Simulation Methods for Electronic Circuits
Assignment on Contents 4

April 2020

Dr. J. E. Rayas-Sánchez

For the buffered 60-Hz band-reject notch filter illustrated below:

a) Simulate the circuit using WinSpice. Plot its AC voltage gain magnitude, $|A_v|$, from 20 Hz to 100 Hz, in linear scale. Use R_1 = 22 KΩ, R_2 = 48.6 KΩ, R_3 = 420 KΩ, R_4 = 200 Ω, R_5 = 4.8 KΩ, C_1 = C_2 = C_3 = 47nF, and Op-Amps LM324.

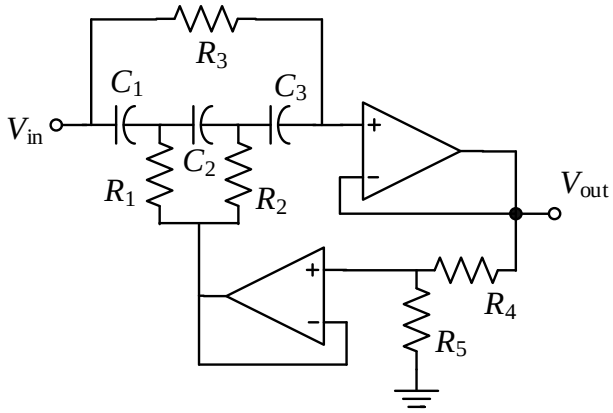
b) Develop a Matlab driver to run the SPICE simulations taking as variable parameters all the resistors (R_1 to R_5) and all the capacitors (C_1 to C_3).

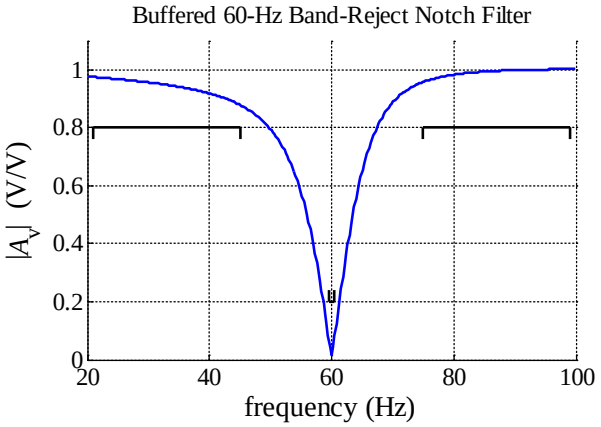
c) Using Matlab and WinSpice, perform a Monte Carlo statistical analysis considering the following tolerances: 2.5% for resistors, and 5% for capacitors. Assume Gaussian probability distribution functions (PDF) for all the parameters.

d) Using a reasonable number of outcomes (explain how you selected this number), perform a yield estimation for the circuit, considering the following design specifications:

$$|A_v| < 0.2 \text{ for } 59.5 \text{ Hz} \leq f \leq 60.5 \text{ Hz}$$

$$|A_v| > 0.8 \text{ for } f \leq 45 \text{ Hz and } f \geq 75 \text{ Hz}$$





Submission deadline: April 29, 2020.

Periférico Sur 8585
45604 Tlaquepaque, Jal., México
Tel +52 33 3669 3598
www.iteso.mx