# Modelling of a second order linear system in SPICE

Stav Zaitsev
Stav.Zaitsev@praxisdynamics.com.au
*Praxis Dynamics, Seaford Meadows, South Australia*

26 May 2012

*Keywords*: Second order system, resonator, SPICE, behavioural modelling

## 1 Introduction

The linear second order ordinary differential equation (ODE) is used to model an incredibly wide range of physical systems. In general, this equation is given by

$$m\ddot{x} + 2m\gamma\dot{x} + m\omega_0^2 x = mf_e(t),$$

where $x = x(t)$ is a variable describing the "position" of the system, $\gamma$ is the damping coefficient, $\omega_0$ is the angular resonance frequency, $m$ is the effective mass (i.e., the measure of the system's inertia), and $mf_e(t)$ is the external excitation force. A dot denotes differentiation with respect to time. This equation is usually written in the standard form as

$$\ddot{x} + 2\gamma\dot{x} + \omega_0^2 x = f_e(t). \tag{1}$$

In engineering, the wast majority of the resonance phenomena are assumed to fit this model, also known as the second order system. The standard example in electrical engineering is the RLC circuit in one of the two configurations: serial or parallel (see Fig. 1). In microelectromechanical systems (MEMS), Eq. (1) describes the behavior of different oscillation modes of mechanical resonators, such as mass sensors and accelerometers, vibrating gyroscopes, and torsional resonators. In optics, the same equation can be shown to apply to various standing modes of optical resonance cavities, and other examples from RF engineering, photonics, and, naturally, mechanical engineering are abundant.

The advantages of using Eq. (1) are well known. First, the equation is linear, and second - there exists an exact solution. For the case of an underdamped system, i.e., $2\gamma < \omega_0$, the solution of Eq. (1) is

$$x(t) = e^{-\gamma t}\left(c_1 \cos\omega t + c_2 \sin\omega t\right) + \frac{e^{-\gamma t}\sin\omega t}{\omega}\int_0^t e^{\gamma\tau} f_e(\tau)\cos\omega\tau\, d\tau - \frac{e^{-\gamma t}\cos\omega t}{\omega}\int_0^t e^{\gamma\tau} f_e(\tau)\sin\omega\tau\, d\tau, \tag{2}$$

where

$$\omega = \sqrt{\omega_0^2 - \gamma^2} \tag{3}$$

is the resonance frequency shifted due to non zero damping, and $c_{1,2}$ are constants determined by the initial conditions. Another important parameter, which is often used instead of the damping coefficient $\gamma$ in underdamped systems, is the quality factor:

$$Q = \frac{\omega_0}{2\gamma}. \tag{4}$$

Due to the wide applicability of the linear second order ODE as a model of different physical systems, a need to simulate it numerically often arises. For example, a simulation of a MEMS based mass detector can include the somewhat idealized model of the mechanical oscillator in the form of the 2nd order system given in Eq. (1), a model of the transducer coupled to the oscillator (e.g., a piezoelectric sensor), a model for the mechanical excitation system (e.g., a piezoelectric or electrostatic actuator), and, finally, the readout and control circuitry. If the electronics designer wishes to verify and optimize his design, he has no alternative but to include at least some approximate models of all the non electronic systems in his SPICE simulations. On the other hand, a cross domain simulation of the exact dynamics of the mechanical (or optical, or any other) system part integrated with the readout and control electronics can give invaluable insights into the gap between the intended and the actual system behavior.

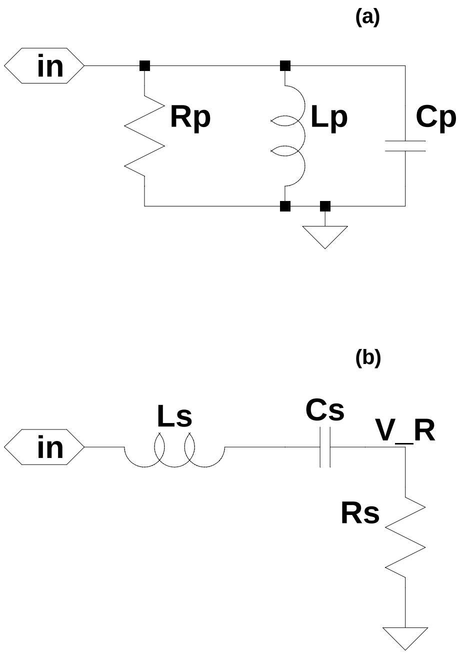


Figure 1: **(a)** Parallel RLC resonant circuit. **(b)** Series RLC resonant circuit.

