Asian Journal of Information Technology 5(5) : 470-475, 2006

A DC Piecewise-Linear Macromodel for the Voltage Regulator Using Experimental Results

Bourouba, N., N. Bouzit and M. Merabat
Department of Electronics Engineering, Ferhat Abbas University, Setif-Algeria

Abstract: In this study, we present a piecewise-linear appraoch in order to construct a DC model of a voltage regulator which will approximate the external behaviour of the the original circuit. The main goal behind the use of this technique is to solve the non-linear behaviour problem encountred with analogue circuits and to give a simplified description of their complexity. In fact, the model obtained from experimental work corresponds to a two port equivalent circuit whose elements are linear such as resistors, controlled and independent voltage sources. Their values are extracted from measured terminal characteristics by applying that approach over all the operating regions. Simple mathematical equations are joined to the model in order to ease a computer analysis.

Key words: Macromodeling, voltage regulator, piecewise-linear, nonlinear behavior, terminal characteristics

INTRODUCTION

The idea and the use of the macromodeling approach have been less common in the field of analogue circuit design. Due to lack of analogue macro simulators, the analysis of large circuits is still confined to a component level. This task is performed by using device level simulators such as, SPICE, ASTAP, etc. Although these simulators are slow, they provide accurate models on which the accuracy of the circuit simulation depends.

Device models are defined in a mathematical form and stored in library files; a designer can then use these models to build his/her own circuits.

In fact devices used for circuit design suffer from parasitic effects. Since these effects may cause changes in circuit performance, they should be also modeled. Thus a complex model of these devices is needed and which may bear little resemblance to those offered by a simulator. For instance, circuit used in this study, is made of prototyping components from the monochip system manufactured by Ferranti Interdesign.The models developed by this manufacturer describe the real electrical characteristics of these transistors and include parasitic effects that result from the device structure. Fortunately, the device level simulator (SPICE) can incorporate these models by means of the subcircuit construct.These subcircuits are treated like " macros" which are transparent for the user and appear as a black box ". Using these models, the simulator, (SPICE) gives accurate predictions of the behaviour of circuits. As circuits are built up with many non-linear devices, the method is far from the correct solution.This is due to the state of these devices which changes from one iteration to another and leads to a variation in the size of the elements from matrix to another. This process can lead to an accumulation of errors and prevent the Newton-Raphson algorithm from converging. Sparse matrix techniques, therefore, are used to overcome the convergence problem. In a successful simulation of large circuits, the convergence is assured after a large number of iteration cycles. This means that matrix solution process is very slow and therefore, a circuit-level simulation is too costly for complex circuits.To overcome these problems, attempts have been made to model large analogue circuits at their macro levels (i.e. functional level)[1]. Although techniques applied in this process are actually different they have given encouraging results.They have drastically reduced the simulation time and require less storage computer memory, (details on these techniques are in the following sections).

In this study, a Piecewise-Linear (P.W.L) appraoch is proposed to model the function of the voltage regulator which is considered as an important analogue circuit in electronic systems. This PWL modeling technique is applied on terminal characteristics measured practically from experimental works and checked with electric simulator results.Taking advantage of fast simulation and controlled loss of accuracy properties that offers this appraoch[2,3], this modeling technique is used to model all the operating regions of the circuit.

The structure of the rest of the study, is as follows: In section 2, we present a brief description on what was done on macromodeling of analogue integrated circuits. Section 3 shows the purpose of the work in this

---

Corresponding Author: Bourouba, N., Department of Electronics Engineering, Ferhat Abbas University, Setif-Algeria

study and introduces a detailled description of the piecewise-Linear (PWL) approximation approach. To demonstrate the efficiency of this technique,a voltage regulator is employed in section 4 to extract its PWL DC model from its terminal characteristics. Section 5 concludes the study.

PREVIOUS WORKS ON IC'S MACROMODELING PROCESS

An integrated circuit can be thought of as an integrated functional block. The bulk of these circuits can be divided into so-called analogue and digital functional sub-circuits,[3]. The concept of the macromodel approach is based on building models that characterise the input/output behaviour of these circuits. It often pays to test a circuit at a high level rather a detailed level, using a functional/behavioural model. Some of these models are given in the forth coming sections.

