

Design of 2nd order low-pass active filters by preserving the physical meaning of design variables

F. Sandoval-Ibarra

*Centro de Investigación y de Estudios Avanzados-Unidad Guadalajara,
Av. Del bosque 1145, Col. El Bajío, 45015 Zapopan, Jalisco, México,
e-mail: sandoval@cts-design.com*

M. Cuesta-Claros, R. Moreno-Espinosa, E. Ortiz-Levy, and L. Palacios-Betancourt
*Mecatronics Engineering, Universidad Panamericana-Campus Guadalajara,
Calzada Circunvalación Poniente 49, Col. Ciudad Granja, 45010 Zapopan, México.*

Recibido el 15 de diciembre de 2008; aceptado el 17 de febrero de 2011

The purpose of this paper is, by one hand, offer to students basics on active filter design by introducing the Butterworth approach as well as some practical examples not only to show the proposed design flow (DF), but also to show that the design flow's stages have physical meaning mainly supported on physical laws. With the help of these laws, further, it is shown how additional filter design specifications can be translated to the physical design without affect neither the design approach nor DF. On the other hand, because any physical implementation suffer of the non-idealities of electronic components, the modeling of some of them based on both experimental results and spice simulations is presented in order to show how unwanted effects may be added to the DF. An advantage of this proposal is that DF preserves the physical meaning of the design variables. The laboratory-based learning adopted in this work has allowed to students be able to understand physical concepts, capture and analyze experimental data, and use design tools in a correct way mainly to avoid "trial and error" approaches.

Keywords: Filters; electric circuits; modeling.

El propósito de este artículo es, por un lado, ofrecer a los estudiantes fundamentos del diseño de filtros activos usando la aproximación Butterworth y presentando ejemplos prácticos no sólo para mostrar el flujo de diseño propuesto, sino también para mostrar que las etapas del diseño tiene un significado físico que está soportado en leyes físicas. Con ayuda de esas leyes se muestra cómo otras especificaciones de diseño del filtro pueden ser incorporadas en el diseño físico sin afectar la aproximación de diseño ni el flujo mismo de diseño. Por otro lado, porque toda implementación física es afectada por las no idealidades de los componentes electrónicos, se presenta el modelado de varios de ellos soportado en datos de simulación spice y resultados experimentales a manera de mostrar cómo esa información se incorpora al flujo de diseño. Una ventaja de esta propuesta es que se conserva el significado físico de toda variable de diseño. El aprendizaje basado en el trabajo de laboratorio, que es la técnica adoptada en este trabajo, ha permitido a los estudiantes comprender conceptos físicos, capturar y analizar información experimental, y usar herramientas de diseño de manera correcta para evitar aproximaciones tipo "prueba y error".

Descriptores: Filtros; circuitos electrónicos; modelado.

PACS: 84.30.Qi; 84.30.-r; 85.40.Bh

1. Introduction

Engineering subjects can be divided into two basic stages: analysis and synthesis. In the analysis stage, the circuits designer translates *input data* (also called *design specifications*) in a mathematical model in order to study the impact of design parameters on the response of the circuit under design (CUD). At this level of abstraction, the design process is supported in both the experience of the designer and software facilities. Experience refers to the knowledge on electronics and related topics as well access to technical data, *i.e.* books, reports, application notes, and so on. Software facilities, on the other hand, include not only free distribution software, but also professional CAD tools (see Fig. 1).

In the synthesis field the starting point is to translate the properties of the mathematical model in an electronic circuit to verify the fulfillment of the design specifications via CAD tools. Unfortunately, the development of a whole design is not as simple as the *analysis and synthesis* meaning. However, in order to accelerate the comprehension on the design

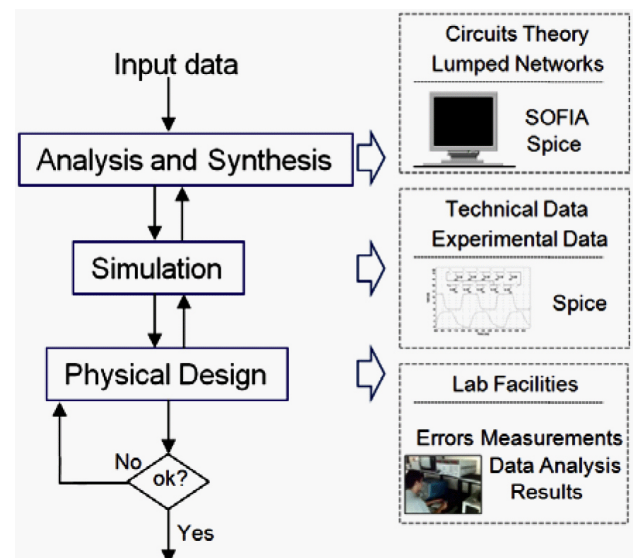


FIGURE 1. Design flow based on three basic design steps.

process any physical implementation can be divided in a set of basic blocks in order to electronically test each one in laboratory; this procedure allows understanding the operation of electronic devices as well as adding laboratory equipment to the DF. Next, because the electronic design without experimental results is a vacancy in electrical engineering activities, in this paper a laboratory-based methodology to design electronic circuits, in the frequency domain, is presented.

As a study case, the design of a Butterworth low-pass filter is used as vehicle to show the advantage of a laboratory-based methodology and also to introduce a formal learning that enhance the students' skills in analog design. This methodology, further, useful for circuits design based on commercial components, is also applicable to the design of fully integrated circuits. According to that focus, Sec. 2 presents the analysis stage, where input data are given in order to find a mathematical model that represents design specifications. In that sense, the analysis of that model, the meaning of each design step, and the discussion of the model's performance in the frequency domain is also given in the same section. The synthesis stage is presented in Sec. 3, where a description based on both Kirchhoff's current law (KCL) and Ohm's law is given in order to analyze a lumped-based RC-active circuit. In the same section, it is demonstrated how the design procedure has a direct impact on each electronic component value without affect the physical meaning of the design variables. In order to verify the fulfillment of the design specifications, spice results are also presented. Sec. 4 presents experimental results of several low-pass filters ranging from 30 kHz up to 135 kHz, where test procedures based on basic laboratory facilities are discussed. At the end of this paper, Sec. 5, conclusions about the proposed DF are given.

2. Analysis

The most important fact of having design specifications is that the designer defines what type of circuit has to be designed according to a mathematical model previously defined. For instance, let us suppose the following set of specifications: Design a second-order Butterworth low-pass filter with a cutoff frequency of 10.0 kHz.

