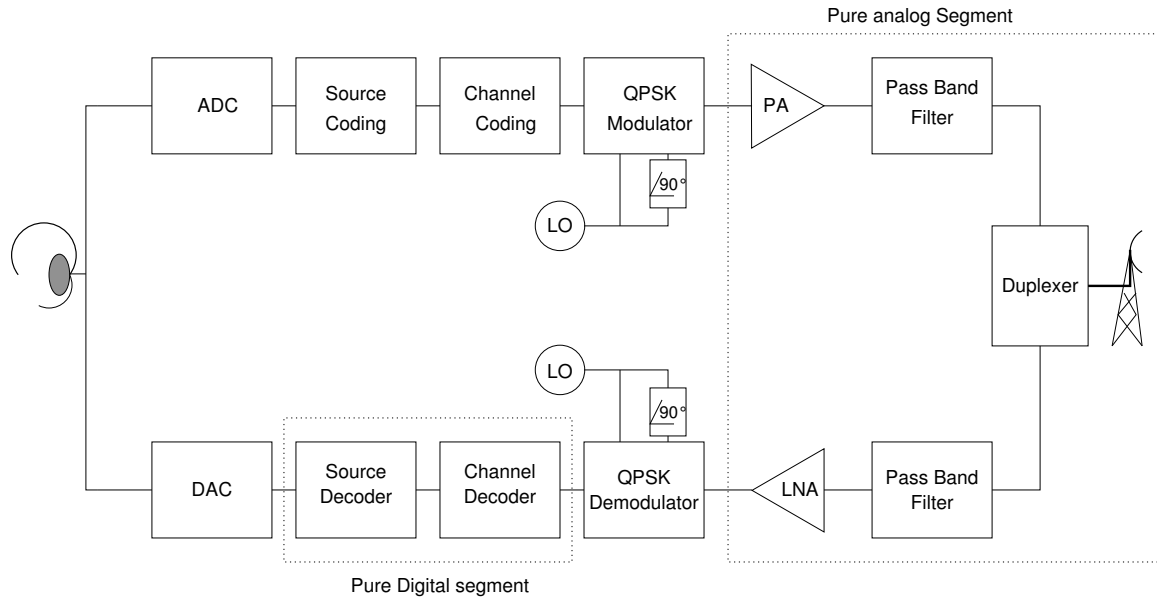
**Figure 3.** Simplified architecture of a communication channel

## 3.2. Frequency synthesizers

Another key circuit of transmissions channels is the frequency synthesizer. Indeed low phase noise of the Local Oscillator is mandatory for minimizing the BER in communications systems or for increasing the spatial and velocity resolution of Radar Systems. While the simulation of phase noise of microwave oscillators is now a well established topic, there is still a lot of work to be performed in order to include those phase noise characteristics into the overall frequency synthesizer which is constituted of both digital frequency dividers, phase-frequency detectors and analog Voltage Controlled Oscillator and loop filter. This kind of circuit is a good example of mixed digital analog circuit. For the modeling of such subsystems the event driven simulation capability of Scicos is very usefull [12] as the overall phase noise can be calculated as the fluctuations of zero crossing times around the perodic sequence $\{T, 2T, \cdots\}$. Thus the phase noise, or equivalently the time jitter, can be simulated by generating random events at the output of the VCO. To do that the dynamic scheduling of events capability of Scicos is used. This kind of simulation provides unique information about the statistical properties of time jitter. The models developped in the framework of frequency synthesizer simulation are used in our laboratory to do research on chaotic self synchronizing chaotic generators in the framework of a collaborative action of CNRS for the use of chaotic techniques in cryptography.

## 3.3. Measurement set-ups

The Scilab environment for circuit-system interactions calculation have to be complemented by measurement interface capabilities. This has lead to the development of a Scilab toolboxes [15] for RF Microwave time domain measurements and analysis [16]. Those toolboxes provide GPIB communications with microwave instruments. Several instrument drivers are available. Most drivers are written in Scilab-language and a GUI has been written using TCL/TK as shown at Fig4. Moreover, new drivers can easily be added with the GPIB toolbox. Several tools are available :

- Noise Power Ratio (NPR) generator (with optimized phase relationship) ;

- Multisine generators ;

- Digital modulations generators (MPSK, MQAM, ...) ;

- Several Analysis : eye's diagram, NPR, Peak to average ratio (PAR), statistic distribution, ... ;

- Files compatibility : CVS, Table, VEE Binblock Complex, S1P and S2P ;

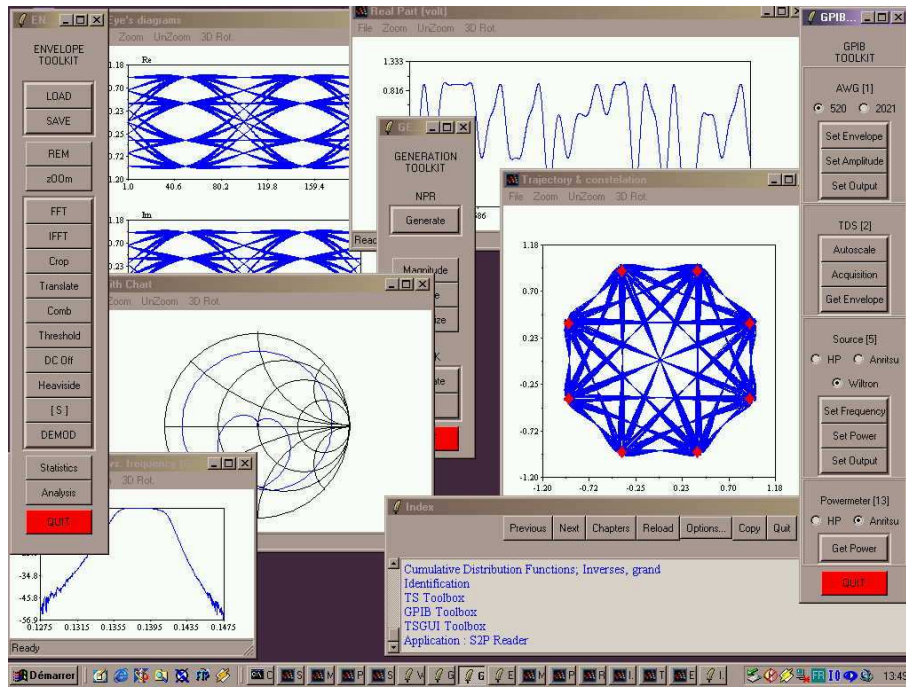- Time-domain envelope calibration procedures ;

**Figure 4.** Screenshot of the Scilab toolbox

## 4. CONCLUSION

Scilab/Scicos has been identified as an efficient tool for high frequency device, circuit and system simulation. Although not presented in this paper it can be used for other applications such as stability analysis [17] or Model Order Reduction [18]. In this paper we have given an overview of the research activities at IRCOM laboratory. IRCOM has got a strong experience since fifteen years in nonlinear circuit simulation and the availability of the Scilab/Scicos environment will allow to develop an open simulation platform for the integration of various tools. To complete this task IRCOM has received strong support from the CNES-Toulouse, the Délégation Générale à l'Armement (DGA) , the CNRS and the Conseil Régional du Limousin. Three PhD sttudents are currently working within this framework.

