



Tibault Reveyrand

email : tibault.reveyrand@laposte.net

web : <http://membres.lycos.fr/treveyrand>

Scilab/Scicos toolboxes for Telecommunications

Sebastien Mons, Alan Layec^{*}, Abderrazak. Benadji, Tibault Reveyrand,
Raphael Sommet, Edouard Ngoya, Raymond Quéré.

XLIM/C2S2 – Université de Limoges, 123 av. A. Thomas, 87060 Limoges, France

^{*}INRIA –Domaine de Voluceau-Rocquencourt, B.P. 105, 78153 Le Chesnay Cedex, France

e-mail : sebastien.mons@xlim.fr

Presented at :

2006 Scilab Contest & International Workshop, Hangzhou, P.R.China
28 - 29 SEPTEMBER 2006

Published in :

S. LI, L.-H. MA, C. GOMEZ (Eds)

“Open source Software for Scientific Computation SCILAB, Research,
Development and Applications”

Tsinghua University Press,

pp. 129-142,

Personal use of this material is permitted. However, permission to reprint/republish this material for advertising or promotional purposes or for creating new collective works for resale or redistribution to servers or lists, or to reuse any copyrighted component of this work in other works must be obtained from Tsinghua University Press.

Scilab/Scicos toolboxes for Telecommunications

Sebastien Mons, Alan Layec*, Abderrazak. Benadji, Tibault Reveyrand, Raphael Sommet, Edouard Ngoya, Raymond Quéré.

XLIM/C2S2 – Université de Limoges, 123 av. A. Thomas, 87060 Limoges, France
*INRIA –Domaine de Voluceau-Rocquencourt, B.P. 105, 78153 Le Chesnay Cedex, France
e-mail : sebastien.mons@xlim.fr

Abstract. 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. This paper presents an overview of research activities on the simulation, characterization and modeling of High frequency devices, circuits and systems using the Scilab/Scicos Environment.

1 Introduction

The strategy of our lab consists on the use of a development platform for both modelling, simulation and instrumentation [1]. The kernel of this platform was developed under Scilab/Scicos environment. 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. The general organisation of this tool can be viewed in three axes :

First, Links between those functions and the electromagnetic, 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-1. This toolbox integrates circuit simulation, physics based simulation of semiconductors equation as well as the 3D thermal simulation.

Section-2 presents two complementary toolboxes dedicated to system level simulation using scilab/Scicos environment. First one is a system level simulation tool for RF and microwave devices which includes many low pass discrete time representation elements as filters, modulators, demodulators, signal and noise generators, samplers but also some analyzers like spectrum analyzers, constellation and eye diagram scopes, or BER calculators. This toolbox is well suited for digital communications system level simulations. The second toolbox concerns respectively the

implementation of PA's behaviour models in system simulation environments and the integration of commercial high frequency circuit simulation tools using API. This will allow the cosimulation of circuits and systems.

This open-source development tool drives also instrumentation algorithms as shown in section 3. Bridges have to be established between measurement test set-ups and the kernel. This is done by integrating GPIB calls in Scilab functions. Toolboxes respectively dedicated to drive a complete Load-Pull measurement setup with pulse capabilities and to generate or measure RF Times envelopes signals are presented here.

Finally, The use of Scilab/Scicos for both characterization and simulation enables an "hardware in the loop" in system level simulations in order to allow an high flexibility for device characterization and modeling processes.

2 Advanced device simulation

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 toolboxes including the best linear libraries like ATLAS, ARPACK, UMFPACK, TAUCS...

1.1 Heterojunction Bipolar Transistor simulation (HBT)

The first toolbox is dedicated to physics-based semiconductor simulation be means of interface functions with our (HBT) simulator. 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 devices, under large signal conditions [2].

1.2 Pin Diode impedance computation

The second toolbox concerns an improvement of the frequency-dependent impedance calculation for p-i-n diodes. Under forward bias condition, for thin p-i-n diodes occur important recombinations in the heavily doped regions which affect seriously the diode impedance. This effect is taken into account to increase the accuracy of existing models and to extend the validity domain to any I-region thicknesses. SCILAB is used to solve nonlinear system as well as linear systems obtained from ambipolar semiconductor equation and recombination current definition [3].

