A New Spice Model Based on the Theoretical Application of Operational Transconductance Amplifier in Didactic Situation Optimization

Mohammad Reza Modabbernia[1], Yekta Nazarpour[2], Seyedeh Shiva Nejati[3], and Syrus Rouhollahi Moghadam[4]

[1]Electrical Engineering Group, Technical and Vocational University, Rasht branch, Rasht, IRAN
[2]Computer Engineering Group, Azad Islamic University, Lahijan Branch, Lahijan, IRAN,
[3]Electrical Engineering Group, Sardar Jangal University, Rasht, IRAN,
[4]Humanity Science Group, Technical and Vocaational University, Rasht branch, Rasht, IRAN,
m_modabbernia@afr.ac.ir, nazarpour_1ta@yahoo.com, s.nejati@sjihe.ac.ir, Moghadam.steve34@gmail.com

Abstract: In this paper, the application of operational transconductance amplifier (OTA), in addition to comparing with the interior structures of the most important available integrated circuits, have been analyzed. In this regard, a model in Spice has been presented to facilitate the trend of education for the undergraduate based on OTA theoretical analysis. The present paper aims to offer a model for the OTA in Spice from which the theoretical features of the said amplifiers can be incorporated. This causes the application of the OTA integrated circuits, under the impression and contribution of computerized simulation, and the accuracy of the theoretical analyses to be instructed and to be studied respectively. At the end, to show the merit and competence of the proposed model, three practical circuits, whose results are compared to the theoretical values and simulation accuracy, are represented. The proposed model covers the high speed of simulation and appropriate numerical convergence.

Keywords: Operational transconductance amplifier; OTA; PSpice; Spice netlist; Simulation

1. Introduction

Education forms the essence and pivotal axis of Industrial, economical and social developments of the societies. Over the past few decades, the researchers, with respect to computerized advancements and internet formation, have represented a lot of new and novel solutions and methods to develop the resources, techniques and educational instruments [1]. The simulation application on Spice has well found its direction, amid electronics-bases sources, promoting and improving the issue of education and also basic concepts elaboration. In this respect, the authors dramatically utilize components modelling and integrated circuits in simulations [2-6]. Selecting a model among diverse types of the represented models for an offered component or IC depends upon the simulation aims. If the aim is to predict the circuit behavior before manufacturing, the elements models should be highly exact. This can be led into more intricacy of the model and makes the simulation more complicated. However, on the condition that the only target is the study of the component behavior performance, view point of a know-how system or application principles, the offered model should be designed simply to accelerate and hasten the trend of simulation making the simulation results comparable to those of the theoretical ones [7-8].

An operational transconductance amplifier (OTA) is basically a voltage-dependent current source whose gain can be adjusted by the control current. Unlikely to operational amplifiers (Opamp), OTA simulation and modelling receives trivial heed and only its application and practical circuits are considered important [5-6].The nonlinear model of OTA can be found in [9-11]. In [12], there is a sub-circuit bipolar transistor for OTA based on the model 17. An OTA linear model for filter simulation and a phase structure model of OTA have been

Received: February 6th, 2014. Accepted: September 27th, 2014

successively studied in [13] and [14]. As well, the IC manufactures, in some extent, have represented a sub-circuit in spice for their own OTAs [15].

In other words, the IC manufacturing companies have offered precise and exact various macro models for better exclusive performance of the operational amplifiers (Opamp) [16-18]. However, in electronic references, a simple model of Opamp has been continuously utilized for the circuits' simulation introduced [4-6]. This satisfies the issue related to the thyristor macro model. The thyristor model is intricate in PSpice and ICAP software [19] for the simulation of power electronic circuits; however, the simple model of thyristor is used for education in [20-22]. This reflects that the application of the complicated models of the components and ICs are not suitable for the under graduated students involved in electronic basic concepts, but it is needed to choose a model which can make a proper compatibility between the theoretical and practical concepts. In case the effects of noise and frequency and also the practical constrains of OTA are foregone, the following items can be considered for the propitious performance of the macro model in electronics undergraduate students education.

1. Preparing a proportionate balance between simplicity and exactness by the model
2. Enjoying a proper convergence while simulating
3. Showing the input and output impedance of the amplifier
4. Enjoying the non-linear relation between the amplifier input and output
5. Having a suitable comparative proportion with the internal circuits of the manufacturing companies

The following pattern has been carried out to instruct OTA application, modeling and its utilization in this paper. At first, the general structure of OTA and its basic block diagram are presented. Forgoing the case study of each element, it is believed that via a basic and comprehensive framework a tremendous number of circuits and identical elements can be demonstrated in the course of education. This can be led into the elevation and declination of the students understanding and fatigue respectively.

Accordingly, the OTA macro model is going to be introduced step by step based on the general structure and block diagram of the OTA. This step by step modelling enjoys a didactic significance from two aspects. First of all, it makes the students implicitly familiarized to spice modelling and its instructions. Secondly, the performance of the OTA configuring layers can be expressed in this regard. Finally under the assistance of the represented OTA spice macro model, the three applicable circuits given by the manufacturing companies [23-25] can be simulated to prepare the ground for a comparison between the simulation result and those stemmed from the circuits' analysis in ICs data sheets. This is accomplished to reflect the model competence and merit.

