EXPERIMENT 8
LUMPED-ELEMENT FILTER DESIGN

OBJECTIVE:

Use Agilent *Genesys* to design a lumped-element microstrip low-pass filter from given specifications. Take the design from theory to a ready-to-implement practical design, with realistic parts and a fabrication-ready microstrip layout. Test an example LPF board on the Network Analyzer.

DESIGN:

1) Design a low pass filter to conform to the following specifications:

F_C (cutoff frequency): 500 MHz (@ -3dB)
Attenuation: >10dB at 725 MHz
Z_S (source impedance): 50 Ohms
Z_L (load impedance): 50 Ohms
Response: **Equal Ripple (Chebyshev)**

It should be a **third-order** design, with two shunt (grounded) capacitors connected by a series inductor, as shown in Fig. 1.

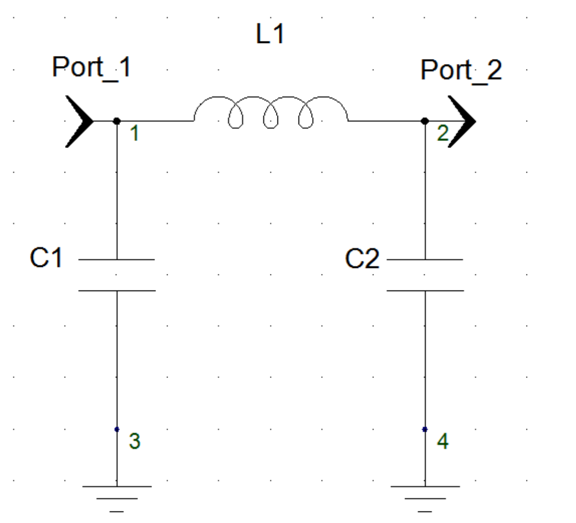

Fig. 1: Basic LPF Schematic

Use the option in *Genesys* to Synthesize a Passive Filter upon opening the program (use Factory Default Values), and enter the specs into the Filter Properties window.

2) Check your response as shown in the Response window to see if it is close to the specs. If not, check your work, as *Genesys* should get you close just by entering the data into the Filter Properties window. Save a screenshot of the output.

3) The pre-made boards you will be testing use a **15nH series inductor** and two **8.2pF shunt capacitors**. Change the values in the schematic to reflect this, and see how the output changes. Again, save a screenshot.

4) Create a new folder in the workspace tree and create a new schematic within that. Design a practical microstrip layout using TLINE transmission lines, and insert a series inductor and two shunt capacitors from the "Johanson Tech" library (these are the "lumped elements"). Look up the parts on the Johanson website and/or datasheets. Pick a standard substrate (RT/duroid 5880, for example). Don't forget to add the input and output ports, and ground the shunt capacitors.

5) You will want 50Ω impedances on the input/output ports and transmission lines, but this will create transmission lines that are too thick for the lumped elements. So, you will need two thicknesses of transmission lines: thick (50Ω) lines on the input and output, and thin lines (approximately 1mm) to connect the parts to the input/output lines and to each other. (You can change the actual thickness of the thin lines AFTER you have converted to Advanced TLINE in Step 6).

6) Convert the schematic into Advanced TLINE, using the substrate you added in Step 4. Make sure it adds "discos" in between the two types of transmission lines. Your final schematic should look similar to Fig. 2 (it doesn't have to be exactly the same, especially regarding the tapered discos).

