# TOOLS FOR MICROWAVE RADIO COMMUNICATIONS SYSTEM DESIGN

H. Abdalla Jr, Paulo Carvalho, Luis. F. Molinaro, Carlos Evangelista, L Bermudez

Departamento de Engenharia Elétrica – Universidade de Brasília – UnB
P.O. Box 04386, Zip Postal: 70.919-970, Brasília – DF – Brasil
Phone: (55 61) 273 5977, Fax: (55 61) 274 6651, abdalla@ene.unb.br

**Abstract**

This paper describes a software package for the design, analysis, and simulation of microwave radio systems. This article is mainly directed to engineering education. The software can also be used to aid in the Hybrid Microwave Integrated Circuit Design needs in industry. Different Design procedures for various types of microwave circuit are used. Special emphasis is given in the progressive use of simulation techniques to stimulate learner activity and overcome some difficulties inherent to the experimental subject and theoretical results. The developed tool has been used in the Microwave Circuits discipline for the graduation course in electric engineering of the University of Brasilia, Brazil.

## 1. INTRODUCTION

Microwaves are generally described as waves with frequencies that range from approximately 500 MHz to 300 GHz or more. Microwave radios propagate signals through Earth's atmosphere between transmitters and receivers often located on top of towers spaced about 90Km to 180 km apart. Therefore, microwave radio systems have the obvious advantage of having the capacity to carry thousands of individual information channels between two points without the need for physical facilities such as coaxial cables or optical fibers. The main advantages of microwave radio systems are:

- Due to their operating frequencies, microwave radio systems can carry large quantities of information;
- High frequencies mean short wavelengths which require relatively small antennas;
- Minimum delay times are introduced;
- Minimal crosstalk exists between voice channels;

Microwave radio systems capacities range from less than 12 voice-band channels to more than 22,000 channels. Early microwave systems carried frequency-division-multiplexed voice-band circuits and used conventional, noncoherent frequency modulation techniques. Most recent developed microwave systems carry pulse-code time-division-multiplexed voice-circuits and use more modern digital modulation techniques, such as phase shift keying (PSK) or quadrature amplitude modulation (QAM).

## 2. MICROWAVE RADIO SYSTEM

In essence, electronic communication is the transmission, reception, and processing of information with the use of electronic circuits. Information is defined as knowledge or intelligence communicated or received. Fig. 1 shows a simplified block diagram of an electronic communication system, which comprises three primary sections: a source, a destination, and a transmission medium.
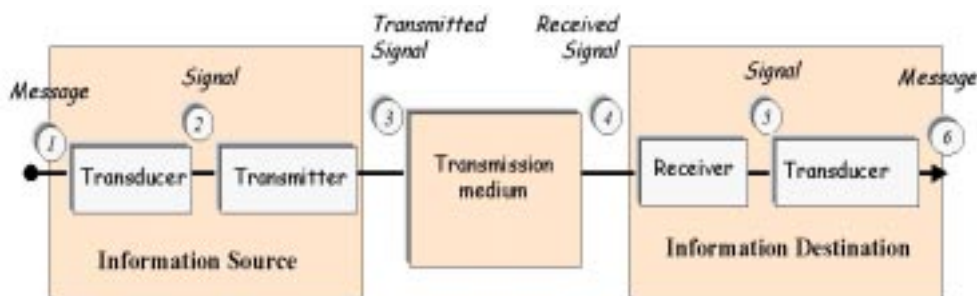


Fig.1. Simplified block diagram for an electronic communication system.

