

ADS Tutorial #4 Filter Design

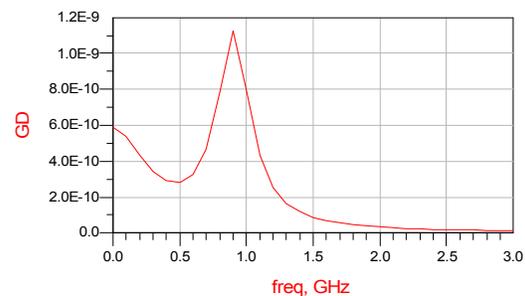
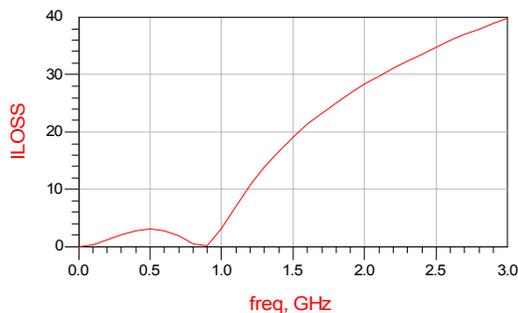
Oklahoma State University

Part I Synthesis Filters with Lumped Circuit Elements

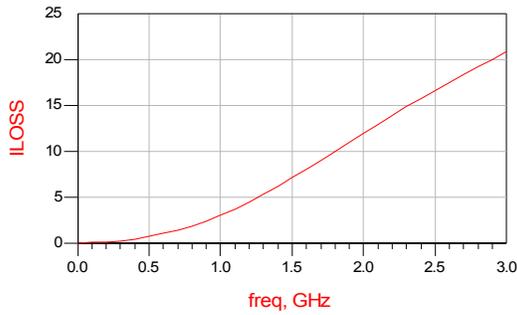
From the top menu click “tool”, and then select “E-Syn” to start it. Click on the “Select Type” button on the top right corner, a menu will pop up and we can select the type of filters. Suppose we need to synthesis 3rd order Chebyshev LPF, and then we can fill the parameter table. Next we click the button for synthesis, and a small window will pop up. Click on the “synthesis” button on the lower left corner, two designs are available, one is CLC circuit and the other is the LCL circuit, where the value of the components are available:

- Design I --- LCL circuit
Input Term: Resistive R = 50 Ohm
IND SE L = 26.64839 nH
CAP PG C = 2.265412 pF
IND SE L = 26.64839 nH
Output Term: Resistive R = 50 Ohm
- Design II --- CLC circuit
Input Term: Resistive R = 50 Ohm
CAP PG C = 10.65936 pF
IND SE L = 5.663531 nH
CAP PG C = 10.65935 pF
Output Term: Resistive R = 50 Ohm

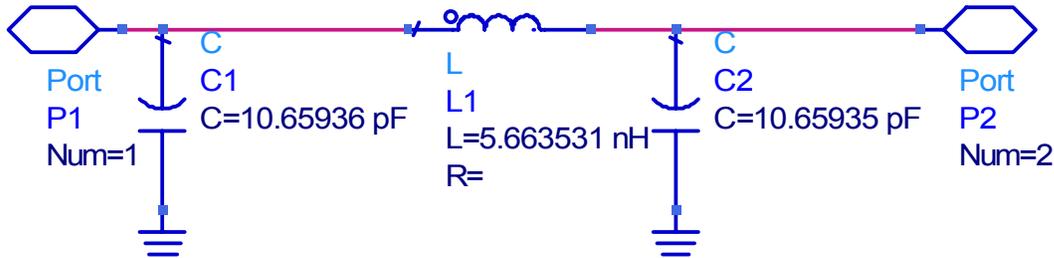
And then one can simulate the design by clicking the “Analyze” button, which is the top one on the right column in the same small window. Another window will pop up, where we can input the simulation parameters. The simulation result can be shown by click the “Data Display” button on top of the large menu, which is the same as the one we used previously. The simulated insertion loss and group delay are show in the following.



