

# Software Tools for Research and Education

Dejan V. Tošić, Milka Potrebić

**Abstract** – We present a comparison of software tools for microwave circuit simulation from the research and educational view point. We highlight salient features of the tools and their relevance and suitability for research tasks and undergraduate teaching process. The accompanying 3D EM simulation functionality, provided by the tools, is considered as the mean for better understanding circuit models and their limitations as taught in microwave engineering curricula or used in research and development process.

**Keywords** – software tool, education and research, Microwave Office, WIPL-D Microwave, Ansoft Designer.

## I. INTRODUCTION

Modern microwave engineering education has many specific issues. Generally, a lot of material should be presented in the teaching process, in one-semester courses. Consequently, appropriate software tools should be used to illustrate microwave circuit simulation as easier as possible. In addition, students should be encouraged to practice computer simulations in their homework or projects [1], [2].

As we move towards a learner-oriented, self-paced, asynchronous system for higher and continuing education, the traditional course-based curriculum structure must be examined for its efficiency [3]. Courses can be restructured into primary concept modules that are interlinked to reflect the logical development of knowledge in the domain of the discipline being studied [4].

Software tools are indispensable in microwave engineering, so the corresponding courses should always address computer aided design and simulation in the teaching process [1].

With the wireless revolution, brought on mostly by cellular radio technologies, microwave applications have come to dominate the global industry. Cost, time to market, and manufacturing capacity are much stronger influences within the microwave engineering. Cost versus performance will always be a trade-off within any engineering project, however, the weighting coefficients have shifted [5]. Therefore, researchers and microwave practitioners naturally rely on software tools for Computer-Assisted Engineering (CAE) or Computer-Aided Design (CAD). Consequently, software tools should be treated as supplements to microwave courses covered by standard textbooks, such as [6] and [7].

In this paper we compare three software tools for microwave circuit simulation that can be used for both research and microwave engineering curricula.

Dejan V. Tošić and Milka Potrebić are with the School of Electrical Engineering, University of Belgrade, Bulevar kralja Aleksandra 73, 11000 Belgrade, Serbia, E-mail: tosic@etf.bg.ac.yu, milka\_potrebic@etf.bg.ac.yu

First, we consider AWR Microwave Office (MWO) [8]. Next, we present WIPL-D Microwave [9], a recently released tool. Finally, we present Ansoft Designer with Nexxim [10]. All these environments are evaluated from the research and educational viewpoint.

Section 2 highlights the importance of software tools and revisits the key concepts. Section 3 presents the MWO environment and Section 4 describes WIPL-D Microwave. Section 5 focuses on Ansoft Designer. Section 6 presents a simulation example. Section 7 presents the comparison of salient features of the tools from the educational view point. Section 8 gives a conclusion about this comparison from the teaching and research view point.

## II. IMPORTANCE OF SOFTWARE TOOLS

Generally speaking, any design that involves using computers may be called CAD. There have been extraordinary recent advances in CAD of RF/Microwave circuits, particularly in full wave electromagnetic (EM) simulations. They have been implemented both in commercial and specific in-house software and are being applied to microwave simulation, modelling, design, and validation. The developments in this area are certainly being stimulated by increasing computer power. Another driving force for the developments is the requirement of CAD for low-cost and high-volume production. In general, besides the investment for tooling, materials and labour mainly affect the cost of production. Labour costs include those for design, fabrication, testing, and tuning. Here the costs for the design and tuning can be reduced greatly by using CAD, which can provide more accurate design with less design iterations, leading to first-pass or tuneless design solutions [11].

More than easing computational burdens, the digital computer has revolutionized the way we design microwave circuits. Any modern treatment of microwave design must address computer-assisted strategies which have become practical including real-time tuning, statistical analysis, sensitivity analysis, design centering, and optimization. It is now feasible to optimize for desired and customized characteristics while simultaneously considering component losses, parasitics and discontinuities [12].

Modern circuit simulators fall into three major categories: linear simulators, SPICE-derived programs, and harmonic balance techniques. Efficient simulators have a few salient features in common: the circuit descriptions may be stored, recalled and edited, simulation is accurate and the process is general, that is, if the user can describe the network the simulator can compute the response.

Linear simulators primarily utilize closed form equations to compute the frequency domain response.

SPICE products are almost invariably based on SPICE2 or SPICE3, developed at the University of California, Berkeley, in the early 1970s. SPICE products solve non-linear differential equations for a network using iterative techniques because a closed-form solution is unknown. Both the frequency and time domains are supported.

Harmonic balance provides non-linear analysis, but is restricted to steady state behaviour.

The disadvantages of SPICE simulation worsen with increasing frequency. Also, time domain data is often of less interest to higher frequency engineers because of the difficulty associated with measuring voltages at high frequencies. Accurate high frequency network analyzers operate in the frequency domain. At lower frequencies, voltage data is less elusive and more insightful. For these reasons, low frequency (below about 10 MHz) engineers typically use SPICE simulators and high frequency engineers use linear simulators. Today, we are seeing more and more cross utilization of these simulator classes and this is healthy. High frequency engineers, when the extra effort of model study and accuracy checking are justified, can benefit from time domain and non-linear simulation. Low frequency engineers can benefit from the real-time tuning and optimization capabilities of a linear simulator to optimize the frequency domain performance of the circuit [12].

Harmonic balance simulators grew out of a need to resolve difficulties associated with SPICE simulation: slow execution, lack of convergence and, at high frequencies, accuracy and elusive active models. Although helpful, harmonic balance is unfortunately a compromise. Transient and dc simulation are unsupported, and accurate active device modelling is still difficult.

A fourth class of simulation, Volterra-Series, has non-linear capability. It is several times slower than linear nodal simulation, and therefore at least an order of magnitude slower than linear two-port simulation. However, this is fast in relation to other non-linear simulators. Unfortunately, only weakly non-linear circuit simulation is accurate, and the time-domain is unsupported. It is better suited for primarily linear devices, such as class-A amplifiers, than for oscillators and class-C amplifiers [12].

