**DESIGN FEATURES**

# Synthesis + Simulation = Enhanced Design Process

*The combination of circuit synthesis and simulation functions within a common CAE design tool can greatly improve the RF/microwave-design process.*

**By Dale D. Henkes**
***President***

*Applied Computational Sciences, 1061 Dragt Pl., Escondido, CA 92029; (760) 612-6988*
*e-mail: LINC2@appliedmicrowave.com, Internet: www.appliedmicrowave.com*

**CIRCUITS can be created in several different ways with computer-aided-engineering (CAE) techniques. With a circuit-synthesis program, a designer starts by entering a list of specifications or design goals and pushing the "synthesize" button. Another approach might be to borrow a circuit topology from an existing design and then start a simulation program optimizing the circuit against a set of design goals. Perhaps the best approach, however, is to combine synthesis and simulation within the same CAE tool.**

The synthesis approach to circuit design is more direct, but it may or may not provide the user enough flexibility in influencing the outcome. However, a well-designed synthesis program will attempt to provide the user with as much control over the final design as possible (or practical). This may include accepting a wide variety of input from the user, a choice of circuit topology, or both. After the design has been synthesized, it can then be analyzed in a circuit-simulation program to verify performance according to the engineer's requirements. Optimization or tuning can be used at this stage to tweak the design. Therefore, a complete set of design tools would include synthesis and simulation.

The design-by-simulation/optimization method is often a trial-and-error approach with no guarantee of success. An impedance-matching problem, for example, may not be solvable with the circuit topology presented to the optimizer. Worse than that is the fact that, because it cannot change the circuit topology (only the component values), the optimizer might get close to meeting an impossible goal by adjusting some circuit component's value to a very large or extremely small value that would not be practical or obtainable.

The MicroLINC™ program from Applied Computational Sciences (Escondido, CA) combines RF and microwave-circuit analysis (simulation) with a set of synthesis tools. One of the most comprehensive circuit-synthesis tools in the package is the "Circles Utility." Primarily a small-signal amplifier or low-noise-amplifier (LNA) design tool, this module is aimed at synthesizing input and output matching networks for the active device. The active device is usually a transistor [bipolar or field-effect-transistor (FET)] that is characterized by a set of S-parameters.

