Quite Universal Circuit Simulator: A Free Mixed Signal Electronic Circuit Simulator

Being a free electronic circuit simulator, this is an efficient tool for an electronics engineer to simulate analogue and digital systems as well as analyse their system behaviour. Try it out in this month's EFY Plus DVD

JAI SACHITH PAUL

Circuit simulation is a critical step in electronic product design. By simulation a design, an engineer will know the behaviour of the circuit on various test cases even before practically implementing it. Getting the design right at the first implementation saves a lot of time and cost for the manufacturer. This, in turn, helps the manufacturer to reduce the time to market.

Quite universal circuit simulator (Qucs) is a free open source integrated circuit simulator with a smart graphical user interface (GUI). The GUI in Qucs is based on Qt, which is a cross-platform application framework. We can run AC, DC and transient analysis, in addition to VHDL/Verilog simulations, very easily with the help of this circuit simulator. Apart from this, we can easily import any existing SPICE model for our circuit simulation.

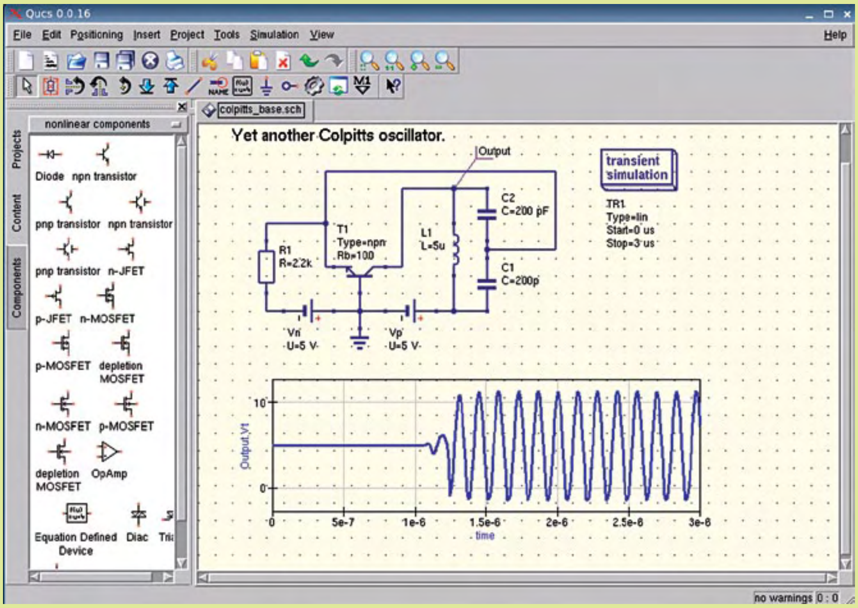

Fig. 1: Schematic and data visualisation

Infobytes

- Qucs is currently developed for GNU/Linux systems using GNU autotools.
- Also available natively on X86 Windows (XP/Vista/7/8) systems and experimentally on X86-64 versions of these. It also gets successfully compiled on other operating systems like Solaris and NetBSD.
- Latest stable release: 0.0.18 on August 31, 2014.

Functions of Various Tool Suites

Tool suite	Function
GUI	Creates schematics and VHDL code, sets up simulations and provides simulation results.
Text editor	Displays netlists and simulation logging information.
Filter synthesis application	Helps in designing various types of filters.
Backend analogue simulator	Simulates the previously set-up schematic. It takes a netlist, checks it for errors, performs required simulation actions and, finally, produces a dataset.
Transmission line calculator	Used to design and analyse different types of transmission lines like microstrips, coaxial cables, etc.
Component library	Holds models of real-life devices such as resistors, diodes, opamps, etc.
Attenuator synthesis application	Designs various types of passive attenuators.
Command line-conversion program	Imports and exports datasets, netlists and schematics from and to other CAD/EDA software.

Features of Qucs

Let us take a look at the main features of Qucs:

Massive component library. Qucs comes up with almost all the components required for an electronic circuit. The library consists of lumped elements like resistors, inductors and capacitors as well as models for sources, probes and transmission lines for electronic circuits.

We can also find non-linear components like diodes and transistors in the library. It includes various transistor models like heterojunction bipolar transistor (HBT), metal semiconductor field effect transistor (MESFET), metal oxide semiconductor field effect transistor (MOSFET) and junction gate field effect transistor (JFET) in the component library. Additionally, users of this tool are given the provision for expanding the libraries as per their requirements.