Computer-aided engineering (CAE) and computer-aided design (CAD) initially gained acceptance in the area of small-signal linear RF/Microwave circuit design. The cost-conscious consumer market forced large-scale product integration and CAD tools need to support a wider range of user base. More and more engineers are becoming generalists instead of being specialized in narrow fields, and they want integrated CAD programs to cover all their needs (e.g. circuit level simulation or full-wave 3D EM simulation) [13].

We cannot overemphasize the importance of systematic computer-aided circuit and system development in today's fierce global competitive environment. Production problems almost always can be traced to poor engineering development practices. Important steps, like finding the optimum circuit solution and performing yield analysis, are frequently overlooked to cut the development-to-market cycle. Design mistakes are expensive to fix in later stages because the cost

increases roughly by tenfold at each step of building the product [13].

In the 1990s several microwave circuit simulators were refined: Advanced Design Systems (ADS) by Agilent Technologies (Hewlett Packard), Ansoft Designer by Ansoft, Microwave Office (MWO) by Applied Wave Research, IE3D by Zeland Software, and Genesis by Eagleware (Agilent).

Most of the RF/Microwave newer circuit simulators now come with a bundled EM simulator instead of exchanging data with a standalone program. Cosimulation – performing circuit (schematic) level and physical circuit layout design concurrently – is becoming the standard in modern CAD. Example of this paradigm shift is recently released (Dec. 2005) WIPL-D Microwave software, which comes with the powerful full-wave 3D EM WIPL-D Pro simulation engine.

The old-fashioned form, where circuit engineers did the schematic-level design and turn it over to layout specialists, has been gradually replaced with the new form, where the circuit engineer has complete control of the circuit from inception through prototype realization until verification. As a result, instead of going through several costly and time-consuming physical prototype iterations that used to take months, the cycles now take place in the computer, in a matter of days [13].

### III. AWR MICROWAVE OFFICE

One of the most popular microwave software environments, in both academia and industry, is AWR Microwave Office (MWO). However, a professional tool like MWO can be over-sophisticated for teaching, so it might be desirable to deploy a more compact software solution instead.

MWO is the linear and nonlinear solution for microwave hybrid, module and monolithic microwave integrated circuit (MMIC) design. It includes linear, harmonic-balance, time-domain, electromagnetic (EM) simulation, and physical layout. It includes linear and nonlinear noise analysis. MWO is, essentially, a frequency domain simulator.

EM simulation is not based on the full-wave analysis (3D solver), but relies on a so called 2.5D solver that is suitable for layered structures, such as microstrip filters and antennas. It has predefined objects, such as rectangle, polygon, path, ellipse, drill hole, edge port, via, and so on. The 2.5D solver has only automatic meshing (subdividing structure into smaller parts for better accuracy) in three levels.

MWO can generate layout (implementation) view from the schematic. The layout view represents the structure that can be analyzed by the 2.5D EM solver. This is important for microwave structures when the circuit model does not take into account mutual couplings, parasitics, and discontinuities. The layout view has many properties and options, so it might be over-sophisticated for introductory microwave courses.

MWO cannot optimize physical structures, but it has a powerful optimizer for circuit model parameters.

Manual optimization, referred to as tuning, is provided so parameter values can be changed interactively while observing the resulting response. This feature is one of the very preferred features in research and industry.

MWO comes with a very useful transmission-line calculator TXLINE that can analyze or synthesize various lines, such as microstrip, stripline or coaxial.

Another MWO practical feature is its Filter Synthesis Wizard that can be used to design several microwave filter structures. The structures can include losses, but care should be taken when analyzing highpass filters: losses are modelled in an unusual way (software bug?) as shown in Fig. 1.

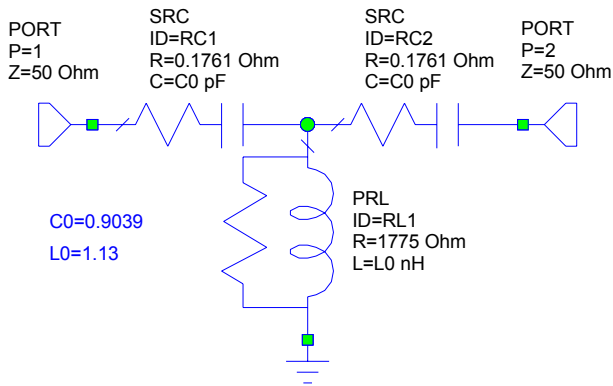


Fig. 1. Filter realization with losses generated by MWO.

#### IV. WIPL-D MICROWAVE

WIPL-D Microwave is a new design and simulation tool for microwave projects involving microwave circuits, components, and antennas. It comes with a full-wave 3D EM solver. It is a new tool in the microwave software arena, so we can describe it in some detail.

Except predefined circuit components, arbitrary composite metallic and dielectric structures can be built interactively. Whenever circuit level simulations are performed, the circuit parameters of the included 3D EM components are computed on-the-fly.

WIPL-D Microwave is intended for a large audience: engineers, practitioners, researchers, academia, and as a teaching tool for microwave engineering curricula.

Unique feature of this tool is the advanced modelling that includes:

- 1) Defining of structures by means of grids
- 2) Using symmetry to facilitate analysis
- 3) Modelling of the end effect and feed area for thick wires
- 4) Coaxial line excitation
- 5) Taking the edge effects into account
- 6) Modelling of layered structures
- 7) Refining the analysis
- 8) De-embedding of circuit parameters from the EM analysis.

WIPL-D Microwave features self-explanatory component palettes. Ideal palette, Fig. 2, contains all basic components relevant for introductory microwave courses, such as short-circuited end, open-circuited end, amplifier, ideal transformer, circulator, symmetric power splitter, and quadrature hybrid coupler.

The student can easily build idealized microwave circuits and networks and demonstrate the principal operation of basic microwave devices.

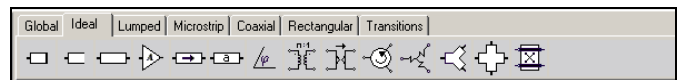


Fig. 2. Ideal components palette.

Technology-related components are grouped in four palettes: microstrip, coaxial, rectangular, and transitions.

WIPL-D Microwave has some microstrip components which are not available, for example in MWO, such as double step, and patch antenna, Fig. 3.