1.3 Model order Reduction

The third toolbox is dedicated to model order reduction of linear differential equation system by means of Ritz vector technique [4]. This technique is a projection technique based on Gram-Schmidt Orthogonalization algorithm. It has been developed to create reduce models from thermal Finite Element Simulation for electrothermal simulation in circuit simulator. Matrices extracted from ANSYS are read in SCILAB thanks to SCILIN toolbox. Sparse format is used. SCILAB linear algebra as well as eigenvalues computation are made in order to derive the reduced model.

2 Advanced circuit-system simulation

A 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 shown fig-1 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 [5] such as eye-diagram scope, constellation scope, spectrum analyser, probability densities scope...

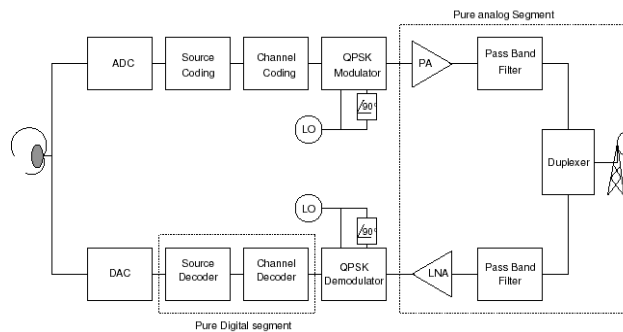


Fig. 1. Simplified architecture of a communication channel

We present a communication toolbox in Scilab dedicated to modelling and simulation of digital transmission. The modelling effort is focused on the improvement of time computation, and on the description in time domain of the constituent components in order to allow future compatibility with models realized in other design environment [6]. A description of a rudimentary communication chain using a DS spread spectrum technique, for a Quadrature Phase Shift Keying (QPSK) transmission shows the returned time domain waveforms in the Scicos environment for a given set of

parameters. The use of the scicos simulate function allows to drive a Scicos diagram from the Scilab prompt, so we are able to realize efficient post-processing operations by directly access to the simulated data. We finally present a laboratory experience where we simulate transmission chains with an heterogeneous network of computers using the Parallel Virtual Machine (PVM) communication toolbox in order to accurate the simulation time of BER estimation.

The « Modnum » toolbox has been developed at our laboratory and is now available [7], its internal architecture is illustrated in fig.-2. It is an open and free built-in Scilab/Scicos toolbox for the modelling and the simulation of communication systems. This toolbox contains several elements : A low-level computational routines library (in C language) including common functions of base band communication and tri-state PFD, a set of Scicos palettes including 71 blocks with computational functions and extended interfacing functions (Communication palette for PSK/QAM transmissions, Phase Locked Loop palette for the designing of integer and fractional frequency synthesizer, sinks palette to display frame-based signal and 3d trajectory in Scicos, Sources palette to compute random process, on-linear palette with common nonlinearities encountered in digital processor, ...).

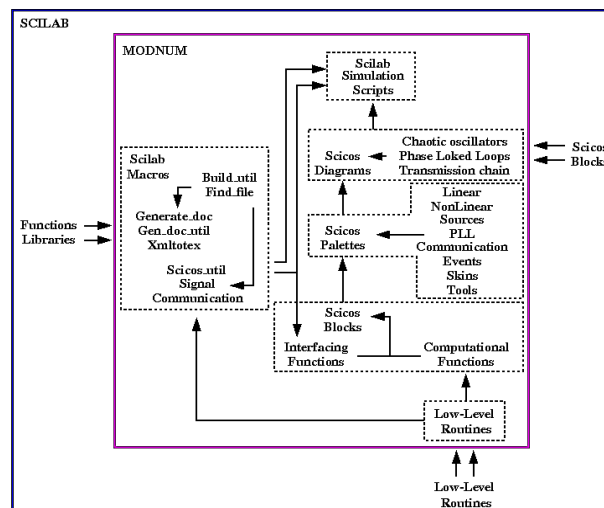


Fig. 2. Toolbox internal architecture

This toolbox proposes blocks, schematics and in-line functions of base-band PSK/QAM modulations in order to build communication chains in Scilab/Scicos environment. Components used to build spread-spectrum communication systems, such Pseudo Noise sequence generators (Quasi-Chaotic, PN and Gold sequence generators) are also included. This toolbox also includes miscellaneous scopes for Scicos, such 3D trajectory Scope and other scopes used for analysis of digital transmission: Eye Diagram Scope, Scattered Diagram Scope,...