2. General Structure of an Operational Transconductance Amplifier (OTA)

Figure 1 shows the block diagram of an operational transconductance amplifier (OTA). Input differential state like all other operational amplifiers should differentiate input signals and make an equivalent current signal in its collector's outputs. Current source provides the required current for biasing of the differential state. This current is generated with respect to external controller factor; therefore, the designer can control it. This issue causes high flexibility in IC suitable for the designer.

Loads of 1, 2 and 3 are current sources which act as active loads. Supposing that the output currents of these sources are equal to that i_{c1} and i_{c2} can be found in the output active loads of 2 and 3 respectively. This causes the OTA output current to be equal to $i_o = i_{c2} - i_{c1}$.

If in the simplest shape ,active loads and current source are considered as a simple mirror current source, general structure of Figure 1 can be considered equal to Figure 2 ,that is a very basic circuit of an OTA(4).

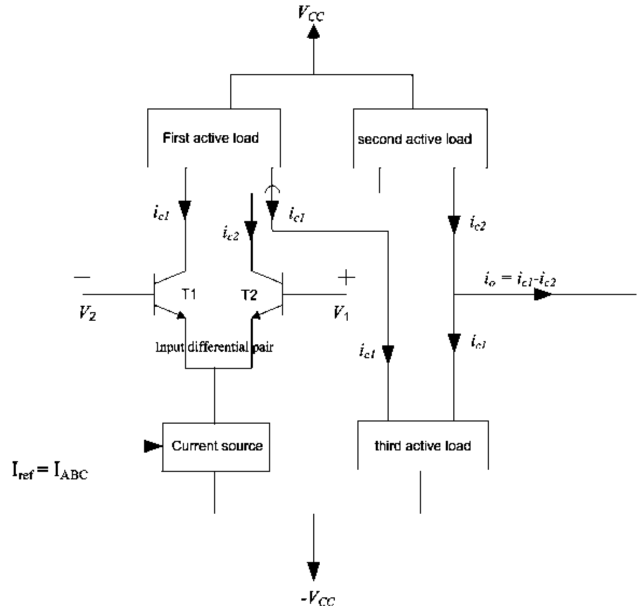

Figure 1. The block diagram of an OTA

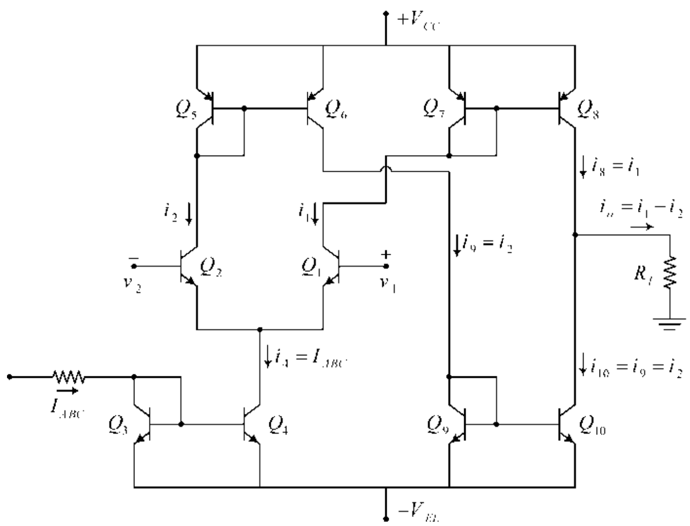

Figure 2. Basic circuit of an OTA

Because, in this paper, all the features of OTAs follow this pattern, and only the used current source type is different in them; therefore, circuit in Figure 2 is studied by detail and the relevant mathematic equations of input and output are obtained. These equations can be easily extended to other internal circuits because other current sources have a better performance than simple current source. By assuming an equal output current and reference current accompanied by big output impedance, it can have more accuracy.

If it is assumed all Figure 2 related transistors are the same and have big β coefficient, then it can be said that output and reference mirror source currents are approximately equal (this equality is more obvious for Wilson current source used in OTA). Q_3 and Q_4 transistors consist of the current source. Q_5 and Q_6 transistors are considered as active load 1. Q_7 and Q_8 are taken into account as active load 2, and Q_9 and $Q_{10}$are active load 3. Supposing output and the references of these current sources are equal, we will have:

$$i_4 = i_3 = I_{ABC} \quad (1)$$

$$i_6 = i_5 \Rightarrow i_6 = i_2 \quad (2)$$

$$i_8 = i_7 \Rightarrow i_8 = i_1 \quad (3)$$

$$i_{10} = i_6 = i_9 \Rightarrow i_{10} = i_2 \quad (4)$$

KCL in output node results in:

$$i_8 = i_{10} + i_o \Rightarrow i_o = i_8 - i_{10} \Rightarrow i_o = i_1 - i_2 \quad (5)$$