In our case the transmission medium is radio frequency, specifically the microwave. The block diagram of a microwave radio system is show in Fig.2.
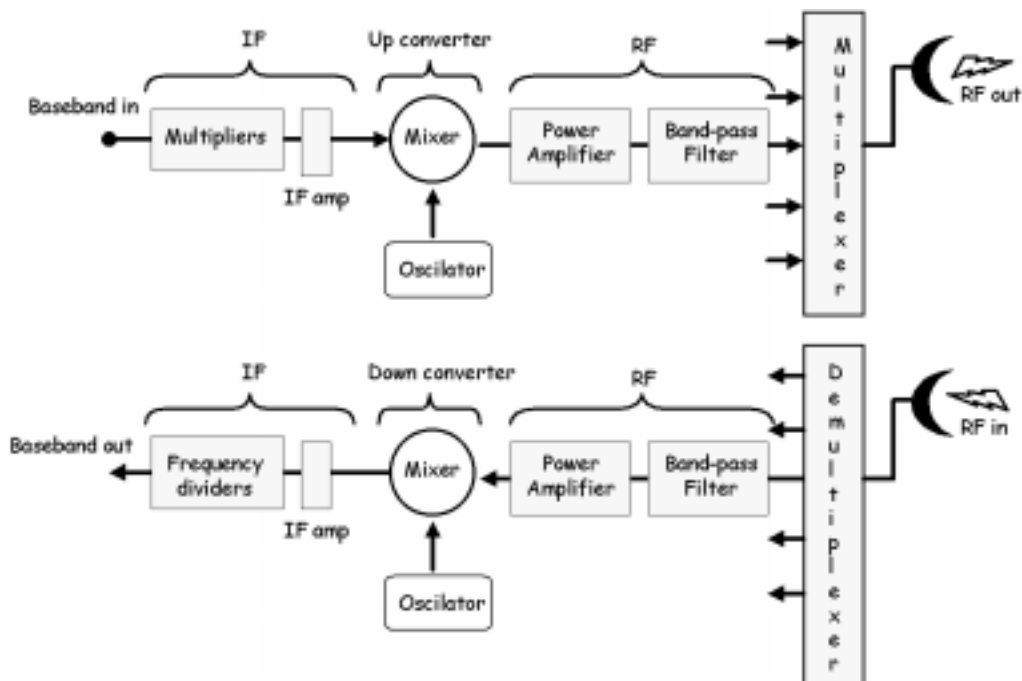


Fig2. Simplified block diagram of a microwave radio system: transmitter and receiver.

In following are the software, which allows each device project and the microwave link analysis – Fig.3.
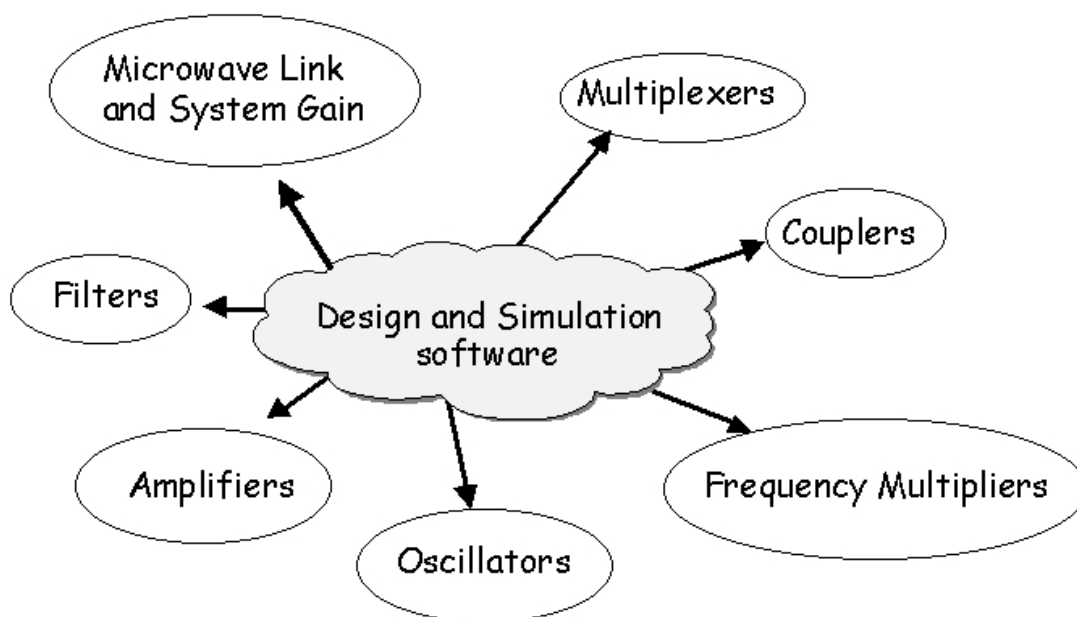
Fig3. Software package for the design, analysis, and simulation of microwave radio system.

## 3. MICROWAVE LINK SIMULATOR

The Microwave Link Simulator is a software developed in Visual Delphi Environment for microwave point-to-point links. This software has two main parts: one dedicated to tower calculus where the tranceptors systems should be installed and the other dedicated to system analyses of power balance throughout the links.

From position data of the tranceptors systems and the obstacle, main frequency of system operation, system dimensions, and atmosphere parameters the software calculates to the user tower heights for a Fresnel first zone visibility system, Fig. 4.

