Practical Session 3

RF Filter Design Practical Session

Aim

The aim of this practical is to provide some experience in the design and optimisation of RF filters using a commercial RF simulation package like Microwave Office.

Task

Using Microwave Office, design a 3 resonator hairpin microstripline filter to have a centre frequency of 1 GHz and a bandwidth with -3 dB attenuation, relative to the insertion loss, of 50 MHz. The filter is to use RO4003 substrate with 1 oz (35 μ) copper and 0.8128 mm thickness. The input and output impedances are to be 50 Ω. Since the filter is to have a flat passband, a Butterworth filter prototype can be used for the initial design.

Procedure

1. Use the filter tables or equations from the lecture notes to determine the K and Q values for the filter.
2. Either: Use equations 11 to 13 in the RF filter lecture notes to calculate Z_{oo} and Z_{oe} for each of the line sections. (Note a calculator can be used or an Excel spreadsheet can be used for the calculations). Use the TxLine program to calculate the required line widths, coupling gaps and resonator lengths.

   Or: Use the design procedure of Figure 52 and 53 in Chapter 7 of the RF_Electronics Lecture notes to determine the required tapping points, coupling gaps and resonator lengths.
3. Realise the schematic for the microstrip filter, observe the resulting layout.
4. Plot the frequency response. Write down the line lengths, track widths and coupling gaps of the resonators. Observe the PCB layout. (Use “select all” and “snap all together” to produce a useable layout.)
5. Insert the bends to change the filter into a hairpin filter. A 3 mm radius bend is fine. If needed the minimum radius can be determined by considering the coupling between two coupled lines as the spacing is varied and ensuring that the coupling is less than -60 dB. Manually tune the resonator line lengths to obtain the correct centre frequency. (If needed the initial values can be restored using the written down values in step 5). Observe the PCB layout.
6. Set up the optimisation goals to satisfy the filter specifications. (S21 < -0.5 dB attenuation over a 40 MHz bandwidth, S21 < -3 dB attenuation over a 50 MHz bandwidth and S21 > -3 dB attenuation outside this frequency range). For the stopband attenuation the use of a sloped optimisation goal is recommended to ensure the correct passband attenuation shape, without spurious resonances.
7. Optimise the filter to meet the specifications. (Different optimisation methods may need to be used.) Observe the PCB layout.

8. Make a note of the frequency response at the harmonics of the passband. Observe the PCB layout.
9. Change the layout to ensure that the input and output can be connected to a BNC PCB connector soldered to the edge of the PCB. (This requires a 3.5 mm spacing between the centre pin and the two ground pins as shown in figure 40 on page 6.24).
10. Place RF shielding on the PCB layout (See figure 40, on page 6.24) and place the BNC connectors so that the filter can be housed in an enclosure.
11. Export the layout as a .dxf file for import into a drawing program or into Protel and as a Gerber plot file .ger, for importing into photo plotters or our PCB milling machine software, ProtoMat.

Note.

Not all students will be able to complete all the steps in the time available for the practical session. These steps do however give a good guide to follow in designing a filter.

Equipment.

Computer with Microwave Office Software.

Further details: please email Keith.Kikkert@jcu.edu.au.