## REFERENCES

1. R.W. Dutton, Z. Yu "Architecture of TCAD For High Frequency Application" Workshop on Nonlinear CAD MTT-S 96, San Fransisco, June 17 1996, pp 2-26.
2. Z. Yu, R. W. Dutton "Large Signal Analysis of RF Circuits in Device Simulation" Workshop on Nonlinear Modeling and Characterization of Microwave Devices MTT-S 99, L. A., June 1999.
3. R. Sommet , C. Chang, P. Dueme, R. Quéré "Electrothermal Models of Transistors based on finite element analysis for RADAR applications", ITHERM 2004, Las Vegas
4. R. Sommet, D. Lopez, and R. Quéré. Electrothermal harmonic balance simulation of an ingap/gaas hbt based on 3d thermal and semiconductor transport models. In GAAS - Paris, pages 516-519, 2000
5. F. Bonani, G. Ghione, M. Pinto, and R. K. Smith, "An efficient approach to noise analysis through multi-dimensional physics-based models," in *IEEE Trans. Electron Devices*, Vol. 45, Jan. 1998.
6. Z. Riah, R. Sommet, and R. Quéré, "Approche clients-serveurs sur la grille de calcul pour la simulation intégrée", in $13^{eme}$ *Journées Nationales Microondes,* LILLE, 2003.
7. R.Sommet, Y. Perreal, R.Quéré "A direct coupling between the semiconductor equations describing a GaInP/GaAs HBT in a circuit simulator for the co-design of microwave devices and circuits" in *GaAs 97 - Bologne,* september 1997
8. Z. Yu and all, "Part1: Pisces- 2ET 2d Device Simulator," in *ICL, Standford University,* California 94305.
9. J.E. Sanchez, "Semiconductor device simulation of low frequency noise under periodic large signal conditions", University of Florida, 2002
10. Fan-Chi (Frank) Hou, "Low frequency bulk and surface generation recombination noise simulations of semiconductor devices", University of Florida, 2002.
11. A. Layec, R. Quéré, J.C Nallatamby, S. Mons and J. Guittard. "Modelling and simulation of a communication chain in the Scilab/Scicos environment" *Conference Scilab2004*, Rocquencourt, 2-3 december 2004

12. A. Layec, R. Quéré, J.C. Nallatamby, S. Mons, and J. Guittard. ”Behavioral model of integer-N frequency synthesizer”. In *International Workshop on Electronics and System Analysis*, Bilbao Espagne, 2004.

13. P. Casas, J. Portilla, R. Quéré, and A. Mallet. ”Implementation of dynamic Volterra model in system simulation environments.” In *International Workshop on Electronics and System Analysis*, Bilbao Espagne, 2004.

14. A. Bennadji, A. Soury, and E. Ngoya. ”Implementation of dynamic Volterra model in system Analysis.” In *International Workshop on Electronics and System Analysis, Bilbao* Espagne, 2004.

15. T. Reyverand, http://membres.lycos.fr/treveyrand/

16. A. Mallet, F. Gizard, T. Reveyrand, L. Lapierre, J. Sombrin ”A new satellite repeater amplifier characterization system for large bandwith NPR and modulated signals measurements ” *IEEE MTT-S Digest, Vol. 3, pp. 2245-2248*, Seattle, June 2002.

17. Aitziber Anakabe , Juan-Mari Collantes, Luc Lapierre, Alain Mallet, http://www.birp.com/hyper/2004/fr/g_e_confsemin.htm

18. P. Casas, J. Portilla, R. Quéré, A. Mallet, and J.F. Villemazet. ”Model-Order Reduction of Linear and Weakly Nonlinear Time-Varying RF and Microwave Circuits.” *Microwave Theory and Techniques, IEEE Transactions on,* 52(9):2262-2273, 2004.

# Overview of research activities on the simulation of high frequency devices, circuits and systems using the Scilab/Scicos Environment

R. Sommet*, R. Quéré*, E. Ngoya*, S. Mons*, J.C Nallatamby*, T.Reveyrand**, A Mallet**

sommet@brive.unilim.fr

*IRCOM UMR N°6615,IUT GEII, 7, rue Jules Vallès 19100 BRIVE

**CNES- Avenue E. Belin 31000 TOULOUSE

# Outline

- Introduction

- General organisation of the research tool for the simulation of devices, circuits and systems

