APPLICATION OF STANDARD CIRCUIT SIMULATORS FOR OPTIMAL MATCHING OF ELECTRONIC CIRCUITS

Assoc. Prof. Dr. Elissaveta Dimitrova Gadjeva
Eng. Borianka Ivanova Mihova
Assoc. Prof. Dr. Lila Andreeva Donevska

Technical University of Sofia, Department of Electronics,
e-mail: egadjeva@vmei.acad.bg

The contemporary development of electronic and communication technology requires the solution of the problems for optimal matching of the signal source to the load. In the present paper, the possibilities of the standard PSpice- like circuit simulators are extended for solving the optimal matching problem.

In the design process of amplifier circuits at RF, the conditions of optimal matching of the signal source to the input stage can be satisfied by simultaneously parameter variation of two inductance elements. Since the *PSpice* simulator allows parametrization of a single element, this variation of two elements is not possible using the standard parametric analysis.

In the present paper, two approaches for optimal matching of electronic circuits are realized using the possibilities of the standard circuit simulator *OrCAD PSpice*. The first approach is based on the independent variation of two parameters in the frequency domain using standard parametric analysis. A nulor model is constructed for including of the optimal matching conditions in the second approach. An approach for optimal matching with respect to the noise is also proposed. The minimal noise factor is obtained corresponding to the optimal matching of the signal source to the input circuit stage is realized with respect to the noise.

I. COMPUTER–AIDED APPROACH FOR OPTIMAL MATCHING BY INDEPENDENT VARIATION OF TWO PARAMETERS

The circuit shown in Fig.1 is used for illustrating the optimal matching approach. The matching elements are the inductances L_G and L_S [1]. The input impedance of the circuit has a form [1]:

$$Z_{in} = \underbrace{\frac{g_m . L_S}{C_{GS}}}_{R_{in}} + j\underbrace{\left[\omega(L_G + L_S) - \frac{1}{\omega C_{GS}}\right]}_{X_{in}}, \qquad (1)$$

where g_m is the transistor transconductance and C_{GS} is the gate-to-source capacitance.

The values of the matching elements L_G and L_S are obtained using the equation (1) and the optimal matching conditions:

$$\mathrm{Re}(Z_{IN})=R_{GOPT} \text{ and } \mathrm{Img}(Z_{IN})=0 \qquad (2)$$

In order to accomplish simultaneous variation of the L_G and L_S elements, the paramet-

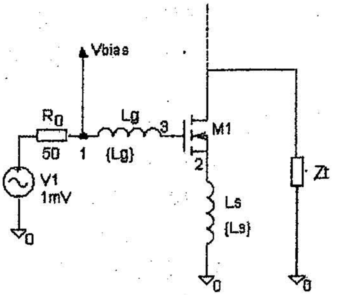


Fig.1 *Example circuit*

ric analysis of the *PSpice* simulator is extended with a possibility for variation of two independent variables val_1 and val_2.

A single parametric *par* is defined as *PSpice* variable parameter by a linear variation from 1 to $n_1 * n_2$, where $n_1(n_2)$ is the number of the step points of the $val_1(val_2)$ parameter.

The current values of val_1 and val_2 are:

$$val_1 = \left(p_{1\min} + \frac{p_{1\max} - p_{1\min}}{n_1 - 1} par_1 \right) \tag{3}$$

$$val_2 = \left(p_{2\min} + \frac{p_{2\max} - p_{2\min}}{n_2 - 1} par_2 \right) \tag{4}$$

where $p_{1min}(p_{2min})$ is the minimal value of $val_1(val_2)$ and $p_{1max}(p_{2max})$ is the corresponding maximal value.

The par_2 value is obtained by the expression:

$$par_2 = \left(\frac{par - p_{1a}}{n_2} \right) \tag{5}$$

$$p_{1a} = \mathrm{mod}(par, n_2) \tag{6}$$

where p_{1a} is the rest of the division $\left(\frac{par}{n_2}\right)$.

