University of New Orleans

# ScholarWorks@UNO

# A Further Look at the 'Reactance of a Parallel RLC Circuit'

Kenneth V. Cartwright
*The College of the Bahamas*

Edit J. Kaminsky
*University of New Orleans*, ejbourge@uno.edu

Part of the Electrical and Electronics Commons

## Recommended Citation

Cartwright, K. and E. Kaminsky, "A further look at the "Reactance of a parallel RLC circuit," Latin-American J. Phys. Educ., vol.5, no. 3, Sept. 2011, pp. 505-508.

# A further look at the "Reactance of a parallel *RLC* circuit"

**Kenneth V. Cartwright[1] and Edit J. Kaminsky[2]**
*[1]School of Mathematics, Physics and Technology, College of The Bahamas,*
*P.O. Box N4912, Nassau, Bahamas.*
*[2]Department of Electrical Engineering, EN 809A Lakefront Campus, University of New Orleans, New Orleans, LA 70148, USA.*

**E-mail:** kvcartwright@yahoo.com

## Abstract

We show that the maximum impedance of the parallel combination of a resistor $R$, an inductor $L$ and capacitor $C$, studied in a previous paper can be found analytically, without using calculus. In fact, we show that the maximum impedance of this parallel combination is just the value of the resistor $R$ and occurs when the driving frequency $\omega$ is equal to $\omega_o = 1/\sqrt{LC}$. These results were only demonstrated graphically in the aforementioned paper. We also provide PSpice simulations which verify the theoretical predictions.

**Keywords:** Parallel *RLC* circuit, Maximum impedance, PSpice simulation.


## Resumen

Demonstramos que la impedancia máxima de la combinación en paralelo de una resistencia *R*, un inductor *L*, y un capacitor *C*, estudiado en una publicación reciente puede ser calculada analíticamente sin usar cálculo. De hecho, mostramos que la impedancia máxima de esta combinación en paralelo es simplemente el valor de la resistencia R y ocurre cuando la frequencia de conducción $\omega$ es igual a $\omega_o = 1/\sqrt{LC}$. Estos resultados fueron demostrados sólo gráficamente en tal publicación. Proveemos también simulaciones en PSpice que verifican las predicciones teoréticas.

**Palabras Clave:** Circuito *RLC* en paralelo, Impedancia máxima, Simulaciones PSpice.

## I. INTRODUCTION

Ma *et al*. studied the interesting parallel *RLC* circuit of Fig. 1 in [1], for two specific cases: (i) $R_1 = R_2 = 0$ and $R_3 = R$ and (ii) $R_1 = R_2 = R_3 = R$. On the other hand, Cartwright *et al*. [2] recently studied the circuit of Fig. 1 in detail for $R_1 = R, R_2 = 0$ and $R_3 = \infty$.
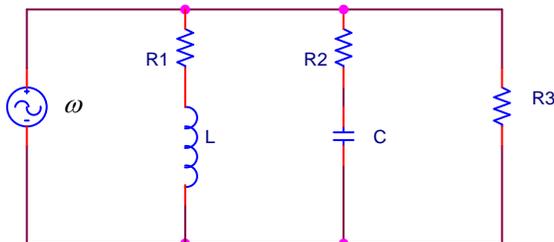