- Device Circuit Interaction

- Circuit System Interaction

- Measurement setups

- Metacomputing /ASP : GASP project

- Conclusion

# Introduction

Interactions between devices and circuits, circuits and systems are key points of the design of high frequency communications or Radar systems :
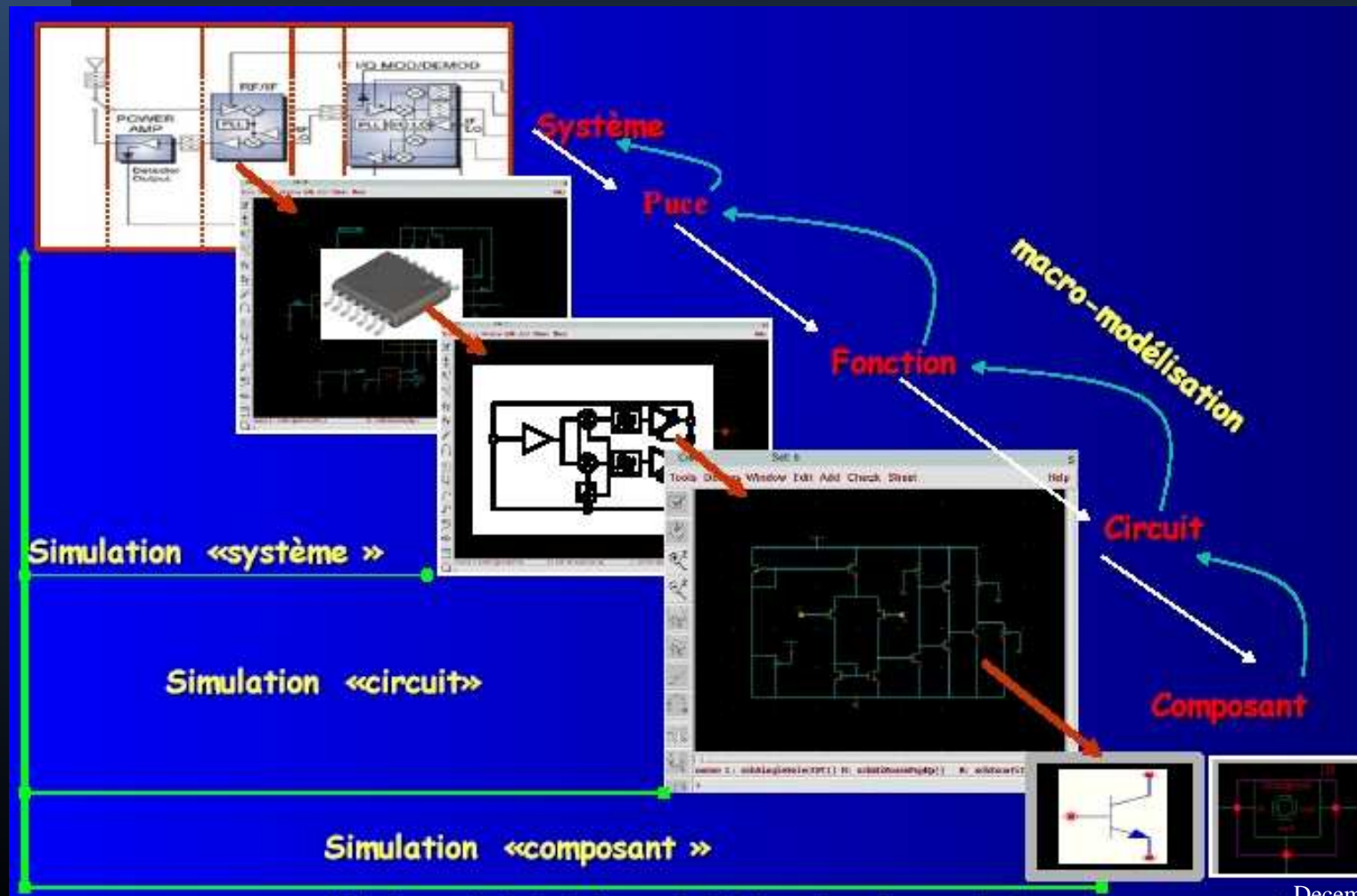
# Introduction

- Need for high level tools
    - Cope with different kinds of formalisms
    - Take into account of different physics-based aspects( semiconductor devices, thermal, electromagnetic,...)
    - Linked to measurements setup
    - cosimulation systems/circuits/devices
- Available tools do not fully correspond to our needs
- Solution : Integration of a number of tools in the high level Scilab/Scicos environment

# Scilab/Scicos environment

Why this choice?

- Offers free and powerful numerical computation capabilities

- An open and extensible computing environment (C, C++, Fortran...)

- High and low level graphical subroutines for post-processing

- Many available various "toolboxes"

- Powerful linear algebra libraries such UMFPACK, ATLAS, ARPACK, TAUCS...

- Easy to interface existing simulators

- Possibility to perform parallel tasks (PVM,...)

# Organisation of research tools
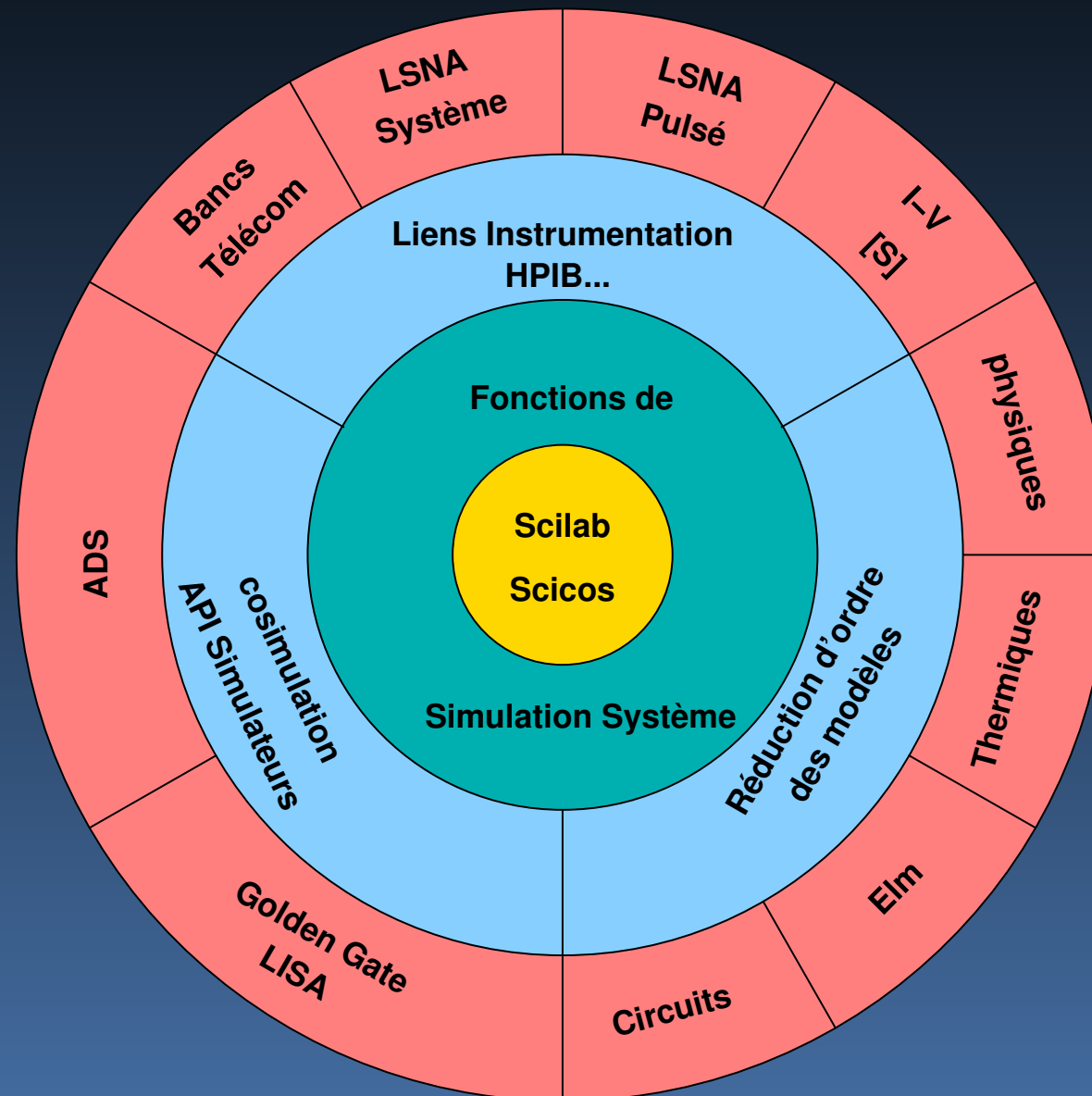
# Key points of the approach

- Allow mixed simulation (event, discrete and continuous)

