

Virtual Instruments Provide Realistic Displays for RF Circuit Simulation

By Alexander Ehle
Electronics Workbench

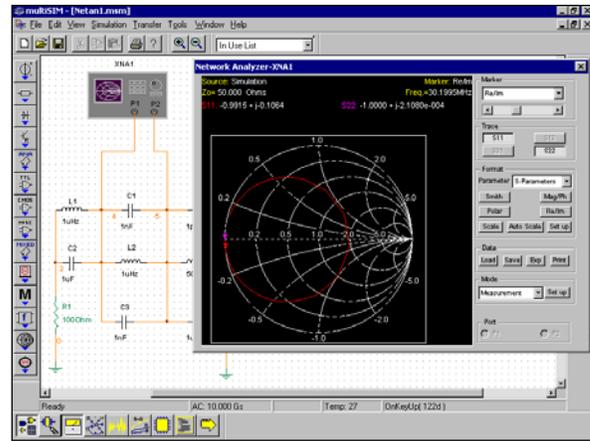
Virtual instruments are a unique method for speeding up analysis of RF circuits within simulation software. These “instruments” look like and can be used like their real counterparts. They make the step from the real lab environment to the software solution easy, requiring less time to learn the software itself.

Multisim V6 from Electronics Workbench has been using virtual instruments for analyzing circuits in the simulation environment since the first launch in 1989. The new version 6 of Multisim offers two new virtual instruments for analyzing RF circuits.

The Network Analyzer, based on the operation of the HP8751A or HP8753E of Agilent Technologies, can measure and view the scattering parameters (*S*-parameters) as well as the *H*, *Y* and *Z* parameters of a circuit. The display can show the values using Smith® chart, polar-chart or standard charts. Simply by clicking the appropriate button on the control area of the instrument, it is possible to change the display format or scaling properties. In addition, the Multisim Network Analyzer can display the stability factor of the connected circuit.

To use the Network Analyzer, it is only necessary to place the symbol of the instrument in the schematic and connect the input and outputs of the symbol with the circuit. The next step is to open the instrument by double-clicking the symbol. After that, the instrument can be configured by clicking on the appropriate controls on the right side. The simulation can be started using the menu or the toolbars power switch. Immediately, the display of the instrument will show the results.

The analysis results of the Network Analyzer



▲ Figure 1. Multisim’s Network Analyzer in action.

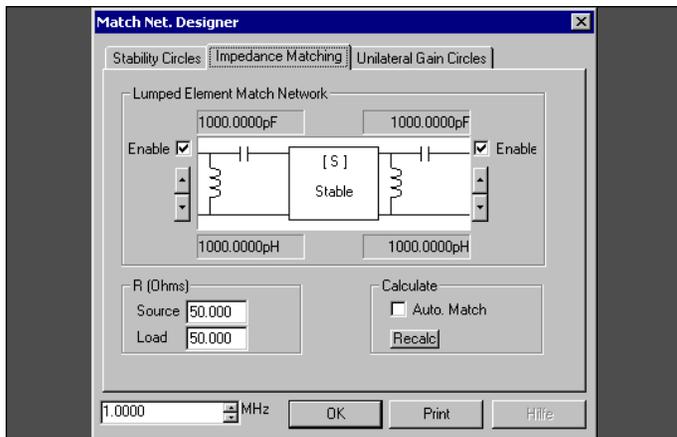
can be stored as plain text files, which can be imported to third party tools such as spreadsheets or mathematical software.

