Technical Article

Designing and Simulating EMC Filters with LTspice

October 19, 2020 by Ignacio de Mendizábal

In this article, we will review the different types of noise that are present in a circuit. We will also discuss how to perform an accurate simulation of an EMC filter with LTspice.

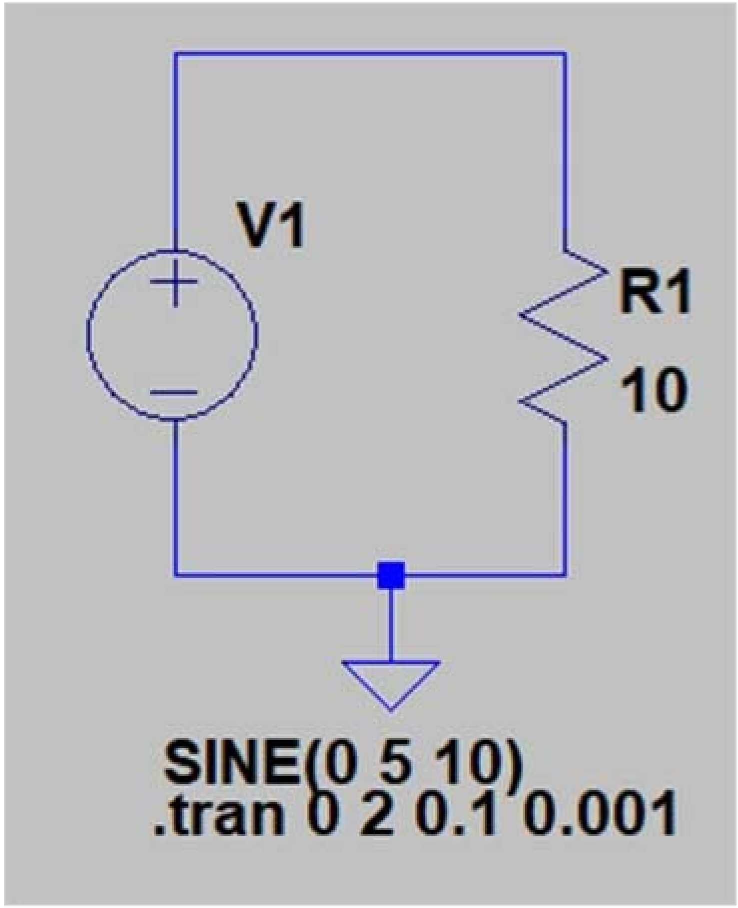
LTspice is a powerful tool for simulating electronic circuits. It can perform simple simulations to verify the functionality of a new design. This tool also completes complex analyses such as worst-case analysis, frequency response, or noise analysis, among others, in a short time.

LTspice is also useful in simulating noise and filters. Filters are critical elements in a circuit for many applications. In particular, electromagnetic compatibility (EMC) filters are used to reduce noise and interference. To obtain accurate results in simulations, real-life circuit phenomena need to be taken into account. Parasitics play a key role in filtering since they can provoke the opposite effect and amplify noise.

In this article, we will review the different types of noise that are present in a circuit. We'll then discuss how to perform an accurate simulation of an EMC filter with LTspice.