The philosophy behind this utility is to give the user as much or as little input into the synthesis processes as desired. In keeping with that philosophy, a substantial amount of analyses and data are available to assist the designer (**Fig. 1**).
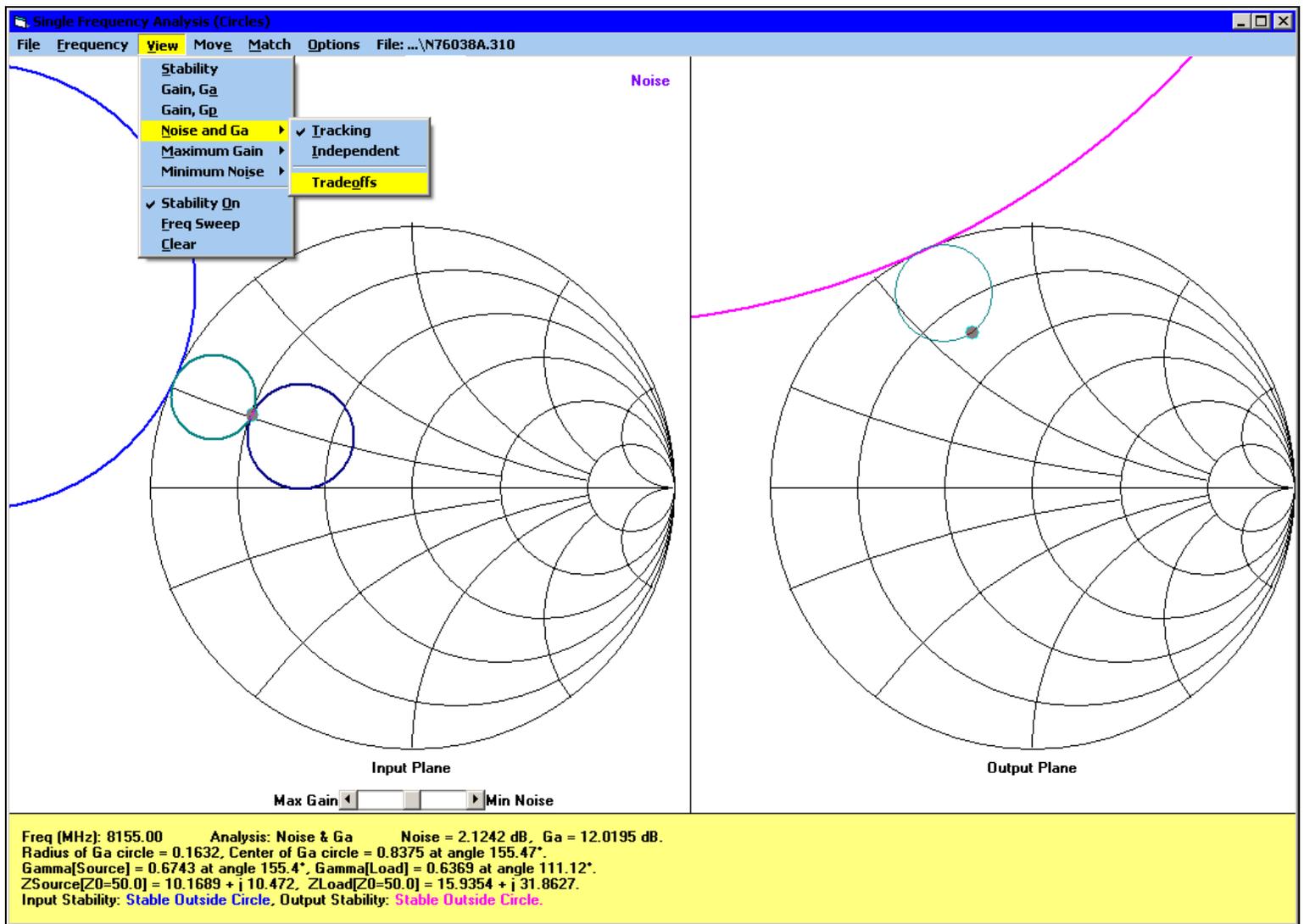
**Fig 1. The MicroLINC CAE program combines synthesis and simulation, providing users with a great deal of analysis power through pull-down menus.**

Thus, a designer experienced in the use of noise, gain, and stability circles can take advantage of the highly automated, yet interactive, nature of these tools as a guide to developing the best design that meets the requirements. On the other hand, a novice might simply input a gain specification and then select a default-matching network from the "Match" menu. The result is the creation of the entire circuit, seamlessly integrated into the simulator for immediate analysis and performance verification. All that remains to be done is to push the "Analyze Circuit" button for generating simulation results.

This is not to imply that a novice designer could not benefit from the circle utilities. Indeed, the interactive nature of the tool encourages experimentation. For example, a tool that proves as useful to the novice as it is to an experienced designer is the "Noise and Gain Trade-offs" tool shown in Fig. 1. When this is selected, gain and noise circles appear on the Smith Chart in the input plane. The circles automatically intersect at a common impedance point representing a typical compromise between maximum gain and minimum noise figure. A slider control below the Smith Chart points toward "Max Gain" on the left or "Min Noise" on the right.