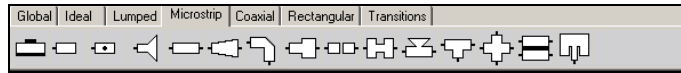


Fig. 3. Microstrip components palette.

Unique feature of WIPL-D Microwave is its coaxial palette with components, such as coaxial taper, band, step, gap, T-junction, and cross, Fig. 4.

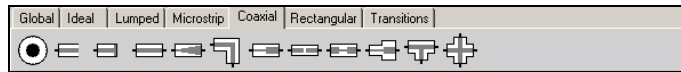


Fig. 4. Coaxial components palette.

Rectangular waveguide components palette contains: E- and H-post, E- and H-band, ET- and HT-junction, E- and H-coupled waveguides, rectangular horn antenna, and magic tee junction, Fig. 5.

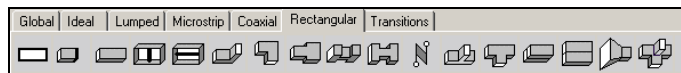


Fig. 5. Rectangular components palette.

Transitions palette makes students better understand interfacing microwave components implemented in different technologies, Fig. 6.

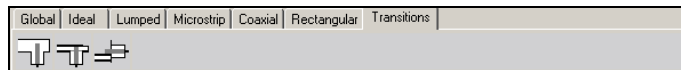


Fig. 6. Transitions components palette.

Ports of each component can be electrically extended, that is, equivalent transmission lines can be added to ports. Consequently, the student or researcher can build schematics with fewer elements. In addition, the student can experiment with shifting the reference planes which define component ports.

Another unique feature of WIPL-D Microwave is the possibility to specify symbolic parameters of geometrical structures and to optimize the 3D EM model by varying these parameters.

Moreover, this 3D EM modelling provides predefined objects, such as wires, plates, dielectric domains, sphere, circle, reflector, and body of revolution (a transition between two coaxial cables, a half-sphere with a hole, a ring with circular cross, a ring with a square cross sections).

Students and researchers can make their own full 3D EM models (with both metallic and dielectric objects) when the component parameters are out of the range over which the analytical model is valid, or when complex models are not available in the component library.

Full-wave 3D EM analysis enables students to explore differences between the results generated by analytical closed-form equations and accurate EM numerical simulations. In the other hand, researchers can further validate their solution by deploying refined EM analysis at final design steps.

In some cases, as microstrip filters and antennas, the edge effect substantially affects the global solution. This effect is taken into account by subdivision of plates in the vicinity of their edges into narrow strips, which is referred to as edging. Edging can be done manually, or automatically. Edging control is suitable for better teaching on parasitics and fringing effects. From the research viewpoint, it gives the user total control over the simulation process and guarantees more accurate results.

This software is provided with multiple component characterization so the user can specify a component as ideal, or analytical, or 3D electromagnetic. This kind of characterization enables students/researchers to make only one circuit schematic with different realization aspects. By overlaying the results of all characterizations, they can observe differences between ideal, analytical, and EM models.

## V. ANSOFT DESIGNER

Ansoft Designer is a high-performance RF/Microwave design and analog/RFIC verification software tool that provides an integrated schematic and design management front-end for complex analog, RF and mixed-signal applications.

By leveraging advanced electromagnetic-field simulators dynamically linked to powerful circuit and system simulation, Ansoft Designer enables engineers to design, optimize and validate component, circuit, and system performance long before building a prototype in hardware.

Ansoft Designer can perform digital-communication-system simulation, nonlinear circuit simulation, frequency domain, and transient analyses, and planar 3D EM simulation. It comprises an integrated IC and PCB layout editor with Java and Visual Basic scripting support, impulse invariance convolution engine, and swept parameter analysis.

Among the important features are support of parameterized subcircuits, global variables, and parameter scoping, support of compiled and interpretive user-defined models.

Design utilities, which come with Ansoft Designer, include real-time tuning, filter and transmission-line synthesis, and load-pull analysis.

This software is an advanced design environment with dynamic project manager and solution caching, and with a dynamic link with HFSS, a full-wave 3D electromagnetic structure simulator authored by Ansoft.

Like MWO, Ansoft represents a microwave circuit as an object which has three views: circuit schematic, physical layout, and planar EM model.

Circuit elements are grouped into abundant libraries that come with software, or that can be accessed or updated online.

Planar EM model can be constructed from predefined objects, such as circle, rectangle, arc, line, polygon, void, hole, via (2.5D or 3D), and cavity. Full 3D EM simulation can be performed and symbolic parameters of 3D EM models can be defined and optimized. Fixed meshing with edge meshing and adaptive meshing are supported.

Except the tuning feature, Ansoft Designer offers (1) optimization to change the design parameter values to meet the goal, (2) parametric analysis to determine how each design variation affects the performance of the design, (3) statistical analysis to determine the distribution of a design's performance, and (4) sensitivity analysis to determine the sensitivity of the design to small changes in variables.

Filter Design Wizard is a comprehensive and versatile module for filter synthesis.

Transmission Line Calculator is a useful module for analysis and synthesis of structures, such as microstrip lines, asymmetric coupled lines, Lange coupler, or coaxial cable.

Estimate module is a unique feature that helps a researcher or student to perform simple microwave computations, such as quarter-wave transformer, tuning stub, rectangular patch or circularly polarized patch antenna, or return loss.

## VI. SIMULATION EXAMPLE

A typical textbook example [11] is used to demonstrate simulation with the three software tools. Consider a lowpass filter with the following specification: cutoff frequency  $f_c = 1\text{GHz}$ , passband ripple 0.1dB, and source/load impedance  $Z_0 = 50\Omega$ .

The filter is realized as a seven-pole microstrip lowpass filter with the Chebyshev response. It is fabricated on a substrate with a relative dielectric constant  $\epsilon_r = 10.8$  and a thickness  $h = 1.27\text{mm}$  with negligible losses. High-impedance lines characteristic impedances are  $Z_{0L} = 110\Omega$  with width  $W_L = 0.1\text{mm}$ . Open-circuited stub has a line width  $W_C = 5\text{mm}$  on the used substrate.

The relevant design parameters of the microstrip lines, are  $l_1 = l_7 = 5.86\text{mm}$ ,  $l_2 = l_6 = 13.32\text{mm}$ ,  $l_3 = l_5 = 9.54\text{mm}$ ,  $l_4 = 15.09\text{mm}$ , Fig. 7.