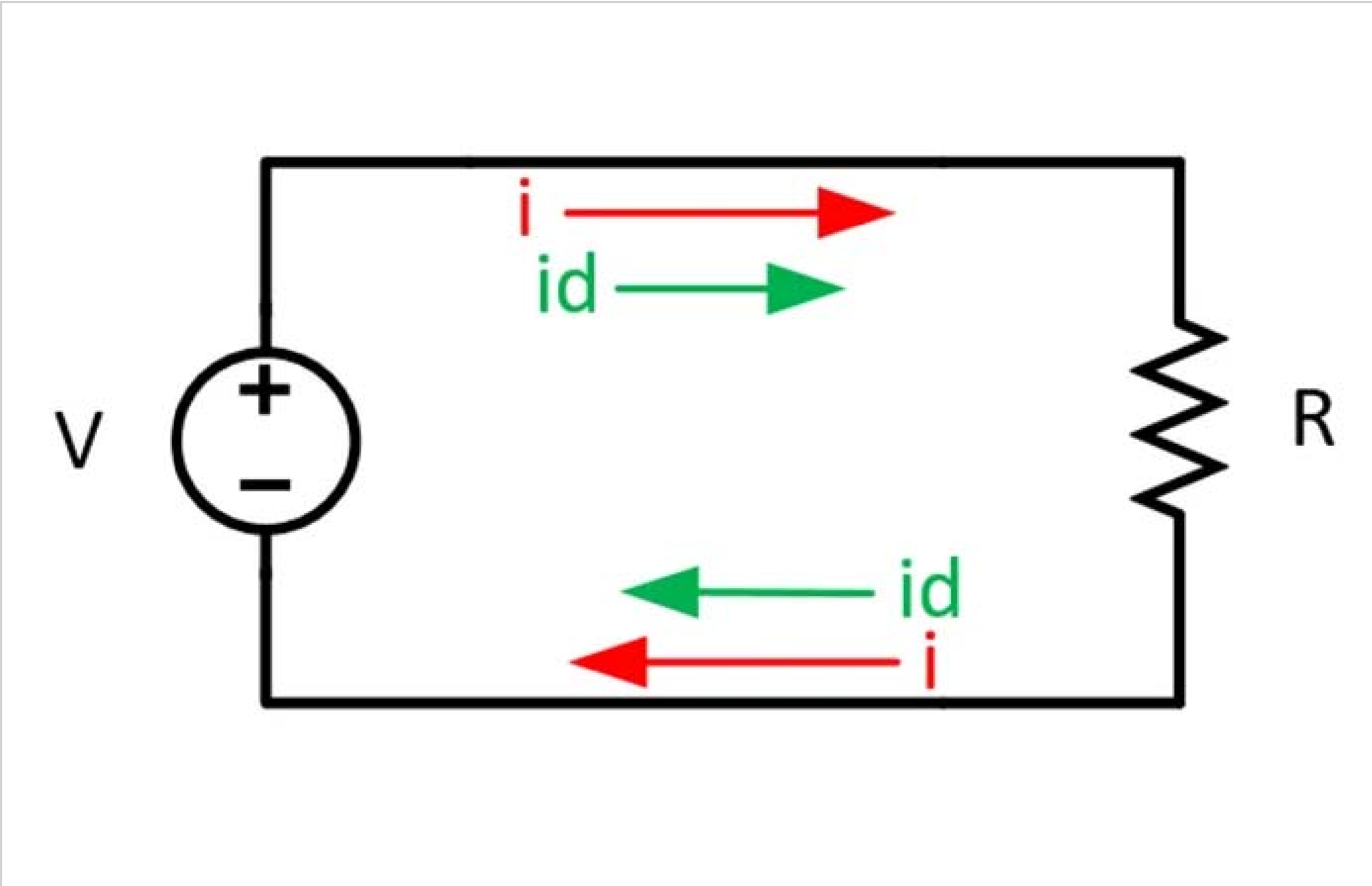
Common-Mode Noise and Differential Noise

Before designing an effective filter, we need to know what kinds of noise can be present in a circuit. The following is a simple circuit that will serve to explain the two types of noise.