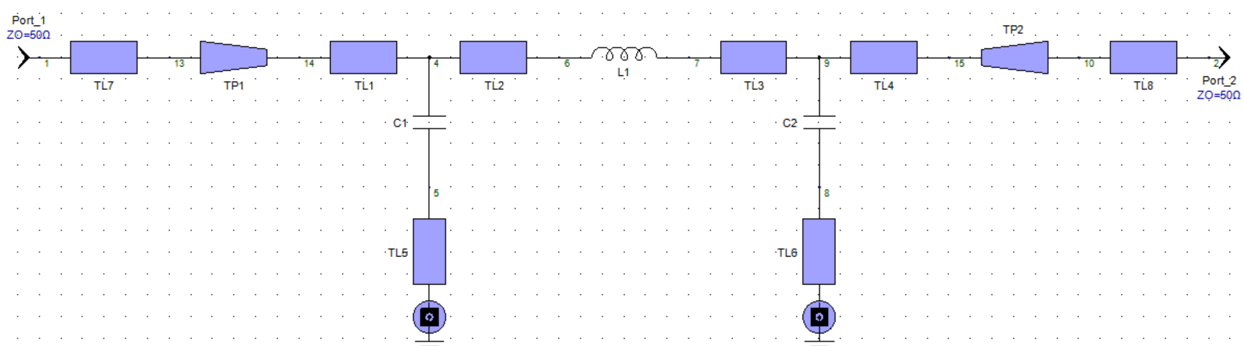

Fig 2: Example LPF Advanced TLINE Schematic

7) Create and run a new linear simulation based on this Advanced TLINE schematic. Save a screenshot of this output.

8) Once you have an acceptable design, prepare it for fabrication. You will need to add a Layout to your new schematic and align the parts. See Fig. 3 for an example (again, it does not have to be exactly the same). Use this to create a Gerber file. Capture a screenshot of the final layout and the Gerber Export Viewer to prove your design is ready for fabrication.

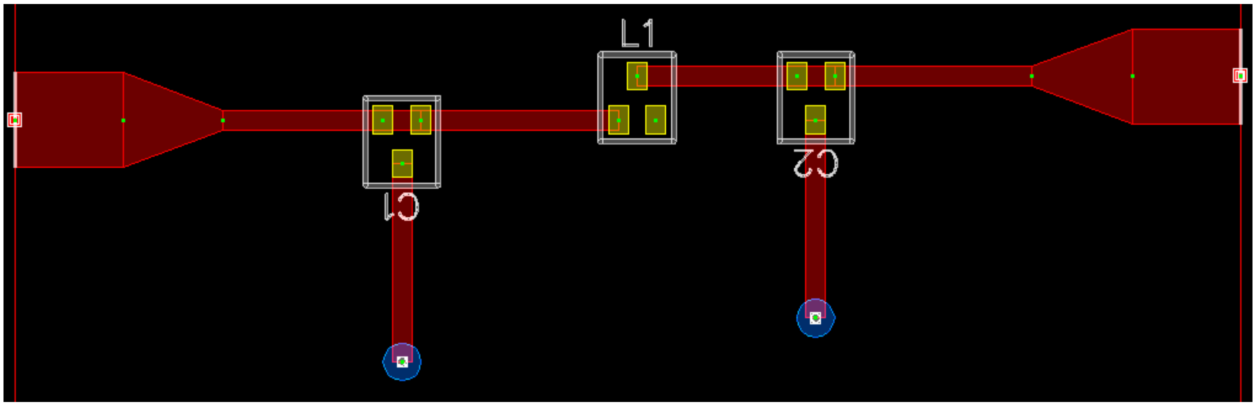

Fig. 3: Example LPF Layout

9) Use the Network Analyzer to test the pre-made board you are given. Show the S_{21} response, and place markers at the following locations:

a) -3 dB point
b) 500 MHz point
c) -10 dB point
d) 725 MHz point

10) Save a plot of this screen for your report.

REPORT:

In your lab report, describe all steps of your design, including screenshots of your schematics, simulations, layouts, Gerber layout, and Network Analyzer output. Include a brief mathematical explanation on the basics of designing this filter as outlined in *Pozar*.

Be sure to specify your final parameters (capacitor and inductor parts, line lengths, etc.).

Discuss possible further improvements.

Thanks to Tim Lindgren for the foundation of this experiment, and for the LPF filter boards.