**FIGURE 1**. Schematic diagram of the parallel *RLC* circuit studied in [1] and [2].

In this paper, we are concerned with case (i) studied by the authors of [1], where they showed how the impedance magnitude $Z$ as seen by the source varied with the angular frequency $\omega$ of the source voltage. In fact, they plotted the normalized impedance $Z/R$ as a function of the normalized angular frequency $\gamma = \omega/\omega_o$, where $\omega_o = 1/\sqrt{LC}$, for various values of the dimensionless parameter given by $\rho = \sqrt{L/C}/R$. For convenience, we reproduce their graph here in Fig. 2.
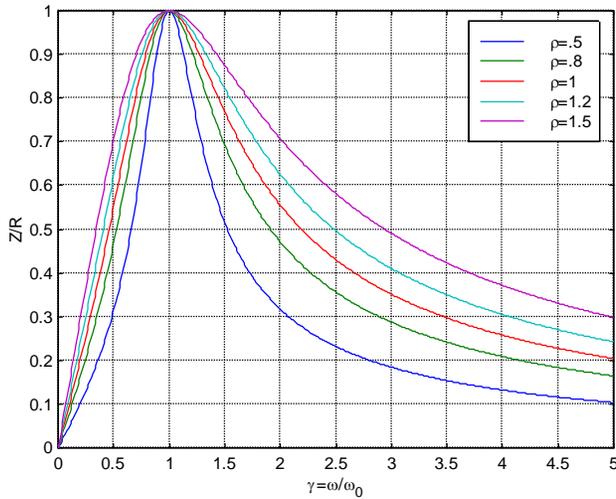
**FIGURE 2.** Relationship between the normalized impedance magnitude $Z/R$ and the normalized frequency $\gamma = \omega / \omega_o$ for the circuit of Fig. 1.

From Fig. 2, it appears that when $\omega = \omega_o = 1/\sqrt{LC}$, the impedance magnitude $Z$ is a maximum with value $R$. This result was apparently unexpected as the authors state [1, p. 163], "It is surprising to see that regardless of the $\rho$ values, $\left| Ze^{j\phi}/R \right|$ reaches to 1 when $\gamma = 1$". Actually, as we show in this paper, this result could be predicted mathematically without the use of calculus, which should be of benefit to the student who has not yet had the opportunity to study calculus. Furthermore, we show that the theoretical results given in Fig. 2 could in fact be obtained by PSpice simulation, which is a popular electrical and electronic circuits simulation software package that is widely used by electrical engineers and some physicists. The latest demo version can be freely obtained from [3].

## II. DERIVATION OF *Z* FOR THE CIRCUIT OF FIGURE 1

The complex impedance $\hat{Z} = Ze^{i\phi}$ of the circuit of Fig. 1 for case (i) is given by

$$Ze^{i\phi} = \left( \frac{1}{R} + \frac{1}{i\omega L} + i\omega C \right)^{-1}. \qquad (1)$$

Eq. (1) can also be rewritten as

$$\frac{\hat{Z}}{R} = \frac{Ze^{i\phi}}{R} = \frac{i\omega L / R}{1 - \omega^2 LC + i\omega L / R}$$

$$= \frac{i\omega \sqrt{LC}\sqrt{L}/\left(R\sqrt{C}\right)}{1 - \omega^2 LC + i\omega \sqrt{LC}\sqrt{L}/\left(R\sqrt{C}\right)}. \qquad (2)$$

$$= \frac{i\gamma\rho}{1 - \gamma^2 + i\gamma\rho}.$$

Hence, using Eq. (2), the normalized impedance becomes

$$\left| \frac{Ze^{i\phi}}{R} \right| = \frac{Z}{R} = \frac{\gamma\rho}{\sqrt{\left(1-\gamma^2\right)^2 + \gamma^2 \rho^2}}. \qquad (3)$$

Note that Eq. (3) can be used to generate the curves in Fig. 2. However, neither Eq. (2) nor Eq. (3) is given in [1], so it is not quite clear what method the authors of [1] used to produce their curves; they likely simply plotted the magnitude of their Eq. (3).

## III. NON-CALCULUS DERIVATION OF THE MAXIMUM Z FOR THE CIRCUIT OF FIGURE 1

Now that a mathematical expression has been determined in Eq. (3) for the impedance, we can show how its maximum value with respect to $\gamma$ can be obtained, without calculus.

We begin by rewriting Eq. (3) as

$$\frac{Z}{R} = \frac{\rho}{\sqrt{\left(\dfrac{1-\gamma^2}{\gamma}\right)^2 + \rho^2}}. \qquad (4)$$

From Eq. (4), it is clear that $Z/R$ is maximized when its denominator is minimized, which happens when $\left(\dfrac{1-\gamma^2}{\gamma}\right)^2$ is minimized, *i.e.*, when $\gamma = 1$, as we expected from Fig. 2. Furthermore, substituting $\gamma = 1$ into Eq. (4) shows that the maximum value of $Z/R$ is one, again as we expected from Fig. 2.