If g_m is defined as the ratio of output signal to the difference between input voltages signal, in order to calculate g_m, $(i_1 - i_2)$ should have a relation with $v_1 - v_2$. With respect to this equation for Q_1 and Q_2 transistors:

$$i_1 = I_s \; e^{\frac{v_{be1}}{V_T}} \quad (6)$$

$$i_2 = I_s \; e^{\frac{v_{be2}}{V_T}} \quad (7)$$

In above equations, $I_s$is reverse saturation current and V_T is thermal voltage. With regard to KCL in emitter node of $Q_1$and Q_2 transistors, we have:

$$i_1 + i_2 = i_4 \Rightarrow I_{ABC} = i_1 + i_2 \quad (8)$$

By filling Equation (8) with Equation (6) and Equation (7), we will have:

$$I_{ABC} = I_s \left(e^{\frac{v_{be1}}{V_T}} + e^{\frac{v_{be2}}{V_T}} \right) \quad (9)$$

or

$$I_s = \frac{I_{ABC}}{e^{\frac{v_{be1}}{V_T}} + e^{\frac{v_{be2}}{V_T}}} \quad (10)$$

so

$$i_1 = I_{ABC} \; \frac{e^{\frac{v_{be1}}{V_T}}}{e^{\frac{v_{be1}}{V_T}} + e^{\frac{v_{be2}}{V_T}}} \quad (11)$$

$$i_2 = I_{ABC} \; \frac{e^{\frac{v_{be2}}{V_T}}}{e^{\frac{v_{be1}}{V_T}} + e^{\frac{v_{be2}}{V_T}}} \quad (12)$$

so it can be said that:

$$i_o = i_1 - i_2 = I_{ABC} \frac{e^{\frac{v_{be1}}{V_T}} - e^{\frac{v_{be2}}{V_T}}}{e^{\frac{v_{be1}}{V_T}} + e^{\frac{v_{be2}}{V_T}}} \tag{13}$$

or

$$i_o = I_{ABC} \times \tanh\left(\frac{v_{be1} - v_{be2}}{2V_T}\right) = I_{ABC} \times \tanh\left(\frac{v_1 - v_2}{2V_T}\right) \tag{14}$$

If, according to Equation (14), i_o current changes curve is plotted based on $v_1 - v_2$ changes, it can be seen that in a very small range of $v_1 - v_2$ changes, the output and input connection is linear, so the system goes to saturation state very fast. OTA circuits obtain hyperbolic tangent function of $v_1 - v_2$; therefore, for a proper performance, it should be assumed that the difference between v_1 and v_2 is very small, i.e. assuming $v_1 - v_2$ is very small $\left((v_1 - v_2) << 2V_T = 50\,\text{mV}\right)$, that is approximately $(v_1 - v_2) << 50\,\text{mV}$, we will have:

$$g_m = \left|\frac{\partial i_o}{\partial (v_1 - v_2)}\right| \tag{15}$$

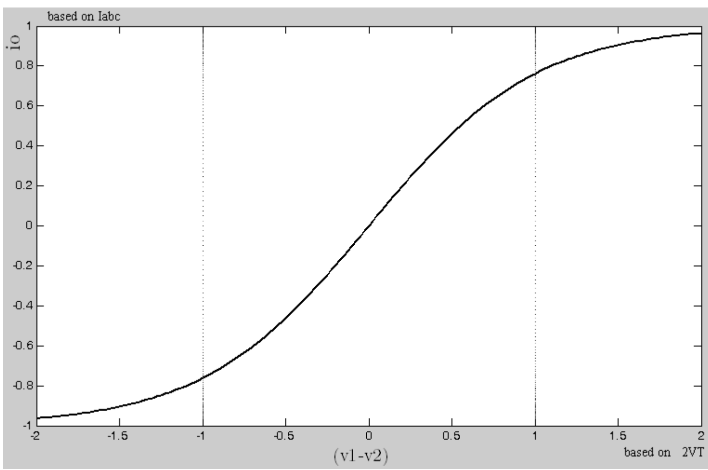

Figure 3. Transfer characteristic curve of i_o per $v_1 - v_2$

$$\tanh\left(\frac{v_1 - v_2}{2V_T}\right) \approx \frac{v_1 - v_2}{2V_T} \tag{16}$$

$$g_m = \frac{I_{ABC}}{2V_T} \tag{17}$$

In Equation (17), V_T is constant and I_{ABC} reference current is determined by additional circuit (an adjustable circuit or a combination of voltage source and variable resistor). Therefore g_m value and thus transfer conductance can be controlled from outside. Voltage gain can be determined as below:

$$A_V = \frac{v_o}{v_1 - v_2} = \frac{i_o\ R_L}{v_1 - v_2} = g_m\ R_L = \frac{I_{ABC}\ R_L}{2V_T} \tag{18}$$

Voltage gain can be controlled by I_{ABC}.