

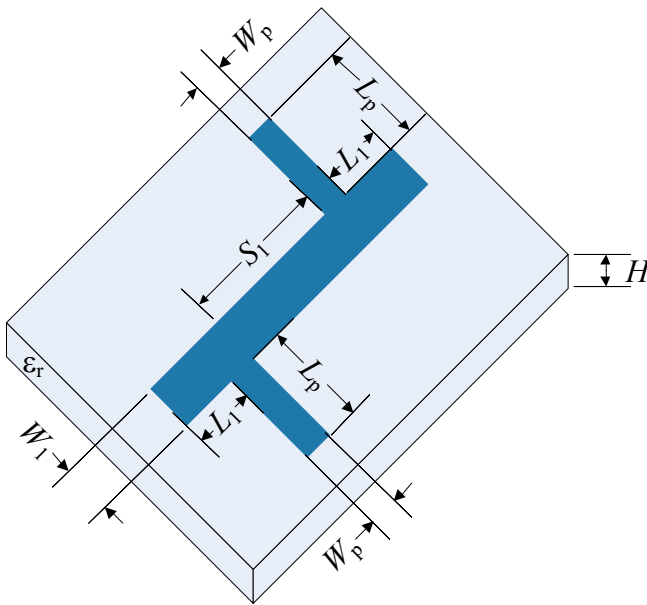


**Optimization-Based Modeling and Design of Electronic Circuits
Assignment 4**

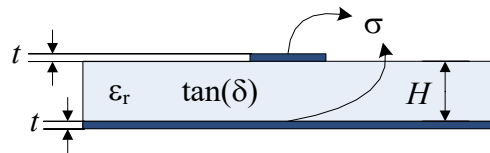
Prof. J. E. Rayas-Sánchez
March 2019

1. Circuit design by optimization

Consider the following microstrip low-pass filter [1,2]. The input-output 50-Ω microstrip lines have a width $W_p = 2.45$ mm, with a length $L_p = 5W_p = 12.25$ mm. These lines are symmetrically separated by a distance S_1 with a microstrip line whose width is W_1 . The separating microstrip line is terminated at both ends by an open stub whose length is L_1 .



The microstrip structure is on a substrate with thickness $H = 0.794$ mm, relative dielectric permittivity $\epsilon_r = 2.2$, and loss tangent $\tan(\delta) = 0.01$. All metals use half-ounce copper with electrical conductivity $\sigma = 5.8 \times 10^7$ S/m and thickness $t = 0.6$ mil = 15.24 μm.



The design specifications for this filter are:

$$|S_{21}| > 0.9 \text{ for } 0.2 \text{ GHz} \leq f \leq 6 \text{ GHz}$$

$$|S_{21}| < 0.1 \text{ for } 8 \text{ GHz} \leq f \leq 10 \text{ GHz}$$

Formulate an optimization problem for this filter to satisfy the previous design specifications. Use as optimization variables the following parameters $\mathbf{x} = [W_1 \ L_1 \ S_1]^T$, keeping fixed the following pre-assigned parameters $\mathbf{z} = [H \ \epsilon_r \ W_p \ L_p \ \tan(\delta) \ \sigma \ t]^T$.

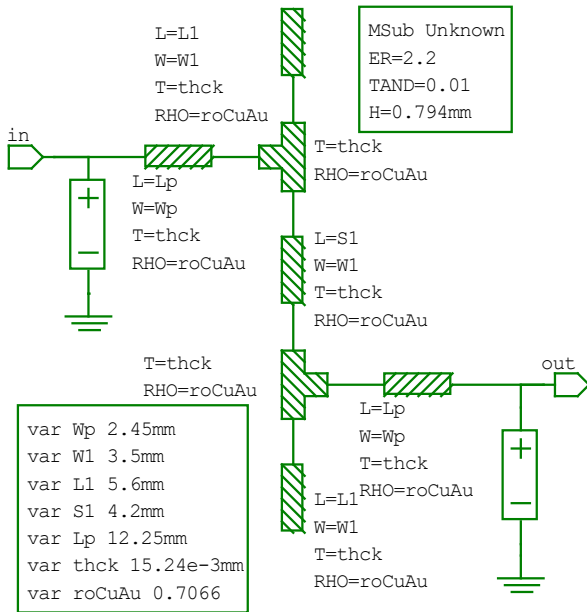
As starting points for optimization, consider the following initial designs:

$$W_1 = 3.5 \text{ mm}, L_1 = 5.6 \text{ mm}, S_1 = 4.2 \text{ mm}$$

$$W_1 = 6.7 \text{ mm}, L_1 = 5.2 \text{ mm}, S_1 = 5.8 \text{ mm}$$

$$W_1 = 1.9 \text{ mm}, L_1 = 4.1 \text{ mm}, S_1 = 3.3 \text{ mm}$$

This filter is implemented in the high-frequency circuit simulator APLAC, as illustrated in the following page.