Gain and noise-figure (NF) trade-offs can be viewed instantly by sliding the control toward Max Gain or Min Noise. As this is performed the gain and noise circles stay locked together, intersecting at the source reflection coefficient ($\Gamma_s$) as it moves along an optimum path between minimum noise and maximum gain. The gain and noise data are updated at the bottom of the screen so the user can determine when the appropriate trade-off has been made. As soon as a suitable trade-off between noise and gain is found, the program has all the data it needs to design the circuit. The next step is to select a set of matching networks from a list of various lumped and distributed topologies. Clicking on one of the items in the list automatically completes the circuit and generates the component values. It is possible to learn an effective design methodology simply by using this feature of MicroLINC.

**DESIGN EXAMPLE**

To further illustrate the design process using CAE techniques, the following design example is proposed:

1. The task involves the design of an amplifier with 12-dB gain at 6.365 GHz using an NEC NE76038 low-noise GaAs metal-semiconductor FET (MESFET) from NEC/California Eastern Laboratories (Santa Clara, CA) biased at a drain-source voltage of $V_{DS}$ = +3 VDC and a drain-source current of $I_{DS}$ = 10 mA.

2. The transistor must be stabilized within the operating band with both ports simultaneously matched [designed for maximum available gain (MAG) after stabilization].

3. The input port must be matched to an impedance of 75 $\Omega$ while the output port is matched to 50 $\Omega$.

4. The amplifier's NF must then be determined when the amplifier has been matched for maximum gain.

5. The amplifier circuit should be synthesized using distributed (transmission-line) matching networks.

6. Simulation should be used to verify the gain and impedance-matching conditions.

Determine the amplifier's NF when matched for maximum gain.

With the MicroLINC Circles Utility, these six apparently complex steps take only a few minutes to accomplish. First, the NE76038 transistor's S-parameter file is opened and 6365 MHz is selected from the "Frequency" menu. Selecting "Maximum Gain...MAG" from the view menu reveals that it is unavailable. Attempting to match both ports to obtain the MAG fails because the device by itself is potentially unstable at this frequency. This is quickly verified by selecting "View Stability" and noting that the stability circles cut into the upper left part of the Smith Chart. Additionally, the stability factor reported is less than 1 (k = 0.799).

