An Innovative CAD Technique for Microstrip Filter Design

Shen Ye, *Member, IEEE,* and Raafat R. Mansour, *Senior Member, IEEE*

***Abstract*— An innovative computer-aided design (CAD) technique for efficient and accurate microstrip filter design is presented in this paper. The technique utilizes full-wave electromagnetic (EM) simulation for the individual circuit elements, while interactions between nonadjacent elements are emulated by introducing circuit components to form extra signal paths. Designs can be accomplished with the accuracy of complete circuit EM simulation while keeping the computational efforts at a cascading simulation level which is crucial for design optimization. The technique has clear physical interpretations and is easy to implement. The authors have successfully applied this technique to design several microstrip filters. Very good filter performance was achieved with good correlation between predicted and measured results.**

***Index Terms*— CAD, EM simulation, filter, microstrip filters, optimization, superconductive filter.**

I. Introduction

IT HAS BEEN recognized that the most effective approach for microstrip filter design is to use a computer-aided desgin (CAD) procedure consisting of an optimization package and an accurate circuit simulator [1]–[3]. While a number of accurate electromagnetic (EM) simulators have become recently available for microstrip circuits, in most cases, their use has been only limited to design verifications. In view of the computation speed of today's computer workstations, the central processing unit (CPU) time required to include such EM simulators in the optimization loop is prohibitively large. More recently, this problem has been tackled in [4] using a space-mapping approach. In this paper, the authors propose an alternative technique which has clear physical interpretation, which is more efficient and easier to implement.

It is computationally superior to use the cascading approach for microstrip circuit simulation, where the circuit sections are simulated individually by the EM simulator and then cascaded together. However, because of the nature of the microstrip-type circuit, stray couplings generate interactions between different parts of the circuit outside the desired signal path. Moreover, performance deviations could exist between an isolated section and the section embedded in the full circuit. The inaccuracy inherited in the cascading approach are usually too large to neglect when it is applied to microstrip circuit design.

Manuscript received December 8, 1996; revised January 18, 1997. This work was supported in part by the Canadian Space Agency (CSA), in part by the Canadian Department of National Defense (DND), and in part by the U.S. Advanced Research Projects Agency (ARPA) under the Technology Reinvestment Program, NASA Cooperative Agreement NCC 3-344.

The authors are with COM DEV, Ltd., Cambridge, Ont., Canada N1R 7H6.

Publisher Item Identifier S 0018-9480(97)03096-2.

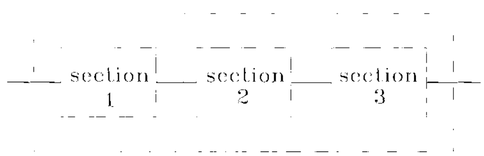
The proposed technique is intended to bridge the gap between the computational efficiency of the cascading approach and simulation accuracy of the full EM simulations. It is an analogy to the calibration concept typically employed in microwave testing procedures. In order to test a device, measurement instruments need to be calibrated to compensate any loss, distortion, etc., inherited in the measurement instruments and setup. In general, a microstrip circuit simulated by the cascading approach can be "calibrated" locally to give the same simulation results as by complete EM simulation.

This technique utilizes EM simulations for individual sections, while circuit components are employed to account for the interactions between the nonadjacent sections. The component parameter values are determined and subsequently modified by very few complete EM simulations of the whole circuit. Design optimization is performed only over the modified cascaded circuit. In this paper, the authors present results of two high-temperature superconductive (HTS) microstrip filters designed using this technique. The authors also provide a detailed description of the steps followed in designing one of these filters.

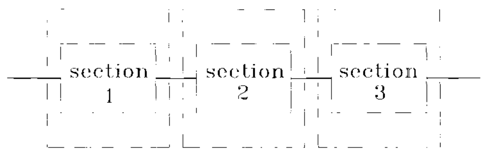
II. Description of the New Design Technique

Consider the simple symbolic example shown in Fig. 1(a). A microstrip circuit consists of three cascaded sections along the signal path. Due to stray coupling effects the discrepancy between cascading simulation shown in Fig. 1(b) and complete EM simulation from Fig. 1(a) could be considerably large. Instead of resorting to complete EM simulation for the design process, one may modify the original cascaded circuit Fig. 1(b) such that it provides the same results as the complete EM simulation. The modification must have physical interpretation. For this particular example, one knows that Section I not only connects to Section II, but also has interactions with Section III. Fig. 2 illustrates one implementation of such a modification: a signal path (Section IV) has been added to emulate the interactions between Sections I and III. Circuit components which form the new signal path are characterized by taking a complete EM simulation as reference, or *calibration standard*.