An important issue, for both teaching and research, is to demonstrate differences between various simulation models: analytical, 2.5D EM, and full-wave 3D EM.

Filter circuit schematic and physical layout were created in all three software tools, Figs. 8-13. First, the circuit level simulation was performed to compute scattering parameters, Fig. 14. Next, electromagnetic simulation was carried out to obtain more accurate results.

Results of the two analyses are summarized in Fig. 15. Obviously, the results significantly differ in the filter stopband! To quantify numerical results, and the difference of the curves, a program Feature Selective Validation Tool [14] was used.

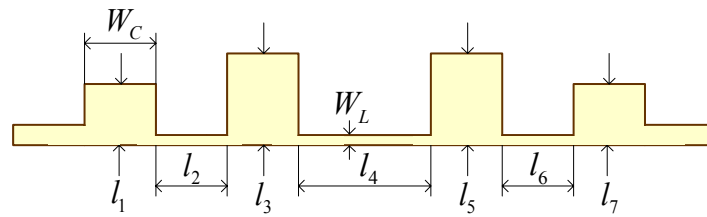


Fig. 7. Example filter implementation in microstrip technology, top view.

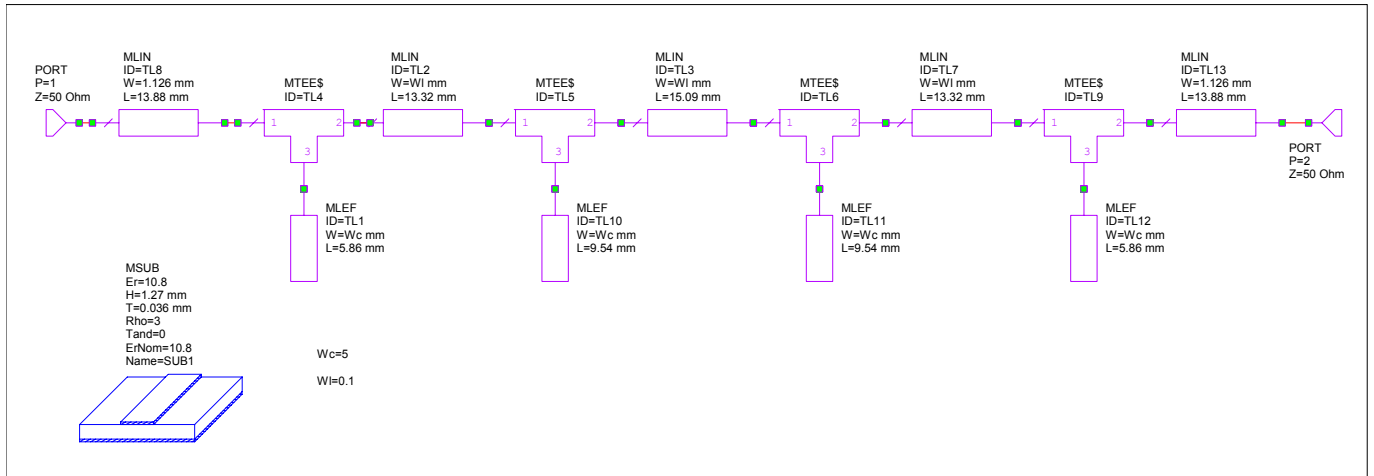


Fig. 8. AWR Microwave Office circuit schematic of the example filter.

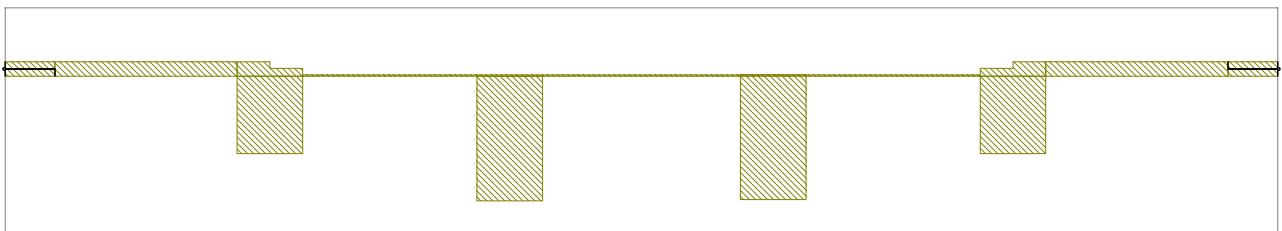


Fig. 9. AWR Microwave Office circuit 2D layout of the example filter.