The Circles Utility provides several ways to stabilize a potentially unstable device. The method used here will be to place a small amount of inductance in the common (source) lead of the transistor. (Reference 1 points out some of the advantages of using this method). Selecting "Common Lead Inductance" from the "Stabilize Device" menu (**Fig. 2**) and clicking the "Optimize" button automatically inserts the minimum amount of inductance needed to stabilize the device at the design frequency. If "Include Noise" is checked, then the noise parameters of the device will be adjusted to include the effects of the inductor.
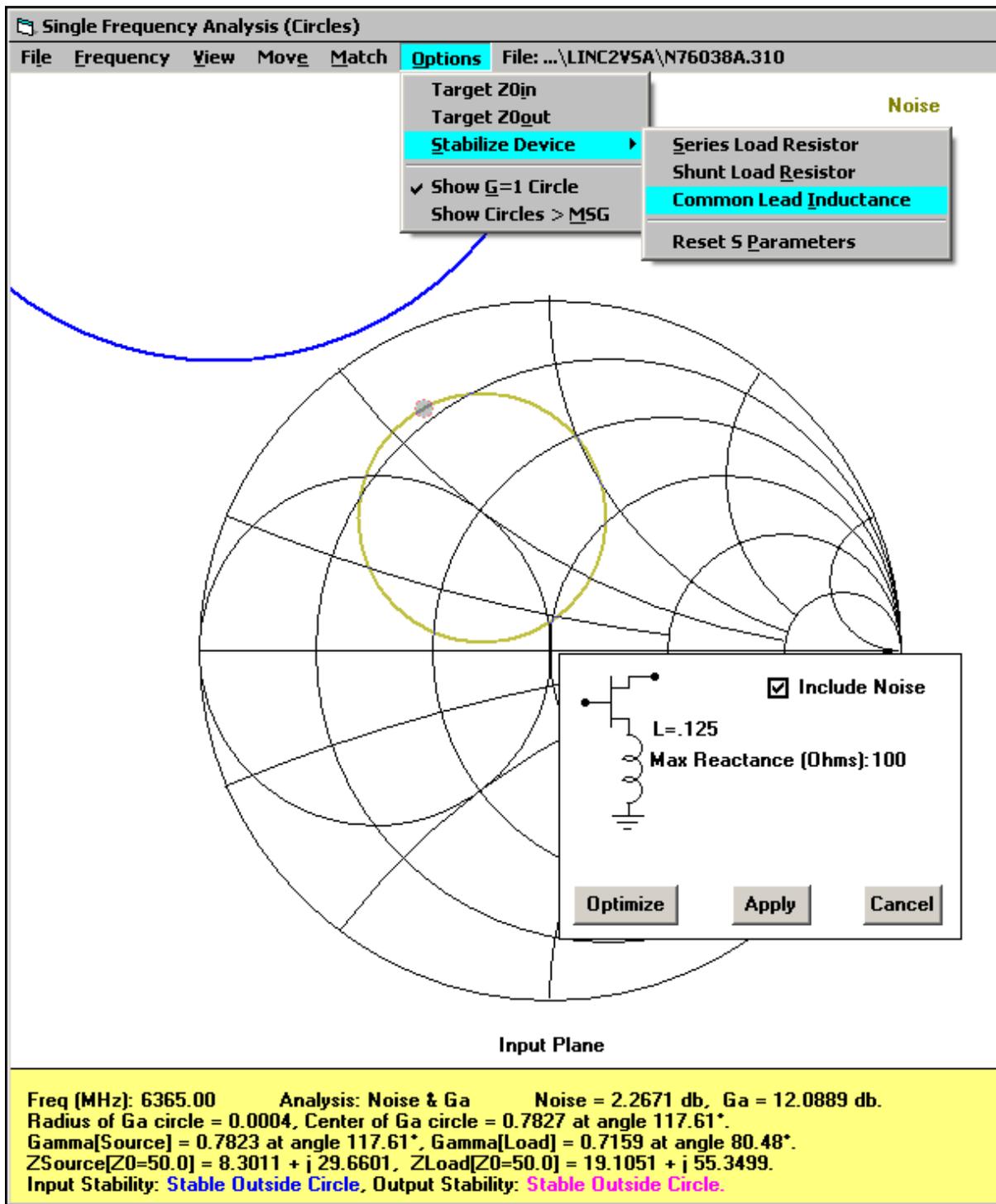
**Fig 2. Stabilizing a high-frequency transistor for use in a microwave amplifier is a simple matter when using the "Stabilize Device" menu selection.**

The next step is to change the input impedance from the 50-$\Omega$ default to 75$\Omega$ via the "Options... Target Z$_{0in}$" menu. Selecting "Maximum Gain... MAG" from the view menu now displays the optimum match points for maximum gain (~12.09 dB). The result is a single point (impedance match) in each of the input and output planes. This is referred to as a bilateral simultaneous conjugate match.[2] It is considered a bilateral match because the program solves the matching problem at both ports simultaneously, taking into account the fact that S$_{12}$ is a nonzero value (the output match affects the input match and vice versa).