Consider the design parameters of the microstrip circuit as $\boldsymbol{x}$. Let the cascade simulation response be defined as $\boldsymbol{R}_{\mathrm{cas}}(\boldsymbol{x})$, while the complete EM simulation response be defined as $\boldsymbol{R}_{\mathrm{em}}(\boldsymbol{x})$. The design parameter $\boldsymbol{x}$ is a vector containing all the microstrip circuit geometry parameters, including fixed parameters $\boldsymbol{x}_{\mathrm{fix}}$ and variable parameters $\boldsymbol{x}_{\mathrm{opt}}$. Cascade simulation is performed by dividing the microstrip



(a)



(b)

Fig. 1. A symbolic three-section microstrip circuit. (a) Circuit schematic. (b) Illustration for cascading analysis.

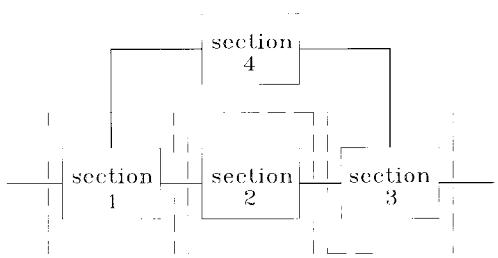


Fig. 2. Modified cascaded circuit from Fig. 1 to include interactions between Sections I and III.

circuit into individual sections and treating them as regular circuit components, e.g., [5]. Nonadjacent interactions among those individual sections are neglected. The complete EM simulation is done by simulating the microstrip circuit as a complete unit, and, therefore, all possible interactions are included. In general,

$$\boldsymbol{R}_{\text{cas}}(\boldsymbol{x}) \neq \boldsymbol{R}_{\text{em}}(\boldsymbol{x}) \tag{1}$$

where the difference between $\boldsymbol{R}_{\text{cas}}(\boldsymbol{x})$ and $\boldsymbol{R}_{\text{em}}(\boldsymbol{x})$ can be attributed to the nonadjacent interactions between the cascade sections and possible performance deviations of individual sections when they are simulated alone and when they are embedded in a circuit.

To compensate the inaccuracy of the cascade simulation, the original cascade circuit is modified to accommodate the nonadjacent interactions, such as the one in Fig. 2, and the inaccuracy of individual section simulations. The modified cascade circuit response is defined as

$$\boldsymbol{R}'_{\text{cas}}(\boldsymbol{x}+\boldsymbol{x}_c, \boldsymbol{y}_c)$$

where $\boldsymbol{x}_c$ is the geometry parameter modification to compensate the characteristic deviation of individual sections, and $\boldsymbol{y}_c$ is the parameter vector characterizing the addition signal paths introduced to model the extra interactions. The objective is to achieve

$$\boldsymbol{R}'_{\text{cas}}(\boldsymbol{x}+\boldsymbol{x}_c, \boldsymbol{y}_c) \approx \boldsymbol{R}_{\text{em}}(\boldsymbol{x}). \tag{2}$$

In other words, $\boldsymbol{x}_c$ and $\boldsymbol{y}_c$ should be obtained by

$$\min_{\boldsymbol{x}_c, \boldsymbol{y}_c} \|\Delta \boldsymbol{R}'(\boldsymbol{x}_c, \boldsymbol{y}_c)\| \tag{3}$$

where

$$\Delta \boldsymbol{R}'(\boldsymbol{x}_c, \boldsymbol{y}_c) = \boldsymbol{R}'_{\text{cas}}(\boldsymbol{x}+\boldsymbol{x}_c, \boldsymbol{y}_c) - \boldsymbol{R}_{\text{em}}(\boldsymbol{x}). \tag{4}$$

It should be emphasized that $\boldsymbol{x}_c$ and $\boldsymbol{y}_c$ are only accurate for a given $\boldsymbol{x}$, i.e., $\boldsymbol{R}'_{\text{cas}}(\boldsymbol{x}+\boldsymbol{x}_c, \boldsymbol{y}_c)$ is a local model. If $\boldsymbol{x}$ is changed to $\boldsymbol{x}+\Delta\boldsymbol{x}$, different $\boldsymbol{x}_c$ and $\boldsymbol{y}_c$ must be calculated to maintain

$$\boldsymbol{R}'_{\text{cas}}(\boldsymbol{x}+\Delta\boldsymbol{x}+\boldsymbol{x}_c, \boldsymbol{y}_c) \approx \boldsymbol{R}_{\text{em}}(\boldsymbol{x}+\Delta\boldsymbol{x}). \tag{5}$$

To utilize $\boldsymbol{R}'_{\text{cas}}(\boldsymbol{x}+\boldsymbol{x}_c, \boldsymbol{y}_c)$ for design purpose, $\varepsilon > 0$ is specified such that if $\|\Delta\boldsymbol{x}\| < \varepsilon$, there exists $\sigma > 0$ and

$$\|\boldsymbol{R}'_{\text{cas}}(\boldsymbol{x}+\Delta\boldsymbol{x}+\boldsymbol{x}_c, \boldsymbol{y}_c) - \boldsymbol{R}_{\text{em}}(\boldsymbol{x}+\Delta\boldsymbol{x})\| < \sigma \tag{6}$$

where σ is the acceptable tolerance. In other words, one accepts $\boldsymbol{R}'_{\text{cas}}(\boldsymbol{x}+\Delta\boldsymbol{x}+\boldsymbol{x}_c, \boldsymbol{y}_c)$ as the accurate circuit response as long as $\|\Delta\boldsymbol{x}\| < \varepsilon$. The model validity region ε can be determined according to available computational resources and the design accuracy requirement.