The software allows equally observation of the various Fresnel zones obstruction, illustrated in Fig. 4. The user can move the obstacle along the line between the origin and destination points. This allows the observation of the non-obstruction Fresnel zones on the entire link. To refresh the values or make new tests, the user can alternate between the windows *Model* and *Diagram.*

In Fig. 5 it is shown the developed environment for system analysis of power balance. In this environment, the user can arrange the desired system from basic elements like: origin and destination tranceptors (  and  ), active and passive repeaters (  and  ) and obstacles (  ). When this happens, it is only needed to point the mouse and click on the icon which represents the desired system element and drag to the project area.
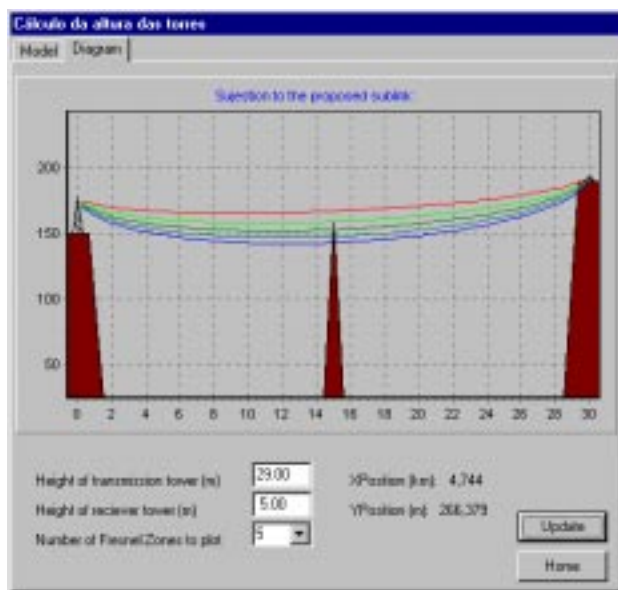
Fig. 4. Tower heights calculus.


When the mouse button is released a dialog box opens to enter the data referred to the element, such as: antenna gain, output power, transmission rate, distance and heights, diversity (spatial and frequencial), etc. The user can insert the amount and variety of elements to build the point-to-point system.
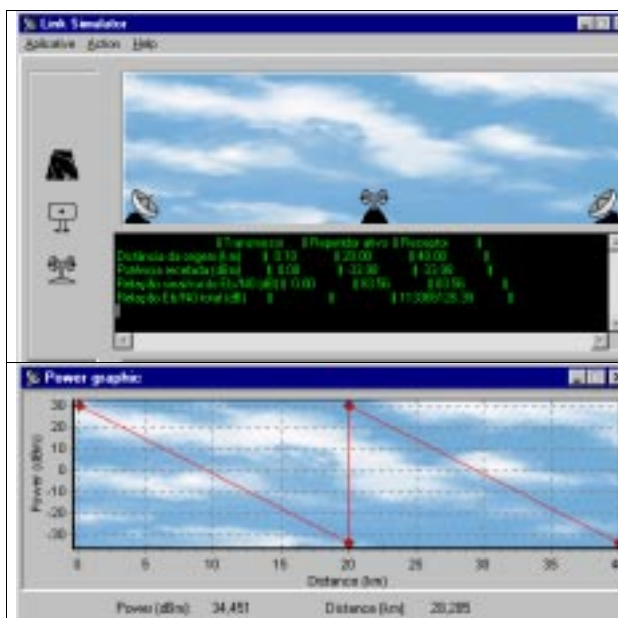


Fig. 5. (a) Link calculus and (b) power balance.


As soon as the system is assembled the user may simulate a link and get the power balance. Power gain and lost values are indicated in each point of the system. The final result is shown in table and graphic, as shown in Fig. 5.b. From these values the user may modify the system parameters and proceed a new analysis till the desired behavior is obtained.

## 4. THE MICROWAVE DIODE FREQUENCY MULTIPLIERS

The CAD of Microwave Diode Frequency Multipliers, called MultFreq, has the following features.

- Calculus of project parameters such as efficiency output power, bias voltage, and input and output diode's impedance.
- Design of the necessary filters for the isolation of the multiplier ports: low-pass and band-pass filters of Chebyshev and Butterworth kinds in distributed and lumped parameters.
- Impedance matching networks using series and parallel stubs.
- Design of $\lambda/4$ transformers of Chebyshev and Butterworth providing dimensions and impedance characteristics of each section.
- Multiplier layout in distributed parameters, including all calculated dimensions.
- Simulation in frequency of each project block: insertion and return losses.
- Poles and zeros analysis of filters.

As an example, a step–by-step design procedure of a varactor frequency multiplier using MultFreq is outlined below.