One of the most successful macromodel approaches applied to analogue circuits was that developed by Boyle[4,5]. This approach aimed to obtain a model of an operational amplifier in order to reduce the simulation time as much as possible and thereby decrease the cost of a simulation process. A simplification technique is employed as a means of simplifying a portion of the circuit. A certain number of elements are replaced by their ideal elements. These are proposed from a built up technique to compose a portion of a circuit without bearing any resemblance to the actual circuit section but meeting circuit performance specifications. The non-linear behaviour of this circuit is achieved using a simpler circuit composed of perfect diodes at its output.The model has been used with device level simulators, such as SPICE, in order to perform the op-amp and it was six times less complexe than the original one[4] .

A functional or behavioural model approach, also known as "black box" approach, is applied to both analogue and digital circuits. It has a reduced network topology and models the terminal behaviours of both these types of circuit with the same degree of accuracy. The model that results from this approach bears no resemblance to the original circuit but should speed up the simulation process. The use of such model can be oriented for performing AC or DC analysis.

One common way of macromodeling is the use of a single transfer function,[6], which is defined in the form of a table of corner points of the input/output voltage characteristics. The model resulting from this technique is shown in Fig. 1 and is composed of nonlinear resistors and controlled voltage sources. The development of this macro has been possible by using an advanced circuit level simulator which is very suitable for solving nonlinear problems.

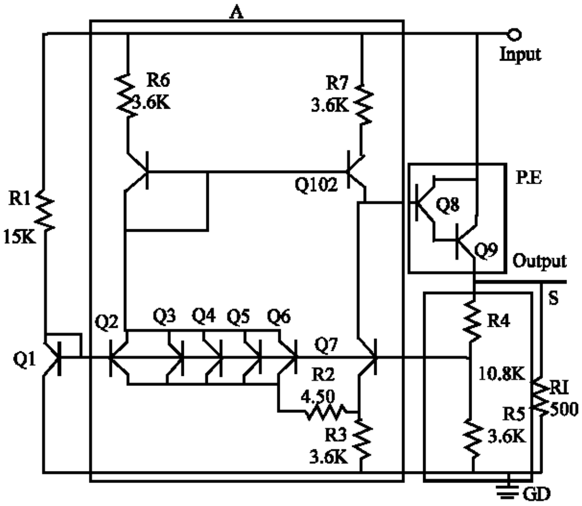


Fig. 1: Two-port equivalent circuit of a voltage regulator

Recently, many techniques for functional modeling based on PWL approaches[7,9] demonstrate several advantages over the Newton-Raphson method used by circuit simulators. It can be used to model strong nonlinearities exhibited at different levels of the circuit, ie from the component to the behavioural and functionnal level. Some of these works are dealing with this kind of circuit behaviour which refer to suden changes and consist of characterizing this behaviour in terms of piecewise linear models.Others techniques are based on a new approach known as hierachical symbolic analysis appraoch. The nonlinear devices are then parametrized PWL modelled and this has led to more compact circuit matrices than that of the traditionnal PWL circuit modeled using ideal diodes.

Our technique proposed here bears a little ressemblance to what has been described in the previous references. It consists of modeling an analogue circuit in term of a simple, uniform and linear equivalent circuit wich differs totally with the original one but characterizing its nonlinear function with a close degree of accuracy. Moreover, this circuit model is established to all its operating regions and a set of linear mathematical equations are joined to it describing its behaviour.

A PIECE-WISE LINEAR, (P.W.L), APPROACH

The P.W.L approach developped in this study, is applied on terminal characteristics obtained from experiments and simulation results. In this technique, one can visualise these characteristics as being composed of several distinct regions in which the terminal parameters of the nonlinear circuits are either essentially constant or at worst, vary in a manner which can be a linear relationship[10].

Description of the P.W.L technique: In performing a P.W.L analysis, the endpoints (breakpoints) of these regions are determined from graphs of different terminal characteristics of the circuit. These breakpoints will drive a good approximation for the characteristics, provided that they have been well selected. Once these points defining regions where the characteristics undergo important changes have been located approximate linear equations can then determine any effect on circuit operation due to parameter changes, and an additional linear model is established.