- Models adopted for analog parts keep the notion of

    - signal/information

    - physics based variables (power, energy, currents, voltages, temperature)

- Cosimulation systems/circuits/devices

- Very large system simulation

    - multi channel systems

    - computation of low Bit Error Rates

# Device Circuit Interaction

Circuit and device interaction determination is one of the major challenge of mobile communication engineering

- **Advantages**
    - A direct analysis which take into account of technological parameters on electrical currents voltages, and or any functional values
    - A fine description of waveforms as well as a direct access to the internal dynamic of the carriers, ...
    - The study of various biasing modes (DC, modulation,...)
    - An investigation tool for circuit with new semiconductor devices
- **Drawbacks**
    - The computation time which is prohibitive
    - The memory required for simulation

# The toolbox

The "intersci" feature of Scilab has been used to interface our physics based device simulator HETSI

- HETSI is one dimensionnal Drift Diffusion Simulator

- Simple and well adapted to HBT transistor

- Available simulation functions

    - I(V) analysis

    - S parameters

    - Noise spectral densities

- Cosimulation of circuit and device interaction

# The Drift Diffusion equations

the nonlinear system to solve

$$-\frac{\partial}{\partial x}\left(\varepsilon\frac{\partial\varphi}{\partial x}\right) = q\left(p - n + N_d - N_a\right)$$

$$\frac{\partial n}{\partial t} = \frac{1}{q}\frac{\partial J_n}{\partial x} + G - R$$

$$\frac{\partial p}{\partial t} = -\frac{1}{q}\frac{\partial J_p}{\partial x} + G - R$$

$$J_n = qn\mu_n nE + qD_n\frac{\partial n}{\partial x}$$

$$J_p = qp\mu_p pE - qD_p\frac{\partial n}{\partial x}$$

# Scilab nonlinear I(V) Simulation

A nonlinear simulation solved in Scilab by a Newton Raphson Algorithm

- The matrix of the problem is build by the call to our simulator HETSI

- The Newton Algorithm is written in the Scilab language

- The Scilab sparse matrix tools are used to solve each linear step of the Newton Raphson algorithm

$$
\begin{bmatrix}
\begin{bmatrix} \times & \times & \times \\ \times & \times & \times \\ \times & \times & \times \\ \times & 0 & 0 \\ \times & \times & 0 \\ \times & 0 & \times \end{bmatrix}^n_n &
\begin{bmatrix} \times & 0 & 0 \\ \times & \times & 0 \\ \times & 0 & \times \\ \times & \times & \times \\ \times & \times & \times \\ \times & \times & \times \end{bmatrix}^n_n &
\begin{bmatrix} \times & 0 & 0 \\ \times & \times & 0 \\ \times & 0 & \times \end{bmatrix}^n & \dots & 0 & 0 \\
0 & . & . & \dots & 0 & 0 \\
. & 0 & . & \dots & . & . \\
0 & . & 0 & \begin{bmatrix} \times & 0 & 0 \\ \times & \times & 0 \\ \times & 0 & \times \end{bmatrix}^n & \begin{bmatrix} \times & \times & \times \\ \times & \times & \times \\ \times & \times & \times \end{bmatrix}^n
\end{bmatrix}
\begin{bmatrix}
(\Delta\psi_1) \\ (\Delta n_1) \\ (\Delta p_1) \\ (\Delta\psi_2) \\ (\Delta n_2) \\ (\Delta p_2) \\ . \\ . \\ (\Delta\psi_{ml}) \\ (\Delta n_{ml}) \\ (\Delta p_{ml})
\end{bmatrix}
= -F^n(1)
$$

# Scilab S parameters simulation

■ A small magnitude perturbation is considered around a bias point.
Pulsation is $\omega$

$$\begin{bmatrix} A & -\omega D \\ \omega D & A \end{bmatrix} \begin{bmatrix} X_r \\ X_i \end{bmatrix} = \begin{bmatrix} -A_p \Delta P \\ 0 \end{bmatrix}$$

■ A : Jacobian matrix of the static case

■ $A_p$ : Partial derivatives versus bias

■ D : Diagonal matrix

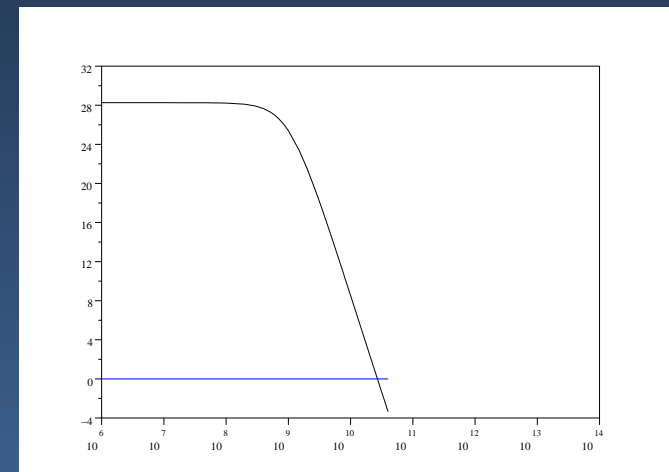■ $\Delta X_r$ et $\Delta X_i$ : Real and Imaginary part of the solution vector