1) Give the diode's specifications and the drive level, as illustrated in Fig.6.
2) Based on Burkhardt method and from initial specifications, all project parameters are calculated. These parameters are: efficiency (r), input power ($P_{in}$), output power ($P_{out}$), bias voltage ($V_{dc}$), input ($R_{in}+jX_{in}$) and output ($R_{out}+jX_{out}$) diode's impedance at input and output frequencies. The Burkhardt's results and the block scheme of the multiplier are shown in another screen, Fig.7. From this block scheme it is possible to access the MultFreq's functionalities that allow the multiplier implementation in distributed parameters.
3) Finally, with all blocks already calculated, it is possible to view the multiplier layout and to verify the dimensions of all circuit sections, Fig.8.
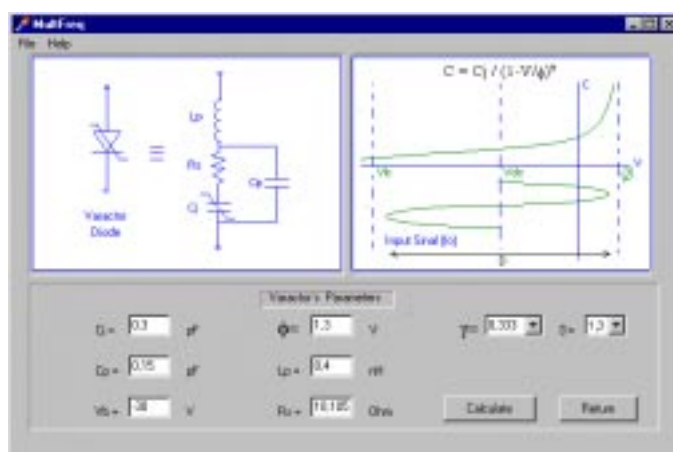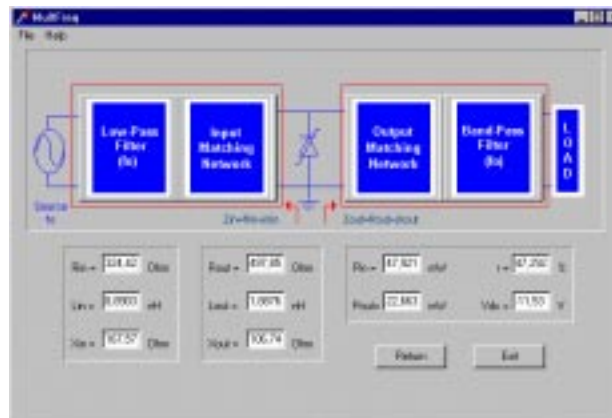


Fig. 6. Screen of varactor's input parameters.

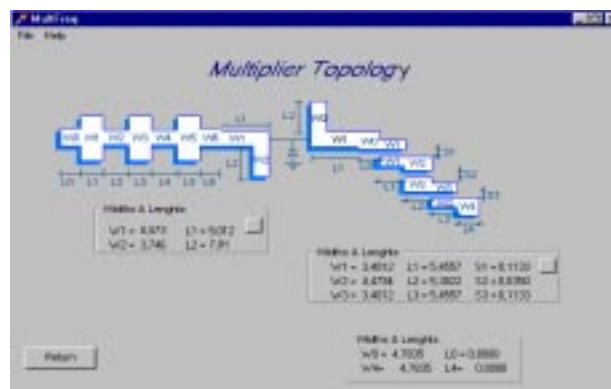Fig. 7. Burkhardt's results and exit to subsequent project sections.



Fig. 8. Reactive multiplier layout.

## 5. THE MICROWAVE FILTER

The CAD for Filter design is composed of two independent modules that execute the follows functions:

- Construction module of the transfer function filter with frequency response of Chebychev or Butterworth. This module also contains a dedicated graphic interface that permits the visualization of the frequency response before of the filter synthesis.
- Synthesis module of the filter in lumped parameters and distributed parameters. This module provides the layout filter, including all calculated dimensions, taking into account the chosen fabrication technology.

A step–by-step design procedure using the software developed is outlined below:

1) Select the kind of frequency response and the structure of the desired transmission. Complete the filter specifications and the basic parameters of the transmission structure based at Fig. 9.
2) With the specifications, it is possible to access the analysis and syntheses modules of the computer program.

These modules allow the construction of the circuit using all necessary data. The Fig. 10 shows a typical output of the frequency response module, in which it is possible to see the insertion loss and the return loss. In the synthesis module, it is possible to obtain the project in lumped and distributed parameters distinctly. The screen with the layout of the tapped miniaturized hairpin line resonator filter is shown in Fig. 11.