## 2 RLC resonators

The straightforward approach for incorporating a general second order system model in a SPICE simulation is to employ one of the RLC configurations shown in Fig. 1. For convenience, the differential equations governing the behaviour of these circuits and the values of the main parameters are summarized in Table 1.

| | Series RLC | Parallel RLC |
|---|---|---|
| ODE | $\frac{d^2I_{in}}{dt^2}+\frac{R_s}{L_s}\frac{dI_{in}}{dt}+\frac{1}{L_sC_s}I_{in}=\frac{1}{L_s}\frac{dV_{in}}{dt}$ | $\frac{d^2V_{in}}{dt^2}+\frac{1}{R_pC_p}\frac{dV_{in}}{dt}+\frac{1}{L_pC_p}V_{in}=\frac{1}{C_p}\frac{dI_{in}}{dt}$ |
| $\omega_0$ | $\frac{1}{\sqrt{L_sC_s}}$ | $\frac{1}{\sqrt{L_pC_p}}$ |
| $Q$ | $\frac{1}{R_s}\sqrt{\frac{L_s}{C_s}}$ | $R_p\sqrt{\frac{C_p}{L_p}}$ |
| $\gamma=\frac{\omega_0}{2Q}$ | $\frac{R_s}{2L_s}$ | $\frac{1}{2R_pC_p}$ |
| $f_e(t)$ | $\frac{1}{L_s}\frac{dV_{in}}{dt}$ | $\frac{1}{C_p}\frac{dI_{in}}{dt}$ |

Table 1: The differential equations governing the behaviour of RLC circuits (see Fig. 1) and the values of the main parameters. $V_{in}$ and $I_{in}$ are respectively the voltage and the current flowing into the `in` pin in Fig. 1.

It is easy to see the main disadvantage of the above approach: the external excitation $f_e(t)$ is applied in the form of a voltage or current derivative. It is possible to calculate the time integral of the desired excitation function either by a special function (in PSPICE) or by integrating a current flowing into a capacitor, and then apply the resulting integral to the input of the RLC circuit. However, this represents an unnecessary complication and diminishes the apparent simplicity of using a regular RLC resonant circuit for modelling of a general second order system. In addition, setting any initial condition (i.e., setting a specific bias point) except the trivial one $x=0$ is somewhat challenging, again due to the fact that the external excitation is present in the form of a voltage or current derivative with respect to time.

A better, Pi type RLC configuration is shown in Fig. 2. In this configuration, the position $x$ is represented by the voltage at node `V`. The differential equation governing the configuration shown in Fig. 2 is

$$\ddot{V}+\frac{R}{C}\dot{V}+\frac{1}{LC}V=\frac{R}{LC}I_{in},$$

and, therefore, $\omega_0=1/\sqrt{LC}$, $Q=\sqrt{L/C}/R$, and $f_e(t)=RI_{in}/LC$. The initial conditions for $x=V$ and $\dot{x}=I_L/C$ can be set as initial voltage at node `V` and initial current through the inductor $L$, respectively.

However, some additional, less obvious caveats exist in using any RLC circuit. First, in many cases, the second order system is used to represent a system developing self-excited oscillations,

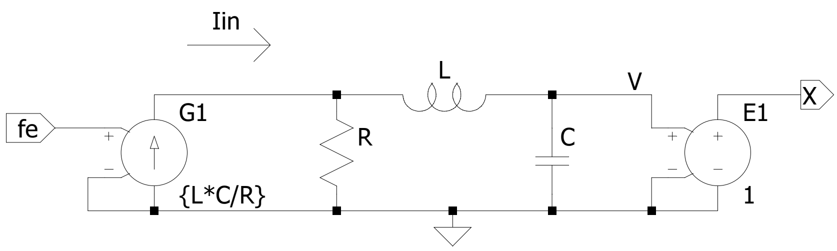


Figure 2: An improved Pi type RLC resonant circuit.

usually in conjunction with some nonlinear amplitude limiting technique. In this case negative linear damping term is essential. However, negative values of $\gamma$ require usage of a negative resistance, which can present serious convergence problem in some simulators.