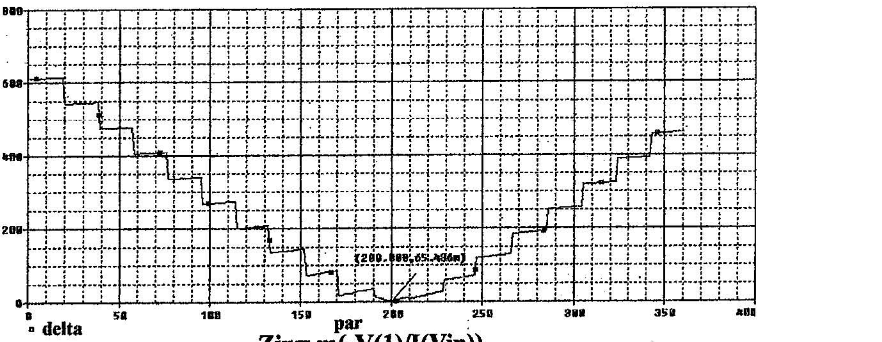
The expression [2]:

$$\mathrm{mod}(a,b) = \frac{b}{\pi}\left(a\tan\left(\tan\left(\frac{a}{b}\pi - \frac{\pi}{2} \right) \right) + \frac{\pi}{2} \right) \tag{7}$$

```
.PARAM
+    p1a={n2*(atan(tan(((par)/(n2))*pi-pi/2))+pi/2)/pi}
+    par1={(p1a-(atan(tan(((p1a)/1)*pi-pi/2))+pi/2)/pi)}
+     par2={(par-p1a)/n2}
+    val1={p1min+(p1max-p1min)/(n1-1)*par1}
+    val2={p2min+(p2max-p2min)/(n2-1)*par2}

.PARAM  Ls={val1}, Lg={val2}
```

Fig. 2. *PSpice description of two independent parameters val_1 and val_2*



Zin= m(-V(1)/I(Vin))

delta=abs(max(Zin)-50)

Fig. 3. *Simulation result for Δ*

is used to obtain the rest using *PSpice*.

The par_1 parameter is obtained by the expression:

$$par_1 = p_{1a} - \text{mod}(p_{1a}, 1) \quad (8)$$

The approach is developed according to the possibilities of the standard circuit simulator *OrCAD PSpice*. The L_G and L_S parameters are defined using the standard *Spice* statement **PARAM** as shown in Fig.2.

The module of the input impedance $|Z_{IN}|$ is used for the computer realization of this optimal matching approach. The difference $\Delta = \left|R_0 - |Z_{IN}|\right|$ is investigated with respect to the values of parameter *par*. The optimal matching conditon is satisfied when Δ has a minimal value.

The respective parameter values L_G and L_S are determined, corresponding to the matched value of Z_{IN}, satisfying condition (2). The simulation result for Δ of the example circuit is shown in Fig.3. The values of val_1 and val_2 parameters corresponding to L_S and L_G respectively are shown in Fig.4. The optimal value of L_S inductance is 1nH and the corresponding optimal value of L_G inductance is 4.1111uH.

II. COMPUTER–AIDED APPROACH FOR OPTIMAL MATCHING USING NULOR MODEL

This optimal matching approach is based on the nulor model, given in Fig.5a [4]. The nulor model is realized in the standard circuit simulators, like *OrCAD PSpice* using the library element- "voltage controlled current source" with a large value of **GAIN** (Fig.5b).

The approach is illustrated using the example circuit, given in Fig.1 by including of the nulor model as shown in Fig.6. The corresponding PSpice realization is given in Fig.7. The required matched value $Z_{IN}=Z_{IN\_OPT}$ is defined by the current source $I_1 = \dfrac{V_{IN}}{Z_{IN\_OPT}}$.

The nullator is connected in parallel to the I_1 source in order to ensure a voltage drop of 0V across the I_1 source. The inductance L_G is defined as a single variable parameter. The norator is connected in parallel to the L_S element. The L_S value corresponding to a

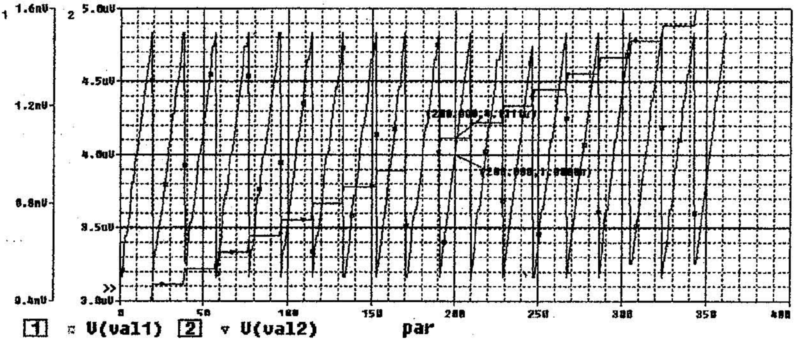


Fig. 4. *The values of val_1 and val_2 parameters corresponding to L_S and L_G respectively*