The selection "View...Noise and Ga" displays the noise circle that intersects the MAG point in the input plane. The NF is reported as ~2.27 dB at a gain of 12.09 dB as shown in Fig. 2. The program now has all the information it needs to complete the circuit. Selecting "Transmission Line...Stub and TRL Cascade" from the "Match" menu produces the circuit shown in **Fig. 3**.
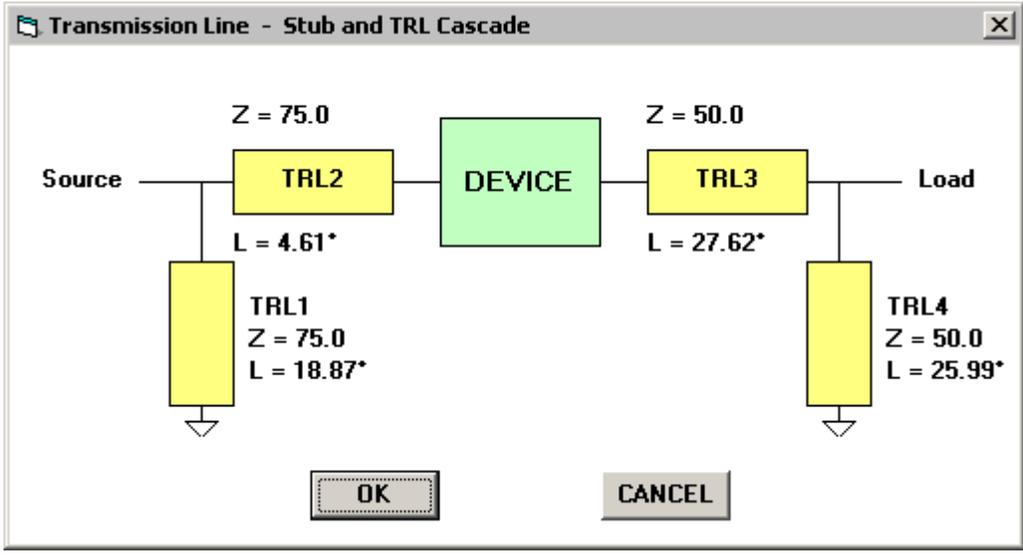


**Fig 3. This circuit contains the matching input and output networks needed to meet the 75- and 50-$\Omega$ input and output conditions, respectively.**

Then, the circuit file is automatically constructed and placed in the Text Editor (**Fig. 4**) for viewing and subsequent analysis.
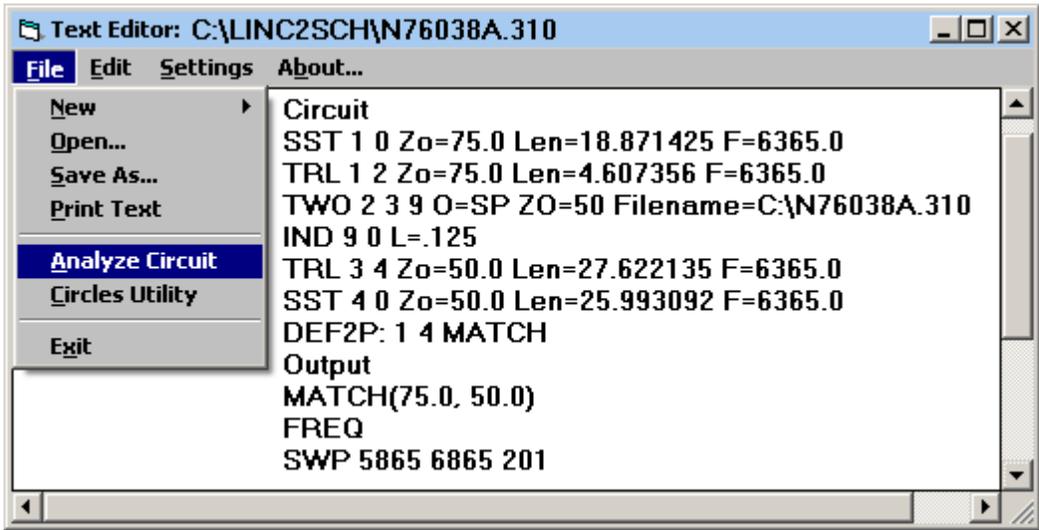


**Fig 4. This circuit file represents the schematic diagram of Fig. 3. It is automatically generated and placed in MicroLINC's Text Editor for viewing and analysis.**