## IV. ANOTHER APPROACH

Rather than deriving Eqs. (1-4), there is a more intuitive approach to showing that the maximum impedance is *R* and occurs when $\gamma = 1$, as we show in this section.

At the resonant frequency $\omega_o = 1/\sqrt{LC}$, it is well-known that the impedance of the inductor/capacitor combination is infinite. Hence, at this frequency, when the inductor/capacitor combination is placed in parallel with the resistor, there will be just be a resistor in parallel with an open circuit, *i.e.*, the total impedance will just be the value of *R*. However, at any other frequency, the impedance of the parallel combination of the inductor/capacitor combination will look like an inductive or capacitive reactance with magnitude of $X = \dfrac{X_L X_C}{|X_C - X_L|}$, where $X_L = \omega L$ and $X_C = (\omega C)^{-1}$. When this combined reactance is placed in parallel with the resistor, the impedance magnitude of the total combination is $\dfrac{RX}{\sqrt{R^2 + X^2}}$ which is lower than *R* for finite *X*.

# V. PSPICE SIMULATED RESULTS

To confirm the theoretical results of Fig. 2, PSpice simulations were performed using the PSpice circuit shown in Fig. 3. Note that we choose the source as a current source rather than a voltage source, but because we are simulating impedance seen by the source, it makes no difference. Also, in this section, we will be using the PSpice notation for voltages and currents in the circuit of Fig. 3. Indeed, the impedance *Z* seen by the current source is V(I1:+)/I(I1), where V(I1:+) is the PSpice voltage at the positive terminal of the ac current source and I(I1) is the PSpice current supplied by the current source. On the other hand, the normalized impedance *Z/R* seen by the current source is given by V(I1:+)/I(I1)/R3. However, R3=V(I1:+)/I(R3), where I(R3) is the PSpice current through R3. Hence, the normalized impedance *Z / R* is given by the ratio of V(I1:+)/I(I1) to V(I1:+)/I(R3), *i.e.*, I(R3)/I(I1). This is what we plot in Fig. 4 on the ordinate, for the same $\rho$ values of Fig. 2.

For the simulations of Fig. 4, we choose to make $L = 1\text{mH}$ and $C = 1.013\mu\text{F}$: hence, $f_o = \omega_o / 2\pi = 5\text{kHz}$, as verified in Fig. 4. Also, we use the PSpice AC Sweep function to vary the frequency from 10 Hz to 25,000Hz, so that the normalized frequency will vary from 1/500 to 5, as given in Fig. 4.

Note Fig. 4 was not created by PSpice; rather, we imported the PSpice simulated data into Matlab. We found that this gave us a more readable graph.
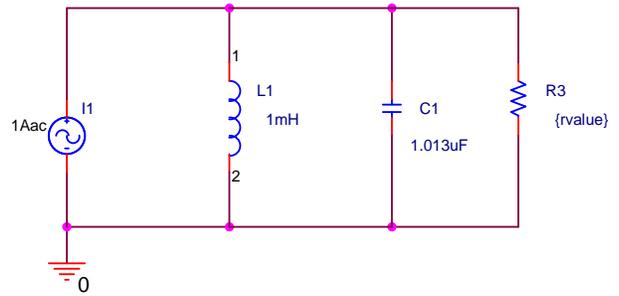