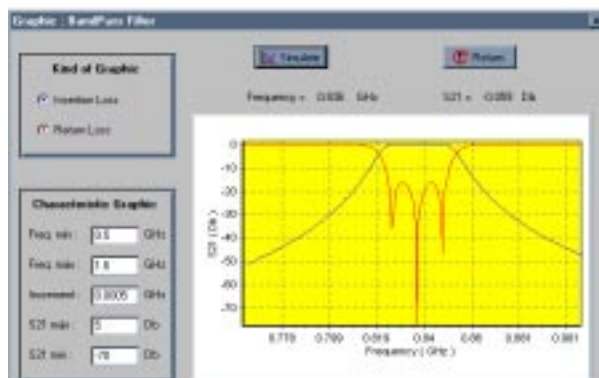Fig. 9. Main screen of the software
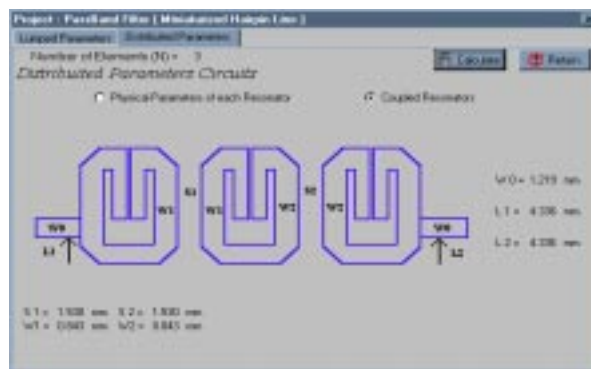


Fig.10. Frequency responses of the filter



Fig.11. Miniatured Hairpin line resonator Filter design

## 6. THE CAD FOR MULTIPLEXER

The software of complementary Diplexer and Triplexer design is composed of the analysis and synthesis modules. The analysis module contains a dedicated graphic interface that permits the visualization of the frequency response and input admittances of each filter. Fig. 12 shows a typical output of the frequency response module, in which it is possible to see the admittances curves, attenuation and the return losses for each filter.
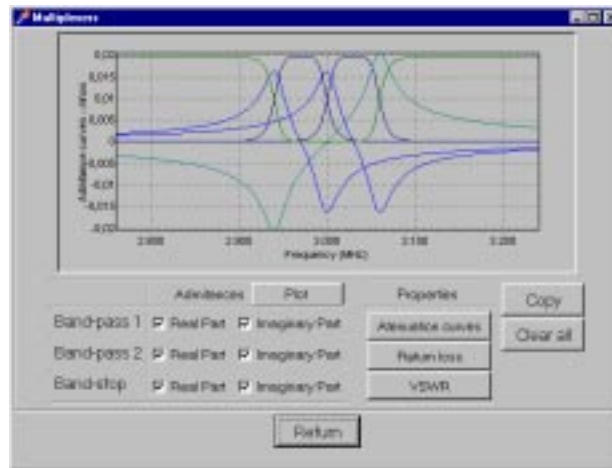


Fig.12. Frequency responses and admittance filters.

The synthesis module provides the layout of the filter, including all calculated dimensions, taking into account the chosen fabrication technology In the synthesis module, it is possible to obtain the project in lumped and distributed parameters. In the program, the distributed parameters design initiates calculating the isolated filters, and in sequence it calculates the interconnection network. The screen with the layout of the stripline triplexer is shown in Fig. 13.
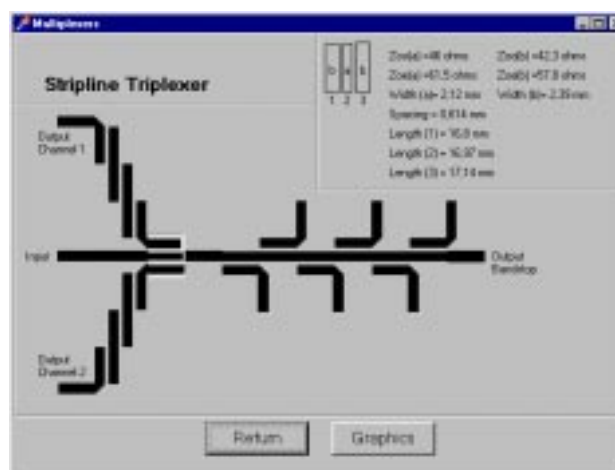


Fig.13. Stipline Triplexer design

# 7. THE CAD FOR OSCILLATOR

In this section we present the developed software operation structure to be used in the computer-supported project to microwave oscillators called *Oscillator Designer*.