General procedure: The procedure to be followed in piece-wise linear analysis is:

- Determine all significant breakpoints
- Make any simplifying approximations which give the degree of accuracy required.
- Use these approximations to determine terminal parameters and to define equations between breakpoints.

This technique is based on the characteristics of the circuit. The format used to represent these terminal characteristics is discussed in the following section.

Selection of dc terminal characteristics: There are number of possibilities that may exist to construct a model. For instance, one may choose to express terminal voltages as function of terminal currents, or some hybrid combinations. In fact, circuits provide a certain number N (N is a finite and natural number) of inputs and outputs. Therefore, there exist 2 to power N combinations from which to choose. Obviously, we do not want to make use of all of these combinations, and therefore, one must select formats which will be of use for the circuit under consideration.

Extraction of macromodel parameters: A function of any electrical circuit is defined by its terminal characteristics. However, these characteristics can not be used directly to model this circuit; they must be transformed to model equations with model parameters. The transformation process of these characteristics from a discrete data format (i.e. numerical table or graphical interpretation of signals) into an explicit model equation is known as "model parameter extraction ".A model can be built by use of appropriate characteristics from which the model parameters will be derived. The emphasis in the macromodel development is then placed on the examination and evaluation of both terminal characteristics of the model and the abstract parameters that are attached to it (i.e. C.V.S and resistors). Traditionally, the evaluation of the model parameters can be done either by hand or by using a circuit simulator. The extraction of the model parameters developed in this study, has been carried out manually, since the circuit simulator (SPICE) can not handle nonlinearities. These parameters are thus voltage gain and impedances.The evaluation of the parameters based on the P.W.L. is very useful since it has drastically reduced simulation run time,[9]. The P.W.L approach originates from a mathematical representation where characteristics can be approximated by linear segments separated by breakpoints. This permits the extraction of circuit parameters over all operating regions of the circuit by applying basic network circuit theorems, such as Thevenin and Norton theorem and/or Khirchoff current and voltage laws. The gain and impedances are actually the slope of the transfer voltage and input/output characteristics respectively. Independent voltage sources correspond to the intercepting points of these curves for a zero value current or voltage.

THE MODEL CONSTRUCTION OF THE VOLTAGE REGULATOR

The voltage regulator is the most required device in all electronic systems. It is designed to accept any unregulated or poorly specified voltage from a power supply and to produce an output invariant versus temperature or input fluctuations. Therefore, it supplies the different parts of the system with this regulated output. The circuit used for a macromodeling purpose, was designed to provide on-chip voltage regulation for semi-custom integrated circuits. As this type of circuit is not commercially available, it is then built using monochip concept. Before developing the model of the voltage regulator, it is interesting to have a brief look at its internal circuit and to understand the operation of its components.

Circuit description of the voltage regulator: The voltage regulator, shown in Fig. 2, is the most common and simplest temperature-stable series regulator and is known as a series band-gap reference.As illustrated in this Figs. the internal circuitry is composed of three different basic elements and are the amplifier, A, the pass element, P.E and the Voltage sampler, S. The amplifier A, of this voltage regulator incorporates a band-gap reference (Q3 - Q7, R2,R3) which establishes a reference voltage, Vref, nearly independent from temperature and input

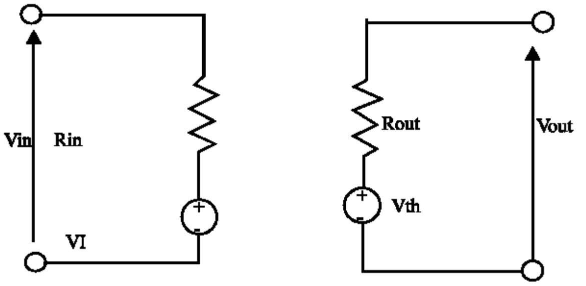


Fig. 2: Circuit configuration of the voltage regulator

voltage fluctuations.This reference, which corresponds to the inverting input of the amplifier, is made up of the base-emitter junction of Q7 and the resistor. The non-inverting input of the amplifier is connected to the diode-connected transistor Q1 which sets the reference voltage from which the amplifier starts to amplify.The output of the circuit as shown in Fig. 2, is a serial combination of the pass-element and voltage sampler. The pass-element is a Darlington amplifier built with transistors Q8 and Q9 which is very convenient for handling high currents. The voltage sampler is used to feed back a fraction of the output of the pass element at the inverting input of the amplifier. This sample of the output voltage is the reference voltage, Vref and can be expressed by the following expression:

$$\text{Vref} = [\text{R5}/(\text{R4}+\text{R5})].\text{Vout} \qquad (1)$$

For small voltages applied to the input of the circuit, the transistor Q1 conducts and turns on the transistor Q2 of the amplifier. Therefore, this transistor provides a current to the current mirror Q101 and Q102 and turns on the Darlington amplifier. The voltage across the resistor R3 and R4 begin to rise. When the voltage at the junction of the resistances R3 and R4 reaches 0.7 Volts which is equal to the voltage drop across the diode Q1, the band-gap starts to conduct and the transistor Q2 turns off. Once the reference voltage reaches a 1.25 Volts, the output stabilises at a voltage of 5Volts which can be fixed through the Equ (1).

The P.W.L. model of the voltage regulator: The voltage regulator model was constructed to describe the external behaviour of the circuit over a input voltage range (1V < Vin < 18 V), wider than that specified above and within the effect of a load, Rl = 500Ω. These two specifications were considered for two main reasons:

- The circuit provides good line and load regulations for a load current of 10 mA. Therefore, this is achieved through the use of a load resistor greater than or equal to 500Ω.

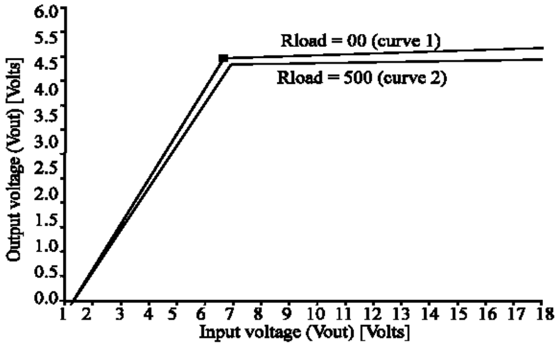


Fig. 3: Voltage transfer characteristic, V_{out}-V_{in}, of voltage regulator under two different load conditions

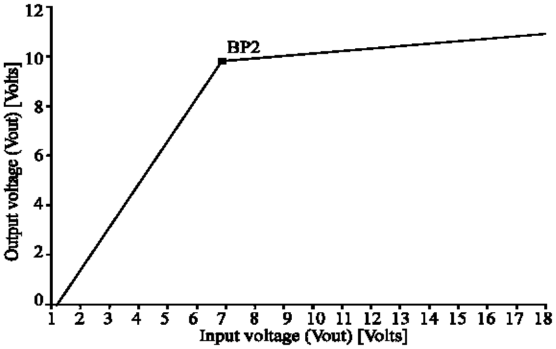


Fig 4 : Input voltage-current characteristic, V_{in}-I_{in}, of the voltage regulator under load condition, R_{load}= 500Ω

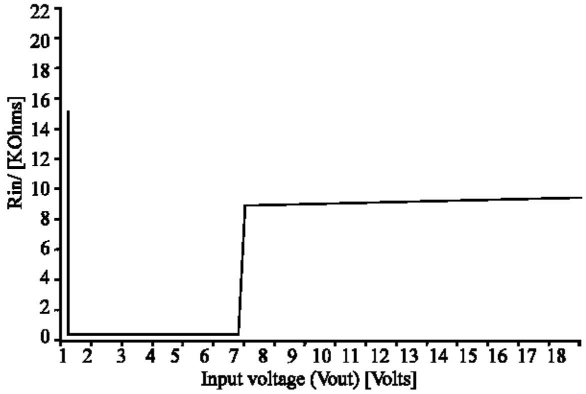
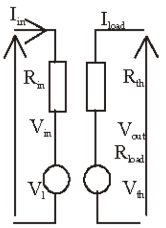


Fig. 5: Input impedance behaviour of a voltage Regulator under load condition, R_{load}500Ω

- The input voltage range was extended down to small voltage values, since it covers all the operating regions of the circuit and therefore, it will lead to a close accurate behaviour.