# [S] parameters [1MHz,40GHz]
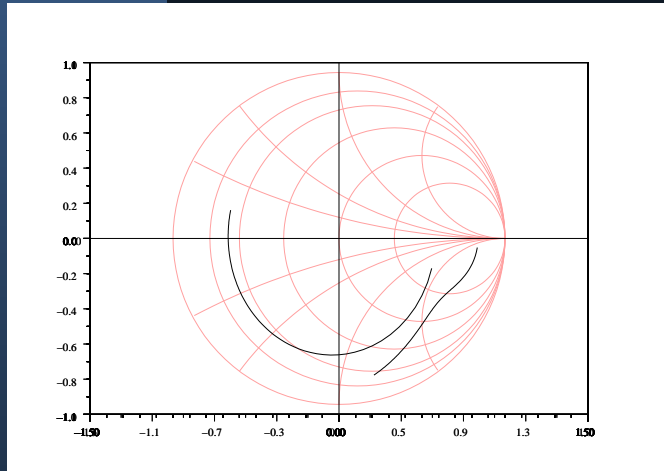


S11 and S22
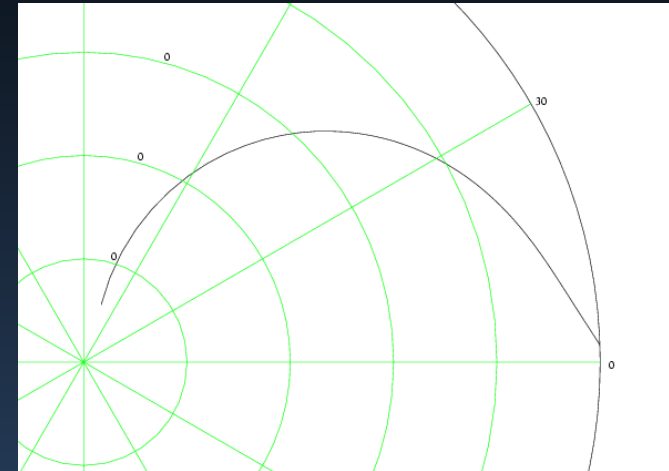


S12
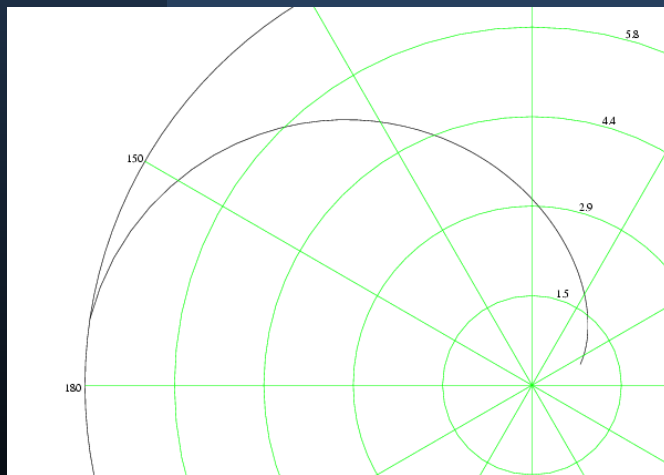


S21



h21 versus frequency

# Scilab Noise Simulation

Based on the Impedance Field method proposed by Schockley, it is possible to compute the Noise Spectral Power Density at the ohmic contact

- Local magnitude of noise is required at mesh nodes Langevin source

- The transfert function between the local node of the mesh and the ohmic contacts, called the Green function is computed

# Some Results : thermal noise

Nyquist relationship

$$S_v(f) = 4kTRe(Z(f))$$

Electrical Model

$$[S_v(f)\delta f]^{\frac{1}{2}}$$
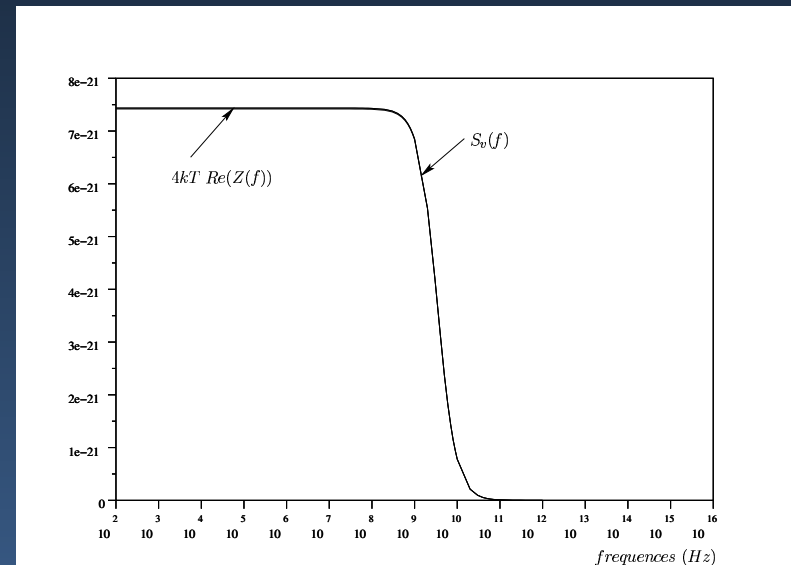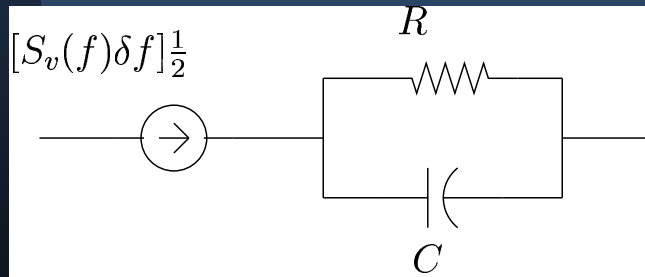


$$Re(Z(f)) = R/(1 + (RC\omega)^2)$$

# Results: Shot noise



(b)



(b)
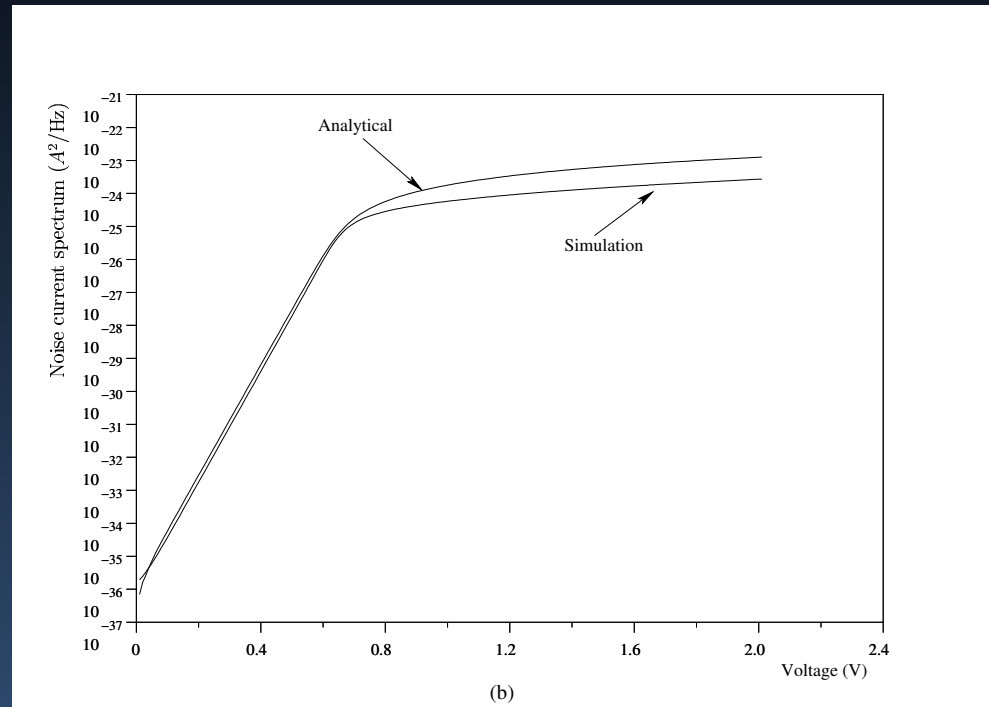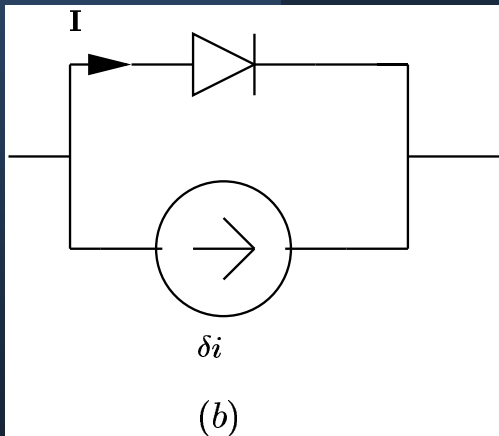
Spectral Density of shot noise (BF)