The whole design procedure could be described as follows.

Step 1—Initial Design: The initial design $\boldsymbol{x}$ can be obtained using optimization and conventional cascade simulation approach.

Step 2—Calibration: Perform an EM simulation $\boldsymbol{R}_{\text{em}}(\boldsymbol{x})$ for the complete circuit; determine $\boldsymbol{x}_c$ and $\boldsymbol{y}_c$ from (3), such that the modified cascaded circuit simulation $\boldsymbol{R}'_{\text{cas}}(\boldsymbol{x}+\boldsymbol{x}_c, \boldsymbol{y}_c)$ gives the same response as that from EM simulation $\boldsymbol{R}_{\text{em}}(\boldsymbol{x})$.

Step 3—Optimization: Optimization is performed to meet the design specifications using $\boldsymbol{R}'_{\text{cas}}(\boldsymbol{x}+\boldsymbol{x}_c, \boldsymbol{y}_c)$ with $\boldsymbol{x}$ as optimization variables; new design $\boldsymbol{x}_1$ is obtained from optimization, where $\boldsymbol{x}_1$ denotes the updated design parameters after optimization.

Step 4—Verification: Compare the parameter values of the new design $\boldsymbol{x}_1$ (Step 3) with the parameter values $\boldsymbol{x}$ (Step 2) where the calibration was done, i.e., calculate $\Delta\boldsymbol{x} = \boldsymbol{x}_1 - \boldsymbol{x}$. If $\|\Delta\boldsymbol{x}\| > \varepsilon$, assign $\boldsymbol{x}_1$ to $\boldsymbol{x}$, and go to Step 2, otherwise the design is complete.

In the case that uniform grid size is required by the EM simulator, e.g., [6], $\boldsymbol{R}_{\text{em}}$ can only be performed on $\boldsymbol{x}_r$ where $x_{ri}, i = 1, \cdots, n$, must be in the multiples of the corresponding grid size, even though in cascade simulation one can use interpolation for each section calculation. Similar to (3)–(6), the design procedure can be modified as follows.

Step 1—Initial Design: Obtain initial design $\boldsymbol{x}$ using optimization and conventional cascade simulation approach.

Step 2—Calibration: Round $\boldsymbol{x}$ to $\boldsymbol{x}_r$, such that

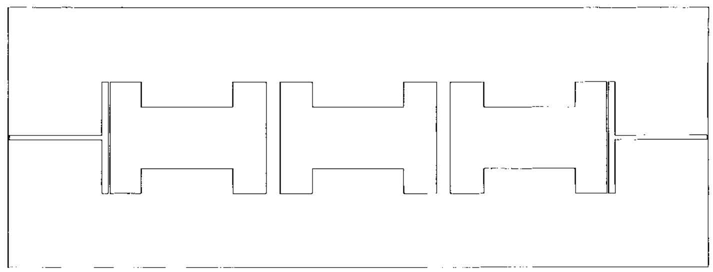
$$\Delta\boldsymbol{x}_r = \boldsymbol{x}_r - \boldsymbol{x} \tag{7}$$

and

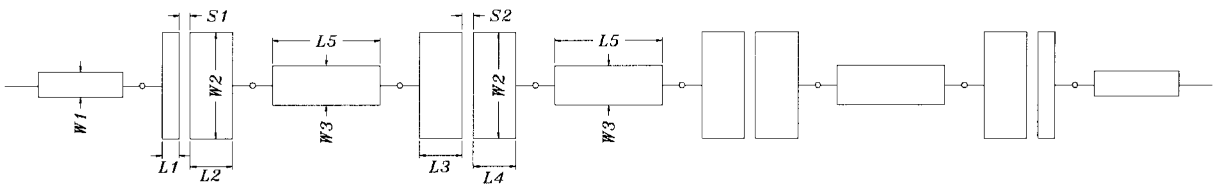
$$\max\{|\Delta\boldsymbol{x}_{ri}|, i = 1, \cdots, n\} \le \frac{\text{(corresponding grid size)}}{2} \tag{8}$$

where n is the total number of geometry parameters. Perform an EM simulation $\boldsymbol{R}_{\text{em}}(\boldsymbol{x}_r)$ for the complete circuit; determine $\boldsymbol{x}_c$ and $\boldsymbol{y}_c$ from

$$\min_{\boldsymbol{x}_c, \boldsymbol{y}_c} \|\Delta \boldsymbol{R}'(\boldsymbol{x}_c, \boldsymbol{y}_c)\| \tag{9}$$



