

AN APPROACH TO AUTOMATIC CIRCUIT DESIGN USING GENERAL-PURPOSE CIRCUIT SIMULATORS

Boriana Ivanova Mihova

Department of Electronics, Technical University of Sofia, Kl. Ohridski No.8 bl.1, 1756 Sofia, Bulgaria, phone: +02 965 37 25, e-mail: bobymih@abv.bg

In this work an approach to automated design of the analog circuits is developed using the standard circuit simulator OrCAD PSpice. The realization of this approach is based on the principles of analog behavioral modeling. The input data, the design conditions and limitations are described using the possibilities of the Spice input language, which allows the inclusion of the complex functional dependencies between the design quantities. The different design stages are presented using the corresponding modules: calculation of the nonlinear distortion coefficient and the standard value determination of a capacitor. These modules are realized by autonomous computer modules and can be used in different design procedures.

Keywords: OrCAD PSpice, automated design, analog circuit

1. INTRODUCTION

The major problem in electronic design automation is the lack of flexible strategies and systematic techniques, which are required for a computer aided approach to the description and design of analog circuits.

Large-scale analog electronic circuits are often constructed from the few well-known basic building blocks, e.g. common-emitter stages, or differential pairs. Although the topology of such a building block is fixed, its elements values may vary strongly depending on the application. Calculating the correct element sizes for given circuit structures and specifications is one of the key tasks in analog circuit design [1].

The analog electronic design can be realized using specialized programs for circuit design, as well as using the possibilities of the standard circuit simulators. The second approach has several advantages. The design procedures are realized by solving linear and nonlinear algebraic systems of equations. In this case the suitable realization of the design procedures is the DC analysis of the standard circuit simulator.

2. MODULE FOR THE CALCULATION OF NONLINEAR DISTORTION COEFFICIENT

The nonlinear distortion coefficient is needed for the realization of the design procedure of the power amplifier, controlled by an operational amplifier [2,3]. A procedure for an automated determination of this coefficient is developed consisting of the following steps:

The current $I_{C_{\max 1}}$ is defined using the dependence $I_{C_{\max 1}} \geq I_{Lm}$. The base current $I_{B_{\max}}$ is determined on the base on the calculated value of the current $I_{C_{\max 1}}$ of the

used transistors from the output characteristic. The collector current i_c' corresponds to the value of the base current $I_{B\max}/2$.

The coefficient of an asymmetry b has a value in the range ($b = 0.05 \div 0.4$).

2.1 Introducing of the input data

The input data for this step are load current I_{Lm} , initial collector current I_C , supply voltage U_{CC} , coefficient of asymmetry b , load R_L .

The input data are introduced using the option *.PARAM* [4]:

```
.param ILm=7.785e-1,IC=7.785e-3,b=0.2,UCC=6.014V,RL=5
```

```
.param ICmax1={1.2*ILm}
```

2.2 Determination of $I_{B\max}$

The nulor model is presented using the controlled source by the gain coefficient of value 10^{10} . The nulor model is used to determinate the current $I_{B\max}$. The equivalent circuit is presented in Fig.1.

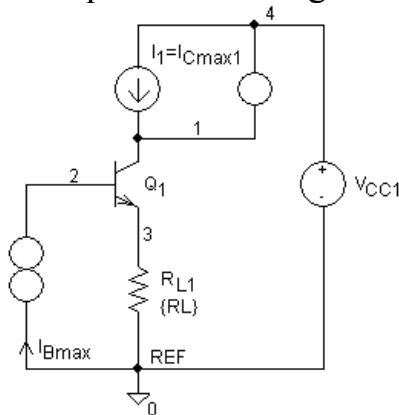


Fig.1. Determination of the current $I_{B\max}$

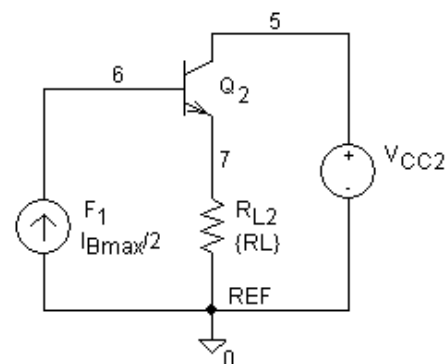


Fig.2. Equivalent circuit for the determination of the collector current i_c'