In Fig. 14 are the elements from which the program is made of. Each of these blocks represents a step, which can be analyzed individually or together.
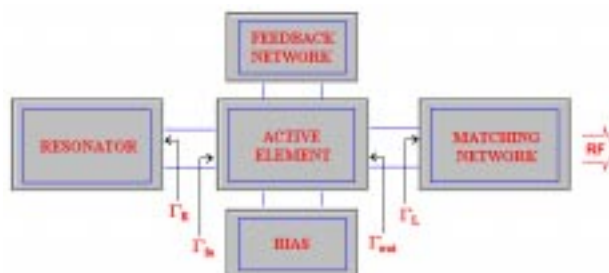


Fig. 14. Basic scheme for an oscillator.

Two types of transistor may be worked on by the oscillator designer: bipolar and FET. They are characterized by their spread parameters, [S], which are available in their manuals or in *s2p files, supplied by the manufacturers.

The software allows the user to choose an existing file or access a transistor editor. It is also possible to simulate every S parameter and line input and output impedance of the device.
The use of transistors implies in an adequate study of the polarization because [S] parameters supplied by the manufacturers are valid only to a single polarization. So the *Oscillator Designer* allows a DC and a RF analysis, i.e., the user may choose structures that guarantee the operation point of the transistor and also structures that protect the RF signal DC source.

Chosen the active element and the adequate topology of polarization, the next step is the feedback structure. An example of this structure proposed by the *Oscillator Designer* is shown in Fig. 15.
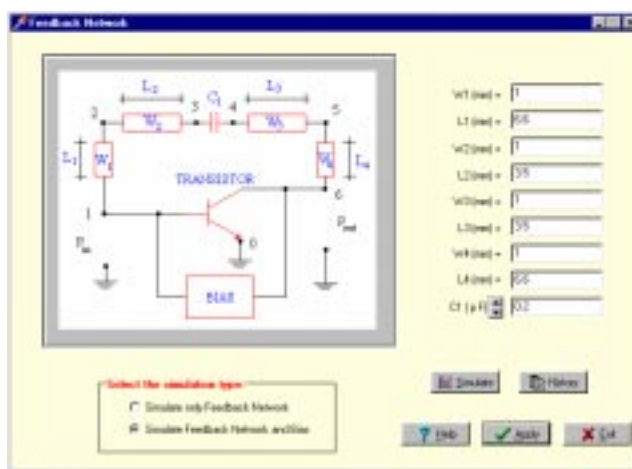


Fig.15. Feedback Network Screen.

Then maximum power should be supplied to the output of the device. This target is reached in the O*scillator Designer* through the impedance match using open or shortcut stubs.

A structure that supplies the desired frequency is chosen after completed all the steps above: active element, polarization circuit, feedback circuit and match circuit. The resonator is the passive element responsible for determining the oscillation frequency. Among the existing topologies, a *microstrip* that supplies a unitary reflection coefficient and opposite phase to the reflection coefficient in the active device gate was chosen. The *Oscillator Designer* allows the user to choose the optimum values for the length and width of the resonator.

The Smith Chart is one of the most used graphic tools that support spread parameters in circuit project. That is why this chart was incorporated in the *Oscillator Designer*. In every step described previously circuits can be simulated and circuit stability may be analyzed by [S] parameters, through what is called stability circles, as shown in Fig. 16. The complete configuration of the oscillator is shown in Fig. 17.
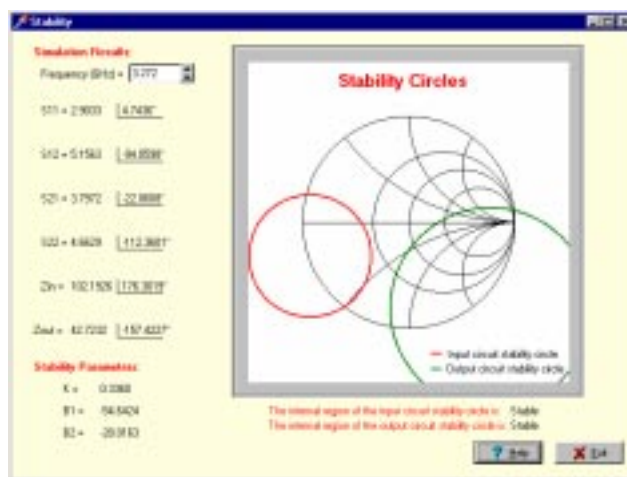