(a)



(b)

Fig. 3. (a) A three-pole 1% Chebyshev microstrip filter. (b) Corresponding sections for cascading analysis.

where

$$\Delta \boldsymbol{R}'(\boldsymbol{x}_c, \boldsymbol{y}_c) = \boldsymbol{R}'_{\text{cas}}(\boldsymbol{x}_r + \boldsymbol{x}_c, \boldsymbol{y}_c) - \boldsymbol{R}_{\text{em}}(\boldsymbol{x}_r). \qquad (10)$$

Step 3—Optimization: Optimization is performed to meet the design specifications using the new cascaded circuit $\boldsymbol{R}'_{\text{cas}}(\boldsymbol{x} + \boldsymbol{x}_c, \boldsymbol{y}_c)$; $\boldsymbol{x}_1$ is obtained from optimization, where $\boldsymbol{x}_1$ denotes the updated design parameters after optimization.

Step 4—Verification: Compare the parameter values of the new design $\boldsymbol{x}_1$ (Step 3) with the rounded parameter values $\boldsymbol{x}_r$ from Step 2, i.e., calculate $\Delta \boldsymbol{x} = \boldsymbol{x}_1 - \boldsymbol{x}_r$. If $\|\Delta \boldsymbol{x}\| > \varepsilon$, assign $\boldsymbol{x}_1$ to $\boldsymbol{x}$, and go to Step 2, otherwise the design is complete.

The design procedures discussed above assume that the designer is knowledgeable in choosing an appropriate $\boldsymbol{R}'_{\text{cas}}(\boldsymbol{x} + \boldsymbol{x}_c, \boldsymbol{y}_c)$, i.e., to configure additional signal paths ($\boldsymbol{y}_c$) which correspond to important stray couplings not modeled by the original cascade circuit.

It should be pointed out that the concept of the technique presented here applies not only to microstrip filters but also to other types of structures where differences between cascade simulation and complete simulation exist and are important issues to the design. The new technique presented here is highly efficient for design optimization of microstrip-type circuits where demanding accuracy is required and employing complete EM simulation in the optimization loop is computationally not feasible.

III. Design Examples

Two HTS filters are designed and tested using this new design technique. The resonator size of the filters are relatively large in order to achieve higher Q and higher power handling capability. However, the use of a larger resonator size significantly increases the cross couplings between nonadjacent resonators.

Fig. 3(a) shows the layout of a three-pole Chebyshev filter with 1% bandwidth. Individual cascaded sections are illustrated in Fig. 3(b) where reference planes of each individual section are represented by circuit nodes and the step discontinuities of each double-patch capacitor were properly considered and deembedded in actual EM simulation. The filter was designed on 20-mil thick $LaAlO_3$ substrate with estimated dielectric constant 23.5.-EM simulation for individual sections and for a complete filter was done using ***em*** [6] which requires uniform grid sizes. Interpolation was implemented to overcome the uniform grid size limitation. ***osa90*** [7] was utilized for cascaded circuit simulation and optimization.

The design parameters of the filter are indicated in Fig. 3(b), where the filter is symmetrical and W_1, W_2, W_3, and L_5 are fixed at 7, 180, 100, and 150 mil, respectively. A grid size of 2 mil $\times$ 2 mil was used in individual section EM simulation and a grid size of 1.5 mil $\times$ 2 mil was used for the complete circuit

TABLE I

		Initial Design	Iteration 1		Iteration 2 (final)	
		cascade	calibrate after rounding to 1.5 x 2.0 grid size	re-optimize using modified cascade circuit	calibrate after rounding to 1.5 x 2.0 grid size	re-optimize using modified cascade circuit
x (mil)	L_1	10.4634	10.5	10.5156	10.5	10.5122
	L_2	50.7438	51.0	51.0	51.0	51.0
	S_1	3.14007	3.0	3.24963	3.0	3.2336
	L_3	56.0262	55.5	55.2983	55.5	55.3088
	L_4	53.9612	54.0	53.7643	54.0	53.772
	S_2	21.1987	21.0	22.3968	22.5	22.4172
x_c (mil)	ΔL_1	N/A	0.0	0.0	0.0	0.0
	ΔL_2	N/A	0.0	0.0	0.0	0.0
	ΔS_1	N/A	0.0	0.0	0.0	0.0
	ΔL_3	N/A	1.49986	1.49986	1.47298	1.47298
	ΔL_4	N/A	0.258669	0.258669	0.25465	0.25465
	ΔS_2	N/A	-0.938043	-0.938043	-1.01134	-1.01134
y_c (pf)	C_1	N/A	-0.097049	-0.097049	-0.094469	-0.094469
	C_2	N/A	0.004863	0.004863	0.004863	0.004863

