High-Frequency Circuits

Lab Experiment 2:
High-Frequency Filter Design

Prof. J. E. Rayas-Sánchez
January-May 2020

Objectives

The objectives of this lab experiment are:

a) to apply the insertion loss method to design a lumped prototype of a low-pass filter starting from its design specifications

b) to implement an ideal lumped prototype of a low-pass filter as a realistic microstrip filter with open stubs using Richard's transformation, Kuroda's identities and Gupta's formulas

c) to observe the performance and limitations of these microstrip filters using a high-frequency circuit simulator and a full-wave electromagnetic (EM) simulator

d) to get more familiar with the main characteristics of the high-frequency circuit simulator Keysight ADS

e) to get familiar with the main characteristics of the full-wave electromagnetic simulator Sonnet.

Components and Instrumentation

A numerical software tool: Matlab, Octave, Scilab, or something similar. The high-frequency circuit simulator Keysight ADS. The full-wave EM simulator Sonnet Lite.

Theoretical and Simulation Procedure

1. Using the insertion loss method, design a low-pass filter prototype with the following specifications: a) cutoff frequency at 6 GHz; b) minimum insertion loss of 30 dB at 10.2 GHz; c) Chebyshev response with ripple of 0.5 dB; d) reference impedance of 50 Ω.
2. Implement the filter prototype with ideal lumped components. Simulate the filter in ADS, plotting $|S_{11}|$ and $|S_{21}|$ from 10 MHz to 12 GHz (linear frequency sweep) in dB as well as in linear scale. Check that the filter satisfies all the design specifications.
3. Using Richard's transformation, implement the filter prototype with ideal lossless transmission lines on air ($\varepsilon_r = 1$). Using Kuroda's identities, transform the filter such that only shunt-connected open stubs are used. Simulate the filter in ADS, plotting $|S_{11}|$ and $|S_{21}|$ from 10 MHz to 12 GHz (linear frequency sweep) in dB as well as in linear scale. Check that the filter satisfies all the design specifications.
4. Neglecting losses and metal thickness, implement the previous filter in microstrip technology using Gupta's formulas [1]. All microstrip lines are on a dielectric substrate with relative dielectric permittivity $\varepsilon_r = 2.2$. The substrate height is $H = 0.794$ mm. Simulate the filter in ADS, plotting $|S_{11}|$ and $|S_{21}|$ from 10 MHz to 12 GHz (linear frequency sweep) in dB as well as in linear scale.
5. Simulate again the previous filter in ADS (using microstrips), but now considering that the loss tangent

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

of the dielectric substrate is tan $\delta = 0.005$, and that all metallic traces (including the ground plane) use copper with conductivity $\sigma_{Cu} = 5.8\times10^7$ S/m and metal thickness $t = 0.65$ mil (half once copper). Plot $|S_{11}|$ and $|S_{21}|$ from 10 MHz to 12 GHz in dB as well as in linear scale.

6. Implement the previous microstrip filter in Sonnet Lite, considering dielectric losses (tan $\delta = 0.005$), metallic losses ($\sigma_{Cu} = 5.8\times10^7$ S/m), and metal thickness $t = 0.65$ mil (half once copper). Select a grid resolution (cell sizes) suitable for simulation in Sonnet Lite (you might need to round the dimensions of the original microstrip traces). Make sure Sonnet's box has adequate dimensions to avoid box resonances in the simulated frequency band. Simulate this filter in Sonnet Lite, plotting $|S_{11}|$ and $|S_{21}|$ from 100 MHz to 12 GHz in dB as well as in linear scale. Compare in the same figure ADS's responses with microstrip lines (from Step 5) and Sonnet's responses (from this step), for both $|S_{11}|$ and $|S_{21}|$ (in two separated figures).

Report

Write a report including all the theoretical calculations, simulation procedures, the required plots, as well as your comments related to the results you obtained, your general conclusions and references. Include in your report the corresponding ADS schematics and Sonnet layout for each circuit you implemented.

Deadline and Assessment

The deadline for submitting the report via email (erayas@iteso.mx) is on Monday May 18, 2020 at 10 AM. The report can be written either in English or in Spanish.

This lab experiment can be realized in teams of up to 4 students. The evaluation of the report will be as follows:

Quality of the report	30%
Accuracy of the theoretical analysis	30%
Simulation procedures	40%

If the report is written in acceptable English, extra points of up to 5% can be granted on the evaluation.

References

[1] K. C. Gupta, R. Garg and R. Chadha, *Computer-Aided Design of Microwave Circuits*. Norwood, MA: Artech, 1981.