## SIMPLE SIMULATION

"Analyze Circuit" starts the simulation, producing the results shown in Figs. 5 and 6.
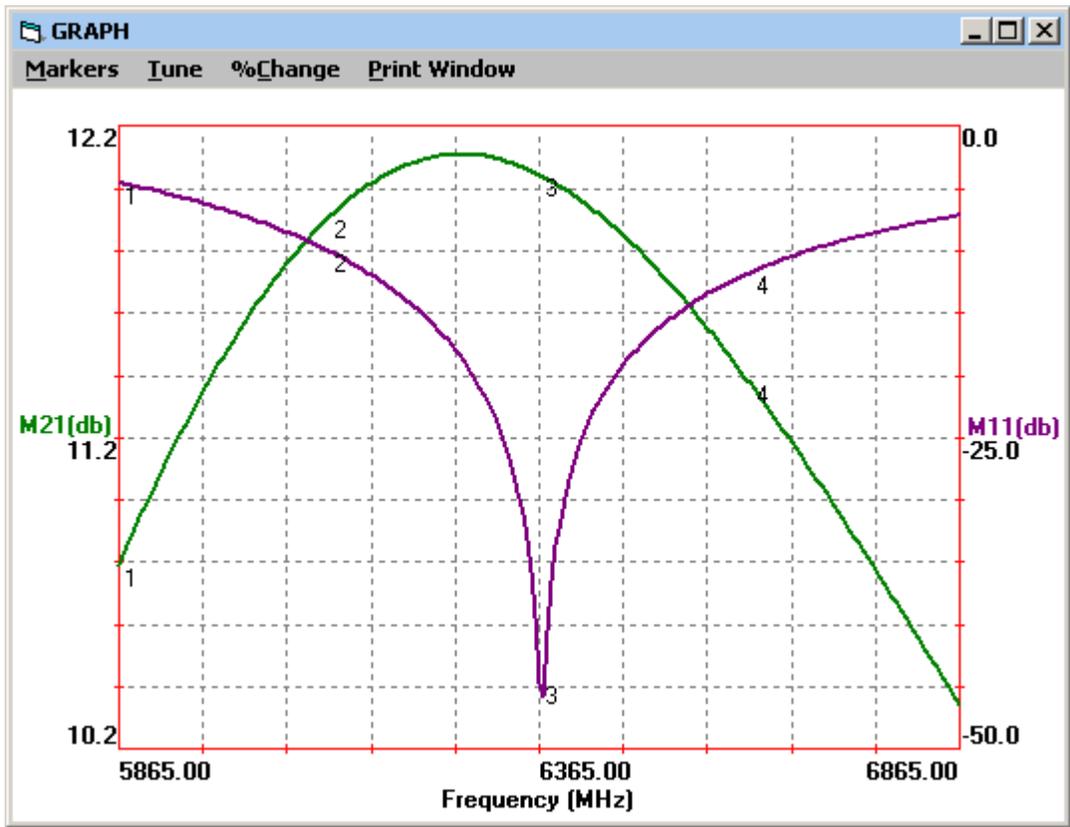
**Fig 5. The simulated performance of the example amplifier is shown here for gain (M$_{21}$) and return-loss (M$_{11}$) characteristics centered at the design frequency.**