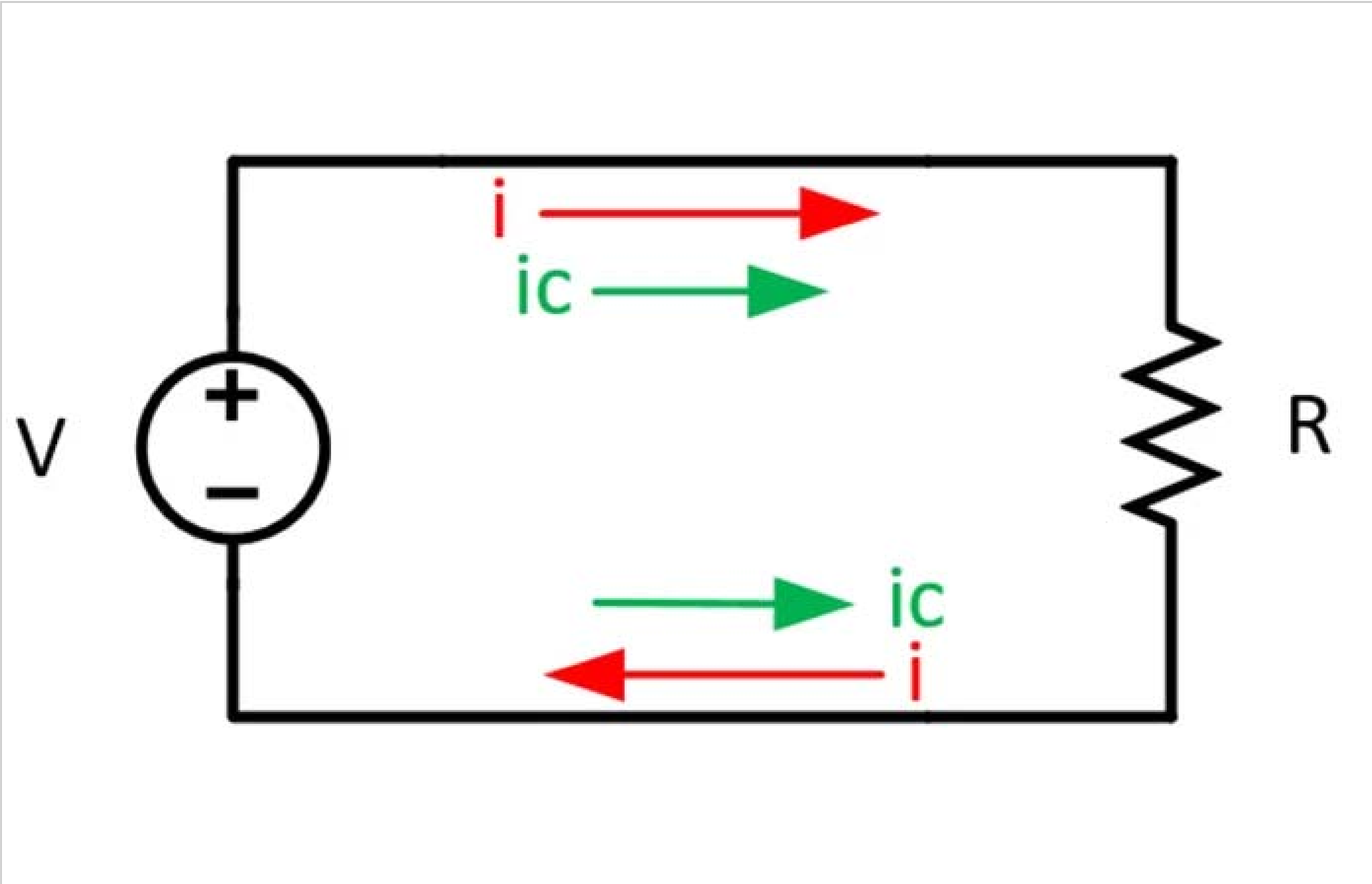


The current generated by the voltage supply V1 will circulate through R1. It will then go back to the source (i.e. ground or the reference voltage).

In ideal conditions, the only current and voltage present in the circuit will be generated by the source V1. As shown below, the noise that follows the same direction as this current is called **differential noise**.



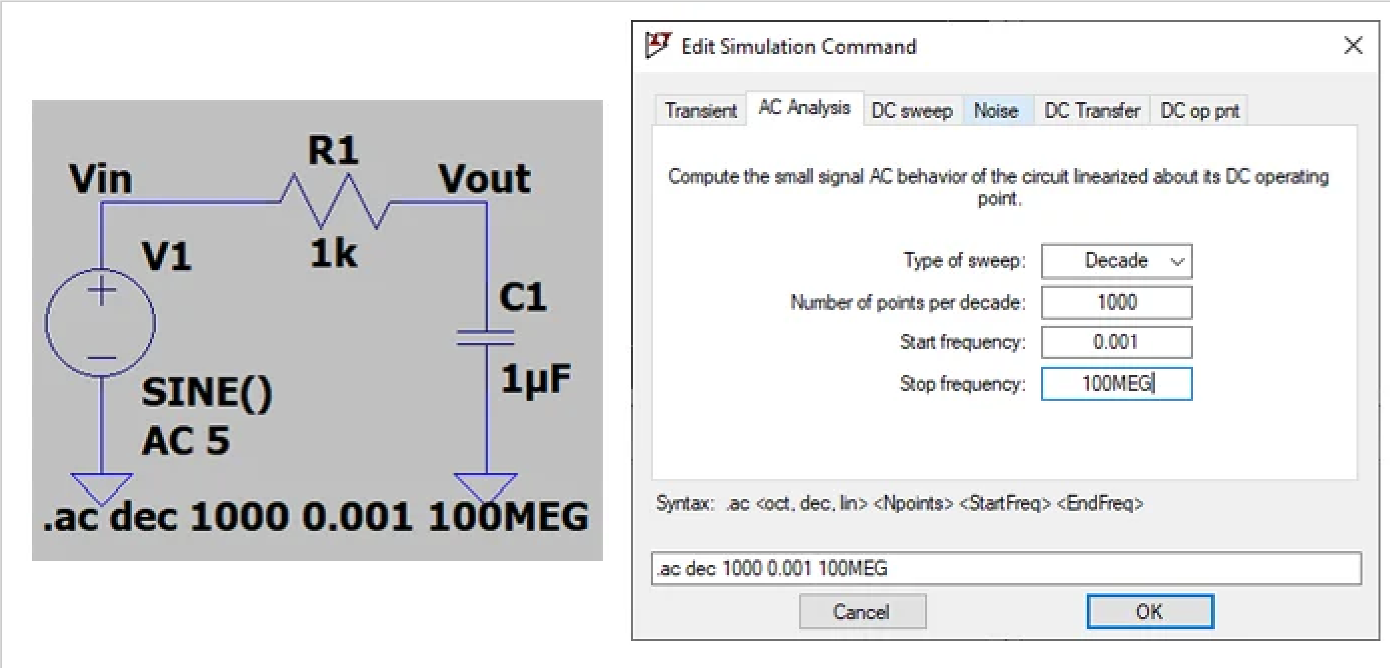
There is a second situation in which **common-mode noise** appears. In this case, the direction of the noise current is opposite to the direction of the return current flowing from the load back to the source.