The voltage $V(1,4)$ is set at zero value by introducing the nullator between nodes 1 and 4. The nulor ensures a current path for the base current $I_{B\max}$ of the transistor, corresponding with the defined collector value $I_{C\max1}$. The independent voltage source *VPROB* type is used as a short-circuit branch.

```
ENOR DOP REF 1 4 1E10
```

```
VPROB 2 DOP DC 0
```

The collector current $I_{C\max1}$ and the supply voltage U_{CC} are applied using the independent current and voltage source of *I* and *V* type.

```
I1 4 1 DC {ICmax1}
```

```
VCC1 4 REF DC {UCC}
```

2.3 Determination of i_c'

The collector current i_c' is determined using the equivalent circuit, shown in Fig.2.

The base current of the transistor Q_2 is equal to the current $I(V_{PROB})$ with the coefficient 0.5. The *Pspice* presentation is realized using the current controlled current source $F1$.

F1 5 REF VPROB 0.5

The current controlled current source $F2$ is used to obtaining the collector current ic' , correspond to $\frac{I_{B\max}}{2}$:

F2 REF Ic1 VCC2 -1

RF2 REF Ic1 1

2.4 Calculation of the harmonic amplitudes

The determination of the harmonic amplitudes is realized using the following equations [2,3]:

$$I_{\max} = (1 + b) * I_{C\max 1}, I_{\min} = -(1 - b) * I_{C\max 1} \quad (1)$$

$$i' = (1 + b) * ic', i'' = -(1 - b) * ic' \quad (2)$$

$$I' = 2 * b * I_C \quad (3)$$

$$I_{1m} = \frac{1}{3} (I_{\max} - I_{\min} + i' + i'') = \frac{2}{3} [I_{C\max 1} - ic'] \quad (4)$$

$$I_{2m} = \frac{1}{4} (I_{\max} + I_{\min} - 2 * I') = \frac{b}{2} [I_{C\max 1} - 2 * I_C] \quad (5)$$

$$I_{3m} = \frac{1}{6} [I_{\max} - I_{\min} - 2 * (i' - i'')] = \frac{1}{3} [I_{C\max 1} - 2 * ic'] \quad (6)$$

$$I_{4m} = \frac{1}{12} [I_{\max} + I_{\min} - 4 * (i' + i'') + 6 * I'] = \frac{b}{6} [I_{C\max 1} - 4 * ic' + 6 * I_C] \quad (7)$$

The harmonic amplitudes I_{1m} , I_{2m} , I_{3m} и I_{4m} are described using the GVALUE elements of *ABM* library, according to the input language of *OrCAD Pspice* and the equations (4), (5), (6), (7).

G11H REF I1H VALUE={0.6667(ICmax1-V(Ic1))}*

R11H REF I1H 1

*G12H REF I2H VALUE={0.5*b*(ICmax1-(2*IC))}*

R12H REF I2H 1

*G13H REF I3H VALUE={ (ICmax1-(2*V(Ic1)))/3}*

R13H REF I3H 1

*G14H REF I4H VALUE={ (ICmax1-(4*V(Ic1))+(6*IC))*(b/6)}*

R14H REF I4H 1

2.5 Determination of the nonlinear distortion coefficient K_h

The nonlinear distortion coefficient is determined by the equation [2,3]:

$$K_h = \frac{\sqrt{I_{2m}^2 + I_{3m}^2 + I_{4m}^2}}{I_{1m}^2} \quad (8)$$

The calculation of the nonlinear distortion coefficient K_h is represented using the *GVALUE* element.

$$GKh \text{ REF } Kh \text{ VALUE} = \{ \text{SQRT}(V(I2H) * V(I2H) + V(I3H) * V(I3H) + V(I4H) * V(I4H)) / (V(I1H) * V(I1H) + 1e-10) \}$$