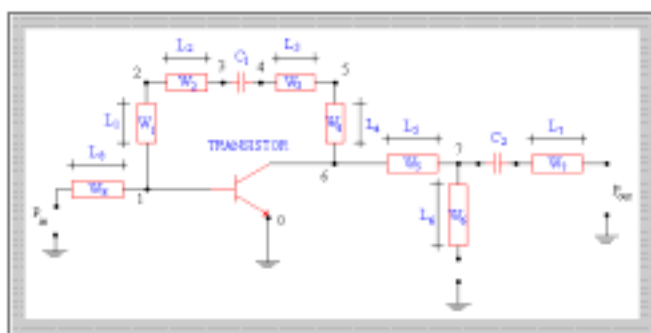Fig. 16 – Stability circles analysis.



Fig. 17. Designed microwave oscillator topology.

## 8. THE CAD FOR AMPLIFIER

Mr. Smith! is a software to design microwave amplifiers. Its friendly interfaces allow from data input, building a library of transistors and amplifiers, to the access of its several functions.
The following steps are the ones to project a single stage microwave amplifier:

1. First the transistor to be used in the project is chosen.
2. Then an input/output adaptation network is defined for a previously determined power gain and frequency or operational frequency range.
3. Chosen the impedances to be synthesized to promote a certain gain, it is time to define input/output adaptation networks.
4. Finally, a polarization network can be defined.

Besides supplying a library with unnumbered transistors the software allows the inclusion of other data-sheets. The transistor is chosen in the standard 'open' window of the Microsoft Windows, Fig. 18.
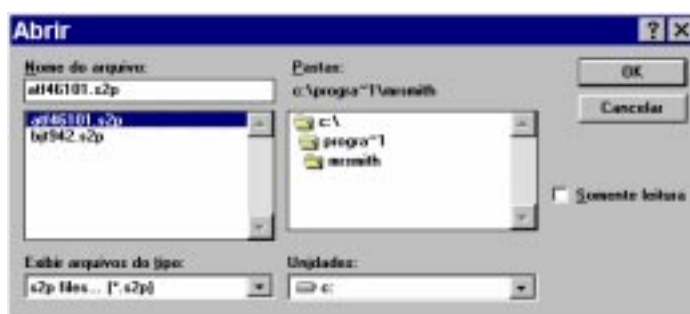


Fig. 18 – Input data.

A system benchmark analysis can be made using the Mr. Smith! simulation tools. The software presents a power gain analysis as a function of frequency: the user enters the regions where the transistor is unconditionally stable and also where the transistor can present power instabilities. Having this analysis as a basis the user may study a unique or multiple frequencies. Fig. 19 illustrates the analysis for the multiple frequencies case. In this case besides allowing a study to several frequencies in an amplifier operational band and to every match, Mr. Smith! displays to the designer the impedance, norm and phase profile that will be synthesized to the project of an amplifier with constant gain in a specific frequency band.
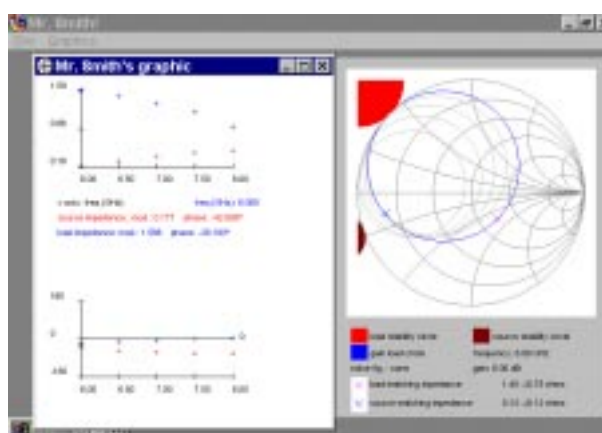


Fig. 19 – Multiple frequencies analysis.

After the impedance profile has been defined, which will allow the choice of an amplifier to a specific frequency; it is time to adjust the matching structures. Figure 20 illustrates a tool to a localized matching structures project.
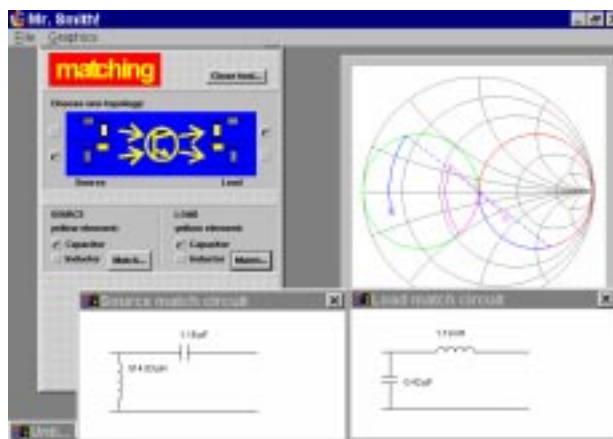


Fig. 20 – Adaptation network in localized technology.

