Designing Bipolar Transistor Amplifiers Using Spice

Kenji Matsumoto[1] Masayoshi Oda[1] Yoshihiro Yamagami[1]
Yoshifumi Nishio[1] Junji Kawata[2] Akio Ushida[2]
(Tokushima University[1] Tokushima Bunri University[2])

1. Introduction

RF power amplifiers are sometimes composed of bipolar transistor circuits [1]. For attaining the maximum gain, we need to set the suitable bias voltages, and choose appropriate load impedances that can be practiced by setting the resistance values. Usually, the optimization can be done by trial and error process using Spice simulator. It is really time-consuming when the design parameters are so many. In this paper, we propose a Spice-oriented optimization algorithm based on the steepest descent algorithm [2-3] whose circuit is realized by ABMs (analog behavior Models) of Spice, and the equilibrium point corresponds to the optimum solution. Firstly, we need to define the *objective function* as follow:

$$\Phi(\mathbf{x},\mathbf{p}), \quad \mathbf{x}\in R^n, \quad \mathbf{p}\in R^k \tag{1}$$

where
x: circuit variables such as voltages and currents.
p: optimization parameters such as bias voltages, resistor's values and so on. The gradient direction is decided by the solutions of sensitivity circuits [3], and the steepest descent algorithm can be realized by the equivalent RC circuits combining with nonlinear controlled current sources.

2. Transistor's sensitivity modules

2.1 Sensitivity analysis: The steepest descent method is the most basic optimization approach, where the gradient direction is decided by the solutions of sensitivity circuit. Using the *Tableau approach*, we have the following Tableau equation to calculate the sensitivities:

$$\begin{bmatrix} \mathbf{K}_i & \mathbf{K}_v & \mathbf{0} \\ \mathbf{0} & \mathbf{1} & -\mathbf{A}^T \\ \mathbf{A} & \mathbf{0} & \mathbf{0} \end{bmatrix} \begin{bmatrix} \mathbf{S}_{i,p_i} \\ \mathbf{S}_{v,p_i} \\ \mathbf{S}_{v_n,p_i} \end{bmatrix} - \begin{bmatrix} \left.\frac{\partial \mathbf{g}(\mathbf{v},\mathbf{i})}{\partial \mathbf{v}}\right|_{\mathbf{v}_0,\mathbf{i}_0} \mathbf{S}_{v,p_i} \\ \mathbf{0} \\ \mathbf{0} \end{bmatrix}$$

$$- \begin{bmatrix} \left.\frac{\partial \mathbf{g}(\mathbf{v},\mathbf{i})}{\partial \mathbf{i}}\right|_{\mathbf{v}_0,\mathbf{i}_0} \mathbf{S}_{i,p_i} \\ \delta(i) \\ \mathbf{A}\delta(i) \end{bmatrix} = \begin{bmatrix} \left.\frac{\partial \mathbf{g}(\mathbf{v},\mathbf{i})}{\partial \mathbf{p}}\right|_{\mathbf{v}_0,\mathbf{i}_0} \delta(i) \\ \mathbf{0} \\ \mathbf{0} \end{bmatrix} \tag{2}$$

where $\delta(i)$ means a delta function satisfying

$$\delta(i) = \begin{matrix} 1 & 2 & \dots & i & \dots & k \end{matrix} \\ [0, \quad 0, \quad \dots, \quad 1 \quad \dots, \quad 0]^T$$

Thus, the sensitivity circuit configuration is equal to the original one except for the nonlinear elements being replaced by the linear incremental resistors at the operating points $\mathbf{V}_0, \mathbf{I}_0$.