$$RKh \text{ REF } Kh \text{ 1}$$

<pre>.lib nom.lib .model TR_npn NPN + (Is=2.447p Xti=3 Eg=1.11 Vaf=100 Bf=208.2 Xtf=5.945 Vtf=10 Rb=.1) .param ILM=7.785e-1,IC=7.785e-3 .param b=0.2,UCC=6.014V .param RL=5,Kx=0.01,ICmax1={1.2*ILM} VGND REF 0 0 Q1 1 2 3 TR_npn I1 4 1 DC {ICmax1} VPROB 2 DOP DC 0 ENOR DOP REF 1 4 1E10 VCC1 4 REF DC {UCC} RL1 3 REF {RL} Q2 5 6 7 TR_npn F1 6 REF VPROB 0.5 VCC2 5 REF DC {UCC} RL2 7 REF {RL} F2 REF Ic1 VCC2 -1 RF2 REF Ic1 1</pre>	<pre>G11H REF I1H VALUE=+ +{0.6667*(ICmax1-V(Ic1))} R11H REF I1H 1 G12H REF I2H VALUE=+ +{0.5*b*(ICmax1-(2*Ic1))} R12H REF I2H 1 G13H REF I3H VALUE=+ +{(ICmax1-(2*V(Ic1)))/3} R13H REF I3H 1 G14H REF I4H VALUE=+ +{(ICmax1-(4*V(Ic1))+(6*IC))*(b/6)} R14H REF I4H 1 GKh REF Kh VALUE=+ +{SQRT(V(I2H)*V(I2H)+V(I3H)*V(I3H)+ +V(I4H)*V(I4H))/(V(I1H)*V(I1H)+1e-10)} RKh REF Kh 1 .print dc V([Kh]) VICmax1 ICmax1 REF DC {ICmax1} RICmax1 ICmax1 REF IG .print DC V([ICmax1]) .end</pre>
---	--

Fig.3. *PSpice* realization of the module for the calculation of nonlinear distortion coefficient

```
NODE VOLTAGE
(Kh) 3.10424040 (I1H) .21421107
(I2H) .09186300 (I3H)-.09719964 (I4H)-.04902293 (Ic1) .61289946 (ICmax1) .93420000
```

Fig.4. Simulation results

The *PSpice* realization of the module for the calculation of nonlinear distortion coefficient is shown in Fig.3 and the simulation results are presented in Fig.4.

3. STANDARD VALUE DETERMINATION OF THE CAPACITOR

The standard value determination of the circuit element is an important part of the electronic circuit design. The selection of the standard value depends on the user-defined tolerance. The module for automated standard value determination is realized in correspondence to the input language of *OrCAD PSpice*.

3.1 Introducing of the input data

The calculated value of the investigated capacitor C_{cal} and the user-defined tolerance ε are introduced as input data using the option *.PARAM* [4]. The calculated

value of the capacitor must be presented in the normalized form: $X \cdot 10^N$, where X is the mantissa in the range $(1 \div 10)$ and N is the exponent.

```
.param Ccal=2.75p, EPS=20
```

3.2 Calculation of the exponent

The parameters C_{CAL_i} $i=12,9,6$ are used to calculate the exponent in accordance with the following equation:

$$C_{CAL_i} = C_{CAL} \cdot 10^i \quad (9)$$

```
.param Ccal12={Ccal*1e12}
```

The value of the parameter C_{CAL_12} is compared with the value -12 . In this case the parameter P_1 (participating in the exponent forming) obtains zero value and -12 otherwise.

The comparison is realized using the operator *IF*. It has a following form:

```
.param P1={if(Ccal12>=1,-12,0)}
```

Similarly, comparisons are made for the range nF und μF using parameter values C_{cal9} and C_{cal6} . The exponent P_3 is calculated by the expression:

```
.param P2={if(Ccal9>=1,3,0)}, P3={P1+P2+if(Ccal6>=1,3,0)}
```

3.3 Determination of the mantissa

The mantissa value is in the range $(1 \div 1000)$ is calculated in the form:

$$C_{CAL_m} = C_{CAL} \cdot 10^{|P_3|} \quad (10)$$

The *PSpice* presentation of the Eq.(10) is:

```
.param Ccalm={Ccal*pwr(10,abs(P3))}
```

In similar way the normalized mantissa in the range $(1 \div 10)$ is calculated using two additional divisions.