It is well known that a second-order design is, by one hand, the simplest model for analyzing a wide variety of physical systems. From the point-of-view of modeling, a second-order model is actually a transfer function, $H(s)=N(s)/D(s)$, that includes model's basic parameters where quality factor (Q), cutoff frequency (ω_0), and low-frequency gain (A_0) are some examples [1]. The function $D(s)=s^2+(\omega_0/Q)s+(\omega_0)^2$, with s the Laplace's variable and $[\omega_0]=\text{rad/s}$, is a second-order polynomial, $n=2$, that defines the order of $H(s)$. As we shall see, the order of $N(s)$ must satisfy the condition $m \leq n$ to sure the model's stability. Further, the function $N(s)$ is responsible to define the model's characteristic in the frequency domain, *i.e.* $N(s)=k_2s^2+k_1s+k_0$. For instance, to obtain a low-pass characteristic $N(s)$ can be rewritten as $N_{LP}(s)=k_0$, where $k_2=k_1=0$, and $k_0 \neq 0$ is the

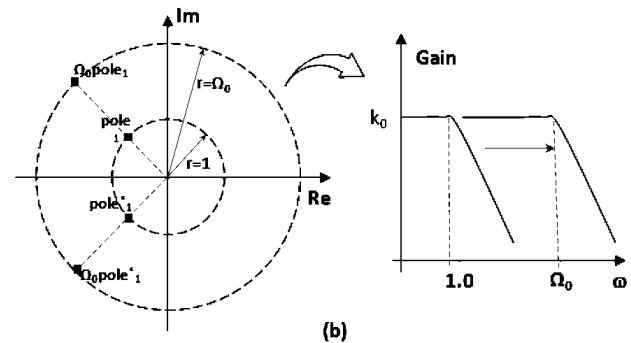
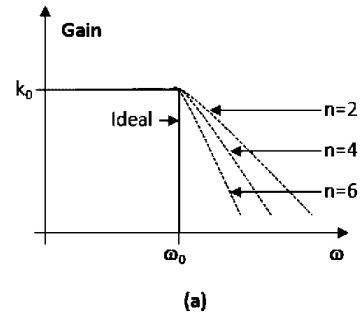


FIGURE 2. Frequency response of Butterworth low-pass filters (a); poles location and their meaning in the frequency domain for a second-order design (b).

TABLE I. Butterworth coefficients.

n	a ₁	a ₂	a ₃	a ₄	a ₅
2	1.414214				
3	2.000000				
4	2.613126	3.414214			
5	3.236068	5.236068			
6	3.863703	7.464102	9.141620		
7	4.493959	10.09783	14.59179		
8	5.125831	13.13707	21.84615	25.68835	
9	5.758770	16.58171	31.16343	41.98638	
10	6.392453	20.43172	42.80206	64.88239	74.23342

trivial solution. Therefore, a second order low-pass filter can be designed with the help of the following mathematical model

$$H(s) = \frac{k_0}{s^2 + \frac{\omega_0}{Q}s + \omega_0^2} \tag{1}$$

In an ideal low-pass filter all signals within the band $0 \leq \omega \leq \omega_0$ are transmitted without loss, whereas inputs with frequencies $\omega > \omega_0$ give zero output (see Fig. 2a). In practice, such a response is unrealizable with physical elements, and thus it is necessary to approximate it. Let us introduce the Butterworth approach, which comprises a set of normalized polynomials. These polynomials are given by $P_n(s)=s^n+a_1s^{n-1}+a_2s^{n-2}+\dots+a_2s^2+a_1s+1$, where the coefficients for n up to 10 are shown in Table I. The Butterworth

TABLE II. Butterworth pole location; these values are call here- after normalized values.

n	Poles	a ₁
2	-0.70711±j0.70711	1.41421
3	-0.50000±j0.86603	1.00000
4	-0.38268±j0.92388 -0.92388±j0.38268	0.76536 1.84776
5	-0.30902±j0.95106 -0.80902±j0.58779	0.61804 1.61804
6	-0.25882±j0.96593 -0.70711±j-0.70711 -0.96593±j0.25882	0.51764 1.41421 1.93186

response for various values of n is plotted in Fig. 2a, where the magnitude of H(s) is down 3 dB at $\omega = \omega_0$ and is monotonically decreasing. As this figure also shows the larger the value of n, the more closely the curve approximates the ideal low-pass response. Unfortunately, a high-order design is an unpractical one because of its excessive cost; power consumption, number of components, PCB area, etc.

On the other hand, the polynomials shown in Table I can be represented by product of quadratic forms, s^2+a_1s+1 , for n even, whereas a linear factor, $s+1$, must added for n odd. The advantage of quadratic forms is that the model's poles are easily calculated (see Table II). Because the poles are on a circumference of radii $r=1$, the Butterworth approach is a maximally flat approximation within a bandwidth of 1.0 rad/s (see Fig. 2b). As Table I shows $D(s)=P_2(s)=s^2+1.4142s+1.0$, where $\omega_0=1.0$ rad/s and $Q=0.7071$, hence $r=1$ is adopted by the independent term in all Butterworth polynomials. So, what about the cutoff frequency (10.0 kHz) requirement? As Fig. 2b also shows, the answer is to increase the radii from 1 to 10^4 by applying a frequency denormalization step via a constant Ω_0 . The latest is defined by

$$\begin{aligned}\Omega_0 &= (\text{actual frequency})/(\text{normalized frequency}) \\ &= 2\pi(10 \text{ kHz})/(1.0 \text{ rad/s}) = 2\pi \times 10^4.\end{aligned}$$

The effect of Ω_0 on the second-order model is added as follows

$$\begin{aligned}H(s)\Big|_{s \rightarrow \frac{s}{\Omega_0}} &= \frac{k_0}{\left(\frac{s}{\Omega_0}\right)^2 + 1.4142\left(\frac{s}{\Omega_0}\right) + 1} \\ &= \frac{k_0\Omega_0^2}{s^2 + 1.4142\Omega_0s + \Omega_0^2}\end{aligned}\quad (2)$$

Note that the model's poles remain in the left-hand s plane and are Ω_0 times their normalized values; the frequency denormalization does not affect the frequency response of the system, it just shift the response up to the actual cutoff frequency (see Fig. 2b). In other words, the band-pass region presents now an equivalent length than that of the radii $r=\Omega_0$.

It is also clear that not only $r=\Omega_0$ is a dimensionless parameter, but also each term in (2) presents a dimension equal to $(\text{rad/s})^2$ making to H(s) a dimensionless function.