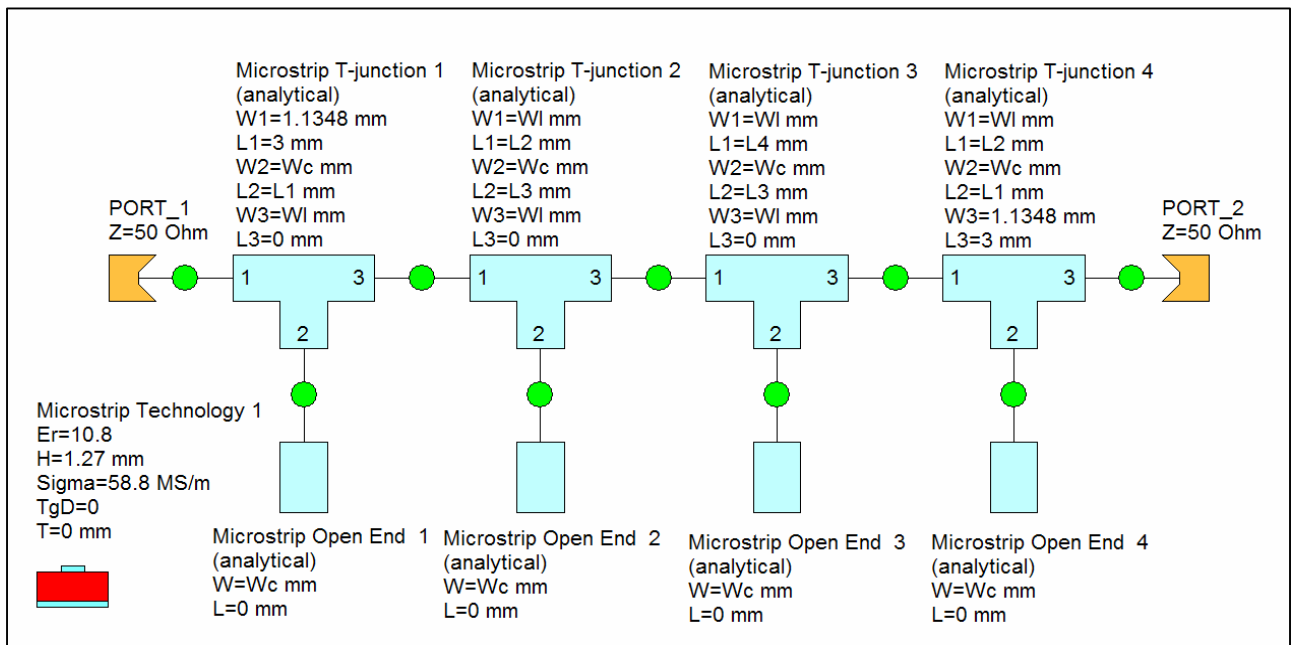


Fig. 10. WIPL-D Microwave circuit schematic of the example filter.

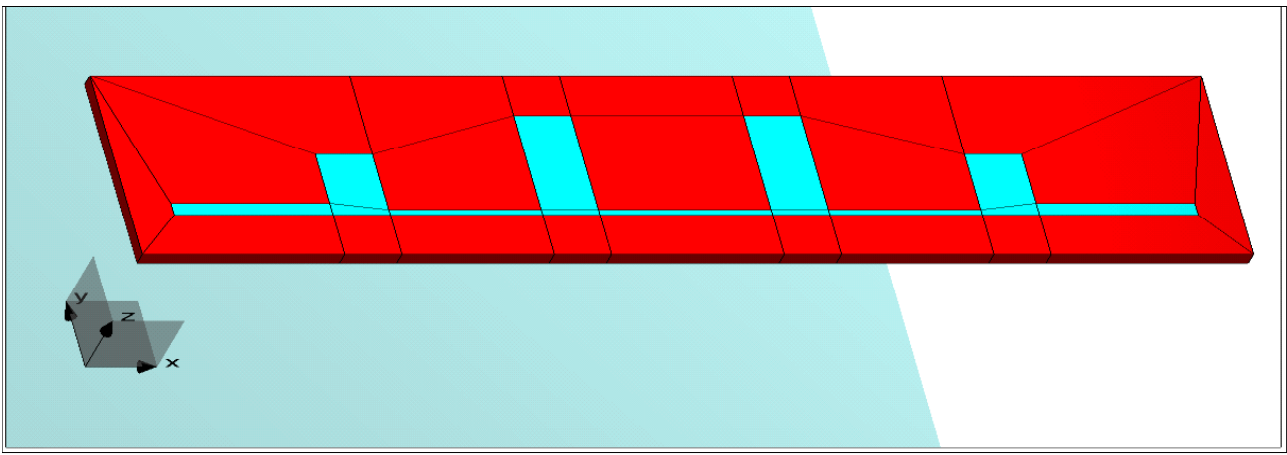


Fig. 11. WIPL-D Microwave circuit 3D layout of the example filter, which is used for full-wave 3D EM analysis.

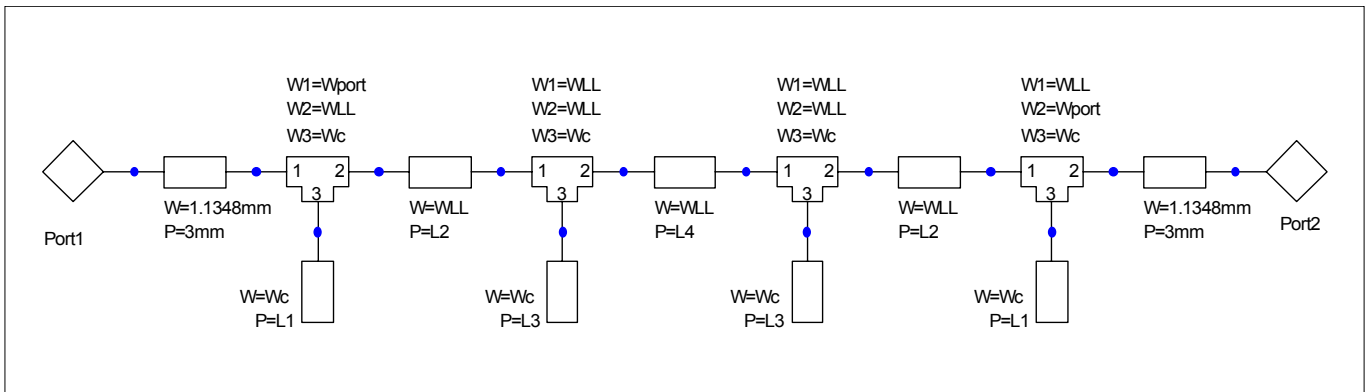


Fig. 12. Ansoft Designer circuit schematic of the example filter.

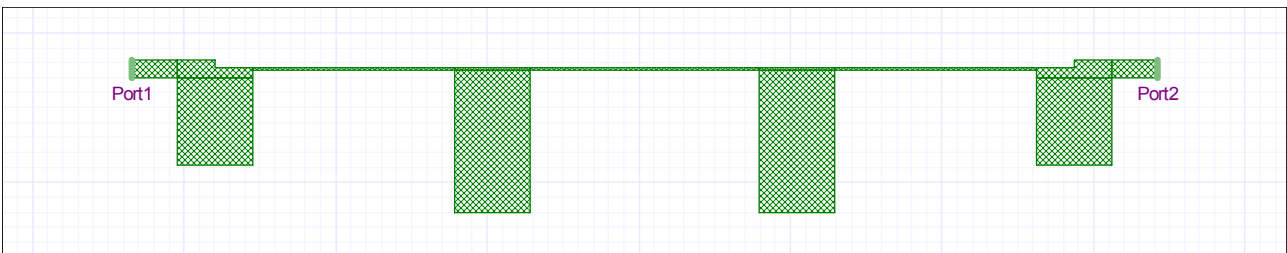


Fig. 13. Ansoft Designer circuit layout of the example filter.

