

## 2<sup>nd</sup> Order RLC Filters

**OBJECTIVE:** Familiarization with the analysis and simulation of filters.

### 1. RLC CIRCUIT

Figure 1.a) shows a parallel RLC circuit whose natural response is described by:

$$D(s) = s^2 + \frac{\omega_o}{Q}s + \omega_o^2 = s^2 + \frac{1}{RC}s + \frac{1}{LC} = 0 \text{ where } \omega_o = \frac{1}{\sqrt{LC}} \text{ and } Q = \omega_o RC.$$

After this network it is possible to implement four filter types, as illustrated in figures 1 b), c), d) and e). All of them present the same natural response and, thus, also the respective poles are the same. That means the polynomials in the denominators of the respective transfer functions are equal to  $D(s)$ . The terminal where the input signal is applied defines the numerator of each one of the respective functions and consequently the type of filter.

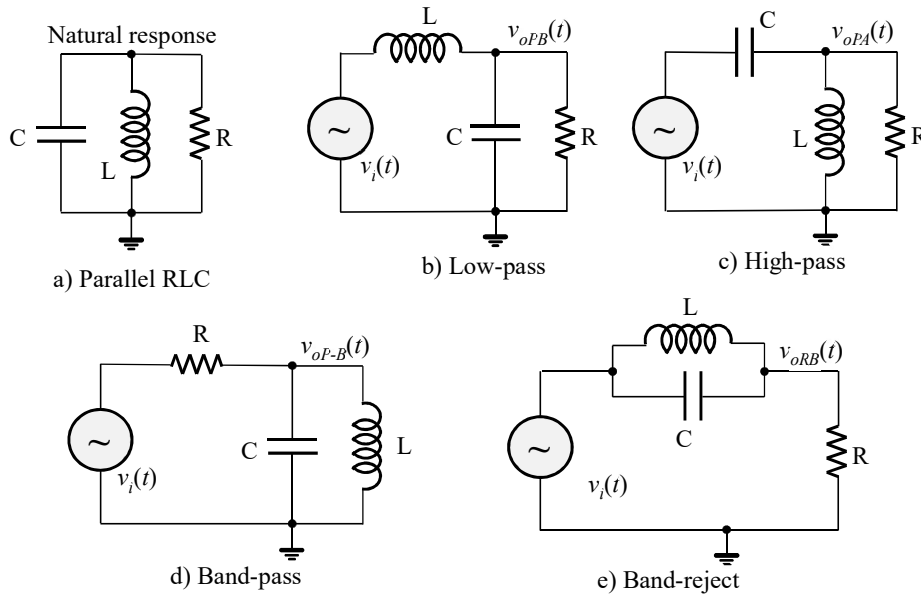


Figure 1 – RLC filters.

- 1.1 Determine R, L e C values for the case where  $D(s) = s^2 + 89 \times 10^3 s + 3,95 \times 10^9$ . Resistor R should show a value of units of k $\Omega$ .
- 1.2 After inspecting, qualitatively, the circuits in figure 1, determine the numerator  $N(s)$  of  $H(s) = N(s)/D(s)$  for each one of the four filter types.
- 1.3 Calculate the most relevant functional characterization parameters of each one (e. g., 3 dB cut-off frequency(ies), phase and amplitude attenuation behaviours, quality factor, amplitude at the natural frequency  $\omega_n$ , ...).

## 2. SIMULATION

**2.1** Using the LTSpice simulator, create a single schematic with the four filters and assign the values calculated in 1.1 to the R, L, and C components. All filters' inputs must be connected to the same voltage source so that the four frequency responses can be obtained after the same simulation.

Note: take advantage of the sub-circuits and hierarchical schematics to create a symbol of the RLC circuit that can be used to create the different instances<sup>1</sup>.

**2.2** Perform an *ac* analysis to obtain the respective frequency responses. For the *.ac* simulation directive use

```
.AC DEC 100 10 10Meg
```

which specifies an *ac* analysis with a frequency sweep in decades, 100 points per decade, from 10 Hz to 10 MHz.

**2.3** After running the simulation, draw the amplitude and phase diagrams of  $V_o(j\omega)/V_i(j\omega)$  of each one of the filters' outputs.

**2.4** Obtain the widths of the respective passband, phase and amplitude attenuation behaviours, quality factor, and amplitude at the natural frequency  $\omega_n$ . To obtain automatically the cutoff frequency you can use the following command:

```
.meas AC fco WHEN v(vo)=1/sqrt(2)
```

The two following measure commands allow you to obtain the bandwidth of the band-pass filter.

```
.meas AC peak max mag(V(vo))  
.meas AC BW trig mag(V(vo))=peak/sqrt(2) rise=1  
+ targ mag(V(vo))=peak/sqrt(2) fall=last
```

The result is given in the simulation log file (View > Spice Output Log).

**2.5** Comment the obtained results and compare with the expected ones.

---

<sup>1</sup> See [https://www.ieee.li/pdf/viewgraphs/ltspice\\_creating\\_a\\_schematic\\_symbol.pdf](https://www.ieee.li/pdf/viewgraphs/ltspice_creating_a_schematic_symbol.pdf)