The Scicos diagram of the transmission chain is shown in Fig. 3. Blocks that don't have event inputs are direct feedthrough marked. The original binary vector, computed by the random binary generator, goes through all the user's blocks in one

time of simulator process (one event). After being separate in two I/Q channel, scale by the over-sampling block, filter by a Root Raised Cosine impulse response, the vector is added to a white gaussian noisy samples vector. The receptor is respectively composed by a RRC filter, an adaptive filter, an under-sampling block and an I/Q demodulator. Final binary vector is compared with the original delayed binary vector and a counter block memorize number of errors.

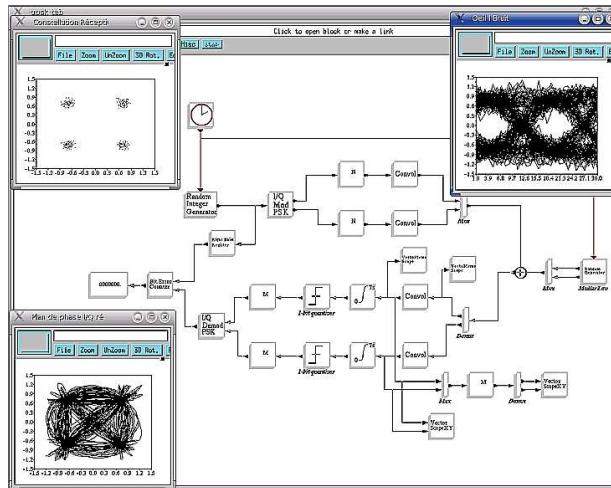


Fig. 3. PSK chain diagram (phase plane, eye & scatter diagram)

The realization of FIR is a critical point : overlap-add method which uses FFT convolution (instead of standard convolution) is used to decrease the execution time. In Scilab we have the scicosim function in order to externally launch the simulation of a pre-compiled Scicos diagram with the Scilab prompt. To launch a parameterized simulation of a Scicos diagram, for example by changing a parameter in the context (such as the variance of the white Gaussian noise random generator or the size of the original binary word), the diagram must be evaluated and compiled before invoking scicosim [8].

The simulation of the BER with this simple chain can tragically become long when we use a single computer. Indeed, to have a good estimation of the BER, we have to simulate a sequence comporting several millions of symbols, according to some values of the variance of the noise source.

Here, the developed idea is to use several computers to simulate a set of ten points of 10^8 symbols each. The laboratory experience is illustrated by the Fig. 4 where we use four computers. All computers are arranged with Scilab and PVM and are connected together via an intranet network correctly configured. They have also got interfacing function and compiled computational function of user's blocks and the Scicos diagram.

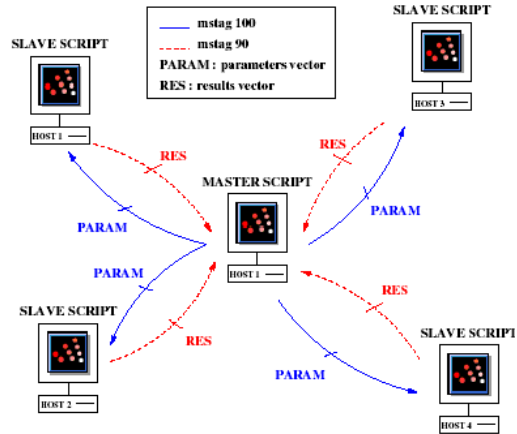


Fig. 4. Laboratory experience

A single master script drive slave scripts that are in charge of executing the simulation. The master script sends parameter vectors, via a first virtual PVM canal, to slave scripts and waits response vectors via a second canal. After realizing the simulation parametrized by the sending vector, slave scripts respond a result vector, and then prevent the master script that they are free to realize another simulation. The resulting BER plot of the preceding experience is shown in Fig. 5. With tree computers instead of one we reduce by a factor two the time calculation.

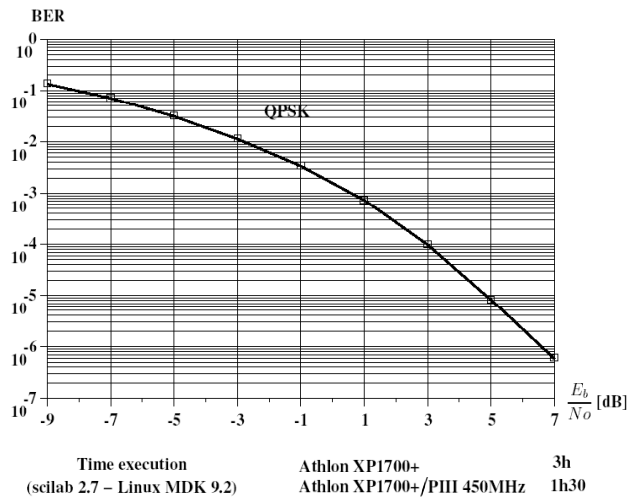


Fig. 5. TEB estimation

The presented transmission chain with the vectorial treatment are able to be completed with other user's blocks. We think now about realizing a spread spectrum transmission chain by adding Gold or chaotic sequences generator to simulate a