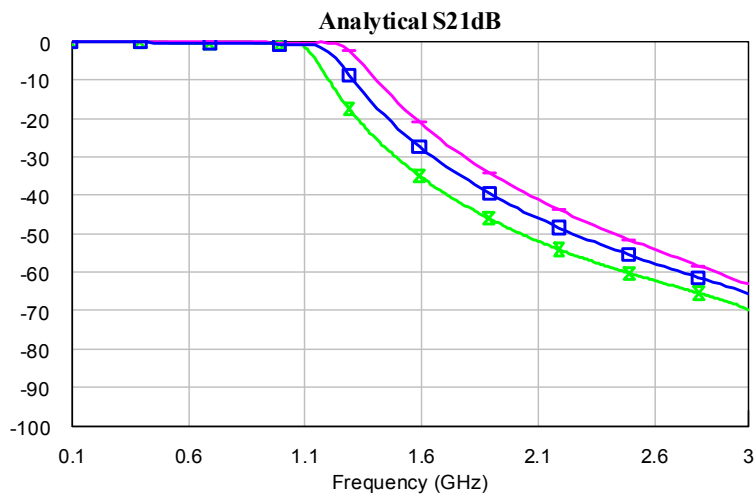


Fig. 14. Simulation of the example filter: results of the circuit-level analysis.

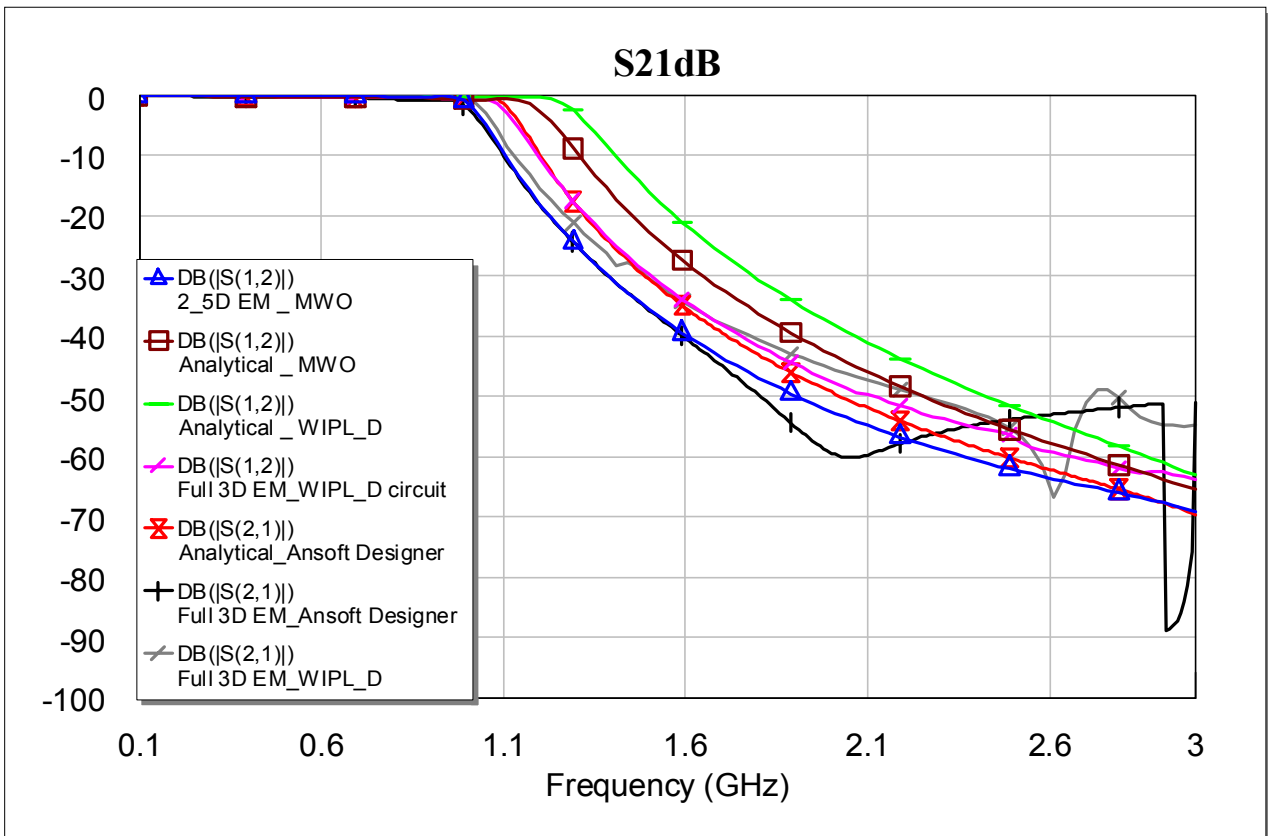


Fig. 15. Simulation of the example filter: results of the electromagnetic analysis.