2.2 Sensitivity modules of bipolar transistor: For simplicity, we consider NPN transistor modeled by Ebers-Moll model as shown in Fig. 3(a), where

$$i_d = I_s(\exp(v_d/v_T)-1), \quad \text{for } v_T = 0.026, \quad \alpha = 0.99 \tag{4}$$

The sensitivity module is shown by Fig. 3(c), where the incremental resistor is given by

$$\frac{1}{r_e} = \left.\frac{\partial i_d}{\partial v_d}\right|_{v_d=v_{d0}} \tag{5}$$

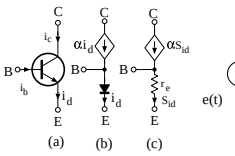

Fig. 3 (a) NPN Transistor, (b) Ebers-Moll Model, (c) The Sensitivity module.

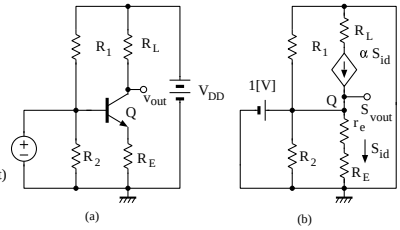

Fig. 4 (a) Common-emitter amp., (b) The sensitivity circuit for calculating the gain.

3. Steepest descent method attaining maximum gain

Now, we consider Spice-oriented optimization technique for attaining the maximum gain of Fig. 4, where resistors R_1, R_2, R_E should be optimized for setting the suitable bias voltage. We need to introduce the numerical differentiation technique as shown in Fig. 5 for the optimization of the gain, because the gain is obtained by the sensitivity circuit shown in Fig. 4(b). The output sensitivities Ss are obtained by Fig. 4(b), and the output p_i controls the resistive value using ABM.

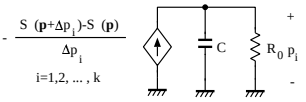

Fig. 5 Spice-oriented optimizing method.

4. Simulation result

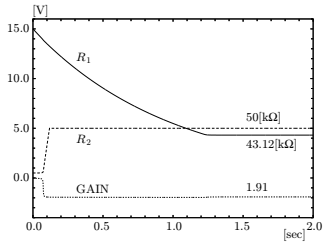

Fig. 6 Simulation result for optimization.

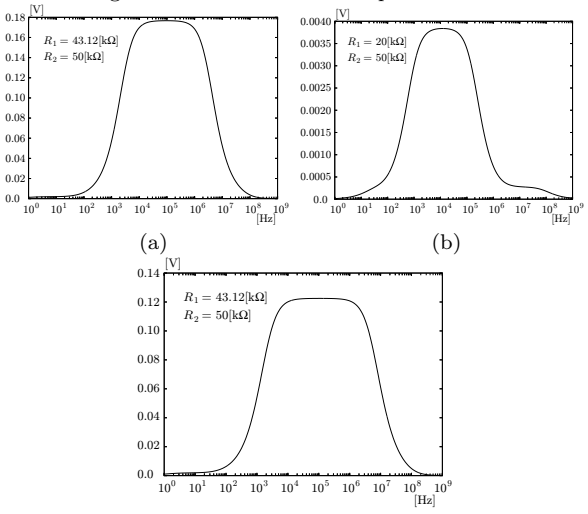

Fig. 7 Simulation result for Vin=0.001[V] (a) optimal resistors (R1=43.12[kΩ], R2=50[kΩ]), (b) non-optimal resistors (R1=20 [kΩ], R2=50[kΩ]), (c) non-optimal resistors (R1=43.12 [kΩ], R2=20[kΩ]).

We optimized R1 and R2 in the circuit of Fig. 4(b). And, the optimal solutions are found as Fig. 6. Time for optimization was 7.72[s]. When the value of R1 and R2 was substituted and the AC analysis was done, the gain became 177. The gain will decrease if either value of R1 or R2 is changed as Fig. 7(b)(c).

5. Conclusion and remarks

In this paper, we obtained the optimal result. Thus, we were able to optimize an amplifier by using steepest descent method on Spice. As a problem in the future, we will try to optimize the multistage amplifier on Spice as the next stage.