**FIGURE 3.** PSpice schematic diagram of the parallel *RLC* circuit used in the simulations.
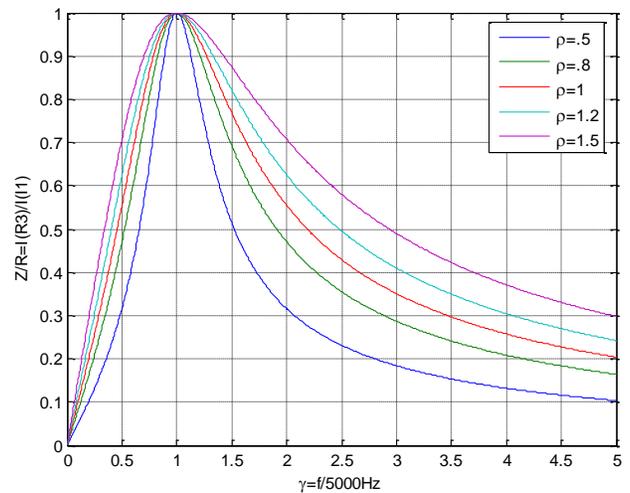


**FIGURE 4.** Plot of the simulated normalized impedance $Z/R = $ I(R3)/I(I1) as a function of normalized frequency $f / f_o$.

Furthermore, we use the fact that $R_3 = \sqrt{L/C} / \rho$. Hence, we vary $R_3$ to achieve the desired $\rho$ value as given in Table I below. In fact, this is the purpose of the PSpice PARAMETERS statement in Fig. 3. We use this statement along with the Parameter Sweep function in PSpice to plot the graph of Fig. 4. (Note the rvalue under the PARAMETERS can be set to any value; here, we choose to set rvalue=$1000 \Omega$).

**TABLE I.** R3 values needed to achieve desired $\rho$ values.

| Desired $\rho$ value | Corresponding R3 value |
|---|---|
| 0.5 | $62.84 \Omega$ |
| 0.8 | $39.27 \Omega$ |
| 1 | $31.42 \Omega$ |
| 1.2 | $26.18 \Omega$ |
| 1.5 | $20.95 \Omega$ |

Kenneth V. Cartwright and Edit J. Kaminsky

From Fig. 4, it is clear that the simulated results do indeed confirm the theoretical results of Fig. 2.

## VI. CONCLUDING REMARKS

We have shown that the maximum impedance of the parallel circuit of Fig. 1 can be determined mathematically without calculus. In fact, we have determined that the maximum value of the normalized impedance magnitude $Z/R$ is one, and occurs when $\gamma = 1$, as the authors of [1] stated, based upon the graphical results of Fig. 2. Furthermore, we provided PSpice simulations in Fig. 4 which confirm the theoretical results of Fig. 2. The advantage of the PSpice simulations is that no complex circuit equations had to be derived to obtain the results in Fig. 4.

Finally, we would like to comment on the difference of terminology of this paper and that of [1].

Clearly, as many (if not most) authors do, we are defining complex impedance to be $\hat{Z} = Ze^{i\phi} = R + iX$, where resistance $R$ is the real part of the complex impedance, whereas the reactance $X$ is the imaginary part of complex impedance. Furthermore, we have defined the impedance $Z$ to be the magnitude of the complex impedance $\hat{Z}$.

On the other hand, the authors of [1] have defined the reactance to be $Ze^{i\phi}$ or indeed just $Z$ throughout the text. However, at the top of Fig. 2 and Fig. 4 of [1], they refer to $Z/R$ as being the impedance.

The reader needs to be mindful of this difference in terminology between our papers.

## REFERENCES

[1] Ma, L., Honan, T. and Zhao, Q., *Reactance of a parallel RLC circuit,* Lat. Am. J. Phys. Educ. **2**, 162-164 (2008). Available from http://www.journal.lapen.org.mx/may08/LAJPE_2008_Vol_2_Num_2.pdf.
[2] Cartwright, K. V., Joseph, E. and Kaminsky, E. J., *Finding the exact maximum impedance resonant frequency of a practical parallel resonant circuit without calculus*, The Technology Interface International Journal **1**, 26-34 (2010). Available from http://www.tiij.org/issues/winter2010/files/TIIJ%20fall-spring%202010-PDW2.pdf.
[3] Orcad PSpice demo download: http://www.cadence.com/products/orcad/pages/downloads.aspx#demo