Two special modes of operation extend the usefulness of the Network Analyzer in Multisim. First, the matching network analysis provides three options:

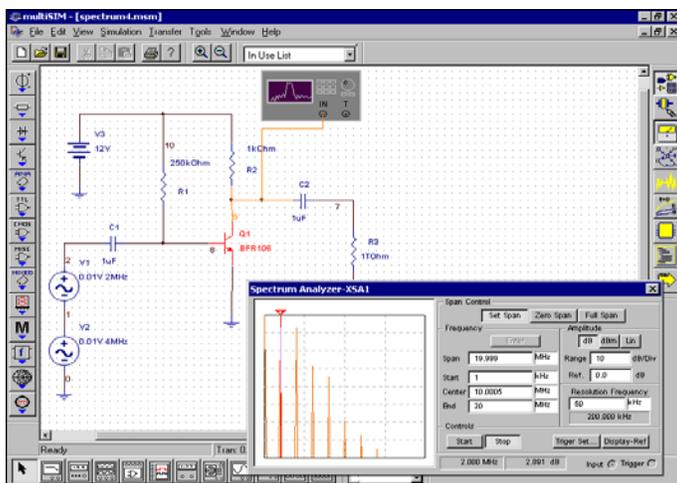
- Stability circles
- Unilateral gain circles
- Impedance matching

The stability circles are used to analyze the stability of a circuit at different frequency points. In practice, a part of the input signal is reflected back to the source. An RF engineer aims to minimize this reflection and deliver maximum signal to the load. The stability circles help to achieve this goal.

To analyze the unilateral property of a circuit, the unilateral gain circles can be used. The



▲ Figure 2. Impedance matching configuration dialog box.



▲ Figure 3. Spectrum analyzer of Multisim at work.

unilateral property of a network is determined by calculating the unilateral figure of merit. The circuit is unilateral if there is no reflection effect.

The other mode of operation is RF characterization analysis, which helps designers study RF circuits in terms of the power gain, voltage gain and input/output impedances. Multisim can calculate the general power gain, the available power gain and the transducer power gain for $Z_0 = 50$ ohms at a given frequency.

Another instrument is the Spectrum Analyzer, which has a bandwidth of 4 GHz and allows the user to analyze the complete frequency range from low frequency to microwave frequency. This virtual instrument contains all important features of a real instrument. To calculate the spectrum, Multisim uses the Radix-2 algorithm.

The adjustment of the Spectrum Analyzer can be done simply with some mouse-clicks. The span control can be adjusted individually or automatically, as can the frequency range. The user can choose between dB, dBm or linear scaling. To influence the results and their quality, it is also possible to change the frequency resolution.

Two cursors allow the display of the frequency and amplitude of each point on the screen. It is also possible to display a reference level, for example, the -3 dB level, to estimate the bandwidth of an amplifier.

Multisim also provides standard SPICE analysis that can be accessed via dialog boxes. In addition to this analysis, Multisim is now able to compute the noise figure (NF) of a circuit. The noise figure shows the signal/noise ratio which displays the quality of a signal. To calculate the noise figure, Multisim uses the equation:

$$NF = N_O / (G * N_S)$$

where N_O is the output noise power, N_S is the thermal noise of the source resistor and G is the circuit's AC gain.

Conclusions

Compared with pure RF simulators, Multisim can be used in a wide range of circuit design. It is not restricted to RF circuits. Enhancement of the Berkeley SPICE3F5 simulator with RF analysis, model generators and libraries and the cooperative simulation of XSPICE code models (created with a C++ compiler) and VHDL offers new simulation capabilities. This simulator is open for the future and provides any designer the freedom to develop circuits and systems.

A free of charge evaluation version of Multisim can be downloaded from www.electronicworkbench.com. A localized version for German users is also available from www.electronicworkbench.de.

Author information

Alexander Ehle is technical director at Electronics Workbench GmbH for the product Multisim in German speaking countries. He has more than 10 years experience in electronic circuit development and simulation. Alexander holds a technician degree in electronics and studied at the Institute of Electronics, Fachhochschule Reutlingen, Germany. His e-Mail is: support@multisim.de.



For more information, contact:

Electronics Workbench (headquarters)
111 Peter Street, Suite 801
Toronto, Ontario
Canada M5V 2H1

Electronics Workbench (U.S. Office)
908 Niagra Falls Blvd., Suite #068
North Tonawanda, NY 14120-2060
Internet: www.electronicworkbench.com

Or circle Reader Service #201