The model of the voltage regulator, as shown in Fig.1, is a two port equivalent circuit.

Table 1: A cricuit model of voltage regulator

Operatig regions	Equations	Elements'values	Linear model
$1V<V_{in}<1.2V$	1) $V_{in} = R_{in} \times I_{in} + V_1$ 2) $V_{th} = G_v \times V_{in} + V_0$ $V_{out} = V_{load} = I_{load} x R_{load}$ V_{th} : thevenin voltage	$R_{in} = 15.2K$ $V_1 = 0.09V$ $G_v = 0.34$ $V_0 = -0.37V$	Rout0 = 24.11K, a = -11.98KΩV 
$1.3V<V_{in}<2V$	- $V_0 = V_{th}$ at V_{in}= 0Volt 3) model of R_{out} -$R_{out} = a \times V_{in} + R_{out0}$ - R_{out0} = Rout at $V_{in} = 0V$	$R_{in} = 522$ $V_1=1.26V$ $G_v = 0.91$ $V_0 = 1.05V$	
$2.1V<V_{in}<6V$	- $R_{out} = R_{th}$ (thevenin resistance calculated for a load, Rload = 500)	$R_{in} = 522$ $V_1 = 1.26V$ $G_v = 0.91$ $V_0 = -1.05V$ $R_{out0} = 84.54K$ a=-10.8OV-1	
$6.9V<V_{in}<1V$		$R_{in} = 9.105K$ $V_1 = -89.17V$ $G_v = 0.02$ $V_0 = 4.79V$ $R_{out0} = 7.77K$ a=-0.19ΩV-1	

Each port is a serial combination of a resistor and a controlled/independent voltage source. The values of these components are derived from terminal characteristics, such as the input and transfer characteristics. Although these characteristics are non-linear (Figs. 3 and 4), they are divided into three regions each approximated by a linear segment. Thus, network theorems, such as Thevenin theorem, K.V.L, K.C.L were applied to the circuit to calculate the value of the model's elements.

The evaluation of the input port parameters: The plot of the input characteristic (Fig. 4), shows that for small values of the input signal (VIN less than 1.2 V), the input circuit behaves as a high impedance, (Rin =15 KΩ) and thereby the current, Iin, supplied to the circuit is very low.

Referring to circuit diagram, of Fig. 2, this impedance is equivalent to the serial combination of the resistor R1 and the dynamic resistor of the conducting diode-connected transistor, Q1.

When the circuit is operating in the region between the Knees of the curve (i.e. between the breakpoints BPI and BP2), the pass element is conducting and causes the input current to increase sharply. This current reaches its maximum (Iin= 10mA) at the second breakpoint, BP2.This increase of current results from the input which acts as low impedance and whose value is as equal as the load resistance (Rin = RL=500Ω). The third region of Vin- Iin plot is the usual region of the voltage regulator operation. The input current was assumed to limit to its previous value (Iin = 10m.A), but this measured characteristic indicates a slight increase of this current when the input voltage approaches its maximum, Vin= 18 Volts.

This small current variation is caused by the change of the input impedance from a low value to a high value (Fig. 5).The behaviour of the input port can be described by the following linear Equ.

$$Vin = Rin. Iin + V1 \qquad (2)$$

Where V1, is the voltage intercept of each straight line of the input characteristic. The values of the elements Rin, and V1 are given in Table 1.

The evaluation of the output port parameters: The output port of the voltage regulator is a Thevenin representation over all regions of circuit operation. It consists of a voltage source, Vth in series with an output impedance Rout and which both of them are related through the following equation;

$$Rout= [Vout-Vload]/Iload \qquad (3)$$

Where Vload and Iload are, respectively the voltage and the current measured across the load resistor, Rload.