CDMA communication system. The Scilab/Scicos environment used with PVM communication toolbox shows that we are in a privileged environment to realize efficient simulation of numerical communication systems

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. Schematics and blocks of integer and fractional frequency synthesizer components (Phase/Frequency Detector, VCO, Delta-Sigma modulators,...) are provided. 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 useful [9] as the overall phase noise can be calculated as the fluctuations of zero crossing times around the periodic sequence $\{T, 2T, \dots\}$.

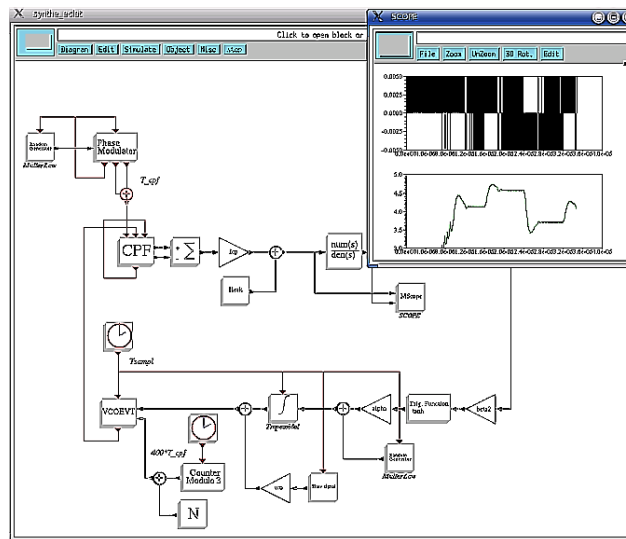


Fig. 6. Frequency synthesiser

This toolbox also focus on the simulation of chaotic systems and gives schematics of simulation of Chua's, Rössler's, Van Der Pol's systems (and others) as shown in fig-7. The models developed 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.

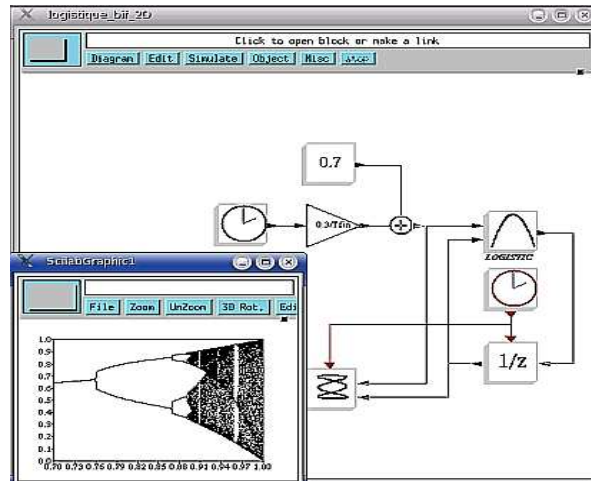


Fig. 7. Chaotic sub-system (bifurcation map)

This toolbox can be installed indifferently on linux (using gcc/g77) or windows (using MSVC++/lcc-win32) platform. Future improvement of builder script should also build the toolbox for Solaris/HP-Unix platform. Source version and binary version of the toolbox include also a English documentation in html/pdf format. Future improvements are Scilab macros to generate scilab simulation script files of scicos diagram with PVM (for BER estimation), Interfacing functions of low-level computational routines, Spectrum analyzer scope for Scicos...

In the design process of modern communication systems, it is more and more required to predict accurately the performances of the full transceiver chain. That means that the needs in terms of system level simulation tools and of macro-models become important. That signifies also that these tools must be associated with models able to describe the behaviors of the different functionalities (amplifiers, mixers, etc.) with a high level of precision. In this context, implementation of the models based on nonlinear convolution integrals in system environment is a critical point

These models have proven to be accurate for all types of signals and their extraction is affordable in common circuit simulators and measurement benches. In an effort to improve modeling accuracy, we have developed a co-simulation interface that allows the system simulator to access a circuit simulator for each time sample. In our example we have considered co-simulation between Scilab/Scicos™ and Xpedion/Goldengate™ circuit simulator. We will briefly present the numerical implementation of advanced model in system-level environment such as Scilab/Scicos and the cosimulation interface between system simulator and circuit simulator.