2.1. Discussion and numerical results

The description of the design methodology given above presents a twofold purpose: 1) translate input data in a mathematical model based on a review of design requirements and, 2) show the design steps included in the *Analysis and Design* stage. In that sense, the model given in (1) is result of the modeling theory where H(s) has been introduced as a vehicle for analyzing the CUD. Next, because design requirements represent an ideal characteristic, a Butterworth approach was used in order to satisfy low-pass characteristics according to the capabilities of a second-order design. Finally, in order to shift the frequency response from 1.0 rad/s to the actual cutoff frequency, a simple process has been described to clarify both the effect of Ω_0 on the model's poles and the meaning of the frequency denormalization step. At this point, the CUD is modeled by (2), where its magnitude is obtained if s is replaced by $j\omega$. Then

$$\begin{aligned}|H(j\omega)| &= \left| \frac{k_0\Omega_0^2}{(j\omega)^2 + 1.4142\Omega_0(j\omega) + \Omega_0^2} \right| \\ &= \frac{k_0\Omega_0^2}{\sqrt{(\Omega_0^2 - \omega^2)^2 + (1.4142\Omega_0\omega)^2}}\end{aligned}\quad (3)$$

where $j^2=-1$. Note that at very low frequencies (3) reduces to k_0 , while at the frequency $\omega = \Omega_0$ the magnitude is $|H(\Omega_0)|=k_0(0.7071)$ that is equivalent to $[20\log(k_0)-3.0]$ dB; $\omega = \Omega_0$ is actually a product given by $(1.0 \text{ rad/s})\Omega_0$. The result given above shows the reason by which $|H(s)|$ down 3 dB at the cutoff frequency. If (3) is to be computed, it is simplified as follows

$$|H(j\omega)| = \frac{k_0}{\sqrt{\left(1 - \frac{\omega^2}{\Omega_0^2}\right)^2 + \left(1.4142\frac{\omega}{\Omega_0}\right)^2}}\quad (4)$$

By defining $x=\omega/\Omega_0$, the response is obtained by evaluating (4) at very specific values, *i.e.* $x=\{10^{-4}, 10^{-3}, 10^{-2}, 10^{-1}, 10^0, 10^1\}$. There is not a figure-of-merit (FOM) to establish how many points must be computed, however, few values lower/higher than $x=1.0$ gives a suitable estimation of the model's performance. Note that the frequency response has to be printed in a semi-log graph to comprise, in a unique representation, all the frequency values. Alternatively, the frequency response may be obtained with the help of free distribution software [2] not only to compare theoretical values, but also to graphically analyze the frequency response.

3. Synthesis

The most important fact of having a mathematical model is that the designer must be able to propose an electronic circuit that performs the function of that model. One of several

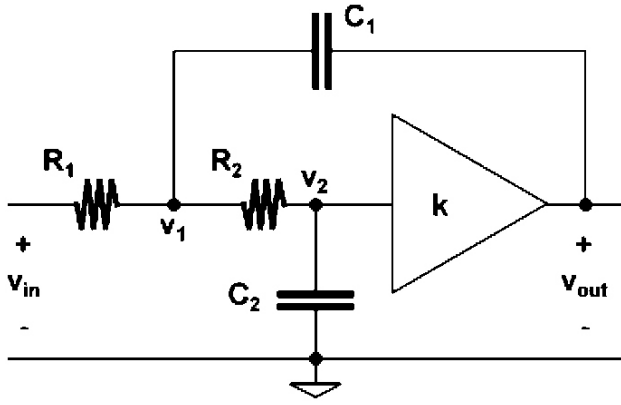


FIGURE 3. Active RC filter.

electronic circuits performing a second-order characteristic is shown in Fig. 3, where k is an amplification factor. In order to obtain the transfer function of the circuit, the following conditions hold: a) electronic components are lumped ones, so that each one represent just a physical characteristic that is unaffected by external/internal effects, *i.e.* temperature, noise, stress, and so on; b) the wavelength λ of signals to be filtered is higher than the physical dimension of the filter by itself. As an example, let us suppose a 20.0 kHz sine signal propagating at $\nu=3.0 \times 10^5$ km/s. The wavelength of that signal is computed from

$$\lambda = \nu / f = (3.0 \times 10^5 \text{ km/s}) / (20 \text{ kHz}) = 15.0 \text{ km,}$$

which is a dimension higher than any physical implementation. Therefore, the filter is a lumped circuit and Kirchhoff's laws can then be used for analyzing it. These laws are based on the Ohm's law that is expressed in the Laplace domain by $V(s)=Z_{eq}(s)I(s)$, where $Z_{eq}(s)$ is the equivalent impedance of each electronic component, *i.e.* R , sL , and $(sC)^{-1}$ for resistors, inductors, and capacitors, respectively. In this work, KCL is widely used because the filter synthesis is based on the so-called voltage-mode design; the current flowing through any electronic component is represented by the voltage across it over its equivalent impedance. By applying KCL, it is easy to demonstrate that design equations are given by

$$(v_1 - v_{in}) \frac{1}{R_1} + (v_1 - v_2) \frac{1}{R_2} + (v_1 - v_{out})sC_1 = 0 \quad (5)$$

$$(v_2 - v_1) \frac{1}{R_2} + v_2sC_1 = 0 \quad (6)$$

$$kv_2 = v_{out} \quad (7)$$

where v_1 and v_2 represent voltage on internal nodes. The first one is the node connecting R_1 , R_2 , and C_1 , whereas the second one connects R_2 , C_2 and the input of the amplifier. Note that (7) is not a current equation but a voltage one. Next, substituting (7) in both (5) and (6), and solving for the transfer function, becomes

$$H(s) = \frac{N(s)}{D(s)} = \frac{\frac{k}{C_1 R_1 C_2 R_2}}{s^2 + s \left[\frac{1}{C_1 R_1} \left(\frac{R_1}{R_2} + 1 \right) + \frac{1-k}{C_2 R_2} \right] + \frac{1}{C_1 R_1 C_2 R_2}} \quad (8)$$

Note that $D(s)$ is an equivalent mathematically model to the second-order Butterworth polynomial. To simplify the analysis it is, therefore, necessary to perform two basic equivalencies: $C=C_1=C_2=1.0$ F and $R=R_1=R_2=1$ Ω . By substituting these values in (8), it follows:

$$H(s) = \frac{k}{s^2 + s(3 - k) + 1} \quad (9)$$

where $(3-k)=1.4142$ or equivalently $k=1.5859$. This result guaranties not only a maximally flat response within a bandwidth of 1.0 rad/s, but also it establishes the value of the amplification factor. By applying the frequency denormalization step to (9), it follows that both (2) and (9) are equivalent models; it is obvious that the constant time $\tau=RC=(1.0 \Omega)(1.0 \text{ F})=1.0$ s is present in both models. Then, the effect of the frequency denormalization step must necessarily modify the value of capacitors. In order to visualize such effect either (2) or (9) can be rewrite as follows:

$$H(s) \Big|_{s \rightarrow \frac{s}{\Omega_0}} = \frac{k_0 \frac{1}{1.0 \left(\frac{1.0}{\Omega_0} \right) 1.0 \left(\frac{1.0}{\Omega_0} \right)}}{s^2 + \frac{1.4142}{1.0 \left(\frac{1.0}{\Omega_0} \right)} s + \frac{1}{1.0 \left(\frac{1.0}{\Omega_0} \right) 1.0 \left(\frac{1.0}{\Omega_0} \right)}} \quad (10)$$

where dimension of all variables have been omitted. It is clear that just capacitors change their value from $C=1.0$ F to

$$C' = C / \Omega_0 = (1.0 \text{ F}) / \Omega_0 \approx 15.9 \mu\text{F}$$

as shown in (10) and also illustrated in Fig. 2b; the constant time is now given by $\tau'=RC'=(1/\Omega_0)$ s. Since the capacitance $C'=15.9 \mu\text{F}$ is not a commercial one, an impedance denormalization step must be performed.