$$S_i(f) = 2qI$$

# Circuit Device cosimulation
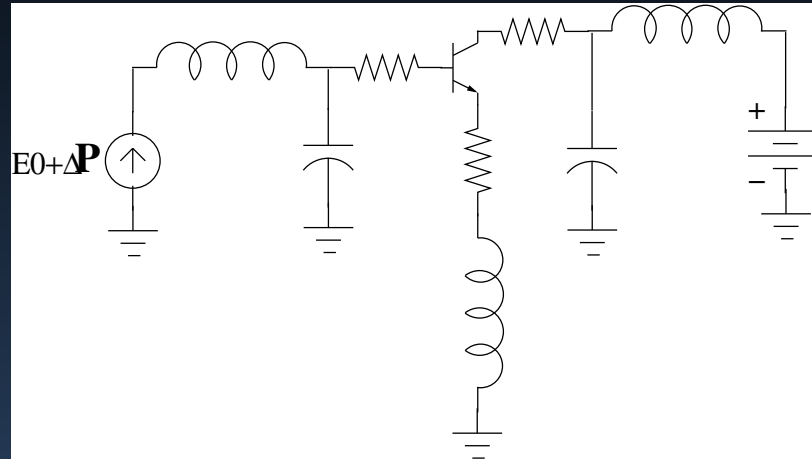


The simulated circuit

- nonlinear case
  Internal mesh nodes inside the transistor are treated as circuit nodes

# Mathematical formulation

- The system equation is written for each sample of the applied signal

- The whole system of equation is sparse and can be solved with an iterative method

$$
\begin{bmatrix}
A_0^n & 0 & 0 & \cdots & 0 & M_{N-1}^n \\
M_1^n & A_1^n & 0 & \ddots & \ddots & 0 \\
0 & M_2^n & \ddots & \ddots & \ddots & \vdots \\
\vdots & \ddots & \ddots & \ddots & \ddots & \vdots \\
\vdots & \ddots & \ddots & \ddots & \ddots & 0 \\
0 & \cdots & \cdots & 0 & M_{N-1}^n & A_{N-1}^n
\end{bmatrix}
\begin{bmatrix}
\Delta X_1^n \\
\vdots \\
\vdots \\
\vdots \\
\vdots \\
\Delta X_{N-1}^n
\end{bmatrix}
=
\begin{bmatrix}
B_1^n \\
\vdots \\
\vdots \\
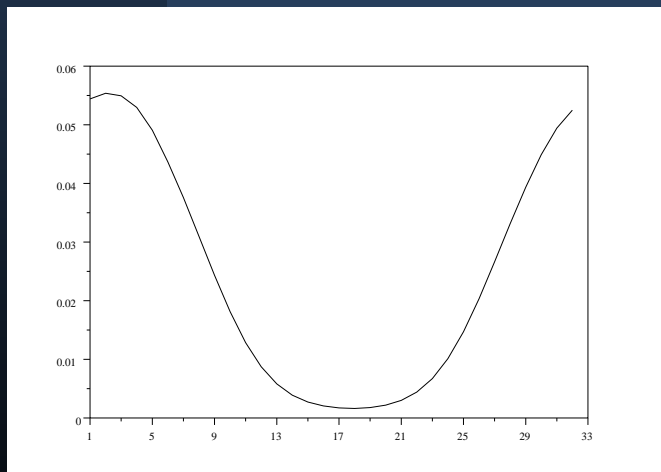\vdots \\
\vdots \\
B_{N-1}^n
\end{bmatrix}
$$

- the matrix sizes : 750x750 for A, N = 32, the whole matrix about $5.85.10^8$, NNZ=168192
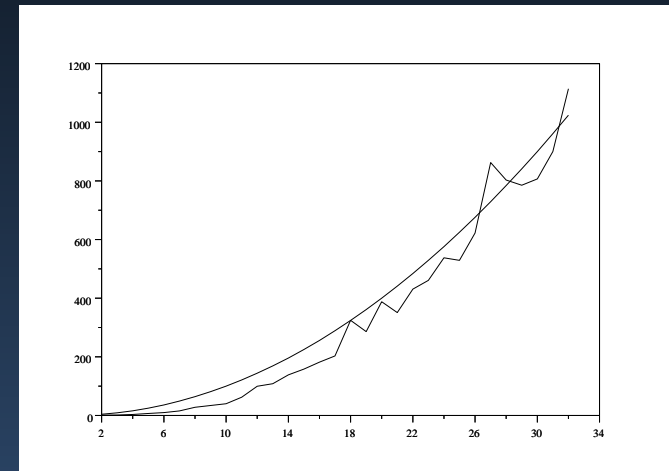
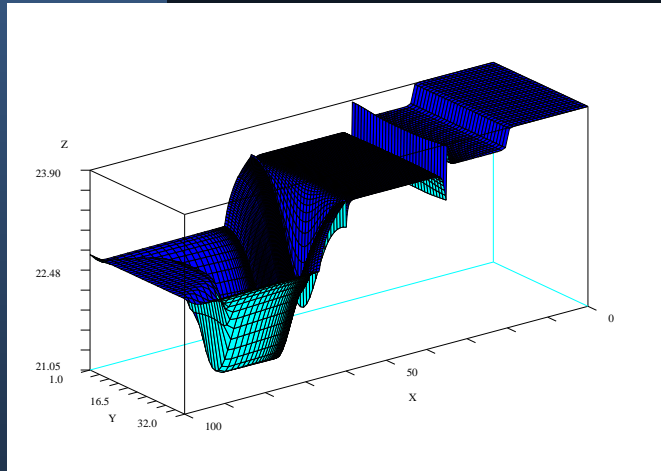# Large signal simulation
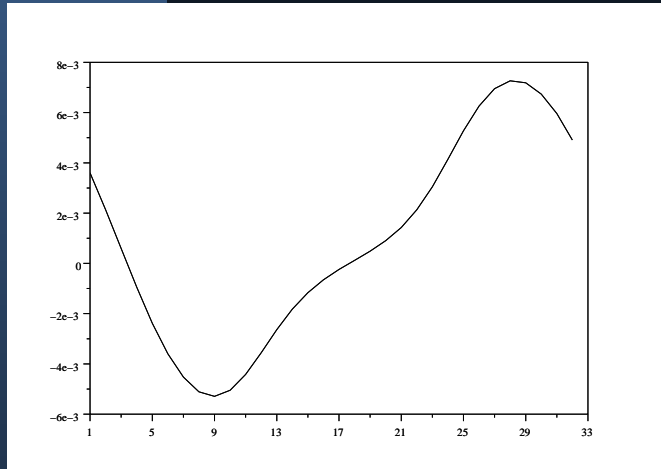