Second, the RLC circuit is not easily extended to represent a nonlinear system, such as the archetypal Duffing equation

$$\ddot{x} + 2\gamma\dot{x} + \left(\omega_0^2 + \alpha_3 x^2\right) x = f_e(t), \tag{5}$$

where $\alpha_3 x^2$ is the nonlinear position dependent correction to the resonance frequency $\omega_0$. This nonlinear equation is the most widely used model for resonant systems that diverge from the completely linear behaviour of a standard second order system described by Eq. (1) . Duffing equation plays an important role in the modelling of MEMS, nonlinear optical materials, etc.

## 3 Improving on RLC resonators using a behavioural model

Considering the disadvantages of the standard RLC resonators described above, it is often more convenient to use a simple and well structured circuit to model a general linear second order system, such as the one presented in Fig. 3. Two parameters must be supplied to this circuit: $f_0$, which is the resonant frequency, and the quality factor $Q$. In addition, the external excitation is supplied as voltage to pin `fe`. In this model, the current $I$ through the behavioural current source `B1` represents the acceleration, $\ddot{x}$. Two integrating capacitors are used to calculate the velocity $\dot{x}$ and the displacement $x$. It is convenient to set the value of both capacitors to $C = 1/\omega_0 = 1/2\pi f_0$. This results in $V_1 = \omega_0\dot{x}$ and $V_2 = \omega_0^2 x$. Finally, the equation

$$I = f_e - \frac{V_1}{Q} - V_2 \tag{6}$$

is used to set the current through `B1`, which is exactly analogous to Eq. (1).

A typical result of an AC analysis simulation of this circuit is shown in Fig. 4. As expected, the resulting frequency response curve precisely coincides with the theoretical Lorentzian.

Unlike the RLC implementations in Fig. 1, the model in Fig. 3 can be easily made to accommodate any initial conditions by applying appropriate initial voltages to nodes `V1` and `V2`. In addition, it is easy to upgrade this schematics to represent a nonlinear system by simply modifying the current $I$ formula given in Eq. (6) to include nonlinear terms. For example, the Duffing

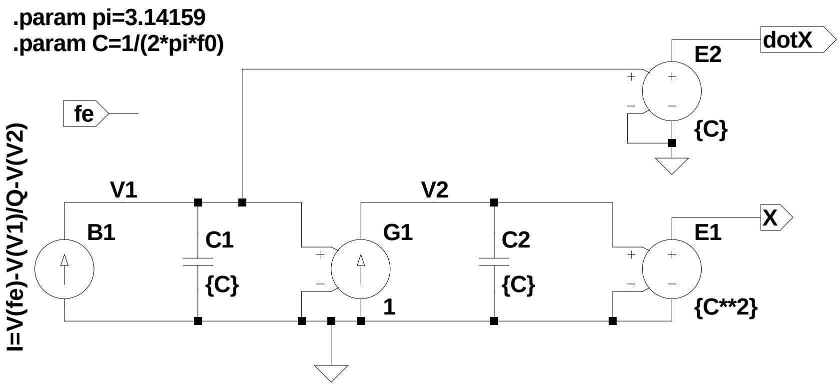


Figure 3: A behavioural model of a general second order system. In addition to the external excitation in the form of a voltage applied to pin `fe`, two parameters must be supplied: the resonance frequency $f_0 = \omega_0/2\pi$, and the quality factor $Q = \omega_0/2\gamma$.