Let us suppose that the capacitance is modified from $C'=15.9 \mu\text{F}$ to a commercial one, $C''=0.1 \mu\text{F}$. So, the impedance denormalization process looks for a constant α relating both quantities, *i.e.* $\alpha=C'/C''=15.9 \mu\text{F}/0.1 \mu\text{F}=159.1$. Therefore, the newest capacitance value is obtained by using the formula $C''=C'/\alpha$, and the newest resistance value is $R' = \alpha R = 159.1(1.0 \Omega) = 159.1 \Omega$ such that the constant time is not affected by α , $\tau'=R'C''=(\alpha R)(C'/\alpha)=RC'$. At this point, the synthesis process ends in the following model

$$H(s)|_{s \rightarrow \frac{s}{\alpha\Omega_0}} = \frac{k_0 \frac{1}{\alpha(1.0)\left(\frac{1.0}{\alpha\Omega_0}\right)\alpha(1.0)\left(\frac{1.0}{\alpha\Omega_0}\right)}}{s^2 + \frac{1.4142}{\alpha(1.0)\left(\frac{1.0}{\alpha\Omega_0}\right)}s + \frac{1}{\alpha(1.0)\left(\frac{1.0}{\alpha\Omega_0}\right)\alpha(1.0)\left(\frac{1.0}{\alpha\Omega_0}\right)}} \quad (11)$$

where the value for each electronic component is clearly indicated. From the mathematical point-of-view (2) and (11) are equivalent models.

3.1. Discussion and simulation results

Two basic design procedures were presented to show their effect on the electronic components' value and illustrate how these values are fitted to commercial ones. However, to visualize the advantage of these procedures via simulation, the design of the amplifier block is still needed.

From the point-of-view of modeling, the simplest lumped amplifier is a voltage-controlled voltage source (VCVS). If this circuit is single-ended type and its input includes a differential port, the circuit is an operational amplifier called commonly opamp. The differential port includes high input resistance (R_{in}), while the output node presents a very low resistance (R_{out}). Figure 4a shows the opamp based on lumped components, where A_0 is the open-loop gain; A_0 for commercial opamps rang between 10^4 and 10^5 , while typical resistance are $R_{in}=10.0 \text{ M}\Omega$ and $R_{out}=100 \text{ }\Omega$ [3,4]. So, the output voltage of the opamp is modeled by $v_{out}=A_0v_d$. Note that this model does not impose any restriction to input signals. As an example, let us suppose a 20.0 GHz sine signal propagating at $\nu=3.0 \times 10^5 \text{ km/s}$. The wavelength of this signal is $\lambda = \nu/f=(3.0 \times 10^5 \text{ km/s})/(20.0 \text{ GHz})=1.5 \text{ cm}$, which is a dimension lower than any filter implementation. According to this result the opamp-based filter under design is not a

lumped circuit but a distributed one [5]. This fact means that for high frequency analysis the filter would be designed with the help of Maxwell's equations instead Kirchhoff's laws. However, because of the intrinsic capacitances of the opamp by itself, other parasite capacitances and additional unwanted effects limit the bandwidth of the opamp making to the filter a lumped design. This is the reason why the filter under design is analyzed via Kirchhoff's laws.

In practice, commercial opamps present a low frequency pole ranging between 100 Hz and 1.0 kHz; the pole is the reason by which the opamp is a limited bandwidth circuit. The effect of the pole on the opamp's performance is added to the lumped model by adding a RC network between the input and output stages as shown in Fig. 4b. The output voltage of the opamp is then modeled by

$$v_{out} = \frac{A_0}{1 + \frac{s}{\omega_p}}v_d \quad (12)$$

where $\omega_p=1/(R_pC_p)$. Note that the proposed lumped circuit includes a VCVS-based output stage with a gain factor $A_1=1$.

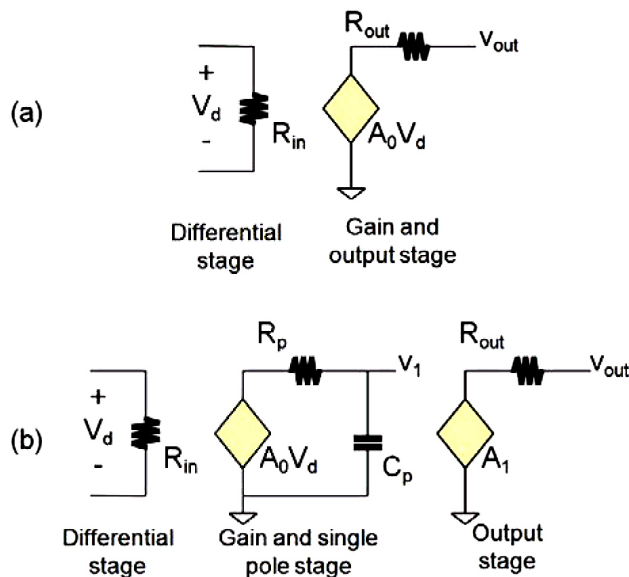


FIGURE 4. Lumped equivalent circuits of opamps.