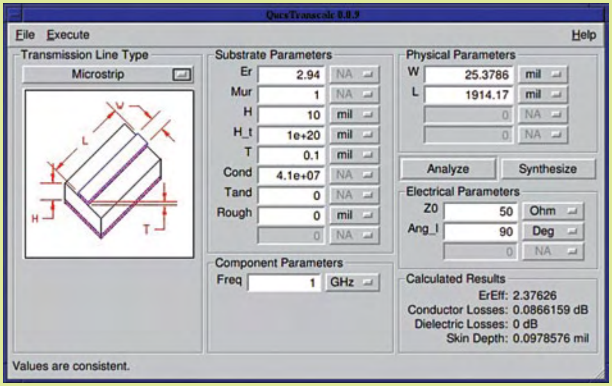

Fig. 2: Transmission line calculator

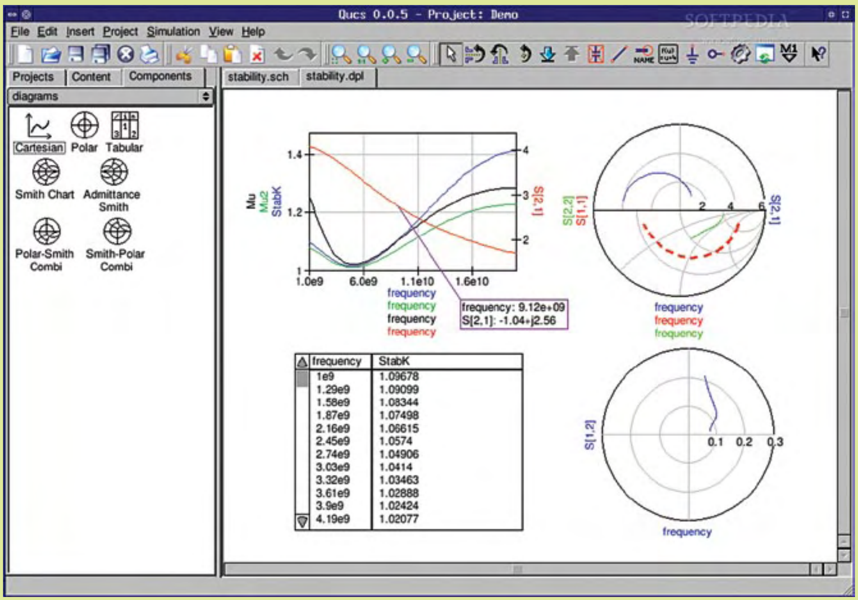

Fig. 3: Plotting data in Qucs

Smart GUI. You have access to a very useful GUI in Qucs that is based on Qt by Digia. This helps us create schematics or write VHDL code using the user interface. Setting up simulations and displaying their results is very easy using this GUI.

The GUI aids in efficient schematic capture by allowing the simulated data to be represented in various forms as Smith chart, tabular, cartesian, polar, Smith-polar combination, 3D-cartesian, locus curve, timing diagram and truth table.

Analysis of results made easy. Various types of analyses are possible with the help of Qucs. We can get the small signal response of a circuit using AC analysis, while the DC sweep analysis in Qucs could be used to calculate the circuit's bias point over a range of values. The transient analysis helps us compute the response of the circuit with respect to time.

Not only can we easily calculate the contribution of noise from the resistor and semiconductor devices in the circuit with the help of noise analysis, we can also vary the range of values of a component in the circuit and note the response using Qucs with the help of parameter sweep analysis. It is easy to obtain scattering parameters by using S parameter simulation in Qucs, while pure digital simulations are supported by VHDL and Verilog HDL in Qucs.

What users think about this tool

The strength of any tool lies in the experience that it can provide its users. Going through the reviews in the online communities, we realised that

Competing tools in mixed signal electronic circuit simulation

GNU circuit analysis package (GNUCAP). GNUCAP is a fully-interactive and command-driven general-purpose circuit simulator, developed as part of GNU project. It can perform non-linear DC and transient analysis, Fourier analysis and AC analysis linearised at an operating point.

Ngspice. Ngspice is yet another open source mixed signal circuit simulator. It is possible to have three classes of analyses using Ngspice—non-linear DC analysis, non-linear transient analysis and linear AC analysis.

Easyeda. Easyeda is a web based EDA tool for electronics engineers. This is an online tool and the user need not download any software. Easyeda helps in online schematic capture, PCB design and Ngspice simulation.

Falstad circuit simulator applet. Circuit simulator is a freely available Java applet that can be used to simulate a wide variety of electrical and electronic circuits. We can either run the simulator as a standalone Java application or embedded within a web page by using a Java-enabled browser.

GeckoCIRCUITS. GeckoCIRCUITS is a circuit simulator for modelling power electronic systems. The software has an extremely-high simulation speed. It can be used to simulate control modelling and thermal simulations. It can also be integrated into MATLAB/Simulink or other programming environments.

users are quite impressed with this product.

A user from sourceforge.net writes, "This is a very nice product. I am a DSP guy, so I do not use circuit simulators often. I have some experience with SPICE and Orchad, but I have a very strong preference for Qucs. It is easier to use and richer in features. For example, the basic OpAmp is just what you need when you want to make some examples in class without drowning in the complexities of a real OpAmp."

Another user from apps.ubuntu.com says, "I have only played with this briefly, but if you are familiar with Agilent Design System (ADS), then you will pretty much be right at home. That being said, if you are not familiar with ADS, then it could be a bit confusing. Features included are: AC, DC, harmonic balance, S parameter sims, Smith charts, as well as cartesian plots for transient sims, SPICE models for transistors and FETs, and file based circuit elements. It looks like it will be adequate for hobby circuit building. In fact, it is probably the most complete open source circuit simulator that I have found."

Why not give it a try

Qucs is already a very popular electronic circuit simulator among design engineers. It is free and an open source simulator. If you are a designer or student who is curious to know how a circuit works, you must give it a try today itself. ●

The author works at EFY