The figure 8 shows the flowchart which recalls the advance of our principle of modeling. The files resulting from measurement or simulation must have a standard format (.dat). They contain all information of the extracted data of the model (Volterra or nonlinear impulse response).

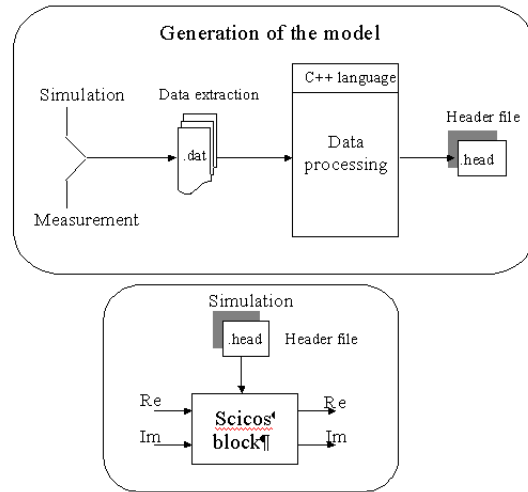


Fig. 8. Principle of modeling

In co-simulation, the system simulator access an Envelope transient [10] circuit simulator to compute the output of the critical part of the system. The interface between the two simulators is based on a parent/child interaction (fig. 9). Scicos acts as the parent simulator, which calls GoldenGate™ at the beginning of the simulation as its child. The co-simulated amplifier block is instantiated into Scicos model thanks to a bloc based on a generic Function, during the simulation, the two simulation kernels are synchronized and share data on their I/O. On the Scicos side a special generic function block acts as the coupling element and it is configured for shared memory communication between Scicos and GoldenGate™, running on a single computer.

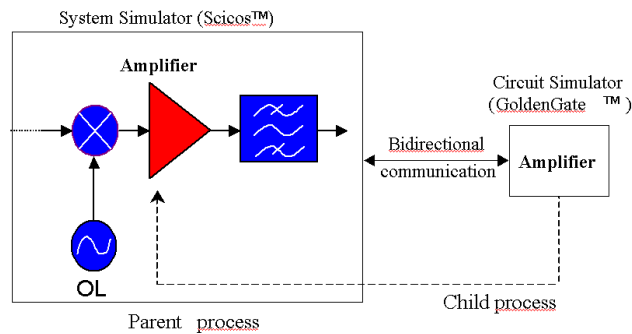


Fig. 9. Principle of modeling

We have run a Scicos simulation using alternatively PA's behavior models developed in our laboratory and the co-simulation interface for a satellite communication chain (Fig. 1). This chain represent a satellite link composed by a Satellite Downlink Transmitter, Downlink Path, and Ground Station Downlink Receiver. The signal

transmitted over the channel is 16-QAM waveform flows with speed of 5 Mbits/s, the carrier frequency of the link equal to 8 GHz. The Volterra kernel and the impulse response of the LNA (operating at 1.96 GHz) placed after the reception antenna were extracted from GoldenGate™ simulator. The simulation performances (CPU) is summarized in table 1. We observe that using the Volterra and nonlinear impulse response models has reduced the CPU time. The performance of nonlinear impulse response model show a simulation speed up sacrificing accuracy.

	Co-simulation	AM/AM AM/PM model	Volterra model	Impulse response model
ACPR	1h. 12 min.	14 sec.	2min. 18 sec.	2 min. 23 sec.

Table 1. Simulations performances

Faster and more accurate system level simulations of transmission chain is more and more important in modern communication system design. We have considered here the modeling and simulation of complete transmission chain as we have demonstrated that this can be achieved with better accuracy using an advanced power amplifier model that accounts memory effects, while keeping the simulation time well reasonable.

3 measurement test 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 for RF and microwave measurements and analysis. Those toolboxes constitute a complete GUI based environment for measurements. The structure of the measurement software is shown in fig. 10. This approach is an open-source alternative to VEE, Labview and Matlab wich enables communications between VISA for Windows (either Agilent or National Instrument) or Linux (NI-VISA) and Scilab in order to control your instruments. The software includes a layer between VISA and Scilab which is freely downloadable [11] : The GPIB toolbox includes several drivers (written in scilab script) for RF and microwave instrumentation.