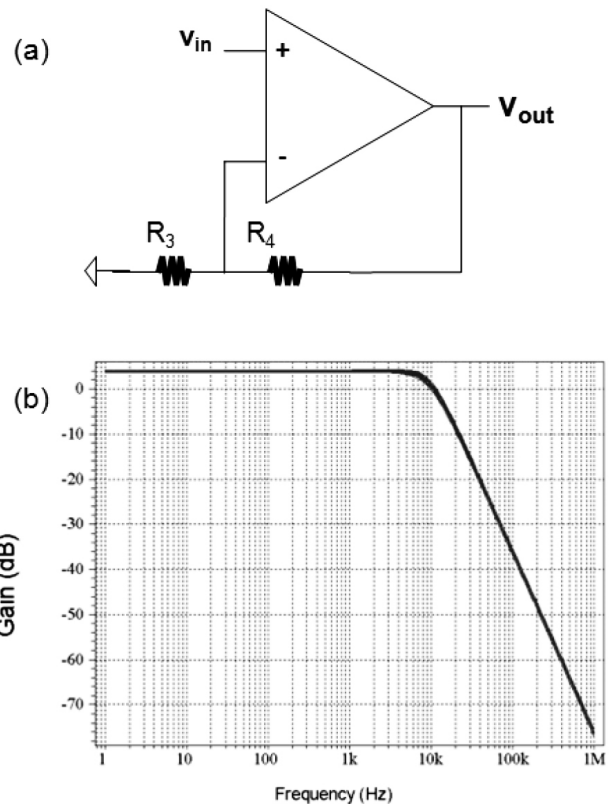


FIGURE 5. Low-pass RC-active filter (a), and the low-pass filter frequency response.

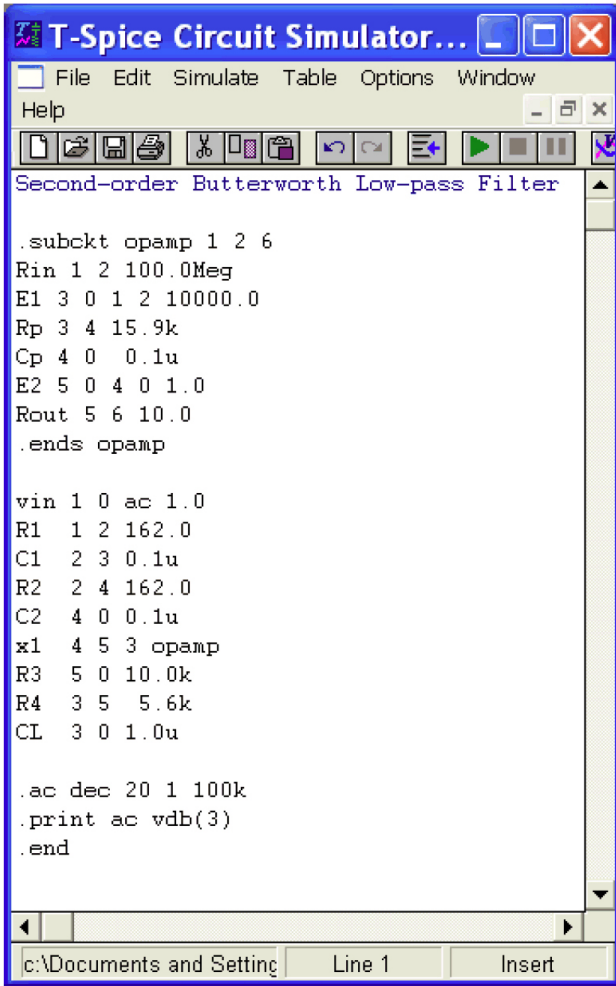


FIGURE 6. Netlist showing the opamp design based on a sub-circuit definition.

In order to model an amplifier with gain k , the circuit shown in Fig. 5a is proposed. It is not only a simple circuit, but also it uses few resistive components. From (12) and taken into account that $A_0 \rightarrow \infty$ and $v^+ = v_{in}$, it is easy to demonstrate that $v^- = v_{in}$. Then, from KCL a unique node equation is obtained:

$$(v_{in} - v_{out}) \frac{1}{R_4} + \frac{v_{in}}{R_3} = 0 \quad (13)$$

or equivalently

$$\frac{v_{out}}{v_{in}} = 1 + \frac{R_4}{R_3} \quad (14)$$

which is a common result reported in literature [6]. Since the resistive value of both R_3 and R_4 are under the designer’s control, by proposing $R_3 = 10 \text{ k}\Omega$ and taken into account $k = 1.5859$ a resistor $R_4 = 5.6 \text{ k}\Omega$ is obtained. Note that both resistors correspond to commercial components. Figure 5b shows the filter’s frequency response obtained from tpsice. The latest is a general-purpose circuit simulator that is easily downloaded by following the procedure described in Ref. 7. As this figure also shows, the low-frequency gain

corresponds approximately to $20 \log(1.5859) = 4 \text{ dB}$, whereas the frequency response presents a loss of -3 dB at a frequency 9.82 kHz . On the other hand, in Fig. 6 the netlist for tpsice simulation is presented; since 159.1Ω is not a commercial value, resistors were fitted to the closer commercial one (162Ω). From the point-of-view of error analysis, such resistance change corresponds to a relative error

$$\varepsilon = (R_{\text{Comm}} - R_{\text{Ideal}}) / R_{\text{Comm}} = 1.17\%.$$

Consequently, the relative error on the cutoff frequency is of the order of 2.0% . Note that the change in the resistance value modifies a little the position of the filter’s poles, *i.e.* the response is actually a quasi-Butterworth one.

4. Simulation

Even when the *Analysis and Synthesis* stage includes simulation as part of the design process, there is a *Simulation* stage in the DF needed to perform simulation runs in order to evaluate the performance of the whole design. As Fig. 1 shows, this design stage uses technical data given by manufacturers and/or experimental data obtained from laboratory activities. Technical data includes generally spice-based electrical models allowing to the designer estimating how parameters of commercial circuits and devices affect the system’s performance. The same is true for those scenarios where temperature effects, via simulation, give also an important qualitative understanding of the system’s performance. On the other hand, experimental data may refer to take into account equivalent circuits to model components’ impedance on the frequency domain, variations of impedances due to temperature effects, or the packaging’s effect on the system performance due to parasitics. As an example the Fig. 7 shows, at bottom, the impedance-frequency characteristic of a capacitor of value $0.3 \mu\text{F}$, where three operation regions are clearly depicted. Experimental values were captured with the help of an impedance analyzer (Agilent, 4192 A). The response for frequencies lower than 4.0 MHz corresponds to capacitive impedance, while for frequencies higher than 4.0 MHz the response is that of inductive impedance. The latest is actually an unwanted effect due to conductive terminals of the capacitor. Note that the figure shows a region on which both capacitive and inductive effects cancel each other. Strictly speaking, the frequency at which the impedance is purely ohmic is called the resonance frequency (f_{res}). From the point-of-view of modeling, the impedance-frequency characteristic can be modeled with the help of the lumped LRC circuit shown in Fig. 7a, where R models the resistance of metallic wires. According to experimental data, R , L , and f_{res} are approximately 0.5Ω , 41 nH , and 4.0 MHz , respectively. The analysis given above is easily translated to the following model

$$Z_{\text{eq}}(s) = R + sL + \frac{1}{sC} = R + \frac{L}{s} \left(s^2 + \frac{1}{LC} \right) \quad (15)$$