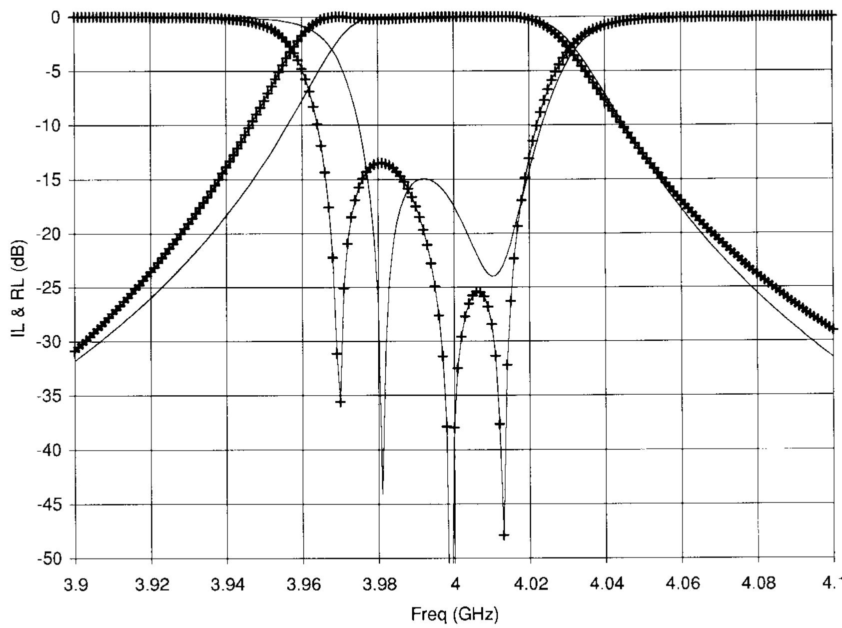
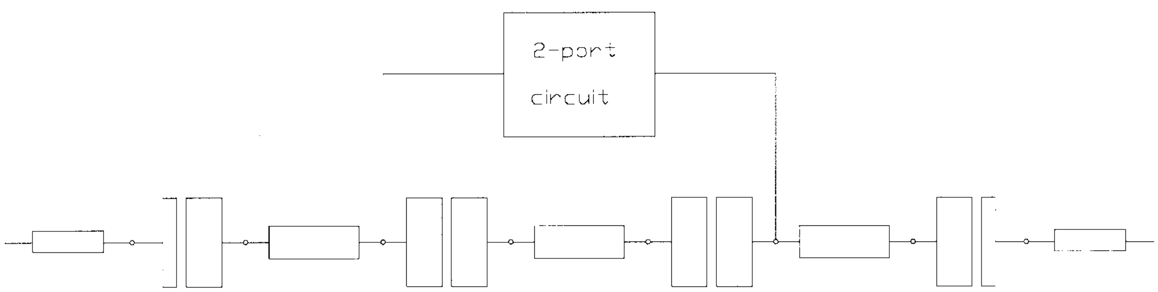
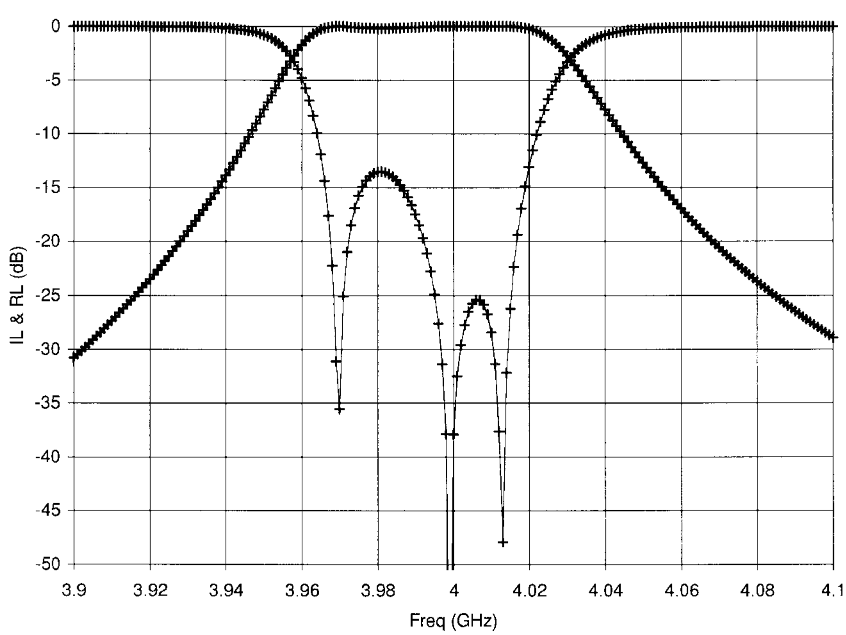


Fig. 4. Comparison between cascading analysis and complete full EM simulation for the circuit show in Fig. 3, where all the dimensions are rounded to a uniform grid size. Solid lines are from cascading circuit simulation and crossed lines are from complete circuit EM simulation.