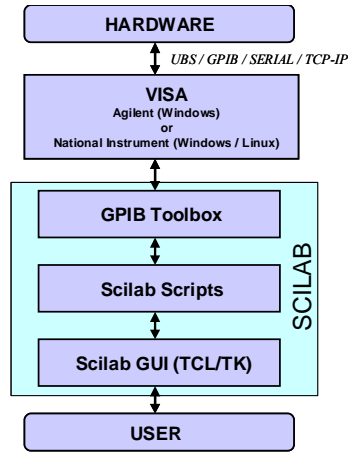


Fig. 10. Structure of the measurement softwares (Scilab)

Three different measurement set-ups are powered with Scilab at our laboratory : a low noise measurement setup [12] ; a time domain envelope load pull setup [13] (as shown in fig. 11) and a VNA based load-pull measurement setup with pulse capabilities in the frequency domain [14] (illustrated on fig. 12).

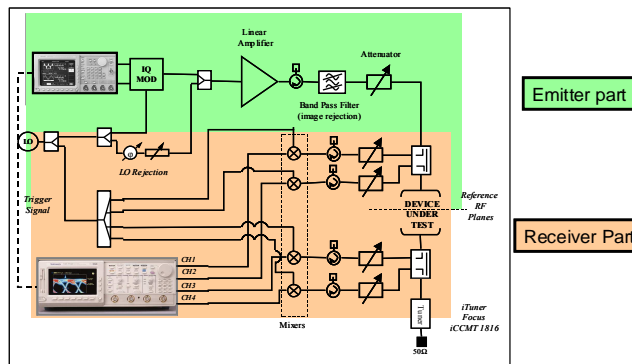


Fig. 11. Time domain envelop load pull measurement setup

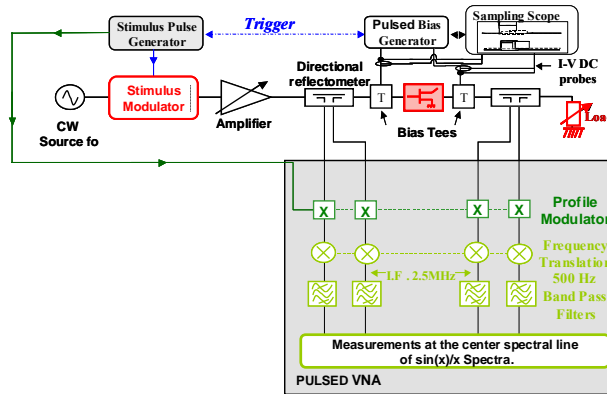


Fig. 12. Frequency domain pulsed load pull measurement setup

If modeling become to complex for a device, one can use either a co-simulation process which enables a link between Scicos and a circuit level simulator (a shown in section 2) or a link in quasi realtime between Scicos and the time domain instrumentation : the “hardware in the loop” process. In this last case, the signal located at the input of the amplifier under simulation is applied to the amplifier under test. This signal is pre-distorted according to the calibration of the emitter part of the bench and sent to the AWG. The calibrated response of the amplifier under test is feed backed to Scicos. Then, the simulation process carry on under the system simulator. This method provides a friendly environment in order to compare easily and simultaneously a device under test and its associated model with a modulated signal excitation.

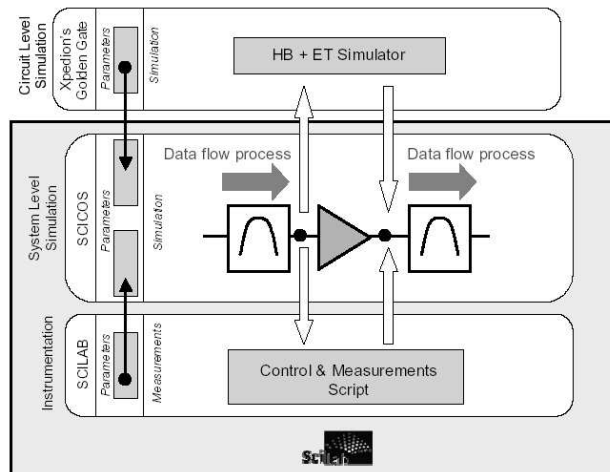


Fig. 13. Dynamic links between simulation tools and instrumentation

The toolboxes for communication are well suited for digital communications system level simulations. Power amplifiers may be included in the chain from their associated

system level behavioral models [15], a circuit level cosimulation principle [16] or a link through measurements [13].