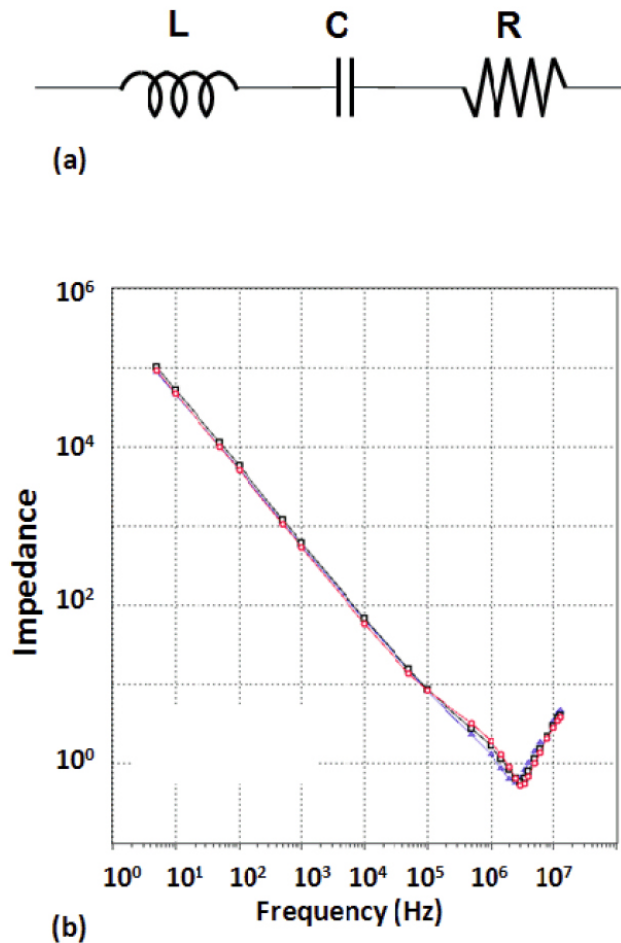


FIGURE 7. The capacitor’s equivalent lumped circuit (a), and its impedance-frequency characteristic (b).

which is obtained by calculating the equivalent impedance (Z_{eq}) of the LRC circuit. As before, if s is replaced by $j\omega$ the magnitude of Z_{eq} is obtained:

$$|Z_{eq}| = \sqrt{R^2 + \left(\frac{L}{\omega}\right)^2 \left(\frac{1}{LC} - \omega^2\right)^2} \quad (16)$$

It is easy to verify that the ohmic impedance is found at the frequency $\omega_0 = (LC)^{-1/2}$ that is actually the resonance frequency given by $\omega_0 = 2\pi f_{res}$. The importance of knowing the resonance frequency value is because it establishes the frequency range on which the impedance corresponds to the ideal electrical characteristic of electronic components. For instance, the capacitor’s performance shown in Fig. 7 indicates that it works correctly as a capacitor from DC to approximately 4.0 MHz, otherwise the impedance-frequency characteristic is due to parasitic effects.

Taking into account unwanted effects, it is possible to perform better simulations by including variations on the components’ values. The frequency response shown in Fig. 5 corresponds to Monte Carlo analysis where all components were allowed to vary no more than 5%. Simulation results show how the filter’s response is not affected by components variation. Note that, at this simulation level, the components

variations do not correspond to a trial-and-error approach but a small variation around the value of each electronic component. The DF takes advantage of Monte Carlo analysis because the design variables of the filter are represented by statistical distributions, which are randomly sampled to produce the filter’s response. Simulation results shown in Fig. 5 were carried out at room temperature.

Another unwanted effect is that of the output capacitive load (C_{Load}). This load is the sum of the intrinsic capacitance of the physical support (protoboard or PCB), the capacitance of the output node by itself, and the probe of the measurement equipment (oscilloscope or spectrum analyzer). Since the filter under design is of the low-frequency class, C_{Load} does not affect the filter’s response. Otherwise, the designer may use simple procedures to estimate the value of C_{Load} and evaluate the circuit’s stability [6]. Note that the netlist shown in Fig. 6 includes a capacitive load of 1.0 μ F, which is enough to ensure the ideal filter’s performance. So, an advice for designing analog circuits is to take into account data sheets of both commercial circuits and related components for writing a netlist as complete as possible representing the physical design of the CUD. In that sense, Martínez-Alvarado in Ref. 2 includes a library based on both commercial opamps and filters topologies such that by choosing each one, the netlist includes the opamp’s electrical model as well as the topology description in spice syntax.

5. Physical design and measurement results

According to the design flow shown in Fig. 1, once the design specifications are fully satisfied via simulation runs, the *Physical Design* is the following design stage. Probably the most popular opamp is that of the 741 family. It is cheap, widely used as a basic building block in many books [6,8], and offers a great variety of applications. In order to verify the correct operation of the CUD, the non-inverting amplifier is easily tested by all students with the help of the setup shown

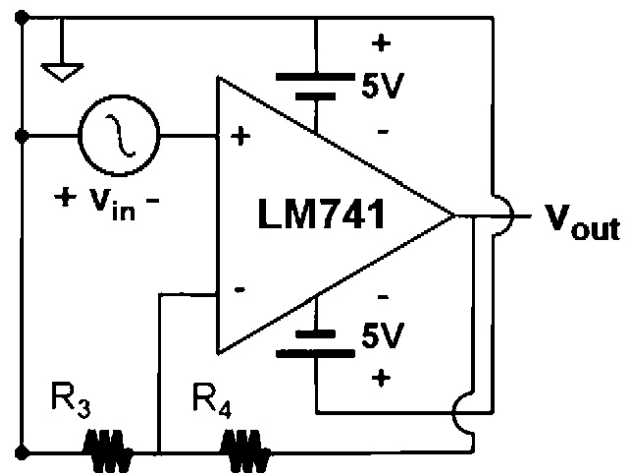


FIGURE 8. Setup for measuring the gain factor $k=1+R_4/R_3$.