**Figure 5** indicates that the gain specification has been met, while **Fig. 6** shows the quality of the input and output match.
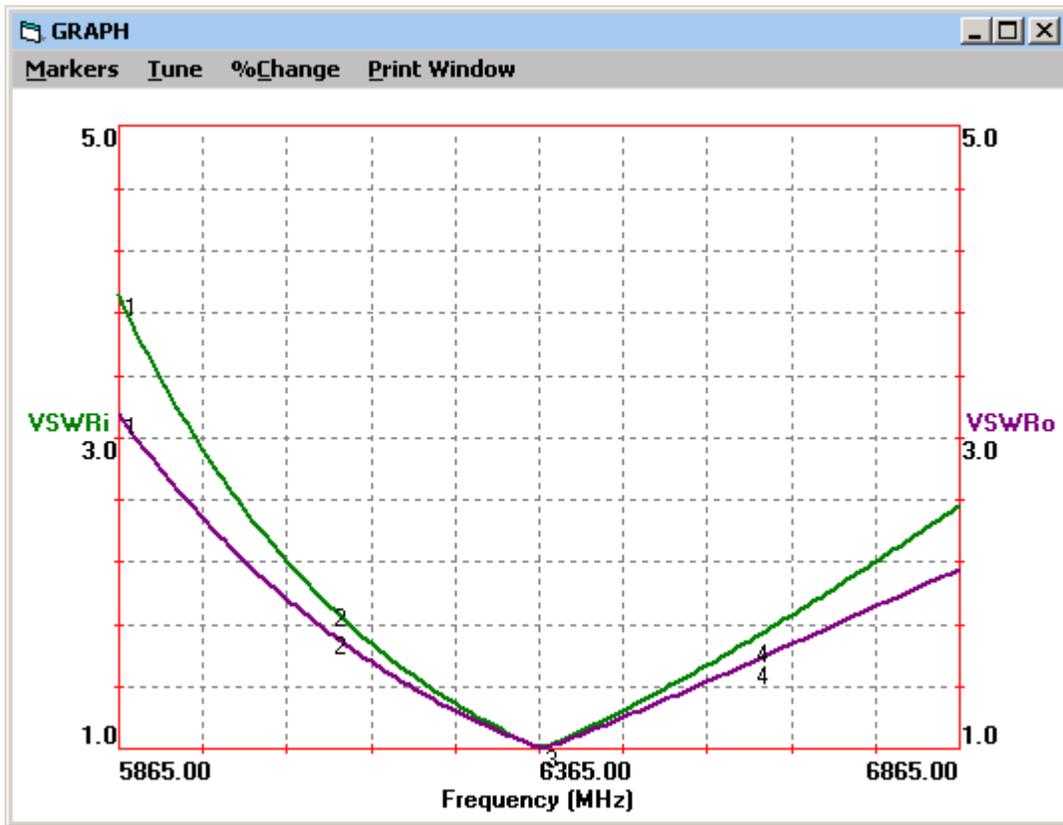
**Fig 6. The plot above shows the quality of the input as well as output impedance-matching networks for the example amplifier.**

The marker at 6365 MHz indicates that the input has been matched to 75$\Omega$ with a VSWR of 1.01:1 and the output is matched to 50$\Omega$ with approximately the same VSWR. An excellent match at both ports was expected (and obtained) because that is what was requested of the synthesizer when it was invoked to perform a MAG design.