(a)



(b)

Fig. 5. (a) Modified cascaded circuit with an added signal path between resonator one and resonator three. (b) Comparison between modified cascading analysis and full EM simulation. Solid lines are from cascading circuit simulation and crossed lines are from complete circuit EM simulation.

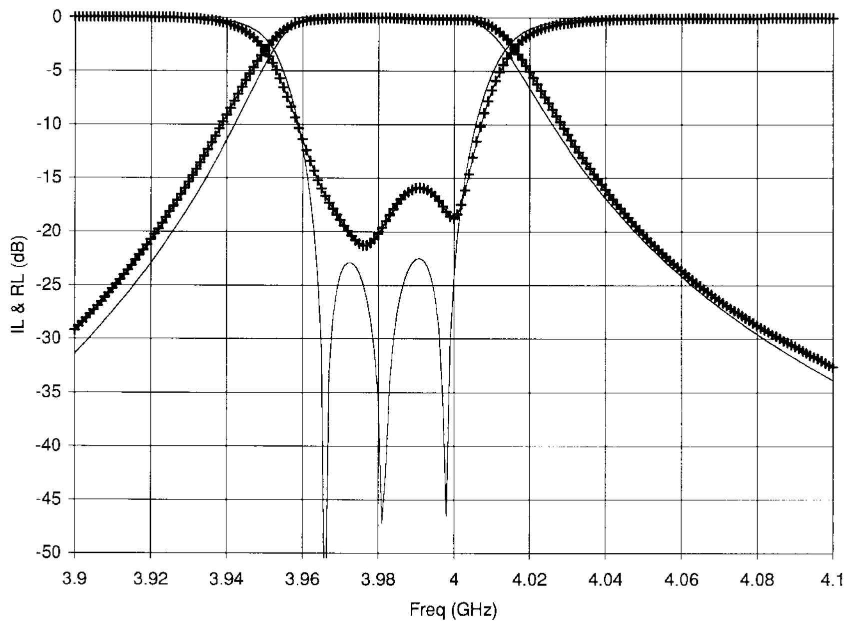


Fig. 6. Comparison between measured and predicted performance of the three-pole Chebyshev microstrip filter. Solid lines are from predicted performance and crossed lines are from measured performance.

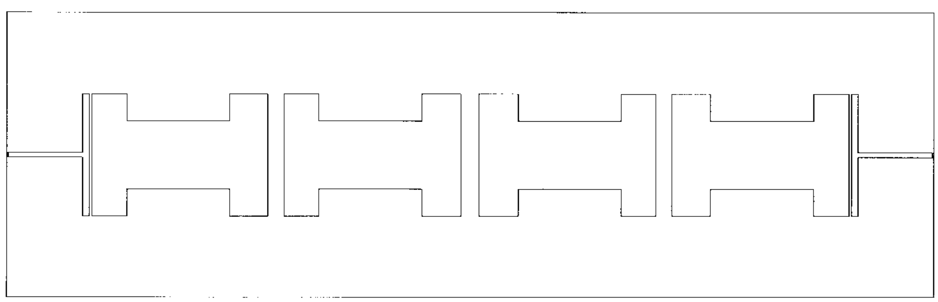


Fig. 7. A four-pole Chebyshev microstrip filter designed using the proposed technique.

EM simulation. Table I provides $\boldsymbol{x}$, $\boldsymbol{x}_c$, and $\boldsymbol{y}_c$ at different stages of the design, where fixed parameters are omitted to simplify the table.

After completing the initial design using a cascade circuit shown in Fig. 3(b), $\boldsymbol{x}$ was obtained as listed under the "cascade" column of "Initial Design" in Table I. This $\boldsymbol{x}$ was then rounded to the values listed under the "calibrate after rounding" column of "Iteration 1." The comparison between cascade circuit simulation $\boldsymbol{R}_{\rm cas}(\boldsymbol{x}_r)$ and full EM simulation $\boldsymbol{R}_{\rm em}(\boldsymbol{x}_r)$ is shown in Fig. 4. Large deviation can be observed.

To apply the new technique discussed in Section II, the original cascade model shown in Fig. 3(b) is modified by adding one new signal path representing stray couplings between resonators one and three as shown in Fig. 5(a). For this particular example, the signal path $\boldsymbol{y}_c$ consists of capacitors C_1, C_2 and a 50-Ω microstrip transmission line with 600-mil fixed length, where C_1 is a series capacitor and C_2 is a shunt capacitor. Following the design procedure presented in Section II, $\boldsymbol{x}_c$ and $\boldsymbol{y}_c$ were determined from (9), as listed under "calibrate after rounding" column of "Iteration 1" in Table I. Fig. 5(b) shows the modified cascade circuit simulation $\boldsymbol{R}'_{\rm cas}(\boldsymbol{x}_r + \boldsymbol{x}_c, \boldsymbol{y}_c)$ versus full EM simulation $\boldsymbol{R}_{\rm em}(\boldsymbol{x}_r)$. The design was then re-optimized according to the new calibrated cascade circuit. The design was finished after two iterations where each iteration involves one full EM simulation to calibrate the cascaded circuit and an optimization using the modified cascade circuit. Note that the second iteration is required since the difference of S_2 between rounded and re-optimized values under "Iteration 1" is greater than half grid size 0.75 mil which was the model validity region specified.