TABLE III. Resistance values and gain factor. The latest is defined by $(k_N - k_{meas})/k_N$

$R_3(\Omega)$	$R_4(\Omega)$	k_N	k_{meas}	$\varepsilon(\%)$
560	330	1.58	1.575	0.33
2.2 k	1.2 k	1.54	1.399	9.15
4.7 k	2.7 k	1.57	1.528	2.67
10 k	5.6 k	1.56	-	-

TABLE IV. Resistance/Capacitance values for calculating the cut-off frequency. The relative error is defined by $(f_N - f_{0,meas})/f_N$

$R_1=R_2(\Omega)$	$C_1=C_2(F)$	$f_0(Hz)$	$f_N(Hz)$	$f_{0,meas}(Hz)$	$\varepsilon(\%)$
100	0.1 μ	16 k	15.915 k	13.6 k	14.5
12 k	440 n	30 k	30.142 k	32 k	5.8
330	10 n	50 k	48.228 k	47 k	2.61
1.2 k	2.2 n	60 k	60.285 k	59 k	2.17
10 k	240 p	70 k	66.314 k	69 k	2.8
1.2 k	1.0 n	130 k	132.6 k	131 k	1.22

in Fig. 8, where the opamp (LM741, National Semiconductor) has been biased with a power supply $V_{DD} = \pm 5 V$ (1626, BK Precision). The input voltage v_{in} is actually a function generator (4040A, BK precision), whereas the output voltage is measured with an oscilloscope (TDS-2002B, Tektronix), therefore the test step is done in the time domain. Table III lists several values of commercial resistors (R_3 and R_4) suitable for generating the gain k .

It is well known that resistors suffer of resistive variation that is commonly indicated by the tolerance range. In other words, because the standard resistance color code provides just the nominal value, a set of measurements is recommended in order to obtain the actual value of all design variables. As an example, Table III shows the average gain factor k_{meas} for each pair of resistors, where k_N represents the gain factor due the nominal value of resistors, and ε is the magnitude of the relative error. Fig. 9 shows a sine input signal (344 mV, 14 Hz) as well as the amplifier’s output response; in this example the gain factor is

$$k = v_{out}/v_{in} = 548 \text{ mV}/344 \text{ mV} = 1.59 \equiv 4 \text{ dB}$$

at 14 Hz. Note that k can be computed for a wide frequency range because the circuit under test is purely resistive; take measurements as much as possible allow to the designer obtain a representative gain factor k_{meas} . As Table III shows, the first pair of resistors is the authors’ choice because of the lowest relative error.

As another example let us suppose the design of a 60 kHz low pass filter, where f_N is approximately 60.28 kHz because of the nominal value of components (see Table IV). In this design $\Omega_0=2\pi(60 \text{ kHz})/(1.0 \text{ rad/s})=1.2\pi \times 10^5$ and $C_1=C_2=2.6 \mu F$ at $R_2=R_1=1 \Omega$, or equivalently $C_1=C_2=2.2 \text{ nF}$ at $R_2=R_1=1.2 \text{ k}\Omega$. In order to obtain the actual cut-off frequency, the signal generator inputs a low frequency sine wave

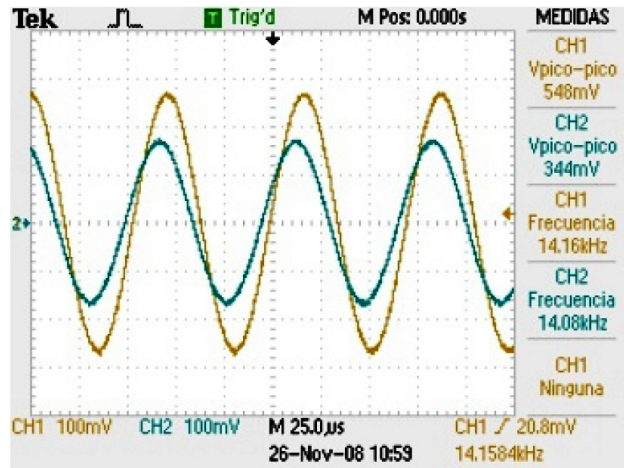


FIGURE 9. The oscilloscope allows estimating the gain factor at a defined frequency.

TABLE V. Magnitude of the relative error for both resistors and capacitors.

$R_N(\Omega)$	$R_{meas}(\Omega)$	$\varepsilon_R(\%)$	$C_N(F)$	$C_{meas}(F)$	$\varepsilon_C(\%)$
10	9.9	1.0	2.2 n	2.3 n	4.54
12	11.8	1.66	10 n	9.97 n	0.3
20	20.4	2.0	0.1 μ	0.12	20
24	23.8	0.83	10 μ	10.28 μ	2.8
100	98	2.0	220 μ	221.3 μ	0.59
220	216.5	1.59			
330	328	0.60			
560	561	0.17			
10 k	9.5 k	5.0			

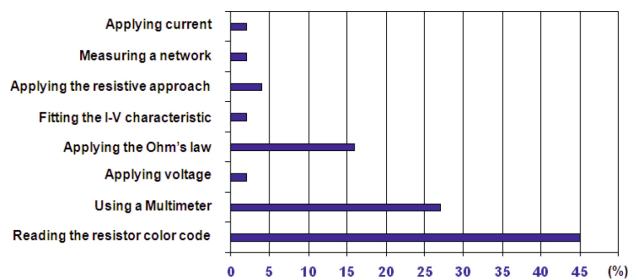


TABLE VI. Magnitude of the relative error for both resistors and capacitors.

with amplitude $V_{pp}=344 \text{ mV}$, and offset voltage 0 V (the reader must remember that the bias is $\pm 5 V$). A low frequency signal is needed for calculating not only the gain factor at that frequency, but also to avoid the attenuation effect of the filter’s pole. Next, for frequencies lower than f_N the amplitude of the output signal is $v_{in}k_{meas}$, and at the cut-off frequency the amplitude is given by $0.7v_{in}k_{meas}$ ($=0.7 \times 1.59 \times 344 \text{ mV}$) as was explained before. Therefore,