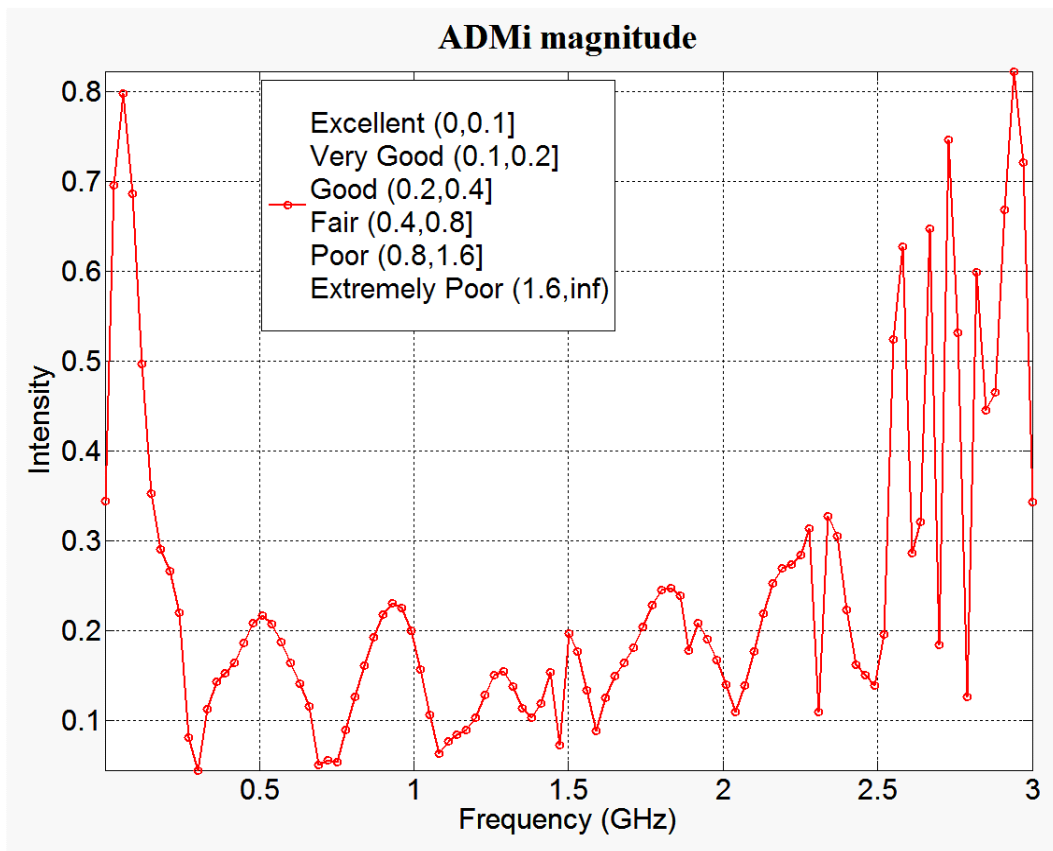


Fig. 16. Quantitative comparison of WIPL-D 3D EM and MWO 2.5D EM analyses by Feature Selective Validation Tool [14].

## VII. COMPARISON OF THE SOFTWARE TOOLS

WIPL-D Microwave is a candidate tool for implementing efficient microwave education. From the teaching view point, it has the following benefits:

- Contains all components and microwave circuit models needed for undergraduate microwave courses
- Ideal microwave elements are grouped into a separate toolbar, so the student easily builds idealized microwave circuits
- Numerous teaching examples are available and are based on the widely used textbooks [7] adopted in many microwave courses
- Comprehensive review of microwave circuit basics is provided, so students can quickly review the scattering matrix properties, element definitions, and other background lessons
- Full-wave 3D EM analysis is available so the student can compare the results generated by analytical closed-form equations and accurate EM numerical simulations
- Multiple component characterization is provided so the student can specify a component as ideal or analytical or 3D electromagnetic
- Technology-related components have integrated parameters and technology descriptors
- Arbitrary metallic-dielectric structures can be characterized by, for example, scattering parameters and incorporated into the schematic when the components parameters are out of the range over which the analytical model is valid
- WIPL-D optimizer can optimize all schematic parameters including the parameters of the embedded 3D EM models
- Ports consist of transmission lines with adjustable length that can be set to an arbitrary value, so a schematic can be built with fewer elements
- WIPL-D is affordable for students because of its low price.

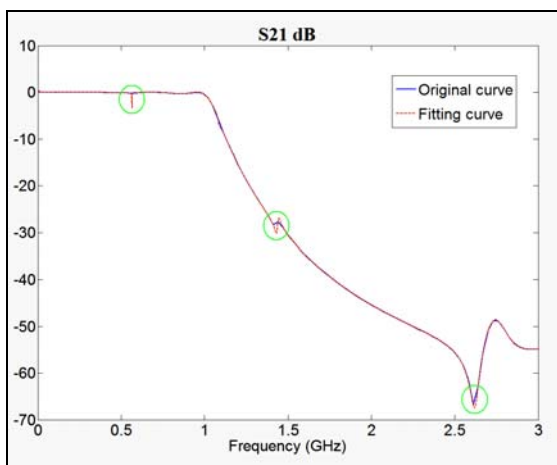


Fig. 17. WIPL-D Microwave curve fitting effects.

Here are some potential disadvantages of this environment:

- Time response cannot be computed
- Nonlinear circuits cannot be simulated
- Subcircuit (hierarchical decomposition) is not available
- Graphical presentation of simulation results by default smoothes data (fitting tool), Fig. 17, that might lead to unexpected curves (peaks); this might confuse students or researchers and cause them to misinterpret the results.

AWR Microwave Office is a professional software tool for advanced users in both academia and industry. Its main features, from the teaching and research view point, are as follows:

- Provides nonlinear microwave circuits and systems and the corresponding analysis (harmonic balance)
- Physical layout representation is assigned to each component in a schematic (sometimes, this option does not give correct layout)
- Arbitrary 2.5D EM multi-layer structures can be incorporated into the schematic
- Cannot optimize physical structure, but has a powerful manual and automatic optimizer for circuit model parameters
- Subcircuits are supported and can be used for building complex schematics
- Computes the time-domain response from the frequency response.

Here are some potential disadvantages of MWO:

- Numerous components and their grouping, sometimes, might be confusing for undergraduate students or novice researchers
- Layered EM structures can be modelled, and simulated by the MWO 2.5D EM simulator, but they cannot be optimized by the MWO optimizer
- Abundant advanced examples may not be suitable for undergraduate teaching process
- High price makes MWO less affordable for students and small research groups.

Ansoft Designer is a sophisticated professional software tool primarily targeted at industrial and research applications and experienced users. The chief benefits of this microwave software are as follows:

- Provides nonlinear models of microwave circuits and systems, and the corresponding analyses (e.g. harmonic balance)
- Physical layout representation is assigned to each component in a schematic and it can be automatically exported to planar electromagnetic model
- Full wave 3D EM analysis with possibility to incorporate arbitrary 3D EM multi-layer structures into the schematic
- Powerful manual and automatic optimizer for circuit and 3D EM model parameters
- Multiple component characterization is provided as analytical or 3D electromagnetic
- Technology-related components have predefined global substrates



- Computes the time-domain response and has a separate Nexxim module for SPICE analysis
- Subcircuits are supported and can be used for building complex schematics
- Microwave calculators are specialized built-in software modules for routine computations, such as microstrip analysis or synthesis and return loss
- Filter Design Wizard is a very power and useful built-in software module for efficient and practical filter design,

more versatile than the corresponding Microwave Office module.