APLAC file:

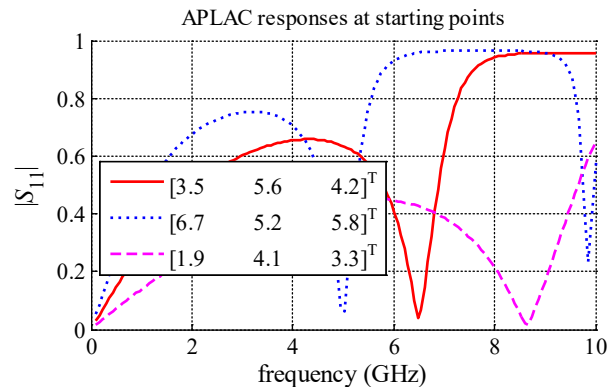
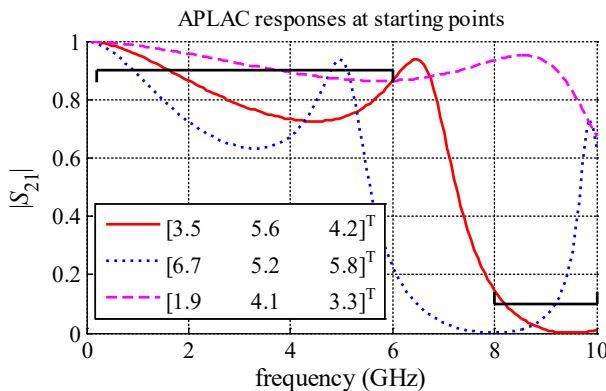
```

$-----
var Wp 2.45mm
var W1 3.5mm
var L1 5.6mm
var S1 4.2mm
var Lp 12.25mm
var thck 15.24e-3mm
var roCuAu 0.7066
MSub Unknown
+ ER=2.2 TAND=0.01 H=0.794mm
Mlin Mlin2 node1 node2
+ L=S1 W=W1 T=thck RHO=roCuAu
Mloc Mloc1 node3
+ L=L1 W=W1 T=thck RHO=roCuAu
Mloc Mloc2 node4
+ L=L1 W=W1 T=thck RHO=roCuAu
Mlin Mlin3 node6 out
+ L=Lp W=Wp T=thck RHO=roCuAu
Mlin Mlin1 in node5
+ L=Lp W=Wp T=thck RHO=roCuAu
Mtee Mtee1 node3 node1 node5
+ T=thck RHO=roCuAu
Mtee Mtee2 node4 node2 node6
+ T=thck RHO=roCuAu
DefNPort nport 2 in GND 50 out GND 50
Sweep "mcsLPF - Circuit Simulation"
+ LOOP 300 FREQ LIN 1KHz 10GHz
+ WINDOW=0 grid Y "S11" "" 0 1
+ WINDOW=1 grid Y "S21" "" 0 1
Show W=0 Y=Mag(S(1,1))
Show W=1 Y=Mag(S(2,1))
EndSweep

```

APLAC uses parameter RHO to define the normalized resistivity of the metal (in this case $\rho_{Cu} = 1/\sigma_{Cu} = 17.2414 \text{ n-}\Omega\text{-m}$) with respect to the resistivity of gold ($\rho_{Au} = 24.4 \text{ n-}\Omega\text{-m}$). Then $RHO = 17.2414/24.4 = 0.7066$ for this particular problem.

APLAC responses at the initial designs are:

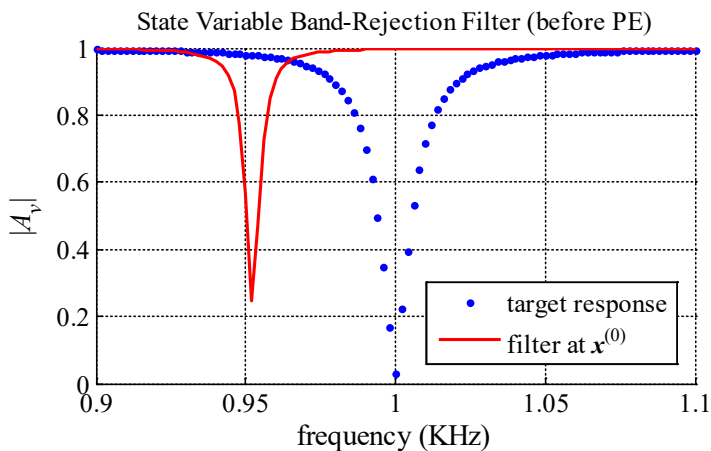
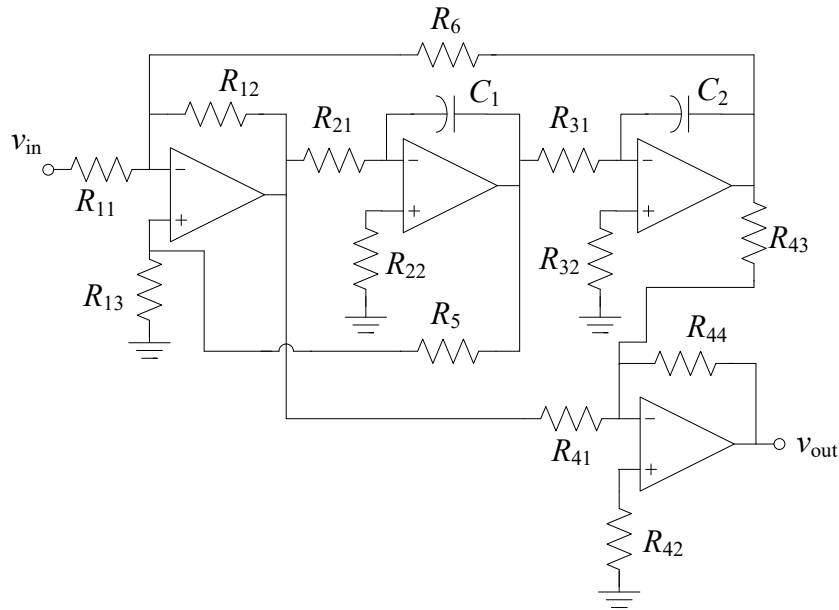


Write down a Matlab script to drive the corresponding APLAC file. Solve the optimization problem using your implementation of the Conjugate Gradient algorithm, your Quasi-Newton algorithm, and the Nelder-mead method available in Matlab (`fminsearch`). Compare results obtained by each algorithm.



2. Circuit parameter extraction by optimization

Consider the following state-variable band-rejection filter [3]. The filter is using Op-Amps $\mu A741$ and is simulated in WinSpice. We want to extract the parameters values for all the resistors and capacitors of the filter such that the magnitude of its voltage gain is as close as possible to the target response shown below.



The initial filter parameters $\mathbf{x}^{(0)}$ are:

- $R_{11} = 20 \text{ K}\Omega$
- $R_{12} = 20 \text{ K}\Omega$
- $R_{13} = 20 \text{ K}\Omega$
- $R_{21} = 20 \text{ K}\Omega$
- $R_{22} = 20 \text{ K}\Omega$
- $R_{31} = 22 \text{ K}\Omega$
- $R_{32} = 20 \text{ K}\Omega$
- $R_{41} = 10 \text{ K}\Omega$
- $R_{42} = 3.3 \text{ K}\Omega$
- $R_{43} = 10 \text{ K}\Omega$
- $R_{44} = 10 \text{ K}\Omega$
- $R_5 = 5.1 \text{ M}\Omega$
- $R_6 = 20 \text{ K}\Omega$
- $C_1 = 7.96 \text{ nF}$
- $C_2 = 7.96 \text{ nF}$

The filter response at the starting point (before parameter extraction) is also shown in the figure above.

Write down a Matlab script to drive the corresponding WinSpice file.

Formulate and solve an optimization problem to extract the filter parameters that make $|A_v|$ as close as possible to the target response. Compare your results using a Manhattan, Euclidean and Chebyshev objective function. To minimize the objective function, use your implementation of the Conjugate Gradient algorithm, your Quasi-Newton algorithm, or the Nelder-mead method available in Matlab (`fminsearch`).



Explore the effects of varying the number of frequency points used in the target response (use text data files provided in the course website).

Include in your report the references and conclusions.

Submission deadline: April 12, 2019.

-
- [1] D. M. Sheen, S. M. Ali, M. D. Abouzahra and J. A. Kong, "Application of the three-dimensional finite-difference time-domain method to the analysis of planar microstrip circuits," *IEEE Trans. Microwave Theory Tech.*, vol. 38, pp. 849-857, July 1990.
 - [2] G. D'Inzeo, F. Giannini, and R. Sorrentino, "Novel microwave integrated low-pass filters," *Electron. Lett.*, vol. 15, no. 9, pp. 258-260, Apr. 26, 1979.
 - [3] W. D. Stanley, *Operational Amplifiers with Linear Integrated Circuits*. Columbus, OH: Merrill, 1989, pp. 373-381.