Base current
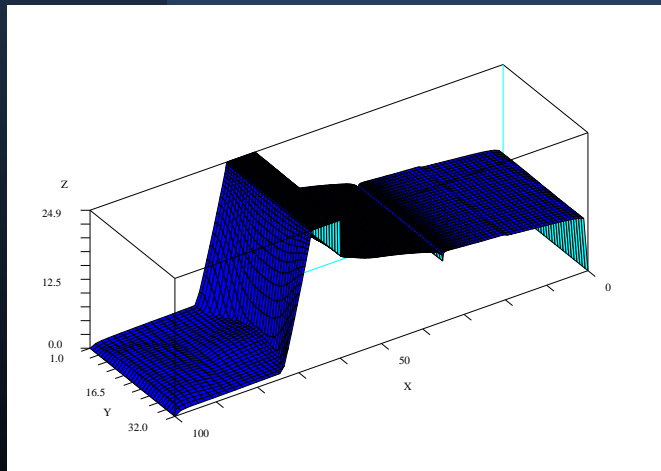


Collector current



Courbe $f(N) = N^2$

important CPU time, but
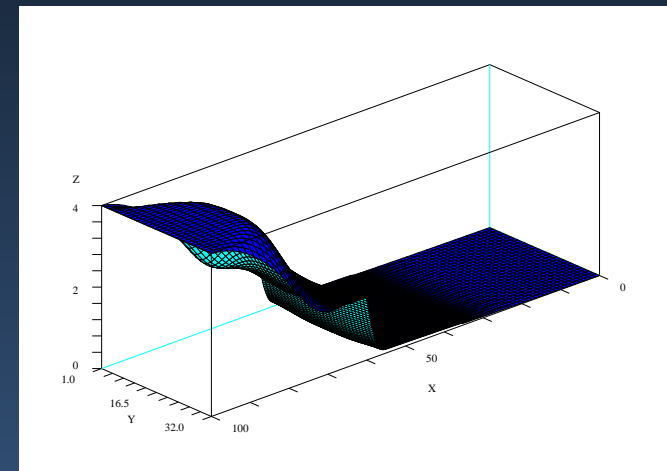may be reduced drastically
by the use of UMFPACK lib

# Internal dynamic of carriers
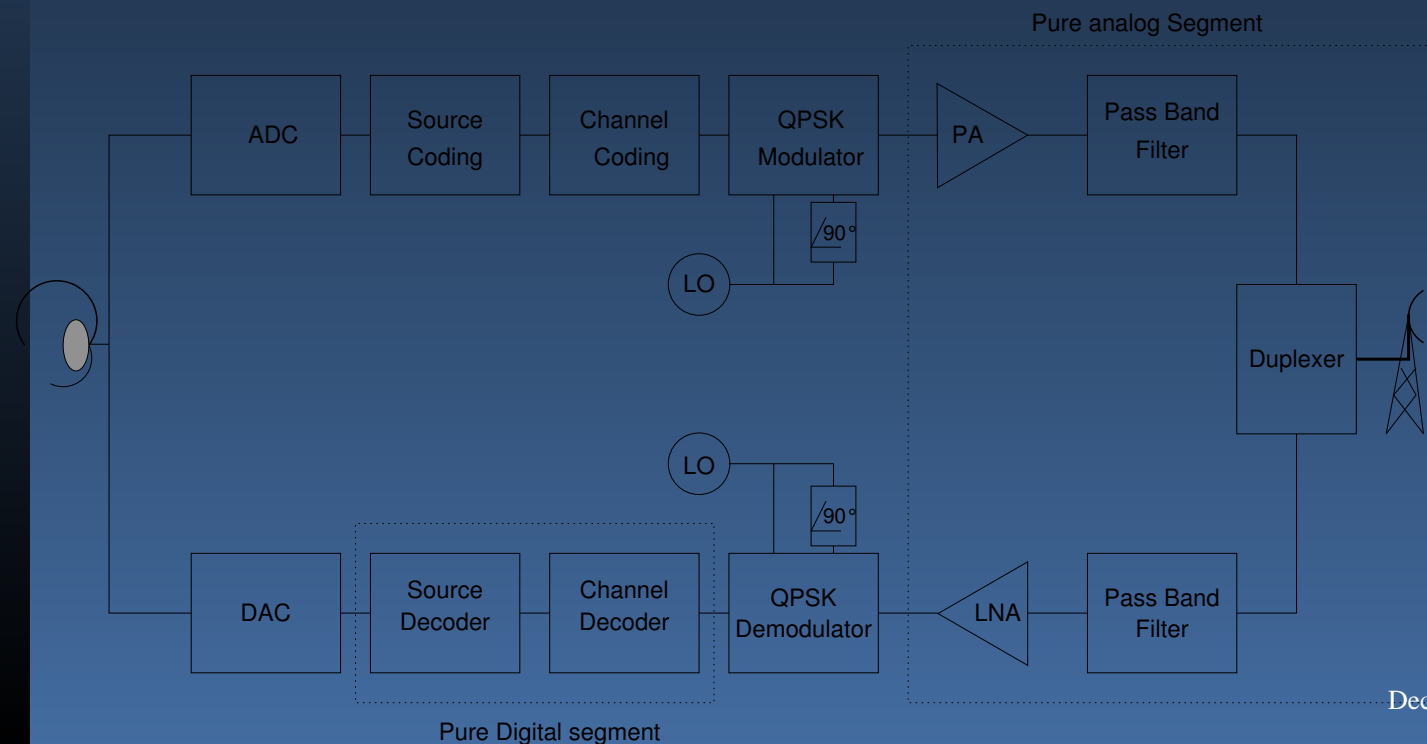


Electron Density



Hole Density



Electrical Potential

# Circuit and system interaction

The use of Scilab/Scicos for interaction between analog circuit and pure digital circuits

- Scicos is very powerful to described mixed circuits

- Implementation in Scicos of models previously developed in C

- Many efforts to develop specific scopes used in telecommunication designs



Pure analog Segment

| ADC | Source Coding | Channel Coding | QPSK Modulator | PA | Pass Band Filter |