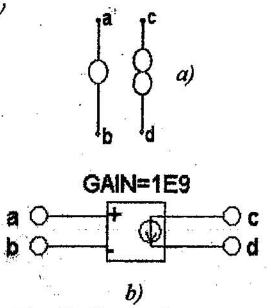


Fig. 5. *Equivalent circuit of the nulor model*

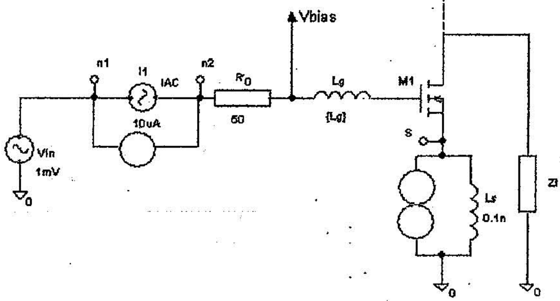


Fig. 6. *Example circuit of Fig. 1 with nulor model*

matched circuit is specified by the investigation of impedance **Zs**, determined as macros: **Zs = max(V(s)/(I(s)*2*pi*freq))**, where I_S is the source current and *freq* is the given frequency.

The optimal value L_{GOPT} corresponds to the real part of Z_S is equal to zero (Z_S is an inductance element– Fig.8a). The corresponding value of L_S is specified using macros:

Ls=max(-img(V(s)/(I(s)*2*pi*freq)).

The optimal value of the inductance L_{SOPT} corresponds to the optimal value of the

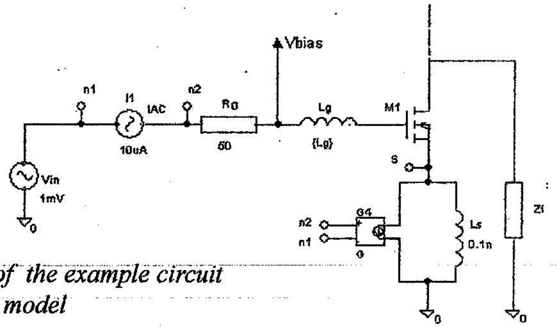
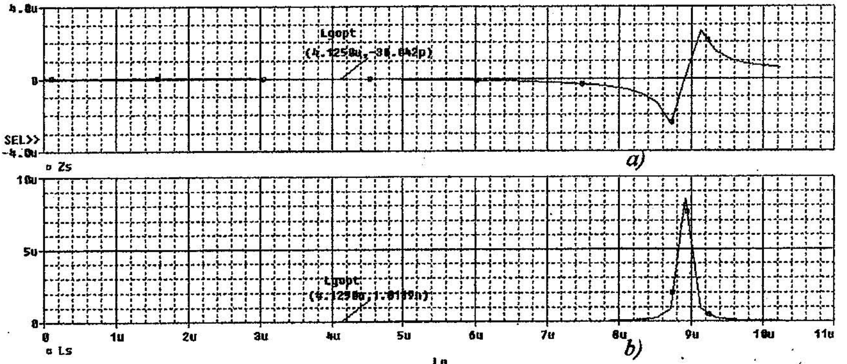


Fig. 7. *PSpice realization of the example circuit using nulor model*



Zs = max(V(s)/(i(Rs)*2*pi*1e8))
Ls=max(-img(V(s)/(I(s)*2*pi*freq))
Fig. 8. *The values of matching inductances L_S and L_G*

inductance L_G- L_{GOPT} (Fig.8b). The obtained L_{SOPT} is 1.0119nH and L_{GOPT} is 4.125uH.

III. AN APPROACH FOR OPTIMAL MATCHING WITH RESPECT TO THE NOISE

The optimal matching with respect to the noise is realized, if the matching noise conditions $R_G=R_{EN}$ and $\mathrm{Img}(Z_G)=-\mathrm{Img}(Z_{IN})$ is satisfied, where R_G is the resistor of the signal source and R_{EN} is the equivalent noise resistor of the circuit.