5 Conclusion

This paper has presented an overview of the current software development and strategy at XLIM laboratory in order to federate different tools within a system level hierarchic design framework. Scilab/Scicos has been identified as an efficient tool for high frequency device, circuit and system simulation. Therefore, some Scilab/Scicos toolboxes were presented respectively dedicated to advanced devices, circuits and system simulation. Those toolboxes are open source and freely usable and distributable. To complete this task XLIM/C2S2 has received strong support from the CNES-Toulouse, the Délégation générale à l'Armement (DGA), the CNRS and the Conseil Régional du Limousin. Two PhD students are currently working within this framework.

References

1. Sommet, R., Quere, R., Ngoya, E., Mons, S., Nallatamby, J.-C., Reveyrand, T., Mallet, A.: Overview of research activities on the simulation of high frequency devices, circuits and systems using the Scilab/Scicos environment. 1st Scilab International Conferences, Rocquencourt (2004)
2. Riah, Z., Sommet, R., Nallatamby, J.-C., Prigent, M., Obregon, J.: Coherent Tools for Physics-Based Simulation and Characterization of Noise in Semiconductor devices oriented to Nonlinear Microwave Circuit CAD. Noise in Devices and Circuits II, SPIE, (2004) 507-518.
3. Submitted in IEEE Microwave and Wireless Components Letters
4. Sommet, R., Chang, C., Dueme, P., Quéré, R.: Electrothermal model of transistors based on finite element analysis for Radar applications. Thermal and Thermomechanical Phenomena in Electronic Systems IThERM (2004).
5. Layec, A., Quere, R., Nallatamby, J.-C., Mons, S., Guittard, J.: Modelling and simulation of a communication chain in the Scilab/Scicos environment. Conference Scilab, Rocquencourt (2004)
6. Bennadji, A., Soury, A., Ngoya, E.: Implementation of dynamic Volterra model in system Analysis. International Workshop on Electronics and System Analysis, Bilbao (2004) 15-18.
7. Layec, A.: available at http://www.lesia.insa-toulouse.fr/nouveau/transchaos/mod_num_web/web/fr/eg.htm
8. *De la compilation à la simulation sous scicos*, Scilab group. [Online]. Available : <http://www.saphir-control.org>.
9. Layec, A., Quere, R., Nallatamby, J.-C., Mons, S., Guittard, J.: Behavioral model of integer-N frequency synthesizer. In International Workshop on Electronics and System Analysis, Bilbao (2004)
10. Ngoya, E., Larcheveque, R.: Envelope Transient Analysis: A New Method for the Transient and Steady State Analysis of Microwave Communication Circuits and Systems. IEEE MTT Symposium Digest (1996) 1365-1368.
11. Reveyrand, T.: available <http://membres.lycos.fr/treveyrand/>

12. Lisboa de Souza, A., Nallatamby, J.-C., Prigent, M.: Low-Frequency Noise Measurements of Bipolar Devices Under High DC Current Density: Whether Transimpedance or Voltage Amplifiers, accepted at European Microwave Week, Manchester (2006)
13. Reveyrand, T., Barataud, D., Neveux, G., Nebus, J.-M., Ngoya, E., Teyssier, J.-P., Quere, R.: Overview on System Level Simulation Environment for Characterization, Modeling and Simulation of RF and Microwave devices. IEEE InMMIC (2006)
14. Gasseling, T., Barataud, D., Mons, S., Nebus, J.-M., Villotte, J.-P., Obregon, J., Quere, R.: Hot small-signal S-parameter measurements of power transistors operating under large-signal conditions in a load-pull environment for the study of nonlinear parametric interactions. Microwave Theory and Techniques, IEEE Transactions on, Vol. 52. (2004) 805-812
15. Soury, A., Ngoya, E., Rousset, J.: Behavioral Modeling of RF and microwave circuit blocs for hierarchical simulation of modern transceivers. IEEE MTT-S International Microwave Symposium (2005)
16. Benadji, A., Soury, A., Ngoya, E., Quere, R.: Implementation of Volterra Model In System Simulation Environments and Rf Circuit/System Co-simulation. European Microwave Week, Manchester (2006)