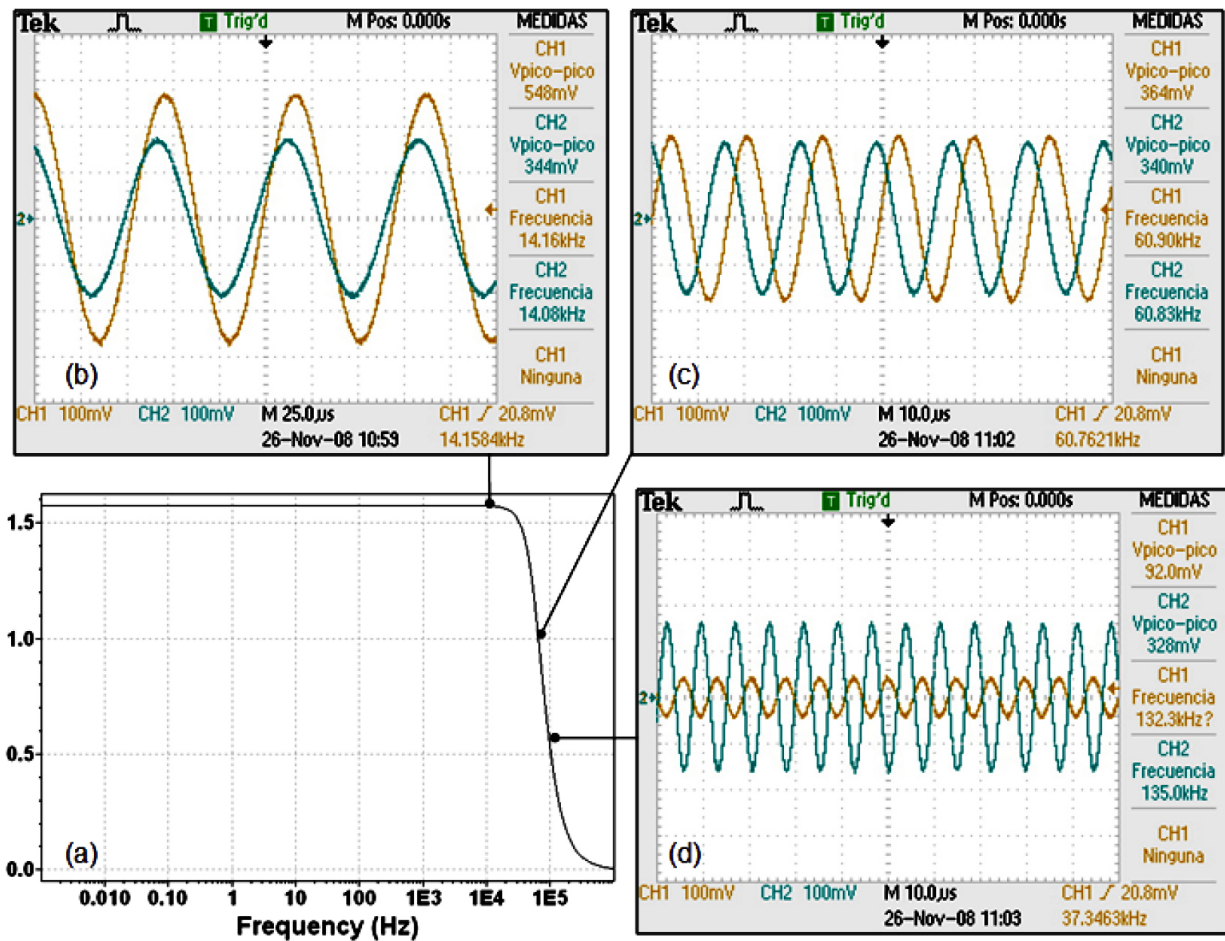


FIGURE 10. Illustrative representation of the output response for frequencies around the filter's cut-off frequency.

the actual cut-off frequency is found by varying the input frequency up to obtain an amplitude equal to 382.87 mV; in this example $f_{0,\text{meas}} \approx 59$ kHz (see Table IV).

Figure 10a shows the gain-frequency characteristic of the 60.0 kHz Butterworth low-pass filter (spice simulation) and the time domain response at three specific frequencies. As we can see, the response satisfying $f \ll 60$ kHz is due to the gain factor only, *i.e.* $v_{\text{out}} = v_{\text{in}} k_{\text{meas}}$ or

$$v_{\text{out}}/v_{\text{in}} = 548 \text{ mV}/344 \text{ mV} = 1.59$$

(see Fig. 10b). Next, at the cut-off frequency $f = 60$ kHz the magnitude of the output voltage is $v_{\text{out}} = 0.7 v_{\text{in}} k_{\text{meas}}$ or $v_{\text{out}}/v_{\text{in}} = 364 \text{ mV}/340 \text{ mV}$ (see Fig. 10c). Finally, the response for frequencies higher than 60 kHz is attenuated, $v_{\text{out}}/v_{\text{in}} = 92 \text{ mV}/132 \text{ mV}$ (see Fig. 10d), as described by theory.

Note that Fig. 10 is a recommended test option mainly when a spectrum analyzer is not a facility in laboratory. This test process could be slow but measuring design variables as much as possible is the correct way for accuracy. Another advantage of this procedure is that students verify by watching the time response how the output voltage's amplitude varies as the frequency of the input signal moves along

the frequency range. This fact makes easy the comprehension of the filtering process.

6. Conclusions

As result of the proposed methodology, students have understood that the filter's response varies due to the tolerance of the passive components, which means (11) is a correct design model. Further, since measurement result is not complete, unless it informs about accuracy, students have measured the actual value (TX1 Multimeter, Tektronix) and have also computed the nominal error (see Table V). This fact is result of the laboratory-based learning and constitutes the base to give a better answer than that given at the beginning of the course (see Table VI); the course, Electronics I, is an undergraduate one offered at the 5th semester of the Mechatronics Engineering career at Universidad Panamericana-Campus Guadalajara.

The proposed methodology allows to the students verify the usefulness of both Ohm's law and Kirchhoff current law in the frequency domain not only to obtain mathematical design models, but also to understand the physical meaning of each term and the geometric meaning of the denormalization

steps as well. An additional goal is the use of basic building blocks, as the opamp is, for doing the synthesis of second order systems from which active filters is just an example. Finally, it is needed to mention that this design methodology, useful for lumped circuits design, is easily extended to other frequency responses (high pass, band pass, band reject), to other design approaches (Chebyshev, Elliptic, etc.), and also to the synthesis of high order systems.

In this paper basics on active filter design by using the Butterworth approach have been presented. Complete design examples for 2nd order low-pass filters were described

by using modeling/measurements of both passive and active components. In order to illustrate the synthesis process an active topology with just one opamp was analyzed. Design consideration and experimental data were presented as vehicle to illustrate how some components deviates from its ideal response. Since this paper is for educational purpose emphasis on spice simulation was done in order to verify design specifications. A free version of T-spice, which is the software used in this paper, can be down loaded at the following www.tanner.com

-
1. Wai-Kai Chen, *Passive and Active Filters: Theory and Implementation* (Chapter 5, John Wiley & Sons, USA, 1986)
 2. L. Martínez-Alvarado, Master thesis *Electrical Engineering* (CINVESTAV-Guadalajara Unit 2002)
 3. E. Sánchez-Sinencio and M.L. Majewski, *IEEE Trans.* (Circuits and Systems, vol. CAS-26, No. 6, June 1979) pp. 395.
 4. M. Alexander and D.F. Bowers, *Op-amp macromodel proves superior in high-frequency regions* (EDN, march 1, 1990). pp. 155.
 5. Ch. Desoer and E. Kuh, *Basic Circuit Theory* (Chapter 1, McGraw-Hill, Singapore, 1993).
 6. J. Forcada, *El Amplificador Operacional* (Chapter 1, Alfaomega, Mexico, 1996).
 7. www.tanner.com/EDA/
 8. D.H. Horrocks, *Circuitos con retroalimentación y amplificadores operacionales* (Capítulo 7, Addison-Wesley Iberoamericana, México, 1994).