Potential disadvantages of this excellent software are the lack of transition components (release 2.2, March 2005) such as coaxial line to rectangular waveguide, and a comparatively high prize. In addition, Ansoft Designer might appear to be over-sophisticated for undergraduate students and novice researchers and practitioners.

Table I summarizes the above comparison.

TABLE I  
COMPARISON OF MICROWAVE SOFTWARE TOOLS

	<b>Feature</b>	<b>MWO</b>	<b>Ansoft</b>	<b>WIPL-D</b>
<b>2.5D &amp; 3D</b>	<b>Full 3D simulation</b>	No	Yes	Yes
	<b>Predefined object</b>	Basic	Basic	Basic & Complex
	<b>Uniform/Non-uniform grid (EM model)</b>	Uniform	Uniform	Uniform & Non-uniform
	<b>Symbolic variables</b>	No	Yes	Yes
	<b>Optimization</b>	No	Yes	Yes
	<b>Symmetry planes</b>	No	No	Yes
	<b>Automatic check</b>	Yes	Yes	Yes
	<b>De-embedding</b>	Yes	Yes	Yes
	<b>Edging</b>	Automatic	Automatic	Automatic & Manual
	<b>EM estimator</b>	No	Yes	No
<b>Circuits &amp; Systems</b>	<b>Nonlinear components</b>	Yes	Yes	No
	<b>Transitions</b>	Only ideal	Only ideal	Yes
	<b>Symbolic variable</b>	Yes	Yes	Yes
	<b>Circuit optimization</b>	Yes	Yes	Yes
	<b>Layout (implementation) view</b>	Yes	Yes	No
	<b>Functional blocks (systems)</b>	Yes	Yes	No
	<b>Transmission line calculator</b>	Yes	Yes	Basic
	<b>Filter design wizard</b>	Yes	Yes, complex	No
<b>Cost Measures</b>	<b>S, Y, Z, G, H, ABCD, V, I, P</b>	Yes	Yes	Not all, such as: G,H, ABCD
	<b>Time domain reflectometry</b>	Yes	Yes	No
	<b>Near/Far field radiation</b>	Far	Near & Far	Near & Far
<b>Cost</b>	<b>Price</b>	Very high	Very high	Low

## VIII. CONCLUSION

We have evaluated and compared three microwave software tools from the educational and research perspective. Two tools are professional software that exist for several years on the market: AWR Microwave Office (version 7.0, release May 2006) and Ansoft Designer with Nexxim (version 2.2, release March 2005). The third software is a new software tool WIPL-D Microwave (version 1.0 Lite, release December 2005).

Our evaluation, and our impressions, might be summarized as follows:

(1) WIPL-D Microwave could be efficiently used for teaching, undergraduate and introductory microwave courses

(2) Ansoft Designer is the most versatile from all view points and it is recommended for experienced researchers, practitioners, professionals, and postgraduate or senior students

(3) AWR Microwave Office has the most convenient user-friendly interface, and it is generally suitable for research, design, and advanced microwave courses

## ACKNOWLEDGEMENT

We thank Ministry of Science and Environmental Protection of the Republic of Serbia for partial support of our research on this topic (Project No. TR-6154).

## REFERENCES

- [1] M. S. Gupta, "Curricular implications of trends in RF and microwave industry," *IEEE Microwave Magazine*, vol. 6, no. 4, pp. 58–70, Dec. 2005.
- [2] M. Potrebić, D. V. Tošić, "Comparison of WIPL-D Microwave and Microwave office software," in Proc. 50th Conf. ETRAN, Belgrade, 2006, in press.
- [3] K. C. Gupta, "Concept maps and modules for microwave education," *IEEE Microwave Magazine*, vol. 1, no. 3, pp. 56–63, Sept. 2000.
- [4] K. C. Gupta, R. Ramadoss, and H. Zhang, "RF and Microwave Network Characterization – A Tutorial," *University of Colorado at Boulder, USA, Concept-Modules LLC*, Boulder, CO, USA. [Online] [http://www.eng.auburn.edu/~ramadra/elec6340/S4-4/Article2/Module-2-with-Voice/Tutorial/index\\_tutorial.htm](http://www.eng.auburn.edu/~ramadra/elec6340/S4-4/Article2/Module-2-with-Voice/Tutorial/index_tutorial.htm)
- [5] J. S. Kenney, "Challenges and opportunities in microwave education," *IEEE Microwave Magazine*, vol. 5, no. 2, pp. 102–103, June 2004.
- [6] D. M. Pozar, *Microwave Engineering*, 3<sup>rd</sup> ed. John Wiley, 2005.
- [7] A. R. Đorđević, D. V. Tošić, *Microwave Engineering*, Akademska Misao, 2005. (in Serbian)
- [8] Microwave Office 2006, *Applied Wave Research*, USA, 2006. [www.appwave.com](http://www.appwave.com)
- [9] B. M. Kolundžija, J. S. Ognjanović, T. K. Sarkar, D. S. Šumić, M. M. Paramentić, B. B. Janjić, D. I. Olčan, D. V. Tošić, M. S. Tasić, *WIPL-D Microwave – Software and User's Manual*, Artech House, USA, 2005.
- [10] Ansoft Designer with Nexxim version 2.2, *Ansoft*, USA, 2005. <http://www.ansoft.com/ansoftdesigner>
- [11] Jia-Shen G. Hong, M. J. Lancaster, *Microstrip Filters for RF/Microwave Applications*, John Wiley, 2001.
- [12] R. W. Rhea, *HF Filter Design and Computer Simulation*, Noble Publishing, Norcross, GA, USA, 1994.
- [13] L. Besser, R. Gilmore, *Practical RF Circuit Design for Modern Wireless Systems*, Volume I—Passive Circuits and Systems, Artech House, 2003.
- [14] Feature Selective Validation (FSV) Tool, UAq EMC Laboratory, Italy, 2005. [Online] [http://ing.univaq.it/uaqemc/public\\_html](http://ing.univaq.it/uaqemc/public_html)