To be effective against interference, filters need to account for both types of noise. The components selected and their location vary depending on the type of noise to attenuate.

Bode Plots in LTspice

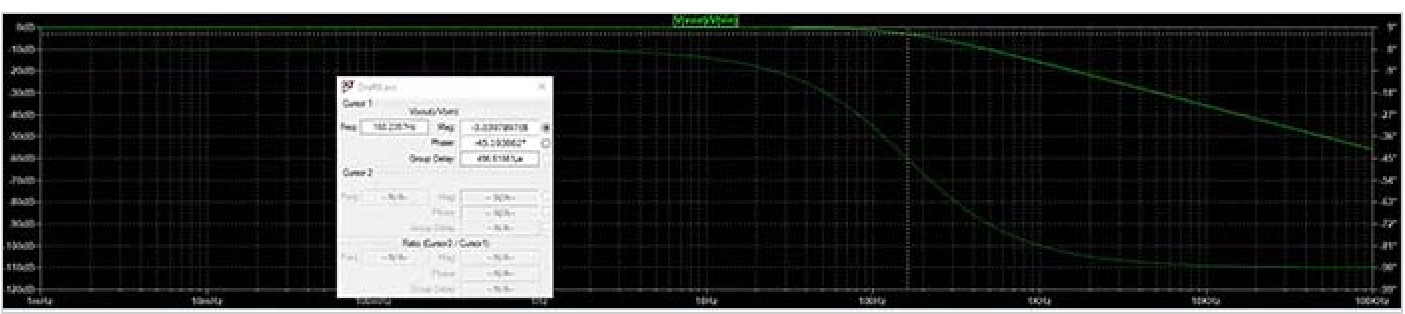
One of the most interesting analyses that LTspice can perform is frequency analysis, also known as AC analysis. We can see its capabilities with a simple low-pass filter.



For an AC analysis, we need to define one AC source. Various parameters can be configured for a voltage source in LTspice, but amplitude is enough for our purposes. We'll also configure four simulation parameters.

After running the simulation, we can plot the output level relative to the input, i.e., the filter transfer function. There are two lines: the continuous one corresponds to the magnitude response and the discontinuous one corresponds to the phase response. We can see that the filter response is essentially flat until it approaches the cut-off frequency:

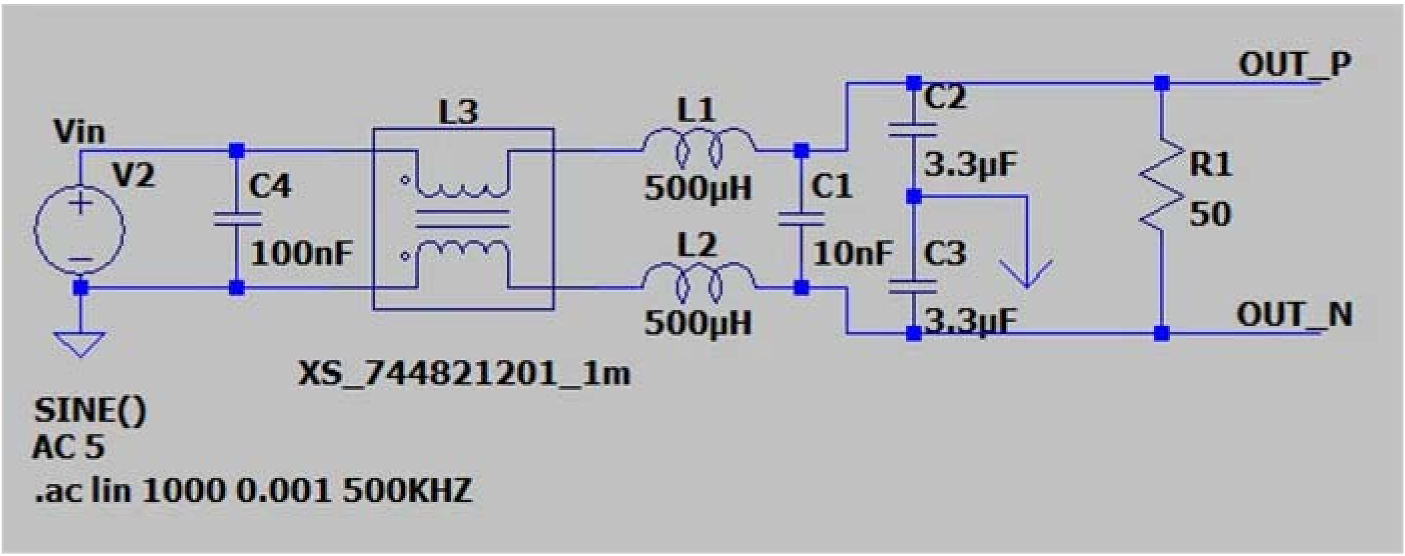
$$f_c = \frac{1}{2\pi * R * C} = \frac{1}{2\pi * 1000 * 10^{-6}} = 159.15Hz$$



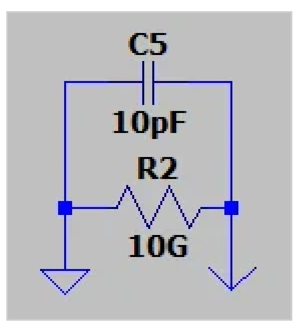
Simulating Noise and Filters in LTspice

It is important to clarify this aspect of LTspice. The software simulates noise that is intrinsic to components using the noise-type simulation. This type of simulation is suitable to verify the functionality of a circuit such as an analog filter. For EMC purposes, we need to simulate common-mode and differential-mode noise, so we can be sure that our filters are effective for filtering both noise types.

The following circuit is an EMC filter comprised of capacitors and a common-mode choke to attenuate common-mode noise (C2, C3, L3). It also includes two inductors and capacitors to attenuate the differential noise (L1, L2, C1, C4).

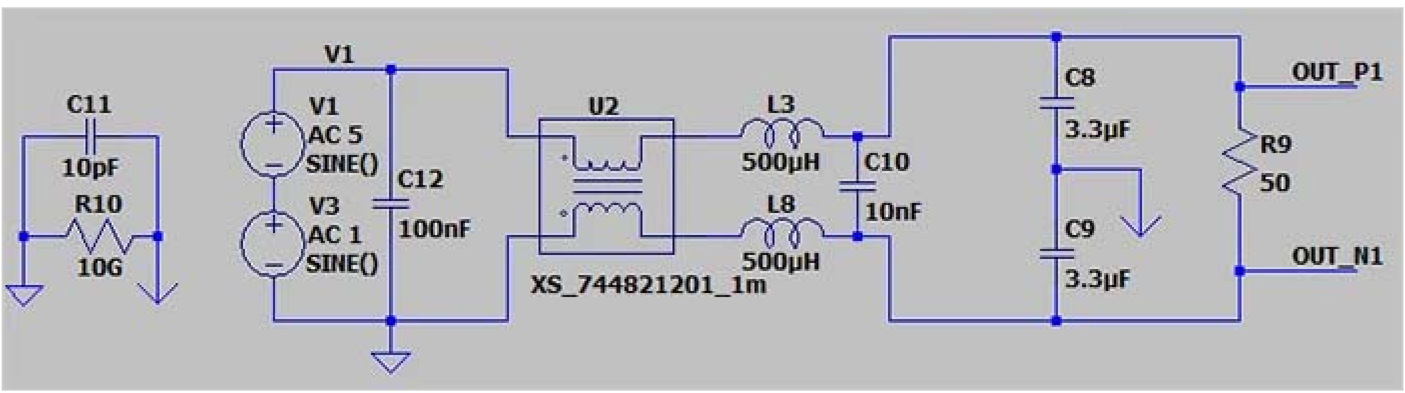


To simulate a protective earth (PE) or a chassis connection, we use a potential that is coupled through a very small capacitance to the regular ground or negative potential.