**Figure 7** shows that the additional goal of in-band transistor stabilization has been met.
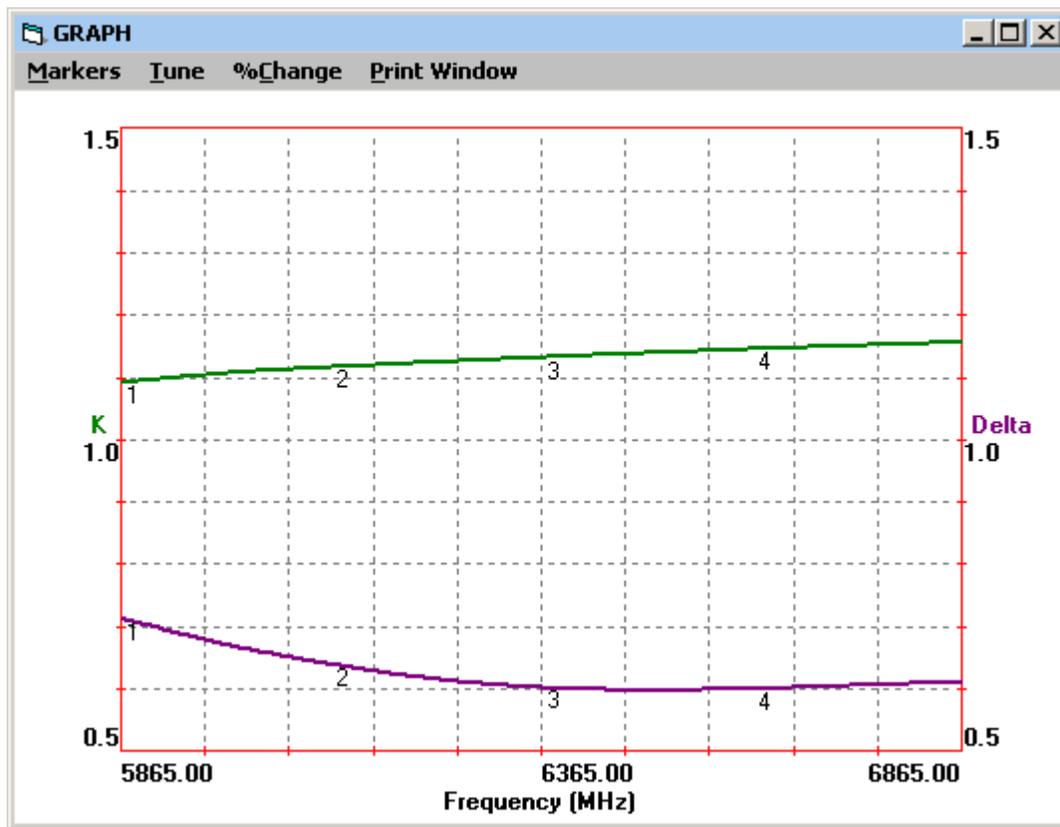
**Fig 7. This plot illustrates the effectiveness of matching the transistor for high gain while maintaining good in-band operating stability.**

As indicated, the Rollett Stability factor (k) is greater than 1 while the absolute value of the determinant of the S-parameter matrix, $|\Delta|$, is substantially less than 1 over the 1-GHz band around the operating frequency. This ensures that the transistor is unconditionally stable at the operating frequency. Of course, additional steps will need to be taken to ensure that the amplifier will be stable at all frequencies. Components can be built into the DC bias feeds in such a way as to enhance stability above and below the operating frequency.

This completes the initial RF portion of the design. The MicroLINC "Transmission Lines" tool can be used to generate the physical dimensions for printed-circuit-board (PCB) microstrip or stripline designs from the electrical parameters shown in Fig. 3.

In closing, it should be noted that simulation provides much more than just verification of the synthesized design. Simulation provides more performance analysis (and insight into circuit behavior) than could be known at the time of circuit synthesis. For example, the return loss ($M_{11}$) in Fig. 5 gives an indication of the narrowband nature of the input match. Additionally, Monte Carlo analysis (from the MicroLINC statistics module) can indicate the amplifier's sensitivity to component tolerances. However, design by simulation alone is a trial-and-error approach. Therefore, when selecting an RF CAE program, it is a good idea to consider a package that provides synthesis and analysis capabilities.

In comparison, a design engineer can spend a great deal of time (and cost) on a combination of separate programs, such as impedance-matching tools, linear circuit simulators, and synthesis software, hoping that the tools mesh and can seamlessly share files. A more practical approach is to select a single tool with multiple functions, so that design files are not lost or outdated when an independent software tool is upgraded.

### References

1. Dale D. Henkes, "LNA Design Uses Series Feedback to Achieve Simultaneous Low Input VSWR and Low Noise," *Applied Microwave & Wireless,* October 1998, p. 26.

2. Guillermo Gonzalez, Chap. 3, Sec. 3.6, "Simultaneous Conjugate Match--Bilateral Case," in *Microwave Transistor Amplifiers,* Prentice-Hall, Englewood Cliffs, NJ, 1984.