The filter was built and tested using TBCCO HTS thin film on $LaAlO_3$ substrate. Fig. 6 gives the designed and measured performances. Excellent results have been obtained. The deviations are attributed to the following major factors:

1) frequency shift due to the dielectric constant difference between the designed and actual;
2) slightly wider bandwidth due to the limited accuracy of the full EM simulation for this particular example.

A similar structure four-pole 1% Chebyshev filter was designed as shown in Fig. 7. The filter was first designed using

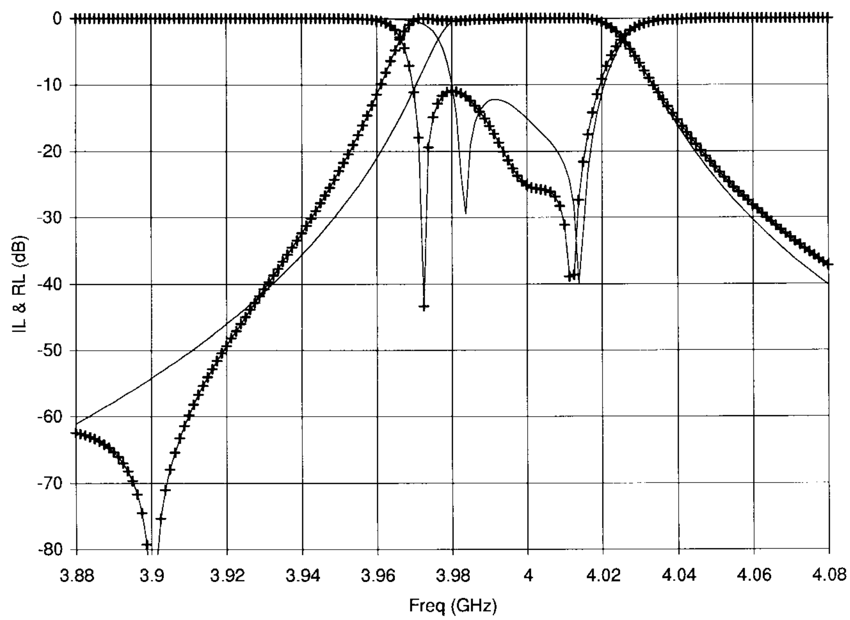


Fig. 8. Comparison between cascading analysis and complete full EM simulation for the circuit shown in Fig. 7 after initial design, where all the dimensions are rounded to a uniform grid size. Solid lines are from cascading circuit simulation and crossed lines are from complete circuit EM simulation.

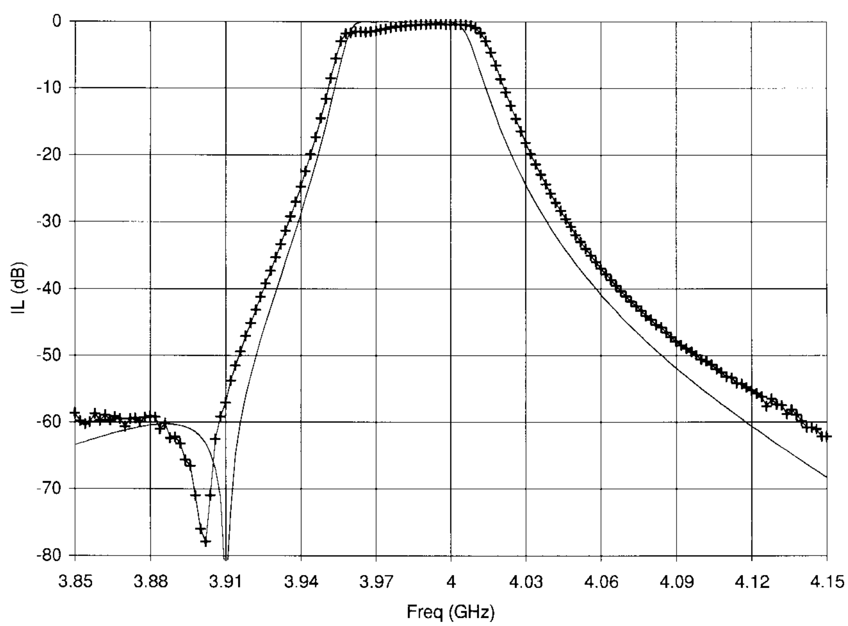


Fig. 9. Comparison between measured and predicted performance of the four-pole Chebyshev microstrip filter. The solid line is from predicted performance using modified cascaded circuit and the crossed line is from measured performance.

regular cascading circuit simulation. A comparison between the regular cascading circuit simulation and full EM simulation is given in Fig. 8. Besides significant differences for the filter center frequency and bandwidth, a transmission zero is shown from full EM simulation which is not possible from the regular cascading circuit simulation. The modified cascade circuit for this four-pole filter includes three additional signal paths using similar configuration to that shown in Fig. 5(a). In this case, signal paths between resonators one and three, between resonators two and four, and between resonators one and four, are included in the analysis. Fig. 9 shows the measured performance and the theoretical performance of the filter designed by applying the design technique presented in this paper. The theoretical results given in this figure were calculated using the modified cascaded circuit. Two full EM simulations were required to complete the design.