The electronic circuits are represented as two-port elements when the noise analysis is activated. They are characterized by the noise factor in the form [3]:

$$F = \frac{INOISE^2}{4kT_0R_G} \tag{9}$$

where k is the Boltzmann's constant $1{,}38.10^{-23}$ (W.sec/°K); T_0 is the temperature 300K; *INOISE* is the total noise at the designed output, referred to the input source; R_G is the signal source resistor.

The equivalent noise resistor of the circuit R_{EN} is specified by investigation of its frequency dependence using the possibilities of the graphical analyser *Probe* to define of macros: **Roptu = V(inoise)*V(inoise)/(4*1.38e-23*300)**. The value of **Roptu** that corresponds to the optimal matching for selected frequency is obtained. The resistor with the same value must be included in the input circuit– Fig.9. The imaginary part of the equivalent impedance, determined by macros:

Loptu=max(1/(2*pi*frequency*img(I(Lg)/V(b0,0))),

where b0 и 0 are the incedent of the L_G element.

The frequency dependence of the noise factor is shown in Fig.10. The minimal noise figure is obtained for the value R_G=50Ω. This value is a sum of the real parts of the source impedance and the equivalent noise resistance of the circuit.

IV. CONCLUSION

An approach for optimal matching has been created in this work using independent

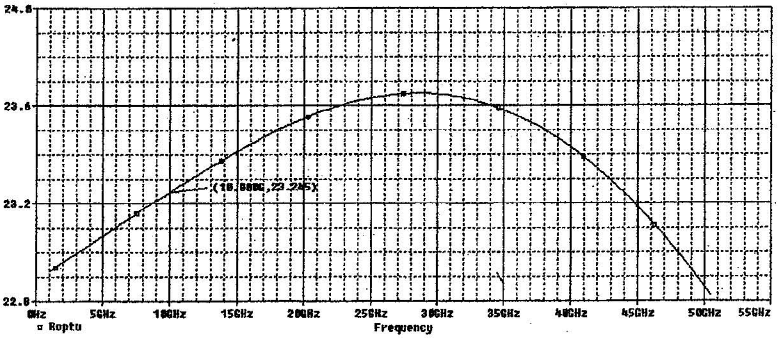
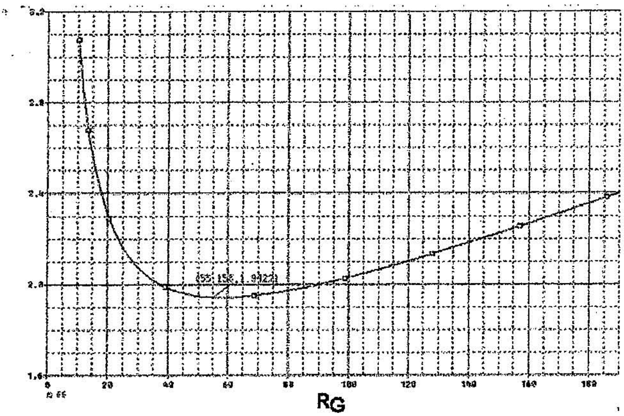


Fig. 9. *Frequency dependence of the optimal source resistance*



FF = max(V(inoise)*V(inoise)/1.656e-20*Roptu)

Fig. 10. *Minimal noise factor*

variation of two parameters.

A *PSpice* nullator-norator model has been developed for introducing the matching conditions.

An approach has been realized for a minimal noise factor determination by optimal matching of the signal source to the input circuit with respect to the noise using the possibilities of the *PSpice* noise analysis.

VII. REFERENCES:

[1] Rafla R., M. El-Gamal, "Design of a 1,5V CMOS Integrated 3GHz LNA", Microelectronics and Computer Systems Laboratory, McGill University, Canada
[2] OrCAD PSpice and Basics. Circuit Analysis Software. OrCAD Inc.,USA, 1998
[3] OrCAD PSpice A/D, Circuit Analysis Software, User's Guide, OrCAD Inc., 1998.
[4] Reed M., S.Seshu, "Linear graphs and electrical circuits", in Russian, Moskva, 1971

Reviewer: Assoc. Prof. Dr. Emanuil Djerassi, BAS-Sofia