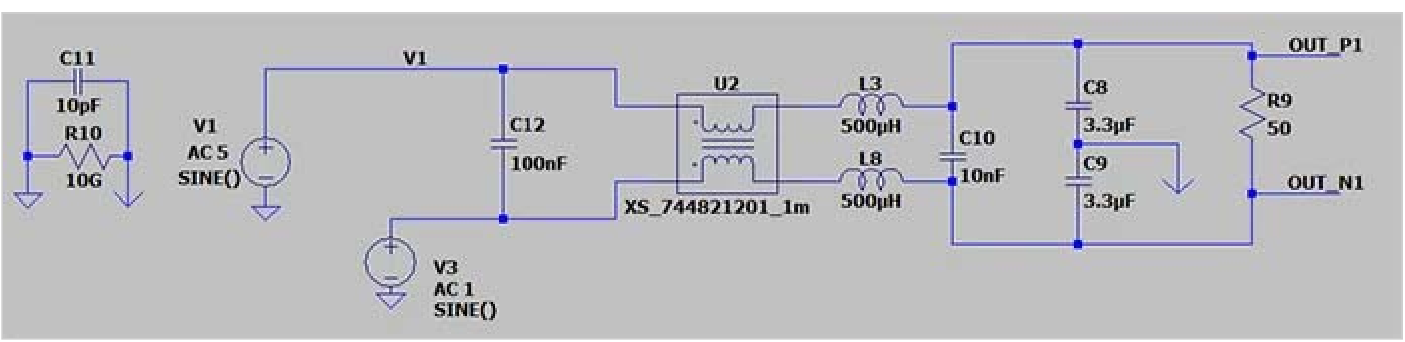


Scroll to continue with content

To simulate differential noise, we can superimpose a voltage source on the signal generator.



For the case of common-mode noise, it can be simulated by adding a voltage source to the return path.



A filter is adequate when both types of noise are sufficiently attenuated, so try to avoid focusing completely on one type of noise and neglecting the other.

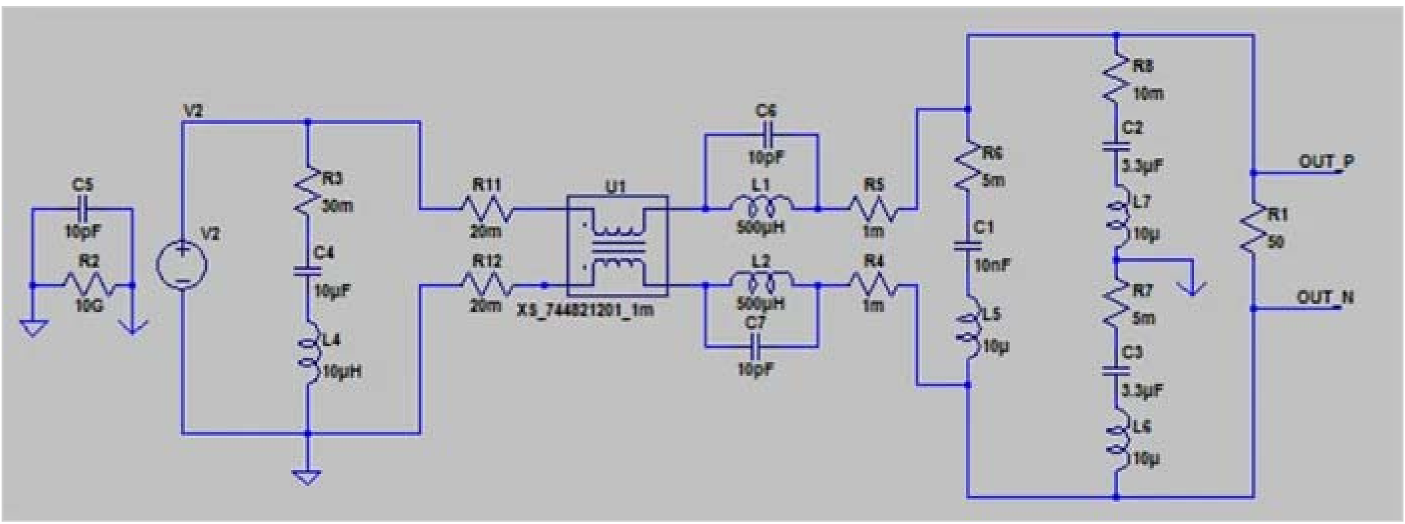
Ideal vs. Real EMC Filters

Unfortunately, real filters do not behave as well as ideal filters. If we want simulations that are close to real results, we need to take into account the parasitic elements of filters and of the board where they will be mounted.