The parameter Vth is theThevenin voltage which is derived from the transfer characteristic of the circuit under no load condition (see curve 1 of Fig. 3). This characteristic was approximated to 3 straight lines, with different slopes. The voltage source was seen as a controlled voltage source defined by the following equ. over each region of operation;

$$Vth = Vin . Gv + V0 \qquad (4)$$

V0 is the intercept point of each straight line and the output voltage axis. As defined by the equation (3), the

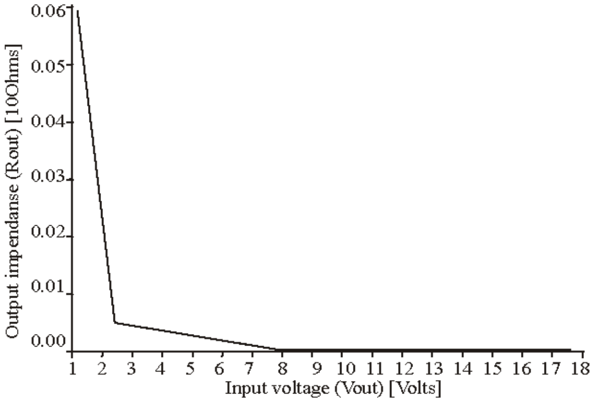


Fig. 6: Output impedance versus input voltage regulator with output load, Rload=500Ω

evaluation of the output impedance, Rout, was carried out over a wide range of input voltage and under the two load conditions (Rload = 500Ω and output open). As plotted in

Fig. 6, This impedance behaves as a non-linear element (Fig, 6). To solve this non-linearity, the characteristic Rout-Vin was approximated by 3 linear segments each described by a linear equ.

$$\text{Rout} = a \,.\, \text{Vin} + \text{Rout } 0 \tag{5}$$

Rout0 is the intercept of each linear segment of the characteristic with Rout axes. The term a, shows the dependence of the output impedance on the input voltage and its dimension

$$[a] = \text{OHM/V} \tag{6}$$

CONCLUSION

The P.W.L approximation approach developed to model analogue circuit such as the series band gap voltage regulator is seen to be very simple in application. It has led to a uniform and linear equivalent circuit. Moreover, the use of terminal characteristics piecewise-linear approximation characterizes the nonlinear behavior in terms of which a linear model is attributed. Therefore, the original circuit is then a set of partial equivalent circuits. It can be noticed from model construction a remarkable reduction of the number of components and hence this will reduce simulation time if the simulation process is carried out. Another interesting point highlighted in this study, is the nonlinear behavior of some of the model's components such as the output impedance Rout. Its measured characteristic, Rout–Vin has been approximated to linear sections characterizing some dependence on input voltage variations.

REFERENCES

1. Johns, D and K. Martin, 1996. Analog Integrated Circuit Design, John Wiley and Sons, New York.
2. Li, X., X. Zeng, D. Zhou and X. Ling, 2001. Behavioral modeling of analogue circuits by wavelet Collocation method ', Proc. Of IEEE/ACM ACCAD, pp: 65-69.
3. Novak, M., 1980. Integrated Functional Blocs, Elsevier Scientific Amsterdam, Edition ISBN.
4. Kiliç, Y and M. Zwolinski, 2004. Behavioral fault modeling and simulation using VHDL-AMS to speed-up analog fault simulation.", Analog Integrated circuits and signal processing, 39: 177-190.
5. Boyle, G.A., D.O. Pederson, B.M. Cohn and J.E. Solomon, 1974. 'Macro modeling of integrated circuit Operational amplifier. IEEE J. of Solid State Circuits; pp: 353-363.
6. Ball, R.J., 1986. Testing electronic Circuits that Contain both analogue and digital elements. M. Phil. Thesis, Lancaster University.
7. Gothoskar, G and A. Doboli, 2003. ' Piecewise-Linear Modeling of Analog Circuits Based on Model Extraction from Trained Neural Networks', Proc. Of the Design Automation and Test in Europe Conference (DATE), Munich.
8. Manthe, A., Z. Li, C.J. Richard Shi and K. Mayaram, 2003. Symbolic Analysis of Nonlinear Analogue Circuits ' Design, Automation and test in Europe, pp: 11108-11109.
9. Yangs, J.S., X.D. Tan and Z.M. Gawecski, 2005. 'Hierarchical Symbolic Piecewise-Linear Circuit Analysis, IEEE International workshop on Behavioral Modeling and simulation (BMAS) San Jose, C CA, pp: 1-6 .
10. Meyer, C.S., D.K. Lynn and J.D. Hamilton, 1969. Analysis and Design of Integrated Circuits, Mc-Graw Hill Book Company, Edition, pp: 208-216.