90°

LO

Duplexer

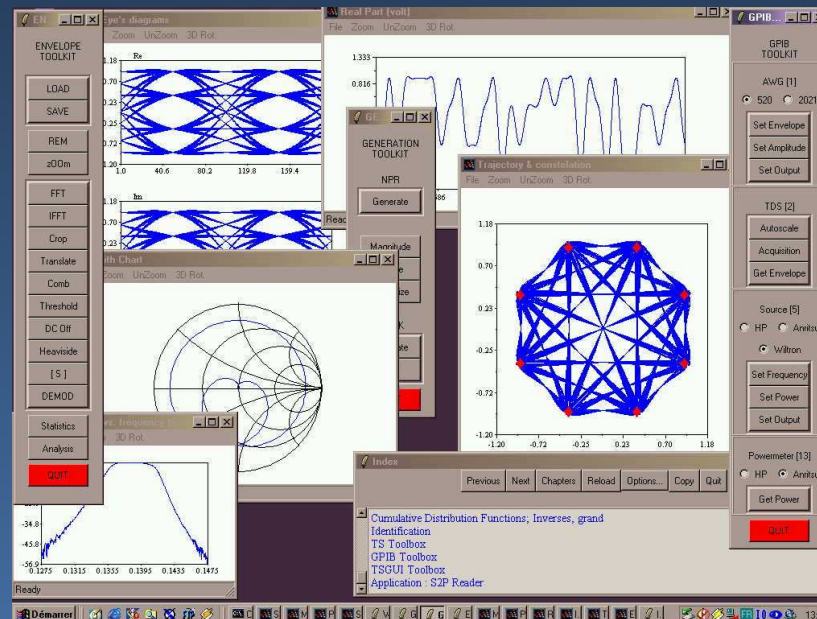| DAC | Source Decoder | Channel Decoder | QPSK Demodulator | LNA | Pass Band Filter |

LO

90°

Pure Digital segment

# Measurements setups

The use of Scilab for driving measurement setups is also very important (Atelier Telecom du Futur, CNES, DGA)

- GPIB communications with instruments

- Calibration procedure

- Signal processing functions

- GUI written in TCL/TK

# Metacomputing /ASP

The future for parallel computation

- Distributed and heterogeneous

- Metacomputing/Grid Computing

- A long time idea : using CPU and memory through the internet

Very promising:

- There is a need in Problem Solving Environments (PSEs) and some Application Service Providers (ASP)

- Applications require more and more powerful computer as well as more and more memory

- Library codes must stay in labs

- Confidentiality must be preserved

- Server must be accessible through very simple interface

# RNTL GASP Project

Based on the lient/Agent/Serveurs Approach

- **RNTL GASP Project**



S1 : Physiscs based Simulator

S2 : Sparse solver

DIET

client
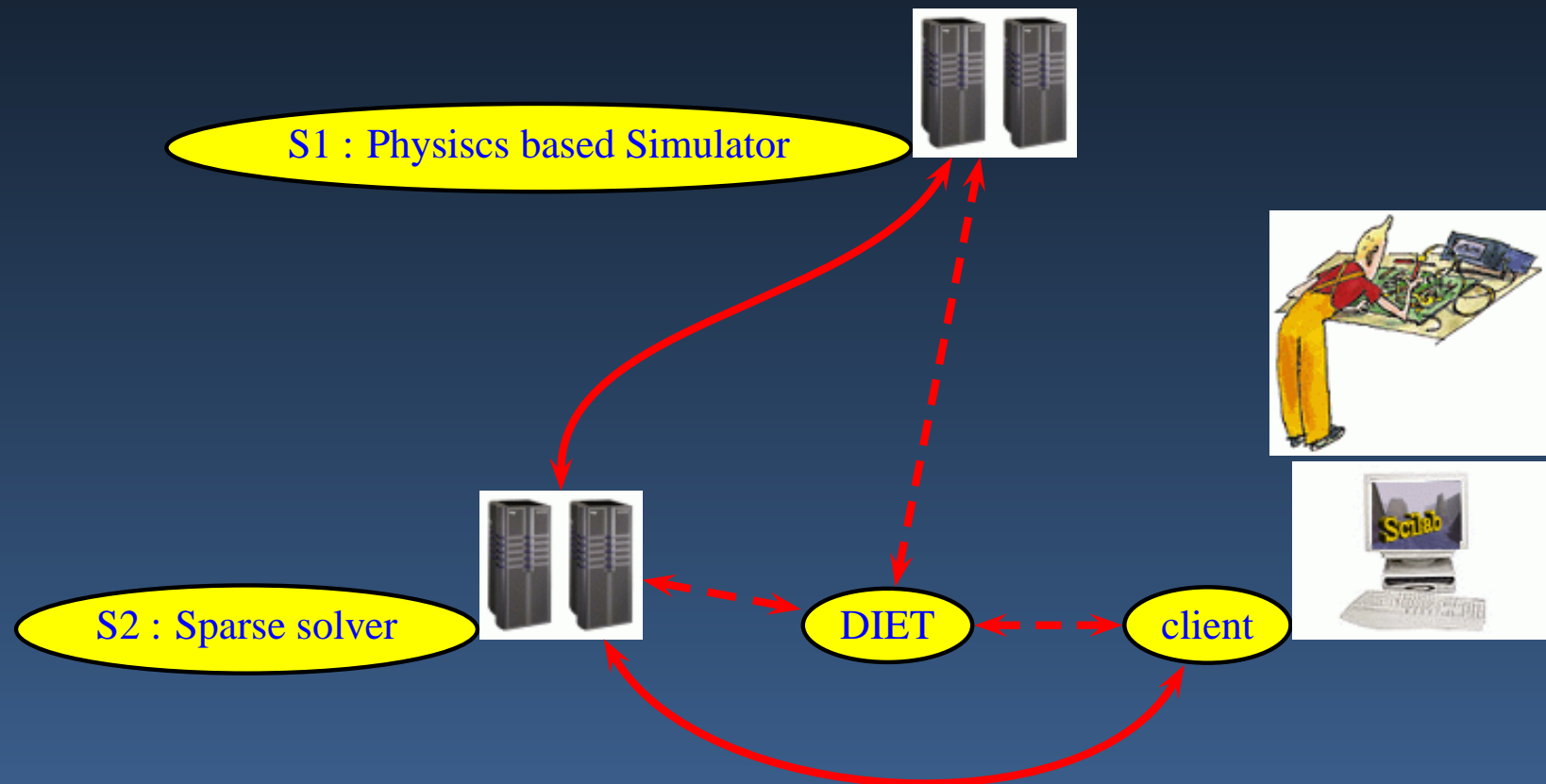
# Conclusion

- Scilab/Scicos has been identified as an excellent tool for high frequency device, circuit and system simulation

- This environment will allow us to develop an open simulation platform to federate the integration of various tools

- Other applications developed in our labs are also implemented on this platform (Stability analysis, Model Order Reduction)

- Three PHD students are currently working with this framework

- We receive a strong support from DGA, CNES an the Conseil Régional du Limousin to complete this task.