3.4 Introducing the standard capacitor values from the data base

The standard values corresponding to $\pm 5\%$, $\pm 10\%$, $\pm 20\%$ tolerance deviations are implemented in the program module. The standard value determination, corresponding to the $\pm 20\%$ derivation, is realized using *.FUNC* function, as shown in Fig.5.

```
.FUNC F20(X)={IF(X<=1.1,1.0,0)+
+IF((X>1.1)&(X<=1.85),1.5,0)+
+IF((X>1.85)&(X<=2.75),2.2,0)+
+IF((X>2.75)&(X<=4.00),3.3,0)+
+IF((X>4.0)&(X<=5.75),4.7,0)+
+IF((X>5.75),6.8,0)}
```

Fig.5. Description of the standard capacitor values for the 20 %tolerance

3.5 Determination of the standard value of capacitor

The standard capacitor value C_{ST} is calculated using the following equation:

$$C_{ST} = F_{20}(C_{END}) \cdot 10^{P_3} \cdot L_{20} + F_{10}(C_{END}) \cdot 10^{P_3} \cdot L_{10} + F_5(C_{END}) \cdot 10^{P_3} \cdot L_5 \quad (11)$$

where $L_i = \{if((EPS=i),1,0)\}$ $i=5,10,20$; EPS is the user-defined tolerance.

The *PSpice* realization of the Eq. (11) has a following form:

```
.param CST={F20(A7)*PWR(10,P5)*L20+
+F10(A7)*PWR(10,P5)*L10+
+F5(A7)*PWR(10,P5)*L5}
```

<pre>.param Ccal=2.75p, EPS=20 .param Ccal12={Ccal*1e12} .param P1={if(Ccal12>=1,-12,0)} .param Ccal9={Ccal*1e9} .param P2={if(Ccal9>=1,3,0)} .param Ccal6={Ccal*1e6} .param P3={P1+P2+if(Ccal6>=1,3,0)} .param Ccalm={Ccal*PWR(10,abs(P3))} .param A5={Ccalm/10} .param P4={P3+if(A5>=1,1,0)} .param A6={A5/10} .param P5={P4+if(A6>=1,1,0)} .param A7={Ccal*pwr(10,abs(P5))} .FUNC F20(X)={IF(X<=1.1,1.0,0)+ +IF((X>5.75),6.8,0)}</pre>	<pre>.FUNC F10(X)={IF(X<=1.1,1.0,0)+ +IF((X>7.5),8.2,0)} .FUNC F5(X)={IF(X<=1.05,1.0,0)+ +IF((X>8.65),9.1,0)} .param B1={20-EPS}, B2={10-EPS}+ +B3={5-EPS} .param L20={if(B1==0,1,0)}+ +L10={if(B2==0,1,0)}+ +L5={if(B3==0,1,0)} .param CST={F20(A7)*PWR(10,P5)*L20+ +F10(A7)*PWR(10,P5)*L10+ +F5(A7)*PWR(10,P5)*L5} VCSTAND C STAND 0 DC {CST} RCSTAND C STAND 0 1G .print DC V({CSTAND}) .end</pre>
---	---

Fig.6. *PSpice* realization of the module for determining the standard value of the capacitor

```
NODE VOLTAGE
(CSTAND) 2.200E-12
```

Fig.7. Calculating value of the capacitor

The *PSpice* realization of the module for the standard value determination of capacitor is shown in Fig.6 and the simulation results are presented in Fig.7.

4. CONCLUSION

In this paper an approach to automated circuit design have been developed using the possibilities of the standard circuit simulator *OrCAD PSpice*. The modules for a calculation of the nonlinear distortion coefficient and the standard value determination of a capacitor are presented. These modules can be used in another design amplifier procedures.

5. REFERENCES

- [1] Hennig, E., R. Sommer, *Application of Computer Algebra Methods to Analog Circuit Sizing*, ECCTD'95 European Conference on Circuit Theory & Design, pp. 83-86, pp. 108-113, 1995.
- [2] Zlatarov, V., S. Kucarov, L. Donevska, D. Stamenov, *User's Guide of electronic circuits Amplifiers*, book 3, Technika, Sofia, 1981 (in Bulgarian)
- [3] Zlatarov, V., D. Stamenov, L. Donevska, I. Nemigenchev, *Electronic Circuits and Systems*, Technika, 1988, Sofia (in Bulgarian)
- [4] OrCAD PSpice A/D. Circuit Analysis Software. Reference Manual. OrCAD Inc., USA, 1998