For comparison, the Bessel (linear phase) LPF is synthesized and simulated in the same way, the results are shown in the following:



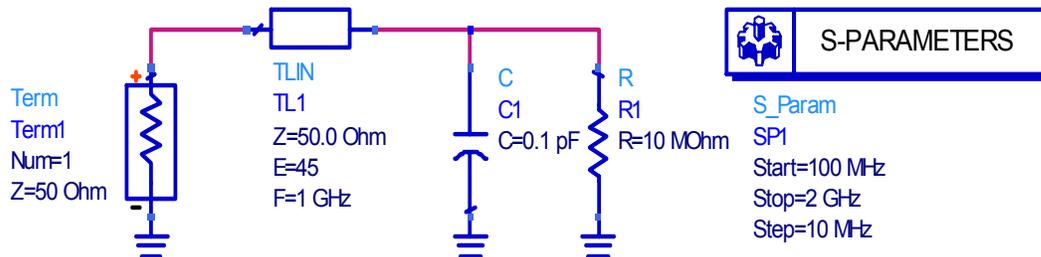
We can generate a customized system level filter, in the schematic design window click *Tool > E-Syn > Place New Design for Synthesized Network*, after we input the name of it, a filter symbol will appear on the schematic design sheet. One can check its subcircuit by clicking the icon with a downward arrow. In the future, we can use it as a system level part, which can be found in the “library”; we can have a check, and put another copy on it onto the design sheet.

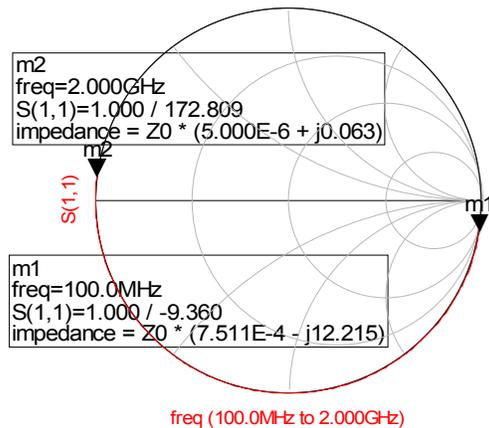


Part II Microstrip Circuit

1. Ideal Transmission Line

The components of the ideal transmission line can be found in the component pallet “Tlines-Ideal”. The following is the simulation of an “open” terminated T-line implemented with microstrip circuit. The length of the T-line is specified by the “electrical length” (the number is the angle in degree) and the frequency. As it is a one port network, one can only get S_{11} (or Γ) in the S-parameter simulation. As the resistance is very high, the termination is equivalent to a pure capacitor, which is reflected in the simulation result. It can also be found that the input impedance looks like a capacitor both at low range, but it becomes an inductor in the high frequency range. Students can change the electrical length to 60 degree, and simulate the circuit again.





2. Example: High-Z/Low-Z Filter

The following circuit is a high-Z/low-Z filter, which means the filter is composed of interchanging high-impedance segment and low-impedance segment of T-lines. This structure is similar to the antireflection coating in optics. The simulation shows it is a low-pass filter.



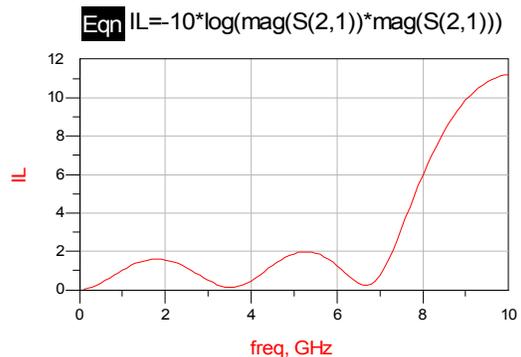
MSub

MSUB
 MSub1
 H=0.76 mm
 Er=3
 Mur=1
 Cond=6.0E+7
 Hu=1.0E+6
 T=50 um
 TanD=0.002
 Rough=5 um