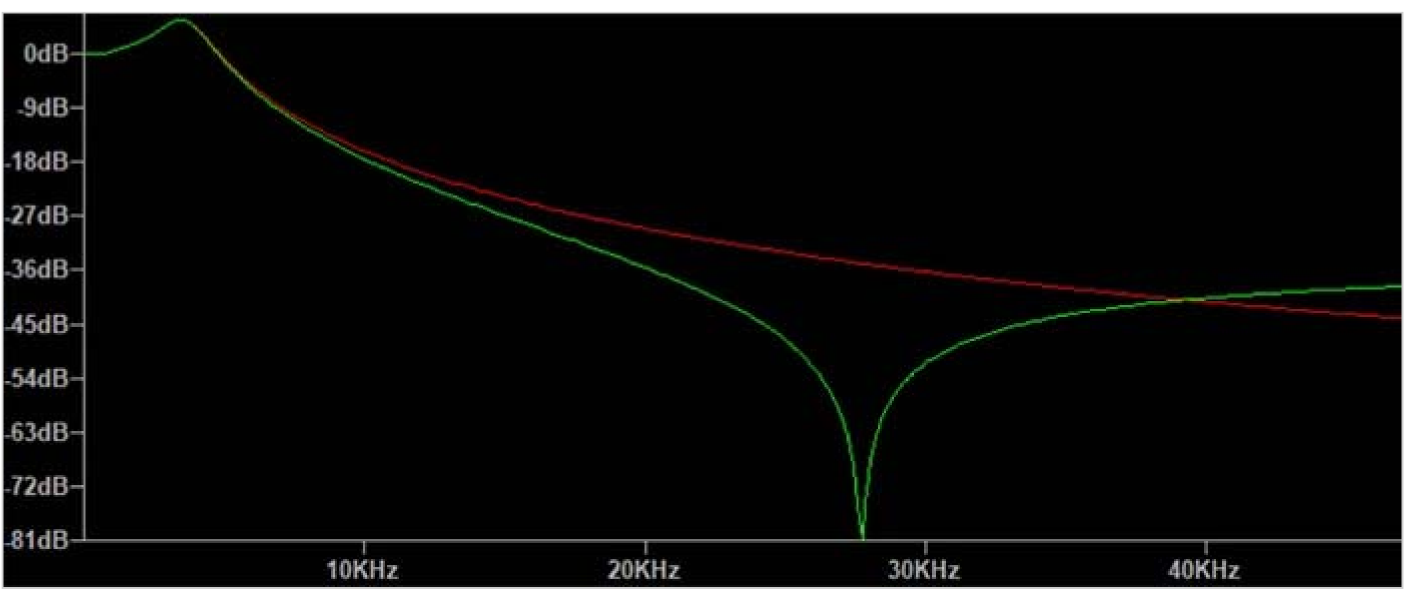
Parasitic components create resonances that can modify the cut-off frequency of an EMC filter. Therefore, if we do not consider parasitics, we might observe that adding a filter makes the situation worse.

Knowing the exact value of each parasitic element can be difficult. This information can sometimes be obtained from datasheets, but if necessary, estimated values can be used.

The following schematic shows the same EMC filter presented before with some parasitics added.



The comparison between the frequency response of the ideal filter (red) and of the real filter (green) is shown below.

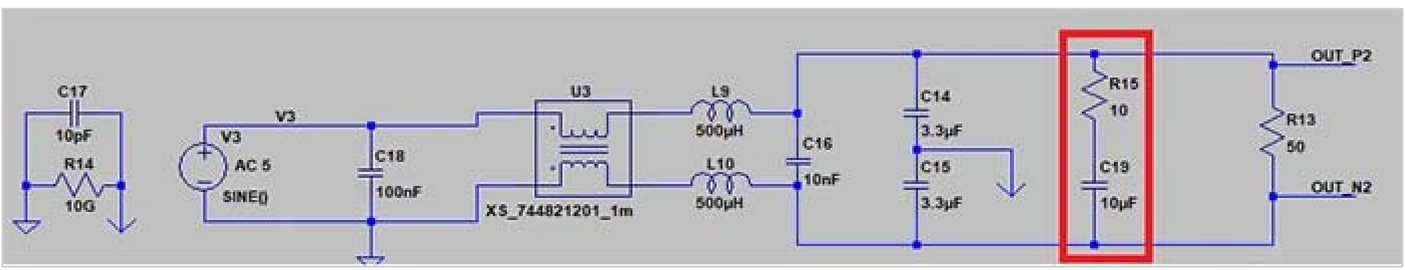


The frequency response of the ideal filter falls smoothly as frequency increases, but the real filter does not exhibit the same behavior. Starting at approximately 40 kHz, it produces less attenuation than the ideal filter. Hence, we can see that performance changes considerably when we approximate a real environment.