Figure 21 shows an adaptation structures project in distributed technology: stripline and microstrip. Again every adaptation procedure is presented in Smith abacus.
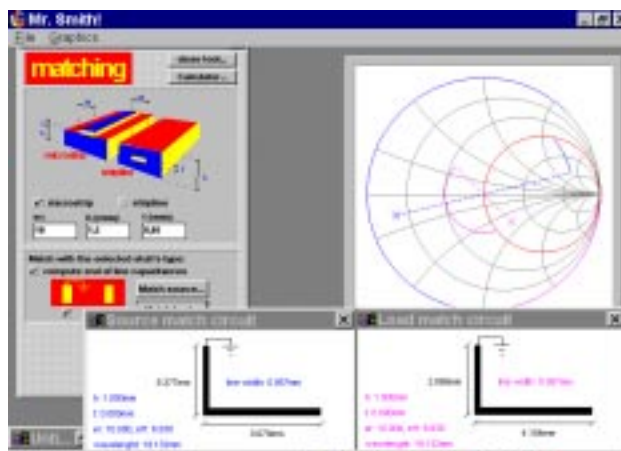


Fig. 21 – Adaptation networks in distributed technology.

Mr. Smith! presents to the user an auxiliary tool to the transistor polarization network project as shown in figure 22.
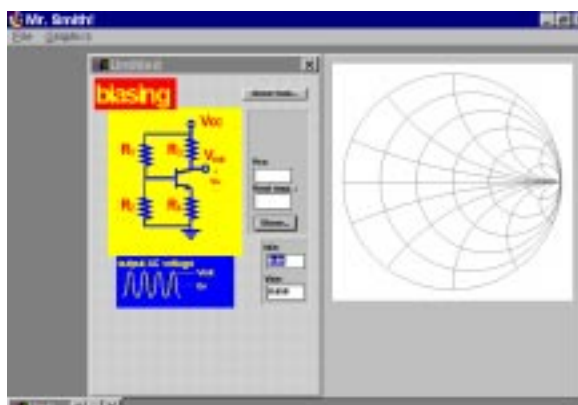
Fig. 22 – Polarization network.


## 9. CONCLUSION

The final product of this work is characterized as a software package for the design, analysis, and simulation, consecrated to the teaching and application of Microwave Circuits. The developed tool has been used in the discipline of Microwave Circuits of the degree course in electric engineering of the University of Brasilia. The positive reactions to this experiment are encouraging us for continuing the development of this type of approach.

It is thus possible to build up a complete, coherent, and stimulating tool that relieves the user of the computation development and lets them concentrate on the application they are interested in. This attractive tool makes learning more efficient when used in conjunction with a textbook.


## REFERENCES

[1] Wayne Tomasi, "Advanced Electronic Communications Systems", fifth edition, 2001, Prentice Hall

[2] H. Abdalla Jr *et al*, "Ambiente Multimídia Como Suporte para o Processo Educacional Interativo"*, I Congreso International "Retos de la Alfabetization Tecnológica en un Mundo en Red*", november 2000, Extremadura, Espanha.

[3] Inder Bahl, Prakash Bhartia, "Microwave Solid State Circuit Design, 1988, Wiley-Interscience

[4] M.F. Iskander, "Development of Multimedia Modules for Education", *Comput. Appl. Eng. Educ.*, vol. 3, No 2, 1995, pp.97-110.

[5] B.P. Lathi, "Modern Digital and Analog Communication Systems, third edition, 1998, Oxford University Press, Inc.

[6] Marek T. Faber, Jerzy Chramiec, Miroslaw E. Adamski. Microwave and Millimeter-Wave Diode Frequency Multipliers". ". Artech House, Inc. USA, 1995.

[7] Gabor C. Temes, Sanjit K. Mitra, "Modern Filter Theory and Design", John Wiley & Sons ,Inc, 1973, pp. 48-53.

[8] Bermúdez, Luís Afonso. *Osciladores de Microondas*. Universidade de Brasília. Departamento de Engenharia Elétrica. Julho, 1990.

[9] Sweet, Allen. MIC & MIC Amplifier and Oscillator Circuit Design. Artech House, 1990.