oscillator, whose equation of motion is given in Eq. (5), can be modelled by

$$I = f_e - \frac{V_1}{Q} - V_2 - \frac{\alpha_3}{\omega_0^6} V_2^3.$$

# 4 Technical considerations

In electronic systems, the quality factors usually encountered in practice are relatively low, i.e., $Q < 100$. In contrast, typical values of quality factors of MEMS devices span from hundreds (in air) to hundreds of thousands (in vacuum). The quality factors of optical resonance cavities can be even higher. In some applications the quality factor can be infinite (i.e., no damping) or even negative, in which case self-sustained oscillations may occur. It is important, therefore, to eliminate any unintended sources of damping that may occur in the model used to describe the oscillator. The usual suspects include small parasitic resistances that are often added by default to pure inductors, capacitors, and voltage sources by SPICE based simulators. For example, the popular LTspice IV simulator routinely adds a minimum $1m\Omega$ series resistor to each inductor, which may not be important in regular electronics or RF simulations, but can significantly change the results if an RLC resonant circuit is used to model a high quality mechanical oscillator in a MEMS gyroscope, for example.

Another possible source of inaccuracy, especially in transient simulations that span many oscillation periods (as is often the case in systems with high $Q$), is numerical damping. This damping effect is mainly due to insufficiently small simulation time steps. Sometimes it is advisable to simulate the system with no damping at all first, in order to see that the oscillation amplitude does not drift unexpectedly.

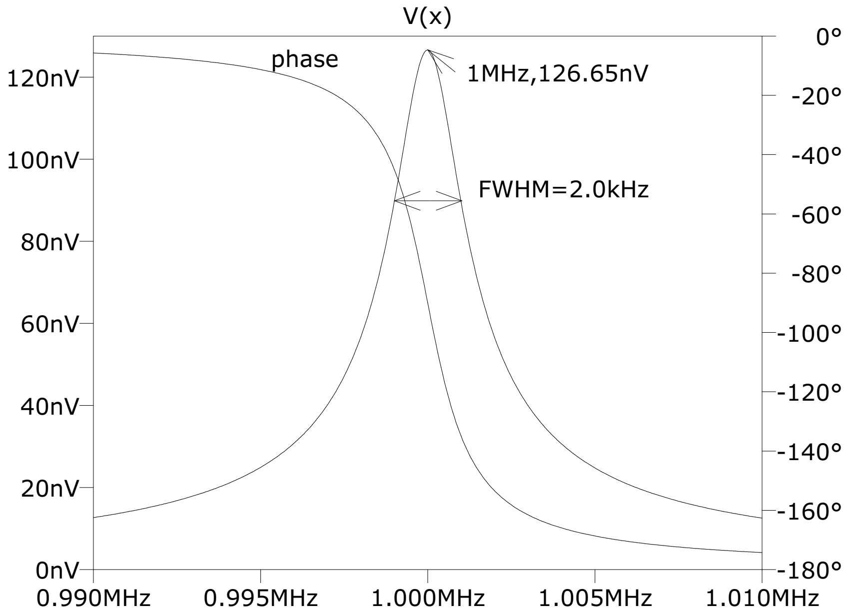


Figure 4: Typical frequency response of the circuit shown in Fig. 3. The system models a typical MEMS resonator, with effective mass $m = 1\,\mathrm{ng}$, resonance frequency $\omega_0 = 2\pi \times 1\,\mathrm{MHz}$, and quality factor $Q = 500$. The excitation amplitude is $mf_e = 10\,\mathrm{nN}$. The expected amplitude at resonance is $x_{\max} = Qf_e/\omega_0^2 = 126.65\,\mathrm{nm}$. The expected "full width at half maximum" (FWHM), i.e., the width of the resonance peak at half the maximum power, namely at $x = x_{\max}/\sqrt{2}$, is $\omega_0/Q = 2\,\mathrm{kHz}$. The results of the simulation are in perfect correspondence with the theoretical predictions.

# 5 Summary

A cross-domain simulation in SPICE requires creation of models that correctly describe the behaviour of mechanical, optical, and other subsystems included in the simulation. One of the most popular models, the linear second order system, which represents an archetypal resonator, can be implemented using a simple RLC circuit. The possible configurations (series, parallel, Pi) are described above, including the governing equations, all relevant parameters, and initial conditions, where applicable. The Pi configuration is by far the easiest configuration to use. If, however, the resonator that is being modelled has some non standard features, such as negative damping, or possible addition of nonlinear terms, a general behavioural model shown in Fig. 3 should be preferred.

All circuits and simulations presented in this paper were created using the LTspice IV simulator developed by Linear Technology Corporation.