S-PARAMETERS

S_Param
 SP1
 Start=0.1 GHz
 Stop=10.0 GHz
 Step=0.1 GHz

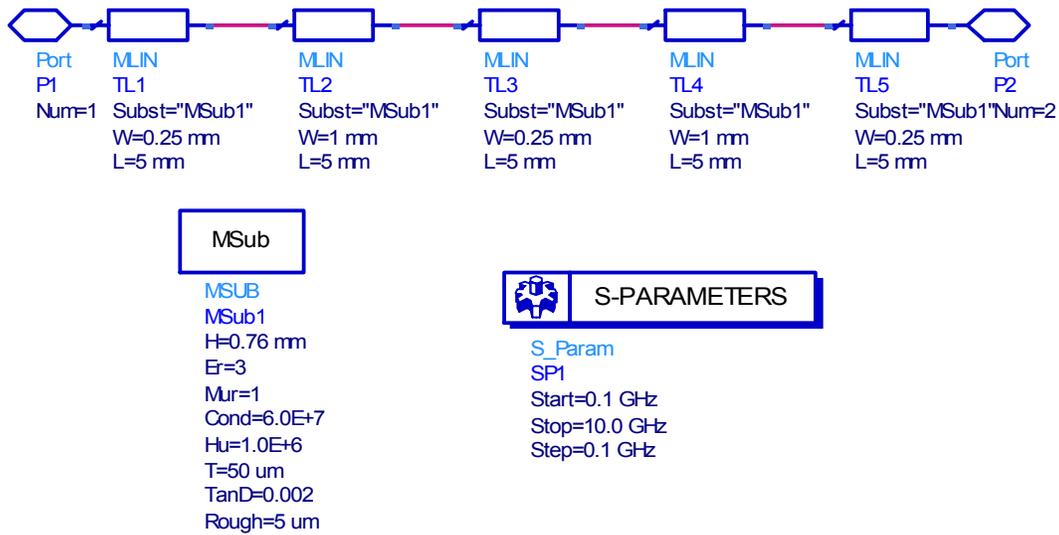
Cond = 4.1E+7



3. Layout and Momentum Simulation

The simulation result shown above is usually inaccurate, one issue is the parasitic elements at the interconnection points between these high- and low-z segments, and another issue is the radiation loss. Therefore, one needs more accurate way to simulate the real microstrip filter. One efficient numerical simulation method is called “momentum method”. As we know the simulation of a real microstrip structure involves solving Maxwell’s equations, which is a partial differential equation in both space and time. In this problem, one can use harmonic function to remove the time dependence. The space-related differential equation can be converted to algebraic equation by means of Fourier transform. In quantum mechanics, the derivative of time corresponds to energy, and the derivative of space corresponds to momentum.

The layout of the filter can be converted from the schematic circuit, which is an advantage if we compared to VLSI design with Cadence. However, before making the conversion we need to add two ports to both ends, icon of the port is just on the left hand side of the “ground”.

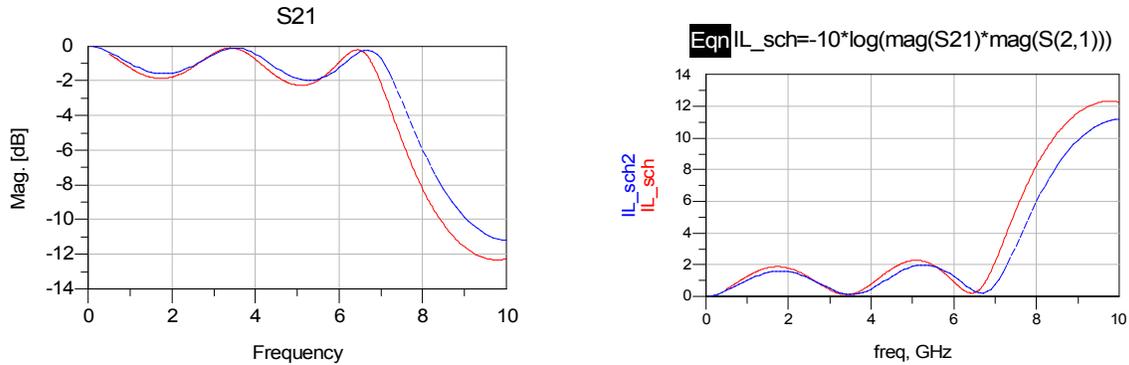


After the schematic circuit has been modified, we can click “Layout” on top of the schematic window and select “Generate/Update Layout”. A black layout window will pop up, and a small window will ask us to choose which port to start with. With this question answered, the layout of the filter will be shown in the layout window.