Note: when downloading LTspice models, you should double-check what is included in the models. Sometimes they already include all the parasitic elements, saving a lot of work.

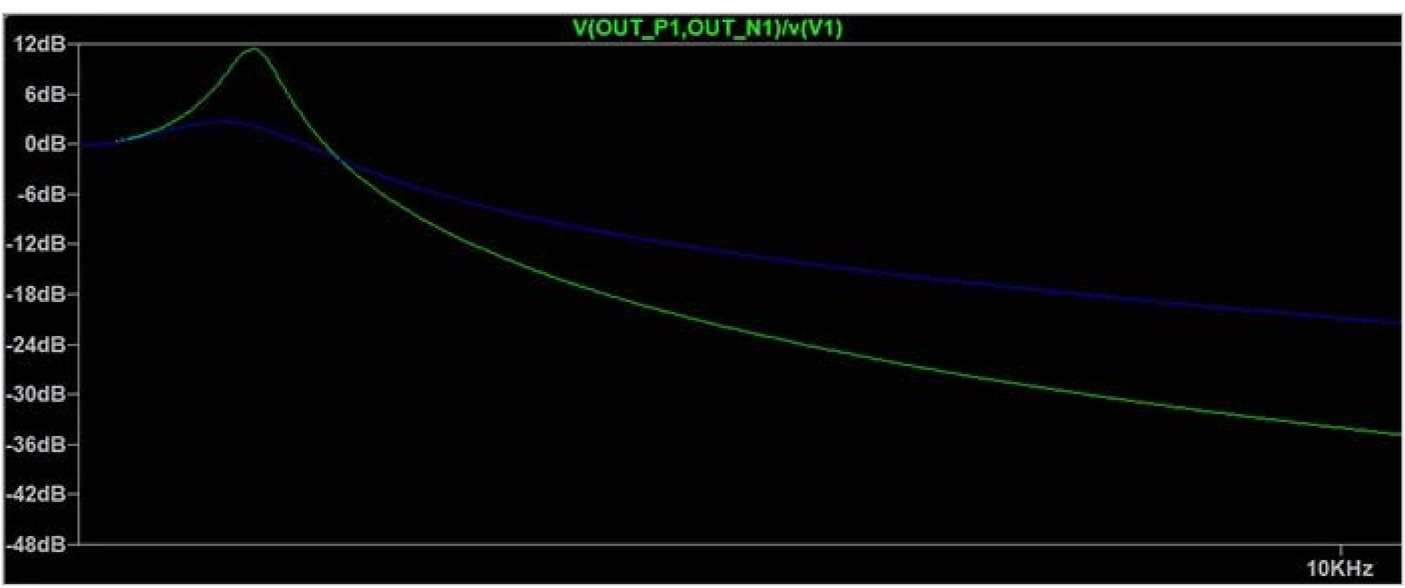
Self-Resonance and Damping

Self-resonances of filters can amplify noise by several dB, provoking an undesired effect. There are some methods to avoid this or at least reduce the negative impact as much as possible. One of them is quite simple and consists of adding one resistance in series with a capacitor as indicated in the diagram below.



The value of the resistor does not need to be huge. We only need to be careful about its power rating. An aspect of capacitors that is normally undesirable is the equivalent series resistance (ESR). However, in this case, we can select a capacitor with a high ESR and thereby avoid using an extra resistor.

The following figure shows the transfer function of a filter without damping resistance (green) and another one, corresponding to the filter with damping resistance (blue).



The difference between them is significant—the amplitude around the resonance frequency is reduced by 9 dB. The downside of this technique is that the filter slope is less pronounced, so all the frequency behavior has to be analyzed to ensure that all the results are acceptable.

Conclusion

LTspice is a powerful tool that saves cost and time in many applications. For instance, EMC filters need to be designed specifically for each application, so simulating them in advance saves a lot of time.

LTspice performs frequency analysis, which allows engineers to generate bode plots, the principal tool to study filters. LTspice can also include real parameters such as parasitics to obtain simulations that are as realistic as possible.

References

- Download LTspice
- LTSpice IV Getting Started Guide
- The LTSPICE IV Simulator, Reference Guide, Gilles Brocard. Würth Elektronik.