To illustrate the efficiency of the authors' proposed technique, cascading simulation of this four-pole filter needs about 1.5 min CPU time for one frequency point while the complete circuit EM simulation needs about 8.25 min CPU time on a HP700 workstation. For the purpose of optimization, the difference in the overall CPU time will be much more significant.

IV. Conclusion

The authors have presented an innovative CAD technique for microstrip filter design. The technique allows design optimization to be achieved with full-wave EM simulation accuracy while keeping the computational effort almost at the cascade design approach level. The authors believe that the concept is not limited only to microstrip filters and can be applied to other design areas. Theoretical and experimental results for two HTS microstrip filters have been presented. Excellent results have been obtained which demonstrates the validity of the proposed technique.

References

[1] J. W. Bandler, S. Ye, R. M. Biernacki, S. H. Chen, and D. G. Swanson, Jr., "Minimax microstrip filter design using direct EM field simulation," in *IEEE Int. Microwave Symp. Dig.*, Atlanta, GA, June 1993, pp. 889–892.

[2] R. R. Mansour, "Design of superconductive multiplexers using single-mode and dual-mode filters," *IEEE Trans. Microwave Theory Tech.*, vol. 42, pp. 1411–1418, July 1994.

[3] S. Ye and R. R. Mansour, "Design of manifold-coupled multiplexers using superconductive lumped element filters," in *IEEE Int. Microwave Symp. Dig.*, San Diego, CA, May 1994, pp. 191–194.

[4] J. W. Bandler, R. M. Biernacki, S. H. Chen, R. A. Grobelny, and R. H. Hemmers, "Space mapping technique for electromagnetic optimization," *IEEE Trans. Microwave Theory Tech.*, vol. 42, pp. 2536–2544, Dec. 1994.

[5] D. G. Swanson, Jr., R. Forse, and B. J. L. Nilsson, "A 10-GHz thin film lumped element high temperature superconductor filter," in *IEEE Int. Microwave Symp. Dig.*, Albuquerque, NM, June 1992, pp. 1191–1193.

[6] *Em User's Manual*, Sonnet Software, Liverpool, NY.

[7] *osa90 User's Manual*, Optimization Systems Assocs., Dundas, Ont., Canada.

Shen Ye (S'88–M'92) was born in Shanghai, China. He received the B.Eng. and M.Eng. degrees from Shanghai University of Technology, Shanghai, China, and the Ph.D. degree from McMaster University, Hamilton, Ont., Canada, all in electrical engineering, in 1982, 1984, and 1991, respectively.

From 1984 to 1986, he was an Assistant Lecturer in the Department of Electrical Engineering, Shanghai University of Technology. In 1991, he was awarded an Industrial Research Fellowship from the Natural Sciences and Engineering Research Council of Canada and joined Optimization Systems Associates Inc., Dundas, Ont, Canada. His major contributions include implementation of microstrip components for the CAD package *osa90/hope* and development of *Empipe* which connects an electromagnetic field solver to *osa90/hope*. In 1993, he joined Com Dev Ltd., Cambridge, Ont., Canada, where he is a Senior Member of Technical Staff. His current work includes in-house CAD program development and microwave circuit design, in particular, high-temperature superconductive thin film circuit design.

Raafat R. Mansour (S'84–M'86–SM'90) was born in Cairo, Egypt, on March 31, 1955. He received the B.Sc. (with honors) and M.Sc. degrees from Ain Shams University, Cairo, Egypt, and the Ph.D. degree from the University of Waterloo, Ont. Canada, all in electrical engineering, in 1977, 1981, and 1986, respectively.

He was a Research Fellow at the Laboratoire d'Electromagnetisme, Institut National Polytechnique, Grenoble, France in 1981. From 1983 to 1986, he was a Research and Teaching Assistant with the Department of Electrical Engineering, University of Waterloo. Since then, he has been with COM DEV, Ltd. Cambridge, Ont., Canada, where he is currently Principal Member of the technical staff in the Corporate Research and Development department. He also holds an appointment of Adjunct Associate Professor at the University of Waterloo. He has authored numerous publications in the area of electromagnetic modeling and high-temperature superconductivity, and holds several patents related to microwave filter design for satellite applications. His present research interests are in the analysis and design of microwave superconductive components and subsystems.