However, this step has just generated the top view of the filter, i.e. the information of “L” and “W”, so we also need to specify the substrate information separately. A convenient way is to transfer the substrate information from the schematic sheet to this layout sheet, this can be done by click the “momentum” button on the top menu in the layout window and select “Substrate” > “Update from schematic”. Before we do the simulation, we need to set up the mesh, which is an

important step for numerical simulation. If the mesh is too sparse, the precision is very low; on the other hand, if the mesh is too dense, it would take very long time to simulate. The following are the simulation results, which is compared to those from schematic simulation (dotted).

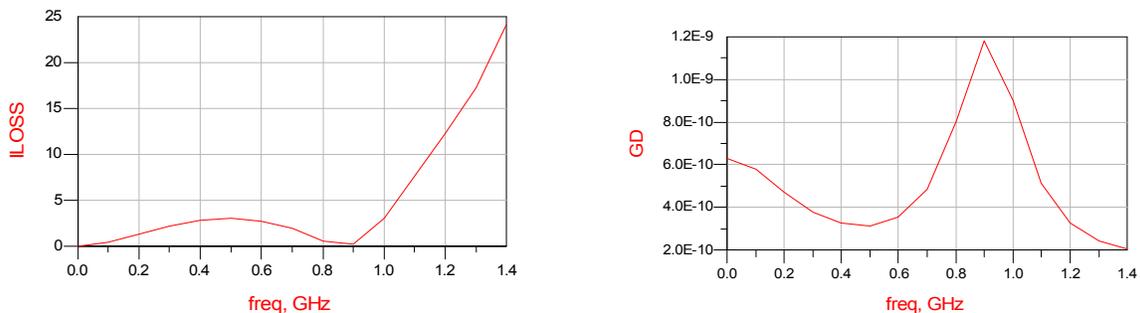


Part III Synthesis Filters with Distributed Circuit

Following basically the same procedure in Part II, one can synthesize the distributed filter; the only difference is to choose “Distributed” instead of “Lumped”. If we still use the Chebyshev LPF as the example, one of the synthesized filters has the following parameters:

Input Term: Resistive R = 50 Ohm
 TLIN SE Z = 166.7548 Ohm E = 60.00000 deg F = 1.000000 GHz
 TLOC PG Z = 160.9265 Ohm E = 60.00000 deg F = 1.000000 GHz
 TLIN SE Z = 166.7548 Ohm E = 60.00000 deg F = 1.000000 GHz
 Output Term: Resistive R = 50 Ohm

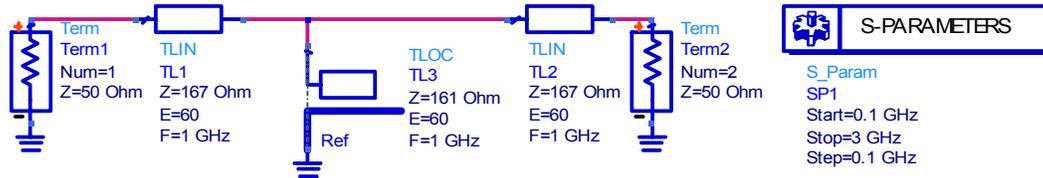
One needs to choose the electrical length judiciously, so that the impedances fall in the range of $10\Omega < Z < 200\Omega$. The simulated insertion loss and group delay are shown below:



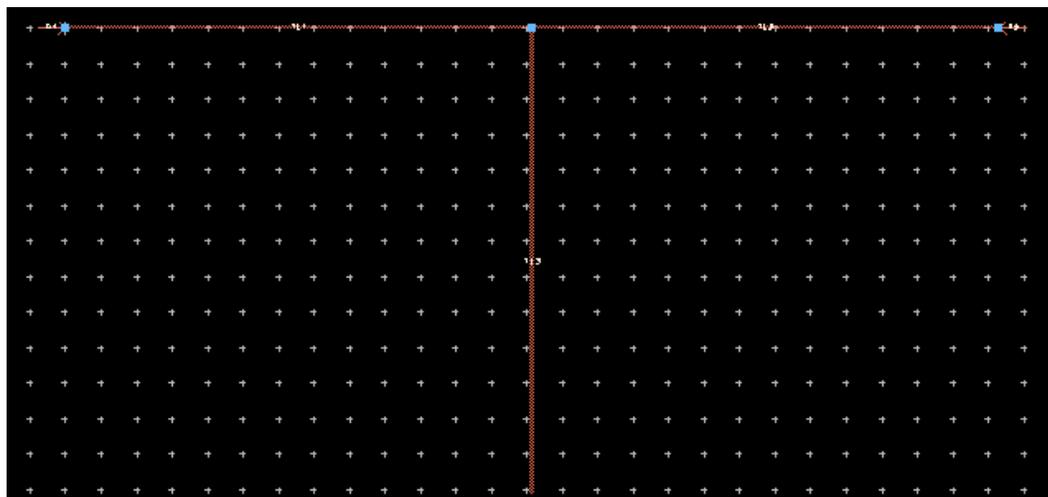
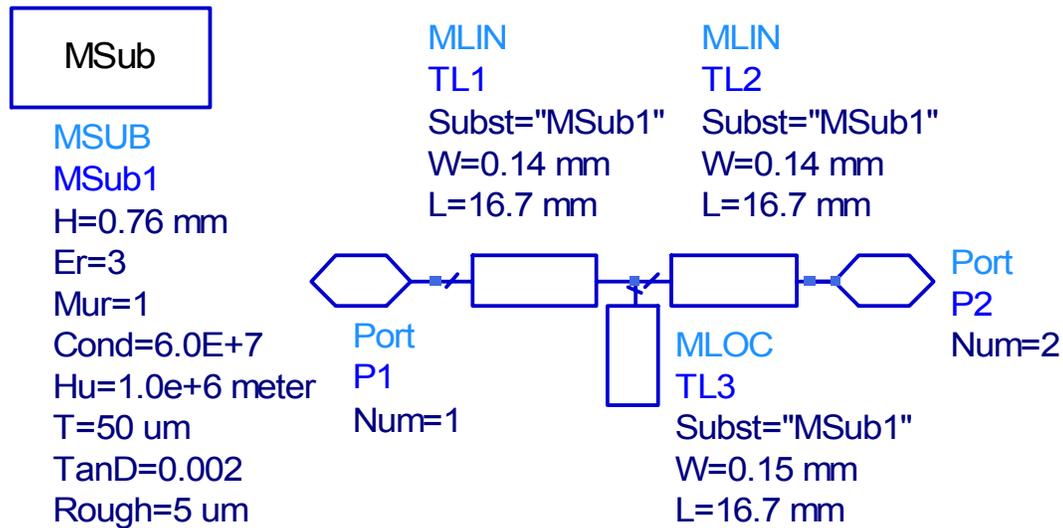
Compared with the simulation result of E-Syn lumped circuits, they match pretty well. However, there is a huge peak in both the insertion loss and the group delay, and that is why the simulation above is terminated at 1.4 GHz. A more accurate simulation can be done with the layout of the filter.

In the schematic window we can generate a customized system level filter by click *Tool > E-Syn > Place New Design for Synthesized Network*, which is the same process used in Part II.

Actually the simulation result above is from the simulation of this schematic circuit; one can remove the two ports and replace them by the terminators and do the S-parameter simulation.



However, this schematic circuit of ideal T-line cannot be directly converted into microstrip layout. In order to do it, we need to design a new schematic circuit. The width of the three segments of the microstrips are determined by the impedance, one can use the equations in section 2.8 in the textbook (Ludwig/Bretchko), or by looking up Fig. 2-20 on page 66 (which is not an accurate way). The physical length can be calculated by the electrical length (one wavelength ~ 360 degree). The following schematic is a crude design; following the procedure in Part III we can generate the layout from the schematic. However, the layout generated in this way might need to be rearranged a little.



With momentum simulation, one can get the following results. Compared with the schematic simulation, the spurious peak in IL at 1.5 GHz is gone.

